

Official Certified SolidWorks Associate (CSWA) Examination Guide

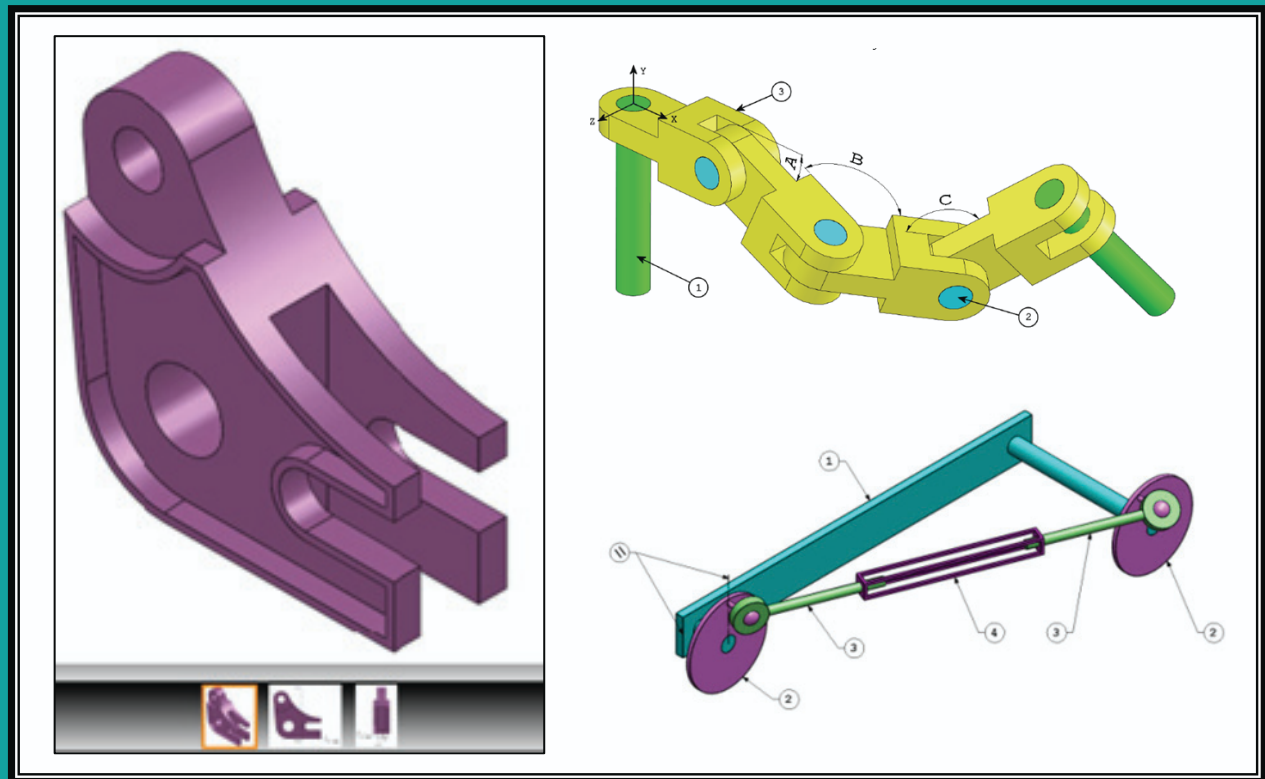
SolidWorks 2009

SolidWorks 2010

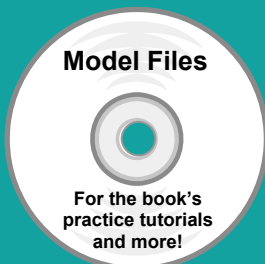
SolidWorks 2011

The only authorized CSWA exam preparation guide

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Examination

CHAPTER 3 - BASIC PART AND INTERMEDIATE PART CREATION AND MODIFICATION

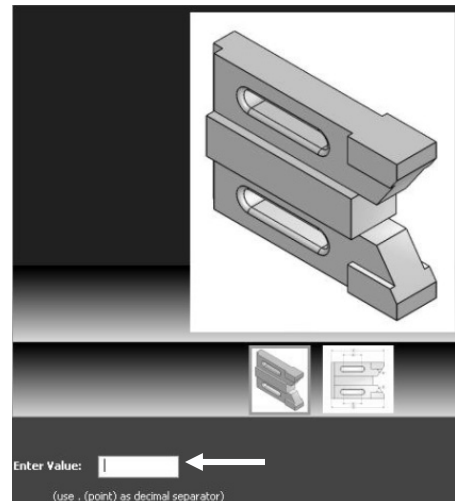
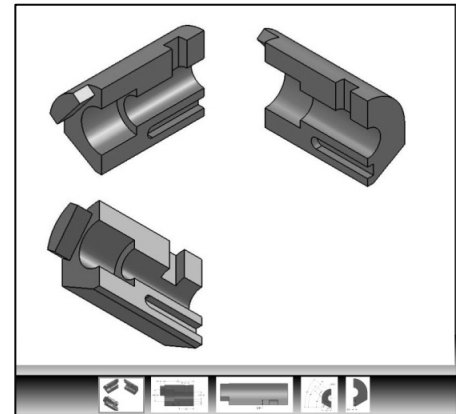
Objectives

Basic Part Creation and Modification and Intermediate Part Creation and Modification are two of the five categories on the CSWA exam. This chapter covers the knowledge to create and modify models for these categories from detailed dimensioned illustrations.

There are two questions on the CSWA exam in the *Basic Part Creation and Modification* category. One question is in a multiple choice single answer format and the other question (Modification of the model) is in the fill in the blank format. Each question is worth fifteen (15) points for a total of thirty (30) points. You are required to build a model, with six or more features and to answer a question either on the overall mass, volume, or the location the Center of mass for the created model relative to the default part Origin location. You are then requested to modify the part and answer a fill in the blank format question.

There are two questions on the CSWA exam in the *Intermediate Part Creation and Modification* category. One question is in a multiple choice single answer format and the other question (Modification of the model) is in the fill in the blank format. Each question is worth fifteen (15) points for a total of thirty (30) points. You are required to build a model, with six or more features and to answer a question either on the overall mass, volume, or the location of the Center of mass for the created model relative to the default part Origin location. You are then requested to modify the model and answer a fill in the blank format question.

The main difference between the *Basic Part Creation and Modification* category and the *Intermediate Part Creation and Modification* or the *Advance Part Creation and Modification* category is the complexity of the sketches and the number of dimensions and geometric relations along with an increase in the number of features.



Basic and Intermediate Part Creation

On the completion of the chapter, you will be able to:

- Read and understand an Engineering document used in the CSWA exam:
 - Identify the Sketch plane, part Origin location, part dimensions, geometric relations, and design intent of the sketch and feature
- Build a part from a detailed dimensioned illustration using the following SolidWorks tools and features:
 - 2D & 3D sketch tools, Extruded Boss/Base, Extruded Cut, Fillet, Mirror, Revolved Base, Chamfer, Reference geometry, Plane, Axis, Calculate the overall mass and volume of the created part, and Locate the Center of mass for the created part relative to the Origin



The complexity of the models along with the features progressively increases throughout this chapter to simulate the final types of models that would be provided on the exam.



FeatureManager names were changed through various revisions of SolidWorks. Example: Extrude1 vs. Boss-Extrude1. These changes do not affect the models or answers in this book.

Read and understand an Engineering document

What is an Engineering document? In SolidWorks a part, assembly or drawing is referred to as a document. Each document is displayed in the Graphics window.

During the exam, each question will display an information table on the left side of the screen and drawing information on the right. Read the provided information and apply it to the drawing. Various values are provided on each question.



If you don't find your answer (within 1%) in the multiple choice single answer format section - recheck your solid model for precision and accuracy.



Copy the corresponding CSWA Model Folder from the CD in the book that matches your release of SolidWorks to your hard drive. Work directly from your hard drive on the tutorials in this book.

Modify the part in SolidWorks.

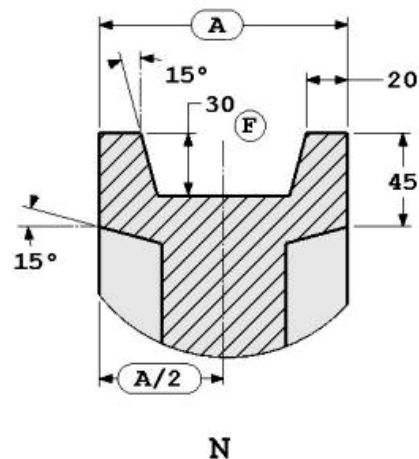
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: Aluminium 1060 Alloy
Density = 0.0027 g/mm³

Modify the part using the following variable values:

A = 140.00
B = 50.00
C = 55 degrees

Note: Assume all unshown dimensions are the same as in the previous question.

What is the overall mass of the part (grams)?

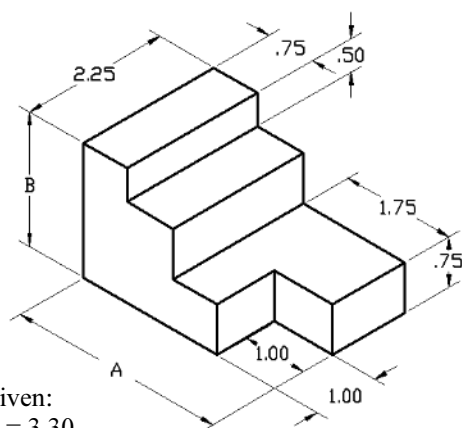


Build a Basic Part from a detailed illustration

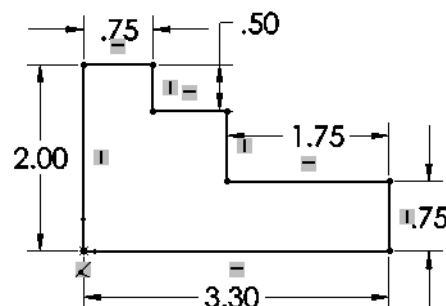
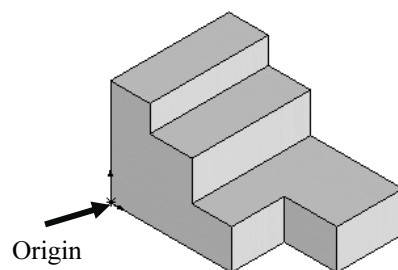
Tutorial: Volume / Center of Mass 3-1

Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated dimensioned model.
The model displays all edges on perpendicular planes. Think about the steps to build the model. Insert two features: Extruded Base (Boss-Extrude1) and Extruded Cut (Cut-Extrude1). The part Origin is located in the front left corner of the model. Think about your Base Sketch plane. Keep your Base Sketch simple.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch1 is the Base sketch. Sketch1 is the profile for the Extruded Base (Boss-Extrude1) feature. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = 2.25in. Identify the extrude direction to maintain the location of the Origin.
6. Create **Sketch2**. Select the Top right face as the Sketch plane for the second feature. Sketch a square. Sketch2 is the profile for the Extruded Cut feature. Insert the required geometric relations and dimensions.
7. Create the **Extruded Cut** feature. Select Through All for End Condition in Direction 1.



Given:
 A = 3.30
 B = 2.00
 Material: 2014 Alloy
 Density = .101 lb/in³
 Units: IPS
 Decimal places = 2



Basic and Intermediate Part Creation

8. **Assign** 2014 Alloy material to the part. Material is required to locate the Center of mass.
9. **Calculate** the volume. The volume = 8.28 cubic inches.



There are numerous ways to build the models in this chapter. A goal is to display different design intents and techniques.

10. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 1.14 inches
- Y: 0.75 inches
- Z: -1.18 inches

11. **Save** the part and name it Volume-Center of mass 3-1.

12. **Close** the model.

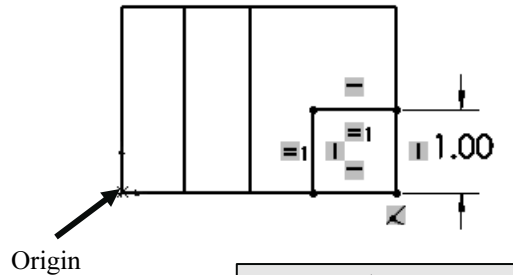


The principal axes and Center of mass are displayed graphically on the model in the Graphics window.

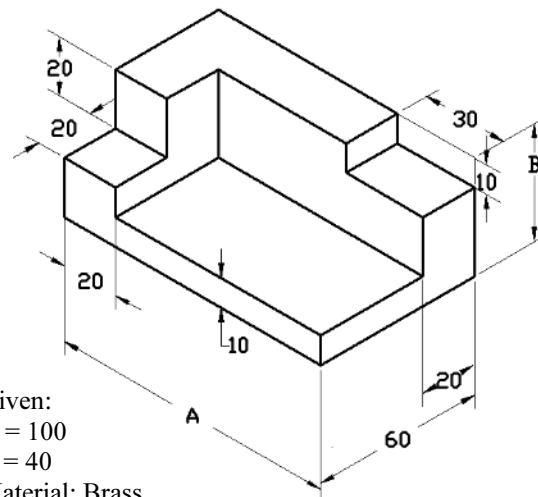
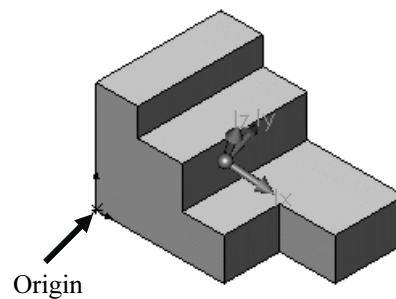
Tutorial: Volume / Center of Mass 3-2

Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.

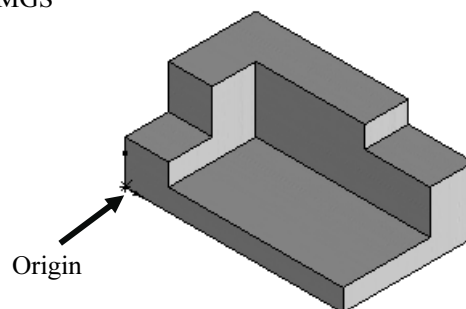
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated dimensioned model. The model displays all edges on perpendicular planes. Think about the steps that are required to build this model. Remember, there are numerous ways to create the models in this chapter.



| |
|--|
| Mass = 0.84 pounds |
| Volume = 8.28 cubic inches |
| Surface area = 29.88 inches ² |
| Center of mass: (inches) |
| X = 1.14 |
| Y = 0.75 |
| Z = -1.18 |



Given:
A = 100
B = 40
Material: Brass
Density = .0085 g/mm³
Units: MMGS

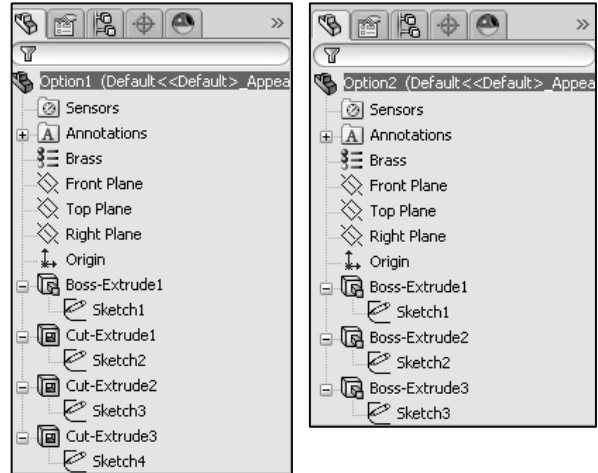




The CSWA exam is timed. Work efficiently.

View the provided Part FeatureManagers. Both FeatureManagers create the same illustrated model. In Option1, there are four sketches and four features (Extruded Base and three Extruded Cuts) that are used to build the model.

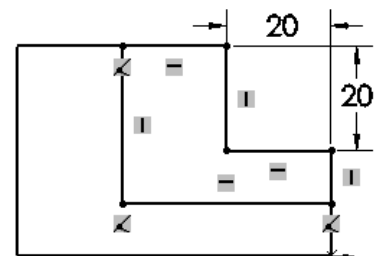
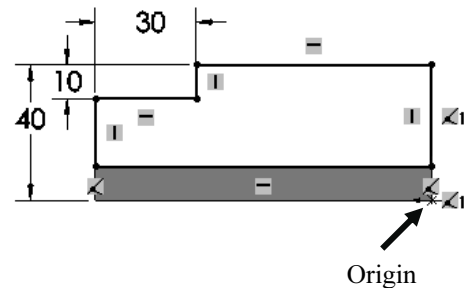
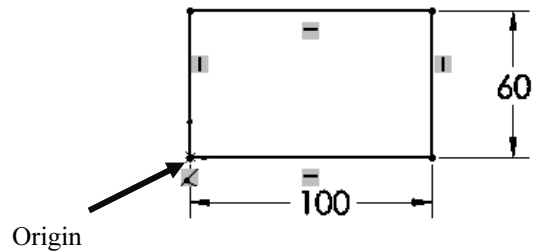
In Option2, there are three sketches and three features (Extruded Boss/Base) that are used to build the model. Which FeatureManager is better? In a timed exam, optimize your time and use the least amount of features through mirror, pattern, symmetry, etc.



Use Centerlines to create symmetrical sketch elements and revolved features, or as construction geometry.

Create the model using the Option2 Part FeatureManager.

3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Sketch a rectangle. Insert the required dimensions.
5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = 10mm.
6. Create **Sketch2**. Select the back face of Boss-Extrude1.
7. Select **Normal To** view. Sketch2 is the profile for the second Extruded Boss/Base feature. Insert the required geometric relations and dimensions as illustrated.
8. Create the second Extruded Boss/Base feature (**Boss-Extrude2**). Blind is the default End Condition in Direction 1. Depth = 20mm. Note the direction of the extrude, towards the front of the model.



Basic and Intermediate Part Creation

9. Create **Sketch3**. Select the left face of Boss-Extrude1 as the Sketch plane. Sketch3 is the profile for the third Extrude feature. Insert the required geometric relations and dimensions.

10. Create the third Extruded Boss/Base feature (**Boss-Extrude3**). Blind is the default End Condition in Direction 1. Depth = 20mm.

11. **Assign** Brass material to the part.

12. **Calculate** the volume of the model. The volume = 130,000.00 cubic millimeters.

13. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 43.36 millimeters
- Y: 15.00 millimeters
- Z: -37.69 millimeters

14. **Save** the part and name it Volume-Center of mass 3-2.

15. **Calculate** the volume of the model using the IPS unit system. The volume = 7.93 cubic inches.

16. **Locate** the Center of mass using the IPS unit system. The location of the Center of mass is derived from the part Origin.

- X: 1.71 inches
- Y: 0.59 inches
- Z: -1.48 inches

16. **Save** the part and name it Volume-Center of mass 3-2-IPS.

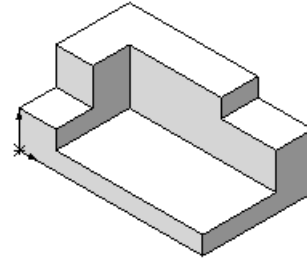
17. **Close** the model.



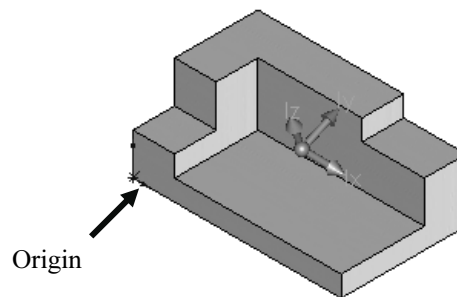
There are numerous ways to create the models in this chapter. A goal is to display different design intents and techniques.



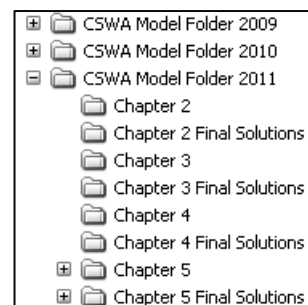
All SW models (initial and final) are provided on the CD in the book. Copy the folders and model files to your local hard drive. Do not work directly from the CD.



| |
|---------------------------------------|
| Mass = 1105.00 grams |
| Volume = 130000.00 cubic millimeters |
| Surface area = 23400.00 millimeters^2 |
| Center of mass: (millimeters) |
| X = 43.46 |
| Y = 15.00 |
| Z = -37.69 |



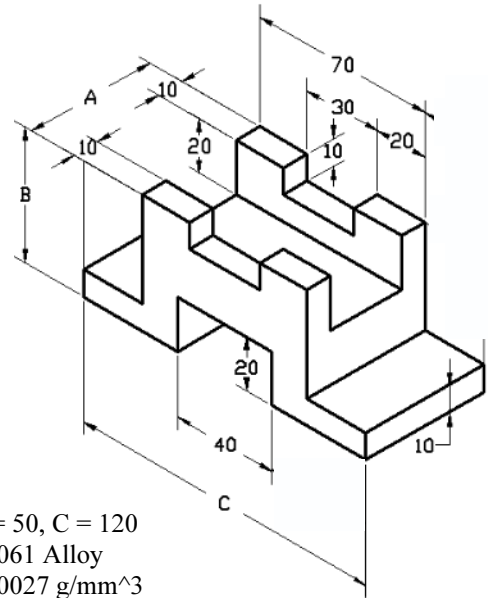
| |
|-------------------------------|
| Mass = 2.44 pounds |
| Volume = 7.93 cubic inches |
| Surface area = 36.27 inches^2 |
| Center of mass: (inches) |
| X = 1.71 |
| Y = 0.59 |
| Z = -1.48 |




Tutorial: Mass-Volume 3-3

Build this model. Calculate the overall mass of the illustrated model with the provided information.

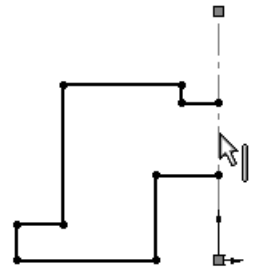
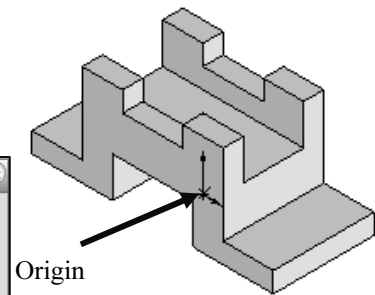
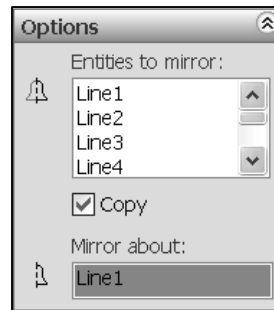
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. The model displays all edges on perpendicular planes. Think about the steps required to build the model. Apply the Mirror Sketch tool to the Base sketch. Insert an Extruded Base (Boss-Extrude1) and Extruded-Cut (Cut-Extrude1) feature.
3. **Set** the document properties for the model.





Given:
 $A = 50$, $B = 50$, $C = 120$
 Material: 6061 Alloy
 Density = $.0027 \text{ g/mm}^3$
 Units: MMGS

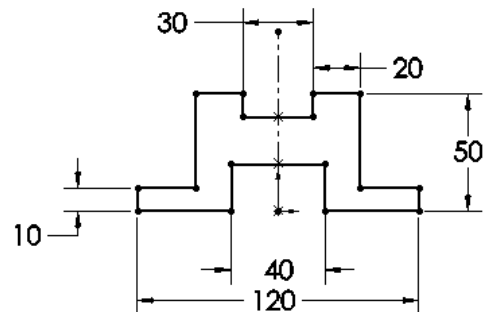
 To activate the Mirror Sketch tool, click **Tools, Sketch Tools, Mirror** from the Menu bar menu. The Mirror PropertyManager is displayed.

4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Apply the Mirror Sketch tool. Select the construction geometry to mirror about as illustrated. Select the Entities to mirror. Insert the required geometric relations and dimensions.



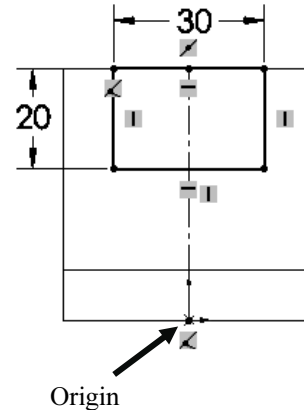
 Construction geometry is ignored when the sketch is used to create a feature. Construction geometry uses the same line style as centerlines.

 When you create a new part or assembly, the three default Planes (Front, Right and Top) are aligned with specific views. The Plane you select for the Base sketch determines the orientation of the part.



Basic and Intermediate Part Creation

5. Create the **Boss-Extrude1** feature. Boss-Extrude1 is the Base feature. Apply the Mid Plane End Condition in Direction 1 for symmetry. Depth = 50mm.
6. Create **Sketch2**. Select the right face for the Sketch plane. Sketch2 is the profile for the Extruded Cut feature. Insert the required geometric relations and dimensions. Apply construction geometry.
7. Create the **Extruded Cut** feature. Through All is the selected End Condition in Direction 1.
8. **Assign** 6061 Alloy material to the part.
9. **Calculate** the overall mass. The overall mass = 302.40 grams.
10. **Save** the part and name it Mass-Volume 3-3.
11. **Close** the model.

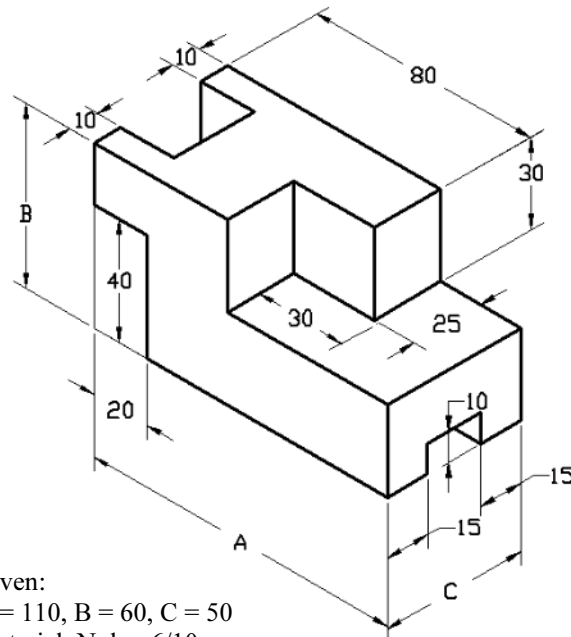
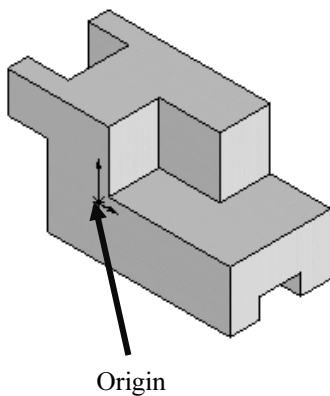


| |
|--|
| Mass = 302.40 grams |
| Volume = 112000.00 cubic millimeters |
| Surface area = 26200.00 millimeters ² |
| Center of mass: (millimeters) |
| X = 0.00 |
| Y = 19.20 |
| Z = 0.00 |

Tutorial: Mass-Volume 3-4

Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.

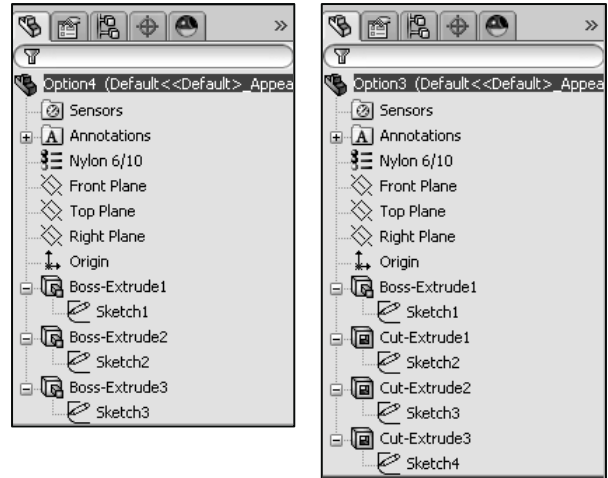
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. The model displays all edges on perpendicular planes.



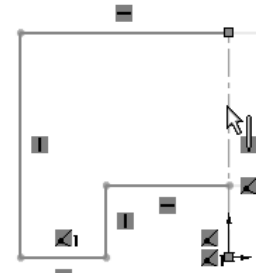
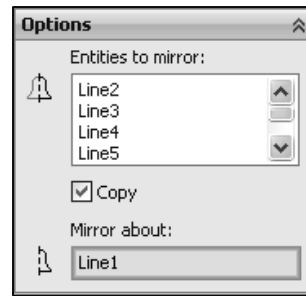
Given:
A = 110, B = 60, C = 50
Material: Nylon 6/10
Density = .0014 g/mm³
Units: MMGS

View the provided Part FeatureManagers. Both FeatureManagers create the same model. In Option4, there are three sketches and three features that are used to build the model.

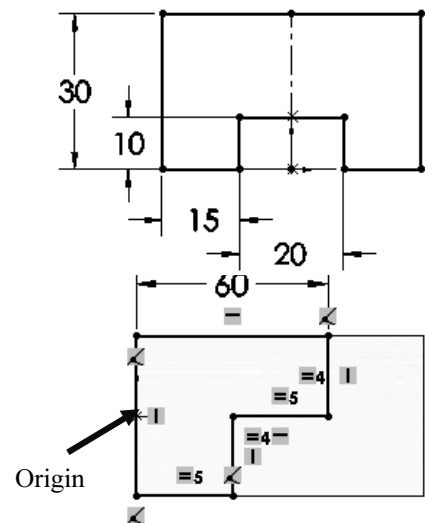
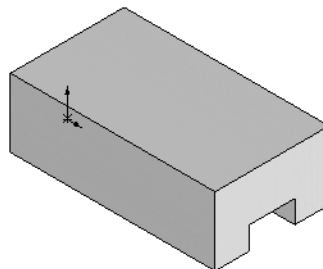
In Option3, there are four sketches and four features that are used to build the model. Which FeatureManager is better? In a timed exam, optimize your design time and use the least amount of features. Use the Option4 FeatureManager in this tutorial. As an exercise, build the model using the Option3 FeatureManager.



3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Right Plane as the Sketch plane. Sketch1 is the Base sketch. Apply the Mirror Entities Sketch tool. Select the construction geometry to mirror about as illustrated. Select the Entities to mirror. Insert the required geometric relations and dimensions.



5. Create the **Boss-Extrude1** feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = $(A - 20\text{mm}) = 90\text{mm}$. Note the direction of the extrude feature.
6. Create **Sketch2**. Select the Top face of Boss-Extrude1 for the Sketch plane. Sketch2 is the profile for the second Extruded Boss/Base feature (Boss-Extrude2). Insert the required geometric relations and dimensions.



Basic and Intermediate Part Creation

7. Create the **Boss-Extrude2** feature. Blind is the default End Condition in Direction 1. Depth = 30mm.
8. Create **Sketch3**. Select the left face of Boss-Extrude1 for the Sketch plane. Apply symmetry. Insert the required geometric relations and dimensions. Use construction reference geometry.



The 20mm dimension for Sketch3 was calculated by:
 $(B - 40\text{mm}) = 20\text{mm}$.

9. Create the **Boss-Extrude3** feature. Blind is the default End Condition in Direction 1. Depth = 20mm. Note the direction of Extrude3.

10. **Assign** Nylon 6/10 material to the part.

11. **Calculate** the volume. The volume = 192,500.00 cubic millimeters.

12. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

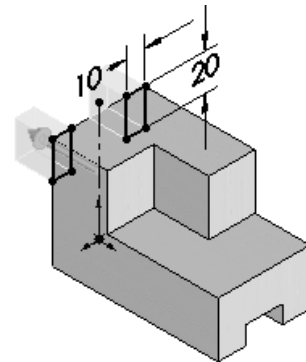
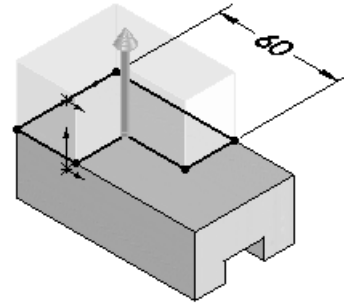
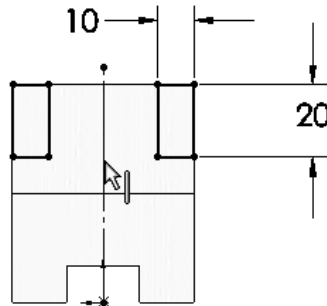
- X: 35.70 millimeters
- Y: 27.91 millimeters
- Z: -1.46 millimeters

13. **Save** the part and name it Mass-Volume 3-4.

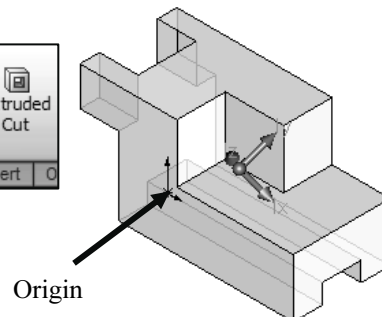
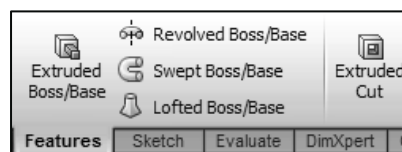
14. **Close** the model.

In the previous section, the models that you created displayed all edges on Perpendicular planes and used the Extruded Base, Extruded Boss, or the Extruded Cut feature from the Features toolbar.

In the next section, build models where all edges are not located on Perpendicular planes.



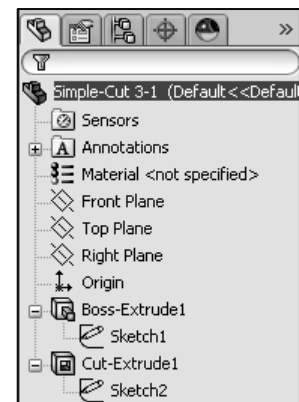
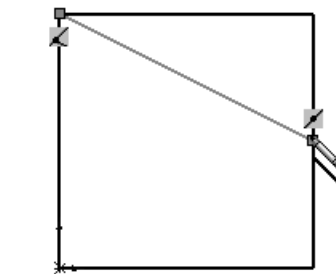
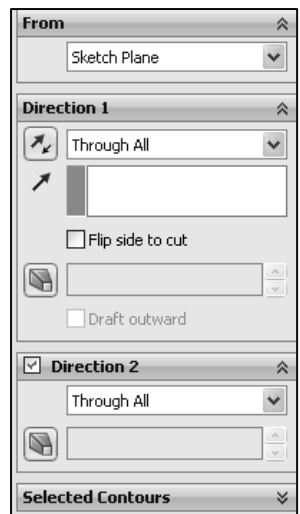
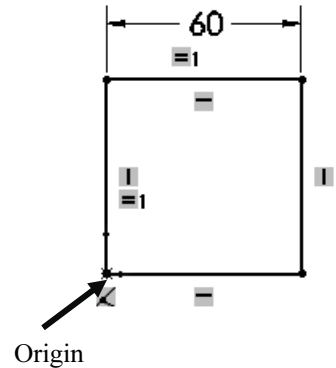
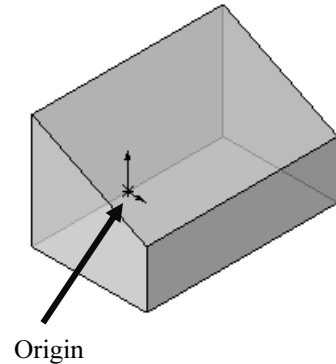
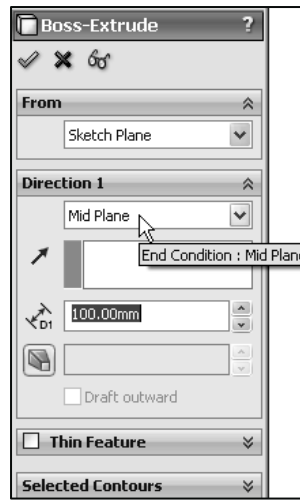
| |
|---------------------------------------|
| Mass = 269.50 grams |
| Volume = 192500.00 cubic millimeters |
| Surface area = 27800.00 millimeters^2 |
| Center of mass: (millimeters) |
| X = 35.70 |
| Y = 27.91 |
| Z = -1.46 |



First, let's review a simple 2D Sketch for an Extruded Cut feature.

Tutorial: Simple Cut 3-1

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model.
Start with a 60mm x 60mm x 100mm block. System units = MMGS. Decimal place = 2. Note the location of the part Origin.
3. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch a square as illustrated. Insert the required dimension. The part Origin is located in the bottom left corner of the model.
4. Create the **Extruded Base (Boss-Extrude1)** feature. Apply the Mid Plane End Condition in Direction 1. Depth = 100mm.
5. Create **Sketch2**. Select the front face as the Sketch plane. Apply the Line Sketch tool. Sketch a diagonal line. Select the front right vertical midpoint as illustrated.
6. Create the **Extruded Cut (Cut-Extrude1)** feature. Through All for End Condition in Direction 1 and Direction 2 is selected by default.
7. **Save** the part and name it Simple-Cut 3-1. View the FeatureManager.
8. **Close** the model.

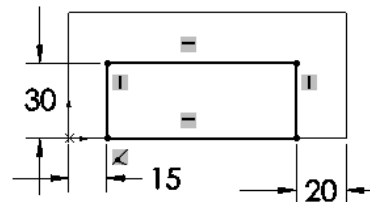
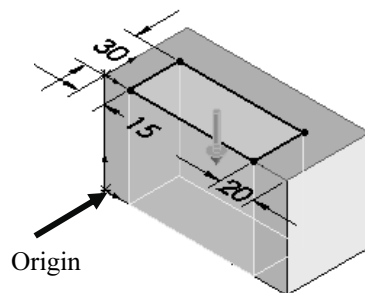
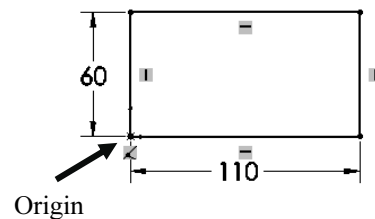
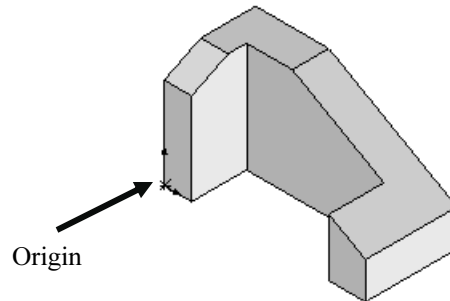
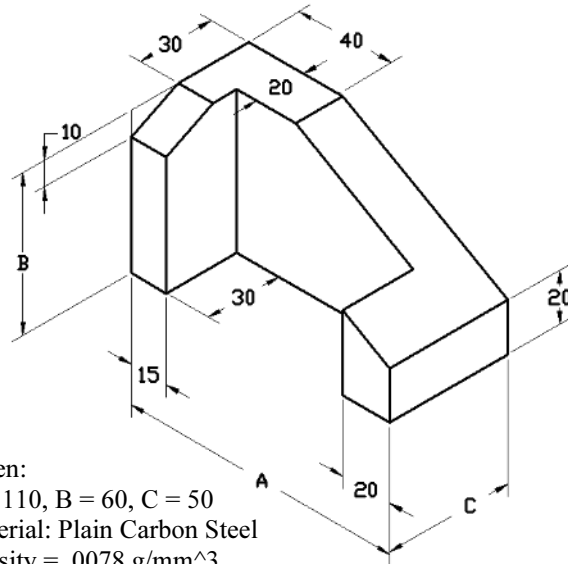


Tutorial: Mass-Volume 3-5

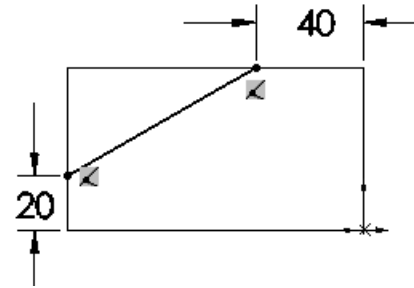
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. All edges of the model are not located on Perpendicular planes. Insert an Extruded Base (Boss-Extrude1) feature and three Extruded Cut features to build the model.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch a rectangle. Insert the required geometric relations and dimensions. The part Origin is located in the left bottom corner of the model.
5. Create the **Extruded Base** feature. Blind is the default End Condition in Direction 1. Depth = 50mm. Note the direction of Extrude1.
6. Create **Sketch2**. Select the top face of Boss-Extrude1 for the Sketch plane. Sketch2 is the profile for the first Extruded Cut feature. Insert the required relations and dimensions.
7. Create the **Extruded Cut** feature. Select Through All for End Condition in Direction 1.

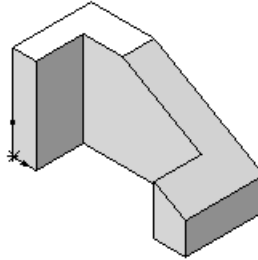
Given:
 $A = 110$, $B = 60$, $C = 50$
 Material: Plain Carbon Steel
 Density = $.0078 \text{ g/mm}^3$
 Units: MMGS



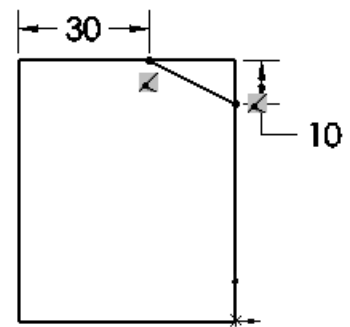
8. Create **Sketch3**. Select the back face of Boss-Extrude1 as the Sketch plane. Sketch a diagonal line. Insert the required geometric relations and dimensions.



9. Create the second **Extruded Cut** feature. Through All for End Condition in Direction 1 and Direction 2 is selected by default. Note the direction of the extrude feature.



10. Create **Sketch4**. Select the left face of Boss-Extrude1 as the Sketch plane. Sketch a diagonal line. Insert the required geometric relations and dimensions.
11. Create the third **Extruded Cut** feature. Through All for End Condition in Direction 1 and Direction 2 is selected by default.



12. **Assign** Plain Carbon Steel material to the part.
13. **Calculate** the overall mass. The overall mass = 1130.44 grams.
14. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

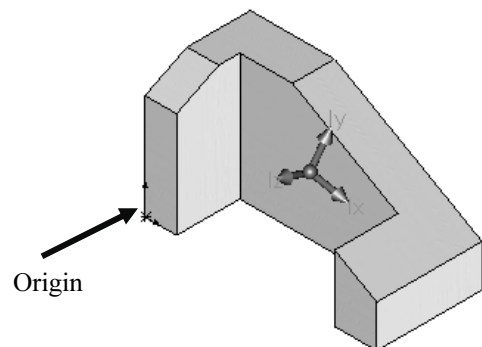
- X: 45.24 millimeters
- Y: 24.70 millimeters
- Z: -33.03 millimeters

| |
|--|
| Density = 0.01 grams per cubic millimeter |
| Mass = 1130.44 grams |
| Volume = 144928.57 cubic millimeters |
| Surface area = 23631.77 millimeters ² |
| Center of mass: (millimeters) |
| X = 45.24 |
| Y = 24.70 |
| Z = -33.03 |

In this category an exam question could read: Build this model. Locate the Center of mass with respect to the part Origin.

- A: X = 45.24 millimeters, Y = 24.70 millimeters, Z = -33.03 millimeters
- B: X = 54.24 millimeters, Y = 42.70 millimeters, Z = 33.03 millimeters
- C: X = 49.24 millimeters, Y = -37.70 millimeters, Z = 38.03 millimeters
- D: X = 44.44 millimeters, Y = -24.70 millimeters, Z = -39.03 millimeters

The correct answer is A.



Basic and Intermediate Part Creation



The principal axes and Center of mass are displayed graphically on the model in the Graphics window.

15. **Save** the part and name it

Mass-Volume 3-5.

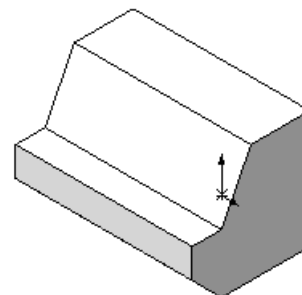
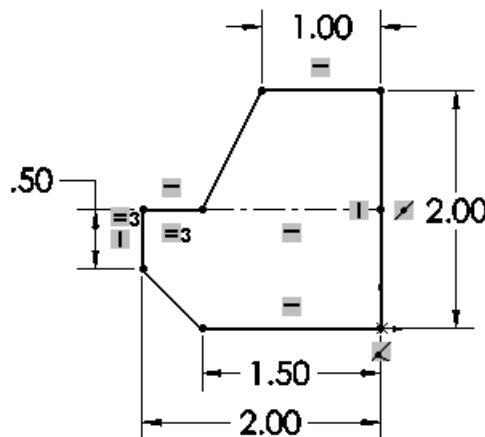
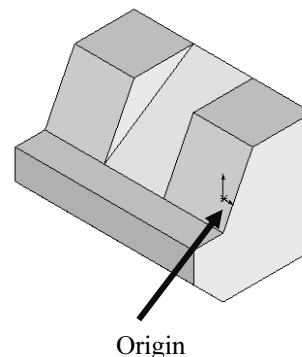
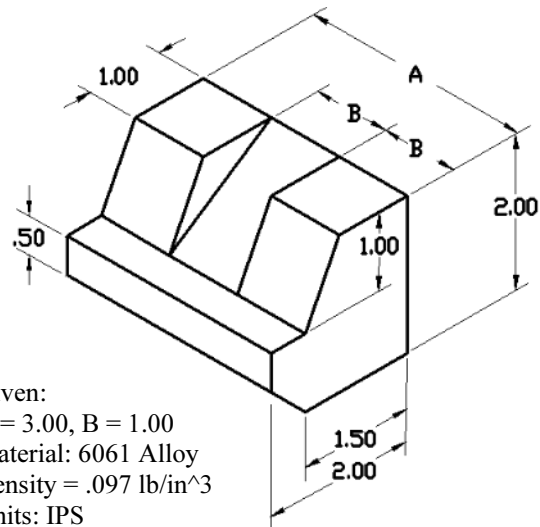
16. **Close** the model.

Tutorial: Mass-Volume 3-6

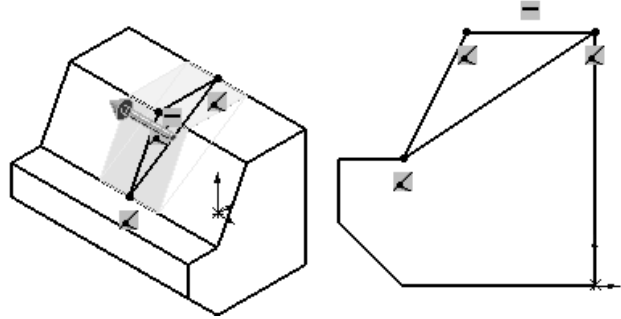
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. All edges of the model are not located on Perpendicular planes. Think about the steps required to build the model. Insert two features: Extruded Base (Boss-Extrude1) and Extruded Cut (Cut-Extrude1).
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Right Plane as the Sketch plane. Apply construction geometry. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Apply symmetry. Select Mid Plane as the End Condition in Direction 1. Depth = 3.00in.

Given:
A = 3.00, B = 1.00
Material: 6061 Alloy
Density = .097 lb/in³
Units: IPS
Decimal places = 2



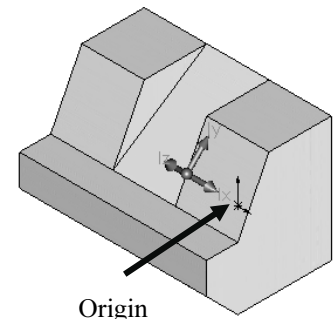
6. Create **Sketch2**. Select the Right Plane as the Sketch plane. Select the Line Sketch tool. Insert the required geometric relations. Sketch2 is the profile for the Extruded Cut feature.
 7. Create the **Extruded Cut (Cut-Extrude1)** feature. Apply symmetry. Select Mid Plane as the End Condition in Direction 1. Depth = 1.00in.
 8. **Assign** 6061 Alloy material to the part.
 9. **Calculate** overall mass. The overall mass = 0.87 pounds.
 10. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
- X: 0.00 inches
 - Y: 0.86 inches
 - Z: 0.82 inches



| |
|-------------------------------|
| Mass = 0.87 pounds |
| Volume = 8.88 cubic inches |
| Surface area = 28.91 inches^2 |
| Center of mass: (inches) |
| X = 0.00 |
| Y = 0.86 |
| Z = 0.82 |

In this category an exam question could read: Build this model. Locate the Center of mass with respect to the part Origin.

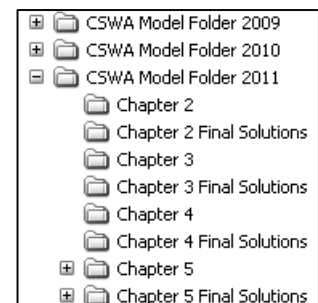
- A: X = 0.10 inches, Y = -0.86 inches, Z = -0.82 inches
- B: X = 0.00 inches, Y = 0.86 inches, Z = 0.82 inches
- C: X = 0.15 inches, Y = -0.96 inches, Z = -0.02 inches
- D: X = 1.00 inches, Y = -0.89 inches, Z = -1.82 inches



The correct answer is B.

11. **Save** the part and name it Mass-Volume 3-6.
12. **Close** the model.

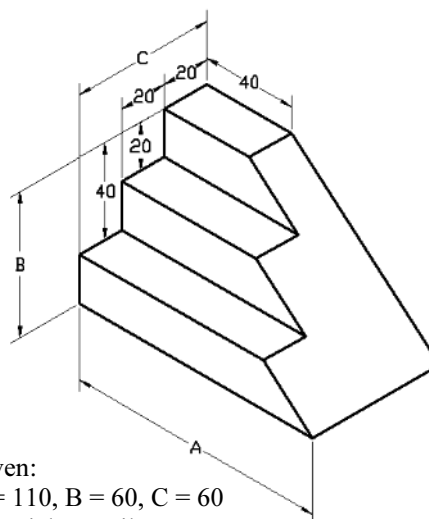
As an exercise, modify the Mass-Volume 3-6 part using the MMGS unit system. Assign Nickel as the material. Calculate the overall mass. The overall mass of the part = 1236.20 grams. Save the part and name it Mass-Volume 3-6-MMGS.



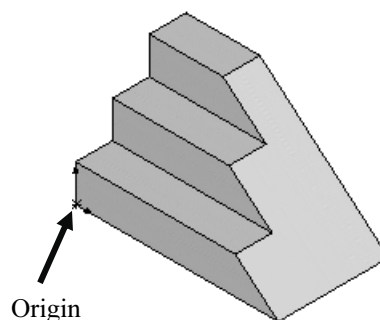
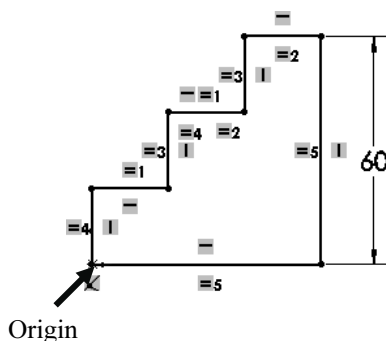
Tutorial: Mass-Volume 3-7

Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

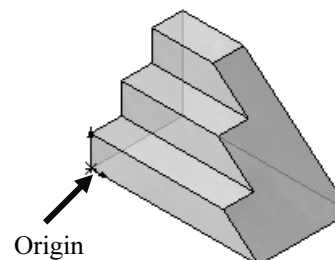
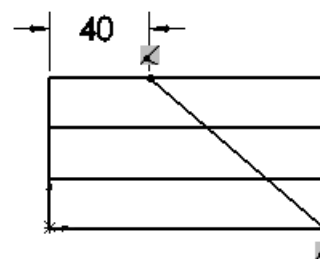
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. All edges of the model are not located on Perpendicular planes. Think about the steps required to build the model. Insert two features: Extruded Base (Boss-Extrude1) and Extruded Cut (Cut-Extrude1).
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Right Plane as the Sketch plane. Apply the Line Sketch tool. Insert the required geometric relations and dimension. The location of the Origin is in the left lower corner of the sketch.



Given:
A = 110, B = 60, C = 60
Material: Ductile Iron
Density = .0079 g/mm³
Units: MMGS



5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = 110mm.
6. Create **Sketch2**. Select the Front Plane as the Sketch plane. Sketch a diagonal line. Complete the sketch. Sketch2 is the profile for the Extruded Cut feature.
7. Create the **Extruded Cut** feature. Through All for End Condition in Direction 1 and Direction 2 is selected by default.

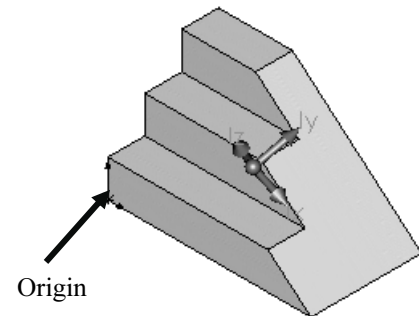


8. **Assign** Ductile Iron material to the part.
9. **Calculate** overall mass. The mass = 1569.47 grams.
10. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
 - X: 43.49 millimeters
 - Y: 19.73 millimeters
 - Z: -35.10 millimeters
11. **Save** the part and name it Mass-Volume 3-7.
12. **Close** the model.

| |
|--|
| Mass = 1569.47 grams |
| Volume = 198666.67 cubic millimeters |
| Surface area = 25487.82 millimeters ² |
| Center of mass: (millimeters) |
| X = 43.49 |
| Y = 19.73 |
| Z = -35.10 |

In this category an exam question could read: Build this model. Locate the Center of mass with respect to the part Origin.

- A: X = -43.99 millimeters, Y = 29.73 millimeters, Z = -38.10 millimeters
- B: X = -44.49 millimeters, Y = -19.73 millimeters, Z = 35.10 millimeters
- C: X = 43.49 millimeters, Y = 19.73 millimeters, Z = -35.10 millimeters
- D: X = -1.00 millimeters, Y = 49.73 millimeters, Z = -35.10 millimeters



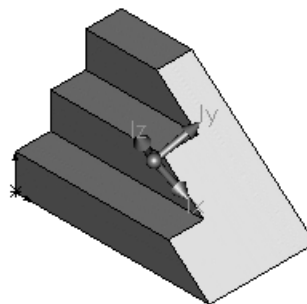
The correct answer is C.

As an exercise, locate the Center of mass using the IPS unit system, and re-assign copper material.

Re-calculate the Center of mass location, with respect to the part Origin. Save the part and name it Mass-Volume 3-7-IPS.

- X: 1.71 inches
- Y: 0.78 inches
- Z: -1.38 inches

| |
|--|
| Mass = 3.90 pounds |
| Volume = 12.12 cubic inches |
| Surface area = 39.51 inches ² |
| Center of mass: (inches) |
| X = 1.71 |
| Y = 0.78 |
| Z = -1.38 |



When you create a new part or assembly, the three default Planes (Front, Right and Top) are aligned with specific views. The Plane you select for the Base sketch determines the orientation of the part.

2D vs. 3D Sketching

Up to this point, the models that you created in this chapter started with a 2D Sketch. Sketches are the foundation for creating features. SolidWorks provides the ability to create either 2D or 3D Sketches. A 2D Sketch is limited to a flat 2D Sketch plane. A 3D sketch can include 3D elements.

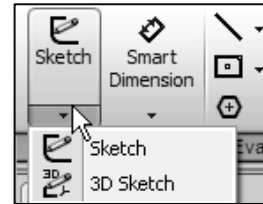
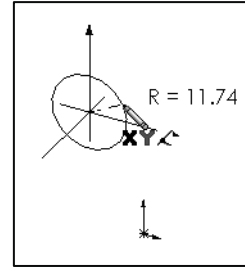


As you create a 3D Sketch, the entities in the sketch exist in 3D space. They are not related to a specific Sketch plane as they are in a 2D Sketch.

You may need to apply a 3D Sketch in the CSWA exam. Below is an example of a 3D Sketch to create a Cut-Extrude feature.

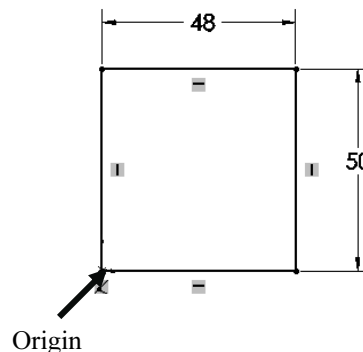
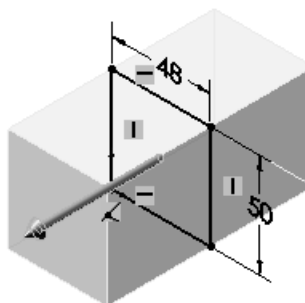
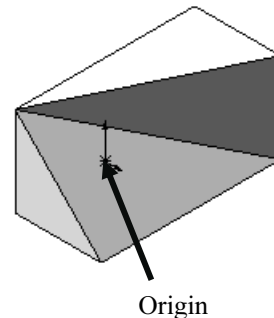


The complexity of the models increases throughout this chapter to simulate the types of models that are provided on the CSWA exam.



Tutorial 3DSketch 3-1

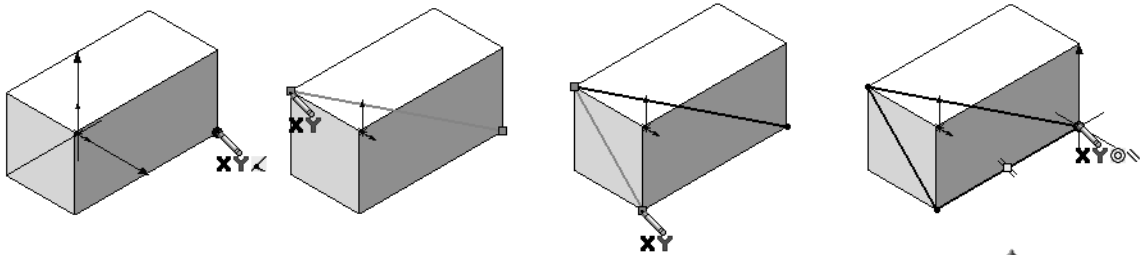
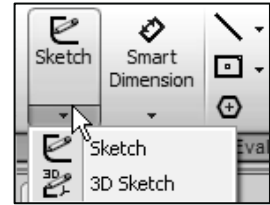
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert two features: Extruded Base and Extruded Cut. Apply the 3D Sketch tool to create the Extruded Cut feature. System units = MMGS.
Decimal place = 2.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch a rectangle. The part Origin is located in the bottom left corner of the sketch. Insert the illustrated geometric relations and dimensions.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Apply symmetry. Select the Mid Plane End Condition in Direction 1. Depth = 100.00mm.



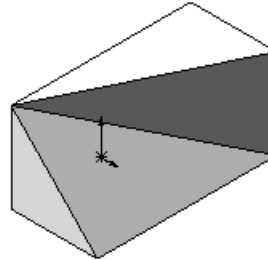


Click **3D Sketch** from the Sketch toolbar. Select the proper Sketch tool.

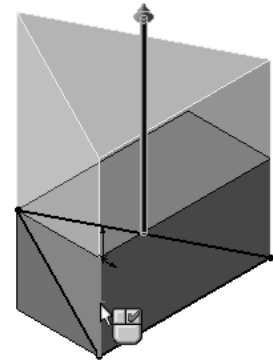
6. Create **3DSketch1**. Apply the Line Sketch tool. 3DSketch1 is a four point sketch as illustrated. 3DSketch1 is the profile for Extruded Cut feature.



7. Create the **Extruded Cut (Cut-Extrude1)** feature. Select the front right vertical edge as illustrated to remove the material. Edge<1> is displayed in the Direction of Extrusion box.



8. **Save** the part and name it 3DSketch 3-1.
9. **Close** the model.

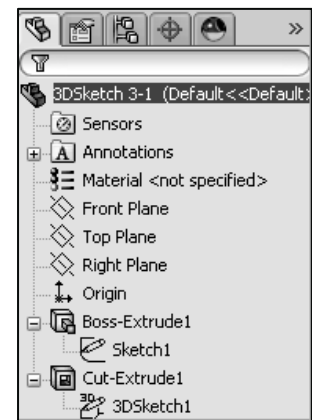
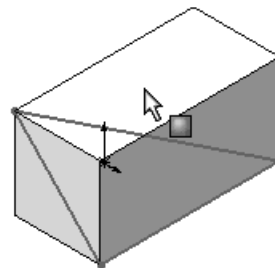


You can either select the front right vertical edge or the Top face to remove the required material in this tutorial.

Use any of the following tools to create 3D Sketches: Lines, Circles, Rectangles, Arcs, Splines, and Points.

Most relations that are available in 2D Sketching are available in 3D sketching. The exceptions are:

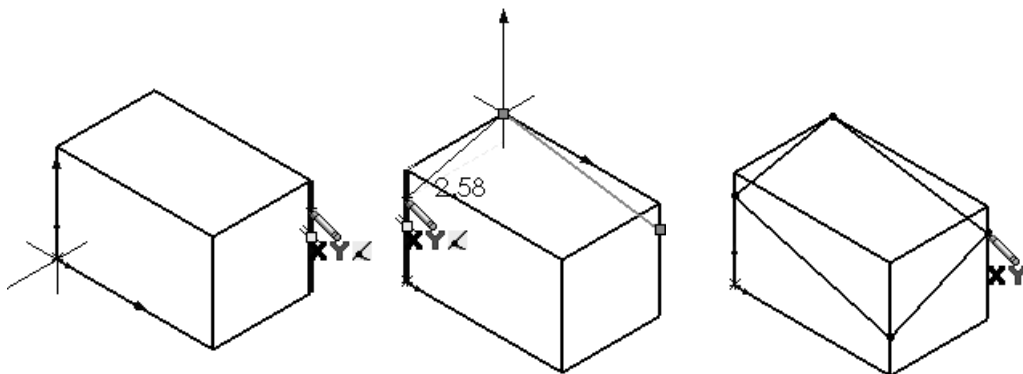
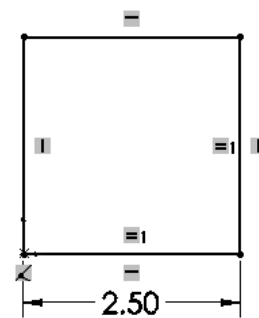
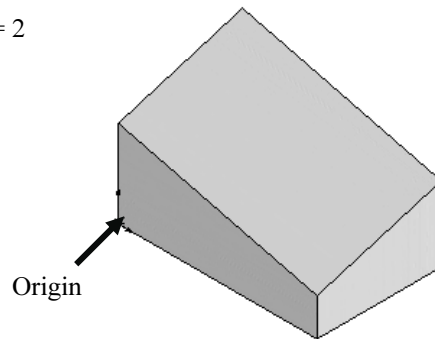
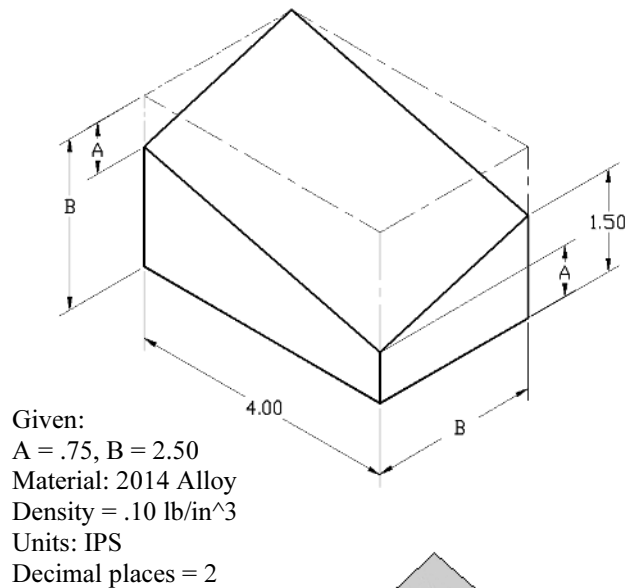
- *Symmetry*
- *Patterns*
- *Offset*



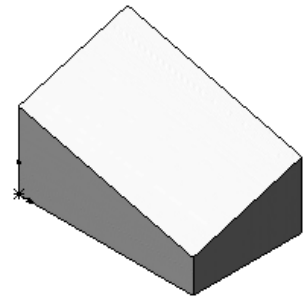
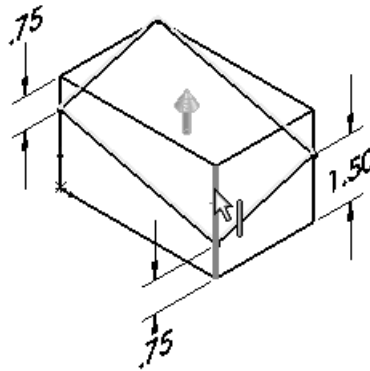
Tutorial: Mass-Volume 3-8

Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. All edges of the model - are not located on Perpendicular planes. Insert two features: Extruded Base (Boss-Extrude1) and Extruded Cut (Cut-Extrude1). Apply a closed four point 3D sketch as the profile for the Extruded Cut feature. The part Origin is located in the lower left front corner of the model.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Right Plane as the Sketch plane. Sketch a square. Insert the required geometric relations and dimension.
5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = 4.00in.
6. Create **3DSketch1**. Apply the Line Sketch tool. Create a closed five point 3D sketch as illustrated. 3DSketch1 is the profile for the Extruded Cut feature. Insert the required dimensions.



7. Create the **Extruded Cut** feature. Select the front right vertical edge as illustrated. Select Through All for End Condition in Direction 1. Note the direction of the extrude feature.



8. **Assign** the defined material to the part.

13. **Calculate** the volume. The volume = 16.25 cubic inches.

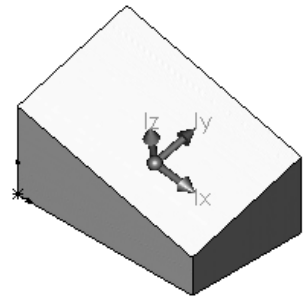
14. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X = 1.76 inches
- Y = 0.85 inches
- Z = -1.35 inches

Mass = 1.64 pounds
 Volume = 16.25 cubic inches
 Surface area = 41.86 inches²
 Center of mass: (inches)
 X = 1.79
 Y = 0.85
 Z = -1.35

In this category an exam question could read: Build this model. What is the volume of the part?

- A: 18.88 cubic inches
- B: 19.55 cubic inches
- C: 17.99 cubic inches
- D: 16.25 cubic inches



The correct answer is D.

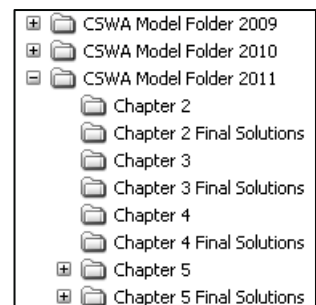
View the triad location of the Center of mass for the part.

13. **Save** the part and name it Mass-Volume 3-8.

14. **Close** the model.

As an exercise, calculate the overall mass of the part using the MMGS unit system, and re-assign Nickel as the material. The overall mass of the part = 2263.46 grams. Save the part and name it Mass-Volume 3-8-MMGS.

Mass = 2263.46 grams
 Volume = 266289.79 cubic millimeters
 Surface area = 27006.69 millimeters²
 Center of mass: (millimeters)
 X = 45.59
 Y = 21.66
 Z = -34.19



Tutorial: Mass-Volume 3-9

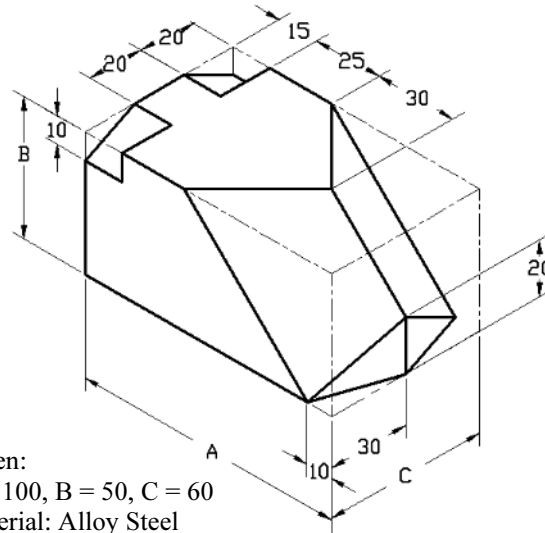
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert five sketches and five features to build the model: Extruded Base, three Extruded Cut features and a Mirror feature.

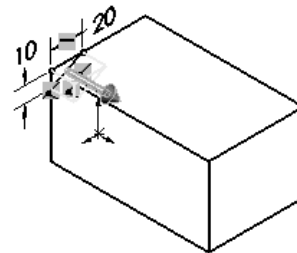
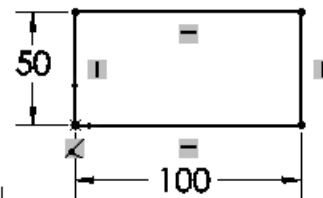
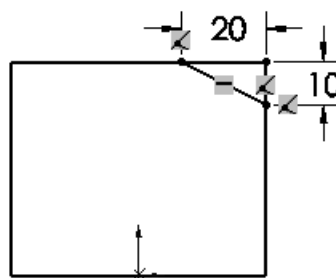
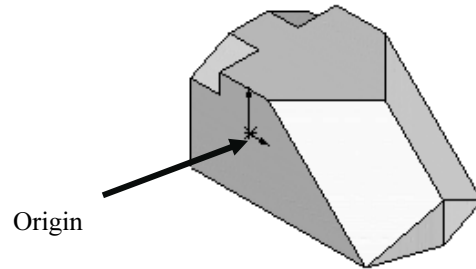


There are numerous ways to build the models in this chapter. A goal is to display different design intents and techniques.

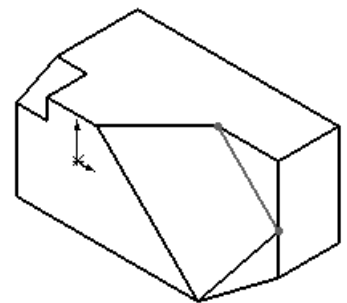
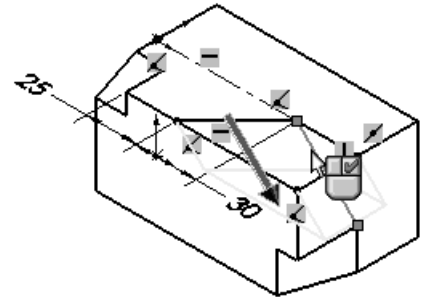
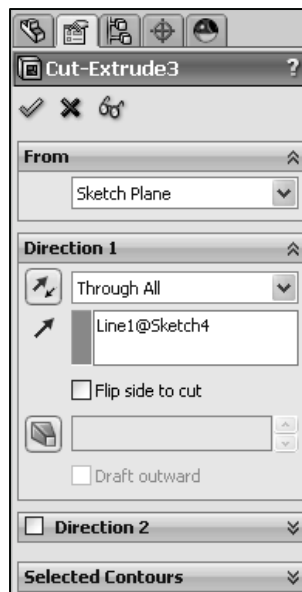
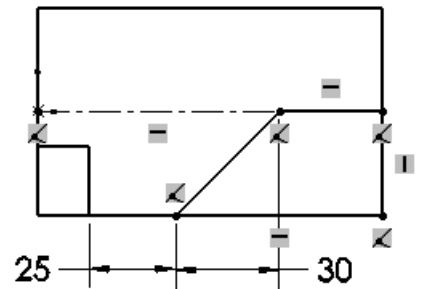
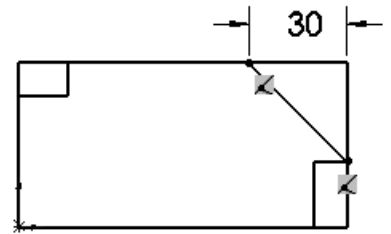
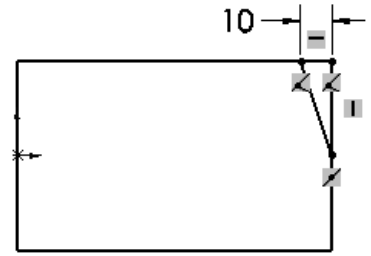
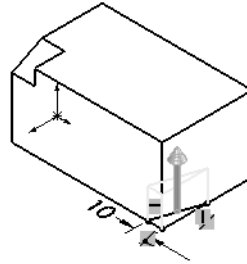
Given:
 $A = 100$, $B = 50$, $C = 60$
 Material: Alloy Steel
 Density = $.007 \text{ g/mm}^3$
 Units: MMGS



3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch a rectangle. Insert the required dimensions. The part Origin is located in the lower left corner of the sketch.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Apply symmetry. Select the Mid Plane End Condition for Direction 1. Depth = 60mm.
6. Create **Sketch2**. Select the left face of Boss-Extrude1 as the Sketch plane. Insert the required geometric relations and dimensions.
7. Create the first **Extruded Cut** feature. Blind is the default End Condition in Direction 1. Depth = 15mm. Note the direction of the extrude feature.



8. Create **Sketch3**. Select the bottom face of Boss-Extrude1 for the Sketch plane. Insert the required geometric relations and dimension.
9. Create the second **Extruded Cut** feature. Blind is the default End Condition in Direction 1. Depth = 20mm.
10. Create **Sketch4**. Select Front Plane as the Sketch plane. Sketch a diagonal line. Sketch4 is the direction of extrusion for the third Extruded Cut feature. Insert the required dimension.
11. Create **Sketch5**. Select the top face of Boss-Extrude1 as the Sketch plane. Sketch5 is the sketch profile for the third Extruded Cut feature. Apply construction geometry. Insert the required geometric relations and dimensions.
12. Create the third **Extruded Cut** feature. Select Through All for End Condition in Direction 1.
13. Select **Sketch4** in the Graphics window for Direction of Extrusion. Line1@Sketch4 is displayed in the Cut-Extrude PropertyManager.

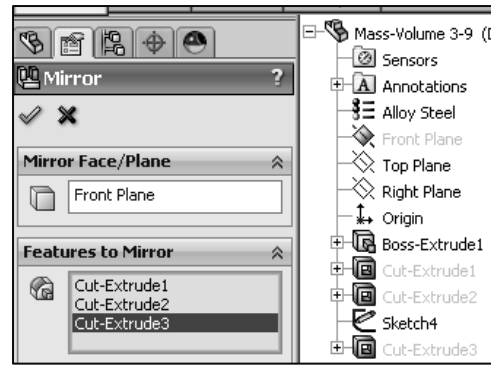
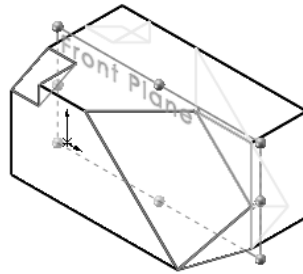


Basic and Intermediate Part Creation

14. Create the **Mirror** feature. Mirror the three Extruded Cut features about the Front Plane. Use the fly-out FeatureManager.

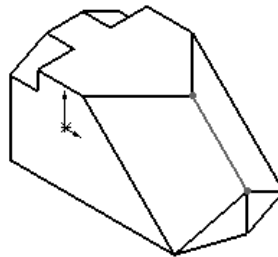
15. **Assign** Alloy Steel material to the part.

16. **Calculate** the overall mass. The overall mass = 1794.10 grams.



17. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X = 41.17 millimeters
- Y = 22.38 millimeters
- Z = 0.00 millimeters

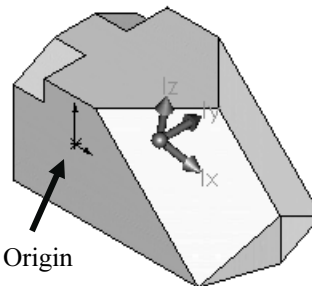


Mass = 1794.10 grams
Volume = 233000.00 cubic millimeters
Surface area = 23060.20 millimeters^2
Center of mass: (millimeters)
X = 41.17
Y = 22.38
Z = 0.00

View the triad location of the Center of mass for the part.

18. **Save** the part and name it Mass-Volume 3-9.

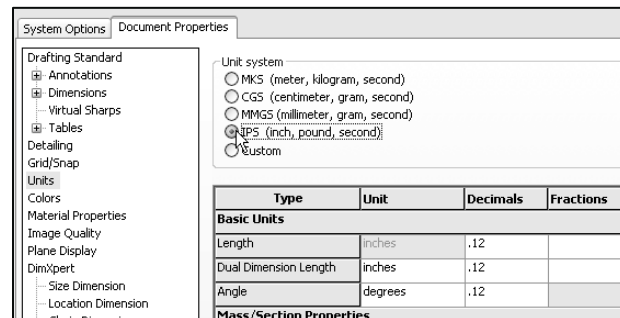
19. **Close** the model.



Set document precision from the Document Properties dialog box or from the Dimension PropertyManager. You can also address: Callout value, Tolerance type, and Dimension Text symbols in the Dimension PropertyManager.



You are allowed to answer the questions in any order you prefer. Use the Summary Screen during the CSWA exam to view the list of all questions you have or have not answered.



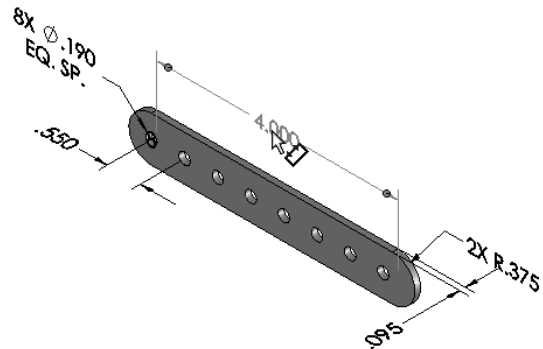
There are no Surfacing or Boundary feature questions on the CSWA exam.

Callout Value

A Callout value is a value that you select in a SolidWorks document. Click a dimension in the Graphics window, the selected dimension is displayed in blue and the Dimension PropertyManager is displayed.

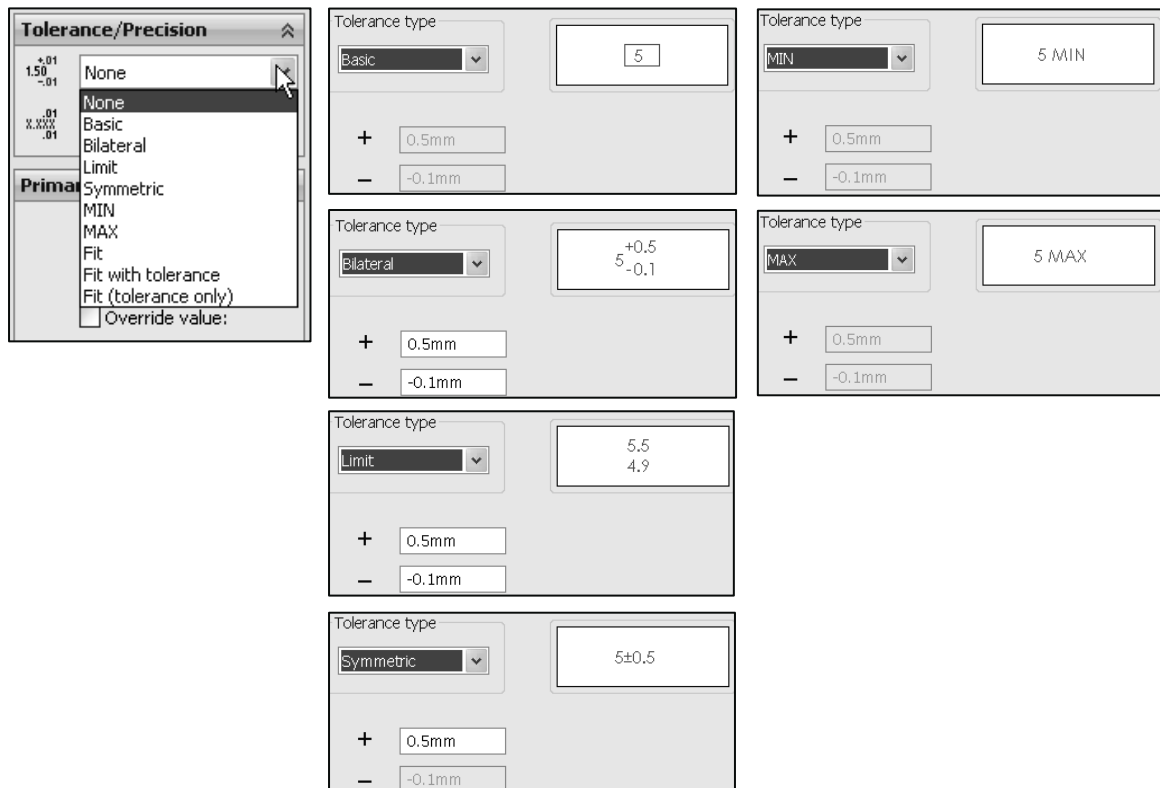


A Callout value is available for dimensions with multiple values in the callout.



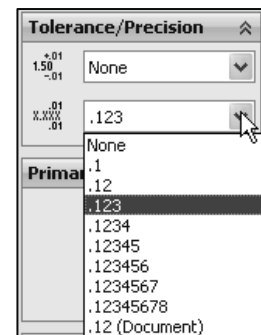
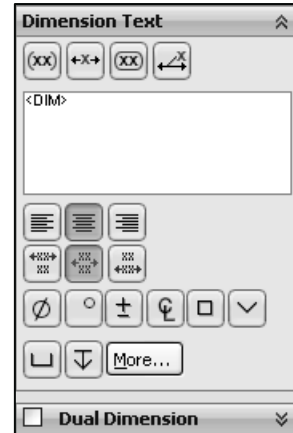
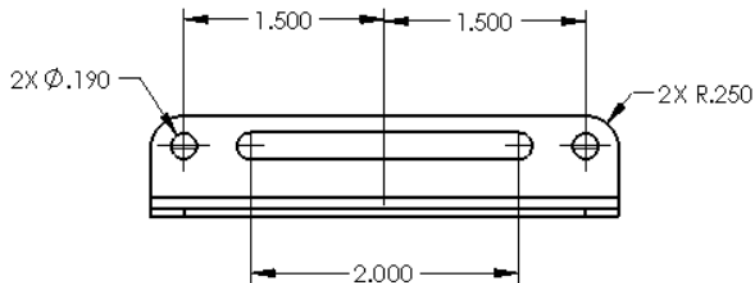
Tolerance Type

A Tolerance type is selected from the available drop down list in the Dimension PropertyManager. The list is dynamic. A few examples of Tolerance type display are listed below:



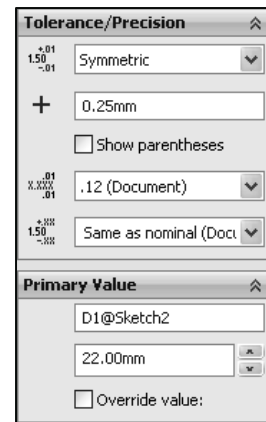
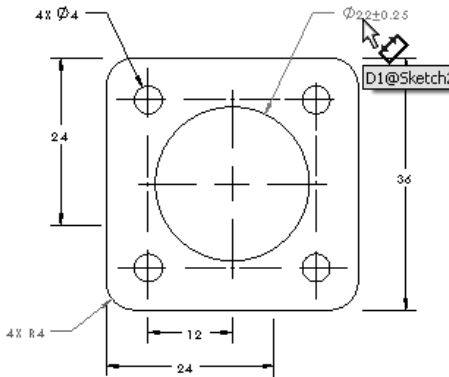
Tutorial: Dimension text 9-1

1. **View** the illustrated model.
2. **Review** the Tolerance, Precision, and Dimension Text.
 - a. 2X Ø.190 - Two holes with a diameter of .190. Precision is set to three decimal places.
 - b. 2X R.250 - Two corners with a radius of .250. Precision is set to three decimal places.



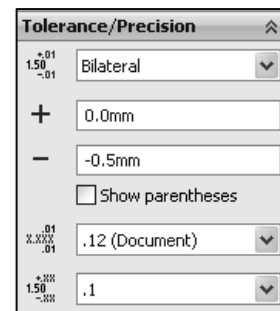
Tutorial: Dimension text 9-2


1. **View** the illustrated model.
2. **Review** the Tolerance, Precision, and Dimension text.
 - a. $\varnothing 22 \pm 0.25$ - The primary diameter value of the hole = 22.0mm. Tolerance type: Symmetric. Maximum Variation 0.25mm. Tolerance / Precision is set to two decimal places.




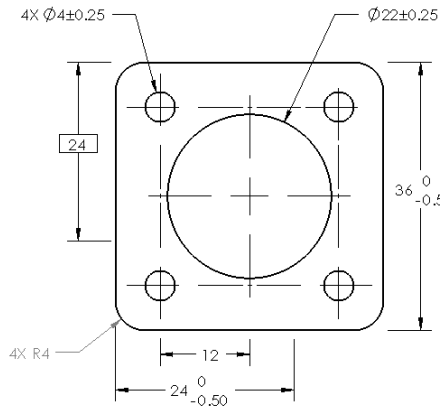
For a Chamfer feature, a second Tolerance/Precision is available.

- b. $36 \begin{smallmatrix} 0 \\ -0.5 \end{smallmatrix}$ - The primary diameter value of the hole = 36mm. Tolerance type: Bilateral. Maximum Variation is 0.0mm. Minimum Variation = -0.5mm. Precision is set to two decimal places. Tolerance is set to one decimal place.



 Trailing zeros are removed according to the ANSI standard.

- c.  - The primary value = 24mm. Tolerance type: Basic. Tolerance / Precision is set to two decimal places.
- d. **4X Ø 4±0.25** - Four holes with a primary diameter value = 4mm. Tolerance type: Symmetric. Maximum Variation = 0.25mm. Precision / Tolerance is set to two decimal places.



Tolerance/Precision

+0.01
-0.01 Basic

+0.01
-0.01 .12 (Document)

+0.01
-0.01 .12

Tolerance/Precision

+0.01
-0.01 Symmetric

+ 0.25mm

☐ Show parentheses

+0.01
-0.01 .12 (Document)

+0.01
-0.01 Same as nominal (Doc)


Primary Value

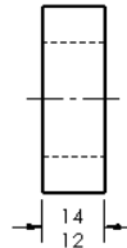
D1@Sketch3

4.00mm

Tutorial: Dimension text 3-3

1. **View** the illustrated model.
2. **Review** the Tolerance, and Precision.

- a.  - The primary value = 12mm. Tolerance type: Limit. Maximum Variation = 2mm. Minimum Variation = 0mm. Tolerance / Precision is set to none.



Tolerance/Precision

+0.01
-0.01 Limit

+ 2mm

- 0mm

+0.01
-0.01 None

+0.01
-0.01 None

Dimension Text Symbols

Dimension Text symbols are displayed in the Dimension PropertyManager. The Dimension Text box provides eight commonly used symbols and a more button to access the Symbol Library. The eight displayed symbols in the Dimension Text box from left to right are: Diameter, Degree, Plus/Minus, Centerline, Square, Countersink, Counterbore and Depth/Deep.

Dimension Text

(XX) +X- (XX) ↗

<DIM>

≡ ≡ ≡

↔ ↔ ↔

↔ ↔ ↔

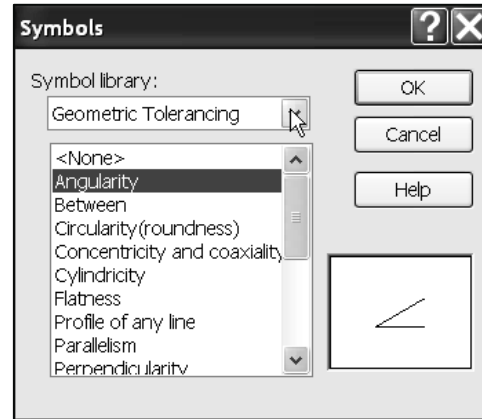
∅ ° ± ∇ □ ∇

↪ ↩ More...

☐ Dual Dimension

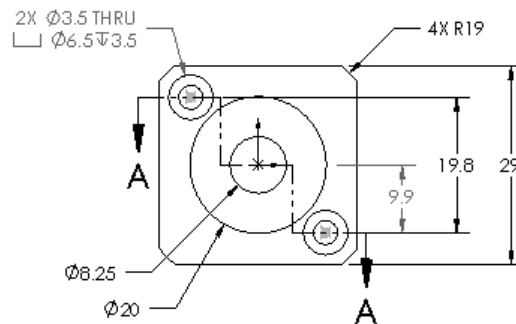
Basic and Intermediate Part Creation

Review each symbol in the Dimension Text box and in the Symbol library. You are required to understand the meaning of these symbols in a SolidWorks document.



Tutorial: Dimension Text symbols 3-1

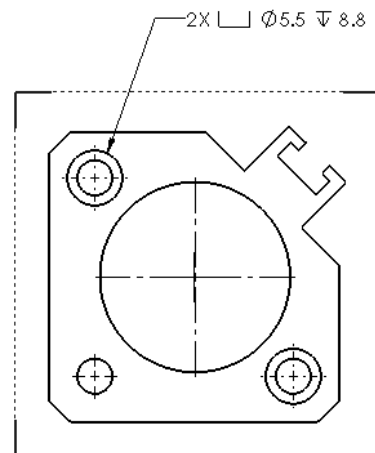
1. **View** the illustrated model.
2. **Review** the Dimension Text and document symbols.



- a. 2X $\varnothing 3.5$ THRU $\varnothing 6.5 \nabla 3.5$
- Two holes with a primary diameter value = 3.5mm, Cbore $\varnothing 6.5$ with a depth of 3.5.

Tutorial: Dimension Text symbols 3-2

1. **View** the illustrated model.
2. **Review** the Dimension Text and document symbols.
 - a. 2X $\varnothing 5.5 \nabla 8.8$ - Two Cbores with a primary diameter value = 5.5mm with a depth of 8.8.



Build Additional Basic Parts

Tutorial: Mass-Volume 3-10

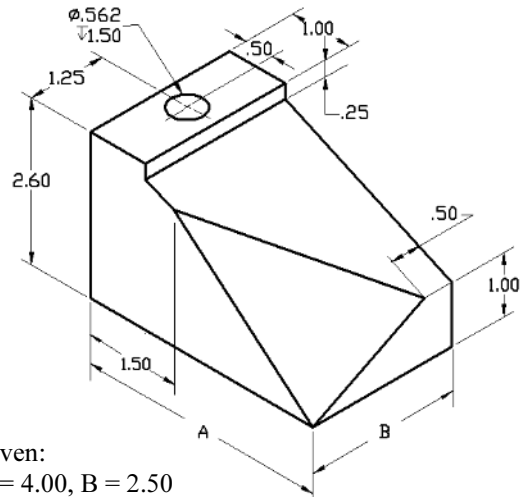
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Note the Depth/Deep ∇ symbol with a 1.50 dimension associated with the hole. The hole $\varnothing.562$ has a three decimal place precision. Insert three features: Extruded Base (Boss-Extrude1) and two Extruded Cuts. Insert a 3D sketch for the first Extruded Cut feature.

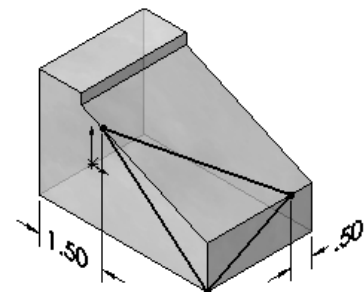
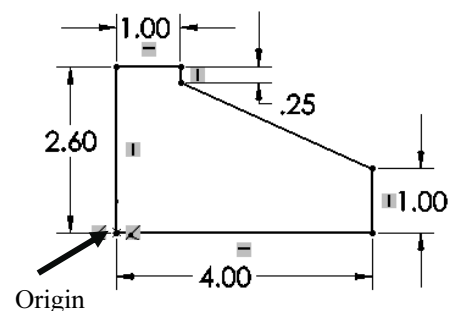
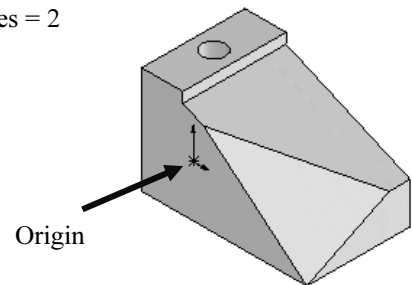


There are numerous ways to build the models in this chapter. A goal is to display different design intents and techniques.

3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. The part Origin is located in the lower left corner of the sketch. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Apply symmetry. Select the Mid Plane End Condition in Direction 1. Depth = 2.50in.
6. Create **3DSketch1**. Apply the Line Sketch tool. Create a closed four point 3D sketch. 3DSketch1 is the profile for the first Extruded Cut feature. Insert the required dimensions.

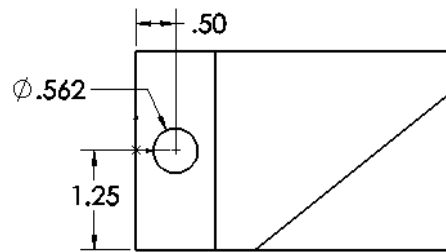
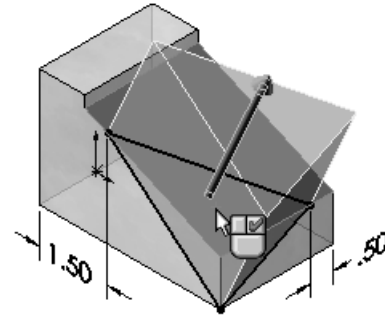


Given:
 A = 4.00, B = 2.50
 Material: Alloy Steel
 Density = .278 lb/in³
 Units: IPS
 Decimal places = 2



Basic and Intermediate Part Creation

7. Create the first **Extruded Cut** feature. Blind is the default End Conditions. Select the top face as illustrated to be removed. Note the direction of the extrude feature.
8. Create **Sketch2**. Select the top flat face of Boss-Extrude1. Sketch a circle. Insert the required geometric relations and dimensions. The hole diameter $\varnothing.562$ has a three decimal place precision.
9. Create the second **Extruded Cut** feature. Blind is the default End Condition. Depth = 1.50in. Note: For the exam, you do not need to insert the Depth/Deep ∇ symbol or note.
10. **Assign** Alloy Steel material to the part.
11. **Calculate** the overall mass. The overall mass = 4.97 pounds.
12. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

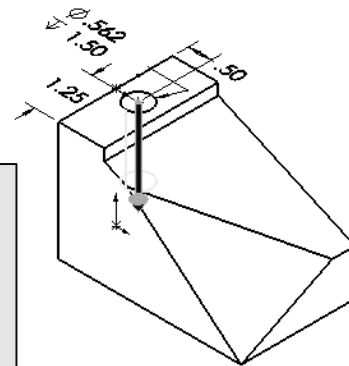


- X: 1.63 inches
- Y: 1.01 inches
- Z: -0.04 inches

View the triad location of the Center of mass for the part.

13. **Save** the part and name it Mass-Volume 3-10.

| |
|--|
| Density = 0.28 pounds per cubic inch Mass = 4.97 pounds Volume = 17.86 cubic inches Surface area = 46.77 inches ² Center of mass: (inches) X = 1.63 Y = 1.01 Z = -0.04 |
|--|



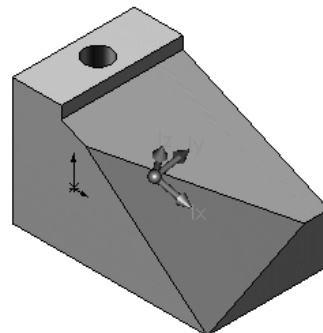
14. **Close** the model.

As an exercise, calculate the overall mass of the part using 6061 Alloy.

Modify the "A" dimension from 4.00 to 4.50. Modify the hole dimension from $\varnothing.562$ to $\varnothing.575$. The overall mass of the part = 1.93 pounds.

Save the part and name it Mass-Volume 3-10A.

| |
|--|
| Mass = 1.93 pounds Volume = 19.77 cubic inches Surface area = 50.66 inches ² Center of mass: (inches) X = 1.83 Y = 0.99 Z = -0.04 |
|--|



Tutorial: Mass-Volume 3-11

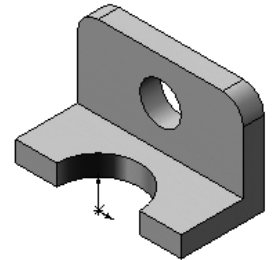
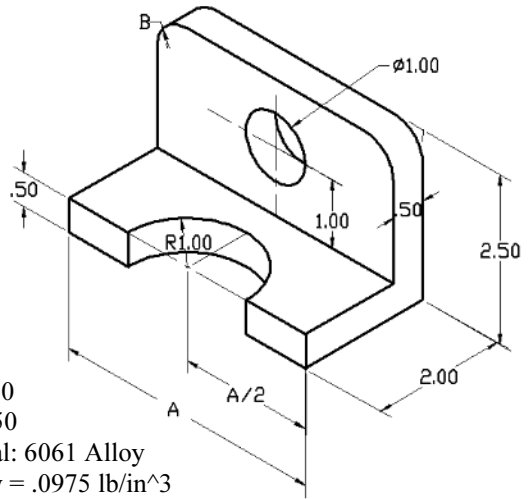
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Think about the required steps to build this part. Insert four features: Extruded Base, two Extruded Cuts, and a Fillet.

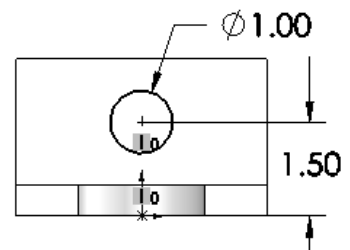
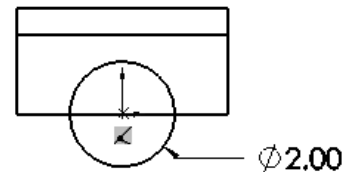
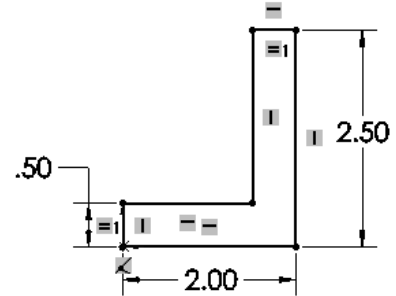


There are numerous ways to build the models in this chapter. A goal is to display different design intents and techniques.

Given:
 $A = 4.00$
 $B = R.50$
 Material: 6061 Alloy
 Density = .0975 lb/in³
 Units: IPS
 Decimal places = 2



3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Right Plane as the Sketch plane. The part Origin is located in the lower left corner of the sketch. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Apply symmetry. Select the Mid Plane End Condition for Direction 1. Depth = 4.00in.
6. Create **Sketch2**. Select the top flat face of Boss-Extrude1 as the Sketch plane. Sketch a circle. The center of the circle is located at the part Origin. Insert the required dimension.
7. Create the first **Extruded Cut** feature. Select Through All for End Condition in Direction 1.
8. Create **Sketch3**. Select the front vertical face of Extrude1 as the Sketch plane. Sketch a circle. Insert the required geometric relations and dimensions.
9. Create the second **Extruded Cut** feature. Select Through All for End Condition in Direction 1.



Basic and Intermediate Part Creation

10. Create the **Fillet** feature. Constant radius is selected by default. Fillet the top two edges as illustrated.
Radius = .50in.



A Fillet feature removes material. Selecting the correct radius value is important to obtain the correct mass and volume answer in the exam.

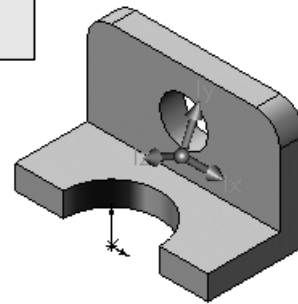
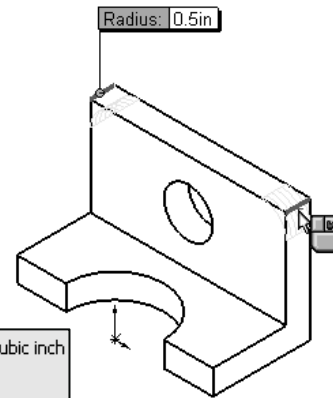
11. **Assign** the defined material to the part.

12. **Calculate** the overall mass. The overall mass = 0.66 pounds.

13. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 0.00 inches
- Y: 0.90 inches
- Z: -1.46 inches

Density = 0.10 pounds per cubic inch
Mass = 0.66 pounds
Volume = 6.76 cubic inches
Surface area = 36.99 inches²
Center of mass: (inches)
X = 0.00
Y = 0.90
Z = -1.46



In this category an exam question could read: Build this model. Locate the Center of mass relative to the part Origin.

- A: X = -2.63 inches, Y = 4.01 inches, Z = -0.04 inches
- B: X = 4.00 inches, Y = 1.90 inches, Z = -1.64 inches
- C: X = 0.00 inches, Y = 0.90 inches, Z = -1.46 inches
- D: X = -1.69 inches, Y = 1.00 inches, Z = 0.10 inches

The correct answer is C. Note: Tangent edges and Origin are displayed for educational purposes.

14. **Save** the part and name it Mass-Volume 3-11.

15. **Close** the model.

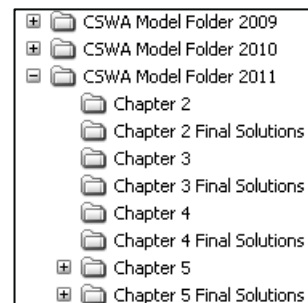
As an exercise, calculate the overall mass of the part using the MMGS unit system, and assign 2014 Alloy material to the part.

The overall mass of the part = 310.17 grams. Save the part and name it Mass-Volume 3-11-MMGS.

Mass = 310.17 grams
Volume = 110774.26 cubic millimeters
Surface area = 23865.83 millimeters²
Center of mass: (millimeters)
X = 0.00
Y = 22.83
Z = -37.11



You are allowed to answer the questions in any order you prefer. Use the Summary Screen during the CSWA exam to view the list of all questions you have or have not answered.



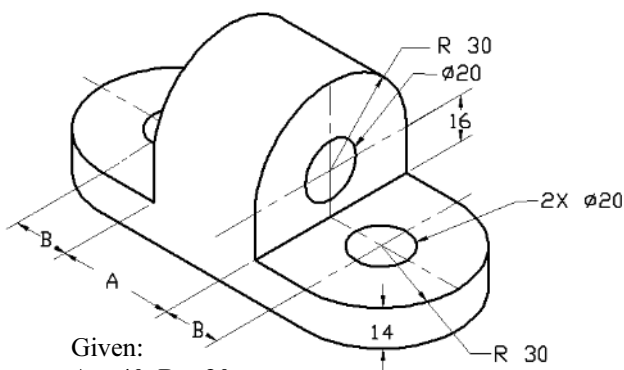
Tutorial: Mass-Volume 3-12

Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

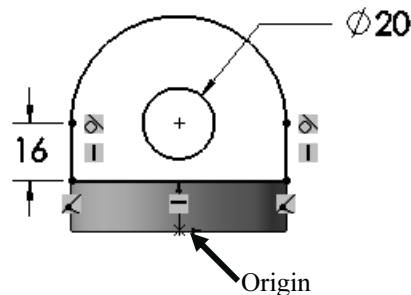
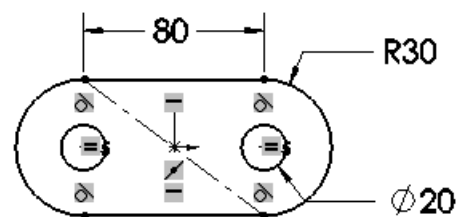
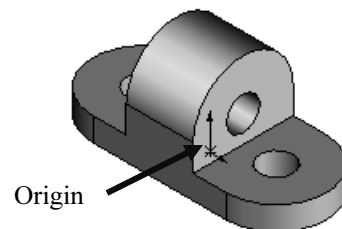
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert two features: Extruded Base (Boss-Extrude1) and Extruded Boss (Boss-Extrude2).
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Apply the Centerline Sketch tool. Locate the part Origin at the center of the sketch. Insert the required geometric relations and dimensions. Note: This is a good case to use the Slot Sketch tool!
5. Create the **Extruded Base (Boss-Extrude1)** feature. Blind is the default End Condition. Depth = 14mm.
6. Create **Sketch2**. Select the Right Plane as the Sketch plane. Insert the required geometric relations and dimensions.
7. Create the **Extruded Boss (Boss-Extrude2)** feature. Apply symmetry. Select the Mid Plane End Condition. Depth = 40mm.
8. **Assign** the defined material to the part.
9. **Calculate** the overall mass. The overall mass = 1605.29 grams.
10. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 0.00 millimeters
- Y: 19.79 millimeters
- Z: 0.00 millimeters

| |
|--|
| Mass = 1605.29 grams |
| Volume = 180369.91 cubic millimeters |
| Surface area = 29918.76 millimeters ² |
| Center of mass: (millimeters) |
| X = 0.00 |
| Y = 19.79 |
| Z = 0.00 |



Given:
 A = 40, B = 20
 All Thru Holes
 Material: Copper
 Density = .0089 g/mm³
 Units: MMGS



Basic and Intermediate Part Creation

11. **Save** the part and name it Mass-Volume 3-12.

12. **Close** the model.

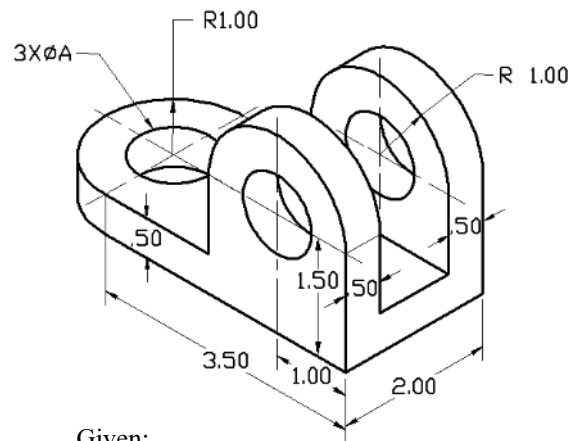


There are numerous ways to build the models in this chapter. Optimize your time. The CSWA is a timed exam.

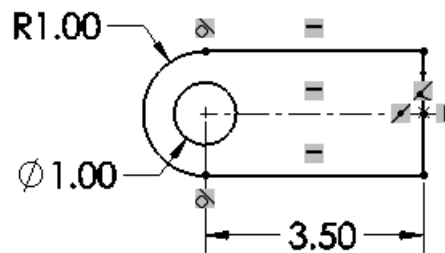
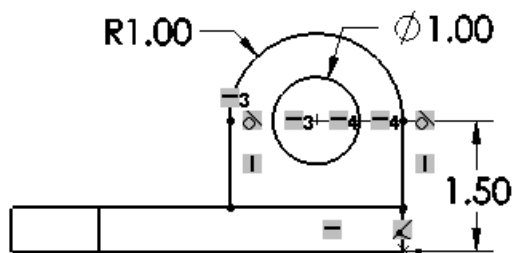
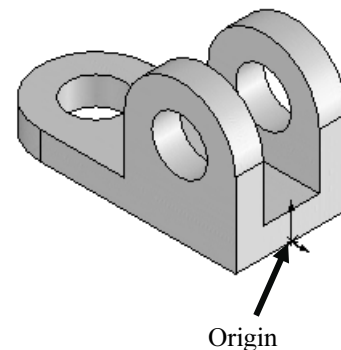
Tutorial: Mass-Volume 3-13

Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.

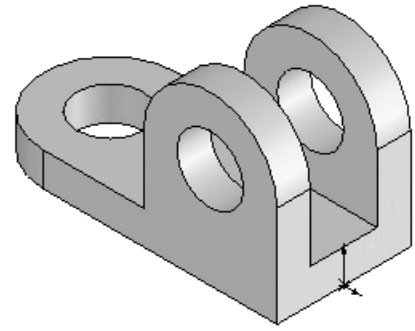
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert three features: Extruded Base (Boss-Extrude1), Extruded Boss (Boss-Extrude2) and Mirror. Three holes are displayed with an $\varnothing 1.00$ in.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Apply the Tangent Arc and Line Sketch tool. Insert the required geometric relations and dimensions. Note the location of the Origin.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Blind is the default End Condition. Depth = .50in.
6. Create **Sketch2**. Select the front vertical face of Extrude1 as the Sketch plane. Insert the required geometric relations and dimensions.



Given:
A = $\varnothing 1.00$
All Thru Holes
Material: Brass
Density = .307 lb/in³
Units: IPS
Decimal places = 2



7. Create the **Extruded Boss (Boss-Extrude2)** feature. Blind is the default End Condition in Direction 1. Depth = .50in. Note the direction of the extrude.
8. Create the **Mirror** feature. Apply Symmetry. Mirror Boss-Extrude2 about the Front Plane.
9. **Assign** the defined material to the part.
10. **Calculate** the volume. The volume = 6.68 cubic inches.
11. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
 - X: -1.59 inches
 - Y: 0.72 inches
 - Z: 0.00 inches



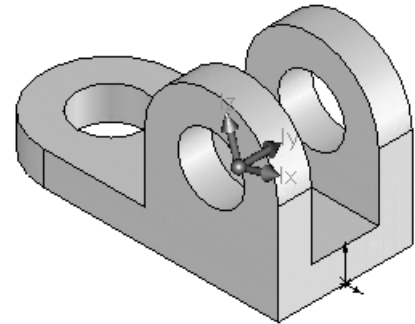
| |
|--|
| Mass = 2.05 pounds |
| Volume = 6.68 cubic inches |
| Surface area = 40.64 inches ² |
| Center of mass: (inches) |
| X = -1.59 |
| Y = 0.72 |
| Z = 0.00 |

In this category an exam question could read: Build this model. What is the volume of the model?

- A = 6.19 cubic inches
- B = 7.79 cubic inches
- C = 7.87 cubic inches
- D = 6.68 cubic inches

The correct answer is D.

View the triad location of the Center of mass for the part.



12. **Save** the part and name it Mass-Volume 3-13.
13. **Close** the model.

As an exercise, calculate the overall mass of the part using the IPS unit system, and assign Copper material to the part. Modify the hole diameters from 1.00in to 1.125in.

The overall mass of the part = 2.05 pounds. Save the part and name it Mass-Volume 3-13A.




The book is design to expose the new user to many tools, techniques and procedures. It may not always use the most direct tool or process.

| |
|--|
| Mass = 2.05 pounds |
| Volume = 6.37 cubic inches |
| Surface area = 39.97 inches ² |
| Center of mass: (inches) |
| X = -1.58 |
| Y = 0.70 |
| Z = 0.00 |


Tutorial: Mass-Volume 3-14

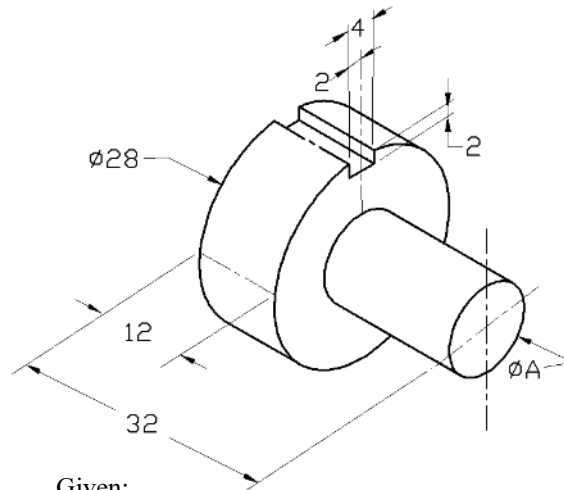
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert a Revolved Base feature and Extruded Cut feature to build this part.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Apply the Centerline Sketch tool for the Revolve1 feature. Insert the required geometric relations and dimensions. Sketch1 is the profile for the Revolve1 feature.
5. Create the **Revolved Base** feature. The default angle is 360deg. Select the centerline for the Axis of Revolution.

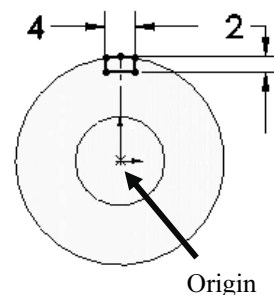
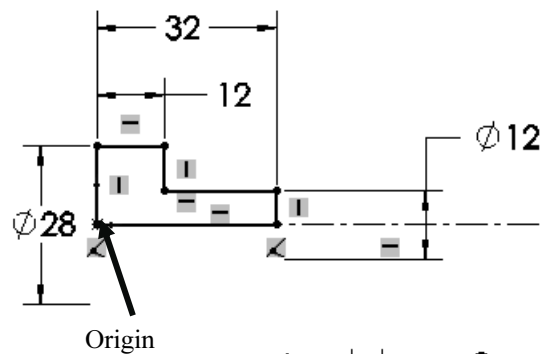
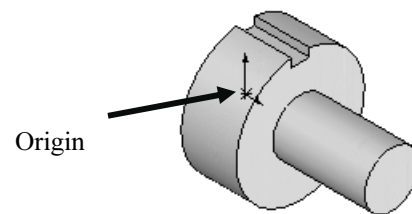
 A Revolve feature adds or removes material by revolving one or more profiles around a centerline.

6. Create **Sketch2**. Select the right large circular face of Revolve1 as the Sketch plane. Apply reference construction geometry. Use the Convert Entities and Trim Sketch tools. Insert the required geometric relations and dimensions.

 You could also use the 3 Point Arc Sketch tool instead of the Convert Entities and Trim Sketch tools to create Sketch2.

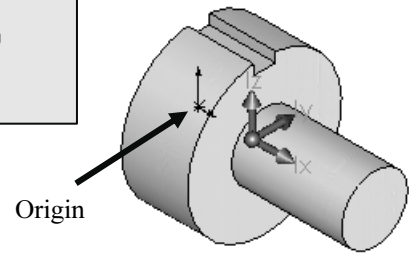
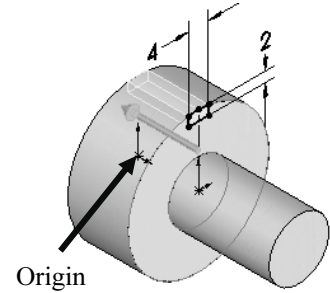


Given:
 $A = \text{Ø}12$
 Material: Cast Alloy Steel
 Density = .0073 g/mm³
 Units: MMGS



7. Create the **Extruded Cut** feature. Select Through All for End Condition in Direction 1.
8. **Assign** the defined material to the part.
9. **Calculate** the overall mass. The overall mass = 69.77 grams.
10. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
 - X = 9.79 millimeters
 - Y = -0.13 millimeters
 - Z = 0.00 millimeters
11. **Save** the part and name it Mass-Volume 3-14.
12. **Close** the model.


| |
|---|
| Mass = 69.77 grams |
| Volume = 9557.27 cubic millimeters |
| Surface area = 3069.83 millimeters ² |
| Center of mass: (millimeters) |
| X = 9.79 |
| Y = -0.13 |
| Z = 0.00 |

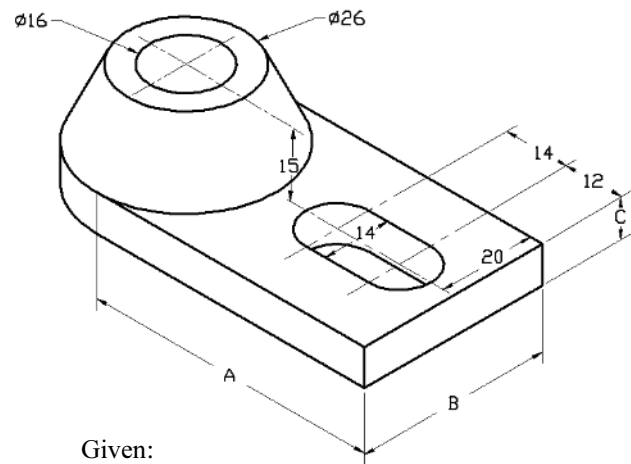


Tutorial: Mass-Volume 3-15

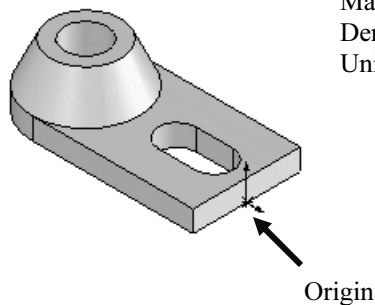
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert two features: Extruded Base (Boss-Extrude1) and Revolved Boss.
3. **Set** the document properties for the model.

 Tangent edges and Origin are displayed for educational purposes.

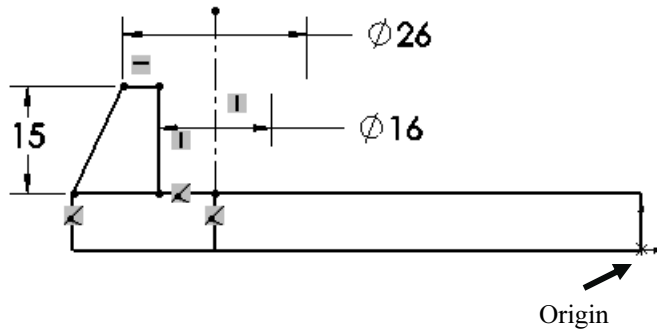
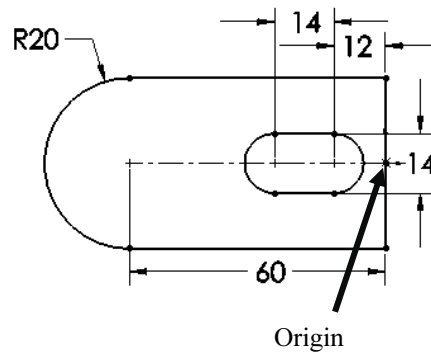


Given:
 A = 60, B = 40, C = 8
 Material: Cast Alloy Steel
 Density = .0073 g/mm³
 Units: MMGS



Basic and Intermediate Part Creation

4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Apply construction geometry. Apply the Tangent Arc and Line Sketch tool. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base** feature. Blind is the default End Condition. Depth = 8mm.
6. Create **Sketch2**. Select the Front Plane as the Sketch plane. Apply construction geometry for the Revolved Boss feature. Insert the required geometric relations and dimension.
7. Create the **Revolved Boss** feature. The default angle is 360deg. Select the centerline for Axis of Revolution.
8. **Assign** the defined material to the part. **Calculate** the overall mass. The overall mass = 229.46 grams.



9. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
- X = -46.68 millimeters
 - Y = 7.23 millimeters
 - Z = 0.00 millimeters

| |
|---|
| Mass = 229.46 grams |
| Volume = 31433.02 cubic millimeters |
| Surface area = 9459.63 millimeters ² |
| Center of mass: (millimeters) |
| X = -46.68 |
| Y = 7.23 |
| Z = 0.00 |

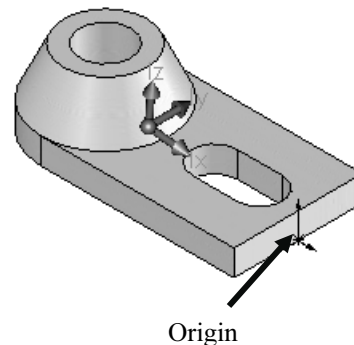
In this category an exam question could read: Build this model. What is the overall mass of the part?

- A: 229.46 grams
- B: 249.50 grams
- C: 240.33 grams
- D: 120.34 grams

The correct answer is A.

10. **Save** the part and name it Mass-Volume 3-15.

11. **Close** the model.

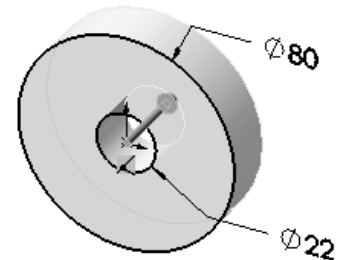
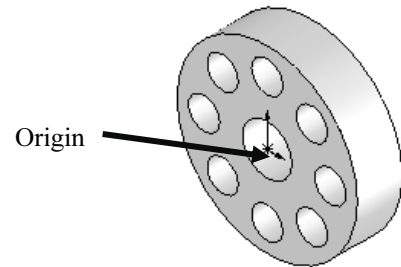
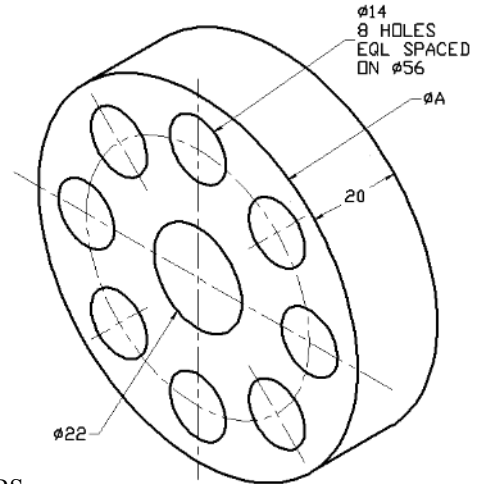



Tutorial: Mass-Volume 3-16

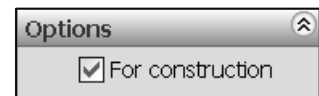
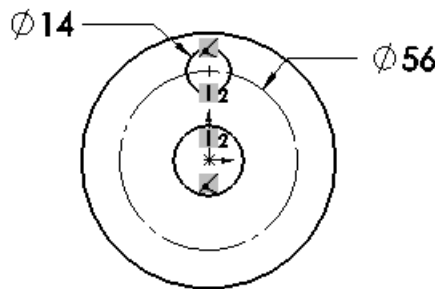
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Insert three features: Extruded Base, Extruded Cut and Circular Pattern. There are eight holes $\varnothing 14\text{mm}$ equally spaced on an $\varnothing 56\text{mm}$ bolt circle. The center hole = $\varnothing 22\text{mm}$.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Sketch two circles. The part Origin is located in the center of the sketch. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base (Boss-Extrude1)** feature. Blind is the default End Condition. Depth = 20mm.
6. Create **Sketch2**. Select the front face as the Sketch plane. Apply construction geometry to locate the seed feature for the Circular Pattern. Insert the required geometric relations and dimensions.

Given:
 $A = \varnothing 80$
 Material: ABS
 Density: $.001 \text{ g/mm}^3$
 Units: MMGS

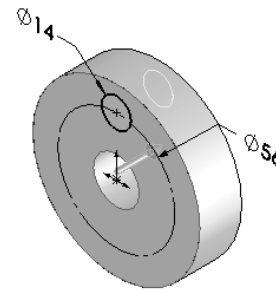
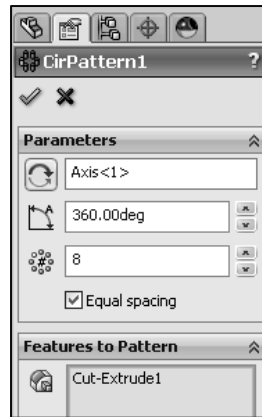


 Apply construction reference geometry to assist in creating the sketch entities and geometry that are incorporated into the part. Construction reference geometry is ignored when the sketch is used to create a feature. Construction reference geometry uses the same line style as centerlines.



Basic and Intermediate Part Creation

7. Create the **Extruded Cut** feature.
Select Through All for End Condition in Direction 1.
8. Create the **Circular Pattern** feature.
Create a Circular Pattern of the Cut-Extrude1 feature. Use the View, Temporary Axes command to select the Pattern Axis for the CirPattern1 feature. Instances = 8. Equal spacing is selected by default.



Apply a circular pattern feature to create multiple instances of one or more features that you can space uniformly about an axis.

9. **Assign** the defined material to the part.
10. **Calculate** the overall mass. The overall mass = 69.66 grams.
11. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

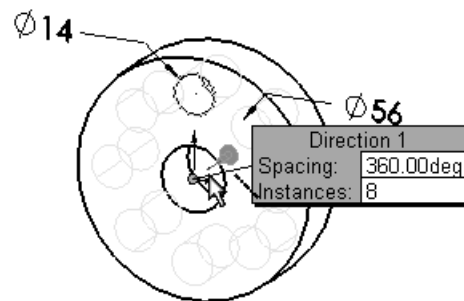
- X = 0.00 millimeters
- Y = 0.00 millimeters
- Z = -10.00 millimeters

12. **Save** the part and name it Mass-Volume 3-16.

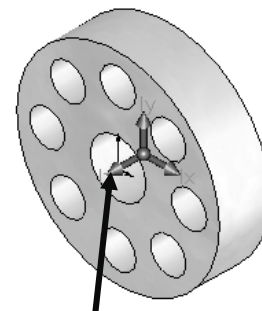
13. **Close** the model.

As an exercise, select the Top Plane for the Sketch plane to create Sketch1. Recalculate the location of the Center of mass with respect to the part Origin: X = 0.00 millimeters, Y = -10.00 millimeters and Z = 0.00 millimeters. Save the part and name it Mass-Volume 3-16-TopPlane.

In the next section, the models represent the feature types and complexity that you would see on the exam.

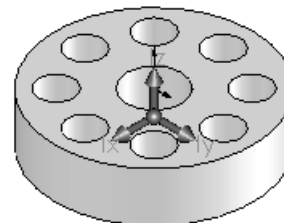


| |
|---------------------------------------|
| Mass = 69.66 grams |
| Volume = 68298.22 cubic millimeters |
| Surface area = 20275.84 millimeters^2 |
| Center of mass: (millimeters) |
| X = 0.00 |
| Y = 0.00 |
| Z = -10.00 |



Origin

| |
|---------------------------------------|
| Mass = 69.66 grams |
| Volume = 68298.22 cubic millimeters |
| Surface area = 20275.84 millimeters^2 |
| Center of mass: (millimeters) |
| X = 0.00 |
| Y = -10.00 |
| Z = 0.00 |



Tutorial: Basic/Intermediate Part 3-1

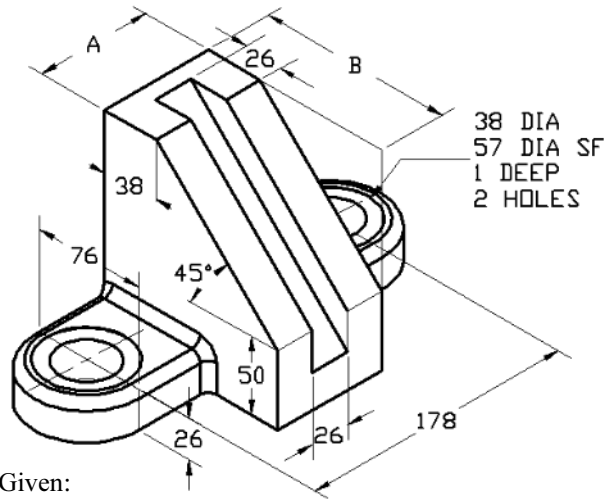
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.

1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Think about the various features that create the model. Insert seven features to build this model: Extruded Base, Extruded Cut, Extruded Boss, Fillet, second Extruded Cut, Mirror and a second Fillet. Apply symmetry. Create the left half of the model first, and then apply the Mirror feature.

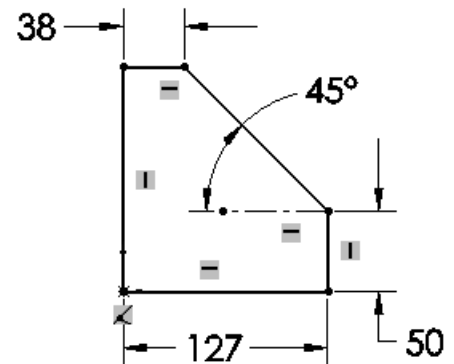
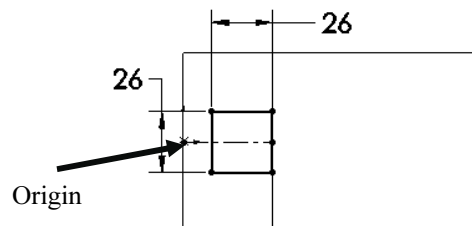
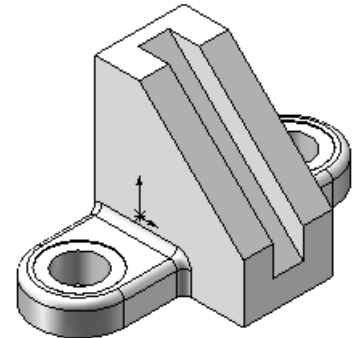
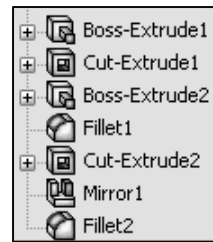


There are numerous ways to build the models in this chapter. The goal is to display different design intents and techniques.

3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Create the main body of the part. The part Origin is located in the bottom left corner of the sketch. Insert the required geometric relations and dimensions.
5. Create the **Extruded Base** feature. Boss-Extrude1 is the Base feature. Select Mid Plane for End Condition in Direction 1. Depth = 76mm.
6. Create **Sketch2**. Select the top flat face of Extrude1 as the Sketch plane. Create the top cut on the Base feature. Apply construction geometry. Insert the required geometric relations and dimensions.

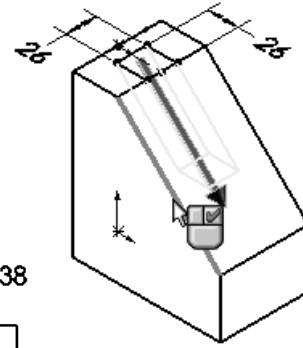


Given:
 A = 76, B = 127
 Material: 2014 Alloy
 Density: .0028 g/mm³
 Units: MMGS
 ALL ROUNDS EQUAL 6MM

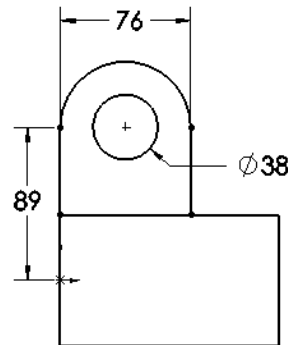


Basic and Intermediate Part Creation

7. Create the first **Extruded Cut** feature. Select Through All for End Condition in Direction 1. Select the illustrated angled edge for the Direction of Extrusion.

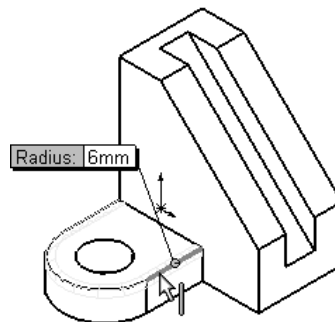


8. Create **Sketch3**. Select the bottom face of Boss-Extrude1 as the Sketch plane. Sketch the first tab with a single hole as illustrated. Insert the required geometric relations and dimensions.

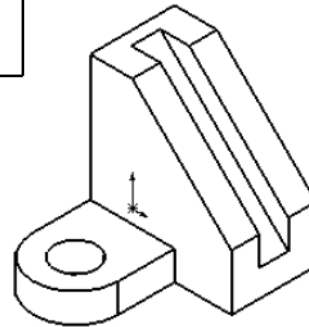


9. Create the **Extruded Boss** feature. Blind is the default End Condition in Direction 1. Depth = 26mm.

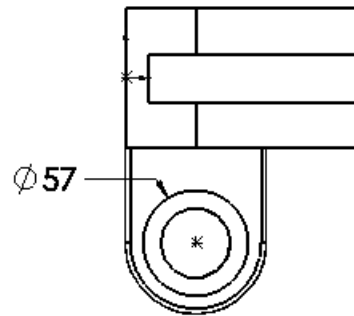
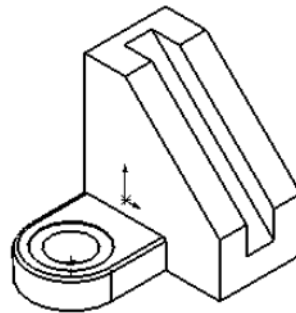
10. Create the first **Fillet** feature. Fillet the top edge of the left tab. Radius = 6mm. Constant radius is selected by default.



11. Create **Sketch4**. Select the top face of Extrude3 as the Sketch plane. Sketch a circle. Insert the required dimension.

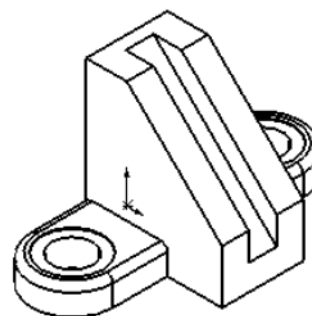
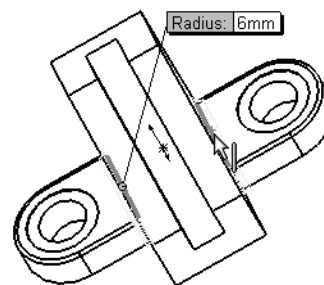


12. Create the second **Extruded Cut** feature. Blind is the default End Condition in Direction 1. Depth = 1mm. The model displayed an Ø57mm Spot Face hole with a 1mm depth.



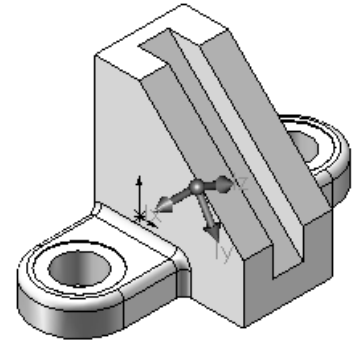
13. Create the **Mirror** feature. Mirror about the Front Plane. Mirror the Cut-Extrude2, Fillet1, and Boss-Extrude2 feature.

14. Create the second **Fillet** feature. Fillet the top inside edge of the left tab and the top inside edge of the right tab. Radius = 6mm.



15. **Assign** the defined material to the part.
16. **Calculate** the overall mass of the part. The overall mass = 3437.29 grams.
17. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.
 - X = 49.21 millimeters
 - Y = 46.88 millimeters
 - Z = 0.00 millimeters
18. **Save** the part and name it Part-Modeling 3-1.
19. **Close** the model.

Mass = 3437.29 grams
 Volume = 1227602.20 cubic millimeters
 Surface area = 101091.11 millimeters^2
 Center of mass: (millimeters)
 X = 49.21
 Y = 46.88
 Z = 0.00



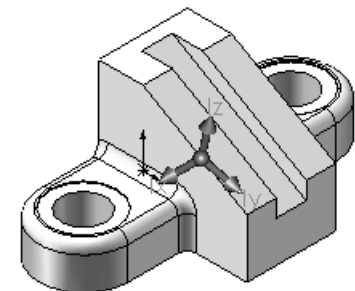
In this category, an exam question could read: Build this model. What is the overall mass of the part?

- A: 3944.44 grams
- B: 4334.29 grams
- C: 3437.29 grams
- D: 2345.69 grams

The correct answer is C.

As an exercise, modify all ALL ROUNDS from 6MM to 8MM. Modify the material from 2014 Alloy to 6061 Alloy.

Modify the Sketch1 angle from 45deg to 30deg. Modify the Extrude3 depth from 26mm to 36mm. Recalculate the location of the Center of mass with respect to the part Origin.



Mass = 3024.12 grams
 Volume = 1120042.82 cubic millimeters
 Surface area = 92861.98 millimeters^2
 Center of mass: (millimeters)
 X = 49.76
 Y = 34.28
 Z = 0.00

- X = 49.76 millimeters
- Y = 34.28 millimeters
- Z = 0.00 millimeters

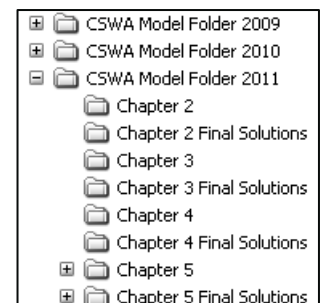
20. **Save** the part and name it Part-Modeling 3-1-Modify.



Tangent edges and Origin are displayed for educational purposes.




When you create a new part or assembly, the three default Planes (Front, Right and Top) are aligned with specific views. The Plane you select for the Base sketch determines the orientation of the part.

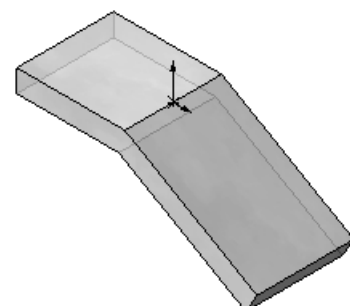
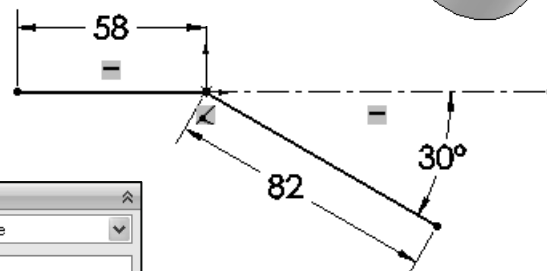
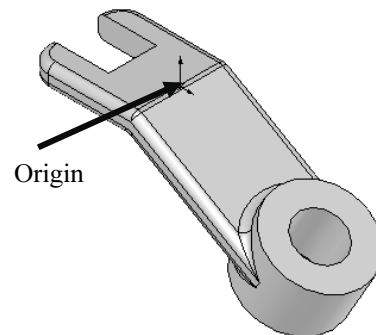
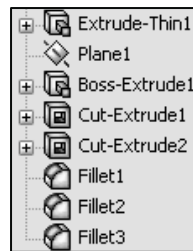
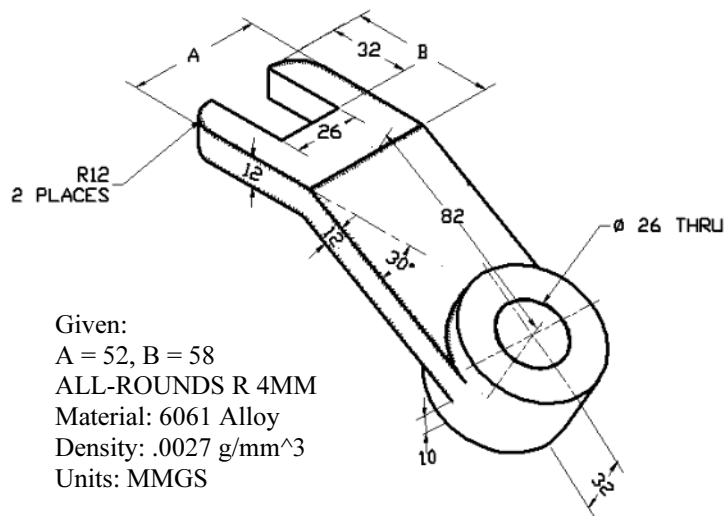


**Tutorial: Basic/Intermediate
Part 3-2**

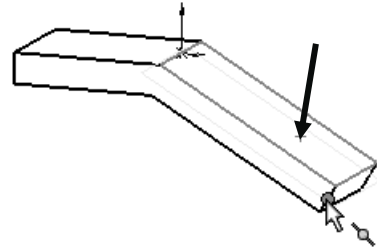
Build this model. Calculate the overall mass of the part and locate the Center of mass with the provided information.


1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Think about the various features that create the part. Insert seven features and a plane to build this part: Extrude-Thin1, Boss-Extrude1, Cut-Extrude1, Cut-Extrude2 and three Fillets. Apply reference construction planes to build the circular features.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Front Plane as the Sketch plane. Apply construction geometry as the reference line for the 30deg angle. Insert the required geometric relations and dimensions. Note the location of the Origin.
5. Create the **Extrude-Thin1** feature. This is the Base feature. Apply symmetry. Select Mid Plane for End Condition in Direction 1 to maintain the location of the Origin. Depth = 52mm. Thickness = 12mm.

 Use the Thin Feature option to control the extrude thickness, not the Depth.

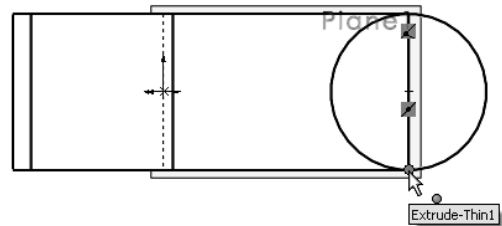



6. Create **Plane1**. Plane1 is the Sketch plane for the Extruded Boss (Boss-Extrude1) feature. Select the midpoint and the top face as illustrated. Plane1 is located in the middle of the top and bottom faces. Select Parallel Plane at Point for option.



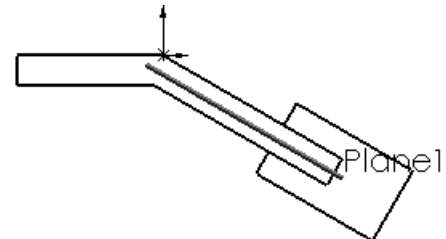
 Create Plane1 to use the Depth dimension of 32mm.


7. Create **Sketch2**. Select Plane1 as the Sketch plane. Use the Normal To view tool. Sketch a circle to create the Extruded Boss feature. Insert the required geometric relations.



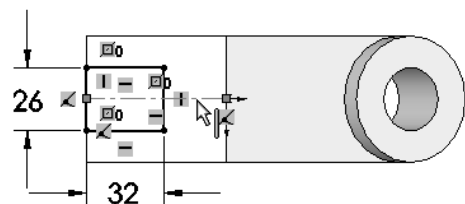
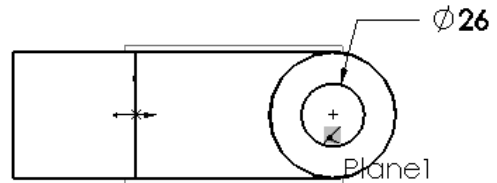
 The Normal To view tool rotates and zooms the model to the view orientation normal to the selected plane, planar face, or feature.

8. Create the **Extruded Boss** feature. Apply Symmetry. Select Mid Plane for End Condition in Direction 1. Depth = 32mm.
9. Create **Sketch3**. Select the top circular face of Boss-Extrude1 as the Sketch plane. Sketch a circle. Insert the required geometric relation and dimension.



 There are numerous ways to create the models in this chapter. A goal is to display different design intents and techniques.

10. Create the first **Extruded Cut** feature. Select Through All for End Condition in Direction 1.
11. Create **Sketch4**. Select the top face of Extrude-Thin1 as the Sketch plane. Apply construction geometry. Insert the required geometric relations and dimensions.
12. Create the second **Extruded Cut** feature. Select Through All for End Condition in Direction 1.



Basic and Intermediate Part Creation

13. Create the **Fillet1** feature. Fillet the left and right edges of Extrude-Thin1 as illustrated. Radius = 12mm.

14. Create the **Fillet2** feature. Fillet the top and bottom edges of Extrude-Thin1 as illustrated. Radius = 4mm.

15. Create the **Fillet3** feature. Fillet the rest of the model; six edges as illustrated. Radius = 4mm.

16. **Assign** the defined material to the part.

17. **Calculate** the overall mass of the part.
The overall mass = 300.65 grams.

18. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 34.26 millimeters
- Y: -29.38 millimeters
- Z: 0.00 millimeters

19. **Save** the part and name it Part-Modeling 3-2.

20. **Close** the model.

As an exercise, modify the Fillet2 and Fillet3 radius from 4mm to 2mm. Modify the Fillet1 radius from 12mm to 10mm. Modify the material from 6061 Alloy to ABS.

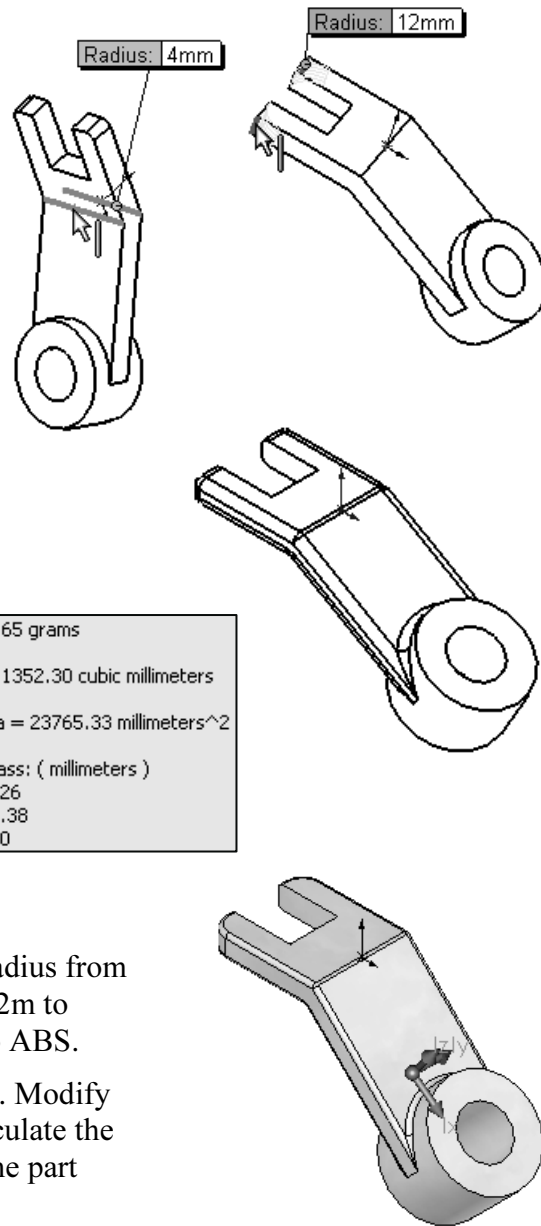
Modify the Sketch1 angle from 30deg to 45deg. Modify the Extrude depth from 32mm to 38mm. Recalculate the location of the Center of mass with respect to the part Origin.

- X = 27.62 millimeters
- Y = -40.44 millimeters
- Z = 0.00 millimeters

21. **Save** the part and name it Part-Modeling 3-2-Modify.



In the exam; you are allowed to answer the questions in any order. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.



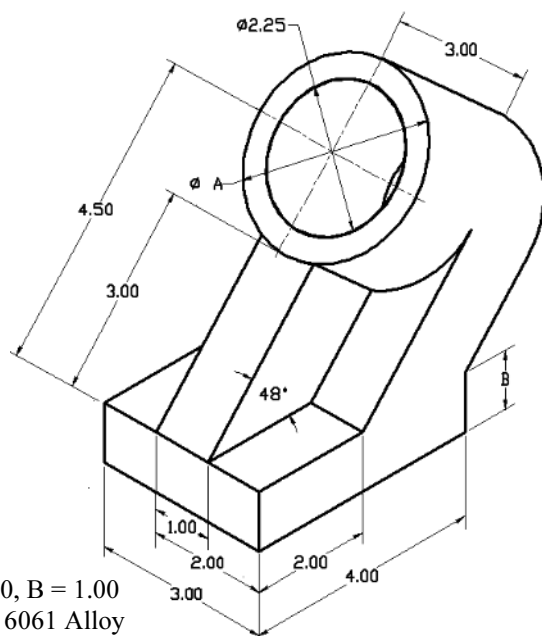
Mass = 300.65 grams
Volume = 111352.30 cubic millimeters
Surface area = 23765.33 millimeters^2
Center of mass: (millimeters)
X = 34.26
Y = -29.38
Z = 0.00

Mass = 123.60 grams
Volume = 121173.81 cubic millimeters
Surface area = 25622.46 millimeters^2
Center of mass: (millimeters)
X = 27.62
Y = -40.44
Z = 0.00

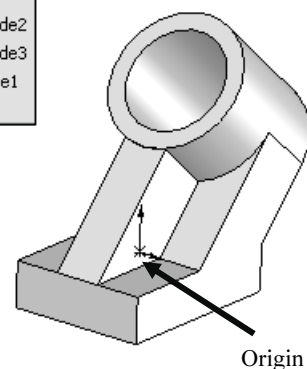
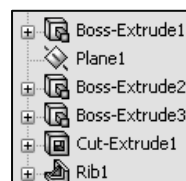
Tutorial: Basic/Intermediate Part 3-3


Build this model. Calculate the volume of the part and locate the Center of mass with the provided information.


1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Think about the various features that create this model. Insert five features and a plane to build this part: Extruded Base, two Extruded Bosses, Extruded Cut and a Rib. Insert a reference plane to create the Boss-Extrude2 feature.
3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Sketch a rectangle. Apply two construction lines for an Intersection relation. Use the horizontal construction line as the Plane1 reference. Insert the required relations and dimensions.
5. Create the **Extruded Base** feature. End Condition in Direction 1. Dep extrude direction is downward.

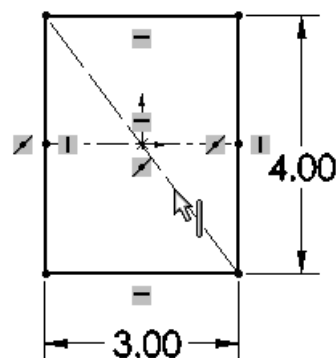


Given:
A = Ø3.00, B = 1.00
Material: 6061 Alloy
Density: .097 lb/in³
Units: IPS
Decimal places = 2



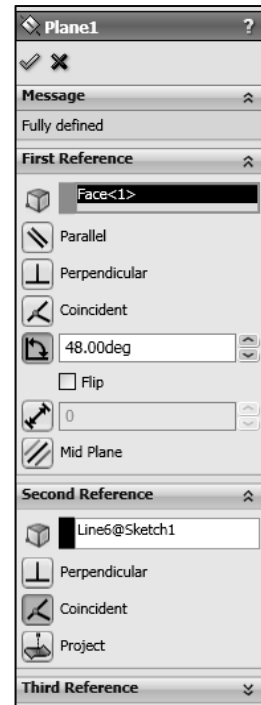
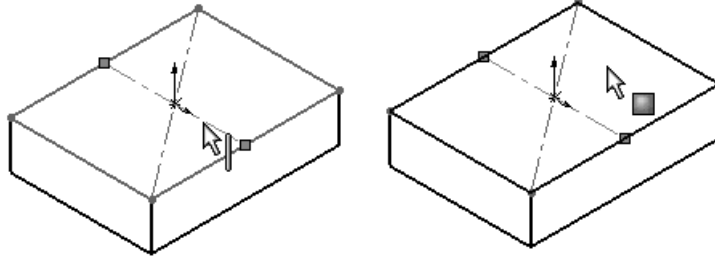
 Create planes to aid in the modeling for the exam. Use planes to sketch, to create a section view, for a neutral plane in a draft feature, and so on.

 The created plane is displayed 5% larger than the geometry on which the plane is created, or 5% larger than the bounding box. This helps reduce selection problems when planes are created directly on faces or from orthogonal geometry.



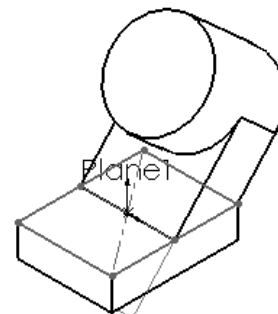
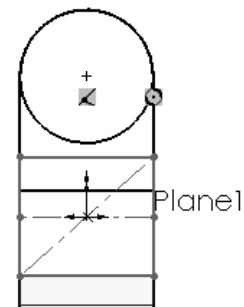
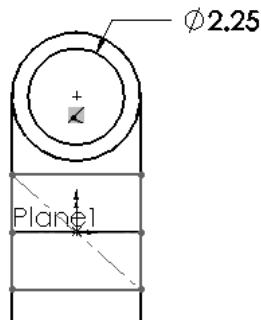
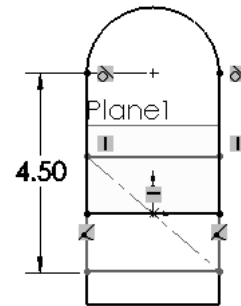
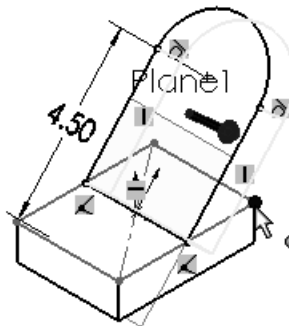
Basic and Intermediate Part Creation

6. Create **Plane1**. Plane1 is the Sketch plane for the Extruded Boss feature. Show Sketch1. Select the horizontal construction line in Sketch1 and the top face of Boss-Extrude1. Angle = 48deg.



Click **View, Sketches** from the Menu bar menu to display sketches in the Graphics window.

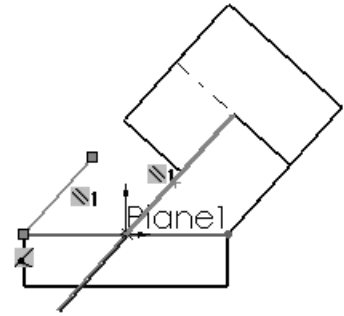
7. Create **Sketch2**. Select Plane1 as the Sketch plane. Create the Extruded Boss profile. Insert the required geometric relations and dimension. Note: Dimension to the front top edge of Boss-Extrude1 as illustrated.
8. Create the first **Extruded Boss** feature. Select the Up To Vertex End Condition in Direction 1. Select the back top right vertex point as illustrated.
9. Create **Sketch3**. Select the back angled face of Boss-Extrude2 as the Sketch plane. Sketch a circle. Insert the required geometric relations.
10. Create the third **Extruded Boss** feature. Blind is the default End Condition in Direction 1. Depth = 3.00in.
11. Create **Sketch4**. Select the front face of Boss-Extrude3 as the Sketch plane. Sketch a circle. Sketch4 is the profile for the Extruded Cut feature. Insert the required geometric relation and dimension.





The part Origin is displayed in blue.

12. Create the **Extruded Cut** feature. Select Through All for End Condition in Direction 1.
13. Create **Sketch5**. Select the Right Plane as the Sketch plane. Insert a Parallel relation to partially define Sketch5. Sketch5 is the profile for the Rib feature. Sketch5 does not need to be fully defined. Sketch5 locates the end conditions based on existing geometry.
14. Create the **Rib** feature. Thickness = 1.00in.



The Rib feature is a special type of extruded feature created from open or closed sketched contours. The Rib feature adds material of a specified thickness in a specified direction between the contour and an existing part. You can create a rib feature using single or multiple sketches.

15. **Assign** 6061 Alloy material to the part.

16. **Calculate** the volume. The volume = 30.65 cubic inches.

17. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 0.00 inches
- Y: 0.73 inches
- Z: -0.86 inches

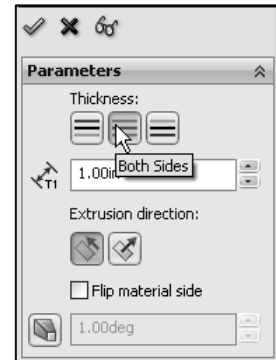
18. **Save** the part and name it Part-Modeling 3-3.

19. **Close** the model.

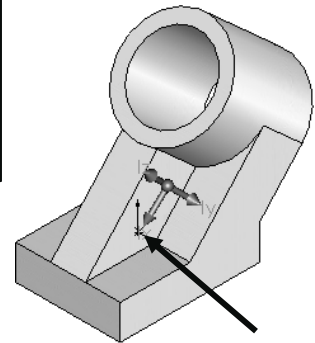
As an exercise, modify the Rib1 feature from 1.00in to 1.25in. Modify the Extrude depth from 3.00in to 3.25in. Modify the material from 6061 Alloy to Copper.

Modify the Plane1 angle from 48deg to 30deg. Recalculate the volume of the part. The new volume = 26.94 cubic inches.

20. **Save** the part and name it Part-Modeling 3-3-Modify.



Mass = 2.99 pounds
Volume = 30.65 cubic inches
Surface area = 100.96 inches²
Center of mass: (inches)
X = 0.00
Y = 0.73
Z = -0.86



Origin

Volume = 26.94 cubic inches
Surface area = 98.18 inches²
Center of mass: (inches)
X = 0.00
Y = 0.58
Z = -0.85

Tutorial:
Basic/Intermediate
Part 3-4

Build this model.
Calculate the volume
of the part and locate
the Center of mass
with the provided
information.

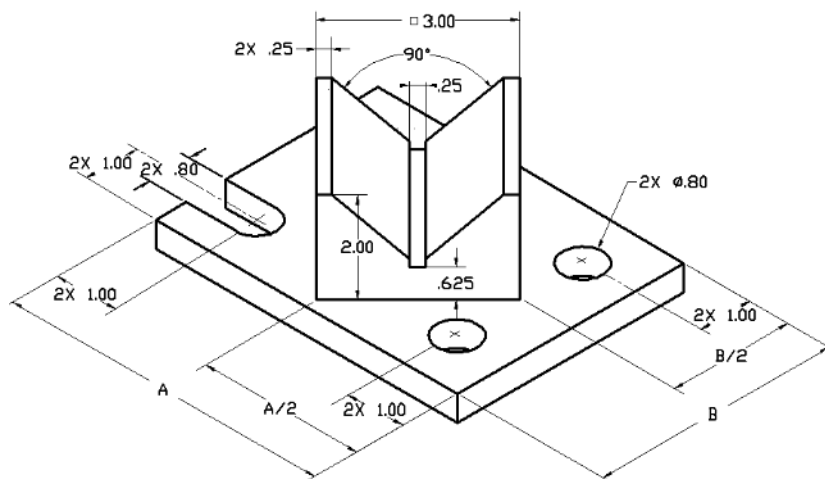
1. **Create** a New part in SolidWorks.
2. **Build** the illustrated model. Apply symmetry. Think about the various features that create the part. Insert six features: Extruded Base, two Extruded Cuts, Mirror, Extruded Boss, and a third Extruded Cut.

3. **Set** the document properties for the model.
4. Create **Sketch1**. Select the Top Plane as the Sketch plane. Apply symmetry. The part Origin is located in the center of the rectangle. Insert the required relations and dimensions.

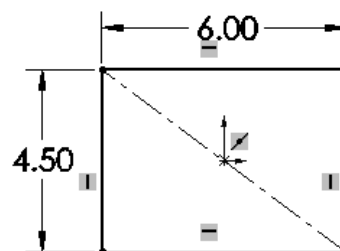
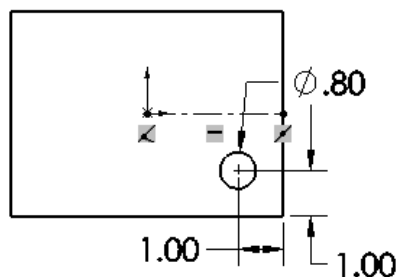
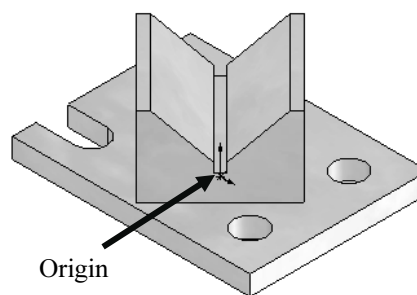
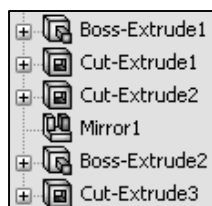
5. Create the **Extruded Base** (Boss-Extrude1) feature. Blind is the default End Condition in Direction 1. Depth = .50in.

6. Create **Sketch2**. Select the top face of Boss-Extrude1 for the Sketch plane. Sketch a circle. Insert the required relations and dimensions.

7. Create the first **Extruded Cut** feature. Select Through All as End Condition in Direction1.



Given:
A = 6.00, B = 4.50
Material: 2014 Alloy
Plate thickness = .50
Units: IPS
Decimal places = 2



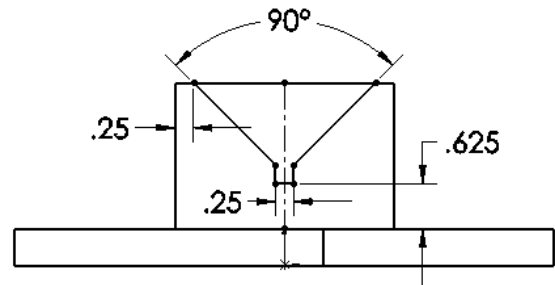
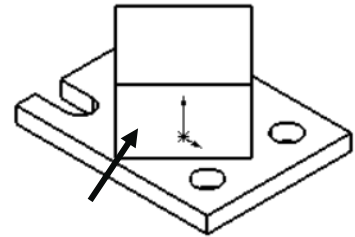
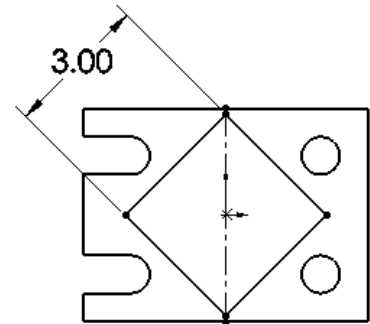
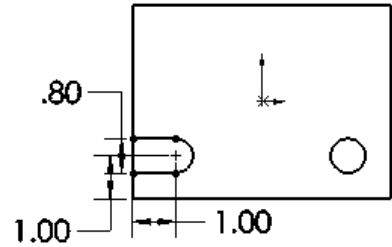
8. Create **Sketch3**. Select the top face of Boss-Extrude1 for the Sketch plane. Insert the required geometric relations and dimensions.



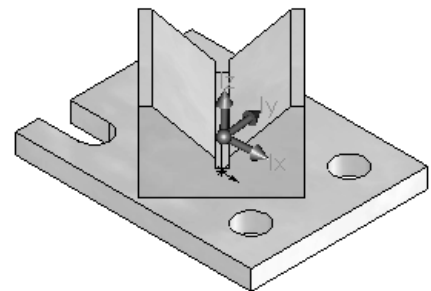
Click **View, Temporary** axes to view the part temporary axes in the Graphics window.

9. Create the second **Extruded Cut** feature. Select Through All as End Condition in Direction1.
10. Create the **Mirror** feature. Mirror the two Extruded Cut features about the Front Plane.
11. Create **Sketch4**. Select the top face of Boss-Extrude1 as the Sketch plane. Apply construction geometry to center the sketch. Insert the required relations and dimensions.
12. Create the **Extruded Boss** feature. Blind is the default End Condition in Direction 1. Depth = 2.00in.
13. Create **Sketch5**. Select the front face of Boss-Extrude as illustrated for the Sketch plane. Sketch5 is the profile for the third Extruded Cut feature. Apply construction geometry. Insert the required dimensions and relations.
14. Create the third **Extruded Cut** feature. Through All is selected for End Condition in Direction 1 and Direction 2.
15. **Assign** 2014 Alloy material to the part.
16. **Calculate** the volume of the part. The volume = 25.12 cubic inches.
17. **Locate** the Center of mass. The location of the Center of mass is derived from the part Origin.

- X: 0.06 inches
- Y: 0.80 inches
- Z: 0.00 inches



| |
|-------------------------------|
| Mass = 2.54 pounds |
| Volume = 25.12 cubic inches |
| Surface area = 88.33 inches^2 |
| Center of mass: (inches) |
| X = 0.06 |
| Y = 0.80 |
| Z = 0.00 |



18. **Save** the part and name it Part-Modeling 3-4.

19. **Close** the model.

Summary

Basic Part Creation and Modification and Intermediate Part Creation and Modification are two of the five categories on the CSWA exam.

There are two questions on the CSWA exam in the *Basic Part Creation and Modification* category. One question is in a multiple choice single answer format and the other question (Modification of the model) is in the fill in the blank format. Each question is worth fifteen (15) points for a total of thirty (30) points.

There are two questions on the CSWA exam in the *Intermediate Part Creation and Modification* category. One question is in a multiple choice single answer format and the other question (Modification of the model) is in the fill in the blank format. Each question is worth fifteen (15) points for a total of thirty (30) points.

The main difference between the *Basic Part Creation and Modification* category and the *Intermediate Part Creation and Modification* or the *Advance Part Creation and Modification* category is the complexity of the sketches and the number of dimensions and geometric relations along with an increase in the number of features.

At this time, there are no modeling questions on the exam that requires you to use Sheet Metal, Loft, or Swept features.



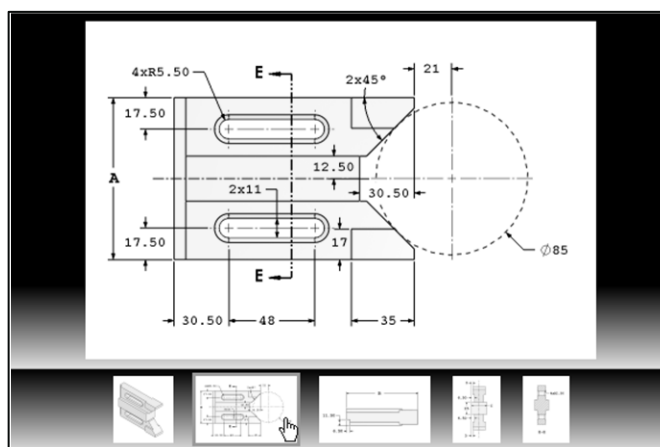
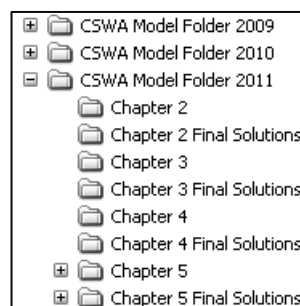
View sample screen shots from an older CSWA exam at the end of the Homework section in this chapter.



There are no Surfacing or Boundary feature questions on the CSWA exam.



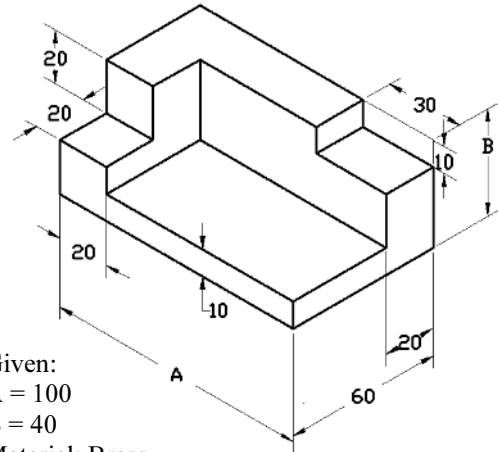
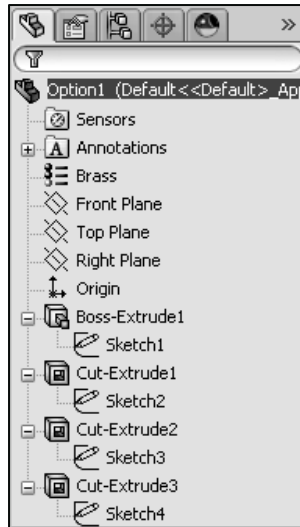
During the CSWA exam; SolidWorks provides various model views. Click on the additional views during the exam to better understand the part and provided information. Read each question carefully. Identify the dimensions, center of mass, units and location of the Origin. Apply needed material.



Questions

1: In Tutorial, Volume / Center of mass 3-2 you built the model using the FeatureManager that had three features vs. four features in the FeatureManager.

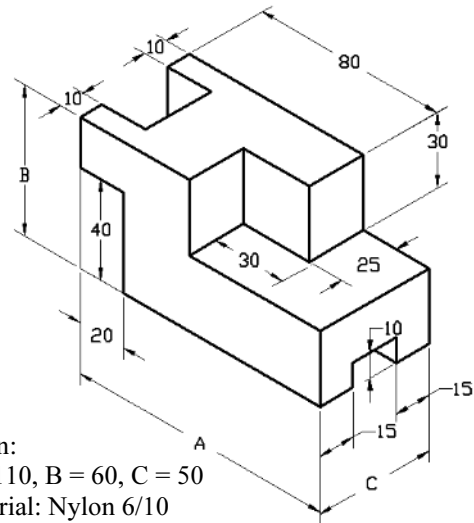
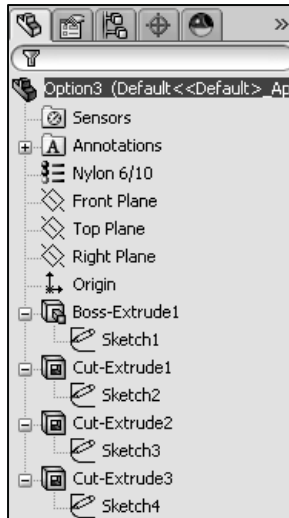
Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information using the Option1 FeatureManager.



Given:
 $A = 100$
 $B = 40$
 Material: Brass
 Density = $.0085 \text{ g/mm}^3$
 Units: MMGS

2. In Tutorial, Mass / Volume 3-4 you built the model using the FeatureManager that had three features vs. four features.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information using the Option3 FeatureManager.

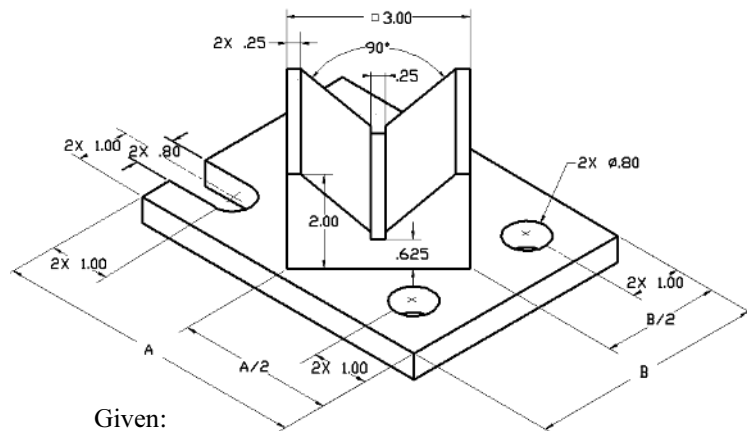


Given:
 $A = 110$, $B = 60$, $C = 50$
 Material: Nylon 6/10
 Density = $.0014 \text{ g/mm}^3$
 Units: MMGS

Basic and Intermediate Part Creation

3. In Tutorial, Basic/Intermediate 3-4 you built the illustrated model. Modify the plate thickness from .50in to .25in. Modify the Sketch5 angle from 90deg to 75deg. Re-assign the material from 2014 Alloy to 6061 Alloy.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information.



Given:

A = 6.00, B = 4.50

Material: 2014 Alloy

Plate thickness = .50

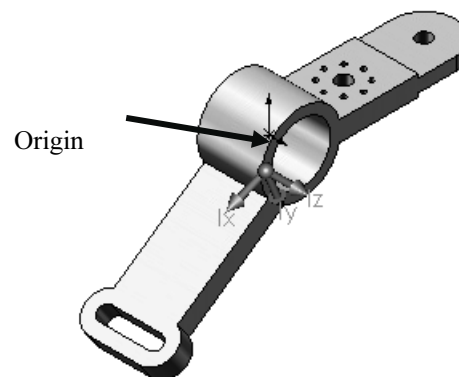
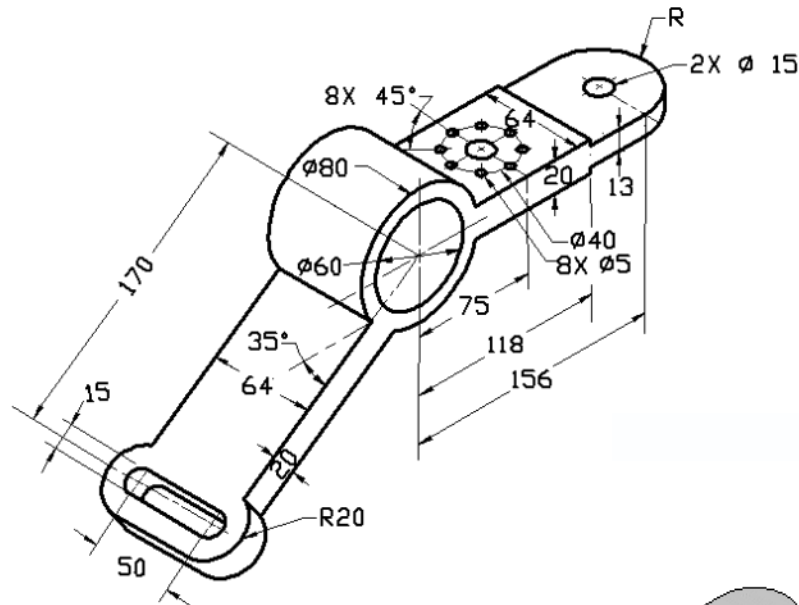
Units: IPS

Decimal places = 2

4. Build this model: Set document properties, identify the correct Sketch planes, apply the correct Sketch and Feature tools and apply material.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided illustrated information.

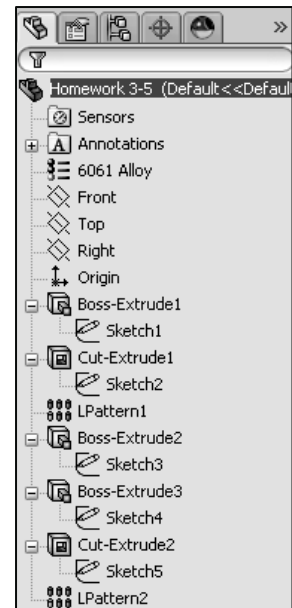
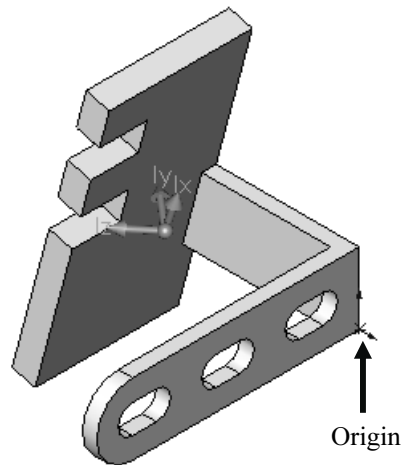
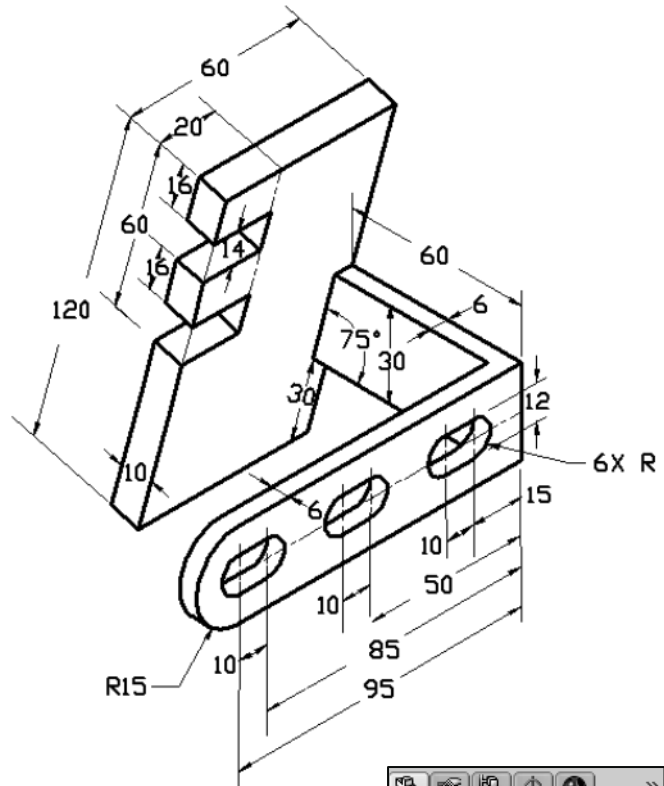
- Material: 6061 Alloy
- Units: MMGS



5. Build this model. Set document properties and identify the correct Sketch planes. Apply the correct Sketch and Feature tools, and apply material.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information.

- Material: 6061 Alloy
- Units: MMGS

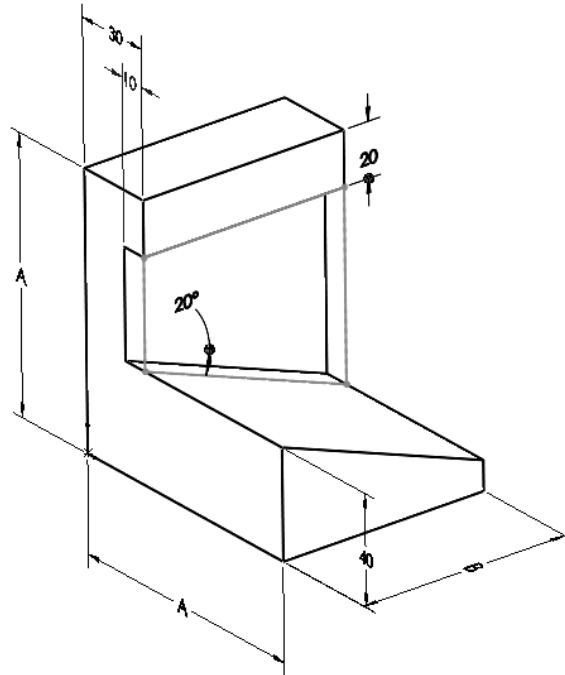


Basic and Intermediate Part Creation

6. Build this model. Set document properties and identify the correct Sketch planes. Apply the correct Sketch and Feature tools, and apply material.

Calculate the overall mass of the part with the provided information. Note: The Origin is arbitrary.

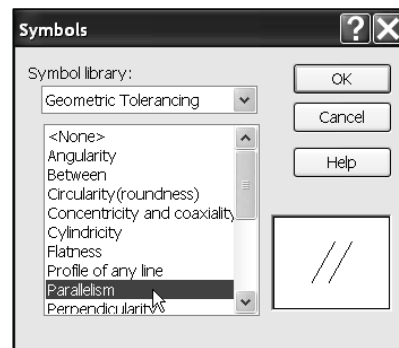
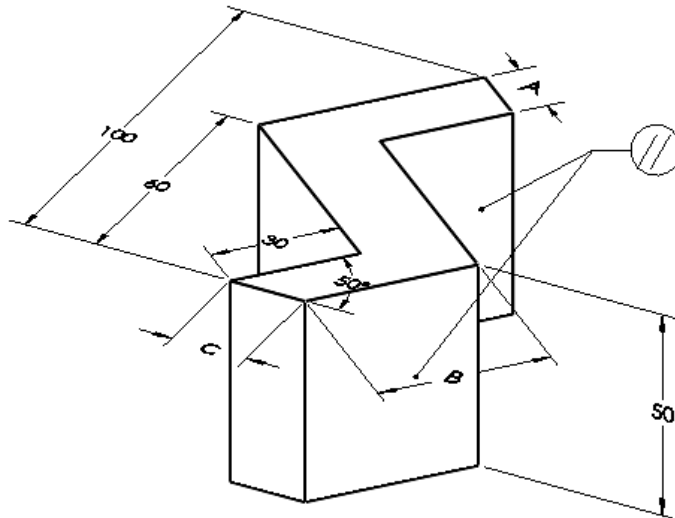
- Material: Copper
- Units: MMGS
- $A = 100$
- $B = 80$



7. Build this model. Set document properties and identify the correct Sketch planes. Apply the correct Sketch and Feature tools, and apply material.

Calculate the overall mass of the part with the provided information. The location of the Origin is arbitrary.

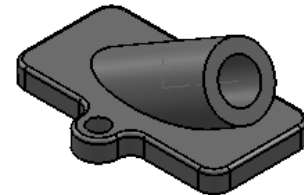
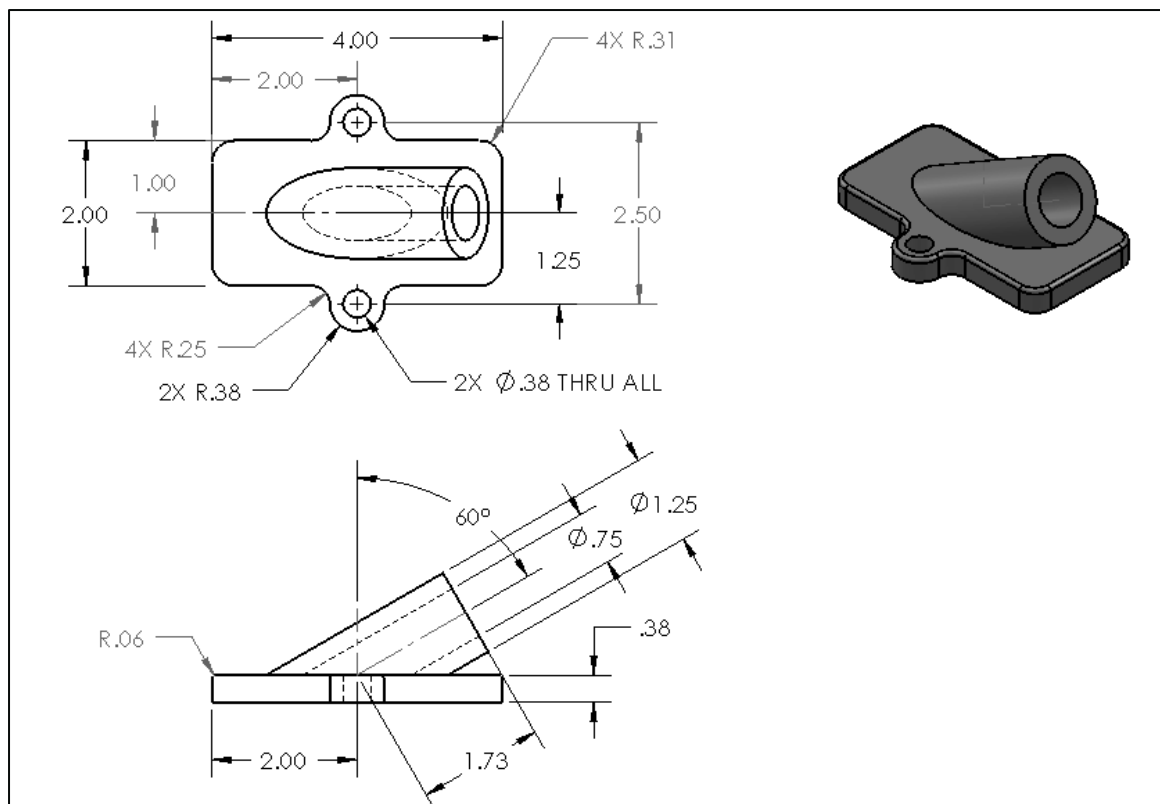
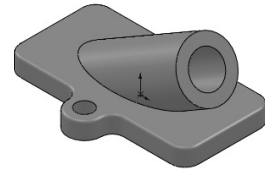
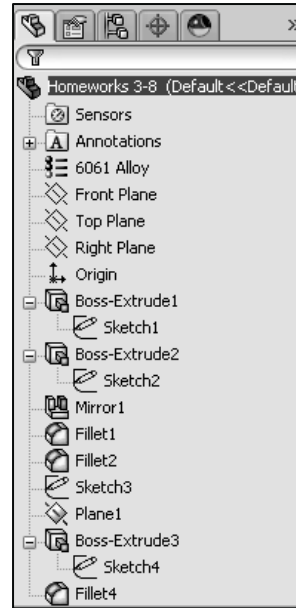
- Material: 6061
- Units: MMGS
- $A = 16$
- $B = 40$
- Side A is perpendicular to side B
- $C = 16$



8. Build this model. Set document properties and identify the correct Sketch planes. Apply the correct Sketch and Feature tools, and apply material.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information.

- Material: 6061
- Units: IPS
- View the provided drawing views for details

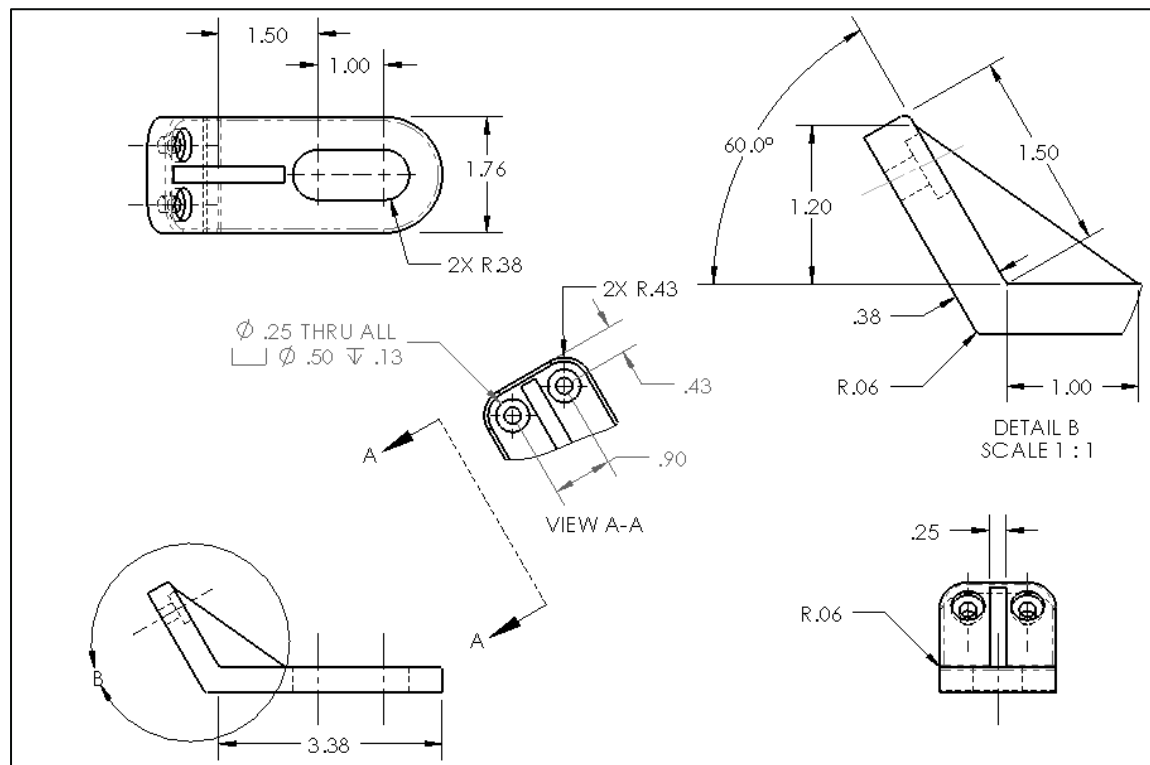
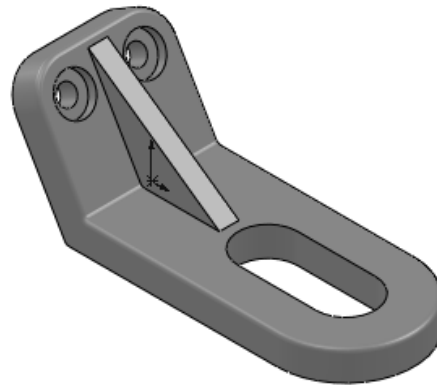
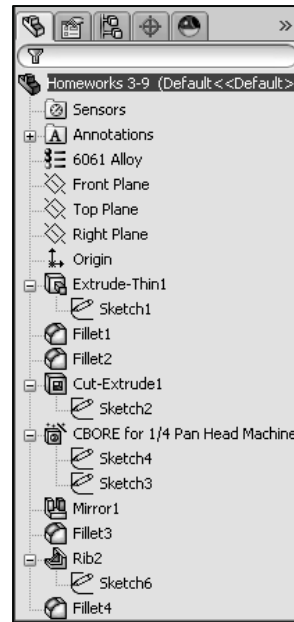


Basic and Intermediate Part Creation

9. Build this model. Set document properties and identify the correct Sketch planes. Apply the correct Sketch and Feature tools, and apply material.

Calculate the overall mass of the part, volume, and locate the Center of mass with the provided information.

- Material: 6061
- Units: IPS
- View the provided drawing views for details

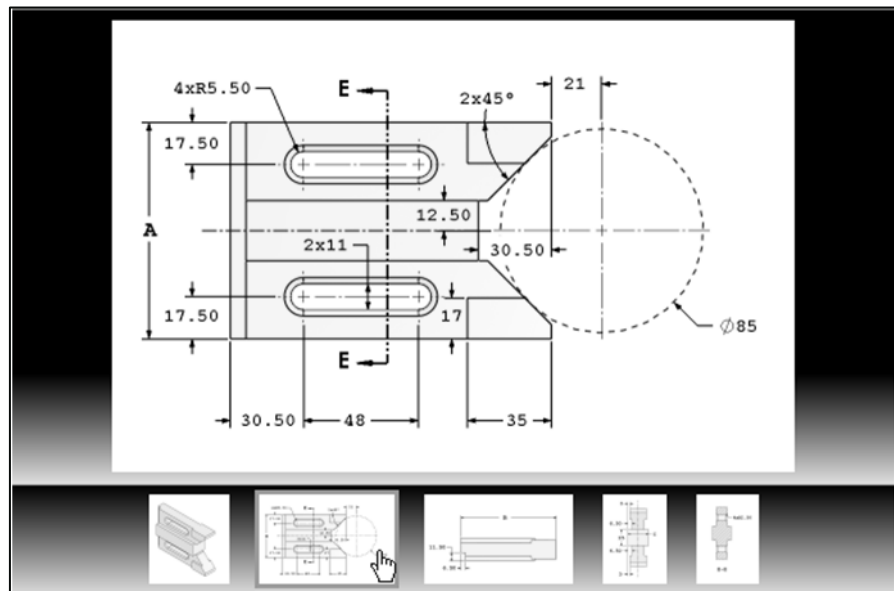
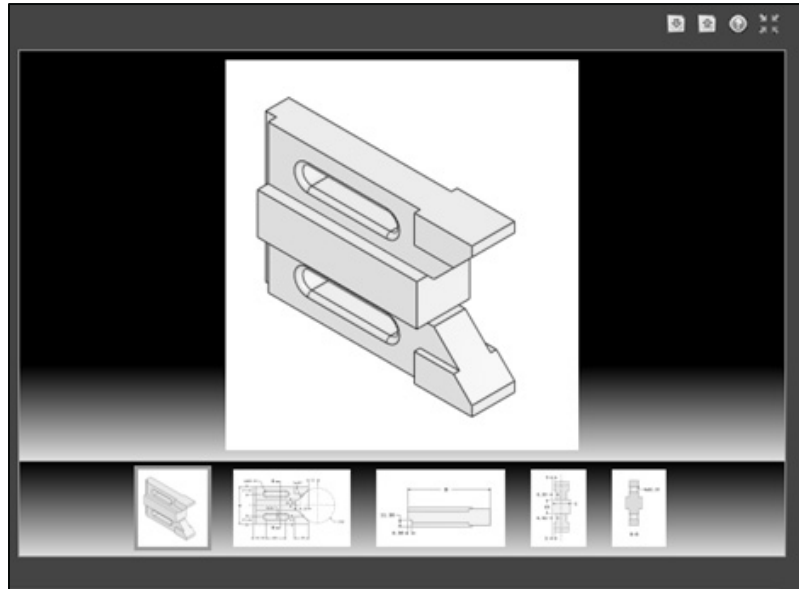




Screen shots from an older CSWA exam for a Basic/Intermediate part.

Click on the additional views to understand the part and to provide information. Read each question carefully.

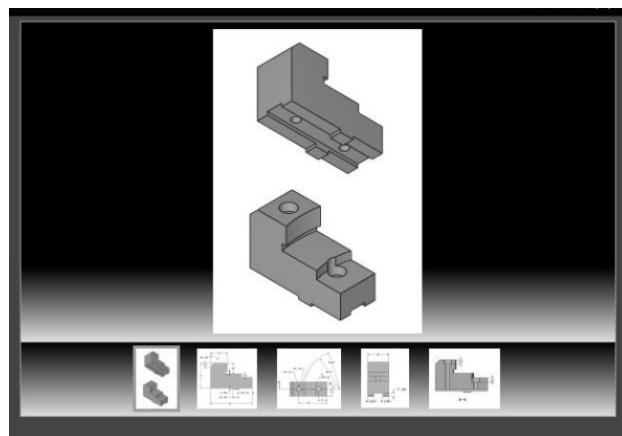
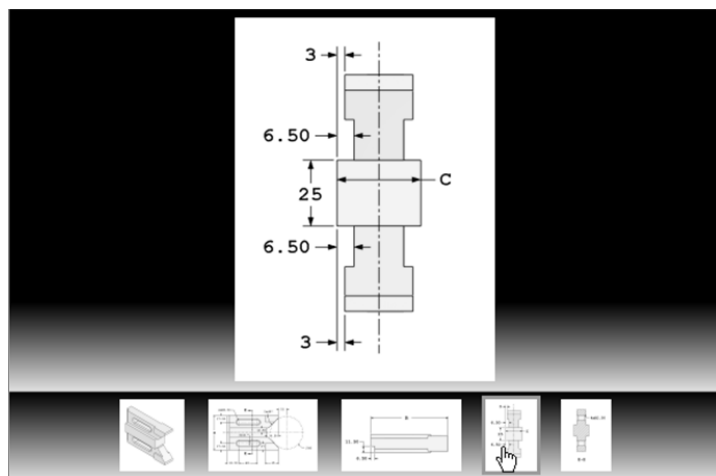
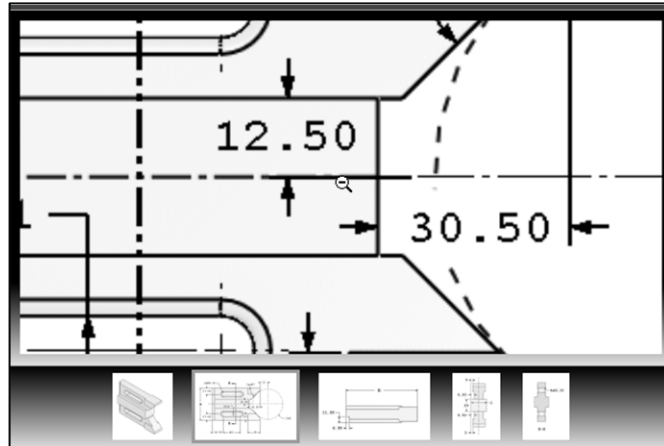
Understand the dimensions, center of mass and units. Apply needed materials.

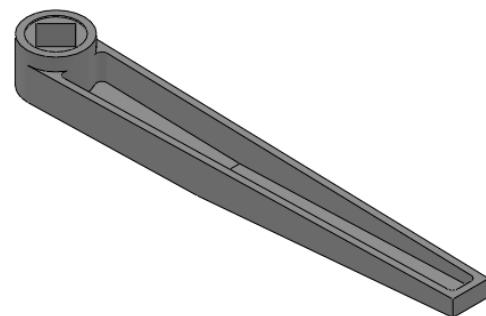
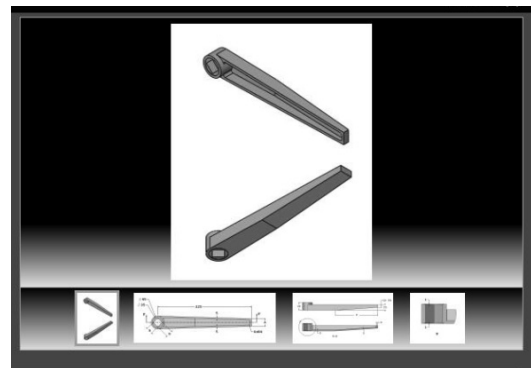
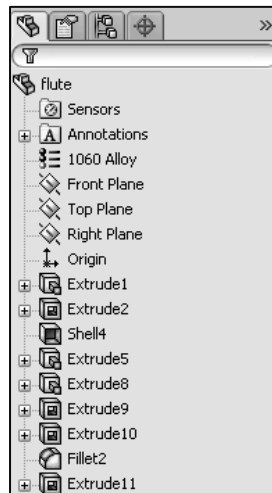
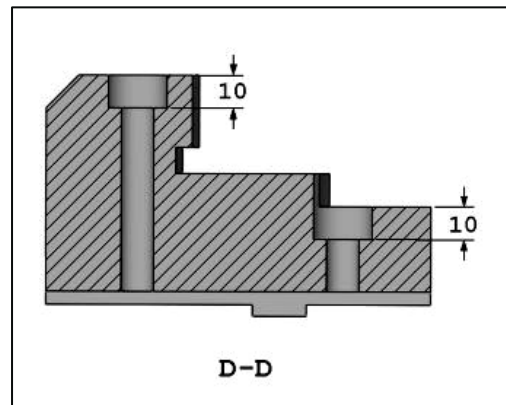
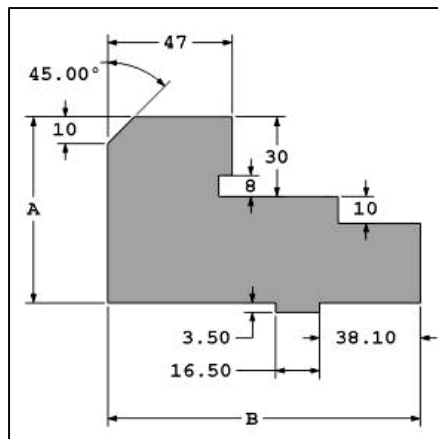
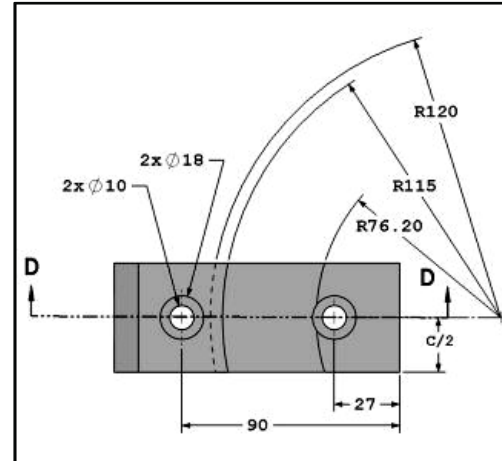
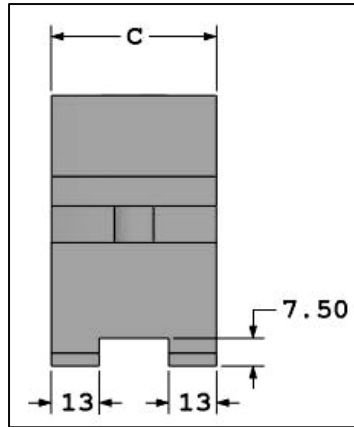


Basic and Intermediate Part Creation



Zoom in on the part or view if needed.





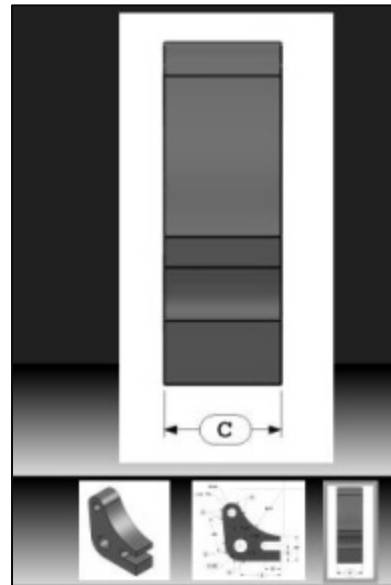
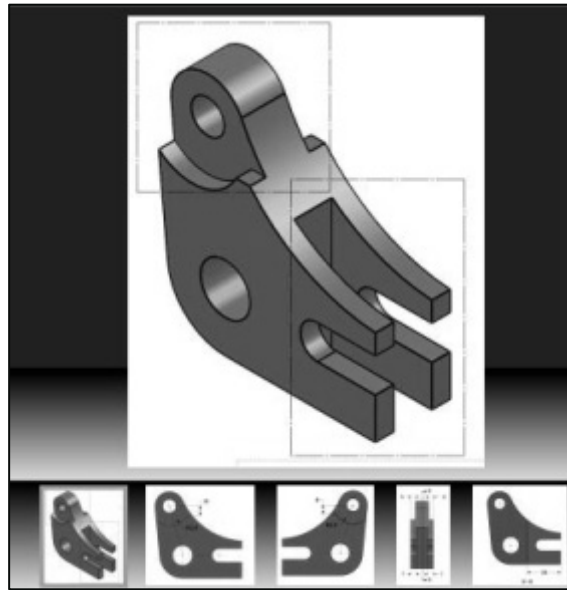
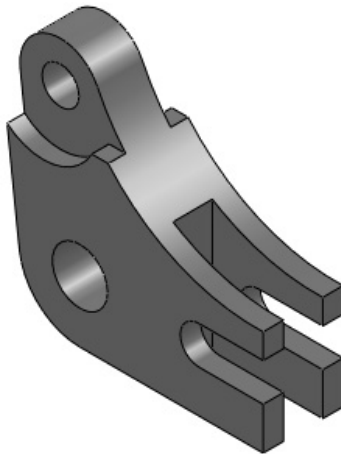
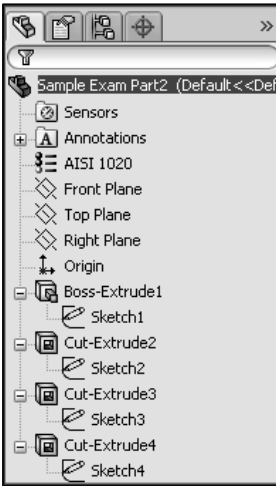
Basic and Intermediate Part Creation



Screen shots from an older CSWA exam for a Basic/Intermediate part.

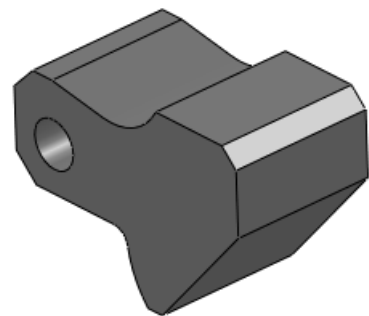
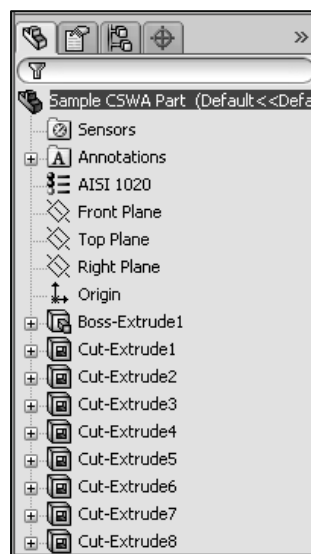
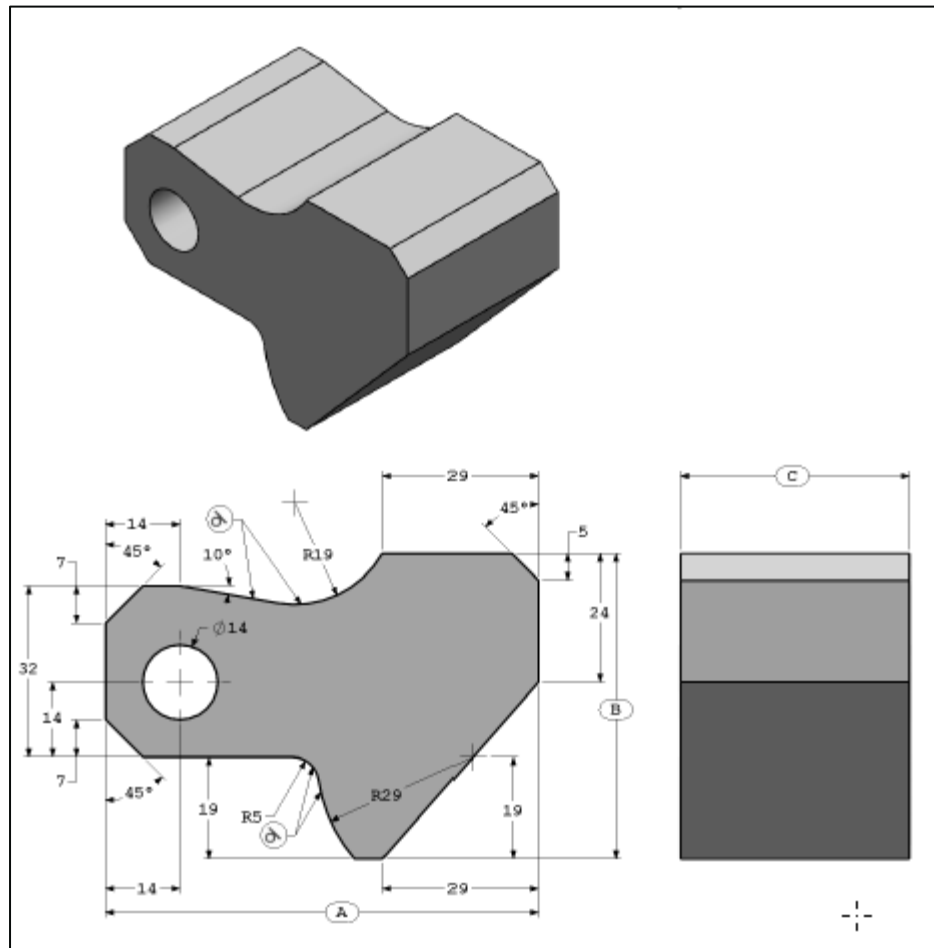
Click on the additional views to understand the part and to provide information. Read each question carefully.

Understand the dimensions, center of mass and units. Apply needed materials.





Screen shot from the new sample CSWA exam for a Basic/Intermediate part. See .pdf file on the CD for additional model information and solution.



Basic and Intermediate Part Creation



Screen shot from the new sample CSWA exam for a Basic/Intermediate part. See .pdf file on the CD for additional model information and solution.

