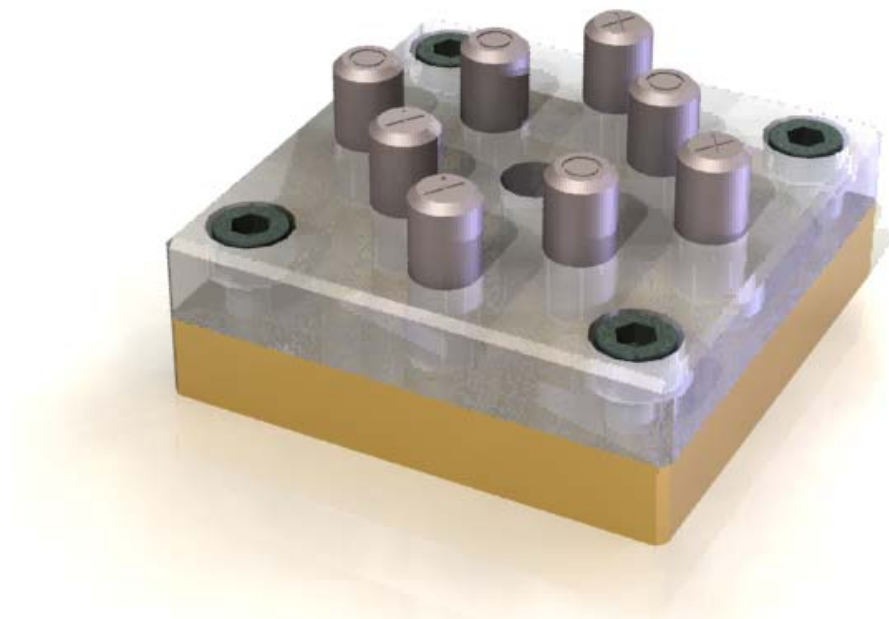


SolidWorks® Tutorial 5

TIC-TAC-TOE



Preparatory Vocational Training
and Advanced Vocational Training



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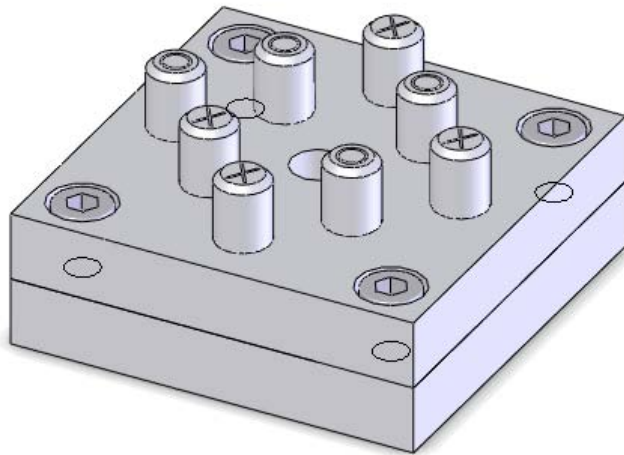
Initiative: Kees Kloosterboer (SolidWorks Benelux)

Educational Advisor: Jack van den Broek (Vakcollege Dr. Knippenberg)

Realization: Arnoud Breedveld (PAZ Computerworks)

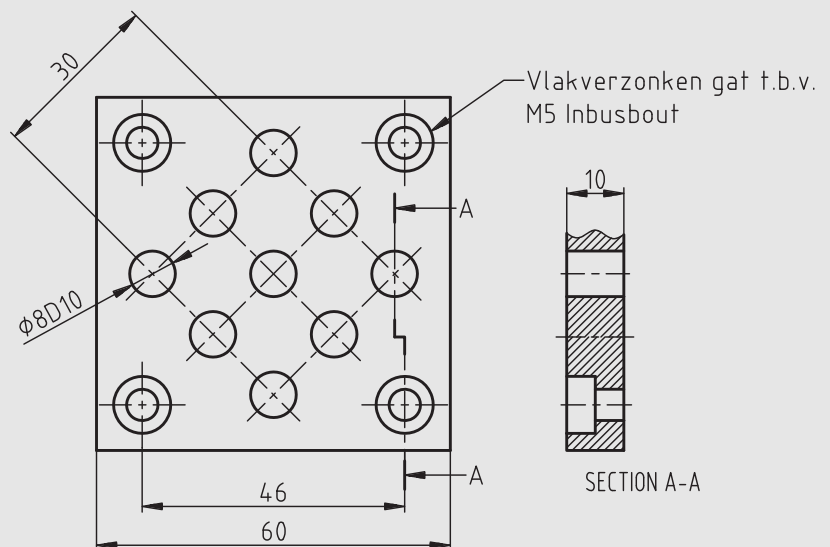
TIC-TAC-TOE

In this tutorial we will create a Tic-Tac-Toe game. The game consists of two plates that are on top of each other. In the top plate, there are holes for inserting small cylinders marked 'X' or 'O'. In this exercise we repeat a lot of tools we already know and add a few others: working with configurations and the use of standard Parts. Some new features in this tutorial include working with tolerances and fittings and working with patterns.



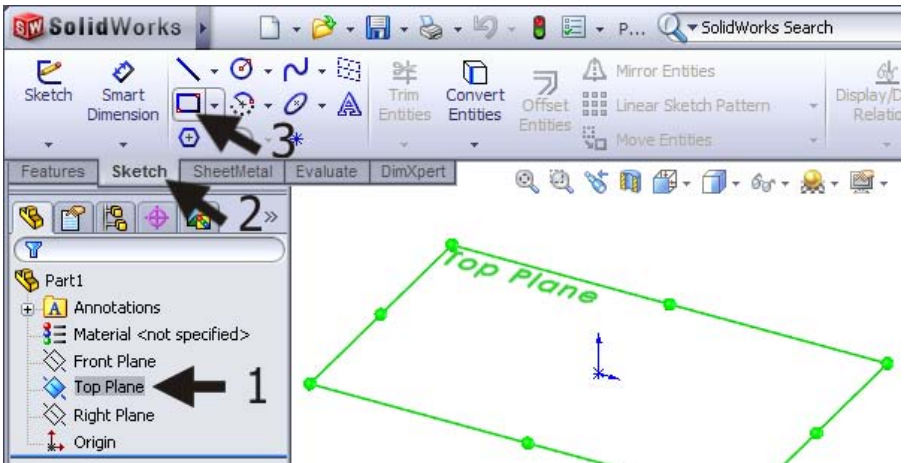
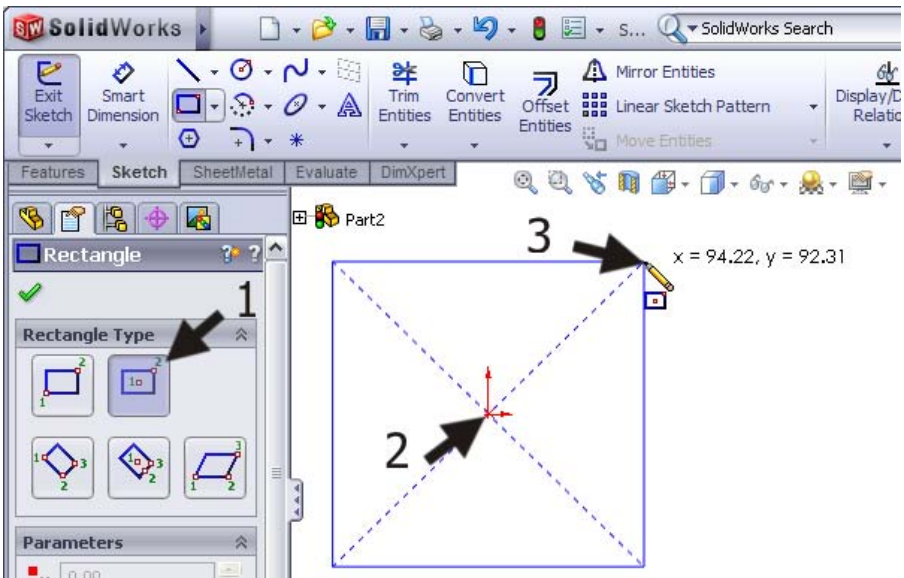
Work plan

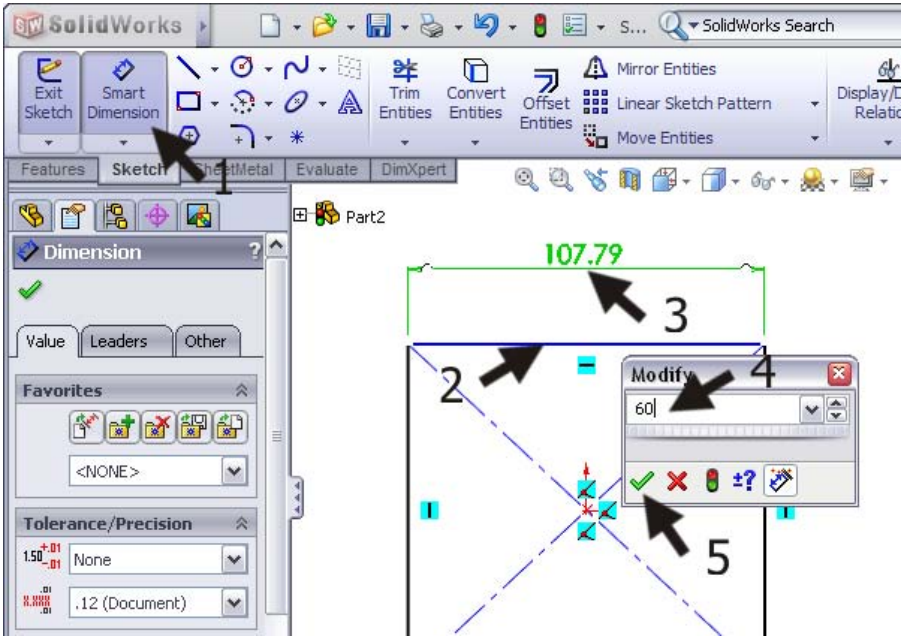
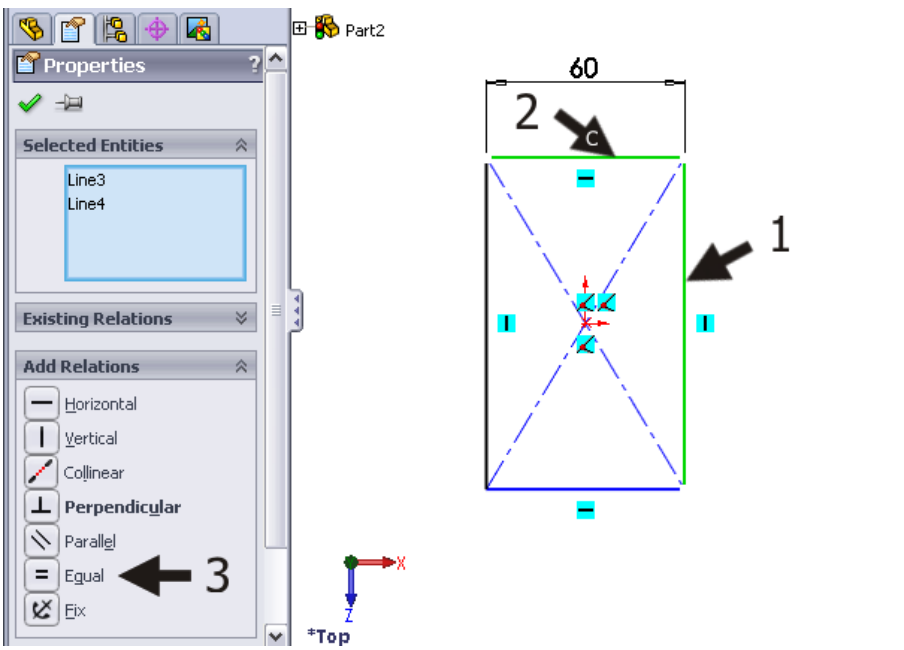
First, we will create the top plate. We will do this according to the drawing below.



We will execute following steps:

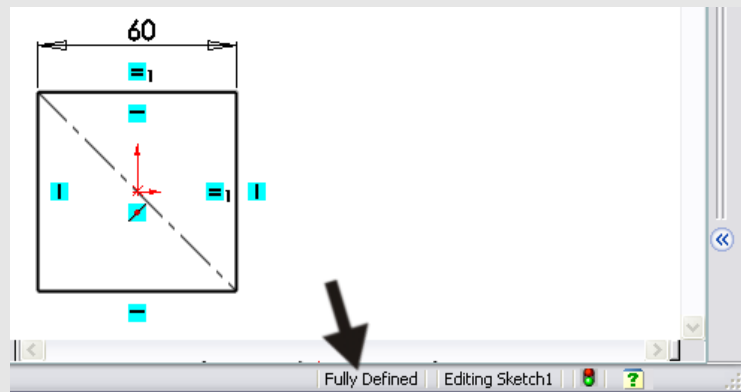
1. First, we will create the top plate first with dimensions 60 x 60 x 10.
2. Then, we will make four counter bore holes.
3. Finally, we will create a pattern of 9 holes.

1	Start SolidWorks and open a new part.	
2	<ol style="list-style-type: none"> 1. Select the 'Top Plane'. 2. Click on 'Sketch' in the CommandManager. 3. Click on Rectangle. 	
3	<p>Draw a rectangle:</p> <ol style="list-style-type: none"> 1. Click on Center Rectangle in the Property-Manager. 2. Click on the origin. 3. Click at a random point to get the second corner. 	

<p>4</p>	<p>Add a horizontal dimension to the sketch, as in the illustration on the right.</p> <p>Change this dimension to 60mm.</p> <p>Push the <Esc> key on the keyboard to end the command.</p>	
<p>5</p>	<p>Set the length of the horizontal and vertical lines to the same length:</p> <ol style="list-style-type: none"> 1. Select a vertical line. 2. Push the <Ctrl> button and click on a horizontal line. 3. Click on 'Equal' in the PropertyManager. 	
<p>Tip!</p>		<p>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).</p> <p>When you see a pink-colored selection field, you do not have to use the Ctrl> key to select more than one element.</p> <p>To remove an element from the list, click on the element in the pink field and push the (delete) key on your keyboard. SolidWorks often asks you if you really want to remove the element from the selection field to prevent inadvertent deletions.</p>
<p>Tip!</p>		<p>The sketch is now fully defined. You can determine this from the color of the lines in the sketch:</p>

- **Blue** means: the sketch is not **fully defined**.
- **Black** means: the sketch is **fully defined**.

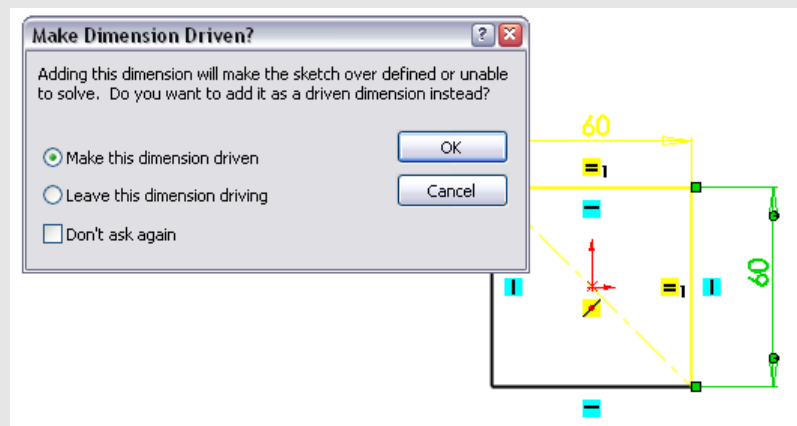
You can check if a sketch is fully defined in the status bar at the bottom of the screen. In SolidWorks it is not *mandatory* to make a fully defined sketch, but it is a good practice to do this because it can help you to avoid a lot of problems when creating a model later.



In addition to the colors 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: set the dimension of the height of the square. The '**Make Dimension Driven?**' message appears:

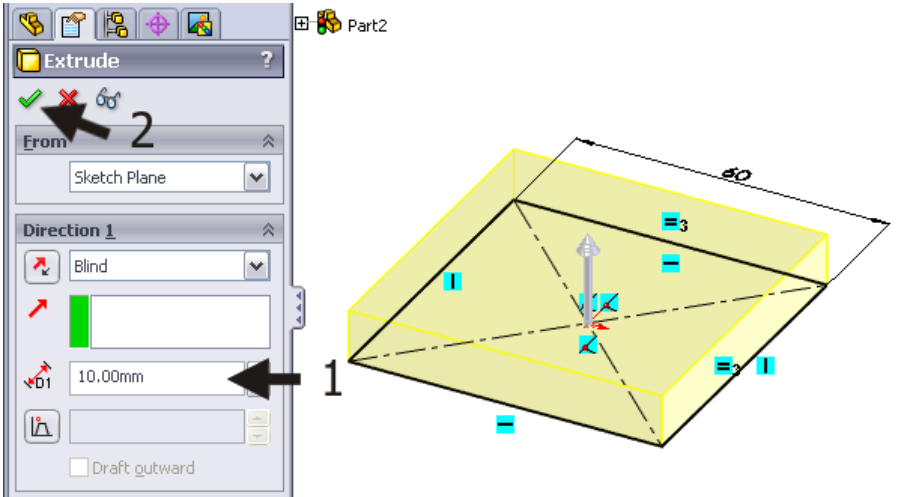
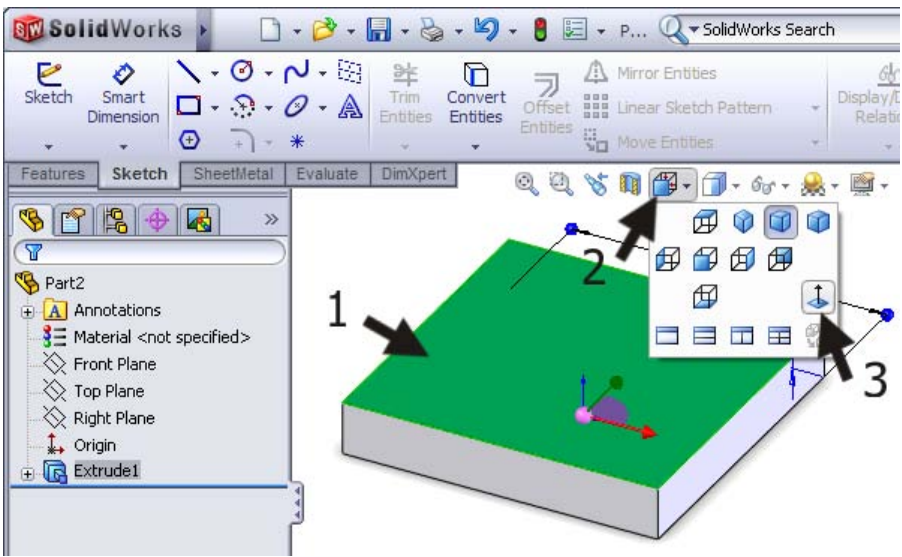
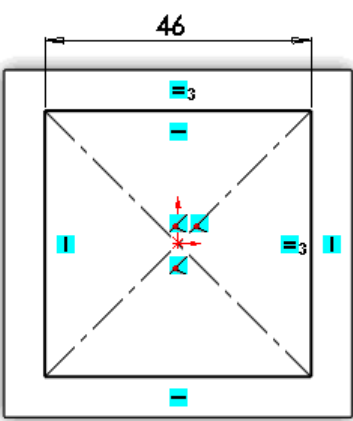


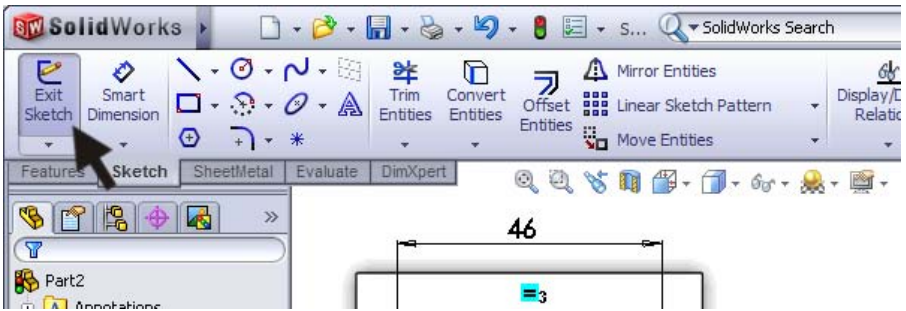
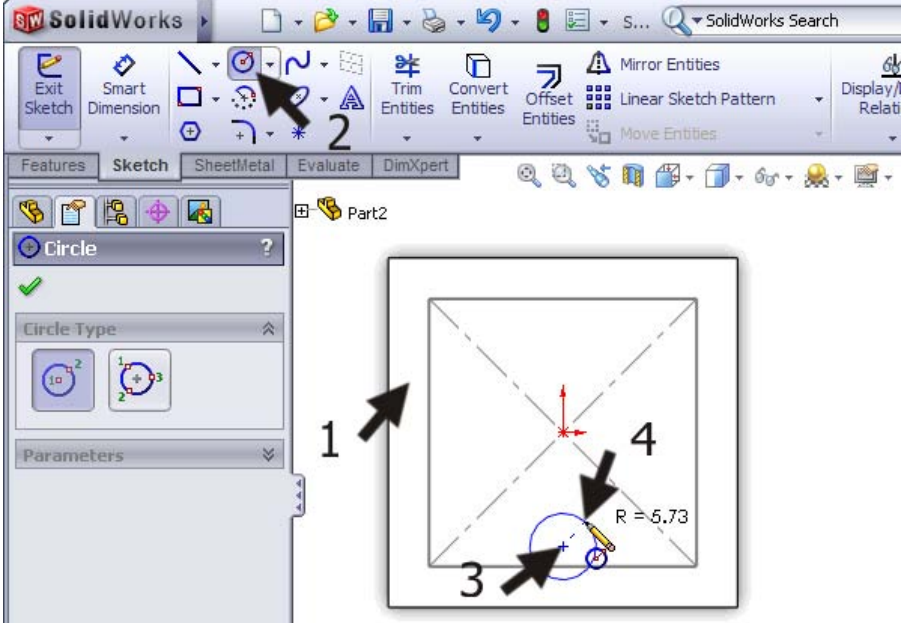
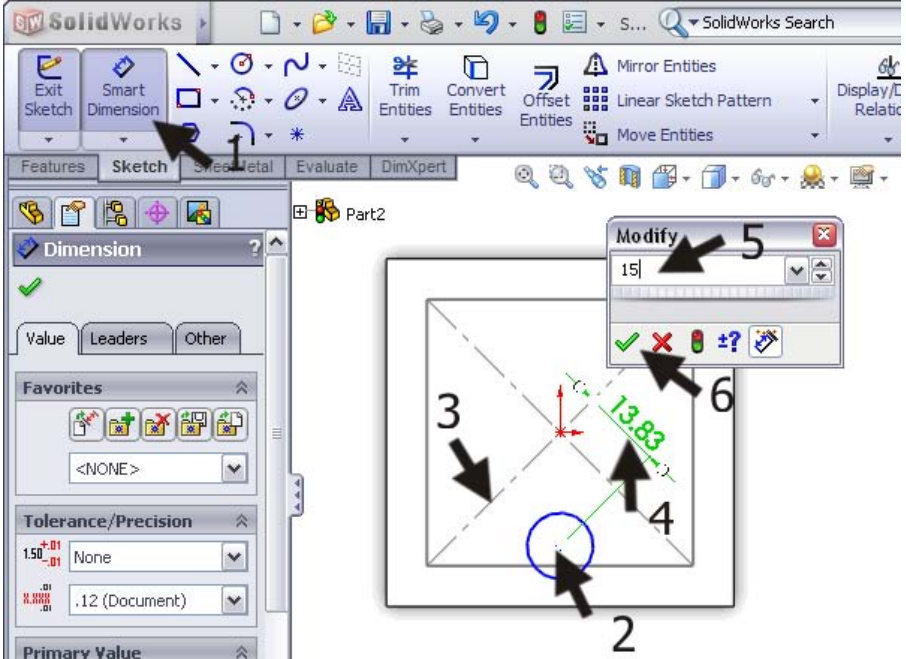
You have entered too much information because:

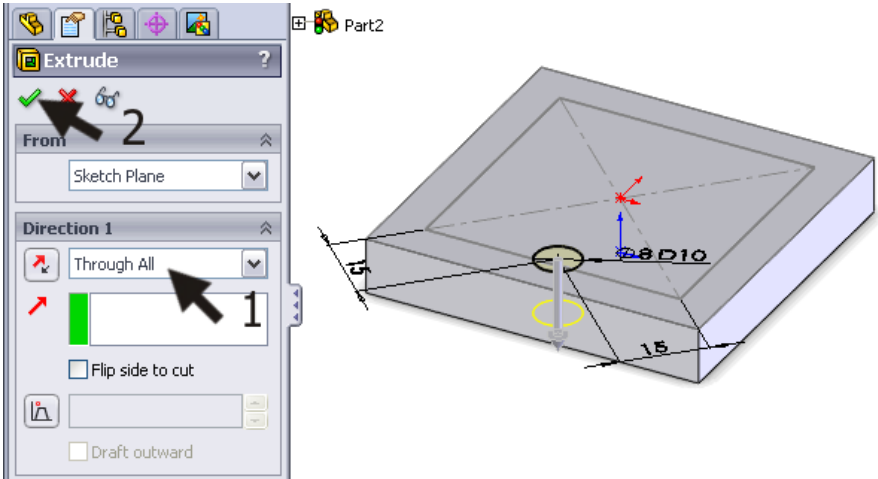
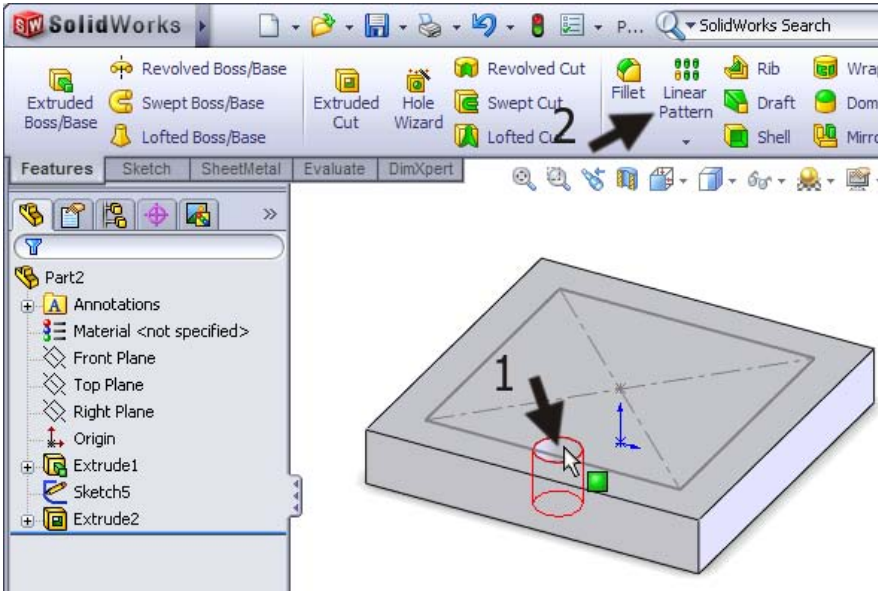
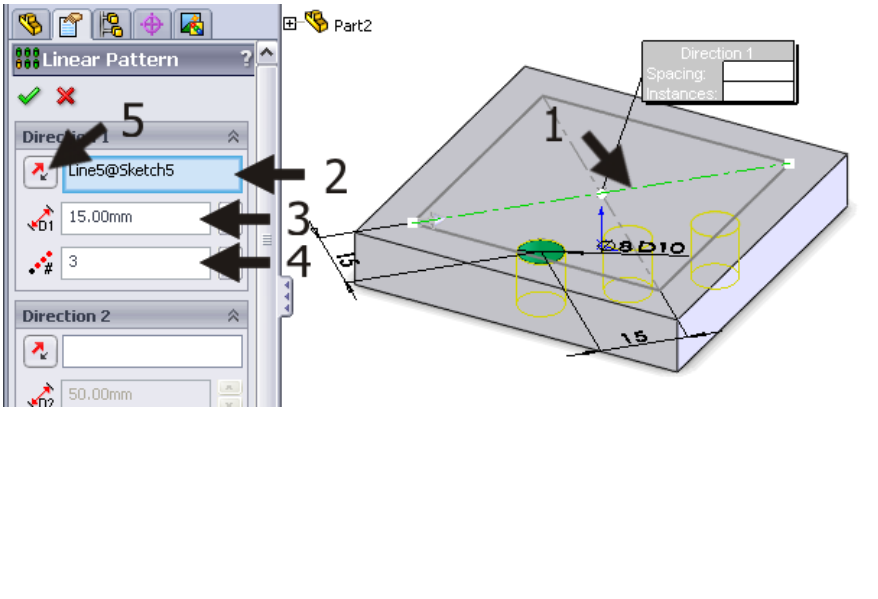
- The dimension you added says the height is 60mm.
- The relation between the two lines you have created before says the height is equal to the width, which is also 60.

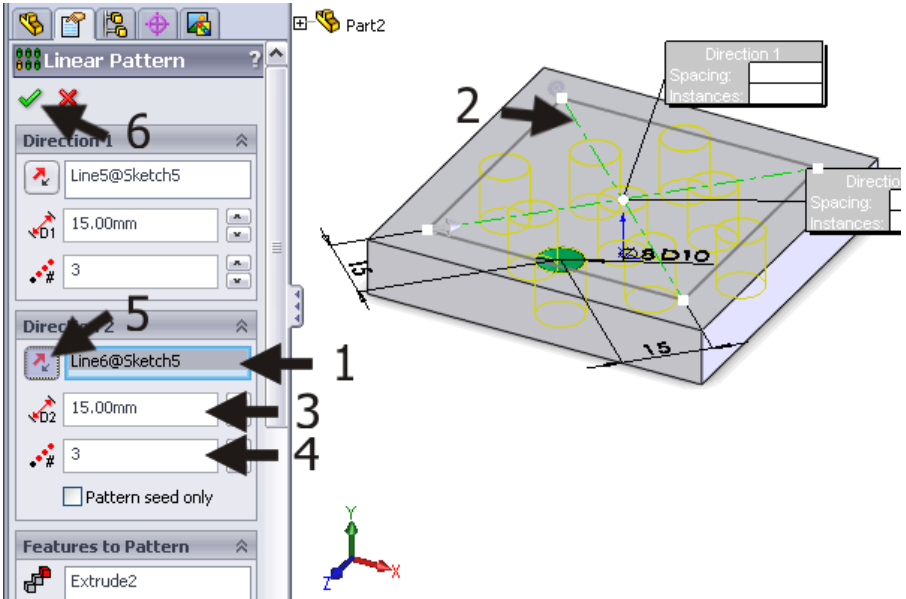
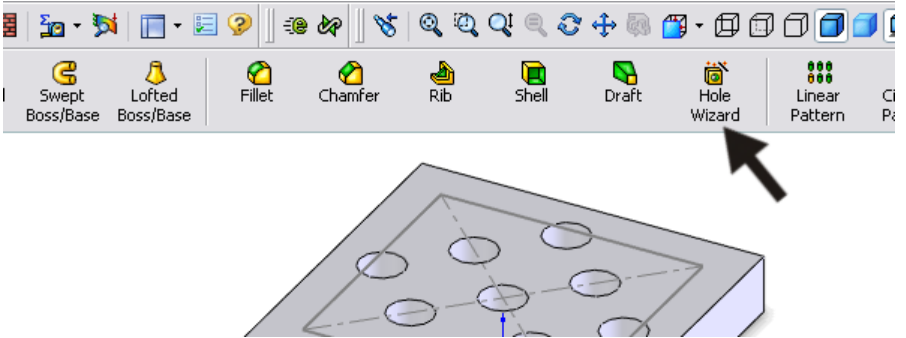
The height is defined twice now, and this creates a conflict in SolidWorks. You must resolve this inconsistency. In the menu that is shown above, the best thing to do is choose '**Cancel**'. The dimension will not be set.

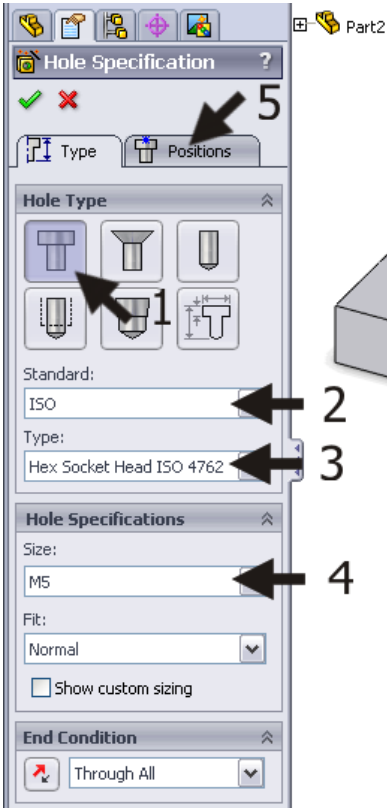
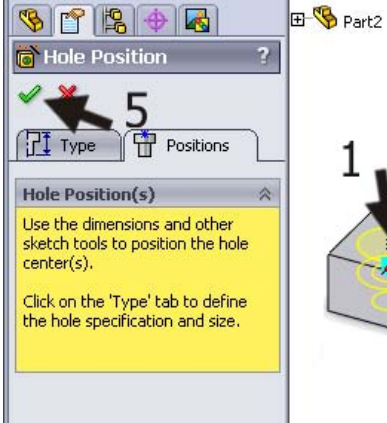
Did you make an **over-defined** sketch anyway? Then, throw away (delete) dimensions and/or relations, so that the sketch is no longer **over-defined**.

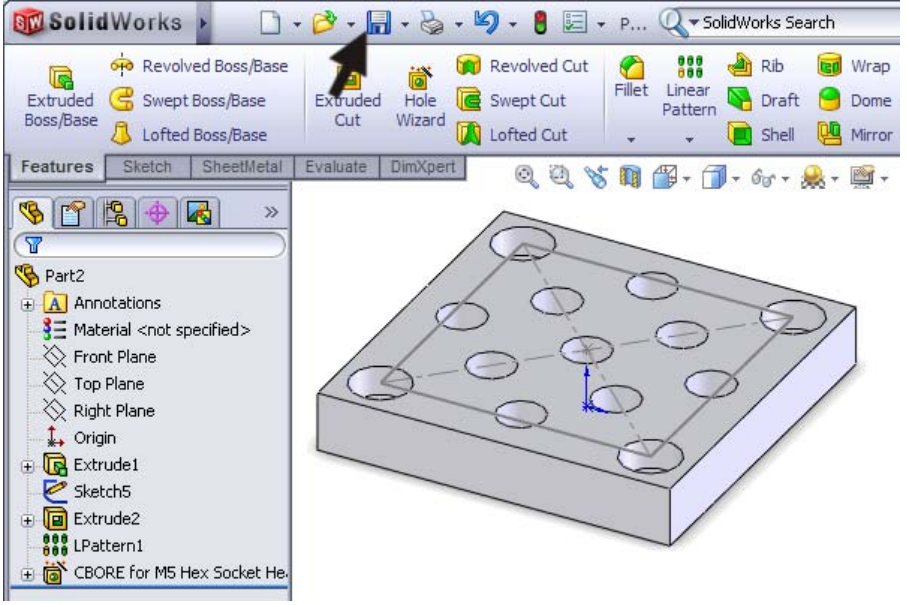
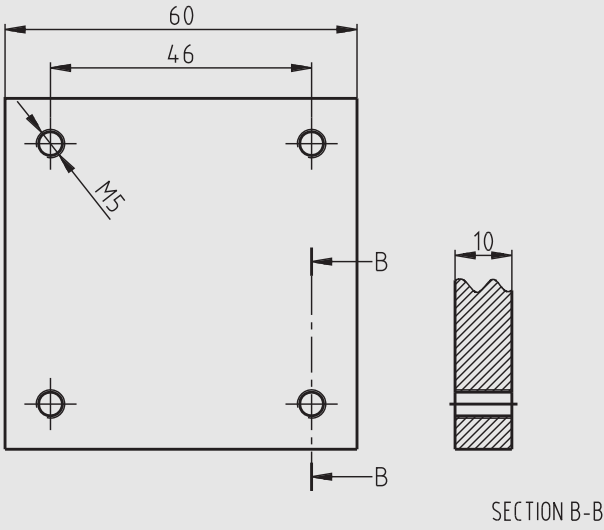
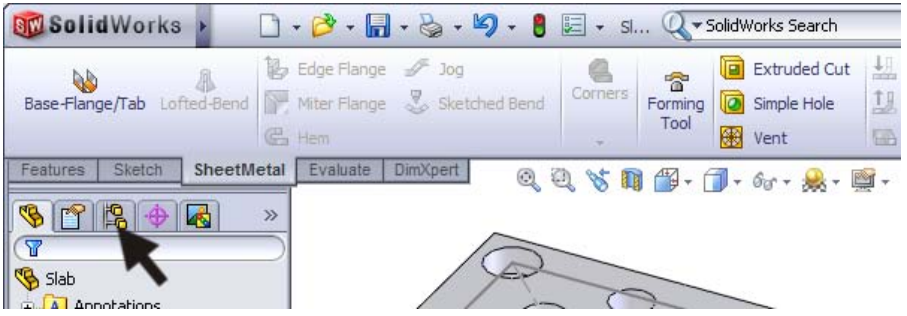
<p>6</p>	<p>Click on 'Features' in the CommandManager, then on 'Extruded Boss/Base'.</p> <ol style="list-style-type: none"> 1. Set the thickness of the plate to 10 mm. 2. Click on OK. 	
<p>7</p>	<p>Next, we will make a sketch in which we determine the exact position of the holes:</p> <ol style="list-style-type: none"> 1. Select the top plane of the plate 2. Click on the View Orientation icon. 3. Click on Normal To. 	
<p>8</p>	<p>Draw another rectangle with a dimension of 46 mm. Follow the steps 3 to 5 again if you need help.</p>	

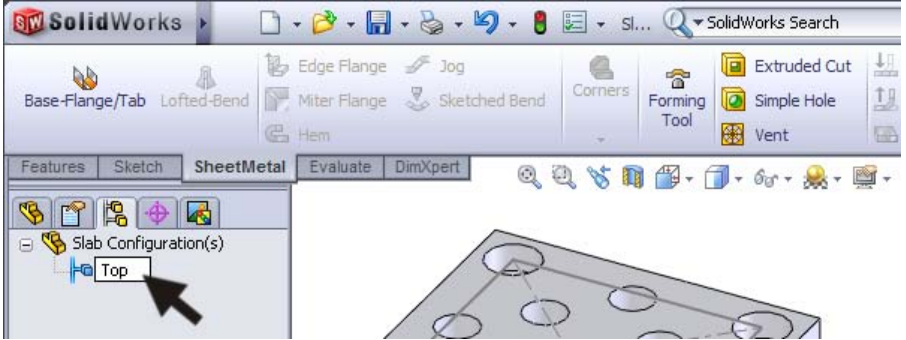
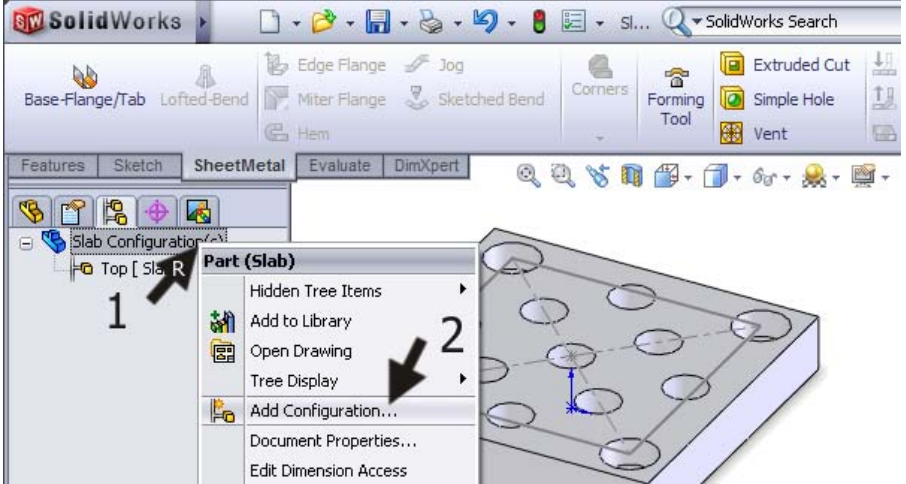
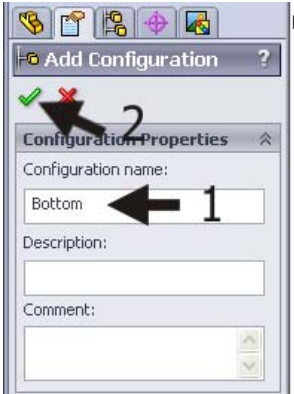
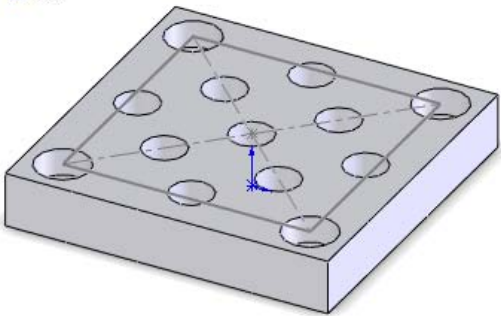

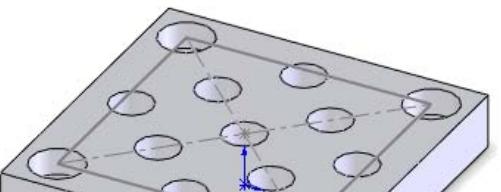
9	<p>Click on 'Exit Sketch' in the CommandManager.</p> <p>We will not use this sketch to make a feature.</p>	
10	<p>Start up a new sketch.</p> <ol style="list-style-type: none"> 1. Select the top plane again. 2. Click on Circle in the CommandManager. 3,4 Draw a circle like the one in the illustration. 	
11	<p>Set the dimension between the circle and one of the diagonal lines that you have drew previously:</p> <ol style="list-style-type: none"> 1. Click on Smart Dimension in the CommandManager. 2. Click on the center of the circle. 3. Click on the diagonal line. 4. Set the dimension. 5. Change it to 15mm. 6. Click on OK. 	

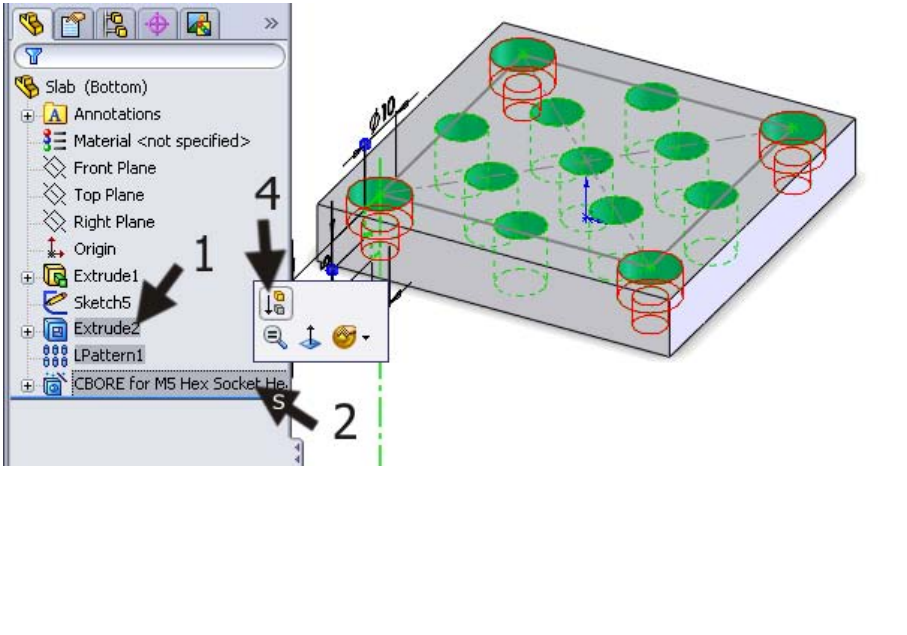

<p>14</p>	<p>Make a hole in this sketch: click on 'Features' in the CommandManager and then on 'Extruded Cut'.</p> <p>Set the depth of the hole in the PropertyManager to 'Through all' and click on OK.</p>	
<p>15</p>	<p>We will complete the hole pattern now.</p> <ol style="list-style-type: none"> 1. Select the hole you just created. 2. Click on the 'Linear pattern' icon in the CommandManager. 	
<p>16</p>	<p>Next, set following features:</p> <ol style="list-style-type: none"> 1. Select ONE of the diagonal lines. 2. Check to make sure that the line appears in the selection field. 3. Set the distance between the copies to 15mm. 4. Set the number of copies to 3. 5. Whenever the copies are placed on the wrong side, click on 'Reverse Direction'. 	

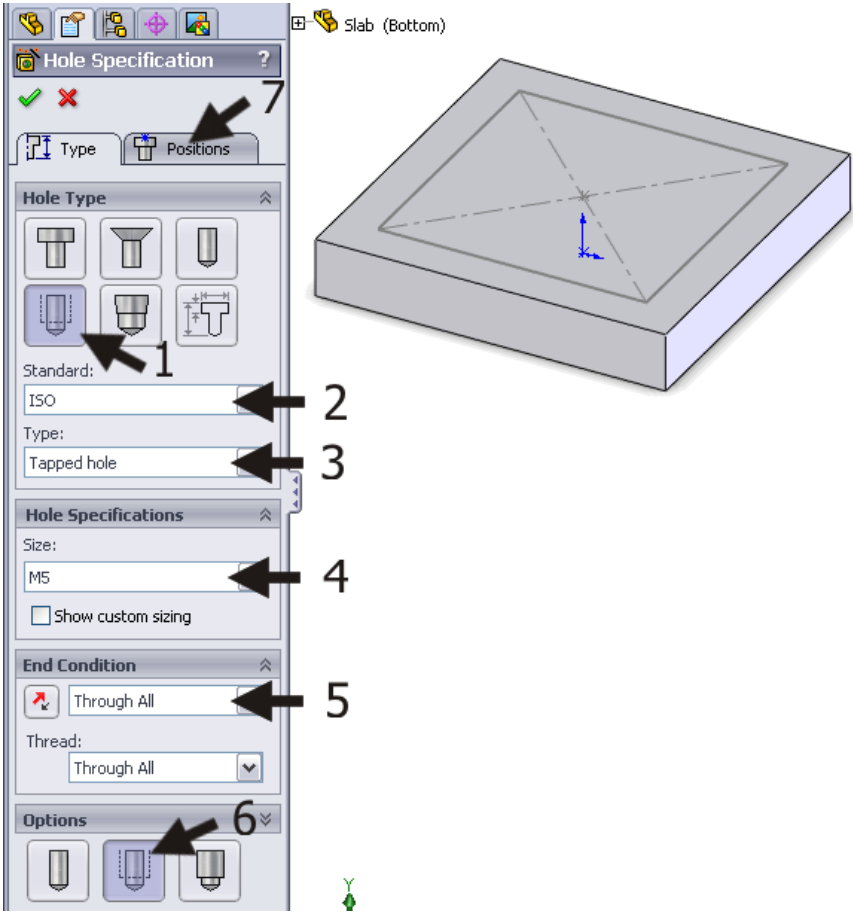
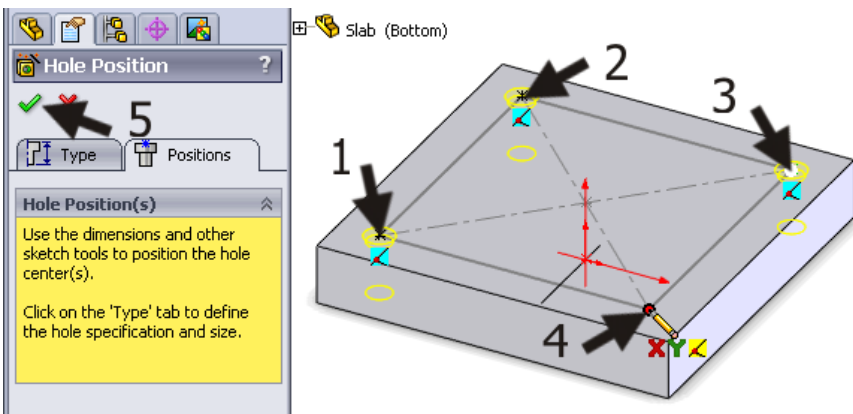
<p>17</p>	<p>Repeat these steps in the area named 'Direction 2'. For this purpose, select the other diagonal line.</p> <p>If the preview looks good to you, click on OK.</p>	 <p>The screenshot shows the 'Linear Pattern' dialog box with 'Direction 1' selected. The 'Line5@Sketch5' is chosen as the patterned feature. The spacing is set to 15.00mm and the number of instances is 3. The 'Features to Pattern' list includes 'Extrude2'. To the right, a 3D model of a rectangular plate shows a grid of holes. Arrows 1 through 6 indicate the sequence of actions: 1 points to the 'Features to Pattern' list, 2 points to the 'Direction 1' dropdown, 3 points to the 'Line5@Sketch5' selection, 4 points to the 'Spacing' field, and 5 and 6 point to the 'Instances' field. Two small pop-up boxes show the 'Direction 1' settings with 'Spacing' and 'Instances' fields.</p>
<p>18</p>	<p>We will now create the mounting holes for the bolts.</p> <p>Click on 'Hole Wizard' in the CommandManager.</p>	 <p>The screenshot shows the SolidWorks CommandManager with the 'Hole Wizard' button highlighted by an arrow. Below the CommandManager, a 3D model of the rectangular plate is shown with a grid of mounting holes.</p>

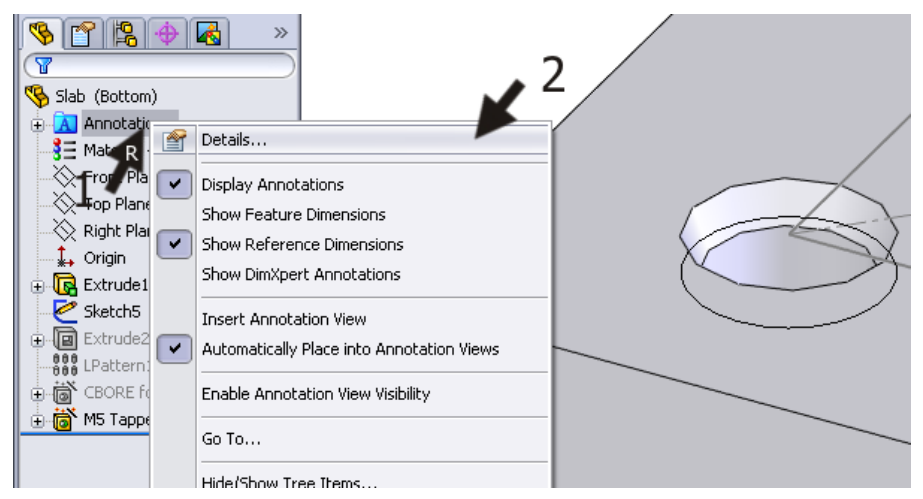
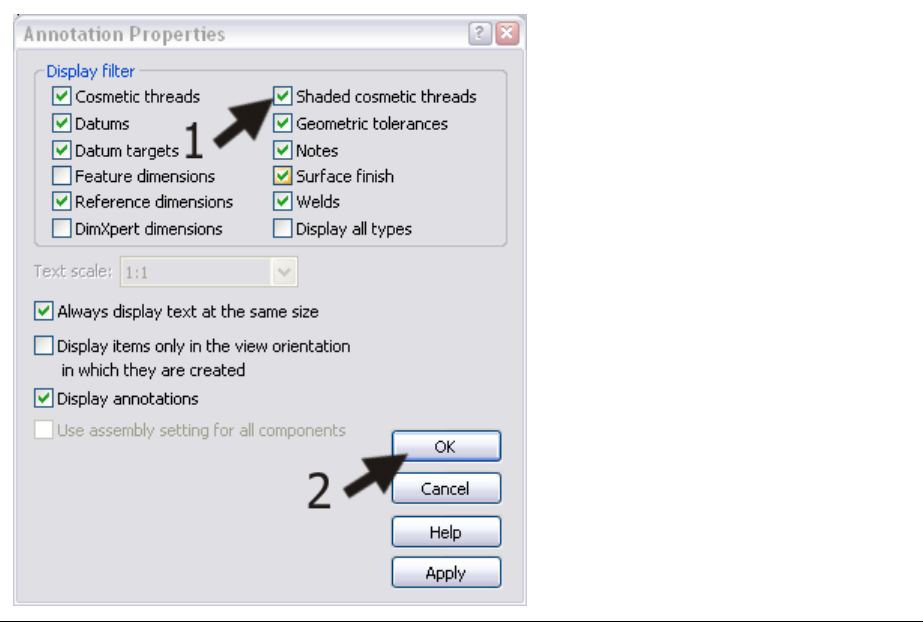
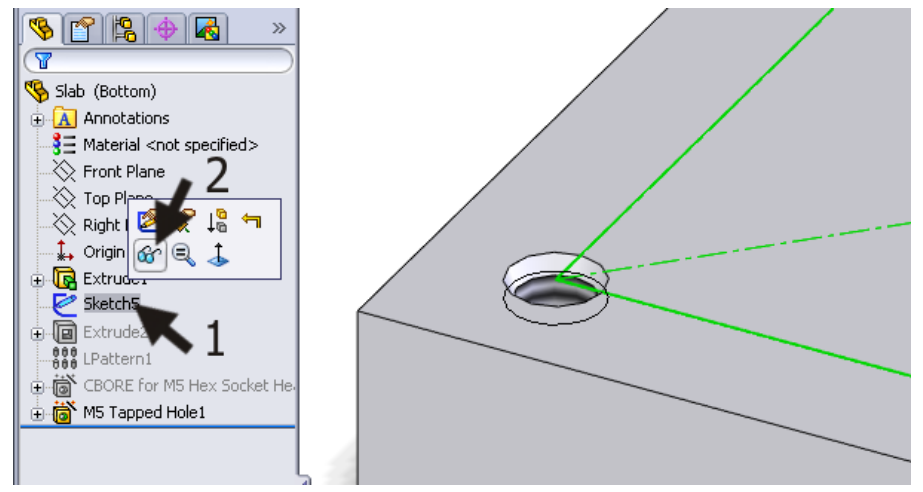
<p>19</p>	<p>Set the following features in the PropertyManager:</p> <ol style="list-style-type: none"> 1. Select the hole type Counter bore. 2. Set the Standard: 'ISO'. 3. Set Type: 'Hex Socket Head ISO 4762'. 4. Set Size: 'M5'. 5. Click on the 'Positions' tab. 	 <p>The screenshot shows the Hole Specification PropertyManager. The Hole Type section has the Counter Bore icon selected (arrow 1). The Standard is set to ISO (arrow 2). The Type is set to Hex Socket Head ISO 4762 (arrow 3). The Size is set to M5 (arrow 4). The Positions tab is selected (arrow 5). To the right is a 3D model of a rectangular plate with a 3x3 grid of 9 holes. A blue dimension line is shown on one of the holes.</p>
<p>20</p>	<p>Next, click at the four corners of the sketch to position the holes.</p> <p>Click on OK.</p>	 <p>The screenshot shows the Hole Position PropertyManager. The Type tab is selected (arrow 1). The Hole Position(s) section contains instructions: "Use the dimensions and other sketch tools to position the hole center(s)." and "Click on the 'Type' tab to define the hole specification and size." To the right is a 3D model of the plate with 9 holes. Four yellow circles with crosshairs are placed at the corners of the hole grid, with arrows 2, 3, and 4 pointing to them. Arrow 5 points to the OK button in the PropertyManager.</p>

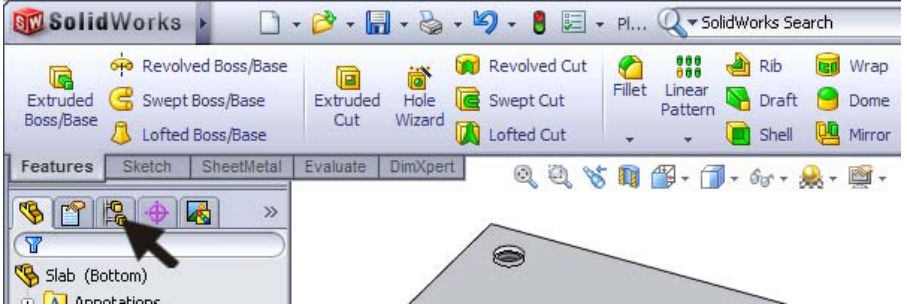
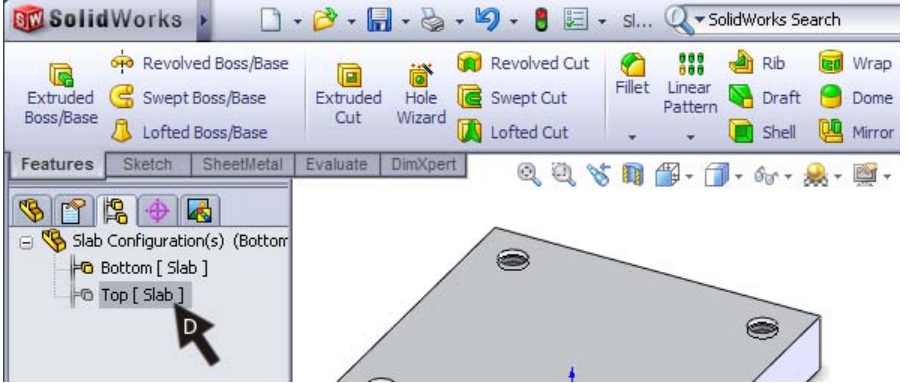
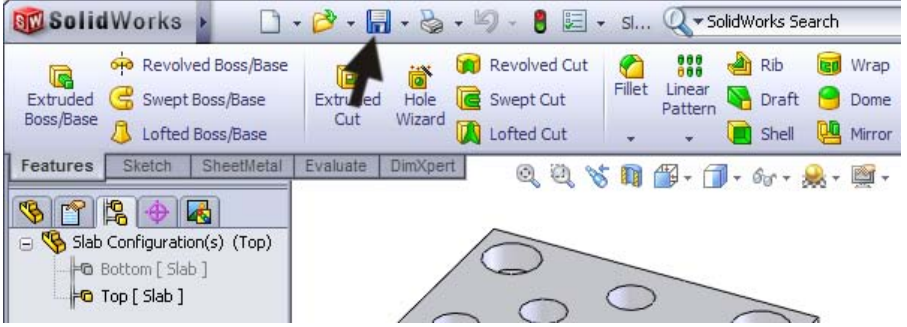
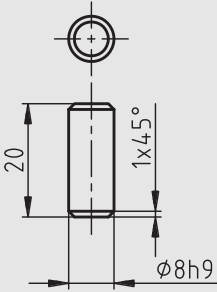
21	<p>The first part, the top plate, is now ready. Save this file as: Slab.SLDPRT.</p> <p>Tip: make a new folder on your computer first. You can arrange all of the files by product.</p>	
	<p>Work plan</p>	<p>We will now create the second part, the bottom plate. We will do this in accordance with the drawing below.</p>  <p>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 produce the second one.</p>
22	<p>Click on the Configuration-Manager tab.</p>	

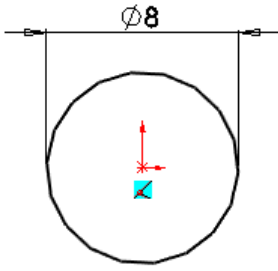
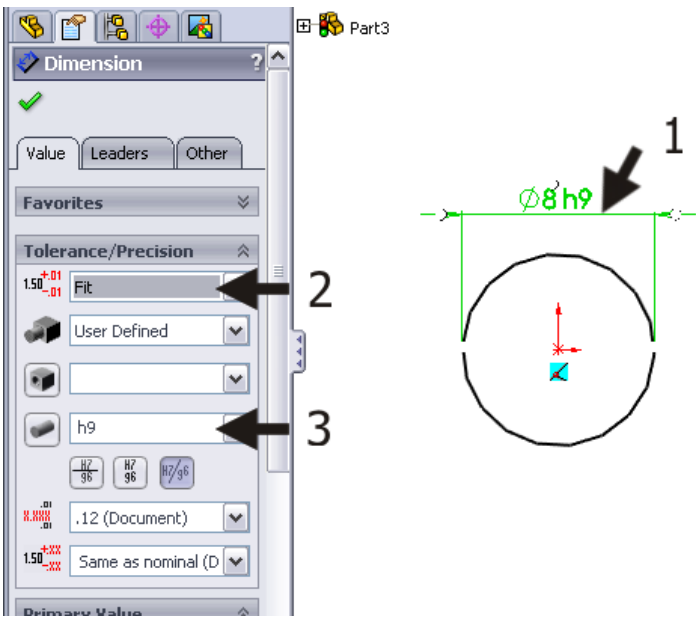
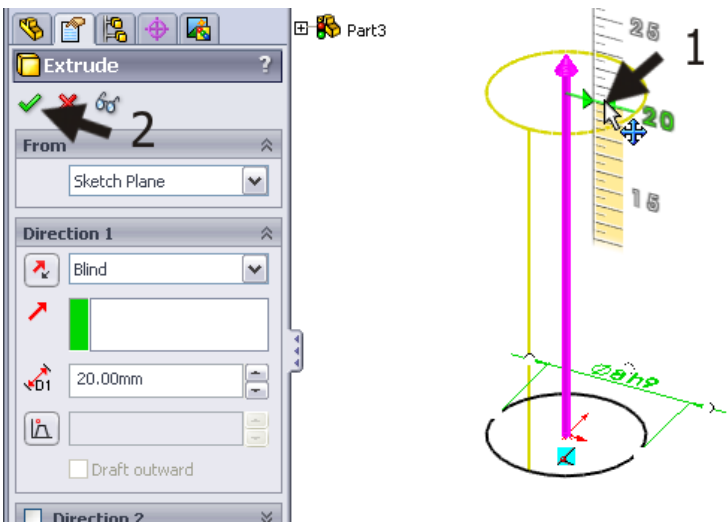
23	<p>The name of the configuration is 'Default'. Double-click on this name to change it to 'Top'.</p>	
24	<ol style="list-style-type: none"> 1. Click your right mouse button on the upper line in the ConfigurationManager. 2. Select 'Add Configuration' from the menu. 	
25	<ol style="list-style-type: none"> 1. Set the name of the new configuration to: 'Bottom'. 2. Click on OK. 	 
26	<p>There are two configurations in the list now: 'Top' (gray, non-active), and 'Bottom' (black, active). We will work with the active configuration.</p> <p>Click on the FeatureManager tab.</p>	 

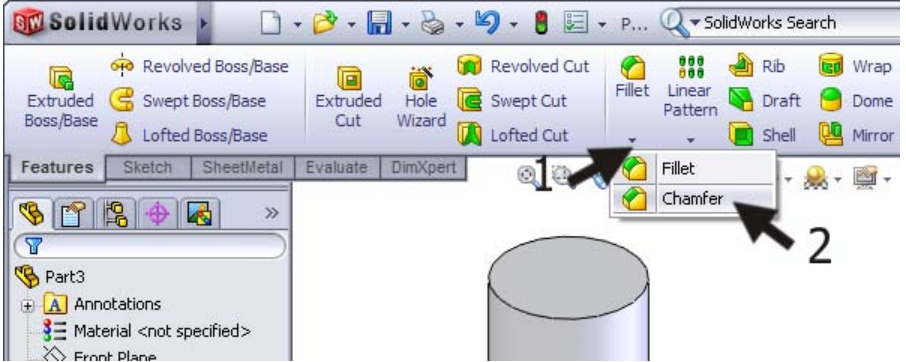
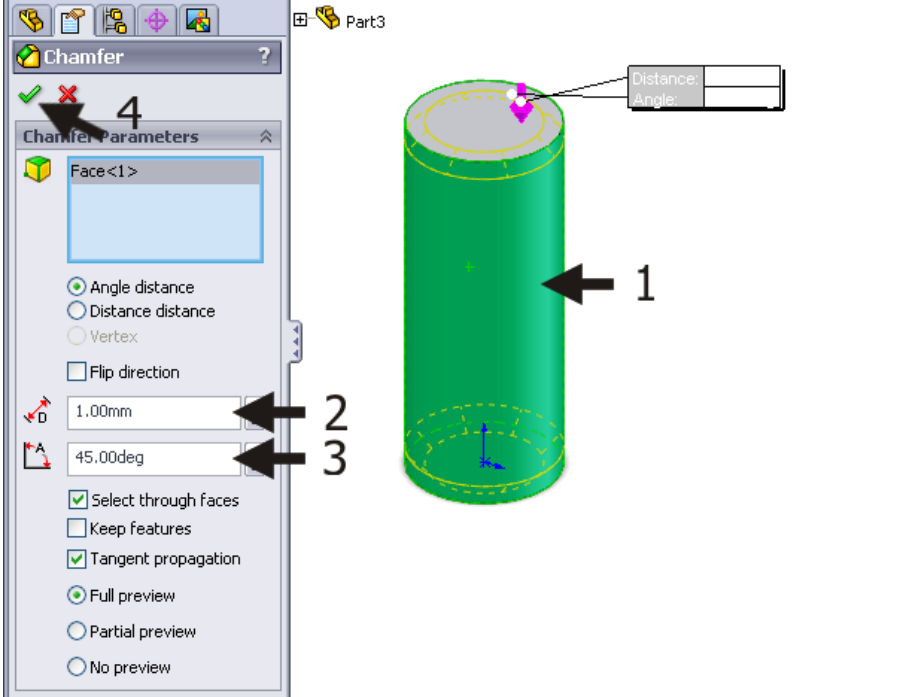
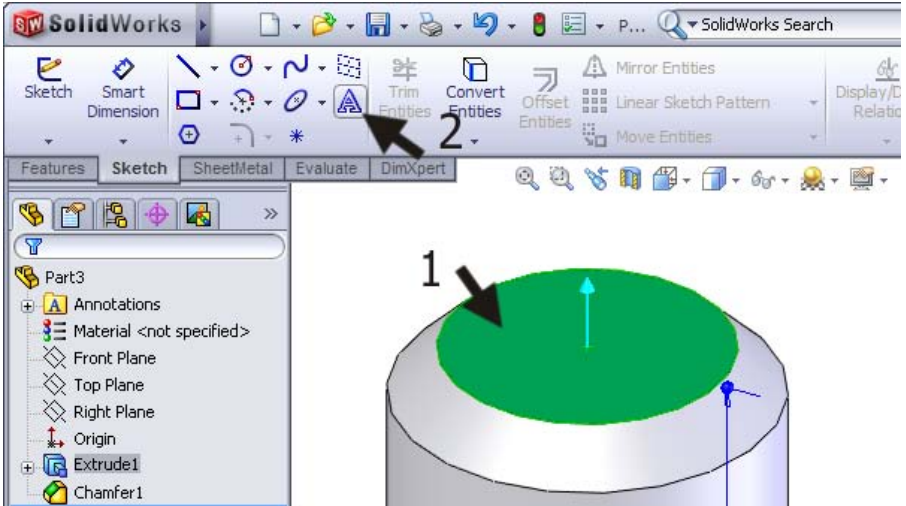
<p>27</p>	<p>Now Suppress the last three features that you just made:</p> <ol style="list-style-type: none"> 1. Click on the feature 'Extrude2'. 2. Hold the Shift key on the keyboard and click on the last feature. 3. Release the Shift key. The last three features are now selected, and a small options menu appears. 4. Select: Suppress in the menu. <p>All holes have disappeared from the model.</p>	
<p>28</p>	<p>Next, we will make some tapped holes with M5 thread.</p> <p>Click on the 'Hole Wizard' in the CommandManager.</p>	

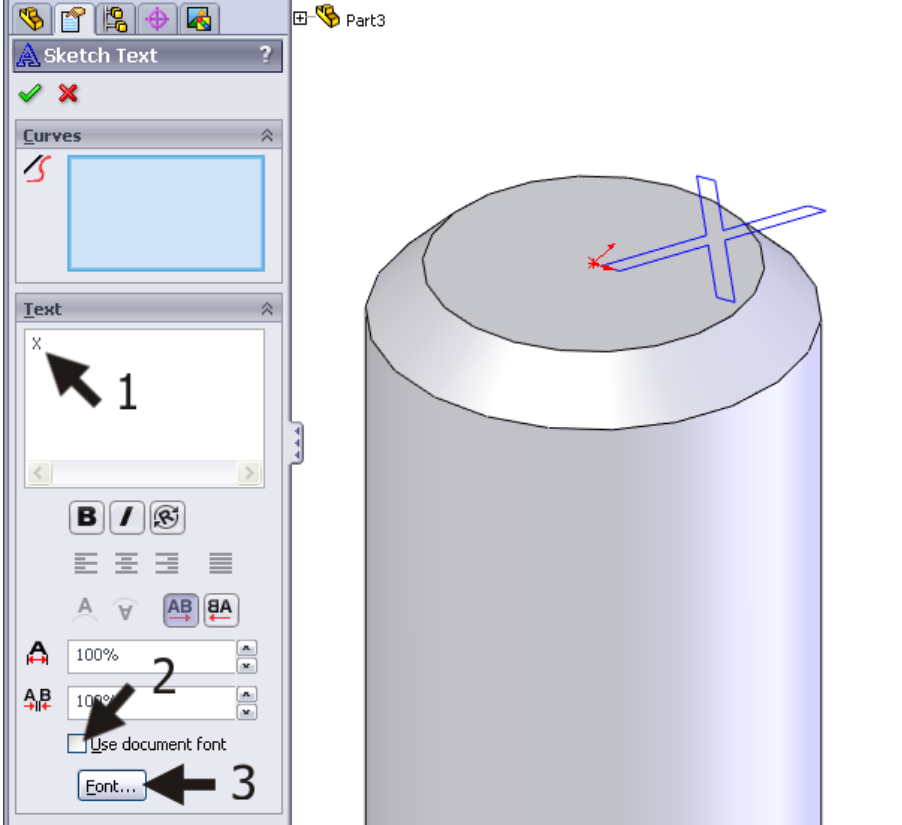
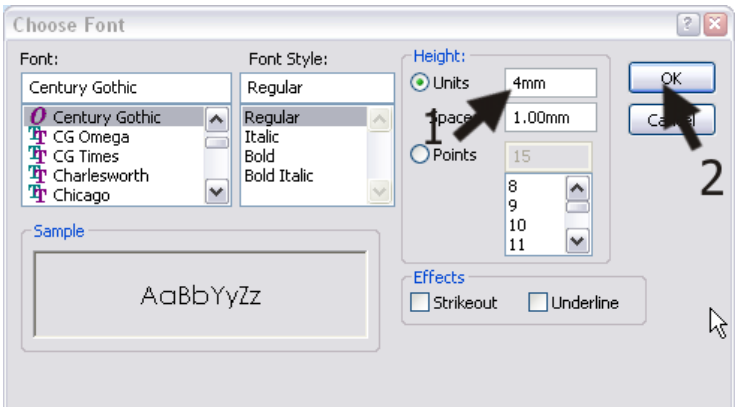
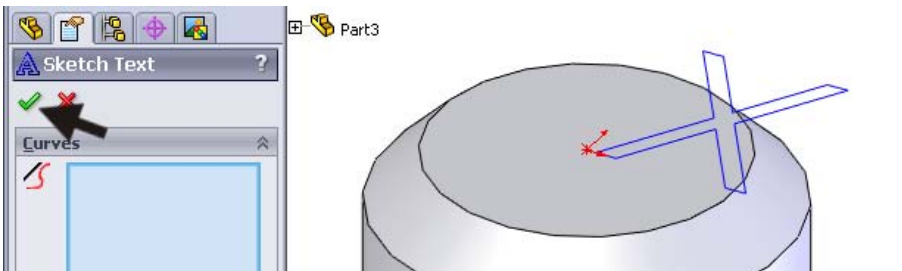
<p>29</p>	<p>Select the hole type Tap in the PropertyManager.</p> <p>Make sure all settings are equal to the settings in the illustration at right.</p> <p>Click on the 'Positions' tab.</p>	 <p>The screenshot shows the Hole Specification PropertyManager with the following settings:</p> <ul style="list-style-type: none"> Hole Type: Tap (indicated by arrow 1) Standard: ISO (indicated by arrow 2) Type: Tapped hole (indicated by arrow 3) Size: M5 (indicated by arrow 4) End Condition: Through All (indicated by arrow 5) Options: (indicated by arrow 6) <p>The 3D model shows a square hole on the top surface of a slab, with a coordinate system at the center. The hole is labeled 'Slab (Bottom)'.</p>
<p>30</p>	<p>Click on the four corners of the sketch to position the holes.</p> <p>Click on OK.</p>	 <p>The screenshot shows the Hole Position PropertyManager with the following settings:</p> <ul style="list-style-type: none"> Hole Position(s): Use the dimensions and other sketch tools to position the hole center(s). Click on the 'Type' tab to define the hole specification and size. (indicated by arrow 5) <p>The 3D model shows four holes positioned at the corners of the square hole on the top surface of a slab, with a coordinate system at the center. The holes are labeled 1, 2, 3, and 4.</p>

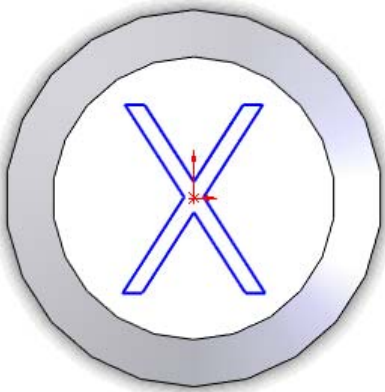
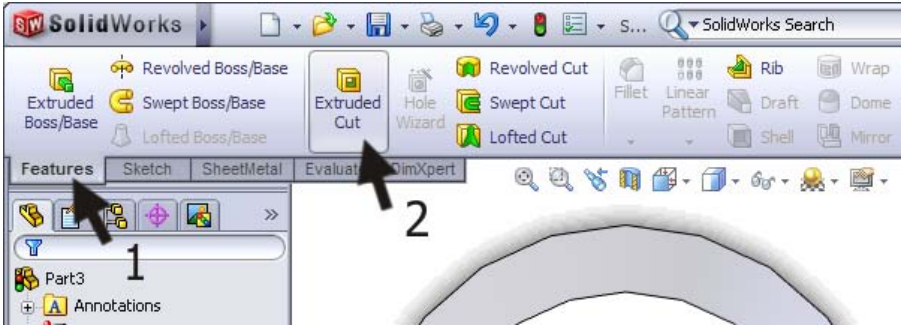
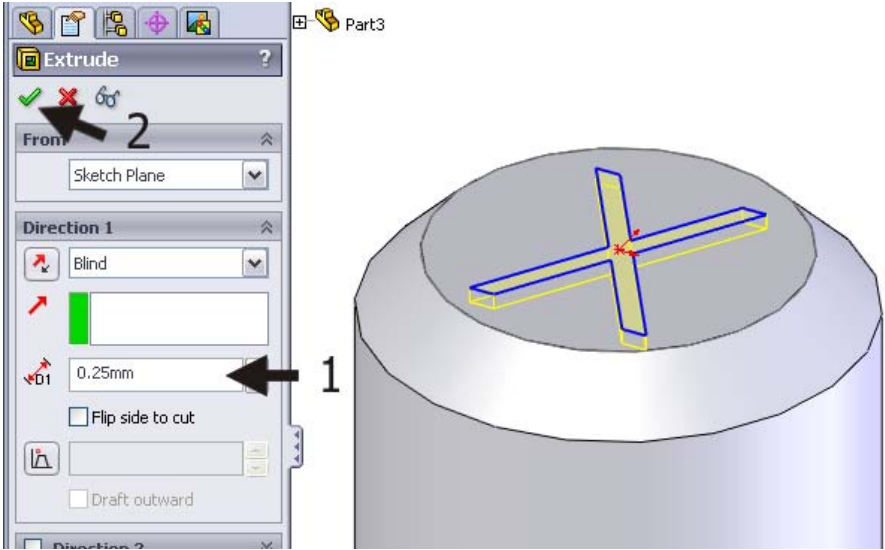
<p>31</p>	<p>Whenever no thread pattern appears in the holes, then change the following settings:</p> <ol style="list-style-type: none"> 1. Click the right mouse button on 'Annotations' in the FeatureManager. 2. Select 'Details'. 	
<p>32</p>	<ol style="list-style-type: none"> 1. Make sure that the option 'Shaded cosmetic threads' is checked. 2. Click on OK. 	
<p>33</p>	<p>Next, we want to hide the sketch we have used to make the holes:</p> <ol style="list-style-type: none"> 1. Click with the right mouse button on the 'Sketch' in the FeatureManager. 2. Select Hide in the menu. 	

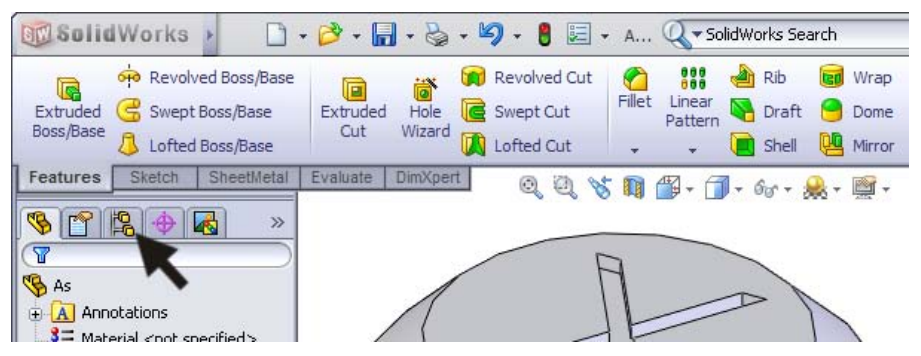
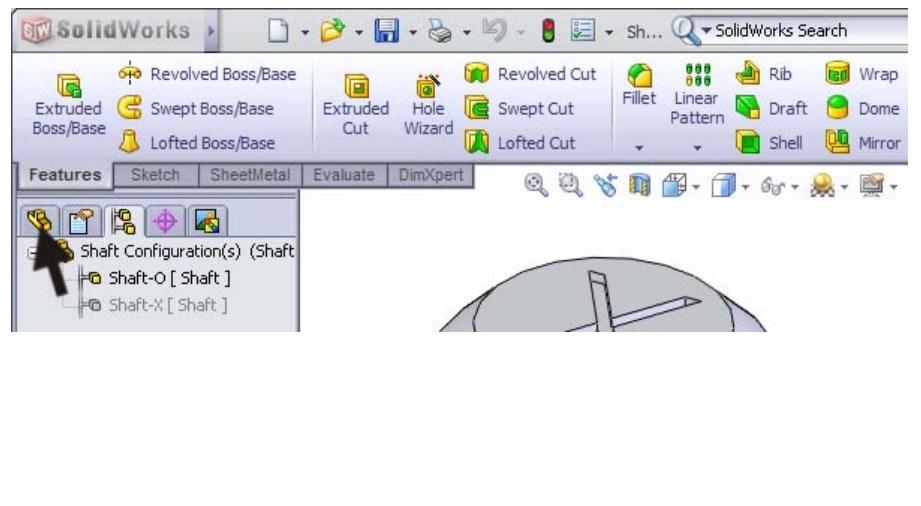
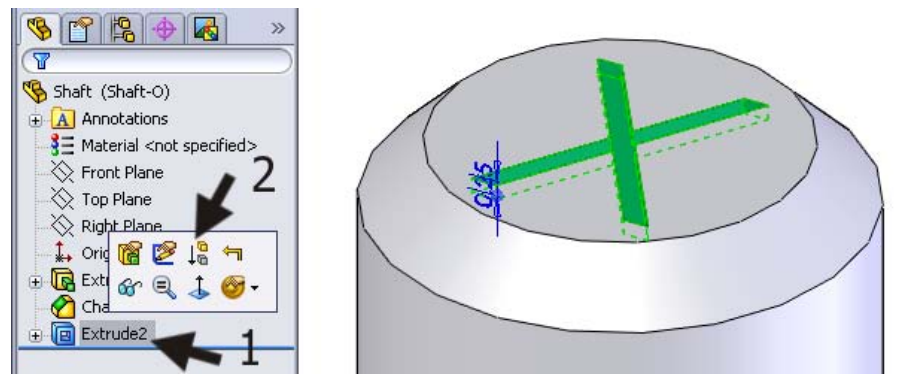
34	<p>Reactivate the configuration of the top plate.</p> <p>Click on the Configuration-Manager tab.</p>	
35	<p>Double-click on the configuration 'Top' in the ConfigurationManager.</p>	
36	<p>Save the file.</p>	
	<p>Work plan</p>	<p>The third part is the cylinder. We will create this by using the dimensions of the drawing below.</p>  <p>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 the cylinder.</p>
37	<p>Open a new part.</p>	

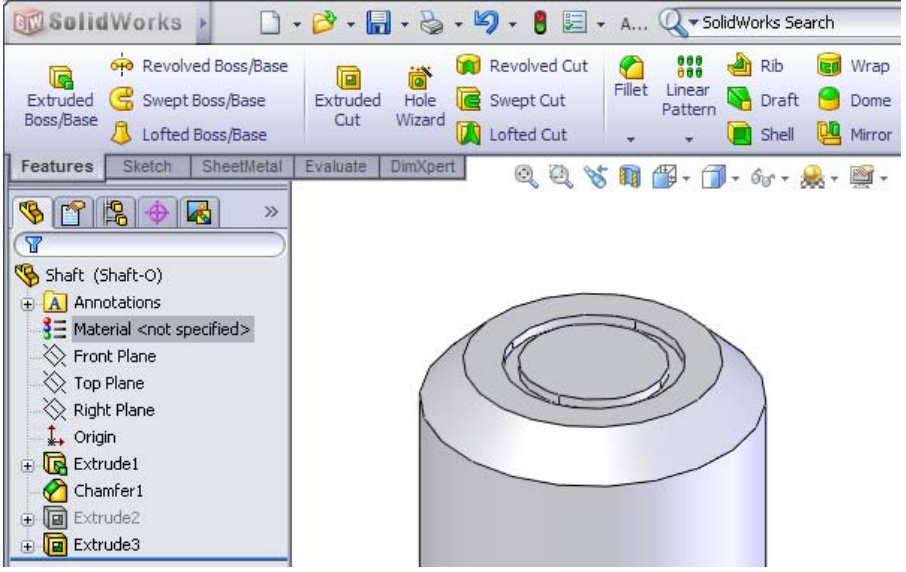
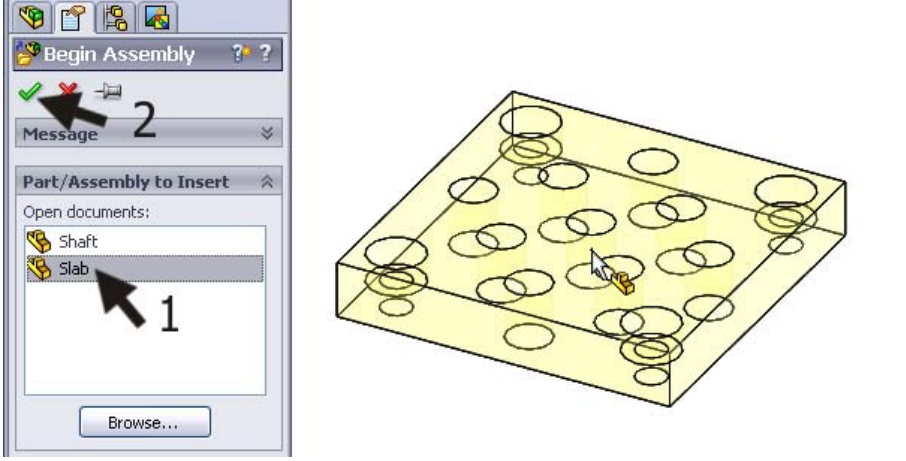
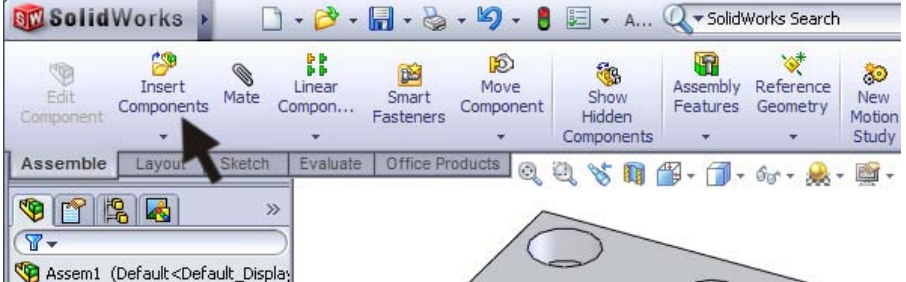
<p>38</p>	<p>Open a sketch in the Top plane.</p> <p>Draw a circle, with the center on top of the origin.</p> <p>Set a dimension $\varnothing 8$.</p>	
<p>39</p>	<p>Set the fitting to h9.</p> <ol style="list-style-type: none"> 1. Select the dimension. 2. Set the Tolerance type to fit in the Property-Manager. 3. Set Shaft fit to h9. 	
<p>40</p>	<ol style="list-style-type: none"> 1. Drag the height of the extrusion to 20mm. 2. Click on OK. 	

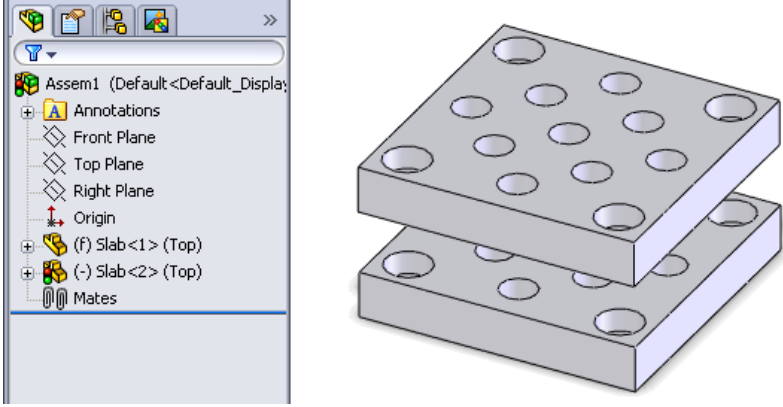
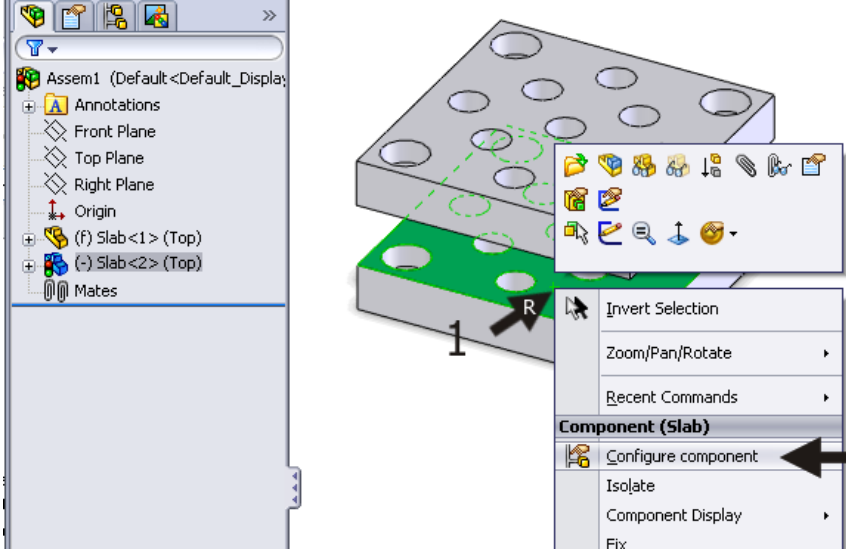
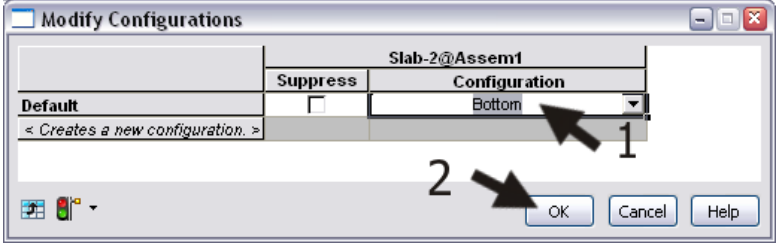
<p>41</p>	<p>We will now make an angled edge at the top and at the bottom of the cylinder with the Chamfer command.</p> <p>Click on 'Chamfer' in the CommandManager.</p>	 <p>The image shows the SolidWorks CommandManager. The 'Features' tab is active. The 'Fillet' dropdown menu is open, and 'Chamfer' is selected. A cylinder is shown below the CommandManager.</p>
<p>42</p>	<ol style="list-style-type: none"> 1. Click on the vertical outside plane of the cylinder. 2. Set the sloped distance to 1mm in the PropertyManager. 3. Check the angle to be 45°. 4. Click on OK. 	 <p>The image shows the SolidWorks PropertyManager for the Chamfer command. The 'Face<1>' is selected. The 'Angle distance' option is selected. The 'Distance' is set to 1.00mm and the 'Angle' is set to 45.00deg. A cylinder is shown with a chamfered edge.</p>
<p>43</p>	<ol style="list-style-type: none"> 1. Select the top plane of the cylinder. 2. Click on Sketch Text in the CommandManager. 	 <p>The image shows the SolidWorks CommandManager. The 'Sketch' tab is active. The 'Sketch Text' command is selected. A cylinder is shown with a sketch on the top plane.</p>

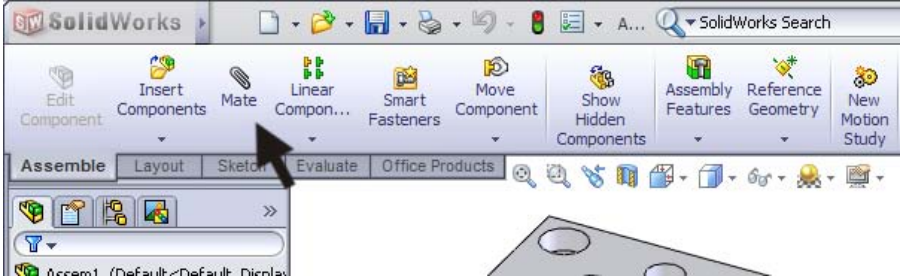
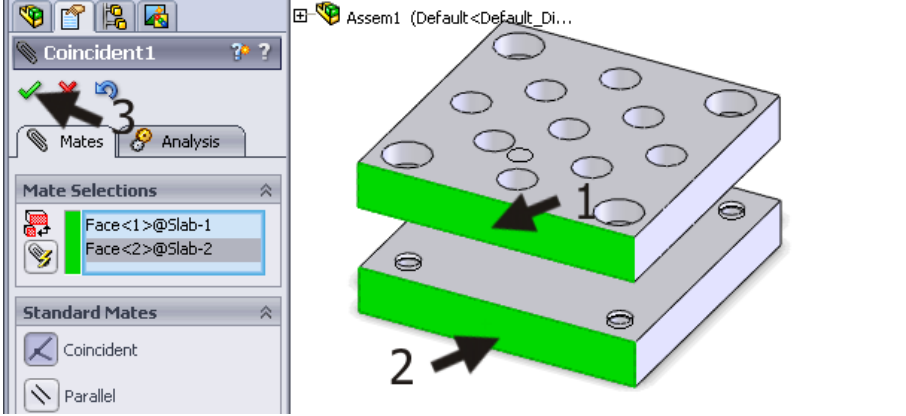
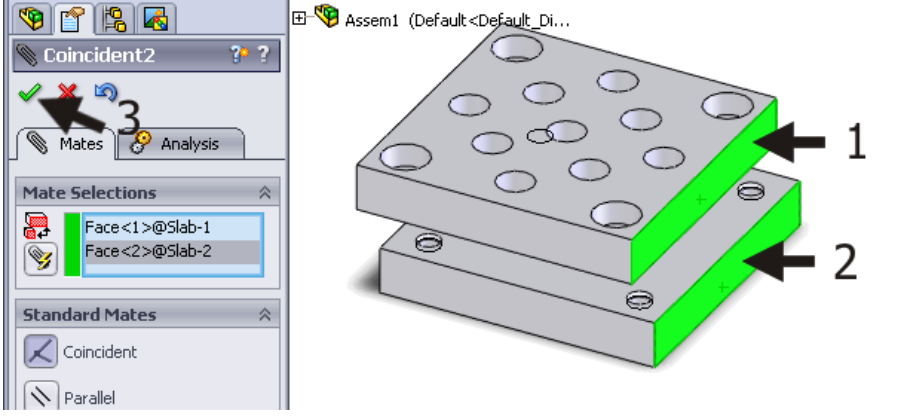
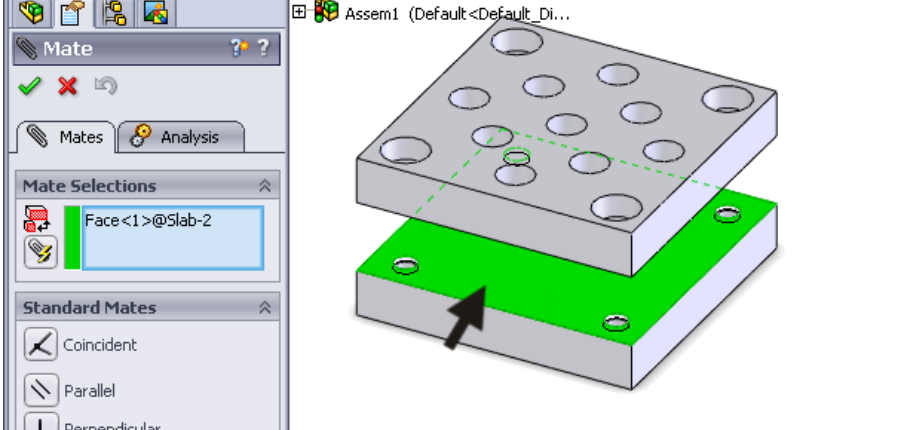
<p>44</p>	<ol style="list-style-type: none"> 1. Type in the capital 'X' in the text field. 2. Uncheck the option 'Use document font'. 3. Click on the 'Font...' button. 	
<p>45</p>	<p>Check in the menu to make sure the text height is set to 4mm, and click on OK.</p>	
<p>46</p>	<p>Click on OK in the PropertyManager.</p>	

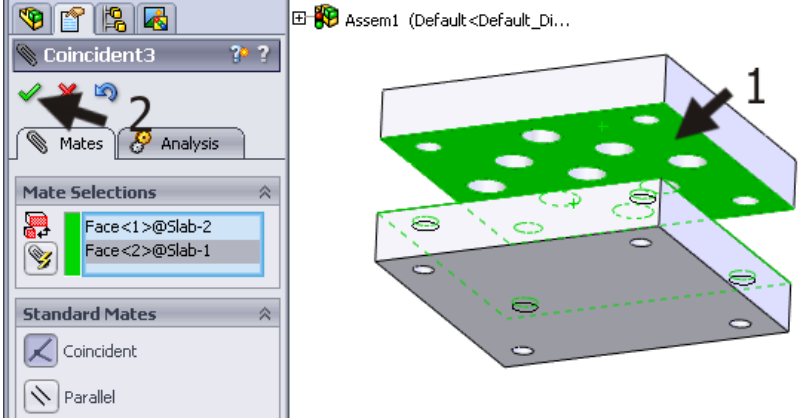
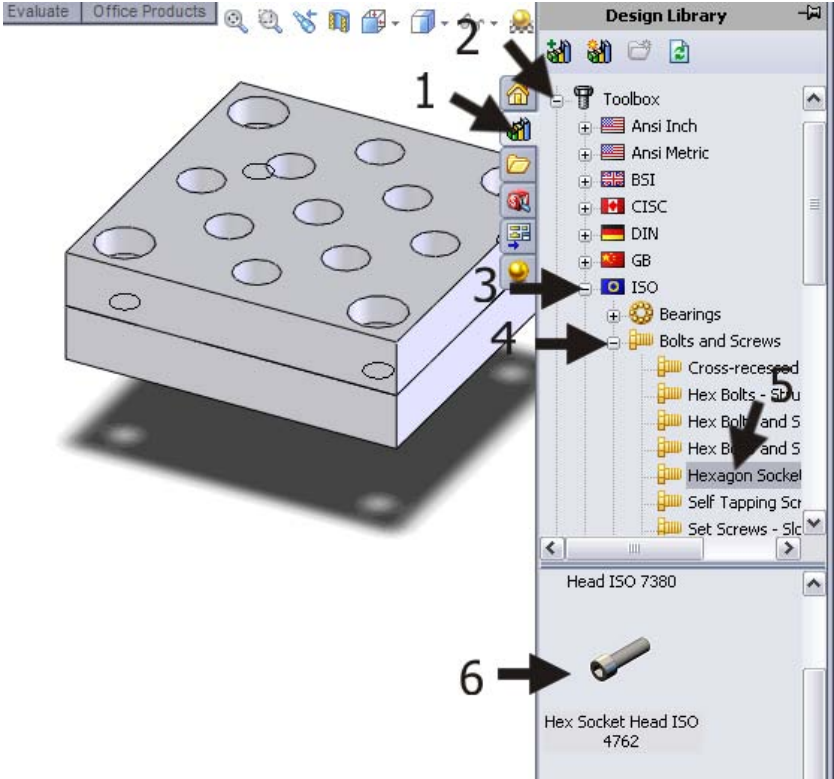
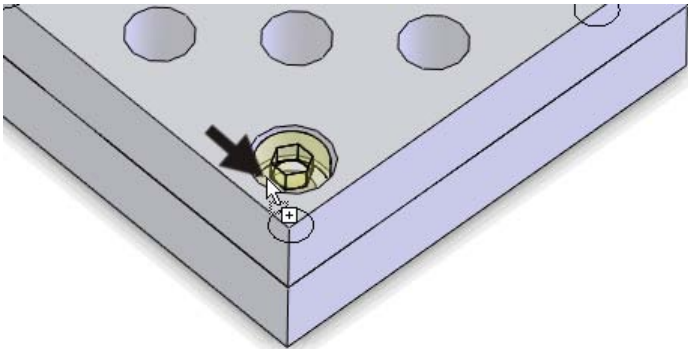
<p>47</p>	<p>Rotate the model with the Normal to command so you can get a good view of the sketch.</p> <p>Drag the letter to the centre of the plane.</p>	
<p>48</p>	<p>Click on 'Features' in the CommandManager and next on 'Extruded Cut'.</p>	
<p>49</p>	<ol style="list-style-type: none"> 1. Set the depth to 0.25mm. 2. Click on OK. 	
<p>50</p>	<p>The cylinder with the 'X' is now ready. Save the file as: Shaft.SLDPRT.</p>	

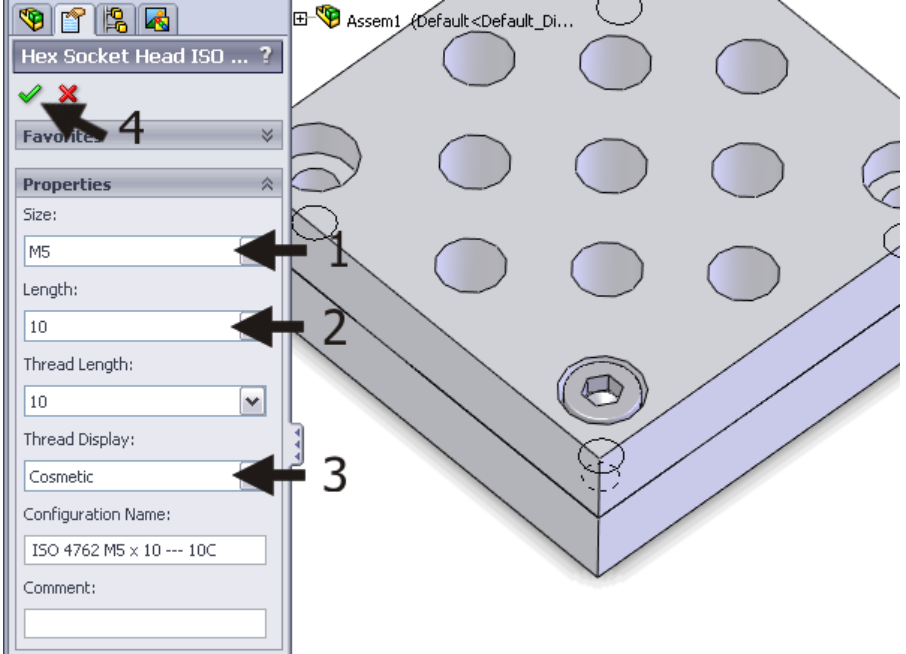
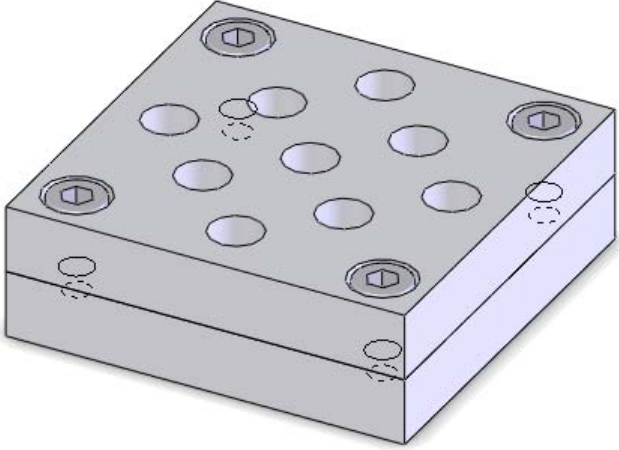

51	<p>To make the cylinder with the 'O' we will use a second configuration.</p> <p>Click on the Configuration-Manager tab.</p>	 <p>A screenshot of the SolidWorks Configuration Manager. The 'Configuration-Manager' tab is selected. The 'Features' tree on the left shows 'As' and 'Annotations'. The 'Features' list on the right shows 'Revolved Boss/Base', 'Extruded Boss/Base', 'Lofted Boss/Base', 'Extruded Cut', 'Hole Wizard', 'Revolved Cut', 'Swept Cut', 'Lofted Cut', 'Fillet', 'Linear Pattern', 'Rib', 'Draft', 'Dome', 'Shell', and 'Mirror'. A black arrow points to the 'Configuration-Manager' tab.</p>
52	<p>Change the name of the current configuration ('Default') to 'Shaft-X'.</p> <p>Create a new configuration called 'Shaft-O'.</p> <p>If necessary, compare these commands to steps 24 to 26.</p> <p>Check to make sure that the configuration 'Shaft-O' is active (black).</p> <p>Click on the FeatureManager tab.</p>	 <p>A screenshot of the SolidWorks Configuration Manager. The 'Configuration-Manager' tab is selected. The 'Features' tree on the left shows 'Shaft Configuration(s) (Shaft)', 'Shaft-O [Shaft]', and 'Shaft-X [Shaft]'. A black arrow points to the 'Shaft-O [Shaft]' configuration. The 'Features' list on the right is the same as in the previous screenshot.</p>
53	<p>With the 'Shaft-O' configuration active, we must hide the letter 'X'.</p> <ol style="list-style-type: none"> Click on the last features which you have made. Select Suppress in the menu that appears. 	 <p>A screenshot of the SolidWorks Feature Tree and a 3D model. The Feature Tree on the left shows 'Shaft (Shaft-O)', 'Annotations', 'Material <not specified>', 'Front Plane', 'Top Plane', 'Right Plane', 'Orig', 'Ext', 'Cha', and 'Extrude2'. A black arrow points to 'Extrude2' (labeled 1) and another black arrow points to the 'Suppress' button (labeled 2). The 3D model on the right shows a cylinder with a green cross on its top surface.</p>

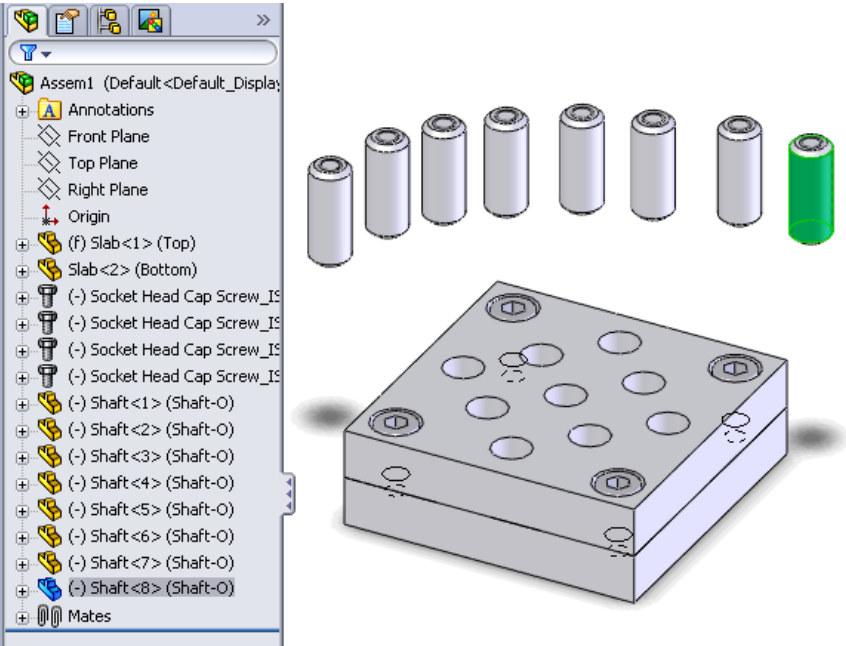
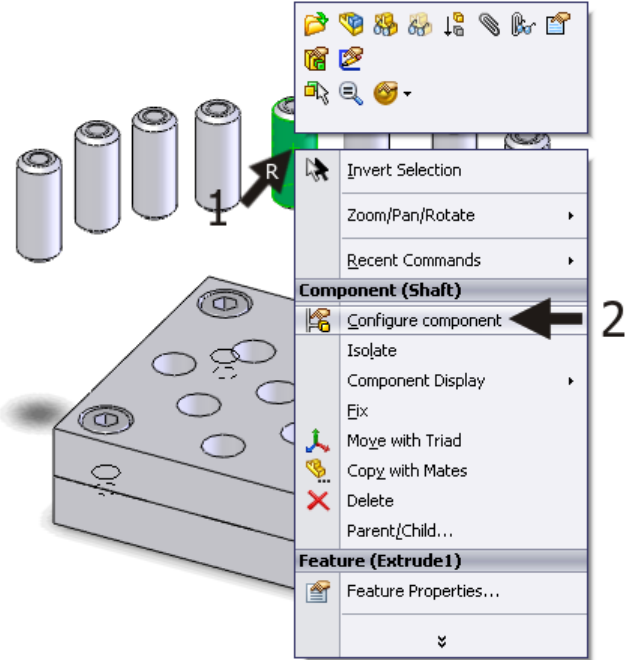
54	<p>Now, put a letter 'O' on the top plane of the cylinder. Do this in exactly the same way as you did before with the letter 'X' in steps 43 to 49.</p>	
55	<p>Save the file. Open a new assembly.</p>	
56	<p>When you did not close the two parts we just created (Slab and Shaft) you will see the image on the right.</p> <ol style="list-style-type: none"> 1. Click on the file 'Slab'. 2. Click on OK. <p>If you did close this file, find it with the 'Browse...' command.</p>	
57	<p>Click on 'Insert Components' in the CommandManager.</p>	

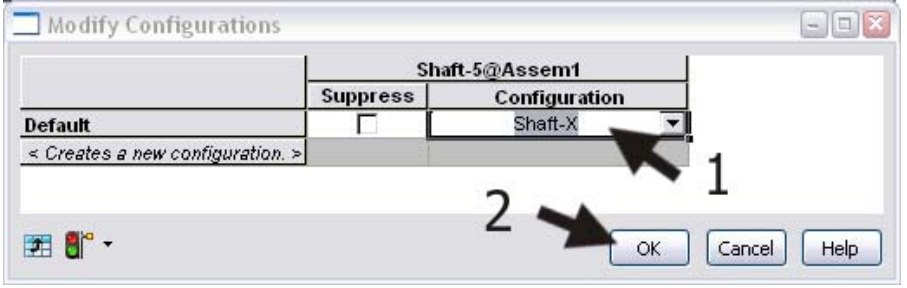
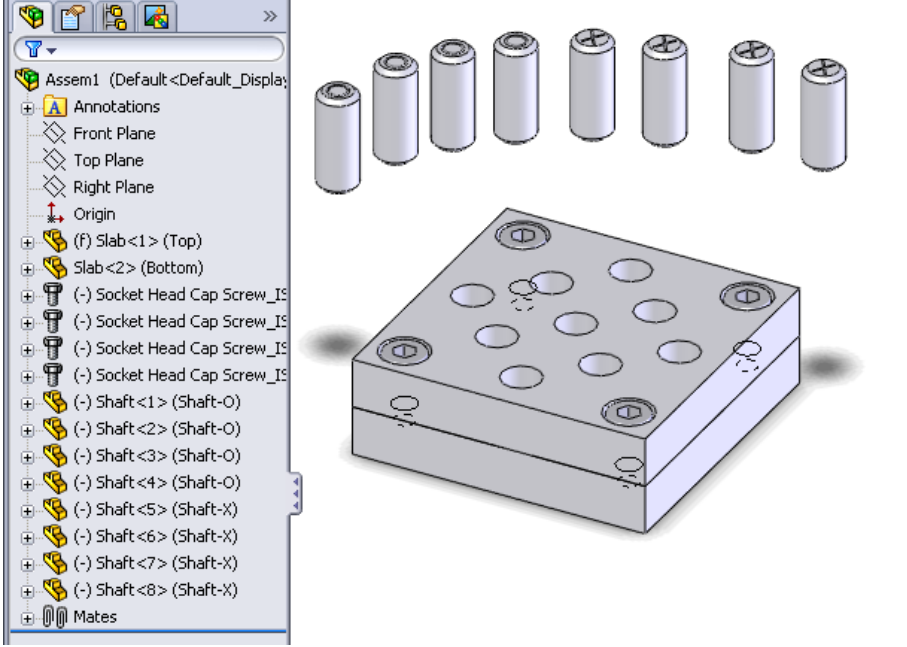

58	Add the same part again. Place it just below the first one.	
59	<p>Next, we have to change the configuration of the bottom plate.</p> <ol style="list-style-type: none"> 1. Click with the right mouse button somewhere on the bottom plate. 2. Select 'Configure Component' in the menu that appears. 	
60	<ol style="list-style-type: none"> 1. Select the Configuration 'Bottom'. 2. Click on OK. 	
	<p>Tip!</p>	<p>When a part is open while added to an assembly, you can only select the desired configuration AFTER putting it in the assembly. That is what we have just done.</p> <p>When a part is closed, click on the PropertyManager and Browse to find it (see step 56). In the menu that appears, you can select the right configuration directly. Therefore, sometimes it is more convenient to use the Browse-function anyway, even though the part is open.</p>

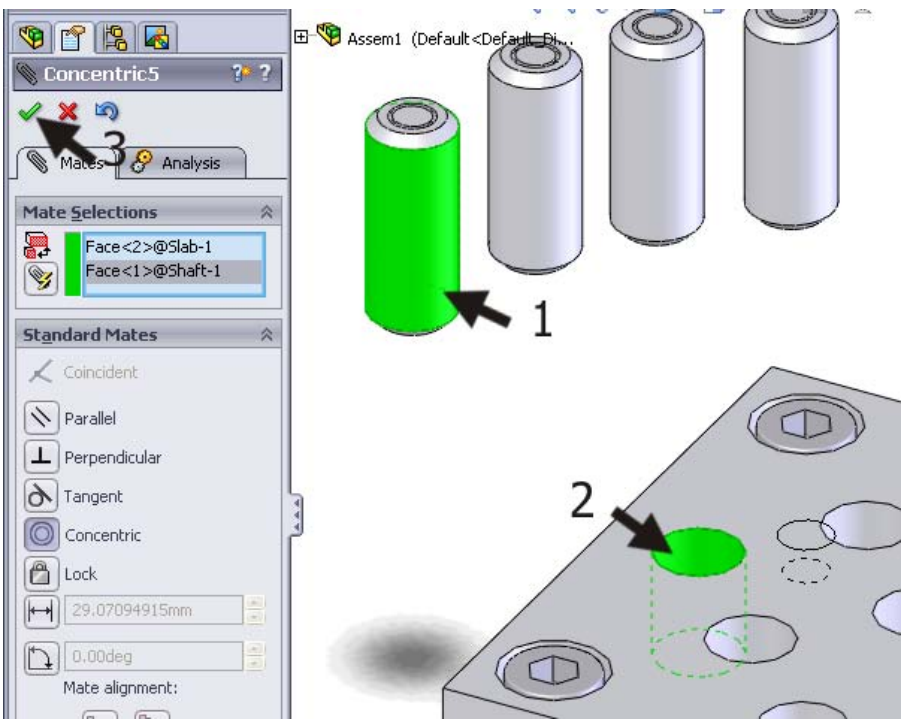
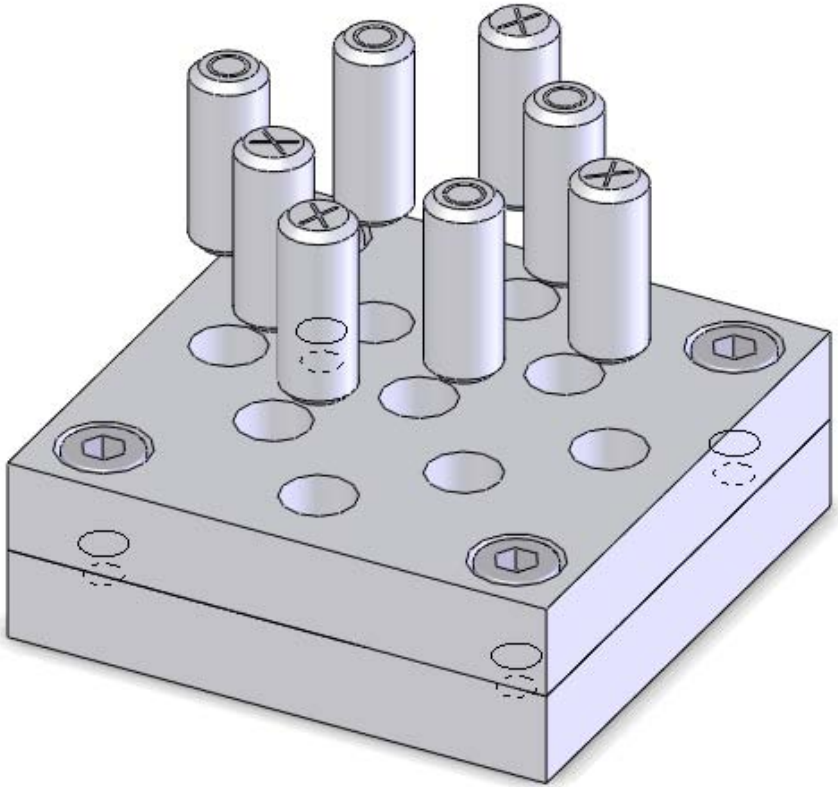
61	<p>Next, we have to align the two parts with the mate command.</p> <p>Click on 'Mate' in the CommandManager.</p>	
62	<p>Select the sides of both parts as shown in the illustration.</p> <p>Click on OK.</p>	
63	<p>Select two other sides of both parts as shown in this illustration.</p> <p>Click on OK.</p>	
64	<p>Select the top plane of the bottom part.</p>	

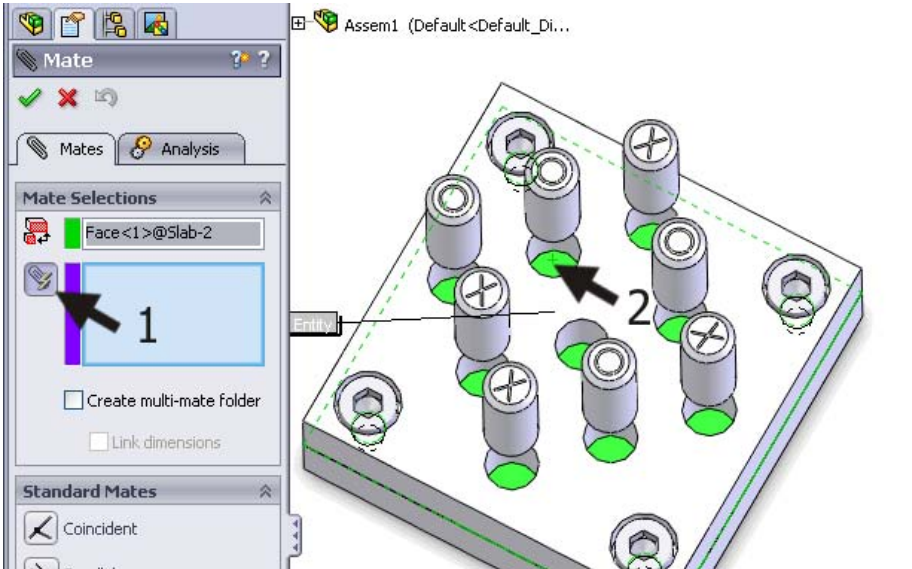
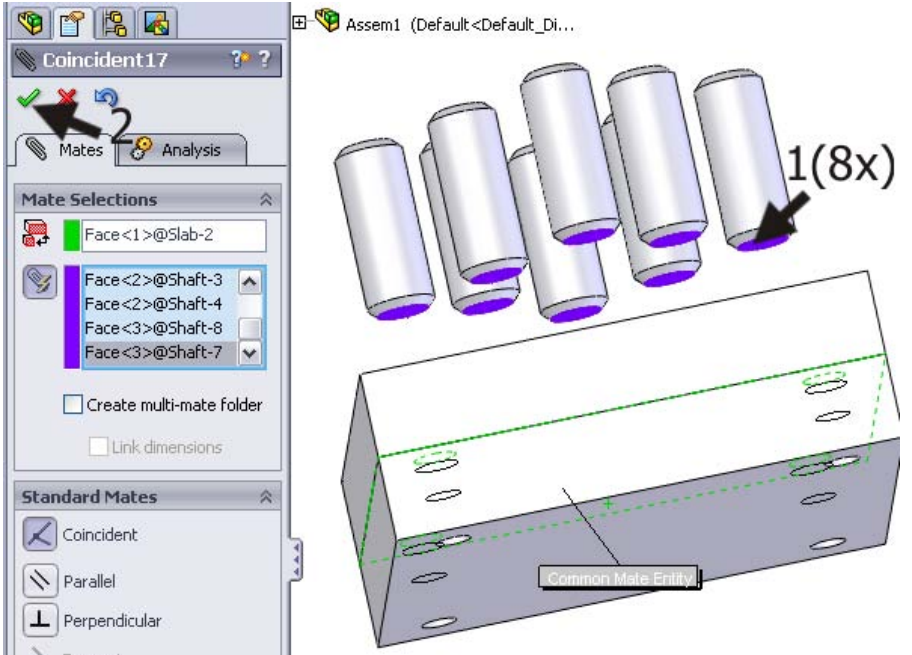
<p>65</p>	<p>Next rotate the model so you get a good view of the bottom of the top part and select the bottom plane.</p> <p>Double-click on OK.</p>	
<p>66</p>	<p>Next we will put the hexagon socket head screws in the model.</p> <ol style="list-style-type: none"> 1. Open the Design Library in the Task Pane. 2. Click on 'Toolbox'. 3. 'ISO'. 4. 'Bolts and Screws'. 5. 'Hexagon Socket Head Screws'. 6. Select: 'Hex Socket Head ISO 4762'. 	
<p>67</p>	<p>Drag the bolt to your model. Release the mouse button at the lower edge of one of the countersink holes.</p>	

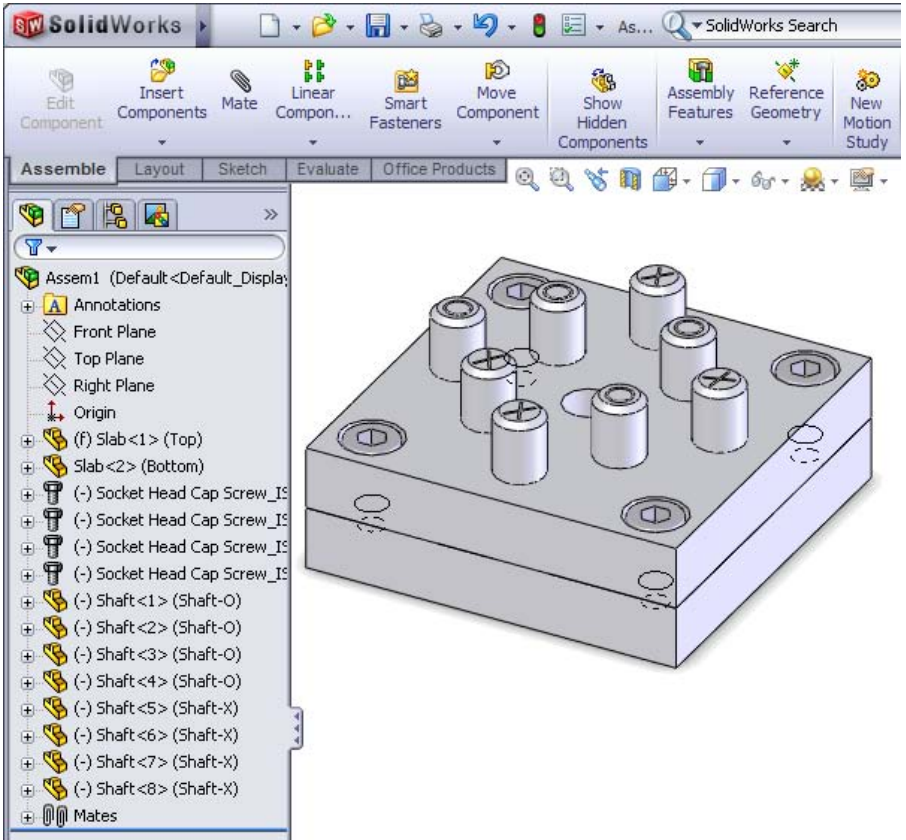
<p>68</p>	<p>Set the following features in the PropertyManager:</p> <ol style="list-style-type: none"> 1. 'Size': 'M5'. 2. 'Thread Length': '10'. 3. 'Thread Display': 'Cosmetic'. 4. Click on OK. 	
<p>69</p>	<p>Put hexagon head screws in the other holes as well.</p>	
<p>70</p>	<p>Finally, the cylinders (pegs) should be placed in the holes.</p> <p>Click on 'Insert Components' in the CommandManager.</p>	

<p>71</p>	<p>Place the cylinder or peg in the assembly 8 times at a random position.</p> <p>Note that it does not matter if you pick an 'X' or 'O' cylinders. We will change four of them later.</p>	
<p>Tip!</p>		<p>You can use the Insert Components command 8 times to insert the pegs, but it is 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.</p>
<p>72</p>	<p>Next, we will change the letter on four of the pegs.</p> <p>Right-click on a peg and select 'Configure component'.</p>	

73	<p>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 the other way around.</p> <p>2. Click on OK.</p>	
74	<p>Repeat this step for three other pegs.</p>	
75	<p>Next, we have to mate the pegs in the holes.</p> <p>Click on 'Mate' in the CommandManager.</p>	

<p>76</p>	<p>Select the two planes as shown in the illustration on the right.</p> <p>Click on OK.</p>	
<p>77</p>	<p>Repeat the last step for all the pegs and select a different hole for every peg. The height of the pegs is not yet been determined. You can still move all of the pegs up and down by dragging them.</p>	

<p>78</p>	<p>We will make the final mate now.</p> <ol style="list-style-type: none"> 1. Click on the Multiple Mate Mode in the PropertyManager. 2. Rotate the model so you get a good view of the INSIDE of a hole. Through the hole you can see the top plane of the bottom part. Select this plane. 	 <p>The screenshot shows the SolidWorks Mates PropertyManager on the left. The 'Mate' tab is active, and 'Multiple Mate Mode' is selected. The 'Mate Selections' list shows 'Face<1>@Slab-2' selected. The 'Standard Mates' list shows 'Coincident' selected. On the right, a 3D model of a Tic Tac Toe board assembly is shown. The board is a rectangular slab with a grid of holes. Eight pegs are already inserted into the holes. A dashed green line indicates the top plane of the bottom part of the board. An arrow labeled '1' points to this plane. Another arrow labeled '2' points to one of the pegs.</p>
<p>79</p>	<p>Rotate the model again so you can see the bottom side of the pegs.</p> <ol style="list-style-type: none"> 1. Select the bottom side of all pegs. 2. Click on OK. 	 <p>The screenshot shows the SolidWorks Mates PropertyManager on the left. The 'Coincident17' tab is active. The 'Mate Selections' list shows 'Face<1>@Slab-2' and 'Face<2>@Shaft-3', 'Face<2>@Shaft-4', 'Face<3>@Shaft-8', and 'Face<3>@Shaft-7' selected. The 'Standard Mates' list shows 'Coincident', 'Parallel', and 'Perpendicular' options. On the right, a 3D model of the Tic Tac Toe board assembly is shown. The board is rotated to show the bottom side of the pegs. A dashed green line indicates the bottom plane of the board. An arrow labeled '1(8x)' points to the bottom side of one of the pegs. A label 'Common Mate Entity' points to the dashed green line.</p>

80	<p>The assembly is ready now. Save the file as: Tic-tactoe.SLDASM.</p>	
	<p>What are the main features you have learned in this tutorial?</p>	<p>In this tutorial we have repeated a lot of what we have seen and done before:</p> <ul style="list-style-type: none"> • Creating simple parts and shapes. • Working with configurations. • Working with standard parts. • Working with the Hole Wizard. <p>We have also learned some new topics:</p> <ul style="list-style-type: none"> • You have set fittings at holes and/or pegs. • You have seen how to use text in a sketch. • You have learned some new tricks.

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