# Official Certified SolidWorks Associate (CSWA) Examination Guide
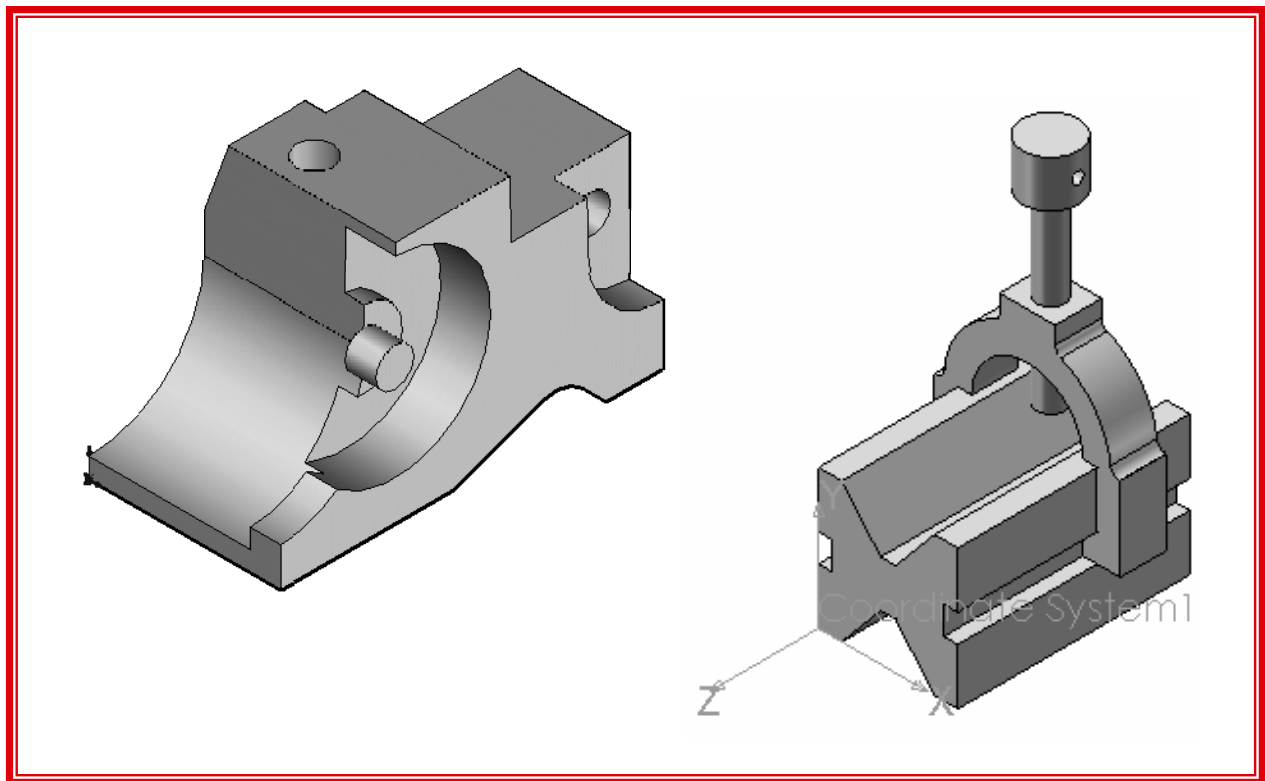
**SolidWorks 2008     SolidWorks 2009     SolidWorks 2010**
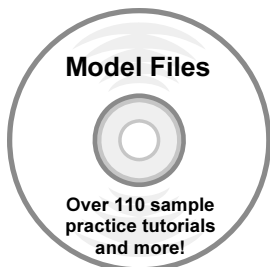
## The only authorized CSWA preparation exam guide

By David C. Planchard & Marie P. Planchard (CSWP)

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# CHAPTER 2: BASIC THEORY AND DRAWING THEORY

## Objectives

| Category |
| --- |
| Drawing Theory |
| Advanced modeling theory and analysis |
| Part Modeling |
| Advanced Part Modeling |
| Assembly Modeling |

Drawing Theory is one of the five categories on the CSWA exam. This chapter covers the general concepts, symbols and terminology (Basic Theory) and then the core element (Drawing Theory) which is aligned to the CSWA exam.

There are two questions on the CSWA exam in the Drawing Theory category. Each question is worth five (5) points. The two questions are in a multiple choice single answer format.

In the Drawing Theory category of the exam, you are not required to perform an analysis on a part, assembly, or drawing but you are required to have general drawing knowledge and understanding of drawing view methods.

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

- Recognize 3D modeling techniques:

  - How parts, assemblies, and drawings are related, Identify the feature type icon, identify parameters, and dimensions, identify the correct standard reference planes: Top, Right, and Front, and determine the design intent for a model

- Identify and understand the procedure for the following:

  - Assign and edit material to a part, Apply the Measure tool to a part or an assembly, Locate the Center of mass, and Principal moments of inertia relative to the default coordinate location, and Origin.

- Calculate the overall mass and volume of a part

- Identify the process of creating a simple drawing from a part or an assembly

- Identify the procedure to create a named drawing view

- Specify Document Properties: Select Unit System, and Set Precision

☼ In the Part Modeling, Advanced Part Modeling and Assembly Modeling categories, you are required to read and interpret all types of drawing views.

☼ The SolidWorks models in this book can be open using either SolidWorks 2008, SolidWorks 2009 or SolidWorks 2010.
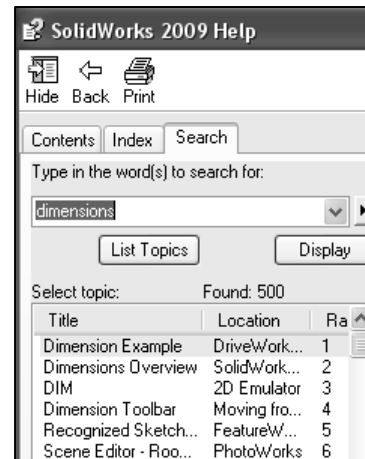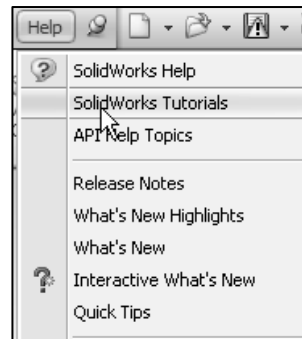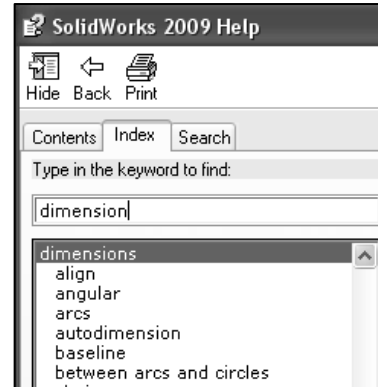
Utilize SolidWorks Help topics during the exam. The SolidWorks Help section is divided into three information tabs:

- **Contents** tab: Contains the SolidWorks Online User's Guide documents.

- **Index** tab: Contains additional information on key entered words.

- **Search** tab: To locate entered information.

Click Help, SolidWorks Tutorials to access the online tutorials. Explore the Tutorials by Category and by Certification as illustrated.

# 3D Modeling techniques

This chapter covers the general concepts, symbols, and terminology (Basic Theory) and then the core element (Drawing Theory) which is aligned to the CSWA exam. We will first address Basic Theory:

- How parts, assemblies, and drawings are related

- Identify the feature type, parameters, and dimensions

- Identify the correct standard reference planes: Top, Right, and Front

- Determine the design intent for a model

In a SolidWorks application, each part, assembly, and drawing is referred to as a document. Each document is displayed in a separate Graphics window.

## How parts, assemblies, and drawings are related

Parts, assemblies, and drawings are associative. This means that changes made in one area are automatically reflected in all the associated areas affected by that change. For example, changes that you make to a part are reflected in the associative assembly and drawing.

Typically, you design each part, combine the parts, "components" into an assembly, and then generate drawings of the assembly for inspection and manufacturing. The associative part, assembly, and drawing share a common data base.
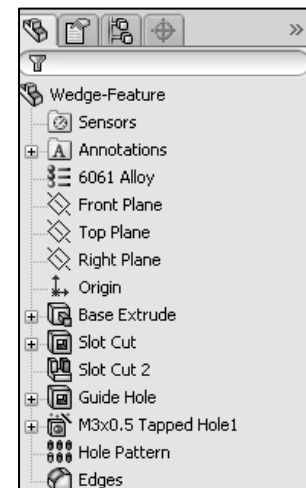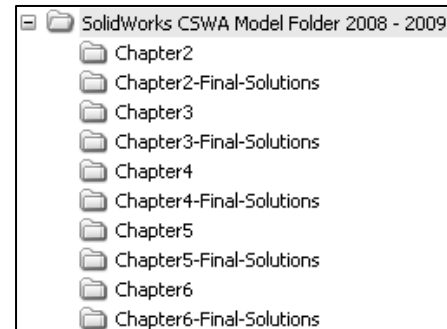
The SolidWorks models in this book can be open using either SolidWorks 2008, SolidWorks 2009 or SolidWorks 2010.

### Tutorial: Associativity 2-1

Verify the association between a part, assembly, and drawing.

1. **Open** the Wedge-Feature part from the SolidWorks CSWA Folder\Chapter2 location as illustrated. The part is displayed in the Graphics window.

2. **Open** the Wedge-Feature drawing from the SolidWorks CSWA Folder\Chapter2 location. View the FeatureManager. View the 120mm length dimension in the Front view.

3. **Open** the Wedge-Tube assembly from the SolidWorks CSWA Folder\Chapter2 location. View the assembly in the Graphics window. The Wedge-Feature part, Wedge-Feature drawing and the Wedge-Tube assembly are active documents.
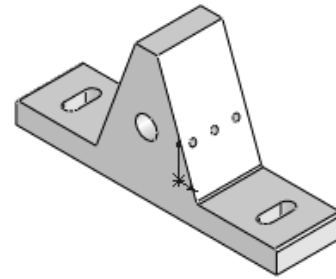
Modify the Extruded Base feature depth dimension of the part.
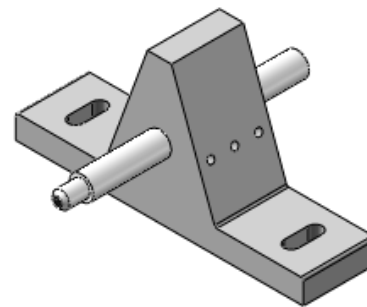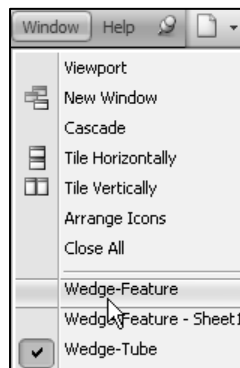
Return to the Wedge-Feature part.
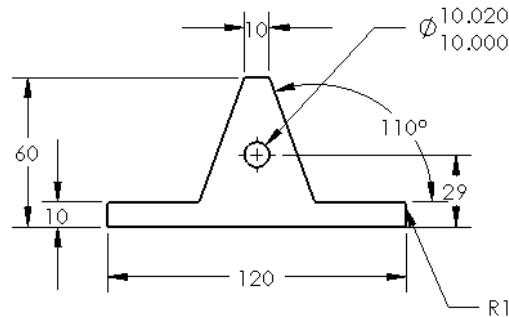
4. Click **Window**, **Wedge-Feature** from the Menu bar menu.

5. Click **Base-Extrude** from the Wedge-Feature Part FeatureManager.
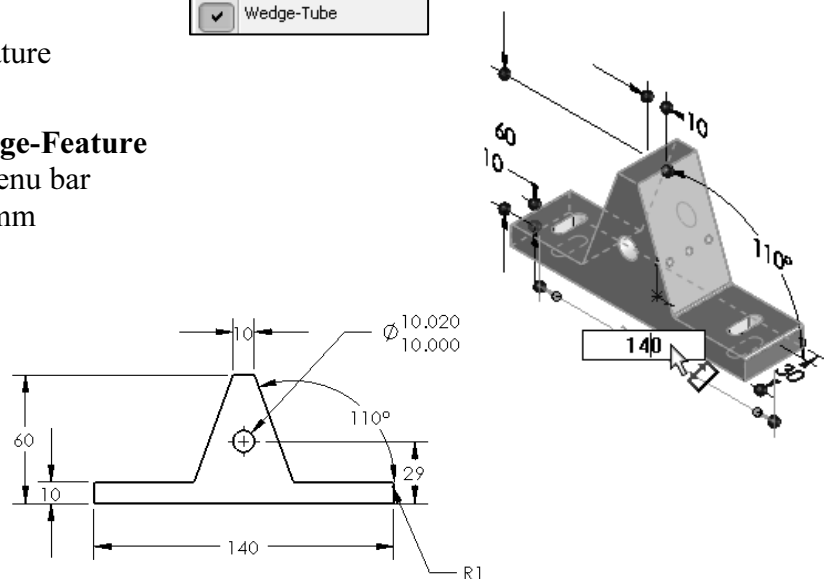
6. **View** the dimensions in the Graphics window.

Modify the Extruded Base feature length dimension.

7. Click the **120**mm length dimension in the Graphics window.

8. Enter **140**.

Return to the Wedge-Feature drawing.

9. Click **Window, Wedge-Feature – Sheet1** from the Menu bar menu. View the 140mm dimension in the drawing.

View the Wedge-Tube assembly.

10. Click **Window, Wedge-Tube** from the Menu bar menu.

11. **View** the length dimension of Base Extrude in the Wedge-Feature component. The length dimension is modified from 120mm to 140mm.

12. **Close** all models. Think about the association between the part, assembly, and drawing.
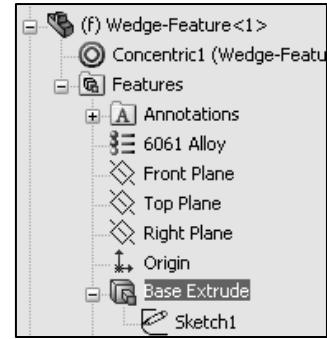
Understand the procedure to open and close a part from a drawing or from an assembly. Understand the process to edit features and sketches from the FeatureManager design tree.
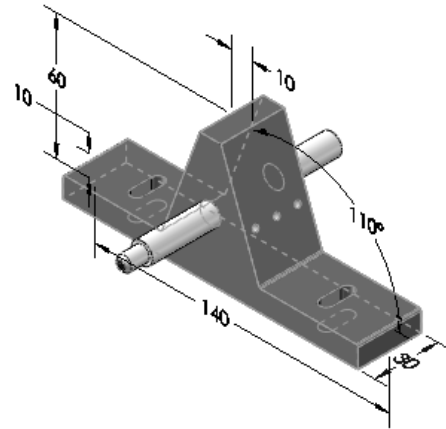
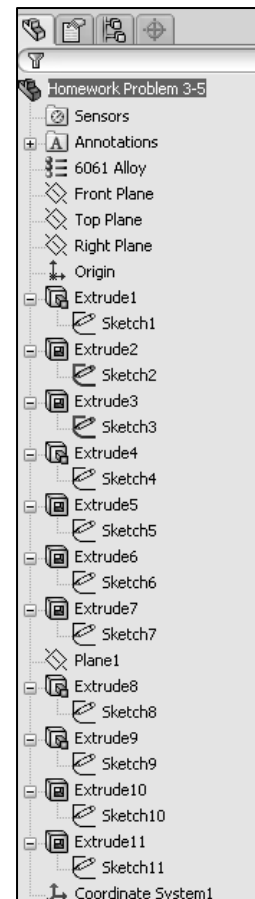# Identify the Feature type by icon

A part is a 3D model, which consists of features. What are features?

- Features are geometry building blocks

- Features add or remove material

- Features are created from 2D or 3D sketched profiles or from edges and faces of existing geometry

You can use the same sketch to create different features.

The CSWA test is a three hour exam. Apply symmetry to decrease design time.

Optimize the number of features required to build your model. In the next three chapters, you will require the knowledge to create a Basic model (5 – 7 features), Advanced model (8 – 13 features), and an Assembly. All sketches should be fully defined!
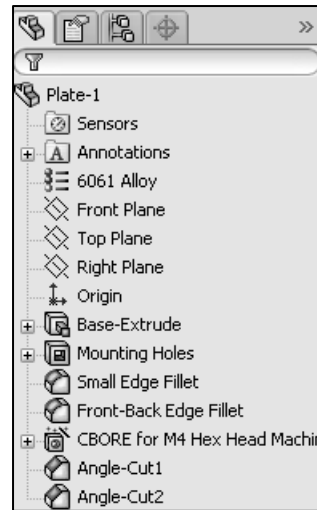
### Tutorial: Identify model features 2-1

Identify the features in the FeatureManager based on the feature icon and understand the procedure to build the Plate-1 part.
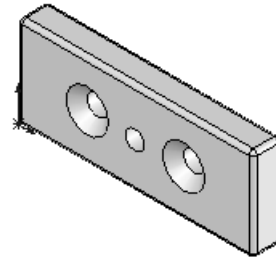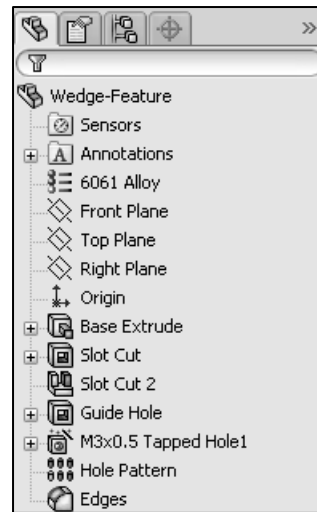


1.  **Open** the Plate-1 part from the SolidWorks CSWA Folder\Chapter2 location. View the illustrated model and FeatureManager design tree. The features were renamed.

    a.  Base-Extrude: Extruded Base feature

    b.  Mounting Holes: Extruded Cut feature for the two outside holes

    c.  Small Edge Fillet: Fillet feature for the top and bottom edges

    d.  Front-Back Edge Fillet: Fillet feature for the front and back side edges

    e.  CBORE: Hole Wizard feature for the center hole

    f.  Angle-Cut1: Chamfer feature for the left front hole

    g.  Angle-Cut2: Chamfer feature for the right front hole

2.  **Close** the model.



### Tutorial: Identify model features 2-2

Identify the features in the FeatureManager based on the feature icon and understand the procedure to build the Wedge-Feature part.
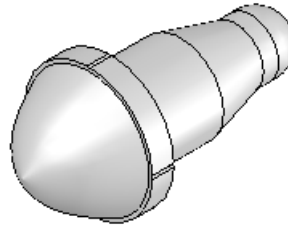


1.  **Open** the Wedge-Feature part from the SolidWorks CSWA Folder\Chapter2 location. The features were renamed.

    a.  Base Extrude: Extruded Base feature

    b.  Slot Cut: Extruded Cut feature

    c.  Slot Cut 2: Mirror feature

    d.  Guide Hole: Extruded Cut feature

    e.  M3X0.5 Tapped Hole1: Hole Wizard feature

    f.  Hole Pattern: Linear Pattern feature

    g.  Edges: Fillet feature

2.  **Close** the model.
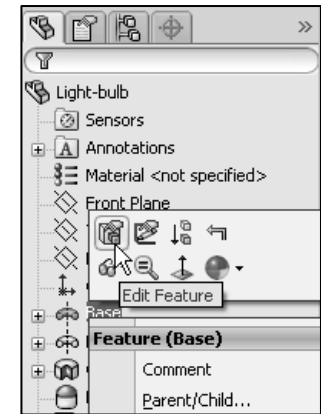
### Tutorial: Identify model features 2-3

Identify the features in the FeatureManager based on the feature icon and understand the procedure to build the Light-bulb part.

1. **Open** the Light-bulb part from the SolidWorks CSWA Folder\Chapter2 location. View the illustrated model and FeatureManager. The features were renamed.

   a. Base: Revolved Base feature

   b. Base-Top: Revolved Boss feature

   c. Cut-Revolve-Thin1: Revolved Cut feature

   d. Back: Dome feature

   e. Seed Cut: Extruded Cut feature

   f. PatternSeedCut: Circular Pattern feature

2. **Close** the model.

Edit the sketches and features in the provided FeatureManager to explore how each model was built.

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

### Tutorial: Identify model features 2-4

Identify the features in the FeatureManager based on the feature icon and understand the procedure to build the Machine Screw part.

1. **Open** the Machine Screw part from the SolidWorks CSWA Folder\Chapter2 location. View the illustrated model and FeatureManager. The features were renamed.

   a. Base: Revolved Base feature

   b. Slot: Extruded Cut feature

   c. Pattern: Circular Pattern feature

   d. Edge: Fillet feature

   e. End: Chamfer feature
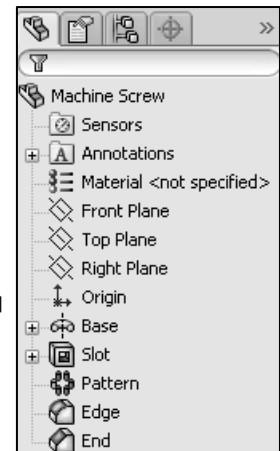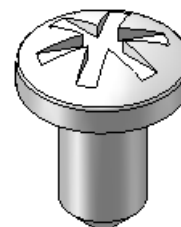
2. **Close** the model.

### Tutorial: Identify model features 2-5

Identify the features in the FeatureManager based on the feature icon and understand the procedure to build the Small O-Ring part.

1. **Open** the Small O-Ring part from the SolidWorks CSWA Folder\Chapter2 location. View the illustrated model and FeatureManager. The features were renamed.

    a.  Base: Lofted Base feature

2. **Close** the model.

All questions on the exam are in a multiple choice single answer format.

☀ SolidWorks states that the first feature you create in a part is the Base. This feature is the basis on which you create the other features for the model. The Base feature can be an extrusion, a revolve, a sweep, a loft, thickening of a surface, or a sheet metal flange.
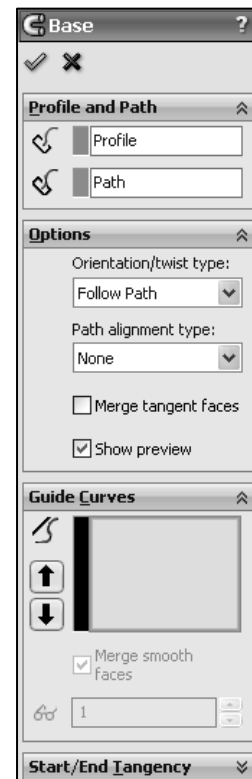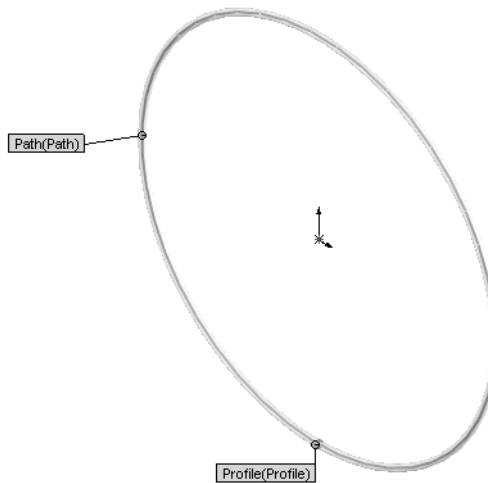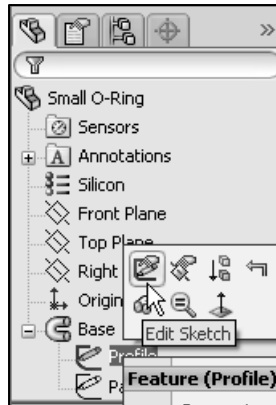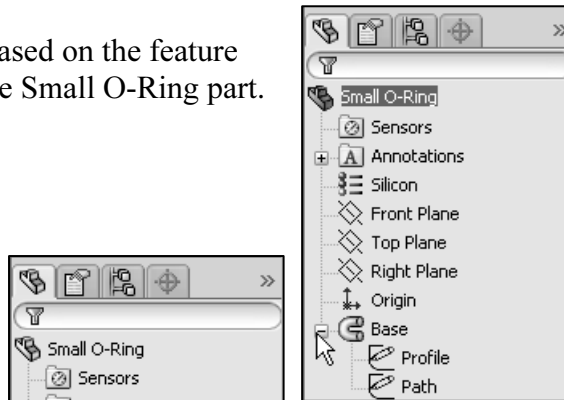
💡 Use the Features toolbar from the CommandManager in SolidWorks to help identify the feature tool icon during the exam.
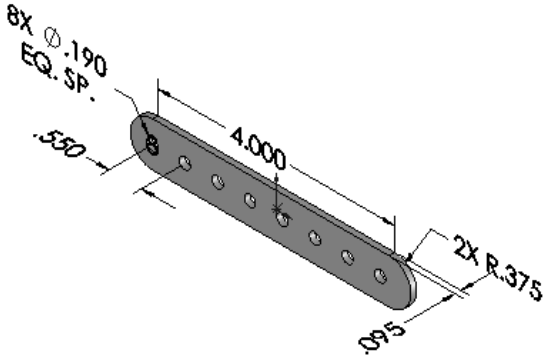


# Identify dimensions and parameters

Create model dimensions as you create each part feature. Then insert the dimensions into the various drawing views. Changing a dimension in the model updates the drawing, and changing an inserted dimension in a drawing view changes the model.

💡 By default, single model dimensions are displayed in black. This includes dimensions that are blue in the part or assembly document (such as the extrusion depth). Reference dimensions are gray by default, design table dimensions are magenta by default.

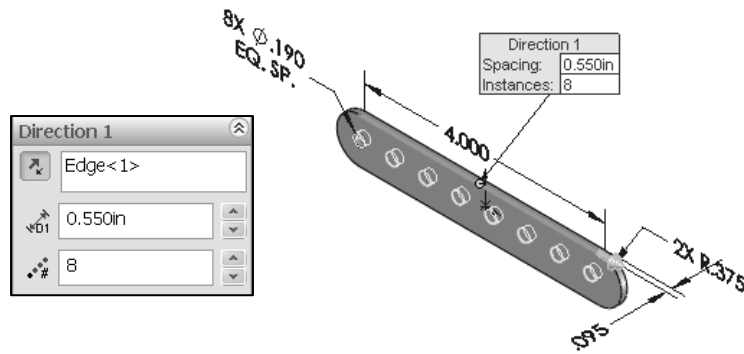### *Tutorial: Identify dimensions and parameters 2-1*

Understand the illustrated dimensions, annotations, and parameters of the model.

1. **Open** the Bar part from the SolidWorks CSWA Folder\Chapter2 location. View the illustrated dimensions, annotations, and parameters of the model.

2. Click **8X Ø .190 EQ. SP** in the Graphics window.

3. Click **OK**.

4. Click **OK**. Understand the procedure to insert dimension text from the Dimension PropertyManager.
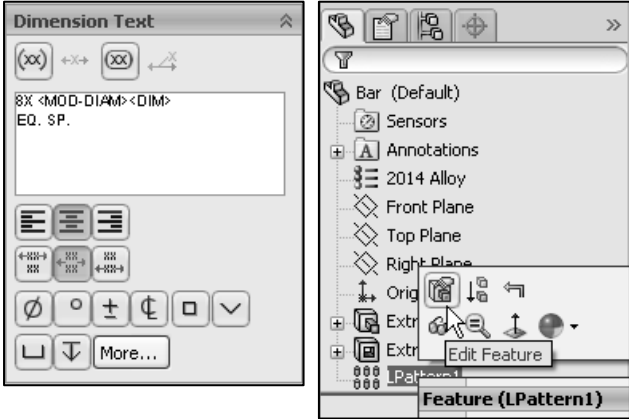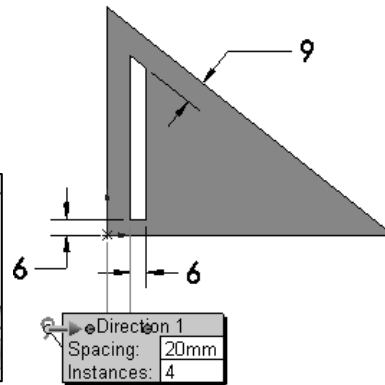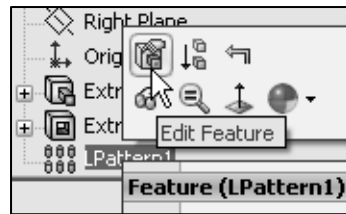
5. **Edit** LPattern1 in the FeatureManager. View the parameters, "number of instances". The number of instances = 8.

6. **Close** the model.

*Direction 1*
Edge<1>
0.550in
8

Direction 1
Spacing: 0.550in
Instances: 8

8X Ø .190
EQ. SP.

4.000

2X R .375

.565

### Tutorial: Identify dimensions and parameters 2-2

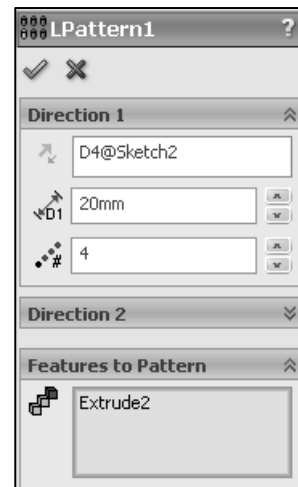Understand the illustrated dimensions and parameters of the model and the procedure to build the Triangle part.

1. **Open** the Triangle part from the SolidWorks CSWA Folder\Chapter2 location.

2. **Click** Extrude2 from the FeatureManager.

3. **View** the Extruded Cut feature dimensions.

4. **Edit** LPattern1 from the FeatureManager.

5. **View** the displayed parameters. Extrude2 is the seed feature for the Linear Pattern. D4@Sketch2 is the Pattern Direction. Spacing = 20mm.

Right Plane
Orig
Extr
Extr
LPattern1
Edit Feature
**Feature (LPattern1)**

9

6

6

Direction 1
Spacing: 20mm
Instances: 4

6. **Close** the model.

☀ The number of instances sets the number of pattern instances for Direction 1 and Direction 2. This number includes the original features or selections.

☀ SolidWorks states that a seed is, "a sketch or an entity (a feature, face, or body) that is the basis for a pattern.

**LPattern1** ?
✓ ✗
**Direction 1**
D4@Sketch2
20mm
4
**Direction 2**
**Features to Pattern**
Extrude2

### *Tutorial: Identify dimensions and parameters 2-3*

Understand the illustrated dimensions, and parameters of the model and the procedure to build the Wheel part.

1. **Open** the Wheel part from the SolidWorks CSWA Folder\Chapter2 location.

2. **Double-click** Extrude1 from the FeatureManager. View the Extruded Base feature dimensions.

3. **Edit** Sketch1 from the FeatureManager.

4. **Modify** the diameter dimension from 3.000in to 3.5000in.

5. **Edit** CirPattern1 from the FeatureManager. Extrude1 and Extrude2 are the seed features for the pattern. Number of instances = 8. Axis<1> is the Pattern Axis. (5) is the pattern instance to skip.

6. **Modify** the number of instances from 8 to 6.

7. **Close** the model.
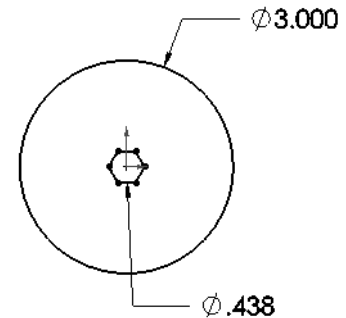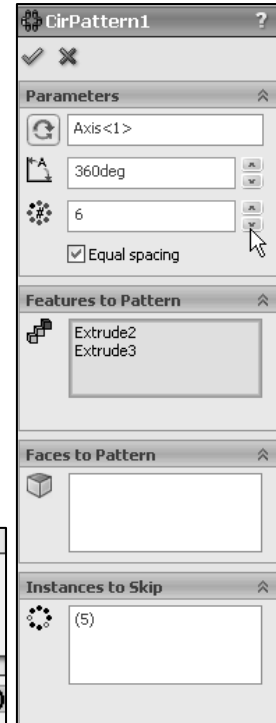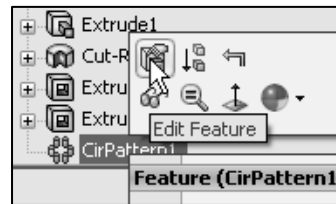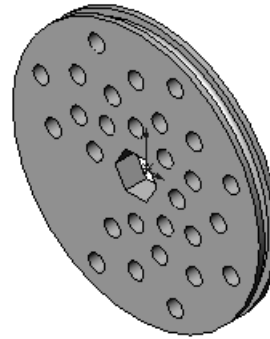
Instances to Skip, eliminates the pattern instances that you select in the Graphics window when you are creating the pattern.

Use the Rollback bar in the FeatureManager to learn how a model was created.
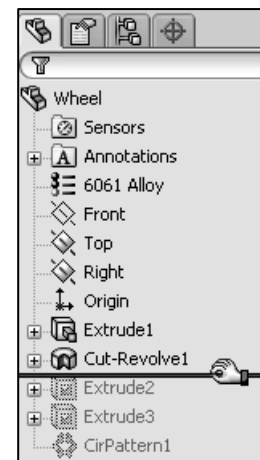
# Identify the correct reference planes: Top, Right, and Front

Most SolidWorks features start with a 2D sketch. Sketches are the foundation for creating features. SolidWorks provides the ability to create either 2D or 3D sketches.
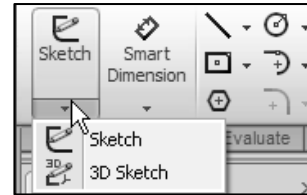
A 2D sketch is limited to a flat 2D Sketch plane. A 3D sketch can include 3D elements. As you create a 3D sketch, the entities in the sketch exist in 3D space. They are not related to a specific Sketch plane as they are in a 2D sketch.
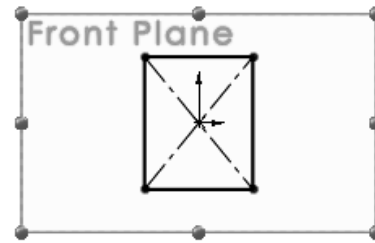
You may need to create a 3D sketch on the exam. The illustrated model displays a 3D sketch, using the Sketch Line tool for an Extruded Cut feature. You will create a 3D sketch in Chapter 3.
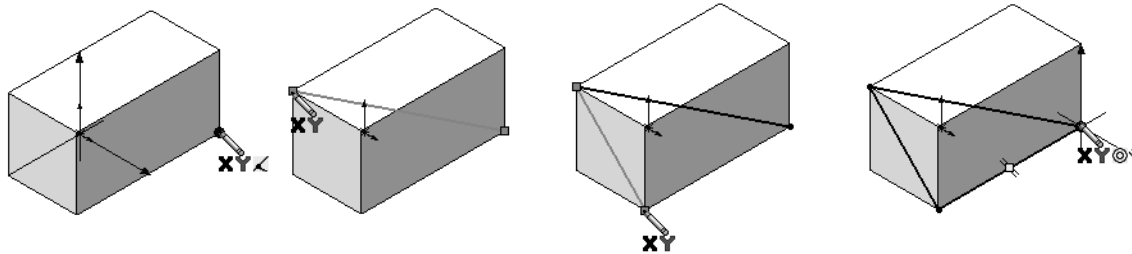
Does it matter where you start sketching a 2D sketch? Yes! When you create a new part or assembly, the three default planes are aligned with specific views.

The plane you select for your first sketch determines the orientation of the part. Selecting the correct plane is very important.

When you create a new part or assembly, the three default planes are aligned with specific views. The plane you select for the Base sketch determines the orientation of the part.
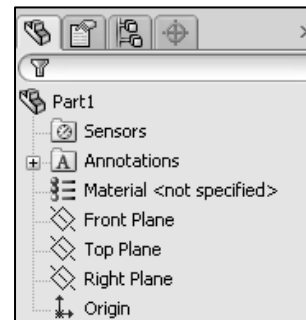
# 2D sketching / reference planes

The three default ⊥ reference planes, displayed in the FeatureManager design tree represent infinite 2D planes in 3D space. They are:

- Front
- Top
- Right

Planes have no thickness or mass. Orthographic projection is the process of projecting views onto parallel planes with ⊥ projectors. The default ⊥ datum planes are:

- Primary
- Secondary
- Tertiary

Use the following planes in a manufacturing environment:

- Primary datum plane: Contacts the part at a minimum of three points
- Secondary datum plane: Contacts the part at a minimum of two points
- Tertiary datum plane: Contacts the part at a minimum of one point

The part view orientation depends on the sketch plane. Compare the Front, Top, and Right Sketch planes for an L-shaped profile in the following illustration.

2D Profile      Front Plane      Top Plane      Right Plane
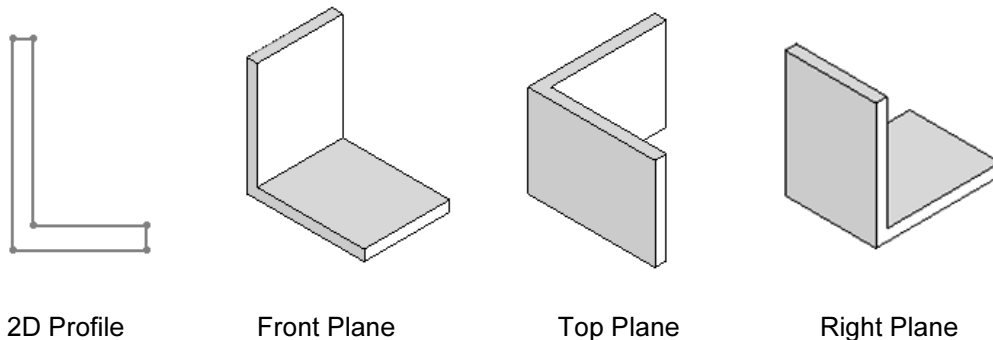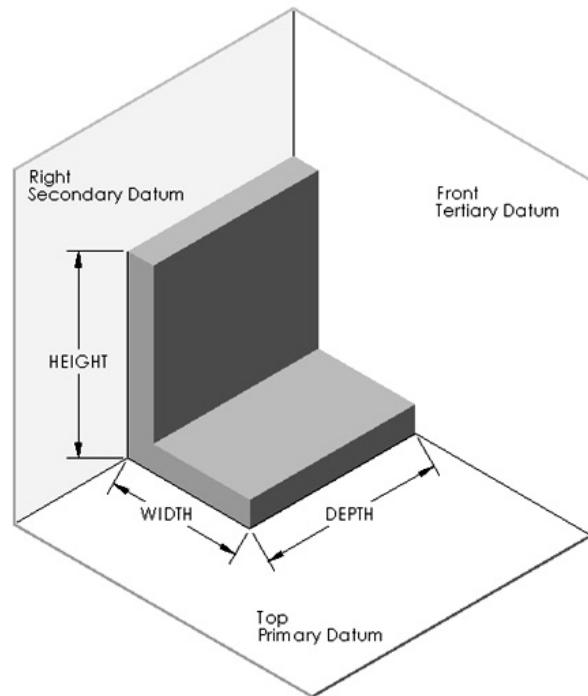
The six principle views of orthographic projection listed in the ASME Y14.3M standard are: Top, Front, Right side, Bottom, Rear, and Left side. SolidWorks Standard view names correspond to these orthographic projection view names.

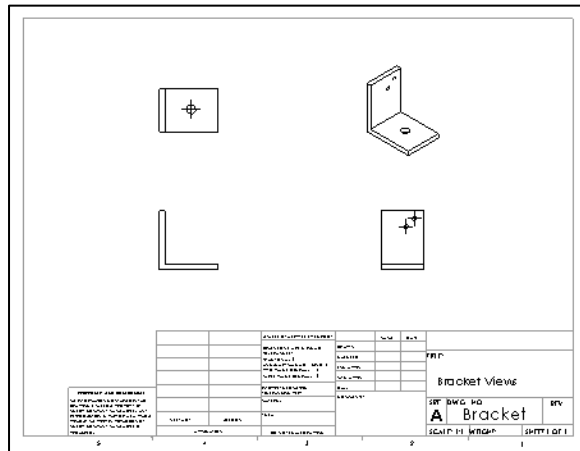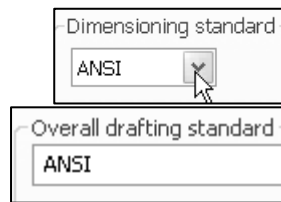| ASME Y14.3M Principle View Name: | SolidWorks Standard View: |
|---|---|
| Front | Front |
| Top | Top |
| Right side | Right |
| Bottom | Bottom |
| Rear | Back |
| Left side | Left |

The standard drawing views in third angle orthographic projection are: Front, Top, Right & Isometric.

☼ ANSI is the default dimensioning (Drafting) standard used in this book.

***Tutorial:***
***Reference plane 2-1***

1. **Create** a New part in SolidWorks as illustrated. Think about the steps that you would take to build the model. Identify the location of the part Origin and the reference plane for Sketch1. Sketch1 is the Base sketch.

Set the document properties for the model.

2. Select **ANSI**.

3. Select the **IPS** unit system.

4. Select **.1** for Length decimal place.

5. Click **OK**.

6. Create **Sketch1**. Select Front Plane as the Sketch plane. Apply the Line Sketch tool. Locate the sketch Origin in the bottom left corner. Insert the illustrated geometric relations and dimensions.
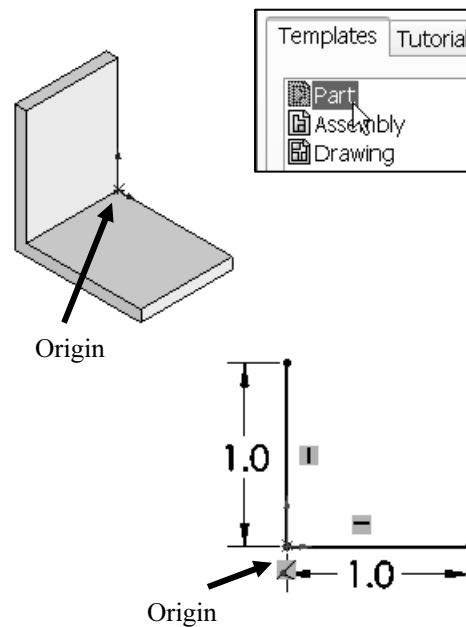
7. Create the **Extrude-Thin1** feature. Blind is the default End Condition in Direction 1. Depth = .8in. One-Direction is the default type in the Thin Feature box. Thickness = .1.

8. **Save** the part and name it Bracket-FrontPlane.

9. **Close** the model.
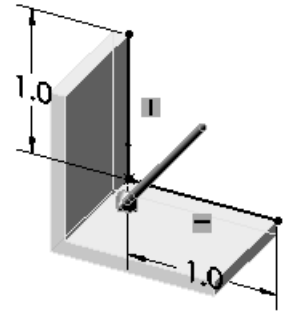
☼ Apply the Thin Features options to control the extrude thickness, not the depth.

### Tutorial: Reference plane 2-2

1. **Create** a New part in SolidWorks. Identify the location of the part Origin and the reference plane for Sketch1. Sketch1 is the Base sketch.

2. Set the document properties for the model. Select **ANSI**. Select the **IPS** unit system. Select **.1** for Length decimal places.

3. Create **Sketch1**. Select the Top Plane as the Sketch plane. Apply the Line Sketch tool. Locate the sketch Origin in the bottom left corner. Insert the illustrated geometric relations and dimensions.
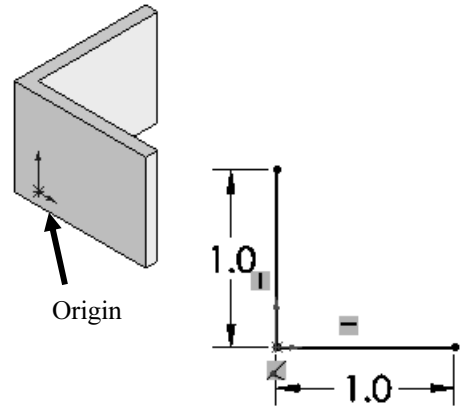
4. Create the **Extrude-Thin1** feature. Blind is the default End Condition in Direction 1. Depth = .8in. One-Direction is the default type in the Thin Feature box. Thickness = .1in.

5. **Save** the part and name it Bracket-TopPlane.

6. **Close** the model.

☼ To modify the Sketch plane in a sketch: Right-click the **Sketch**<#> in the FeatureManager, select **Edit Sketch Plane**. Select a new plane from the fly-out FeatureManager. The name of the plane is displayed in the Sketch Plane/Face box. Click **OK** from the Sketch Plane PropertyManager.
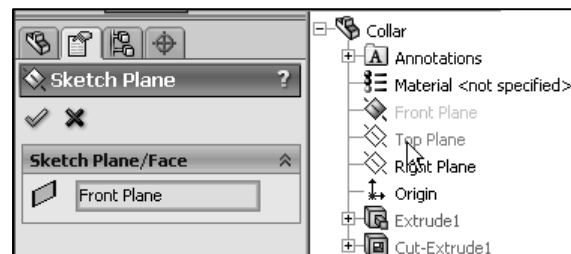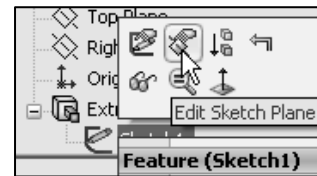
Question 1: Identify the Sketch plane for the Extruded Base feature.

- A: Top Plane

- B: Front Plane

- C: Right Plane

- D: Left Plane

The correct answer is A.

Origin

Question 2: Identify the Sketch plane for the Extruded Base feature.

- A: Top Plane

- B: Front Plane

- C: Right Plane

- D: Left Plane
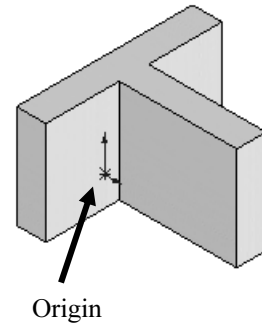
The correct answer is B.

Origin

Question 3: Identify the Sketch plane for the Extruded

Base feature.

- A: Top Plane

- B: Front Plane

- C: Right Plane

- D: Left Plane

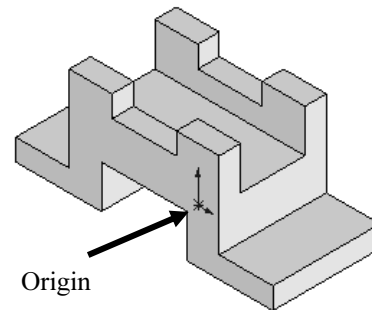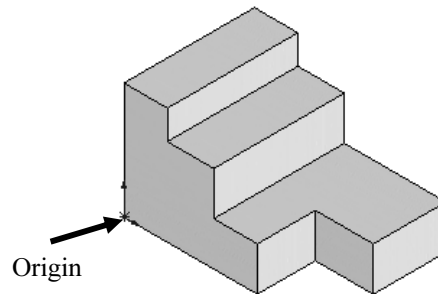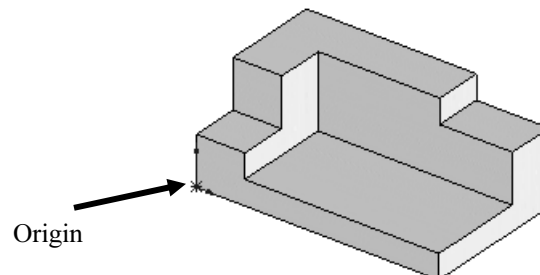The correct answer is B.

Origin

Question 4: The _____ is the Sketch plane for the Extruded Base feature.

The correct answer is Front.

🔅 There are numerous ways to create the models in this and other chapters.

Origin

# Determine the design intent for a model

What is design intent? Design intent is the intellectual arrangements of features and dimensions of a design. Design intent governs the relationship between sketches in a feature, features in a part, and parts in an assembly.

The SolidWorks definition of design intent is the process in which the model is developed to accept future modifications. Models behave differently when design changes occur.

Design for change! Utilize geometry for symmetry, reuse common features, and reuse common parts. Build change into the following areas that you create: sketch, feature, part, assembly, and drawing.

### Design intent in a sketch

Build design intent in a sketch as the profile is created. A profile is determined from the Sketch Entities. Example: Rectangle, Circle, Arc, Point, etc.

Apply symmetry in a profile through a sketch centerline, mirror entity, and position about the reference planes and Origin.
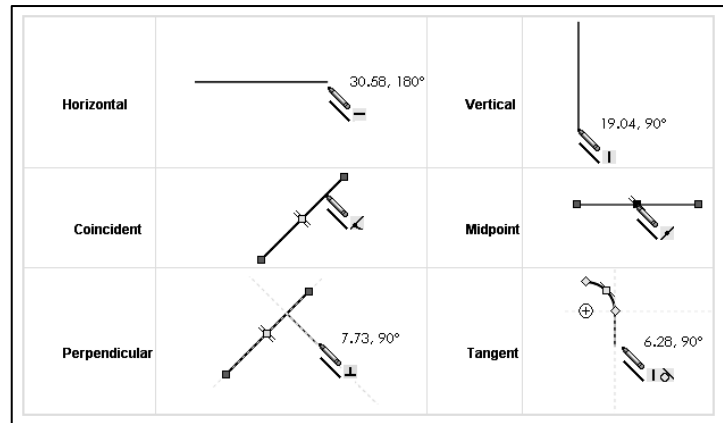
Build design intent as you sketch with automatic geometric relations. Document the decisions made during the up front design process. This is very valuable when you modify the design later.

A rectangle contains Horizontal, Vertical, and Perpendicular automatic geometric relations. Apply design intent using added geometric relations. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel, etc.
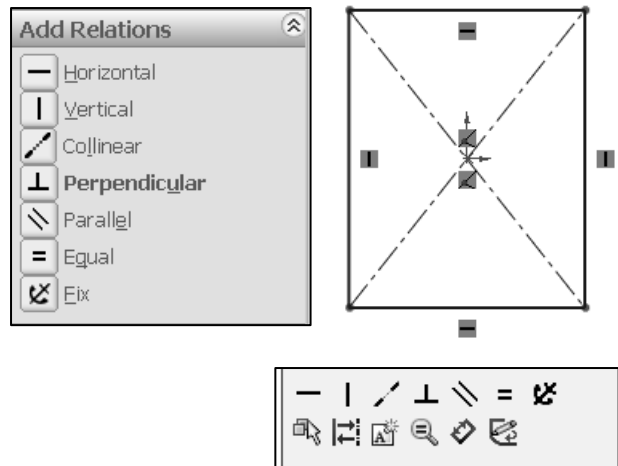
The Properties PropertyManager and the Context toolbar are displayed when you select multiple sketch entities in the Graphics window.

Example A: Apply design intent to create a square profile. Sketch a rectangle (Corner Rectangle tool) with the Origin approximately in the center. Insert a construction reference centerline. Add a Midpoint relation. Add an Equal relation between the two vertical and horizontal lines. Insert a dimension to define the square.
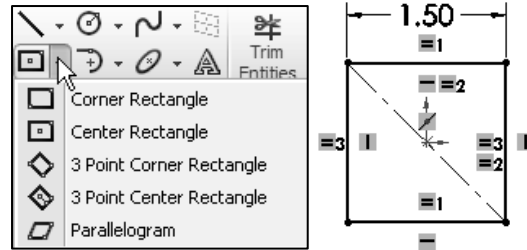
The Center Rectangle tool eliminates the need to create a centerline, midpoint and equal relation.

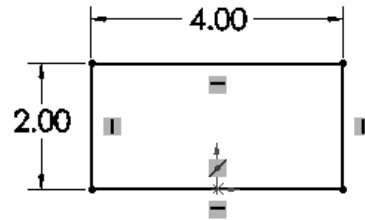Example B: Develop a rectangular profile. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Sketch a rectangle. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the rectangle.

## Design intent in a feature

Build design intent into a feature by addressing symmetry, feature selection, and the order of feature creation.

Example A: The Base Extrude feature remains symmetric about the Front Plane. Utilize the Mid Plane End Condition option in Direction 1. Modify the depth, and the feature remains symmetric about the Front Plane.
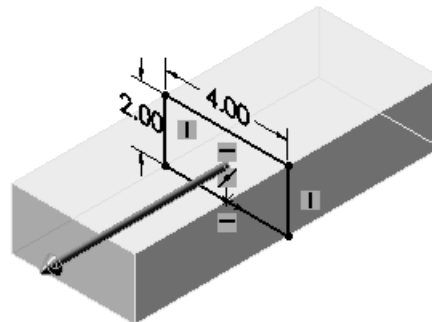
Example B: Create 34 teeth for a Circular Pattern feature. Do you create each tooth separate using the Extruded Cut feature? No. Create a single tooth and then apply the Circular Pattern feature. Modify the number of teeth from 32 to 24.
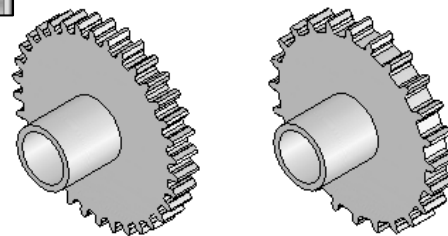
## Design intent in a part

Utilize symmetry, feature order and reusing common features to build design intent into a part.

Example A: Feature order. Is the entire part symmetric? Feature order affects the part. Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.
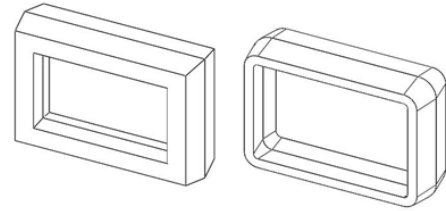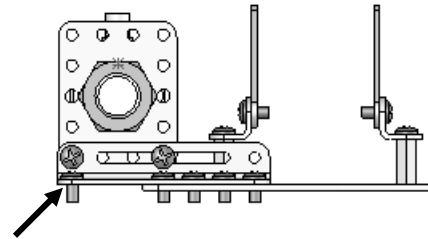
## Design intent in an assembly

Utilizing symmetry, reusing common parts and using the Mate relation between parts builds the design intent into an assembly.

Example A: Reuse geometry in an assembly. The assembly contains a linear pattern of holes. Insert one screw into the first hole. Utilize the Component Pattern feature to copy the machine screw to the other holes.

## Design intent in a drawing

Utilize dimensions, tolerance and notes in parts and assemblies to build the design intent into a drawing.

Example A: Tolerance and material in the drawing. Insert an outside diameter tolerance +.000/-.002 into the Pipe part. The tolerance propagates to the drawing.

Define the Custom Property Material in the Part. The Material Custom Property propagates to your drawing.

Create a 2D sketch on any of the default planes: Front, Top, and Right or a created plane.
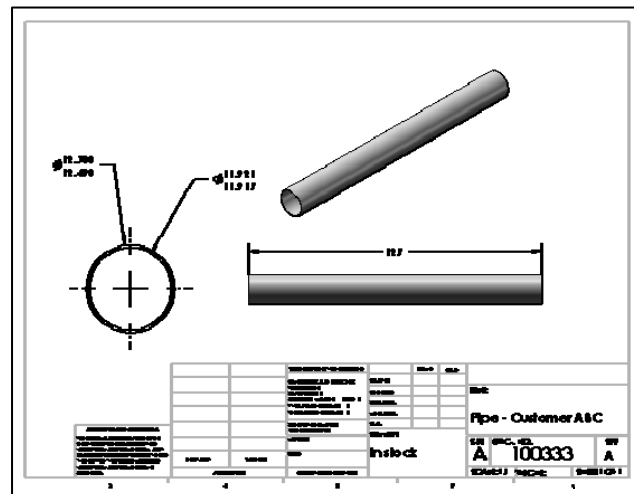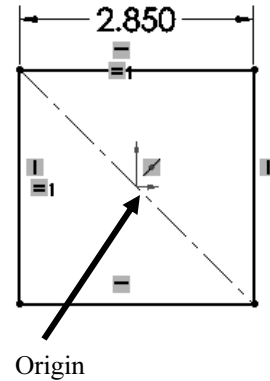
### *Tutorial: Design intent 2-1*



🔆 Design intent is an important consideration when creating a SolidWorks model. Plan your sketches and features.

1. **Open** the Battery part from the SolidWorks CSWA Folder\Chapter2 location. Drag the rollback bar up under the Base feature in the FeatureManager.

2. **Edit** Sketch1 from the FeatureManager. Top Plane is the Sketch plane. View the geometric relations and dimensions. Drag the rollback bar down under Terminals-Top in the FeatureManager.

The first feature of the Battery is Extrude1. Extrude1 is the Base feature and requires a Base sketch. The Base sketch plane determines the orientation of the Base feature. Note: Extrude1 was renamed to Base.



The Sketch plane locates the sketch profile on any plane or face. The sketch for the feature was created using symmetry. The sketch is a rectangle profile on the Top Plane, centered at the Origin using the Sketch Centerline tool with a Midpoint and Equal relation. The Equal relation was applied to the two perpendicular line segments.
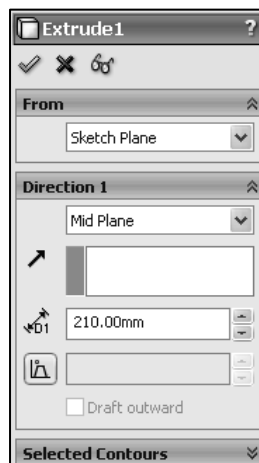
3. **Close** the model.

🔆 Click **View**, **Sketch Relations** from the Menu bar menu to view the relations in a sketch.

### *Tutorial: Design Intent 2-2*

1. **Open** the Box part from the SolidWorks CSWA Folder\Chapter2 location.

2. **Edit** Extrude1 from the FeatureManager. Extrude1 is the Base feature. Mid Plane was selected as the End Condition in Direction 1. Both sides of the Box are equal. The design intent was symmetry, and the Mid Plane option was selected.
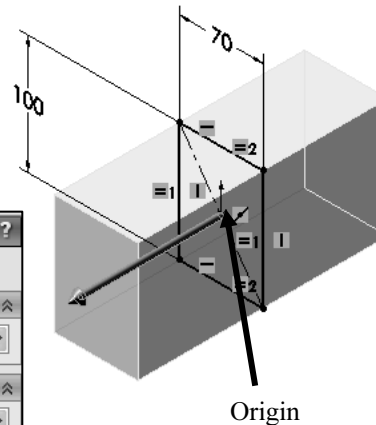


3. **Close** the model.

# Identify material, measure, and mass properties

Understand the process and procedure of the following:

- Assign and edit material to a part

- Apply the Measure tool to a part or assembly

- Locate the Center of mass, and Principal moments of inertia relative to the default coordinate location, Origin.

- Calculate the overall mass and volume of a part

## Assign and edit material

Use the Material dialog box to assign or edit material in a part. Material is required to calculate the correct Mass Properties of a part or an assembly. You will need this knowledge in the exam.

The Material icon is displayed in the FeatureManager regardless of whether a material is applied. Right-click the icon to view a list of the ten most recently used materials.
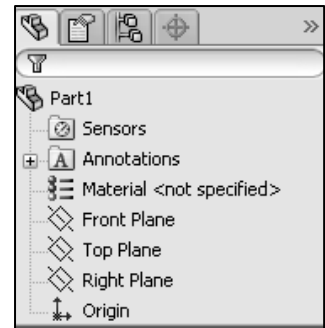
### *Tutorial: Assign and edit material 2-1*

Assign material to the Machine Screw part.

1. **Open** the Machine Screw part from the SolidWorks CSWA Folder\Chapter2 location.

2. **Assign** 6061 Alloy to the part.

3. **View** the available materials.

4. **View** the physical properties of the 6061 Alloy in the Physical Properties box.

5. **Close** the model.
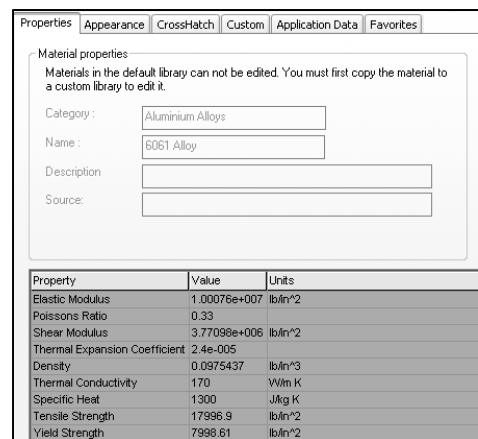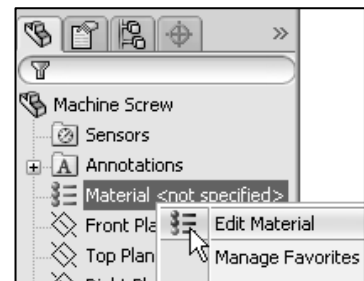
💡 Know the available default materials and their location.

Question 1: Identify the material category for 6061 Alloy.

- A: Steel

- B: Iron

- C: Aluminum Alloys

- D: Other Alloys

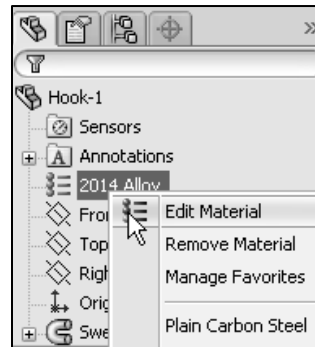- E: None of the provided

The correct answer is C.

### *Tutorial: Assign and edit material 2-2*

Edit the material for the Hook-1 part.

1. **Open** the Hook-1 part from the SolidWorks CSWA Folder\Chapter2 location.

2. **Edit** the material from 2014 Alloy to 6061 Alloy.

3. **Close** the model.

## Measure tool

The Measure tool measures distance, angle, radius, and size of and between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings.

When you measure the distance between two entities, the delta X, Y, and Z distances can be displayed. When you select a vertex or sketch point, the X, Y, and Z coordinates are displayed.

💡 Activate the Measure tool from the Evaluate tab in the CommandManager or click **Tools**, **Measure** from the Menu bar menu.
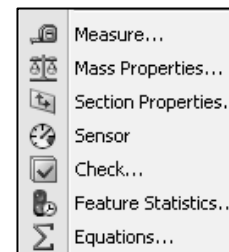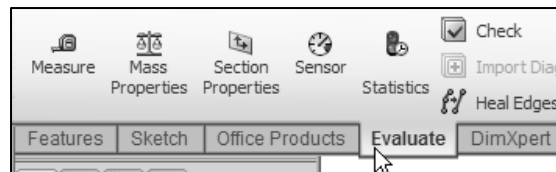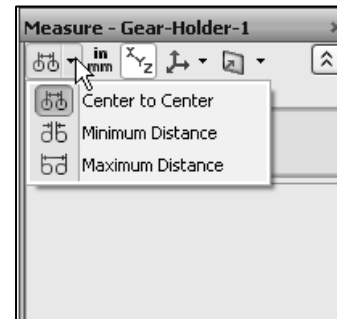
Select the Units/Precision option to specify custom measurement units and precision for your model.

### Tutorial: Measure tool 2-1

1. **Open** the Gear-Holder-1 part from the SolidWorks CSWA Folder\Chapter2 location.

Activate the Measure dialog box.

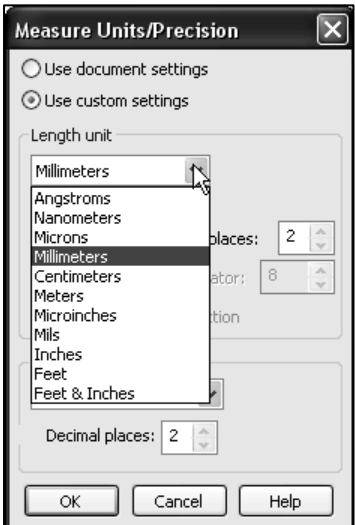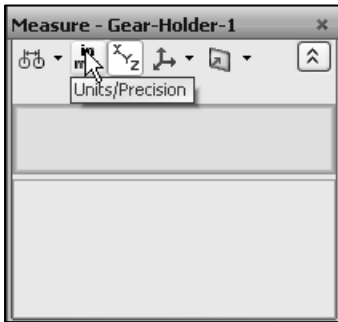2. Click the **Measure** tool from the Evaluate tab. The Measure dialog box is displayed.

3. **Measure** the overall length of the Holder. Select the two edges as illustrated.

4. **Close** the Measure dialog box.

5. **Close** the model.

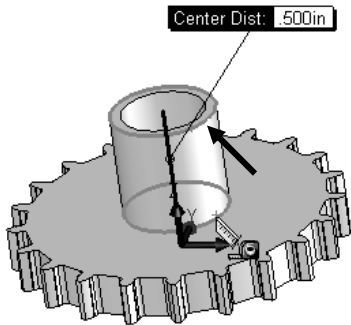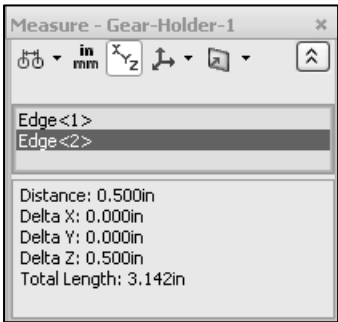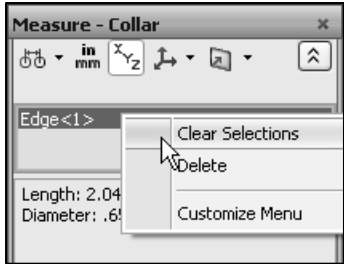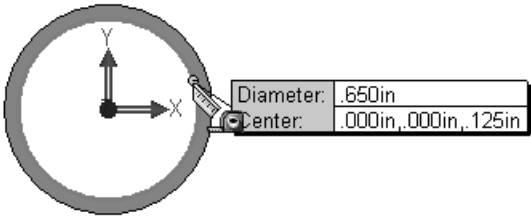### Tutorial: Measure tool 2-2

1. **Open** the Collar part from the SolidWorks CSWA Folder\Chapter2 location.

Activate the Measure dialog box.

2. Click the **Measure** tool from the Evaluate tab. The Measure dialog box is displayed.

3. **Measure** the inside diameter of the Collar. Select the inside circular edge as illustrated. The inside diameter = .650in. Length = 2.042in.
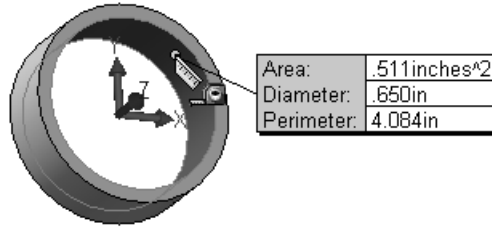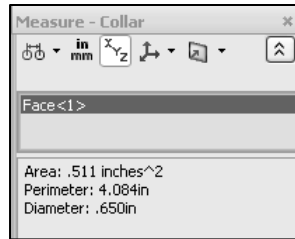
4. **Clear** all selections in the Measure - Collar dialog box.

Measure the inside area, perimeter, and diameter of the Collar.

5. Click the **inside face** as illustrated. View the results.

6. **Clear** all selections.

7. **Close** the Measure dialog box.

8. **Close** the model.

# Locate the Center of mass, and Principal moments of inertia relative to the default coordinate location

The Mass Properties tool displays the mass properties of a part or assembly model, or the section properties of faces or sketches.

The results are displayed in the Mass Properties dialog box. The principal axes and Center of mass are displayed graphically on the model in the Graphics window.

Origin
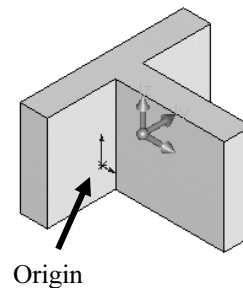
On the CSWA exam, you may see the word centroid. What is a centroid? Let's view a simple example of a triangle. In the plane of any triangle ABC, let:

- D = midpoint of side BC

- E = midpoint of side CA

- F = midpoint of side AB

From the sketch, the lines AD, BE, CF come together to a single point. This point is called the centroid of the triangle ABC. There are many other points that are called triangle centers, but unlike most of them, "centroid" works on arbitrary shapes.
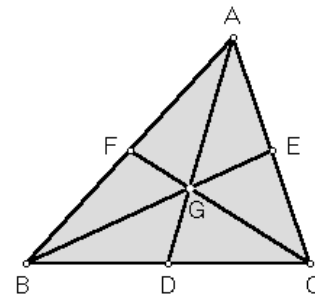
Centroid is sometimes referred to as the Center of mass or Center of gravity. The Center of mass, (center of gravity) of a solid is similar to the Centroid of a Solid. However, calculating the Centroid involves only the geometrical shape of the solid. The center of gravity is equal to the Centroid if the body is homogenous, "constant density".

### Tutorial: Mass properties 2-1

Use the Mass Properties tool to calculate the density, mass, volume, surface area, and Center of mass for a part relative to the default coordinate location.

1. **Open** the Gear-Mass part from the SolidWorks CSWA Folder\Chapter2 location. 6061 Alloy is the applied material.
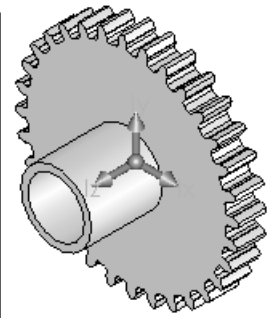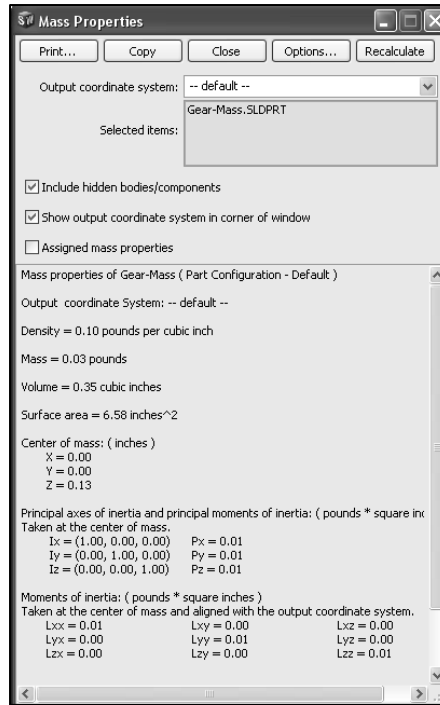
Activate the Mass Properties dialog box.

2. Click the **Mass Properties** tool from the Evaluate tab in the CommandManager. The Mass Properties dialog box is displayed. View the various Mass Properties of the Gear-Mass part.

3. **Close** the Mass Properties dialog box.

4. **Close** the model.

☀ By default, the Centroid is relative to the part or assembly default coordinate location.

View the Center of mass based on the default part Origin location. The density, mass, volume, surface area, Center of mass, Principal axes of inertia and principal moments of inertia, and Moments of inertia are displayed.

☀ To evaluate components or solid bodies in an assembly or multi-body part documents, click the **component** or **body**, and click **Recalculate**. If no component or solid body is selected, the mass properties for the entire assembly or multi-body part are reported.
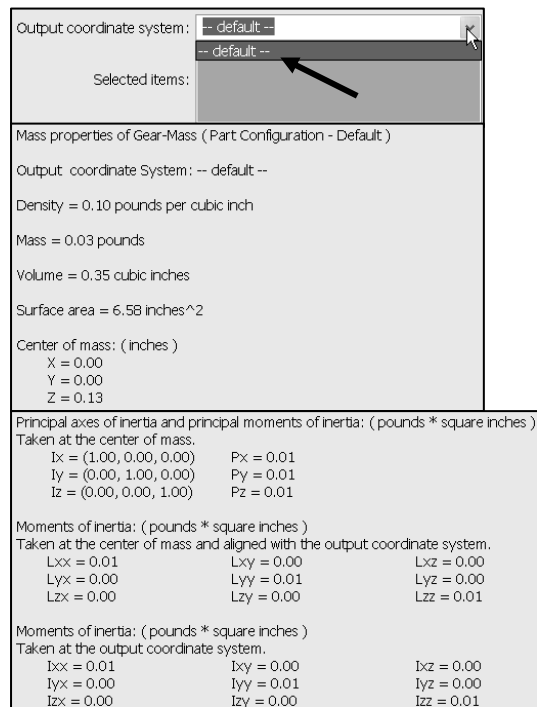
### Tutorial: Mass properties 2-2

Calculate the density, overall mass, volume, surface area, and Center of mass for an assembly relative to the default coordinate location.
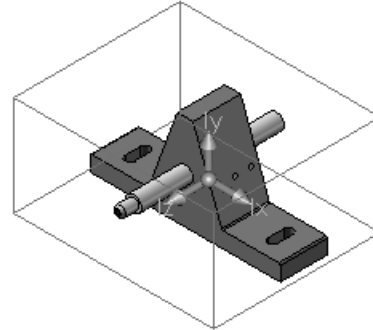
1. **Open** the Wedge-Tube-Mass assembly from the SolidWorks CSWA Folder\Chapter2 location.

Activate the Mass Properties dialog box.

2. Click the **Mass Properties** tool from the Evaluate tab. The Mass Properties dialog box is displayed. The Centroid is displayed relative to default assembly Origin. The density, volume, surface area, moments of inertia, etc. are displayed in the Mass Properties dialog box.

Obtain the Mass Properties of the Tube component in the Wedge-Tube-Mass assembly.

3. **Clear** all selected items.

4. Click **Tube** from the FeatureManager. Tube-1 is displayed in the Selected items box.

5. Click **Recalculate** from the Mass Properties dialog box.

6. **View** the Tube properties. The Centroid location is relative to Tube part Origin.

7. **Close** the Mass Properties dialog box.

8. **Close** the model.

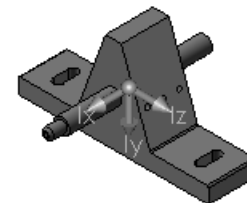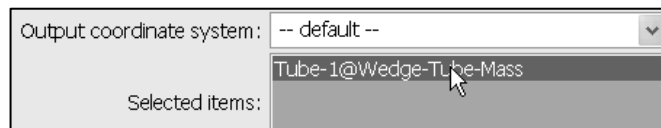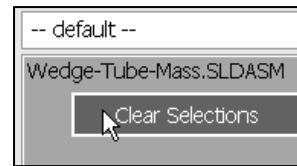☀ You will require this knowledge in the Assembly Modeling category of the CSWA exam.

```
-- default --
Wedge-Tube-Mass.SLDASM
      Clear Selections
```

```
Output coordinate system:   -- default --
                            Tube-1@Wedge-Tube-Mass
           Selected items:
```

```
Mass properties of Tube ( in Assembly Configuration - Default )

Output  coordinate System: -- default --

The center of mass and the moments of inertia are output in the coordinate system of Wedge-Tube-Mass
Mass = 18.41 grams

Volume = 6817.65 cubic millimeters

Surface area = 3896.89 millimeters^2

Center of mass: ( millimeters )
     X = 0.00
     Y = 29.00
     Z = 5.22
```

# File formats for input and export

SolidWorks offers various File formats for input and export. SolidWorks provides three basic document types with their own extensions:

- A part file ends with an extension (.sldprt)

- An assembly file ends with an extension (.sldasm)

- A drawing file ends with an extension of (.slddrw)

When you save a new document, the Save As dialog box is displayed to enter a filename or accept the default filename.

### Tutorial: File formats 2-1

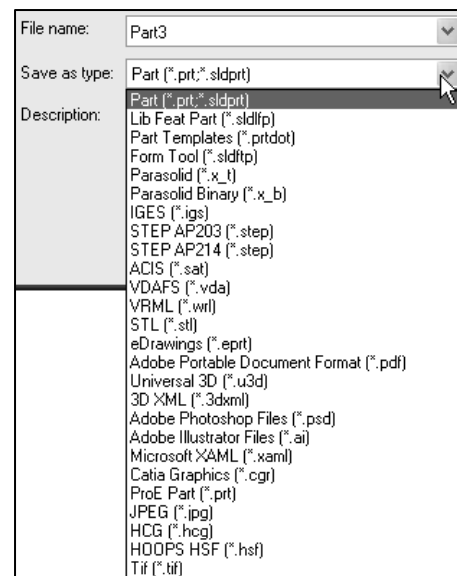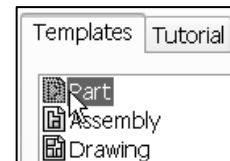View the available file formats for a part.

1. **Create** a New part in SolidWorks.

2. Click **Save As** from the Menu bar toolbar.

3. Click the **drop-down arrow** from the Save as type box.

4. **View** the available file formats.

5. **Close** the model.

Question 1: What is the default Part file format in SolidWorks?

- A: *.sldfop

- B: *.sldprt

- C: *.hjyy

- D: *.pprt

The correct answer is B.

### *Tutorial: File formats 2-2*

View the available file formats for an assembly.

1. **Create** a New assembly in SolidWorks.

2. Click **Save As** from the Menu bar toolbar.

3. Click the **drop-down arrow** from the Save as type box.

4. **View** the available file formats.

5. **Close** the model.
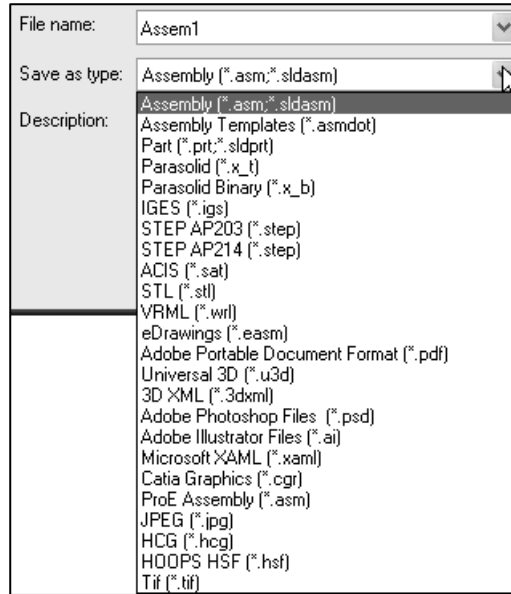
### *Tutorial: File formats 2-3*

View the available file formats for a drawing.

1. **Create** a New drawing in SolidWorks.

2. Click **Save As** from the Menu bar toolbar. Click the **drop-down arrow** from the Save as type box.

3. **View** the available file formats.

4. **Close** the model.

Question 1: Which is not a valid Drawing format in SolidWorks?
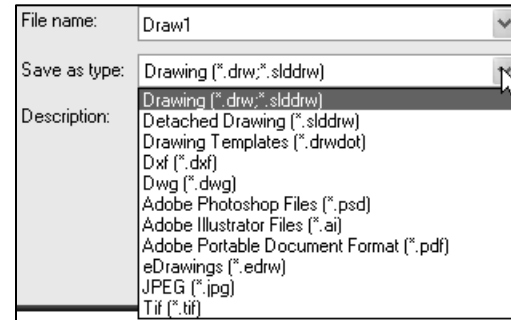
- A: *.tif

- B: *.jpg

- C: * dwgg

- D: * dxf

The correct answer is C.

# Process of creating a simple drawing from a part or an assembly

You will need to recognize the process and be acquainted with the procedure for the following:

- Insert the 3 standard views: Front, Top, and Right

- Insert an Isometric named view

- Add a drawing sheet

- Insert drawing annotations

- Modify a drawing view with the scale and view mode option

🔅 You are not required to create a drawing from a part or an assembly in the CSWA exam at this time. You do need the ability to understand the procedure and process to create a named drawing view.

🔅 Think of a drawing view as a container. Generally, the contents are views of a model. When you sketch in a drawing, or insert annotations or blocks, the entities belong to the active drawing view or drawing sheet.

### Tutorial: Drawing process 2-1

Understand the process to create a drawing from an existing part.

1. **Create** a New drawing. The Sheet Format/Size dialog box is displayed.

2. Select **A-Landscape**.

3. Select **Standard sheet size**.

4. Click **OK** from the SheetFormat/Size dialog box. The Model View PropertyManager is displayed, if the Start command when creating new drawing option is check.

5. Click **Browse** from the Part/Assembly to Insert box.

6. Double click the **Gear** part from the SolidWorks CSWA Folder\Chapter2 location.
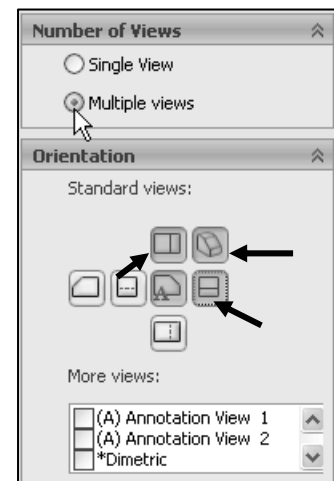
Insert views. Note: Third angle projection is used in this book.

7. Select **Multiple views** from the Number of Views box.

8. Select **\*Top**, **\*Right**, and **\*Isometric** view. Note: \*Front view is selected by default.

9. Click **OK** from the Model View PropertyManager. The views are displayed on Sheet1.

10. Click **Yes**.
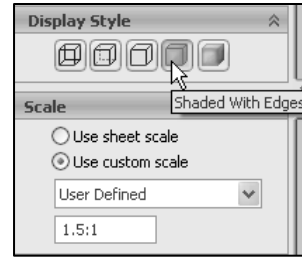
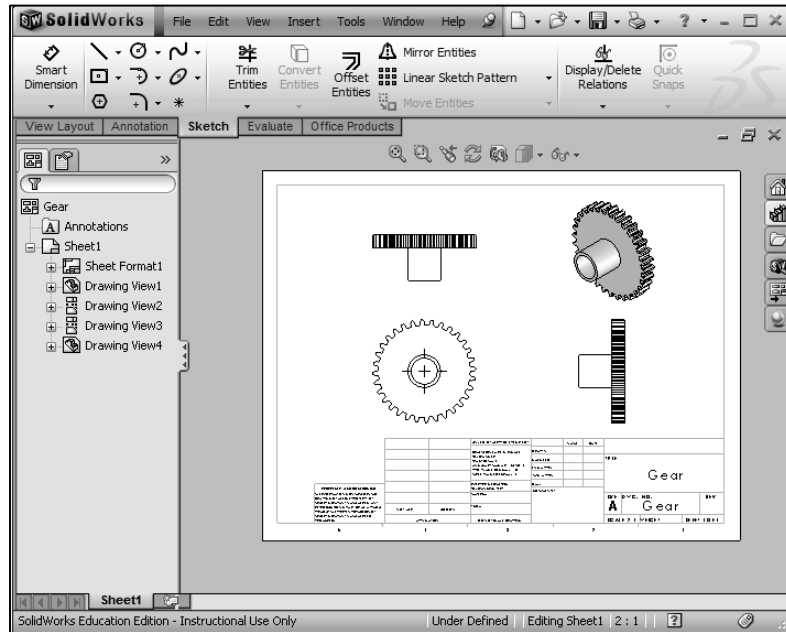11. **Click** inside the Isometric view boundary.

12. **Apply** Shaded With Edges to the Isometric view.

13. **Modify** the scale of the four views to 1.5:1.

14. **View** the four views in the Gear drawing.

15. **Insert** a Sheet in the drawing. Sheet2 is displayed.

🔅 The Annotations toolbar provides tools for adding notes and symbols to a drawing, part, or assembly document. Only those annotations that are appropriate for the active document are available; the other tools are displayed in gray.

16. **Return** to Sheet1. Click the Sheet1 tab.

17. **Click** the Note tool from the Annotation toolbar.
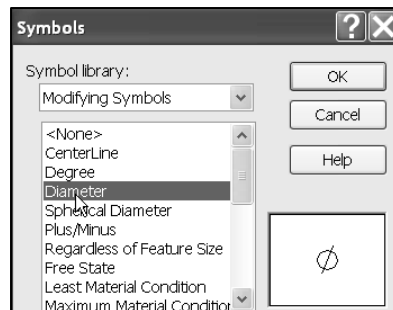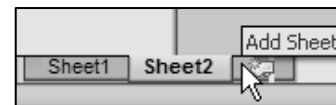
18. **Insert** the following note as illustrated above the Front View.

19. **Save** the drawing and name it Gear.

20. **Close** the drawing.
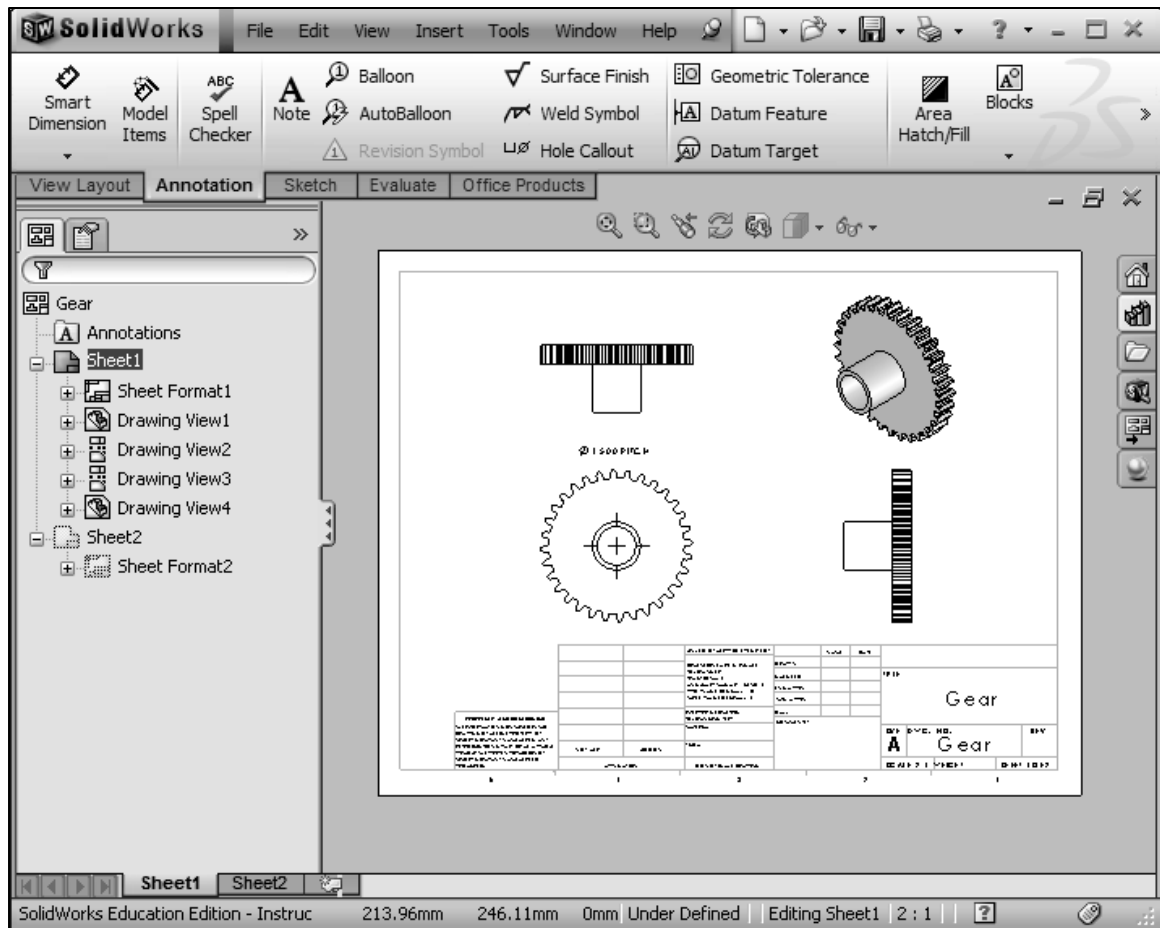


💡 You can add all types of annotations to a drawing document. You can add most types in a part or assembly document, then insert 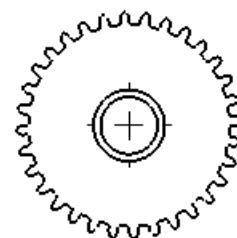them into a drawing document. However, there are some types; such as Center Marks and Area Hatch that you can add only in a drawing document.

💡 In the Part Modeling, Advanced Part Modeling and Assembly Modeling categories, you are required to read and interpret all types of drawing views.

### *Tutorial: Drawing process 2-2*

Understand the process to create an Isometric drawing view from an Assembly.

1. **Create** a New drawing in SolidWorks. The Sheet Format/Size dialog box is displayed.

2. Select **A-Landscape**.

3. Select **Standard sheet size**.

4. Click **OK**. The Model View PropertyManager is displayed.

5. **Click** Browse.

6. Double click the **Wedge-Tube** assembly from the SolidWorks CSWA Folder\Chapter2 location.

7. **Insert** an Isometric view in a Shaded mode with a 0.9:1 scale. Note: Third angle projection is used in this book.

8. Select the **Edit Sheet Format** mode.

9. Add a Title: **Customer Wedge-Tube** using **12** Font.

10. **Return** to the Edit Sheet mode.

11. **Save** the drawing.

12. **Close** the drawing.
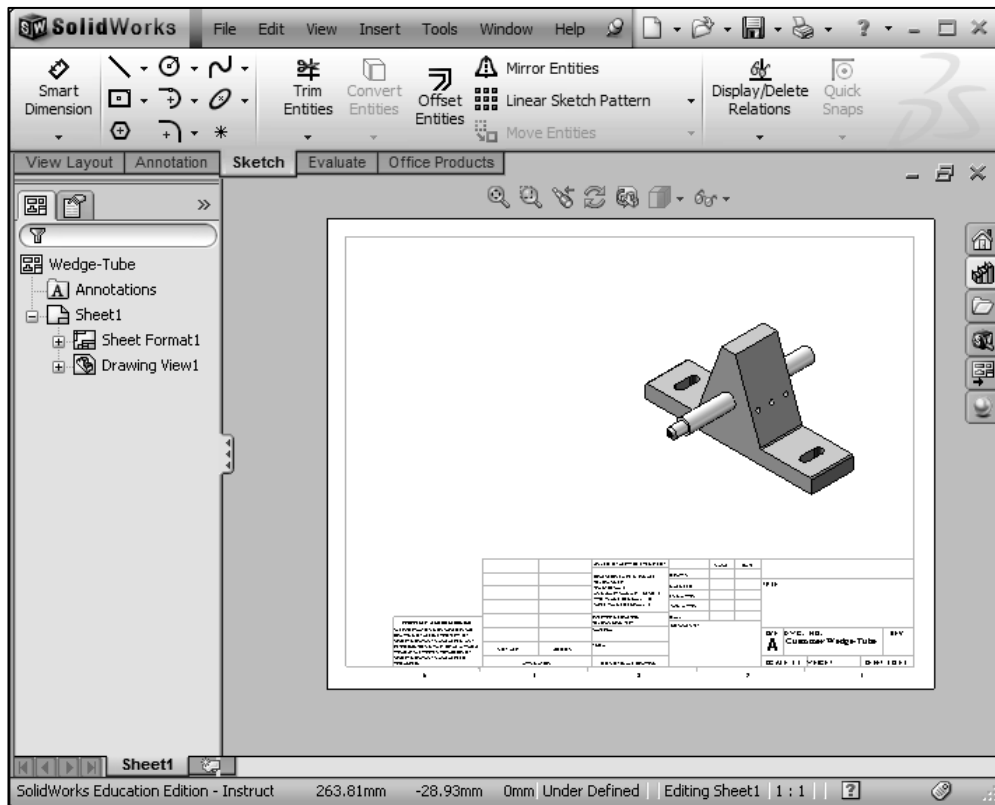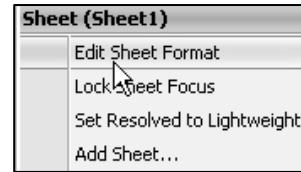
# Drawing section - icons

You need the ability to recognize the drawing name view type by it's icon and the procedure to create the drawing view: *Standard 3 View, Model View, Projected View, Auxiliary View, Section View, Aligned Section View, Detail View, Broken-out Section, Break, Crop View, and Alternate Position View*.
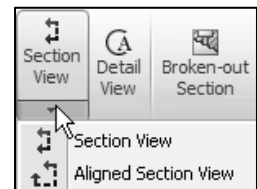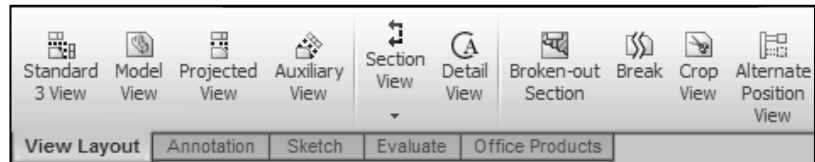
### Tutorial: Drawing name view type 2-1

Identify the drawing name view type in the Drawing FeatureManager. Understand the drawing view procedure.

1. **Open** the Plate drawing from the SolidWorks CSWA Folder\Chapter2 location.

2. **View** Sheet1. Identify the drawing name view type that was used to create the drawing view from the FeatureManager icon. The views have been renamed.
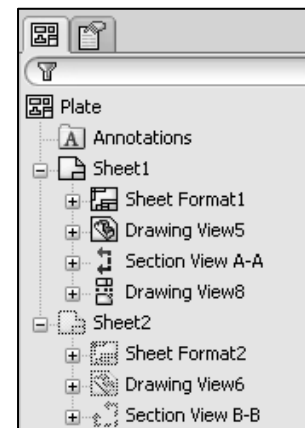
   a. Drawing View5: Model View

   b. Section View A-A: Section View

   c. Drawing View8: Projected View

   Create a Section View in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

3. **View** Sheet2. Identify the drawing name view type that was used to create the drawing from the icon.

   a. Drawing View6: Model View

   b. Section View B-B: Aligned Section View

4. **Close** the drawing.

   Create an Aligned section view in a drawing through a model, or portion of a model, that is aligned with a selected section line segment. The Aligned Section view is similar to a Section View, but the section line for an aligned section comprises two or more lines connected at an angle.
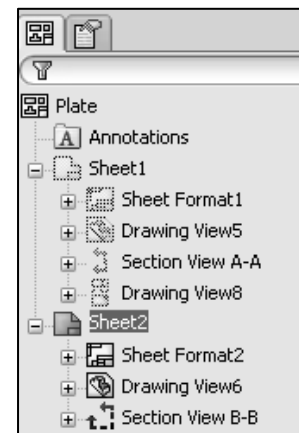
💡 Create a detail view in a drawing to show a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, a non-planar (isometric) view, a section view, a crop view, an exploded assembly view, or another detail view.

### *Tutorial: Drawing name view type 2-2*

Identify the drawing name view type in the Drawing FeatureManager. Understand the drawing view procedure.

1. **View** the illustrated Drawing FeatureManager. Identify the drawing name view type that was used to create the drawing from the icon.

   a. Drawing View1: Model View

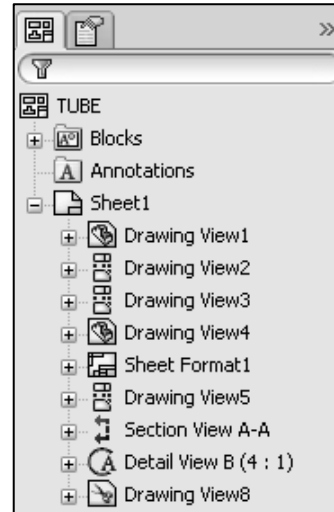   b. Drawing View2: Projected View

   c. Drawing View3: Projected View

   d. Drawing View4 : Model View

   e. Drawing View5 : Projected View

   f. Drawing View6 : Section View

   g. Drawing View7: Detail View

   h. Drawing View8: Crop View

💡 Crop any drawing view except a Detail View, a view from which a Detail View has been created, or an exploded view. To create a Crop view, sketch a closed profile such as a circle or spline. The view outside the closed profile disappears as illustrated.

💡 Create a Detail view in a drawing to display a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, a non-planar (isometric) view, a section view, a crop view, an exploded assembly view, or another detail view.
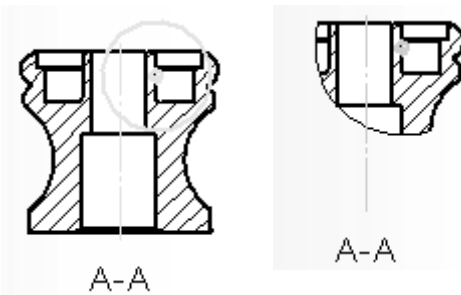
In the Drawing Theory section, an exam question could read:

Question 1: Identify the following Drawing View icon .

- A: Projected View
- B: Standard View
- C: Section View
- D: None of the provided

The correct answer is A.

Question 2: Identify the following Drawing View icon .

- A: Projected View
- B: Trim View
- C: Cut View
- D: Crop View

The correct answer is D.

Question 3: Identify the following Drawing View icon .

- A: Section View
- B: Broken View
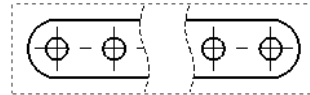- C: Break View
- D: Two Section View

The correct answer is C.

# Procedure to create a Named Drawing view

You need the ability to identify the procedure to create a named drawing view: *Standard 3 View, Model View, Projected View, Auxiliary View, Section View, Aligned Section View, Detail View, Broken-out Section, Break, Crop View, and Alternate Position View*.

### *Tutorial: Drawing named procedure 2-1*

Identify the drawing name view and understand the procedure to create the name view.

1.  **View** the illustrated drawing views. The top drawing view is a Break View. The Break View is created by adding a break line to a selected view.
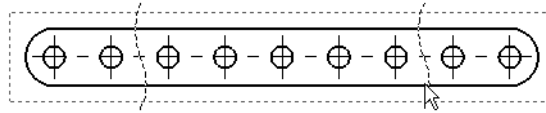
💡 Broken views make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.
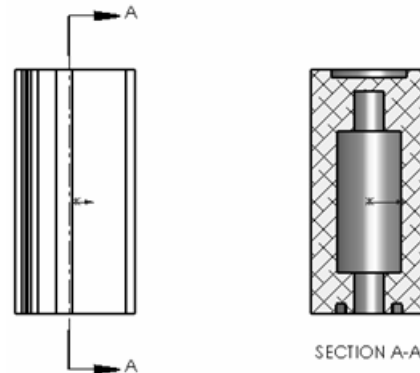
💡 In views with multiple breaks, the Break line style must be the same.

### *Tutorial: Drawing named procedure 2-2*

Identify the drawing name view and understand the procedure to create the name view.

1.  **View** the illustrated drawing views. The right drawing view is a Section View. The Section View is created by cutting the parent view with a section line.

💡 Create a Section View in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

### *Tutorial: Drawing named procedure 2-3*

Identify the drawing name view and understand the procedure to create the name view.

1.  **View** the illustrated drawing views. The Top drawing view is an Auxilary View of the Front View. Select a reference edge to create an Auxiliary View.
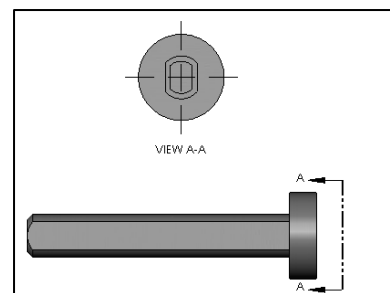
💡 An Auxiliary View is similar to a Projected View, but it is unfolded normal to a reference edge in an existing view.
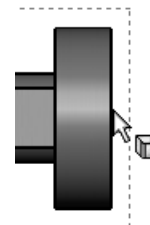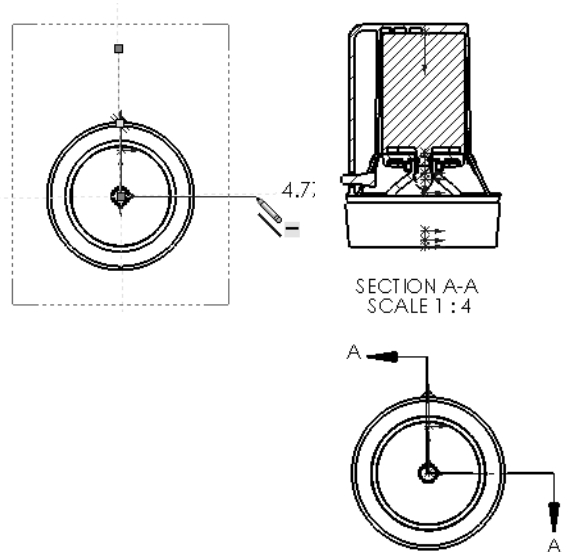
## Tutorial: Drawing named procedure 2-4

Identify the drawing name view and
understand the procedure to create the
name view.

1. **View** the illustrated drawing views.
   The right drawing view is an Aligned
   Section View of the bottom view. The
   Section View is created by using two
   lines connected at an angle. Create an
   Aligned Section View in a drawing
   through a model, or portion of a
   model, that is aligned with a selected
   section line segment.

   The Aligned Section View is very
   similar to a Section View, with the
   exception that the section line for an
   aligned section comprises of two or more
   lines connected at an angle.

## Tutorial: Drawing named procedure 2-5

Identify the drawing name view and
understand the procedure to create the
name view.

1. **View** the illustrated drawing views.
   The left drawing view is a Detail
   View. The Detail View is created by
   sketching a circle with the Circle
   Sketch tool. Click and drag for the
   location.
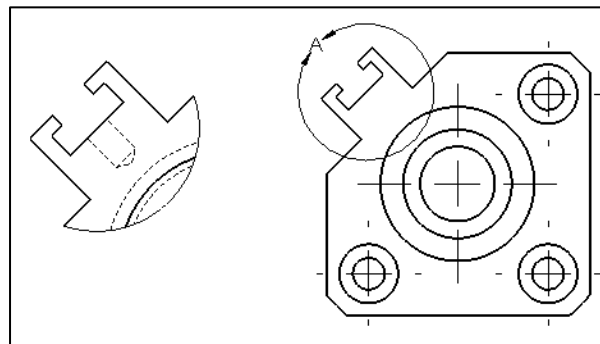
   The Detail View tool provides
   the ability to add a detail view to display
   a portion of a view, usually at an
   enlarged scale.

   To create a profile other than a
   circle, sketch the profile before clicking
   the Detail View tool. Using a sketch entity tool, create a closed
   profile around the area to be detailed. You can add dimensions or
   relations to the sketch entities to position the profile precisely
   relative to the model.
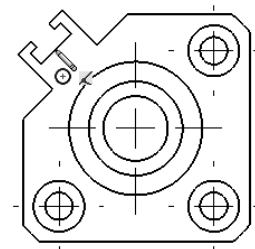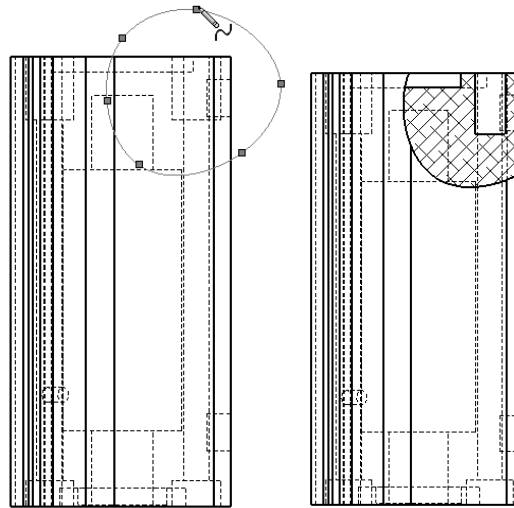
### *Tutorial: Drawing named procedure 2-6*

Identify the drawing name view and understand the procedure to create the name view.

1. **View** the illustrated drawing views. The right drawing view is a Broken-out Section View. The Broken-out Section View is part of an existing drawing view, not a separate view. Create the Broken-out Section View with a closed profile, usually by using the Spline Sketch tool. Material is removed to a specified depth to expose inner details.
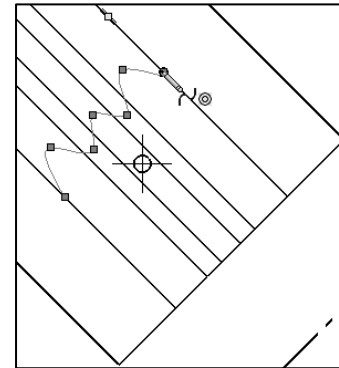
### *Tutorial: Drawing named procedure 2-7*

Identify the drawing name view and understand the procedure to create the name view.

1. **View** the illustrated drawing view. The top drawing view is a Crop View. The Crop View is created by a closed sketch profile such as a circle, or spline as illustrated.

The Crop View provides the ability to crop an existing drawing view. You can not use the Crop tool on a Detail View, a view from which a Detail View has been created, or an exploded view.

☀ Use the Crop tool to save steps. Example: instead of creating a Section View and then a Detail View, then hiding the unnecessary Section View, use the Crop tool to crop the Section View directly.
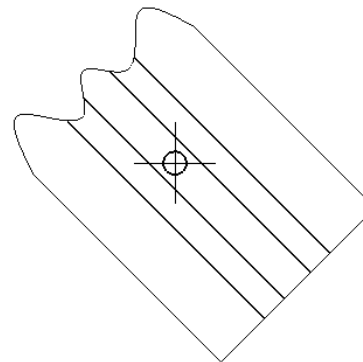
### Tutorial: Drawing named procedure 2-8

Identify the drawing name view and understand the procedure to create the name view.

1. **View** the illustrated drawing view. The drawing view is an Alternate Position View.

   The Alternate Position View tool ⊞ provides the ability to superimpose an existing drawing view precisely on another. The alternate position is displayed with phantom lines.

☼ Use the Alternate Position View is display the range of motion of an assembly. You can dimension between the primary view and the Alternate Position View. You can not use the Alternate Position View tool with Broken, Section, or Detail views.
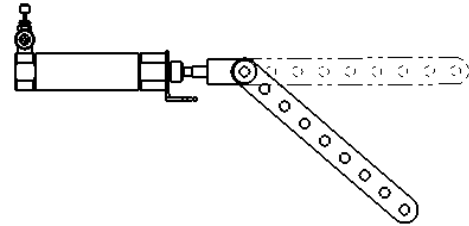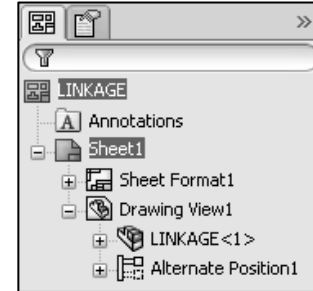
# Engineering Documentation Practices

A 2D drawing view is displayed in the Advanced Part and Assembly modeling section of the CSWA exam to clarify dimensions and details.

The ability to interpret the 2D drawing views is required.

- Example: The provided 2D detail drawing view illustrates that eight holes are required. The hole diameters are .19. The equal spacing between the holes is .55. Note: Units are provided in the CSWA exam problem.
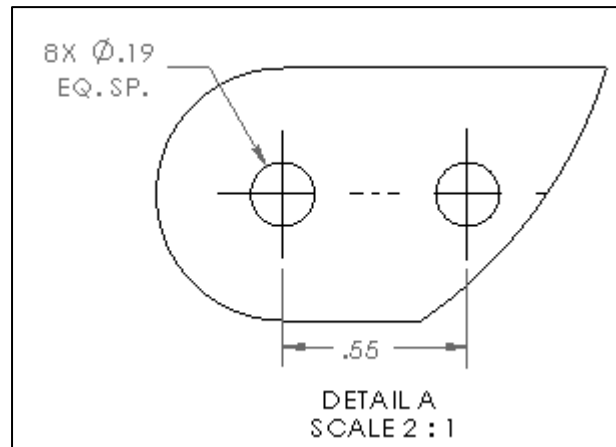
# Document Properties

You need the ability to identify the procedure to select system units and precision of a SolidWorks model using the Document Properties section. Access the Document Properties tab from the Options tool located in the Menu bar toolbar.
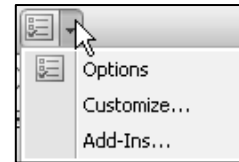
You are required to set the precision for the selected unit system and to set the Tolerance / Precision using the Dimension PropertyManager or the Document Properties dialog box.

Document properties apply to the current document. The Document Properties tab is only available when a document is open.
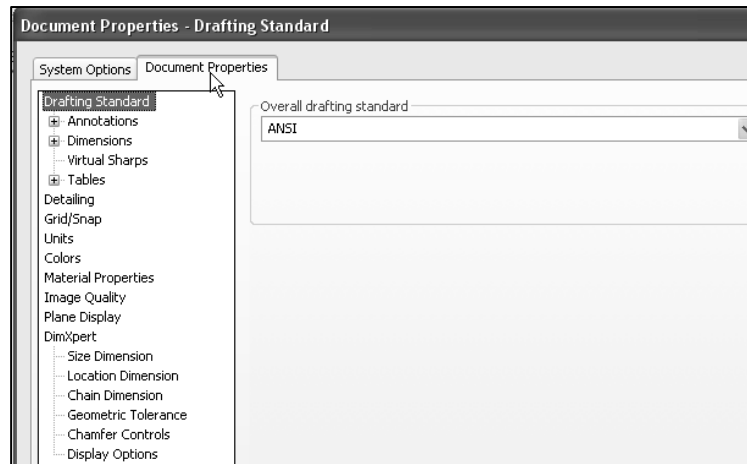
New documents get their document settings (such as Units, Image Quality, etc.) from the document properties of the template used to create the document.
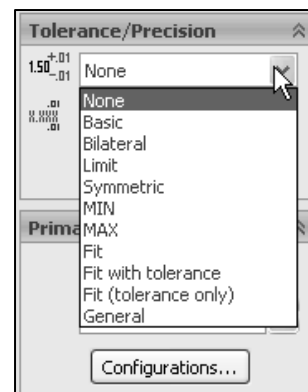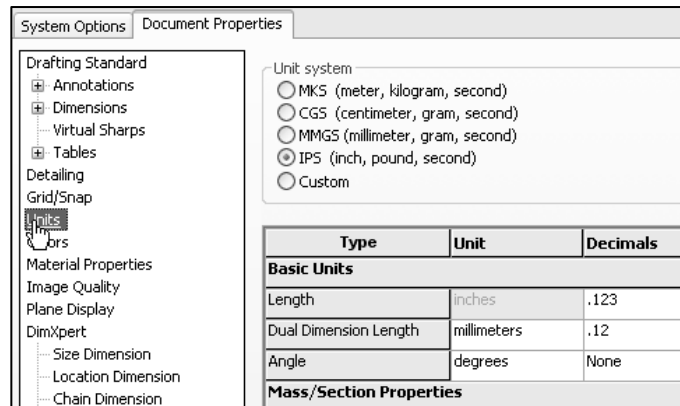
Know how to set the Unit system, and Tolerance/Precision option in a document for the CSWA exam.

### Tutorial: Document properties 2-1

Set dimensioning (drafting) standard, system units, and precision.

1. **Create** a New part in SolidWorks.

2. Click **Options**, **Document Properties** tab from the Menu bar toolbar.

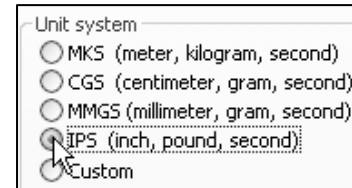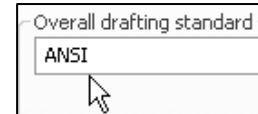3. Select **ANSI** for Dimensioning (drafting) standard.

4. Click **Units**.

5. Select **IPS** for Unit system.

6. Select .**123** Decimal places for Length units.

7. Select **None** Decimal places for Angular units.

8. **Close** the part.
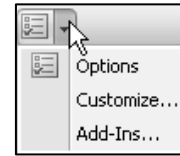
### Tutorial: Document properties 2-2

Set dimensioning standard, system units, and precision.

1. **Create** a New part in SolidWorks.

2. Click **Options**, **Document Properties** tab from the Menu bar menu.

3. Select **ANSI** for Dimensioning (drafting) standard.

4. Select **Units**.

5. Click **Custom** for Unit system.

6. Select **centimeters** from the length units drop down box.

7. Select .1**2** Decimal places for Length units.

8. Select **None** Decimal places for Angular units.

9. **Close** the part.

# Summary

Drawing Theory is one of the five categories on the CSWA exam. In this chapter, you covered general concepts, symbols, and terminology and then the core element (Drawing Theory) that is aligned to the CSWA exam.

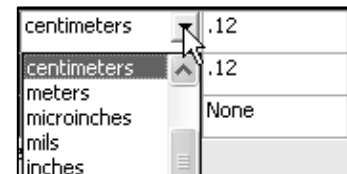There are two questions on the CSWA exam in this category. Each question is worth five (5) points. The two questions are in a multiple choice single answer format.

Spend no more than 10 minutes on each question in this category. This is a timed exam. Manage your time.

Part Modeling is the next chapter in this book. Chapter 3 covers the knowledge to identify the part Origin, design intent, and key features to create a simple part from a detailed dimensioned illustration.

The complexity of the models along with the features progressively increases throughout chapter 2 to simulate the final types of parts that could be provided on the CSWA exam.

There is one question on the CSWA exam in the Part Modeling category. The question is worth thirty (30) points. You are required to create 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 relative to part Origin.

☀ No PhotoWorks, Motion Study, Display Pane, or assembly configuration questions on the CSWA exam as this time.

☀ View sample screen shots from an older CSWA exam for a Drawing Theory question at the end of the Homework section in this chapter.

# Terms

- *Aligned section view.* A drawing view that is aligned with a selected section line segment. The Aligned Section view is similar to a Section View, but the section line for an aligned section comprises two or more lines connected at an angle.

- *Alternate position view.* A drawing view in which one or more views are superimposed in phantom lines on the original view. Alternate position views are often used to show range of motion of an assembly.

- *Base feature.* The first feature of a part is called the Base feature.

- *Base sketch.* The first sketch of a part is called the Base sketch. The Base sketch is the foundation for the 3D model. Create a 2D sketch on a default plane: Front, Top, and Right in the FeatureManager design tree, or on a created plane.

- *Crop view.* You can crop any drawing view except a Detail View, a view from which a Detail View has been created, or an exploded view. Create a Crop view, sketch an closed profile such as a circle or spline. The view outside the closed profile disappears.

- *ConfigurationManager.* Located on the left side of the SolidWorks Graphics window is a used to create, select, and view multiple configurations of parts and assemblies in a document.

- *Constraints*. Geometric relations such as Perpendicular, Horizontal, Parallel, Vertical, Coincident, Concentric, etc. Insert constraints to your model to incorporate design intent.

- *Coordinate system*. A system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.

- *Design intent*. Design intent is the intellectual arrangements of features and dimensions of a design. Design intent governs the relationship between sketches in a feature, features in a part, and parts in an assembly.

- *Design Library*. Located in the Task Pane, the Design Library provides a central location for reusable elements such as parts, assemblies, etc.

- *Detail view*. A portion of a larger view, usually at a larger scale than the original view.

- *Drawing*. A 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.

- *Document*. A file containing a part, assembly, or drawing.

- *Features*. Individual shapes created by Sketch Entities tools: lines, circles, rectangles, etc. that when combined, creates the part. Features are geometry building blocks, they add or remove material, and are created from 2D or 3D sketched profiles or from edges and faces of existing geometry.

- *FeatureXpert*. Powered by SolidWorks Intelligent Feature Technology (SWIFT™), helps manages fillet and draft features; only for constant radius fillets and neutral plane drafts.

- *FeatureManager design tree*. Located on the left side of the SolidWorks Graphics window and provides an outline view of the active part, assembly, or drawing. The FeatureManager design tree and the Graphics window are dynamically linked. You can select features, sketches, drawing views, and construction geometry in either pane.

- *Fly-out FeatureManager design tree*. Allows you to view and select items in the PropertyManager and the FeatureManager design tree at the same time.

- *Mass Properties tool*. Displays the mass properties of a part or assembly model, or the section properties of faces or sketches.

- *Measure tool*. Measures distance, angle, radius, and size of and between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings.

- *Model view*. A drawing view of a part or assembly

- *Parameter*. A value used to define a sketch or feature, often a dimension or instance.

- *Predefined view*. A drawing view in which the view position, orientation, and so on can be specified before a model is inserted. You can save drawing documents with predefined views as templates.

- *Rollback*. Suppresses all items below the rollback bar.

- *Section view*. A drawing view in which you cut the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

- *Sketch Status*. Sketch states include the following: Over Defined, Under Defined, Fully Defined, Driven, Dangling, Invalid, and Not Solved.

- *Standard 3 views*. The three orthographic views; front, right, and top that are often the basis of a drawing.

# Questions

1: Identify the following Feature icon  .

- A: Extruded Cut feature

- B: Mirror feature

- C: Fillet feature

- D: Chamfer feature

2: Identify the following Feature icon  .

- A: Extruded Boss/Base feature

- B: Round Pattern feature

- C: Circular Pattern feature

- D: Chamfer feature

3. Identify the following Feature icon  .

- A: Hole Wizard feature

- B: Simple Hole feature

- C: Circular Pattern feature

- D: Top feature

4. Identify the following Feature icon .

- A: Shell feature

- B: Loft feature
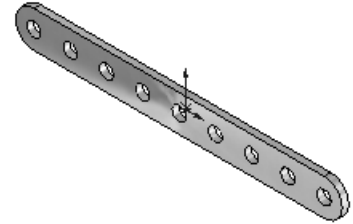
- C: Round feature

- D: Rib feature

5. Identify the following Feature icon .

- A: Shell feature

- B: Loft feature
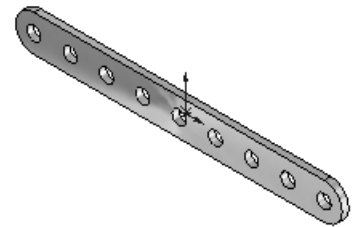
- C: Round feature

- D: Revolved Cut feature

6. Identify the number of instances in the illustrated model.

- A: 7

- B: 5

- C: 9

- D: None

7. Identify the Sketch plane for the Extrude1 feature.

- A: Top Plane

- B: Front Plane

- C: Right Plane

- D: Left Plane

8. Identify the following Sketch Entities icon ⤵.

- A: Center Arc tool
- B: Tangent Arc tool
- C: 3 Point Arc tool
- D: Point Arc tool

9. Identify the following Sketch Entities icon ⌢.

- A: Center Arc tool
- B: Tangent Arc tool
- C: 3 Point Arc tool
- D: Point Arc tool

10. Identify the following Sketch Entities icon ⤴.

- A: Centerpoint Arc tool
- B: Circle tool
- C: 3 Point Arc tool
- D: Singlepoint Arc tool

11. A fully defined sketch is displayed in what color?

- A: Blue
- B: Black
- C: Red
- D: None of the listed

12. What symbol does the FeatureManager display before the Sketch name in an under defined sketch?

- A: (-)
- B: (?)
- C: (+)
- D: No symbol

13. Which is not a valid drawing format in SolidWorks?

- A: *.tif
- B: *.jpg
- C: * .dwgg
- D: * .dxf

14. Which is a valid assembly format in SolidWorks?

- A: *.tiffe
- B: *.jpgg
- C: * .assmy
- D: * .stl

15. Identify the following Drawing View icon  .

- A: Projected View
- B: Trim View
- C: Cut View
- D: Crop View

16. Identify the following Drawing View icon  .

- A: Section View
- B: Broken View
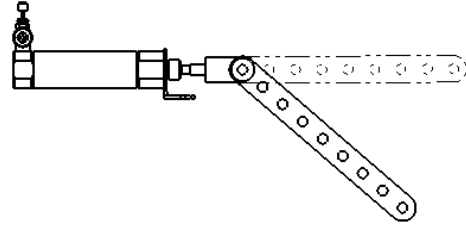- C: Break View
- D: Aligned Section View

17. Identify the following Drawing View icon  .

- A: Projected View
- B: Standard 3 View
- C: Break View
- D: Aligned Section View

18. Identify the illustrated Drawing view.

- A: Projected View

- B: Alternative Position View

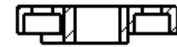- C: Extended View

- D: Aligned Section View

19. Identify the illustrated Drawing view.

- A: Crop View

- B: Break View

- C: Broken-out Section View
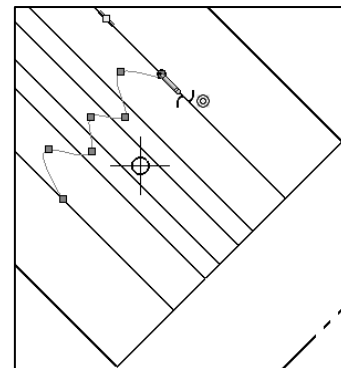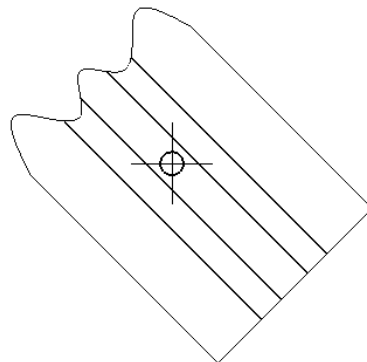
- D: Aligned Section View
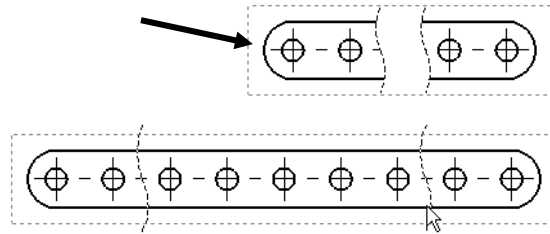
20. Identify the illustrated Drawing view.

- A: Section View

- B: Crop View

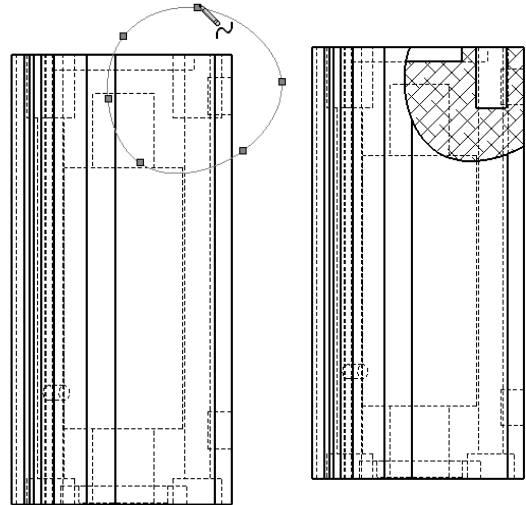- C: Broken-out Section View

- D: Aligned Section View
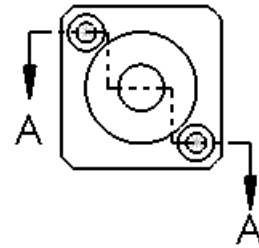
21. Identify the view procedure. To create the following view, you need to insert a:

- A: Rectangle Sketch tool

- B: Closed Profile: Spline

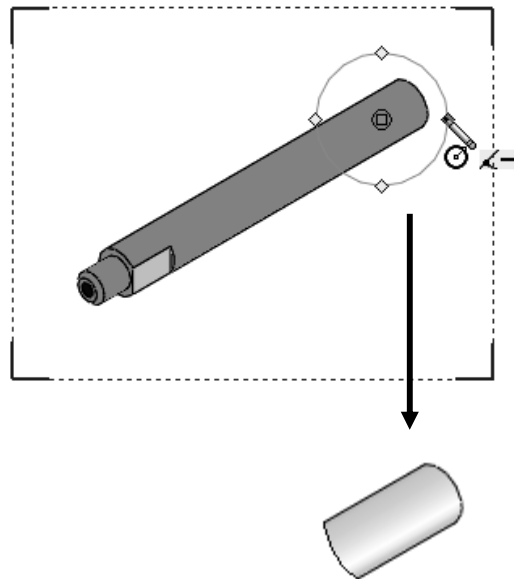- C: Open Profile: Circle

- D: None of the above

22. Identify the view procedure. To create the following view, you need to insert a:

- A: Open Spline
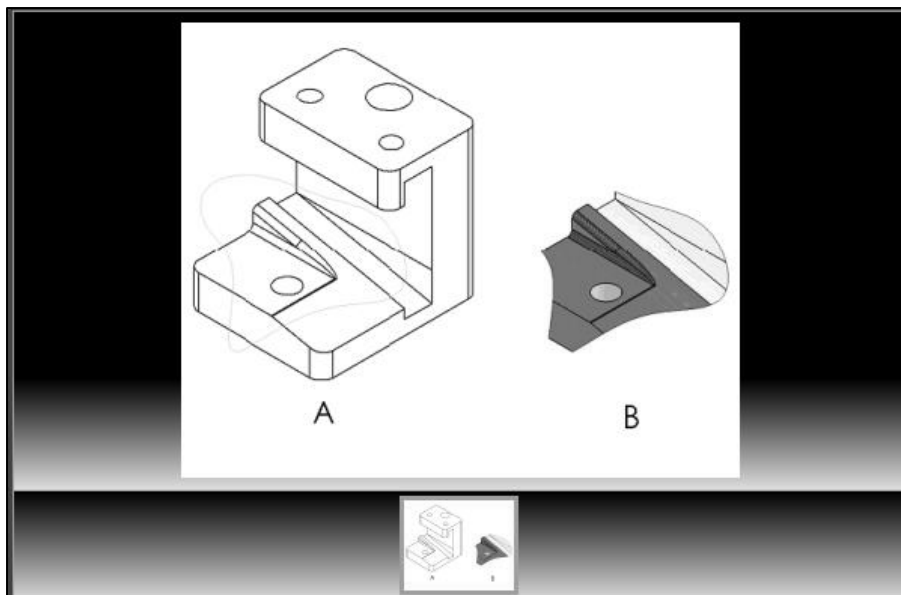- B: Closed Spline
- C: 3 Point Arc
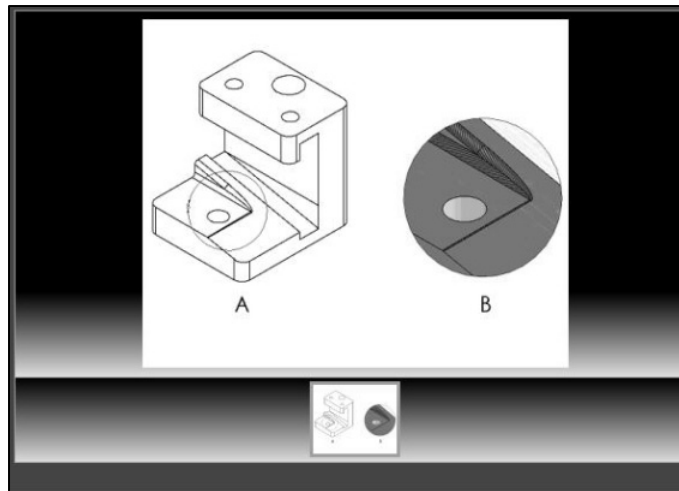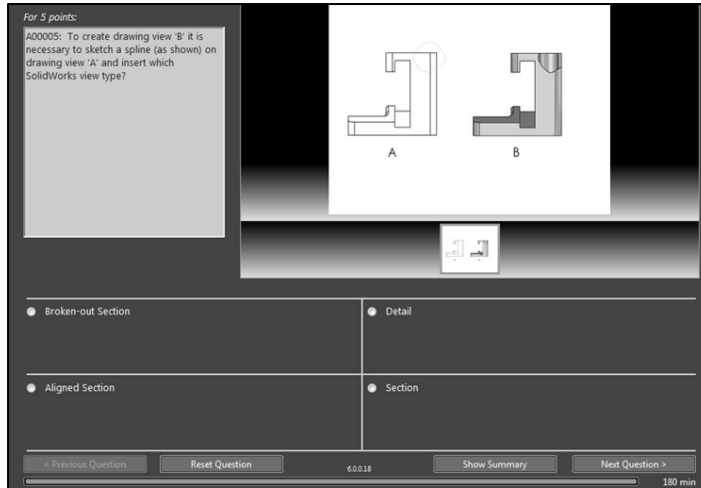- D: None of the above

23. Identify the illustrated view type.

- A: Crop view
- B: Section view
- C: Projected view
- D: None of the above

 Sample screen shots from an older CSWA exam for the Drawing Theory category. Read each question carefully. Use SolidWorks help if needed.

Zoom in on the part or view if needed.