

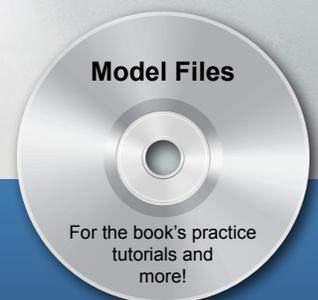
# Official Guide to Certified SolidWorks® Associate Exams: CSWA, CSDA, CSWSA-FEA

SolidWorks 2012 - 2013

An authorized CSWA preparation exam  
guide with additional information on the  
CSDA and CSWSA-FEA exams



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# CHAPTER 2 - CSWA

## INTRODUCTION AND DRAFTING

### COMPETENCIES

#### Introduction

DS SolidWorks Corp. offers various types of certification. Each stage represents increasing levels of expertise in 3D CAD: *Certified SolidWorks Associate CSWA*, *Certified SolidWorks Professional CSWP* and *Certified SolidWorks Expert CSWE* along with specialty fields in Simulation, Sheet Metal, Weldments, Molds, Surfacing, and Sustainable Design.

The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The main requirement for obtaining the CSWA certification is to take and pass the on-line proctored 180 minute exam (minimum of 165 out of 240 points). The new CSWA exam consists of fourteen questions in the following five categories and subject areas:

- *Drafting Competencies*: (Three questions - multiple choice - 5 points each).
  - Questions on general drawing views: Projected, Section, Break, Crop, Detail, Alternate Position, etc.
- *Basic Part Creation and Modification*: (Two questions - one multiple choice / one single answer - 15 points each).
  - Sketch Planes:
    - Front, Top, Right
  - 2D Sketching:



A00006: Drafting Competencies - To create drawing view 'B' it is necessary to select drawing view 'A' and insert which SolidWorks view type?

B22001: Basic Part (Hydraulic Cylinder Half) - Step 1  
Build this part in SolidWorks.  
(Save part after each question in a different file in case it must be reviewed)  
  
Unit system: MMGS (millimeter, gram, second)  
Decimal places: 2  
Part origin: Arbitrary  
All holes through all unless shown otherwise.  
Material: Aluminium 1060 Alloy

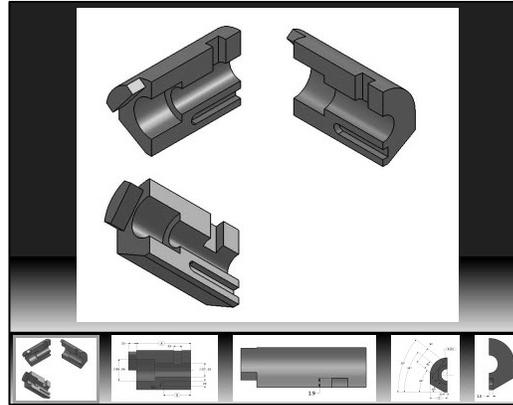
Screen shots from the exam

## CSWA Introduction and Drafting Competencies

- Geometric Relations and Dimensioning
- Extruded Boss/Base Feature
- Extruded Cut feature
- Modification of Basic part



In the *Basic Part Creation and Modification* category there is a dimension modification.



- *Intermediate Part Creation and Modification:* (Two questions - one multiple choice / one single answer - 15 points each).
  - Sketch Planes:
    - Front, Top, Right
  - 2D Sketching:
    - Geometric Relations and Dimensioning
  - Extruded Boss/Base Feature
  - Extruded Cut Feature
  - Revolved Boss/Base Feature
  - Mirror and Fillet Feature
  - Circular and Linear Pattern Feature
  - Plane Feature
  - Modification of Intermediate Part:
    - Sketch, Feature, Pattern, etc.
  - Modification of Intermediate part
- *Advanced Part Creation and Modification:* (Three questions - one multiple choice / two single answers - 15 points each).
  - Sketch Planes:
    - Front, Top, Right, Face, Created Plane, etc.
    - 2D Sketching or 3D Sketching
  - Sketch Tools:
    - Offset Entities, Convert Entities, etc.

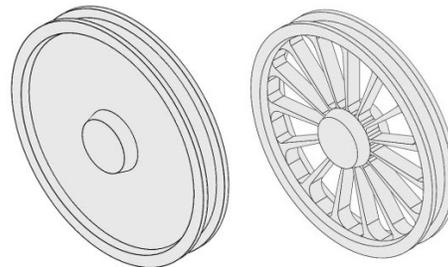
D12801: Intermediate Part (Wheel) - Step 1  
Build this part in SolidWorks.  
(Save part after each question in a different file in case it must be reviewed)

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

A = 134.00  
B = 890.00

Note: All geometry is symmetrical about the plane represented by the line labeled "F" in the M-M Section View.

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



C12801: Advanced Part (Bracket) - Step 1  
Build this part in SolidWorks.  
(Save part after each question in a different file in case it must be reviewed)

Unit system: MMGS (millimeter, gram, second)  
Decimal places: 2  
Part origin: Arbitrary  
All holes through all unless shown otherwise.  
Material: AISI 1020 Steel  
Density = 0.0079 g/mm<sup>3</sup>

A = 64.00  
B = 20.00  
C = 26.50

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

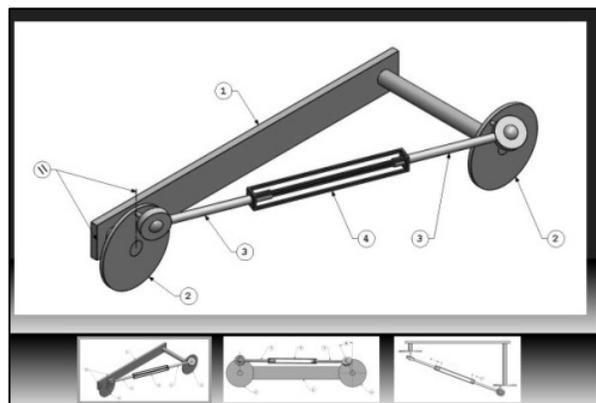
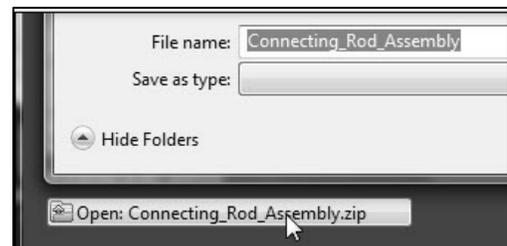
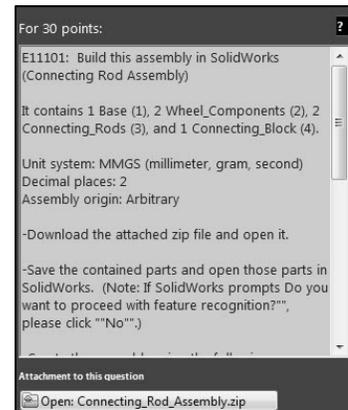
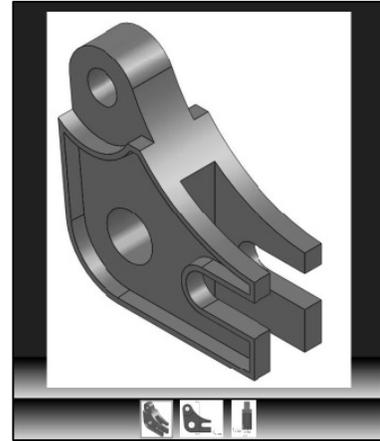
Screen shots from the exam

- Extruded Boss/Base Feature
- Extruded Cut Feature
- Revolved Boss/Base Feature
- Mirror and Fillet Feature
- Circular and Linear Pattern Feature
- Shell Feature
- Plane Feature
- More Difficult Geometry Modifications
- *Assembly Creation and Modification:* (Two different assemblies - four questions - two multiple choice / two single answers - 30 points each).
  - Insert the first (fixed) component
  - Insert all needed components
  - Standard Mates
  - Modification of key parameters in the assembly

Download the needed components in a zip folder during the exam to create the assembly.

Note: To apply for CSWA Provider status for your institution, go to [www.SolidWorks.com/cswa](http://www.SolidWorks.com/cswa) and fill out the CSWA Provider application. It is as easy as that.

 Illustrations may vary depending on your SolidWorks version and operating system.



Screen shots from the exam

## CSWA Introduction and Drafting Competencies

A total score of 165 out of 240 or better is required to obtain your CSWA Certification.



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.

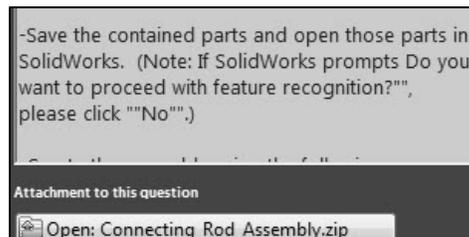
During the exam, use the control keys at the bottom of the screen to:

- *Show the Previous the Question.*
- *Reset the Question.*
- *Show the Summary Screen.*
- *Move to the Next Question.*



Do NOT use feature recognition when you open the downloaded components for the assembly in the CSWA exam. This is a timed exam. Manage your time. You do not need the additional model information.

Question (double-click to go to)	Points	State
1 A00001: Drafting Competencies - To create drawing view 'B' it is necessary to sk...	5	Answered
2 A00003: Drafting Competencies - To create drawing view 'B' from drawing view ...	5	Answered
3 A00007: Drafting Competencies - To create drawing view 'B' it is necessary to se...	5	Answered
4 B21601: Basic Part (Hydraulic Cylinder Half) - Step 1	15	Answered
5 B21602: Basic Part (Hydraulic Cylinder Half) - Step 2	15	Answered
6 D11901: Intermediate Part (Wheel) - Step 1	15	Answered
7 D11902: Intermediate Part (Wheel) - Step 2	15	Answered
8 E11501: Build this assembly in SolidWorks (Connecting Rod Assembly)	30	Answered
9 E11502: Modify this assembly in SolidWorks (Connecting Rod Assembly)	30	Answered
10 E22101: Build this assembly in SolidWorks (Linkage Assembly)	30	Answered
11 E20902: Modify this assembly in SolidWorks (Linkage Assembly)	30	Answered
12 C11801: Advanced Part (Bracket) - Step 1	15	Answered
13 C11802: Advanced Part (Bracket) - Step 2	15	Answered
14 C11803: Advanced Part (Bracket) - Step 3	15	Answered



Screen shots from the exam

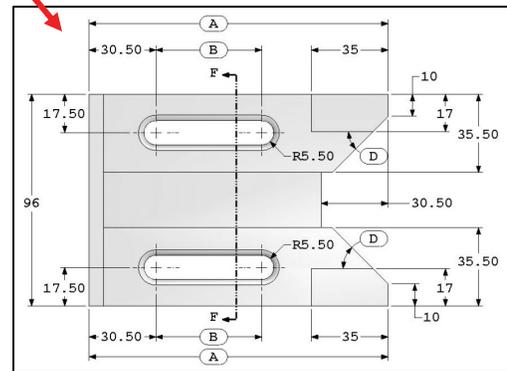
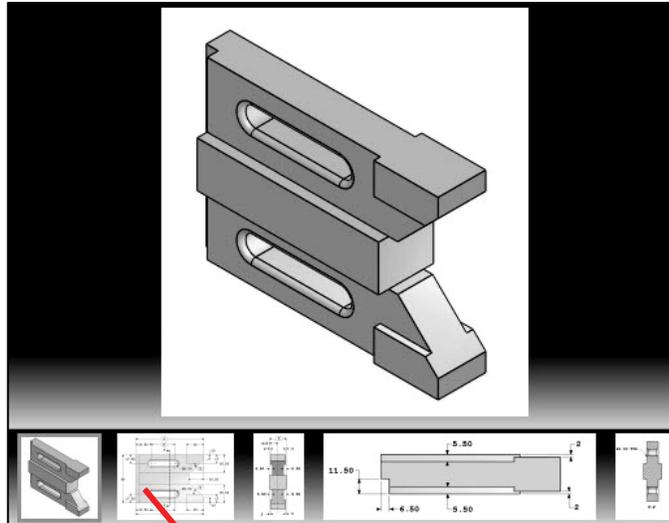
During the exam, SolidWorks provides the ability to click on a detail view below (as illustrated) to obtain additional details and dimensions during the exam.

 No Simulation (CosmosXpress) questions are on the CSWA exam.

 No Sheetmetal questions are on the CSWA 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.

 No Surface questions are on the CSWA exam.



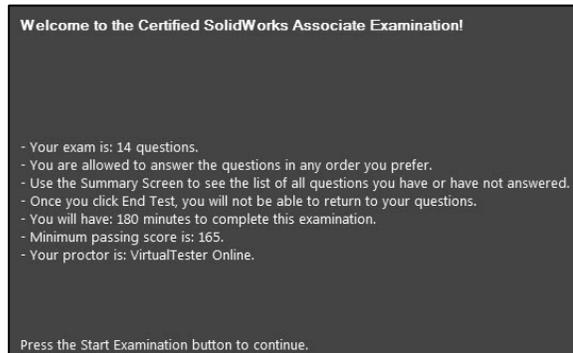
Screen shots from the exam

## Goals

The primary goal is not only to help you pass the CSWA exam, but also to ensure that you understand and comprehend the concepts and implementation details of the CSWA process.

The second goal is to provide the most comprehensive coverage of CSWA exam related topics available, without too much coverage of topics not on the exam.

The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA exam.



Screen shots from the exam

# Objectives

Drafting Competencies is one of the five categories (*Drafting Competencies, Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advance Part Creation and Modification, and Assembly Creation and Modification*) on the CSWA exam. This chapter covers the general concepts, symbols and terminology used in the exam and then the core element (*Drafting Competencies*) which is aligned to the new CSWA exam.

There are three questions (total) on the CSWA exam in the *Drafting Competencies* category. Each question is worth five (5) points. The three questions are in a multiple choice single answer format. You are allowed to answer the questions in any order you prefer. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.

In the *Drafting Competencies* category of the exam, you are **not required** to create or perform an analysis on a part, assembly, or drawing but you are required to have general drafting / drawing knowledge and understanding of various drawing view methods.

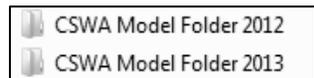
On the completion of the chapter, you will be able to:

- Recognize 3D modeling techniques:
- 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:
  - Projected view, Section view, Break view, Crop view, Detail, Alternate Position view, etc.
- Specify Document Properties: Select Unit System, and Set Precision



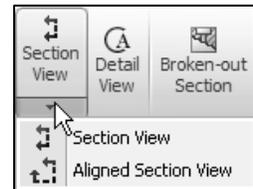
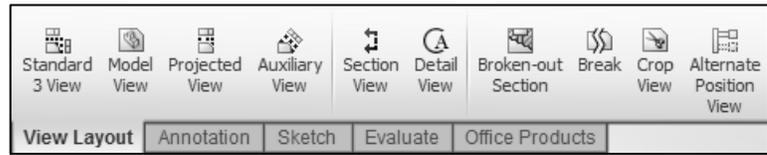
In the *Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advanced Part Creation and Modification and Assembly Creation and Modification* categories, you are required to read and interpret various types of drawing views and understand various types of drawing annotations.

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



## 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*.

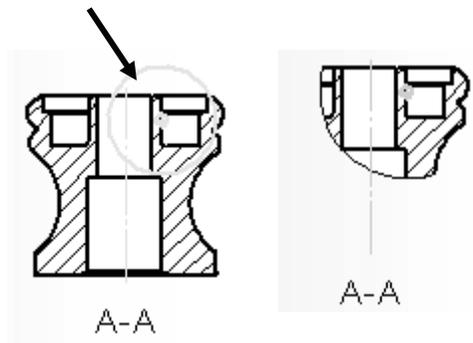


Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

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.

 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.

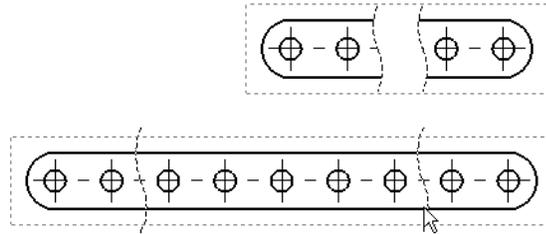


 Create a Detail view in a drawing to display a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, a non-planar (isometric) view, a Section view, a Crop view, an Exploded assembly view, or another Detail view.

**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.



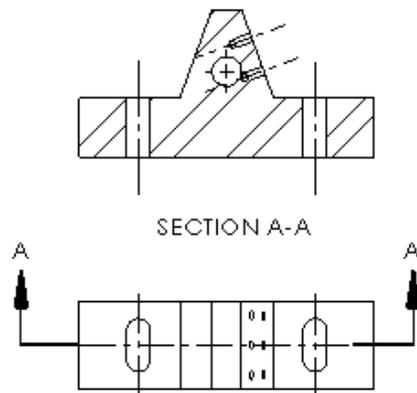
 Broken views make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.

 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 cutting section line.

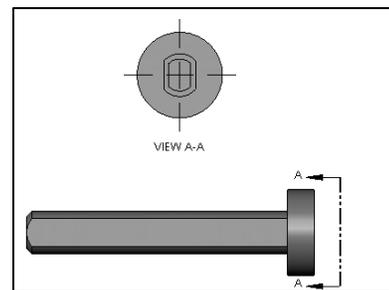


 Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

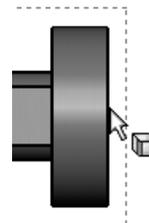
**Tutorial: Drawing Named Procedure 2-3**

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 Auxiliary view of the Front view. Select a reference edge to create an Auxiliary view.



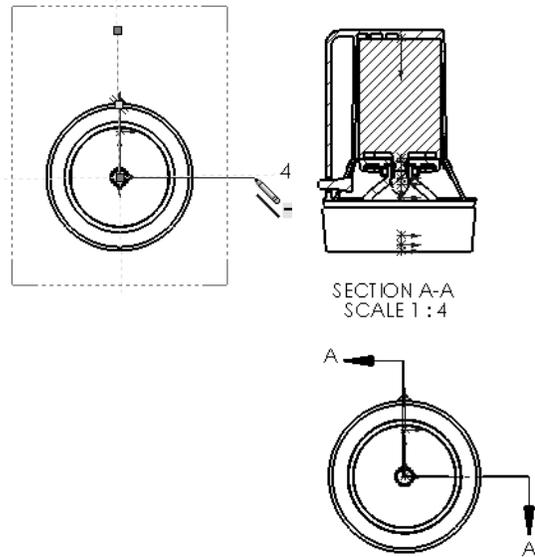
 An Auxiliary view is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view.



**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.

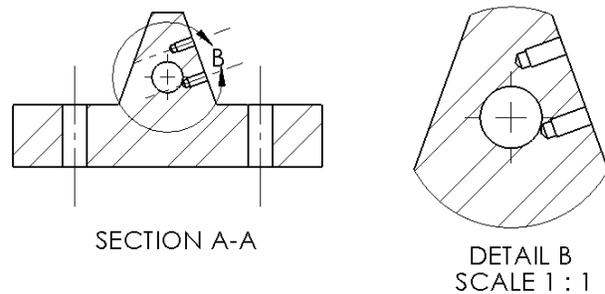


 The Aligned Section view is very similar to a Section View, with the exception that the section line for an aligned section comprises of two or more lines connected at an angle. The procedure was modified in SolidWorks 2013.

**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 of the Section 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.

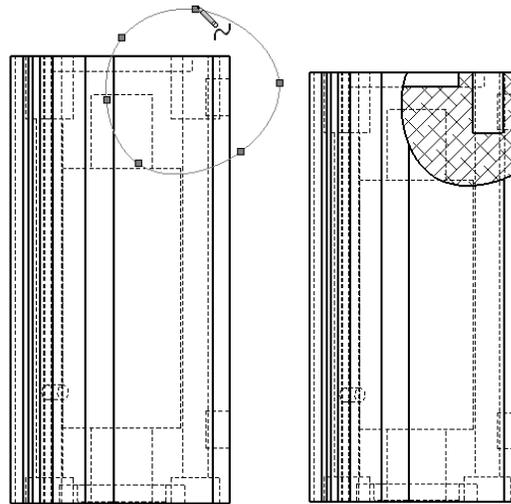
 To create a profile other than a circle, sketch the profile before clicking the Detail view tool. Using a sketch entity tool, create a closed profile around the area to be detailed.



**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.



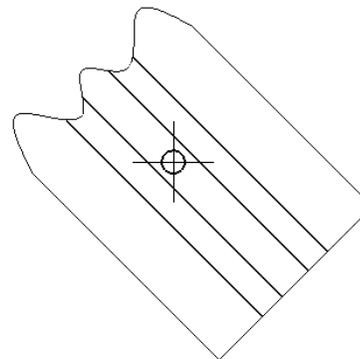
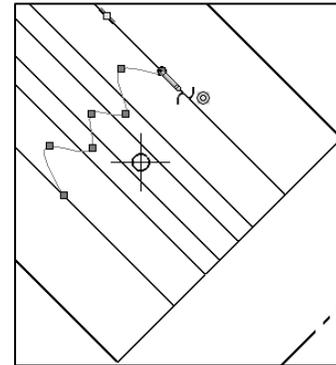
**Tutorial: Drawing Named Procedure 2-7**

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.



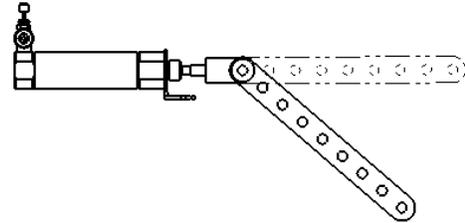
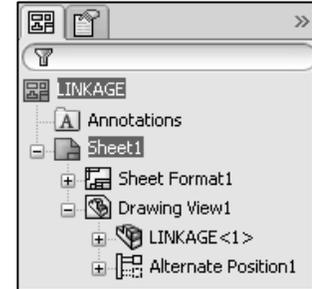
In the exam; you are allowed to answer the questions in any order. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.

**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.



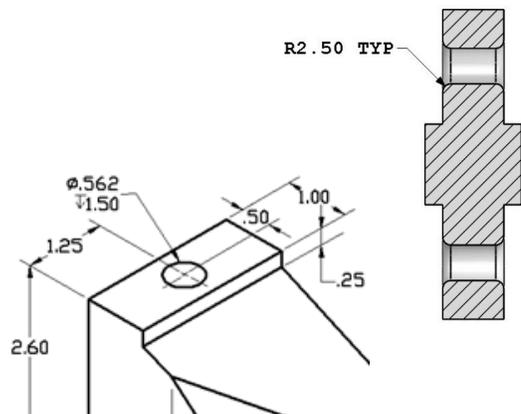
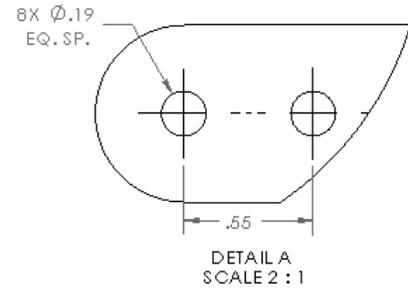
 Use the Alternate Position view to display the range of motion of an assembly. You can dimension between the primary view and the Alternate Position view. You can not use the Alternate Position view tool with Broken, Section, or Detail views.

## Engineering Documentation Practices

A 2D drawing view is displayed in the *Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advance Part Creation and Modification, and Assembly Creation and Modification* categories of the CSWA exam to clarify dimensions and details.

The ability to interpret a 2D drawing view is required.

- Example 1: *8X Ø.19 EQ. SP.* Eight holes with a .19in. diameter are required that are equally (.55in.) spaced.
- Example 2: *R2.50 TYP.* Typical radius of 2.50. The dimension has a two decimal place precision.
- Example 3:  $\nabla$ . The Depth/Deep  $\nabla$  symbol with a 1.50 dimension associated with the hole. The hole  $\text{Ø}.562$  has a three decimal place precision.



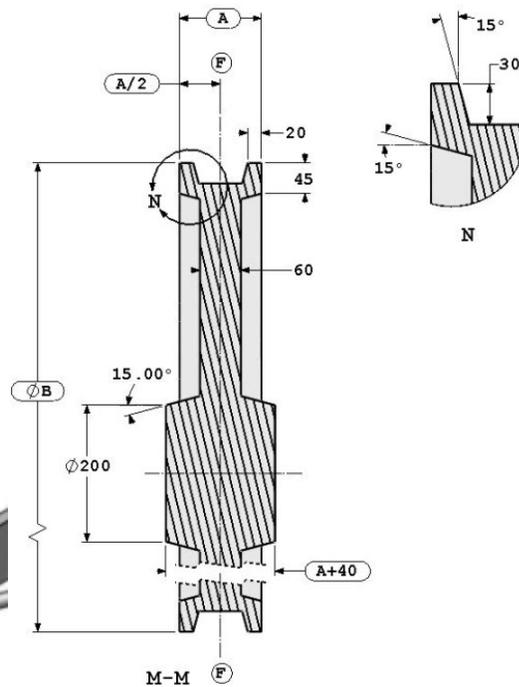
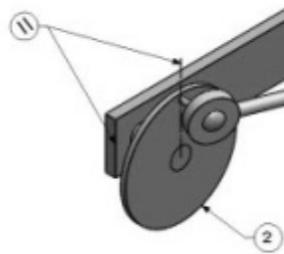
## CSWA Introduction and Drafting Competencies

- Example 4:  $A+40$ . A is provided to you on the CSWA exam.  $44\text{mm} + A$ .

 N is a Detail view of the M-M Section view.

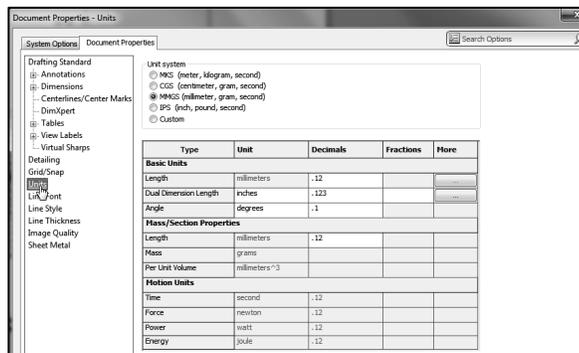
- Example 5:  $\varnothing B$ . Diameter of B. B is provided to you on the exam.

- Example 6: . Parallelism.



## 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.



 Set precision for the selected unit system during the exam as illustrated.

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 model or drawing.

Build this part in SolidWorks.  
(Save part after each question in a different file in case it must be reviewed)

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

A = 136.00  
B = 48.00  
C = 45 degrees

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

Screen shot from the exam

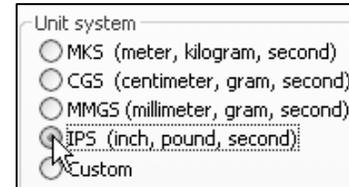
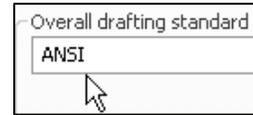
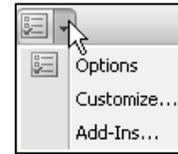


Know how to set the Unit system, and Tolerance/Precision option in a document for the CSWA exam.

**Tutorial: Document Properties 2-1**

Set Document Properties for a part.

1. **Create** a New part in SolidWorks.
2. Click **Options, Document Properties** tab from the Menu bar toolbar.
3. Select **ANSI** for the Overall drafting standard drop-down menu.
4. Click the **Units** folder.
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.

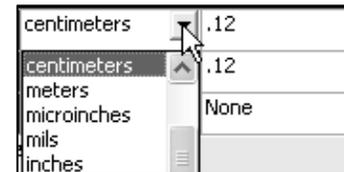


Type	Unit	Decimals
<b>Basic Units</b>		
Length	inches	.123
Dual Dimension Length	inches	.12
Angle	degrees	None

**Tutorial: Document Properties 2-2**

Set Document Properties (Custom) for a part.

1. **Create** a New part in SolidWorks.
2. Click the **Options, Document Properties** tab from the Menu bar menu.
3. Select **ANSI** for Overall drafting standard from the drop-down menu.
4. Click the **Units** folder.
5. Click **Custom** for Unit system.
6. Select **centimeters** from the length units drop down box.
7. Select **.12** Decimal places for Length units.
8. Select **None** Decimal places for Angular units.
9. **Close** the part.



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## Summary

*Drafting Competencies* is one of the five categories on the CSWA exam. There are three questions on the CSWA exam in this category. Each question is worth five (5) points. The three 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.

*Basic Part Creation and Modification* and *Intermediate Part Creation and Modification* is the next chapter in this book.

This chapter covers the knowledge to create and modify models for these categories from detailed dimensioned illustrations.

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

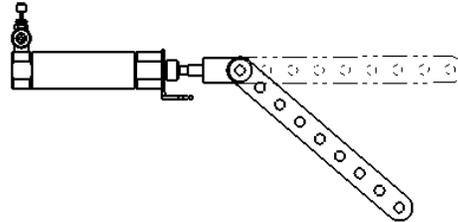
 Screen shots in the book were made using SolidWorks 2012 SP2 and SolidWorks 2013 SP0 running Windows® 7 Ultimate.

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## Questions

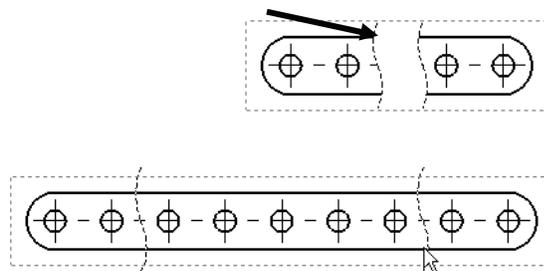
1. Identify the illustrated Drawing view.

- A: Projected
- B: Alternative Position
- C: Extended
- D: Aligned Section



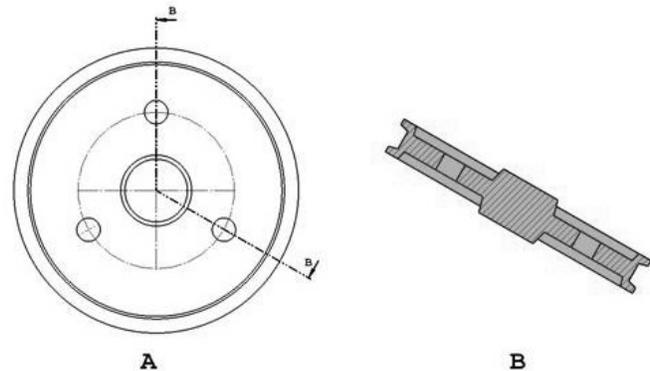
2. Identify the illustrated Drawing view.

- A: Crop
- B: Break
- C: Broken-out Section
- D: Aligned Section



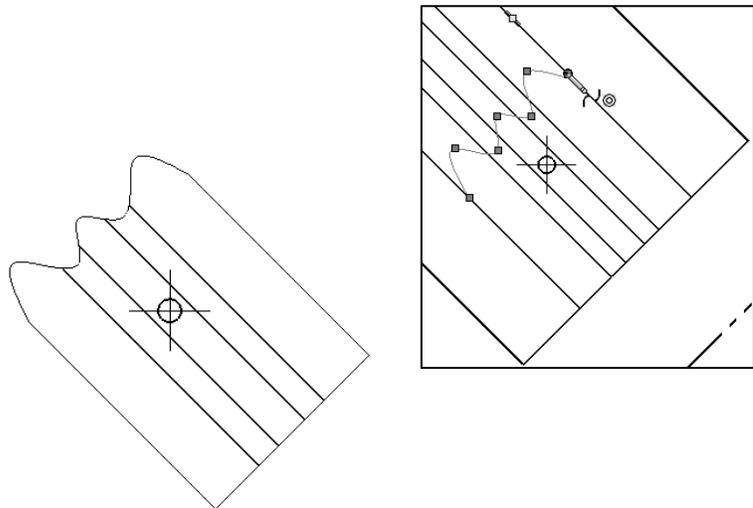
3. Identify the illustrated Drawing view.

- A: Section
- B: Crop
- C: Broken-out Section
- D: Aligned



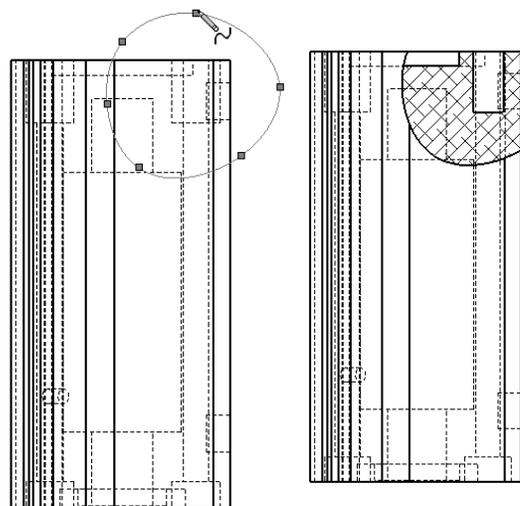
4. Identify the view procedure. To create the following view, you need to insert a:

- A: Rectangle Sketch tool
- B: Closed Profile: Spline
- C: Open Profile: Circle
- D: None of the above



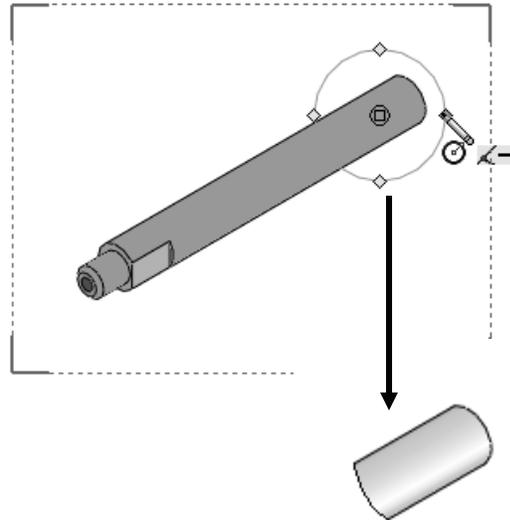
5. Identify the view procedure. To create the following view, you need to insert a:

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



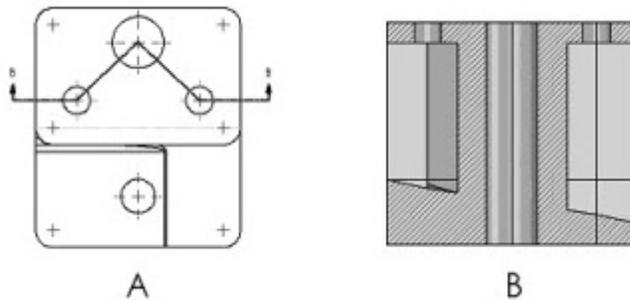
6. Identify the illustrated view type.

- A: Crop
- B: Section
- C: Projected
- D: Detail



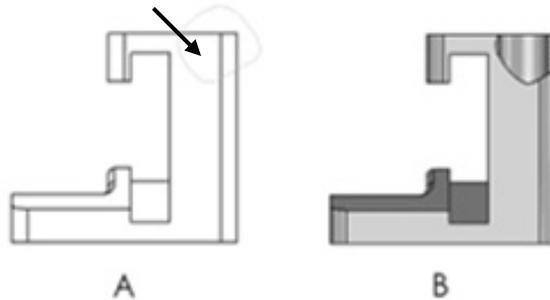
7. To create View B from Drawing View A insert which view type?

- A: Crop
- B: Section
- C: Aligned Section
- D: Projected



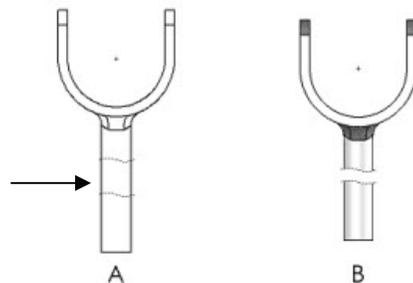
8. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

- A: Broken out Section
- B: Detail
- C: Section
- D: Projected

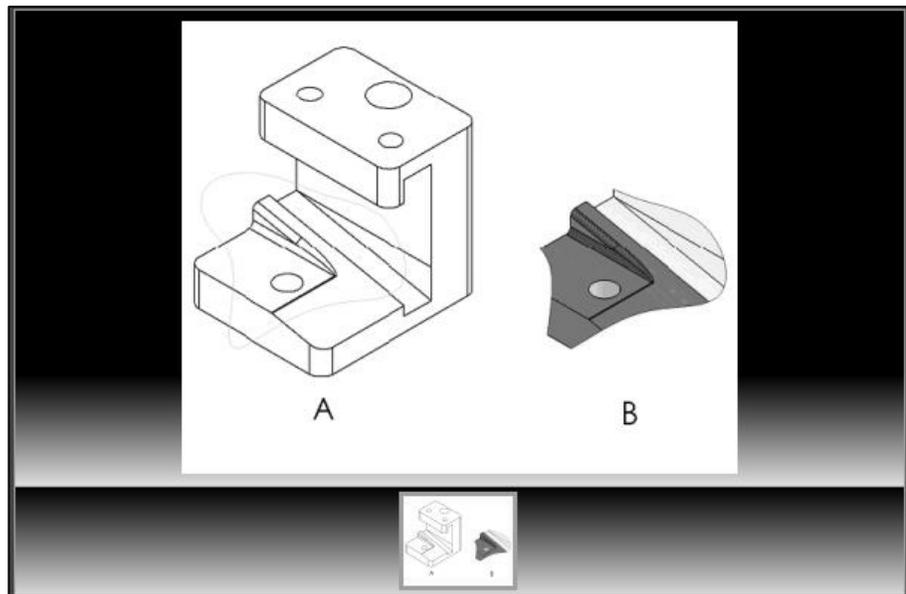
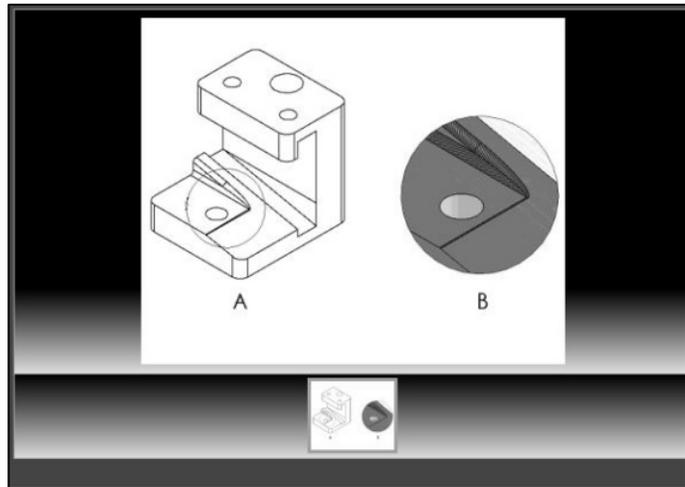


9. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

- A: Horizontal Break
- B: Detail
- C: Section
- D: Broken out Section

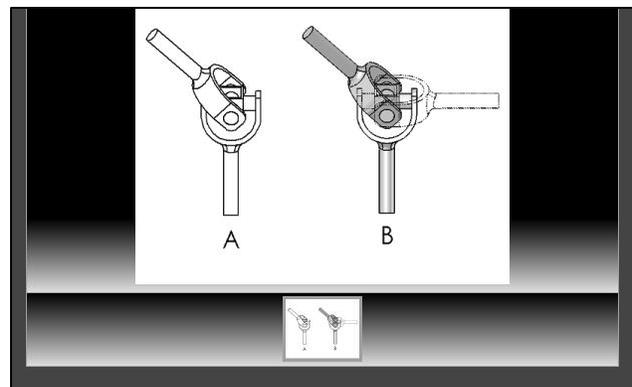


💡 Screen shots from an older CSWA exam for the *Drafting Competencies* category. Read each question carefully. Use SolidWorks help if needed.



💡 Zoom in on the part or view if needed.

**Alternative Position View:**

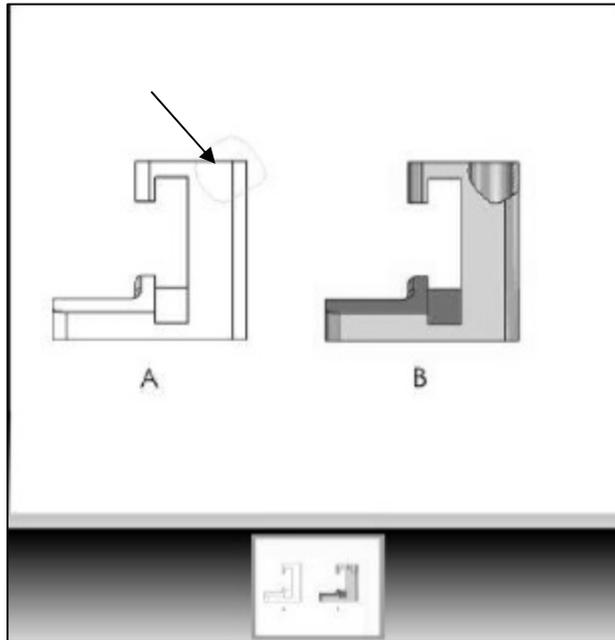


Screen shots from the exam

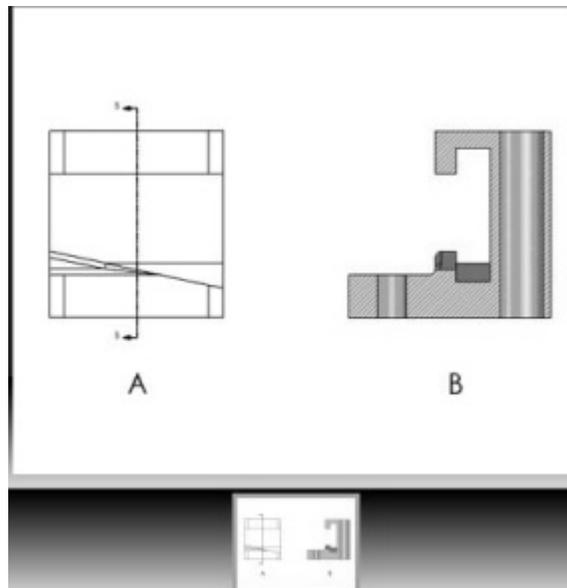
## CSWA Introduction and Drafting Competencies

 Screen shots from an older CSWA exam for the *Drafting Competencies* category. Read each question carefully. Use SolidWorks help if needed.

### Broken out Section View:



### Section View:



Screen shots from the exam