CHAPTER 1 - SEGMENT 1 OF THE CSWP CORE EXAM

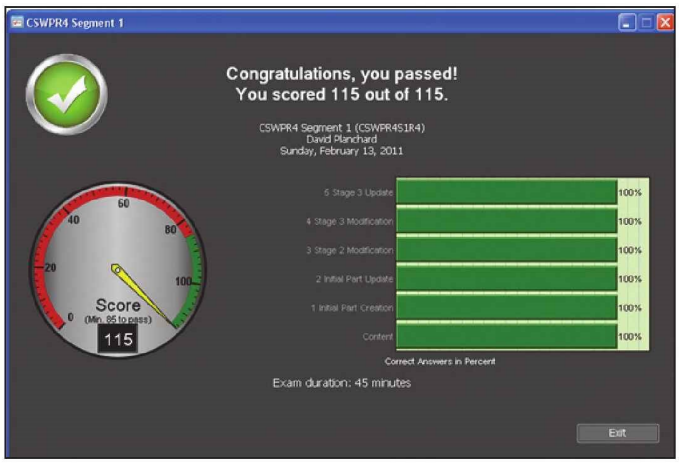
Introduction

DS SolidWorks Corp. offers various stages of certification. Each stage represents increasing levels of expertise in 3D CAD design: *Certified SolidWorks Associate CSWA, Certified SolidWorks Professional CSWP and Certified SolidWorks Expert CSWE* along with specialty fields in Drawings, Simulation, Sheet Metal, Surfacing and more.

The CSWP Certification exam is offered in three separate segments. The book covers the three separate segments of the CSWP CORE exam. At SolidWorks World, there is a single three hour CSWP exam only for registered users.

The first segment is not a prerequisite for the second and the second segment is not a prerequisite for the third. You can take the segments in any order.

Each segment covers a different set of disciplines. This chapter addresses the first segment of the CSWP CORE exam. All segments are timed. You will be tested on data found in the Mass Properties section of SolidWorks. It is important to be familiar with accessing Mass Properties and interpreting them correctly.

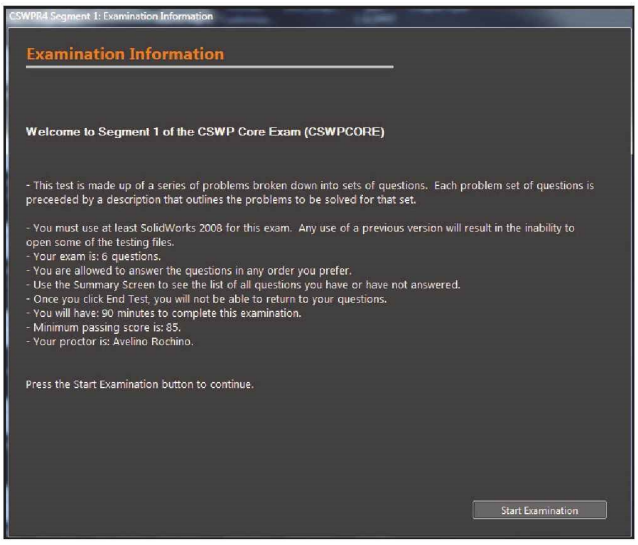


The first segment is 90 minutes with six (6) questions. The format is either multiple choice or single fill in the blank.

The first question in each segment is typically in a multiple choice format. A total score of 85 out of 115 or better is required to pass the first segment.

You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question. If you don't have the exact answer of the first question, you will most likely fail the following question. This is crucial as there is no partial credit.

In the first segment, you will create a part and modify various dimensions. A question is presented to you in multiple steps and you need to obtain the correct answer at each step to get the question correct.



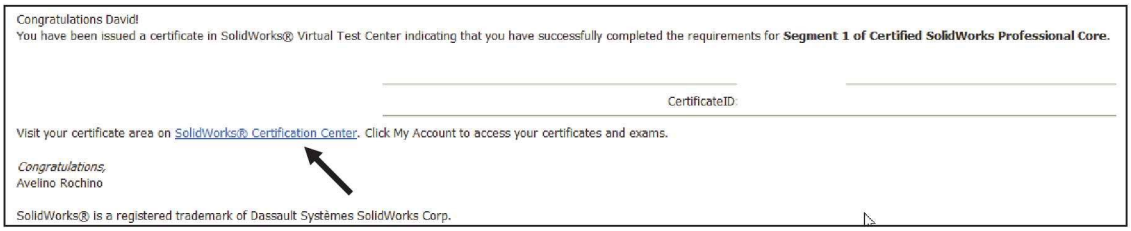
Actual CSWP exam format

The Examination Information dialog box states that you are allowed to answer the questions in any order that you prefer. Is this true?

Yes and No! Segment 1 consists of six (6) questions. In those 6 questions there are three major stages to be modeled: Stage 1 (question 1), Stage 2 (question 4) and Stage 3 (question 5). Questions 2, 3 and 6 then ask the user to make a modification to those stages.

Strategically it is best if a user does questions 1, 4 and 5 to model the major stages and compare their Mass Properties to the multiple choice answer. You should be within 1% of the stated value in the multiple choice section before you go back to questions 2, 3 and 6 (fill in the blank format) to make the changes.

When you pass the first segment of the CSWP CORE exam you will receive the following email. Click on the SolidWorks Certification Center hyperlink to login, activate and view your certificate.



For these tasks, review the text and the videos on the DVD that correspond to the competencies you need to know for Segment 1 of the CSWP CORE exam.

Before you start, review the Read and understand an Engineering document section. Engineering drawing views with annotations are presented to you in all segments of the CSWP CORE exam.

Read and understand an Engineering document

What is an Engineering document? In SolidWorks a part, assembly or drawing is referred to as a document. Each document is displayed in the Graphics window.

During the exam, each question will display an information table on the left side of the screen and drawing information on the right. Read the provided information and apply it to the drawing. Various values are provided in each question.

 If you do not find your answer (within 1%) in the multiple choice single answer format section - recheck your solid model for precision and accuracy.

 SolidWorks Mass Properties calculates the center of mass for every model. At every instant of time, there is a unique location (x, y, z) in space that is the average position of the mass in the system. The CSWP exam asks for center of gravity. For the purpose of calculating the center of mass and center of gravity near to earth or on earth, you can assume that the center of mass and the center of gravity are the same.

 SolidWorks views present illustrations that are not proportional to the given dimensions.

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: Aluminium 1060 Alloy
Density = 0.0027 g/mm^3

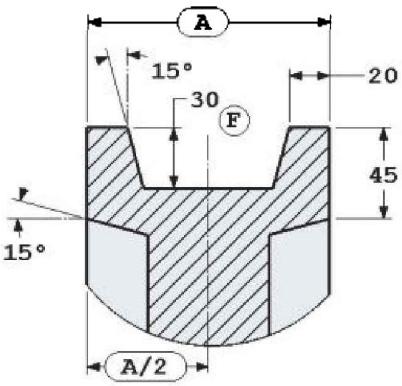
Modify the part using the following variable values:

A = 140.00
B = 50.00
C = 55 degrees

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

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

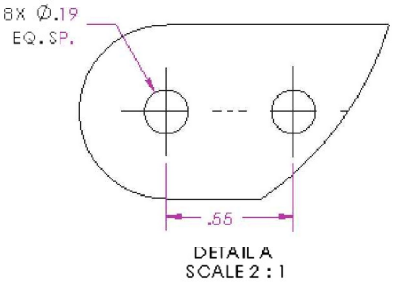
Actual CSWP exam format



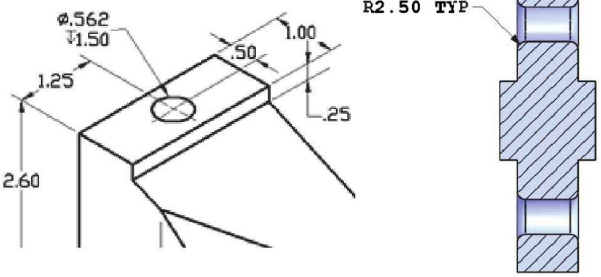
N

Engineering Documentation Practices

2D drawing views are displayed in the CSWP exam.
The ability to interpret a 2D drawing view is required.

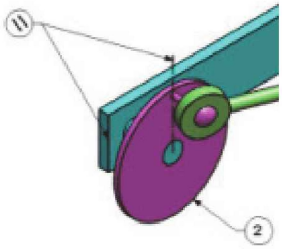


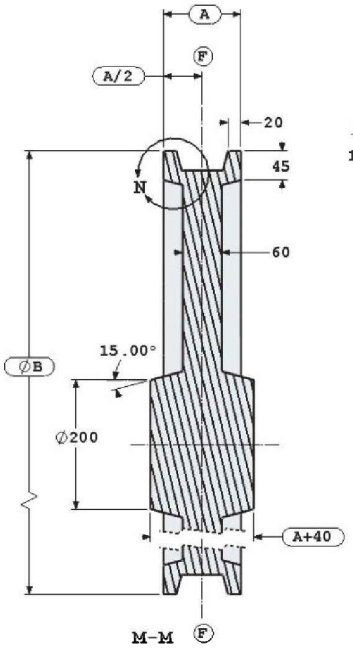
- Example 1: *8X Ø.19 EQ. SP*. Eight holes with a .19in. diameter are required that are equally (.55in.) spaced.
- Example 2: *R2.50 TYP*. Typical radius of 2.50. The dimension has a two decimal place precision.
- Example 3: ↧. The Depth/Deep ↧ symbol with a 1.50 dimension associated with the hole. The hole Ø.562 has a three decimal place precision.
- Example 4: *A+40*. A is provided to you on the exam. 44mm + A.

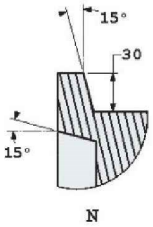


N is a Detail view of the M-M Section view.

- Example 5: *ØB*. Diameter of B. B is provided to you on the exam.
- Example 6: ⫽. Parallelism.





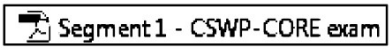


Build a part from a detailed illustration

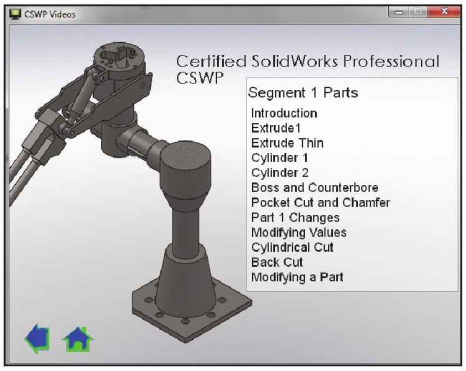
Segment 1 of the CSWP CORE exam - First question

Segment 1 of the CSWP CORE exam is one of three sections. In it, you are asked to create and modify a single part. Segments 2 and 3 of the exam require you to download part and component files using the testing client. The question setup in segment 1 is similar to the sample test posted by SolidWorks. Below is the needed information and steps to correctly create the provided model in the sample exam.

View the provided pdf file if needed for this segment.



Utilize segment videos and model folders to follow along while using the book. Each segment video provides valuable information to successfully pass the CSWP CORE exam.



Provided information:

Initial part - Stage 1: Build this part in SolidWorks.

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Part origin: Arbitrary

Material: Alloy Steel

Density: 0.0077 g/mm^3

All holes through all unless shown otherwise.

Use the following parameters and equations which correspond to the dimensions labeled in the images:

A = 213 mm

B = 200 mm

C = 170 mm

D = 130 mm

E = 41 mm

F = Hole Wizard Standard: ANSI Metric - Counterbore

Type: Hex Bolt - ANSI B18.2.3.5M

Size: M8

Fit: Close

Through Hole Diameter: 15.00 mm

Counterbore Diameter: 30.00 mm

Counterbore Depth: 10.00 mm

End Condition: Through All

X = A/3

Y = B/3 + 10mm

Hint #1: The dimensions that are to be linked or updated, and are variable, will be labeled with letters. Any dimensions that are simple value changes from one stage to another will be circled in the images.

Hint #2: To save the most time, make use of linked dimensional values and equations.

Measure the mass of the part.

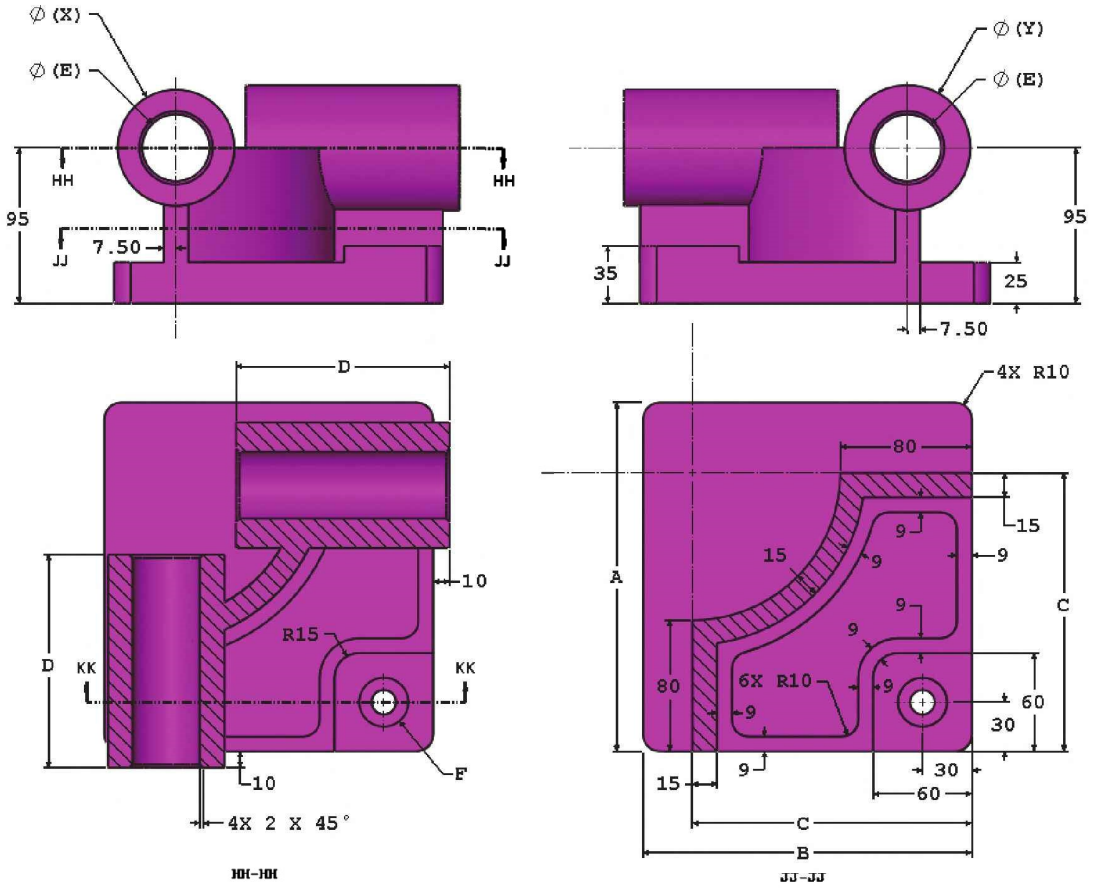
What is the mass of the part (grams)?

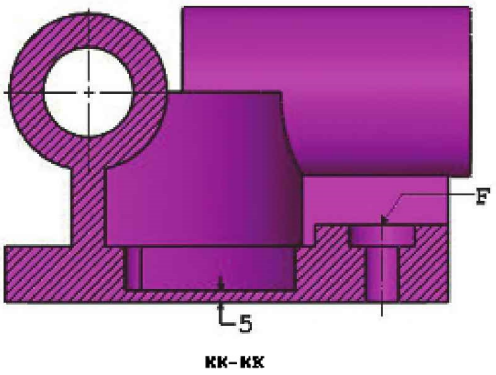
a) 14139.65

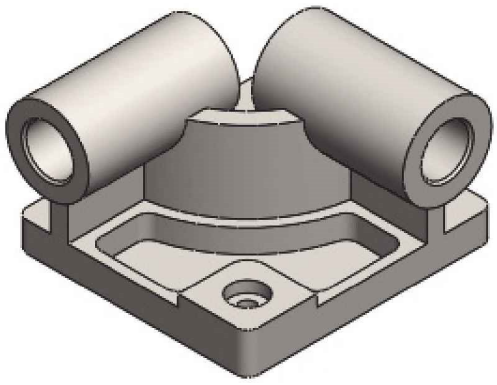
b) 14298.56

c) 15118.41

d) 14207.34





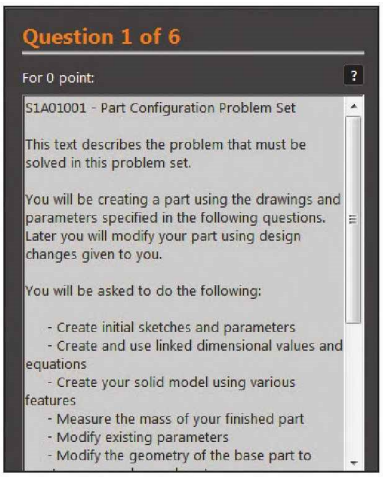


When you begin segment 1 of the CSWP CORE exam, you will be presented with a variety of drawings and parameters specified in the question. Take your time to first identify the drawing views and to better understand the provided geometry that is needed to create the initial part.

When you create the initial part, think about using Global variables, equations or design tables. This is crucial in this section of the book due to time constraints.

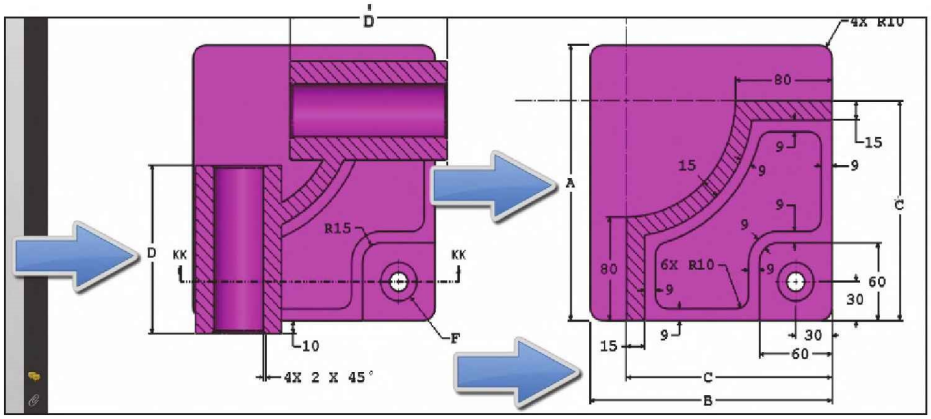
In this section when using SW 2013 & 2014, I use Global Variables and equations. I stay away from design tables.

Note where the variables (A, B, C, D & E) are located in the provided drawing views. F is a Hole Wizard hole.

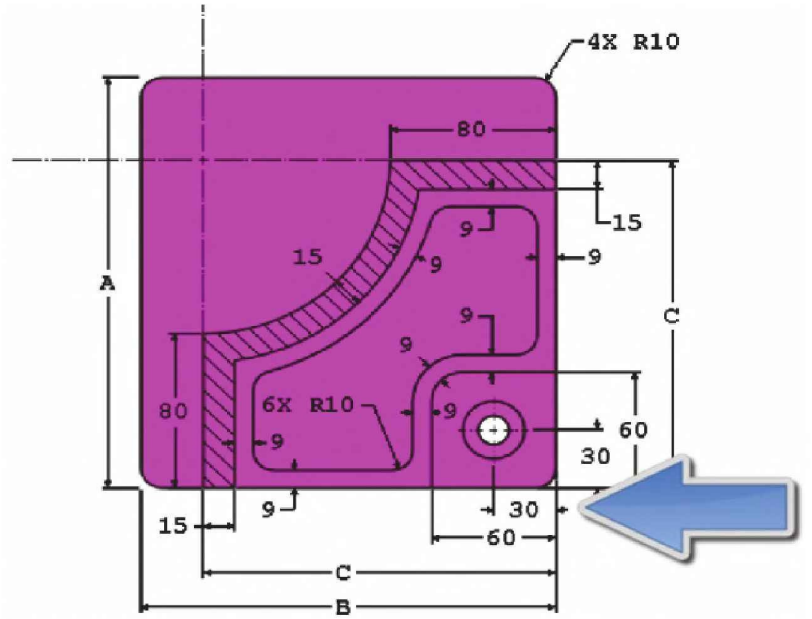


Actual CSWP exam format

You will see anywhere between 5 to 6 variables and 2 equations in this segment of the exam.



Observe where the dimensions are referenced. In the illustrated example, the dimensions are referenced from the lower right-hand corner. This is where you should begin with the origin of the rectangle for the Base Sketch (Sketch1) in this example.

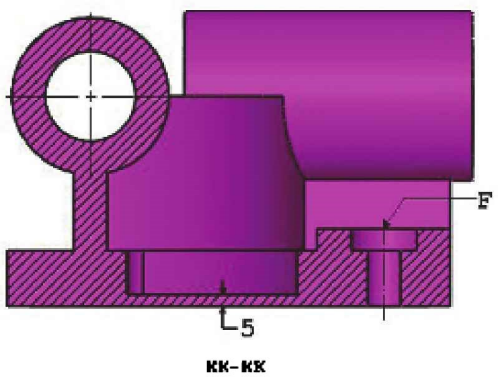


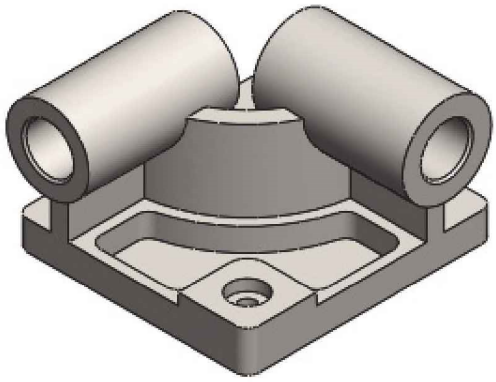
Most questions ask for a center of gravity or mass in grams. For the former it is crucial that the model is oriented exactly as illustrated. For the latter it does not matter as much, but do not take the chance.

SolidWorks previously used a tool called the Link Value, but this is no longer used. For users familiar with older versions of SolidWorks (2012 or older), the Link Value was what we now think of as a Global Variable. View the provided video on this section to learn more about Link Values.

The text in this section is focused on SolidWorks 2013 and 2014 using Global Variables with the new additional equation drop-down menus in the Modify dialog box and in the Depth box in the PropertyManager.

A Global Variable is just a name assigned to a dimension, a reference measurement, or an entire equation. Global Variables are assigned in the Equations dialog box or in the Modify dimension box as simply the variable name equaling an expression of a value.





Be aware of what is symmetrical and what is different. The two cylinders look the same, but they are not. They have different diameters. It would not make sense to create a pattern or to mirror the two.

Look at the provided variables. Ask yourself what requires a Global Variable in the Equation folder to address speedy part modification in a timed exam?

Understand the provided Hole type.

Based on the provided information, create an Equation folder and input the provided variables.

SolidWorks displays a circle, ellipse or square around the areas and features that require modification from the original part.

Remember, the purpose of this book is not to educate a new or intermediate user on SolidWorks; but to inform you on the types of questions, layout and what to expect when taking the three segments of the CSWP CORE exam.

In this section, you can address part modification through an Equation folder using Global Variables, Equations or by using a design table.

Perform the procedure that you are the most comfortable with.

A = 213 mm

B = 200 mm

C = 170 mm

D = 130 mm

E = 41 mm

F = Hole Wizard Standard: Ansi Metric Counterbore

Type: Hex Bolt - ANSI B18.2.3.5M

Size: M8

Fit: Close

Through Hole Diameter: 15.00 mm

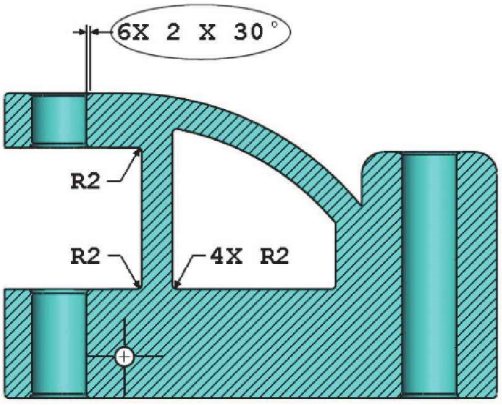
Counterbore Diameter: 30.00 mm

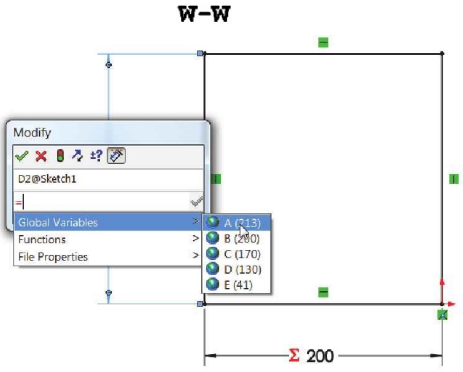
Counterbore Depth: 10.00 mm

End Condition: Through All

X = A/3

Y = B/3 + 10mm





The Modify dialog box can accept equations (SW 2013 or newer). You can also use it to create on-the fly Global Variables. To start an equation in the Modify box, start by replacing the numeric value with an = sign in the value box. When you do this, you will see a drop-down menu for functions and file properties. You can also use the dimension entry boxes in the PropertyManager.

Let's begin.

1. **Create** a folder to save your models.
2. **Create** a new part.
3. **Set** document properties (drafting standard, units and precision) for the model.

Start with setting the Global variables. The provided variables are:

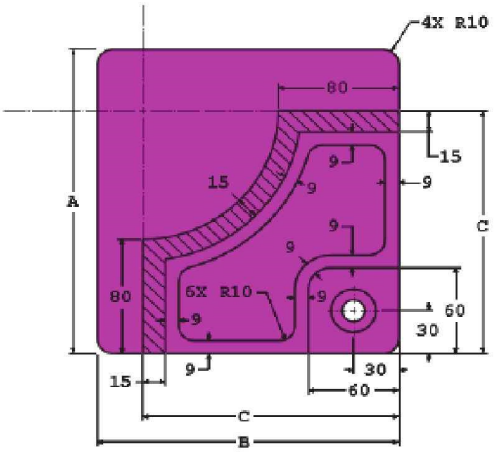
A = 213 mm

B = 200 mm

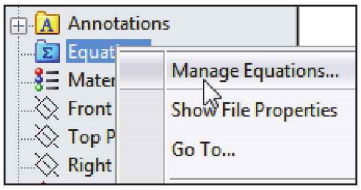
C = 170 mm

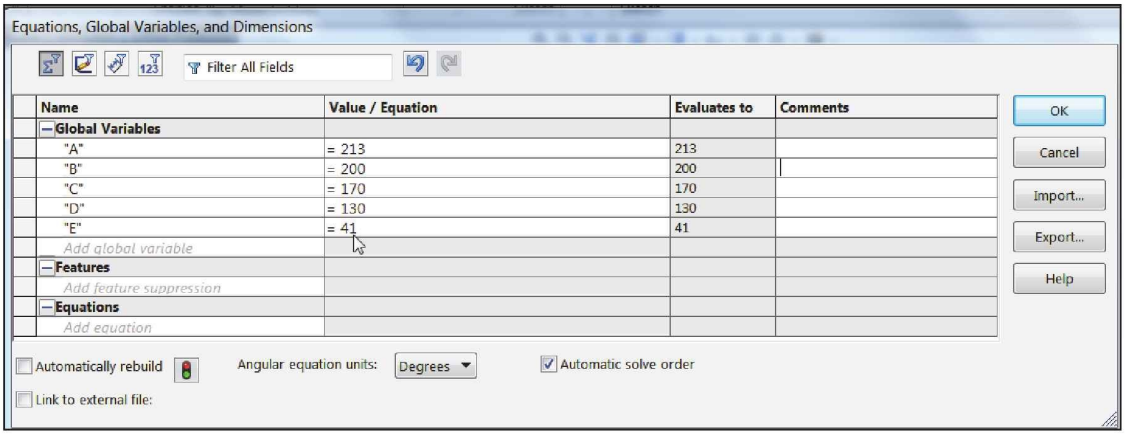
D = 130 mm

E = 41 mm

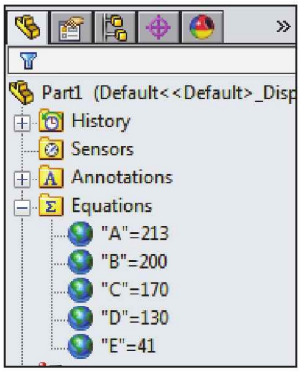


4. **Display** the Equation folder in the FeatureManager.
5. **Display** the Equations, Global Variables, and Dimension dialog box.
6. **Enter** the five Global Variables (A, B, C, D & E) as illustrated.





7. **Exit** the dialog box.

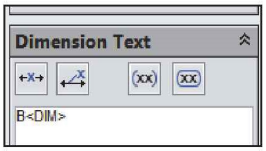


View the created Global Variables.

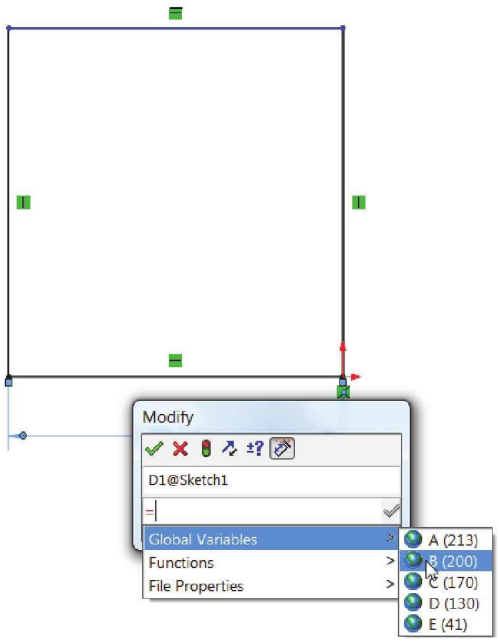
8. **Expand** the Equation folder in the FeatureManager.

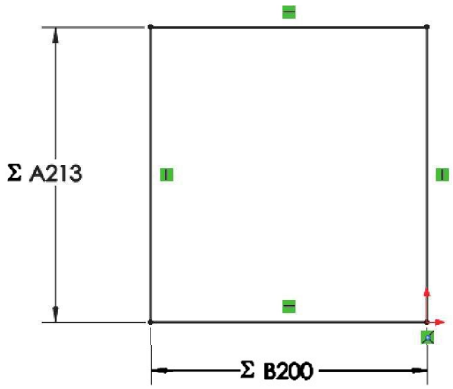
Next create the Base Sketch.

9. **Create** Sketch1. Select the Top Plane as the Sketch plane. Sketch1 is the profile for the Extruded Base (Boss-Extrude1) feature. Apply the Corner Rectangle Sketch tool. Click the origin and a position in the upper left section of the Graphics window. Most of the dimensions in the provided drawing view are referenced from this location.
10. **Insert** the horizontal dimension using the Global Variable B(200) from the Modify dialog box.

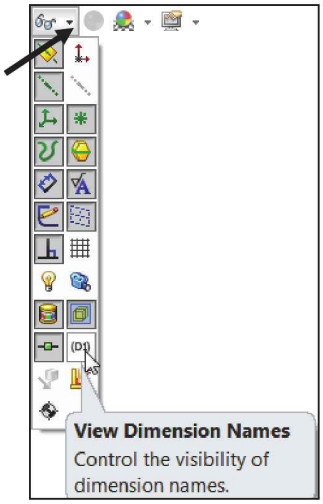


11. **Enter** B for Dimension Text. This will help you keep track of the variables.
12. **Insert** the vertical dimension using the Global Variable A(213) from the Modify dialog box. Sketch1 is fully defined.
13. **Enter A** for Dimension Text.





14. **Display** Primary values. Click View Dimension Names from the Heads-up toolbar. View the results in the Graphics window.



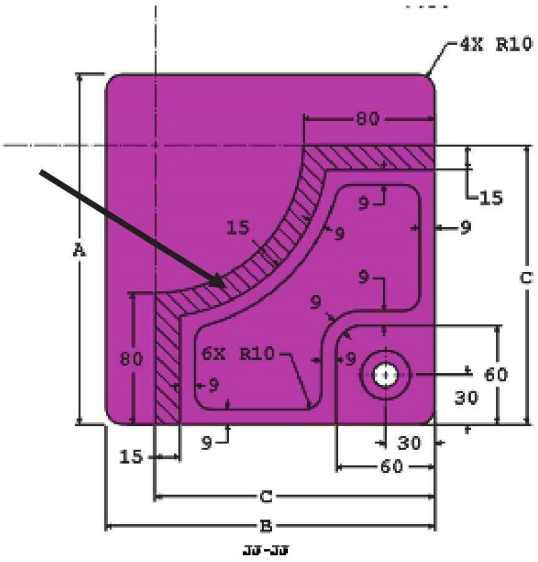
15. **Create** the Extruded Base feature. Boss-Extrude1 is the Base feature. Blind is the default End Condition in Direction 1. Depth = 25mm. Note the direction of the extrude feature.

16. **Assign** Alloy Steel material to the part.

Create the Extrude-Thin feature. The Extrude-Thin feature is controlled by the variable C and two dimensions that are equal (80mm). The height of the feature from the bottom is 95mm.

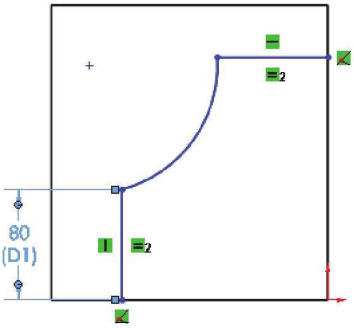
Start from the Top face and subtract 25mm from 95mm to obtain the correct depth for the Thin-Extrude feature.

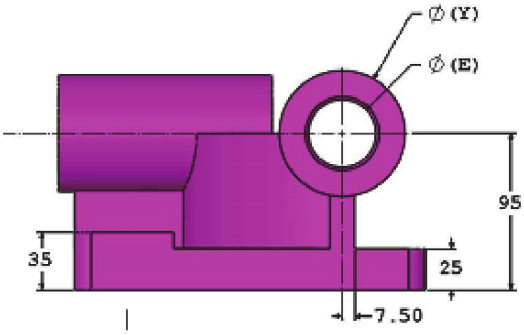
17. **Create** Sketch2. Select the Top face. Sketch2 is the profile for the Extrude-Thin feature. Utilize the Line Sketch tool to create a vertical and horizontal line. Utilize the 3 Point Arc Sketch tool to complete the sketch as illustrated.



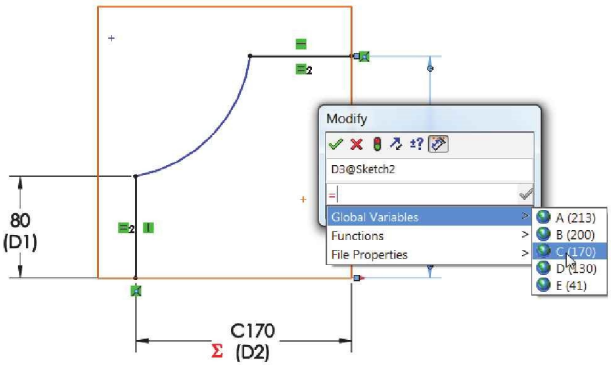
18. **Add** an Equal relation between the vertical and horizontal line of Sketch2.

19. **Insert** the 80mm dimension on the vertical line.

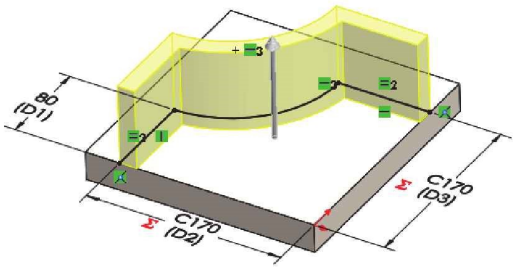


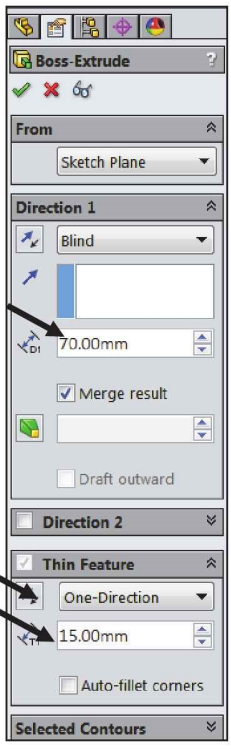


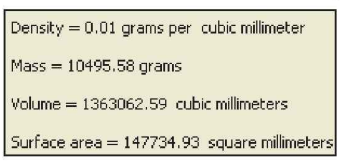
20. **Insert** the horizontal Global Variable C dimension as illustrated.
21. **Enter** C for Dimension Text.
22. **Insert** the vertical Global Variable C dimension as illustrated.
23. **Enter** C for Dimension Text.
24. **Insert** a horizontal relation between the centerpoint of the arc and the horizontal sketch line end point if needed. The sketch is fully defined and is displayed in black.



25. **Insert** the Extrude-Thin feature. The Extrude-Thin feature is controlled by the Global Variable C and two dimensions that are equal (80mm). The overall height of the part is 95mm. Start from the Top face. Subtract 25mm from 95mm to obtain the correct depth for the Thin-Extrude feature. Click No in the Close Sketch With Model Edges dialog box. You want an open profile. If needed click the Reverse Direction in the Thin Feature box. Enter 15mm for Thickness. Enter 70mm for Depth in the Direction 1 box. Blind is the default End Conditions. Click OK from the Boss-Extrude PropertyManager. Extrude-Thin1 is displayed in the FeatureManager.



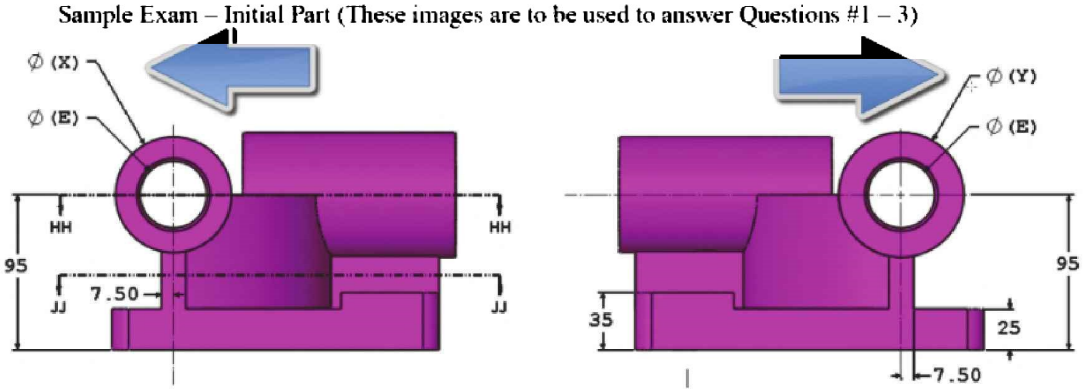




At this time - your model should have a mass of **10495.58 grams**. You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question.

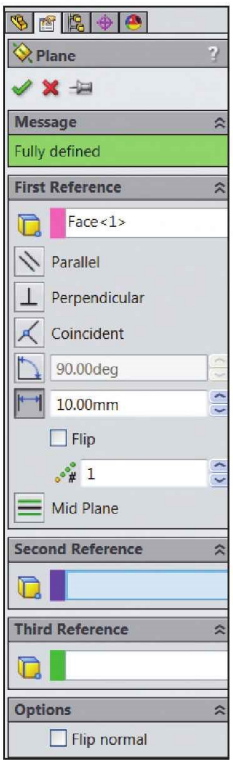
Create the first cylinder (remember the two cylinders are not the same). The first cylinder outside diameter is controlled by equation X. X = A/3. The inside diameter is E = (41mm) and the depth is D = (130mm).

Sample Exam – Initial Part (These images are to be used to answer Questions #1 – 3)

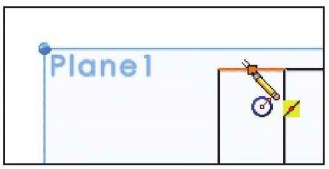


The first cylinder is offset 10mm from the Front Plane or face.

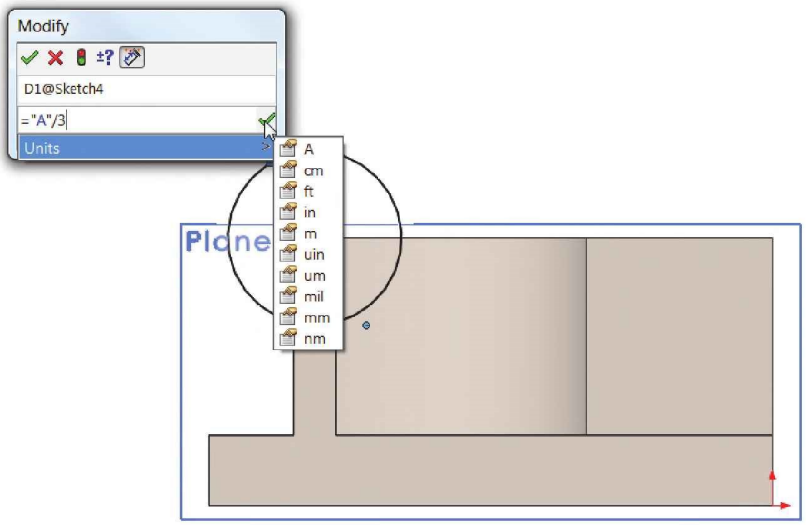
26. **Create** a Plane offset from the Front Plane (front face) 10mm. Plane1 is created.



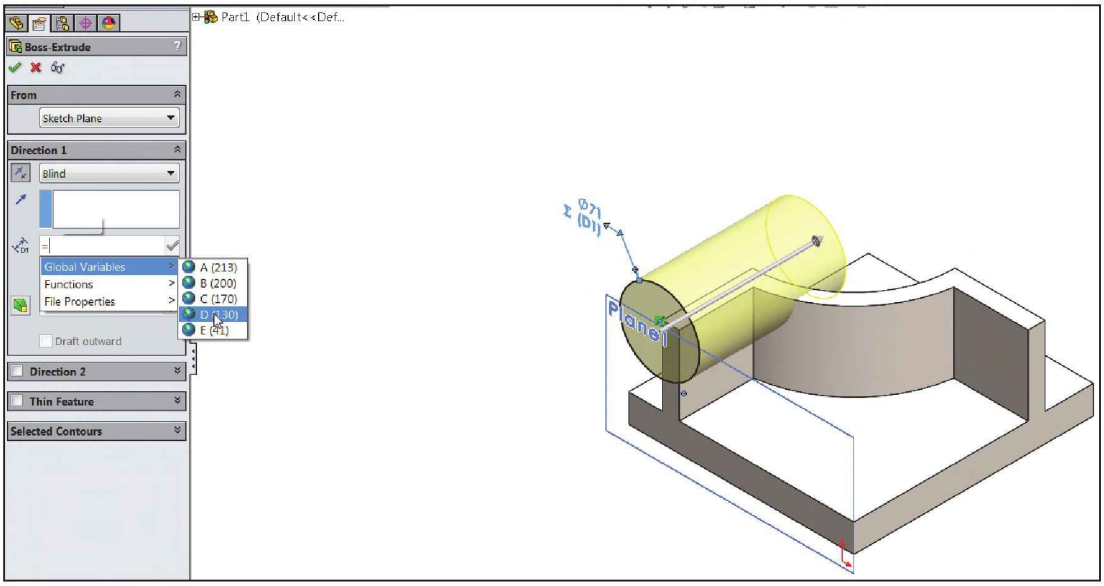
27. **Create** Sketch3 on Plane1. Sketch a circle with the centerpoint Coincident at the midpoint of the Extrude-Thin1 feature. The dimension is driven by the X equation.



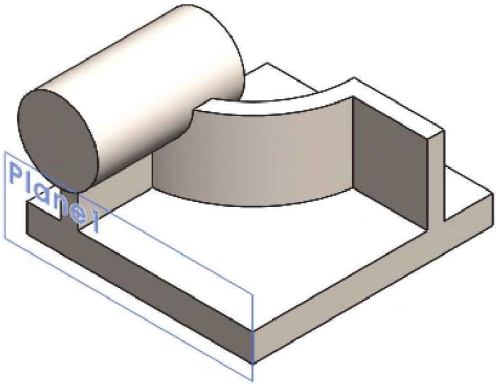
28. **Enter** the equation for X in the Modify dialog box as displayed.



29. **Create** the Extruded Boss feature from Sketch3. Enter the extruded distance of the Global Variable D. Depth D = 130mm. Click Reverse direction if needed as illustrated.



30. **Rebuild** your part. View the results in the Graphics window.

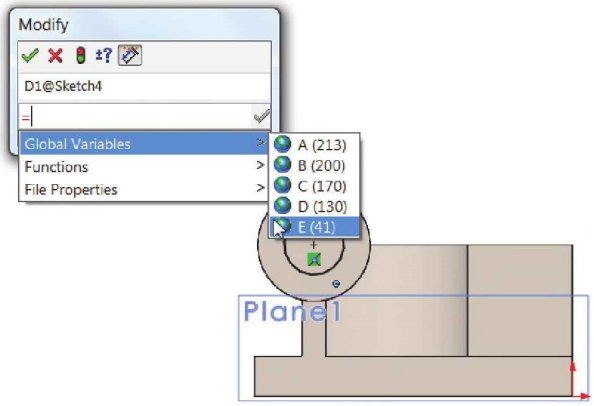


Prior to SolidWorks 2013, a method commonly used to define variables for CSWP exam problems was to use Linked Values. Linked Values, also called linked dimensions, connect two or more dimensions without using equations or relations. Linked Values can still be used today in SolidWorks 2013 and SolidWorks 2014. The method to apply Linked Values is described in videos included in the book's DVD.

For SolidWorks 2013 and SolidWorks 2014, the method of using Global Variables to define variables for the CSWP exam problems is documented in the book. Global Variables are easier to find and change in the CSWP exam problems.

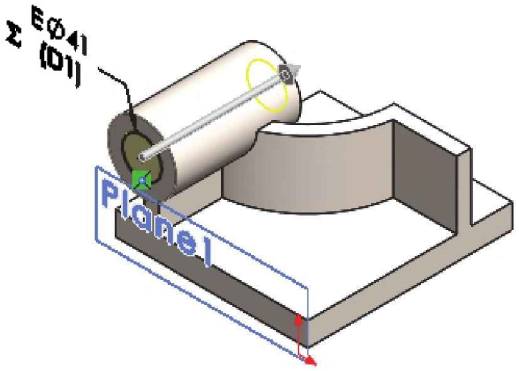
There are numerous ways to build the model in this section. A goal is to display different design intents and techniques.

31. **Create** Sketch4 on the front face of the cylinder to create the Extruded Cut feature. Use the Circle Sketch tool. Click the centerpoint of the Extruded Boss feature for the centerpoint.
32. **Dimension** Sketch4. Insert the Global Variable E. E = 410mm.
33. **Enter** E for Dimension Text.



There are numerous ways to build the model in this section. A goal is to display different design intents and techniques.

34. **Create** an Extruded Cut feature using Sketch4. Select Through All for End Condition to address any future design change for depth.



At this time, your model should have a mass of **12722.39 grams**. You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question.

Create the second cylinder, (remember the two cylinders are *not* the same).

The second cylinder outside diameter is controlled by equation Y.

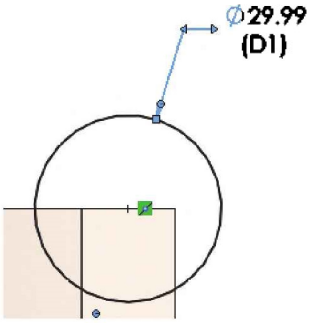
Y = B/3 +10mm.

The inside diameter is E = (41mm).

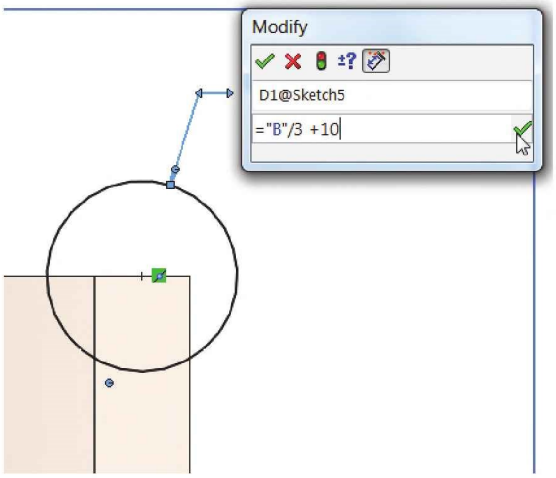
The depth is D = (130mm).

The second cylinder is offset 10mm from either the Right Plane or right face.

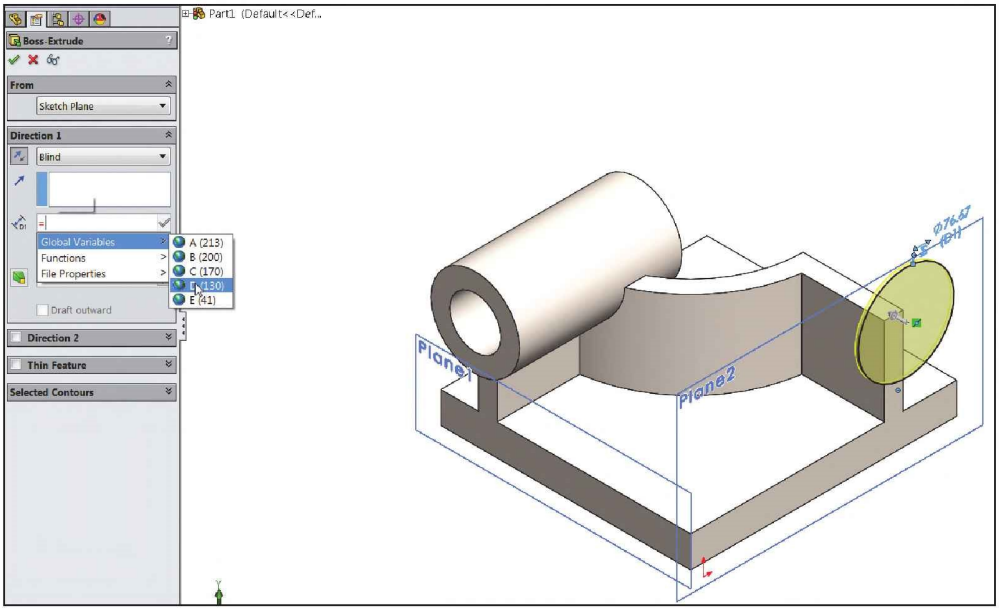
35. **Create** a Plane (Plane2) offset from the Right Plane (right face) 10mm.
36. **Create** Sketch5 on Plane2. Sketch a circle with the centerpoint Coincident at the midpoint of the Extrude-Thin1 feature. The dimension is driven by the Y equation.



37. **Dimension** Sketch5. Enter the equation for Y in the Modify dialog box as displayed.

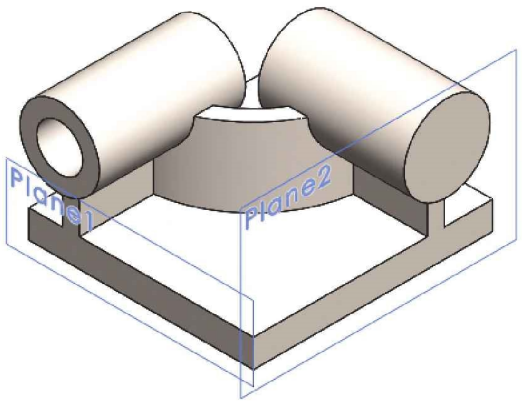


38. **Create** the Extruded Boss feature from Sketch5. Enter the extruded distance of the Global Variable D. Depth D = 130mm. Click Reverse direction if needed as illustrated.

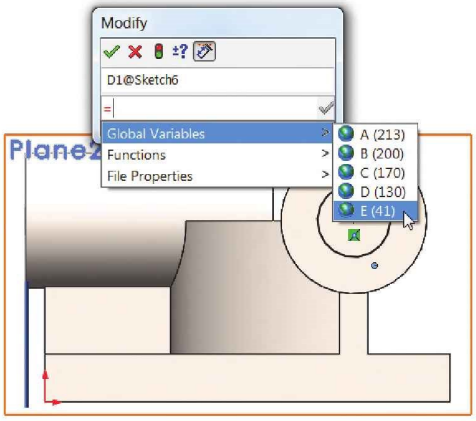


39. **Rebuild** your part. View the results in the Graphics window.

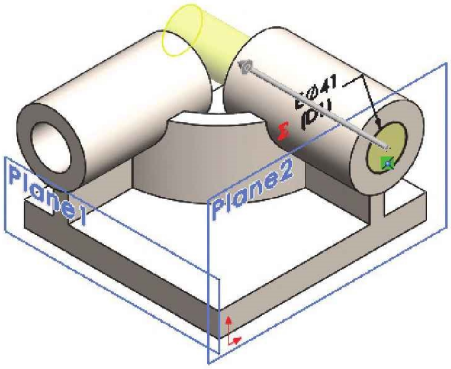
40. Save the part.



41. **Create** Sketch6 on the Right face of the second cylinder to create the Extruded Cut feature. Use the Circle Sketch tool. Click the centerpoint of the Extruded Boss feature for the centerpoint.
42. **Insert** the Global Variable E. E = 41mm.
43. **Enter** E for Dimension Text.
44. **Create** an Extruded Cut feature from Sketch6. Select Through All for End Condition to address any future design change in depth.

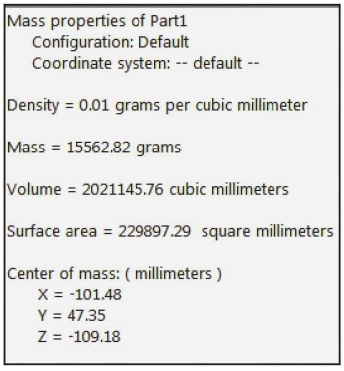


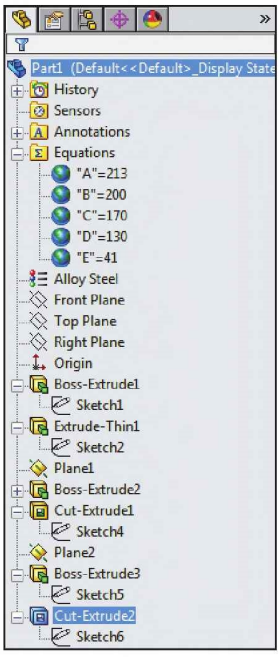
At this time, your model should have a mass of **15562.83 grams**.



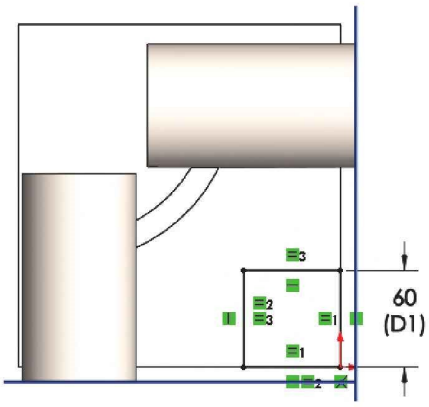
Always enter the needed decimal places in the answer field.

Each segment video provides valuable information to successfully pass the CSWP CORE exam.





Create the Extruded Boss, Counterbore and Fillet features on the right side of the model.

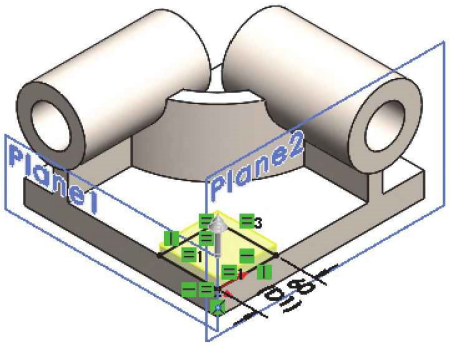


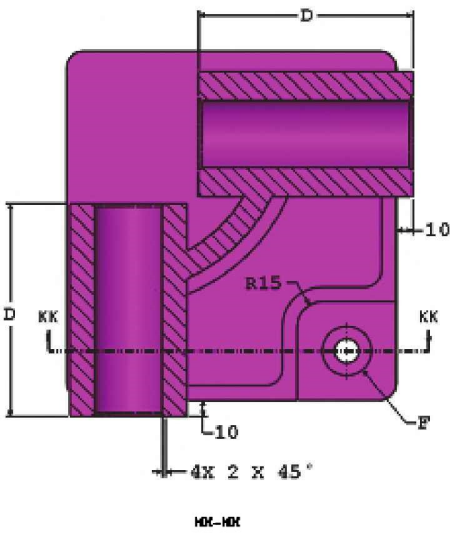
45. **Create** Sketch7 on the top face of Boss-Extrude1. Apply the Corner Rectangle tool from the right hand side. Insert an Equal geometric relation. Enter 60mm for dimension.

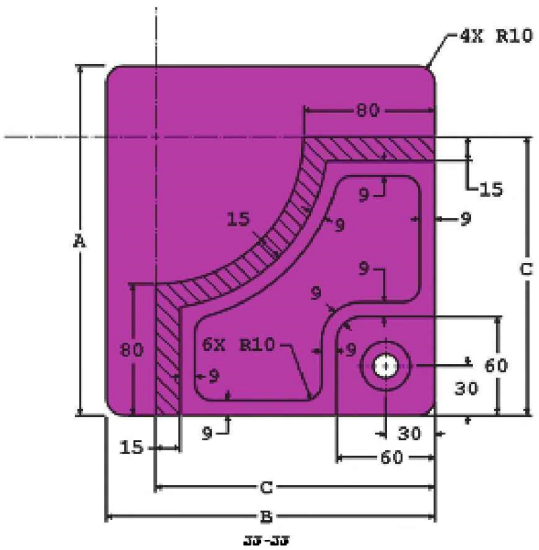
46. **Create** Boss-Extrude4 from Sketch7. Depth = 10mm. (35mm - 25mm) = 10mm from the provided information in the question.

Create the Counterbore hole using the Hole Wizard feature tool. Remember the provided information.

The Hole Wizard feature tool was modified in SolidWorks 2014. Additional Hole Types and other features were added.



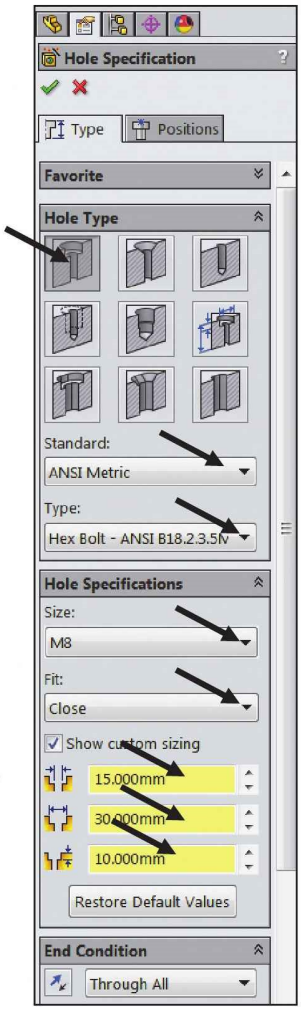




47. **Create** the Counterbore hole using the Hole Wizard on the top face of Boss-Extrude4.

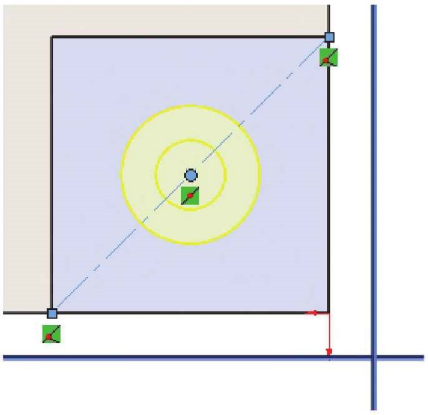
There are numerous ways to build the models in this chapter. A goal is to display different design intents and techniques.

48. **Enter** the provided information in the Hole Specification PropertyManager. This is a Through All End Condition hole.



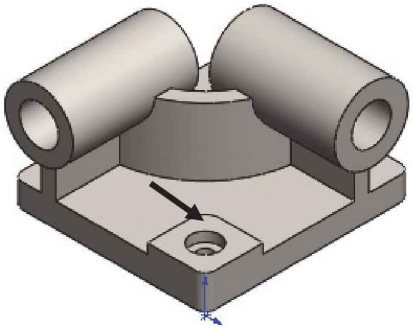
49. **Create** Sketch9. Use the Centerline Sketch tool with a Midpoint relation to location the center of the hole on the Boss-Extrude4 face. This saves time from creating two dimensions. The hole is complete.

50. **Create** the first Constant radius Fillet (15mm) on the inside corner of the Boss-Extrude4 feature. You can use the Multiple radius fillet option to create all needed fillets, but for design intent and future modifications in the exam; insert two separate fillet features.

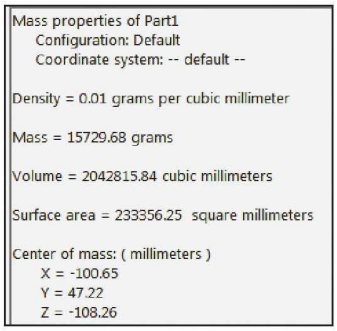


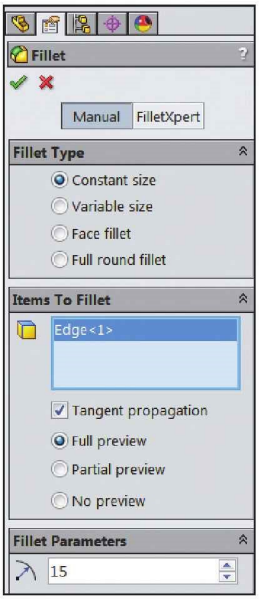
Use caution when taking the CSWP exam. Fillets are modified often. The Fillet PropertyManager was modified in SW 2013 & 2014.

51. **Create** the second Constant radius Fillet (10mm) on the four outside edges of Boss-Extrude1 as illustrated.



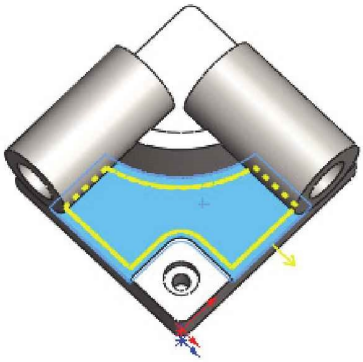
At this time, your model should have a mass of **15729.68 grams**.

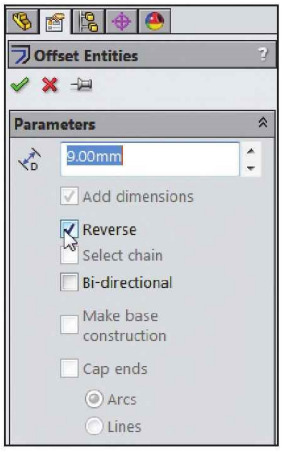




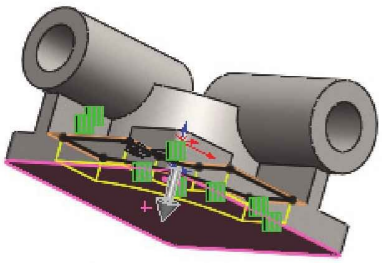
Create the Extruded Cut feature (pocket) on the top face of Boss-Extrude1 using the Offset Entities Sketch tool; then create the Chamfer feature.

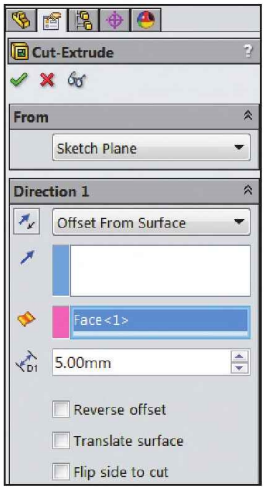
52. **Create** Sketch10. Use the Offset Entities Sketch (9mm Offset distance) tool. Note the direction of the offset.



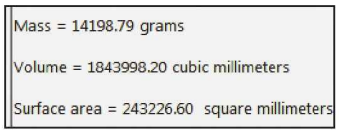


53. **Create** the Cut-Extrude3 feature based on the bottom face of the model. Utilize the Offset from Surface End Condition. Enter 5mm from the provided model information.

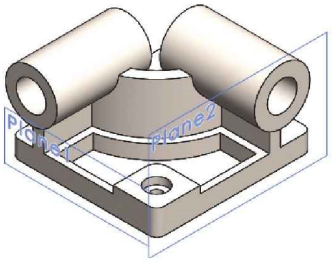




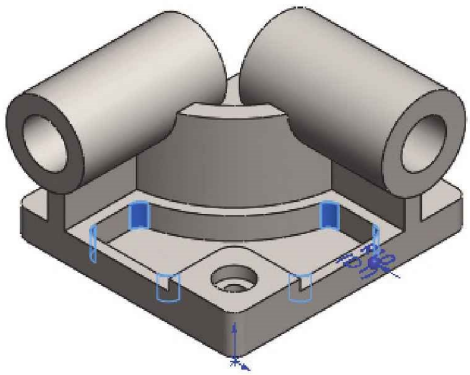
At this time, your model should have a mass of **14198.79** grams.



If you use faces vs. dimensions when creating features, this can help when you need to create a design change to maintain the design intent of the part.

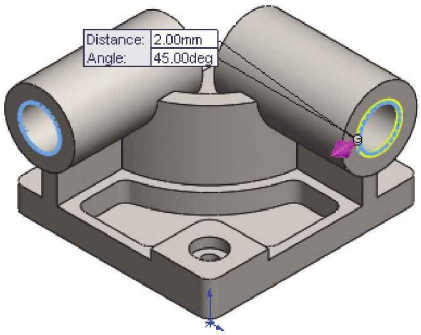


54. **Insert** six (6) Constant radius Fillet features (10mm) on the Cut-Extrude3 feature per the provided information.



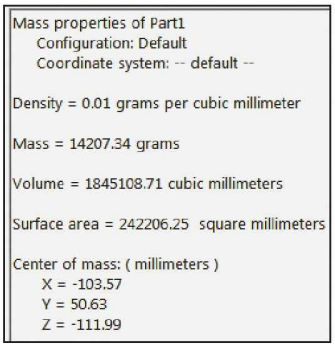
Insert the last feature for the part - Chamfer1.

55. **Create** a Chamfer (Angle distance) feature on the internal diameter edges of the cylinders. Angle = 45. Distance = 2mm. Note: select four edges.



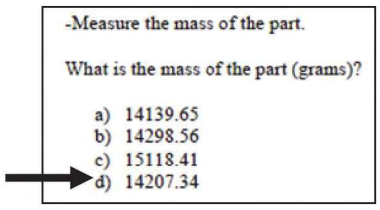
56. **Calculate** the Mass Properties of the model.

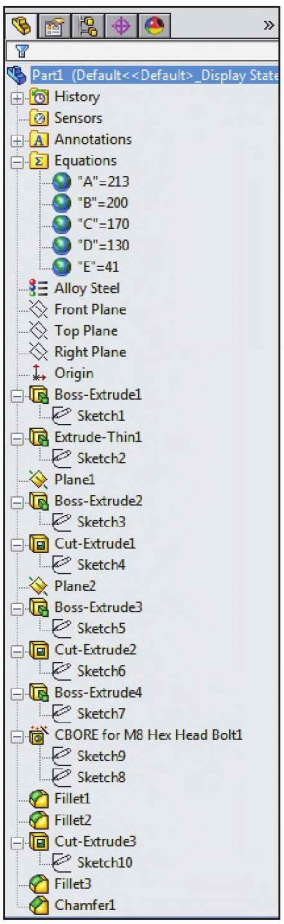
57. **Select 14207.34** grams for the answer in this section. The number matches the answer of d. You should be within 1% of the stated value before you move to the next section to modify the original part.



58. **Save** the part.

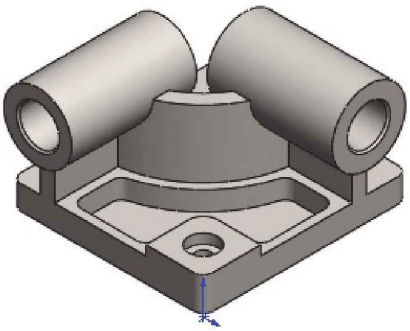
59. **Rename** Part1 to Part2 for the second question. You are finished with this section.





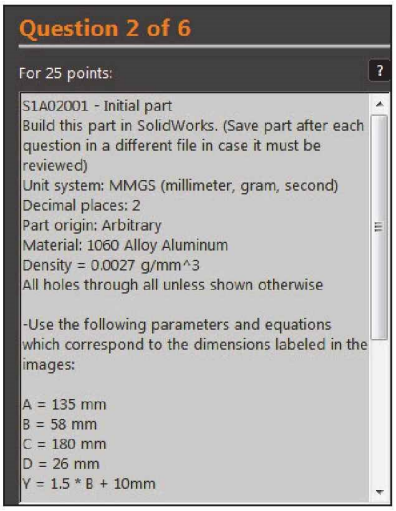
Always enter the needed decimal places in the answer field.

This section presents a representation of the types of questions that you will see in this segment of the exam.



At the time of this writing, there were slight variations in Mass Properties between SolidWorks 32 bit versus 64 bit. These variations are still within the 1% required for the CSWP exam segments. Always save your models to verify your results.

Segment 1 of the CSWP CORE exam - Second question



Actual CSWP exam format

In this section, modify the original part using Global Variables. The material and units are the same but the variables A thru E are different from their original values.

The Hole Wizard feature remains the same and the equation X and Y remain the same from the original part.

Read the question carefully. This section provides a single fill in the blank format, not a multiple choice format.

Provided Information:

Update parameters of the initial part.

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Part origin: Arbitrary

Material: Alloy Steel

Density: 0.0077 g/mm^3

All holes through all unless shown otherwise.

Use the following parameters and equations which correspond to the dimensions labeled in the images:

A = 225 mm

B = 210 mm

C = 176 mm

D = 137 mm

E = 39 mm

F = Hole Wizard Standard: ANSI Metric - Counterbore

Type: Hex Bolt - ANSI B18.2.3.5M

Size: M8

Fit: Close

Through Hole Diameter: 15.00 mm

Counterbore Diameter: 30.00 mm

Counterbore Depth: 10.00 mm

End Condition: Through All

X = A/3

Y = B/3 + 10mm

Hint #1: The dimensions that are to be linked or updated and are variable will be labeled with letters. Any dimensions that are simple value changes from one stage to another will be circled in the images.

Hint #2: To save the most time, make use of linked dimensional values and equations.

Measure the mass of the part.

What is the mass of the part (grams)?

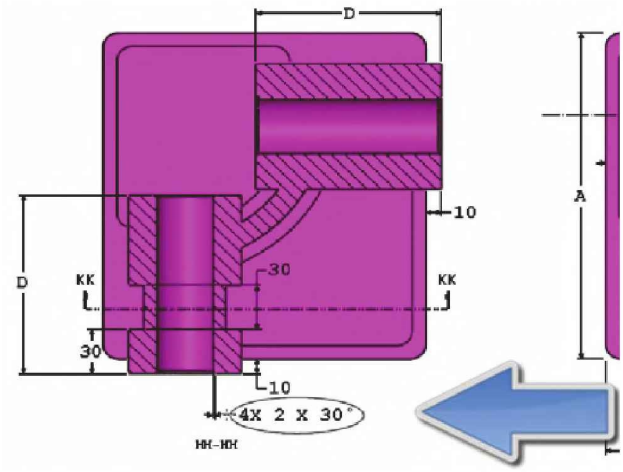
SolidWorks displays a circle, ellipse or a square around the areas and features that require modification from the original part.

This section presents a representation of the types of questions that you will see in this segment of the exam.

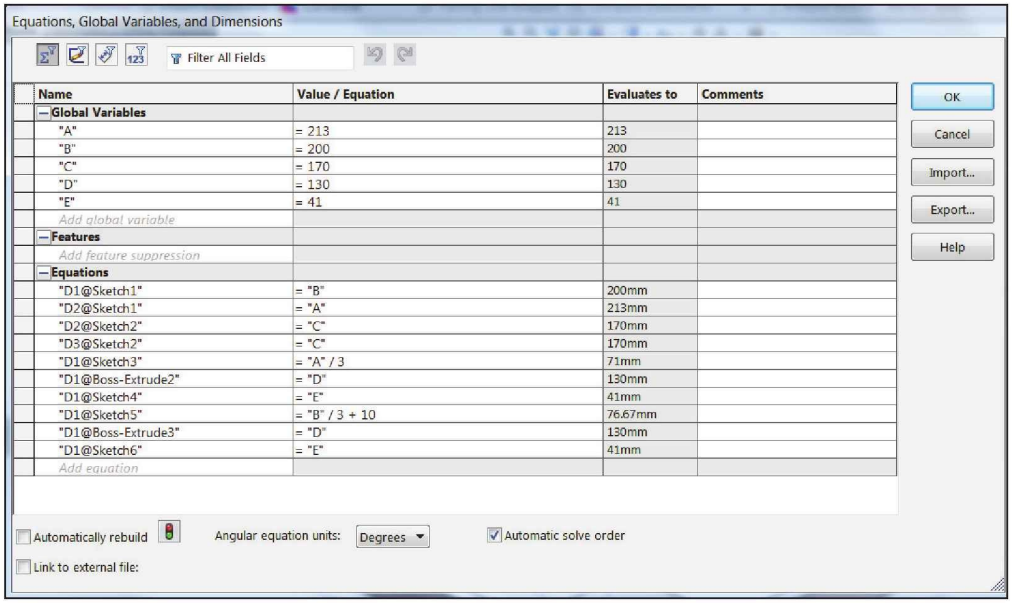
The images displayed on the exam are not to scale due to differences in the parts being downloaded for each tester.

Prior to SolidWorks 2013, a method commonly used to define variables for CSWP exam problems was to use Linked Values. Linked Values, also called linked dimensions, connect two or more dimensions without using equations or relations. Linked Values can still be used today in SolidWorks 2013 and SolidWorks 2014. The method to apply Linked Values is described in videos included in the book's DVD.

For SolidWorks 2013 and SolidWorks 2014, the method of using Global Variables to define variables for the CSWP exam problems is documented in the book. Global Variables are easier to find and change in the CSWP exam problems.



Segment 1 - CSWP-CORE exam



SolidWorks 2014 screen shot Part 1

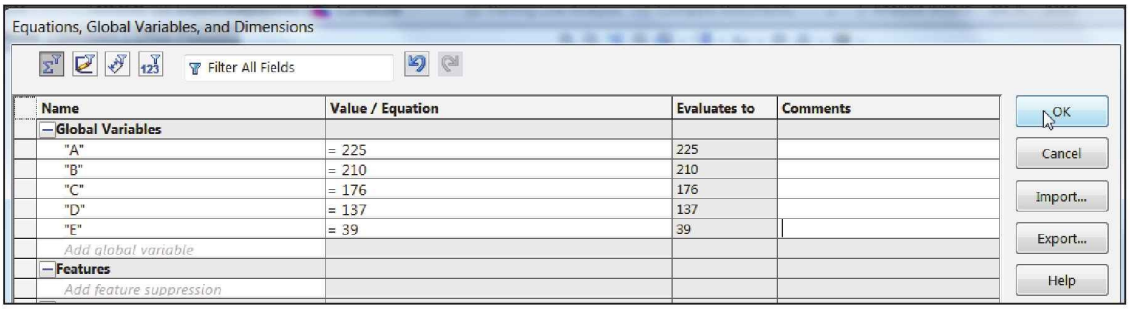
Let's begin.

Modify the variables that are different from Part1.

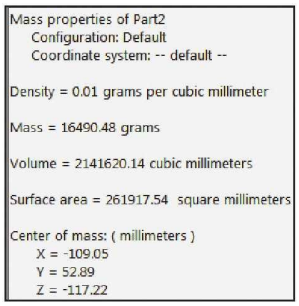
1. **Display** the Equations, Global Variables, and Dimension dialog box.
2. **Enter** the five Global Variables (A, B, C, D & E) as illustrated.

A = 213 mm	A = 225 mm
B = 200 mm	B = 210 mm
C = 170 mm	C = 176 mm
D = 130 mm	D = 137 mm
E = 41 mm	E = 39 mm

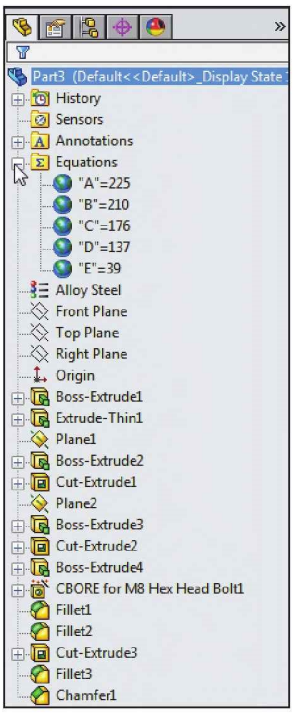
X = A/3
Y = B/3 + 10mm



Note you can double-click on a feature to modify the Global Variable.

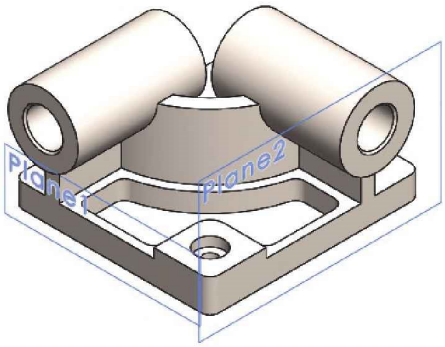


3. **Calculate** the mass of the model in grams.
4. **Enter 16490.48** grams. In the CSWP exam you will need to enter this number exactly. You need to be within 1% of the stated value in the single answer format to get this question correct.
5. **Save** the part.
6. **Rename** Part2 to Part3 for the third question of the exam.

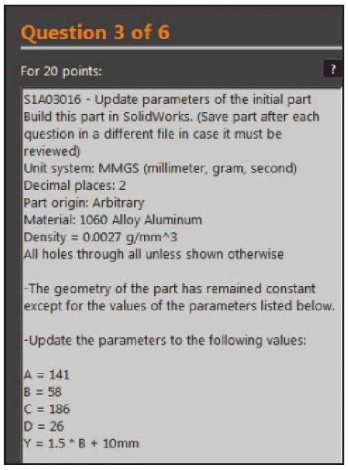


Always enter the needed decimal places (in this case 2) in the answer field.

You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question.



Segment 1 of the CSWP CORE exam - Third question



Actual CSWP exam format

Update various parameters of the part. Modify the dimensions under Global Variables. Read the question carefully. Identify what variables and equations are the same vs. different. Has the material changed? Did a feature change? A question will be presented to you in multiple steps and you need to get each step correct to get the question correct.

Provided Information:

Update parameters of the initial part.

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Part origin: Arbitrary

Material: Alloy Steel

Density: 0.0077 g/mm^3

All holes through all unless shown otherwise.

Use the following parameters and equations which correspond to the dimensions labeled in the images:

A = 209 mm

B = 218 mm

C = 169 mm

D = 125 mm

E = 41 mm

F = Hole Wizard Standard: ANSI Metric - Counterbore

Type: Hex Bolt - ANSI B18.2.3.5M

Size: M8

Fit: Close

Through Hole Diameter: 15.00 mm

Counterbore Diameter: 30.00 mm

Counterbore Depth: 10.00 mm

End Condition: Through All

X = A/3

Y = B/3 + 10mm

Hint #1: The dimensions that are to be linked or updated, and are variable, will be labeled with letters. Any dimensions that are simple value changes from one stage to another will be circled in the images.

Hint #2: To save the most time, make use of linked dimensional values and equations.

Measure the mass of the part.

What is the mass of the part (grams)?

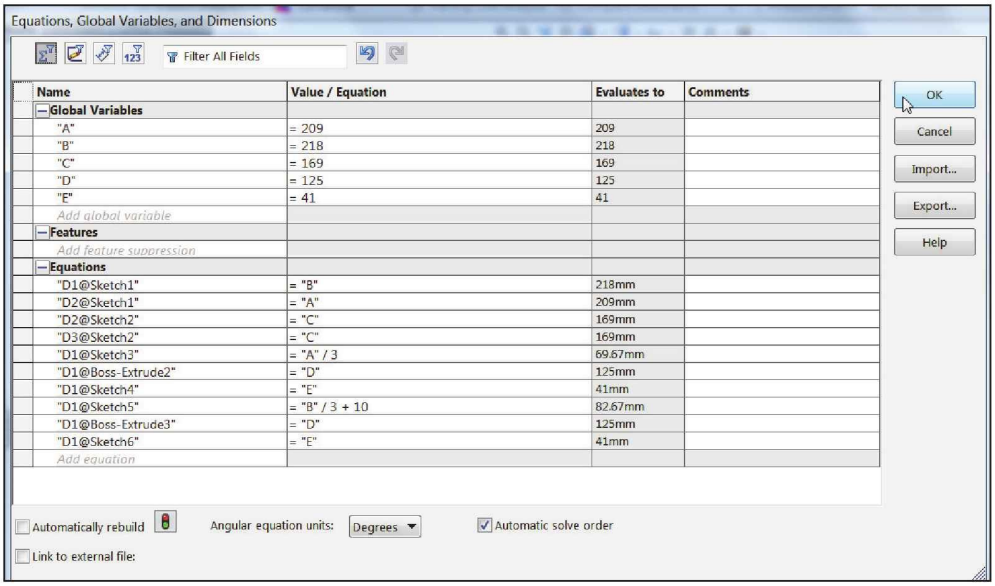
To display all dimensions, right-click Annotations folder from the FeatureManager and check the Display Annotations box.

Let's begin.

1. **Display** the Equations, Global Variables, and Dimension dialog box.
2. **Enter** the five Global Variables (A, B, C, D & E) as illustrated. Modify the variables that are different.

A = 225 mm	A = 209 mm
B = 210 mm	B = 218 mm
C = 176 mm	C = 169 mm
D = 137 mm	D = 125 mm
E = 39 mm	E = 41 mm

X = A/3
Y = B/3 + 10mm

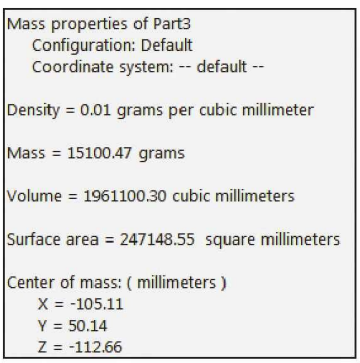


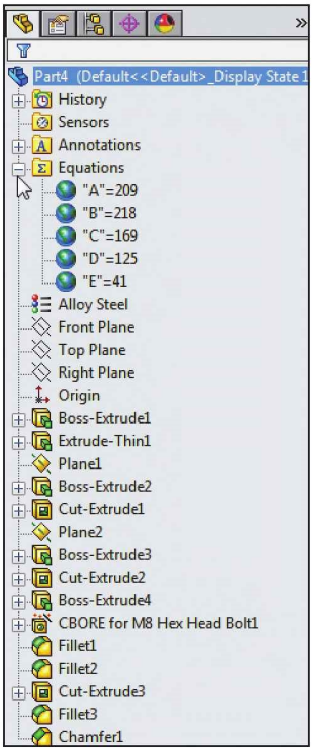
Name	Value / Equation	Evaluates to	Comments
Global Variables			
"A"	= 209	209	
"B"	= 218	218	
"C"	= 169	169	
"D"	= 125	125	
"E"	= 41	41	
Add global variable			
Features			
Add feature suppression			
Equations			
"D1@Sketch1"	= "B"	218mm	
"D2@Sketch1"	= "A"	209mm	
"D2@Sketch2"	= "C"	169mm	
"D3@Sketch2"	= "C"	169mm	
"D1@Sketch3"	= "A" / 3	69.67mm	
"D1@Boss-Extrude2"	= "D"	125mm	
"D1@Sketch4"	= "E"	41mm	
"D1@Sketch5"	= "B" / 3 + 10	82.67mm	
"D1@Boss-Extrude3"	= "D"	125mm	
"D1@Sketch6"	= "E"	41mm	
Add equation			

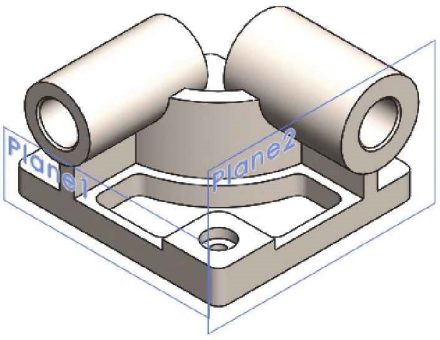
The equations (**X** = A/3, **Y** = B/3 + 10mm) have not been modified between the first, second or third question.

The CSWP exam in this section provides variables that either increase or decrease from the original part question. Design for this during the exam.

3. **Calculate** the mass of the model in grams.
4. **Enter 15100.47** grams.
5. **Save** the part.
6. **Rename** Part3 to Part4 for the fourth question of the exam.

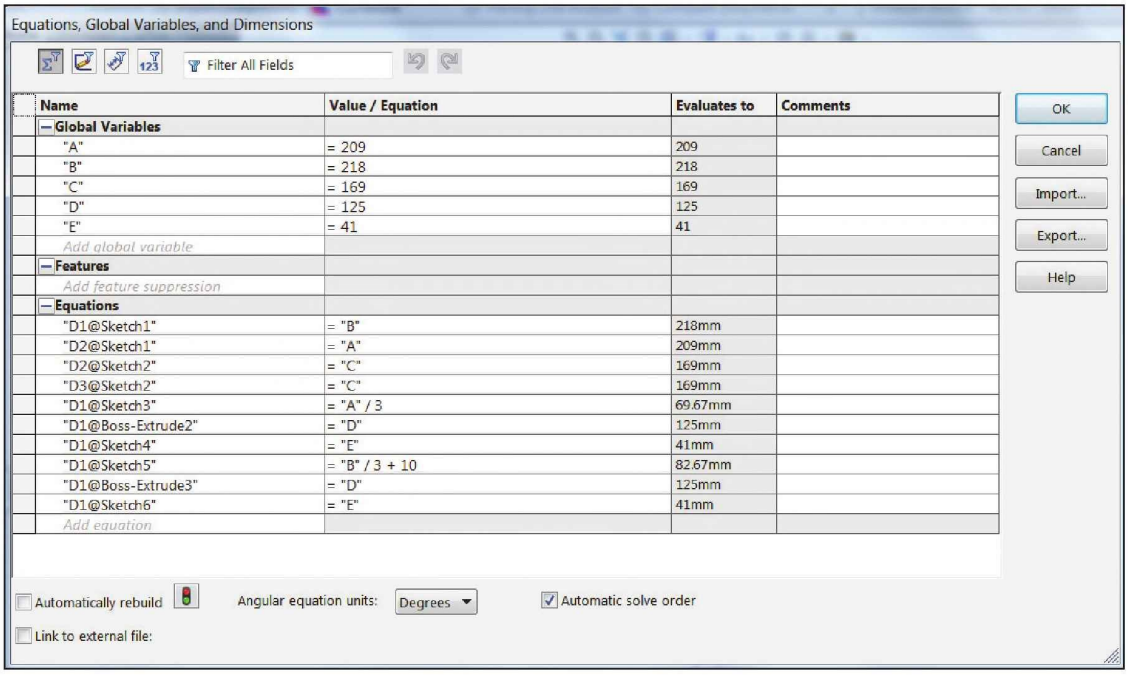






Always enter the needed decimal places (in this case 2) in the answer field.

You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question.



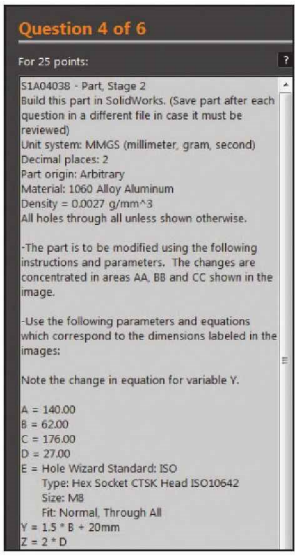
Name	Value / Equation	Evaluates to	Comments
Global Variables			
"A"	= 209	209	
"B"	= 218	218	
"C"	= 169	169	
"D"	= 125	125	
"E"	= 41	41	
Add global variable			
Features			
Add feature suppression			
Equations			
"D1@Sketch1"	= "B"	218mm	
"D2@Sketch1"	= "A"	209mm	
"D2@Sketch2"	= "C"	169mm	
"D3@Sketch2"	= "C"	169mm	
"D1@Sketch3"	= "A" / 3	69.67mm	
"D1@Boss-Extrude2"	= "D"	125mm	
"D1@Sketch4"	= "E"	41mm	
"D1@Sketch5"	= "B" / 3 + 10	82.67mm	
"D1@Boss-Extrude3"	= "D"	125mm	
"D1@Sketch6"	= "E"	41mm	
Add equation			

Segment 1 of the CSWP CORE exam - Fourth question

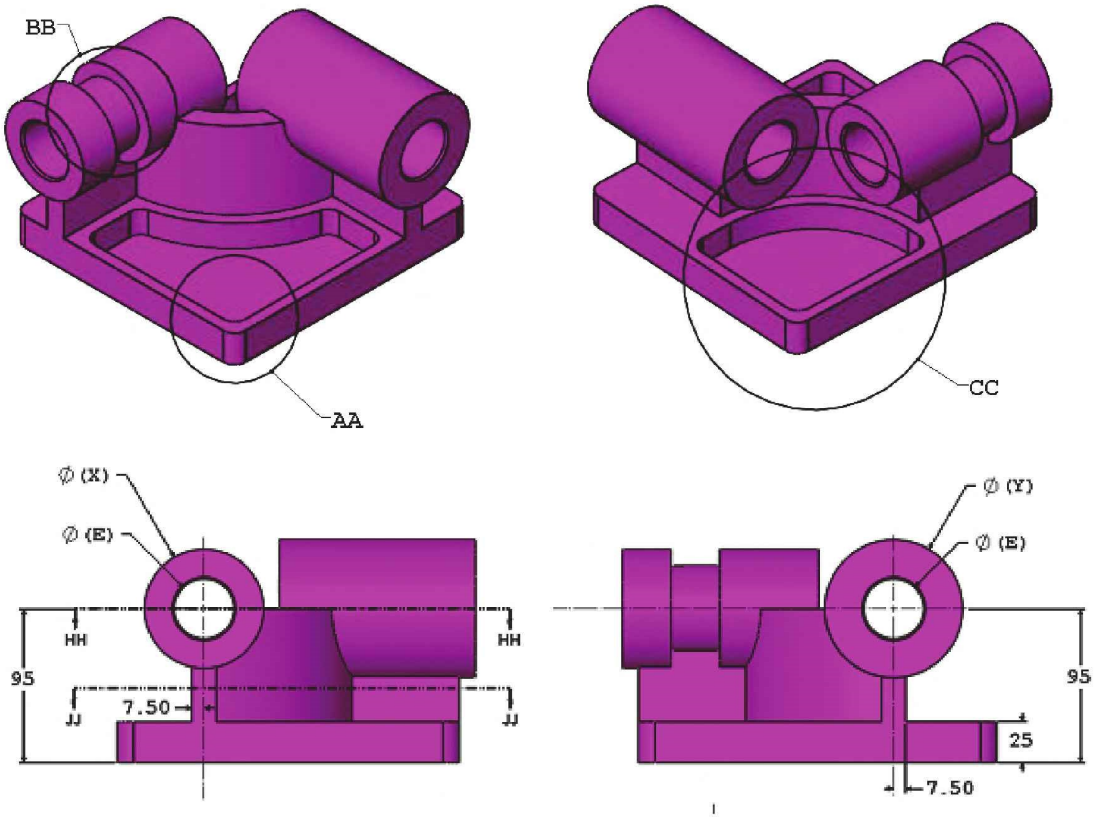
Stage 2: Modify the part using the following dimensions (These images are to be used to answer questions 4 and 5).

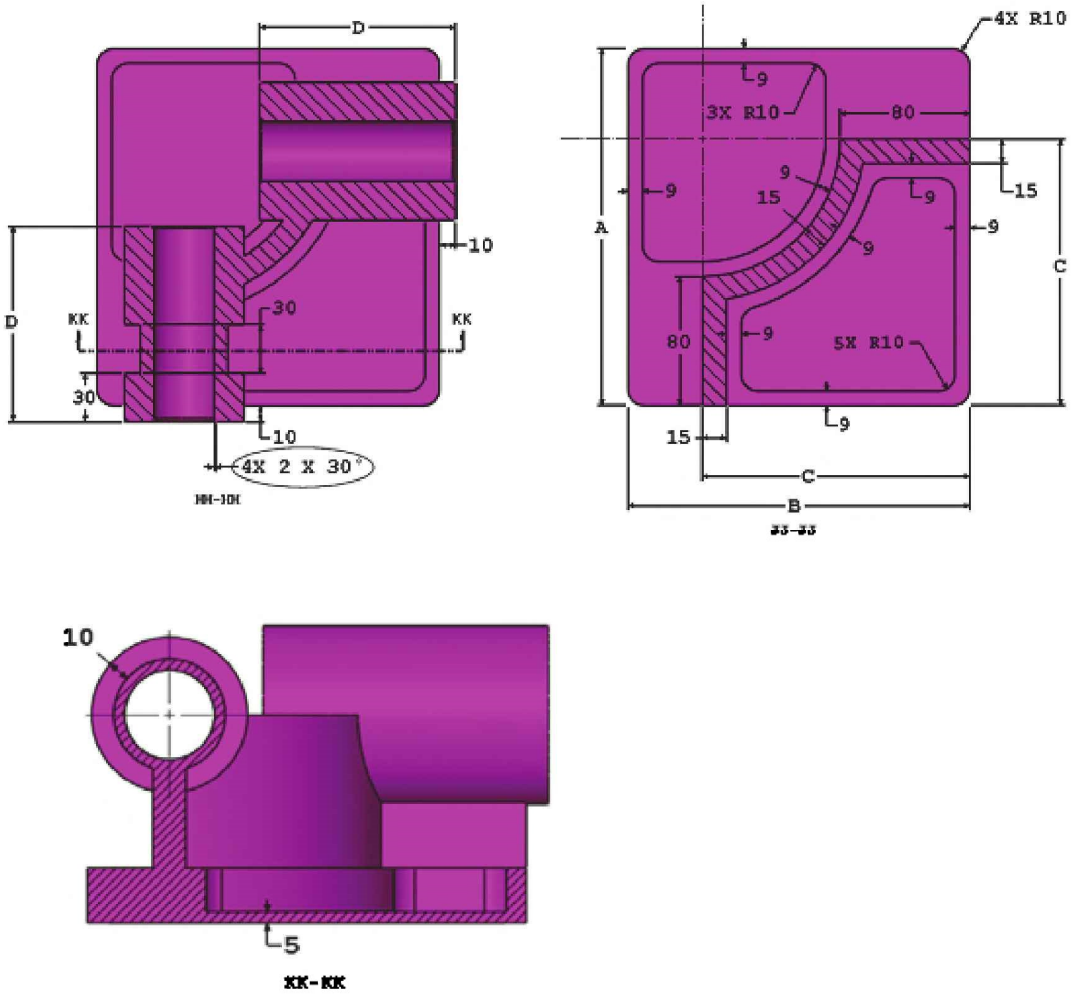
Note: The changes from the initial part are concentrated in areas AA, BB and CC shown in the first two images.

The needed modifications AA, BB, and CC are displayed with a circle, ellipse or a square in the CSWP exam. Compare the information with your existing part. This question provides a multiple choice answer.



Actual CSWP exam format





Provided Information:

4. Stage 2

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Part origin: Arbitrary

Material: Alloy Steel

Density: 0.0077 g/mm^3

All holes through all unless shown otherwise.

Use the following parameters and equations which correspond to the dimensions labeled in the images:

A = 221 mm

B = 211 mm

C = 165 mm

D = 121 mm

E = 37 mm

X = A/3

Y = B/3 + 15mm

Note: The equation for Y has changed from the initial part.

Hint #1: The dimensions that are to be linked or updated, and are variable, will be labeled with letters. Any dimensions that are simple value changes from one stage to another will be circled in the images.

Hint #2: To save the most time, make use of linked dimensional values and equations.

Measure the mass of the part.

What is the mass of the part (grams)?

a) 13095.40

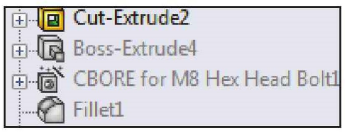
b) 13206.40

c) 13313.35

d) 13395.79

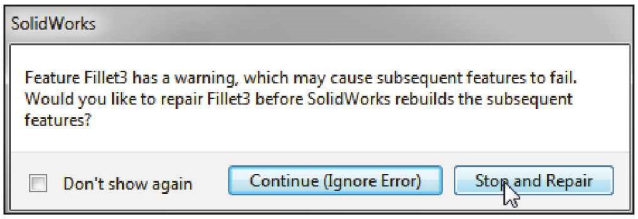
The material and units are the same. Equation A is the same. Variables A thru E are different. The Hole Wizard feature is removed with a few other features as illustrated. Equation Y is different. Read the question carefully.

Address AA modification in the model. Create a single pocket and remove the Hole Wizard feature along with the Boss-Extrude4 feature and a few fillets. Recover and repair from missing items.

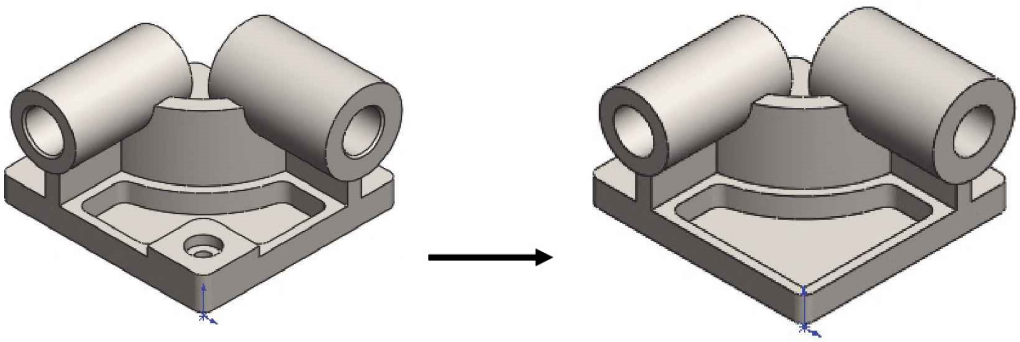


Let's begin.

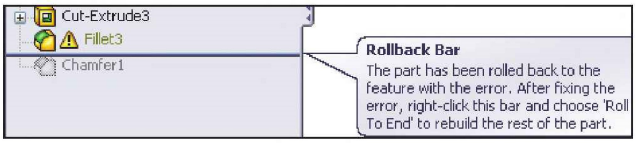
1. **Suppress** Fillet1, CBORE and Boss-Extrrude4. A dialog box is displayed.
2. **Press** the Stop and Repair button from the SolidWorks dialog box. Fillet3 has a reference issue. There are missing items in the existing feature.



As a general rule, suppress features before you delete them. This will inform you if there are any rebuild or feature errors during modification in the exam.

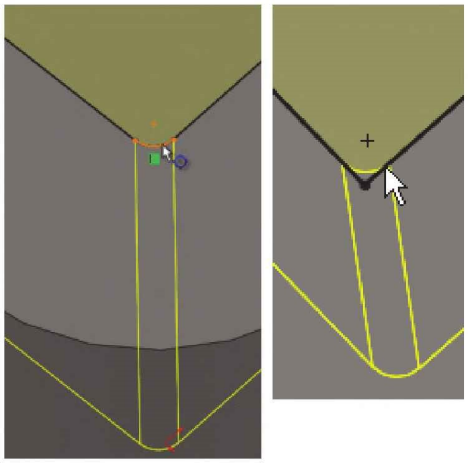


3. **Edit** the Fillet3 feature and repair. Delete any missing edges and insert the new edge.

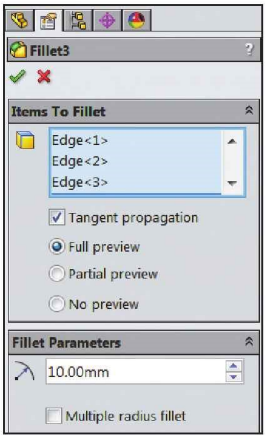


But wait, you can't insert the new edge (as illustrated) because it is part of the offset from the original sketch when you created the part.

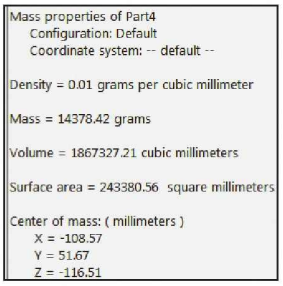
Create a square corner for Sketch10 to select the needed edge for Fillet3.

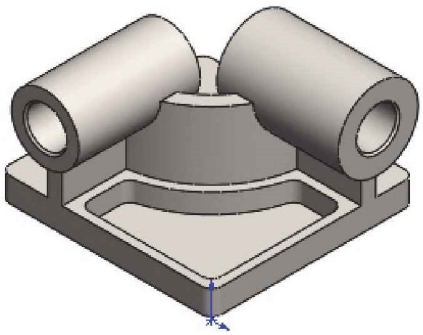


4. **Edit** Sketch10 from the Cut-Extrude3 feature. Delete the fillet.
5. **Insert** the needed edge for Fillet3 using the Trim Entities sketch tool (Corner option).
6. **Edit** the Fillet3 feature and add the needed edge that you just created. Verify that you have 5 fillets with the radius of 10mm. There are no errors displayed in the FeatureManager.
7. **Delete** all suppressed features and any unneeded sketch in the FeatureManager: Fillet1, CBORE, Boss-Extrude4 and Sketch7. Suppress features before you delete them. This will inform you if there are any rebuild or feature errors during modification in the exam.



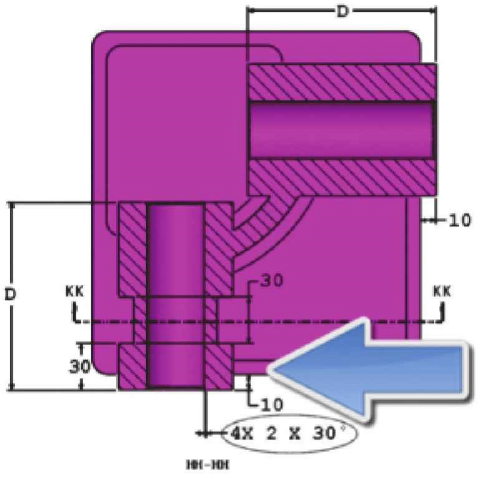
8. **Roll** back the Rollback bar in the FeatureManager.
9. **Calculate** the mass of the model in grams.
10. Your mass at this time should be **14378.42 grams**.

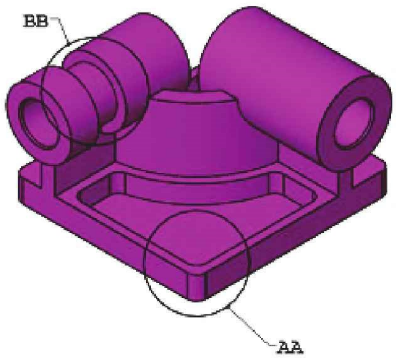




You should have the exact answer (within 1% of the stated value in the multiple choice section) before you move on to the next question.

11. **Save** the part.



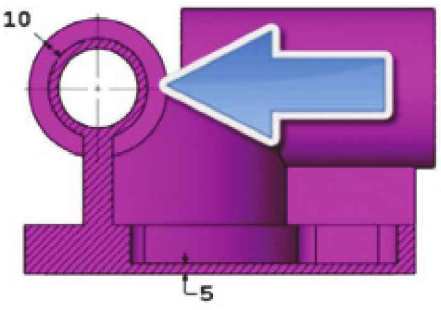


Address the BB modification in the model. Create a circular cut as illustrated on the left cylinder. The circular cut is offset 30mm from the front face of the cylinder. The depth of the cut is 30mm.

From the front view, the circular cut is offset 10mm and the circular cut does not go completely through the Extrude-Thin1 feature.

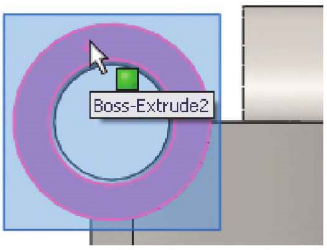
Last, modify the Chamfer feature of the cylinders.

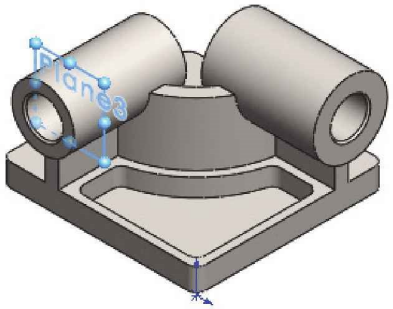
In this section utilize the Offset Entities, Convert Entities, Line and Trim Sketch tools to create the Base Sketch for the Extruded Cut feature.



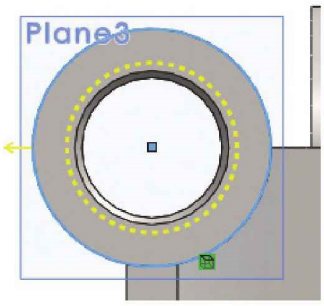
Let's begin.

12. **Create** a Sketch plane (30mm) offset from the front face of the cylinder as illustrated. Plane3 is displayed in the FeatureManager.



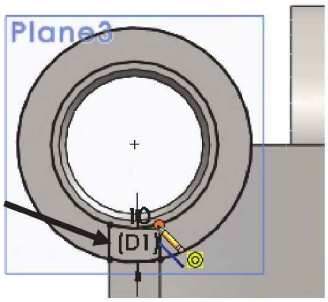


13. **Create** the Base Sketch on Plane3. Apply the Convert Entities Sketch tool to utilize the outside cylindrical geometry of the tube.



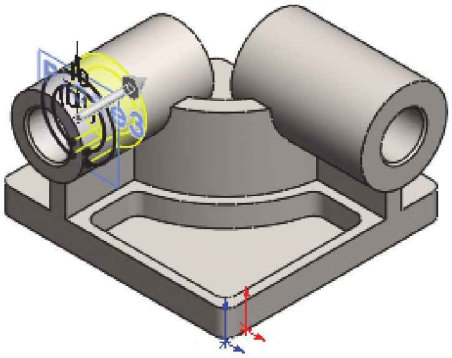
14. **Apply** the Offset Entities Sketch tool with an offset distance of 10mm. Click the outside cylindrical edge and reverse the direction if needed. You created an inside and outside ring for the Extruded Cut feature on your Base Sketch (Sketch11).

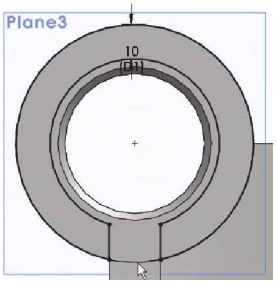
15. **Create** two vertical lines from the outside cylindrical edge to the inside cylindrical edge of the ring. The sketch is fully defined.



16. **Trim** any unwanted sketch geometry to finish the sketch for the Extruded Cut feature. Your Base sketch should consist of two arcs and two vertical lines.

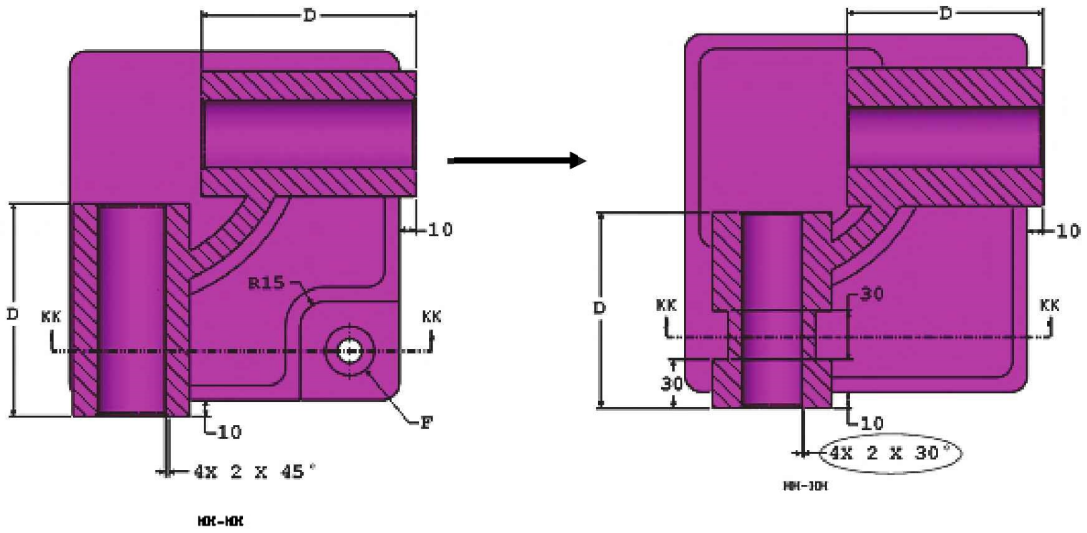
17. **Create** an Extruded Cut feature with a depth of 30mm. The Extruded Cut feature removes the needed material and keeps the Extrude-Thin1 feature unbroken.



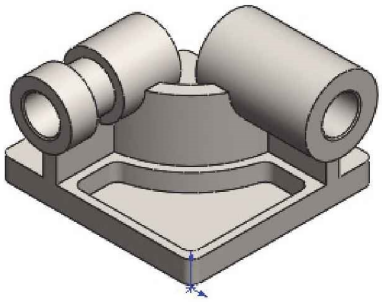


Next, address the modification of the Chamfer feature in the front face of the cylinder.

18. **Modify** the cylinder Chamfer feature from 45 degrees to 30 degrees.



At this time your model should have a mass of **13983.95 grams**.



19. **Save** the part.

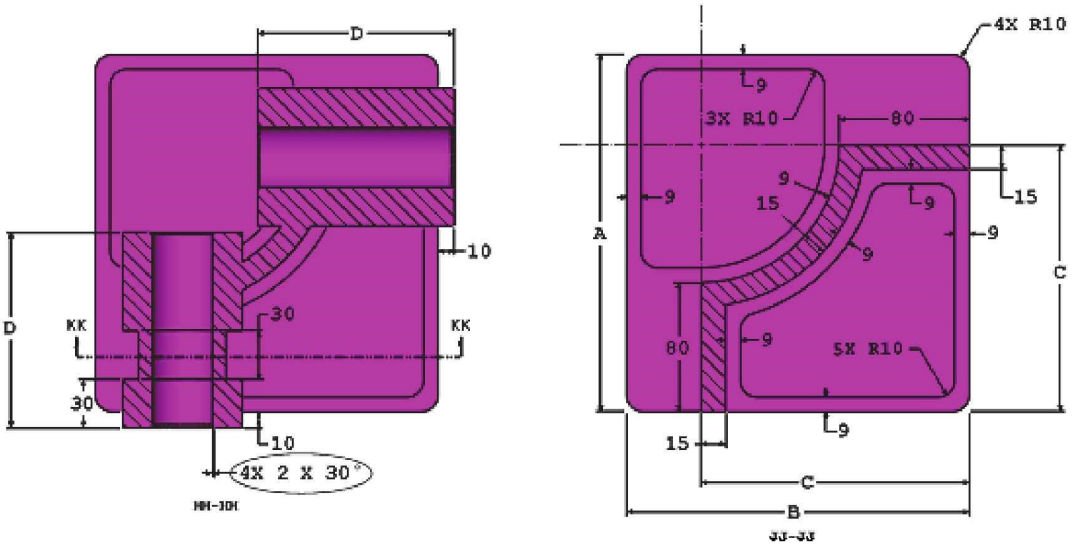
Always enter the needed decimal places (in this case 2) in the answer field.

Mass properties of Part4A
Configuration: Default
Coordinate system: -- default --

Density = 0.01 grams per cubic millimeter

Mass = 13983.95 grams

Volume = 1816096.79 cubic millimeters

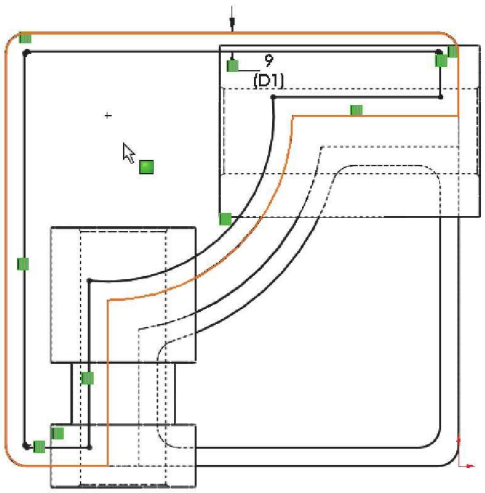


Address the Extruded Cut and fillet feature and then address the Global Variables A thru E and equation Y.

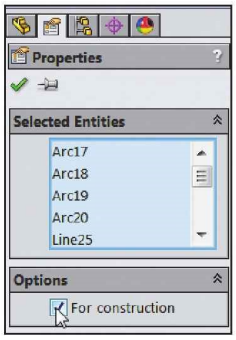
There are numerous ways to create the sketch for the Extruded Cut feature. In this case, utilize Construction geometry. Construction geometry helps you create a sketch but is not part of the feature.

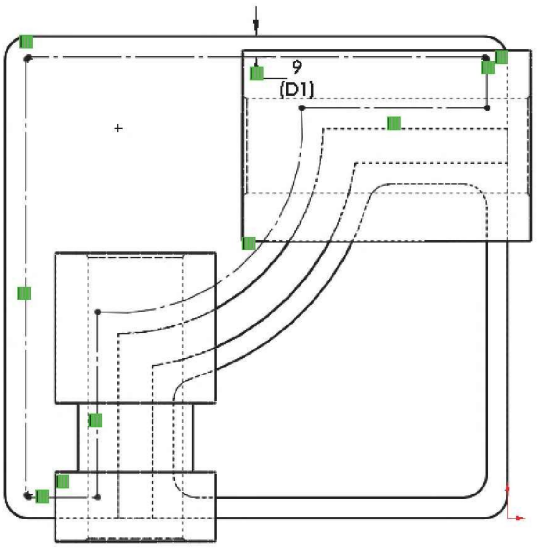
View the DVD for an alternative method using the Rollback bar and existing geometry to create the sketch for the Extruded Cut feature.

20. **Create** a sketch using the Offset Entities Sketch tool on the top back face. Enter 9mm for Offset distance. Reverse the direction if needed.



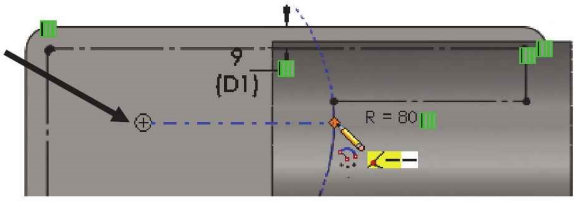
21. **Window-select** the part. Check the For construction box. Again, construction geometry helps create a sketch but is not part of the feature.

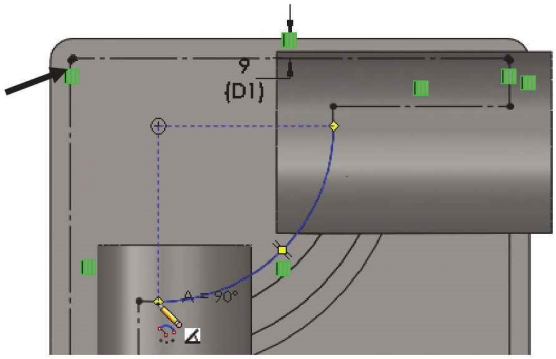




Create a 90^0 Arc and inference the vertical line from the Extrude-Thin1 feature.

22. **Create** a 90^0 Arc. Use the Centerpoint Arc Sketch tool. Click the centerpoint as illustrated, drag directly to the right until you see the inference symbol on the construction arc. Click the start point. Drag downward and click the end point to create the 90^0 Arc. Note the mouse pointer displays A = 90 for angle feedback.

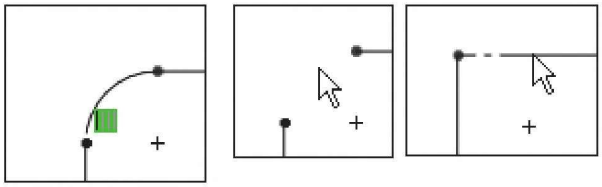




Sometimes when you convert sketch geometry, unwanted arcs and points are created. In the next section, delete any unwanted sketch geometry to cleanly create Sketch12.

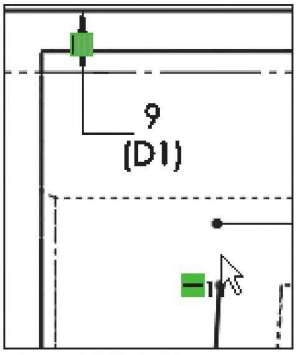
23. **Remove** the top left fillet with the Trim Entities Sketch tool.

24. **Restore** the corner of the fillet feature with the Trim Entities Sketch tool (Corner option) or by just dragging the endpoints together.



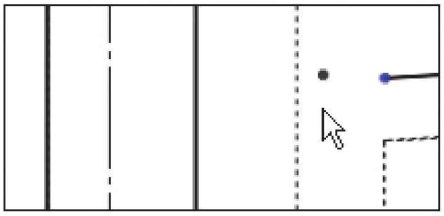
25. **Remove** all unwanted sketch geometry around the start point of the 90^0 Arc.
26. **Remove** all unwanted sketch geometry around the end point of the 90^0 Arc.

Now you can utilize the horizontal and vertical construction geometry with the 9mm offset with new lines and the Arc is created correctly. Again there are other ways to create the sketch for the Extruded Cut feature.

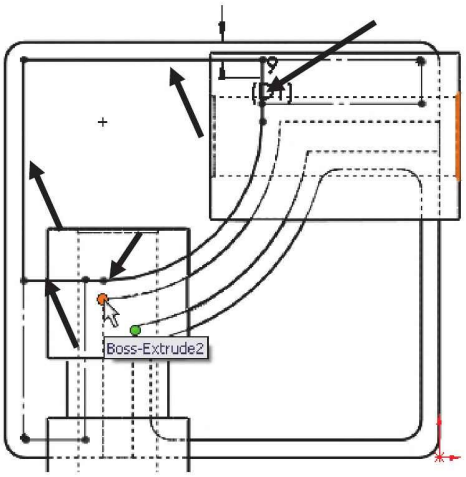


27. **Complete** the close sketch profile with the Line Sketch tool.

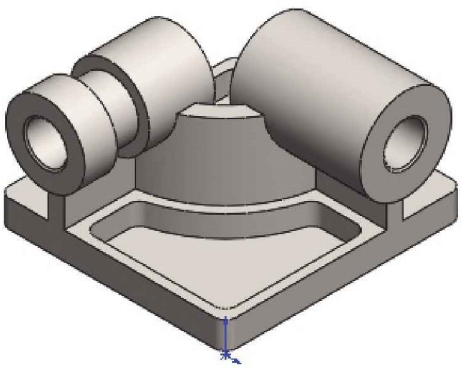
Insert all needed Geometric relations.

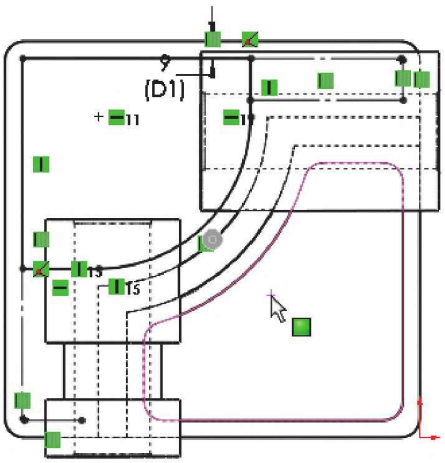


28. **Insert** a vertical relation between the end point of the Arc and the corner point of the Thin-Extrude1 feature. The sketch should be fully defined and displayed in black.
29. **Create** the Extruded Cut feature. Apply the Up To Surface End Condition and click the inside face of Cut-Extrude3. The two surfaces provide a similar dimension.
30. **Apply** the Fillet feature. Insert three fillets per the provided drawing. 10mm is the fillet radius.



At this time your model should have a mass of **12154.09 grams**.

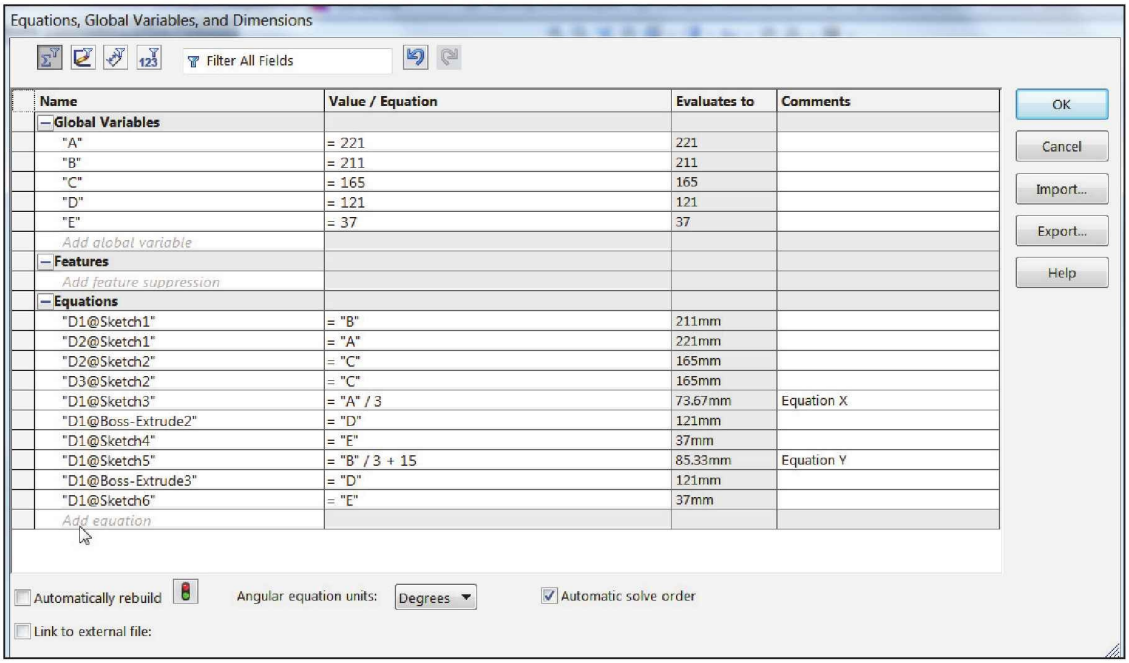




31. **Display** the Equations, Global Variables and Dimension dialog box.
32. **Enter** the five new Global Variables (A, B, C, D & E) and the new Y equation as illustrated.

A = 221 mm
B = 211 mm
C = 165 mm
D = 121 mm
E = 37 mm
X = A/3
Y = B/3 + 15mm

Note: The equation for Y has changed from the initial part.



33. **Calculate** the mass of the model.
34. **Select 13206.40 grams**. The number matches the answer of b. You should be within 1% of the stated value before you move to the next section to modify the part.
35. **Save** the part.
36. **Rename** Part4 to Part5.

Mass properties of Part5
Configuration: Default
Coordinate system: -- default --

Density = 0.01 grams per cubic millimeter

Mass = 13206.40 grams

Volume = 1715116.48 cubic millimeters

Surface area = 253729.21 square millimeters

Center of mass: (millimeters)
X = -97.24
Y = 57.94
Z = -114.16

What is the mass of the part (grams)?

a) 13095.40
b) 13206.40
c) 13313.35
d) 13395.79

Always save your models to verify your results.

Use the Comments box to label your equations.

Confirm that your math is correct. Use the Measure tool during the exam.

Segment 1 of the CSWP CORE exam - Fifth question

Compare the provided information with your existing part. The Global Variables A thru E change, and the equations are the same from the last question but equation Y has changed from the initial part. This question provides a fill in the blank format.

Provided Information:

5. Stage 2 - Update Parameters

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Part origin: Arbitrary

Material: Alloy Steel

Density: 0.0077 g/mm^3

All holes through all unless shown otherwise.

Use the following parameters and equations which correspond to the dimensions labeled in the images:

A = 229 mm

B = 217 mm

C = 163 mm

D = 119 mm

E = 34 mm

X = A/3

Y = B/3 + 15mm

Hint #1: The dimensions that are to be linked or updated and are variable will be labeled with letters. Any dimensions that are simple value changes from one stage to another will be circled in the images.

Hint #2: To save the most time, make use of linked dimensional values and equations.

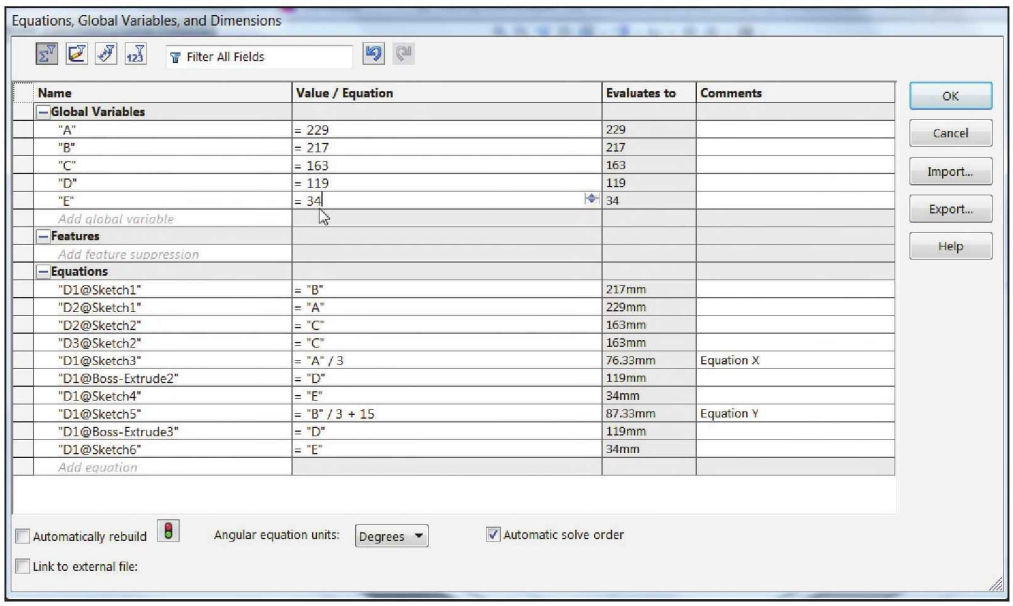
Measure the mass of the part.

What is the mass of the part (grams)?

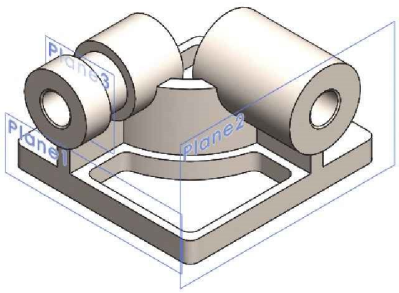
Let's begin.

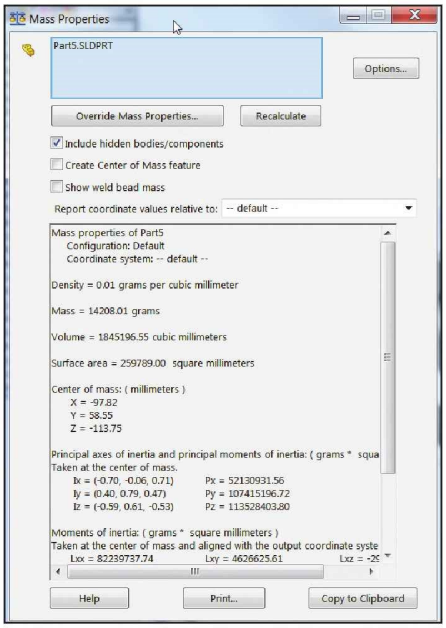
1. **Display** the Equations, Global Variables and Dimension dialog box.
2. **Enter** the five new Global Variables (A, B, C, D & E) as illustrated.

A = 229 mm
B = 217 mm
C = 163 mm
D = 119 mm
E = 34 mm
X = A/3
Y = B/3 + 15mm

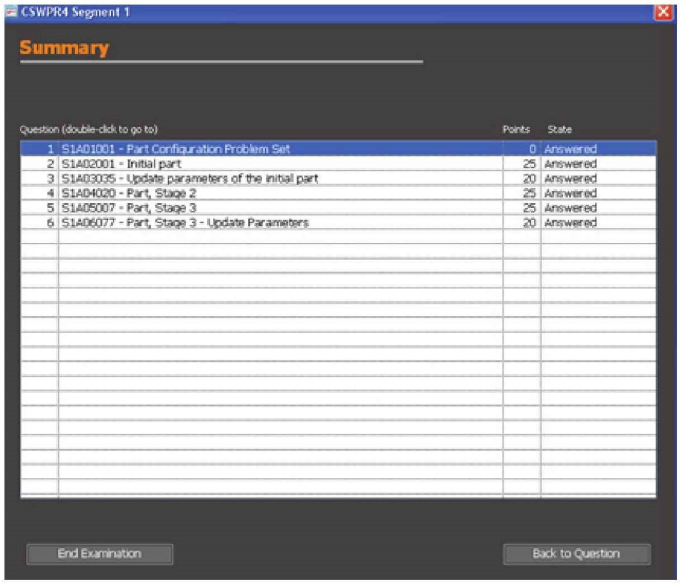


3. **Calculate** the mass of the part.
4. **Enter 14208.01 grams**.
5. **Save** the part.





At the end of the exam, view the Summary dialog box. Press the End Examination button only if you are finished.

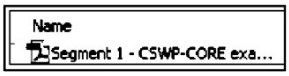


Actual CSWP exam format

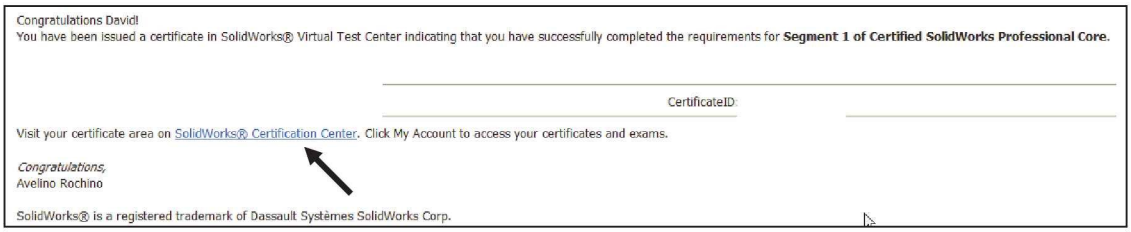
Always enter the needed decimal places in the answer field.

The sample CSWP exam only covers segment 1. Time yourself on the practice exam. You should be able to finish the sample Segment 1 CSWP CORE exam in approximately 60 - 75 minutes.

Prior to SolidWorks 2013, a method commonly used to define variables for CSWP exam problems was to use Linked Values. Linked Values, also called linked dimensions, connect two or more dimensions without using equations or relations. Linked Values can still be used today in SolidWorks 2013 and SolidWorks 2014. The method to apply Linked Values is described in videos included in the book's DVD.



When you pass the first segment of the CSWP CORE exam you will receive the following email. Click on the SolidWorks Certification Center hyperlink to login, activate and view your certificate.


Congratulations David!
You have been issued a certificate in SolidWorks® Virtual Test Center indicating that you have successfully completed the requirements for **Segment 1 of Certified SolidWorks Professional Core.**

CertificateID:

Visit your certificate area on SolidWorks® Certification Center. Click My Account to access your certificates and exams.

Congratulations,
Avelino Rochino

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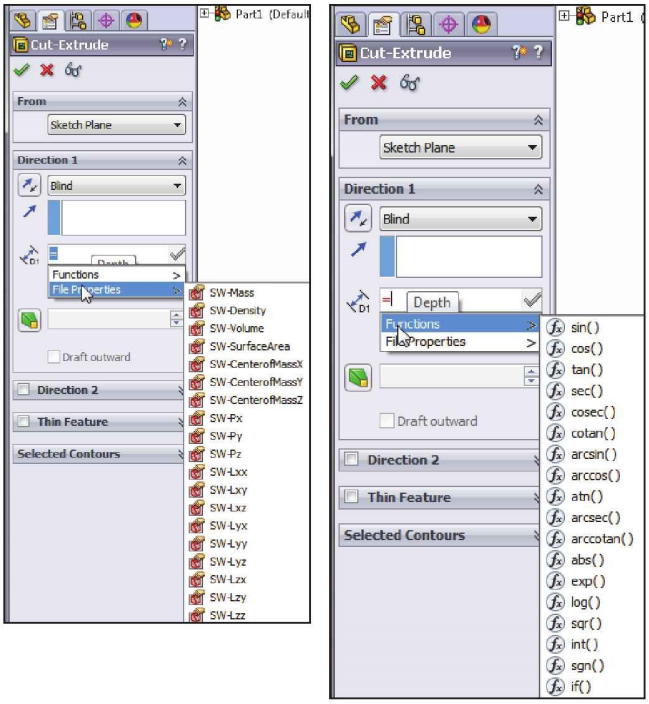
Use the SolidWorks Tutorials and SolidWorks Help to find additional information on Global Variables and equations before the exam. A novice using help files during the exam will not have enough time.



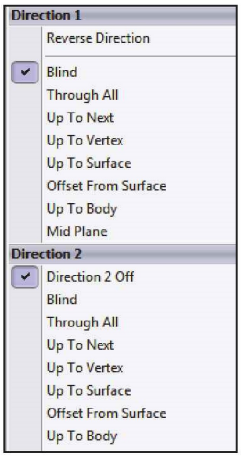
For many features in SolidWorks 2013 and newer (Extruded Boss/Base, Extruded Cut, Simple Hole, Revolved Boss/Base, Revolved Cut, Fillet, Chamfer, Scale, Shell, Rib, Circular Pattern, Linear Pattern, Curve Driven Pattern, Revolved Surface, Extruded Surface, Fillet Surface, Edge Flange and Base Flange) you can enter and modify equations directly in the PropertyManager fields that allow numerical inputs.

You can create equations with global variables, functions, and file properties without accessing the Equations, Global Variables and Dimensions dialog box.

For example, in the PropertyManager for the Extruded Boss/Base feature, you can enter equations in:

- Depth fields for Direction 1 and Direction 2
- Draft fields for Direction 1 and Direction 2
- Thickness fields for a Thin Feature with two direction types
- Offset Distance field

To create an equation in a numeric input field, start by entering = (equal sign). A drop-down list displays options for global variables, functions, and file properties. Numeric input fields that contain equations can display either the equation itself or its evaluated value.



You can toggle between the equation and the value by clicking the Equations or Global Variable button that appears at the beginning of the field.

In SolidWorks 2013 and newer, right-click anywhere on an extruded feature and change the end condition from the shortcut menu. Click in empty space, on geometry, or on the handle. The shortcut menu provides all the options for Direction 1 and Direction 2. Note: Options are document dependent.