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Official Guide to Certified **SOLIDWORKS® Associate Exams:** **CSWA, CSWA-SD, CSWA-S, CSWA-AM** SOLIDWORKS® 2022 - 2025

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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.

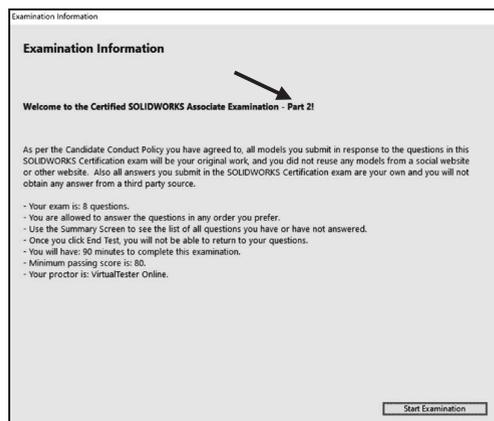
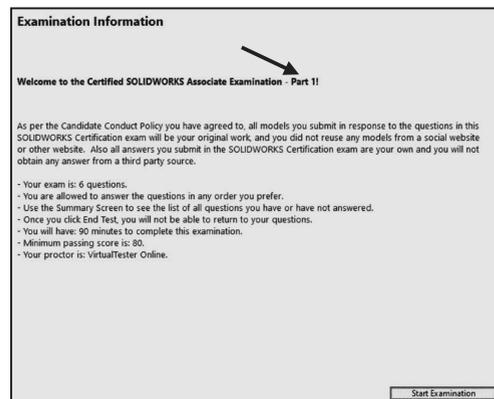
There are two questions in the Basic Part Creation and Modification category, two questions in the Intermediate Part Creation and Modification category and two questions in the Assembly Creation and Modification category.

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 3-hour Academic exam consists of 14 questions 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”.

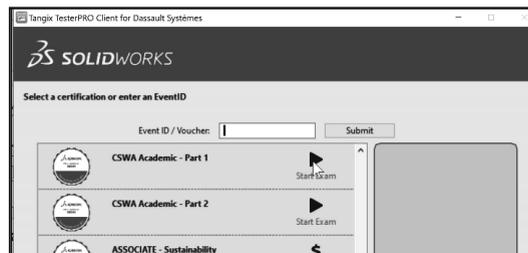
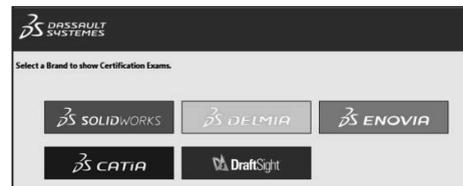
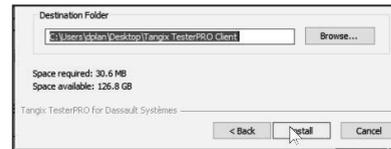
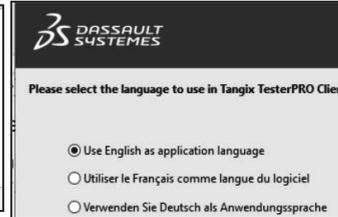
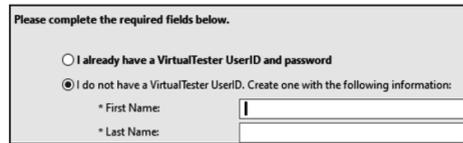
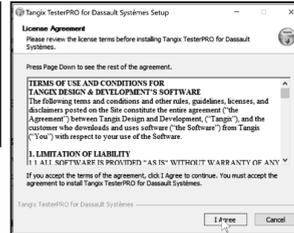
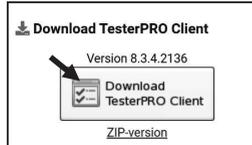
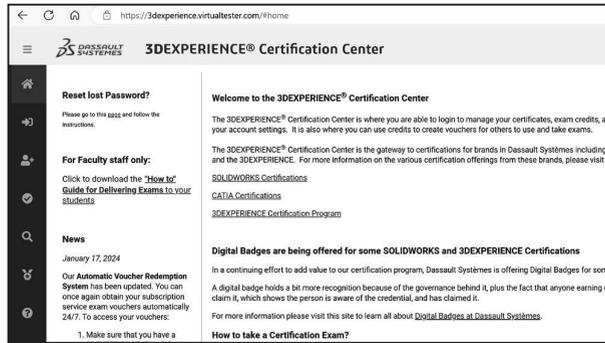
Login. Click Continue.

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.

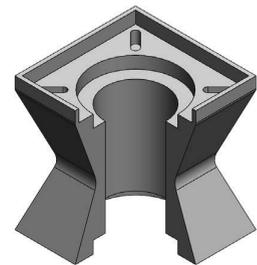
[Start Examination](#)

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 procedure:

- Read the question.
- Understand the question.
- Identify the correct Base sketch.
- Identify and apply the correct Sketch tools: Line, Circle, Rectangle, Offset Entities, Convert Entities, etc.
- Insert the correct geometric relations and or dimensions.

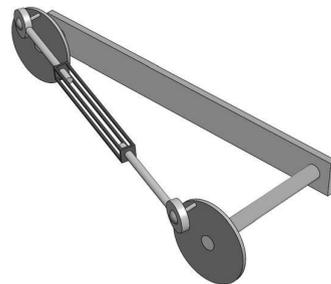
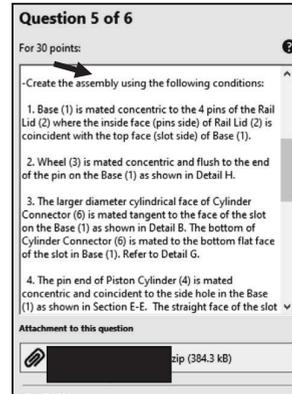


CSWA Introduction and Drafting Competencies

- Identify and apply the correct Feature tools: Extruded Boss/Base, Extruded Cut, Revolved Boss/Base, Mirror, Shell, Fillet, Circular, Linear, etc.
- Create the correct offset Plane.
- Apply the Mass Properties tool.
- Apply the Measure tool.
- Show/Hide Axes.
- Apply material.
- Perform the requested design modification.

Segment 1 & 2: Assembly Creation and Modification procedure:

- Read the question.
- Understand the question.
- Download the Assembly zip folder.
- Save the Assembly zip folder to a local hard drive.
- Un-zip (5 - 6) the components.
- Open all parts (components) from the folder.
- View the parts (components).
- Create a new SOLIDWORKS Assembly document.
- Set document properties (units and precision).
- Identify the first component.
- Insert the first component into the Assembly.
- Fix the first component to the origin of the assembly.
- Insert all needed components.
- Rotate the components if needed in the assembly.
- Insert needed mates: Coincident, Concentric, Perpendicular, Parallel, Tangent, Distance, Angle, Advanced Distance, Advanced Angle, and Aligned, Anti-Aligned option
- Apply Mass Properties.
- Apply the Measure tool (Distance/Angle).
- Create a new Coordinate System.
- Modify the assembly.



Segment 1

Question 1: Build a basic SOLIDWORKS part. Multiple-choice single answer format. Read the question. Understand the question.

View the stated units and precision. View the Hint.

Build the part. 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.

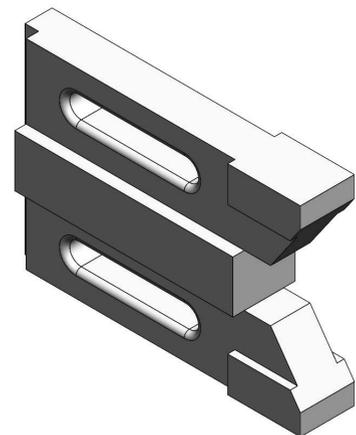
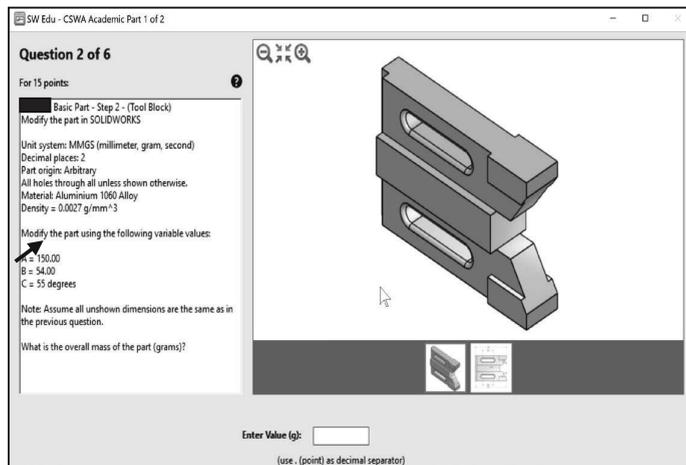
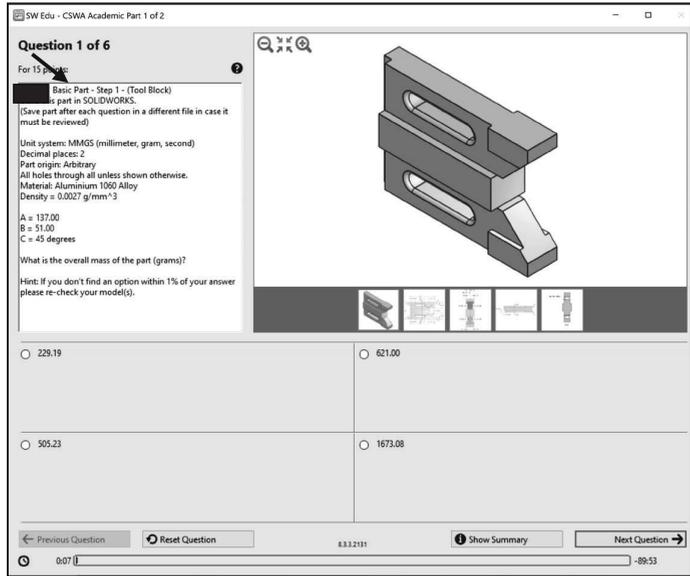
Question 2: Modify question 1. Single answer format.

Read the question. Understand the question.

Does your answer make sense?

Input the answer.

Click Next Question. **Note:** Review the Basic and Intermediate chapter in this book for additional details and exam exercises.



CSWA Introduction and Drafting Competencies

Question 3: Create a new SOLIDWORKS Intermediate part. Multiple-choice single answer format. Read the question. Understand the question. Build the part. 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. **Note:** Review the Basic and Intermediate chapter in this book for additional details and exam exercises examples.

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. Understand the question. Download the zip folder. Un-zip the components.

Identify the first component. Fix the first component to the assembly. 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. **Note:** Review the assembly chapter in this book for additional details and exam exercises.

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).

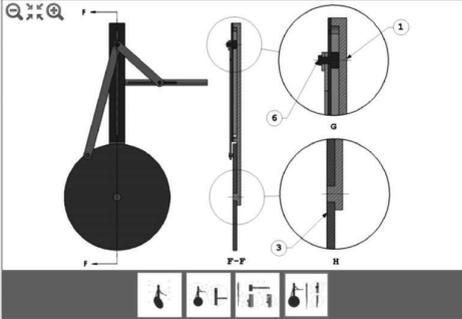
A = 6.50 degrees

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)



17.35 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.

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.

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)?

Examination Information

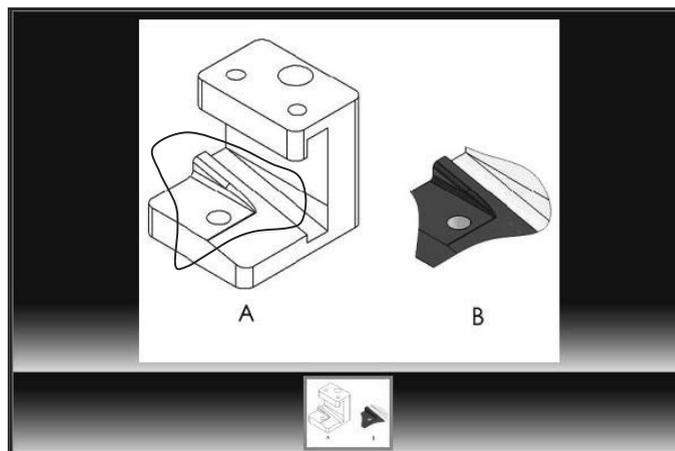
Examination Information

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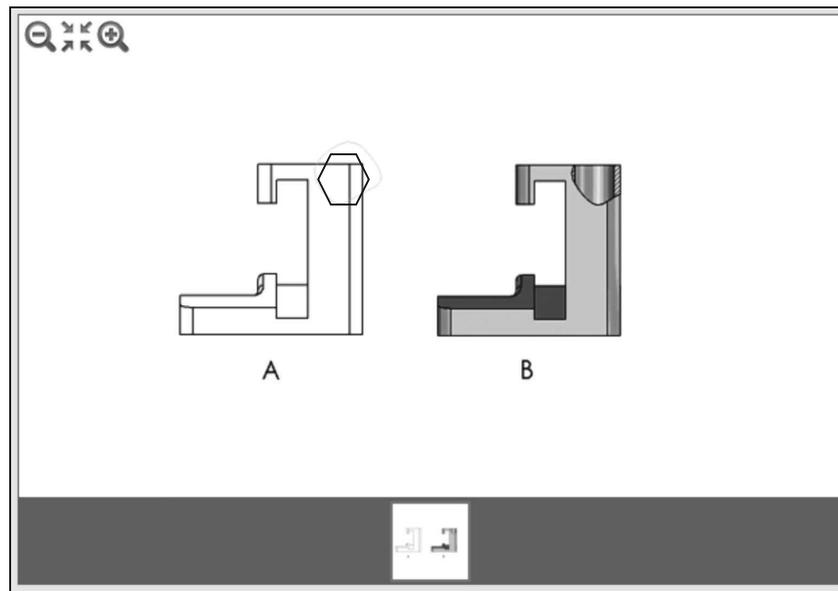
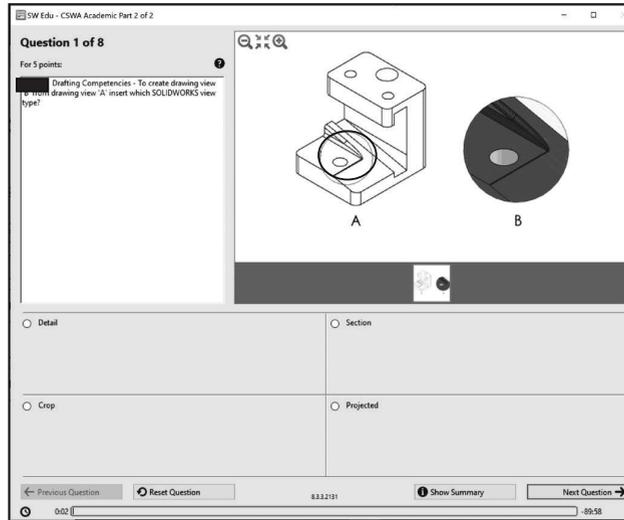
- Your exam is: 8 questions.
- You are allowed to answer the questions in any order you prefer.
- Use the Summary Screen to see the list of all questions you have or have not answered.
- Once you click End Test, you will not be able to return to your questions.
- You will have: 90 minutes to complete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

Start Examination



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.

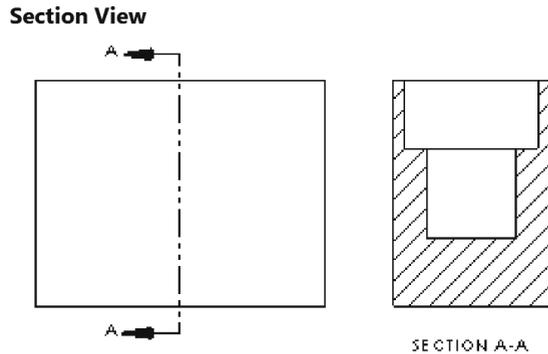
Create a Section view in a drawing by cutting the parent view with a section line. The Section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

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.

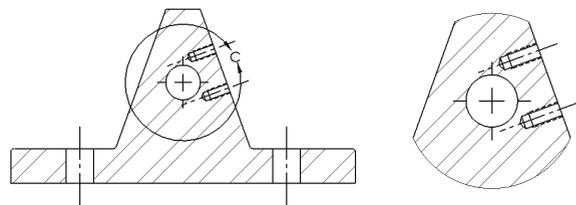
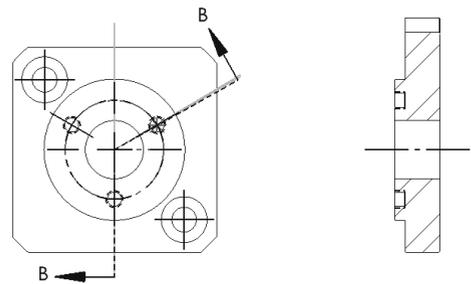
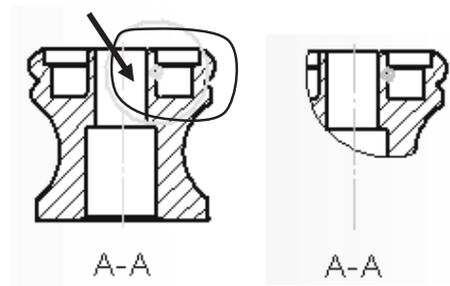
Create a Crop view, sketch a closed profile such as a circle or spline. The view outside the closed profile disappears as illustrated.

 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.



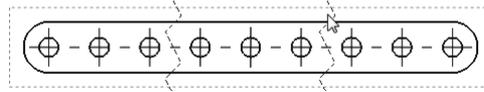
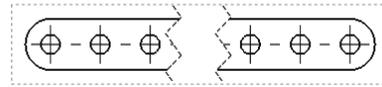
Adds a section view, aligned section view, or half section view by cutting the parent view with a section line.



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.



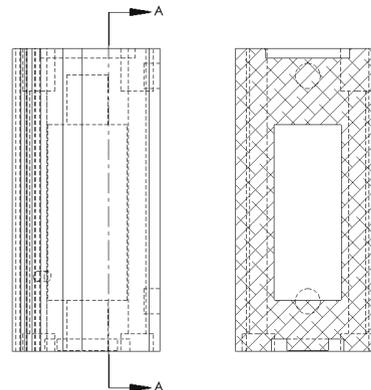
 Broken views make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.

 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.

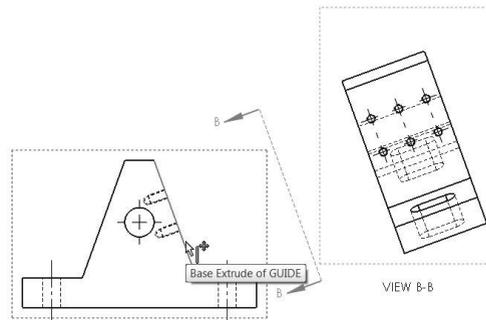


 Create a Section view in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include Concentric arcs.

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.

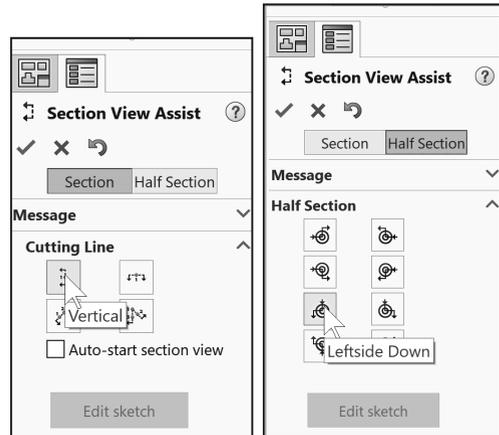
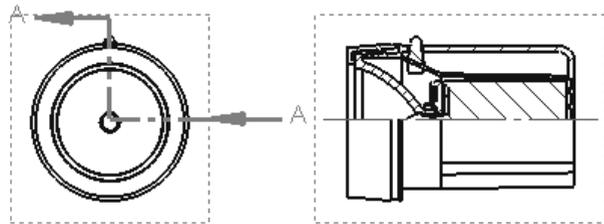


 An Auxiliary view is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view.

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.

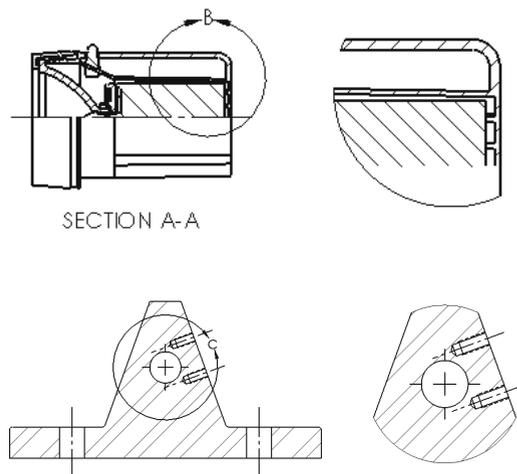


 The Aligned Section view is very similar to a Section View, with the exception that the section line for an aligned half section is comprised of two or more lines connected at an angle.

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.



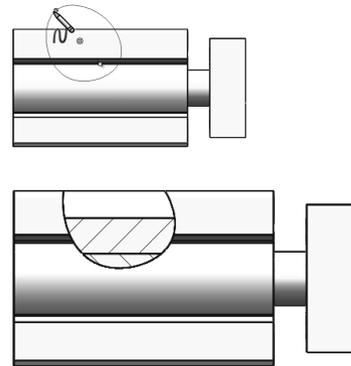
The Detail view  tool provides the ability to add a Detail view to display a portion of a view, usually at an enlarged scale.

 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 crop the name view.

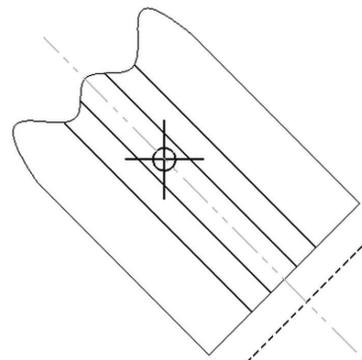
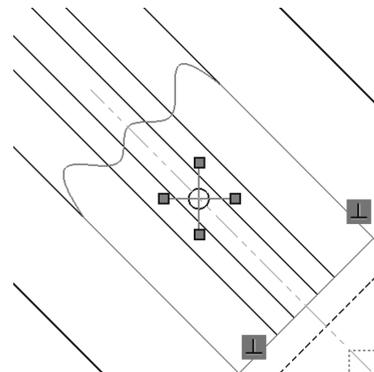
1. **View** the illustrated drawing view. The top drawing view is a Crop view. The Crop view is created by a closed sketch profile such as a circle, or spline as illustrated.

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.



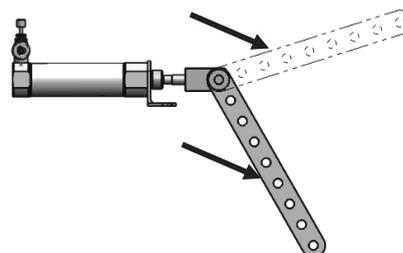
In the exam, you are allowed to answer the questions in any order. Use the Summary Screen during the exam to view the list of all questions you have or have not answered.



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.



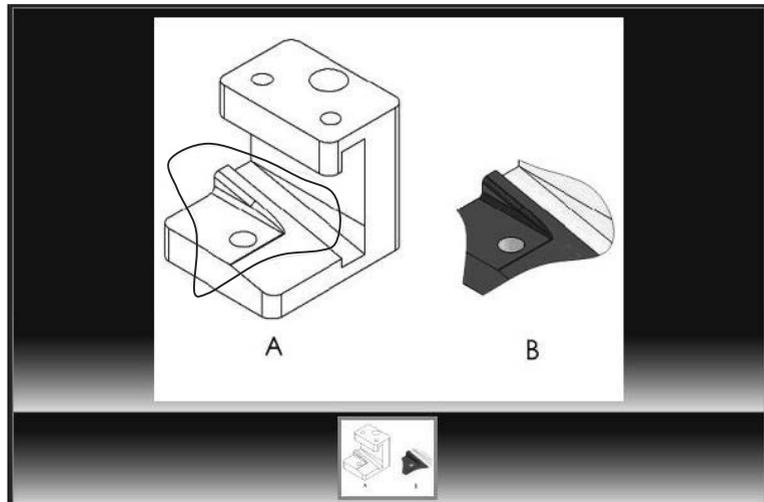
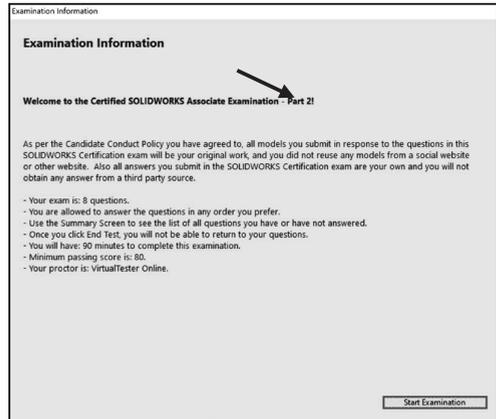
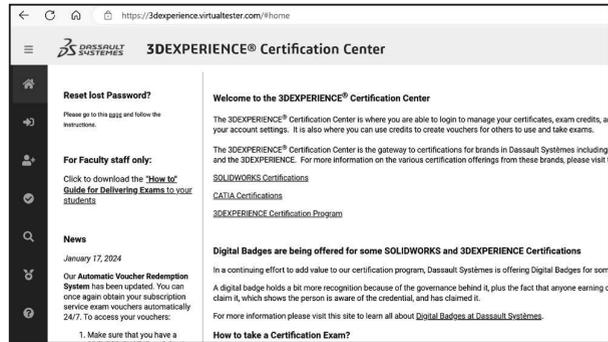
Summary

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.

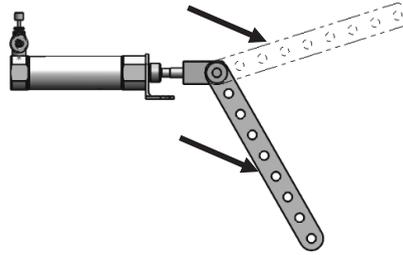
Each question is worth five (5) points. The three questions are in a multiple-choice single answer format.



Questions:

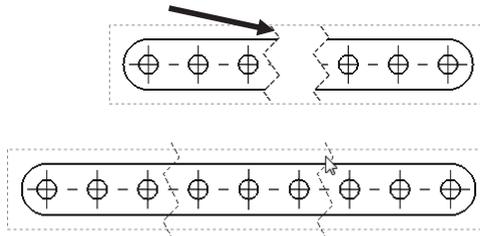
1. Identify the illustrated Drawing view.

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



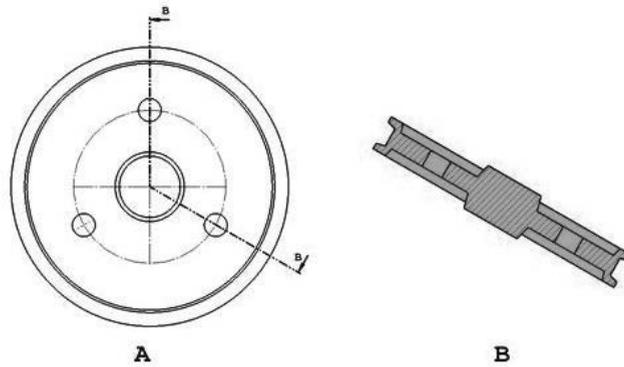
2. Identify the illustrated Drawing view.

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



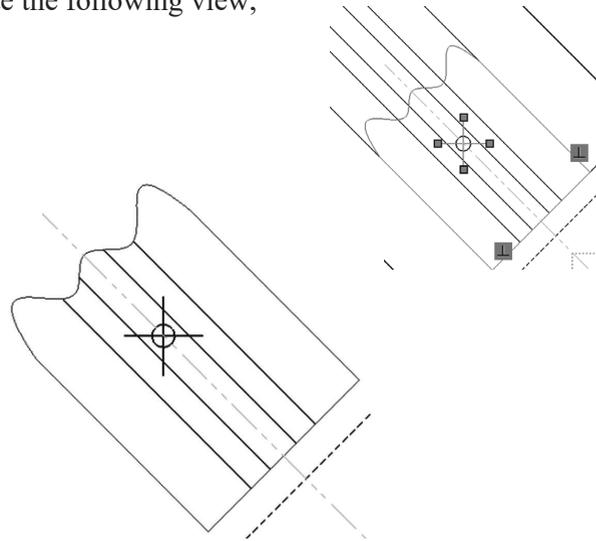
3. Identify the illustrated Drawing view.

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



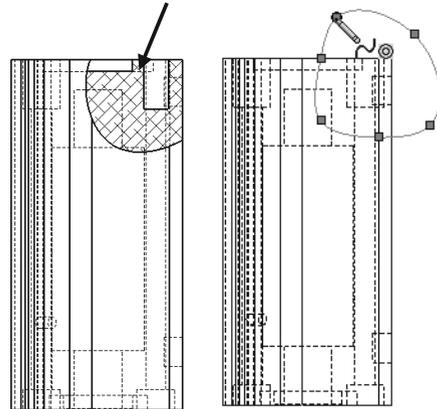
4. Identify the view procedure. To create the following view, you need to insert a:

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



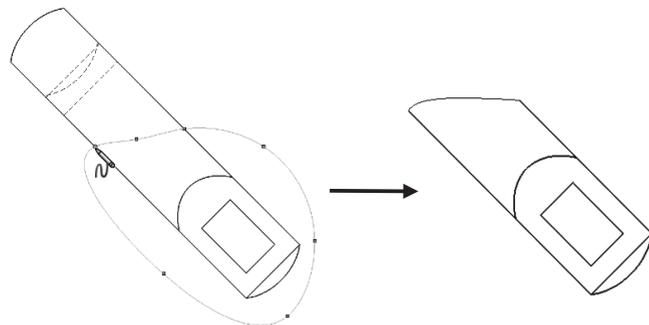
5. Identify the view procedure. To create the following view, you need to insert a:

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



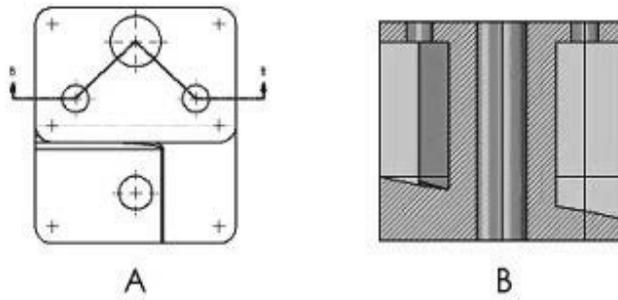
6. Identify the illustrated view type.

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



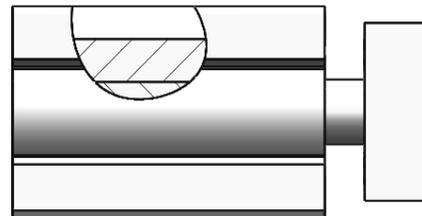
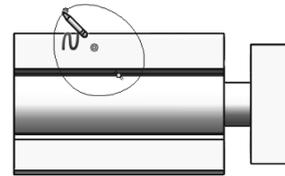
7. To create View B from Drawing View A insert which View type?

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



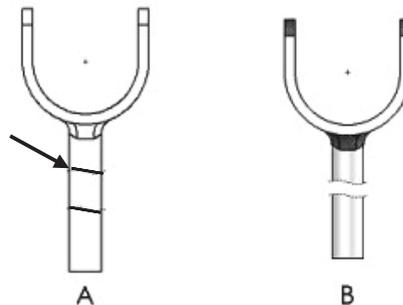
8. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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

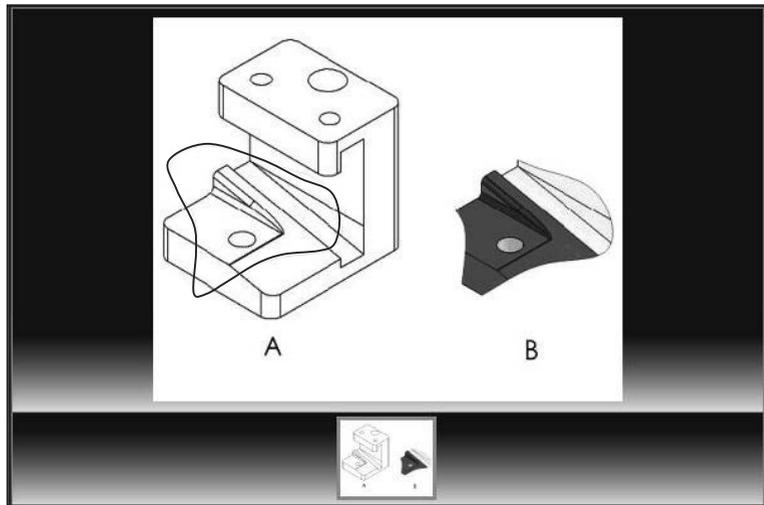


9. To create View B it is necessary to sketch a closed spline on View A and insert which View type?

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

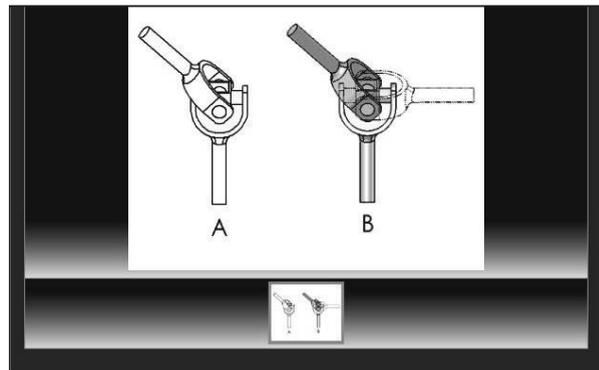


💡 Screen shots from an older CSWA exam for the Drafting Competencies category. Read each question carefully. Use SOLIDWORKS help if needed.



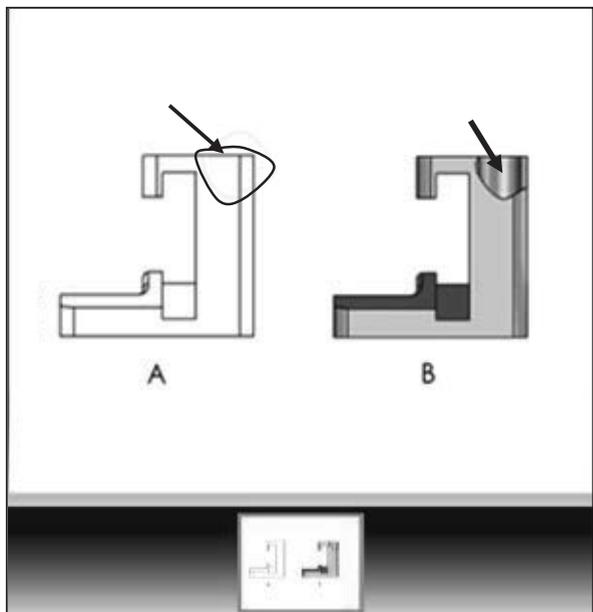
💡 Zoom in on the part or view if needed.

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:

