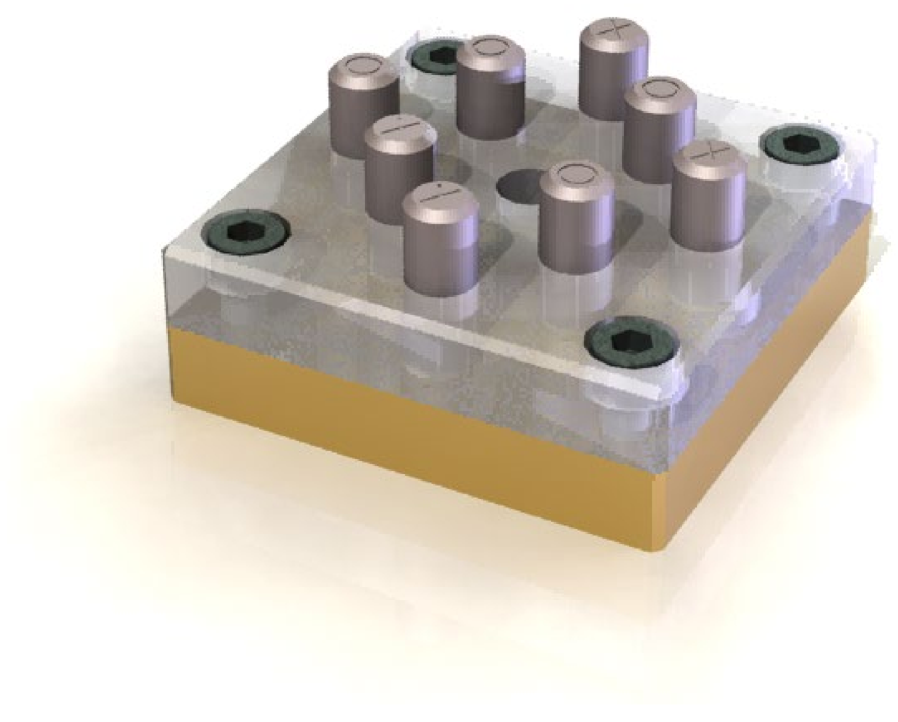
SOLIDWORKS® tutorial 5

TIC-TAC-TOE



Prepatory and Advanced Vocational Training

To be used with SOLIDWORKS® Educational Release 2018-2019

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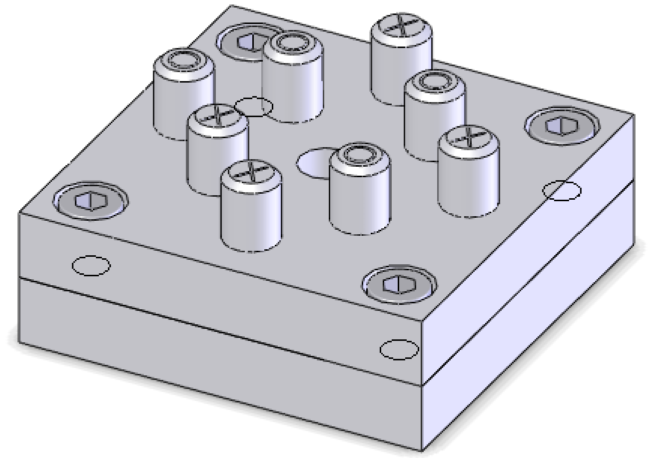
Initiative: Kees Kloosterboer (SOLIDWORKS Benelux)
Educations advisor: Jack van den Broek
Realisation: Arnoud Breedveld (PAZ Computerworks)

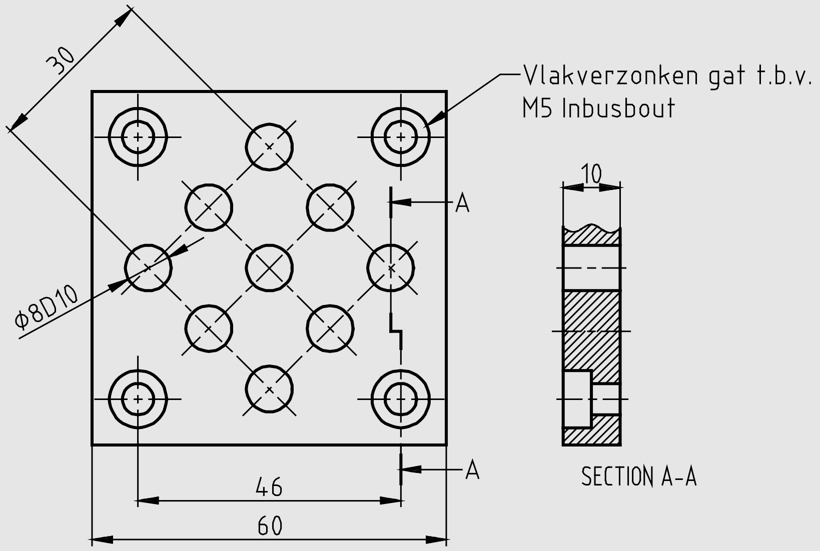
Tic-Tac-Toe

In this tutorial we will create a game called Tic-Tac-Toe. It consists of two plates which are mounted on top of each other. In the top plate there are holes to insert small cylinders marked X or O. In this exercise we repeat a lot of features we already know, amongst others: working with configurations and the use of standard parts. New in this tutorial is that you are going to work with tolerances and fittings and you will be working with patterns.



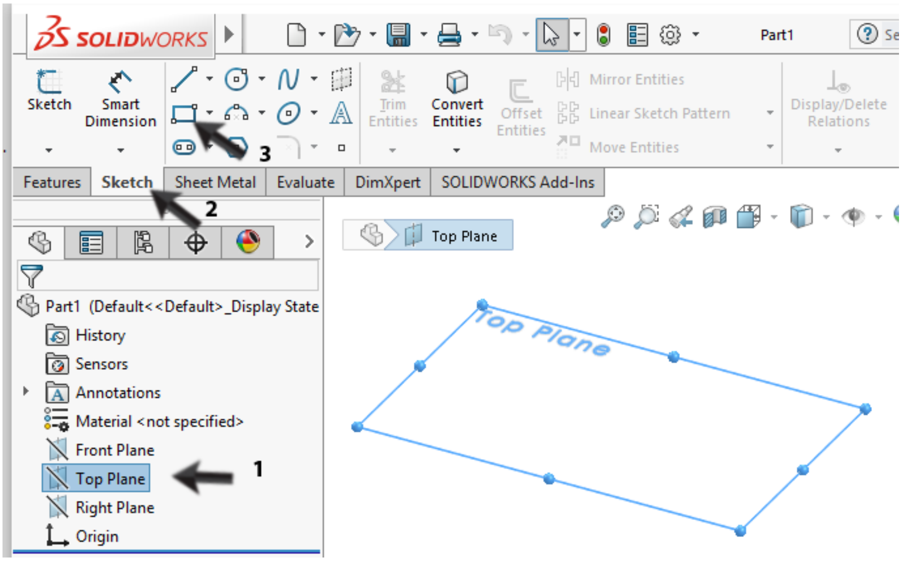
Work plan

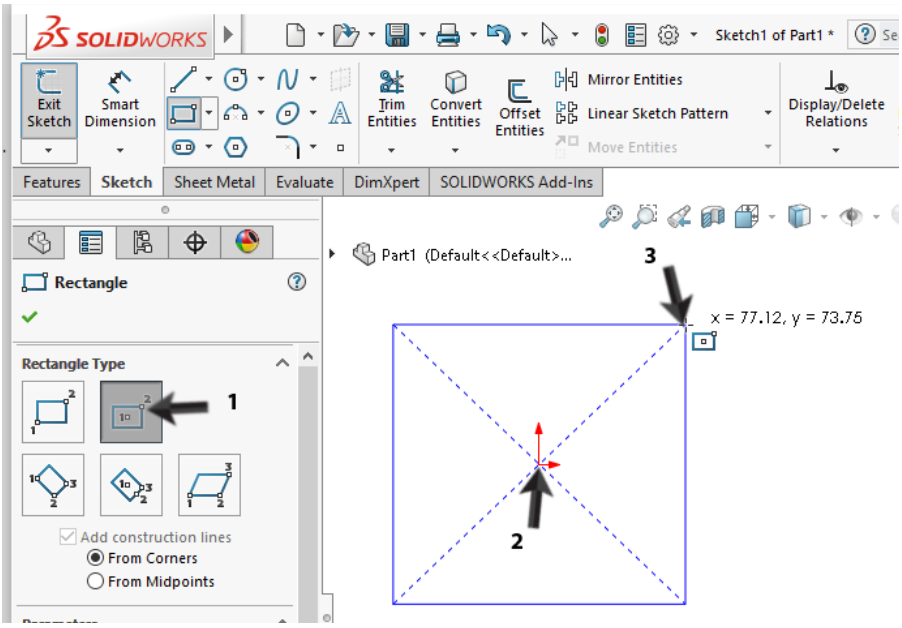
First we will create the top plate. We will model it according to drawing below.



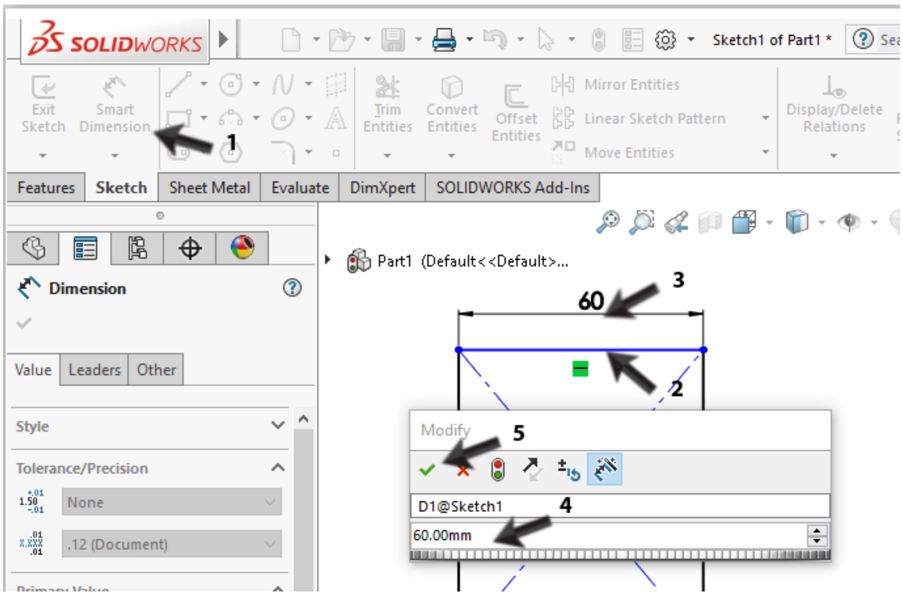
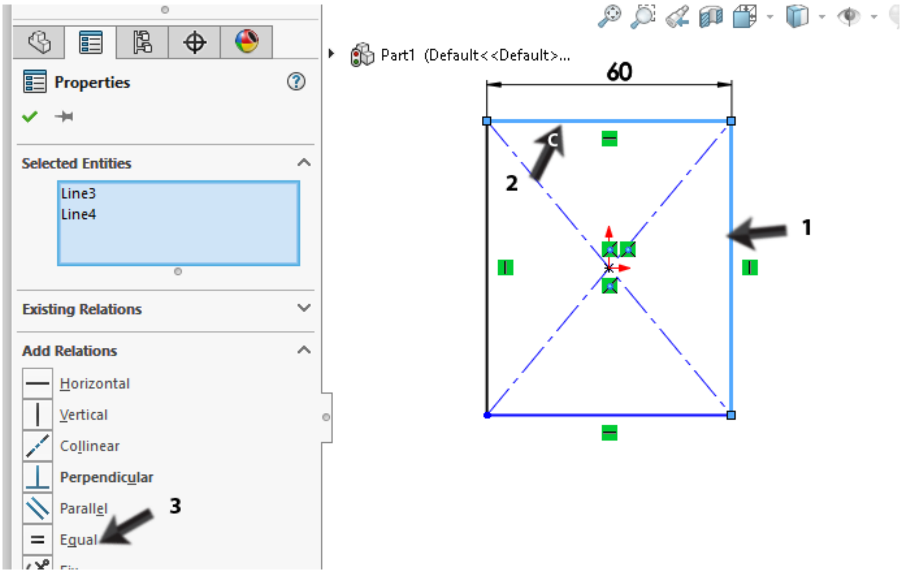
We will execute following steps:

1. First we will create the top plate first with dimensions 60 x 60 x 10.
2. After that we will make four counter bore holes.
3. Finally we will create a pattern of 9 holes.

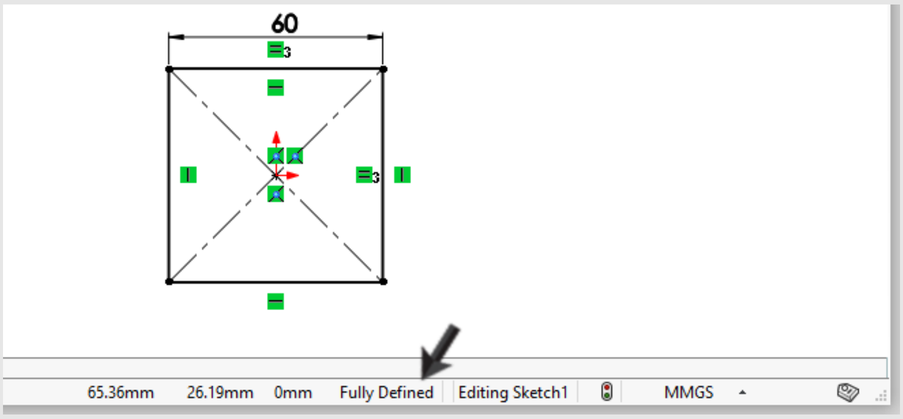
<table>
<tr>
<td>1</td>
<td>Start SOLIDWORKS and open a new part.</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>
1. Select the Top Plane
2. Click on Sketch in the CommandManager
3. Click on Rectangle.
</td>
<td>


</td>
</tr>
<tr>
<td>3</td>
<td>
Draw a rectangle:
1. Click on Center Rectangle in the PropertyManager
2. Click on the origin
3. Click at a random point to get the second corner.
</td>
<td>


</td>
</tr>
</table>

4	Add a horizontal dimension to the sketch, like in the illustration on the right. Change this dimension to 60mm. Push the <esc> key on the keyboard to end the command.	
5	Set the length of the horizontal and vertical lines at the same length: 1. Select a vertical line. 2. Push the <ctrl>-button and click on a horizontal line. 3. Click on Equal in the PropertyManager	
	Tip!	Remember that a blue field in the PropertyManager is a selection field. You can add elements by clicking on them in your model and you can also delete elements from it. (e.g. when you have selected a wrong element) When you see a pink-colored selection field, you don not have to use the ctrl>-key to select more than one element. To remove and element from the list, click on the element in the pink field and push the <del> key on your keyboard. SOLIDWORKS often asks you if you really want to remove the element from the selection field for safety reasons.
	Tip!	The sketch is now fully defined. (Fully defined). You can determine this from the color of the lines in the sketch: - **Blue** means: Sketch is not fully defined - **Black** means: Sketch is fully defined In the status bar at the bottom of the screen you can check is the sketch is

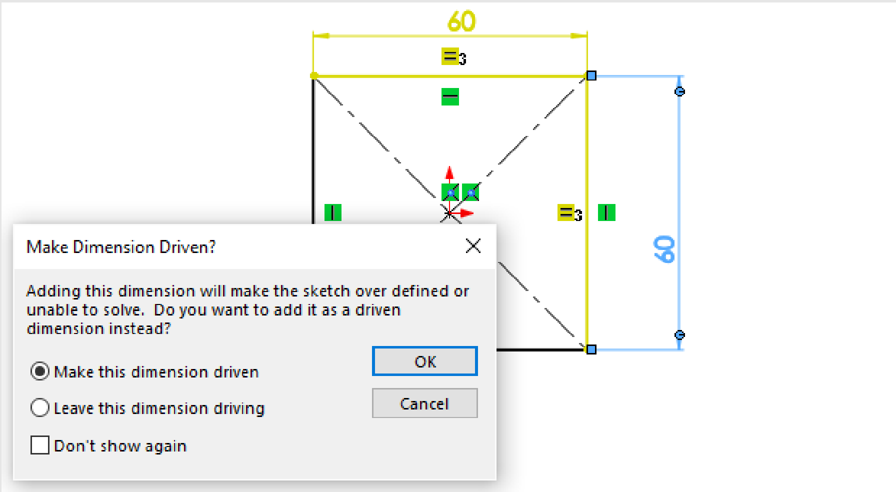
fully defined. In SOLIDWORKS it is not *mandatory* to make a fully defined sketch, but it is a good habit to do this. This can avoid a lot of problems when creating a model later.



Next to Blue and Black a line in a sketch can turn red or yellow.

- **Red** or **Yellow** means: the Sketch is over defined

Try the following: add a dimension of the height of the square. The next message appears:



You have given to much information because:

- The dimension you added says the height is 60mm,
- The relation between the two lines you have created before says the height is equal to the width, which is also 60.

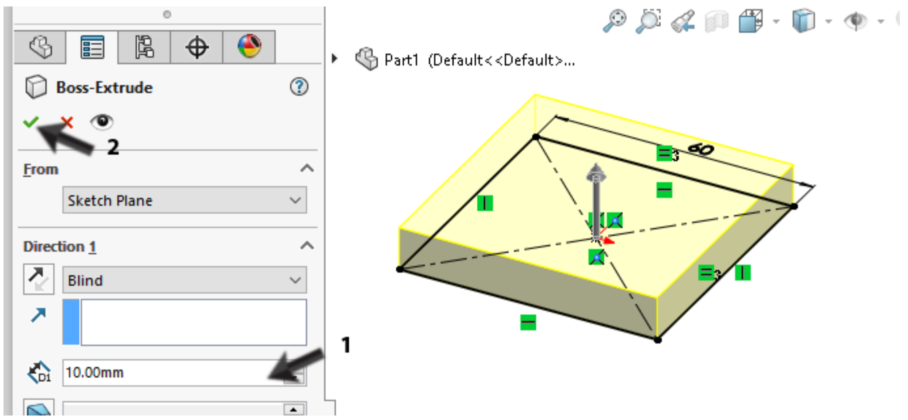
The height is defined twice now and SOLIDWORKS has a problem with that. You must solve this. In the menu which is shown above the best thing to do is choose Cancel. The dimension will not be added to the sketch then.

Did you make an over defined sketch anyway, then throw away (delete) dimensions and/or relations, just as long as the sketch is no longer over de-fined.

6

Click on Features in the CommandManager, next on Extruded Boss/Base.

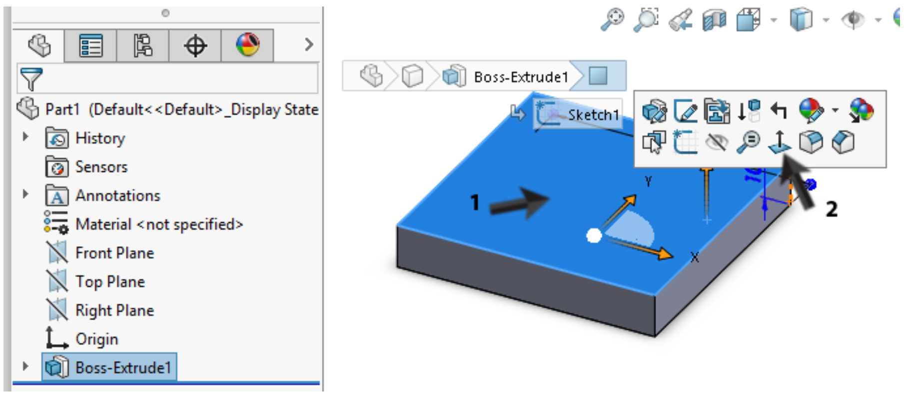
1. Set the thickness of the plate to 10 mm.
2. Click OK.



7

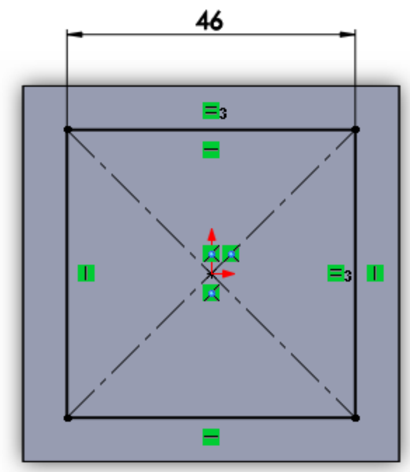
Next we will make a sketch in which we determine the exact position of the holes:

1. Select the top plane of the plate
2. Click on de View Orientation
3. Click on Normal To



8

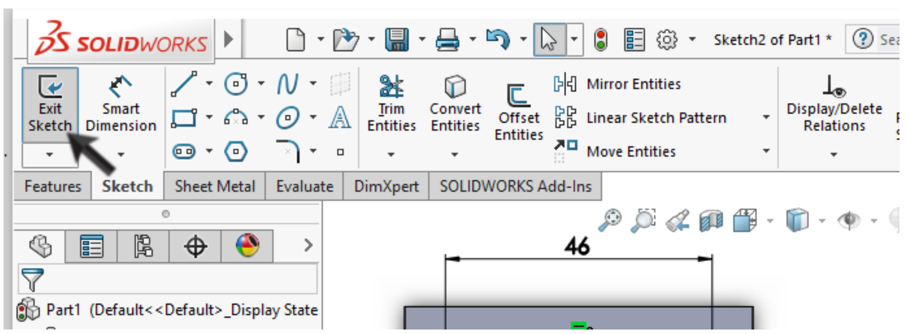
Draw another rectangle with a dimension of 46 mm. Follow the steps 3 to 5 again if you need help.

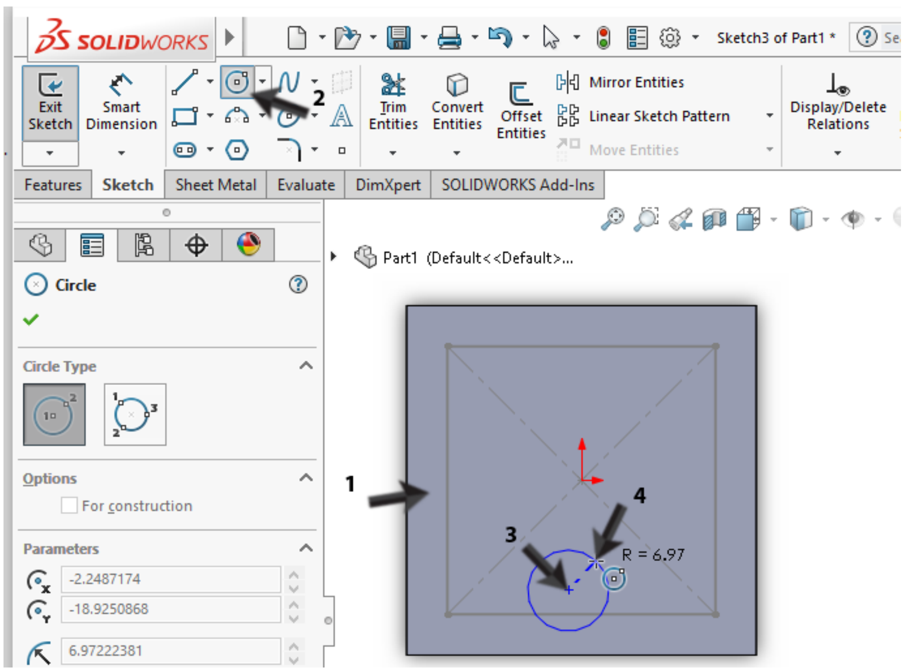
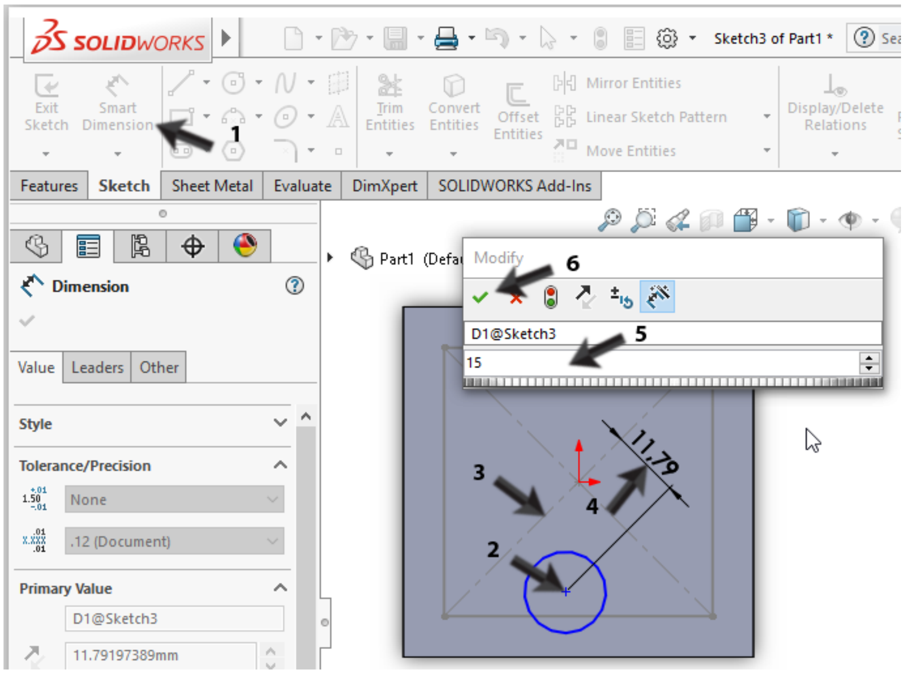


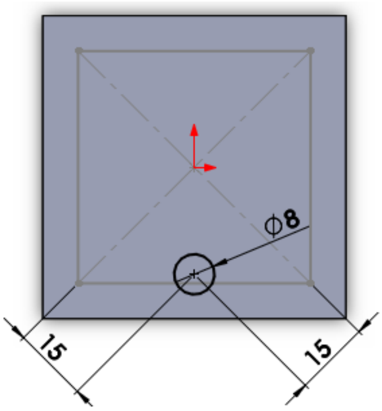
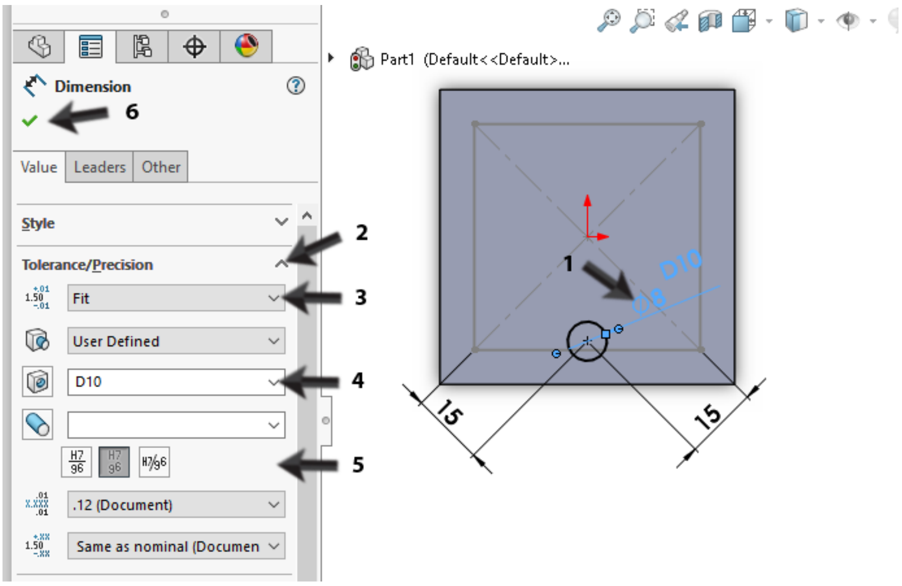
9

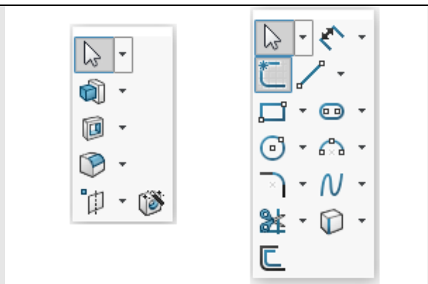
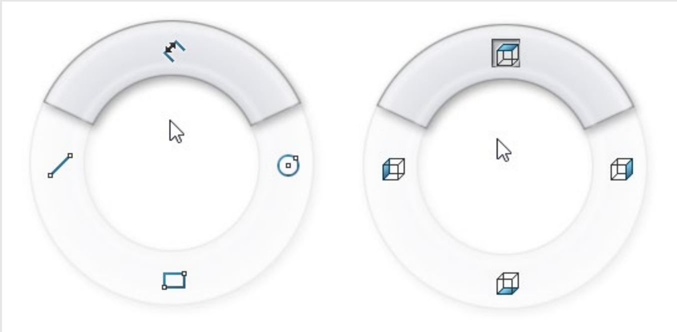
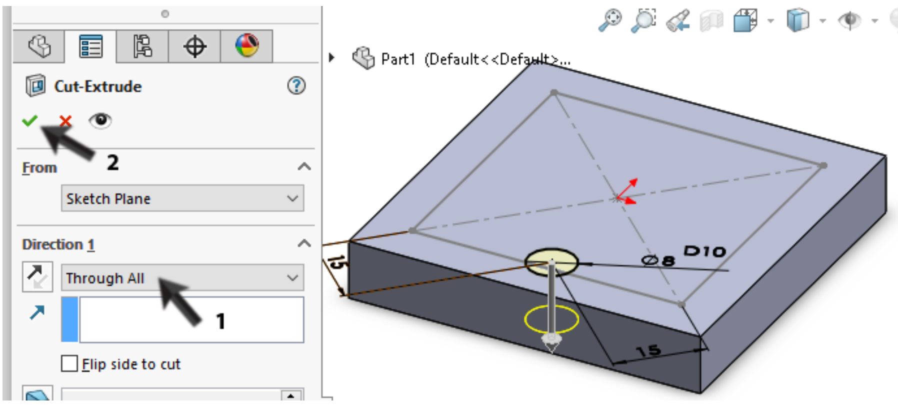
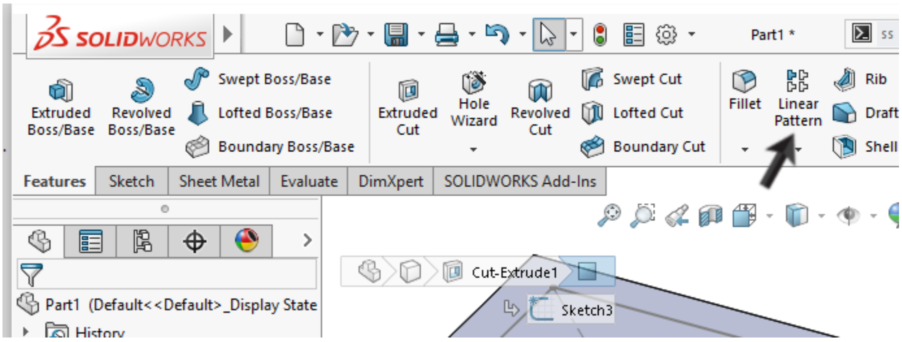
Click on Exit Sketch in the CommandManager.

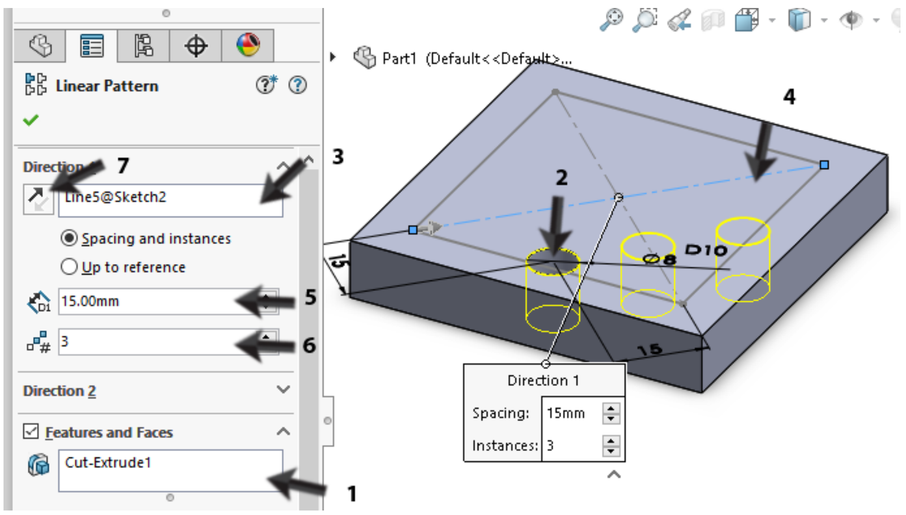
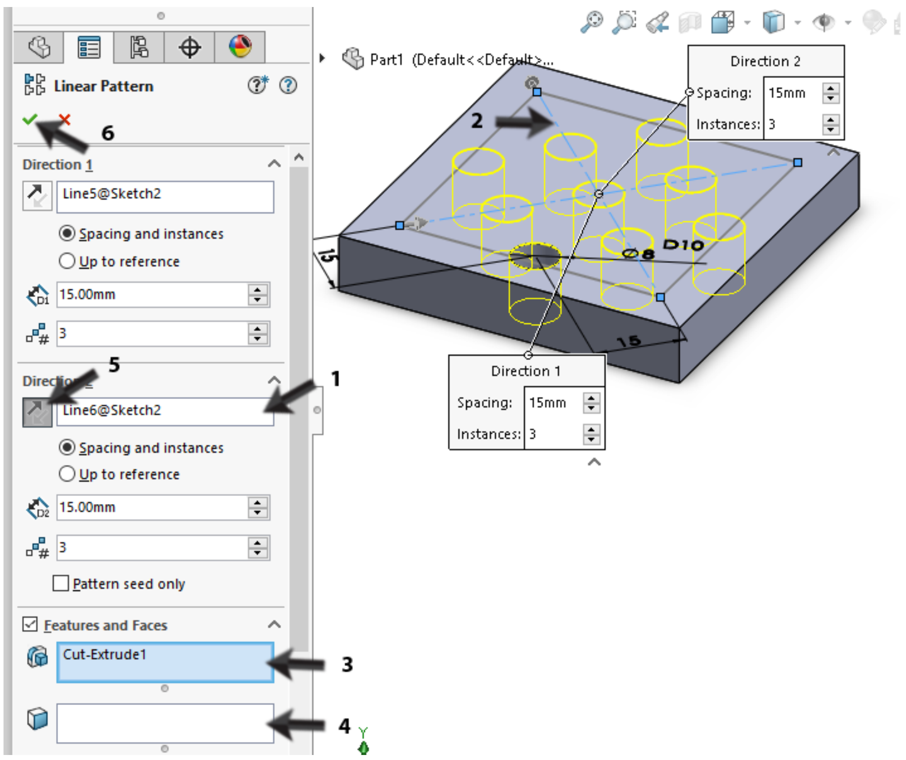
We will not use this sketch to make a feature.



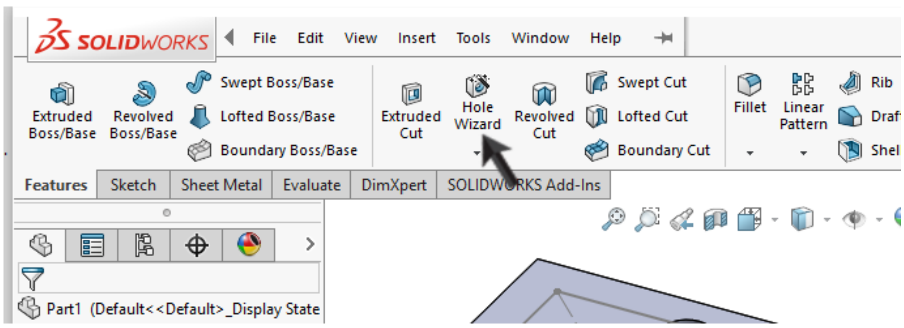
10	Start up a new sketch. 1. Select the top plane again. 2. Click on Circle in the CommandManager. 3,4 Draw a circle like the one in the illustration.	
11	Add a dimension between the circle and one of the diagonal lines which you have drawn before: 1. Click on Smart Dimensions in the CommandManager. 2. Click on the centre of the circle. 3. Click on the diagonal line. 4. Place the dimension. 5. Change it to 15mm. 6. Click OK.	

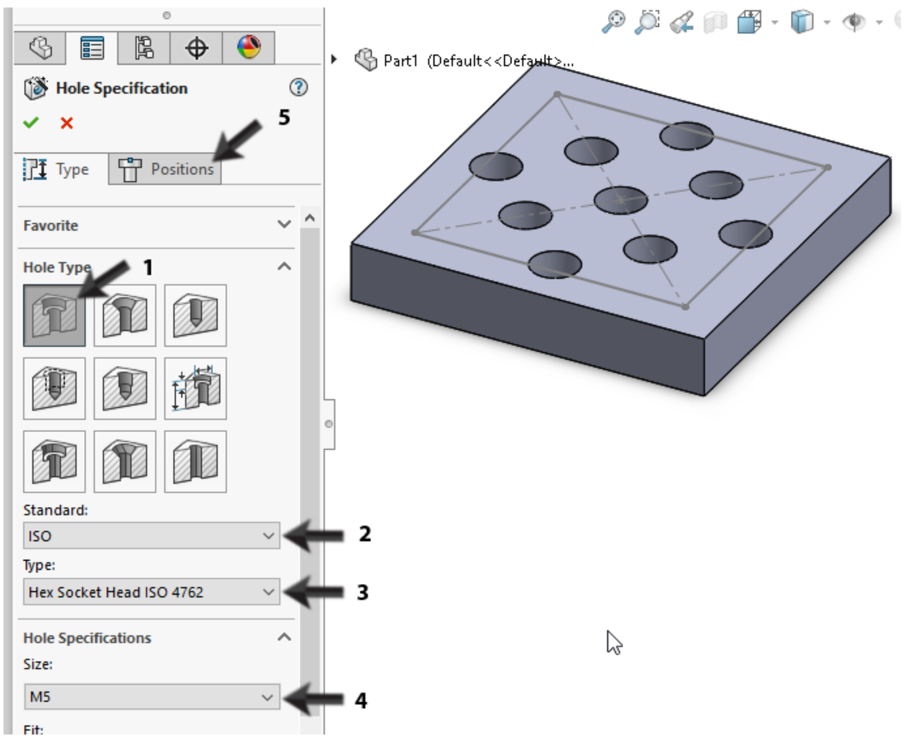
12	Next add the dimension to the other diagonal line (15mm) and the diameter of the circle (Ø8mm). Push <Esc> to close the Smart Dimension command.	
13	To set an exact fitting to the hole (Ø8), follow the next steps: 1. Select a dimension (it turns blue) 2. Be sure that the area called Tolerance/Precision is visible in the PropertyManager. Click on the arrows to reveal it. 3. Set Tolerance type to Fit 4. Select a fitting of D10 in the Hole Fit field. 5. Click on linear Display so that the tolerance will be placed directly after the dimension. 6. Click OK.	
	Tip!	In this and the next tutorials we will be picking the commands from the CommandManager. Now that you are getting used working with SOLIDWORKS, you might find it more convenient to use the quick menu. This quick menu can be activated by pressing the 'S' on the keyboard. The most important and mostly used commands will appear. The menu is context driven: if you are working in a sketch, the sketch commands will be shown, otherwise you will see the feature commands.

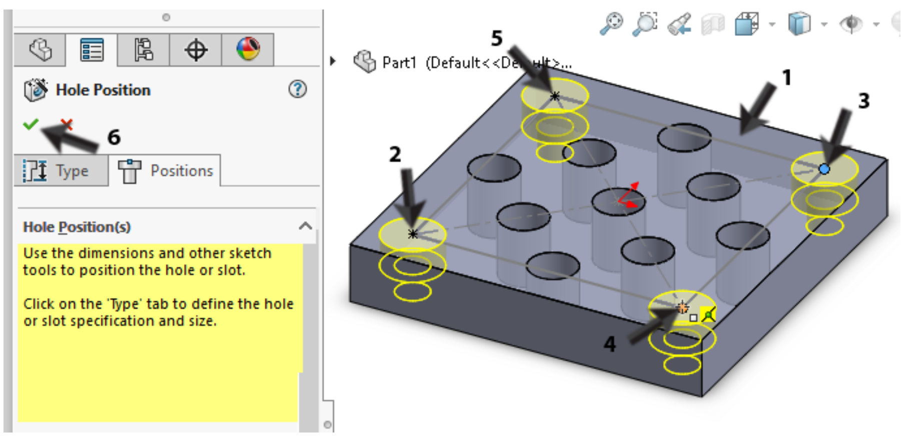
		 Another way to quickly select commands, is by mouse gestures: 1. Click and hold the right mouse button. 2. Now move the mouse a little (with the right button still pressed) A circle will appear around the cursor, with 4 commonly used commands. Again, the menu is context driven. 3. Move the cursor over the desired command out of the circle, and release the mouse button. The selected command is activated.  Once you're used to mouse gestures, it's a very quick way to select commands, especially when you are working in a sketch.
14	Make a hole in this sketch: click on Features in the CommandManager and next on Extruded Cut. Set the depth of the hole in the PropertyManager to Through all and Click OK.	
15	Now we will create the hole pattern. Click on Linear pattern in the CommandManager	

16	Next set following features: 1. Activate the selection field under 'Features and Faces'. 2. Select the hole we creates in the previous steps 3. Activate the selection field at 'Direction 1' 4. Select one of the diagonal lines. 5. Set the distance between the copies to 15mm 6. Set the number of copies to 3. 7. When the copies are place in at the wrong side, click on Reverse Direction.	
17	Repeat these steps in the area named Direction 2. For this purpose, select the other diagonal line. If the preview looks good to you, click OK.	

<table>
<tr>
<td>18</td>
<td>We will now create the mounting holes for the bolts.

Click on Hole Wizard in the CommandManager.</td>
<td>
</td>
</tr>
<tr>
<td>19</td>
<td>Set the following features in the PropertyManager:

1. Select the hole type Counter bore.
2. Set the Standard: ISO.
3. Set Type: Hex Socket Head ISO 4762.
4. Set Size: M5
5. Click on the Positions tab.</td>
<td>
</td>
</tr>
<tr>
<td>20</td>
<td>First, select the plane on which the holes must be placed. Next click at the four corners of the sketch to position the holes.

Click OK.</td>
<td>
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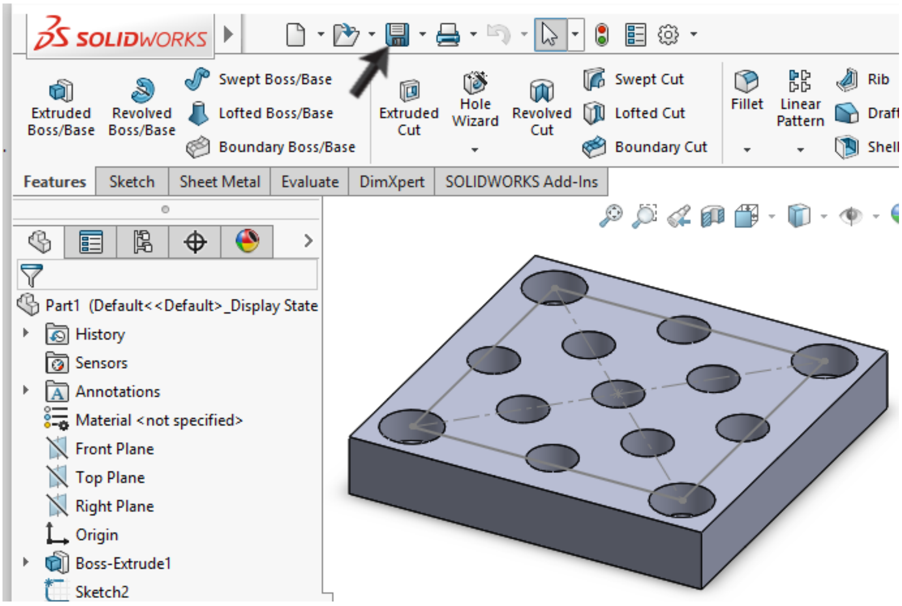
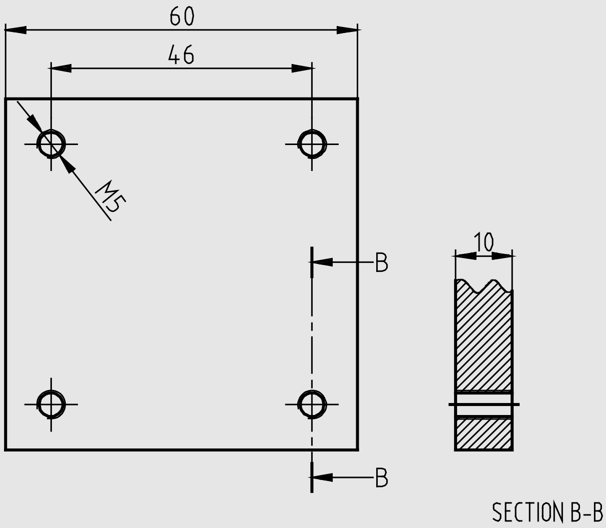
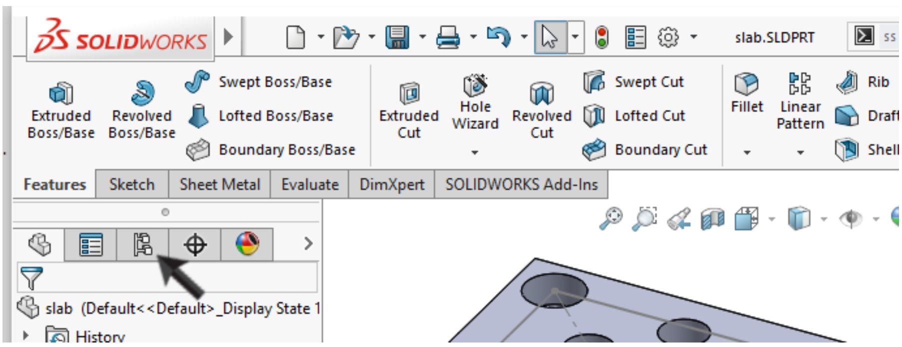
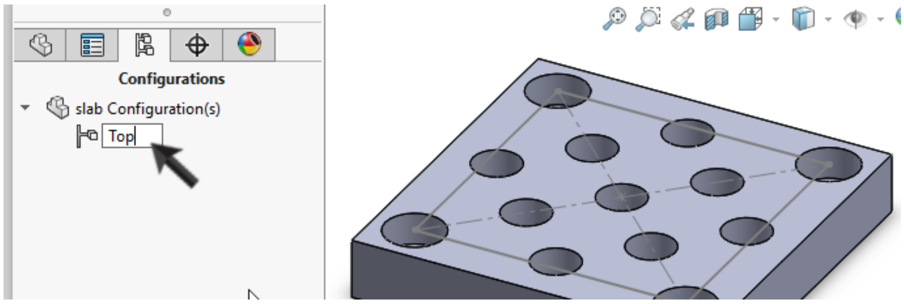
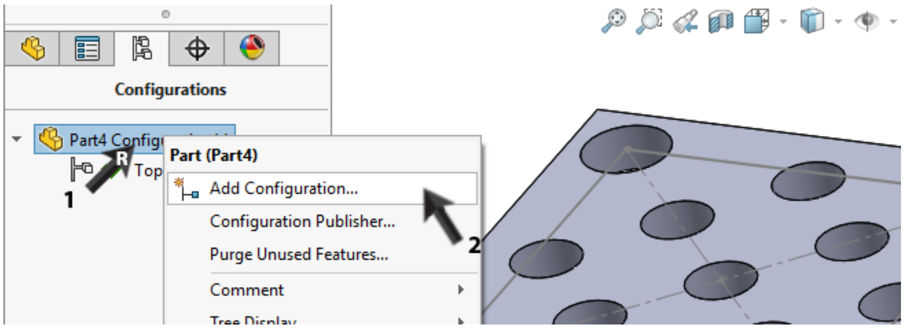
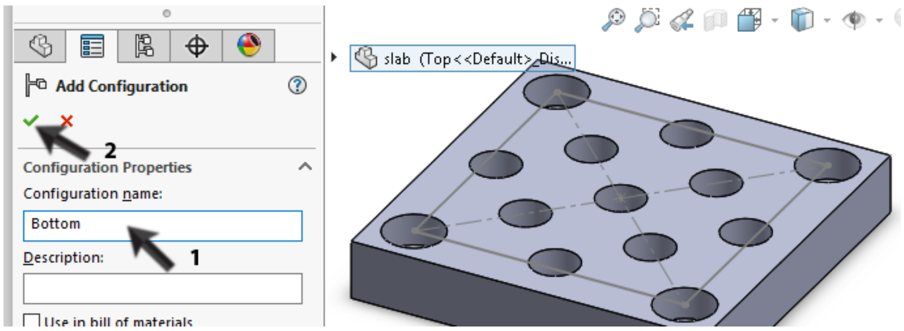
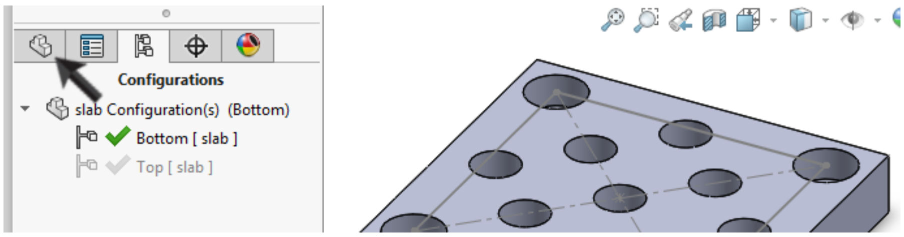
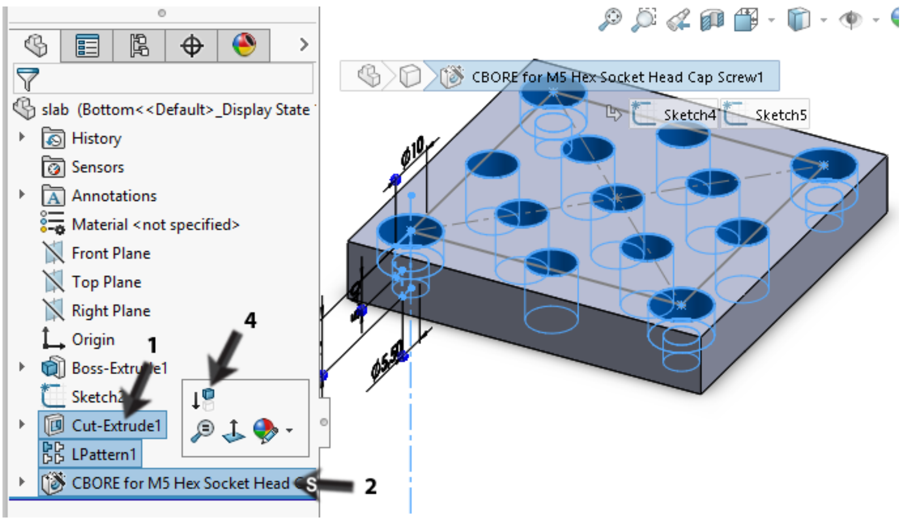
21	The first part, the top plate, is ready now. Save this file as: Slab.sldprt Tip: make a new folder in your computer first. You can arrange all the files by product.	
	Work plan	We will now create the second part, the bottom plate. We will do this in accordance with the drawing below.  Notice that this part looks very much like the first one. The perimeter dimensions and the position of the mounting holes are the same. That is why we will create a configuration from the first part to get the second one.
22	Click on the Configuration-Manager tab	

23	The name of the configuration is 'Default'. Double-click on this name to change it to 'Top'.	
24	1. Click your right mouse button at the upper line in the ConfigurationManager. 2. Select Add Configuration in the menu.	
25	1. Set the name of the new configuration to: Bottom 2. Click OK.	
26	In the list there are two configurations now: Top (grey, non-active), and Bottom (Black, active). We work in the active configuration. Click on the FeatureManager tab.	

<table>
<tr>
<td>27</td>
<td>Now Suppress the last three features which you made just before:
1. Click on the Feature Extrude2.
2. Hold the Shift-key on the keyboard and click on the last feature.
3. Release the Shift-key, the last three features are selected now and a small options menu appears.
4. Select: Suppress in the menu.
All holes have disappeared from the model.</td>
<td>
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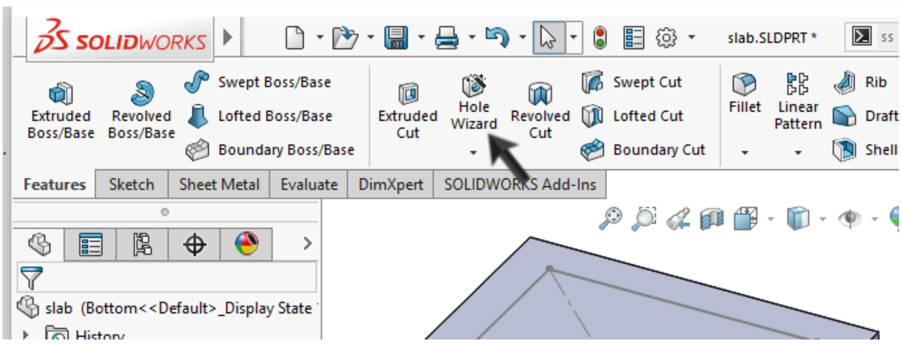
<tr>
<td>28</td>
<td>Next we will make some tapped holes with M5 thread.
Click on the Hole Wizard in the CommandManager.</td>
<td>
</td>
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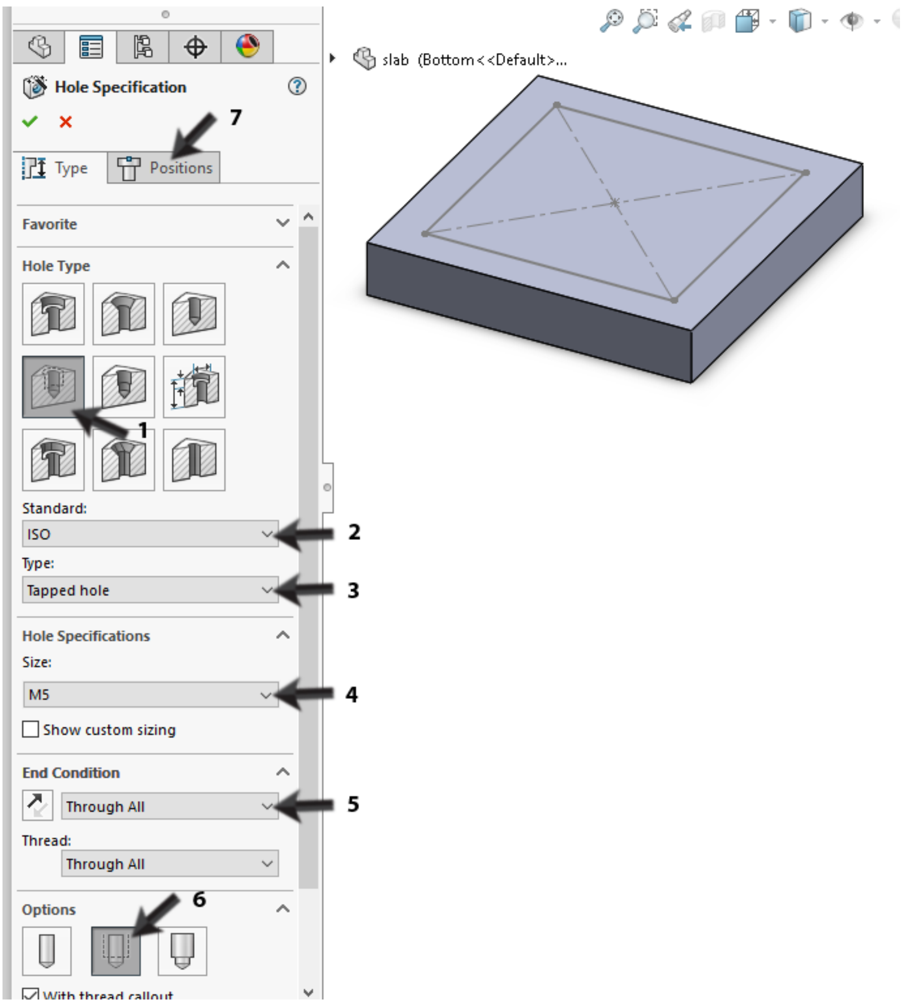
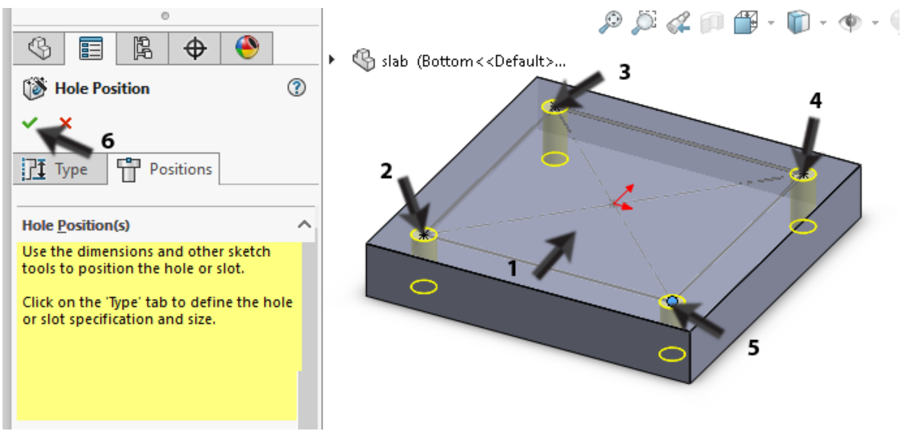
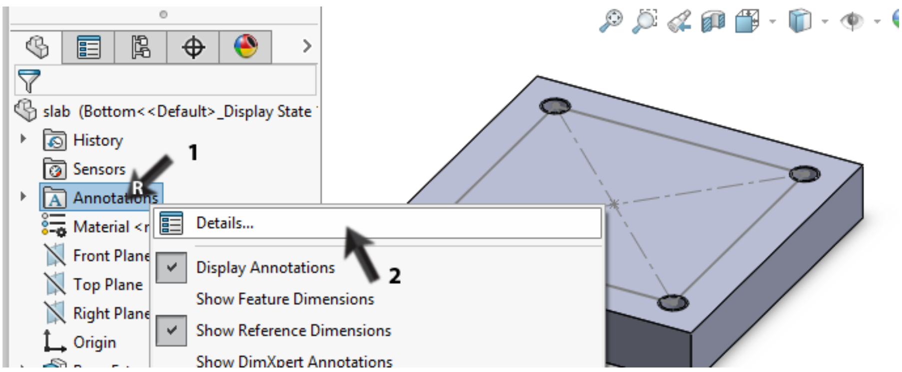
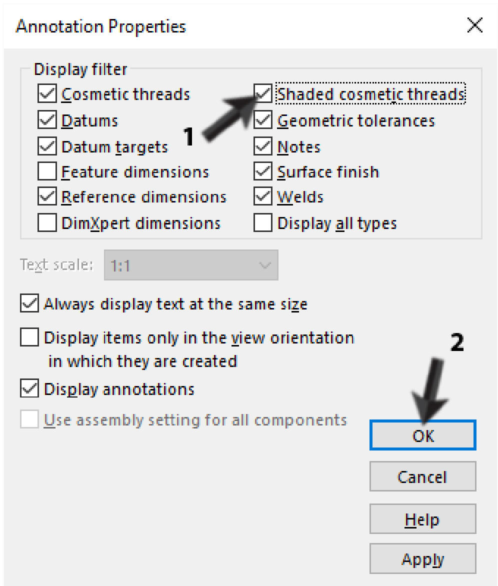
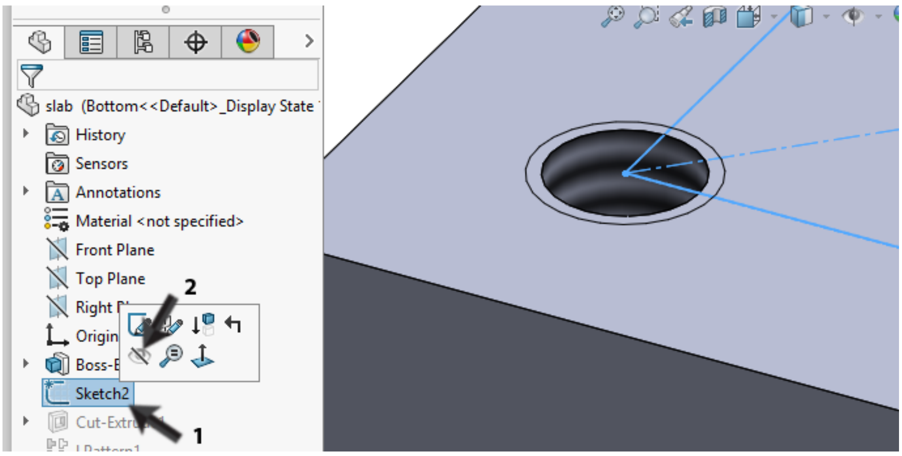
29	Select the hole type Tap in the PropertyManager. Make sure all settings are equal to the settings in the illustration at the right. Click on the Positions tab.	
30	First select the plane where the holes will be placed, then click on the four corners of the sketch to position the holes. Click OK.	

<table>
<tr>
<td>31</td>
<td>When no thread is visible in the holes, then change the next settings:
1. Click the right mouse button on Annotations in the FeatureManager
2. Select Details.</td>
<td>
</td>
</tr>
<tr>
<td>32</td>
<td>1. Make sure that the option Shaded Cosmetic Threads is checked.
2. Click OK.</td>
<td>
</td>
</tr>
<tr>
<td>33</td>
<td>Next we want to hide the sketch we have used to make the holes:
1. Click with the right mouse button on the sketch in the FeatureManager.
2. Select Hide in the menu.</td>
<td>
</td>
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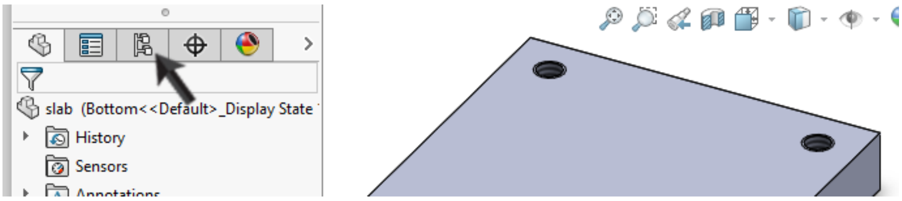
<tr>
<td>34</td>
<td>Re-activate the configuration of the top plate.
Click on the ConfigurationManager tab</td>
<td>
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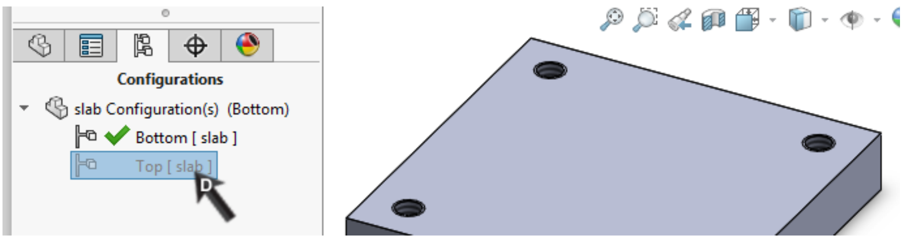
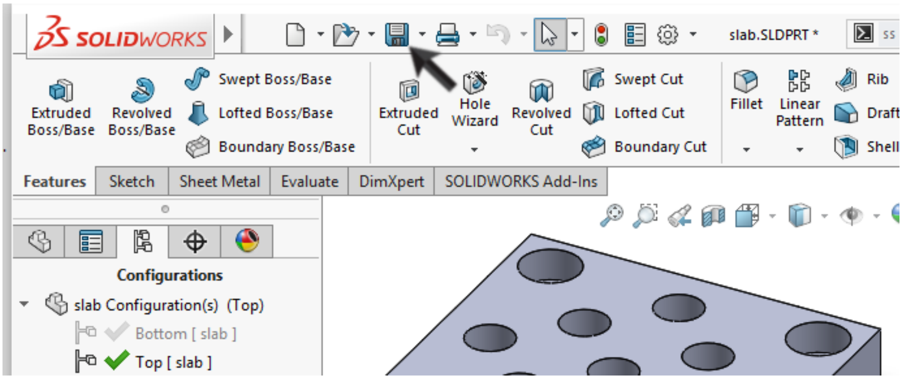
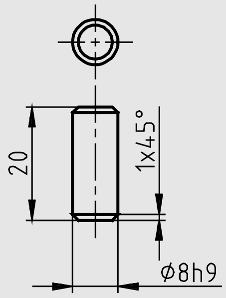
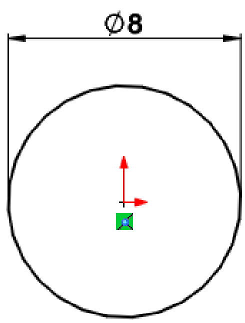
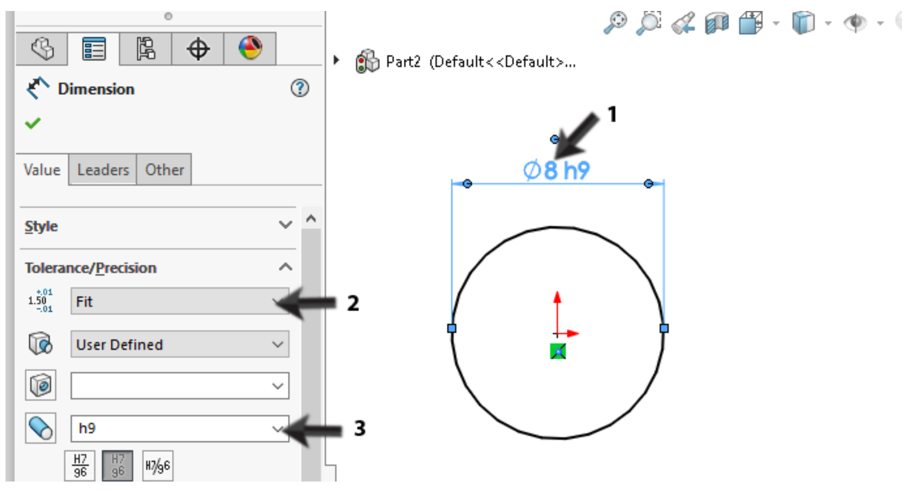
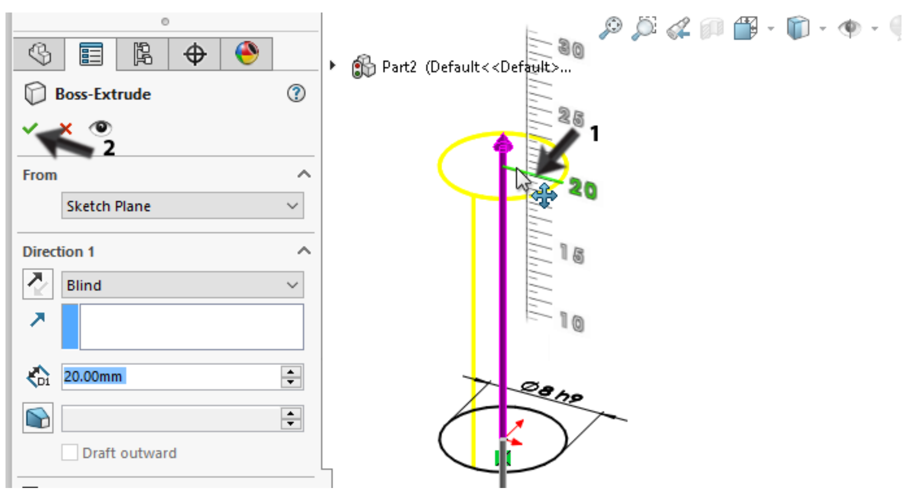
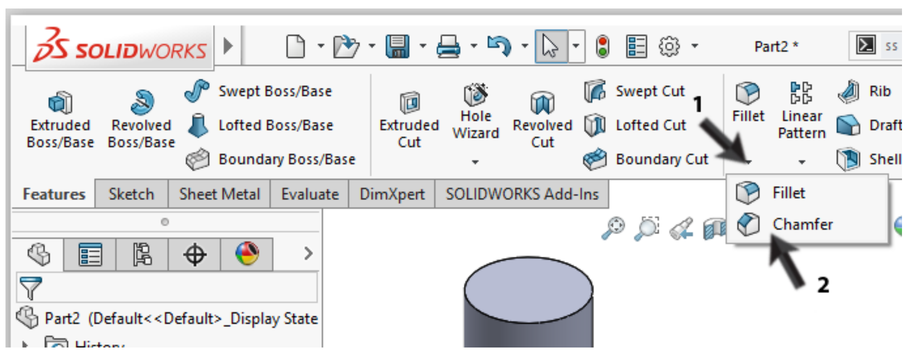
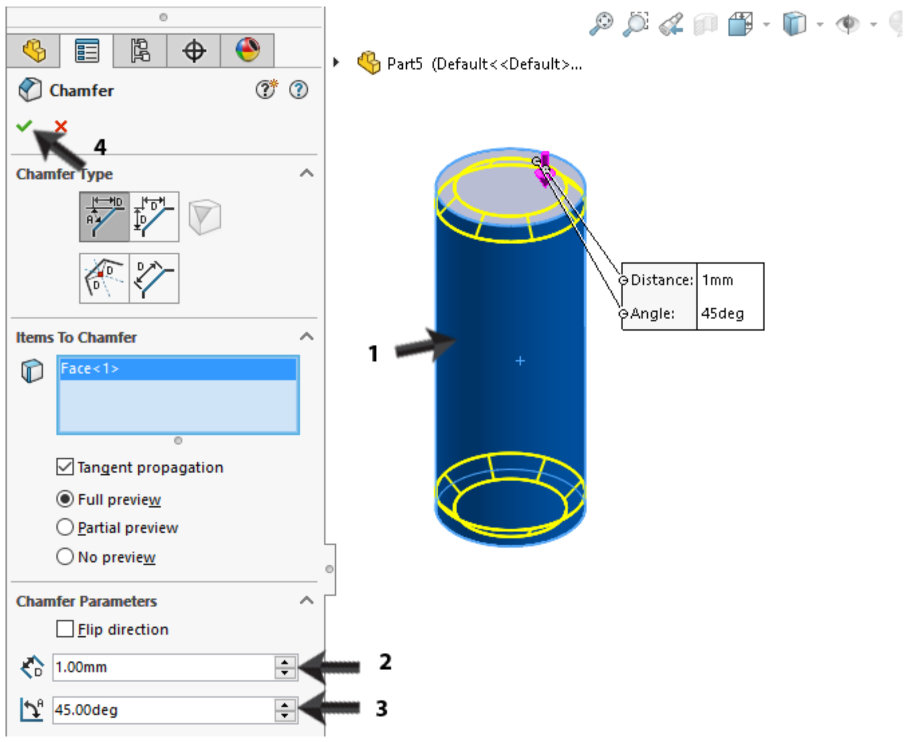
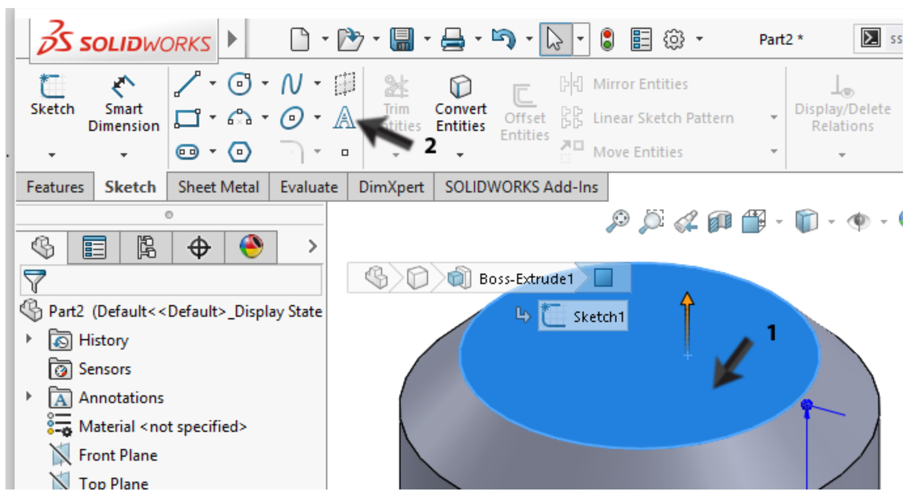
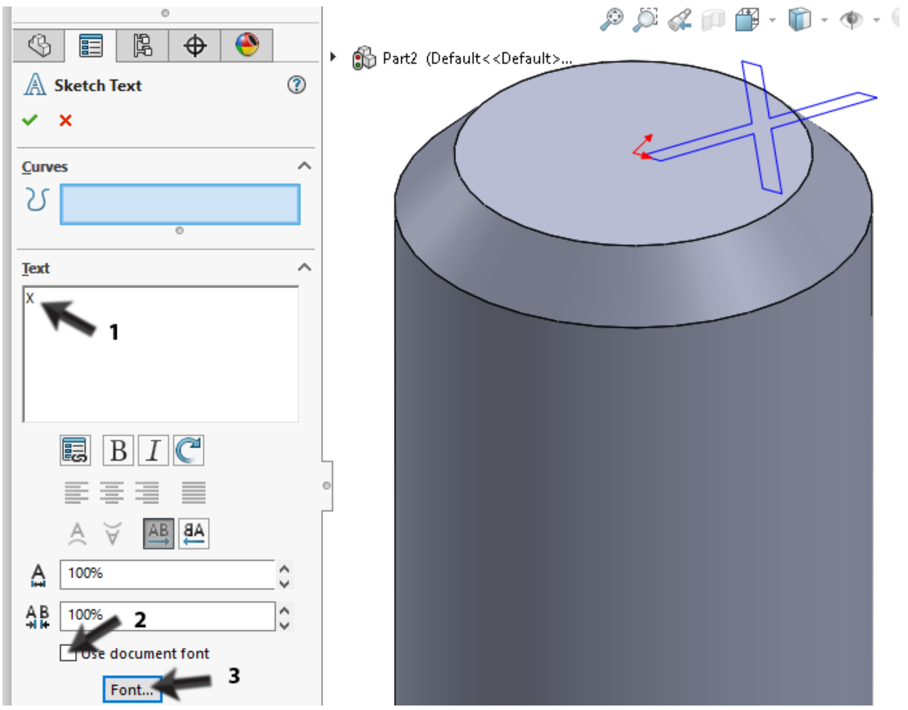
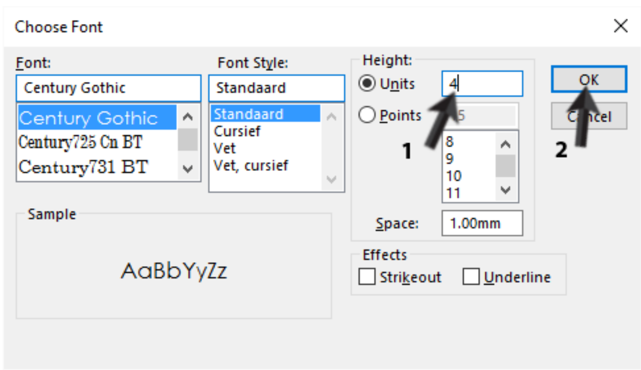
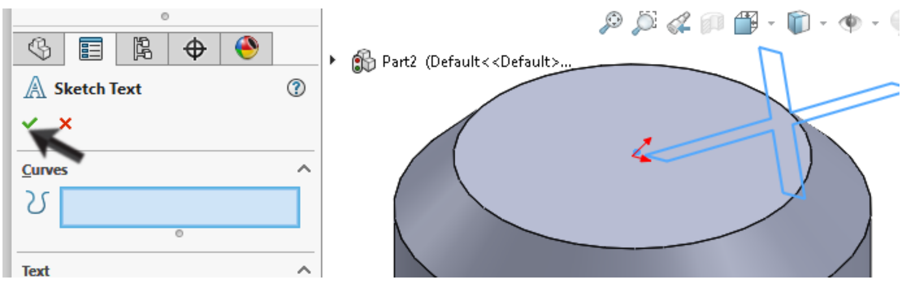
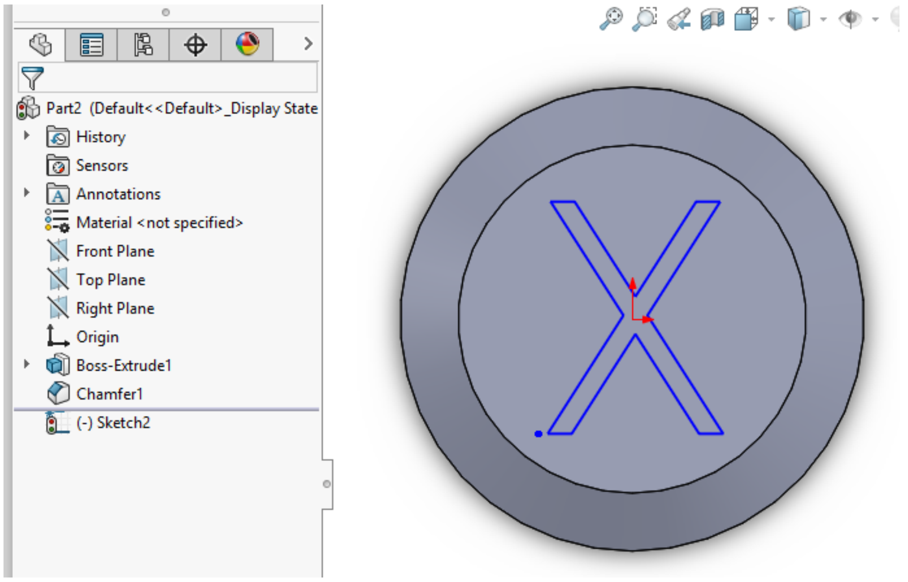
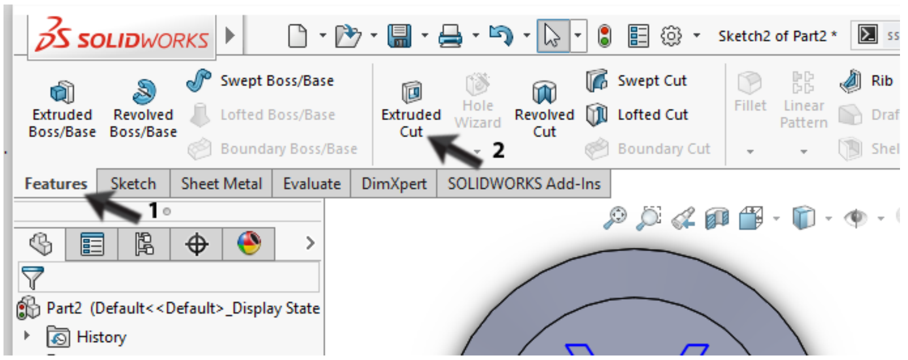
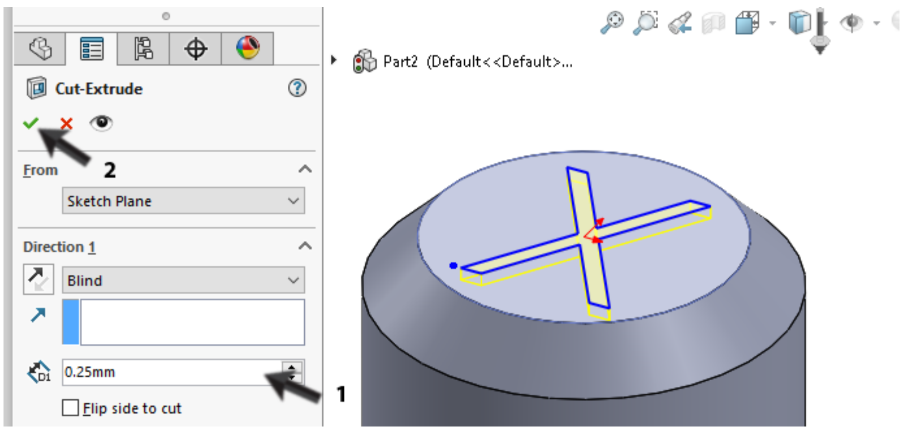
35	Double-click on the configuration 'Top' in the ConfigurationManager.	
36	Save the file.	
	Work plan	The third part is the 'cylinder'. We will create this by using the dimensions of the drawing below.  To be able to play Tic-Tac-Toe we need to insert an X or an O at the top of each cylinder. We will do this by making two configurations of this cylinder.
37	Open a new part.	
38	Open a sketch in the Top-plane. Draw a circle, with the centre on top of the origin. Add the dimension Ø8.	

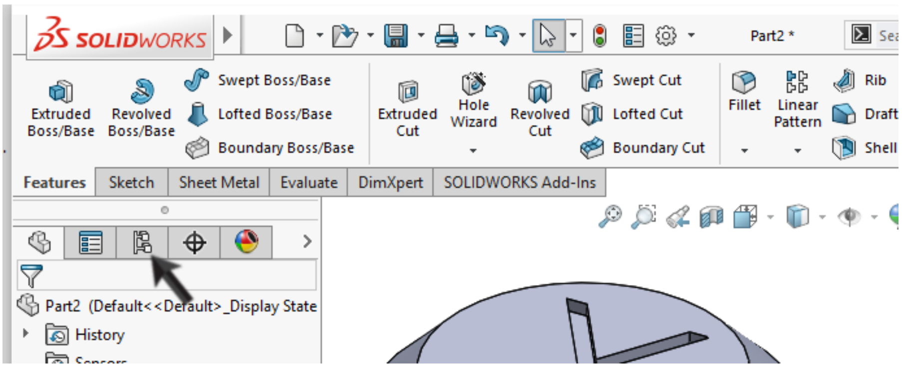
39	Set the fitting to h9. 1. Select the dimension 2. Set the Tolerance type to fit in the Property-Manager. 3. Set Shaft fit to h9.	
40	Select the Extrude Boss/Base command in the CommandManager 1. Drag the height of the extrusion to 20mm 2. Click OK.	
41	We will now make an angled edge at the top and at the bottom of the cylinder with the Chamfer command. Click on Chamfer in the CommandManager.	

<table>
<tr>
<td>42</td>
<td>1. Click on the vertical outside plane of the cylinder.
2. Set the sloped distance to 1 mm in the PropertyManager.
3. Check the angel to be 45°
4. Click OK.</td>
<td>
</td>
</tr>
<tr>
<td>43</td>
<td>1. Select the top plane of the cylinder.
2. Click on Sketch Text in the CommandManager.</td>
<td>
</td>
</tr>
</table>

44	1. Type in the capital X in the text field. 2. Uncheck the option Use Document Font. 3. Click on the Font button.	
45	Check in the menu if the text height is set to 4mm, and Click OK.	
46	Click OK in the PropertyManager.	

47	Rotate the model with the Normal to command so you can get a good view at the sketch. Drag the letter to the centre of the plane.	
48	Click on Features in the CommandManager and next on Extruded Cut.	
49	1. Set the depth to 0.25mm. 2. Click OK.	
50	The cylinder with the X is ready now. Save the file as: Shaft.sldprt	

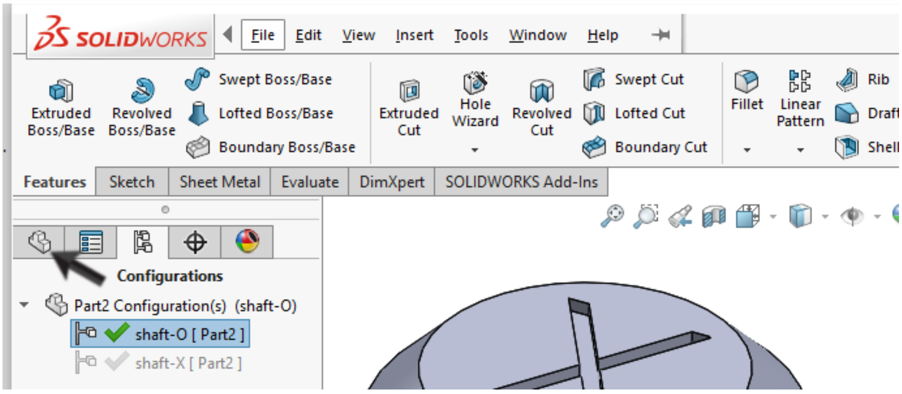
<table>
<tr>
<td>51</td>
<td>To make the cylinder with the O we will use a second configuration.

Click on the Configuration-Manager tab</td>
<td>
</td>
</tr>
<tr>
<td>52</td>
<td>Change the name of the current configuration (default) to Shaft-X.

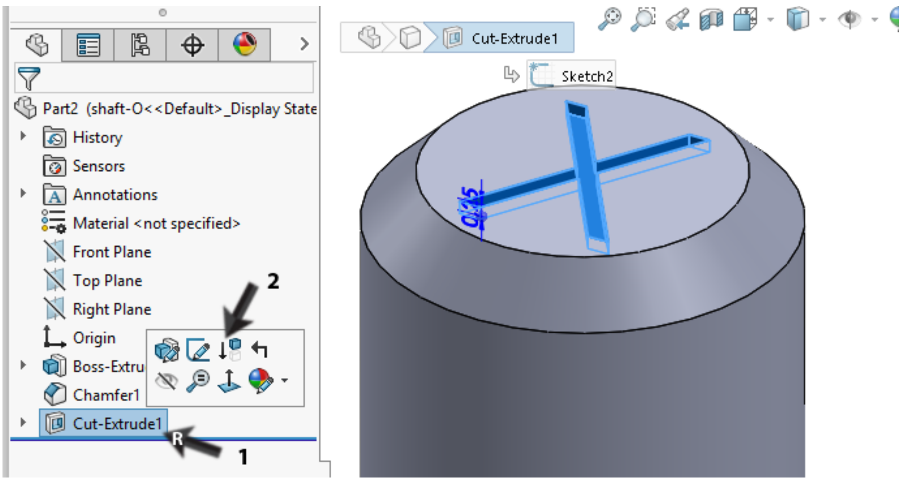
Create a new configuration called Shaft-O.

Check these commands in steps 24 to 26.

Check if the configuration Shaft-O is active (black).

Click on the FeatureManager tab.</td>
<td>
</td>
</tr>
<tr>
<td>53</td>
<td>With the Shaft-O configuration active, we must hide the letter X.

1. Click on the last features which you have made.

2. Select Suppress in the menu that appears.</td>
<td>
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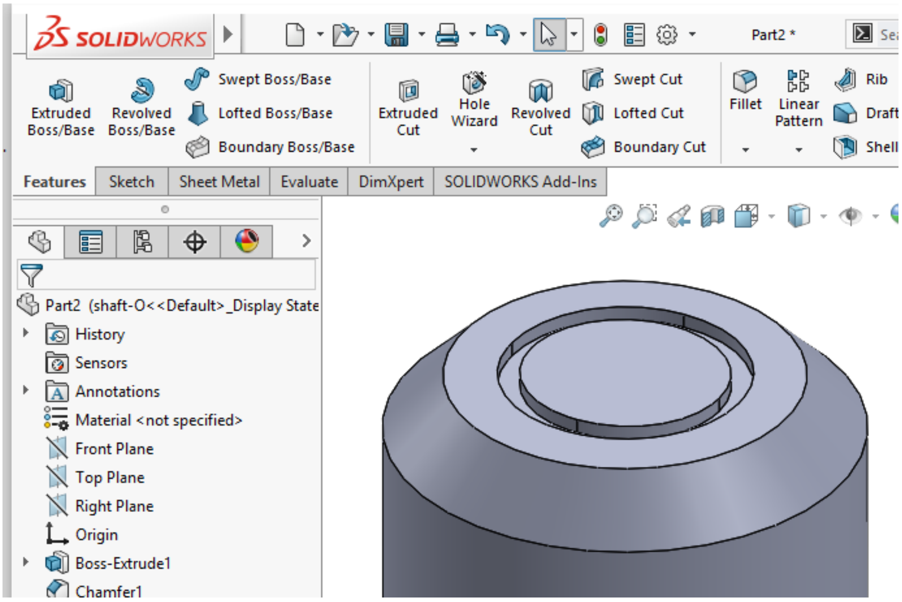
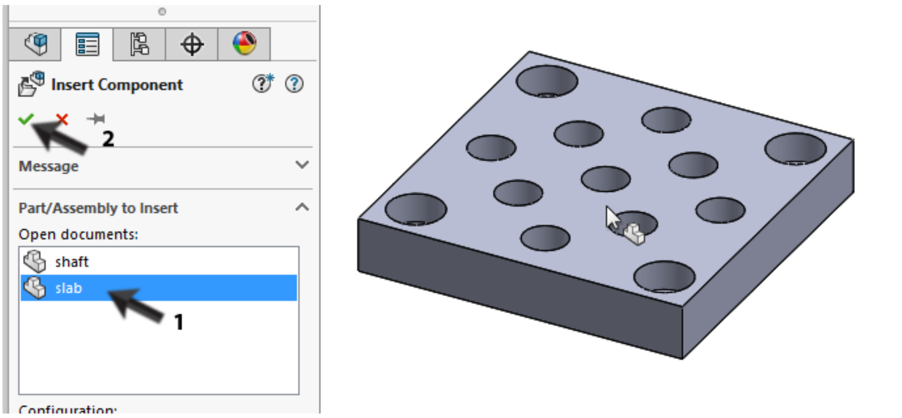
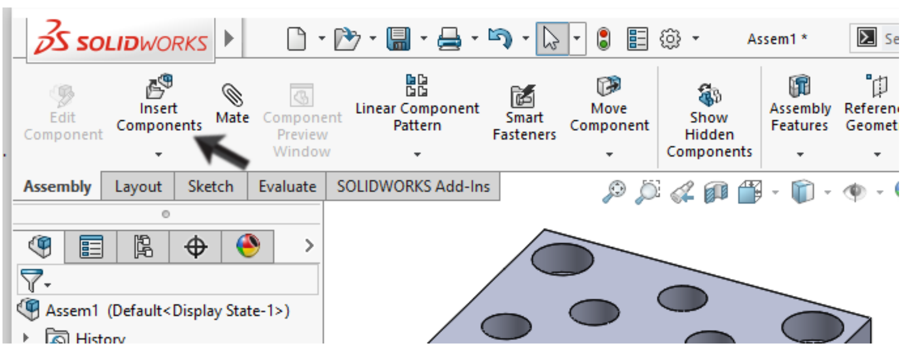
<table>
<tr>
<td>54</td>
<td>Now put a letter O on the top plane of the cylinder. Do this in exactly the same way as you did before with the letter X. (steps 43 to 49)</td>
<td>
</td>
</tr>
<tr>
<td>55</td>
<td>Save the file.

Open a new assembly.</td>
<td></td>
</tr>
<tr>
<td>56</td>
<td>When you did not close the two parts we just created (Slab and Shaft) you will see the image on the right.

1. Click on the file Slab.
2. Click OK.

If you did close this file, find it with the Browse command.</td>
<td>
</td>
</tr>
<tr>
<td>57</td>
<td>Click on Insert Component in the CommandManager.</td>
<td>
</td>
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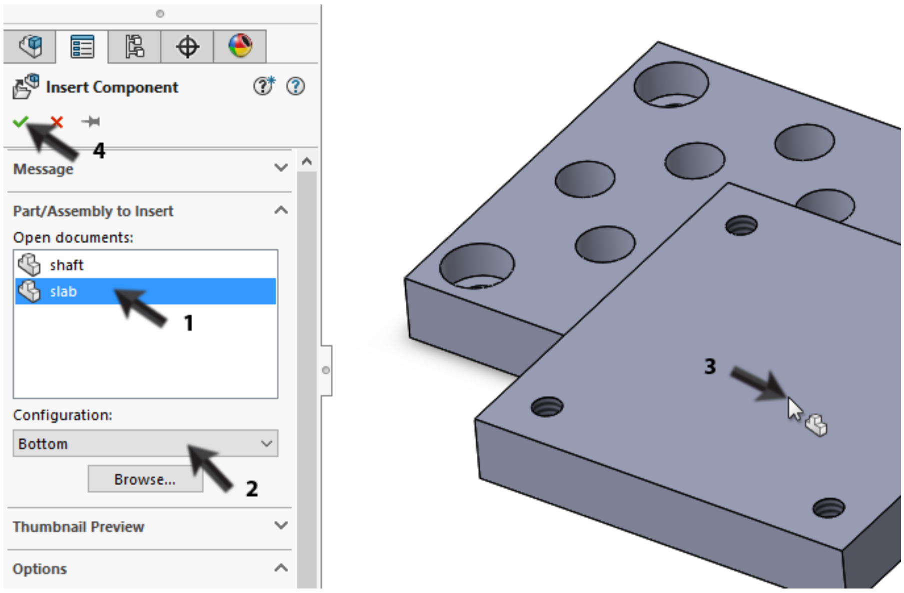
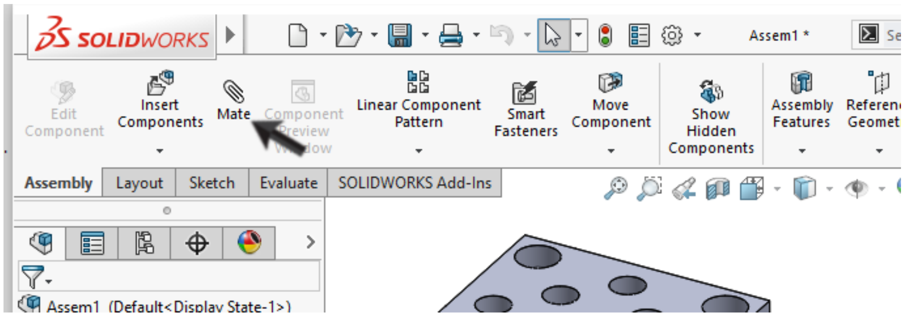
</table>

<table>
<tr>
<td>58</td>
<td>
Insert the same part again, but now with the other configuration.

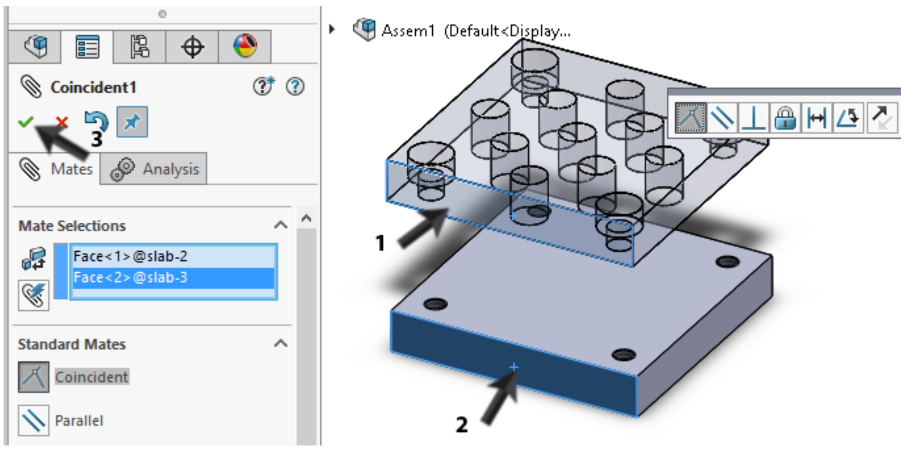
1. Select the part
2. Select the right configuration in the PropertyManager
3. Place the part in the assembly
4. Click OK

If necessary, shift the part so that it is more or less in the right position
</td>
<td>


</td>
</tr>
<tr>
<td>59</td>
<td>
Next we have to align the two parts with the mate command.

Click on Mate in the CommandManager.
</td>
<td>


</td>
</tr>
<tr>
<td>60</td>
<td>
Select the sides of both as shown in the illustration.

Click OK.
</td>
<td>


</td>
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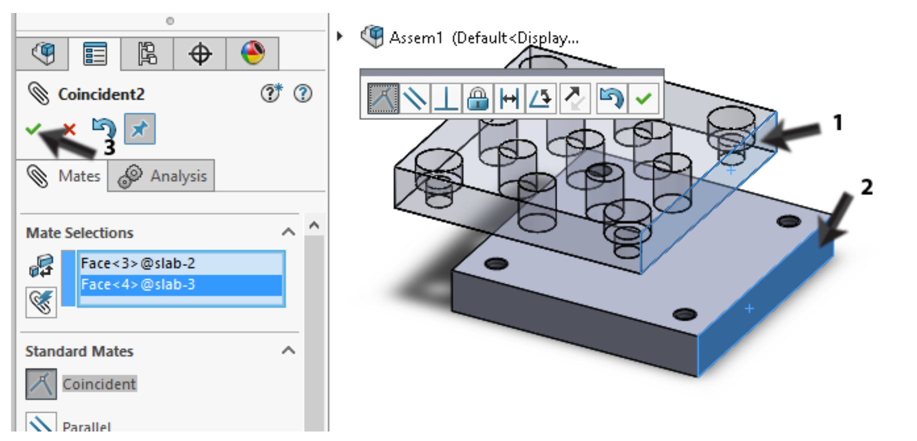
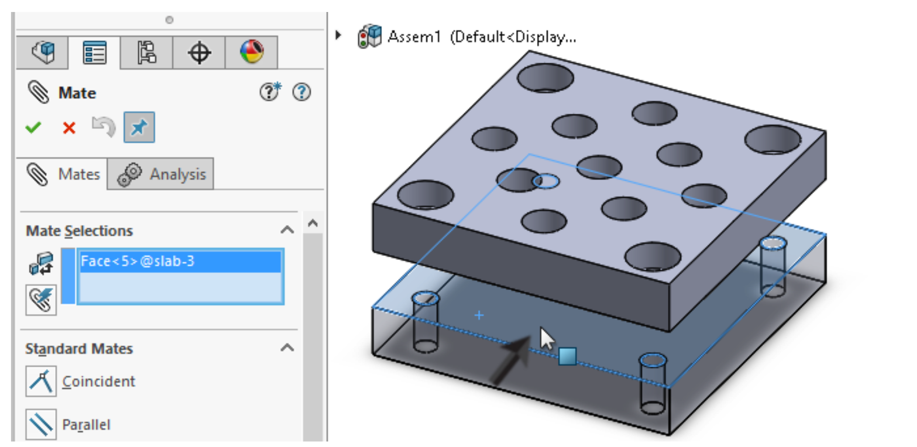
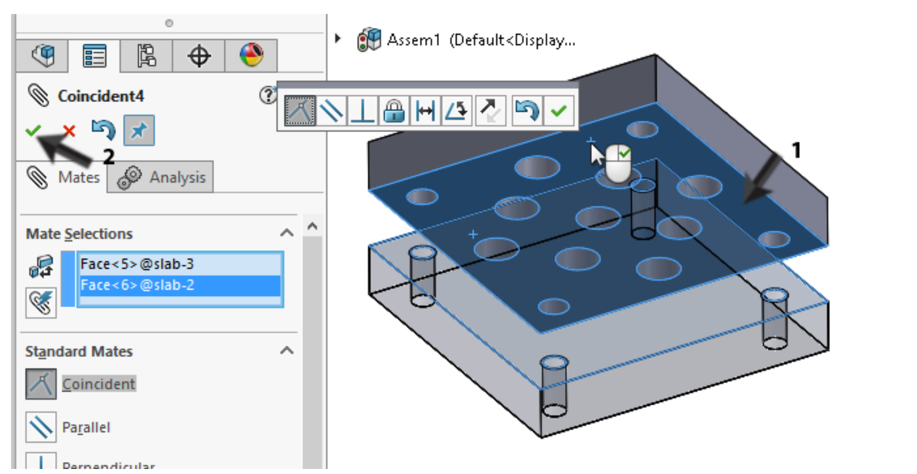
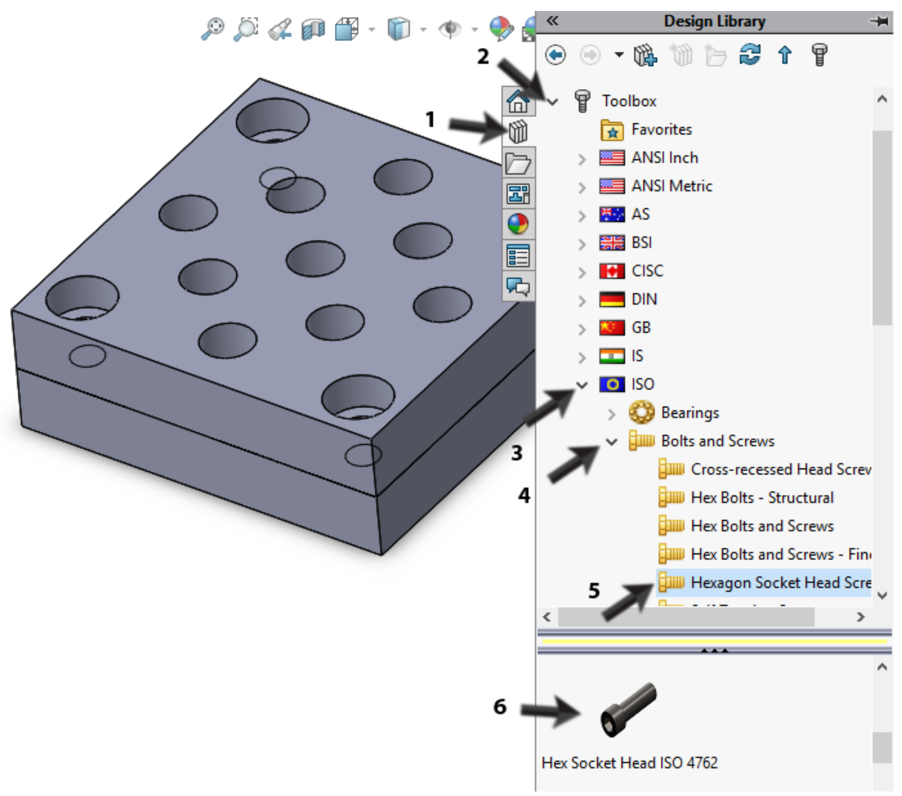
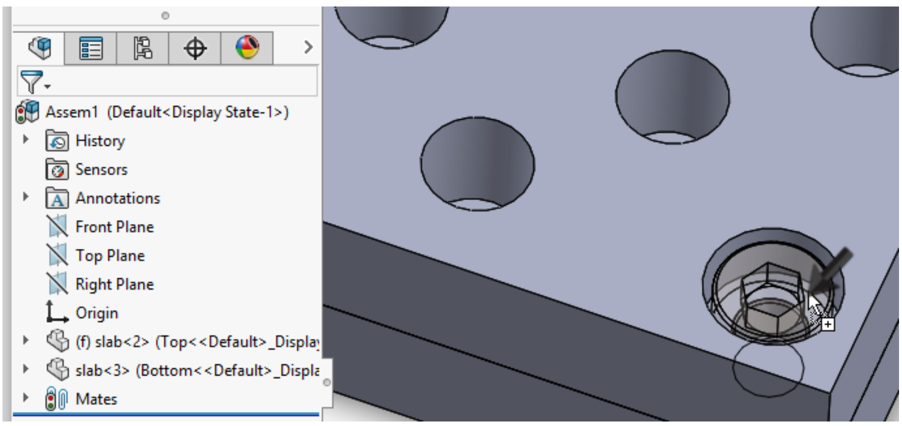
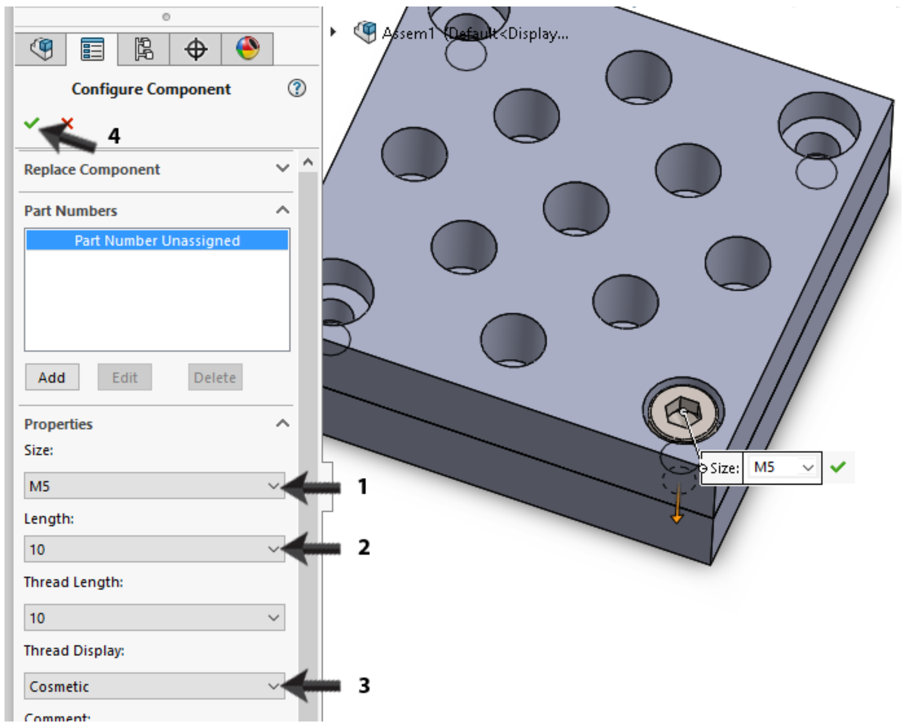
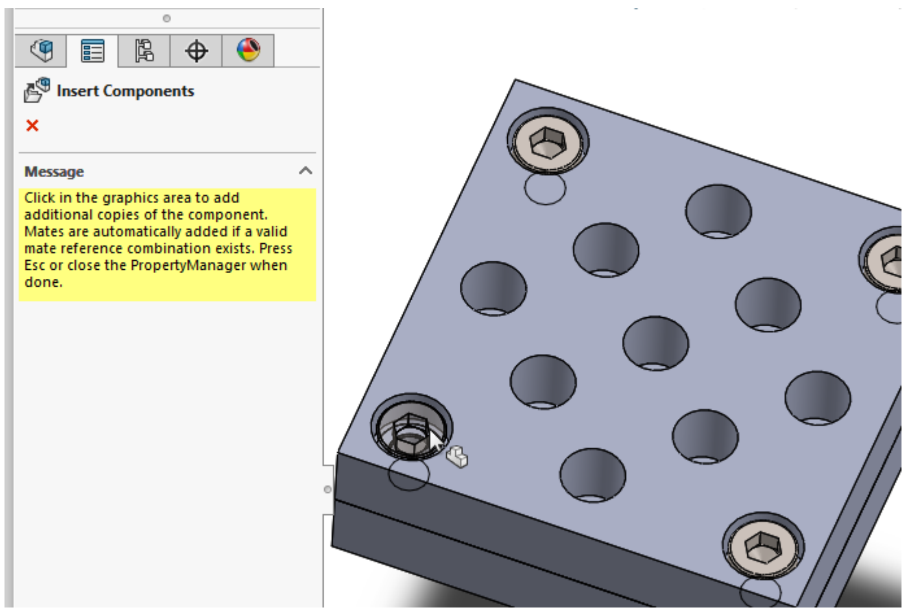
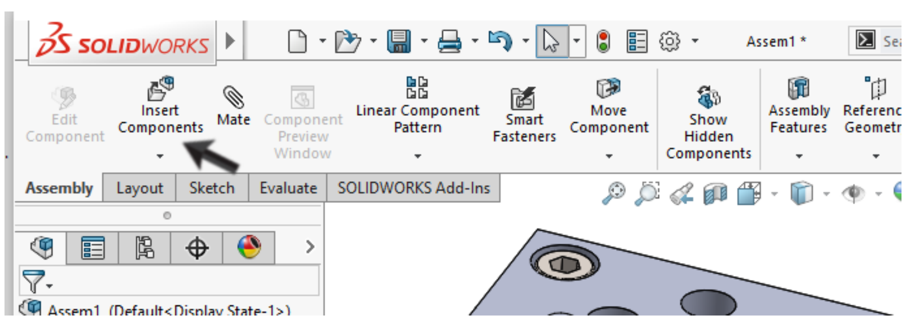
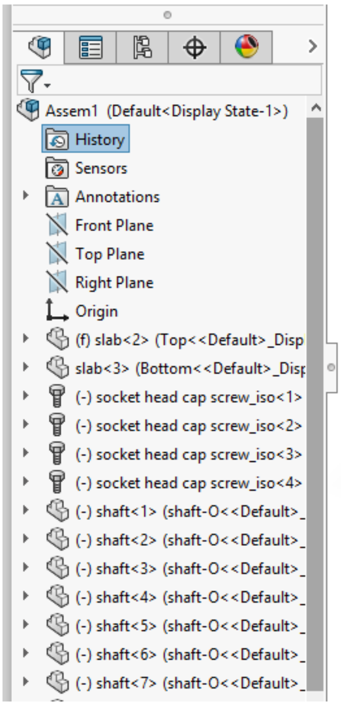
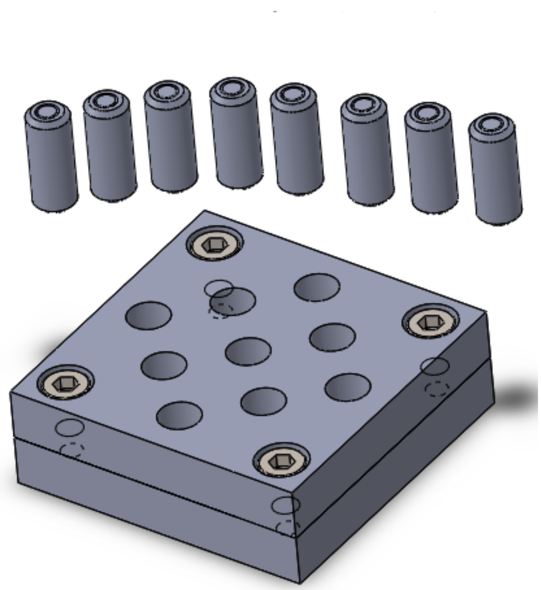
61	Select two other sides of both parts as shown in this illustration. Click OK.	
62	Select the top plane of the bottom part.	
63	Next rotate the model so you get a good view at the bottom of the top part and select the bottom plane. Double Click OK.	

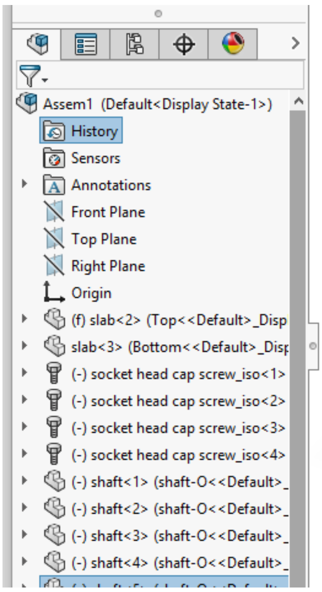
64	Next we put the hexagon socket head screws in the model. 1. Open the Design Library in the Task Pane. 2. Click on Toolbox 3. ISO 4. Bolts and Screws 5. Hexagon Socket Head Screws 6. Select: *Hex Socket Head ISO 4762*	
65	Drag the bolt to your model. Release the mouse button at the lower edge of one of the countersink holes.	

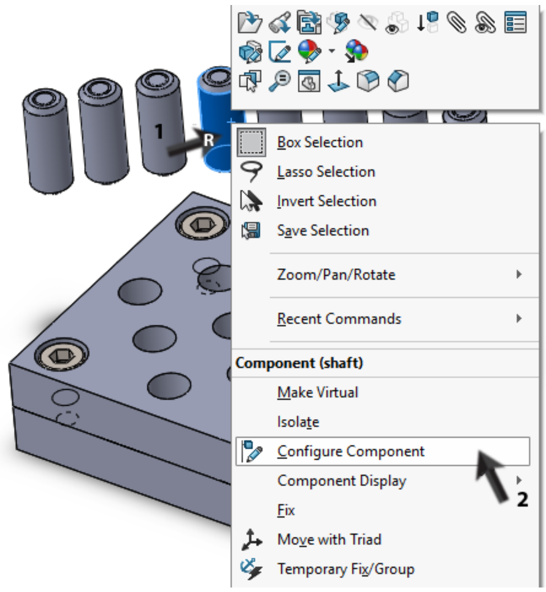
66	Set the following features in the PropertyManager: 1. Size: M5 2. Length: 10 3. Thread display: Cosmetic 4. Click OK.	
67	Put hexagon head screws in the other holes as well.	
68	Finally the cylinders (pens) should be placed in the holes. Click on Insert Component in the CommandManager.	

<table>
<tr>
<td>69</td>
<td>Put the cylinder or pen 8 times in the assembly at a random position.

Notice: it does not matter is you pick an X or O cylinder. We will change four of them later on.</td>
<td>

</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>You can of course use the Insert Component command 8 times to insert the pens, but it will be much quicker to drag the part from the FeatureManager, holding the <ctrl>-key. A copy of the part is made every time you do so.</td>
</tr>
<tr>
<td>70</td>
<td>Next we will change the letter on four of the pens.

Right click on a pen and select Configure Component.</td>
<td>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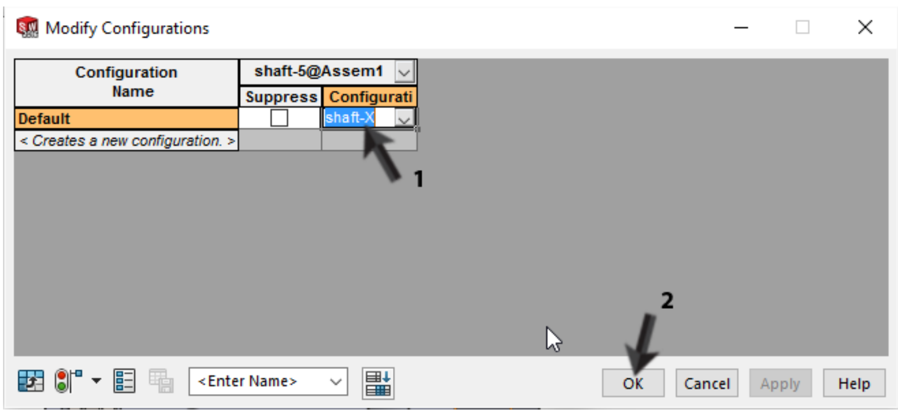
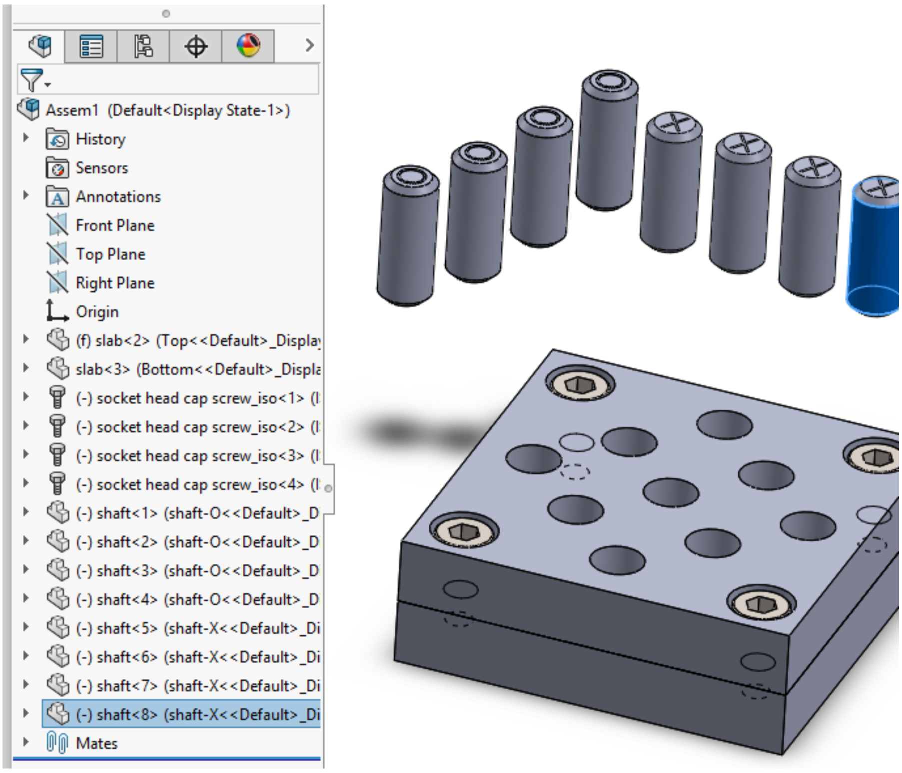
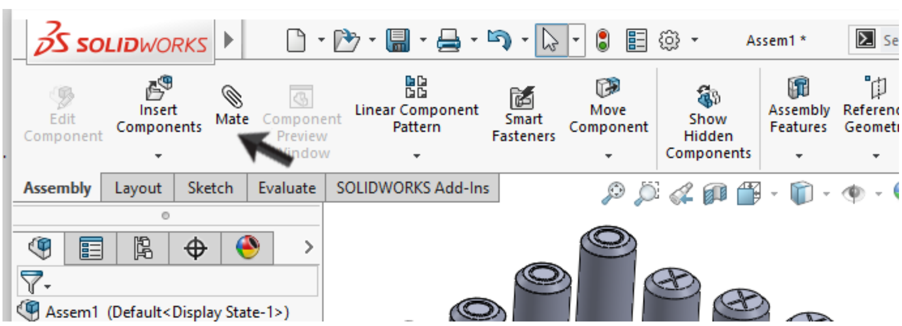
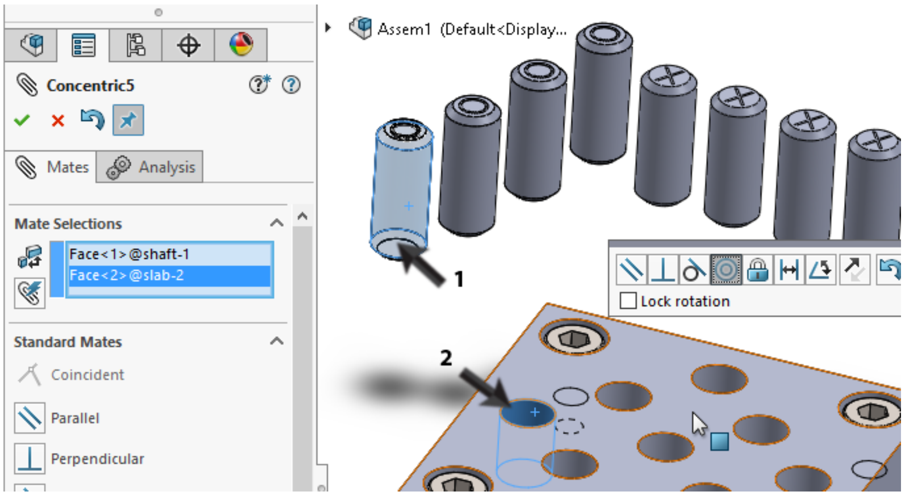
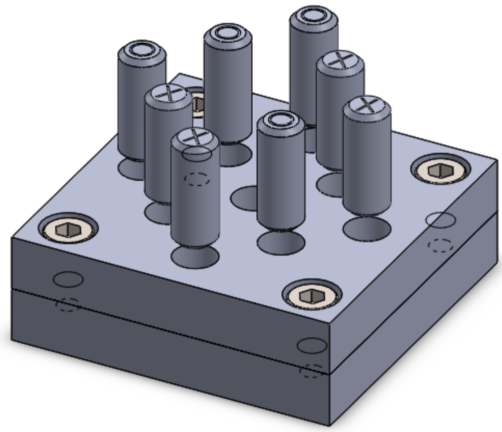
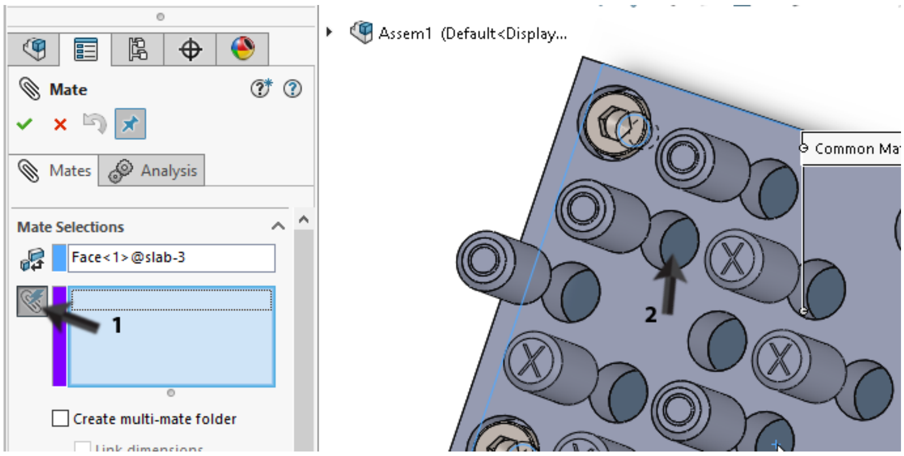
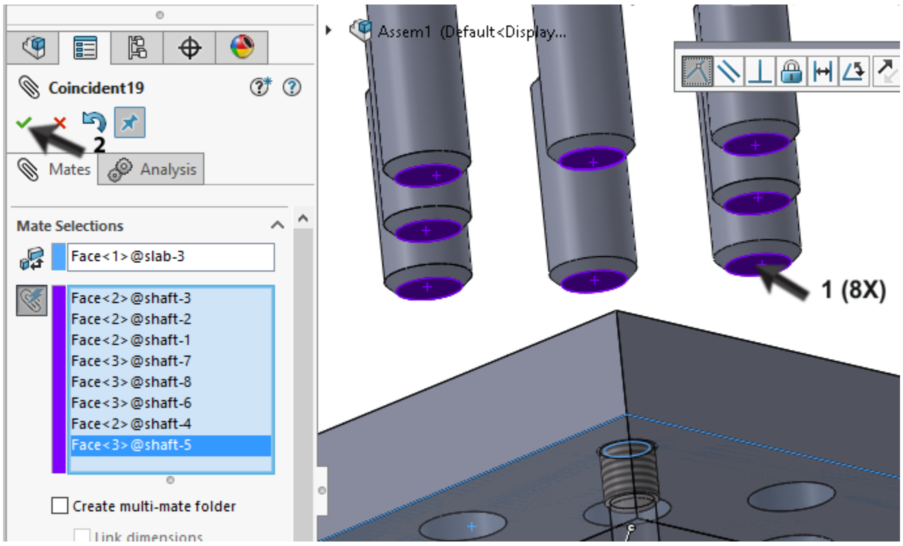
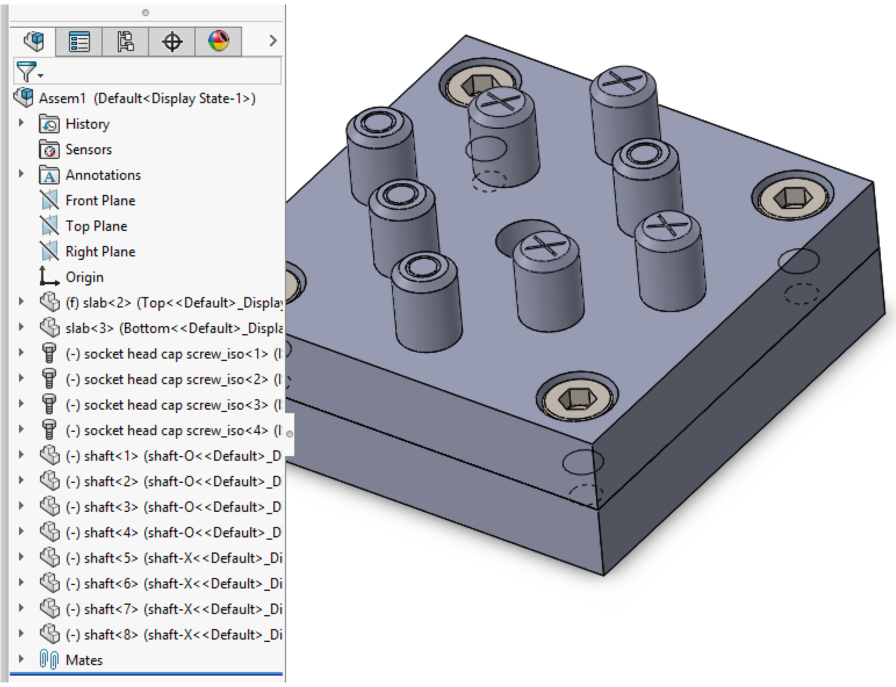
</table>

71	1. Select the desired configuration in the menu that appears: when a cylinder has an O on top, select the X- configuration or do this just the other way around. 2. Click OK.	
	Tip!	There are several ways to get the part with the right configuration in the assembly. Here we first inserted the part and changed the configuration afterwards. At step 58 we selected the right configuration before inserting the part. Just choose the way that works best for you!
72	Repeat this step for three other pens.	
73	Next we have to mate the pens in the holes. Click on Mater in the CommandManager	

74	Select the two planes like it is shown in the illustration on the right. Click OK.	
75	Repeat the last step for all the pens and select a different hole for every pen. The height of the pens is not determined yet. You can still move al the pens up and down by dragging them.	
76	We will make the final mate now. 1. Click on the Multiple Mate Mode in the PropertyManager. 2. Rotate the model so you get a good view at the INSIDE of a hole. Through the hole you can see the top plane of the bottom part. Select this plane.	

77	Rotate the model again so you can see the bottom side of the pens. 1. Select the bottom side of all pens. 2. Click OK.	
78	The assembly is ready now. Save the file as: Tictactoe.SLDASM.	
	What are the main features you have learned in this tutorial?	In this tutorial we have repeated al lot of what we have seen and done before: • Creating simple parts and shapes. • Working with configurations. • Working with standard parts. • Working with the Hole Wizard. We have also learned some new topics: • You have set fittings at holes and/or pens. • You have seen how to use text in the sketch. • You have learned some new tricks.

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