

Official Guide to Certified **SOLIDWORKS® Associate Exams:** CSWA, CSWA-SD, CSWA-S, CSWA-AM

SOLIDWORKS® 2019 - 2021

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CHAPTER 2 - CSWA

INTRODUCTION AND DRAFTING

COMPETENCIES

Chapter Objective

Provide a basic introduction into the curriculum and categories of the Certified SOLIDWORKS Associate (CSWA) exam. Awareness of the exam procedure, process, and required model knowledge. The five exam categories are Drafting Competencies, Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advanced Part Creation and Modification, and Assembly Creation and Modification.

Introduction

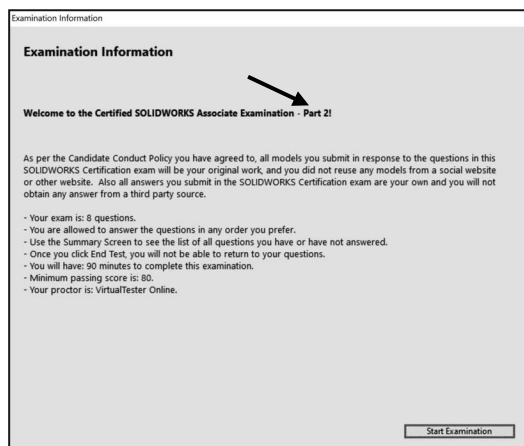
The CSWA Academic exam is provided either in a single 3-hour segment, or 2 - 90-minute segments.

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.



CSWA Part 1:

Basic Part Creation and Modification, Intermediate Part Creation and Modification

There are **two questions** on the CSWA Academic exam in the *Basic Part Creation and Modification* category and **two questions** in the *Intermediate Part Creation and Modification* category.

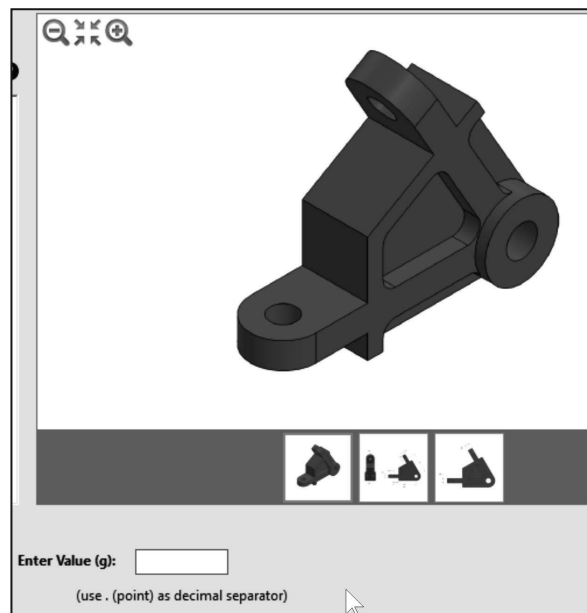
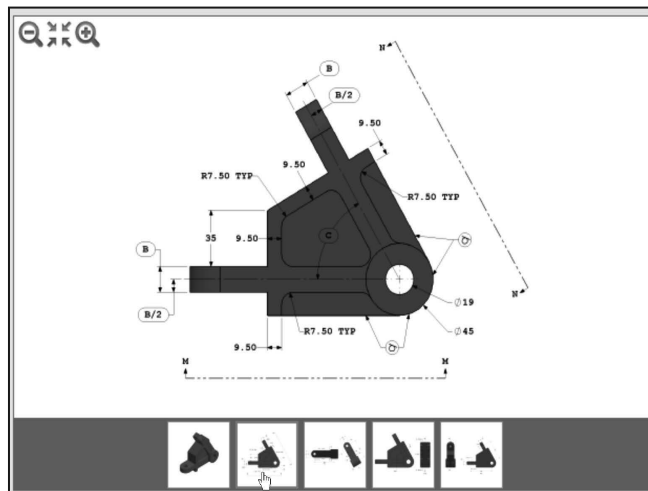
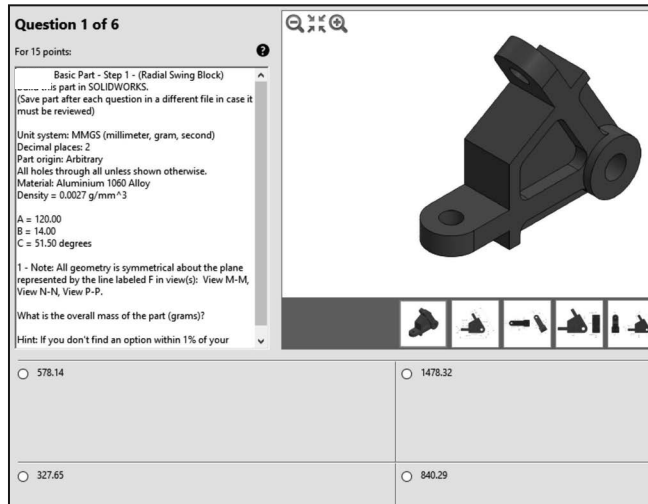
The first question is a multiple-choice single answer format. You should be within 1% of the multiple-choice answer before you move on to the modification single answer section (fill in the blank format).

Each question is worth fifteen (15) points for a total of thirty (30) points.

You are required to build a model with six or more features and to answer a question either on the overall mass, volume, or the location of the Center of mass for the created model relative to the default part Origin location.

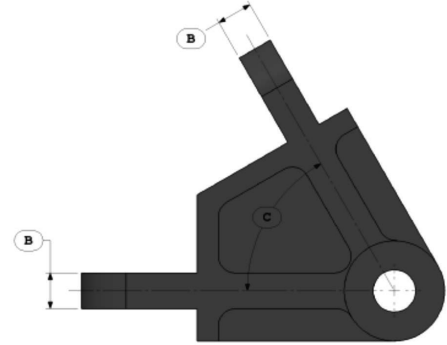
You are then requested to modify the part and answer a fill in the blank format question.

- *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
 - Basic Sketch tools



Screen shot from an exam

- Extruded Boss/Base feature
- Extruded Cut feature
- Revolved Boss/Base feature
- Fillet and Chamfer feature
- Apply Material
- Modification of basic part
- Mass Properties tool



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, face, plane, etc.
 - Sketch Tools:
 - Line, Circle, Rectangle, Offset Entities, Convert Entities, etc.
 - 2D Sketching:
 - Geometric relations and dimensioning.
 - Extruded Boss/Base feature
 - Extruded Cut feature
 - Revolved Boss/Base feature
 - Mirror, Chamfer, and Fillet feature
 - Circular and Linear Pattern feature
 - Plane feature
 - Mass Properties and Measure tool
 - Show/Hide Axes
 - Modification of Intermediate part

You should be within 1% of the multiple-choice answer before you go on to the modification single answer section.

Question 3 of 6

For 15 points: ?

Intermediate Part - Step 1 - (Tank)

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
Density = 0.0027 g/mm³

A = 375.00
B = 412.50

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).

Question 4 of 6

For 15 points: ?

Intermediate Part - Step 2 - (Tank)

Modify the part in SOLIDWORKS by adding top handle and bottom platform.

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

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

2 - The top platform has 3 cutouts the exact same size and spaced 90 degrees from each other.

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

Screen shots from an exam

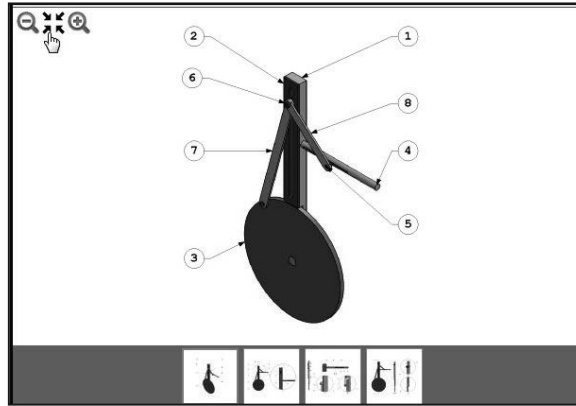


If you do not pass the certification exam (either segment), you will need to wait 30 days until you can retake that segment of the exam.

Assembly Creation and Modification

There are four questions on the CSWA Academic exam (2 questions in Part 1, 2 questions in Part 2) in the Assembly Creation and Modification category: (2) different assemblies - (4) questions - (2) multiple choice / (2) single answer - 30 points each.

- *Assembly Creation and Modification:* (Two different assemblies - four questions - two multiple choice/two single answers - 30 points each).
 - Download Assembly zip file
 - Save zip file to a local hard drive
 - Un-zip components
 - Create a new Assembly document
 - Set document properties
 - Insert first component. Fix first component to the origin of the assembly
 - Insert all needed components
 - Insert needed mates: Coincident, Concentric, Perpendicular, Parallel, Tangent, Distance, Angle, Advanced Distance, Advanced Angle, and Aligned, Anti-Aligned option
 - Apply Mass Properties and Measure tool
 - Show/Hide Axes
 - Create a new Coordinate System
 - Modify the assembly



Question 5 of 6

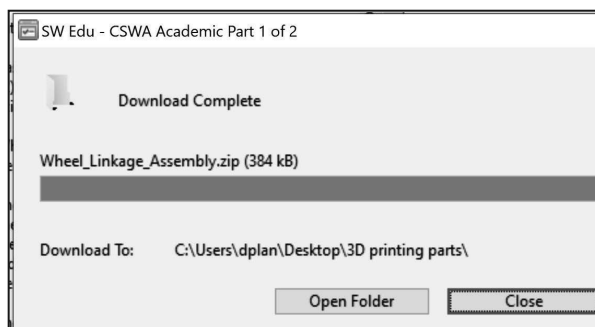
For 30 points: ?

-Create the assembly using the following conditions:

1. Base (1) is mated concentric to the 4 pins of the Rail Lid (2) where the inside face (pins side) of Rail Lid (2) is coincident with the top face (slot side) of Base (1).
2. Wheel (3) is mated concentric and flush to the end of the pin on the Base (1) as shown in Detail H.
3. The larger diameter cylindrical face of Cylinder Connector (6) is mated tangent to the face of the slot on the Base (1) as shown in Detail B. The bottom of Cylinder Connector (6) is mated to the bottom flat face of the slot in Base (1). Refer to Detail G.
4. The pin end of Piston Cylinder (4) is mated concentric and coincident to the side hole in the Base (1) as shown in Section E-E. The straight face of the slot

Attachment to this question

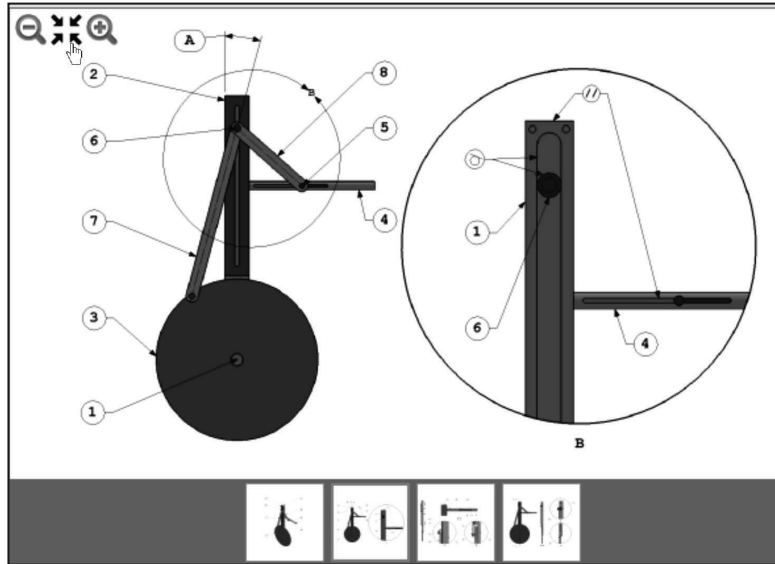
Wheel_Linkage_Assembly.zip (384.3 kB)



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



In the Assembly Creation and Modification category (Part 1 and Part 2), expect to see five to eight 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.



CSWA Part 2: Introduction and Drafting Competencies

There are **three questions** on the CSWA Academic 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.

Examination Information

Examination Information

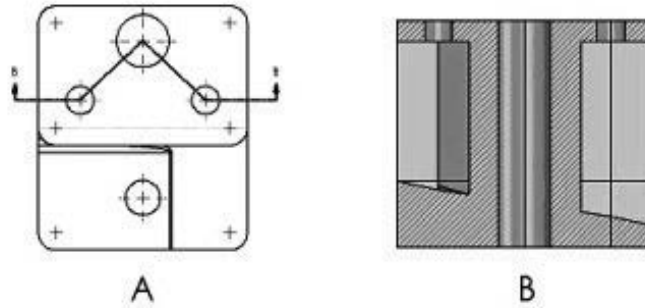
Welcome to the Certified SOLIDWORKS Associate Examination - Part 2!

As per the Candidate Conduct Policy you have agreed to, all models you submit in response to the questions in this SOLIDWORKS Certification exam will be your original work, and you did not reuse any models from a social website or other website. Also all answers you submit in the SOLIDWORKS Certification exam are your own and you will not obtain any answer from a third party source.

- Your exam is: 8 questions.
- You are allowed to answer the questions in any order you prefer.
- Use the Summary Screen to see the list of all questions you have or have not answered.
- Once you click End Test, you will not be able to return to your questions.
- You will have: 90 minutes to complete this examination.
- Minimum passing score is: 80.
- Your proctor is: VirtualTester Online.

CSWA Introduction and Drafting Competencies

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.



The questions are on general drawing views: Projected, Section, Break, Crop, Detail, Alternate Position, etc.


Advanced Part Creation and Modification

There are **three questions** on the CSWA Academic exam (Part 2) in the Advanced Part Creation and Modification category.

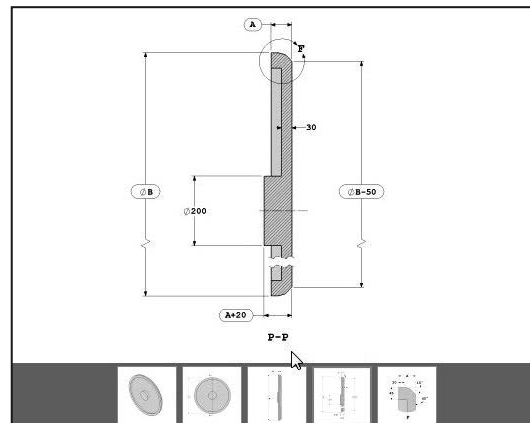
The first question is in a multiple-choice single answer format.

The other two questions (Modification of the model) are in the fill in the blank format.

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

 FeatureManager and CommandManager tabs and tree folders will vary depending on system setup, version, and Add-ins.

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



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)

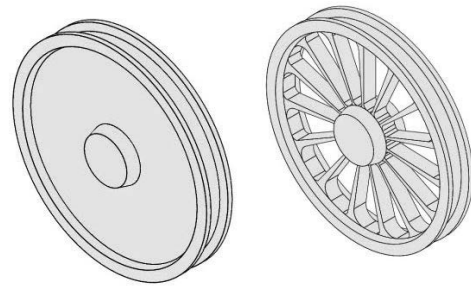
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: AISI 1020 Steel
Density = 0.0079 g/mm³

A = 64.00
B = 20.00
C = 26.50

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

Screen shots from the exam

- *Advanced Part Creation and Modification:*
(Three questions - one multiple choice/two single answers - 15 points each).



- Sketch Planes:
 - Front, Top, Right, face, plane, etc.
 - 2D Sketching or 3D Sketching
- Sketch Tools:
 - Line, Circle, Rectangle, Offset Entities, Convert Entities, etc.
- Extruded Boss/Base feature
- Extruded Cut feature
- Revolved Boss/Base feature
- Mirror, Chamfer, and Fillet feature
- Circular and Linear Pattern feature
- Shell feature
- Plane feature
- Mass Properties and Measure tool
- Show/Hide Axes
- More difficult geometry modifications

Advanced Part - Step 1 - (Connector)
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: Cast Stainless Steel
Density = 0.0077 g/mm³

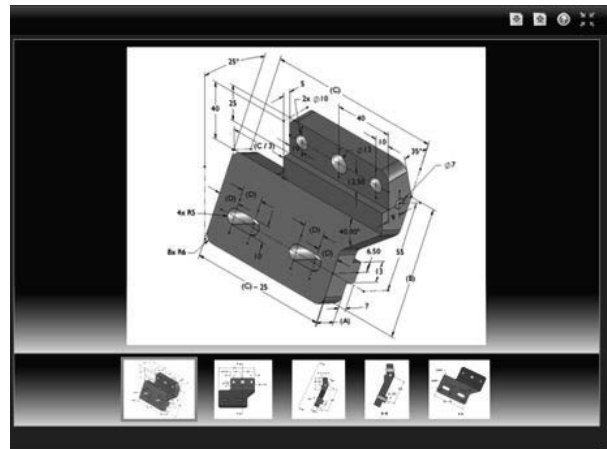
A = 104.00
B = 100.00
C = 20.00

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.



Screen shots from the exam

Each question is worth fifteen (15) points for a total of forty five (45) points.

You are required to build a model, with six or more features and to answer a question either on the overall mass, volume, or the location of the Center of mass for the created model relative to the default part Origin location. You are then requested to modify the model and answer fill in the blank format questions.

Assembly Creation and Modification

There are four questions on the CSWA Academic exam (2 questions in part 1, **2 questions** in part 2) in the Assembly Creation and Modification category: (2) different assemblies - (4) questions - (2) multiple choice / (2) single answer - 30 points each.

The first question is in a multiple-choice single answer format. You should be within 1% of the multiple-choice answer before you move on to the modification single answer section (fill in the blank format).

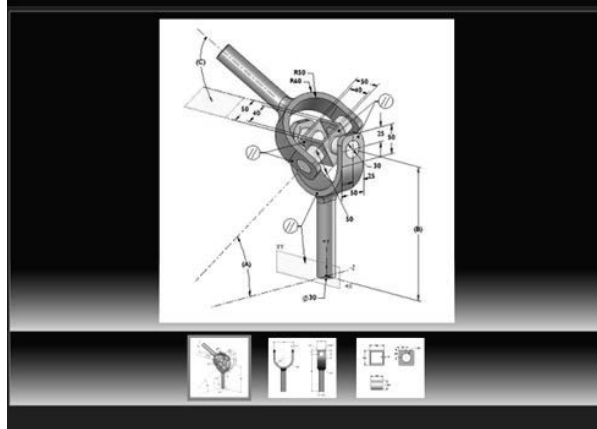
You are required to download the needed components from a provided zip file and insert them correctly to create the assembly.

In the Assembly Creation and Modification category, expect to see five to eight 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.

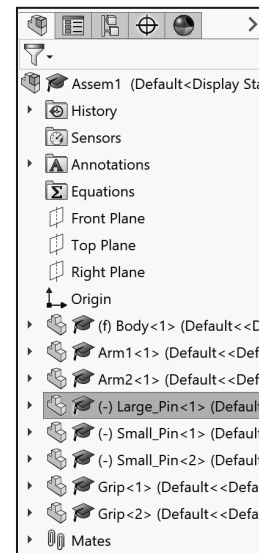
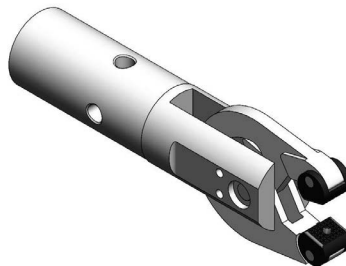
Know how to create and apply an Advance distance and angle mate. Use the Measure tool in the assembly on the components.



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



Screen shots from the exam



Question 7 of 7

For 30 points:

Modify this assembly in SolidWorks (Linkage Assembly)

Unit system: MMGS (millimeter, gram, second)
 Decimal places: 2
 Assembly origin: Arbitrary

-Using the same assembly created in the previous question modify the following parameters:
 A = 80.00

What is the center of mass of the assembly (millimeters)?

Enter Coordinates: X:

Y:

Z:

(use . (point) as decimal separator)

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.

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.*

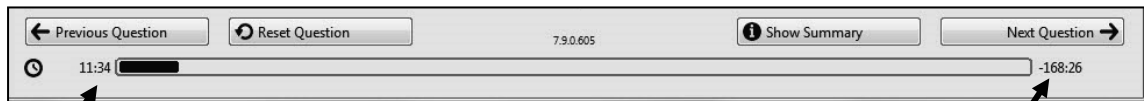
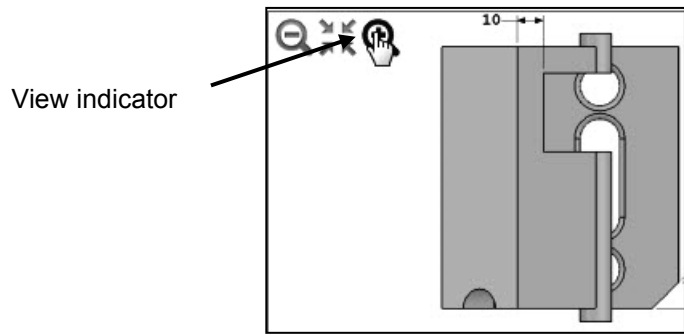
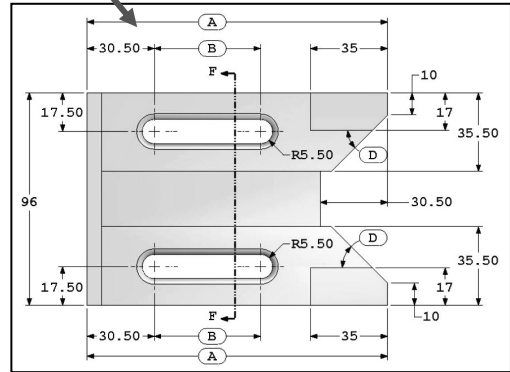
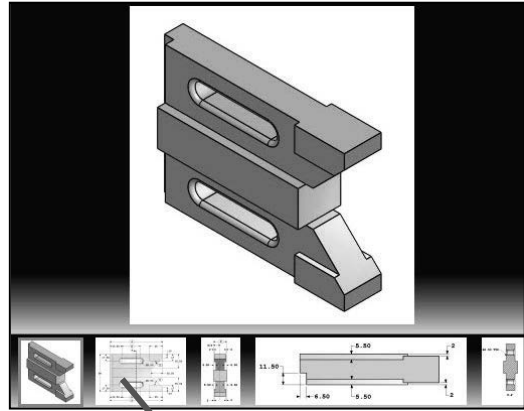
When you are finished, press the End Examination button. The tester will ask you if you want to end the test. Click Yes.

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

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.

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

 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.



Drafting Competencies

Drafting Competencies is one of the five categories 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.

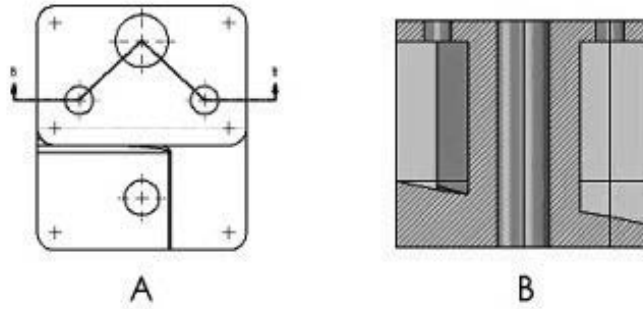
In the *Drafting Competencies* category of the exam, you are **not required** to create or perform an analysis on a part, assembly, or drawing but you are required to have general drafting/drawing knowledge and understanding of various drawing view methods.

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

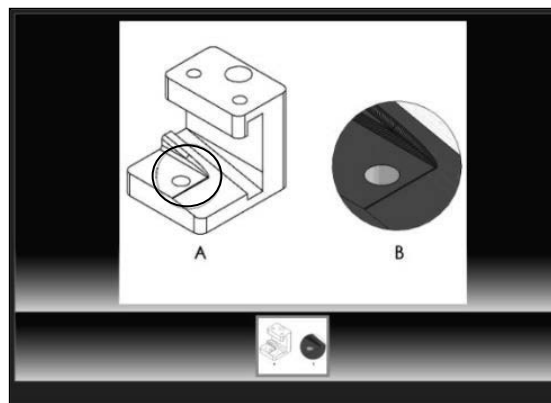
- Identify the procedure to create a named drawing view: Projected view, Section view, Break view, Crop view, Detail, Alternate Position view, etc.



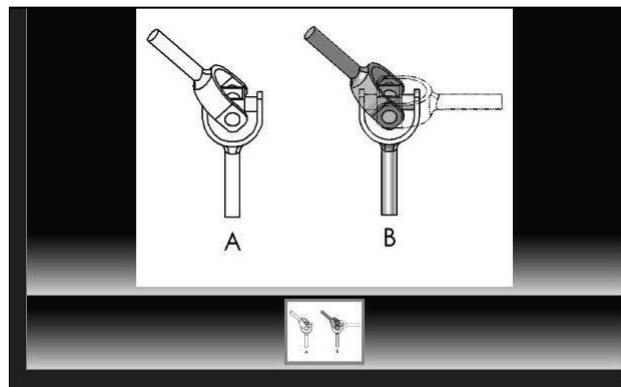
In the *Basic Part Creation and Modification*, *Intermediate Part Creation and Modification*, *Advanced Part Creation and Modification* and *Assembly Creation and Modification* categories, you are required to read and interpret various types of drawing views and understand various types of drawing annotations.



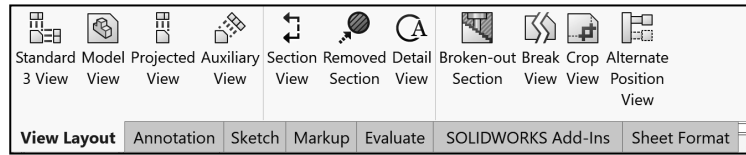
Drafting Competencies - To create drawing view 'B' it is necessary to select drawing view 'A' and insert which SolidWorks view type?



Screen shot from the exam



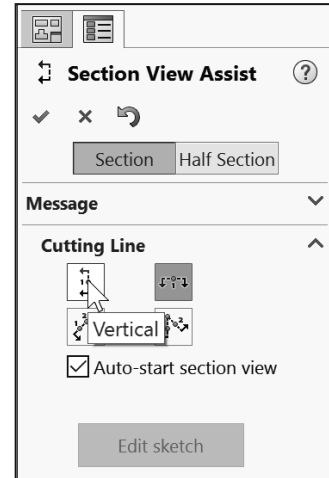
Procedure to Create a Named Drawing view



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.*

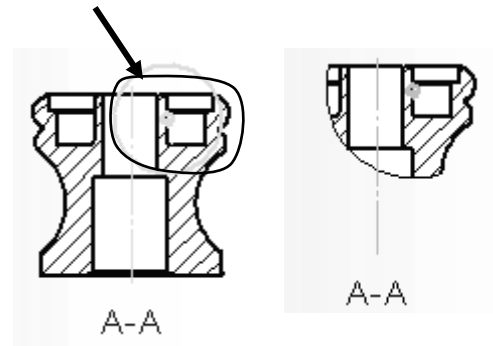
Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

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.

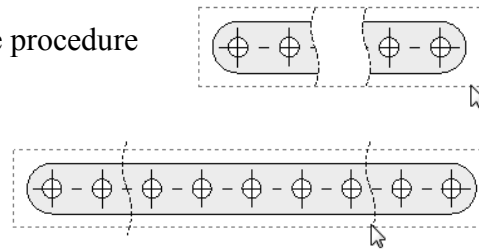


Create a Detail view in a drawing to display a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, a non-planar (isometric) view, a Section view, a Crop view, an Exploded assembly view, or another Detail view.

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.

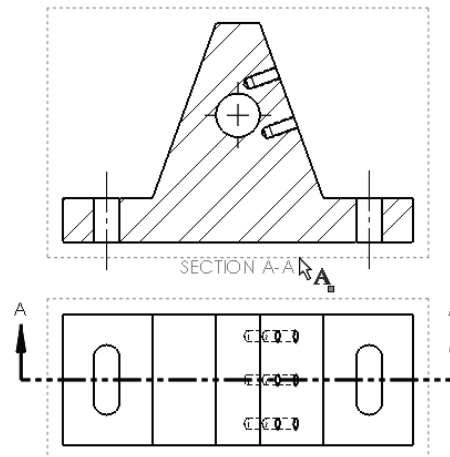


In views with multiple breaks, the Break line style must be the same.

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.



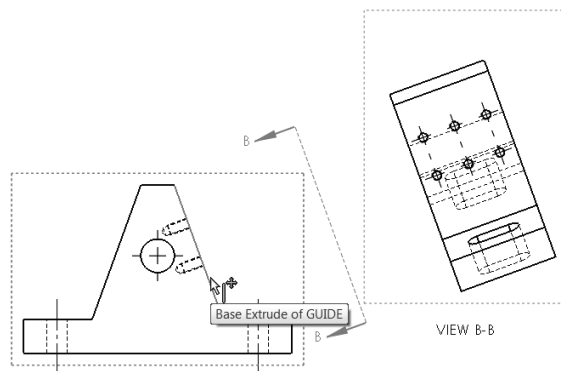
Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

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.

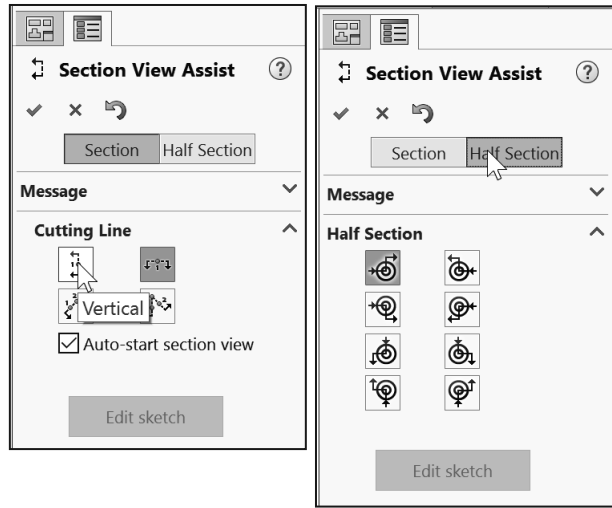
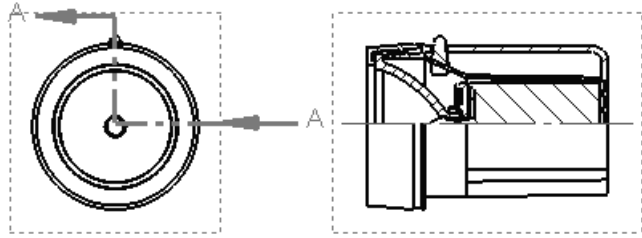
An Auxiliary view is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view.



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.

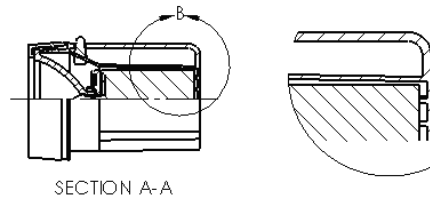



The Aligned Section view is very similar to a Section View, with the exception that the section line for an aligned half section is comprised of two or more lines connected at an angle.

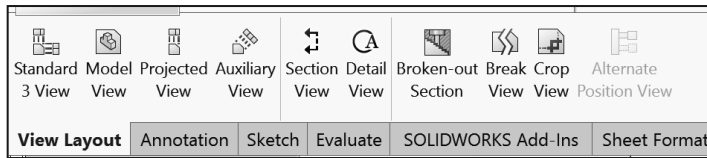
Tutorial: Drawing Named Procedure 2-5


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.

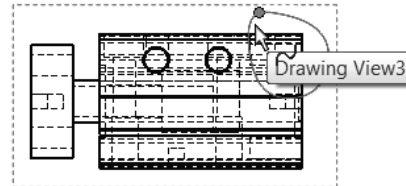
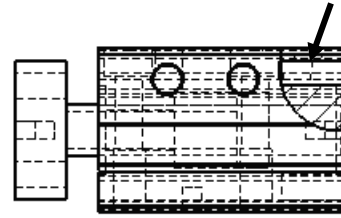


 To create a profile other than a circle, sketch the profile before clicking the Detail view tool. Using a sketch entity tool, create a closed profile around the area to be detailed.

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.

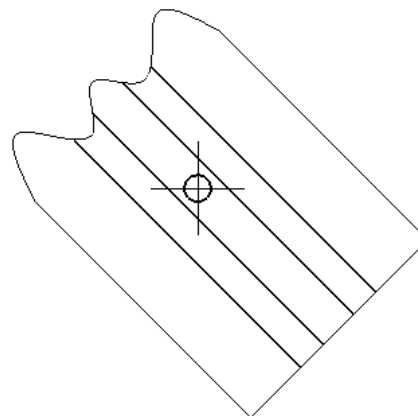
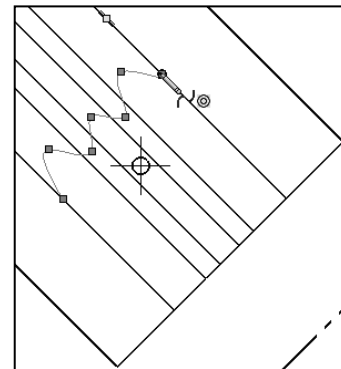
1. **View** the illustrated drawing view. The top drawing view is a Crop view. The Crop view is created by a closed sketch profile such as a circle, or spline as illustrated.

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.

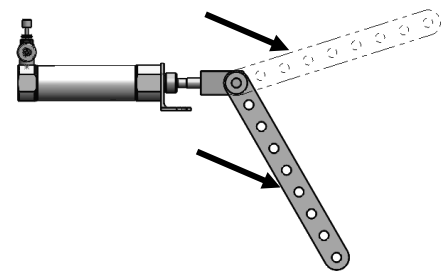
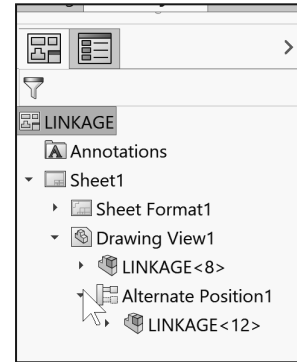


Tutorial: Drawing Named Procedure 2-8

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.



Summary

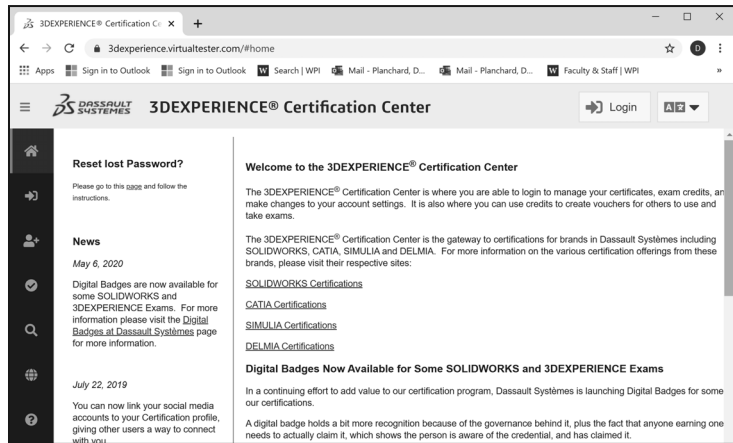
Drafting Competencies is one of the five categories on the CSWA exam. The CSWA Academic exam is provided either in a single 3-hour segment, or 2 - 90-minute segments.

The CSWA exam for industry is provided only 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.

 To obtain additional CSWA exam information, visit <https://3dexperience.virtualtester.com/#home>





CSWA Introduction and Drafting Competencies

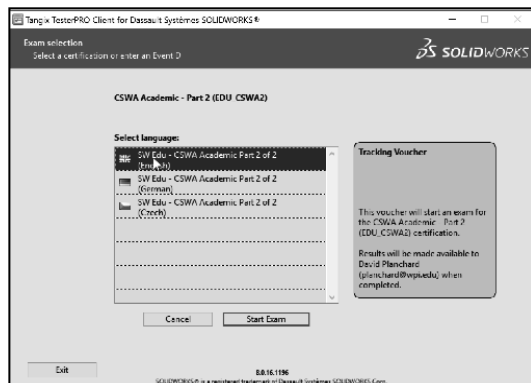
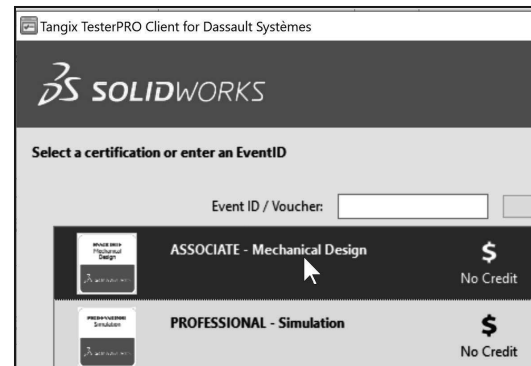
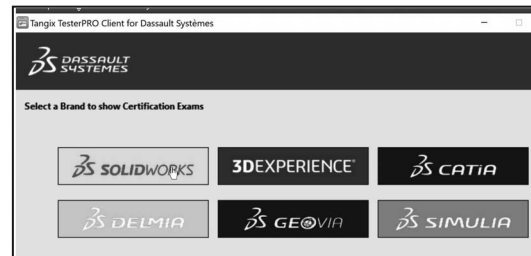
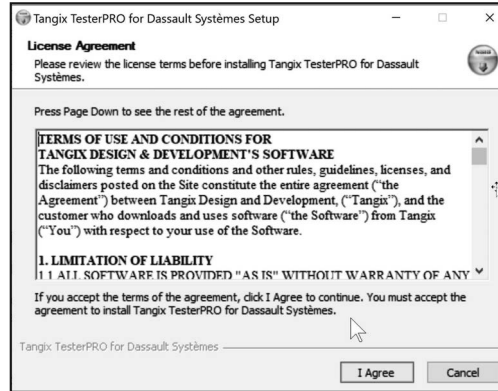
Spend no more than 10 minutes on each question in this category. This is a timed exam. Manage your time.

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 models that could be provided on the exam.

 Screen shots in the book were made using SOLIDWORKS 2019 SP4, SOLIDWORKS 2020 SP4 and SOLIDWORKS 2021 SP0 Windows® 10.

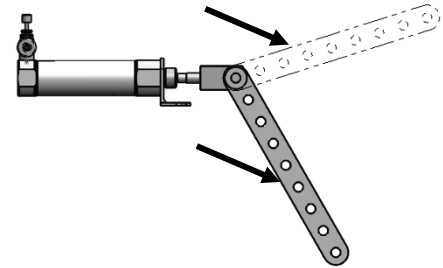
 FeatureManager and CommandManager tabs and tree folders will vary depending on system setup, version and Add-ins.



Questions

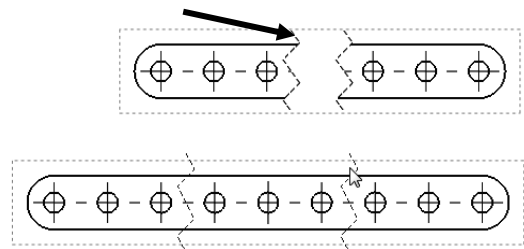
1. Identify the illustrated Drawing view.

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



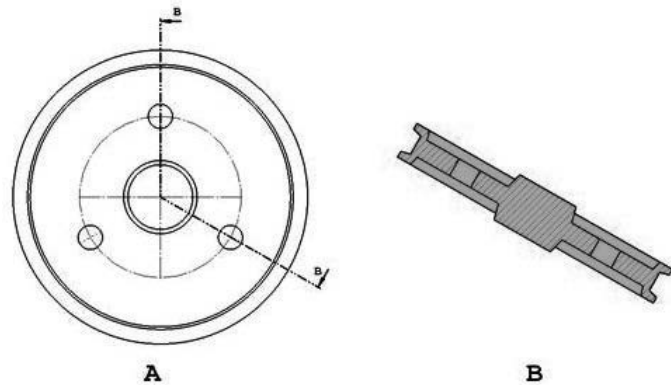
2. Identify the illustrated Drawing view.

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



3. Identify the illustrated Drawing view.

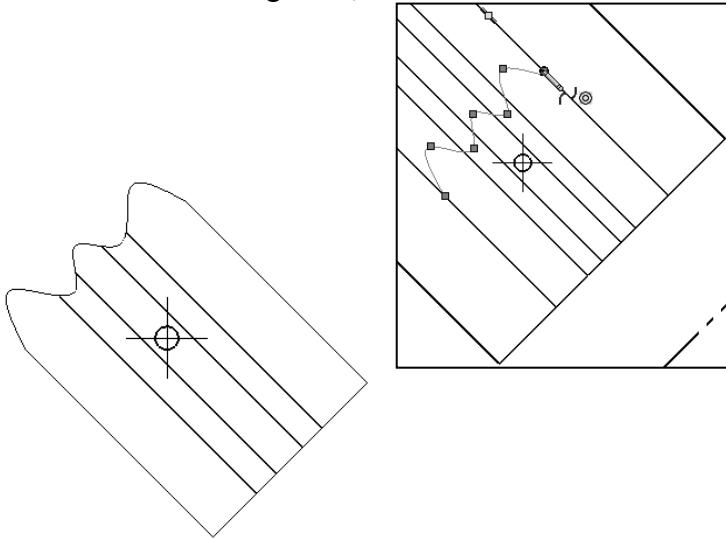
- A: Section
- B: Crop
- C: Broken-out Section
- D: Aligned



CSWA Introduction and Drafting Competencies

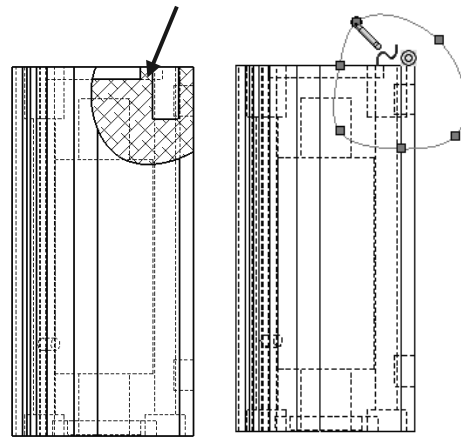
4. Identify the view procedure. To create the following view, you need to insert a:

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



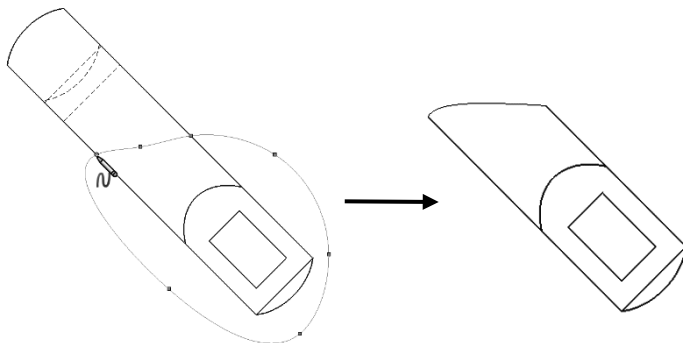
5. Identify the view procedure. To create the following view, you need to insert a:

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



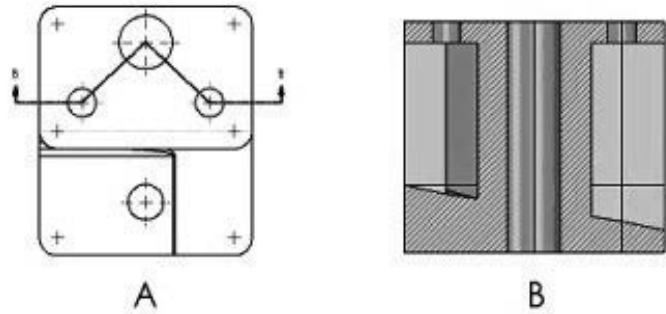
6. Identify the illustrated view type.

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



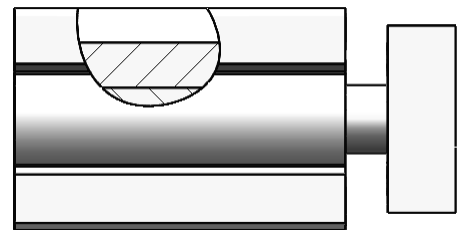
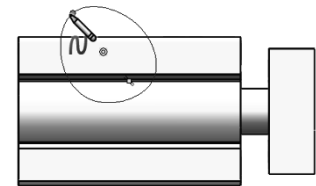
7. To create View B from Drawing View A insert which View type?

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



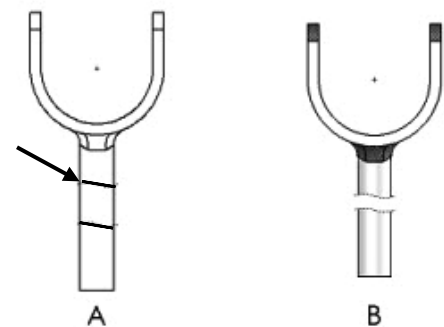
8. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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




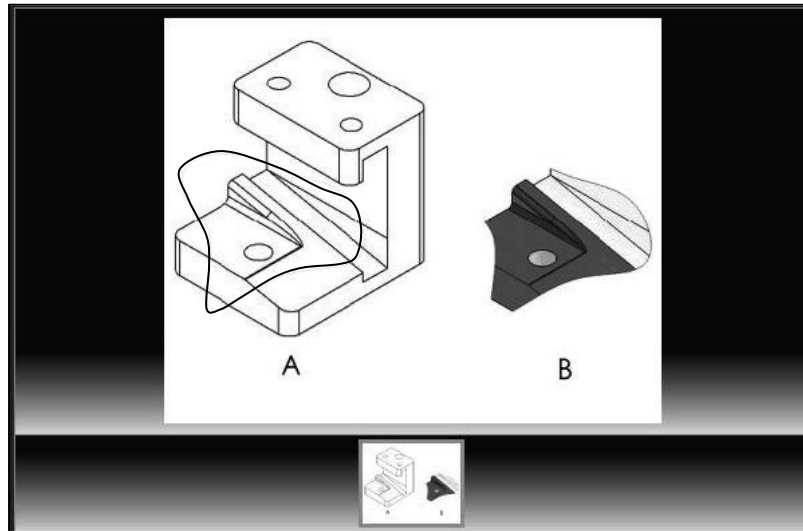
9. To create View B it is necessary to sketch a closed spline on View A and insert which View type?


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



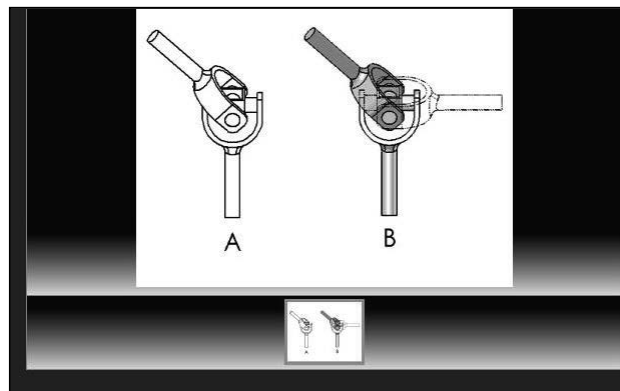
CSWA Introduction and Drafting Competencies


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



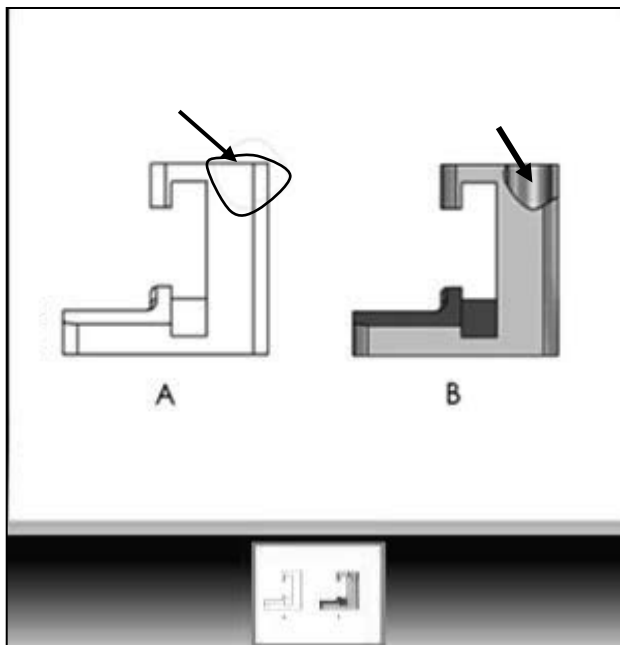
 Zoom in on the part or view if needed.

Alternative Position View:

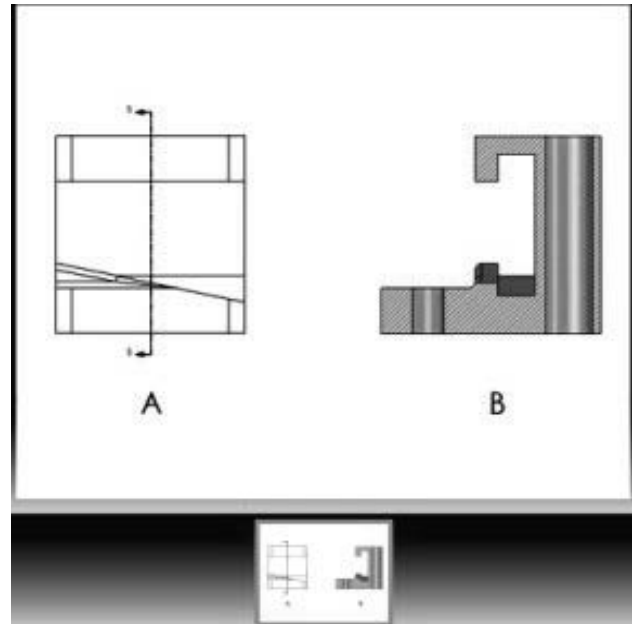


 Screen shots from an older CSWA exam for the *Drafting Competencies* category. Read each question carefully. Use SOLIDWORKS help if needed.

Broken out Section View:



Section View:



CSWA Introduction and Drafting Competencies

Notes: