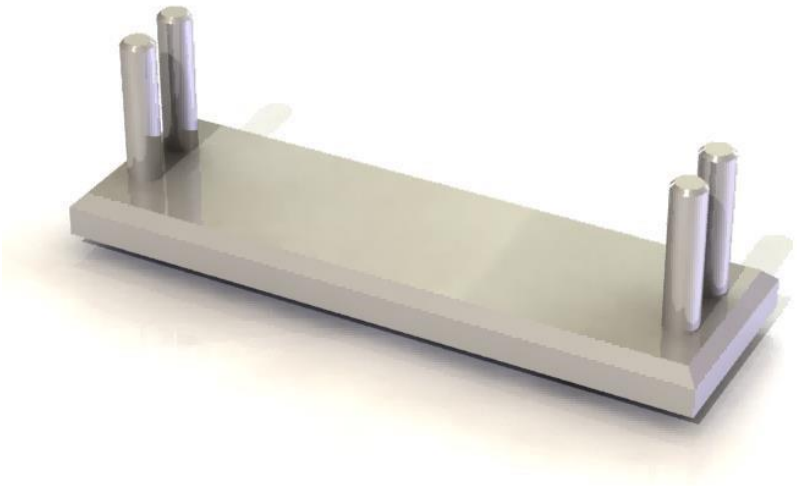
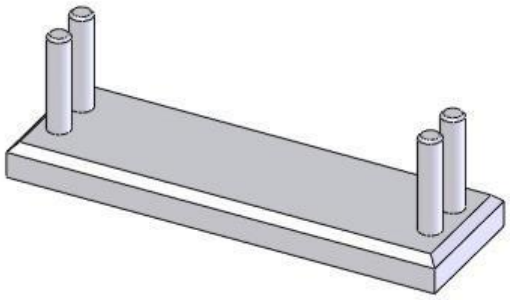
SOLIDWORKS tutorial 2

PICTURE HOLDER



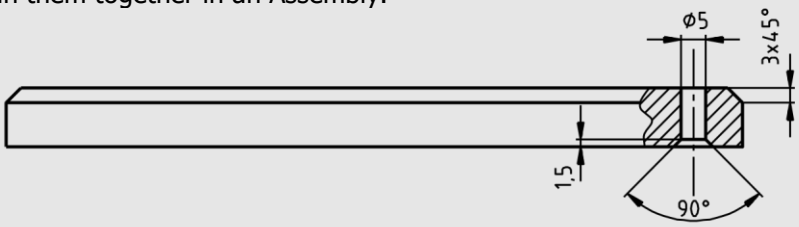
Picture holder

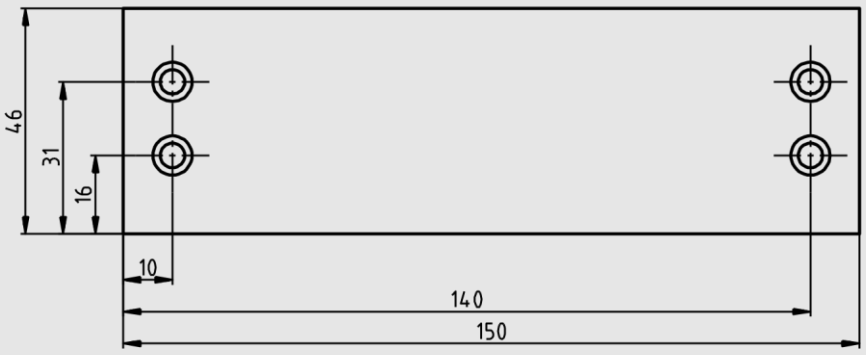
In this tutorial you will be making a picture holder, consisting of a rectangular base with 4 vertical axes on it. You will get to know some new features, e.g. the "chamfer" command. You will also get to know the Assemblies command.



Work plan

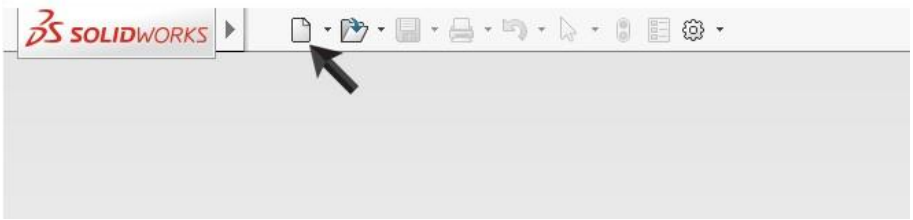
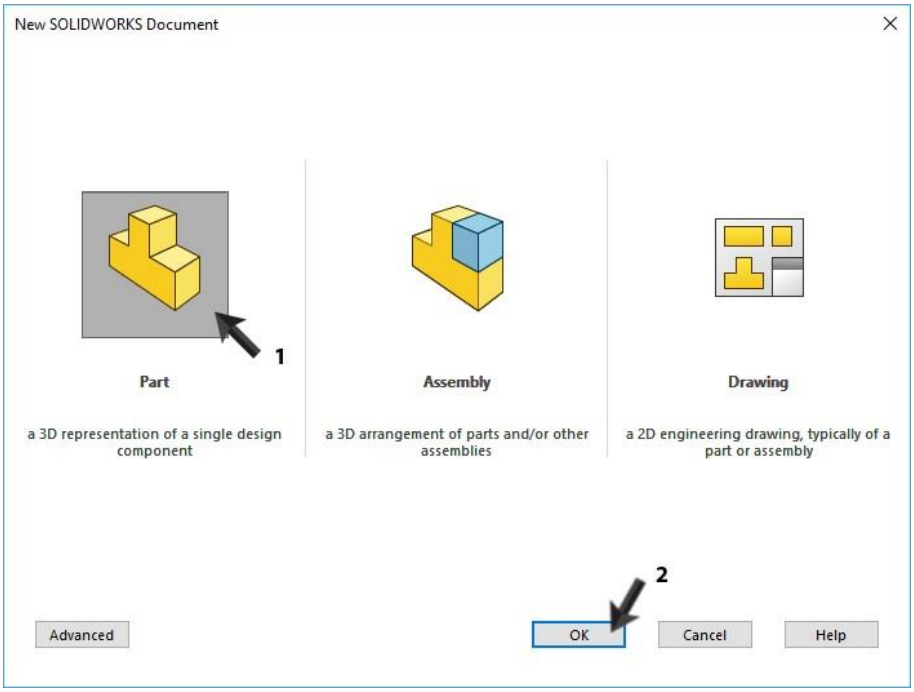
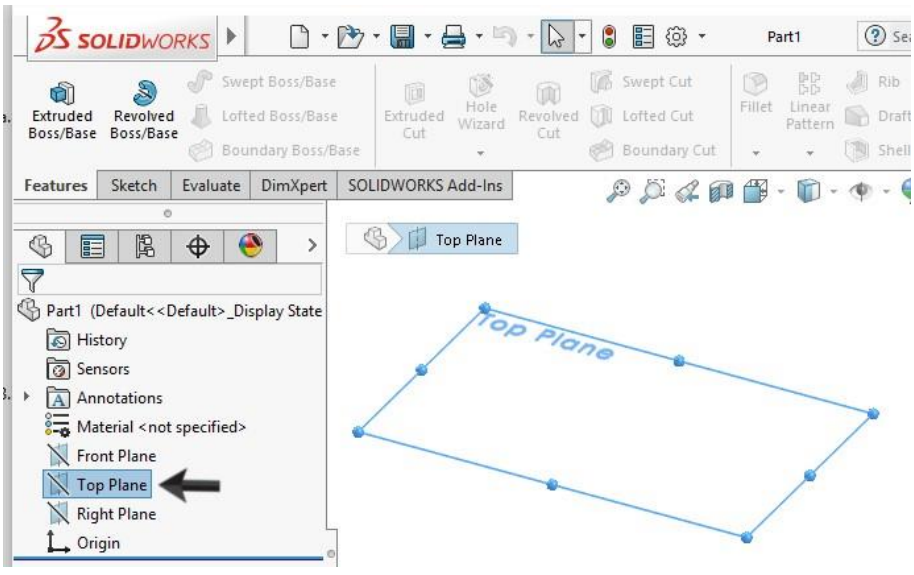
This time also we will be examining how to shape this product. It has two different parts, which we will design separately from each other. After that, we will join them together in an Assembly.

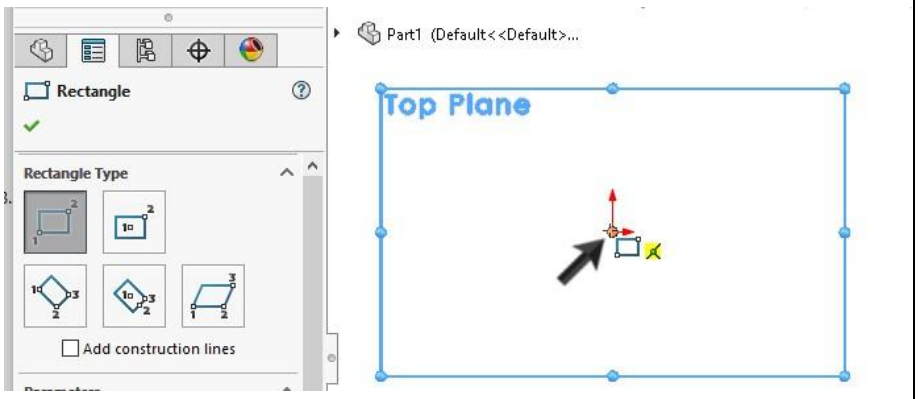
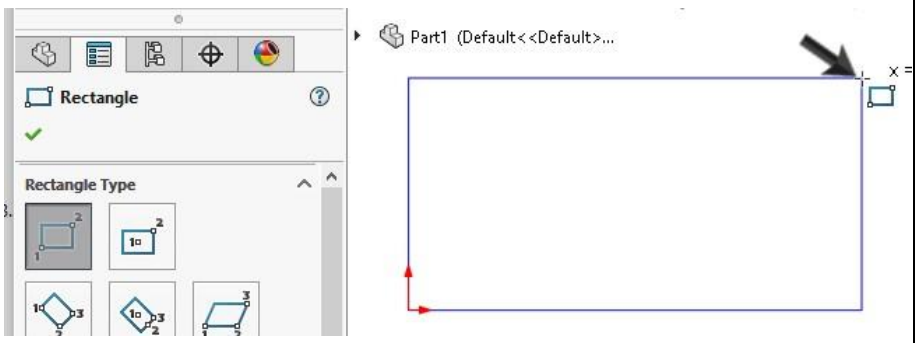
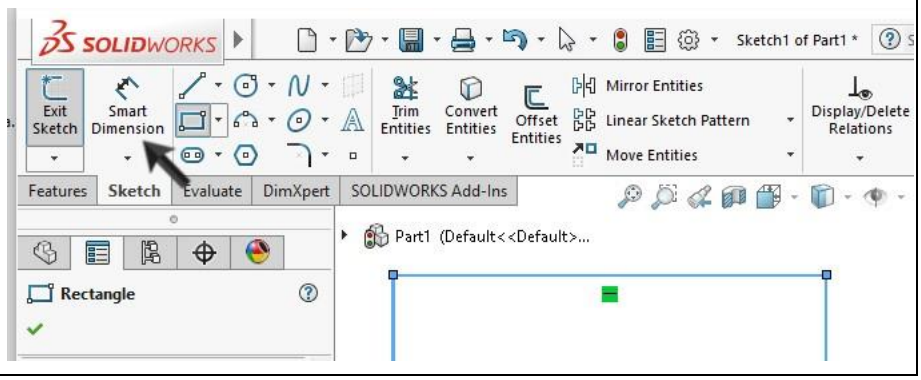


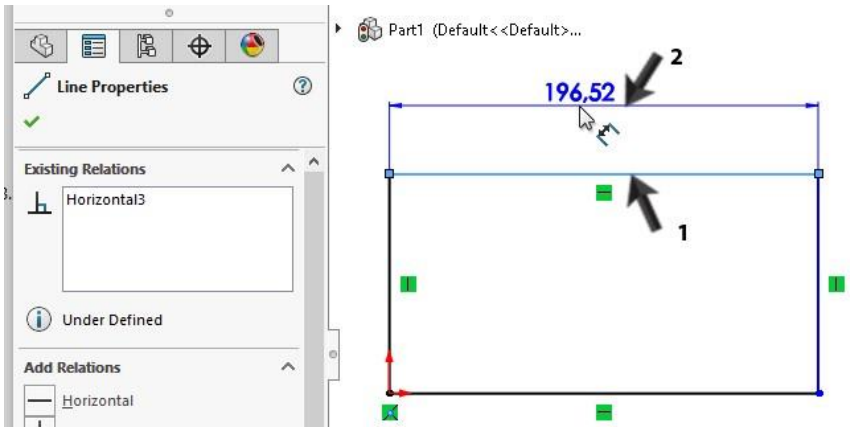
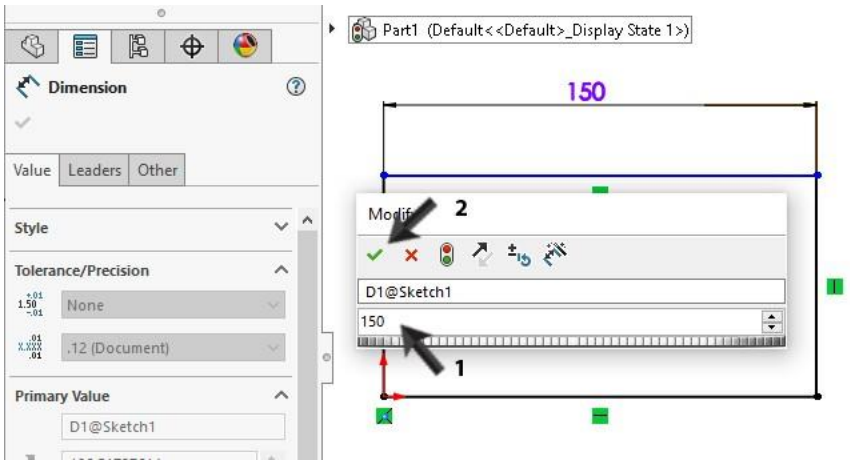
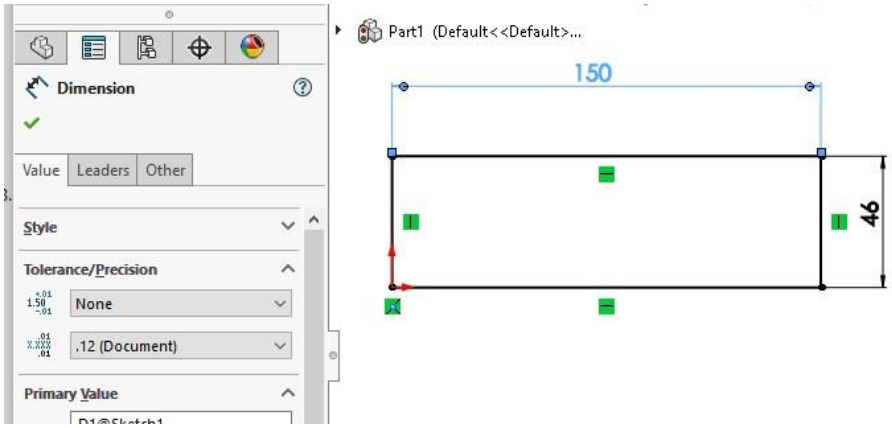


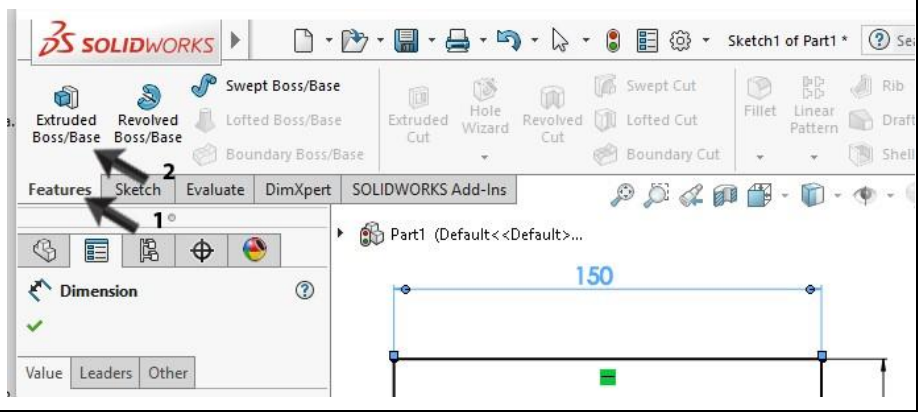
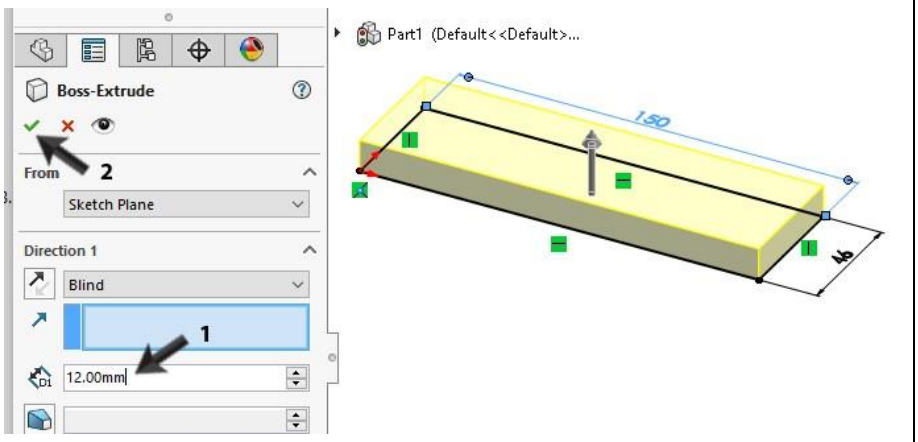
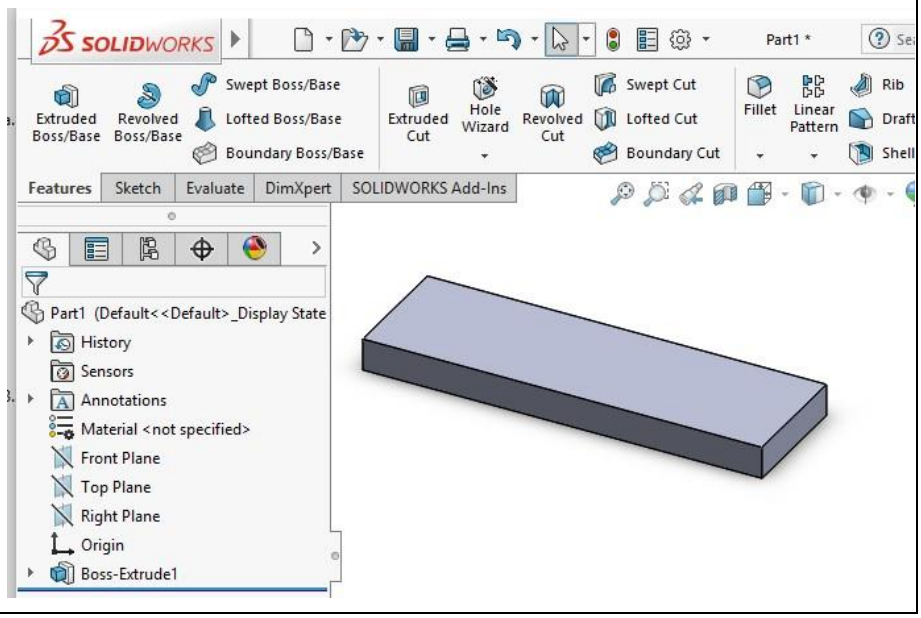
We will start with the base. We will use the same working order as we would do in the workshop:

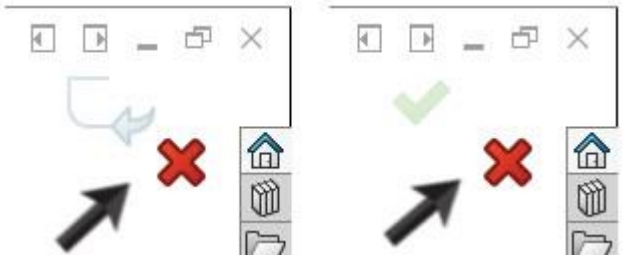
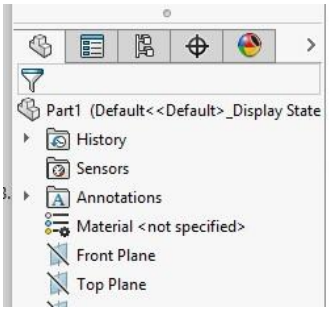
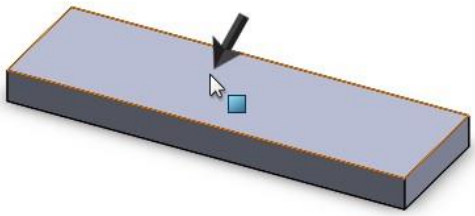
1. use a piece of material with following dimensions: 150x46x12
2. chamfer the ribs of the top plane
3. drill four holes with a diameter of Ø5
4. counter bore the holes at the bottom plane

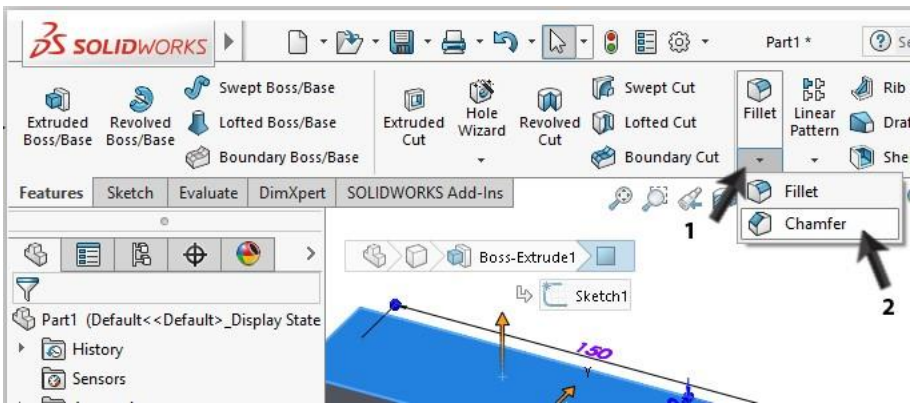
1	Start SOLIDWORKS and open a new file: click on New.	
2	Of course we will start the making of a part first. 1 Click on the 'Part' button in the menu first. 2 Then click on 'OK'.	
3	Click on 'Top Plane' in the left column of the Feature Manager. On this plane we will make a sketch.	

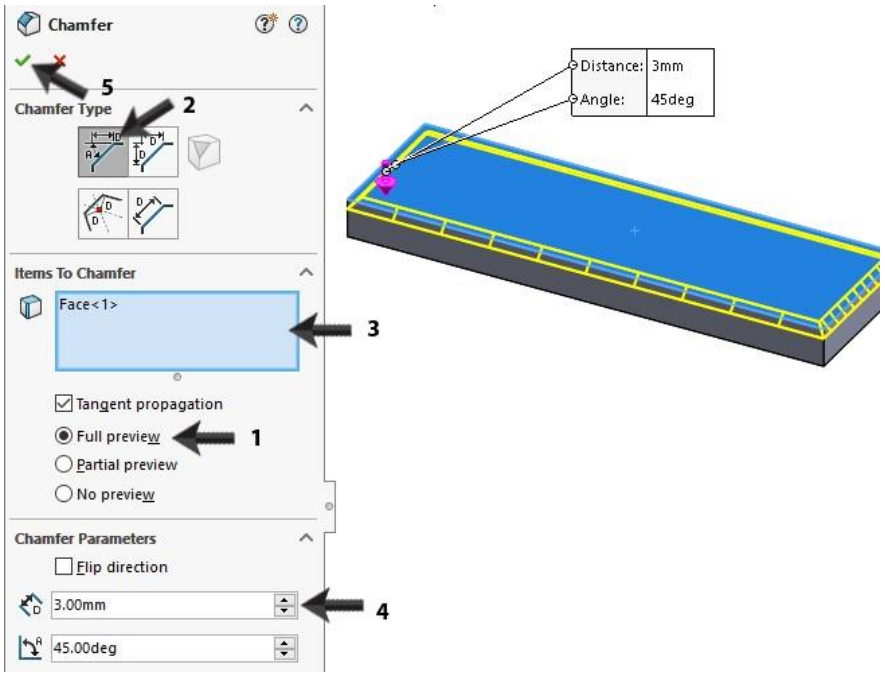
4	Click on 'Sketch' in the CommandManager (which is the menu at the top of the screen) to show the right buttons. Then click on Rectangle to draw a rectangle.	
5	Put the mouse right above the origin and it will change shape like in the view on the right. Click once.	
6	Move the mouse away from the origin. At the cursor the dimensions of the rectangle you are drawing will appear. The accurate dimensions are not important yet. Click again to draw the rectangle.	
7	Now we will set the accurate dimensions: click on 'Smart Dimension' in the CommandManager.	

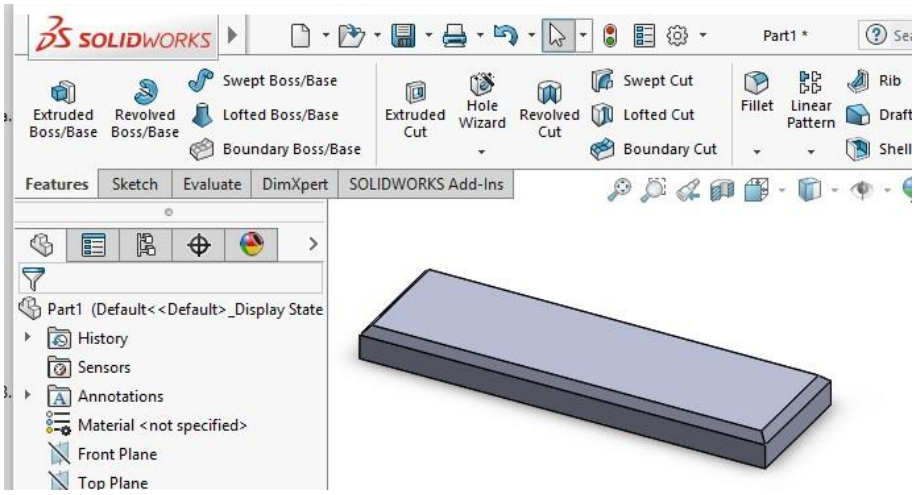
8	Click on the upper horizontal line. Move the cursor up and click at a random position to place the dimension.	
9	A menu will automatically appear in which you can set the accurate dimension. Change the dimension to 150 and click on OK (the green 'check' icon).	
10	Do the same with the vertical side of the rectangle. Change this dimension to 46. The sketch now looks like in the illustration on the right.	

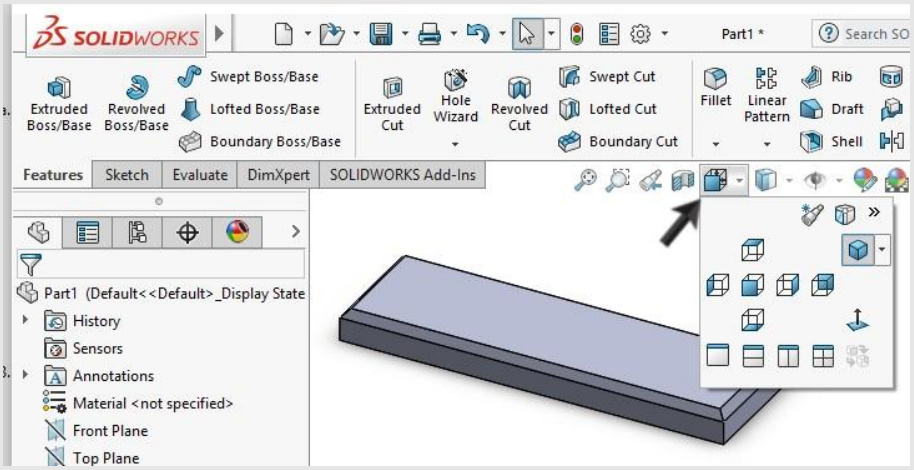
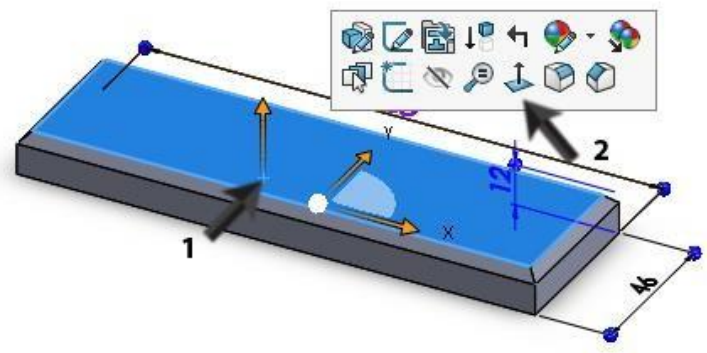
11	The sketch is ready and we will transform this into a rectangular piece of material. Click on 'Features' in the CommandManager and then on Extruded Boss/Base.	
12	Enter a height of 12 at the left side of the screen and click on OK.	
13	There, the first feature is finished already!	

14	Before we continue: check if no sketch or feature is still opened. Watch the right top corner of your screen. If you see one of the views on the right, then click on the red 'X' to close any opened commands.	
15	Next we will create the chamfer at the top plane. To do so, you do not have to make a sketch first. Click on the top plane of the block to select it.	 

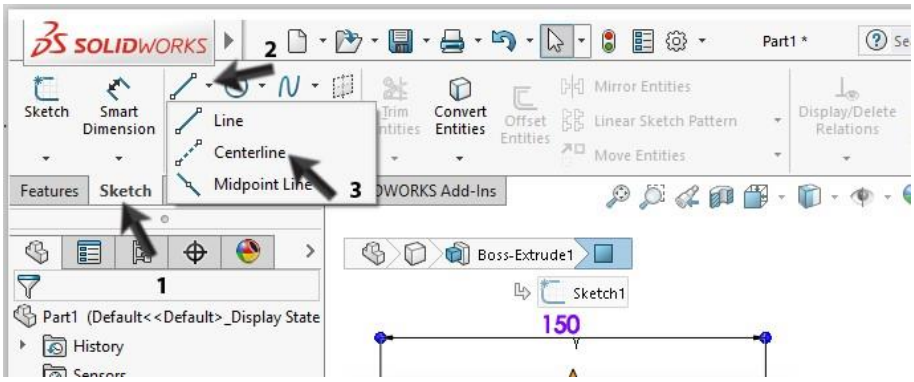
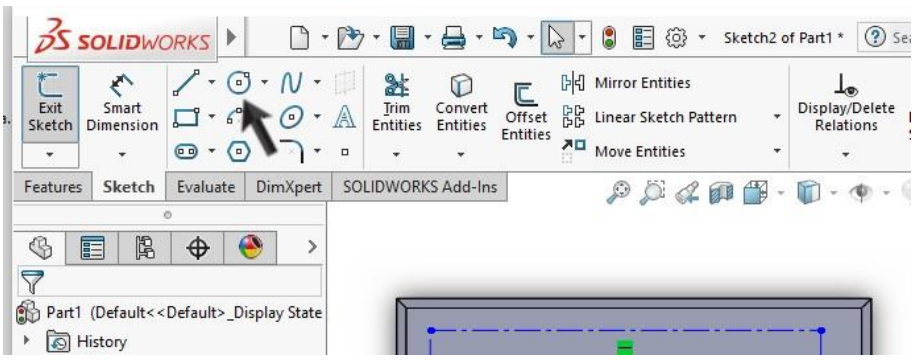
16	1. Click on the arrow directly below the Fillet button in the CommandManager to show the roll-down menu. 2. Click on Chamfer.	

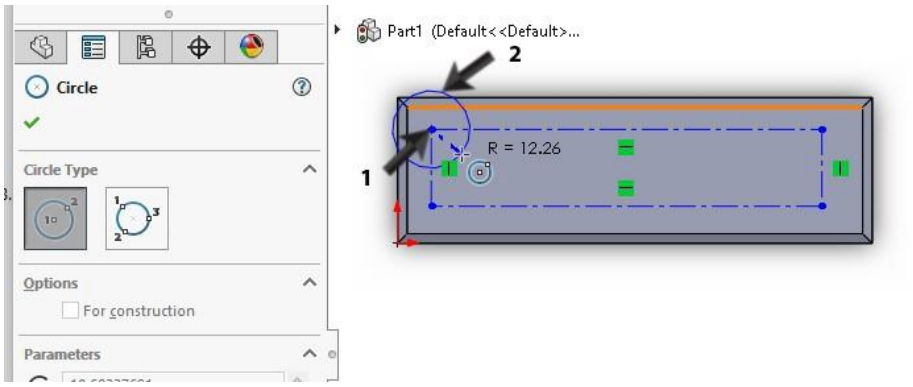
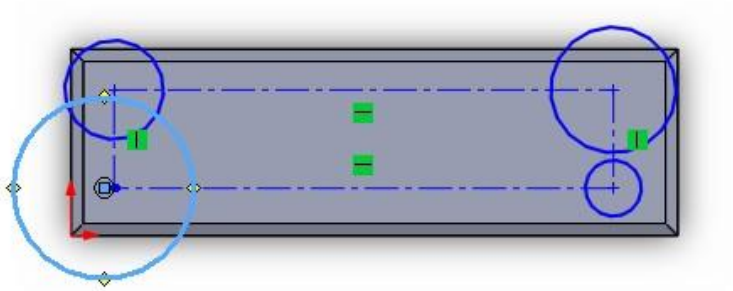
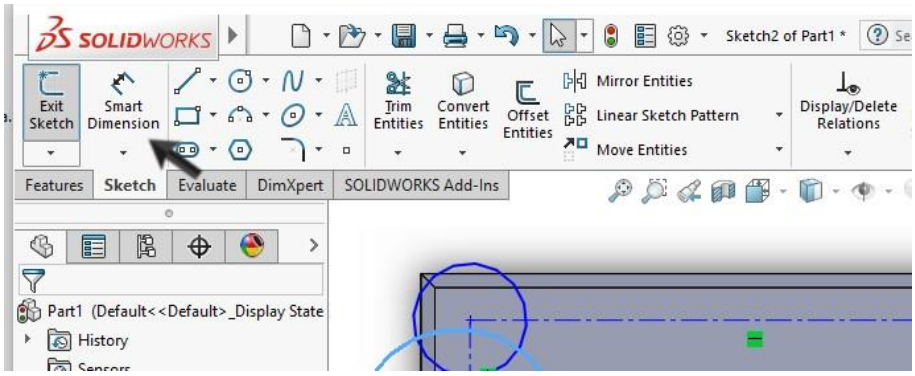
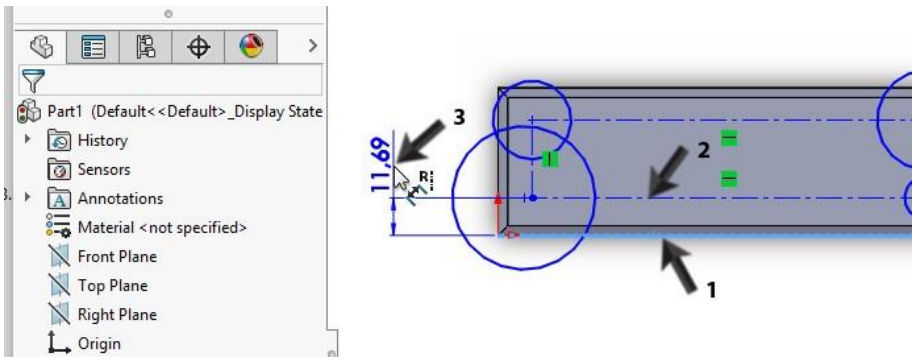
17	Now you must check and set a number of items. 1. Be sure the options 'Full preview' is selected. This will give you a good idea of what the chamfer will look like. 2. Make sure that in the field Chamfer Type, the option 'AngleDistance' has been selected. 3. If everything went correct, only one 'Face' (plane) is selected in the blue field. (read the Tip below) 4. Set a chamfer distance of 3mm and 45 deg. 5. If everything is set, click on OK.	
	Tip!	In SOLIDWORKS you will often see a blue selection field, like in step # 17. In this field you will see the elements of a part on which a command will be executed. You can **remove** elements by selecting them and using the <Delete>button. You can **add** elements by selecting them in the part. In case you have more than one selection field, there will always be only one active field (blue). To activate another one, click inside of the desired field.

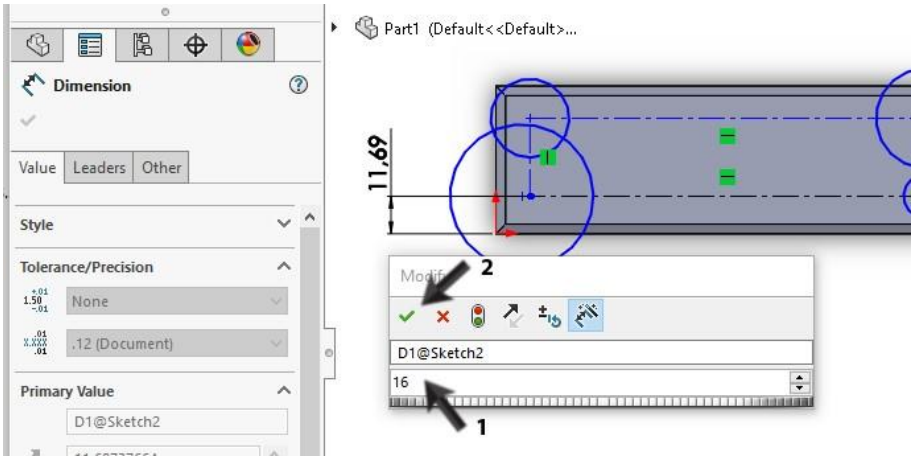
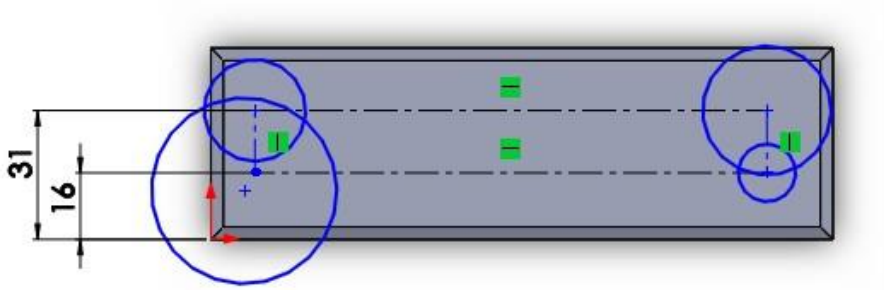
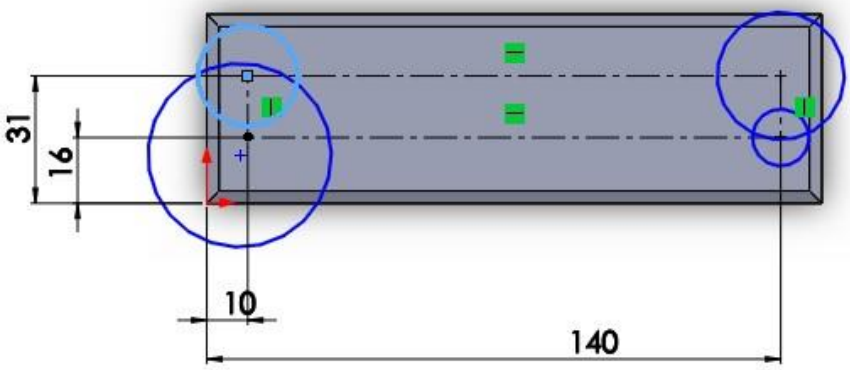
18	The chamfer is done now.	

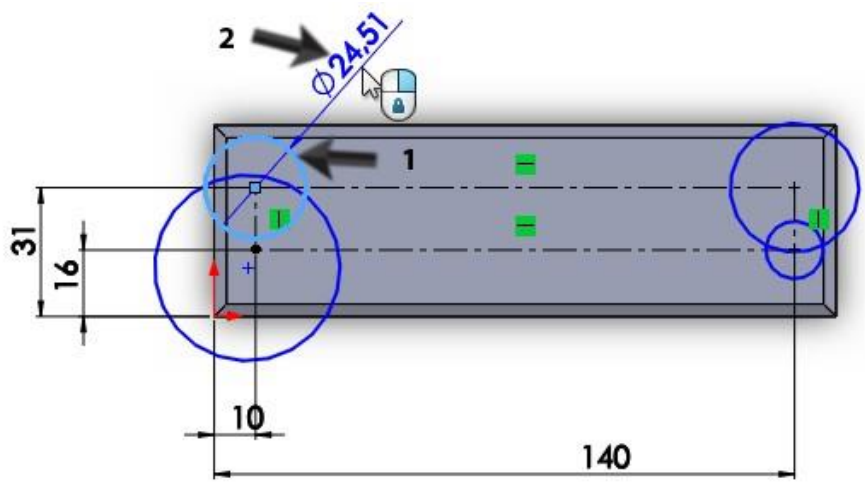
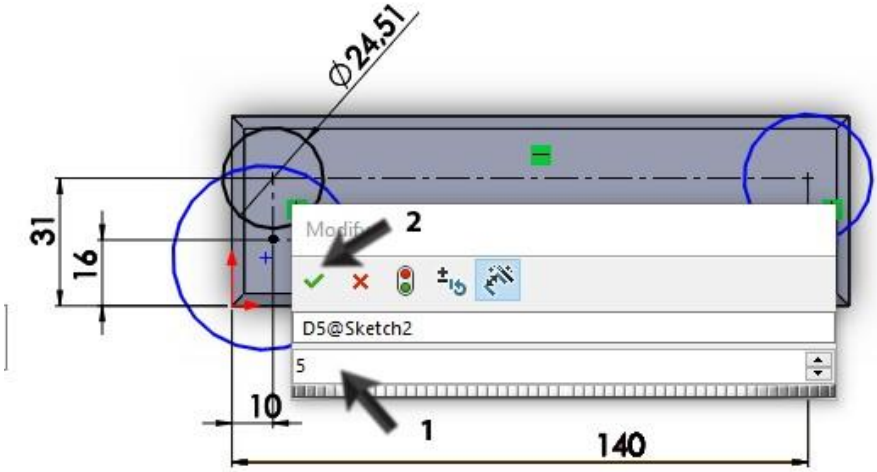
	Tip!	Remember that you can zoom in- and out at all times, or you can rotate the model to get just the right view: • Zooming in- and out is done by **turning** the scroll-wheel of the mouse. • Rotating is done by **pressing** the scroll-wheel of the mouse and moving the mouse. You can also use the button View Orientation to put your model in the right position directly. 
19	We are now going to create the holes. Select the top plane of the block by clicking on it. In the popup menu, click Normal To. The model will rotate the selected plane towards you. This makes it easier to create a sketch. The option Normal To can also be found under View	

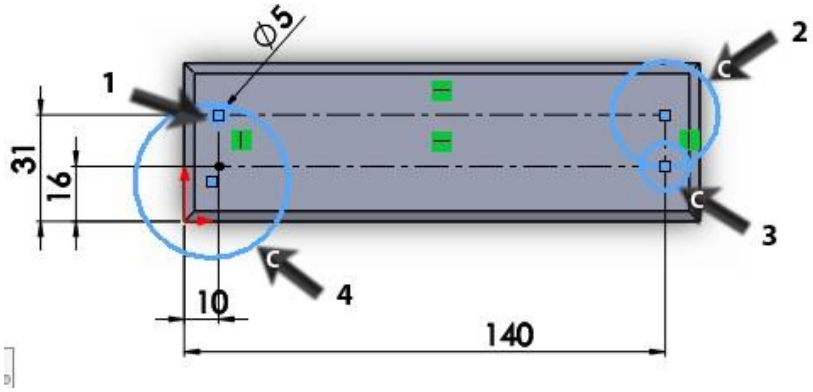
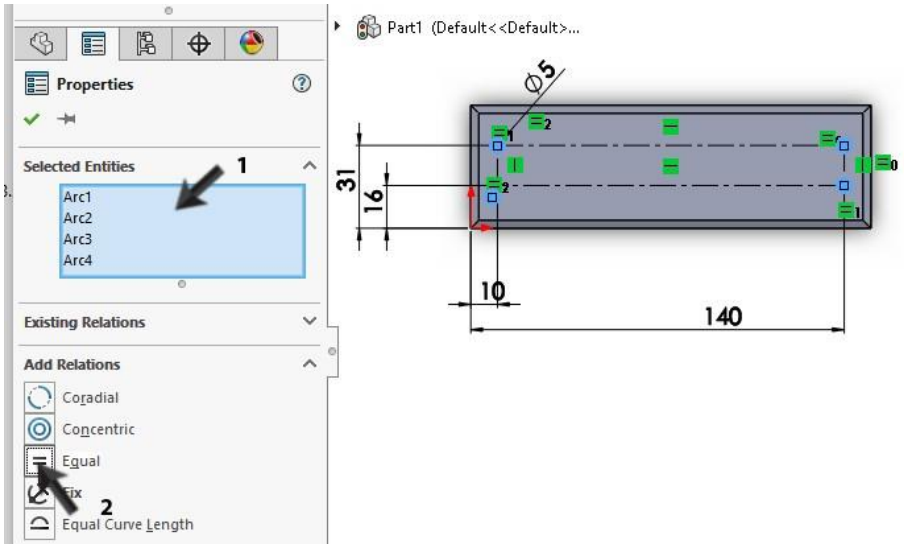
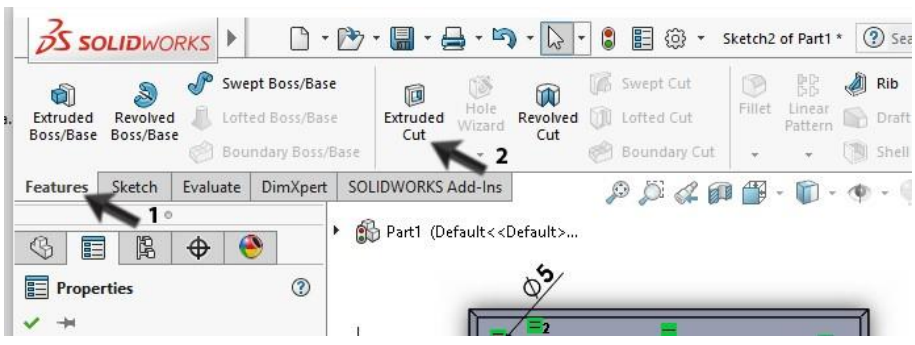
	Orientation	

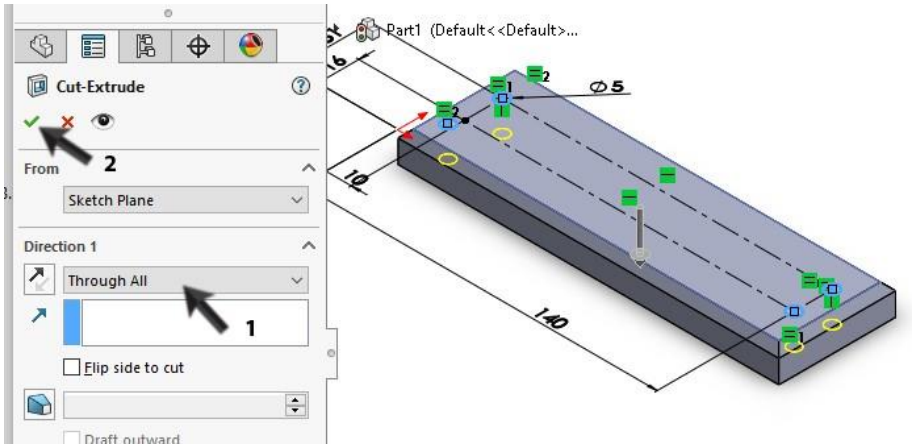
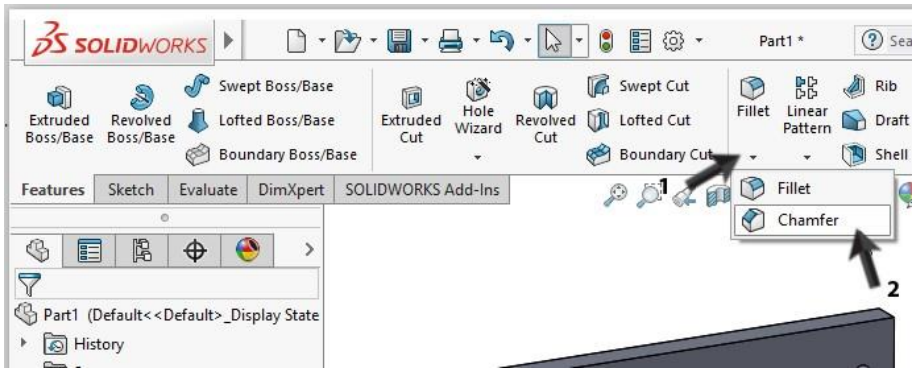
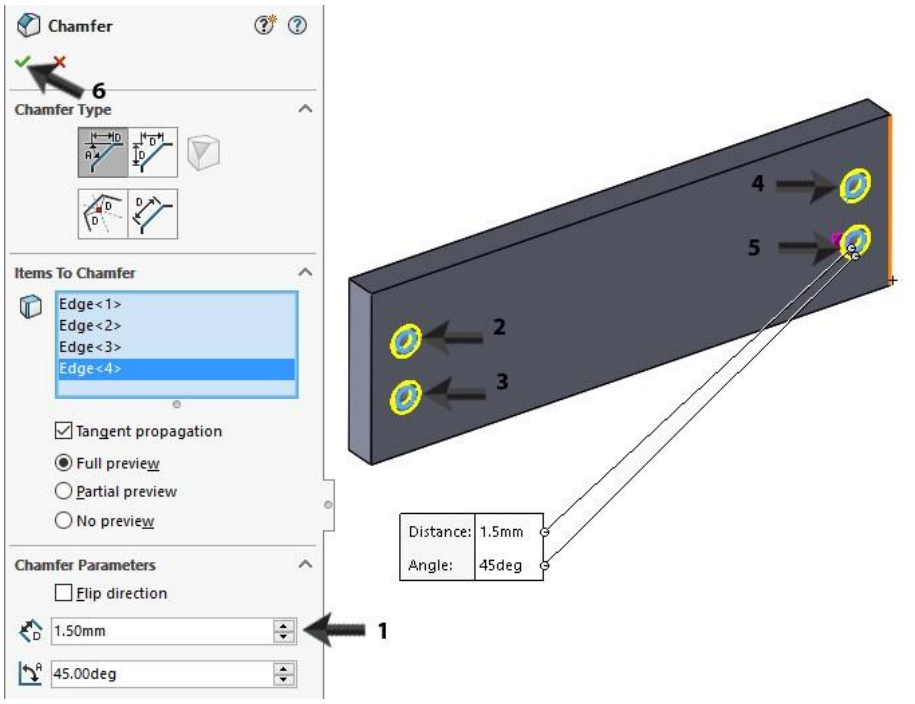
20	1 Click on Sketch in the CommandManager. 2 Click on the arrow next to Line 3 Click on Centerline Centerlines are construction lines which can help you with the design of a part.	
21	Next draw a rectangle by using four lines. • Mind the construction lines which will appear all the time. These will help you to draw horizontal and vertical lines and make sure that the fourth corner will exactly fit underneath the first one. (look at the drawing on the right) In this way you will get a closed rectangle. • Make sure that the corners of the rectangle are not set directly above or on top of another element (e.g. the edge of a plane). • After you have drawn the last line you must press the <Esc(ape)>button on your keyboard to end the command.	
22	Next draw the holes. Click on Circle in the CommandManager.	

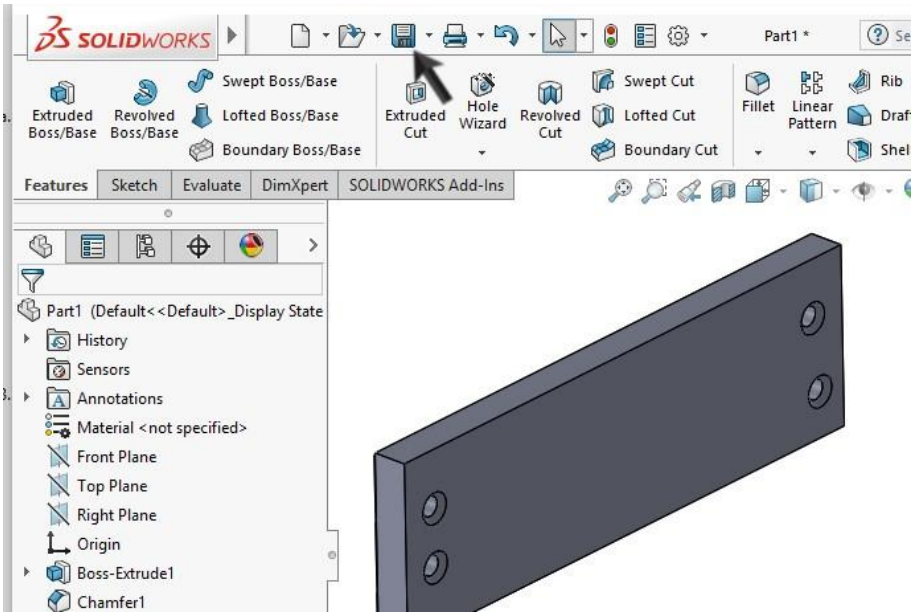
23	Click on one of the corners of the rectangle. Move the cursor up and click it again (**do not** click right on another element) to draw the circle. The exact dimension of the circle is to be set later.	
24	Use this method to draw a circle on every corner of the rectangle. After drawing all four circles, press the <Esc>button on your keyboard to end the command.	
25	Next we want to add the dimensions to the sketch. Click on Smart Dimensions.	
26	Add the first dimension: 1 Click on the lower horizontal line of the model 2 Next click on the bottom construction line of the rectangle you have just drawn. 3 Next click beside the model to position the dimension.	

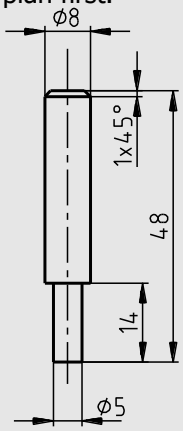
27	In the menu that appears you can enter a value of 16 and then click OK.	
28	Use this method to add a dimension between the bottom line of the model and the top construction line of the rectangle. This dimension is set to 31.	
29	Next, in exactly the same way you will add two horizontal dimensions to determine the distance between the left side of the model and the left and right construction line of the rectangle. Set these dimensions to 10 and 140.	

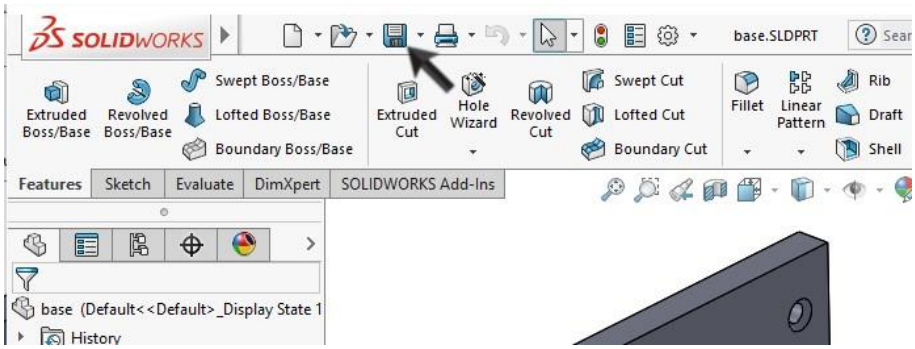
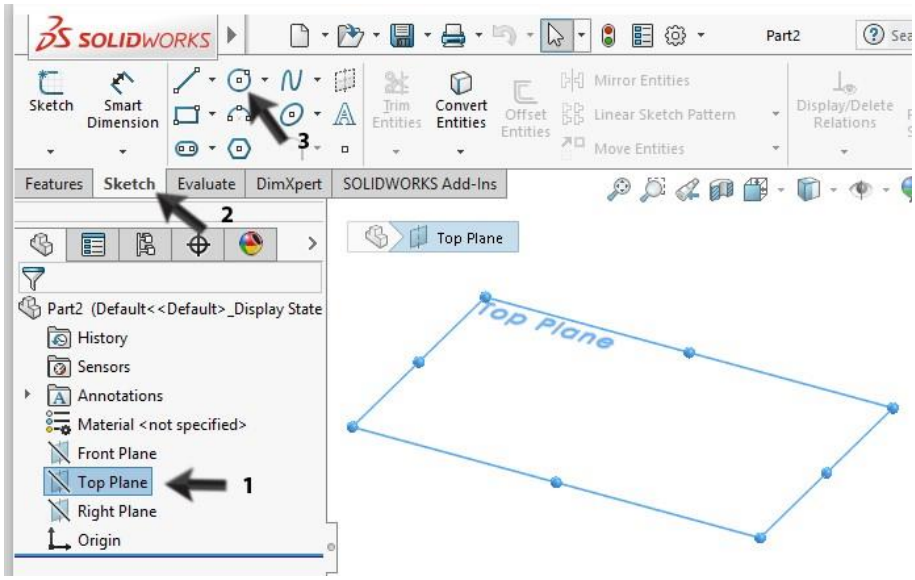
30	The diameter of the holes must be added now. Stay in the Smart Dimension command. Click on a circle and click beside the model to position the dimension.	
31	Enter a dimension of 5 for the circle and click on the OK-icon. Press the <Escape>-button on the keyboard to close the Smart Dimensions command.	

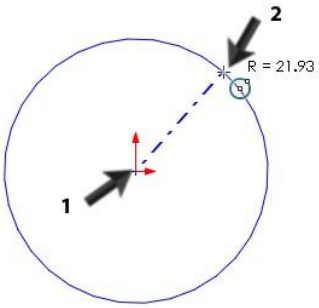
32	To set the same dimension for all circles you do the following: 1 Click on one of the circles. 2-4 Press and hold the <Ctrl>-button on your keyboard. Next click on the other circles one by one. 5 Release the <Ctrl>button. If you did this properly, all four circles are now selected (and turned blue). If not, click beside the model to un-select everything and try again.	
33	1 Check the blue field in the PropertyManager if you have selected the four circles and nothing else. In the field four times 'Arc' will be visible. 2 If so, click on Equal. You have now added a relation. This relation makes sure that the four holes will always be the same size, even if the size of the first circle is changed.	
34	The sketch is finished and we can continue by making the holes. Click on Features in the CommandManager and then on Extruded Cut.	

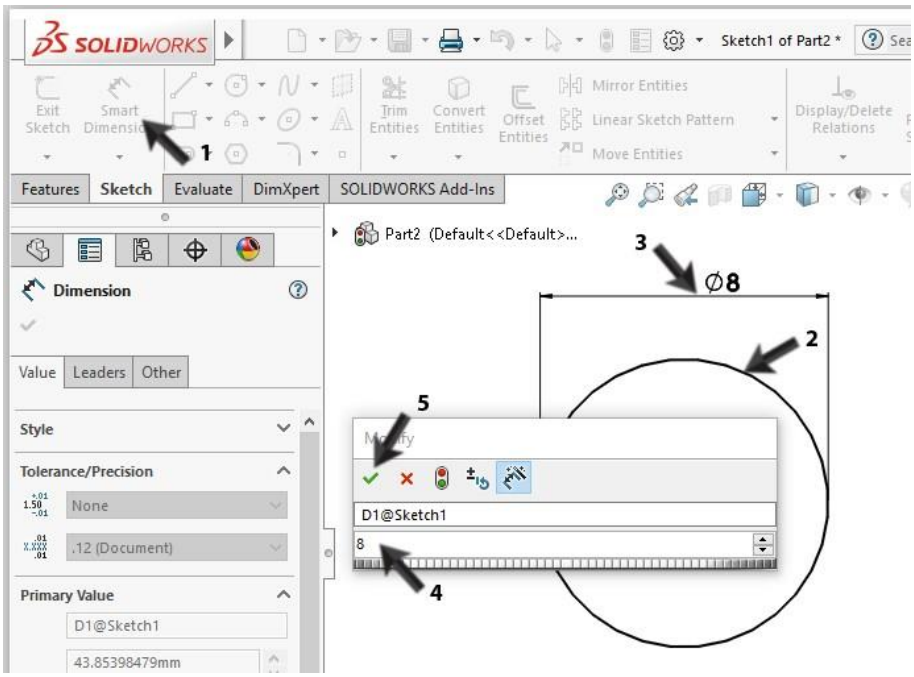
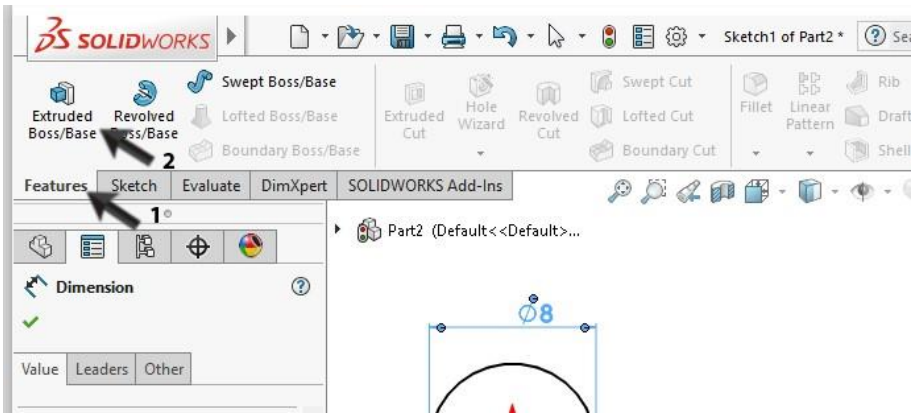
35	Rotate the model (press the scroll-wheel and move your mouse) so you can get a better view. 1. Set the depth of the holes to 'Through All': the holes will go through the complete depth of the material. 2. Click on OK.	
36	Finally we have to countersink the holes. Rotate the model so you have a good look at the bottom plane. 1. Click on the arrow underneath the Fillet button in the CommandManager. 2. Click on Chamfer.	
37	To set the slope you do the following: 1. Set the characteristics of the slopes to 1.5mm and 45deg. 2-5 Select the edges of the four holes. ONLY select the edges and not the faces. In the blue field you will read Edge<…> four times. If you have selected a wrong element, click on it in the blue field and press the <Delete>-button on your keyboard. Try to select the right element again. 6. When you have selected the right elements, click on OK.	

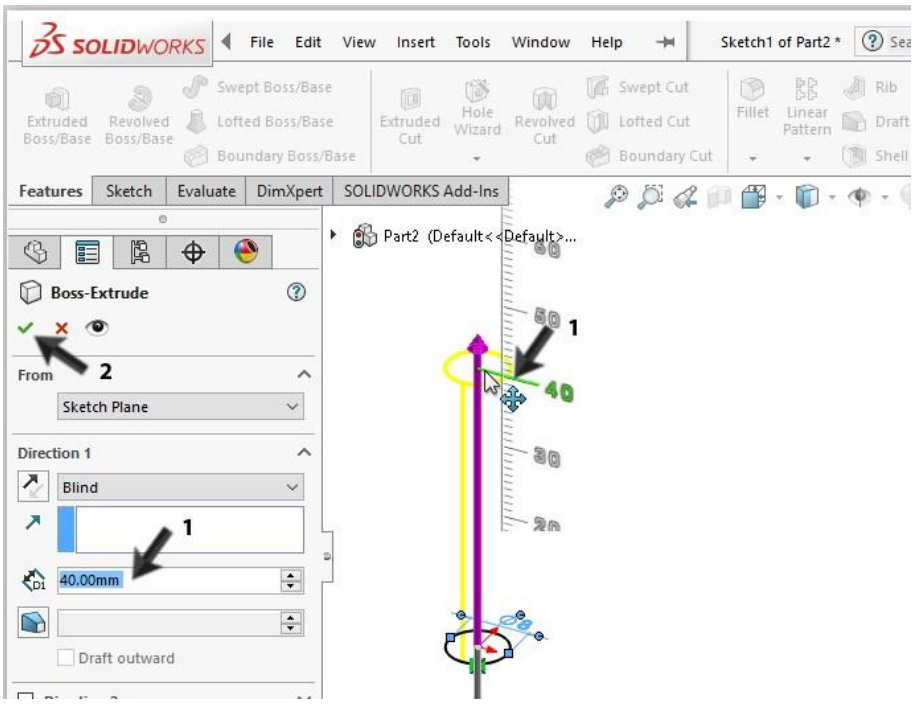
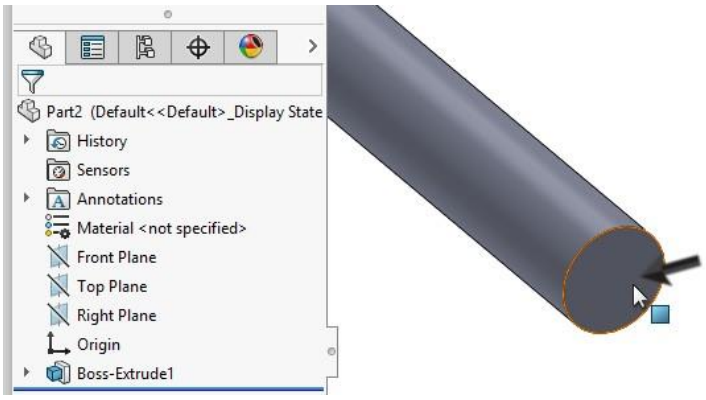
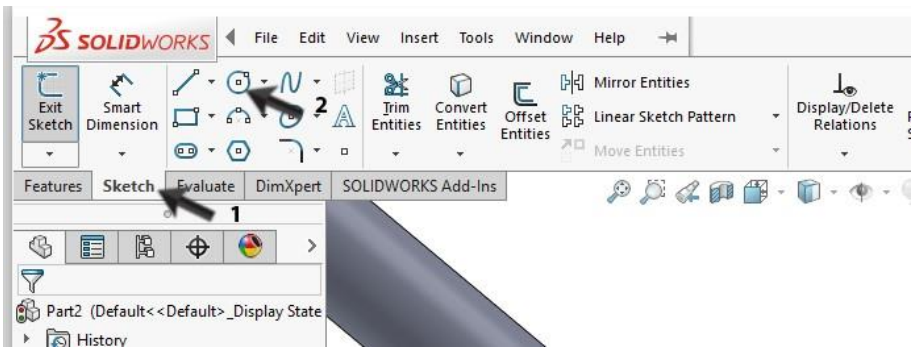
38	The holes now are countersunk and the first part of this model is ready. Click on 'Save" in the upper menu and save your model as: **base.sldprt**	

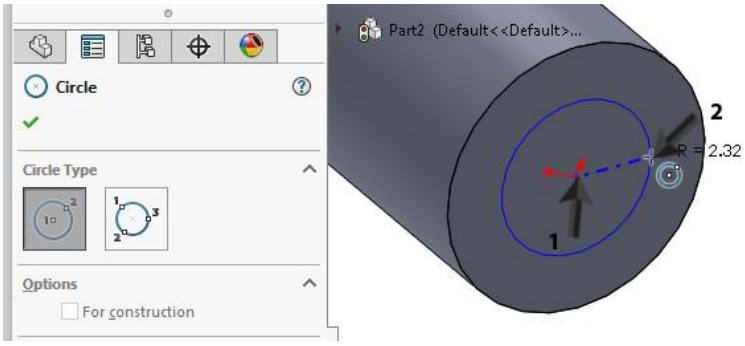
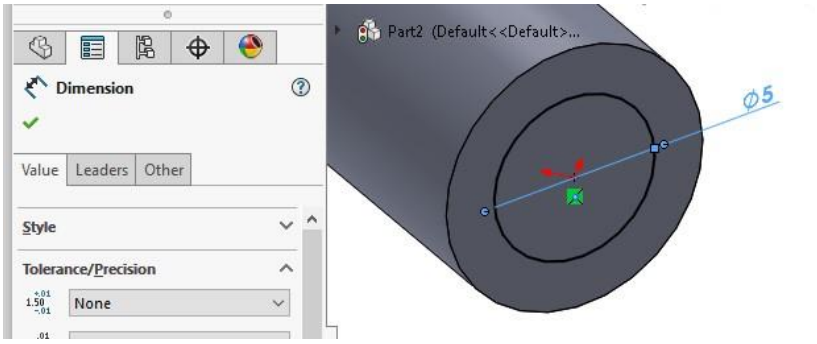
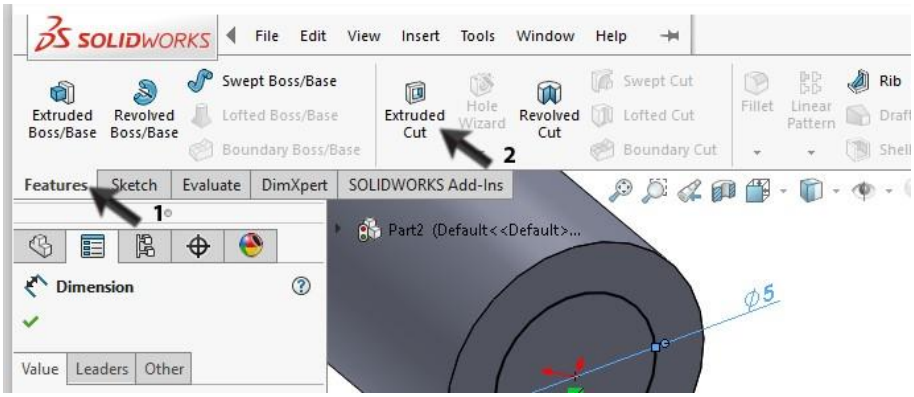
	Work plan	Next we will be making the second part, the axis. Again we will make a work plan first.  We will create this model in three steps: 1. We take the basic material of Ø8 x 48 2. At the bottom we will make the axis thinner: Ø5 over 14mm 3. At the top we will make a chamfer. We have seen all these steps before. Therefore, try to make the axis without using the description which follows!

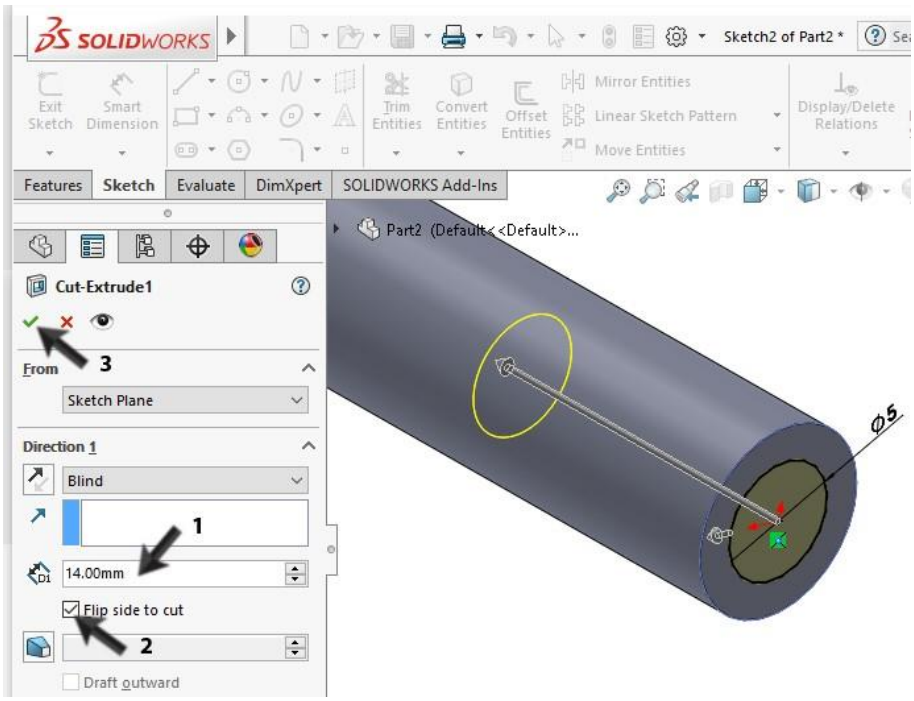
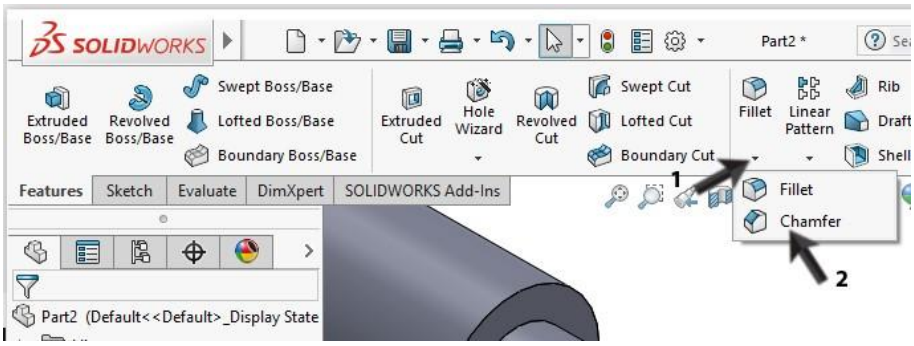
39	Start a new part. Click on 'New' in the upper menu and chose 'Part'.	
40	We will use the Top-plane to make the first sketch: 1. Select the Top-plane in the Feature Manager. 2. Click on Sketch in the CommandManager to reveal the right buttons. 3. Click on Circle.	

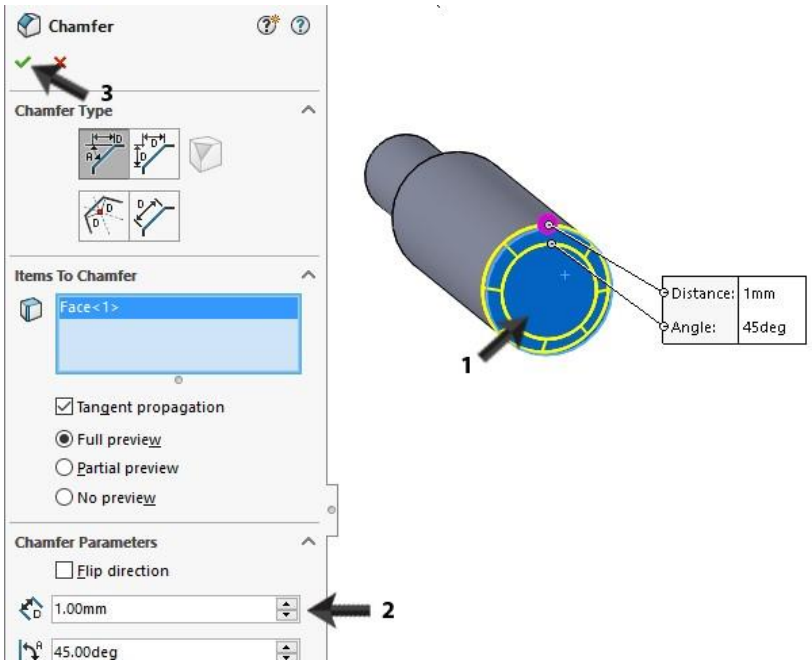
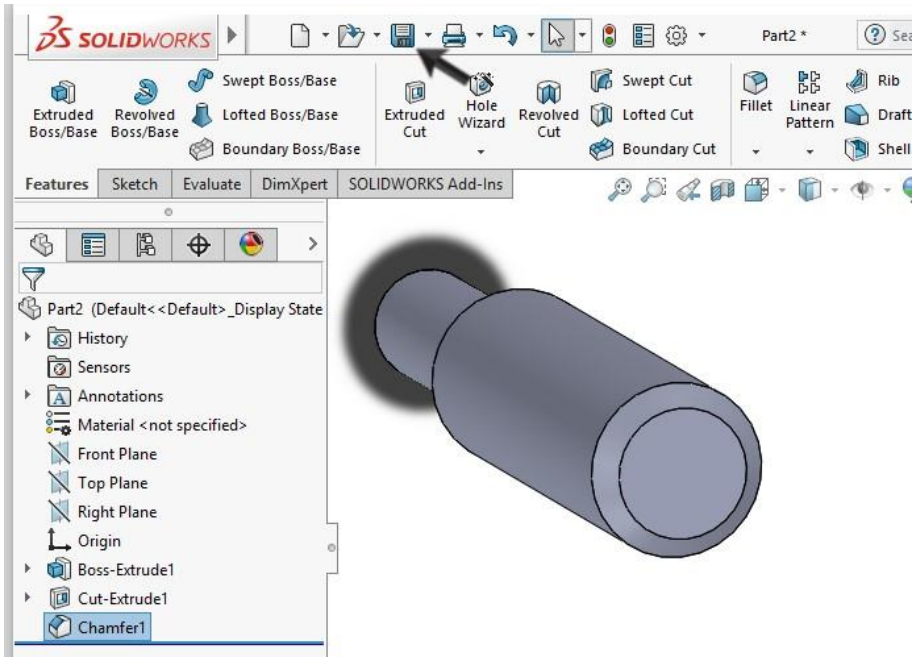
41	Draw a circle. Click on the origin and next move the mouse away from the origin and click again to draw a random circle.	

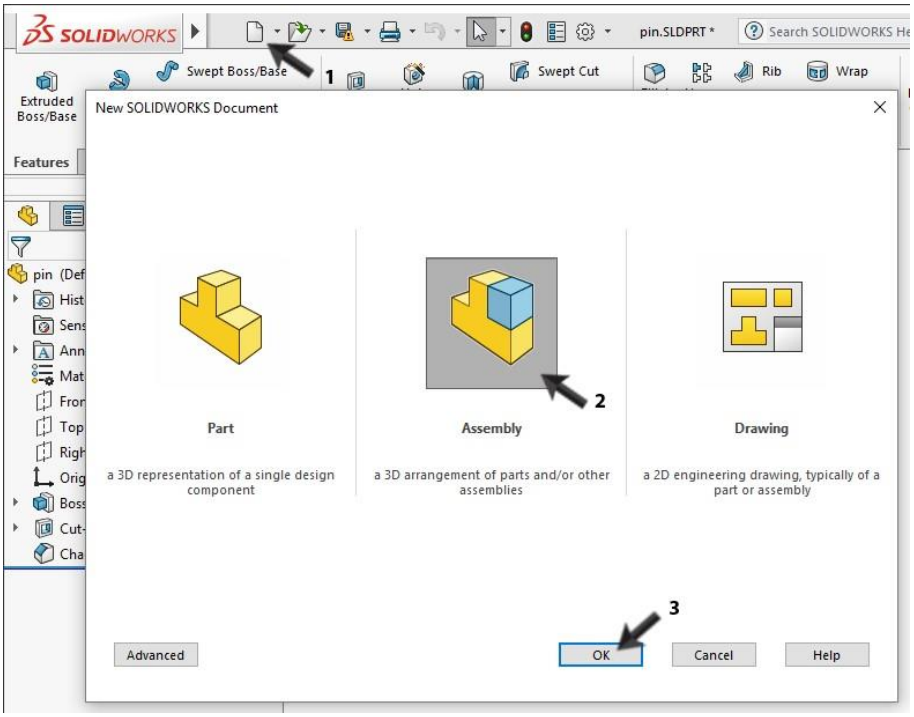
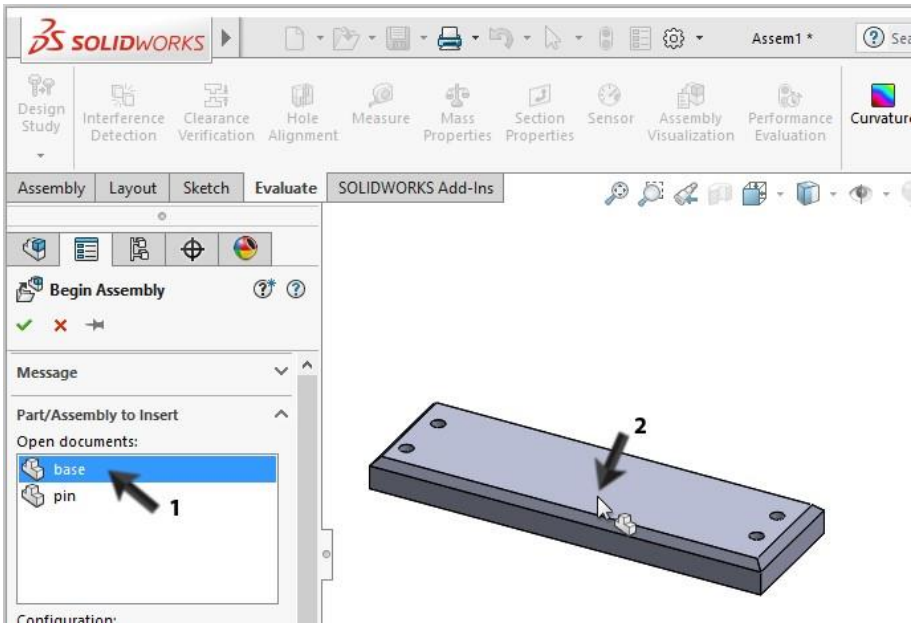
42	Add the dimension with Smart Dimensions: 1. Click on Smart Dimensions in the CommandManager. 2. Click on the circle. 3. Place the dimension by clicking beside the circle. 4. Change the size to 8mm in the menu. 5. Click on OK.	
43	Click on Features in the CommandManager and then on Extruded Boss/Base	

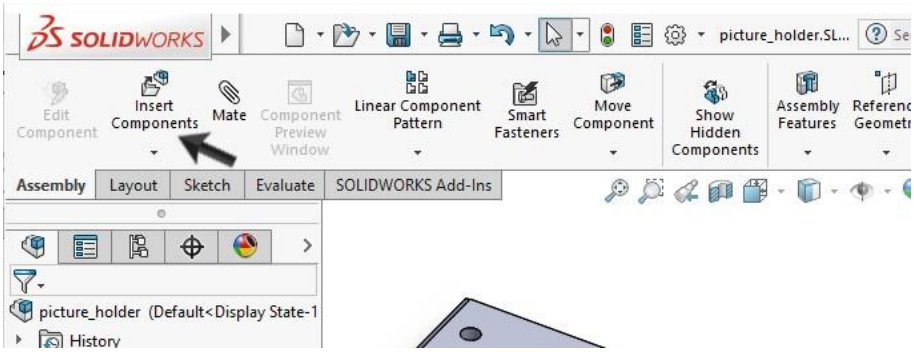
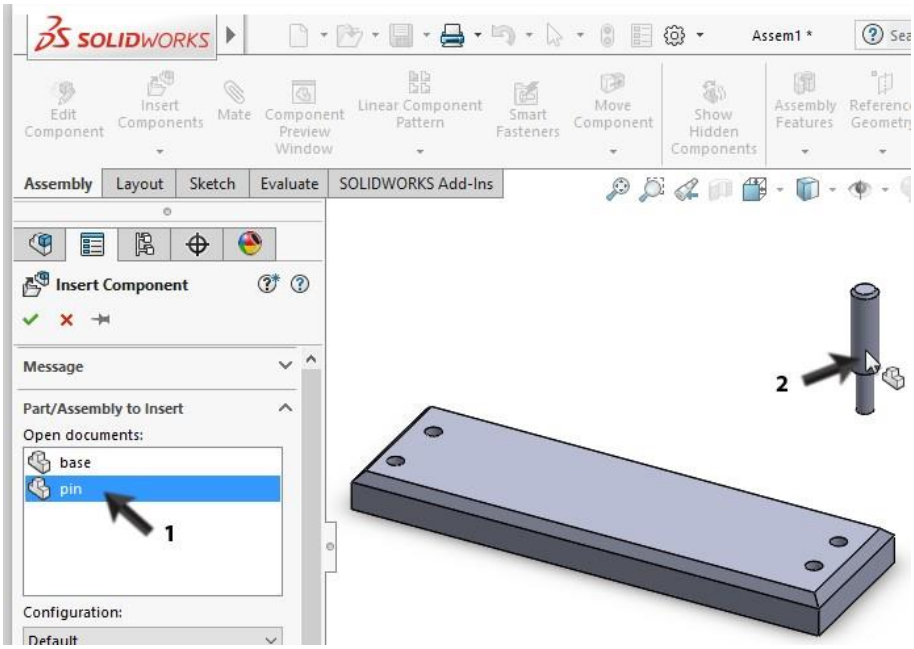
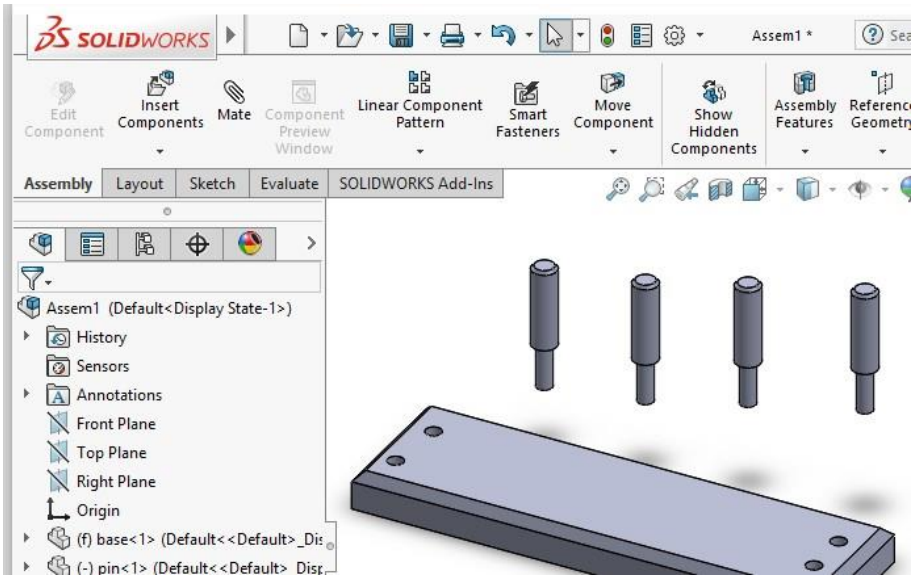
44	1 Drag the arrows in the model to a length of 48mm. Of course you can also do this by entering the value of 48 in the PropertyManager. 2 Click on OK.	
45	Rotate the model to get a good view at the bottom of the part. (Use the scrollwheel of the mouse). Click on this plane to select it.	
46	Click on Sketch in the CommandManager and then on Circle.	

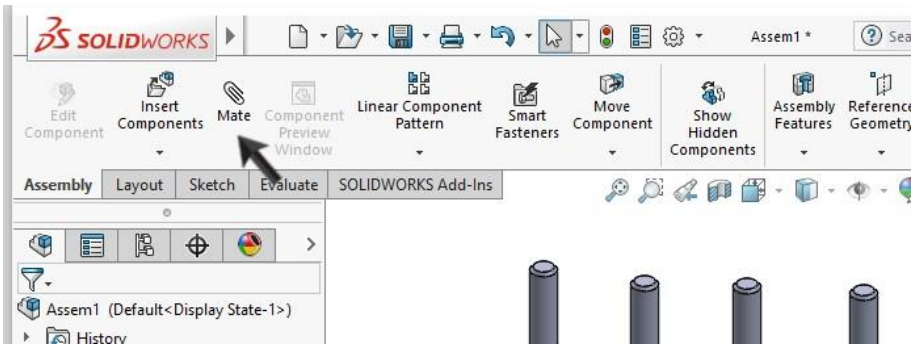
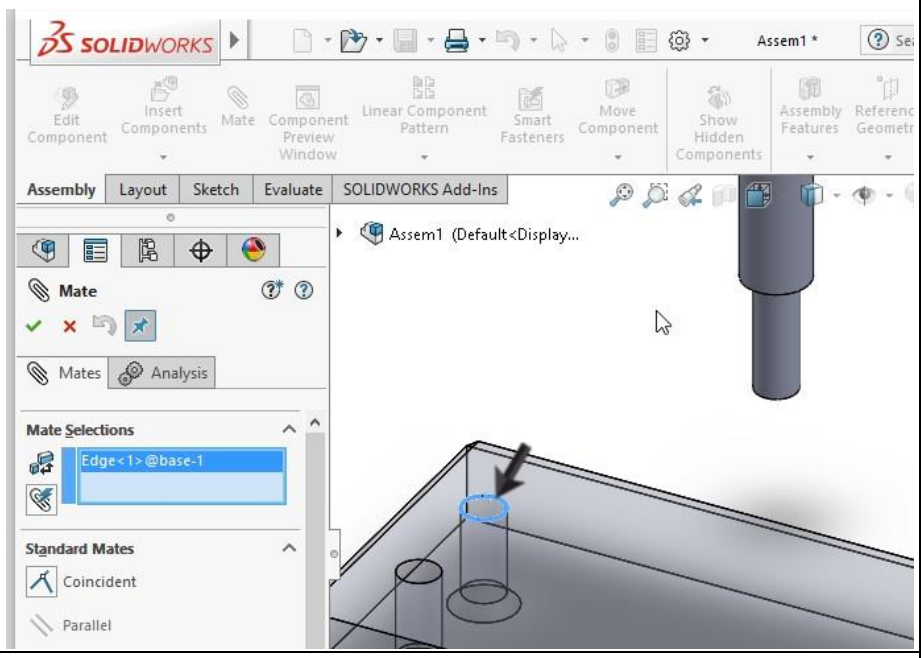
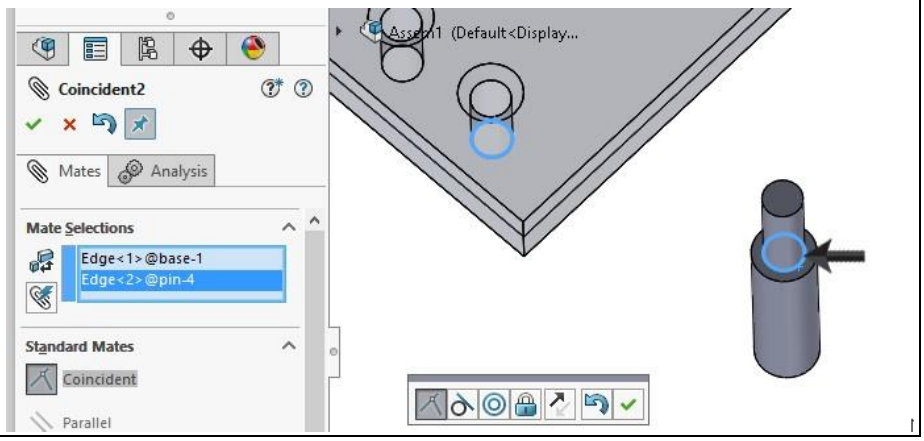
47	Draw a circle in the selected plane. Click on the origin for the centre of the circle. Next move the mouse to draw a circle with a random dimension and click again.	
48	Add a dimension for the circle and change it to 5 mm.	
49	Click on Features in the CommandManager and then on Extruded Cut.	

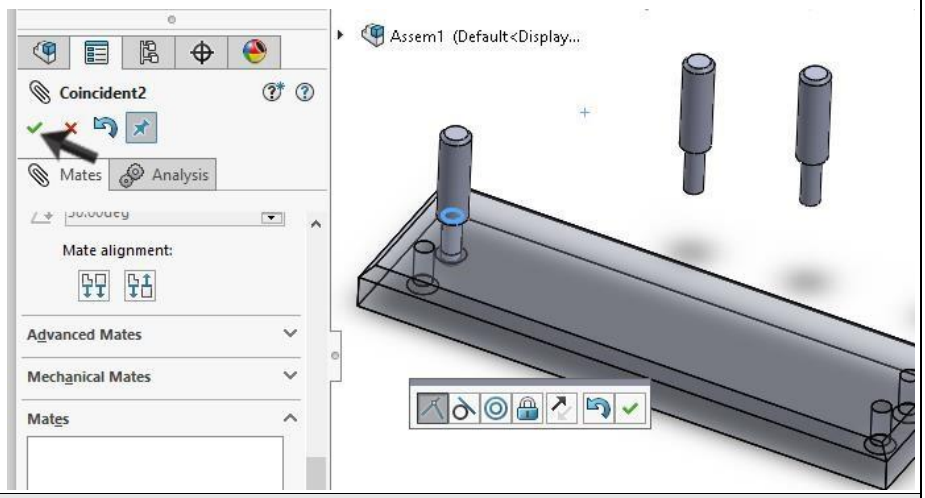
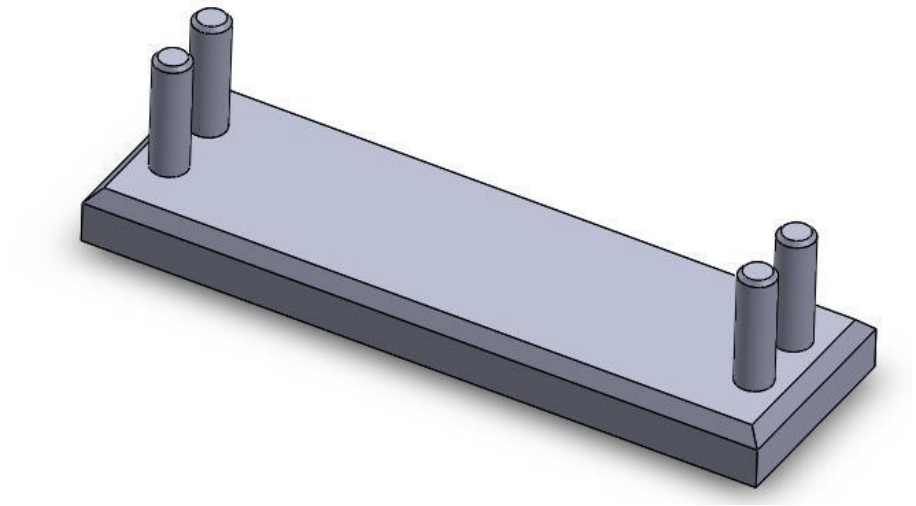
50	1 Set the depth to 14mm. 2 Check 'Flip Side to Cut' to cut away the outer material. 3 Click OK.	
51	The last feature that we have to make is the chamfer at the top of the axis. Rotate the model so you can get a good view on the top plane. Click on Chamfer in the CommandManager.	

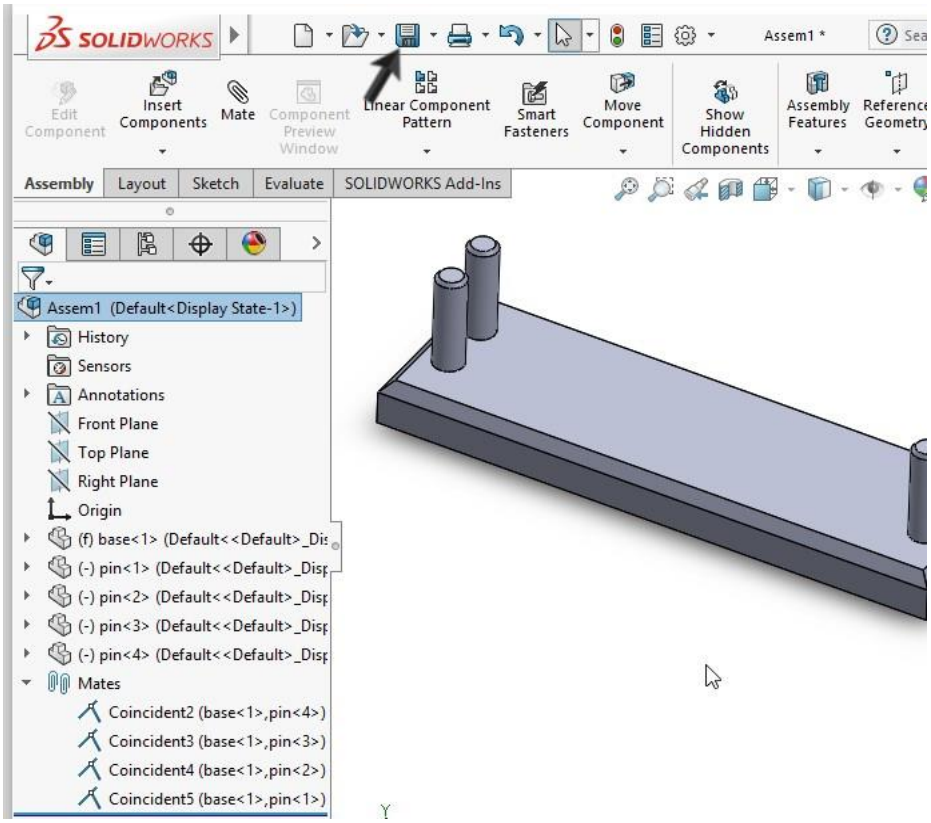
52	Check and set the following features: 1. Select the top plane of the axis. 2. Set the distance of the chamfer to 1mm 3. Click OK. Be sure the option Full Preview is active or else you do not have a clear view on what is happening.	
53	Save the file as **pin.sldprt**	

54	The two parts for the picture holder are ready. We are going to assemble them in an 'Assembly', to get a complete product. 1 Click on New in the menu. 2 Select 'Assembly' 3 Click on OK.	
55	1 Click on 'base' in the PropertyManager. This is the first part we created. 2 Click at a random point in the drawing field. The part is placed in the assembly. **Pay attention**: does this step not work properly? Read the tip which follows next.	
	Tip	In the last step some command may not work as described. • When the left column looks different from the example as shown in step 55, the 'Insert Components' command has not started automatically. When so, click on 'Insert Components' in the CommandManager. • When the parts 'Base' and 'Pin' are not in the list, you apparently closed these parts. When so, click on 'Browse' and find the right files. After doing so, you can put them in the assembly as described.

56	Click on Insert Component in the CommandManager to add the first pin.	
57	Select 'pin' in the menu on the left of the screen and click at a random point in the drawing field to place the part. When you have closed the file pin.sldprt before, it will not be in the list. (read the last tip again) If so, click on Browse and find the file.	
58	Repeat the last step three times in order to place four pins in the drawing. All pins are at a random position.	

59	Next we will place the pins at their **accurate** position. Click on Mate in the CommandManager.	
60	At this point, you will have to select two elements to 'Mate'. You must do this with the greatest accuracy! Zoom in on one of the holes in the base part. Select the **edge** of the hole (Pay attention: it must be an 'edge' and not a 'face' (=plane)). In the blue field in the PropertyManager (at the left of your screen) the description: Edge<1>@base1 will appear.	
61	Rotate the model (press the Scroll-wheel, remember?) so you can get a good view of the bottom of the pins. Zoom in if necessary. Select the edge from the pin like illustrated in the right view. Make sure you do not select a plane.	

62	When the two edges have been selected, the pin will move into the hole. When this is done and the result looks good, click on OK.	
	Tip!	It is very important to select the right elements when making a mate. If you select something else than described in the last steps, something completely different will happen or maybe nothing will happen. When, by accident, a wrong element is selected, think about what has been said about the blue fields. You can delete a wrong element by clicking on it and press the <Delete> button on the keyboard. After that, you can add another element.
63	Repeat the last three steps for every pin, so eventually each pin is placed in one of the holes.	
	Tip!	Every mate you create will be visible in the FeatureManager as you can see in the example below. Do you want to remove a mate? Select it and press the <Delete> key on the keyboard. You can change a mate by clicking on it with the **right mouse** button and chose Edit Feature.

64	You have just created your first Assembly in SOLIDWORKS! Congratulations. Save the file as: **picture_holder.sldasm**	

	What are the most important things you have learned in this tutorial?	In the part section you used some new commands: • You drilled holes. • You have copied the dimension of one hole to other holes using the Equal-relation. • You have made sloped edges with the Chamfer-feature Next to that you have made an assembly: • You have assembled several parts to a complete product. • You have placed the components in their right positions using the mate command. You have reached a next level in SOLIDWORKS. In the next tutorials we will use what you now already.

Please follow for more tutorials