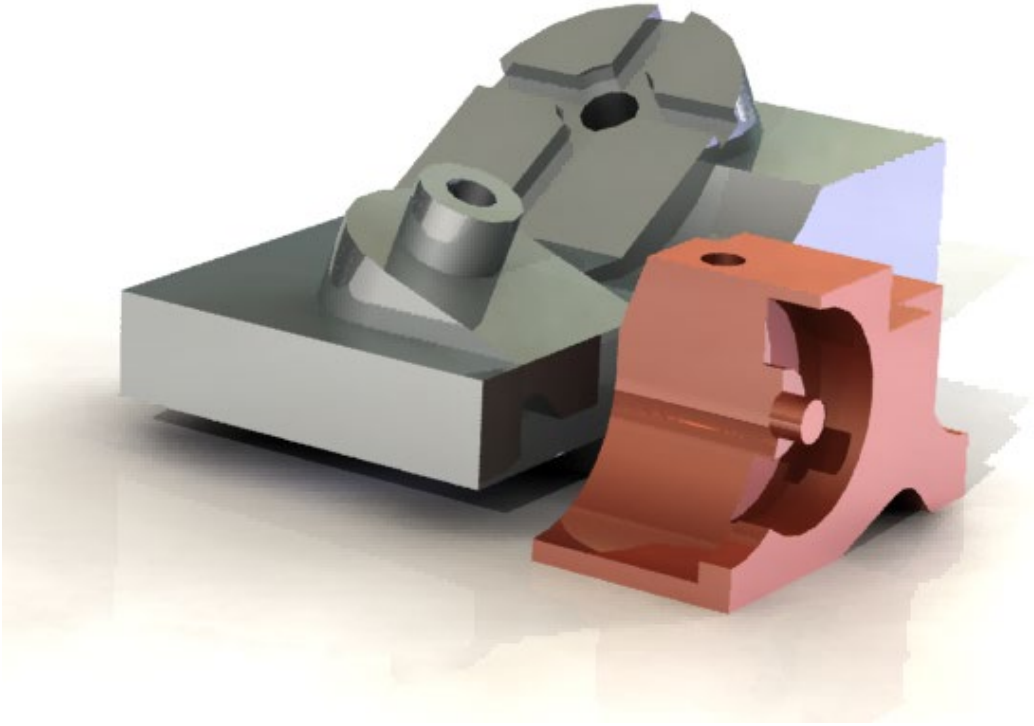
SOLIDWORKS® tutorial 12

CSWA



Prepatory and Advanced Vocational Training

To be used with SOLIDWORKS® Educational Release 2018-2019

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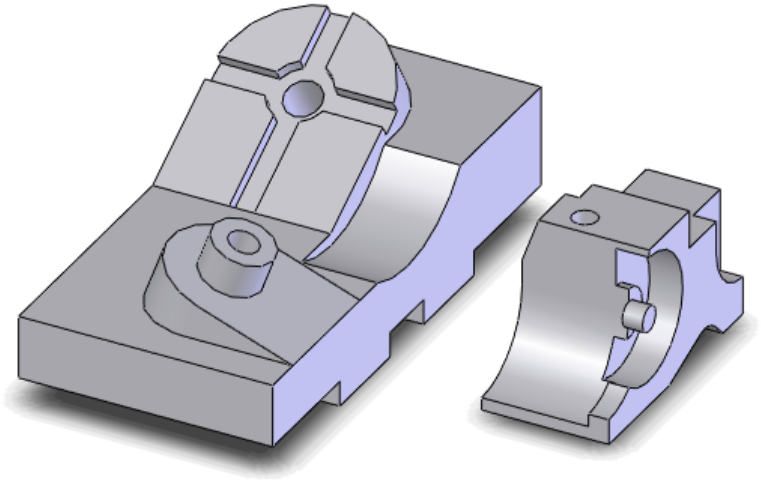
Initiative: Kees Kloosterboer (SOLIDWORKS Benelux)
Educations advisor: Jack van den Broek
Realisation: Arnoud Breedveld (PAZ Computerworks)

CSWA

CSWA (Certified SOLIDWORKS Associate) is a certificate you can receive after you have trained yourself in working with SOLIDWORKS. When you have to apply for a job in the future, you can simply prove that you know how to use SOLIDWORKS rather well by showing this certificate. It is certainly interesting to get it. Do you want more information about how to get this certificate, please ask your teacher for it.

Did you do all tutorials in this array and practiced some more exercises, you should be able to get the CSWA-certificate. To get more acquainted to the questions from a CSWA-test, we will practice two exercises from the CSWA test. You will not get any new topics, but you will find out how to make such an exercise.

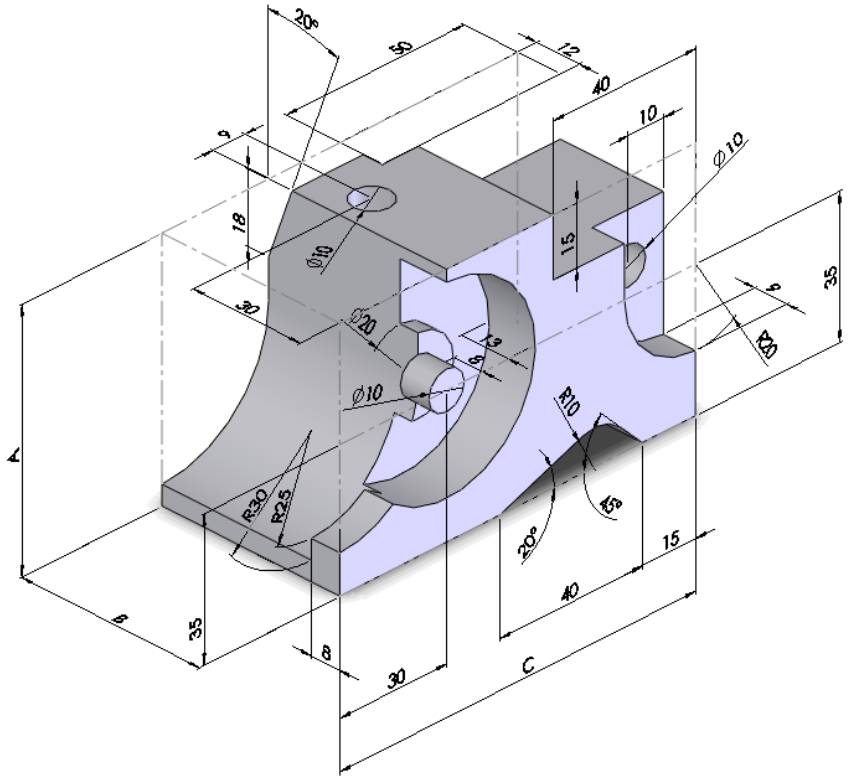


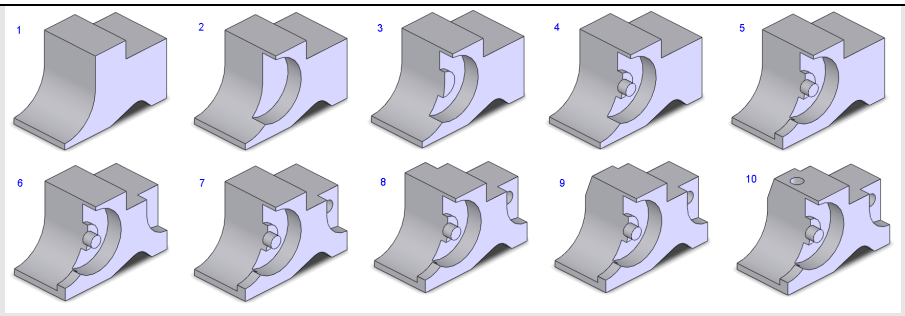
Assignment

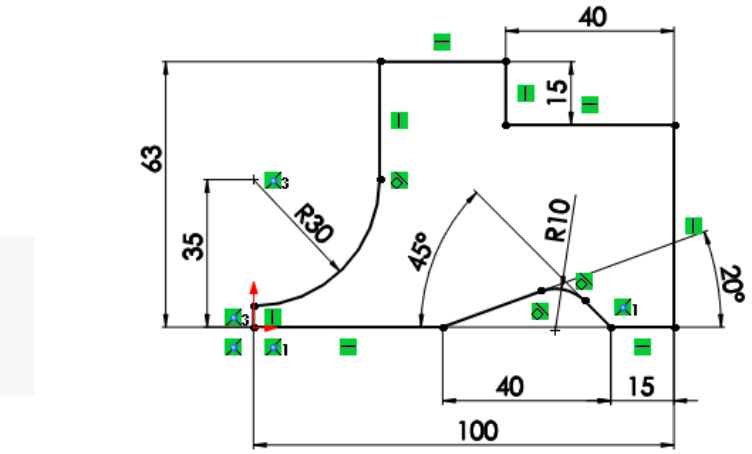
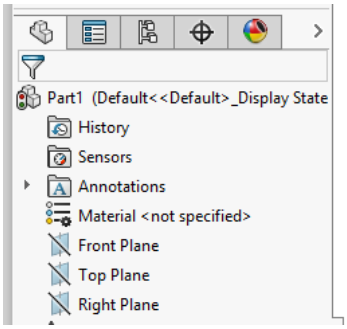
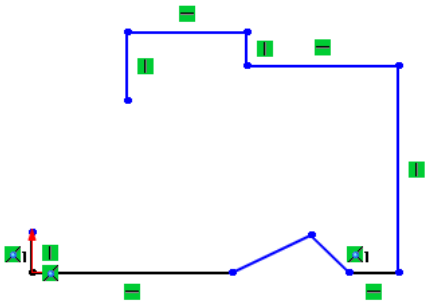
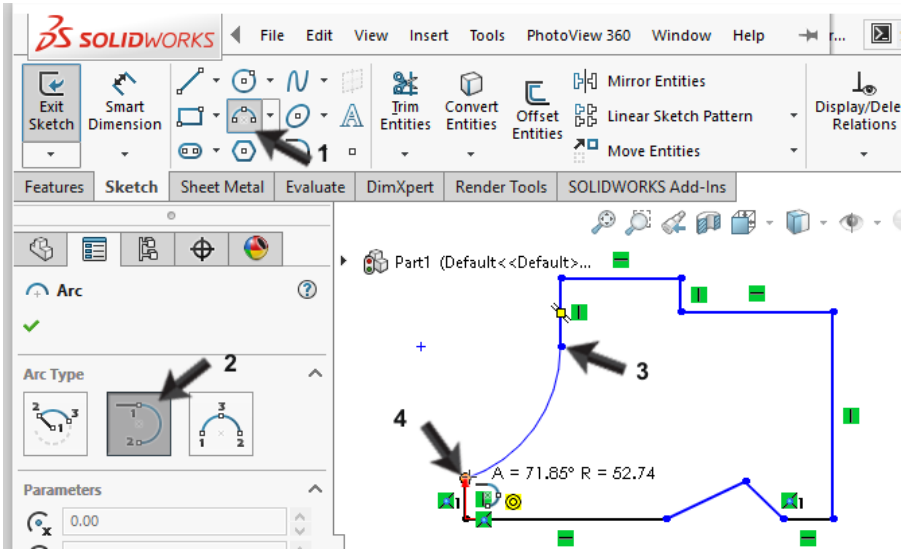
Available time: 45 min.

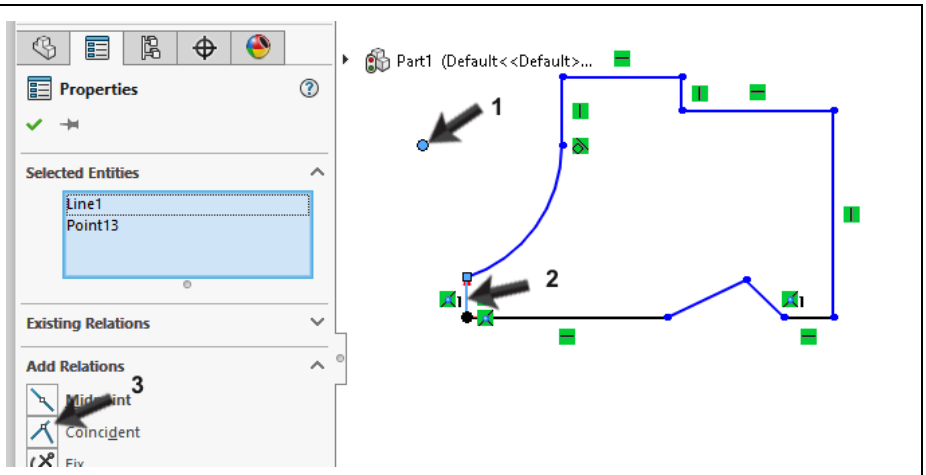
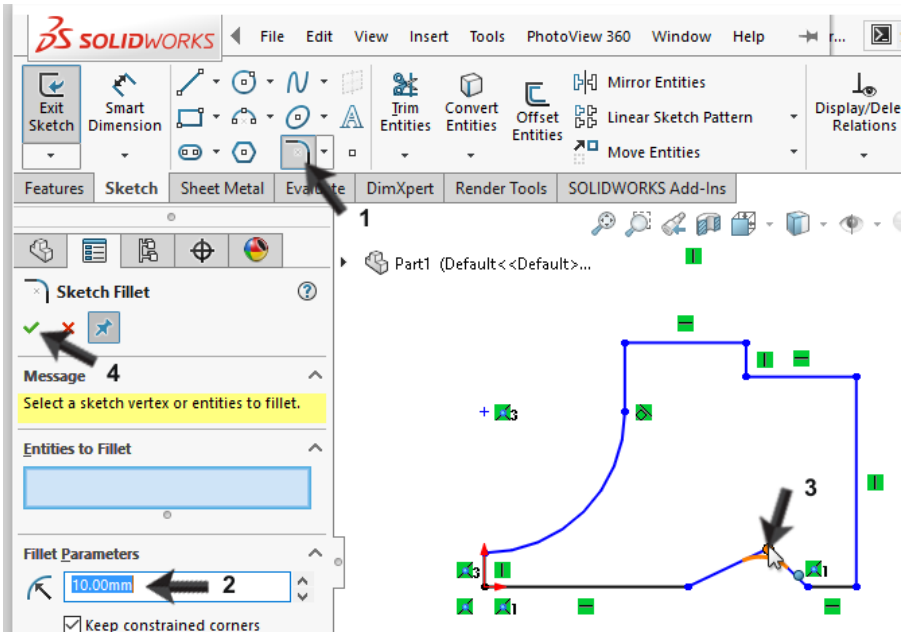
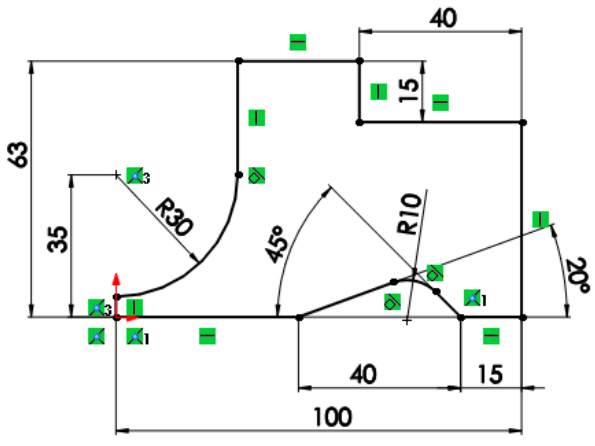
We will show you the assignment as described in the test

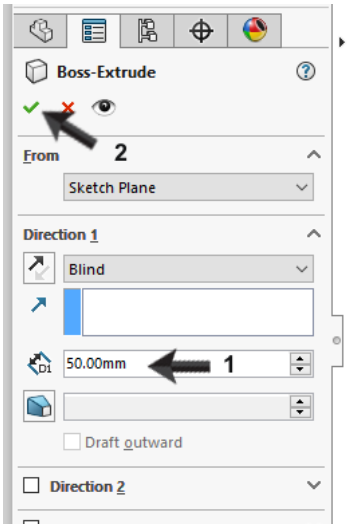
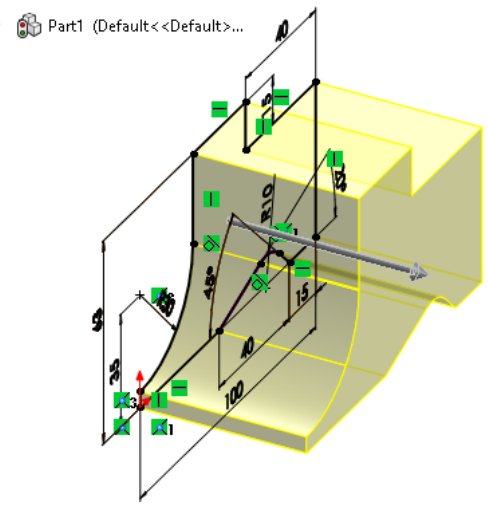
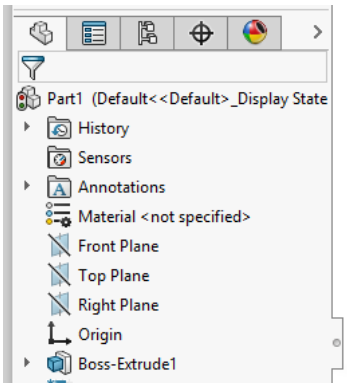
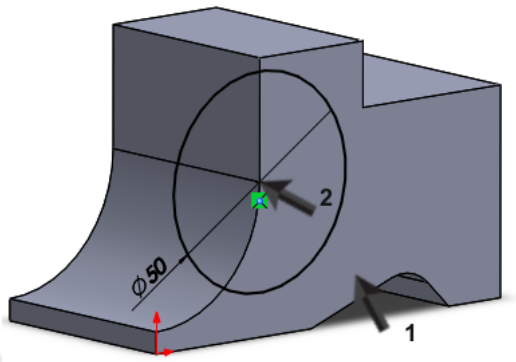
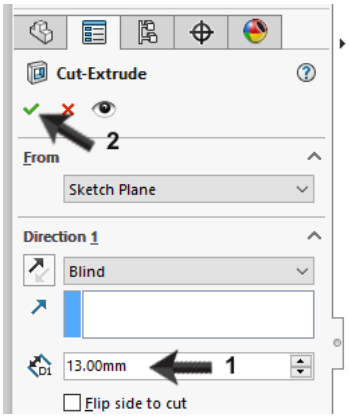
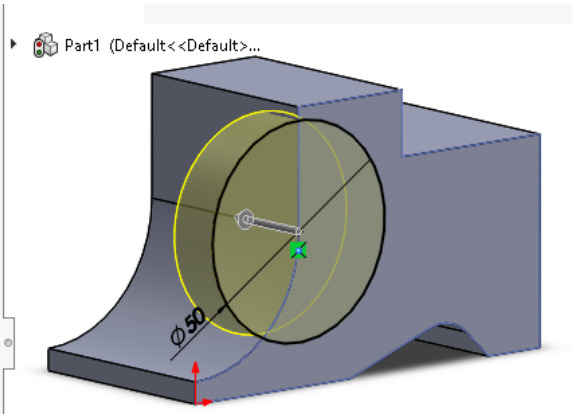
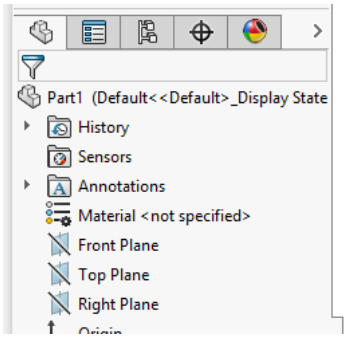
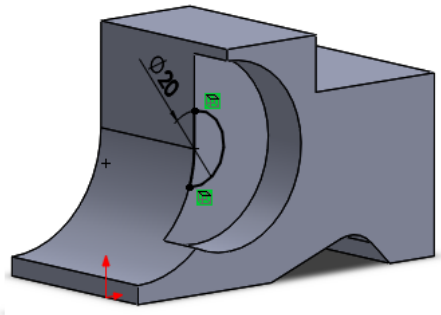
Build this part in SOLIDWORKS.	Your assignment is: build a part in SOLIDWORKS.
Unit system: MMGS (millimeter, gram, second)	Because inches are often used in the USA it says here that we work in millimeters and grams (metrical system). This is default in Europe, except for in England.
Decimal places: 2	We will work with two decimals: this is a default setting too.
Part Origin: Arbitrary	The Origin is at a random position, although in some assignments the position of the origin is determined.
A=63mm, B=50mm, C=100mm	Some dimensions are indicated in the model with the letters A, B or C. You will replace them with the values as given on the left.
All holes through all, unless other-	All holes will go through the whole

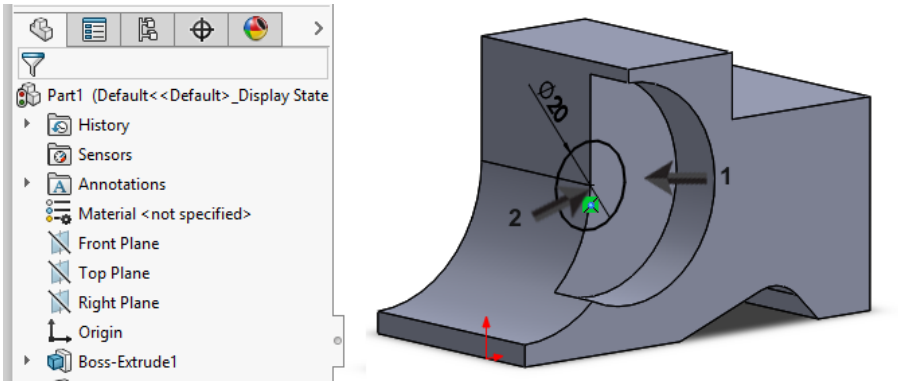
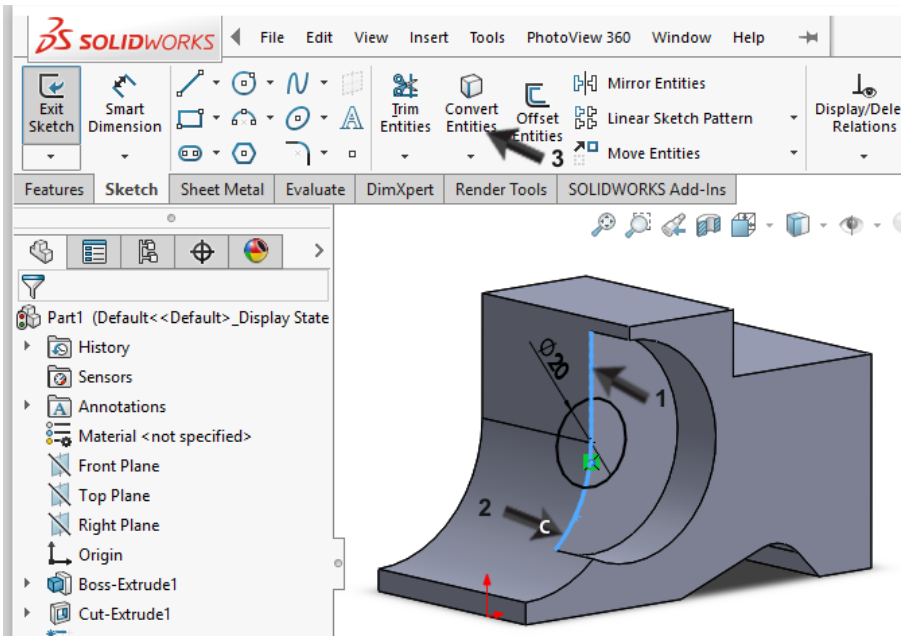
		wise specified	model, unless otherwise specified (this is often not visible in the drawing or illustration)
		Part material: Copper Density = 0.0089 g/mm^3	The part is made out of copper in this example The specific weight of copper is 0.0089gram per mm³
		What is the overall mass of the part in grams? a. 1205 b. 1280 c. 144 d. 1108	The question here is: What is the total weight of the part. It is a multiple choice question with four possible answers.
			
	Work plan	Although the shape of this assignment looks pretty complicated at first glance, you will see that it is build up form boss-extrude and cut-extrude commands. The hardest part is how to make a plan for it. Look at the shape very precisely and try to divide it in different features. **It is very important to do this before you start modeling!** Down below you can see the steps we have used to build the model. Every step is a feature that we will make. In total there are 10 features as you can see. Once you have made your plan, the modeling is rather simple.	

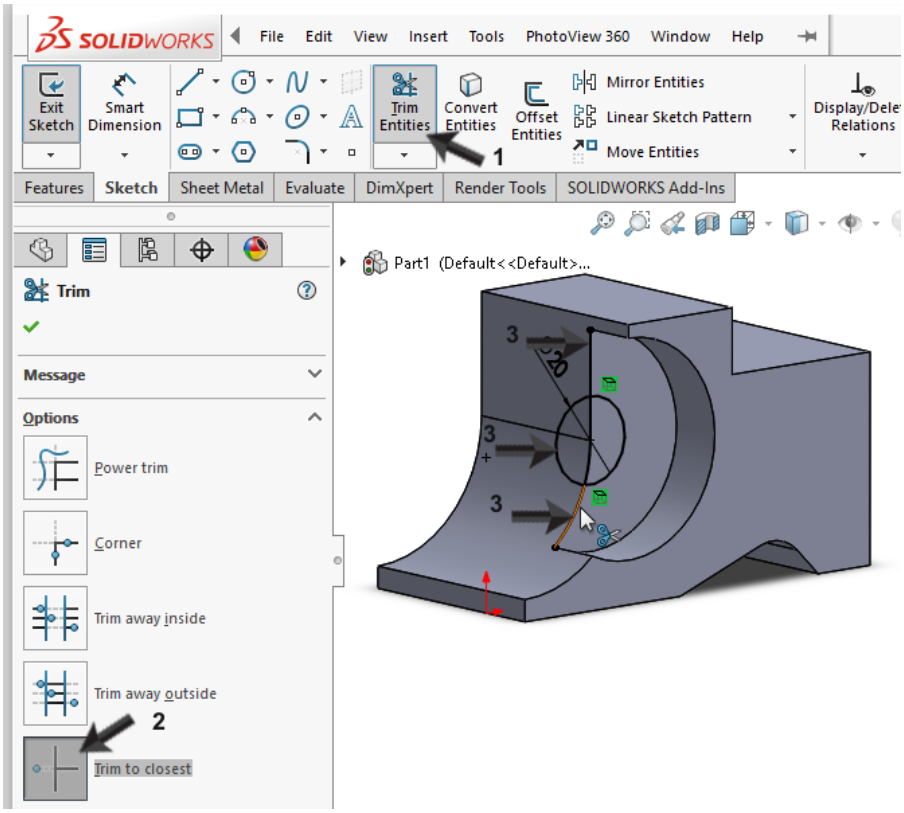
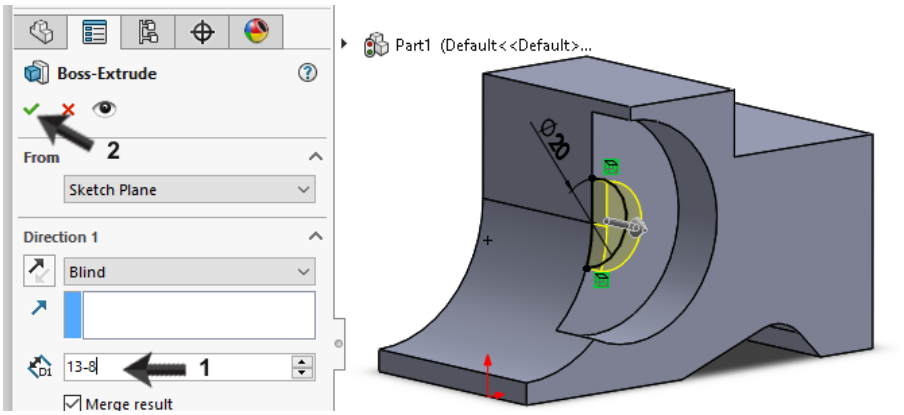
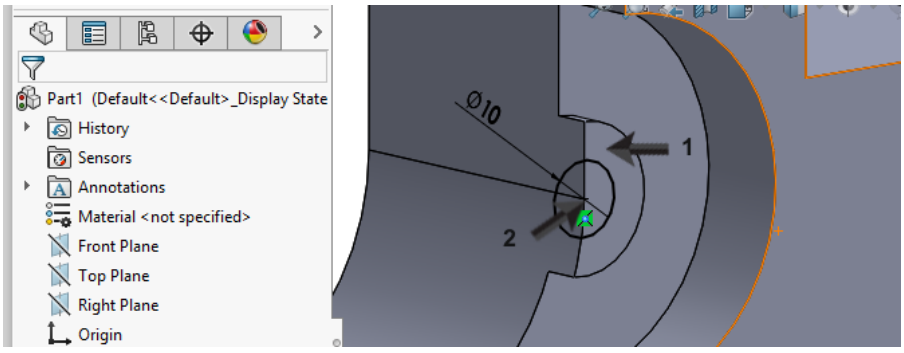
		 You can build your model in another way of course. There is not only one right way, but you have to do it as simple as possible, using as little features as possible. Let us have a look how to build the model from here.

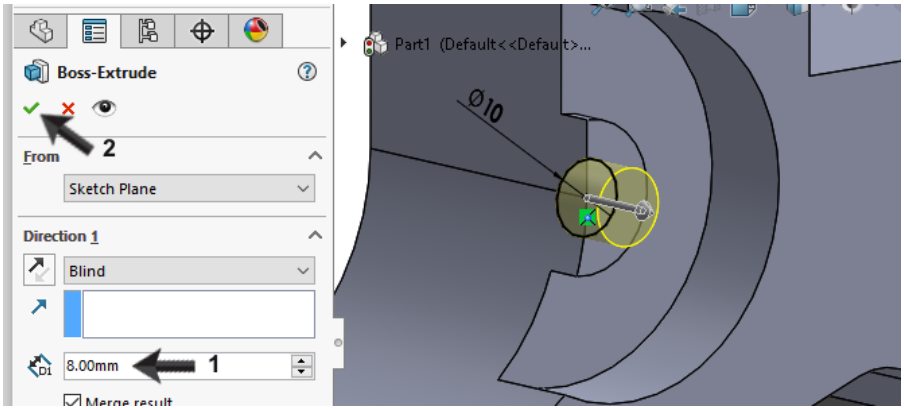
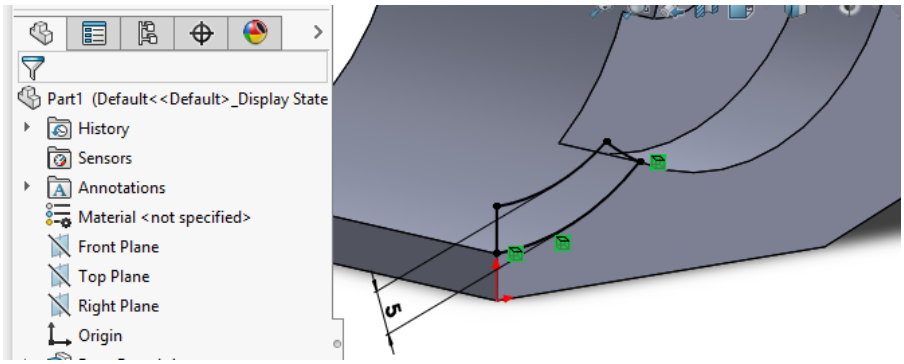
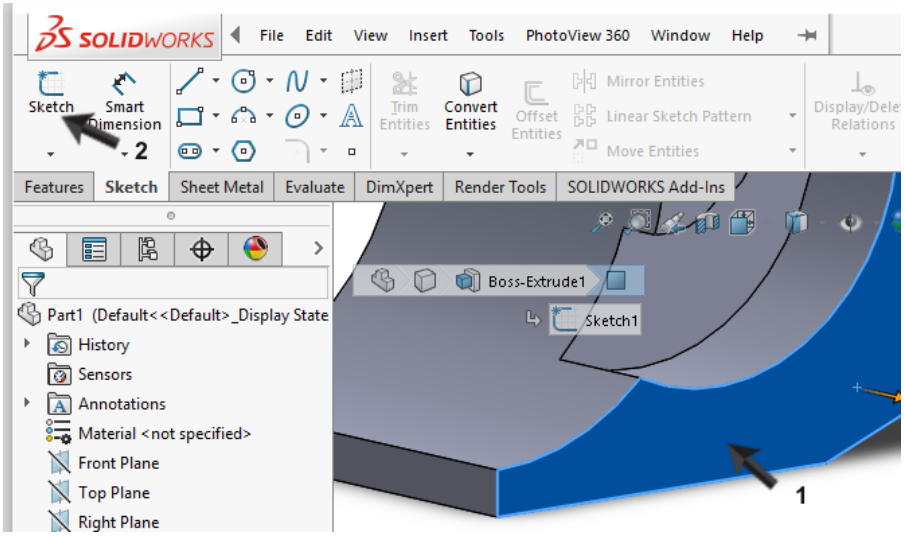
1	Start SOLIDWORKS and open a new part.	
2	Select the Right Plane and make a sketch s shown on the right. Can you make this sketch yourself already? Very good. Continue with step 8 now. If you do not succeed doing it yourself, try using the next few steps.	
3	Draw a shape as you see on the right. Make sure the line from the origin runs horizontally to the right and has a length of about 50 mm. With this as a base the proportions will be right.	 
4	Draw the arc now: 1. Click on Arc in the CommandManager. 2. Click on Tangent Arc in the PropertyManager 3. Click on the lower end of the vertical line as shown on the right 4. Click on the upper end of the vertical line as shown on the right.	

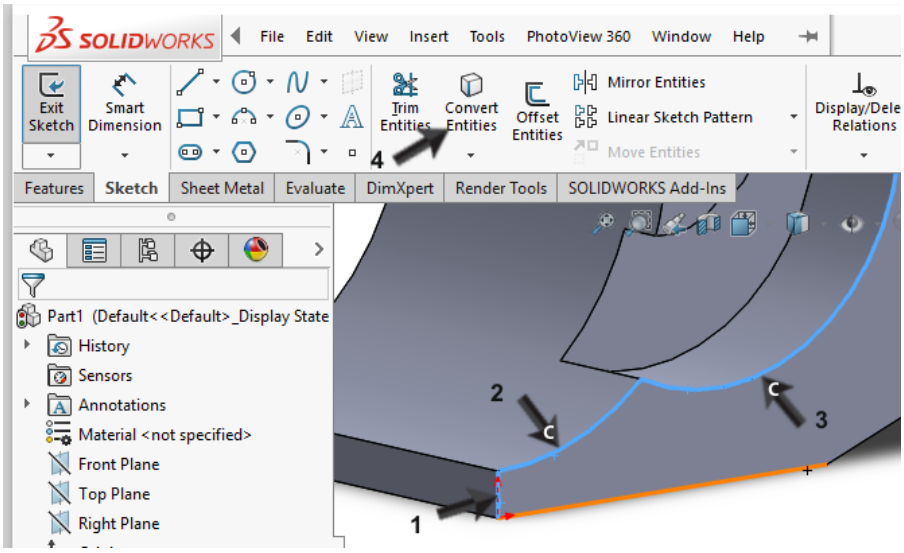
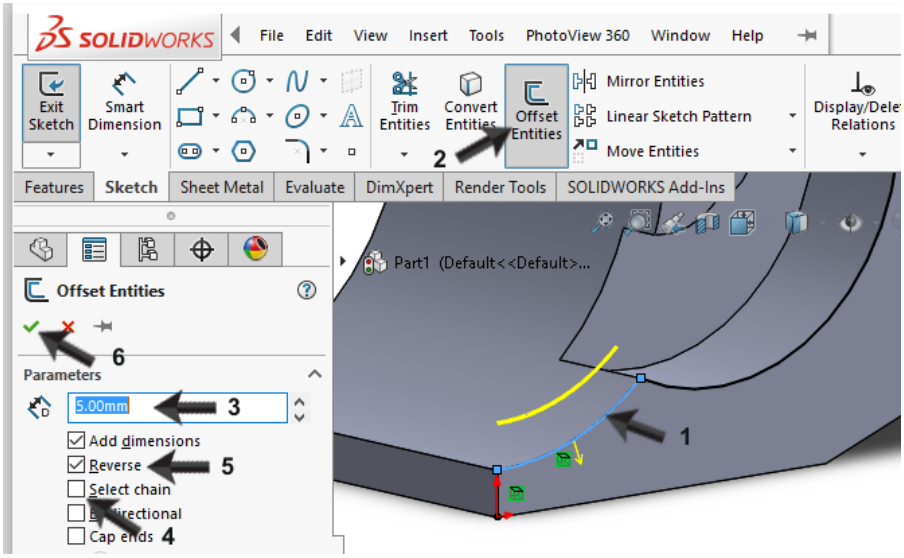
5	Put the midpoint of the arc you have just drawn exactly on the left vertical line. By doing so, you are sure the arc is always 90°. 1. Select the midpoint of the arc. 2. Select (holding the <ctrl>-key) the left vertical line. 3. Click on Coincident in the CommandManager.	
6	Make a fillet at the bottom of the sketch: 1. Click on Sketch Fillet in the CommandManager 2. Check if you have set a radius of 10 mm. in the PropertyManager (this is the default value) 3. Click on the corner you want to fillet in the sketch.	
7	Add the dimensions in the sketch as shown on the right.	

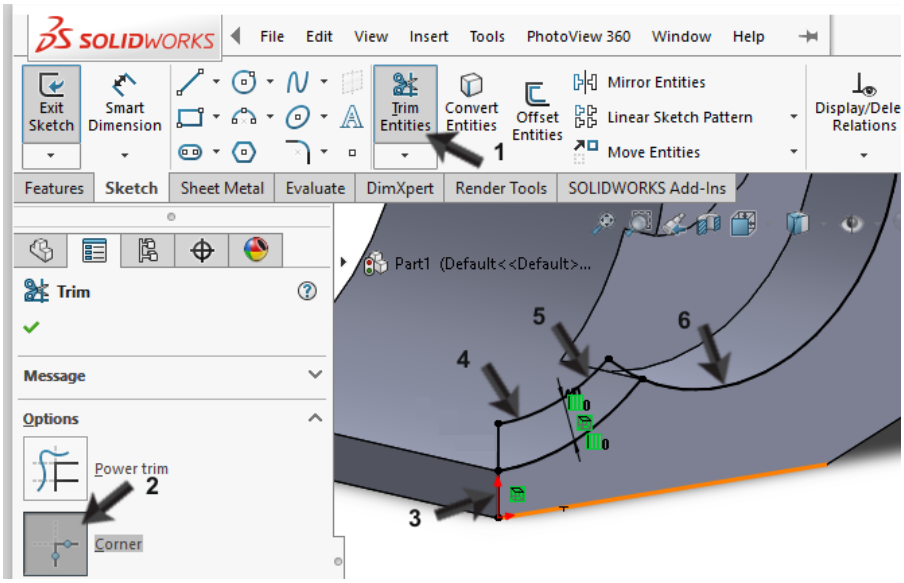
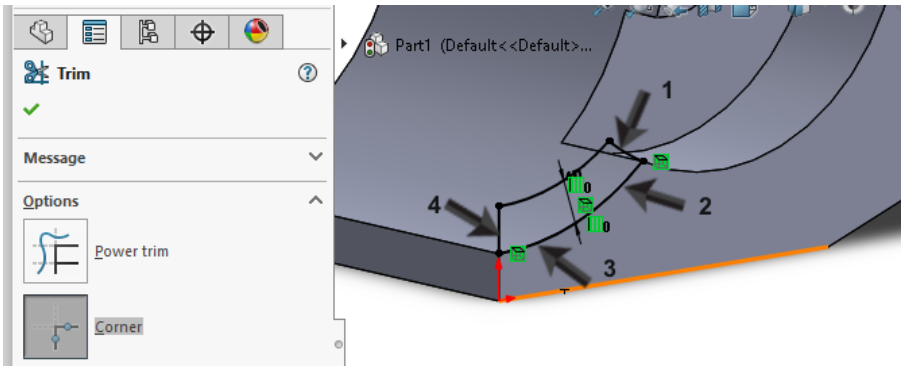
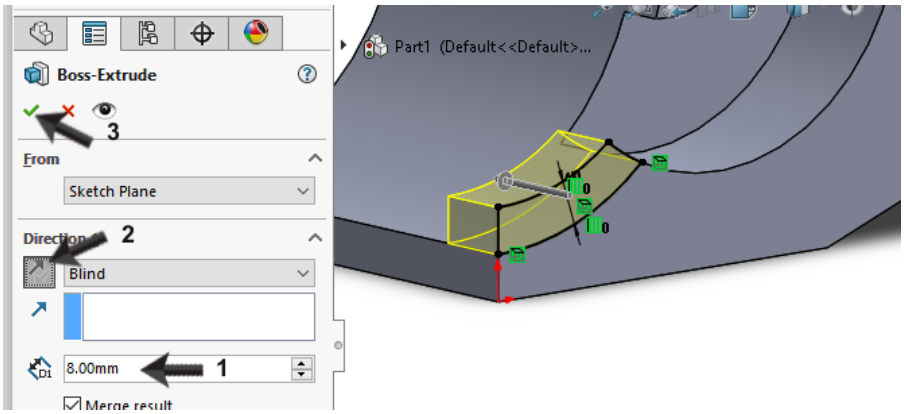
8	Extrude the sketch over 50mm.	 
9	Next make a sketch, as shown on the right. 1. Select the front surface of the model to draw a new sketch on it 2. Click on the point where the line converts into the arc. Draw the circle and add the dimension to the sketch.	 
10	Make an Extruded Cut from the sketch, setting the depth to 13mm.	 
11	Make a sketch as shown on the right. Can you do it yourself? Proceed to step 15. If this does not work out, watch the following steps which tell you how to handle this.	 

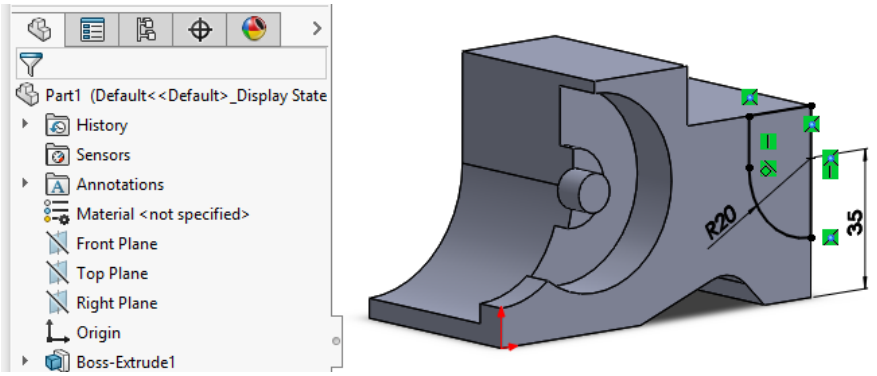
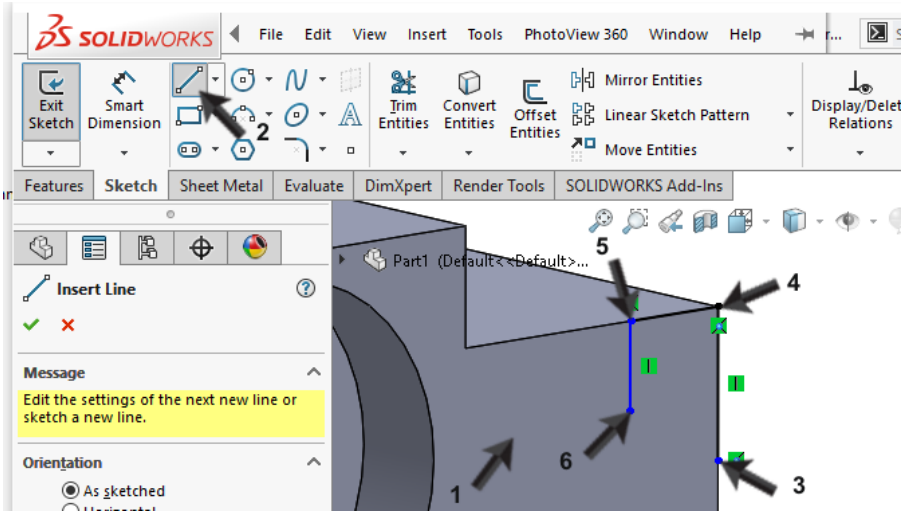
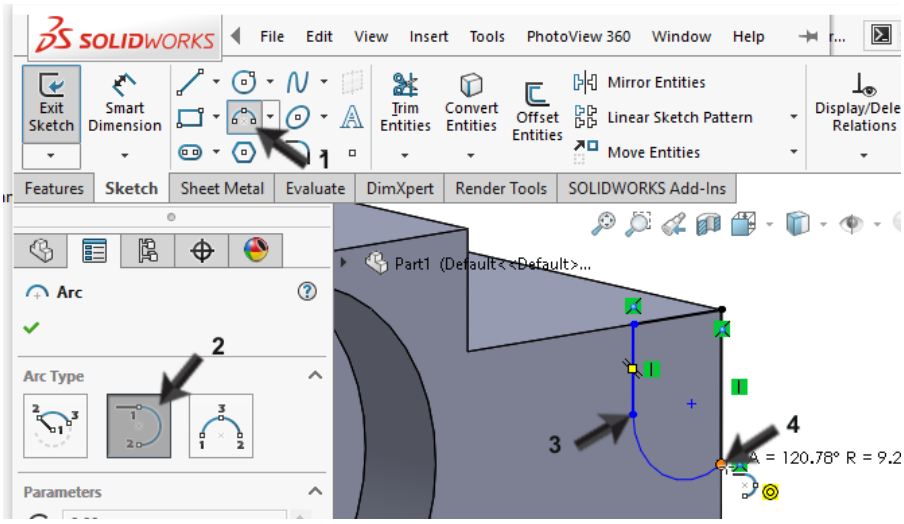
12	1. Select the deeper plane first. On this surface we will make a new sketch. 2. Draw a circle and make sure the midpoint is exactly at the point where the straight line converts in to an arc. 3. Set the size of the circle to Ø20mm.	
13	Push <esc> on your keyboard to end the Smart Dimensions-command. 1,2 Select the line and the arc as shown on the right. 3. Click on Convert Entities in the CommandManager	

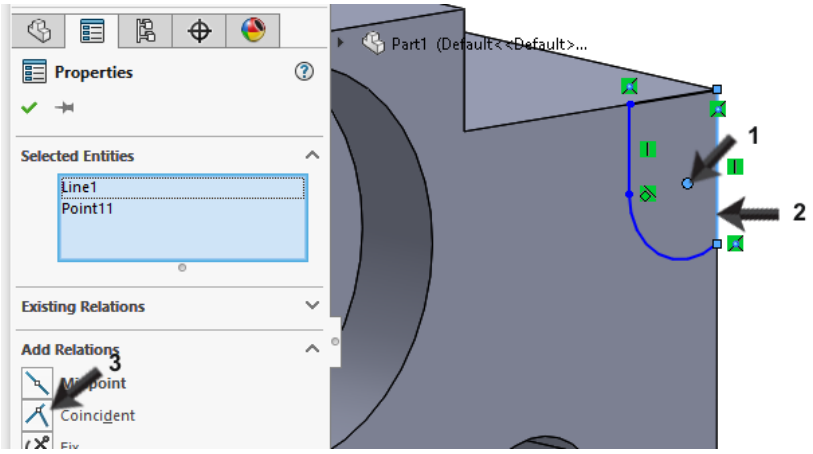
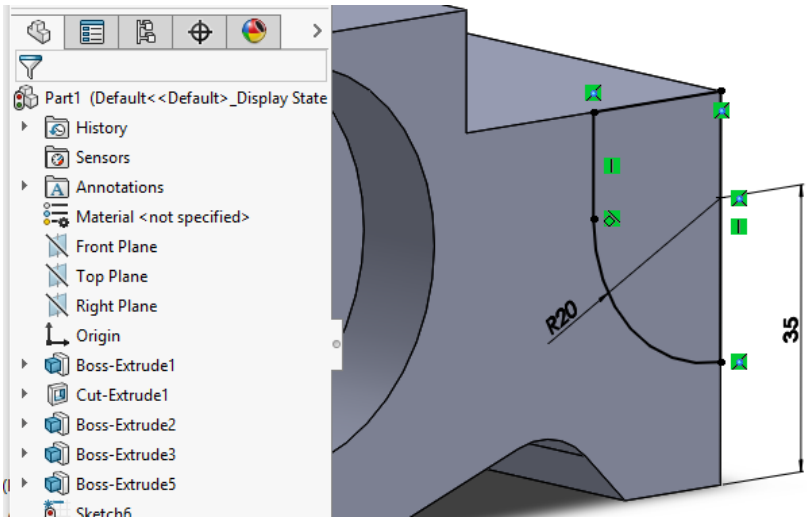
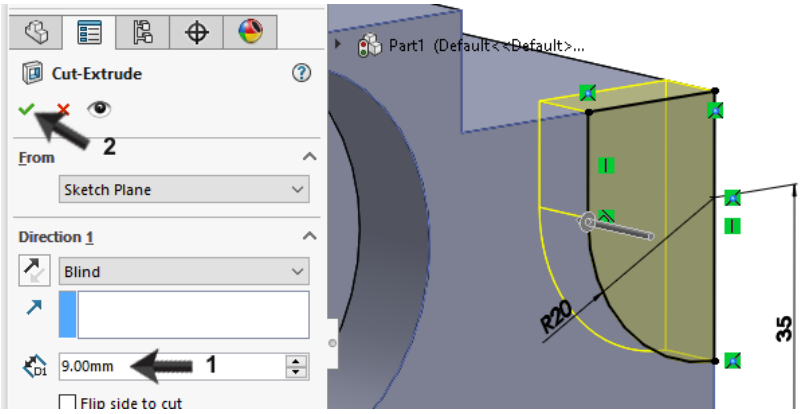
14	1. Click on Trim Entities in the CommandManager 2. Click on Trim to closest in the PropertyManager 3. Click on the three parts of the sketch that need to be removed.	
15	Extrude this sketch to a depth of 5mm.	
16	Make the sketch as in the illustration on the right. 1. Select the plane to draw a sketch on. 2. Draw a circle. Make sure the midpoint is exactly on the point where the straight line converts into an arc. 3. Set the size of the circle to Ø10mm.	

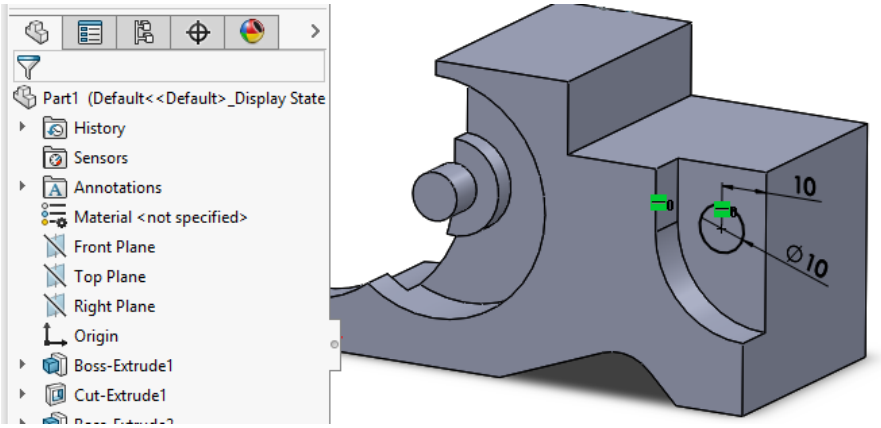
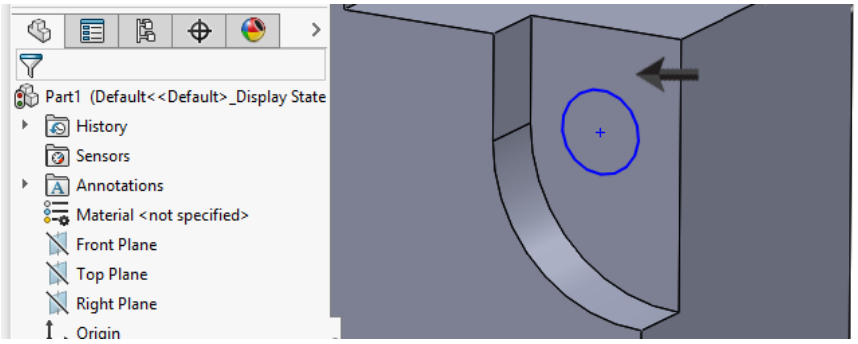
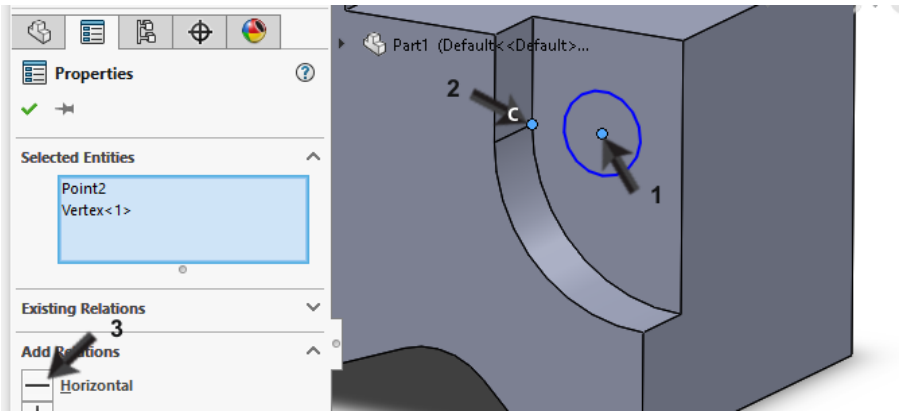
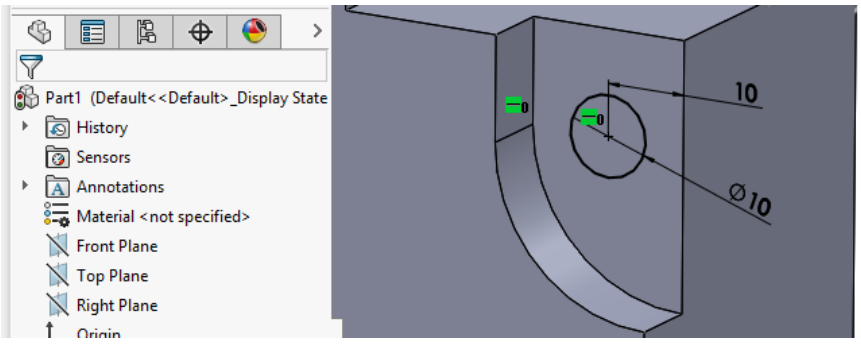
17	Extrude the sketch to a depth of 8mm.	
18	Make the sketch as drawn on the right. Can you manage yourself? If you can, proceed to step 24. If you can not do it all by yourself, follow the next steps below.	
19	1. Select the plane you want to make a sketch on. 2. Click on Sketch in the CommandManager to open the sketch.	
	Tip!	Most times when we want to make a sketch we select a plane and start drawing a line or circle. SOLIDWORKS will automatically open the sketch then. In the last step you have opened the sketch explicitly. Why? We will use the Convert Entities command first and to be able to use this command the sketch must be open already. That is the reason for this action.

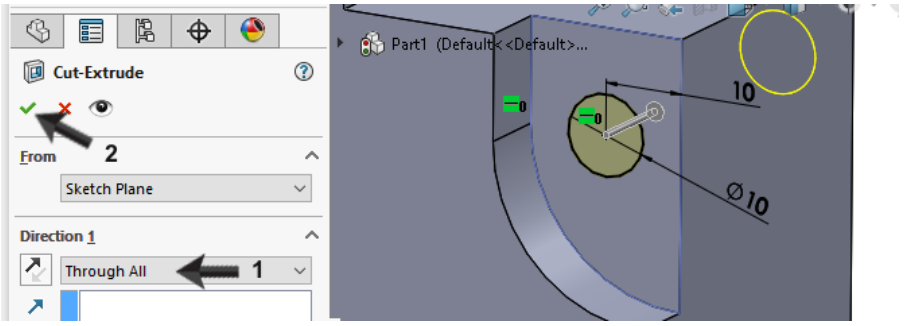
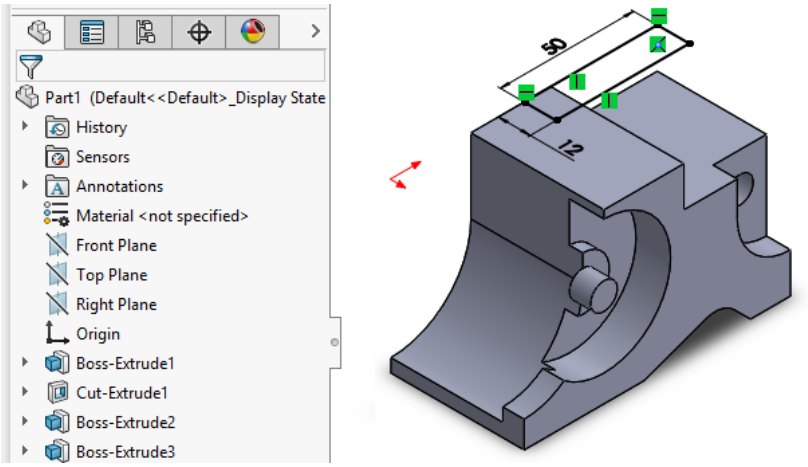
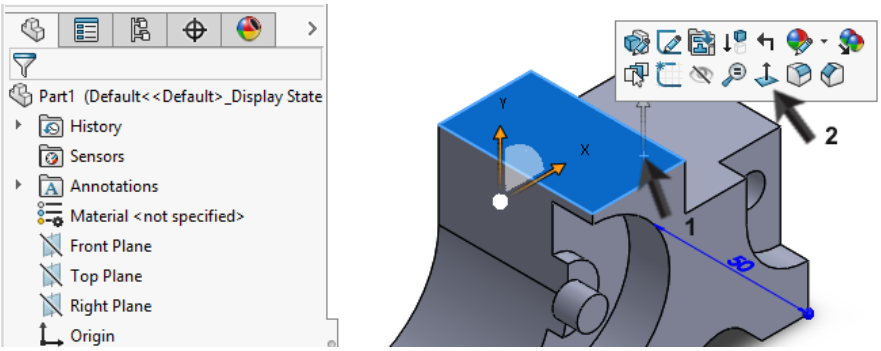
20	1-3 Select the three edges in the model as sown on the right. 4. Click on Convert Entities in the CommandManager. 5. In the PropertyManager, click OK.	
21	1. Select the edge as shown in the illustration 2. Click on Offset Entities in the CommandManager 3. Set the distance to 5mm in the PropertyManager 4. Uncheck the option Select Chain. 5. Check the option Reverse to be sure the copy will be put at the right side. 6. Click OK.	

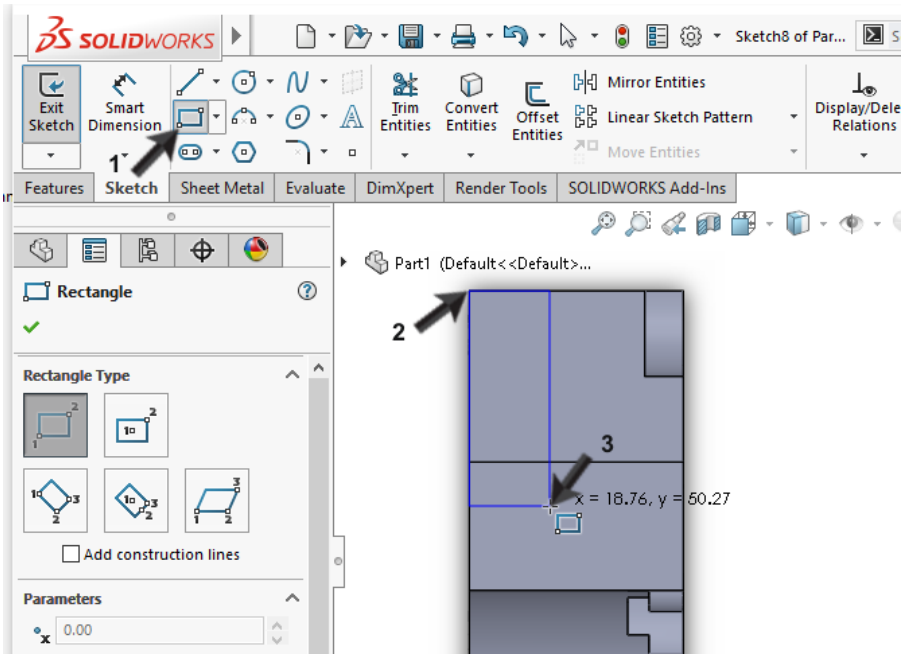
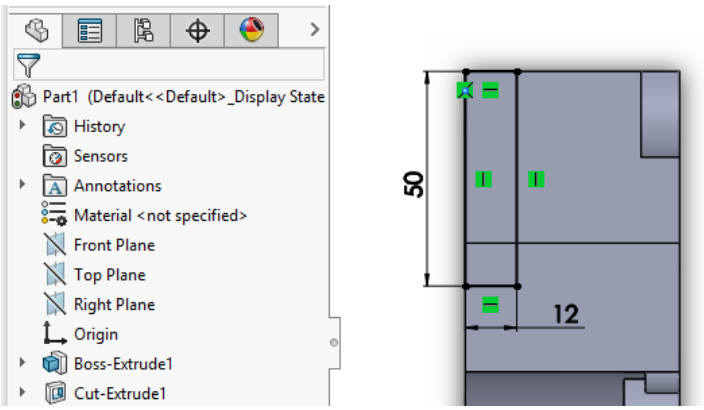
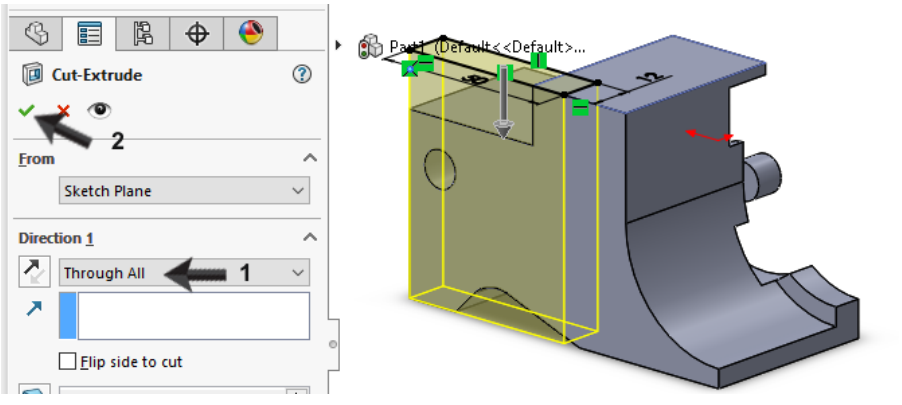
22	1. Click on Trim Entities in the CommandManager 2. Select the option Corner in the PropertyManager. 3-6 Make the upper corners by clicking as indicated in the illustration.	
23	Next make the bottom corner points by clicking as shown on the right.	
24	Extrude this sketch over 8mm. Use the Reverse Direction key to make sure the extrusion extends in the right direction.	

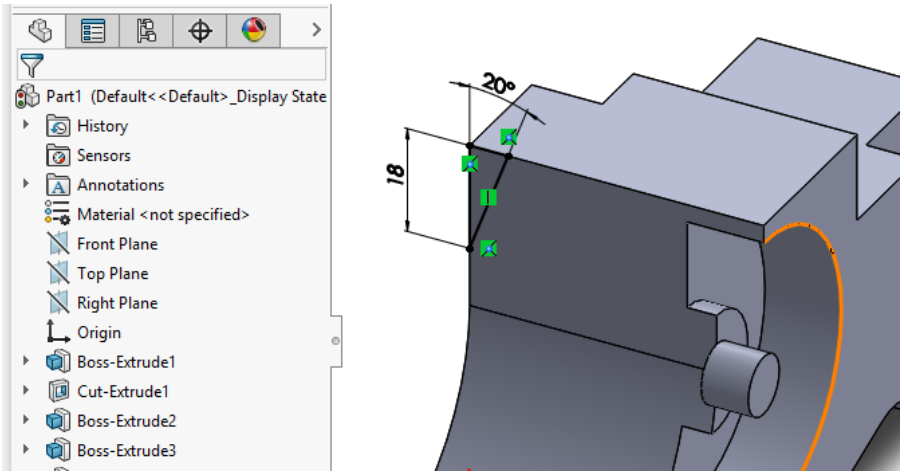
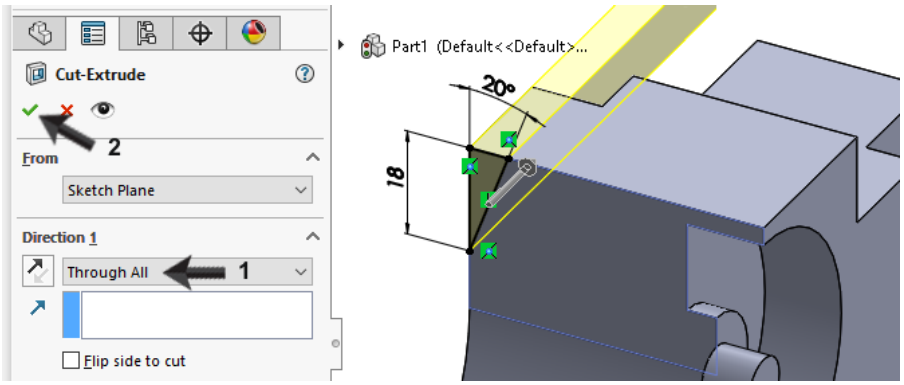
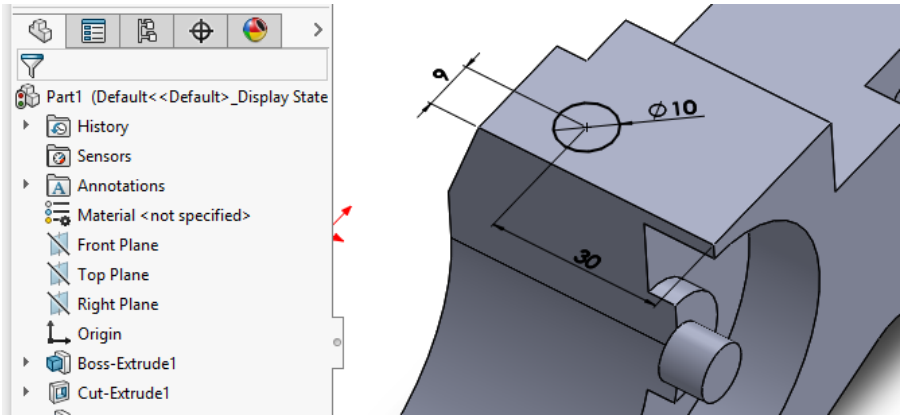
25	Make the sketch as shown. Can you manage this your-self? Continue at step 30. If not, follow the next few steps.	
26	Select the plane on which you want to make a sketch. Draw three straight lines as shown in the illustration.	
27	1. Click on Tangent Arc in the CommandManager 2. Click at the bottom end of the left vertical line. 3. Click on the bottom end of the right vertical line.	

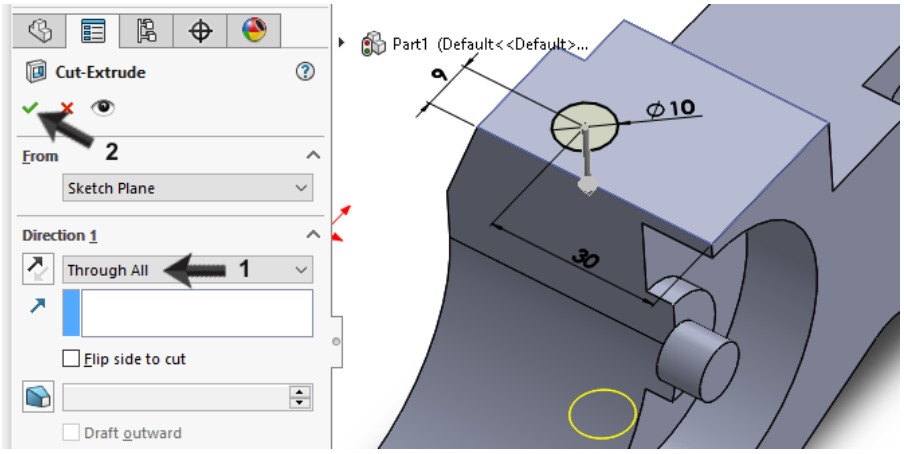
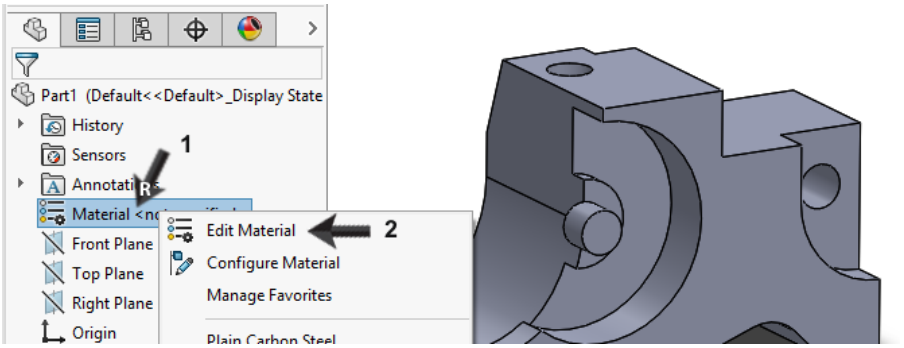
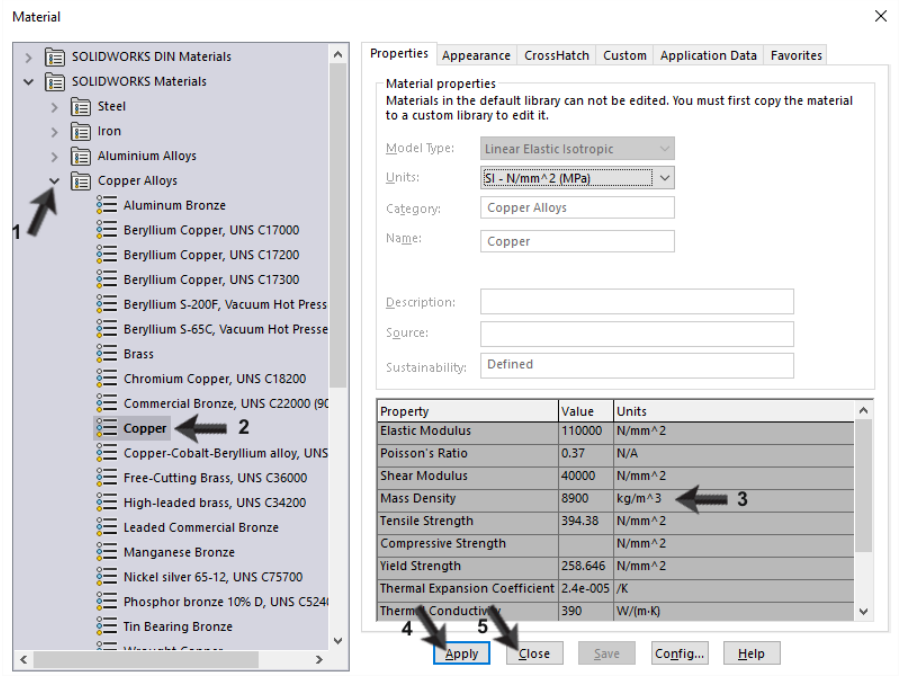
28	1. Select the midpoint of the arc. 2. Hold the <ctrl>-key and select the right vertical line too. 3. Click on Coincident in the PropertyManager.	
29	Add the two dimensions as shown.	
30	Make an Extruded Cut from this sketch with a depth of 9mm.	

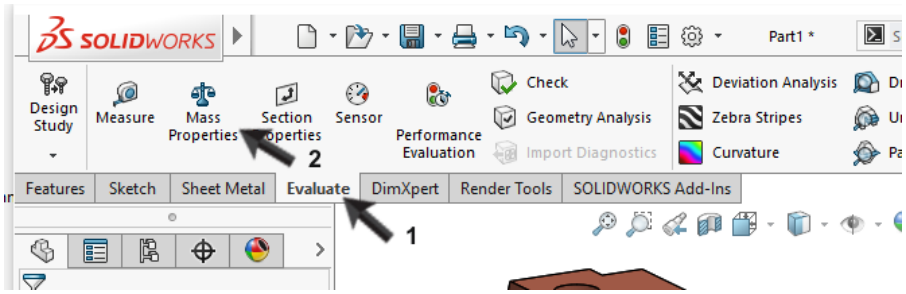
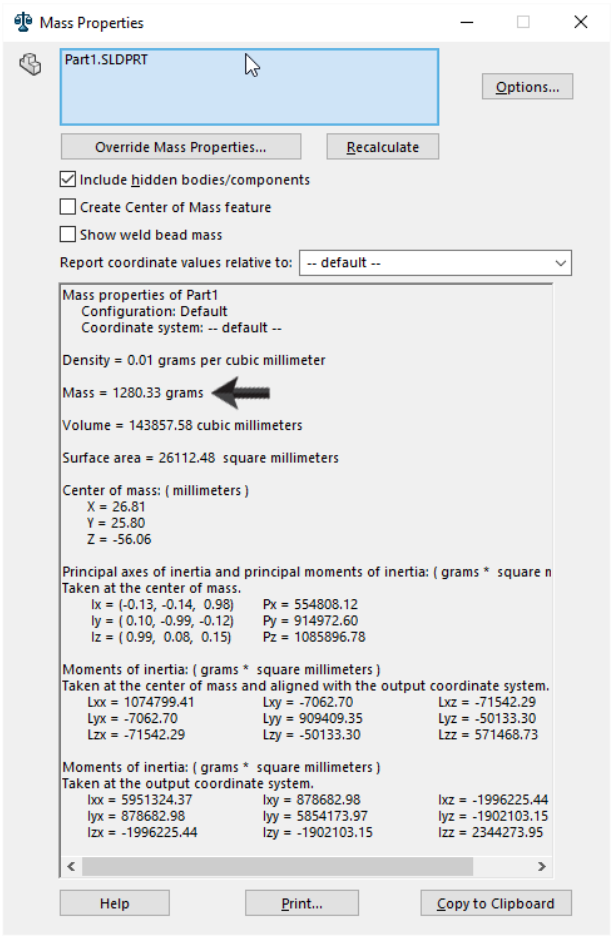
31	Make the sketch as shown and continue to step 35. If you can not manage yourself, follow the next few steps.	
32	1. Select the plane to make the next sketch as shown on the right. 2. Draw a circle, just about the size and position as in the illustration.	
33	1. Select the midpoint from the circle. 2. Hold the <ctrl>-key and click on the point as shown on the right. 3. Click on Horizontal in the PropertyManager.	
34	Set the sizes as shown in the illustration.	

35	Make an Extruded Cut from this sketch. Select the option Through All.	
36	Make the sketch as shown on the right and continue to step 40. If you can not manage yourself, follow the next few steps.	
37	1. Select the upper surface from the model 2. Click on Normal To in the popup-menu.	

38	1. Click on Rectangle in the CommandManager 2. Draw the rectangle as shown in the illustration on the right.	
39	Set the two sizes as shown.	
40	Make an extruded Cut from this sketch and set the depth to: Through all.	

41	Make the sketch as shown.	
42	Make an Extruded Cut from this sketch and set the depth to: Through All.	
43	Finally: Make the sketch as shown in the illustration on the right.	

44	Make an Extruded Cut from this sketch and set the depth to: Through All.	
45	The model is ready now. We have to select the kind of material: the assignment says 'copper'. 1. Right-click on Material in the FeatureManager. 2. When Copper is in the list, you can click on it. If not, click on Edit Material.	
46	If you chose Edit Material in the previous step, the menu will apear in which you can select a material for the model. 1. Click on 'Copper Alloys'. 2. Selecteer Copper. 3. Just to be sure: check the density at Physical Properties. Is it the same as in the assignment? 4. Click OK. 5. Click on Close	

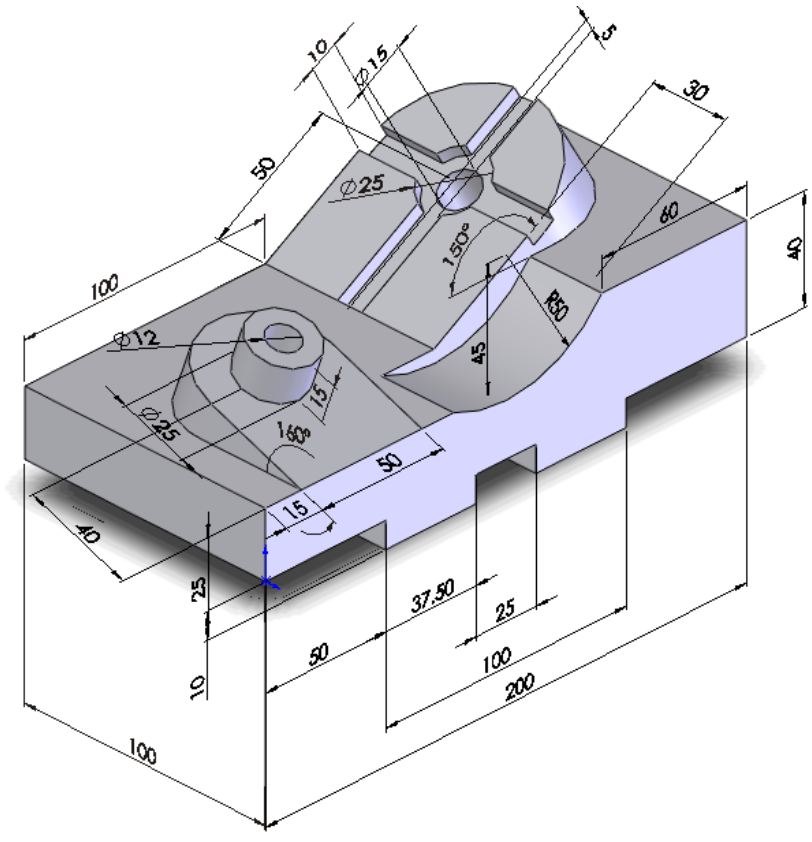
47	We need to know the weight of this part: 1. Click on the tab Evaluate in the CommandManager. 2. Click on Mass Properties.	
48	In the popup-menu you can read the weight: 1280.33 grams. This is answer **B** from the assignment.	

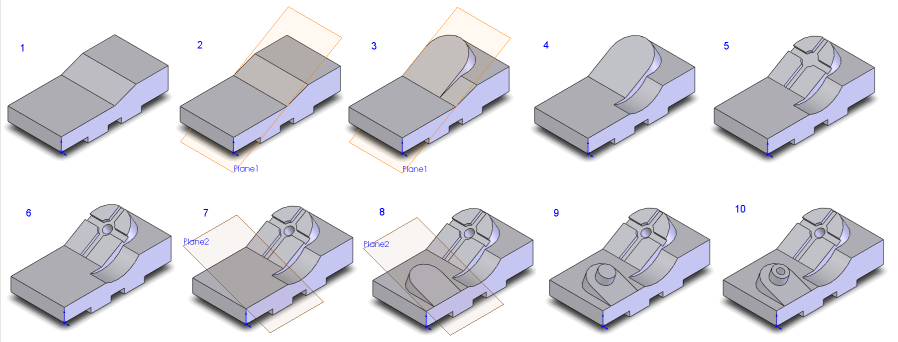
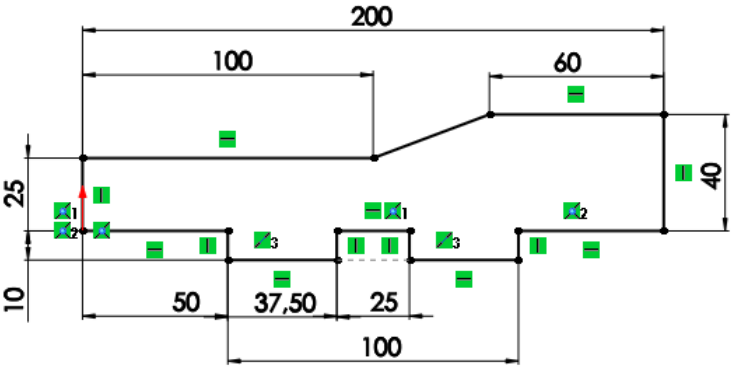
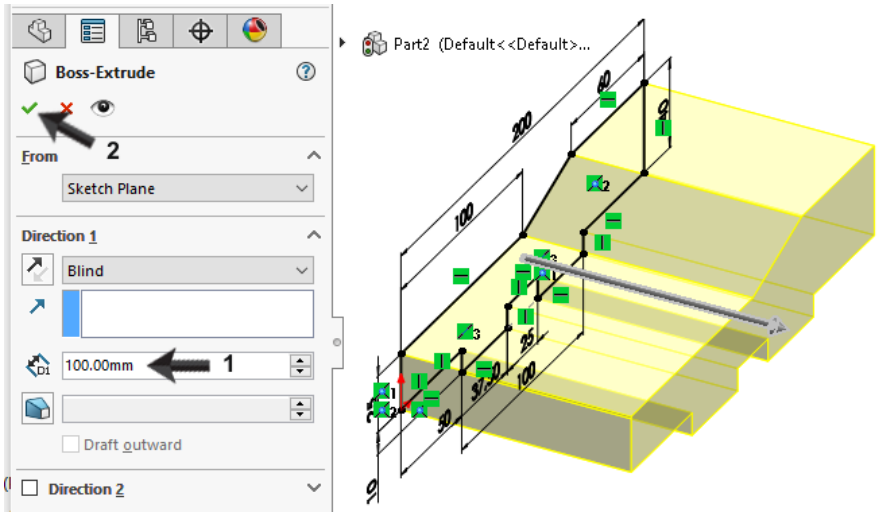
Assignment

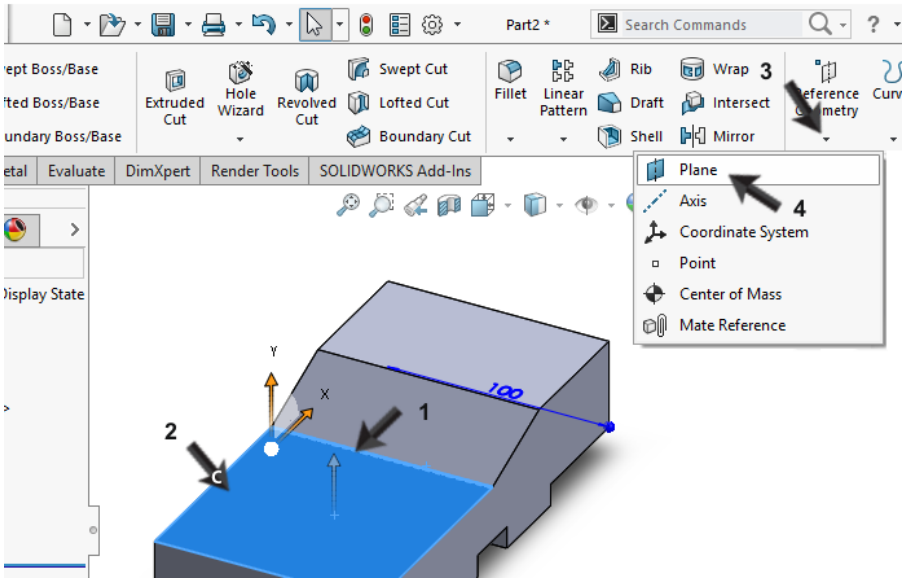
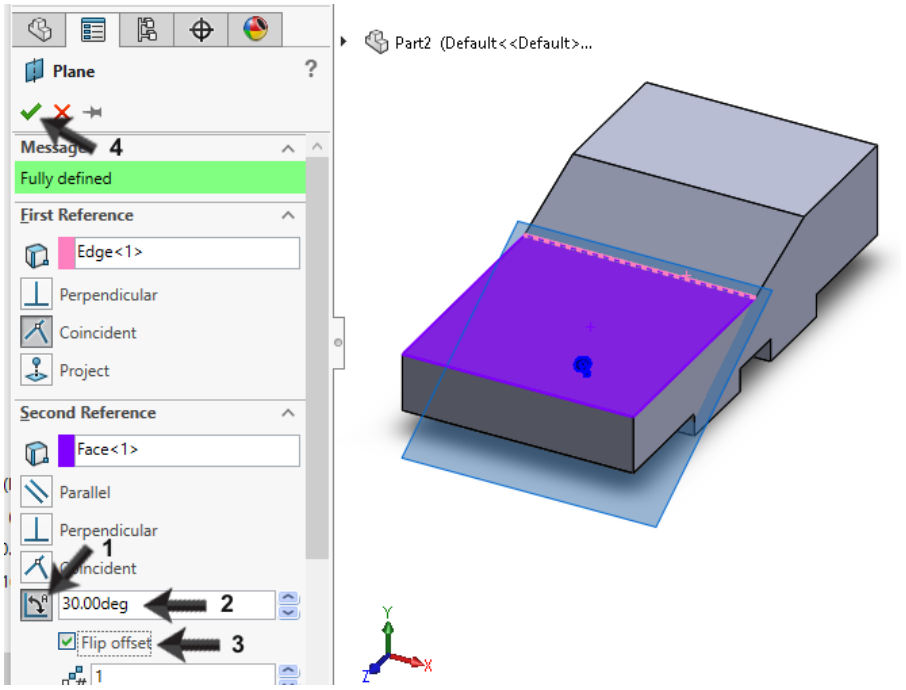
Available time: 45 min.

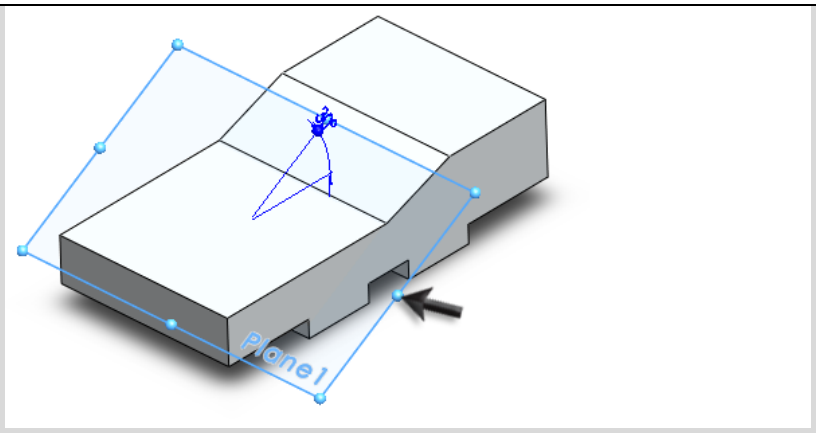
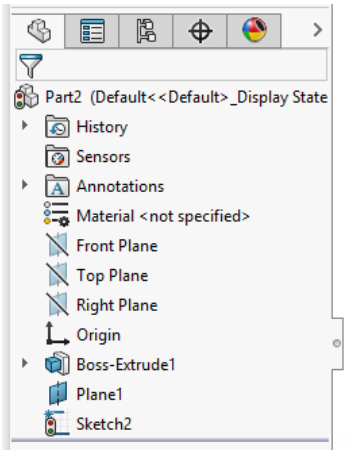
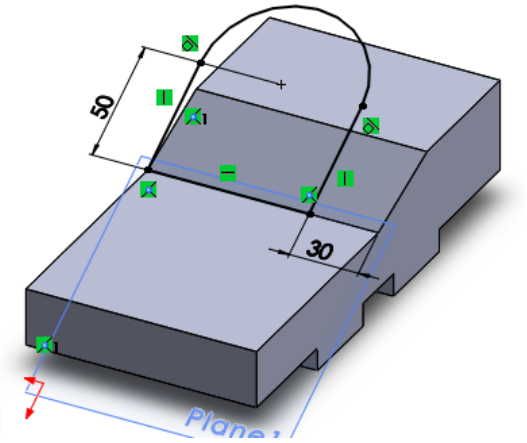
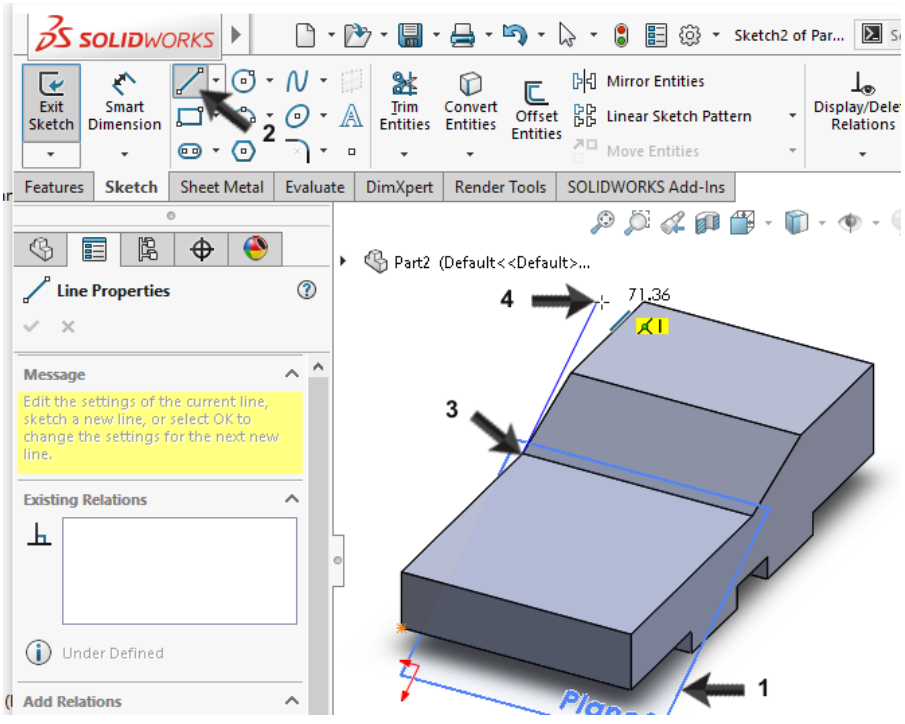
We are going to build a second model. Again, there is an assignment to this model. It is similar to the first one.

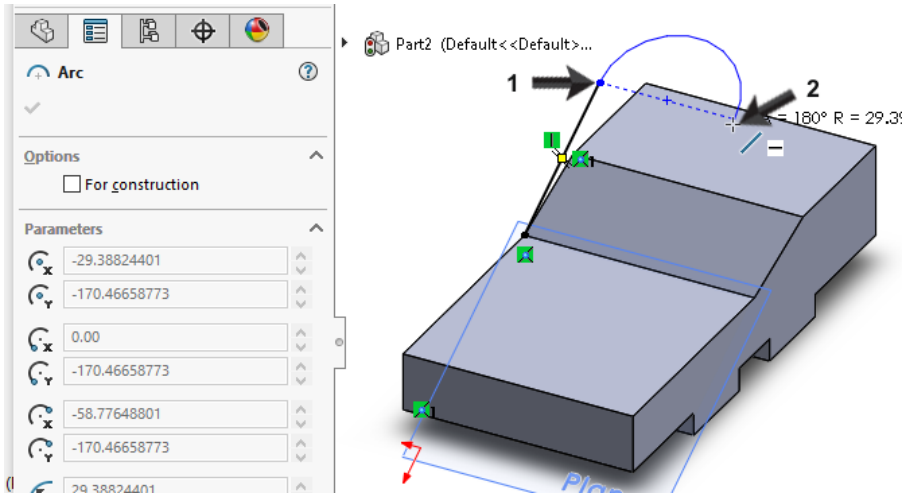
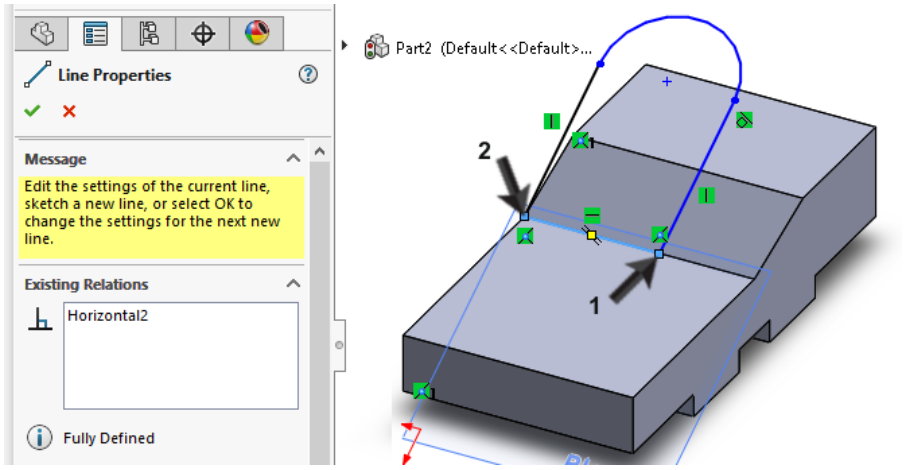
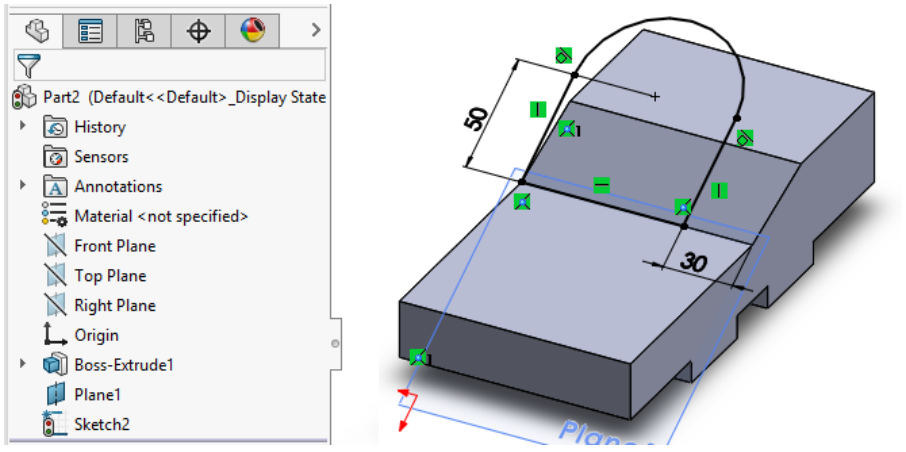
Build this part in SOLIDWORKS.
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part Origin: Arbitrary
All holes through all, unless otherwise specified
Part material: 6061 Alloy Density = 0.0027 g/mm^3
What is the overall mass of the part in grams? a. 2040.57 b. 2004.57 c. 102.63 d. 1561.23

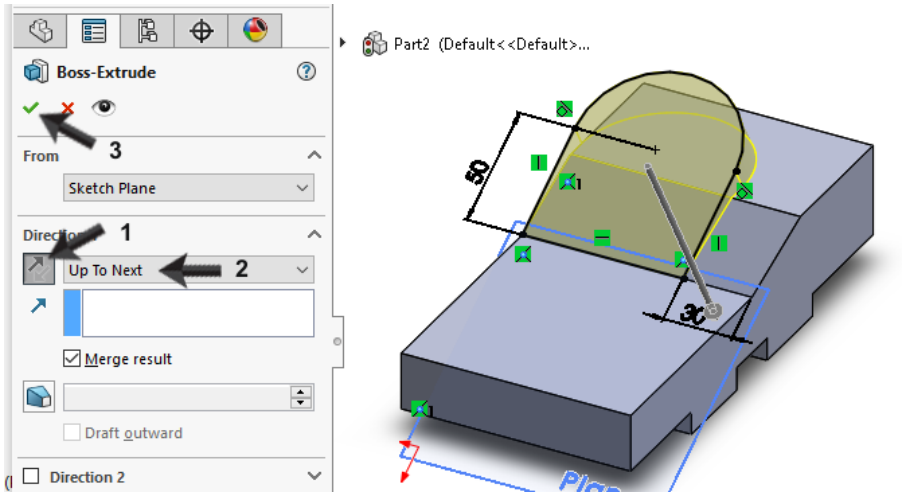
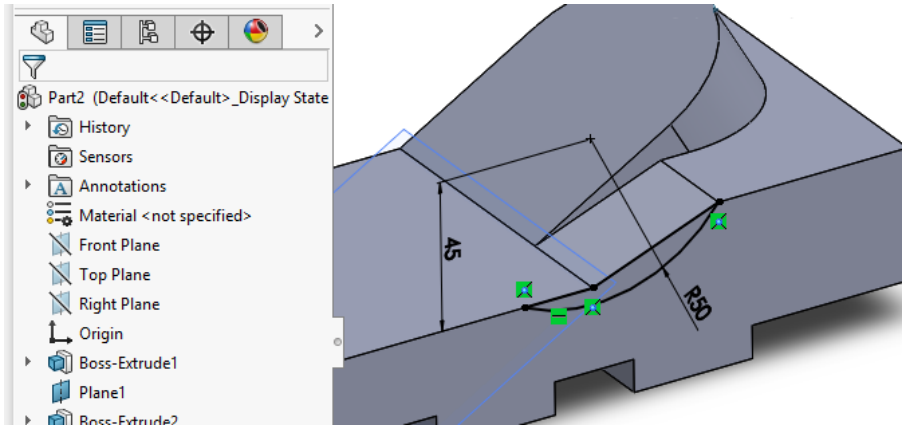
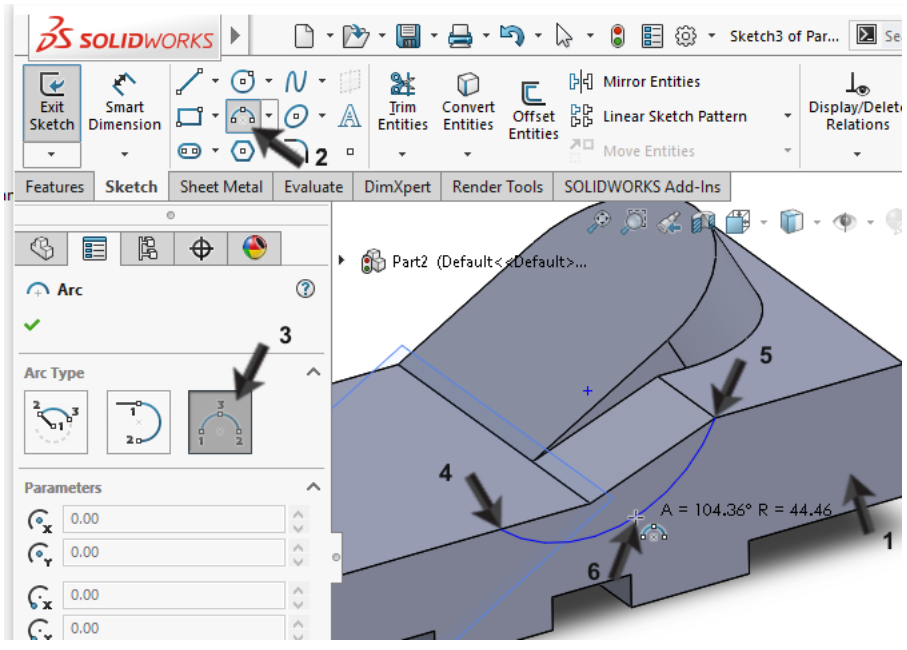


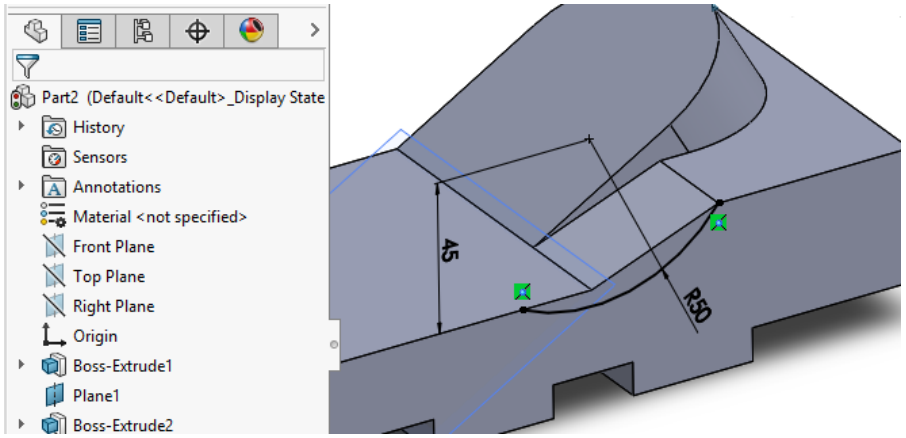
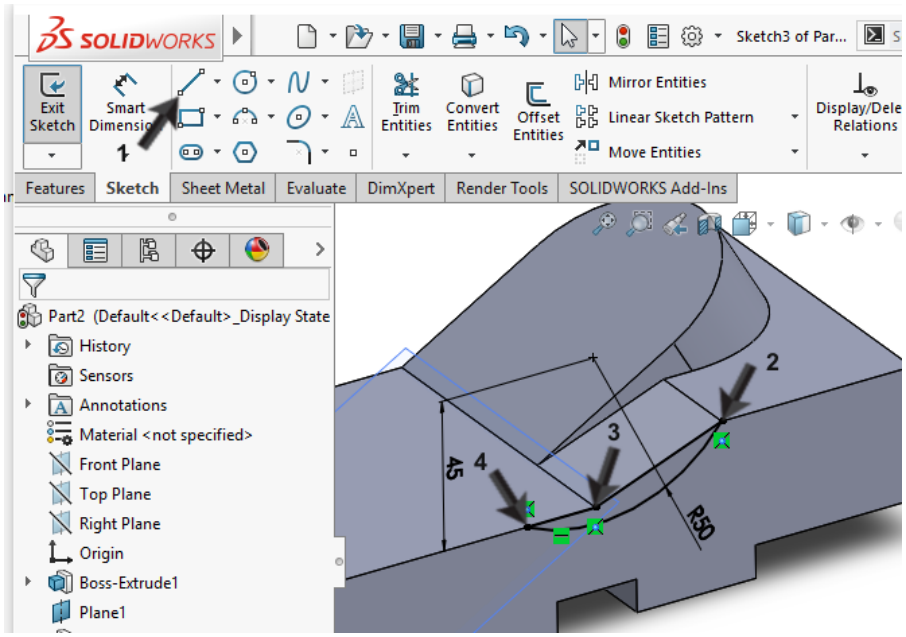
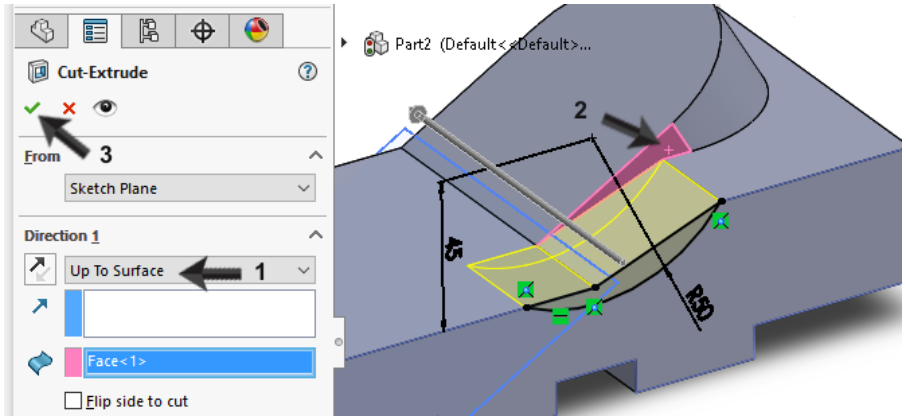
	Work plan	Again, you have to think about the way you are going to build this model. Down below are the steps you must take. Every step is a feature. 
49	Open a new part and make the sketch as shown on the right on the Right Plane.	
50	Extrude the sketch to 100 mm	

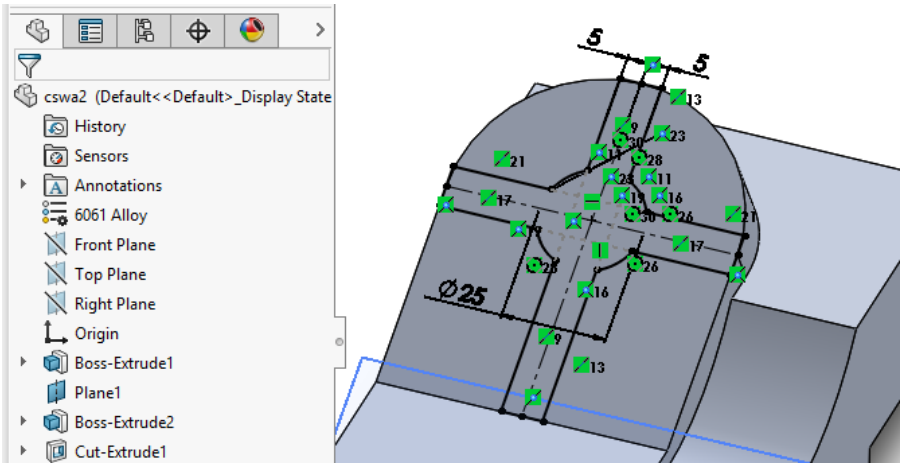
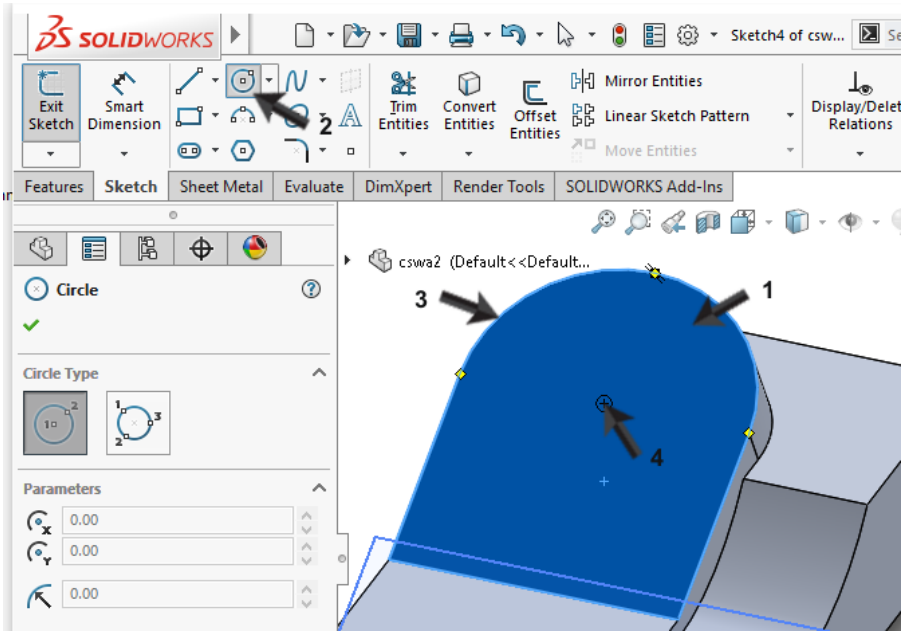
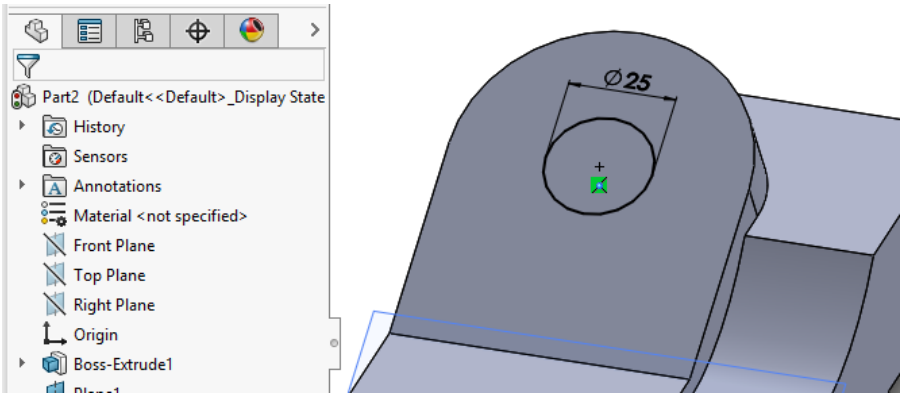
51	We will create the first reference plane: 1. Select the edge as shown. 2. Hold the <ctrl>-key and select the plane as shown in the illustration too. 3. Click on the arrow beneath Reference Geometry in the CommandManager. 4. Click on Plane.	
52	1. In PropertyManager, click Angle. 2. Set angle of the new plane to 30 °. 3. Select Flip. 4. Click ok.	
	Tip!	If the edge of the reference plane is exactly collinear with the edge of the plane you selected in step 51, you'd best enlarge the reference plane a bit. This makes it easier to select it later on. Select the plane, and drag the circles that appear near the edges. By doing so, the position of the plane won't change. It will just be shown a bit larger.

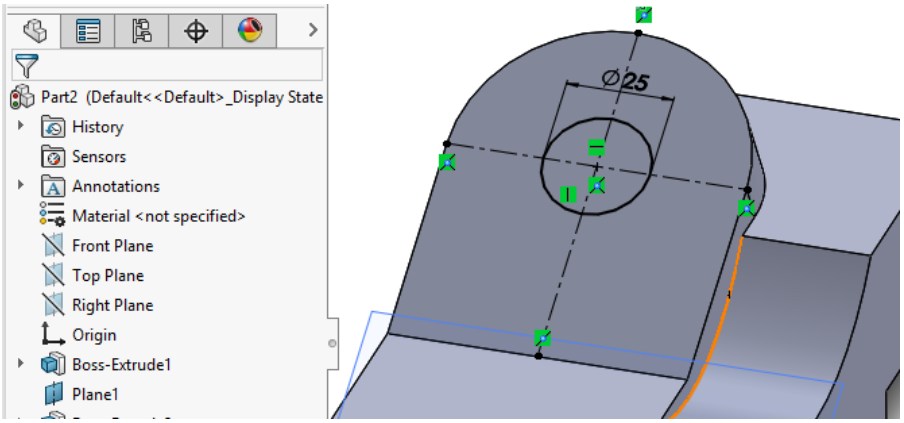
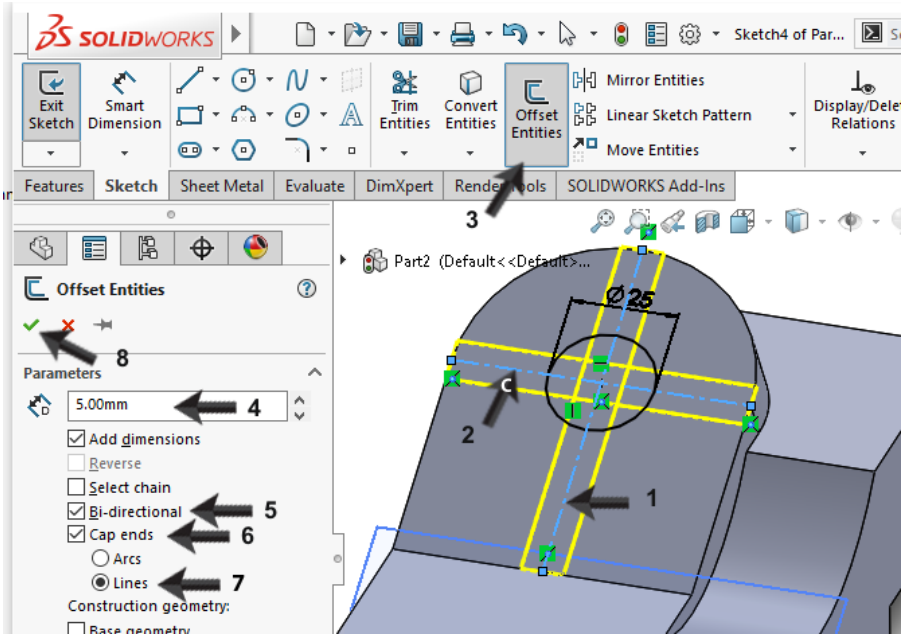
		
53	Now draw the sketch that you see next to, and proceed to step 58. Is not it possible to make this sketch? Then follow the steps below.	 
54	1 Select the help level you just created. 2 In the CommandManager, click Line 3 Click for the first point of the line as you see next to it. 4 Also click for the second point of the line as you see next to it.	

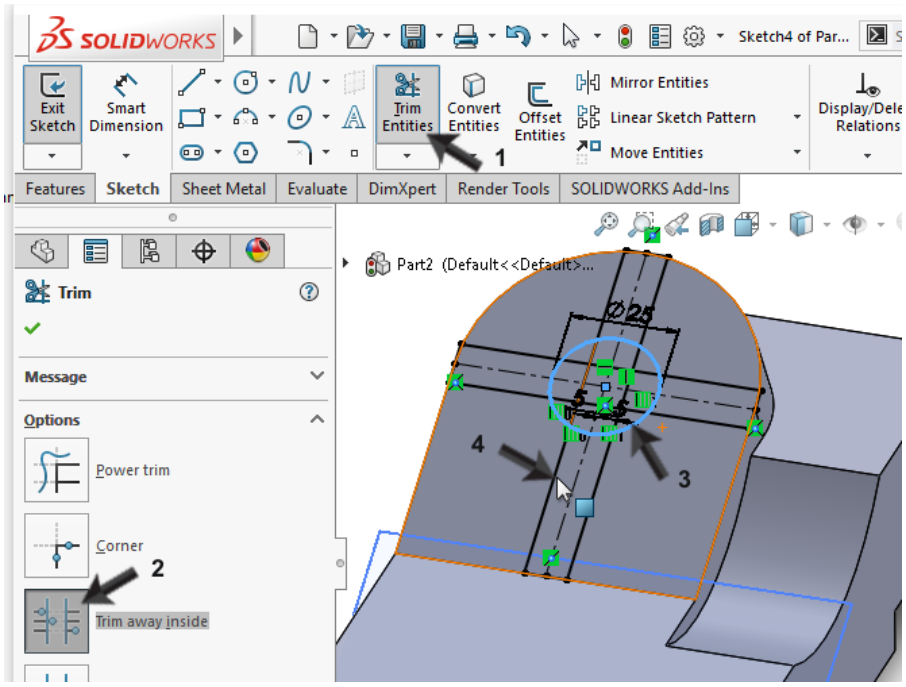
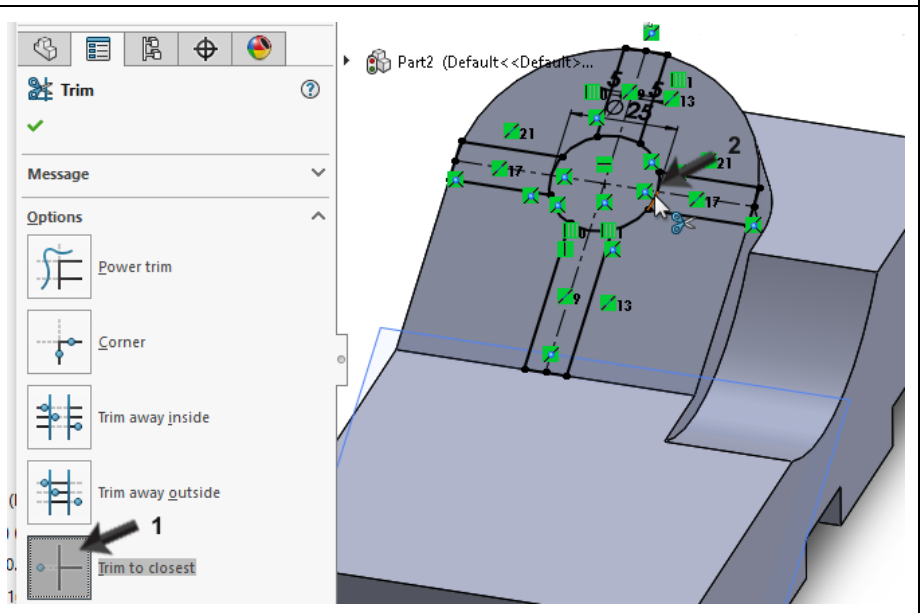
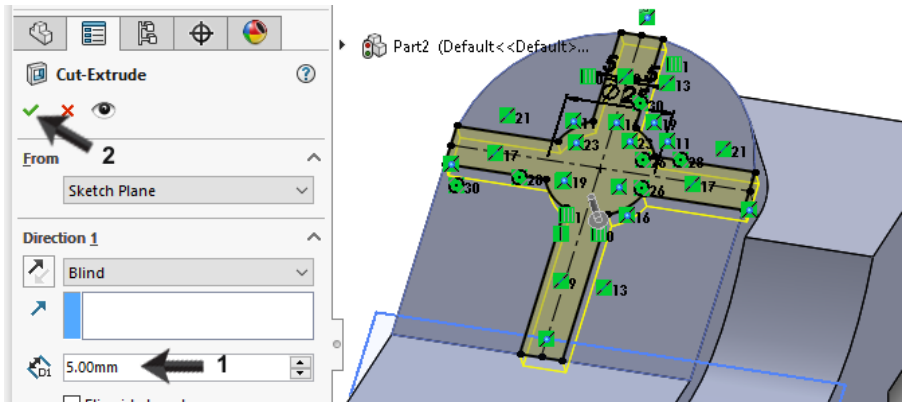
55	1. Move the cursor away from the last point and then return to it (do NOT click!) SOLIDWORKS starts drawing an arc now. 2. Click as shown to get the second point of the arc. Make sure to draw half a circle. Note the dashed construction line.	
56	SOLIDWORKS will automatically draw lines again. Draw the two last lines.	
	Tip!	You saw an 'automatic' change of function between the Line and Circle command. This is called Autotransitioning in SOLIDWORKS and is very convenient if you want to build a sketch from lines and coincident circles.
57	Add the two dimensions as shown with Smart Dimensions.	

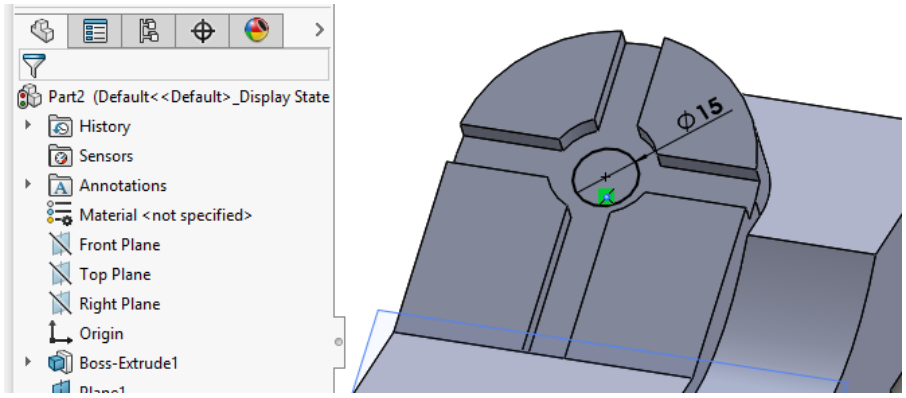
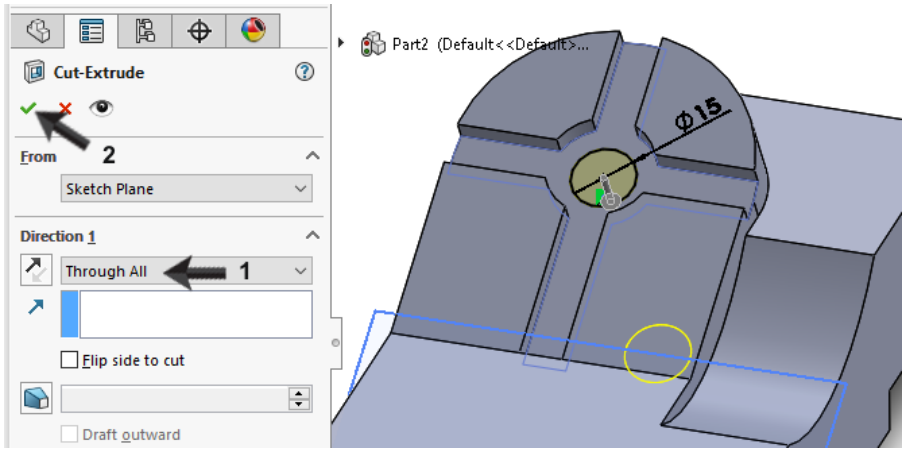
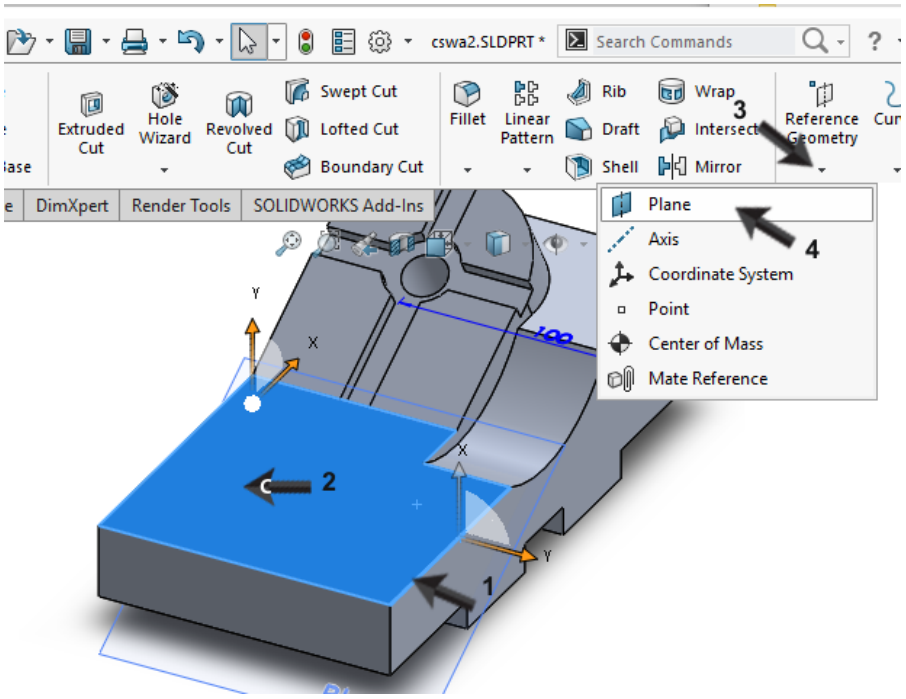
58	Make an extrusion from this sketch. 1. Click on Reverse Direction in the PropertyManager to make sure that the extrusion goes downwards and not upwards. 2. Select Up to Next to set the depth. 3. Click OK.	
59	Make the sketch as shown in the illustration on the right and continue to step 63. If you can not make this sketch by yourself, then follow the next few steps.	
60	1. Select the plane to make a sketch on. 2. Click on Arc in the CommandManager. 3. Click on 3 Point Arc in the PropertyManager 4. Set the first arc point at the corner as shown 5. Set the second point on the edge 6. Set the third point at a random position.	

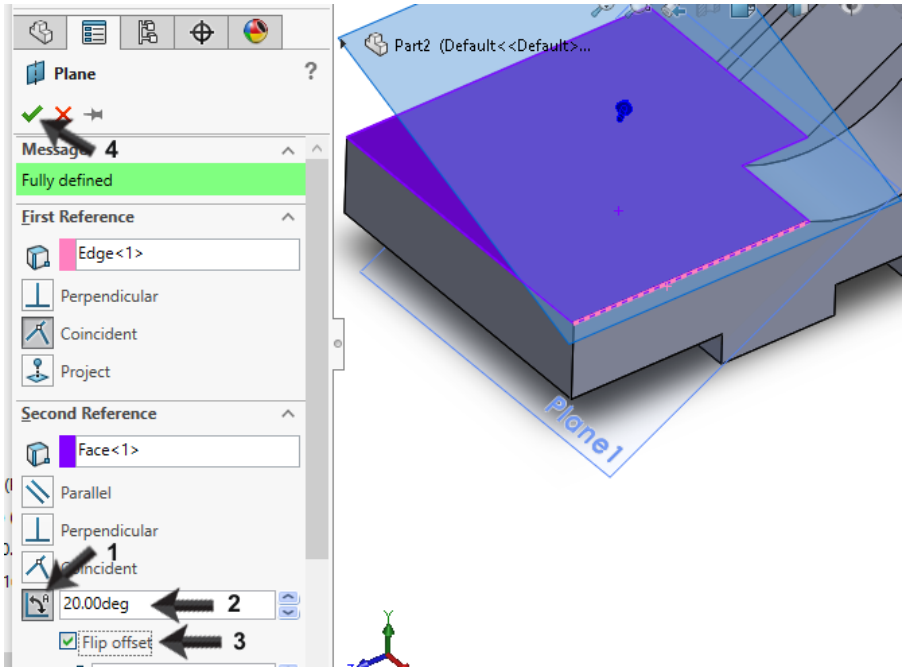
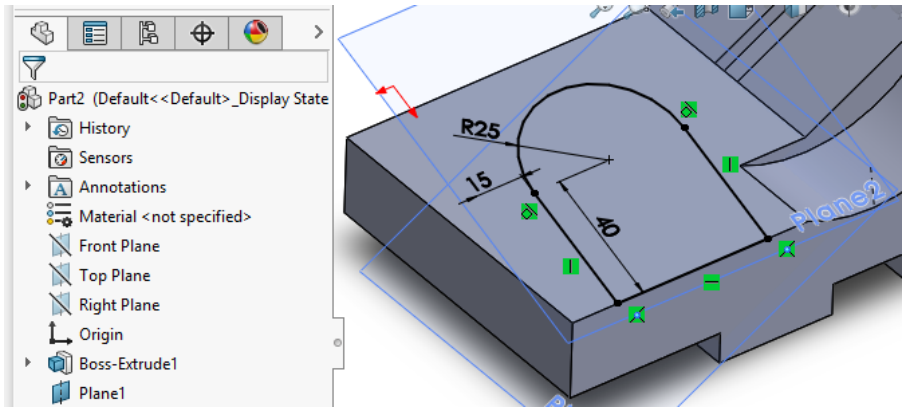
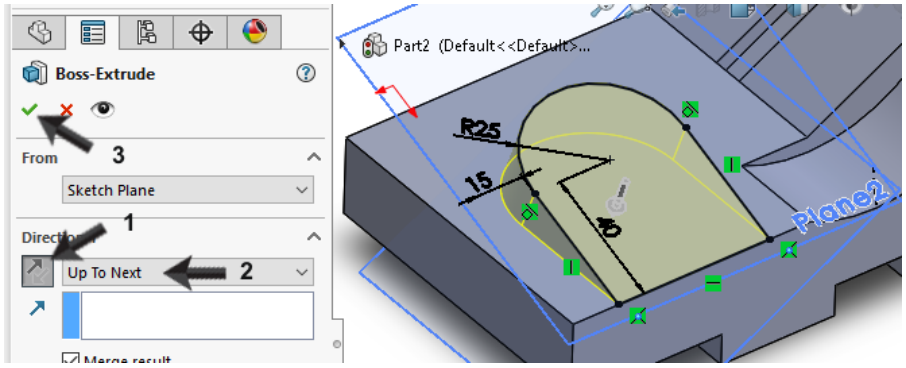
61	Insert the two dimensions as shown.	
62	Draw two small lines above the arc as shown.	
63	Make an Extruded Cut from this sketch. 1. Select the option Up To Surface to set the depth 2. Click on the plane which indicates the end of the Cut Extrude.	

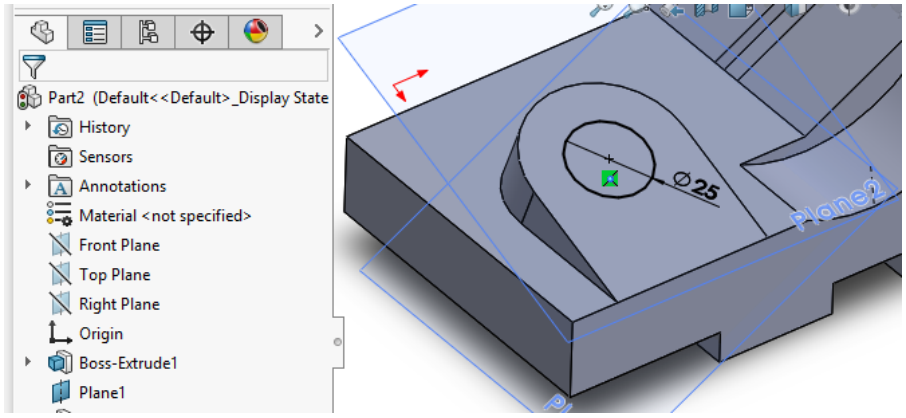
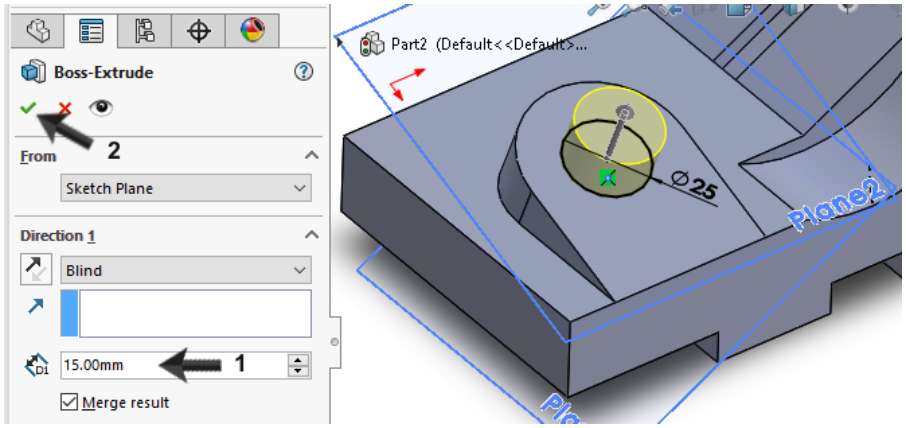
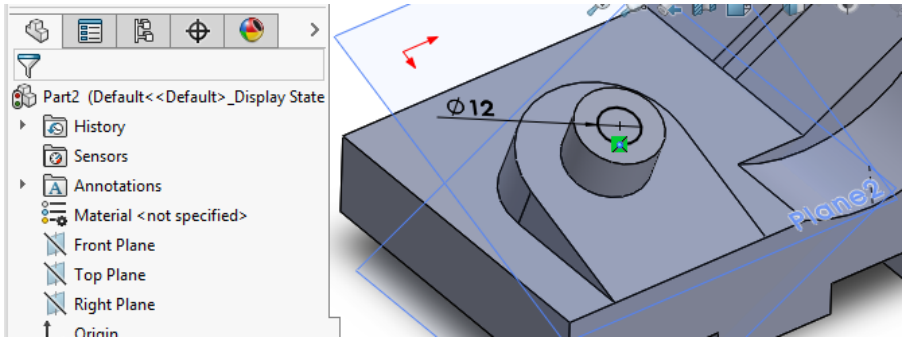
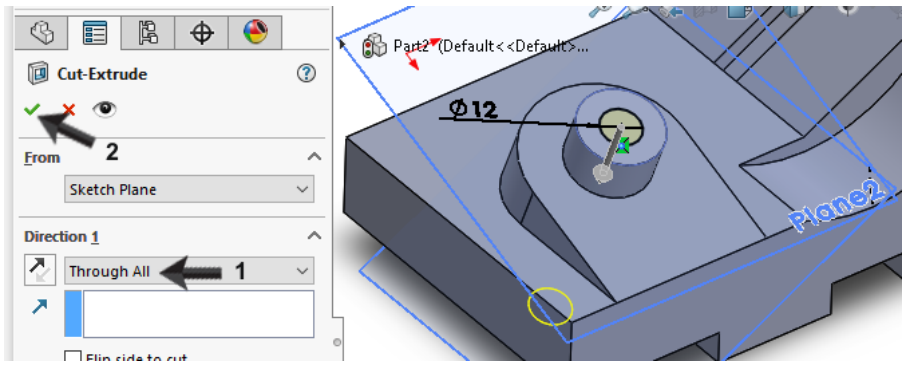
64	Make the sketch on the sloped plane as shown in the illustration on the right and continue to step 58. If you can not make this sketch by yourself, then follow the next few steps.	
65	1. Select the sloped plane first to make a sketch on. 2. Click on Circle in the CommandManager 3. Keep the cursor still just above the rounded edge at the top of the plane. **Do NOT click!** 4. The midpoint of the edge appears. Click on that to set the middle of the circle.	
66	Draw the circle and add the dimension.	

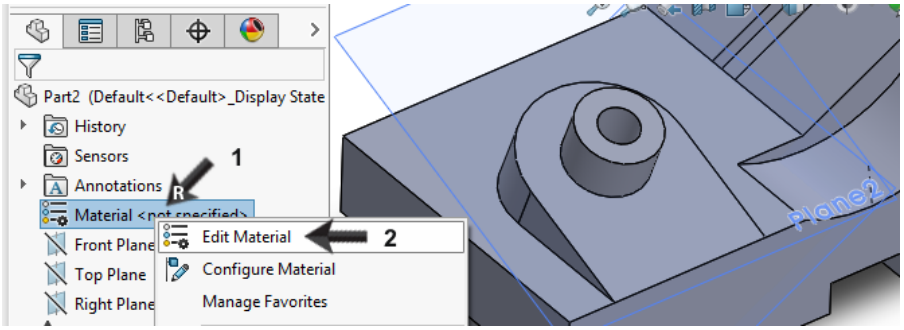
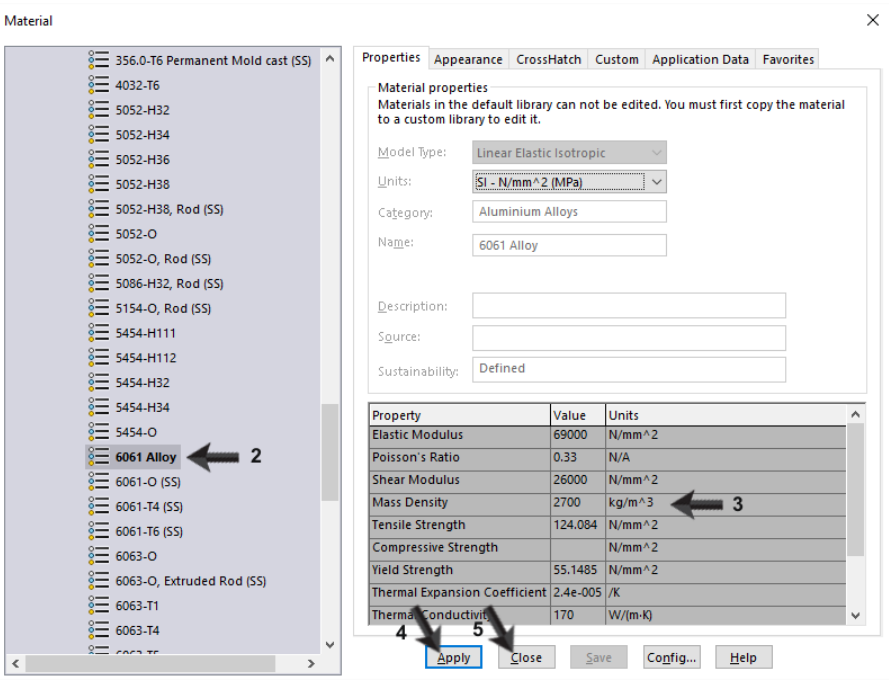
67	Draw two centerlines as shown on the right. Push <esc> after you have drawn the first centerline, next you have to draw the second centerline.	
68	1,2 Select both centerlines (use the <ctrl>-key) 3. Click on Offset in the CommandManager 4. The distance is 5mm 5. Select the option Bi-directional. 6. Select the option Cap ends. 7. Select the option Lines. 8. Click OK.	

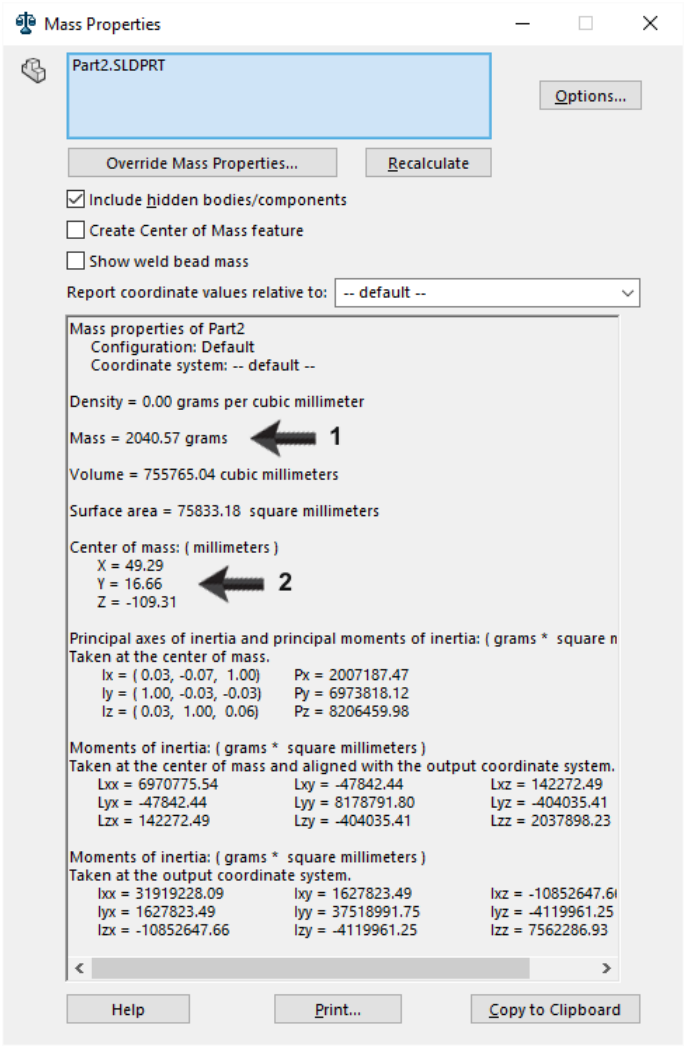
69	1. Click on Trim Entities in the CommandManager. 2. Click on the option Trim away inside in the PropertyManager. 3. Click on the circle 4. Click on all four lines that run through the circle. The pieces at the inside of the circle will be removed.	
70	1. Click on Trim to Closest in the PropertyManager 2. Click on the parts of the circle that you want to be removed.	
71	Did you trim everything, then you can make an Extruded Cut from the sketch. Set the depth to 5mm.	

72	Make the sketch as shown in the illustration on the right.	
73	Make an Extruded Cut Through All.	
74	We will now make the second construction plane. 1. Select the edge as shown. 2. Hold the <ctrl>-key and select the plane as shown in the illustration. 3. Click on the arrow beneath Reference Geometry in the CommandManager. 4. Click on Plane.	

75	1. In the PropertyManager, select the option Angle 2. Set the angle of the new plane to 20° 3. Click on Flip offset, so the plane extends in the right direction. 4. Click OK.	
76	Make a sketch as shown on the plane that you have just created. In steps 54 to 56 you have already made a similar sketch. If you want, you can check these steps to see how it is done.	
77	Make an extrusion from the sketch. 1. First click on Reverse Direction in the PropertyManager to extend the extrusion downwards. 2. Select the option Up To Next. 3. Click OK.	

78	Make the sketch as shown on the right.	
79	Extrude the sketch with a height of 15mm.	
80	Make the sketch as shown on the right.	
81	Make a Cut Extrude Through All from this sketch.	

82	The model is finished now. We will select the kind of material now. 1. Right-click on Material in the FeatureManager. 2. Click on Edit Material.	
83	1. Open the list of Aluminum Alloys in the PropertyManager. 2. Select 6061 Alloy 3. Verify if the density is the same as the one in the assignment. 4. Click OK.	

84	Finally we want to know the total mass from this part. Click on the tab Evaluate in the CommandManager and next on Mass Properties. In the popup-menu you can read a weight of 2040.57 grams. So this is answer **A** from the assignment. You can also see the Center of mass. This value is displayed in an X-, Y- and Z-coordinate in relation to the origin. In the model itself the center of mass is indicated too.	
	What are the main features you have learned in this tutorial?	As we have said in the introduction of this tutorial you did not learn a lot of new features. You have seen a few smart gadgets, though: • Some of the option from the offset-command • Some of the options from the Trim-command • The automatic change between a line and an arc shaped line while creating a sketch. More important is that you have seen and have practiced with two examples of CSWA-exams. You have seen how to make a plan when modeling a more complex part. By practicing a lot with this kind of assignments you can gain the necessary routine for passing the CSWA-exam and hence receiving a CSWA-certificate.

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