

SOLIDWORKS Certification Exam Guide & Practice Test



**CSWA: Certified
SOLIDWORKS Associate**

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About This Guide

This guide contains information, advice, and practice for becoming a Certified SOLIDWORKS Associate. Here, you will find:

- An overview of SOLIDWORKS Certifications
- Details about the CSWA Exam
- Advice on preparing for and taking the CSWA Exam
- A Practice CSWA Exam and answer key
- Directions to useful SOLIDWORKS websites

The CSWA Certification

SOLIDWORKS Certifications are a benchmark to measure your knowledge and competency with SOLIDWORKS software. A certification helps you stand out from the crowd and showcases your expertise to businesses and professionals alike—a valuable asset in a competitive job market.

A few popular SOLIDWORKS Certifications include:

- **CSWA:** Certified SOLIDWORKS Associate
- CSWP: Certified SOLIDWORKS Professional
- CSWPA: Certified SOLIDWORKS Professional Advanced:
 - Sheet Metal
 - Weldments
 - Surfacing
 - Mold Tools
 - Drawing Tools
- CSWE: Certified SOLIDWORKS Expert
- CEPA: Certified Enterprise PDM Administrator

Why take the CSWA Exam?

Being a CSWA is the difference between saying “I’ve used SOLIDWORKS” and “I know SOLIDWORKS.” The information in the CSWA Exam covers basic must-have skills for a serious SOLIDWORKS user. The CSWA may also be your stepping stone towards higher certifications, like the CSWP and the CSWPAs.

CSWA Exam Details

The following is the topic and question breakdown of the CSWA exam:

Drafting Competencies (3 Questions of 5 Points Each):

- Miscellaneous questions on drafting functionality

Basic Part Creation and Modification (2 Questions of 15 Points Each):

- Sketching
- Extrude Boss
- Extrude Cut
- Modification of Key Dimensions

Intermediate Part Creation and Modification (2 Questions of 15 Points Each):

- Sketching
- Revolve Boss
- Extrude Cut
- Circular Pattern

Advanced Part Creation and Modification (3 Questions of 15 Points Each):

- Sketching
- Sketch Offset
- Extrude Boss
- Extrude Cut
- Modification of Key Dimensions
- More Difficult Geometry Modifications

Assembly Creation (4 Questions of 30 Points Each):

- Placing of Base Part
- Mates
- Modification of Key Parameters in Assembly

Total Questions: 14

Total Points: 240

Points Needed to Pass: 165

Maximum Time: 180 min

The sample test in this guide shows the basic format of the CSWA exam in three sections:

- Drafting Competencies
- Part Modeling
- Assembly Creation

How to Prepare for the Exam

Don't let the exam questions take you by surprise! We recommend that you practice creating and editing drawings, models, and assemblies beforehand, and that you ask for help from SOLIDWORKS users.

No details to the solutions for either this sample exam or the real test will be shared by the SOLIDWORKS Certification team. Please consult your SOLIDWORKS reseller, your local user group, or the on-line SOLIDWORKS forums at forum.solidworks.com to review any topics on the CSWA exam.

A great resource is the SOLIDWORKS website (solidworks.com).

- For training courses, go to **Home > Support > Training > SOLIDWORKS Courses**.
- To download models to practice on, go to **Home > Support > Training > SOLIDWORKS Training Files**
- You'll find a wealth of access to videos, tutorials, blogs, events, and fellow users in the SOLIDWORKS Community at **Home > Resource Center**

You can also log into my.solidworks.com to browse lessons, forums, models, and much more.

Last but not least, make use of this exam guide to review realistic exam questions, look over important topics, and familiarize yourself with the exam procedure.

To simulate test conditions, you can take the sample exam in the testing client software. For instructions, go to the 'Taking the Real Exam' section.

How to Take This Practice Exam

1. You must be running SOLIDWORKS on the computer, both for the practice and actual test. SOLIDWORKS can only be installed with the Windows operating system.
2. To simulate real conditions, it is best NOT to print this exam.
In the real test, the VirtualTester client window runs concurrently with SOLIDWORKS, requiring you to switch between applications. Keep this document open and consult it while running SOLIDWORKS.
3. After each question, save a version of your model in a different file for later reference. This may also help you fix errors later on in the test.
4. The multiple choice will help you check that your model is on the right track. If your answer is not listed in the selections offered, it is likely that there is something wrong with your model.
5. This guide includes an answer key after the practice exam.
6. If you can complete this exam and get at least 6 out of 8 questions correct in less than 90 minutes, you should be ready to take the real CSWA exam.
7. This practice exam is also available to take on-line for free in the testing client software. To take the exam in the client software, follow the instructions in the next section.

Taking the Real Exam

The real exam can be taken on your personal computer at a time of your choosing. The CSWA Exam is administered through the Tangix TesterPRO Client, an application that you may download from the SOLIDWORKS VirtualTester website.

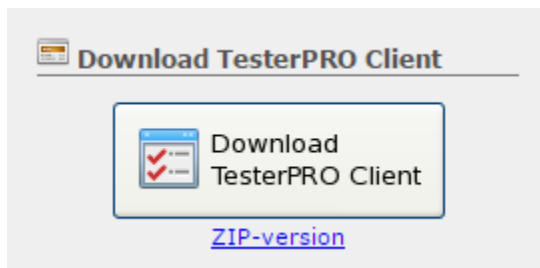
The sample exam in this packet can also be taken in the Tangix TesterPRO Client.

To learn the testing procedure in VirtualTester, if you have a MySolidWorks account, please refer to this video:

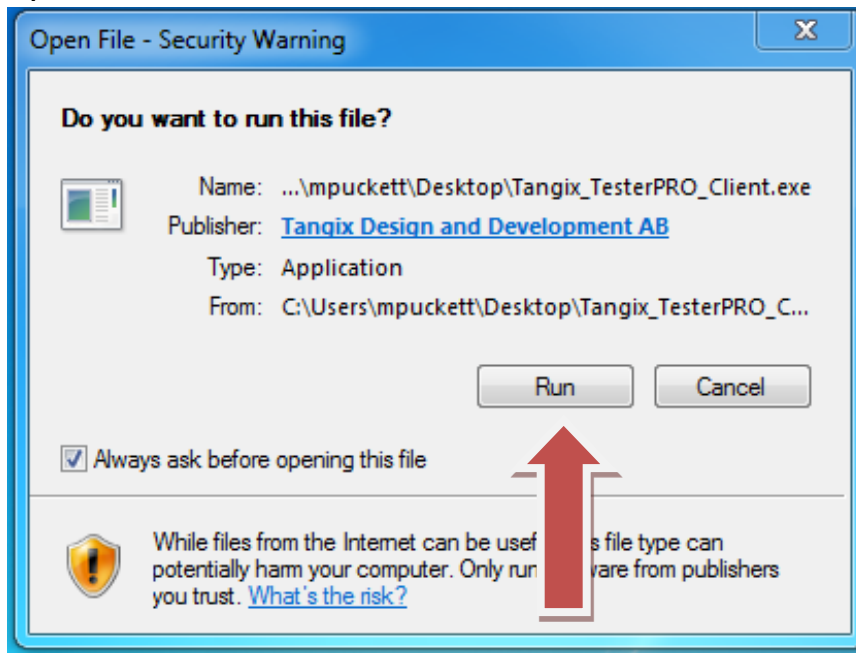
<http://my.solidworks.com/training/master/103/installing-and-viewing-the-test-software>

Alternatively, you may follow these steps:

1. Visit the SOLIDWORKS VirtualTester Certification Center at <https://SOLIDWORKS.virtualtester.com/>.
2. Locate and click the button (right hand side of page) to download the Tangix TesterPRO Client:



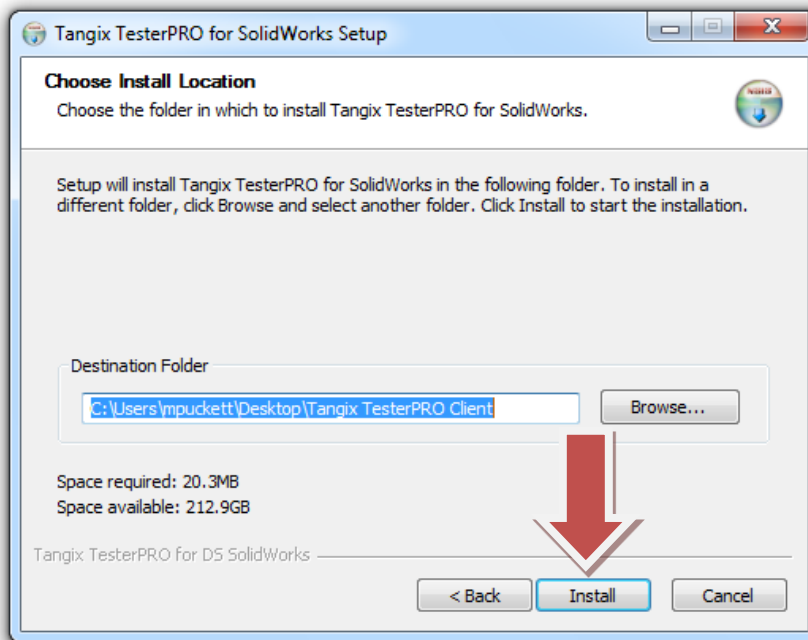
3. Open the downloaded folder and run the executable file:



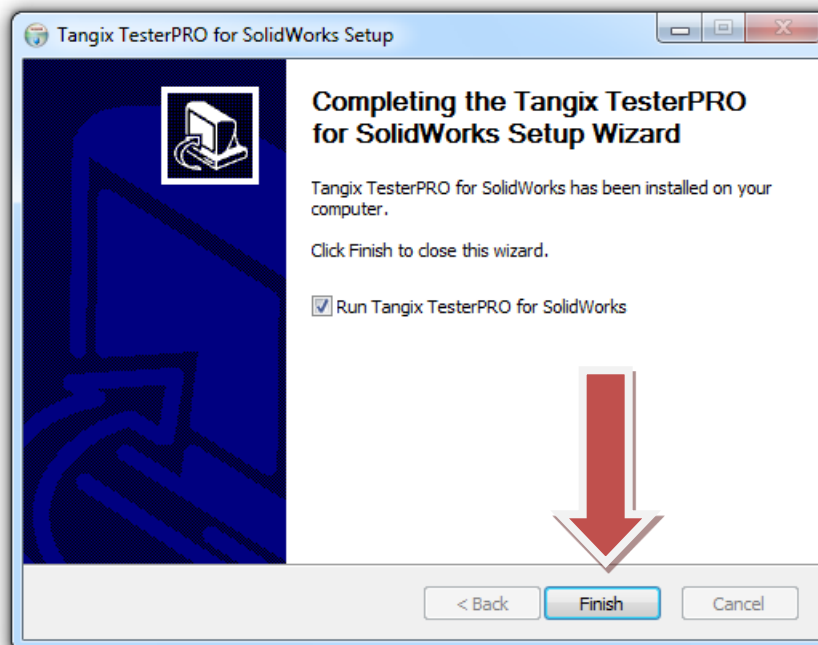
4. Agree to the License Agreement.



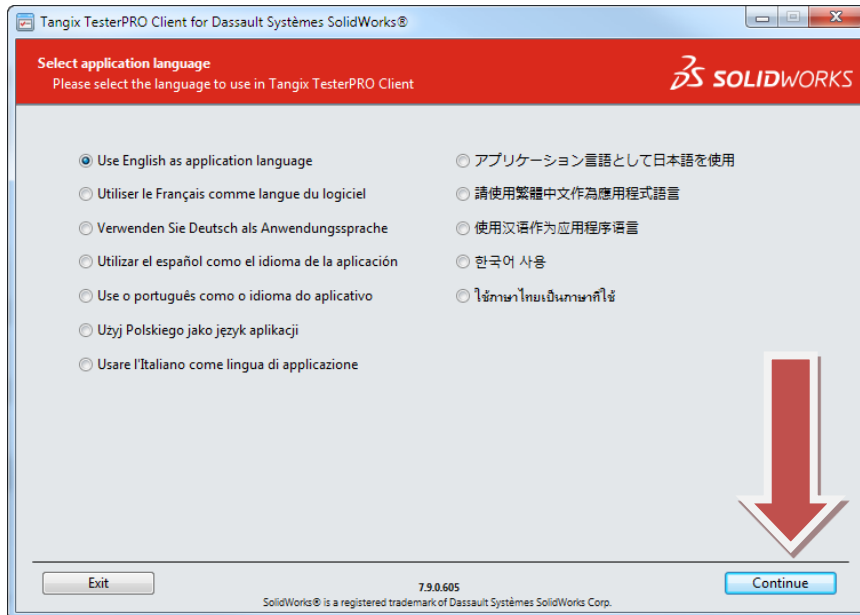
5. Click the install button.



6. Click the finish button.



7. When you run the program, select your language and click Continue.



If you have already taken a SOLIDWORKS Certification Exam, skip to step 9.

8. If this is your first exam, you will need to create an account. Fill in the required fields and click continue:

Write down your login credentials for future use!

Skip to step 10.

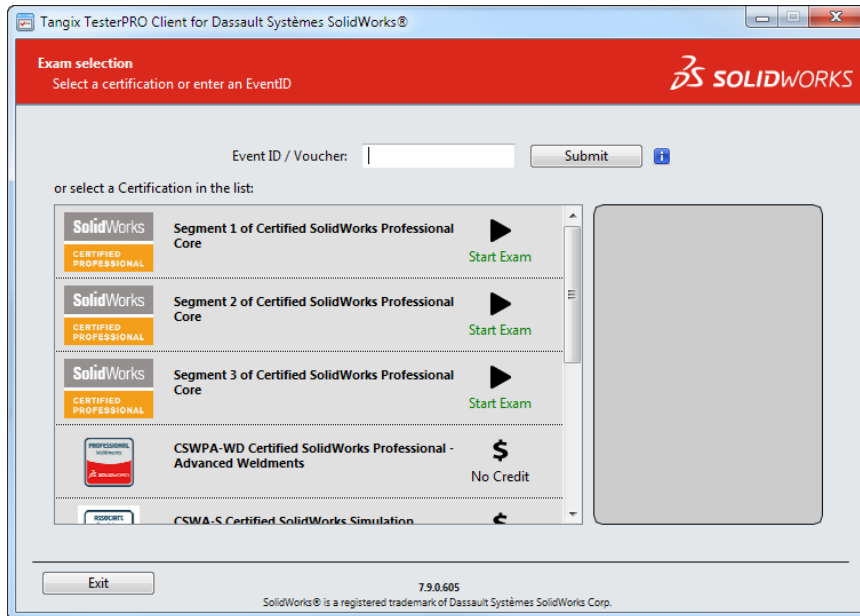
9. If you have already taken a SOLIDWORKS Certification Exam, select the option below, fill in your credentials, and click continue.

The screenshot shows the 'Candidate Information' window of the Tangix TesterPRO Client. The window has a red header bar with the SolidWorks logo and the text 'Candidate Information' and 'Connect to VirtualTester'. Below the header, there are two radio button options. The first option, 'I already have a VirtualTester UserID and password', is selected. Below this option are input fields for 'E-mail:' and 'Password:'. The second option is 'I do not have a VirtualTester UserID. Create one with the following information:'. At the bottom of the window, there is an 'Exit' button on the left, a version number '7.9.0.605' in the center, and a 'Continue' button on the right. A small copyright notice is visible at the very bottom: 'SolidWorks® is a registered trademark of Dassault Systèmes SolidWorks Corp.'

10. If you are taking a sample exam online, scroll down and select the exam with a gift icon.

The screenshot shows the 'Exam selection' window of the Tangix TesterPRO Client. The window has a red header bar with the SolidWorks logo and the text 'Exam selection' and 'Select a certification or enter an EventID'. Below the header, there is an input field for 'Event ID / Voucher:' with a 'Submit' button and an information icon. Below this, it says 'or select a Certification in the list:'. There is a list of four certifications, each with a small icon on the left, the certification name in the middle, and a status on the right. The first two are 'CSWA-S Certified SolidWorks Simulation Associate - FEA' and 'CSWPA-DT Certified SolidWorks Professional - Advanced Drawing Tools', both with a '\$' icon and 'No Credit' status. The third is 'CSWS-CCO Specialist - Customer Center Onboarding' with a play button icon and 'Start Exam' status. The fourth is 'CSWA Sample Exam' with a gift icon and 'Free Exam' status. The 'Free Exam' option is circled in red. To the right of the list is a large empty rectangular box. At the bottom of the window, there is an 'Exit' button on the left, a version number '8.0.3.1091' in the center, and a copyright notice: 'SOLIDWORKS® is a registered trademark of Dassault Systèmes SOLIDWORKS Corp.'

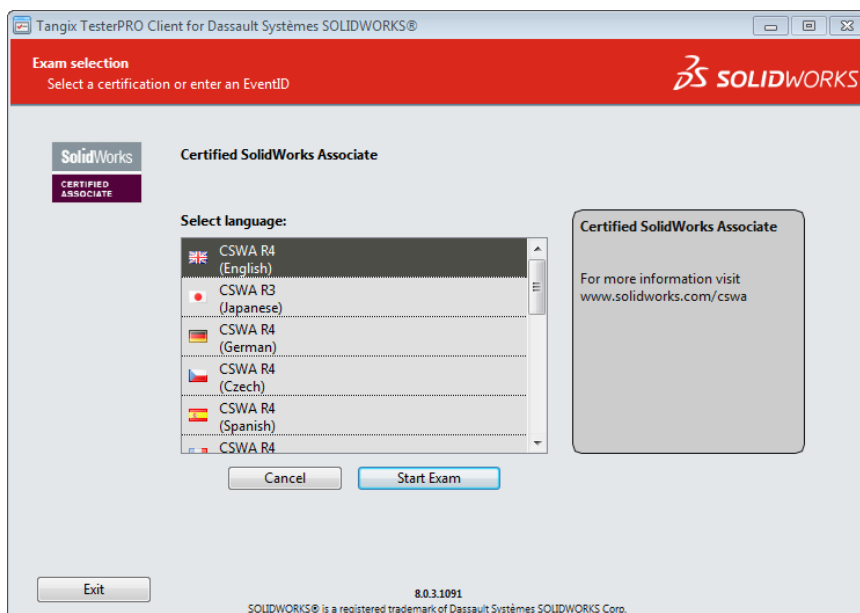
11.To take an exam from a credit, select an exam that has the words 'Start Exam' next to it (only exams that you have credits for will show as available):



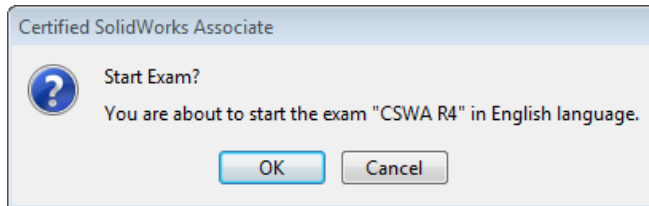
If you do not have credit for the exam, do one of the following:

- If you have an Event ID or Voucher, type it into the field on this page and click "Submit."
- Purchase an exam credit online. This may be found on solidworks.com: Click **Support > Certification > Exam Registration**.

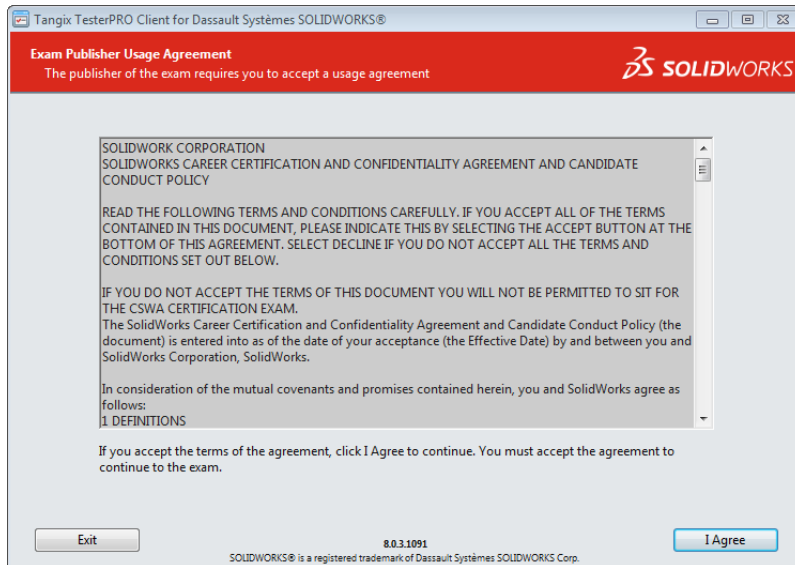
12.You may be asked to select a language. Select a language and click "Start Exam."



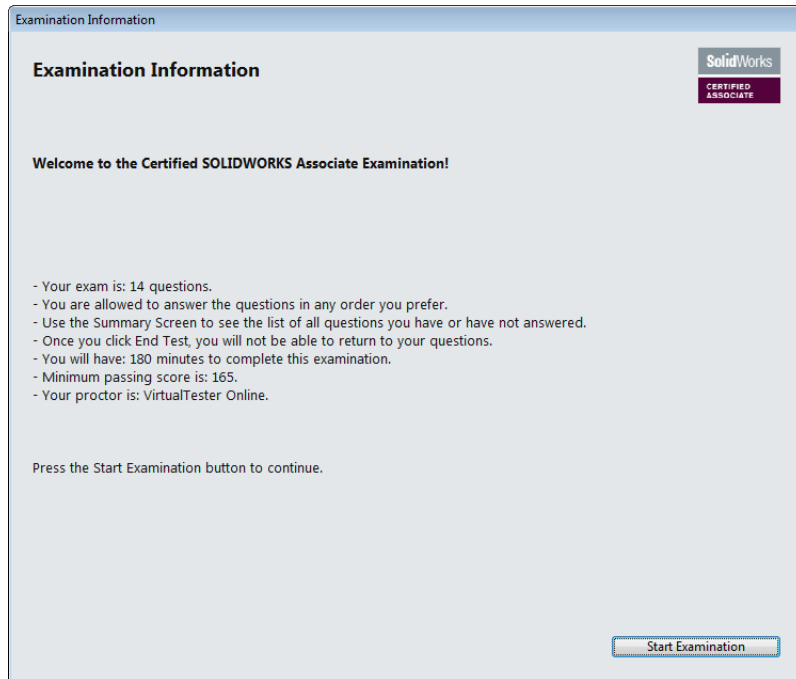
13. Confirm the exam you are about to begin:



14. Agree to the Confidentiality Agreement and Candidate Conduct Policy.



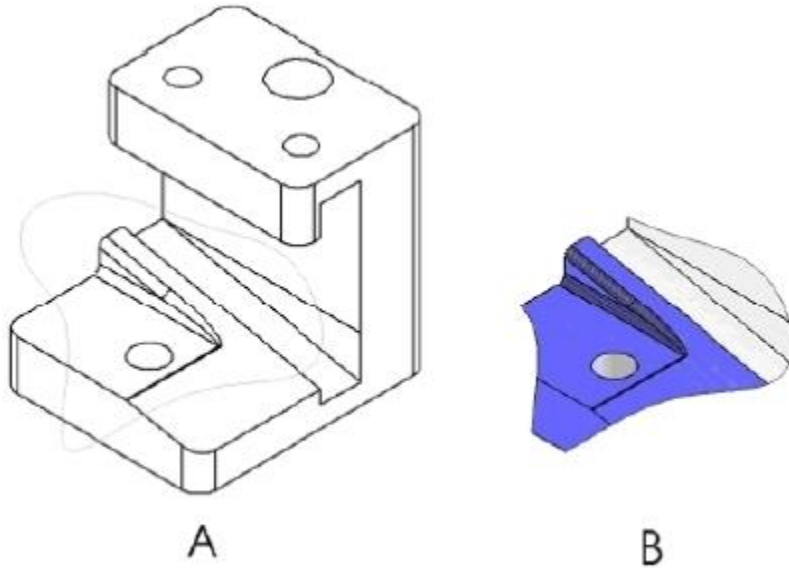
15. On the Examination Information page, read the information and then click "Start Examination", this will begin the timed portion of your exam.



The practice exam begins on the next page.

Test Questions

Part 1: Drafting Competencies



1. To create drawing view 'B' it is necessary to sketch a spline (as shown) on drawing view 'A' and insert which SOLIDWORKS view type?

a) Section

To crop a view:

b) Crop

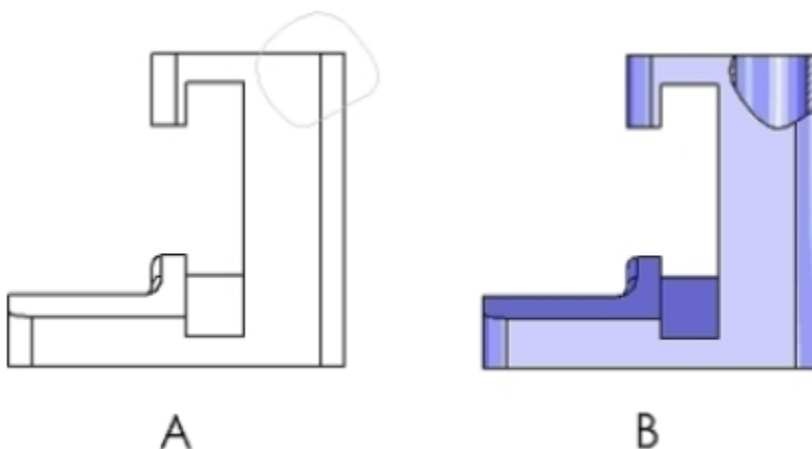
In a drawing view, sketch a closed profile such as a circle.

Click Crop View tool_Crop_View_Drawing.gif (Drawing toolbar), or click Insert > Drawing View > Crop. The view outside the profile disappears.

c) Projected

d) Isometric

A circle is drawn on this Section View After cropping, only the view inside the circle is displayed.

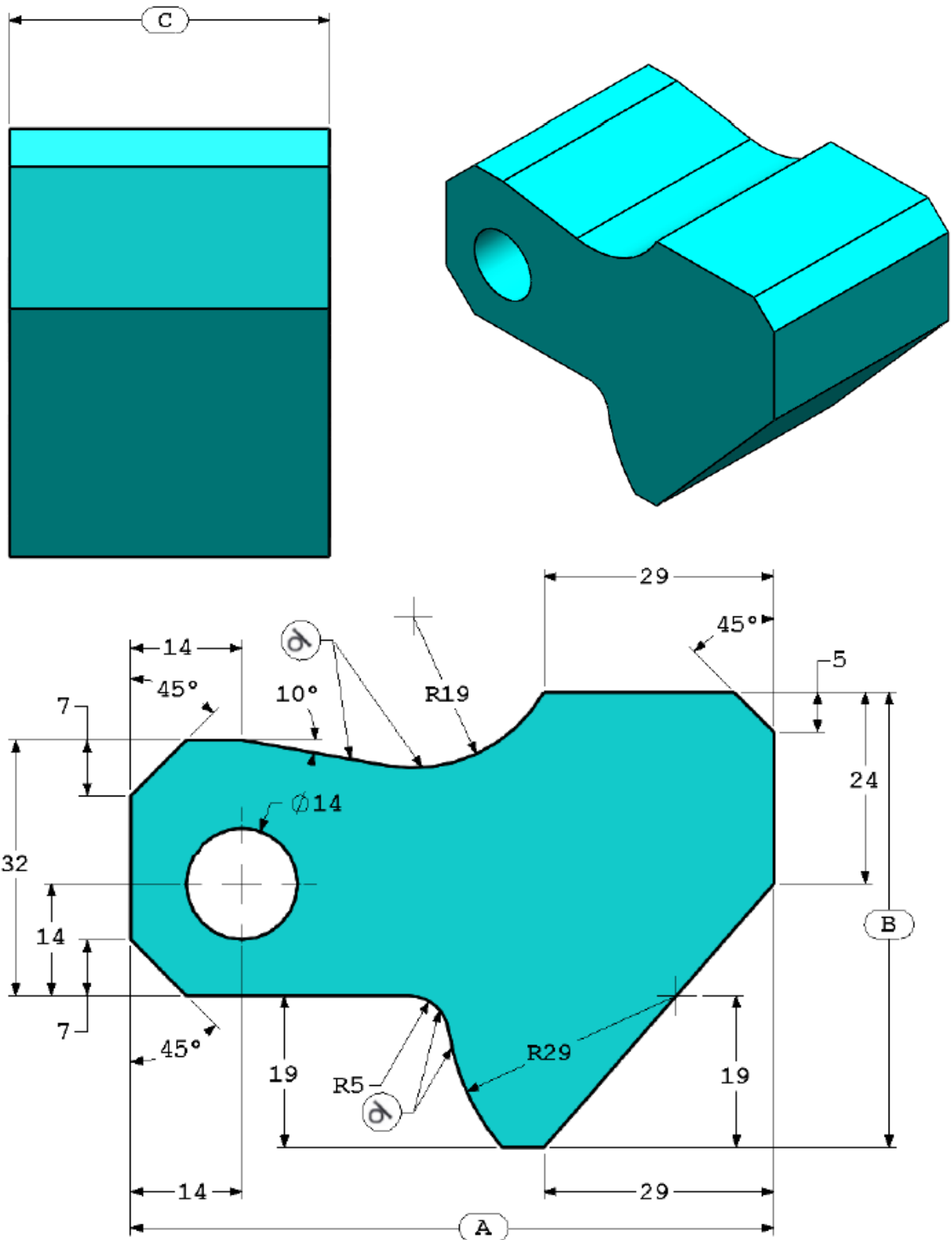


2. To create drawing view 'B' it is necessary to sketch a spline (as shown) on drawing view 'A' and insert which SOLIDWORKS view type?

- a) Aligned Section
- b) Detail
- c) Broken-out Section
- d) Section

You will need to specify the depth that you can (sort of like extrude cut) and you can specify by clicking on another view and edge, or numerically.

The following images are to be used to answer Questions #3 – 4.



For material: Click on material on the left side list: edit material, find AISI
For variables: Tools--> Equations->global variables -> "A" = 81

3. Part (Tool Block) - Step 1

Build this part in SOLIDWORKS.

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 [How do I change the density?](#)

A = 81.00

B = 57.00

C = 43.00

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 solid model.

a) 1028.33

b) 118.93

c) 577.64

d) 939.54

REMINDER: Save your part in a different file after each question in case it must be reviewed.

4. Part (Tool Block) - Step 2

Modify the part in SOLIDWORKS.

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

Use the part created in the previous question and **modify it by changing** the following parameters:

A = 84.00

B = 59.00

C = 45.00

Note: Assume **all other dimensions are the same** as in the previous question.

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

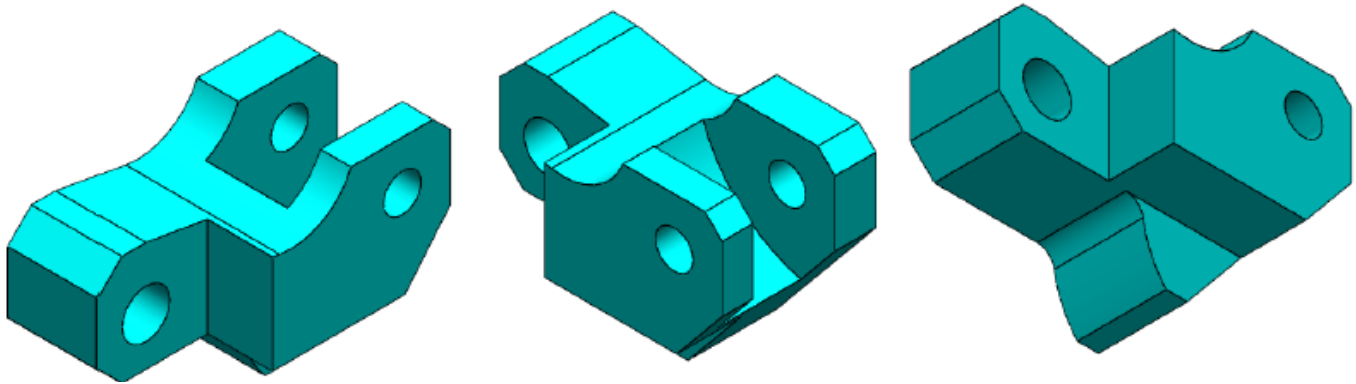
Mass = 1032.32 grams

use coma as decimal

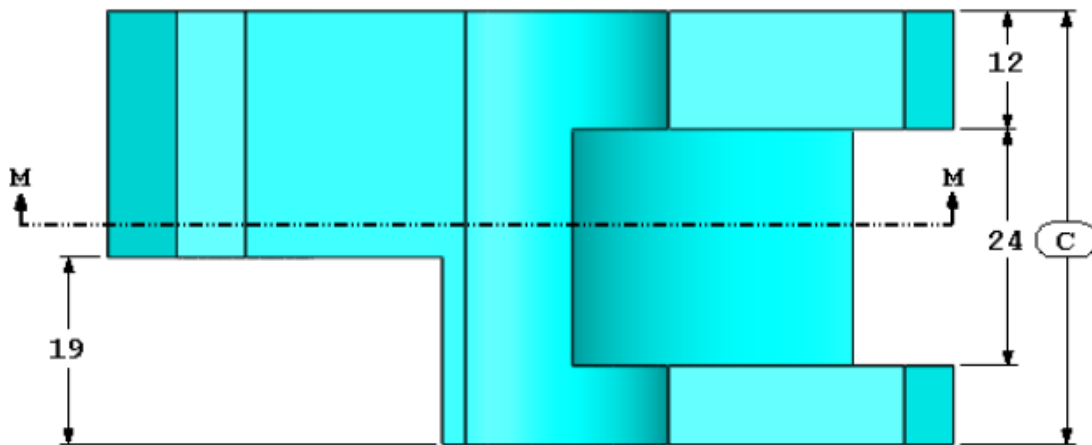
Mass = 1032,32 grams

Part Modeling (continued)

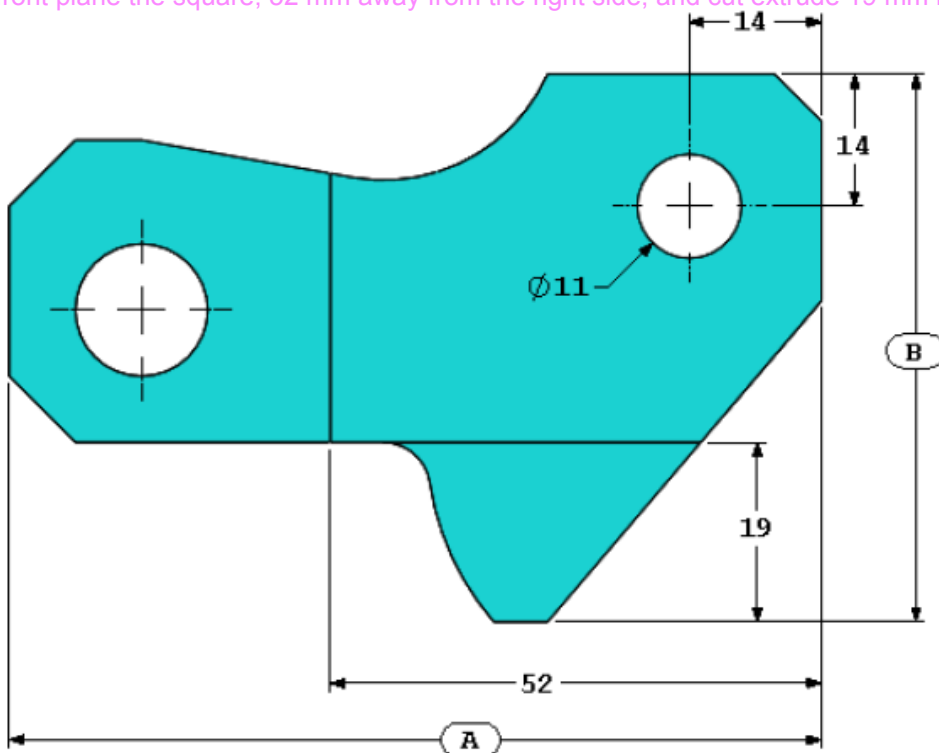
These images are to be used to answer Question #5.

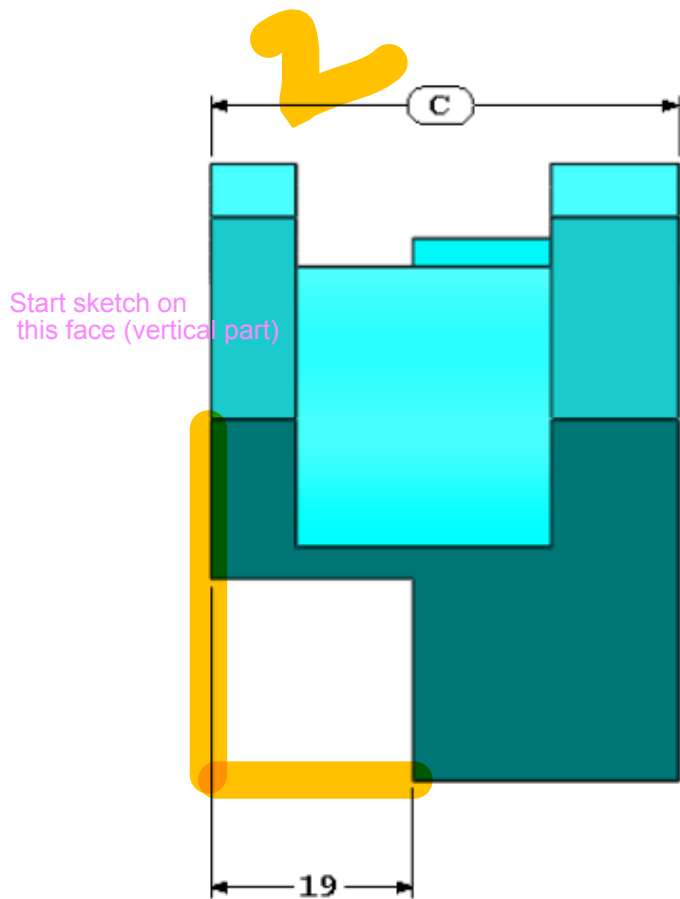


Create a plane with Reference geometry, add how far away by modifying the offset



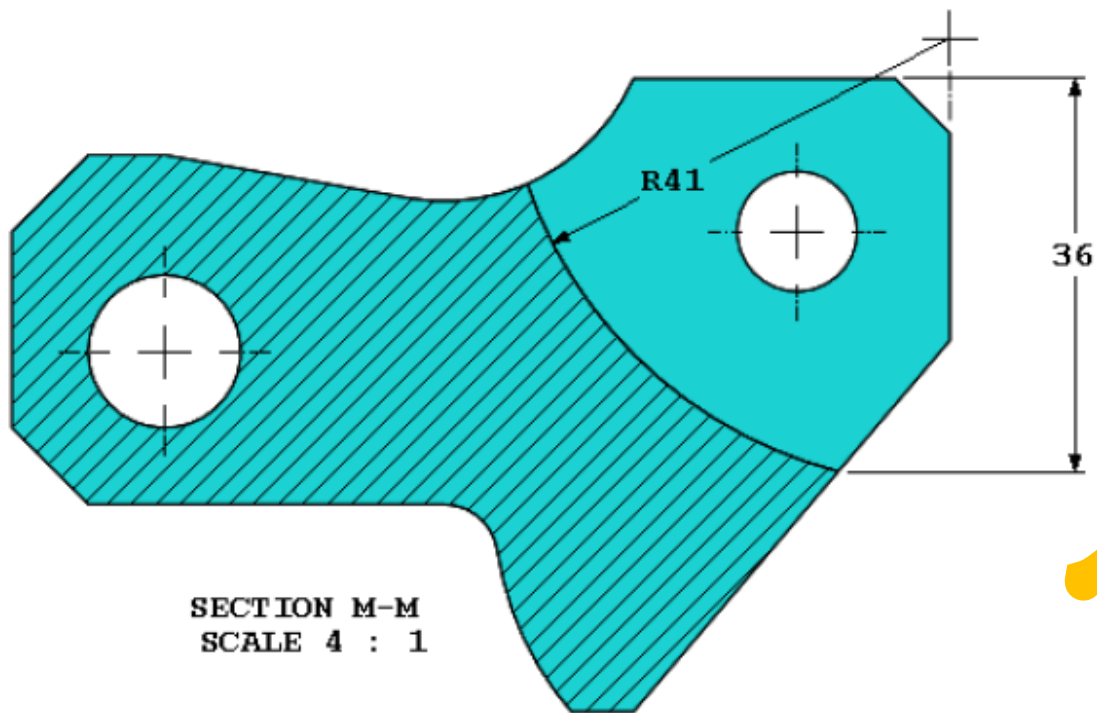
Sketch on the front plane the square, 52 mm away from the right side, and cut extrude 19 mm blind





Convert edges on entities so you can have guidelines for dimensions. and draw a square of 19 mm from there

Cut through all



Sketch on the new plane, the circle of R41, the center of the circle and the last edge are coincident
Convert front face into entities to mark a dimension on it, we need to create a point that intersects the circle and the diagonal, so we can add the dimension of 36
then extrude cut 24mm

5. Part (Tool Block) - Step 3

Modify the part in SOLIDWORKS.

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

Use the part created in the previous question and modify it by removing material and also by changing the following parameters:

A = 86.00

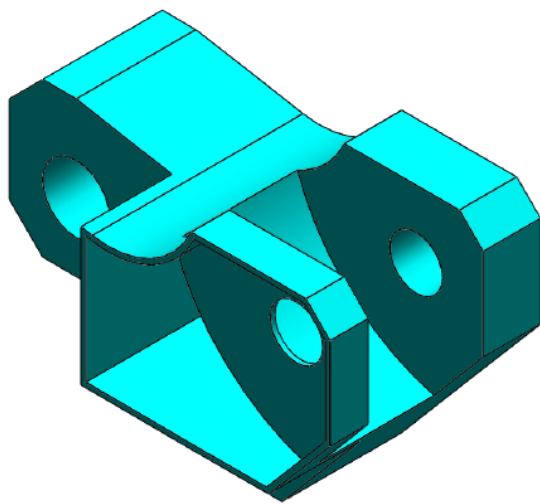
B = 58.00

C = 44.00

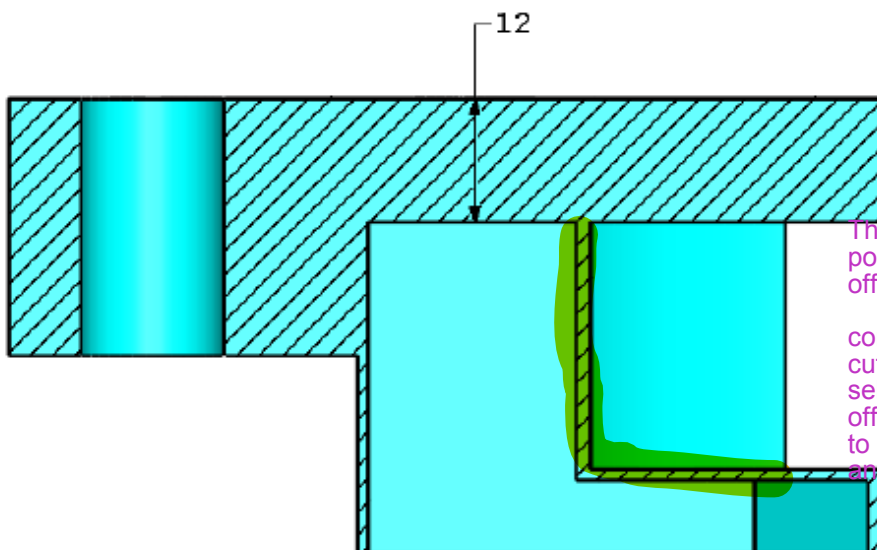
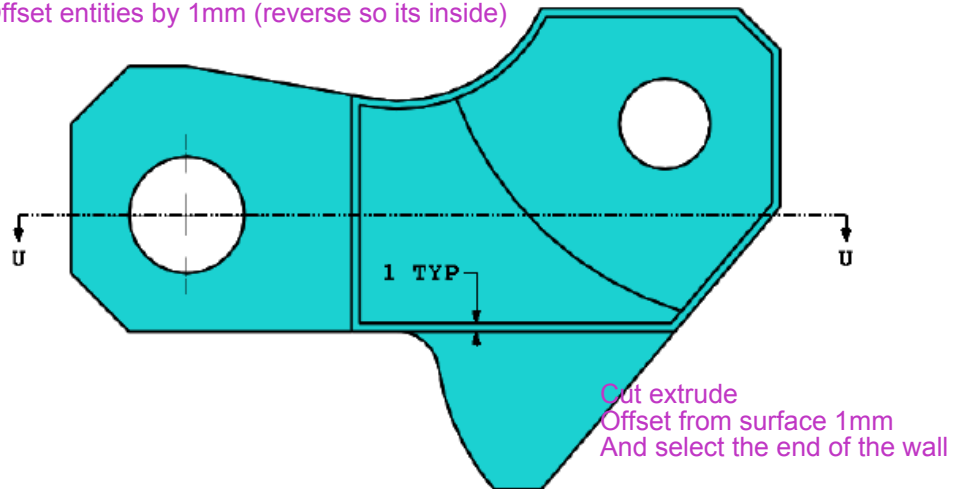
What is the overall mass of the part (grams)? Mass = 628.18 grams

Part Modeling (continued)

These images are to be used to answer Question #6.

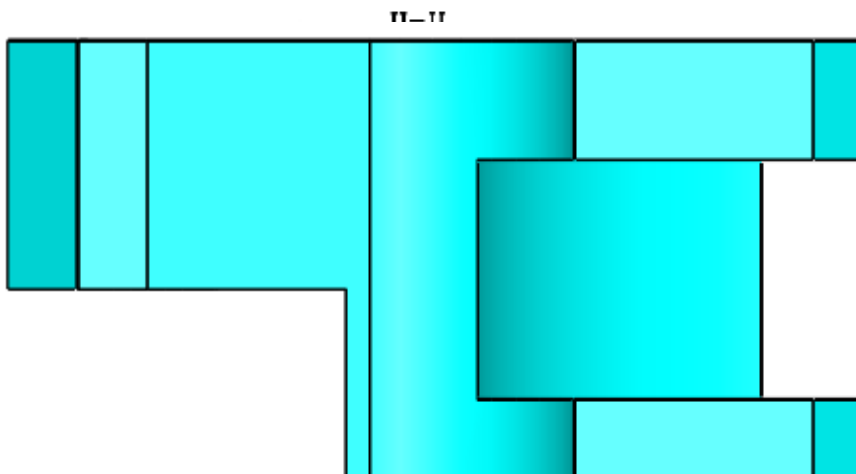


Sketch on this side first
Offset entities by 1mm (reverse so its inside)



Then you can do the extra cut byt sketching on the pocket. Convert identities of the arch and offset it 1mm too so we can have the wall thickness

convert sketch plane into entities
cut extrude
select the lower area
offset from surface
to other end of the object
and do 12mm



6. Part (Tool Block) - Step 4

Modify this part in SOLIDWORKS.

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

Use the part created in the previous question and modify it by adding a pocket.

Note 1: Only one pocket on one side is to be added. This modified part is not symmetrical.

Note 2: Assume all unspecified dimensions are the same as in the previous question #5.

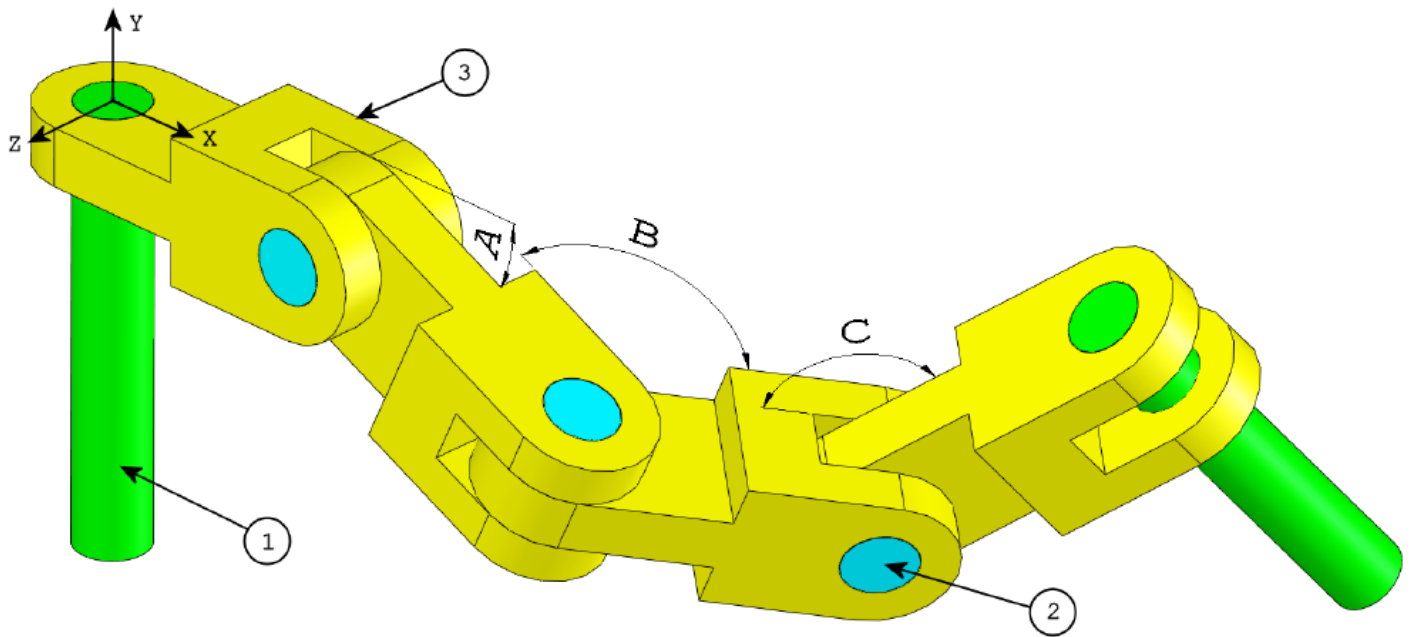
What is the overall mass of the part (grams)?

Mass = 432.58 grams

<https://www.youtube.com/watch?v=eu1zbl1t7zc>

Part 3: Assembly Modeling

This image is to be used to answer Questions #7 – 8.



Reference geometry-> coordinate system-> align the arrows with the edges of your assembly

When checking the center of mass make sure you select your own coordinate system

7. Build this assembly in SOLIDWORKS (Chain Link Assembly)

- Download the attached zip file and open it.
- Save the contained parts and open those parts in SOLIDWORKS. (Note: If SOLIDWORKS prompts "Do you want to proceed with feature recognition?" please click "No".)
- IMPORTANT: Create the Assembly with respect to the Origin as shown in isometric view. (This is important for calculating the proper Center of Mass)
- Create the assembly using the following conditions:

1. Pins are mated concentric to chain link holes (no clearance).
2. Pin end faces are coincident to chain link side faces.

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Assembly origin: As shown in image

You add angle mates from one face to the other

A = 25 degrees I had to do 155 so it matched. ($180-25=155$)

B = 125 degrees 125 worked

C = 130 degrees 130 worked

I think it depends of the point of view

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

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

a) X = 348.66, Y = -88.48, Z = -91.40

b) X = 308.53, Y = -109.89, Z = -61.40

c) X = 298.66, Y = -17.48, Z = -89.22

d) X = 448.66, Y = -208.48, Z = -34.64

Click evaluate-> mass properties-> create center of mass feature-> relative to coordinate system not default

Review how to add point and coordinate system

8. Modify the assembly in SOLIDWORKS. (Chain Link 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 = 30 degrees

B = 115 degrees

C = 135 degrees

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

END OF TEST

Answer Key

Answers:

1. b) Crop

2. c) Broken-out Section

3. d) 939.54 g

4. 1032.32 g

5. 628.18 g

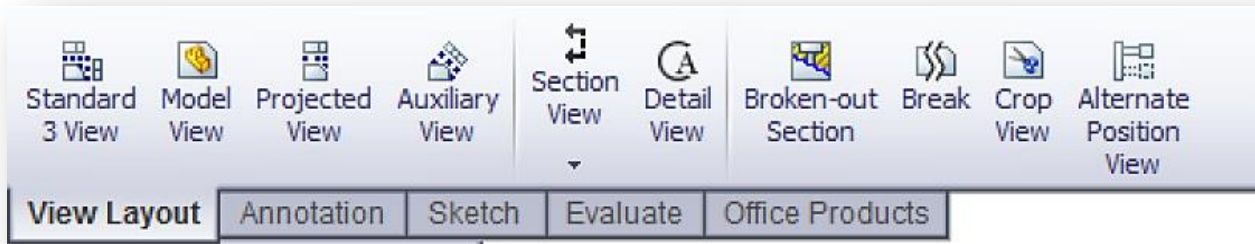
6. 432.58 g

7. a) $X = 348.66$, $Y = -88.48$, $Z = -91.40$

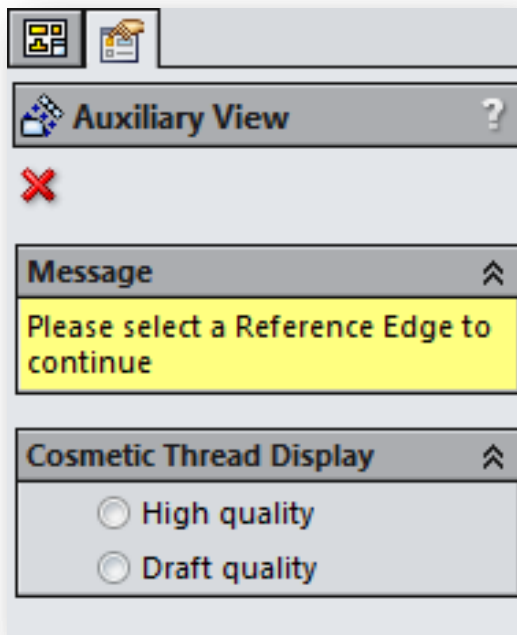
8. $X = 327.67$, $Y = -98.39$, $Z = -102.91$

Hints and Tips

- 1) To prepare for the Drafting Competencies section of the CSWA, review all the drawing views that can be created. These commands can be found by opening any drawing and going to the **View Layout** command manager toolbar or in the menu **Insert > Drawing View**.



- 2) For a detailed explanation of each View type, access the individual feature Help section by selecting the **Help** icon in the **Feature Manager** for that View Feature.



Frequently Asked Questions

Below is a series of common questions regarding SOLIDWORKS Certifications. To peruse more FAQs, please visit the SOLIDWORKS Certification FAQ page under **Home > Support > Certification > FAQ**.

1. What resources do I need to take the CSWA/CSWP?

The CSWA/P test is an on-line test that can be taken on any computer that has SOLIDWORKS running and a connection to the Internet. **You are responsible for providing your own working copy of SOLIDWORKS. A commercial license of SOLIDWORKS or the SOLIDWORKS Student Edition will be adequate for the exams.**

The test runs in its own client window separate from SOLIDWORKS. You can either run the exam software on the same computer as your SOLIDWORKS software or in a separate computer next to the one running SOLIDWORKS. Please note that if you are taking the test on a separate computer that you have some means to transfer files from the computer running the testing client to the computer running SOLIDWORKS. Dual monitors are recommended but not required.

As stated above, an internet connection is required for the CSWA/P exams. At this point there are no provisions for a stand-alone exam.

2. What is the policy on retaking the CSWA/CSWP?

There is a 30-day waiting period before retaking the CSWA and any Advanced CSWP exam. There is a 14-day waiting period before retaking any CSWP segment exam. Also the user will be required to purchase another exam credit to retake the test.

3. What do I receive when I pass the exam?

You will receive certificates for each exam that you pass. You receive an email that directs you to our electronic certificate access page. There you can login and download your electronic certificate(s) and electronic business card logo(s).

4. Can the exam be paused?

No. Once started, exams cannot be paused.

5. I took the exam and my Internet connection failed when the test ended. How do I report my results now?

The tester software will retry to connect for ~2 minutes. If the connection is still not available, an error message will be displayed and a TXT file will be saved on your desktop. The name of the file is: Tangix_TesterPro_Error_Date_Time.

Please send this file to support@tangix.com

6. Will my answers be lost if my computer crashes? Will I lose my testing credit/have to pay again if my computer crashes?

You will have to start the examination over and no refund will be available. Once you click 'take exam' your exam credit will be redeemed and we cannot refund your payment or voucher. If your computer crashes, your answers are not recoverable.

7. Will a certificate be mailed out to me once I pass an exam?

SOLIDWORKS no longer mails out certificates to individuals who have passed an exam. We have made it a simple download that you can print from your computer so you no longer have to wait to receive it. To print your certificate, simply visit www.virtualtester.com/solidworks/user.php. On that page, simply click the "Print My Certificate" button and follow the on-screen instructions. Please note that not all exams have printable certificates.

Helpful Sites

SOLIDWORKS Home:

www.solidworks.com

SOLIDWORKS Resource Center:

<http://www.solidworks.com/sw/resources.htm>

SOLIDWORKS YouTube Channel:

<https://www.youtube.com/user/solidworks>

MySOLIDWORKS:

my.solidworks.com

Training Pages:

<http://www.solidworks.com/sw/support/software-training-certification.htm>