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Official Guide to Certified **SOLIDWORKS® Associate Exams:** CSWA, CSWA-SD, CSWA-S, CSWA-AM SOLIDWORKS® 2020 - 2023



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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.

Segment 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.

Segment 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.



Taking the CSWA Exam (Segment 1 or 2)

Go to the 3DEXPERIENCE® Certification Center at <https://3dexperience.virtualtester.com/#home>.

The 3DEXPERIENCE® Certification Center is where you are able to log in to manage your certificates, take certifications, and make changes to your account settings.

If your school is an academic certification provider, your instructor can allocate a free CSWA Segment 1 and Segment 2 exam credit.

The instructor will require your .edu email address.

Download the TesterPRO Client.

Un-zip the Tester PRO Client.

Read the License Agreement. Agree to the Candidate Conduct Policy. Click I Agree.

Install the Tester PRO Client. Click install. Click Finish.

Select Test Language. Click Continue.

Create an account. If you already have an account, select “I already have a VirtualTester UserID and password”.

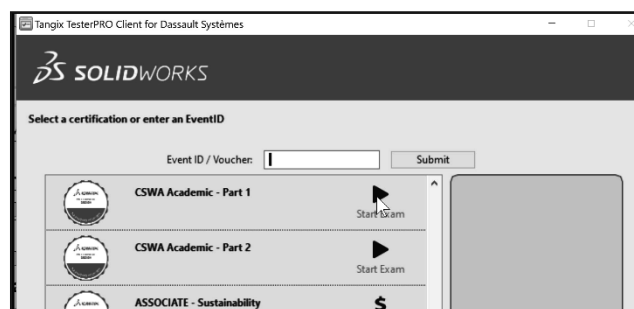
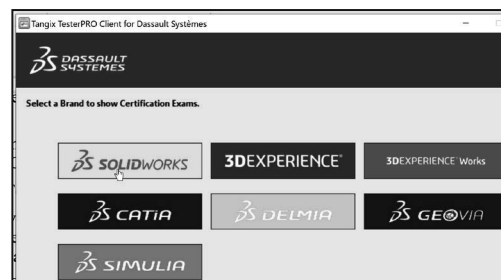
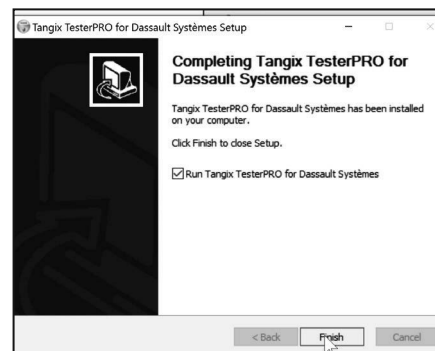
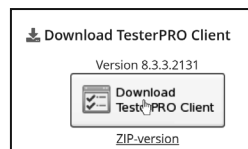
Log in. Click Continue.

Click SOLIDWORKS.

Select either CSWA Academic - Part 1 or CSWA Academic - Part 2.


This will depend on the exam credit that your instructor provided. Click Start Exam. Select language. Click Start Examination.

Read the instructions. Agree to the Confidentiality Agreement and Candidate Conduct Policy. Click Start Examination.



Examination Information

Examination Information



Welcome to the Certified SOLIDWORKS Associate Examination - Part 1!

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: 6 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.

[Start Examination](#)

Segment 1: 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.

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

Segment 1 & 2: Part Creation and Modification requires knowledge of the following:

- 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

CSWA Introduction and Drafting Competencies

- Mirror, Chamfer, and Fillet feature
- Circular and Linear Pattern feature
- Plane feature
- Mass Properties and Measure tool
- Show/Hide Axes
- Part Modification
- Apply Material

Segment 1 & 2: Assembly Creation and Modification requires knowledge in the following:

- Download an 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

 [Redacted] zip (384.3 kB)



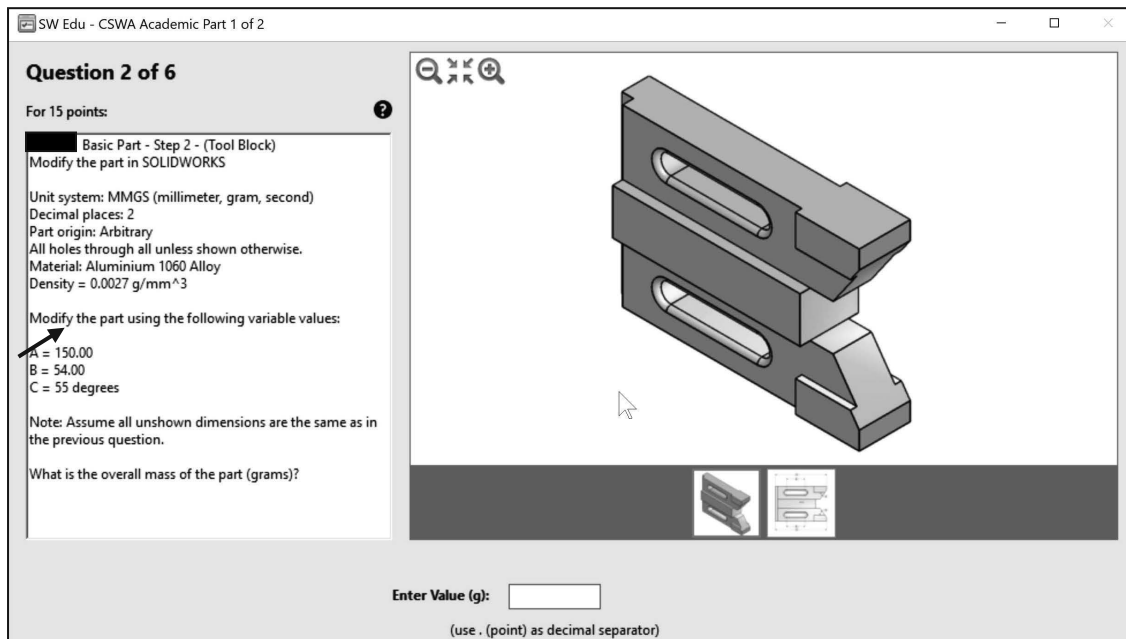
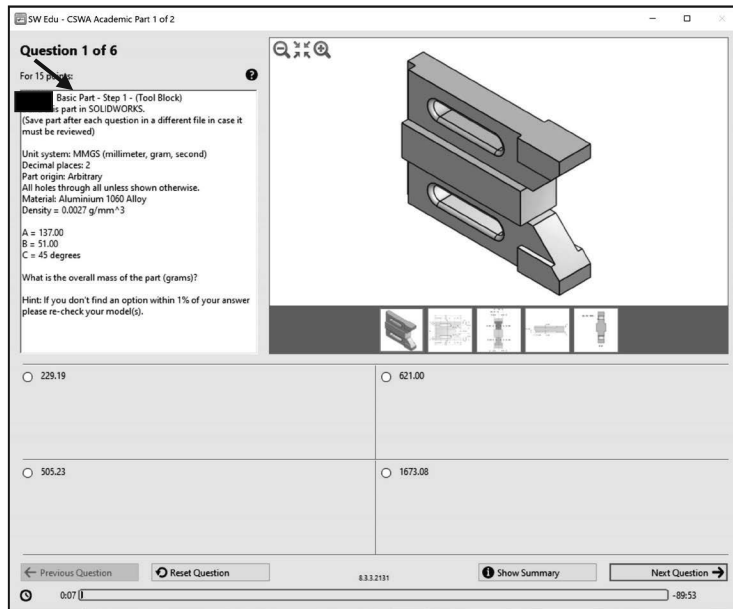
Segment 1

Question 1: Build a basic SOLIDWORKS part. Multiple-choice single answer format. Read the question. Select an answer. Click Next Question. Save all parts.

Your answer should be within 1% of the multiple-choice answer before you move on to the second question.

All answers are in the MMGS unit system. Decimal place 2.

Question 2: Modify question 1. Single answer format. Read the question. Input the answer. Click Next Question.



CSWA Introduction and Drafting Competencies

Question 3: Create a new SOLIDWORKS Intermediate part. Multiple-choice single answer format. Read the question. Select an answer. Click Next Question. Your answer should be within 1% of the multiple-choice answer before you move on to the fourth question. Part origin: Arbitrary. All holes through all unless shown otherwise. Decimal place: 2.

Question 4: Modify the SOLIDWORKS Intermediate part. Single answer format. Read the question. Input the answer. Click Next Question.

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)?

Question 5: Create a SOLIDWORKS assembly. Multiple-choice single answer format. Read the question. Download the zip file. Create an assembly per the instructions. Save the assembly for question 6. Select an answer. Click Next Question. Your answer should be within 1% of the multiple-choice answer before you move on to the final question.

SW Edu - CSWA Academic Part 1 of 2

Question 5 of 6

For 30 points: ?

of Rail Lid (2) as indicated in Detail C.

7. The opposite hole in Large Link (7) is concentric to the pin on the Wheel (3).

8. One hole in Small Link (8) is mated concentric to Cylinder Connector (6) with the bottom face of Small Link (8) coincident to the top face of Large Link (7) as indicated in Detail C.

9. The opposite hole in Small Link (7) is concentric to the protruding end of the Piston (5).

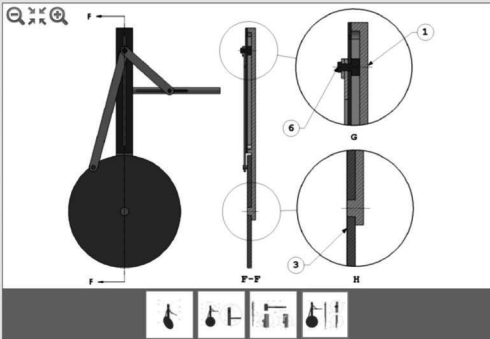
A = 6.50 degrees

What is the measured distance X (millimeters)?

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

Attachment to this question

Wheel_Linkage_Assembly.zip (384.3 kB)



17.55

35.36

38.16

15.21

Question 6: Modify the SOLIDWORKS assembly from question 5. Single answer format. Read the question. Input the answer. Click Next Question. All answers are in the MMGS unit system. Decimal place 2. This is the final question in Segment 1.

Segment 2

There are eight questions in Segment 2. Three of the questions are in the Drafting Competencies category. Each question is worth five (5) points.

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.



Download all needed model files from the SDC Publication website www.SDCpublications.com/downloads/978-1-63057-567-0.

Question 6 of 6

For 30 points: ?

E30402: Modify this assembly in SOLIDWORKS (Wheel 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 = 16.50 degrees

What is the measured distance X (millimeters)?

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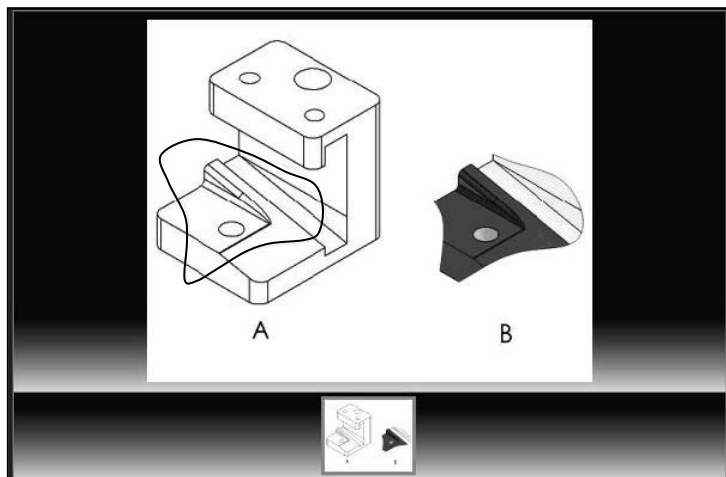
- Your exam is: 8 questions.
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- SOLIDWORKS 2020
- SOLIDWORKS 2021
- SOLIDWORKS 2022
- SOLIDWORKS 2023

Drafting Competencies

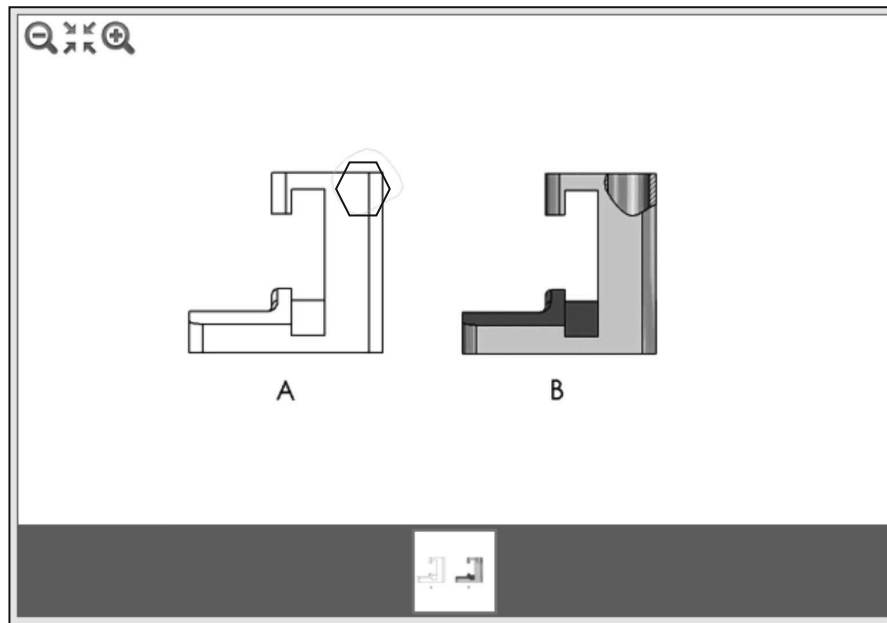
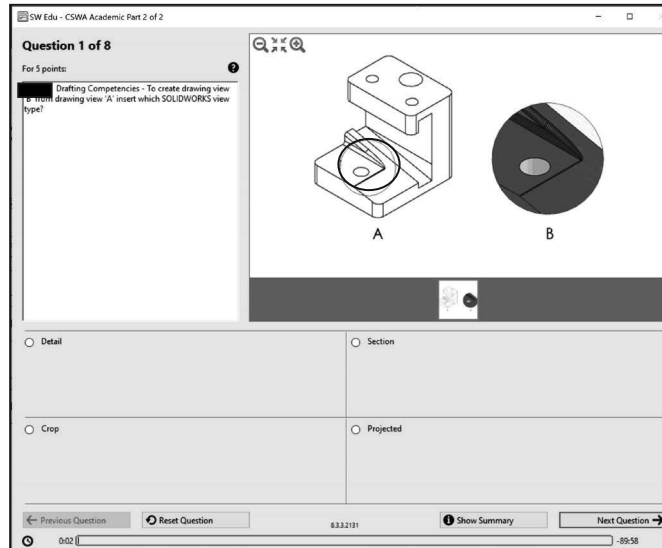
Drafting Competencies is one of the five categories on the CSWA exam.

There are three questions (total) on the CSWA exam in the Drafting Competencies category. Each question is worth five (5) points.



CSWA Introduction and Drafting Competencies

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.



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, Removed Section, 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.

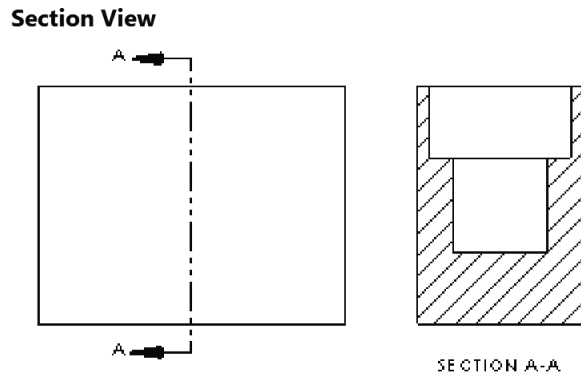
Create an Aligned Section view in a drawing through a model, or portion of a model, that is aligned with a selected section line segment. The Aligned Section view is similar to a Section view, but the section line for an aligned section comprises two or more lines connected at an angle.

Create a Crop view, sketch a closed profile such as a circle or spline. The view outside the closed profile disappears as illustrated.

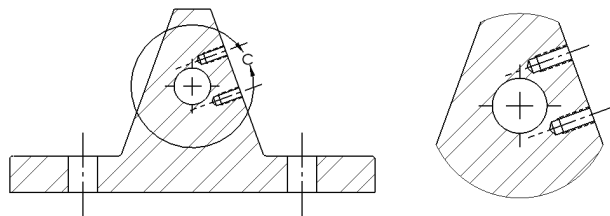
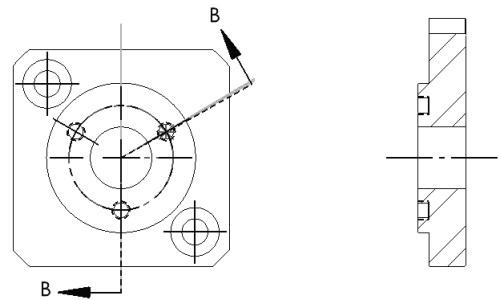
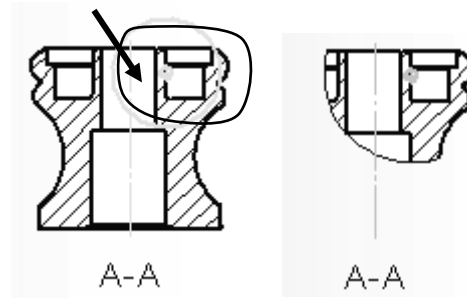


Crop any drawing view except a Detail view, a view from which a Detail view has been created, or an Exploded view.

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.



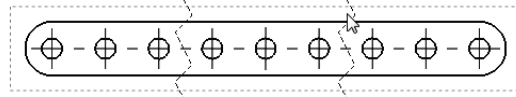
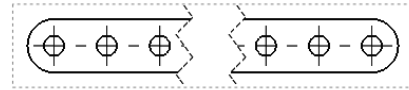
Adds a section view, aligned section view, or half section view by cutting the parent view with a section line.





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.



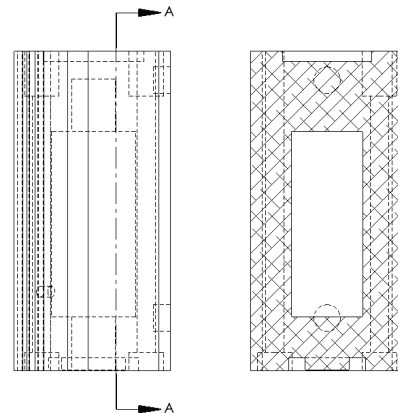
 Broken views make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.


 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 top drawing view is a Section view of the bottom view. The Section view is created by cutting the parent view with a cutting section line.

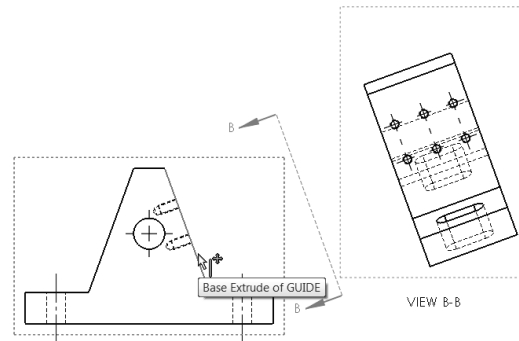



 Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

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.

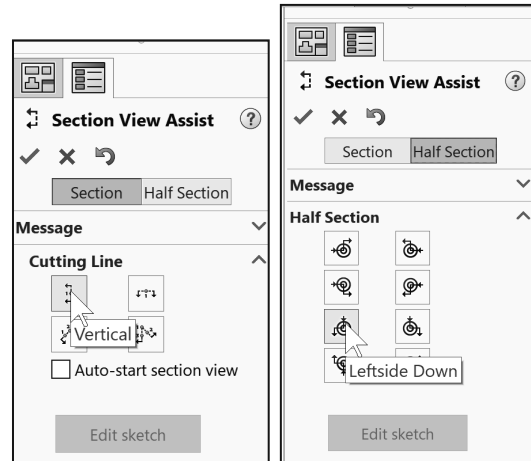
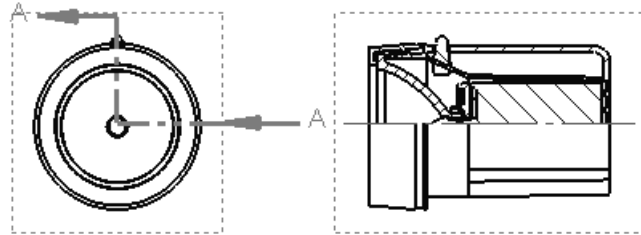



 An Auxiliary view is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view.

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.

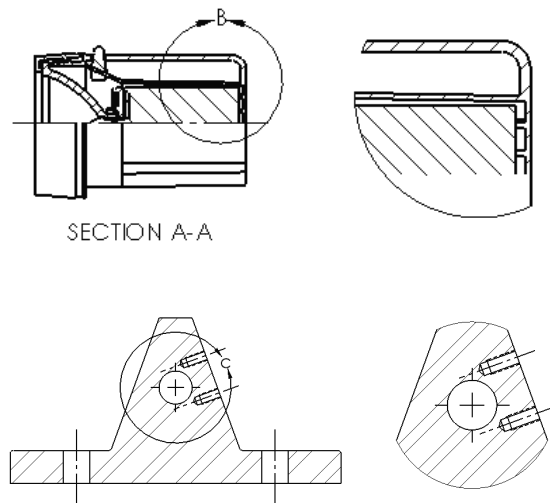



 The Aligned Section view is very similar to a Section View, with the exception that the section line for an aligned half section is comprised of two or more lines connected at an angle.


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.



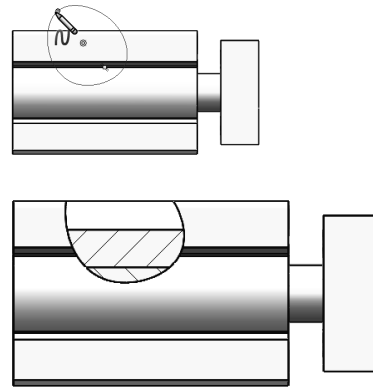
The Detail view  tool provides the ability to add a Detail view to display a portion of a view, usually at an enlarged scale.

 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 top drawing view is a Broken-out Section view of the bottom drawing 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.

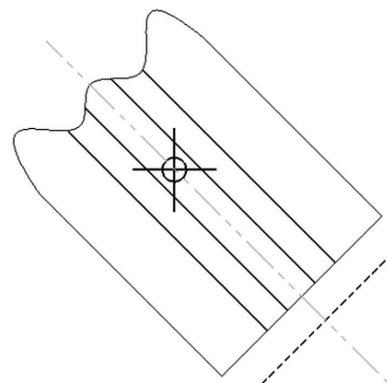
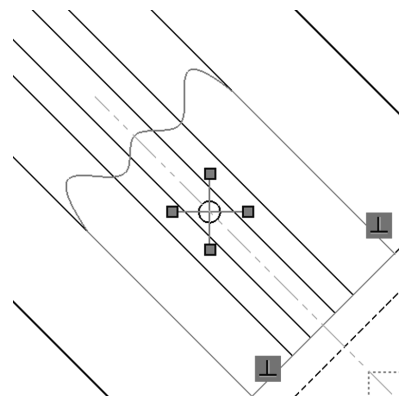
1. **View** the illustrated drawing view. The top drawing view is a Crop view. The Crop view is created by a closed sketch profile such as a circle, or spline as illustrated.

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.




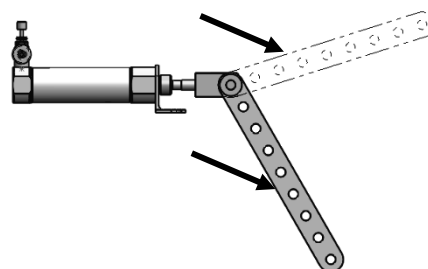
In the exam, you are allowed to answer the questions in any order. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.



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.



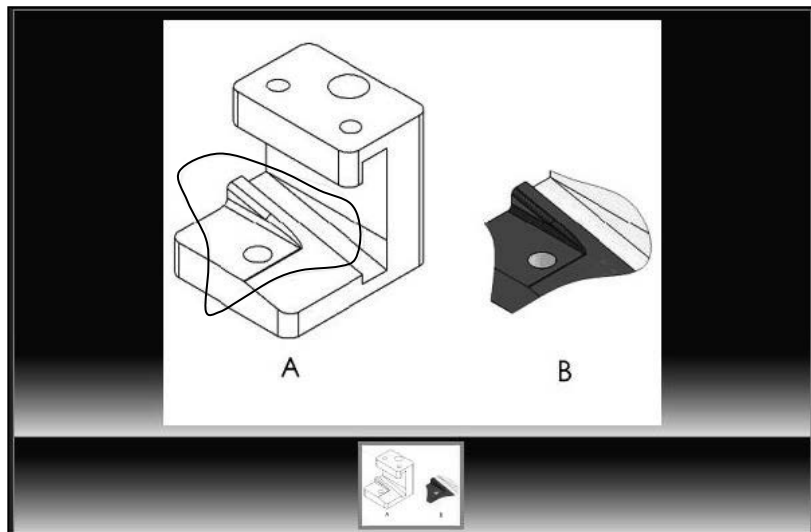
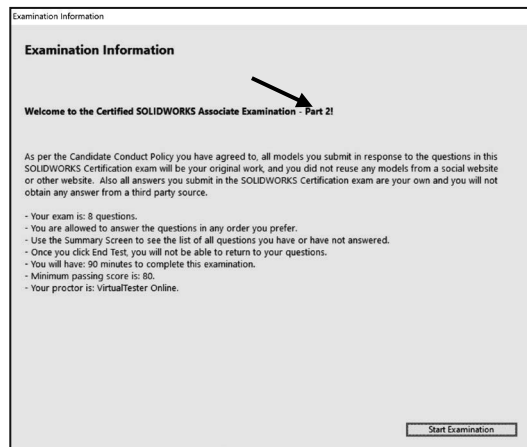
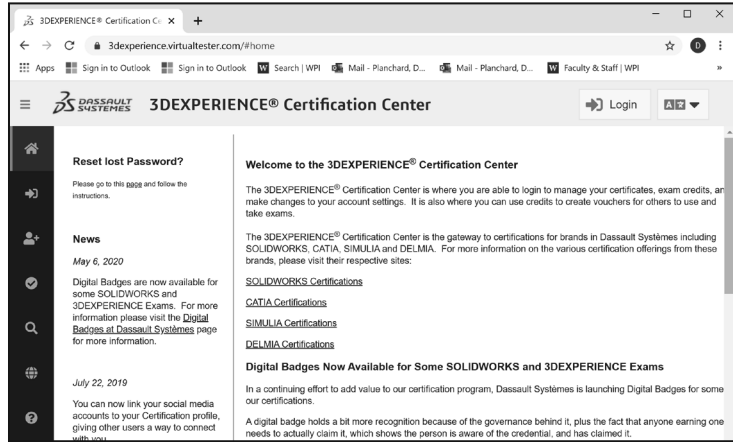
Summary

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.

Segment 2 of the CSWA Academic exam is 90 minutes, minimum passing score is 80 with 8 questions.

Three of the eight questions are in the Drafting Competencies category.

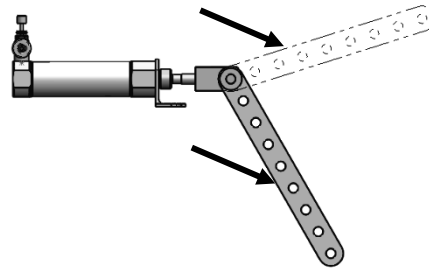
Each question is worth five (5) points. The three questions are in a multiple-choice single answer format.



Questions:

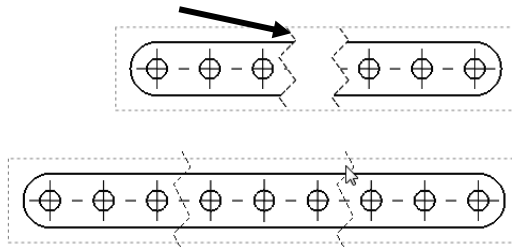
1. Identify the illustrated Drawing view.

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



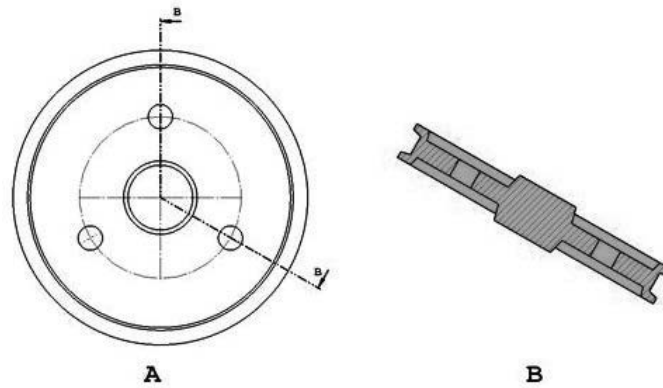
2. Identify the illustrated Drawing view.

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



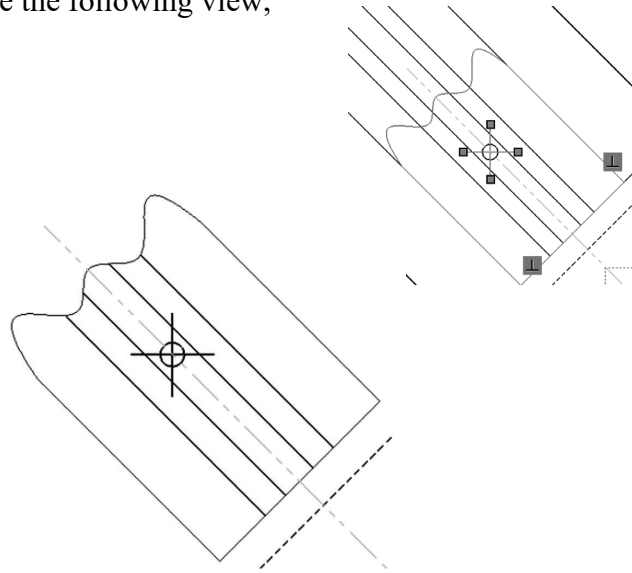
3. Identify the illustrated Drawing view.

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



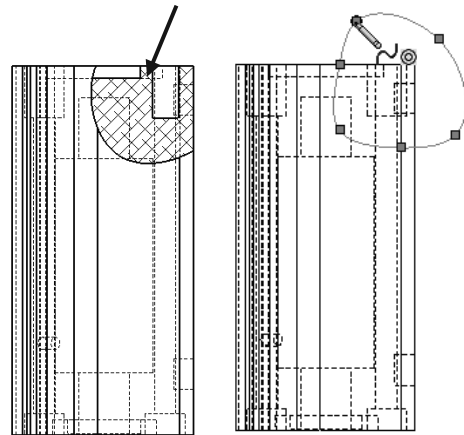
4. Identify the view procedure. To create the following view, you need to insert a:

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



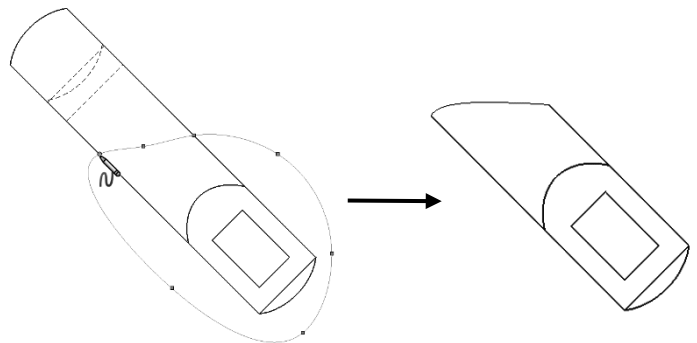
5. Identify the view procedure. To create the following view, you need to insert a:

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



6. Identify the illustrated view type.

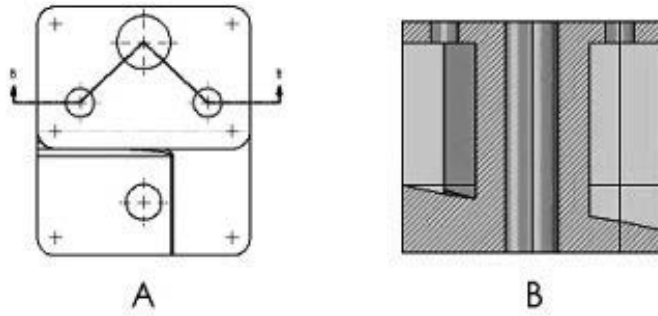
- A: Crop
- B: Section
- C: Projected
- D: Detail



CSWA Introduction and Drafting Competencies

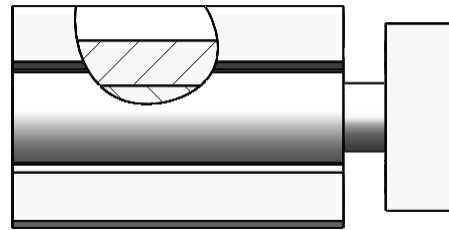
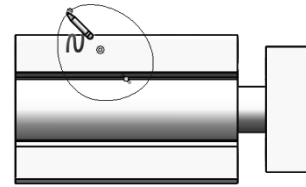
7. To create View B from Drawing View A insert which View type?

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



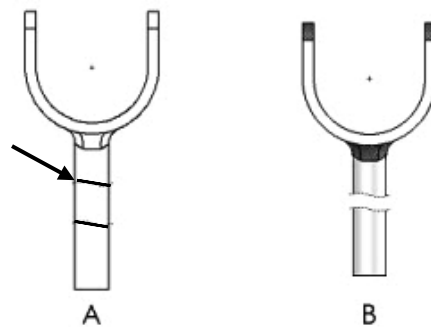
8. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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

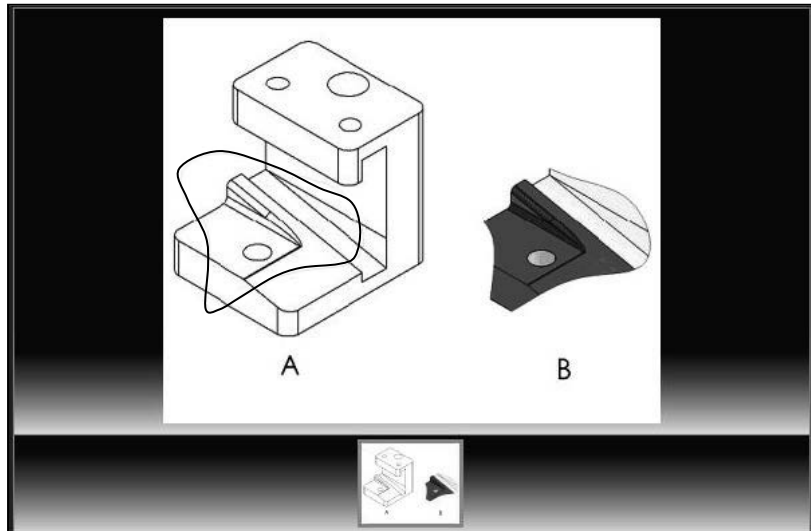


9. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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

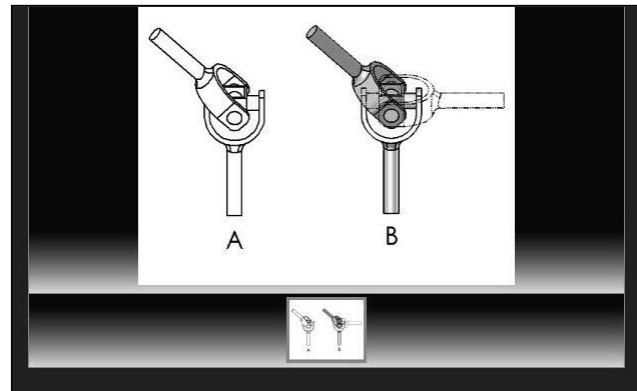


💡 Screen shots from an older CSWA exam for the Drafting Competencies category. Read each question carefully. Use SOLIDWORKS help if needed.



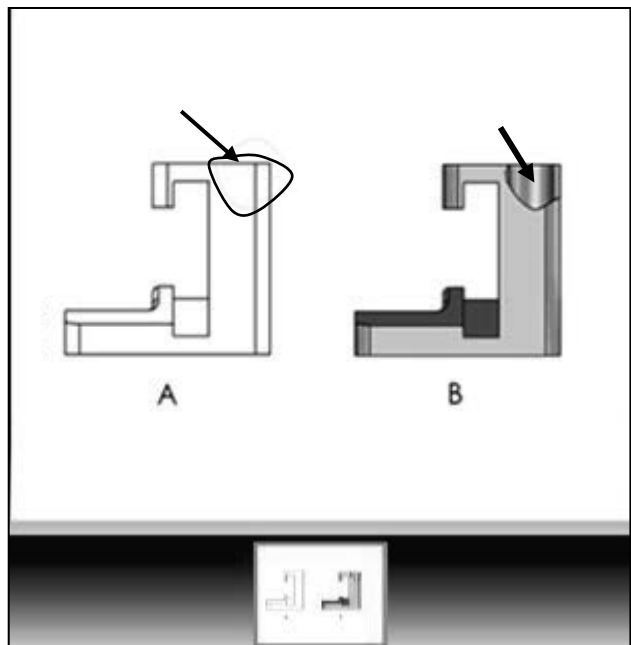
💡 Zoom in on the part or view if needed.

Alternative Position View:



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

Broken out Section View:



Section View:

