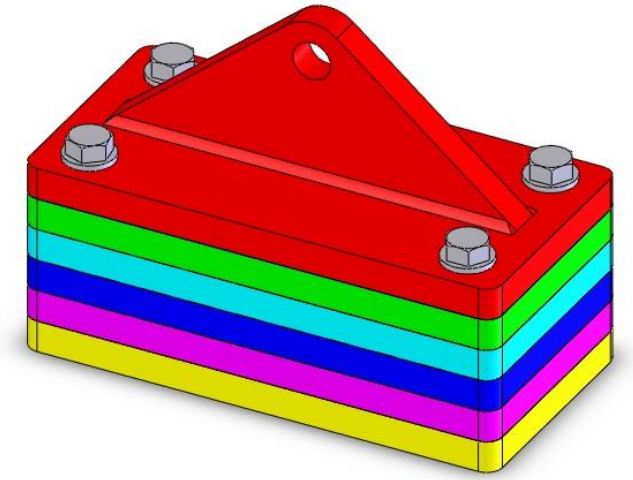
SOLIDWORKS® tutorial 3

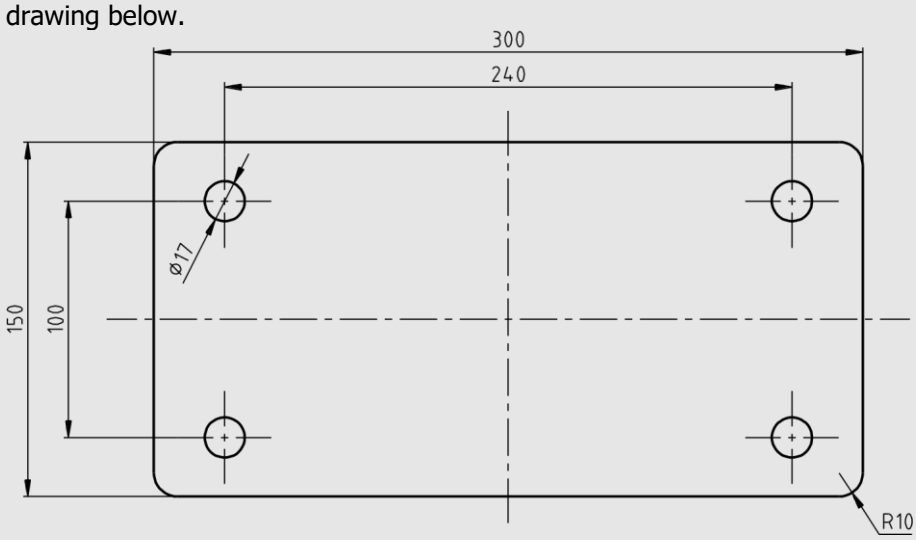
MAGNETIC BLOCK

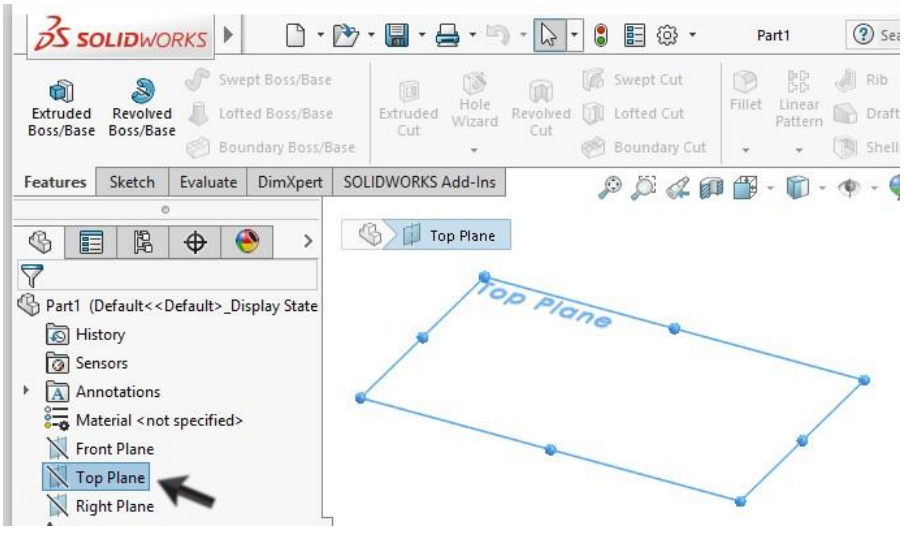
Magnetic block

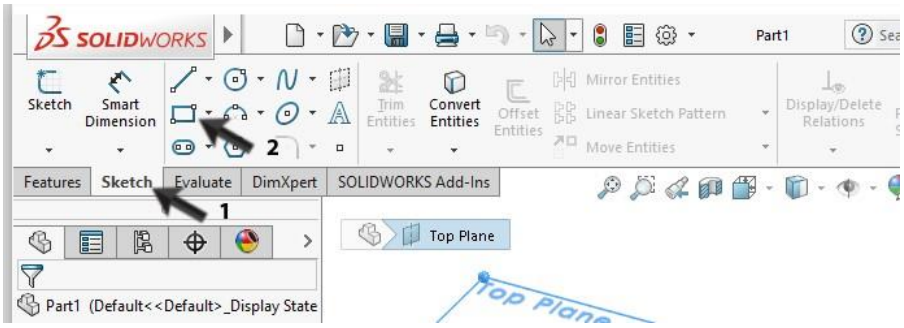
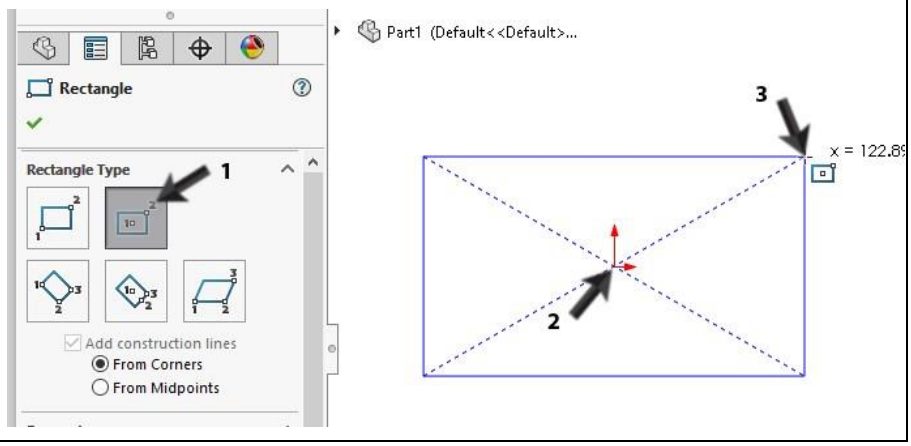
In this exercise you are making a magnetic block. To do so, you will be creating a few parts, which you will assemble. In this tutorial you will learn following new commands:

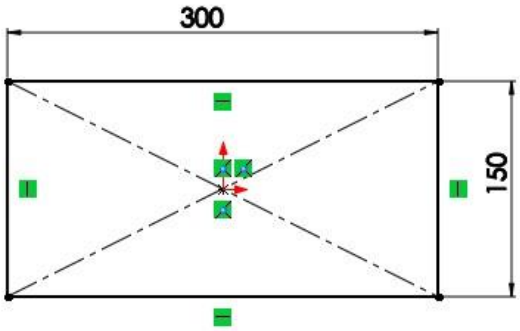
- You will make two configurations of a part You will weld the parts together.
- You will be making holes with the aid of the Hole Wizard You will be using standardized parts from the parts-library.
- You will give different components different colors.

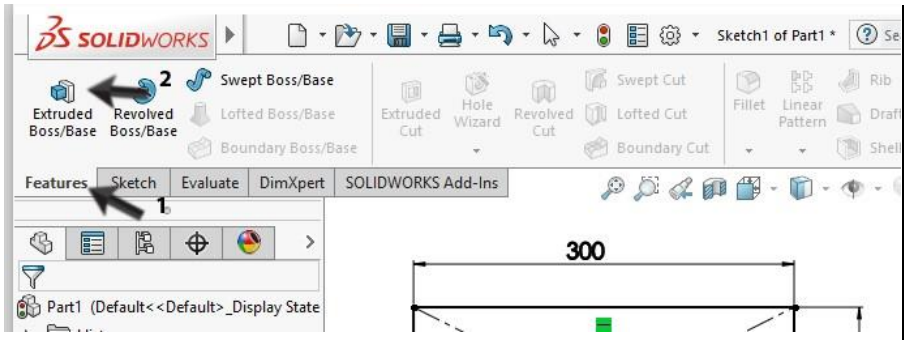
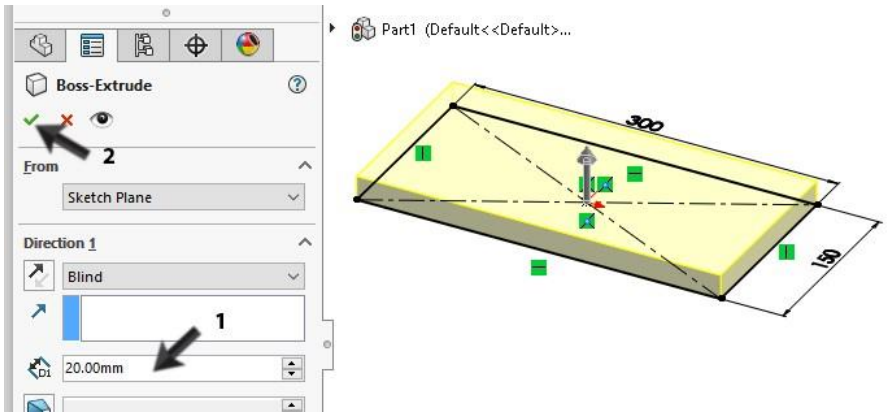
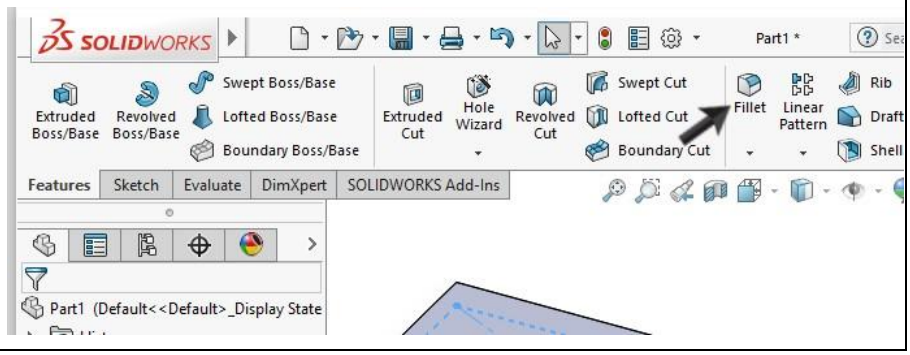


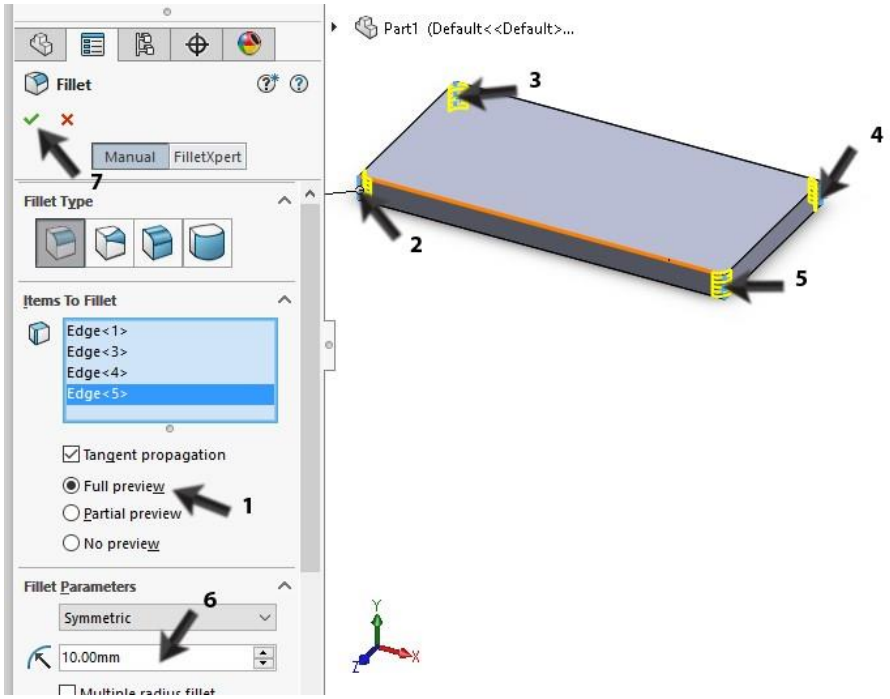
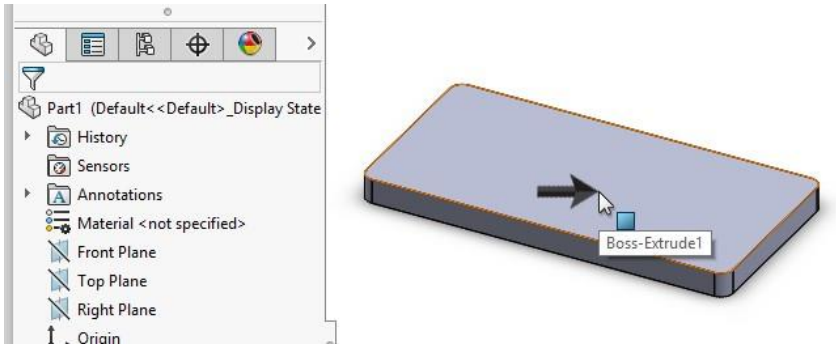
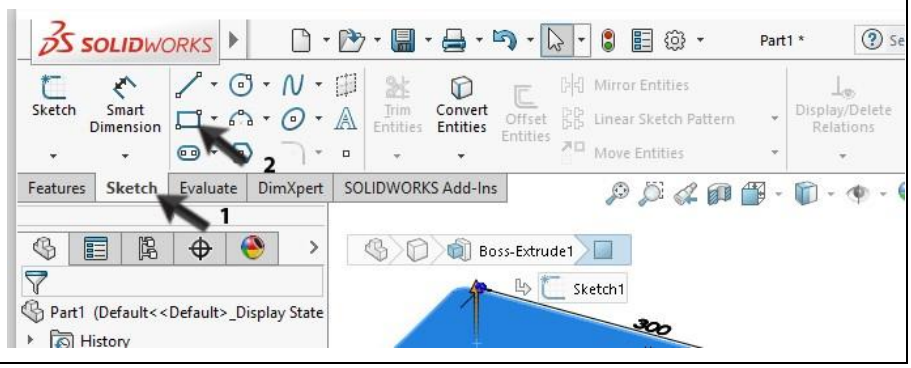
	Work plan	To make this product, we will have to make some parts first. We will start with a simple rectangular base with a thickness of 20mm according to the drawing below.  We will perform following steps: 1. take a piece of material of 150x300x20 2. round of the four corners with a radius of 10 mm 3. drill four holes of Ø17

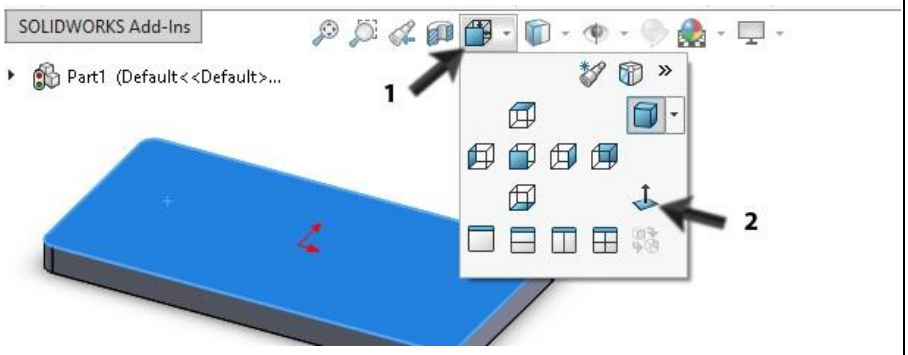
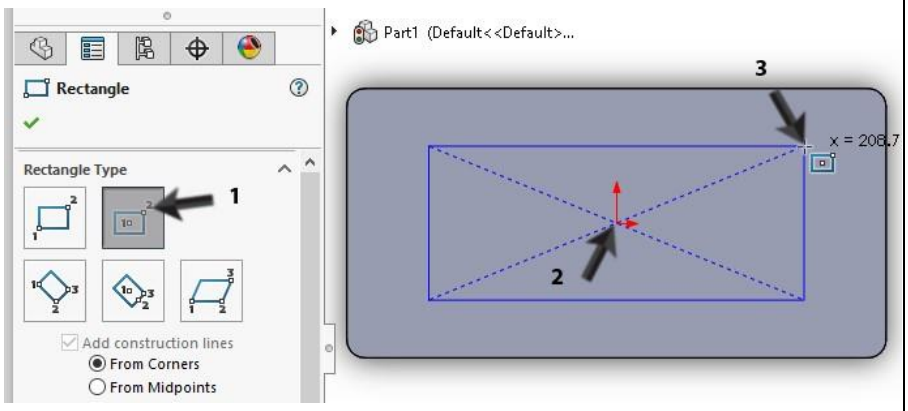
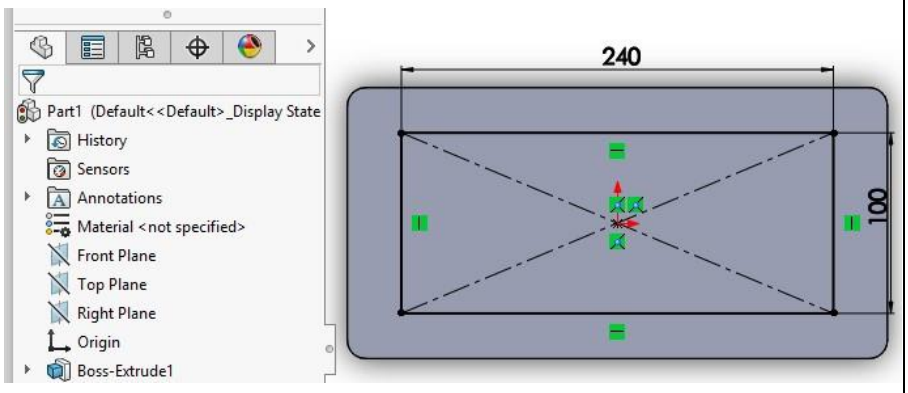
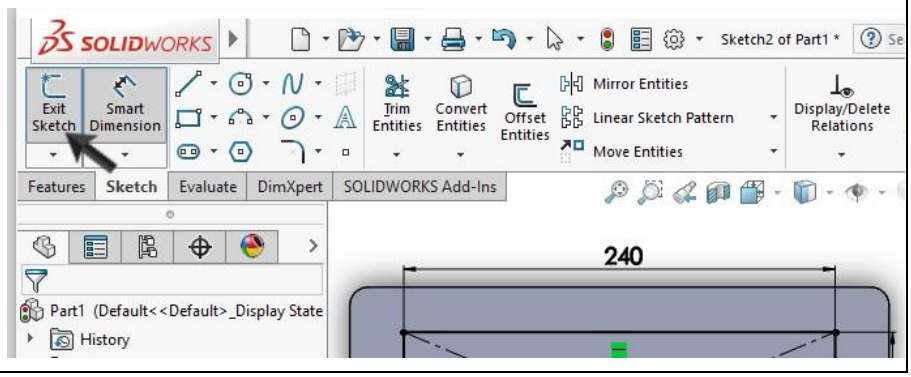
1	Start SOLIDWORKS and open a new part.	
2	Click on Top Plane in the FeatureManager (the left column at your screen in which all the parts of your model are listed). On this plane we will be making a sketch.	

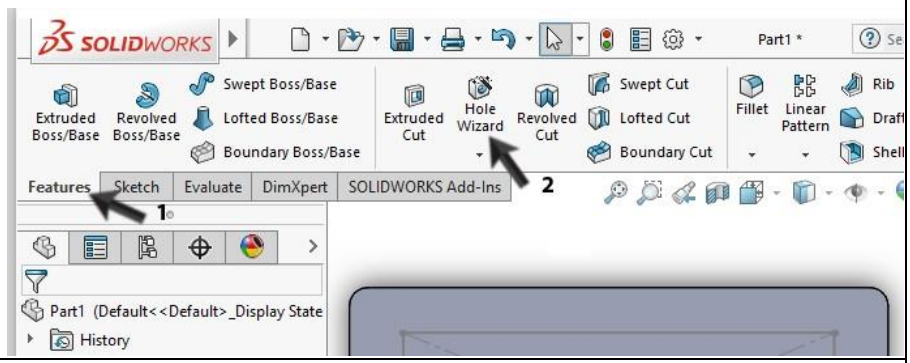
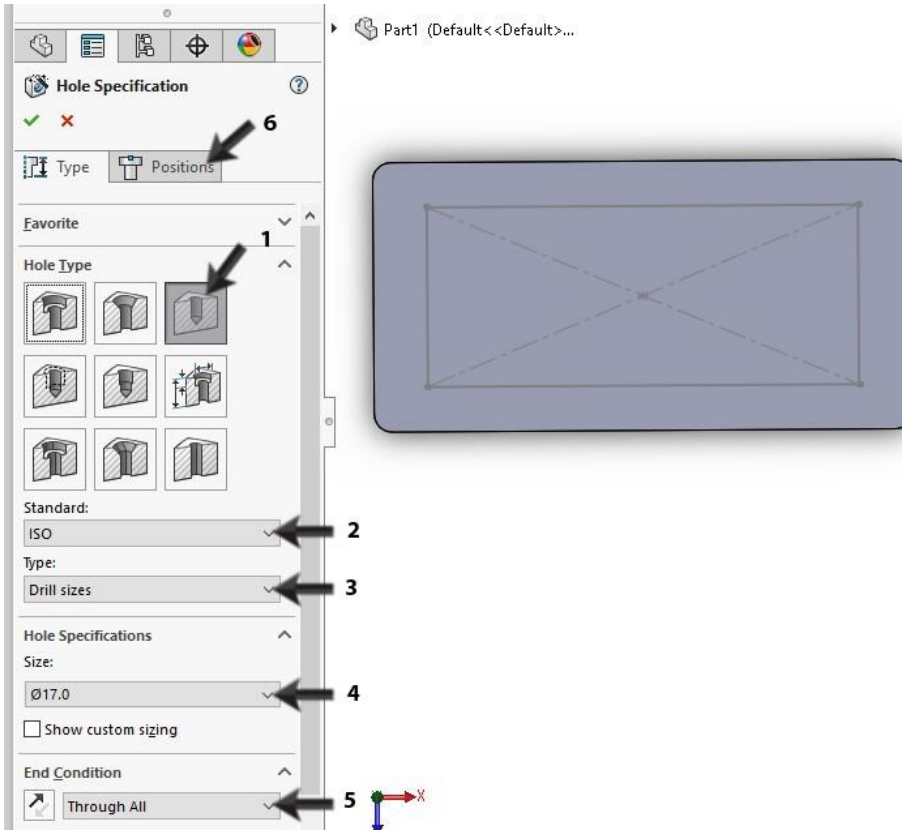
3	Click on Sketch in the CommandManager to reveal the right buttons and next on Rectangle to draw a rectangle.	
4	1. Click on Center Rectangle in the CommandManager. 2. Click on the origin. 3. Click at a random point like in the view at the right to draw a rectangle.	

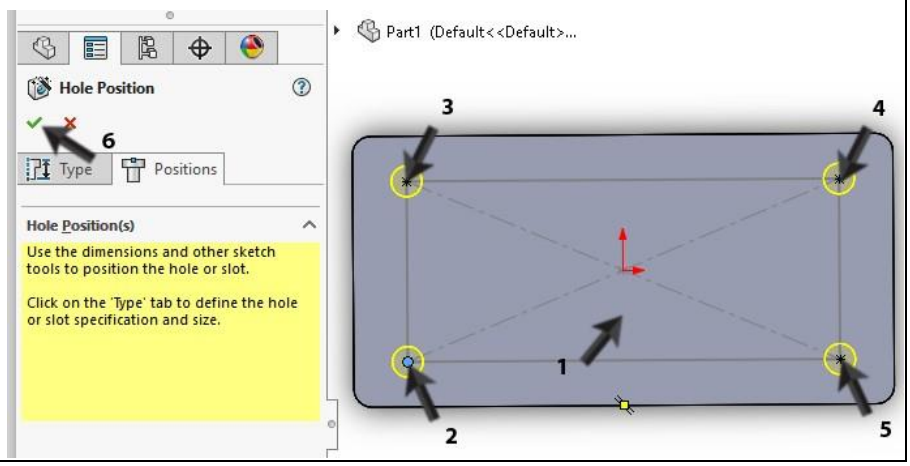
5	Next use the command Smart Dimension to place two dimensions at the sides of the rectangle: 150 and 300. You have used Smart Dimension before. Can you remember this? If not, look it up again in tutorial 2, step 7 to 10.	
6	1. Select the tab Features in the CommandManager.	

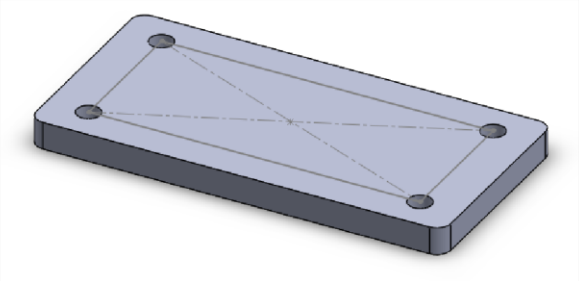
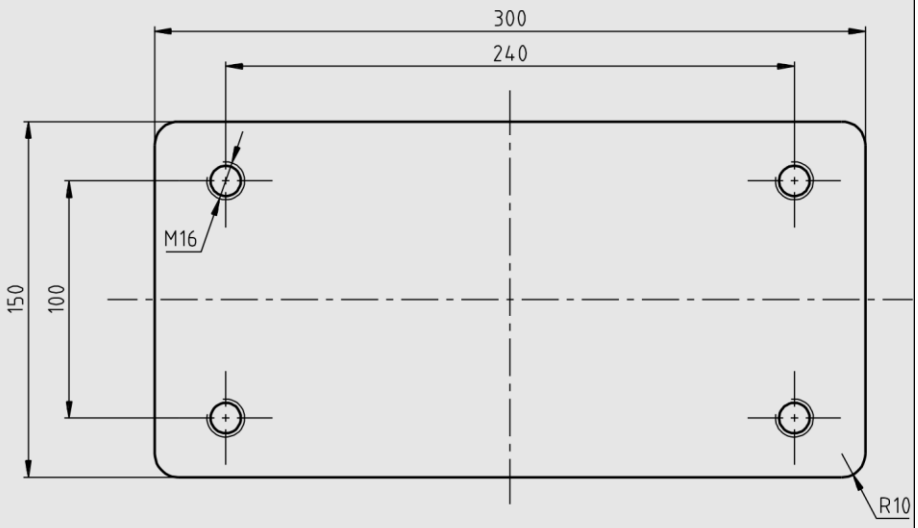
	2. Click on Extruded Boss/Base.	
7	1 Set the thickness at 20mm 2 Click OK.	
8	Next we will round of the corners. Click on Fillet in the CommandManager. The Fillet-command looks similar to the Chamfercommand which we have used before.	

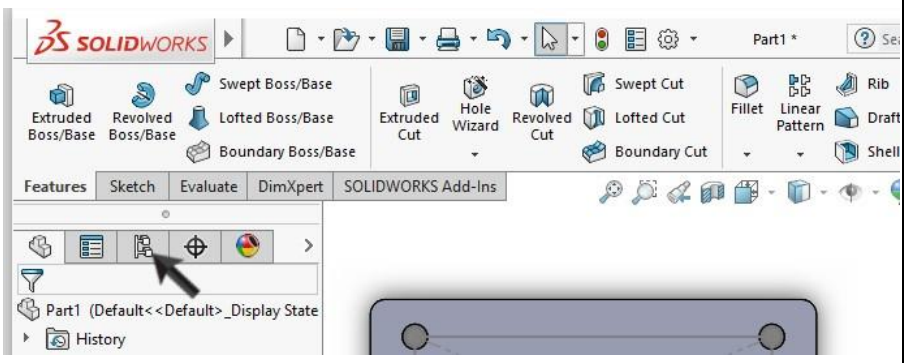
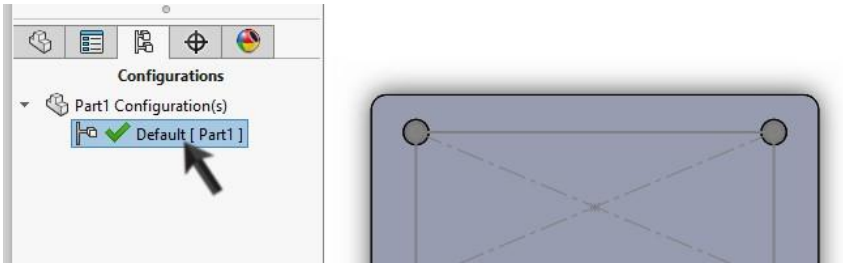
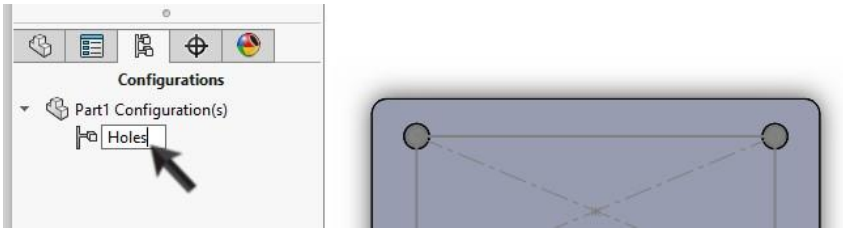
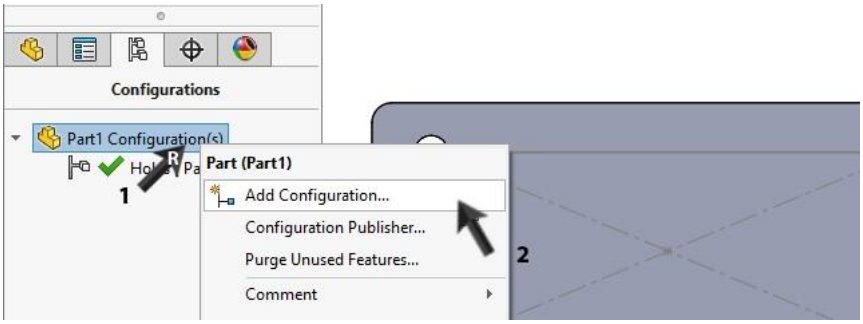
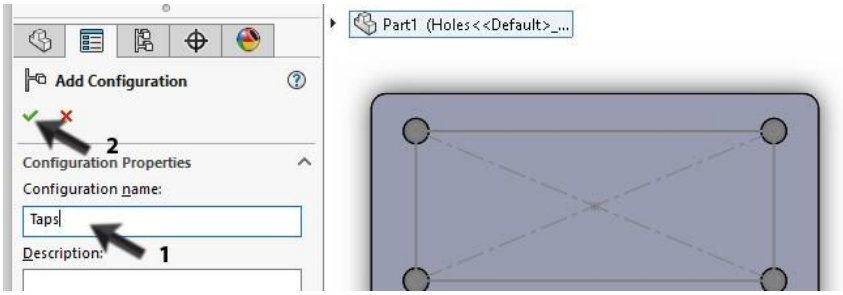
9	1. Make sure the option Full preview is selected. 2-5 Next select the four edges you want to round of. 6. Set the radius at 10mm 7. Click OK.	
10	Next select the top plane of the model just by clicking it.	
11	Click on Sketch and next on Rectangle to draw a rectangle.	

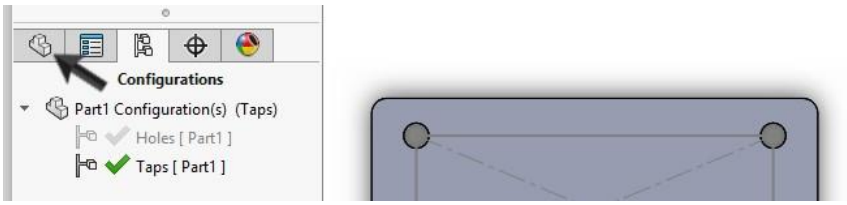
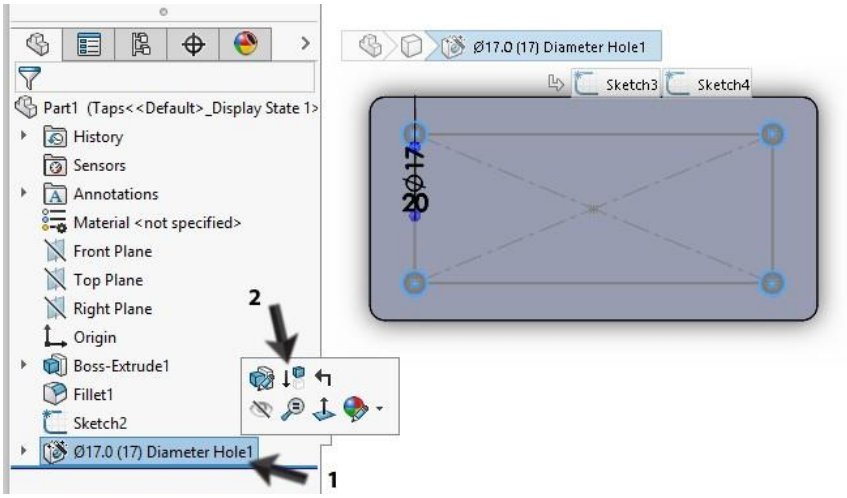
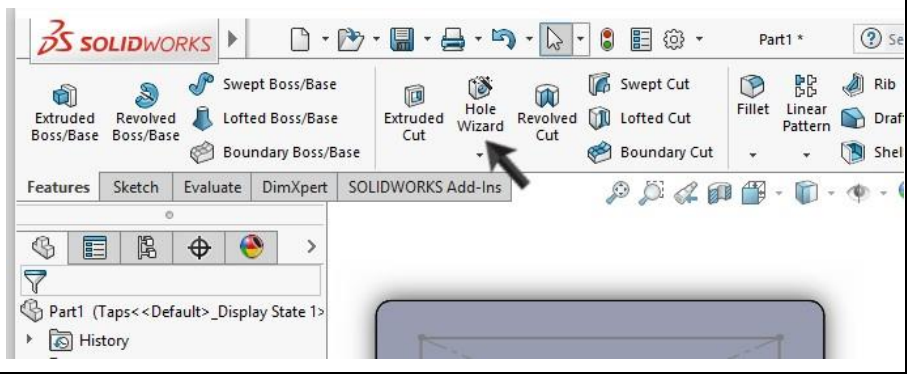
12	Click on the button Standard Views at the top of the screen and then on Normal To. The model now turns itself so you can have a straight view at the plane on which we are making the sketch. It does not matter if the model is horizontally or vertically at your screen.	
13	1. Click on Center Rectangle in the Property Manager. 2. Click on the origin. 3. Click at a random point like in the view at the right to draw a rectangle.	
14	Next add two more dimensions with the command Smart Dimension: the horizontal dimension of 240 and the vertical dimension of 100.	
15	Click on Exit Sketch in the CommandManager. The sketch remains visible, but turns grey. Notice that we made a sketch, but we did NOT make a feature of it. Later you will see how we will use the sketch after all.	

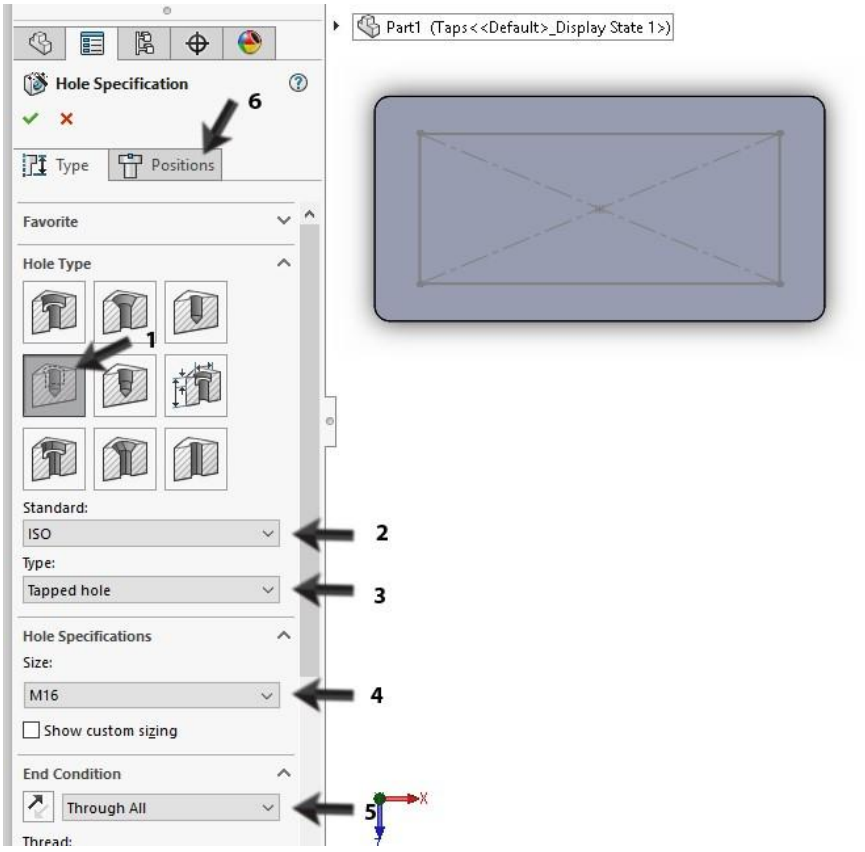
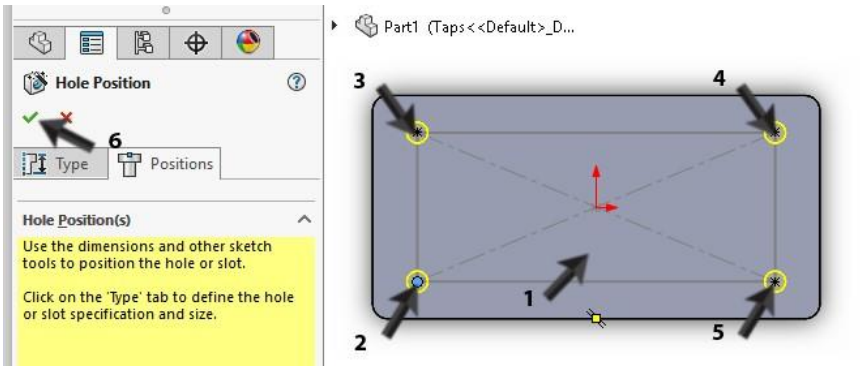
16	Select the tab Features in the CommandManager and then click Hole Wizard.	
17	You will have to set the features of the holes in the Property Manager. 1 Chose a hole type: chose Hole 2 Check if the standard is set at ISO 3 Check if the Type is set at Drill sizes 4 Set the diameter at Ø17mm 5 Set the End Condition at Through All	

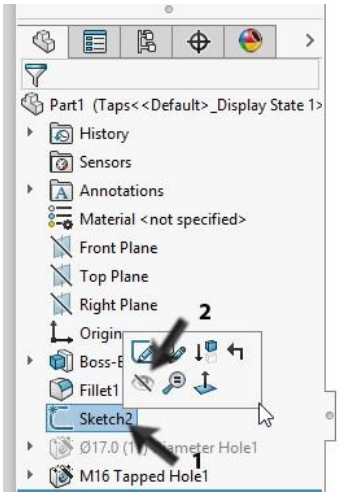
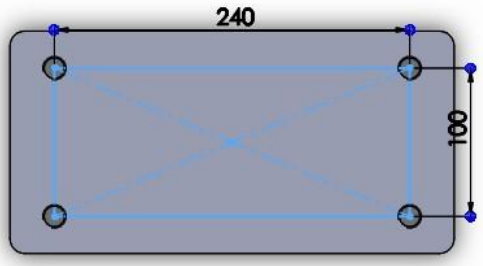
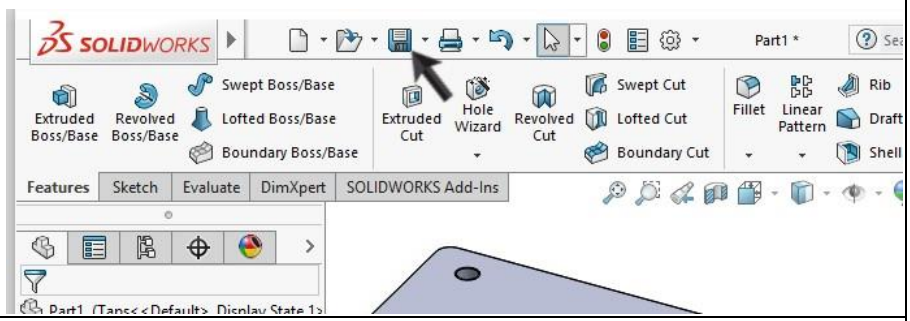
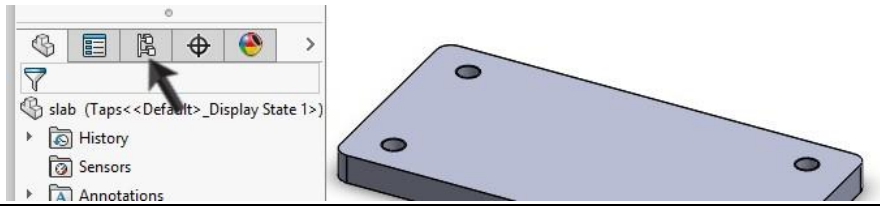
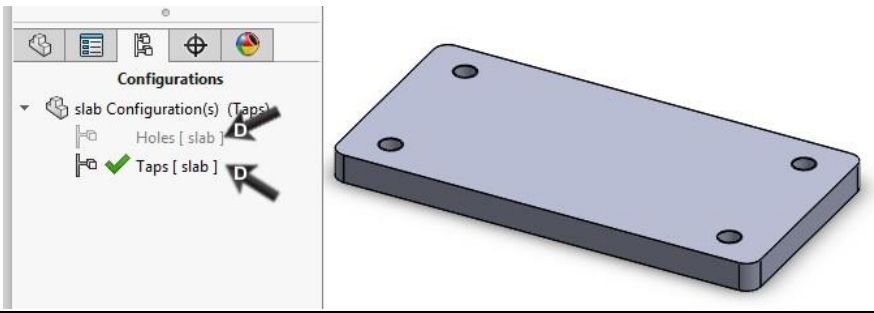
	6 Click on the tab page Positions	
18	1 Click on the plane where the holes are to be placed 2-5 Click on each of the four corners of the rectangle you've drawn before 6 Click OK	

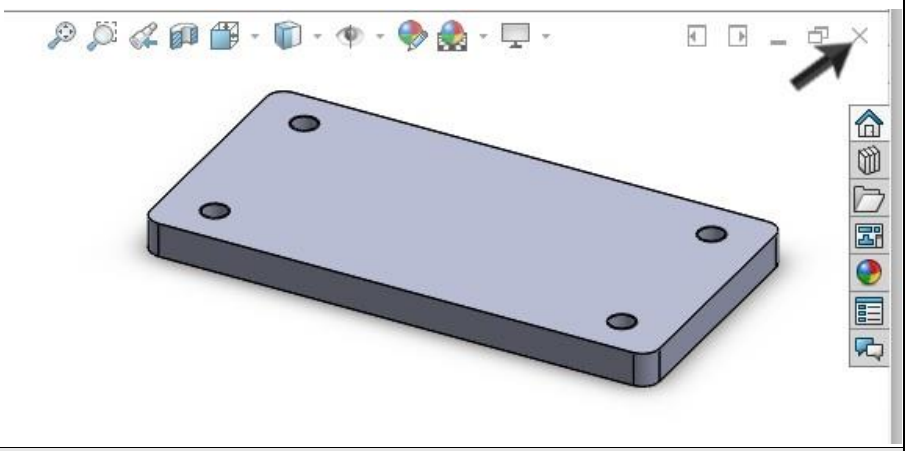
	Tip!	 The first part is ready now. The holes we just made, could also have been created with the Extruded Cut feature. The Hole Wizard we have used now, is often very convenient, even more if the holes you want to make a bit more complicated. Later on we will see an example of this.
	Werkplan	The second part we need looks very much like the last one. Instead of the normal holes we now need tapped holes. You could create a whole new part now, but it is much easier to make a second version within this part. We call this a Configuration.  We will do following: 1. create a new configuration 2. remove the normal holes in the new configuration 3. make tapped holes instead. If you experience any problems in working with configuration, you can always create a new part in exactly the same way as the first part. Use step 27 instead of step 17.

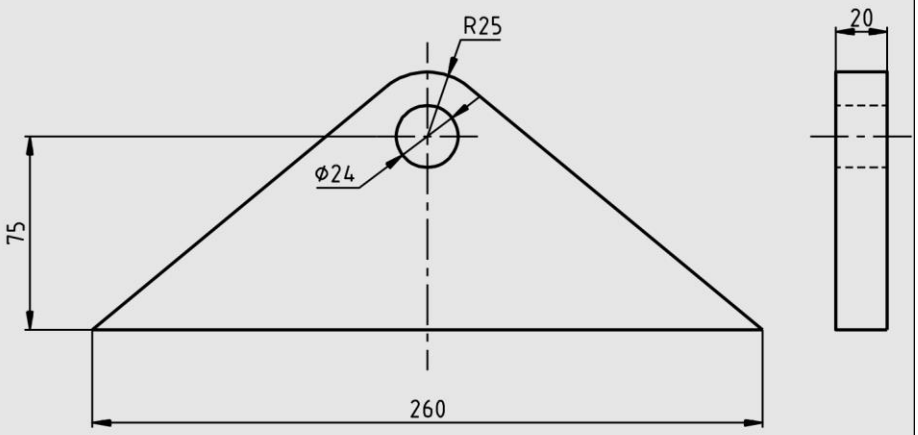
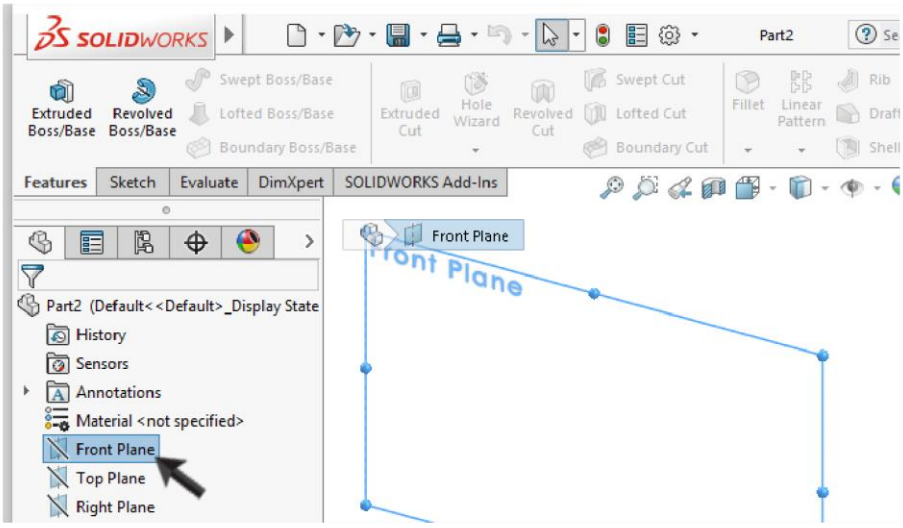
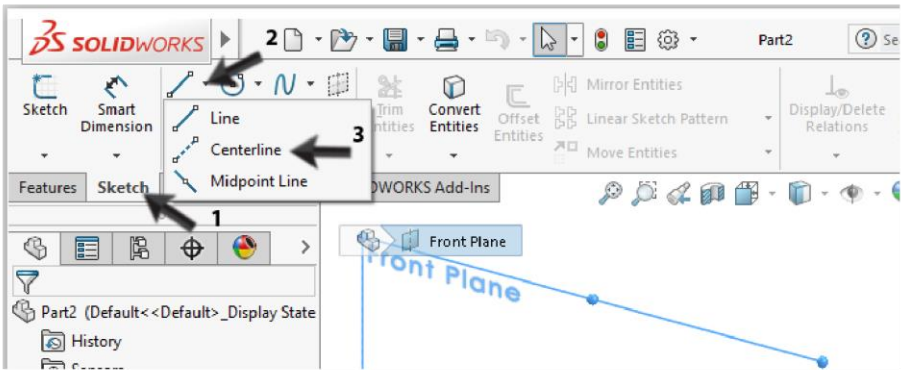
19	Click on the third tab in the FeatureManager. Instead of the FeatureManager or the Property Manager, the ConfigurationManager now appears.	
20	There is only one configuration, named 'Default [Part1]'. Click slowly on the name once or twice to change the name.	
21	Rename this item as: 'Holes'. Press <Enter>	
22	Next make a new configuration: 1 Right click on the top line of the list (Part1 Configuration(s)) 2 Select Add Configuration in the menu.	
23	Fill in the name of this configuration in the Property Manager: 'Taps', and click on OK.	

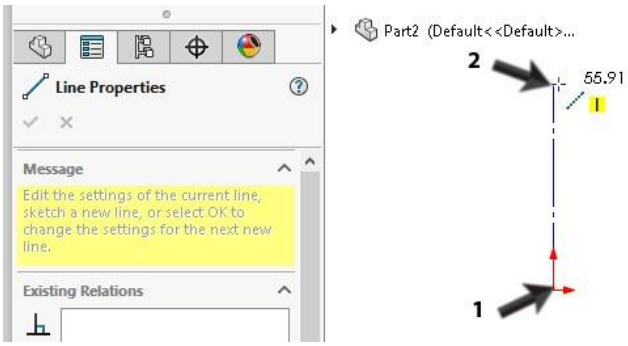
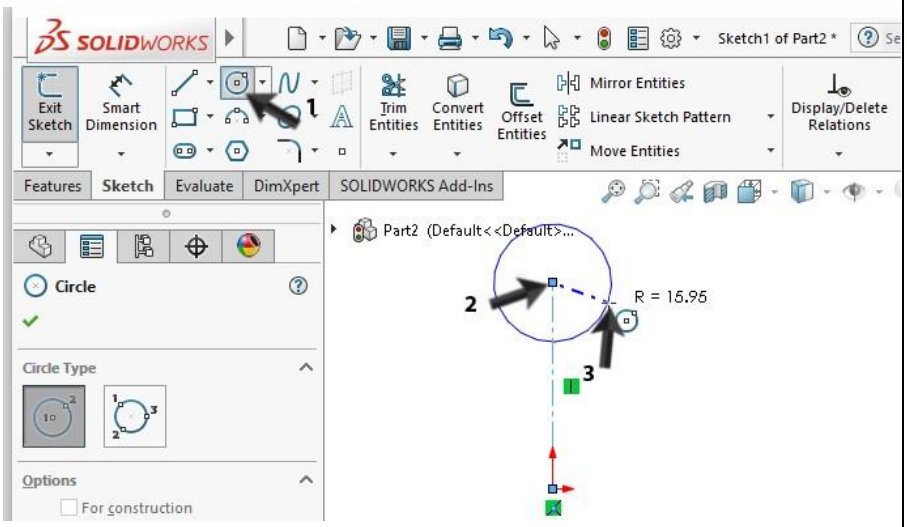
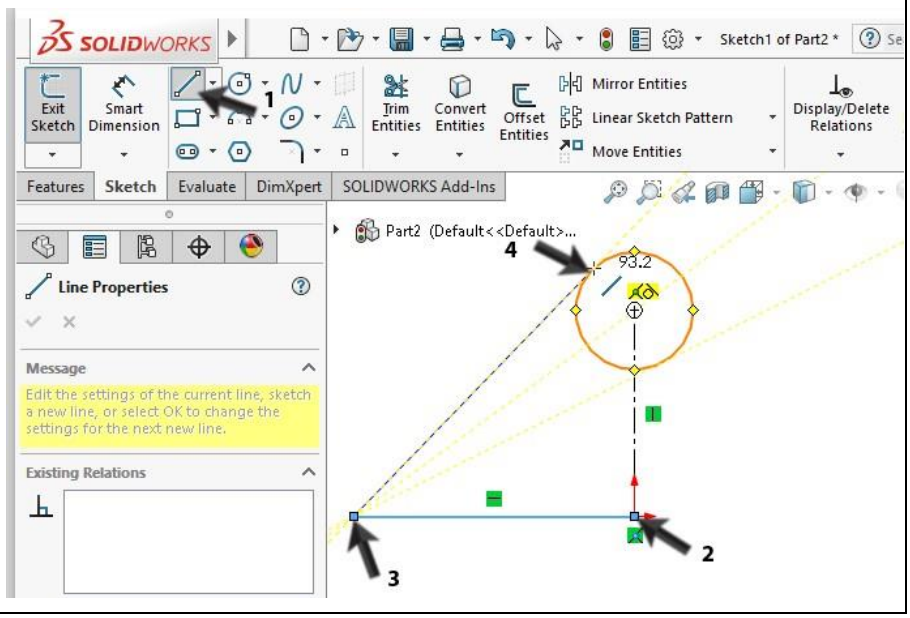
24	Click on the first tab of the ConfigurationManager to go to the.	
	Tip!	At this point we have two configurations: only one is active, this is the one we are working in. • In the ConfigurationManager you recognize the active configuration because it is printed in black. (check this at step 24) • In the FeatureManager the name of the active configuration is at the top of the list, behind the name of the part (check step 25).
25	1. Click on the last feature you created (the holes). 2. Click on Suppress in the menu. The holes now disappear from the model and are printed grey in the FeatureManager.	
	Tip!	Instead of clicking on a feature with your left mouse button, you can also use the right mouse button. You will see a much more extended menu.
26	Click on Hole Wizard in the CommandManager.	

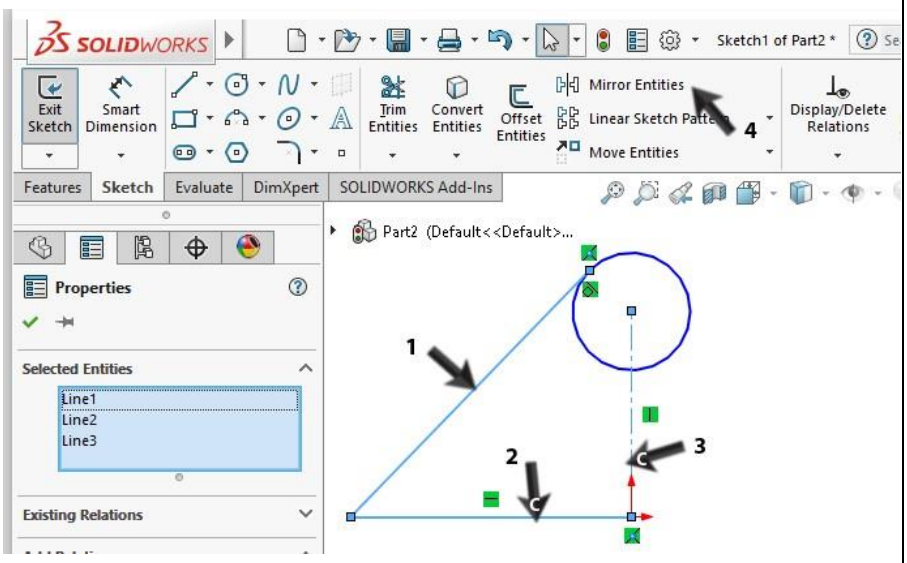
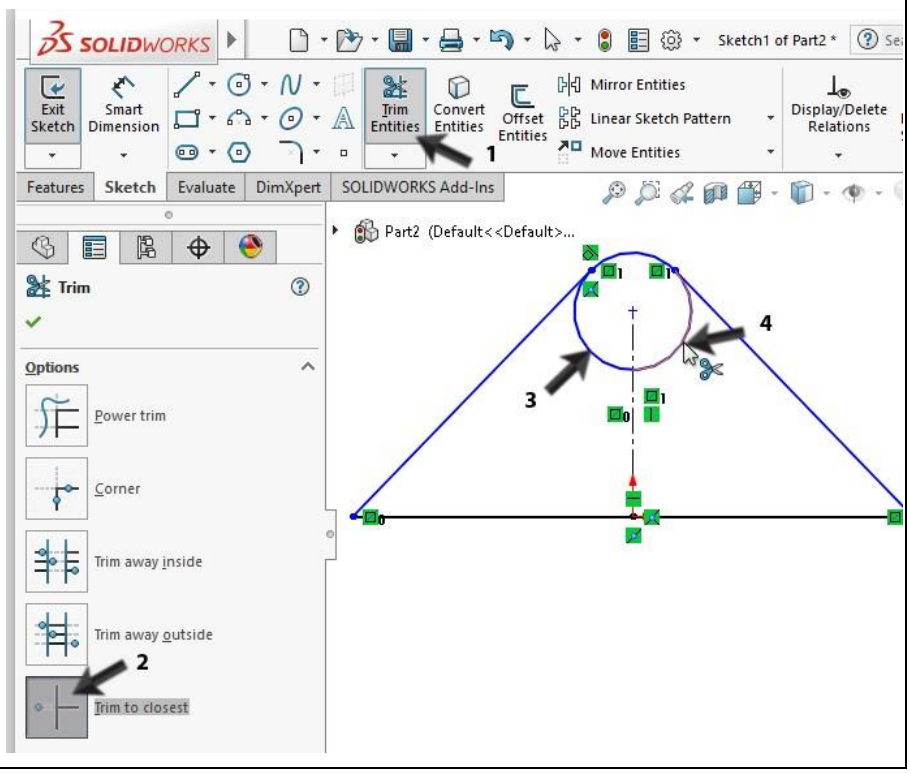
27	Set the properties of the holes in the Property Manager. 1 Chose hole type: Tap 2 Check if the standard is set at ISO 3 Check if the Type is set at Tapped hole 4 Set the dimension at M16 5 Set the End Condition at Through All 6 Click on the tab Positions	
28	Position the holes in the same way you did in step 18	

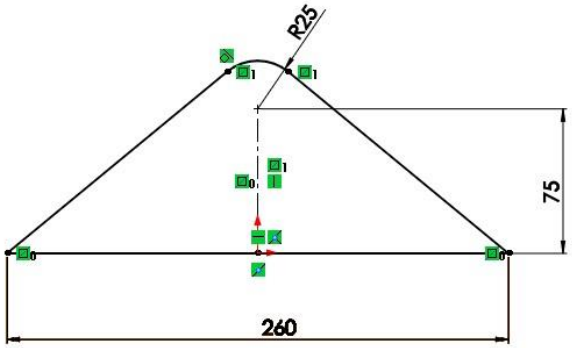
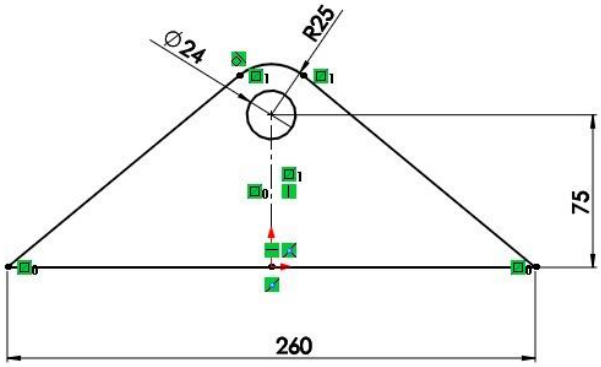
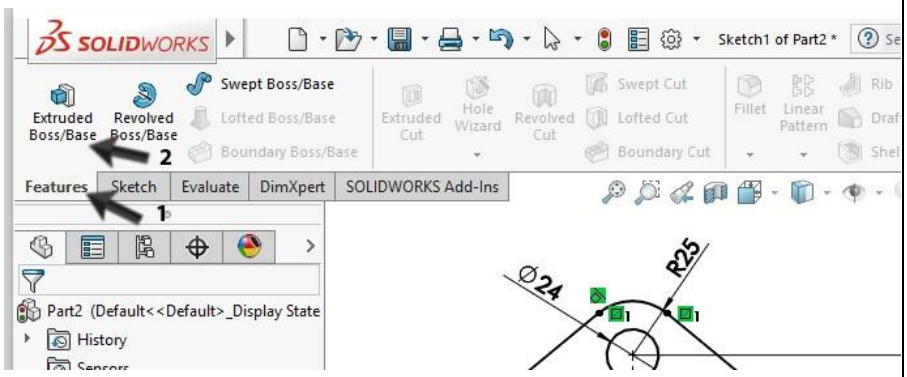
29	Now click on the sketch which you have used to position the holes. Probably it is named 'Sketch2' or 'Sketch3'. The number can be different. Click on Hide in the menu which appears.	 
30	Save the file as: slab.sldprt	
31	Next click on the third tab at the top of the FeatureManager to go to the ConfigurationManager.	
32	There are now two versions (configurations) of the base model: one with normal holes and one with tapped holes. Only one of these two is active (and visible). By double clicking on a configuration in the ConfigurationManager you will make the configuration active. Try this now.	

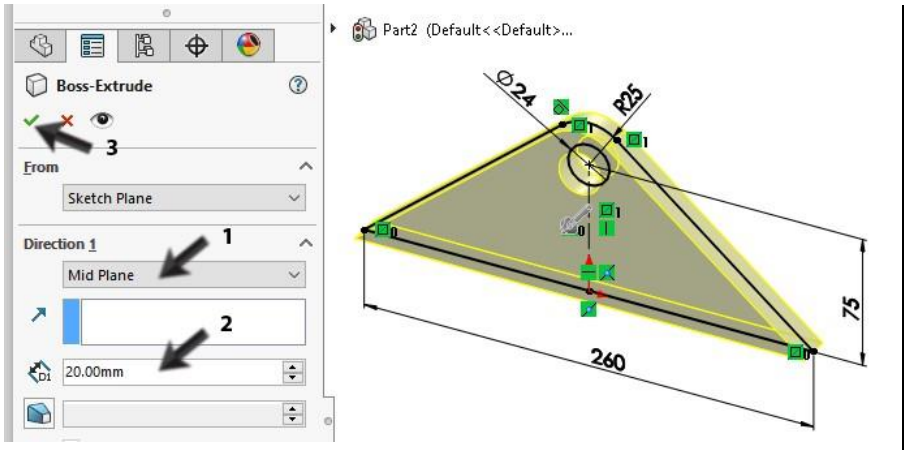
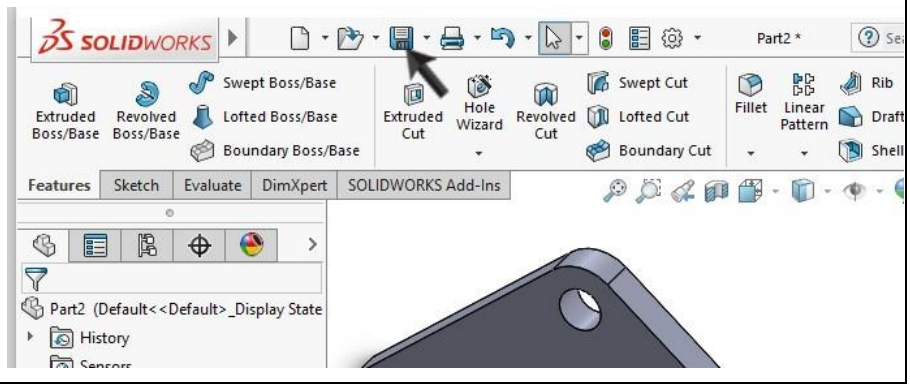
33	Close the file by clicking on File and next on Close.	
	Tip!	In this product we need two plates of material. These are the same of course, only the hole properties are different from each other. Of course we could have created a second plate, but then we had to do a certain amount of command for a second time. This was not needed while we used configurations. So, in a case like this, it is a good idea to work with the configurations command. Within a single part you create different 'versions' of the same product or part. In the ConfigurationManager you can make a choice which

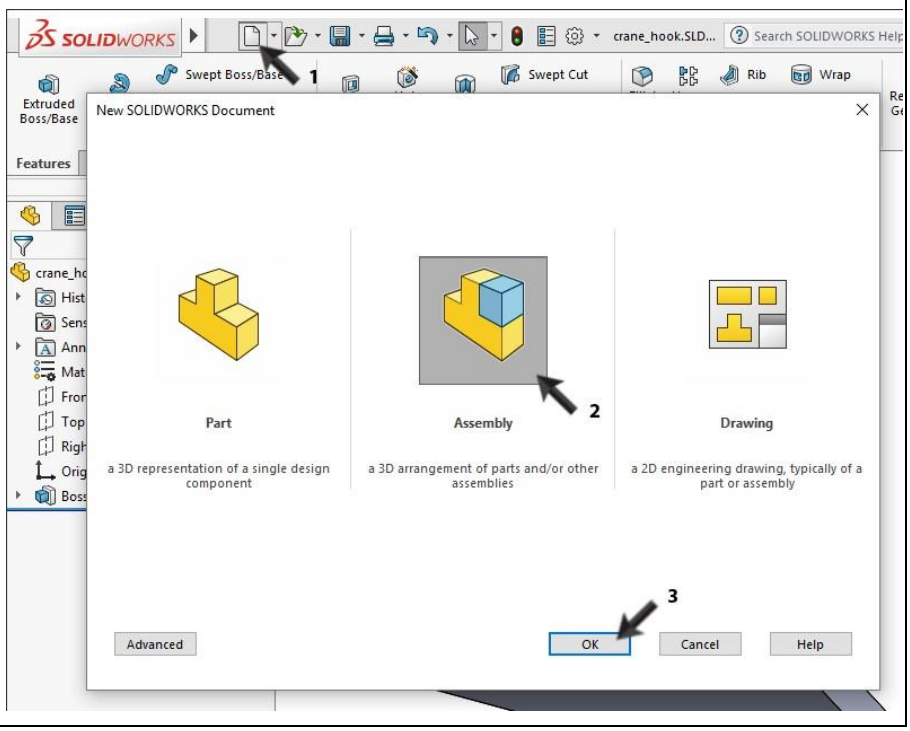
		version is active: this is the version you work in and changes the features. Within every configuration you can make features invisible (suppressed) or visible (unsuppressed). By doing so, we create more than one version, and in every version you have different features visible, like the normal holes or the tapped holes in the two versions we have just done. Of course there are also many features which have to be visible in every version, like in the first part you have created. By changing a dimension in one version, the other versions will be changed immediately!
	Work plan	The next part we have to create is the bracket on top of crane hook.  To create this part we only have to make a sketch and extrude it.
34	Open a new part, select the Front Plane and create a sketch.	
35	Click on Sketch in the CommandManager and select Centerline.	

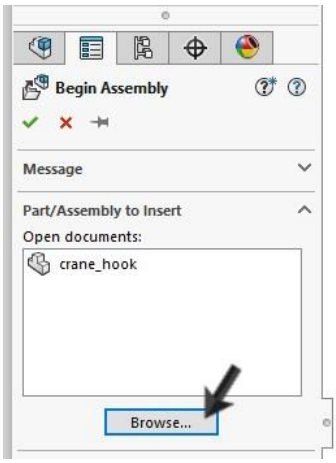
36	Draw a centerline from the Origin straight up.	
37	Next draw a circle. Click on the top end of the centerline. Move the mouse and click again to create a circle with a random radius.	
38	Now draw two lines: 1 Click on Line in the CommandManager. 2 Click on the origin. 3 Move the mouse horizontally to the left and click again to set a second point. (check the view on the right) 4 Move the mouse towards the circle. Move the mouse over the circle until the two yellows icons show like in the illustration on the right. When this is the case, you click to create a line which is in contact with the circle.	

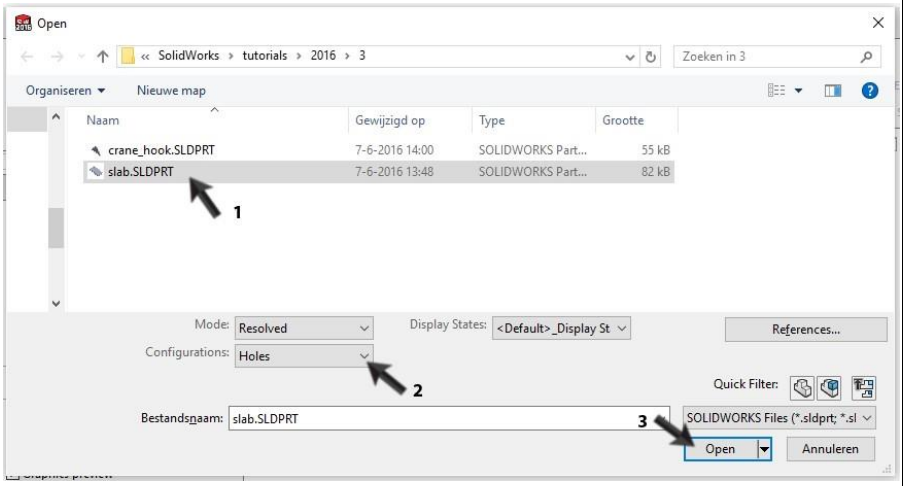
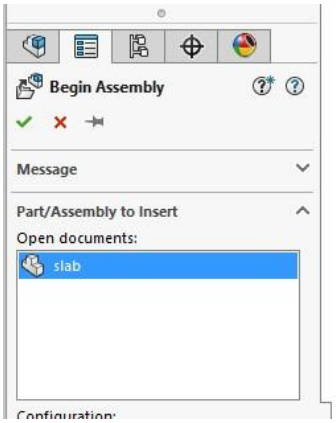
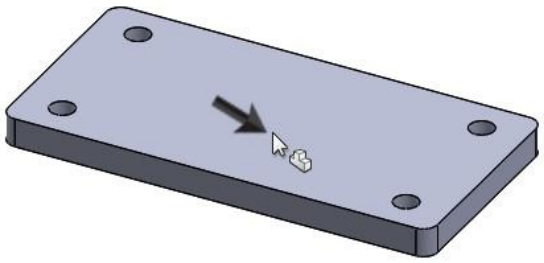
39	Next we will mirror two lines. Push <esc> on your keyboard to end the line command. 1. Select the first line 2. Hold the <Ctrl>-key and select a second line 3. Keep the <Ctrl>-key down and select the centerline 4. Click on Mirror Entities in the CommandManager.	
40	The bottom part of the circle has to be removed. 1 Click on Trim Entities in the CommandManager. 2 Select the option Trim to Closest in the Property Manager. 3,4 Next click on the two parts of the circle which have to be removed.	

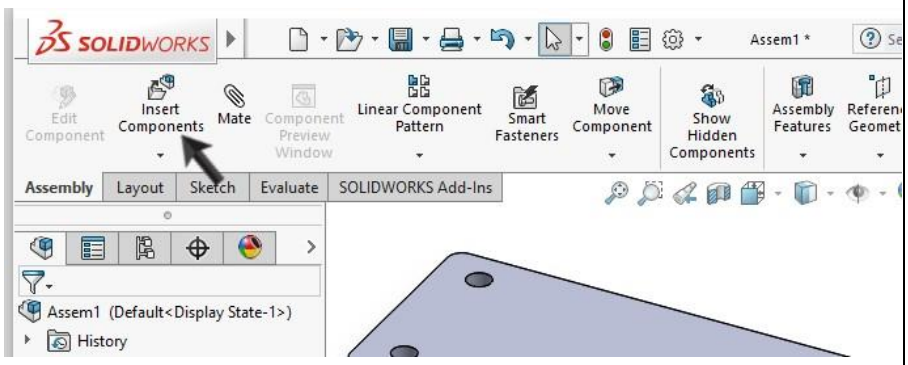
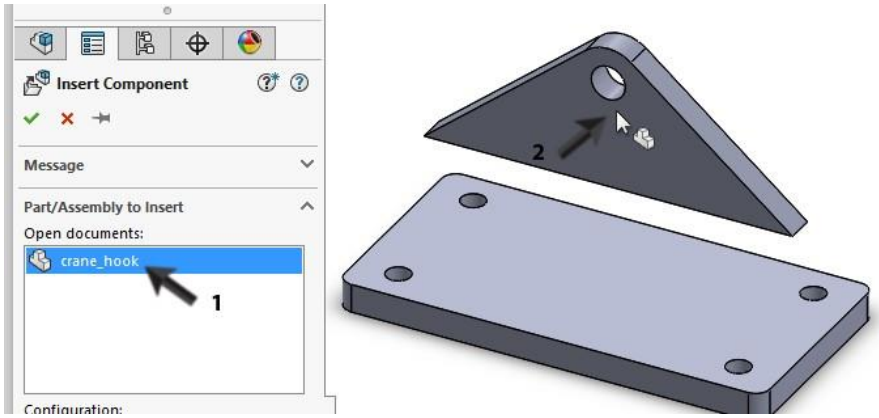
41	Add three dimensions to the sketch using Smart Dimension Check the illustration on the right.	
42	Finally, draw another circle to make a hole with a diameter of Ø24.	
43	rial can extrude the mateof the sketch now. 1 Click on Features in the CommandManager 2 Click on Extruded Boss/Base.	
44	1 Select the option Mid Plane at Direction1 in the Property Manager 2 Set the thickness at 20mm.	

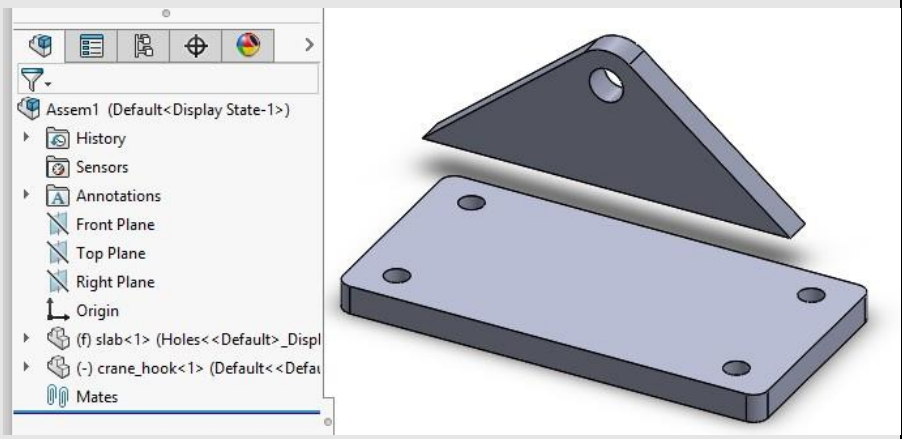
	3 Click on OK.	
45	Save the file as crane_hook.sldprt	

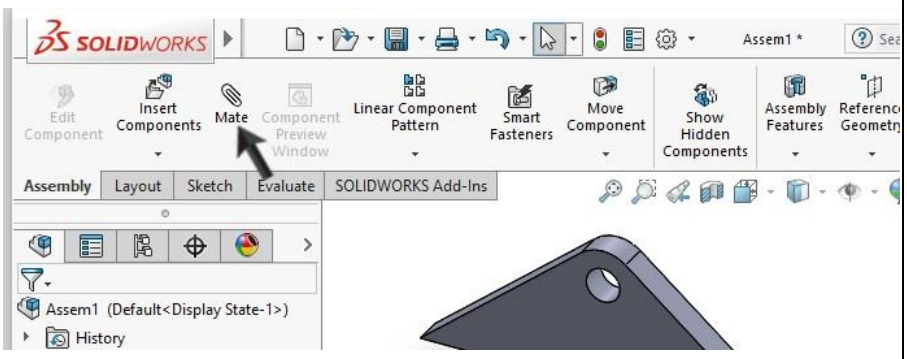
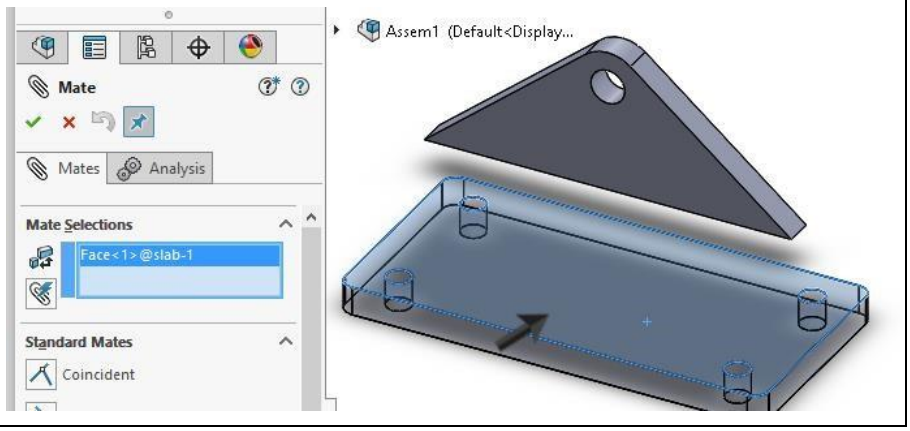
46	The parts are ready for the assembly. 1 Click on New in the toolbar 2 Select file type Assembly 3 Click on OK.	

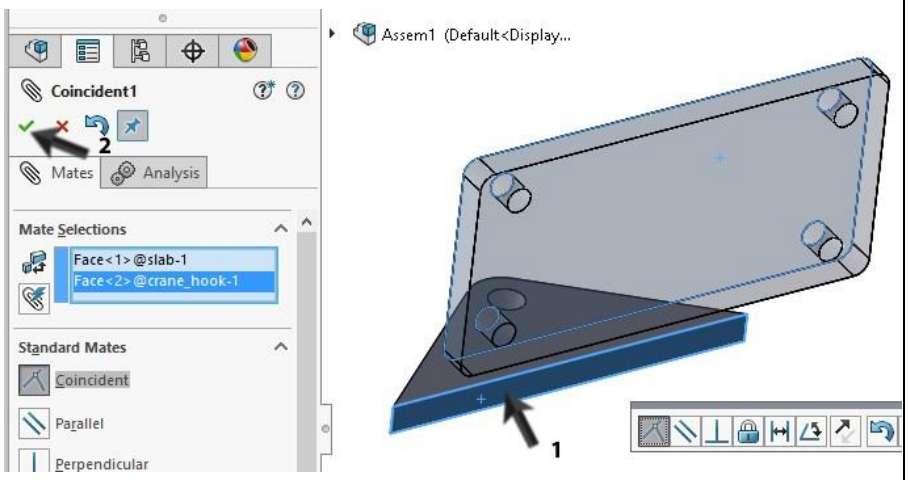
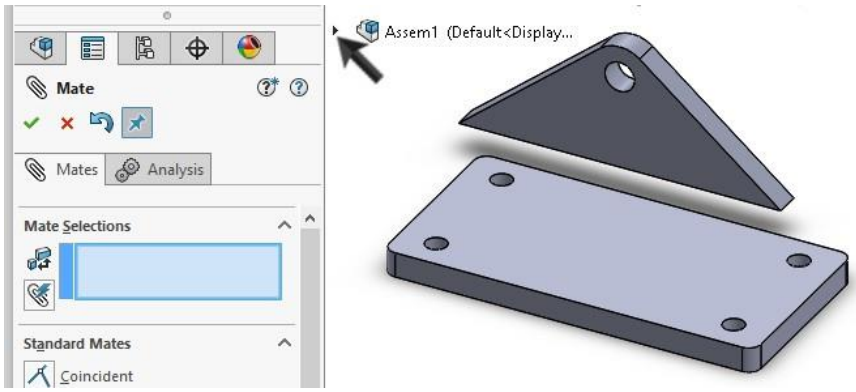
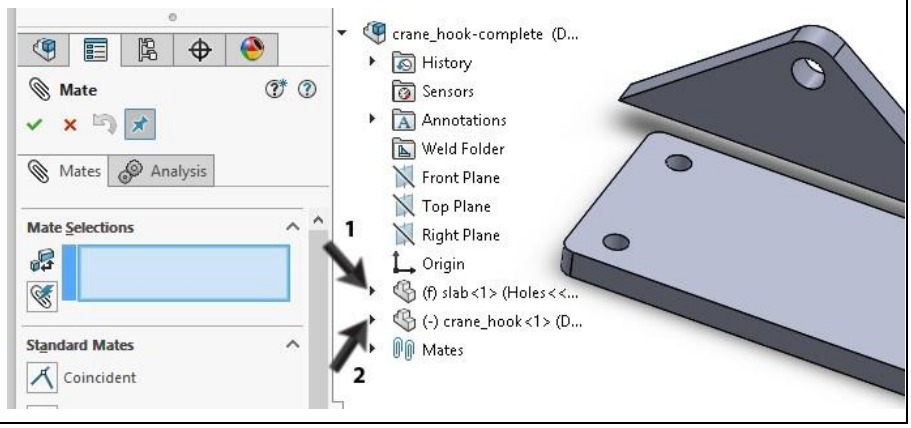
47	We have closed the file slab.sldprt, for this reason it is not in the list in the Property Manager. Click on Browse… Pay attention! Even when the file is not closed and is in the list, click on Browse. If you do not do this, you will not be able to select the right configuration.	
	Tip!	Normally, the Insert Component command starts automatically when a new assembly is opened. If this does not happen, click on Insert Component in the CommandManager.

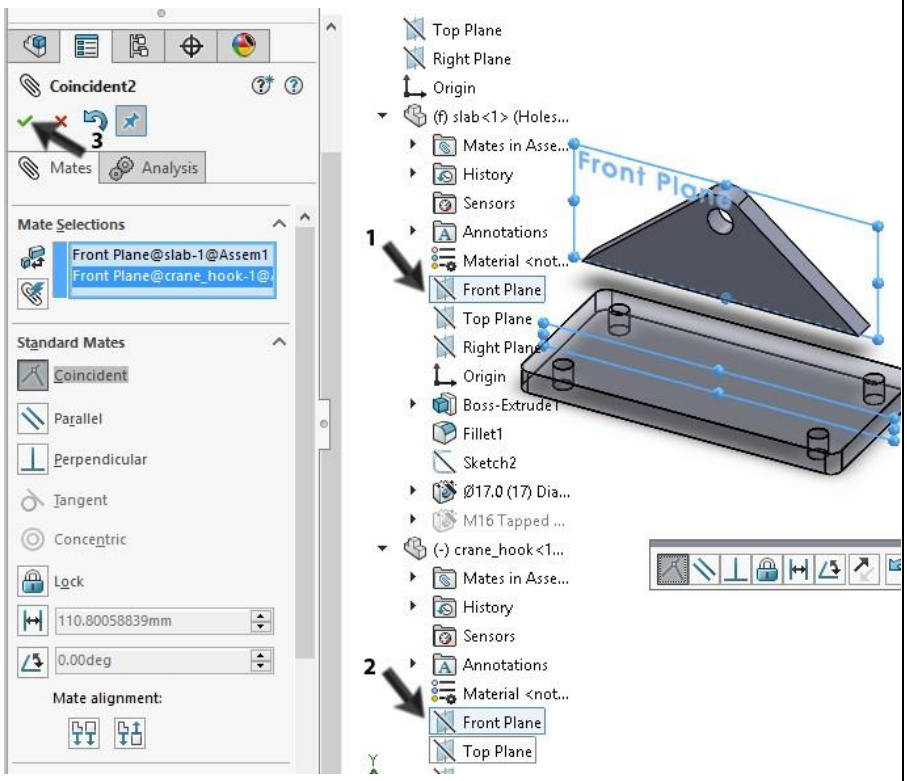
48	Find the file slab.sldprt, which we made earlier. 1 Select the file. 2 This file contains more than one configuration so you have to choose which configurations you will be using. Select 'Holes'. 3 Click on Open.	
49	Now the part is fixed to the cursor. Do not click in the graphical area, but click on OK in the Property Manager.	 

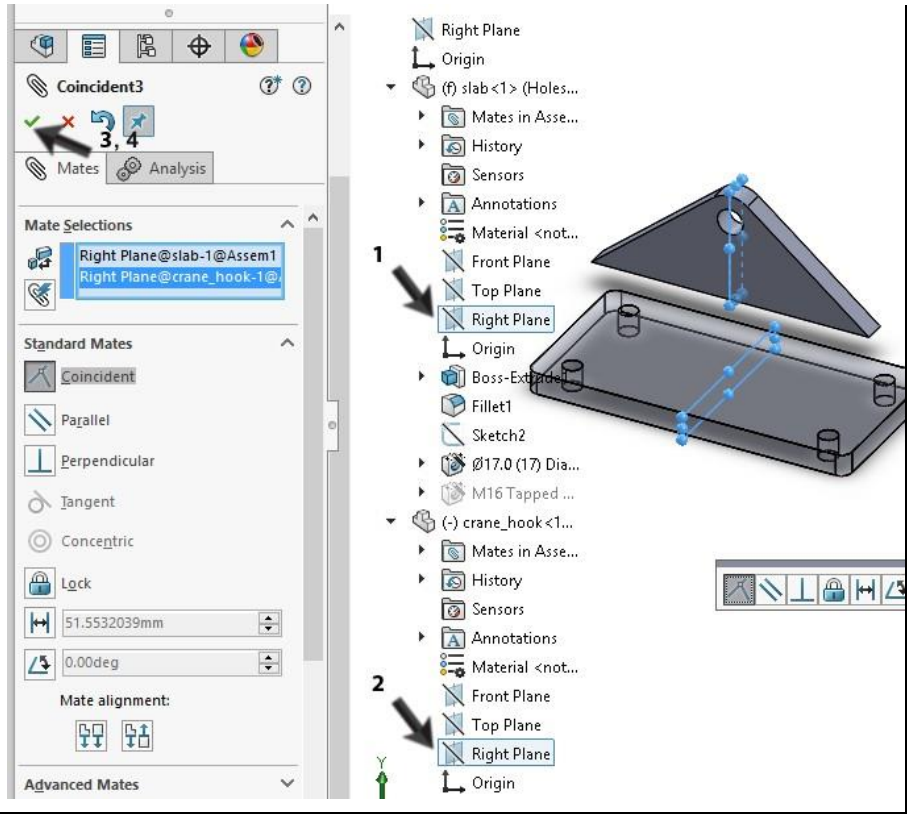
50	To add the next part, click on Insert Component in the CommandManager.	
51	1 Select the file 'Crane_hook' in the list, 2 Place the part at a random position in the assembly.	

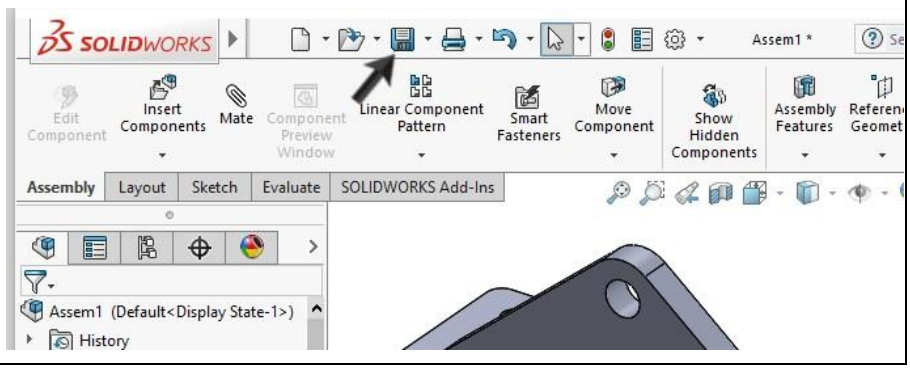
	Tip!	Did you execute the steps correctly until now, you will notice that the base part cannot be moved, whilst the crane hook can be moved around. This is because the first part you chose is Fixed. In the FeatureManager you can verify this because in front of the filename Slab is an **(f)**, and before the 

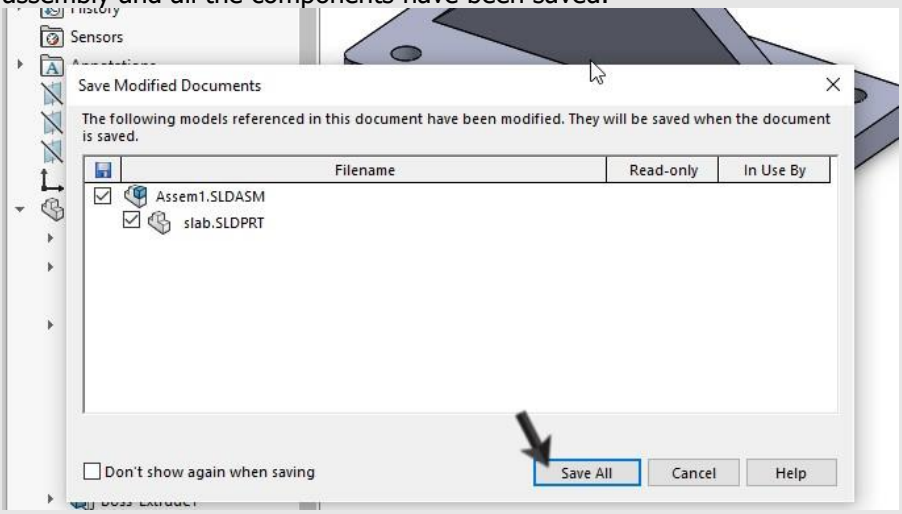
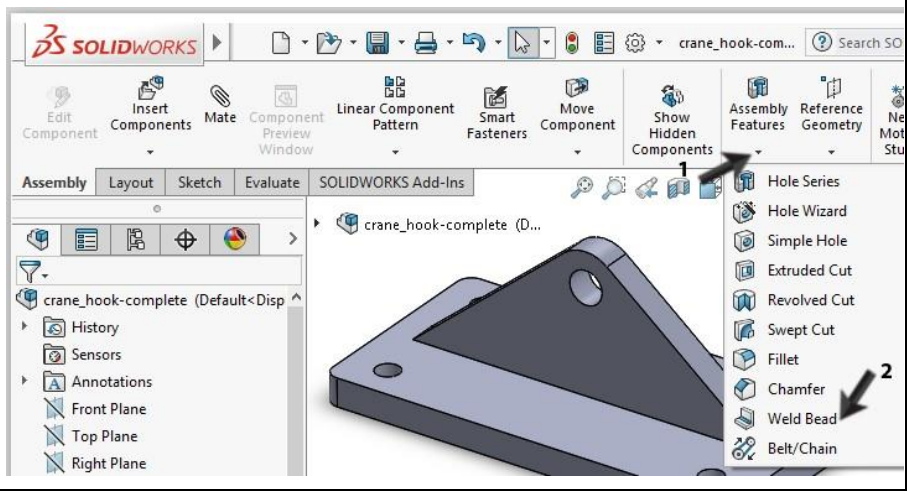
		Crane_hook a **(-)**. This part with an (f) is a floating part and can be moved around. Be sure at all times that ONE part is Fixed; the rest is connected to this with the mate command. You can make any part Fixed or Floating by clicking on it with the right mouse buttons and chose Fix or Float.
52	Click on Mate in the CommandManager.	
53	Click on the upper surface of the part.	

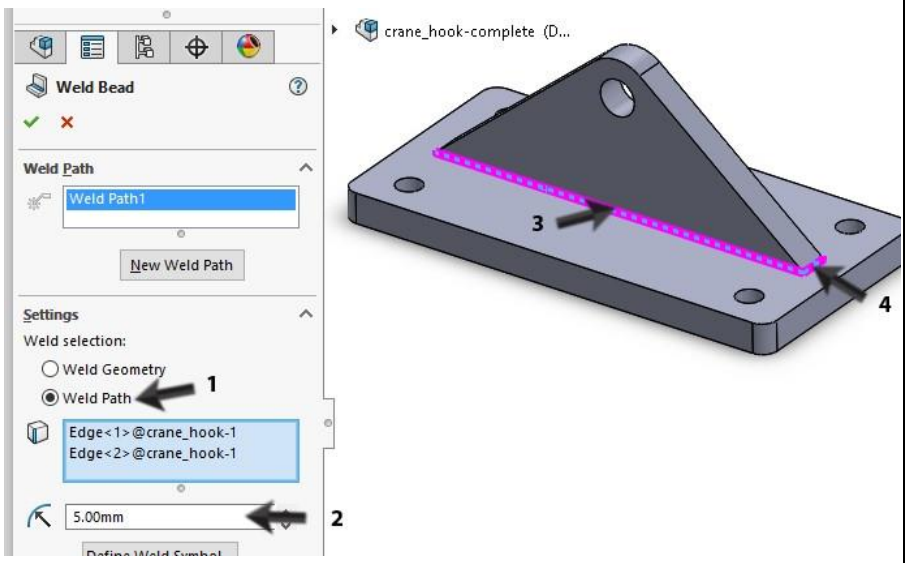
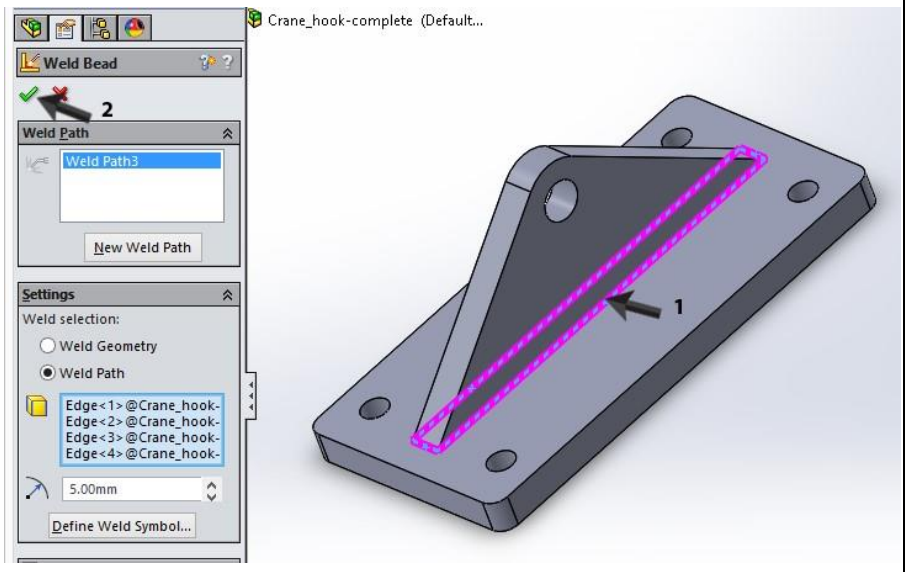
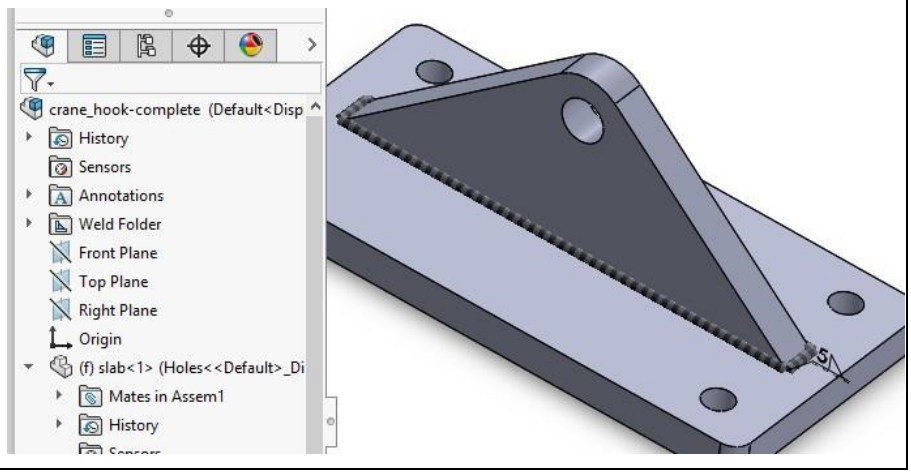
54	Rotate the model so you get a clear view at the bottom side of the crane hook. Press the scrollwheel and move your mouse to rotate. 1 Click on the bottom of the crane hook. The now parts now move to each other. 2 Click on OK.	
55	The selection field in the Property Manager is now emptied and you can start with the next mate immediately. To center the crane hook we use the standard planes Front Plane and Right Plane. You cannot select them in the model however, only in the FeatureManager. Because now the Property Manager is visible and not the FeatureManager, you must use the FeatureManager in the graphical area. Click on the + directly in front of the file name.	
56	Next click on the + in front of both parts. Pay attention: after clicking on the first + the list expands.	

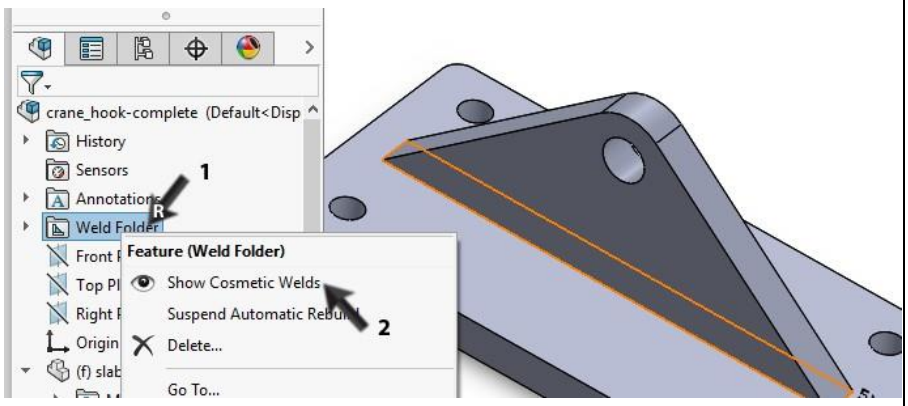
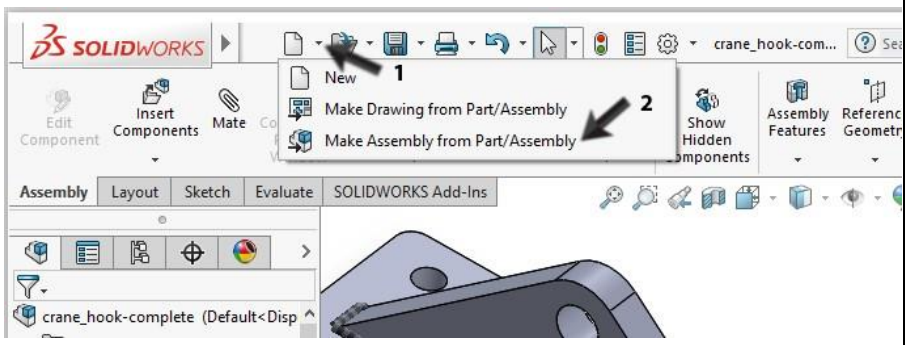
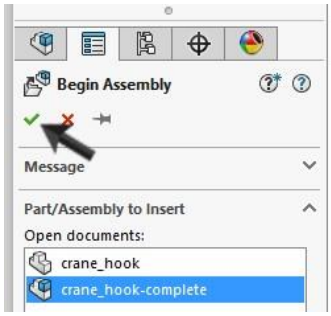
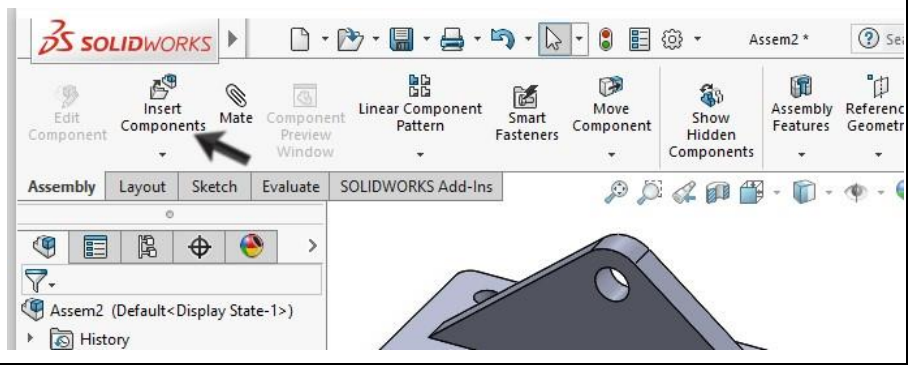
57	1 Next select the Front Plane within the part 'Slab' 2 Also select the Font Plane within the part 'Crane_hook'. 3 Click OK.	
58	1 Select the Right Plane within the part 'Slab'. 2 Also select the Right Plane within the part 'Crane_hook'. 3 Click on OK.	

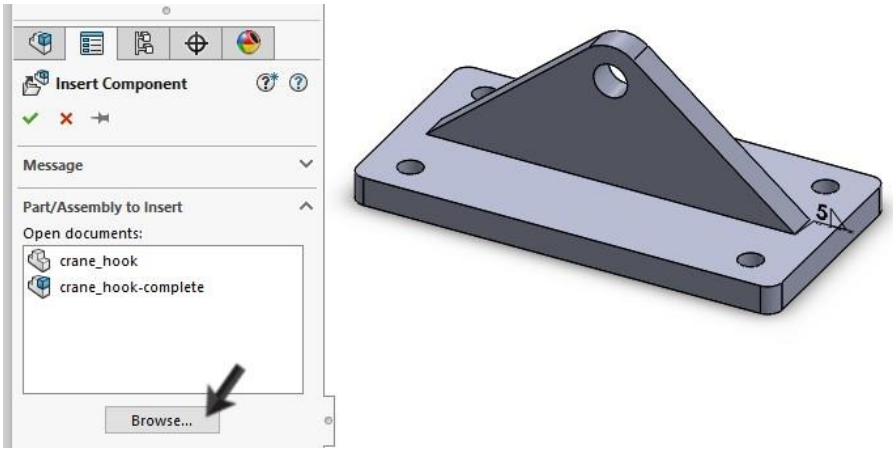
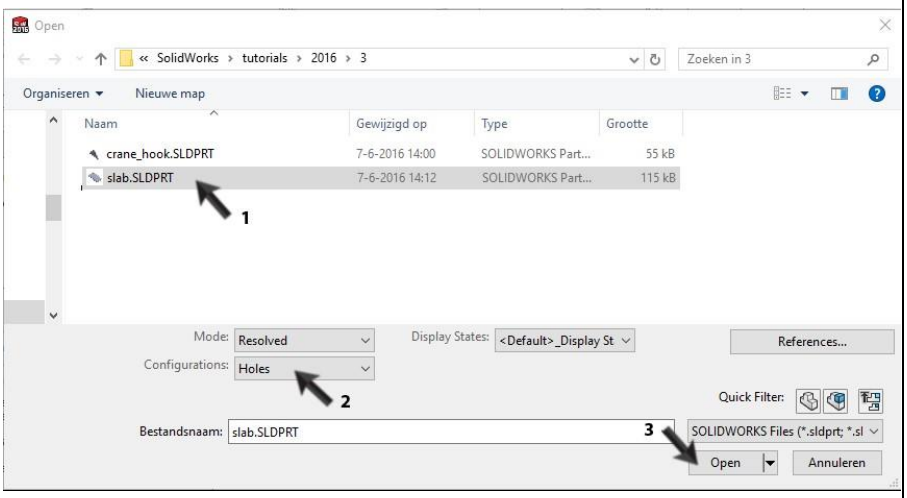
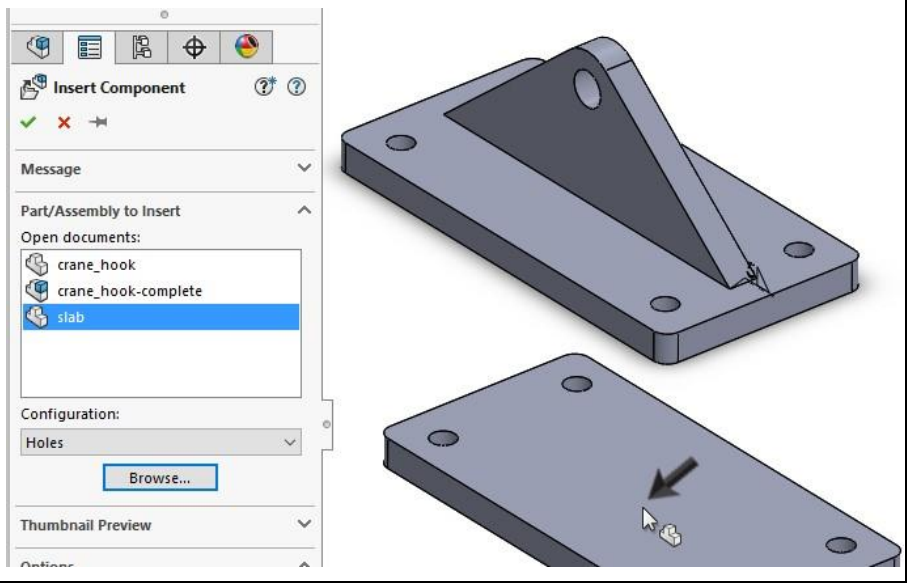
	4 Click on OK again to confirm the mate, and again to close down the mate-command.	

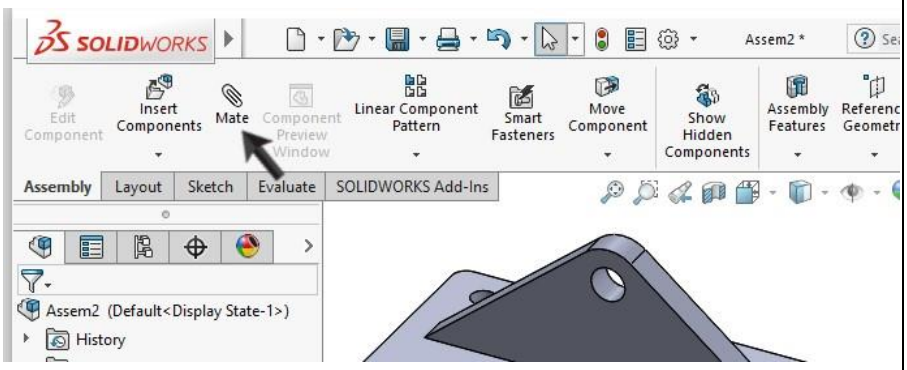
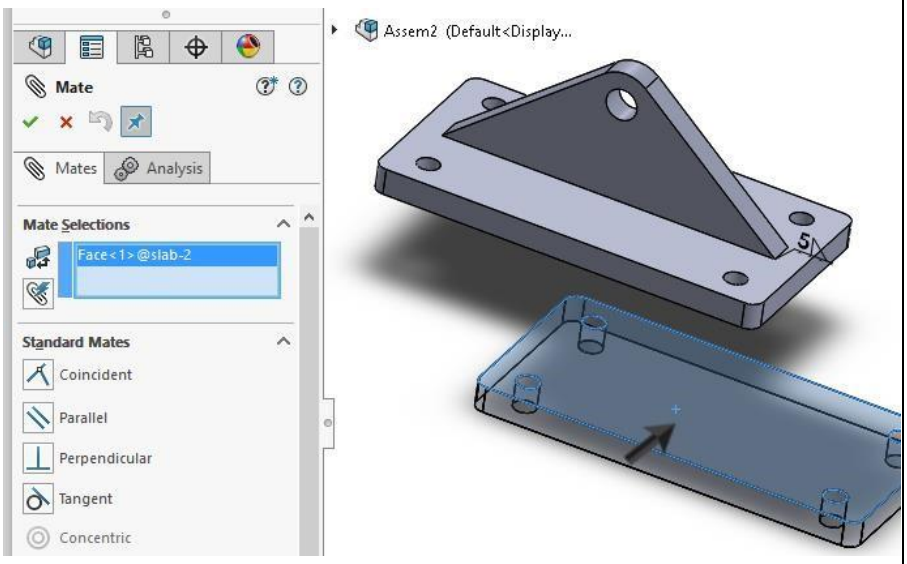
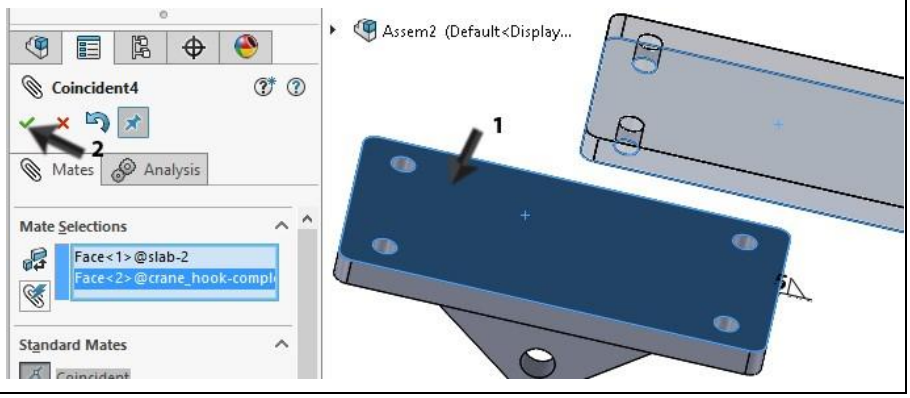
59	Save the assembly as: crane_hookcomplete.sldasm	

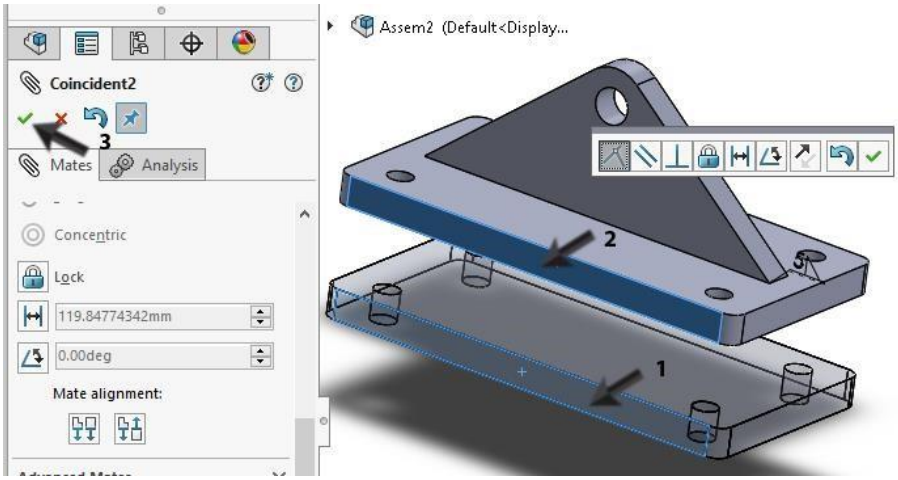
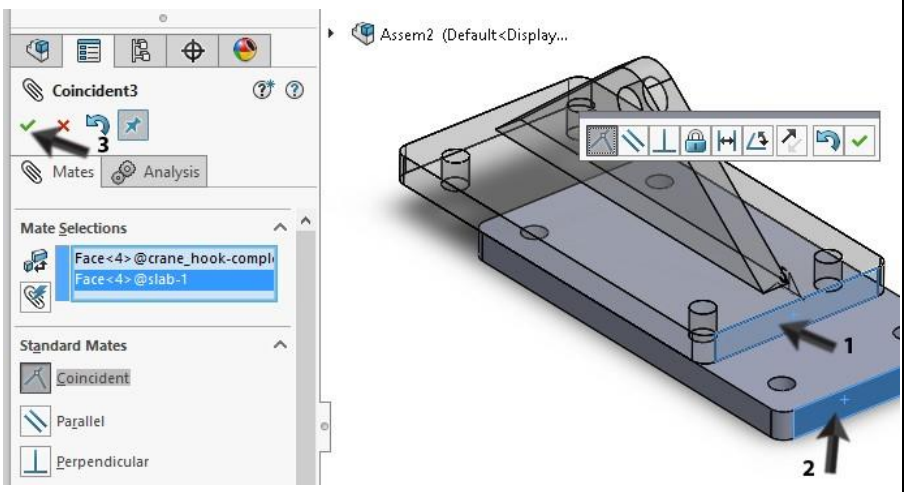
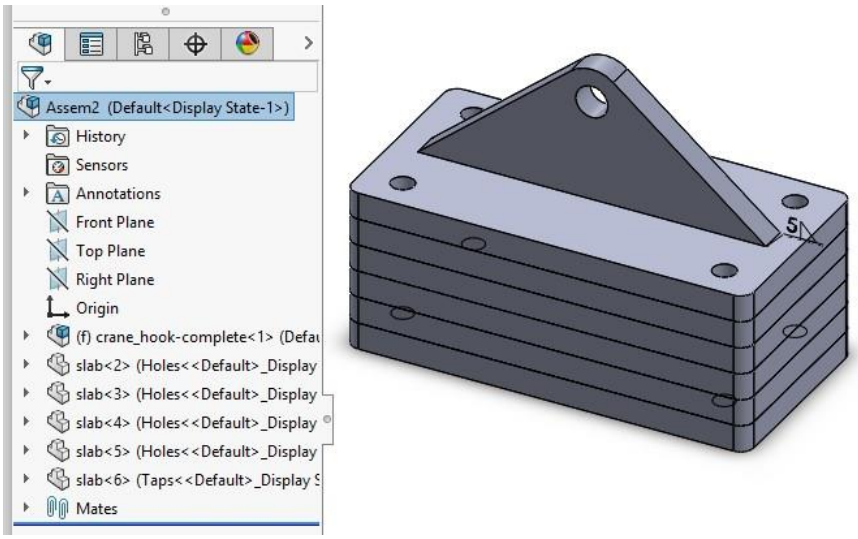
	Tip!	When saving an assembly, you could see a screen like shown below. This is because SOLIDWORKS checks if any components in the assembly have been changed. If so, these components must be saved first. Usualy in the sceen below you will simply click 'Save All'. Then you can be assured that the assembly and all the components have been saved. 
60	We are going to weld the parts together. 1 Click on the arrow below the Assembly Features in the CommandManager 2 Click on the Weld Symbol	

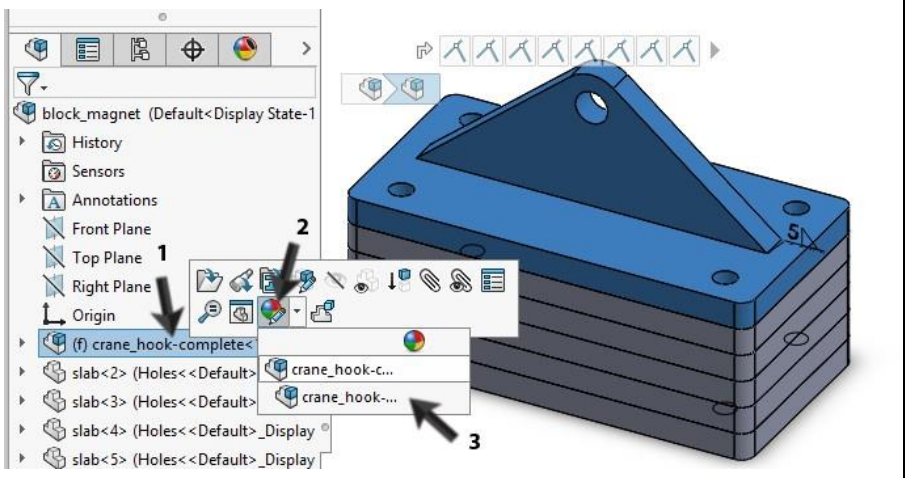
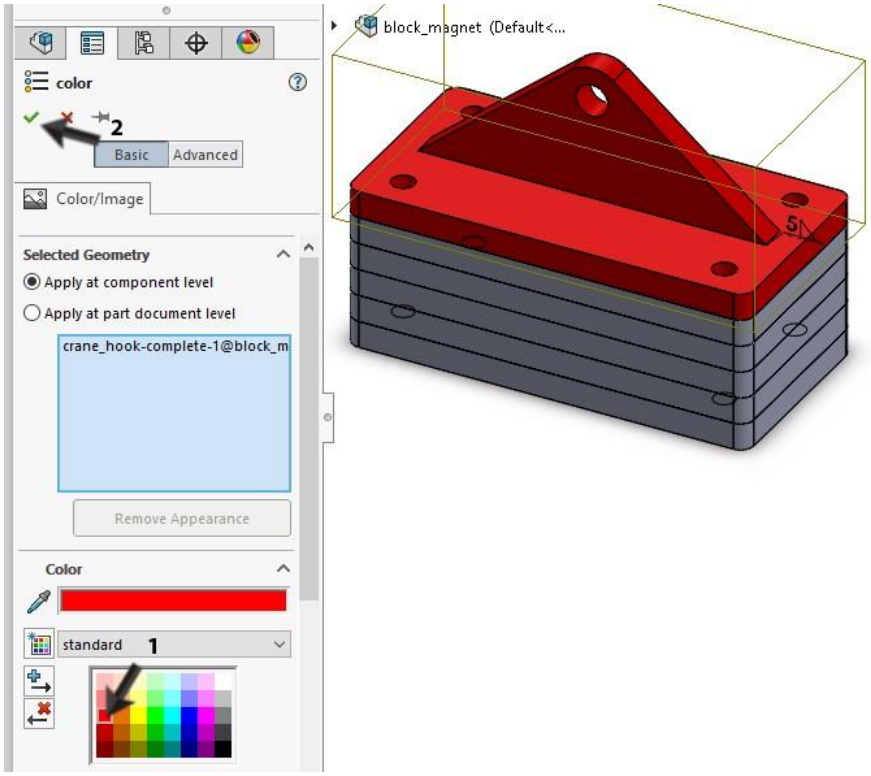
61	1. Select Weld Path 2. Enter 5mm as the dimension of the weld Now we have to select the weld path. 3-4 Select de edges along which have to be welded (as far as you can click on them).	
62	Rotate the model 1 Select the edges you couldn't select before. 2 Click on OK.	
63	The weld now has been made. In the model you will see the weld symbol and in the FeatureManager you will see the 'Weld Folder', in which all welds can be found. You can open and edit the welds there.	

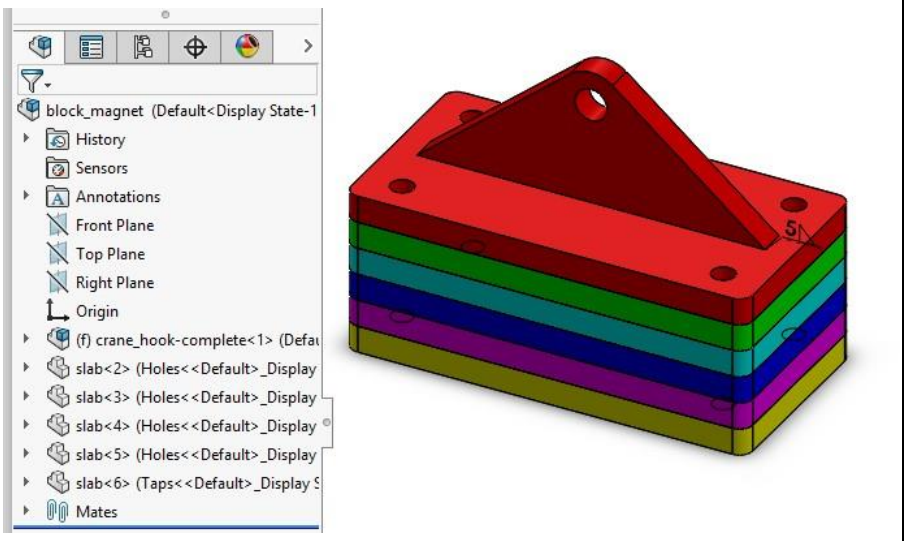
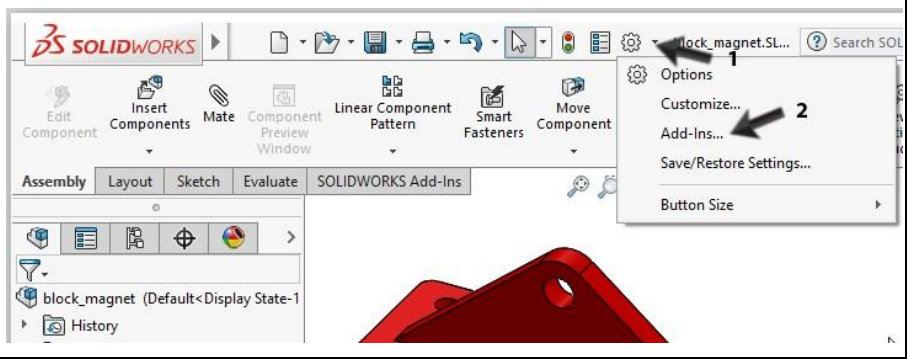
64	If you do see the weld symbol, but not the weld bead itself, you could do the following: 1. Right click on the Weld Folder 2. Click on Show 'Cosmetic Welds'	
65	Save the assembly.	
66	We are going to use the last assembly in the mainassembly. Click on Make Assembly from Part/Assembly in the toolbar.	
67	A new assembly appears in which the last assembly is added directly. Click on OK.	
68	Click on Insert Component in the CommandManager	

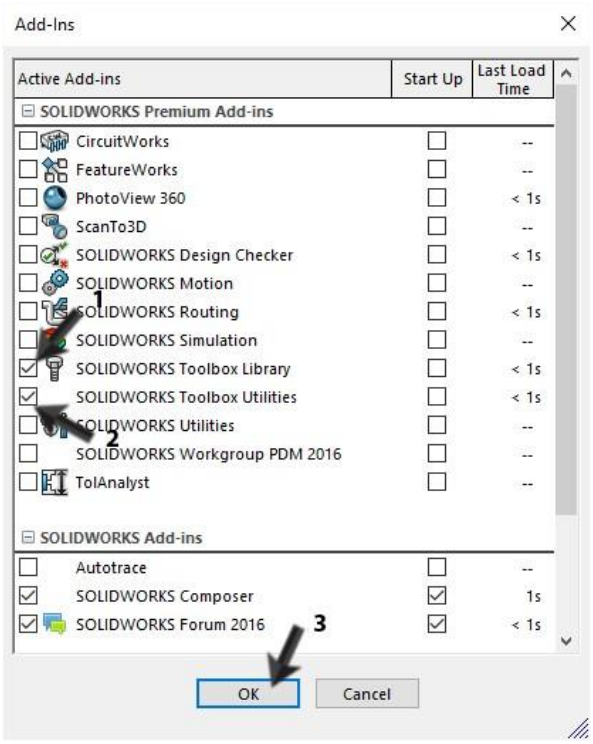
69	Click on Browse in the PropertyManager …	
70	3. Select the file Slab.sldprt, 4. Select the configuration 'Holes'. 5. Click on Open.	
71	Click at a random position to place the plate. Click on OK.	

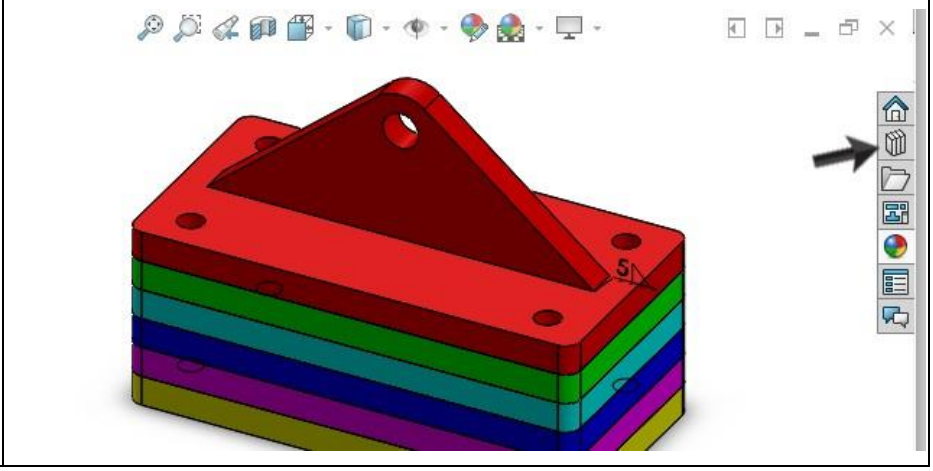
72	Click on Mate in the CommandManager	
73	Select the upper plane of the plate first.	
74	Next rotate the model (by pushing the scroll-wheel of the mouse) and select the bottom plane of the crane hook. Click on OK	

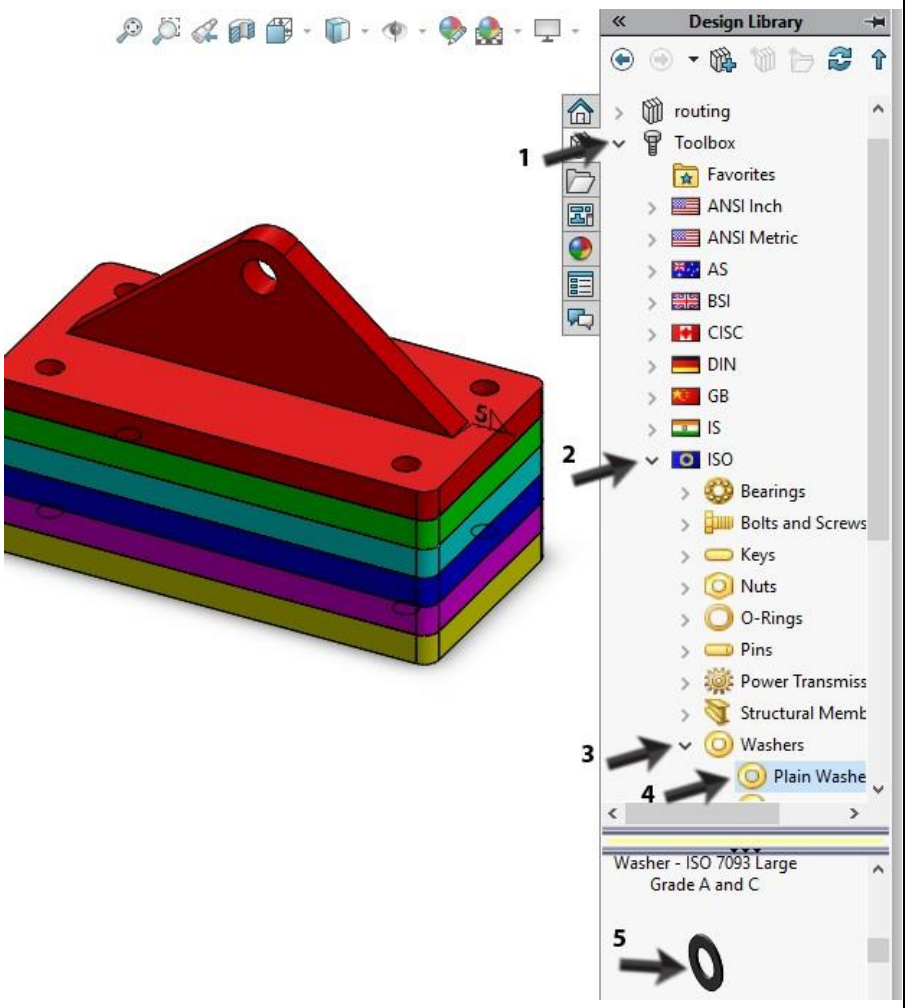
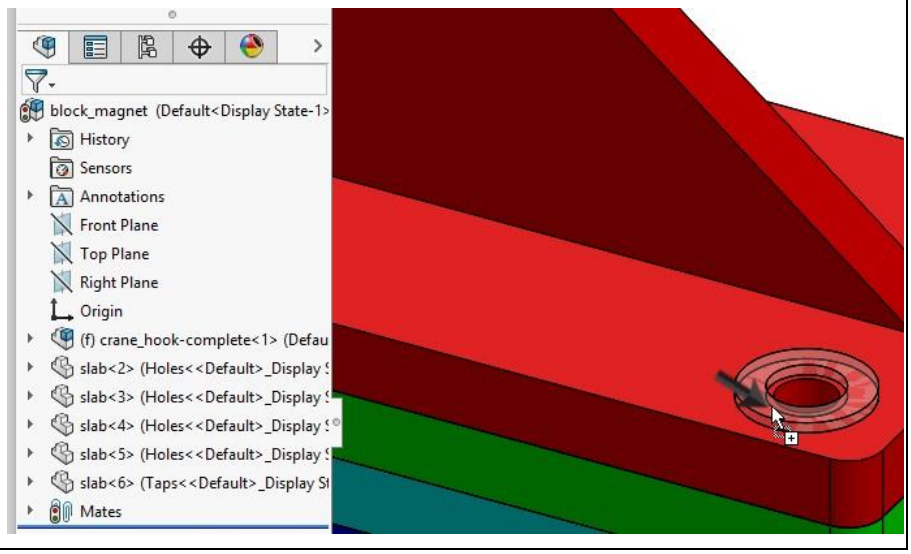
75	To make the next mate you select the long side planes of both parts and click on OK.	
76	To make the final mate you select the short sides planes of both parts and click OK. Click OK again to end the Mate-command	
77	In the same way, add three more similar parts with holes to the assembly. The last part must be a plate with tapped holes. So do exactly the same again, only now you select the configuration 'Taps' when adding this part.	
78	Save the assembly as 'Block_magnet.sldasm'.	

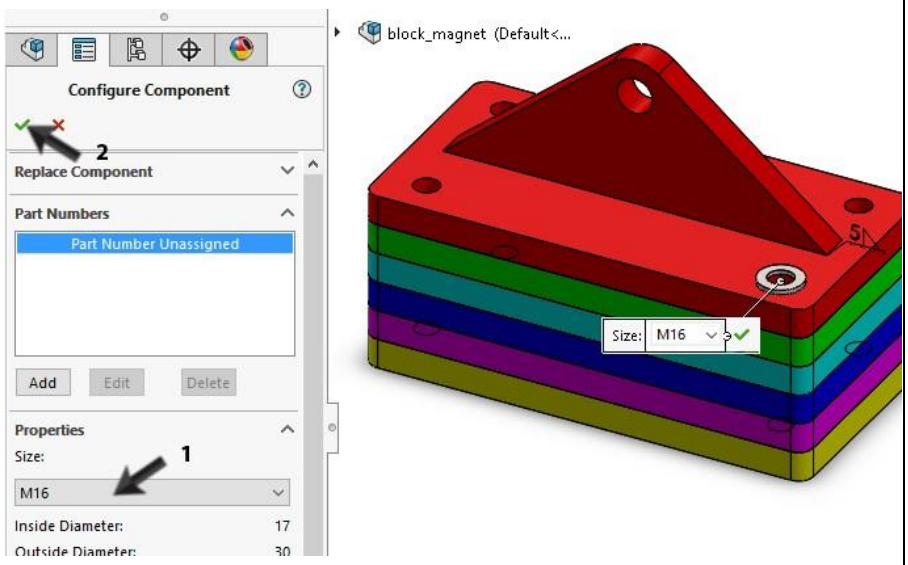
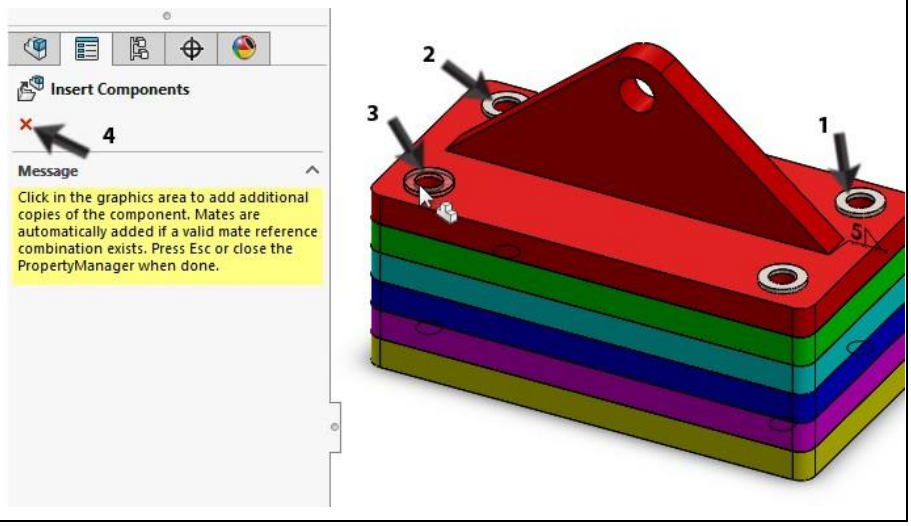
79	Now we will be adding some colors to our model. 1 Click on the first part (Crane_hookcomplete) in the FeatureManager. 2 Click on 'Appearance callout' in the menu that appears. 3 Click on the part in the bottom line.	
80	First click on 'Apply changes at assembly component level' in the Property Manager. Select a color and click OK. The whole part will be colored now.	

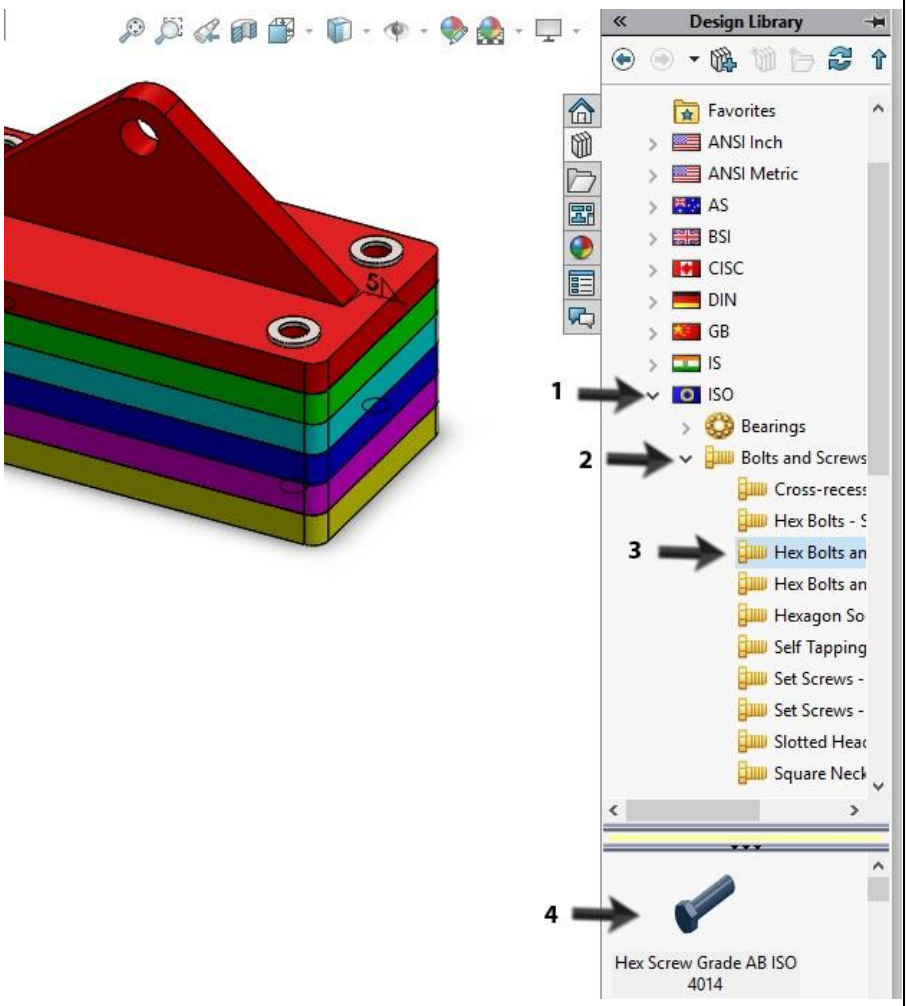
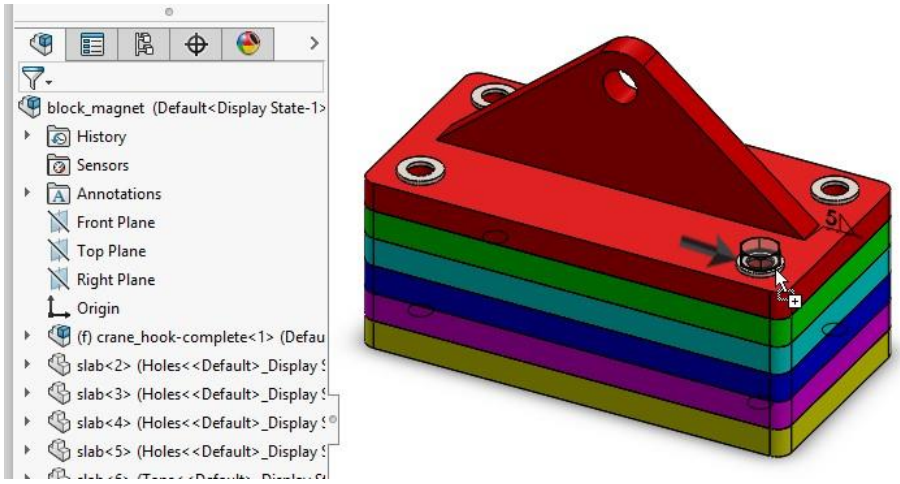
81	Select another color for each part of the magnetic block.	
82	We will be adding some washers and bolts now. We will use a part of SOLIDWORKS which is called Toolbox. Before you can use this, you must first check if Toolbox is already installed AND activated on your computer. Click on Add-Ins… in the CommandManager.	

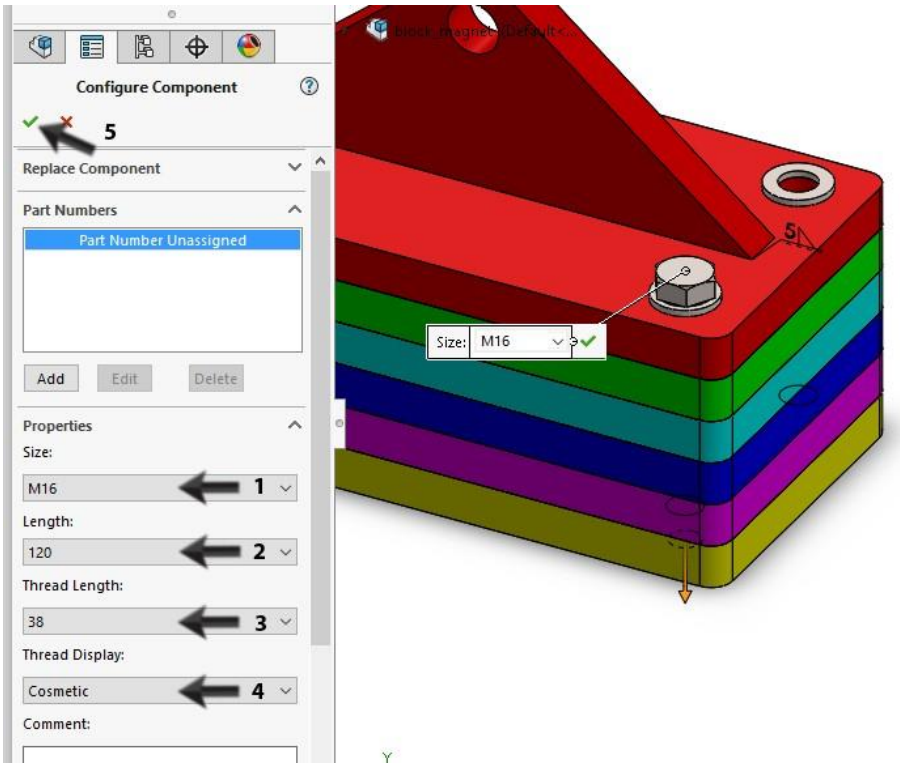
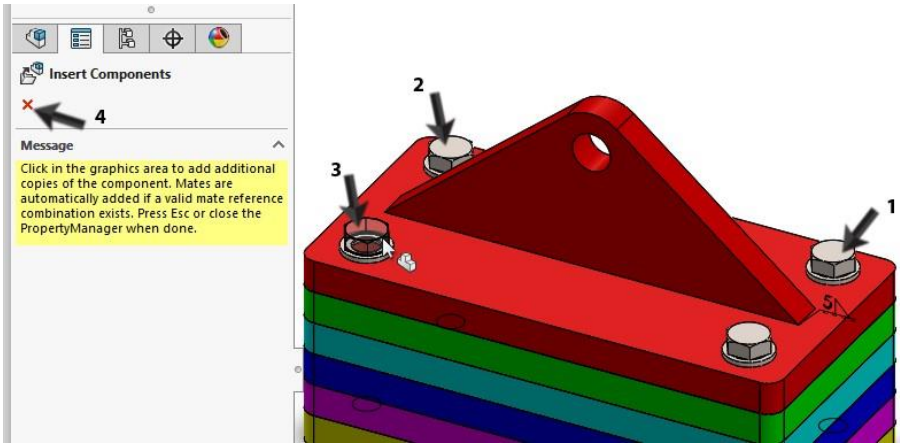
83	Make sure that the options SOLIDWORKS Toolbox and SOLIDWORKS Toolbox Browser both are selected with a 'check' symbol. Are these options not visible or available, read the next tip.	
	Tip!	It may be that you are using a version of SOLIDWORKS in which Toolbox is not available. In that case you can't finish this tutorial. If you still want to finish your model, you can download these parts (e.g. bolts and washers) from www.SOLIDWORKS.nl. You do not use Toolbox, but put the bolts and washers in the assembly like you would do the same with any other part.
	Tip!	In step 83 you could also have checked the two checkboxes in the right column next to SOLIDWORKS Toolbox and SOLIDWORKS Toolbox Browser. If you did, these add-ins will be loaded automatically every time SOLIDWORKS is started. So you do not have to activate the Toolbox over and over again.

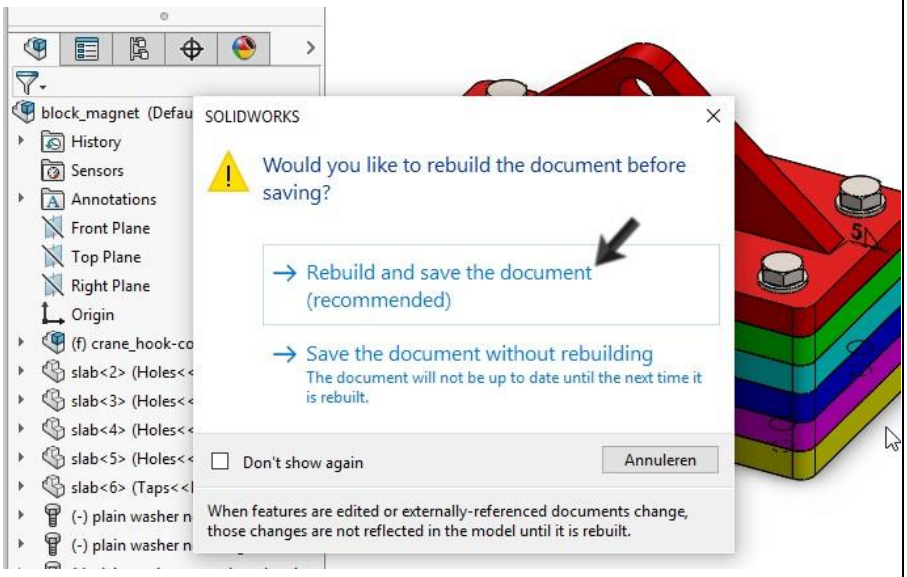
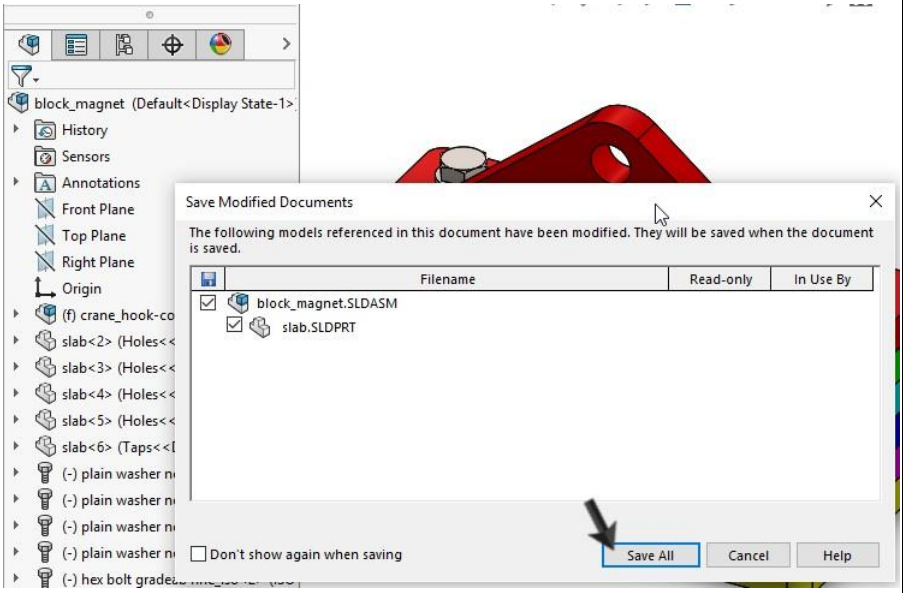
84	Click on the symbol of the Design Library in the Task Pane (at the right of the screen)	

85	The Task Pane unfolds itself and you can see the Toolbox now. We are going to add some washers. Double click following elements one after another: 1. Toolbox 2. ISO 3. Washers 4. Plain Washers In the lower part of the Task Pane the available washers appear. 5. Find the washer: Washer – ISO 7089 Normal Grade A.	
86	Next drag this washer form the Task Pane to your model with the left mouse button. As soon as the washer is above one of the holes, it will find its way to the right position. At that moment, release the mouse button. The size of the washer will be adjusted automatically, but could also be adjusted manually later on.	

87	If necessary, change the size of the washer to M16 in the Property Manager, and click OK.	
88	The ring is now attached to your mouse and you can place it on the other holes. After you have finished placing all the washers, click on Cancel.	

89	Open the Task Pane again and go to: 1 Toolbox 2 ISO 3 Bolts and Screws 4 Hex Bolts and Screws 5 Select this bolt: Hex Screw Grade AB ISO 4014	
90	Again, drag this component to one of the holes. Pay attention: release the mouse button when the cursor is above one of the washers. This is important, because when the cursor is above the plane, the bolt will be positioned TO LOW. (At the surface of the plane and NOT on top of the washer.	

91	In the Property Manager you can set the features of the bolt. 1. Diameter is M16 2. Length of the bolt is 120mm 3. Length of the thread is 38mm 4. The thread is displayed as 'Cosmetic' 5. Click OK.	
92	Now the bolt is attached to the cursor, so you can place it in the other holes too. Pay attention to click on the washer and NOT on of in the hole!	

93	Save the assembly. If the screen appears like in the image, click on Rebuild and save the document.	
94	Is a screen appears like shown here, Click on Save All	
	Which are the main features you have learned in this tutorial?	In this exercise we have executed a lot of new command. • You have created parts from a symmetrical axis • You have use a number of new sketch-tools, like Mirror and Trim • You have used the Hole Wizard to make complicated holes • You have made a welded connection in the assembly • You have colored part • You have used standard parts from the Toolbox. You have reached the next level in SOLIDWORKS, and you have seen some powerful tools.

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