

# Official Guide to Certified **SOLIDWORKS® Associate Exams:** CSWA, CSWA-SD, CSWSA-FEA, CSWA-AM

SOLIDWORKS® 2017 - 2019

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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.

The CSWA Academic exam is provided either in a single 3 hour segment, or 2 - 90 minute segments.

Part 1 of the CSWA Academic exam is 90 minutes; minimum passing score is 80, with 6 questions.

There are two questions in the Basic Part Creation and Modification category, two questions in the Intermediate Part Creation and Modification category and two questions in the Assembly Creation and Modification category.

Part 2 of the CSWA Academic exam is 90 minutes; minimum passing score is 80 with 8 questions. There are three questions on the CSWA Academic exam in the Drafting Competencies category, three questions in the Advanced Part Creation and Modification category and two questions in the Assembly Creation and Modification category.

The CSWA exam for industry is only provided in a single 3 hour segment. The exam consists of 14 questions in five categories worth a total of 240 points. All exams cover the same material.

- *Drafting Competencies* (Three questions - multiple choice - 5 points each).
  - Questions on general drawing views: Projected, Section, Break, Crop, Detail, Alternate Position, etc.

Question (double-click to go to)	Points	State
1 Basic Part - Step 2 - (Hydraulic Cylinder Shaft)	15	Answered
2 Basic Part - Step 2 - (Hydraulic Cylinder Shaft)	15	Answered
3 Intermediate Part - Step 1 - (Wheel)	15	Answered
4 Intermediate Part - Step 2 - (Wheel)	15	Answered
5 Build this assembly in SOLIDWORKS (Connecting Rod Assembly)	30	Answered
6 Modify this assembly in SOLIDWORKS (Connecting Rod Assembly)	30	Answered

Question (double-click to go to)	Points	State
1 Drafting Competencies - To create drawing view 'B' from drawing view ...	5	Answered
2 Drafting Competencies - To create drawing view 'B' from drawing view ...	5	Answered
3 Drafting Competencies - To create drawing view 'B' from drawing view ...	5	Answered
4 Build this assembly in SOLIDWORKS (Gripper Assembly)	30	Answered
5 Modify this assembly in SOLIDWORKS (Gripper Assembly)	30	Not Answered
6 Advanced Part - Step 1 - (Connector)	15	Answered
7 Advanced Part - Step 2 - (Connector)	15	Answered
8 Advanced Part - Step 3 - (Connector)	15	Answered

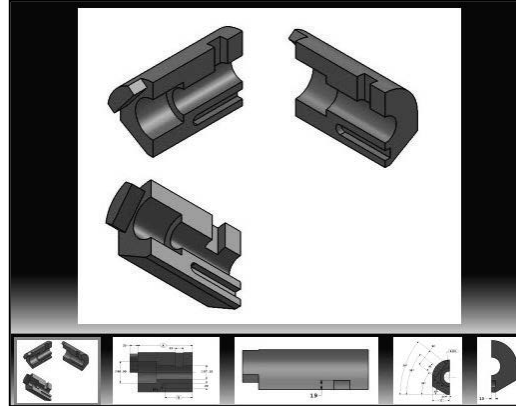


Drafting Competencies - To create drawing view 'B' it is necessary to select drawing view 'A' and insert which SolidWorks view type?

Screen shot from an exam

## CSWA Introduction and Drafting Competencies

- *Basic 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.
  - Modification of Basic part.



In the *Basic Part Creation and Modification* category there is a dimension modification question based on the first (multiple choice) question. You should be within 1% of the multiple-choice answer before you go on to the modification single answer section.

- *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.

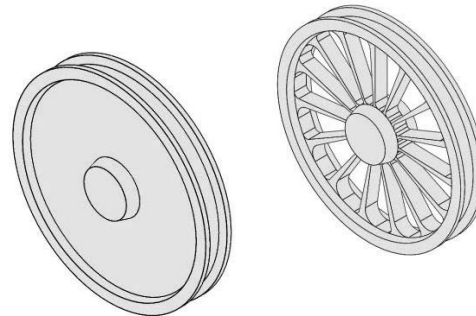
Intermediate Part (Wheel) - Step 1  
Build this part in SolidWorks.  
(Save part after each question in a different file in case it must be reviewed)

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)?



Hint: If you don't find an option within 1% of your answer please re-check your model(s).

Screen shots from an exam



In the *Intermediate Part Creation and Modification* category; there are two dimension modification questions based on the first (multiple choice) question. You should be within 1% of the multiple-choice answer before you go on to the modification single answer section.

- *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.
  - 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.

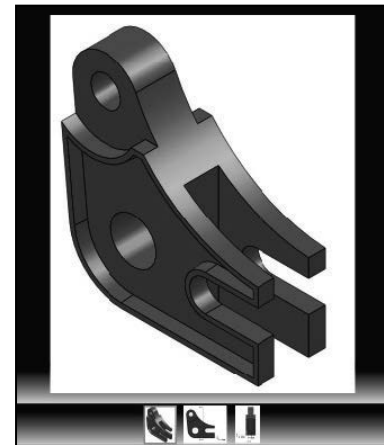
Advanced Part (Bracket) - Step 1

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)?



Hint: If you don't find an option within 1% of your answer please re-check your model(s).



In the *Advanced Part Creation and Modification* category, there are two dimension modification questions based on the first (multiple choice) question. You should be within 1% of the multiple-choice answer before you go on to the modification single answer section.

- *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.



Screen shots from an exam

## CSWA Introduction and Drafting Competencies

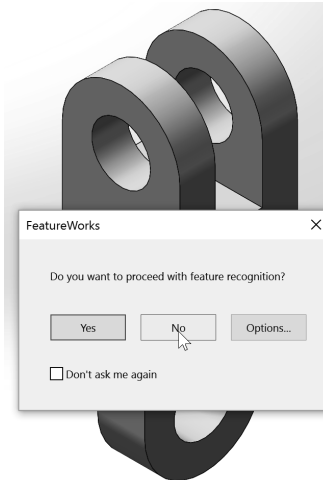
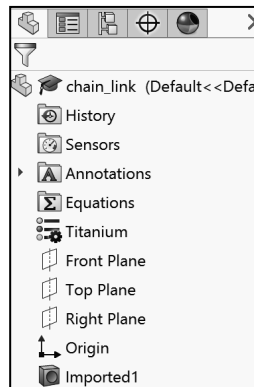
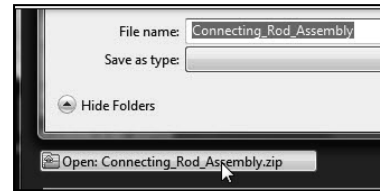


In the *Assembly Creation and Modification* category; expect to see five to seven components. There are two dimension modification questions based on the first (multiple choice) question. You should be within 1% of the multiple-choice answer before you go on to the modification single answer section.

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

Do not use feature recognition when you open the downloaded components for the assembly. This is a timed exam.

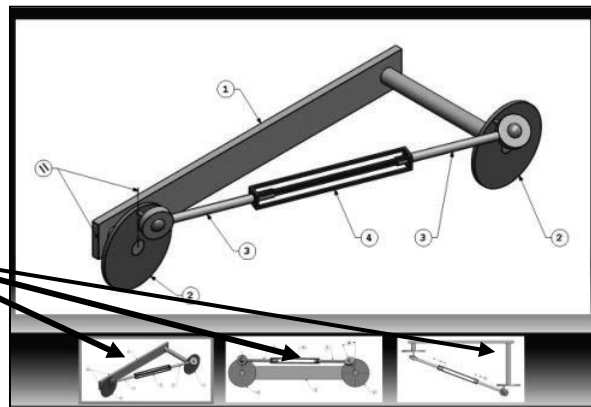
Additional model information is not needed in the exam.



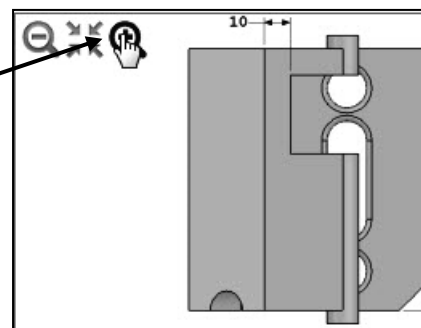
Illustrations may vary depending on your SOLIDWORKS version and system setup.

Use the view indicator to increase or decrease the active model in the view window.

View indications



View indicator



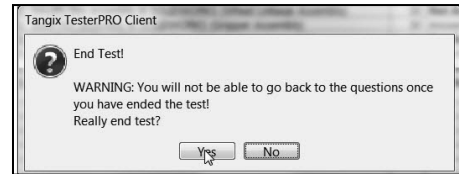
Screen shots from an exam

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.

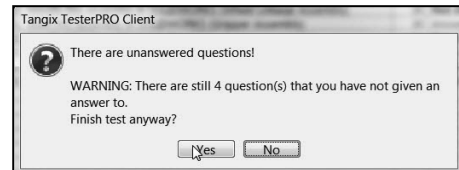
During the exam, use the control keys at the bottom of the screen to:

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

When you are finished, press the End Examination button. The tester will ask you if you want to end the text. Click Yes.



If there are any unanswered questions, the tester will provide a warning message as illustrated.

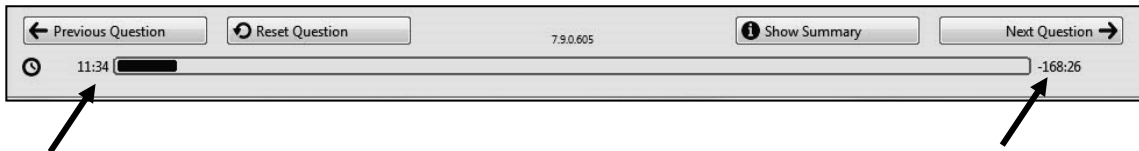


If you do not pass the certification exam (either segment), you will need to wait 30 days until you can retake that segment of the exam.



To obtain additional CSWA exam information, visit the SOLIDWORKS VirtualTester Certification site at <https://SOLIDWORKS.virtualtester.com/>.

Use the clock in the tester to view the amount of time that you used and the amount of time that is left in the 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.


There are three questions (total) on the CSWA exam in the *Drafting Competencies* category. Each question is worth five (5) points. Drafting Competency questions are addressed in part 2 of the CSWA exam.

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.

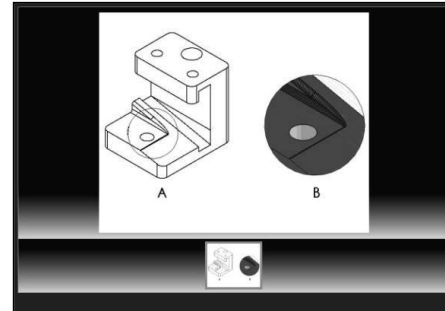
In the *Drafting Competencies* category of the exam, you are **not required** to create or perform an analysis on a part, assembly, or drawing but you are required to have general drafting/drawing knowledge and understanding of various drawing view methods.

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




- Identify the procedure to create a named drawing view: Projected view, Section view, Break view, Crop view, Detail, Alternate Position view, etc.










 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. Download the corresponding folder that matches your release of SOLIDWORKS.



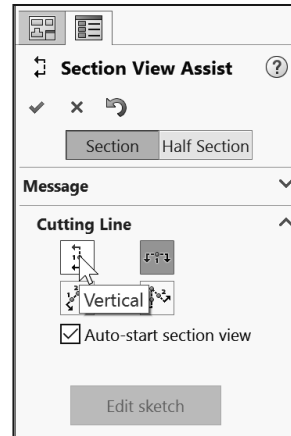
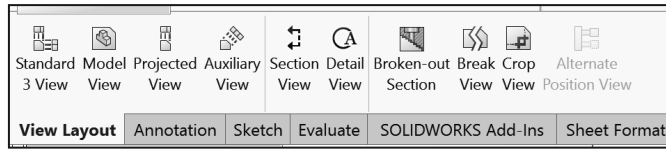
Screen shot from the exam

-  SOLIDWORKS 2017
-  SOLIDWORKS 2018
-  SOLIDWORKS 2019

-  Chapter 2
-  Chapter 2 Final Solutions
-  Chapter 3
-  Chapter 3 Final Solutions
-  Chapter 4
-  Chapter 4 Final Solutions
-  Chapter 5
-  Chapter 5 Final Solutions
-  CSWASampleExam

## Procedure to Create a Named Drawing view

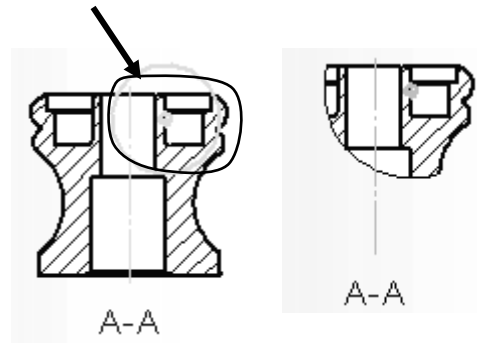
You need the ability to identify the procedure to create a named drawing view: *Standard 3 View, Model View, Projected View, Auxiliary View, Section View, Aligned Section View, Detail View, Broken-out Section, Break, Crop View and Alternate Position View.*



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.

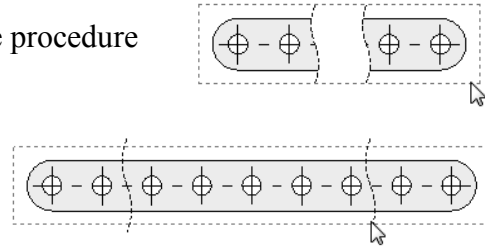



Create a Detail view in a drawing to display a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, a non-planar (isometric) view, a Section view, a Crop view, an Exploded assembly view, or another Detail view.


**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.



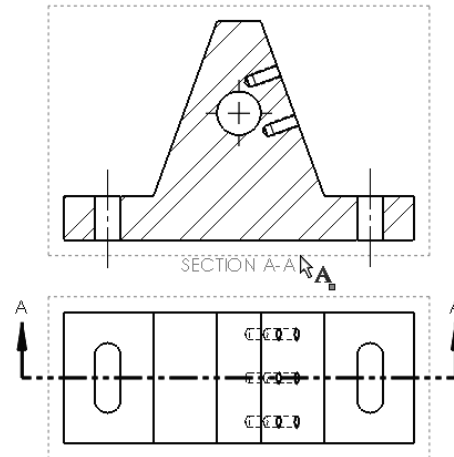
 Broken views make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.


 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.



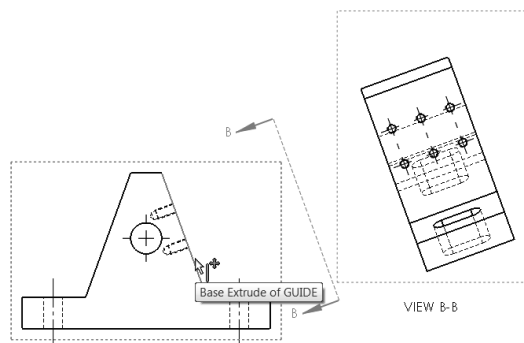
 Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

**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 view to the right is an Auxiliary view of the Front view. Select a reference edge to create an Auxiliary view as illustrated.

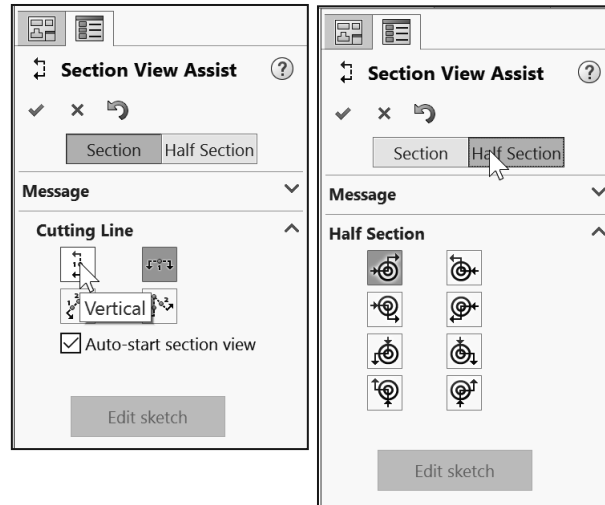
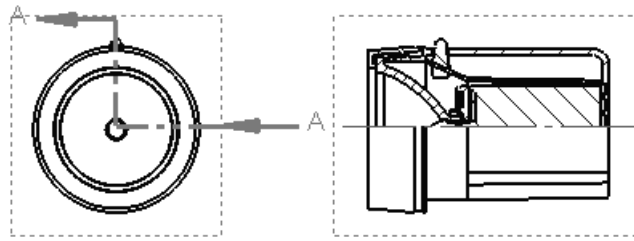
An Auxiliary view is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view.




**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 half Section view of the view to the left. The Section view is created by using two lines connected at an angle. Create an Aligned half Section view in a drawing through a model, or portion of a model, that is aligned with a selected section line segment.

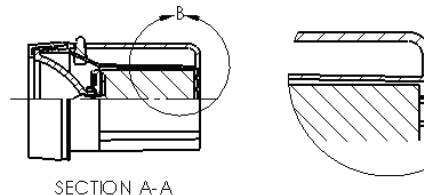



 The Aligned Section view is very similar to a Section View, with the exception that the section line for an aligned half section is comprised of two or more lines connected at an angle.

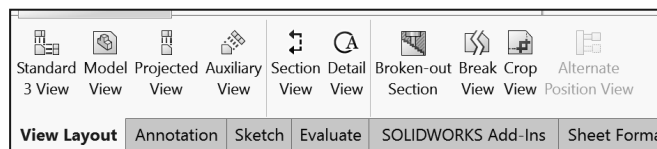
**Tutorial: Drawing Named Procedure 2-5**


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 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.

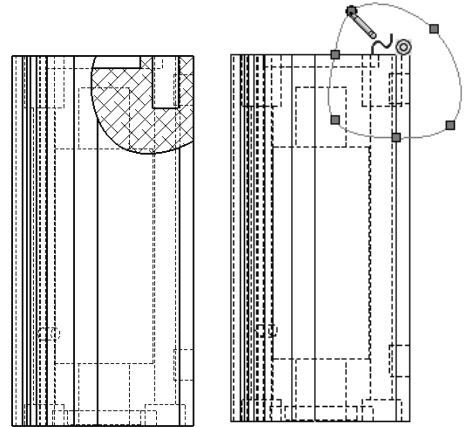


 To create a profile other than a circle, sketch the profile before clicking the Detail view tool. Using a sketch entity tool, create a closed profile around the area to be detailed.

**Tutorial: Drawing Named Procedure 2-6**

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

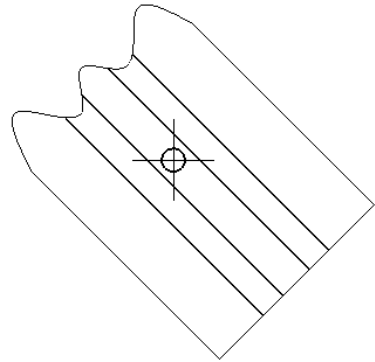
1. **View** the illustrated drawing views. The right drawing view is a Broken-out Section view. The Broken-out Section View is part of an existing drawing view, not a separate view. Create the Broken-out Section view with a closed profile, usually by using the Spline Sketch tool. Material is removed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



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

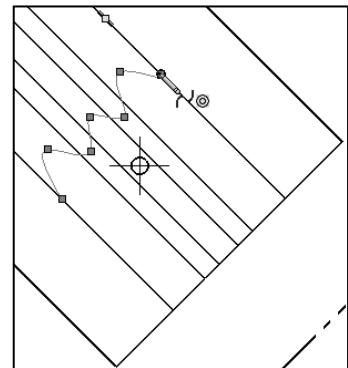
Identify the drawing name view and understand the procedure to create the name view.


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 cannot 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.



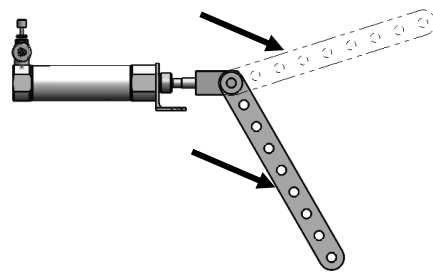
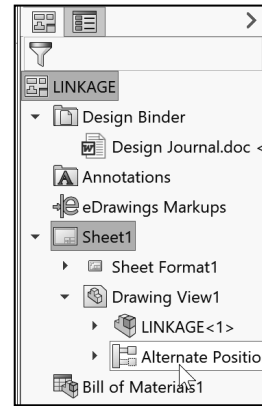
 In the exam you are allowed to answer the questions in any order. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.

**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 cannot use the Alternate Position view tool with Broken, Section, or Detail views.



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## Summary

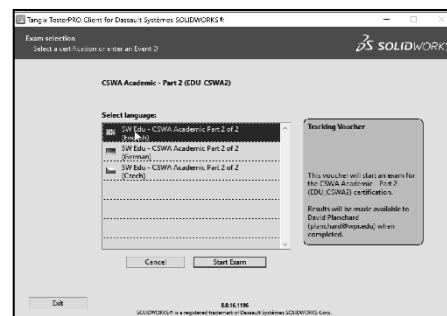
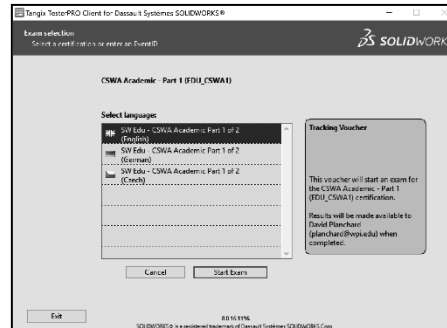
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The CSWA exam for industry is only provided in a single 3 hour segment (14 questions - 5 categories). All exams cover the same material.

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.


Drafting Competency questions are addressed in part 2 of the CSWA Academic exam.


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 the book. The 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.

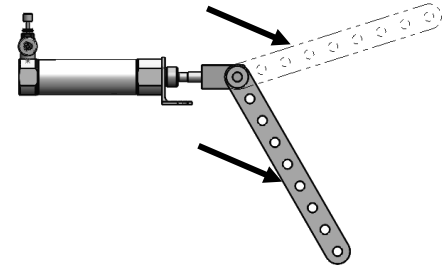
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 Screen shots in the book were made using SOLIDWORKS 2018 SP2 and SOLIDWORKS 2019 SP0 running Windows® 10.

## Questions

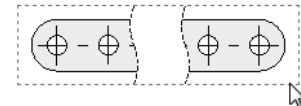
1. Identify the illustrated Drawing view.

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



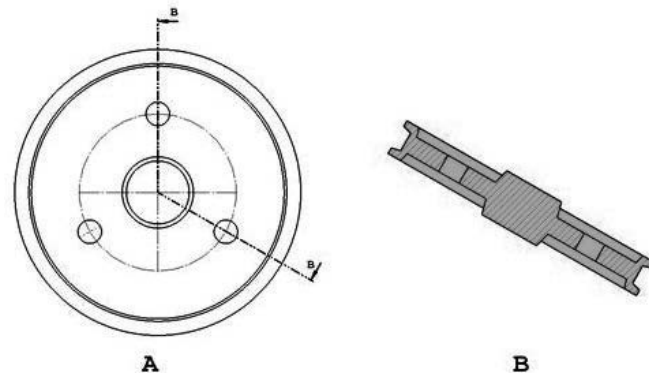
2. Identify the illustrated Drawing view.

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



3. Identify the illustrated Drawing view.

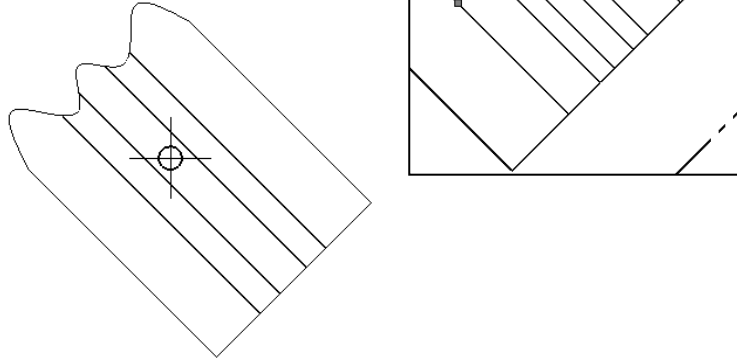
- A: Section
- B: Crop
- C: Broken-out Section
- D: Aligned Section



## CSWA Introduction and Drafting Competencies

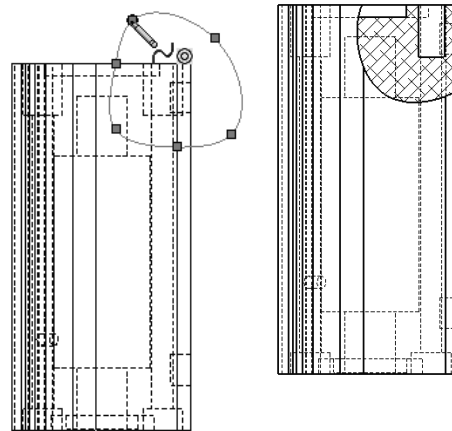
4. Identify the view procedure. To create the following view, you need to insert a:

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



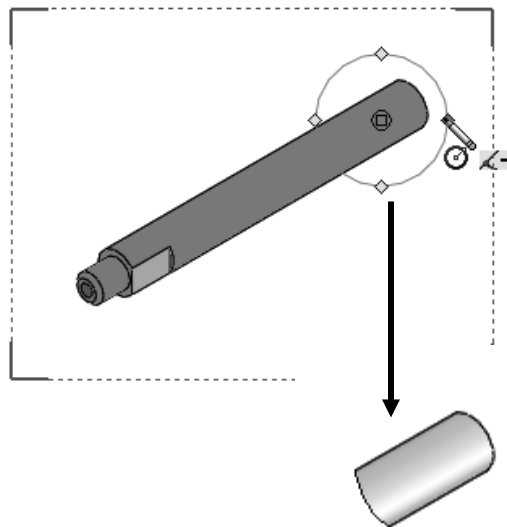
5. Identify the view procedure. To create the following view, you need to insert a:

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



6. Identify the illustrated view type.

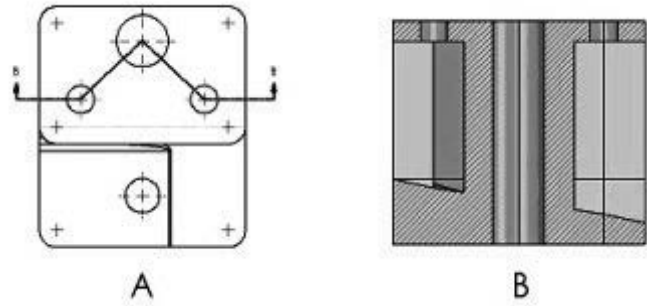
- A: Crop
- B: Section
- C: Projected
- D: Detail





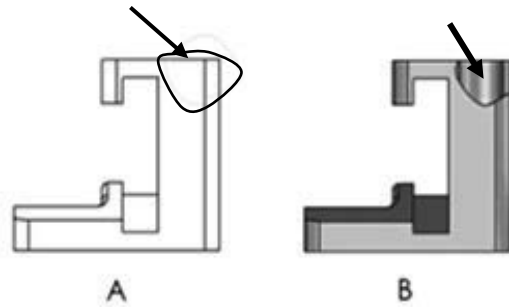
7. To create View B from Drawing View A insert which view type?

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



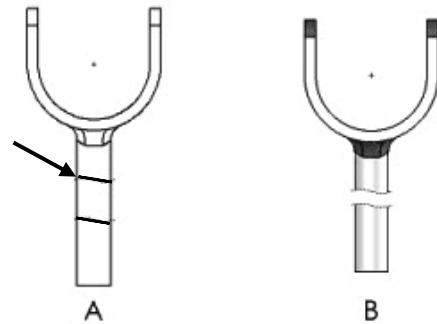
8. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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



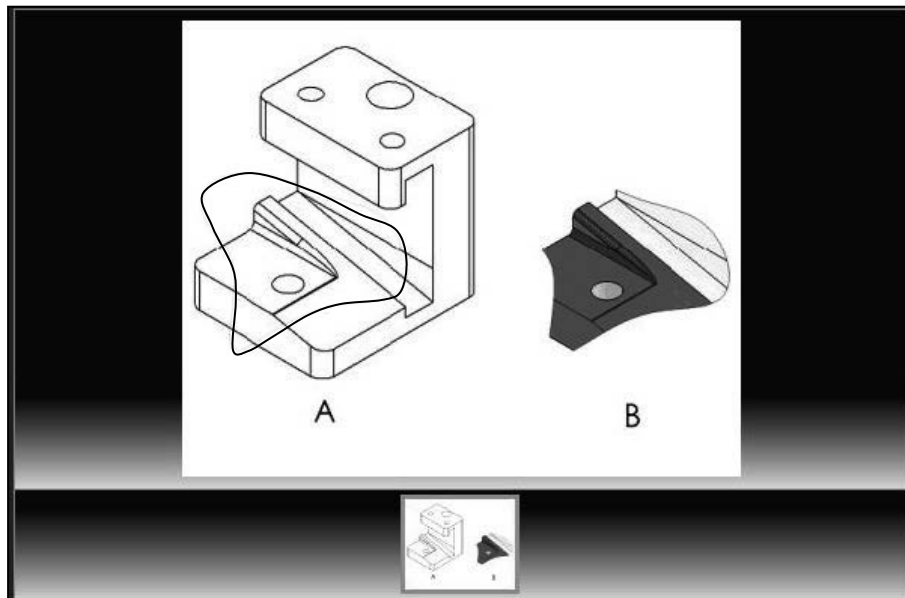
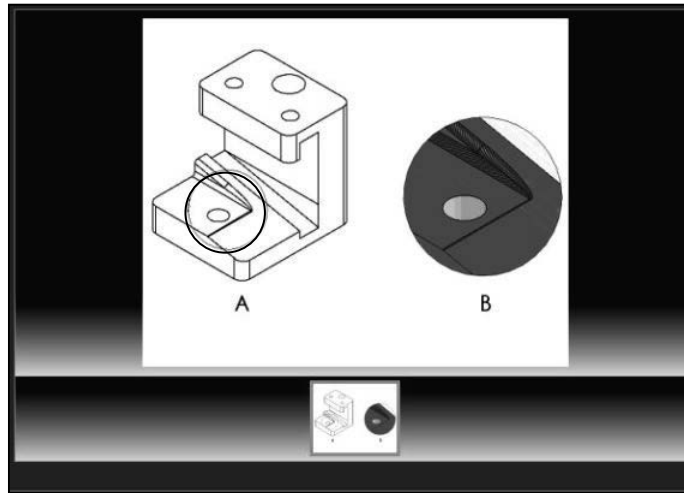
9. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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



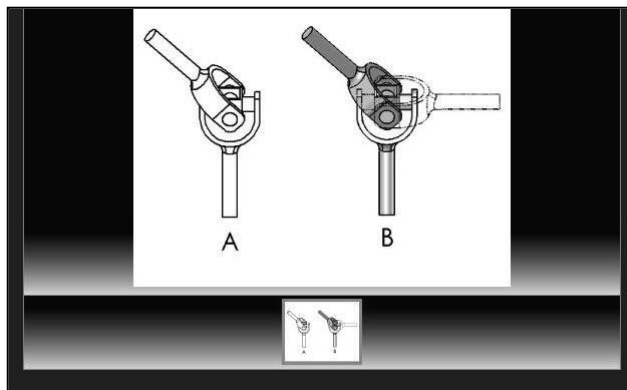
**CSWA Introduction and Drafting Competencies**

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



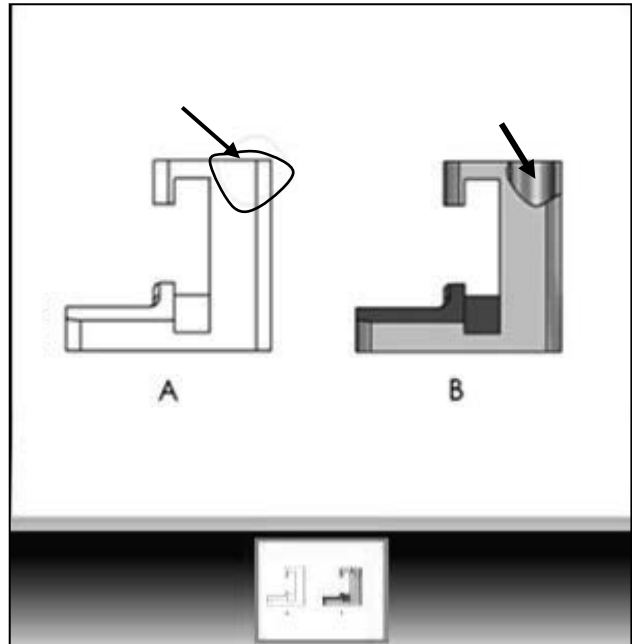
💡 Zoom in on the part or view if needed.

**Alternative Position View:**

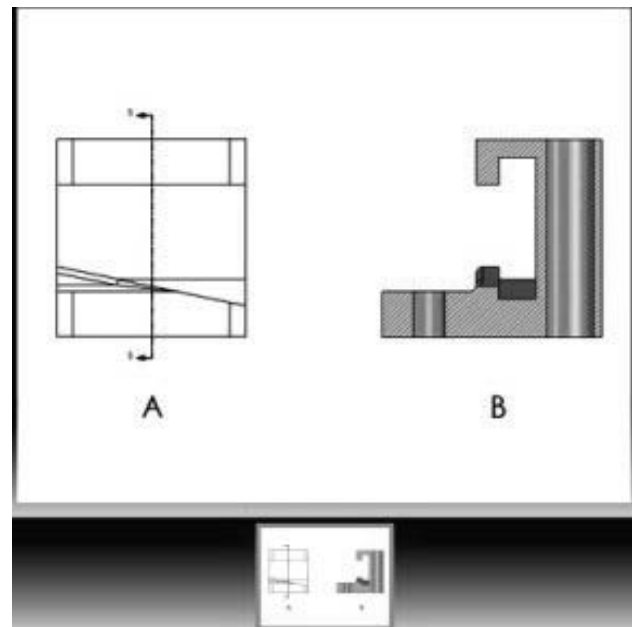


💡 Screen shots from an older CSWA exam for the *Drafting Competencies* category. Read each question carefully. Use SOLIDWORKS help if needed.

**Broken out Section View:**



**Section View:**



## **CSWA Introduction and Drafting Competencies**

**Notes:**