SolidWorks® Tutorial 3

MAGNETIC BLOCK

Preparatory Vocational Training
and Advanced Vocational Training

To be used with SolidWorks® Educational Release 2008-2009

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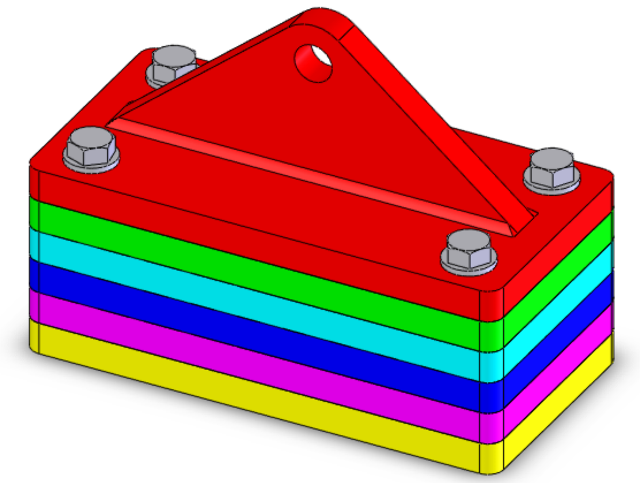
Magnetic Block

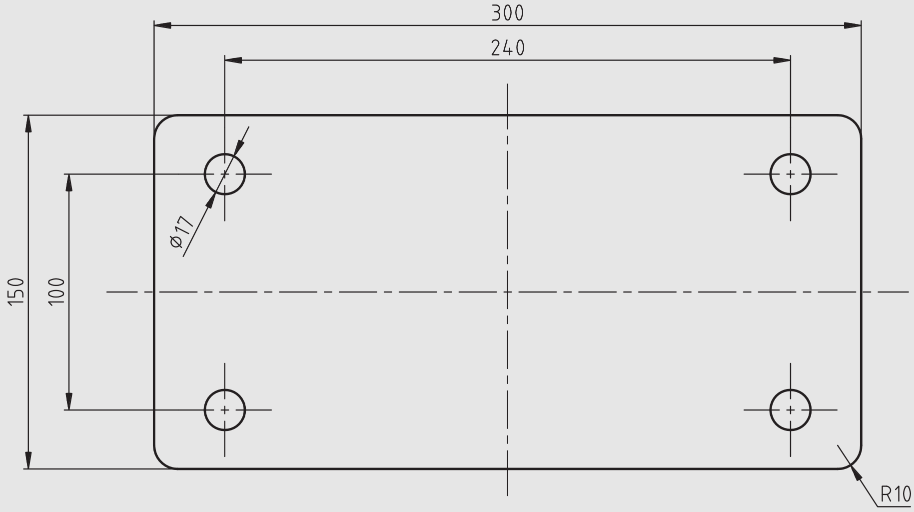
In this exercise you will make a magnetic block. To do so, you will create a few parts, which you will assemble. You will learn the following new applications in this tutorial:

- You will make two configurations of a part.
- You will weld the parts together.
- You will make holes using the Hole Wizard.
- You will use standardized parts from the Parts Library.
- You will assign different colors to different parts.



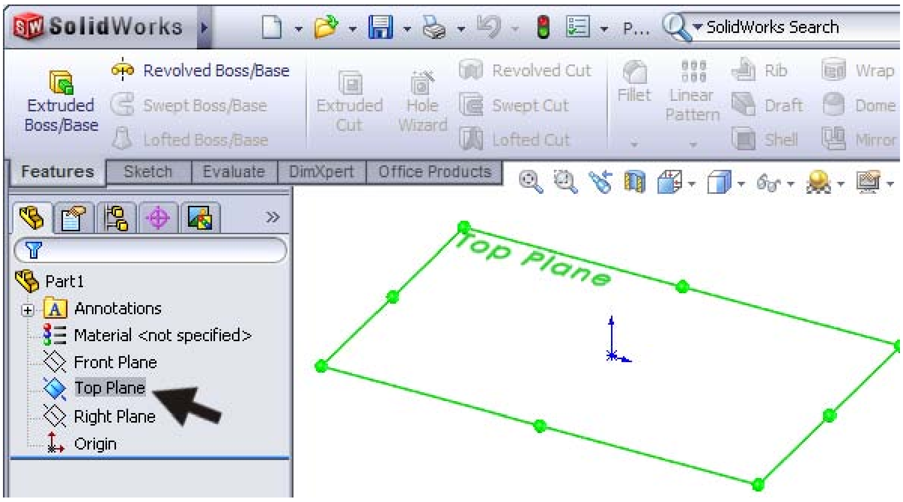
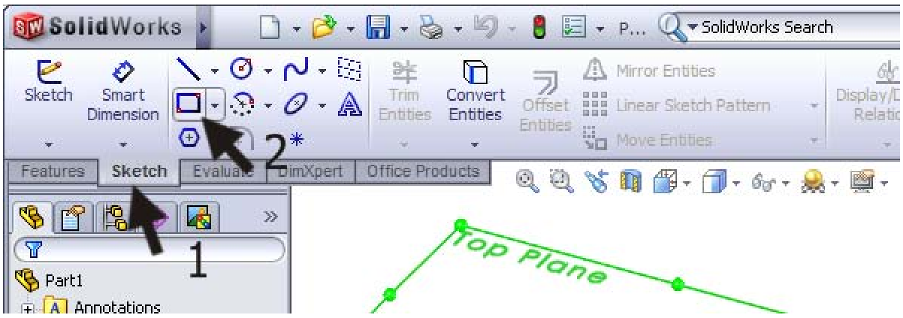
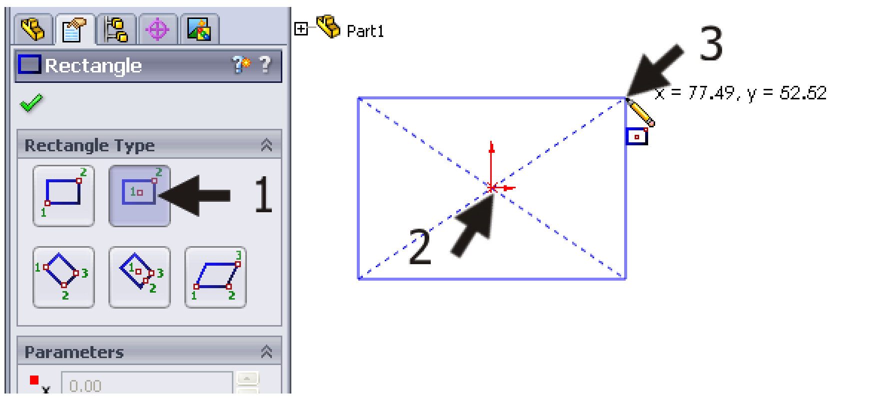
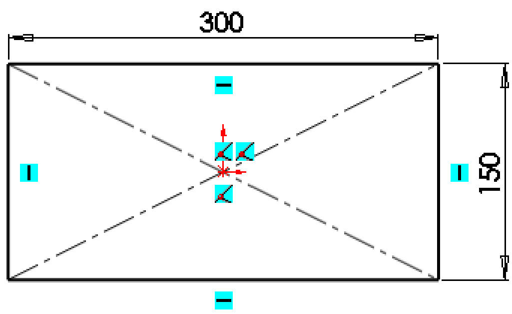
Work plan

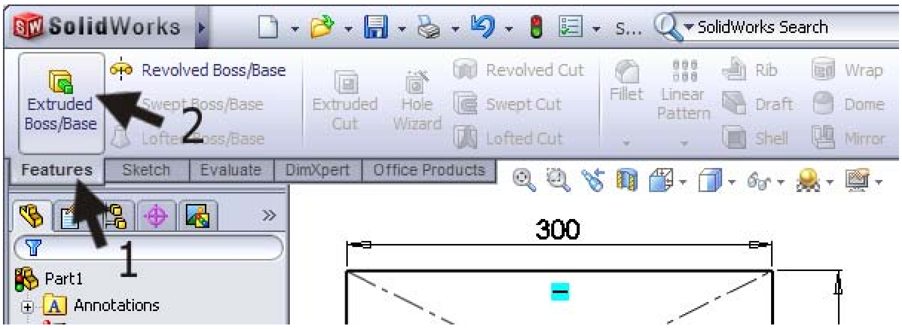
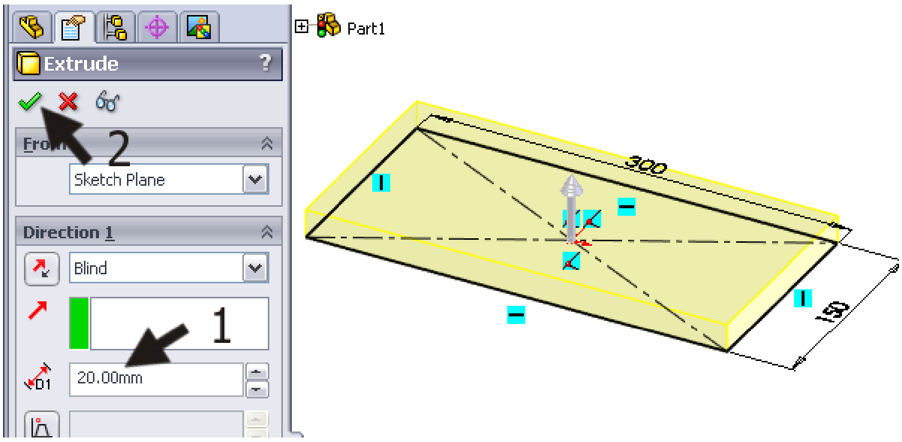
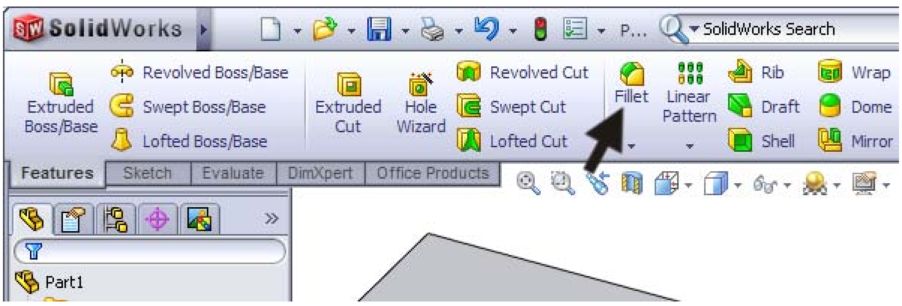
To make this assembly, you will have to make several parts. We will start with a simple rectangular base with a thickness of 20mm per the drawing below.

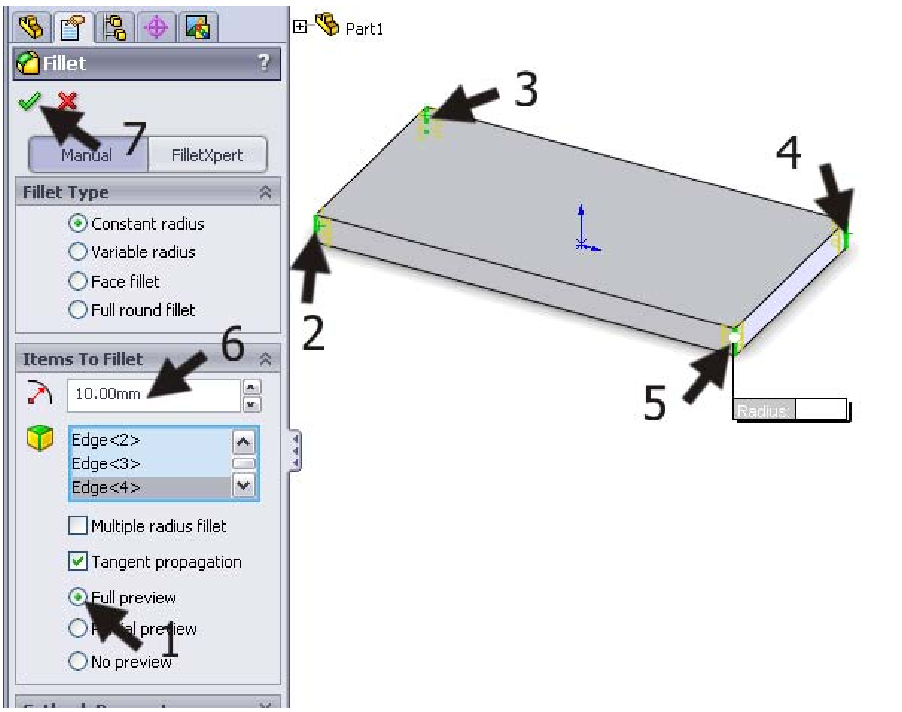
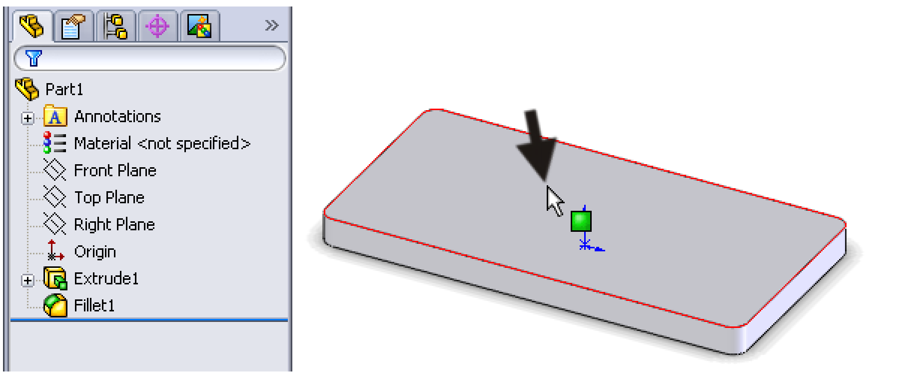
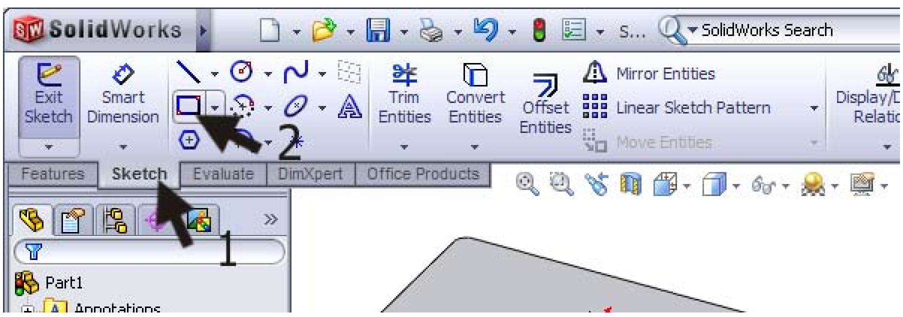


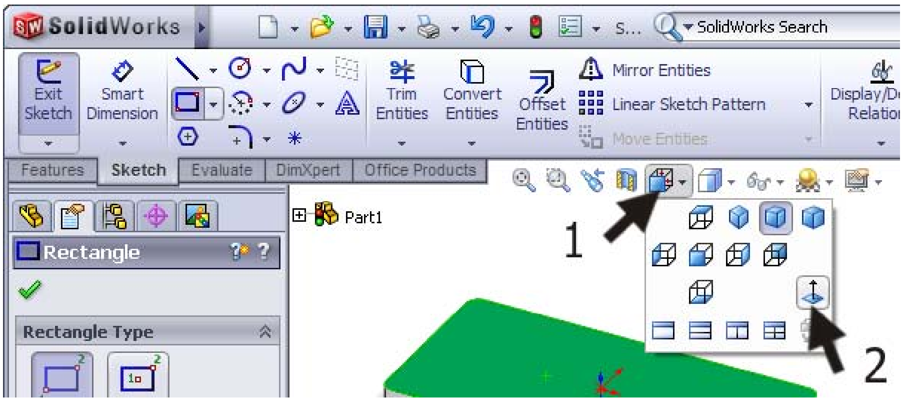
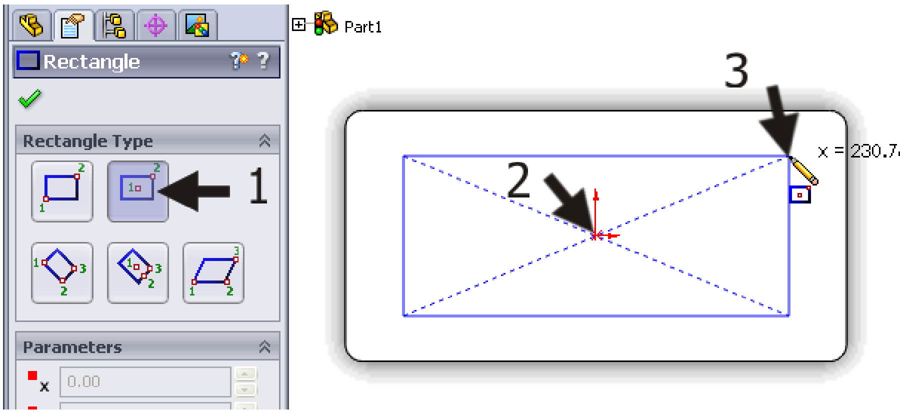
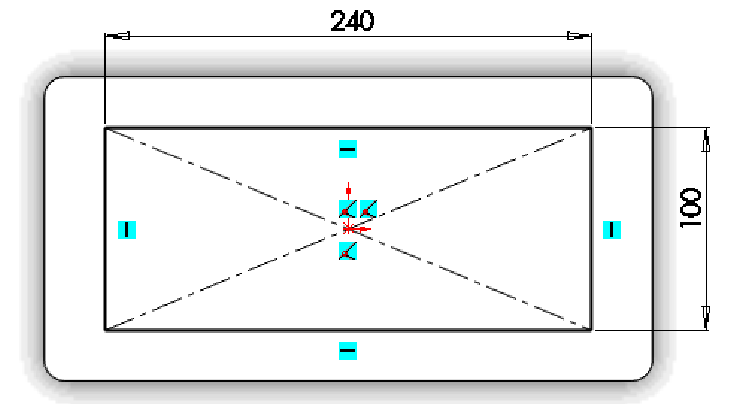
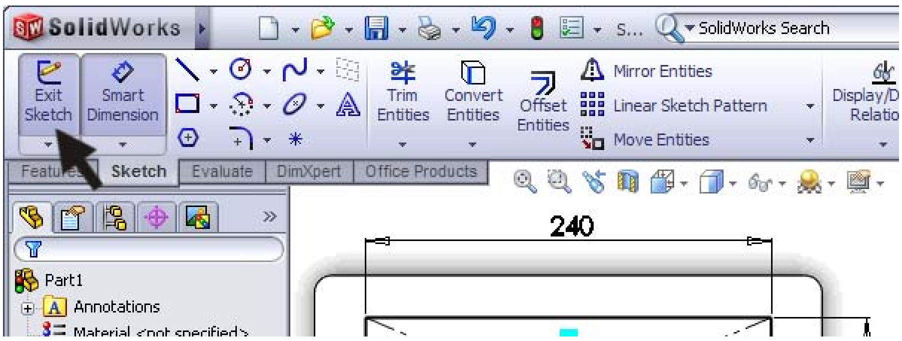
We will perform the following steps:

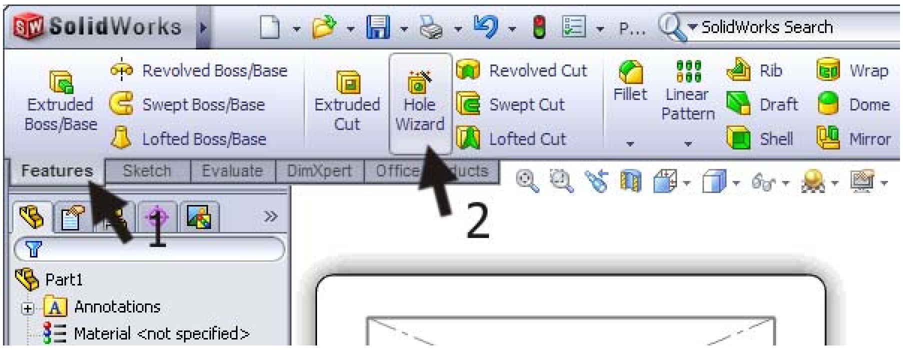
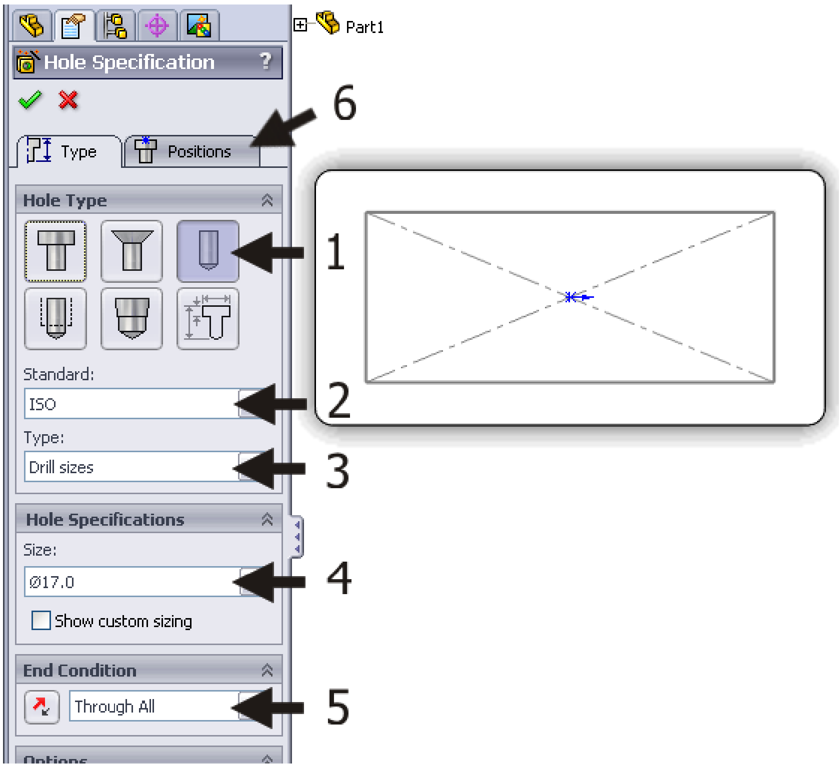
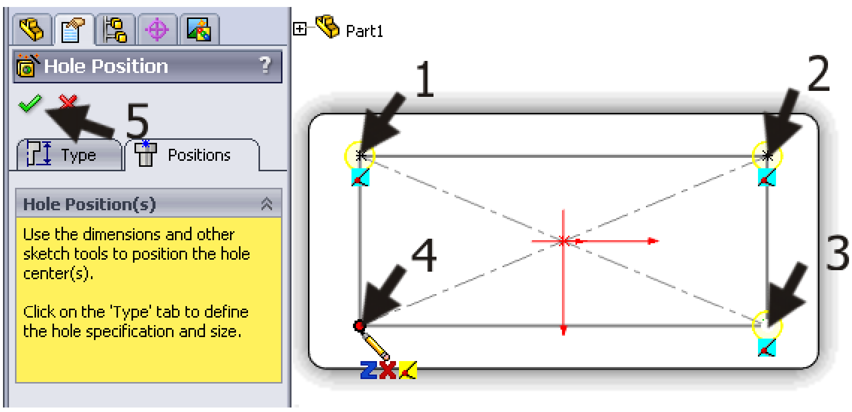
1. Take a piece of material of 150x300x20.
2. Round off the four corners with a radius of 10 mm.
3. Drill four holes of Ø17.

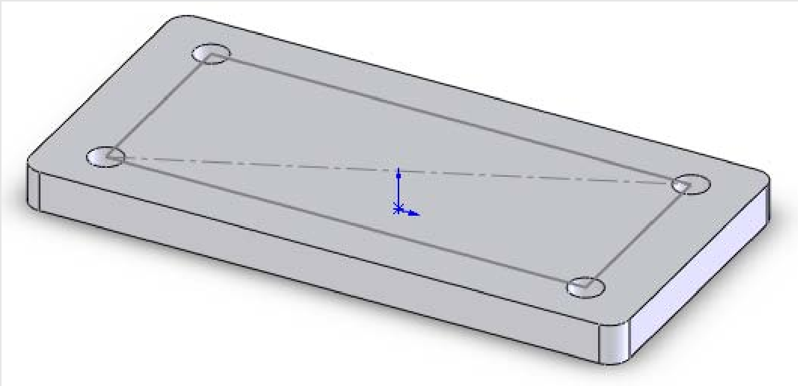
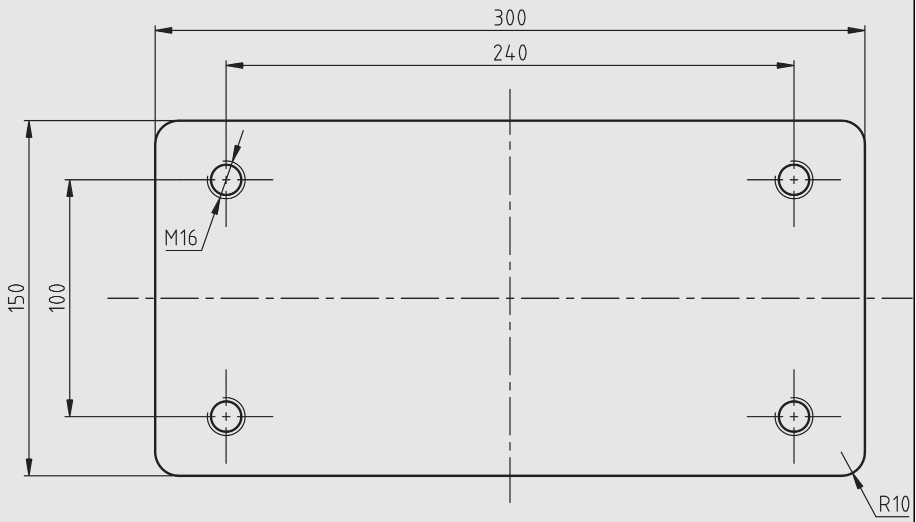
1	Start SolidWorks and open a new part.	
2	Click on ‘Top Plane’ in the FeatureManager (the left column of your screen in which all the parts of your model are listed). In this plane we will be making a sketch.	
3	Click on ‘Sketch’ in the CommandManager to reveal the correct buttons and next on Rectangle to draw a rectangle.	
4	1. Click on Center Rectangle in the CommandManager. 2. Click on the origin. 3. Click at a random point as in the view at the right (#3) to draw a rectangle.	
5	Next use the command Smart Dimension to determine two dimensions at the sides of the rectangle: 150x300. You have used Smart Dimension before. Can you remember this? If not, look it up again in Tutorial 2, steps 7 to 10.	

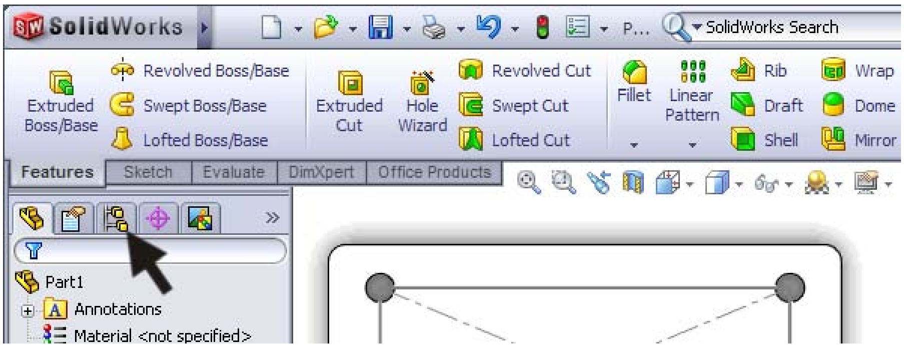
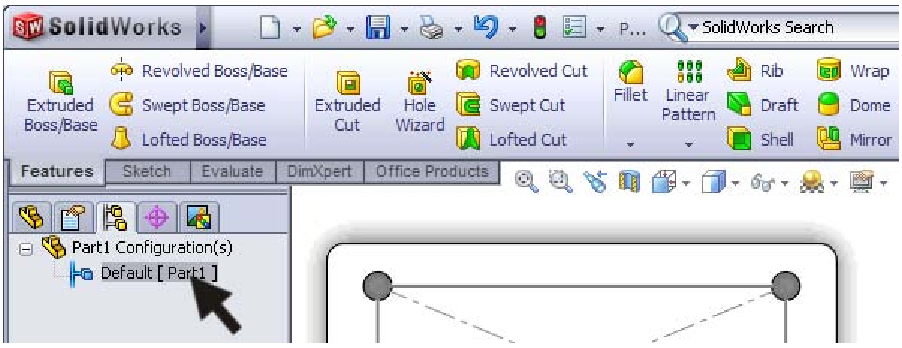
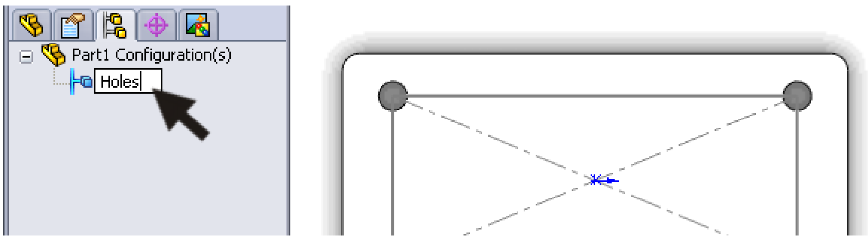
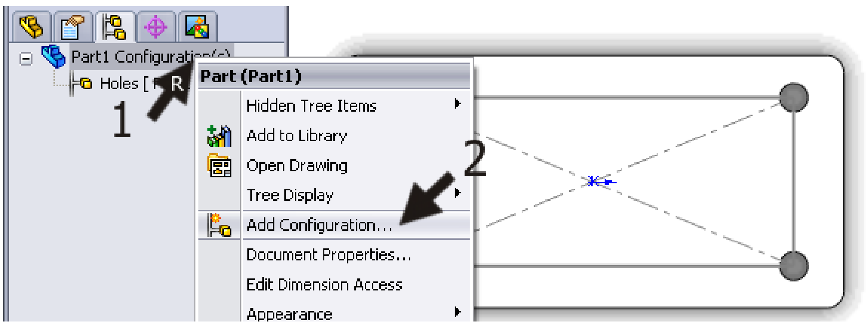
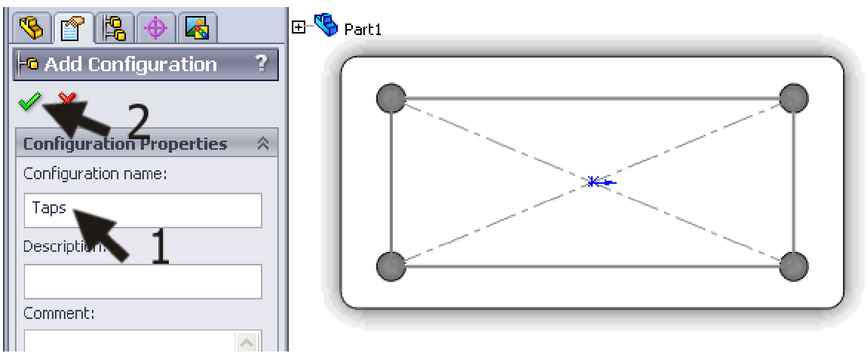
6	1. Click on ‘Features’ in the CommandManager. 2. Click on ‘Extruded Boss/Base’.	
7	1 Set the thickness at 20mm. 2 Click on OK.	
8	Next, we will round off the corners. Click on ‘Fillet’ in the CommandManager. The ‘Fillet’ command looks similar to the ‘Chamfer’ command that we used previously.	

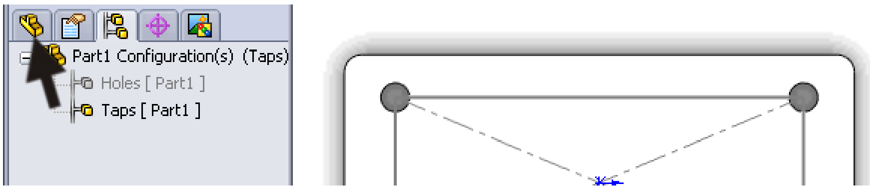
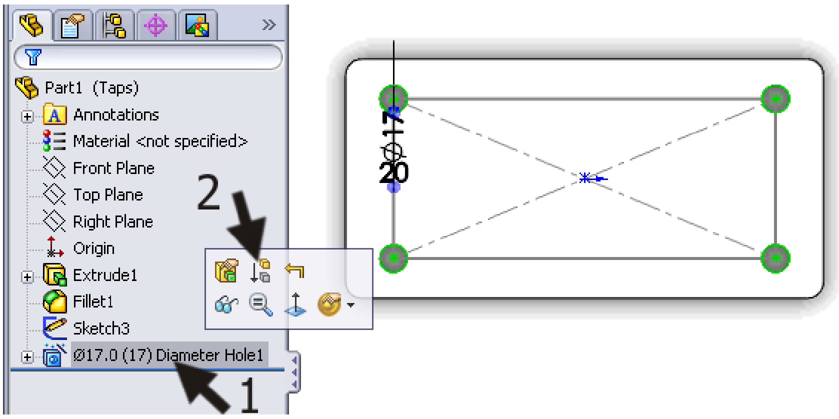
9	1. Make sure the option 'Full preview' is selected. 2-5 Next select the four edges you want to round off. 6. Set the radius at 10mm. 7. Click on OK.	
10	Next, select the top plane of the model just by clicking it.	
11	Click on 'Sketch' and next on Rectangle to draw a rectangle.	

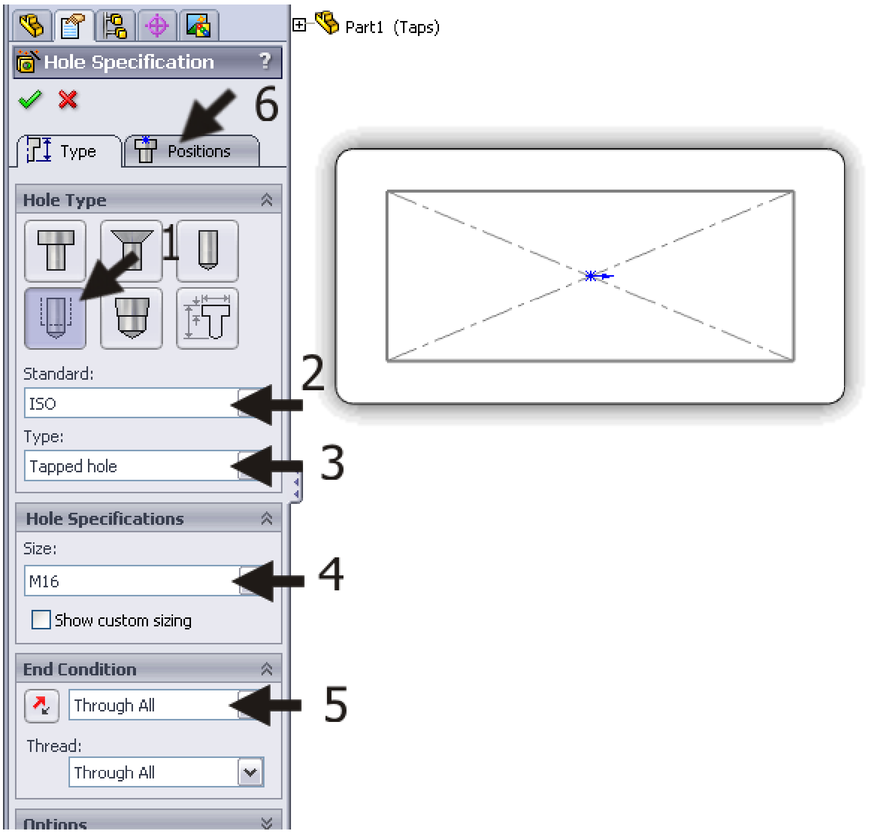
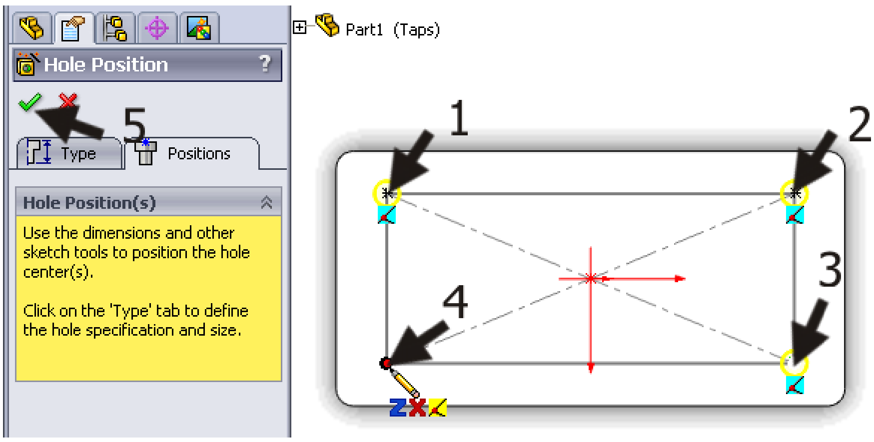
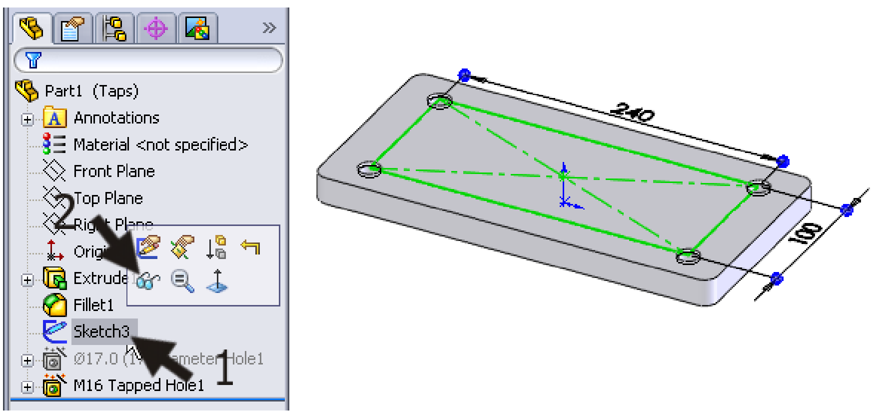
12	Click on the Standard Views button at the top of the screen and next on Normal To. The model now rotates itself to provide a straight-on view of the plane on which we are making the sketch. It does not matter if the model is positioned horizontally or vertically on your screen.	
13	4. Click on Center Rectangle in the PropertyManager. 5. Click on the origin. 6. Click at a random point like in the view at the right (#3) 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	Next click on 'Exit Sketch' in the CommandManager. The sketch remains visible, but turns grey. Notice that we will make a sketch, but do NOT make a feature of it. Later, you will see how we will use sketch like this.	

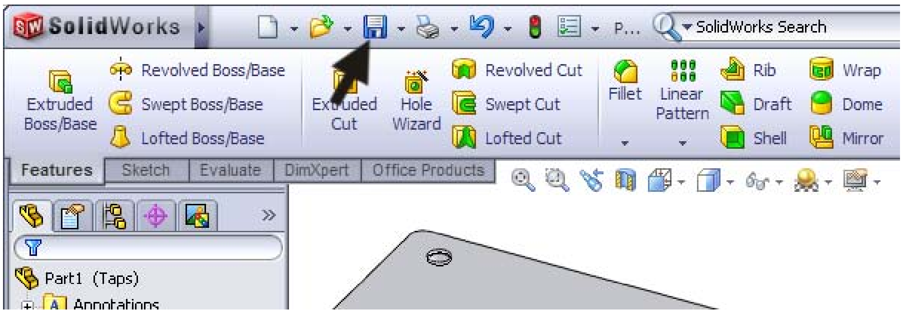
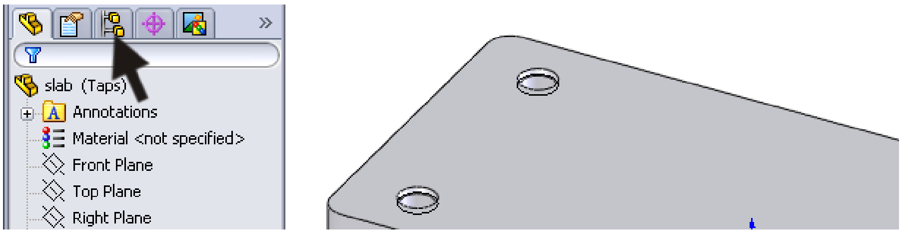
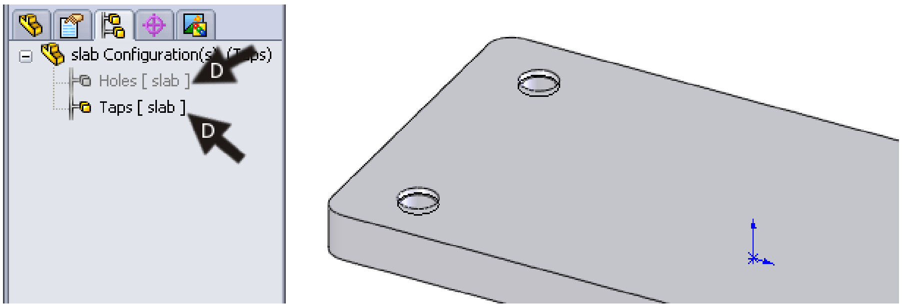
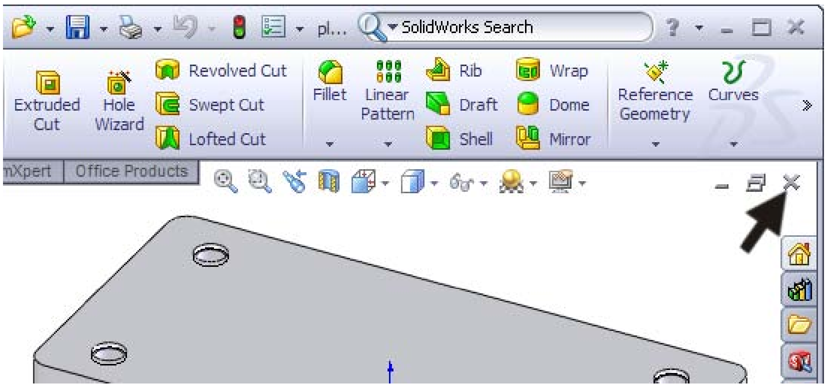
16	First, click on 'Features' in the CommandManager and next on 'Hole Wizard'.	
17	You will have to set the features of the holes in the PropertyManager. 1 Choose a 'Hole Type': choose Hole. 2 Check that the 'Standard' is set at 'ISO'. 3 Check that the 'Type' is set at 'Drill sizes'. 4 Set the diameter at Ø17mm. 5 Set the 'End Condition' at 'Through All'. 6 Click on the tab page 'Positions'.	
18	Next, click on the four corners of the rectangle you have drawn before and then click OK.	

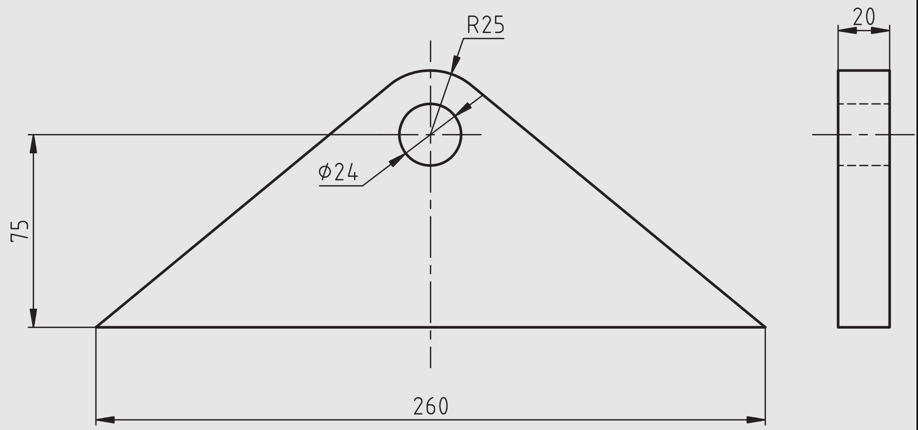
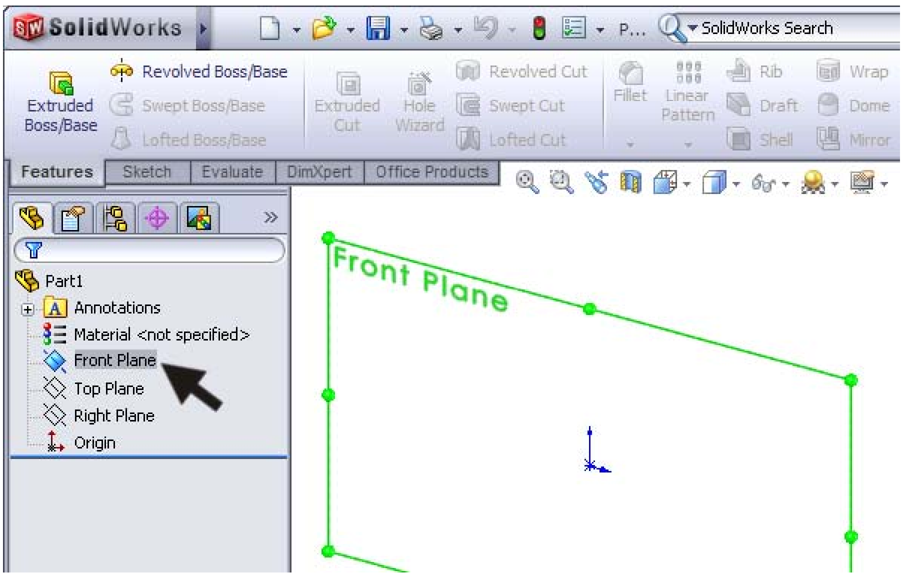
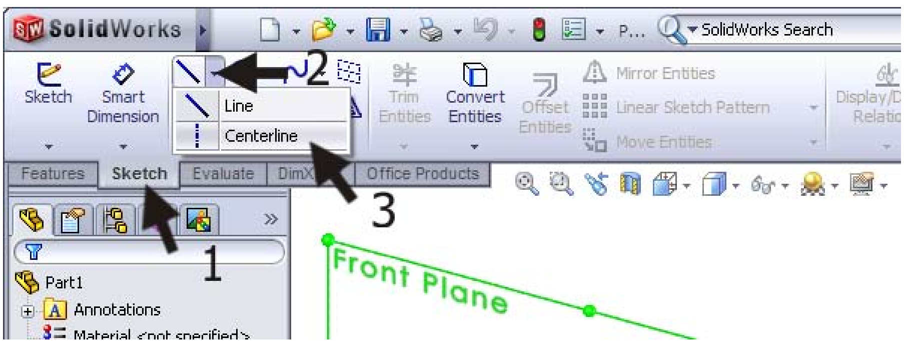
	Tip!	 The first part is ready now. We could also have created the holes we just made with the Extruded Cut feature. However, the Hole Wizard we just used is often very convenient, even more so if the holes you want to make area bit more complicated. Later on, we will see an example of this.
	Work plan	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, 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 configurations, you can always create a new part in exactly the same way as the first part. Use step 27 instead of step 17.

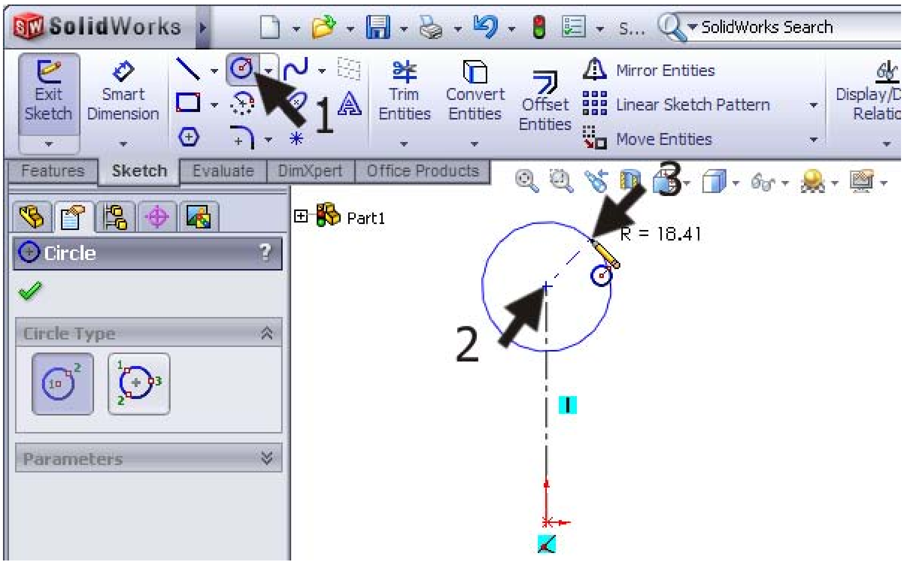
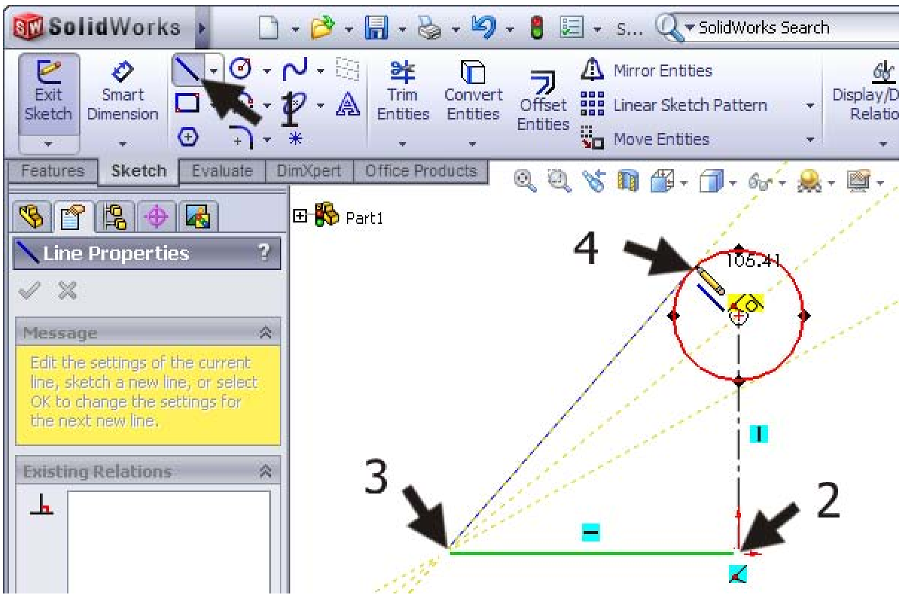
19	Click on the third tab in the FeatureManager. Instead of the FeatureManager or the PropertyManager, the ConfigurationManager now appears.	
20	There is only one configuration, named 'Default [Part1]'. Click slowly on the name once or twice to change the name.	
21	Rename this item as: 'Holes'. Push the <Enter> key on your keyboard.	
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 PropertyManager as 'Taps', and then click OK.	

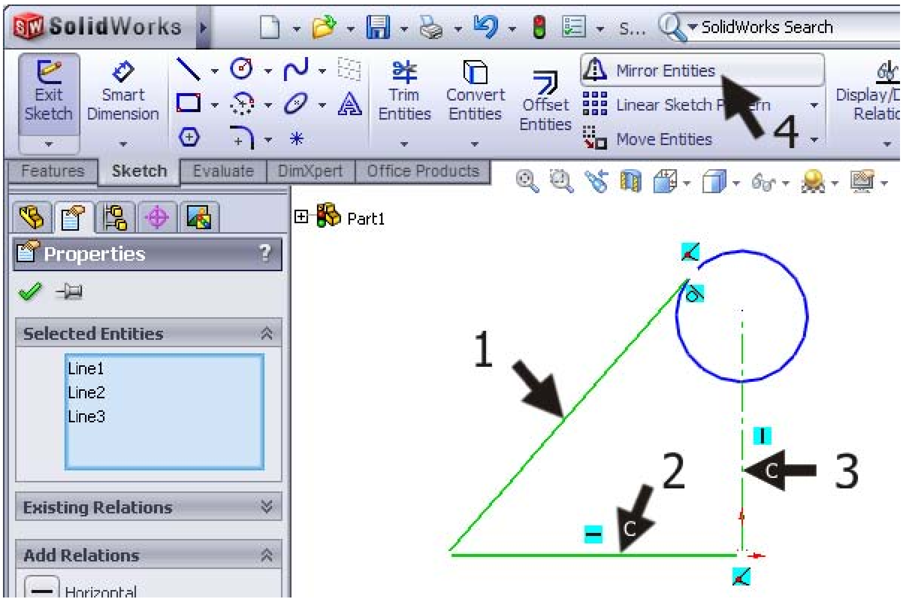
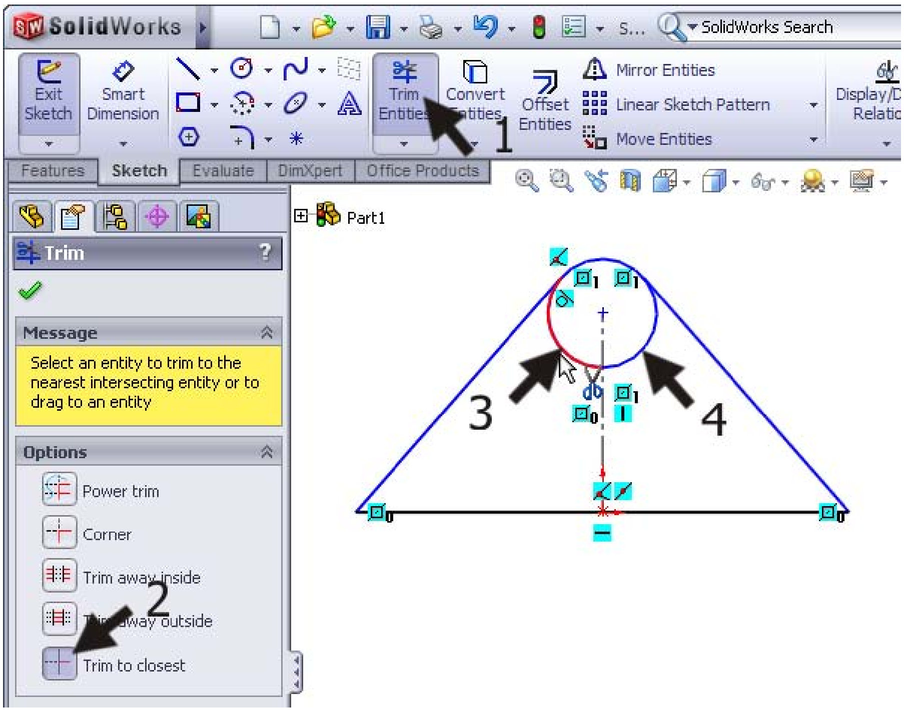
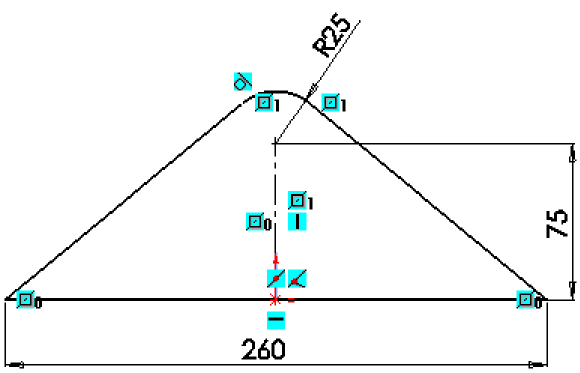
24	Click on the first tab of the ConfigurationManager to go to the FeatureManager.	
	Tip!	At this point we have two configurations but only one is active: the one we are working in. • In the ConfigurationManager you can recognize the active configuration because it is printed in black (check this at step 24). • In the FeatureManager the name of the active configuration is at the top of the list, behind the name of the created part (check this at step 25).
25	Click on the last feature you created (the holes). 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.	

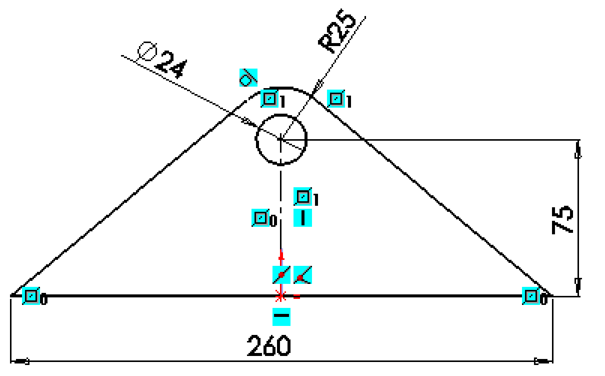
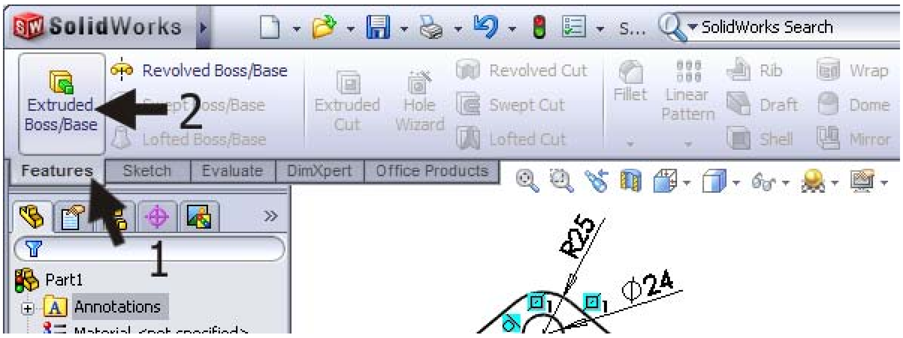
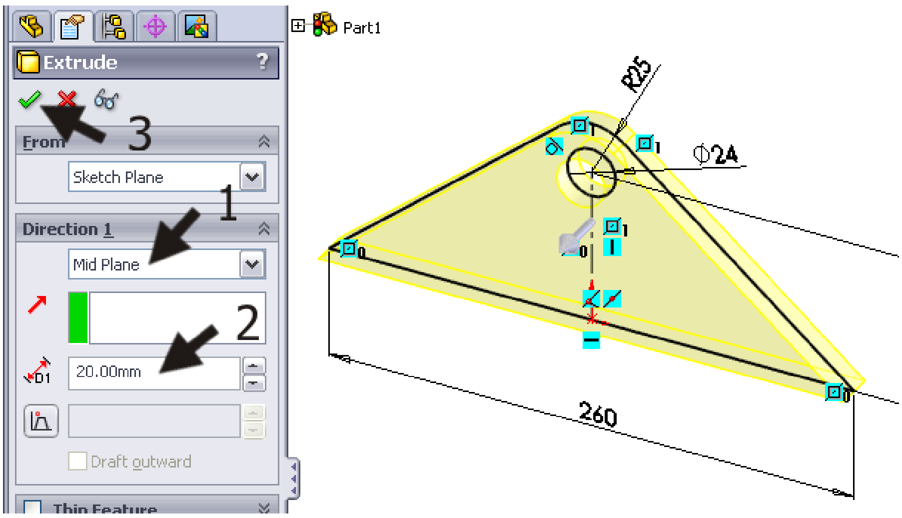
27	Set the properties of the holes in the PropertyManager. 1 Choose 'Hole Type': Tap. 2 Check that the 'Standard' is set at 'ISO'. 3 Check that the 'Type' is set at 'Tapped hole'. 4 Set the dimension at M16. 5 Set the 'End Condition' at 'Through All'. 6 Click on the tab 'Positions'.	
28	Click on the four corners of the rectangle to position the holes and then click on OK.	
29	Now click on the sketch that you have used to position the holes. Usually it is named 'Sketch2' or 'Sketch3'. The number can vary. Click on Hide in the menu that appears.	

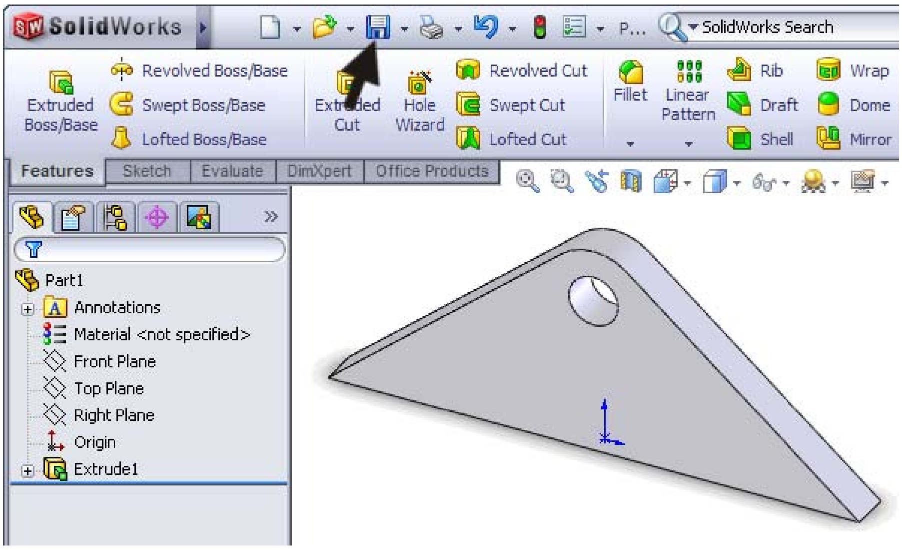
30	Save the file as slab.SLDPRT.	
31	Next click on the third tab at the top of the FeatureManager to go to the ConfigurationManager.	
32	There are now two versions (configurations) of the base model: one with normal holes and one with tapped holes. Only one of these two is active (and visible). By double-clicking 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. It is not necessary to save the file again when the program asks for it.	
	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 number of commands a second time. This was not necessary because 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 choose which version is active: this is the version you work with to change the features.

		Within every version 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 completed. 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 automatically!
	Work plan	The next part we have to create is the bracket on top for the 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 next on '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	Next, 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 appear as in the illustration on the right. When this is the case, you click to create a line which is in contact with the circle.	

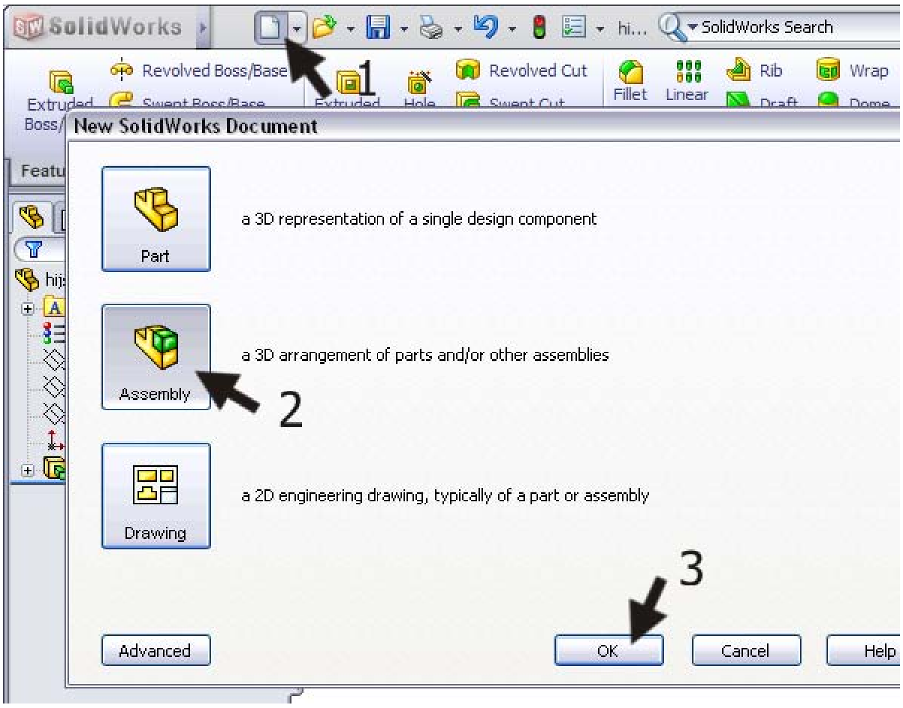
39	Next, we will copy two lines. Push the <Esc> key on your keyboard to end the line command. 1. Select the first line. 2. Hold the <Ctrl> key and select a second line. 3. Keep the <Ctrl> key down and select the centerline. 4. Click on 'Mirror Entities' in the CommandManager.	
40	The bottom part of the circle has to be removed. 1 Click on 'Trim Entities' in the CommandManager. 2 Select the option 'Trim to closest' in the PropertyManager. 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 dimension of Ø24.	
43	We can extrude the material of 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 PropertyManager. 2 Set the thickness at 20mm. 3 Click on OK.	

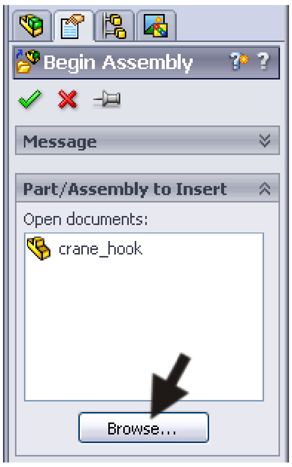
<table>
<tr>
<td>45</td>
<td>Save the file as crane_hook.SLDPRT.</td>
<td>
</td>
</tr>
<tr>
<td>46</td>
<td>The parts are ready for the assembly.

1 Click on New in the toolbar.

2 Select file type ‘Assembly’.

3 Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>47</td>
<td>We have closed the file slab.SLDPRT. For this reason it is not in the list in the PropertyManager.

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.</td>
<td>
</td>
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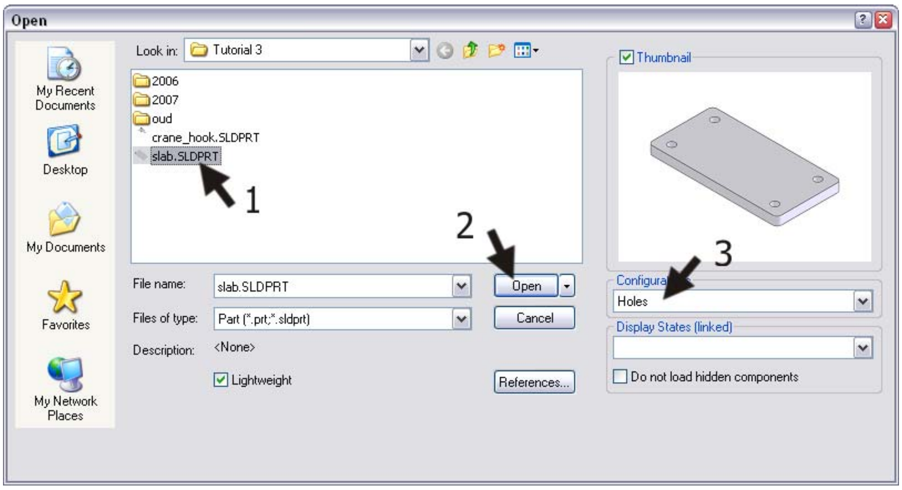
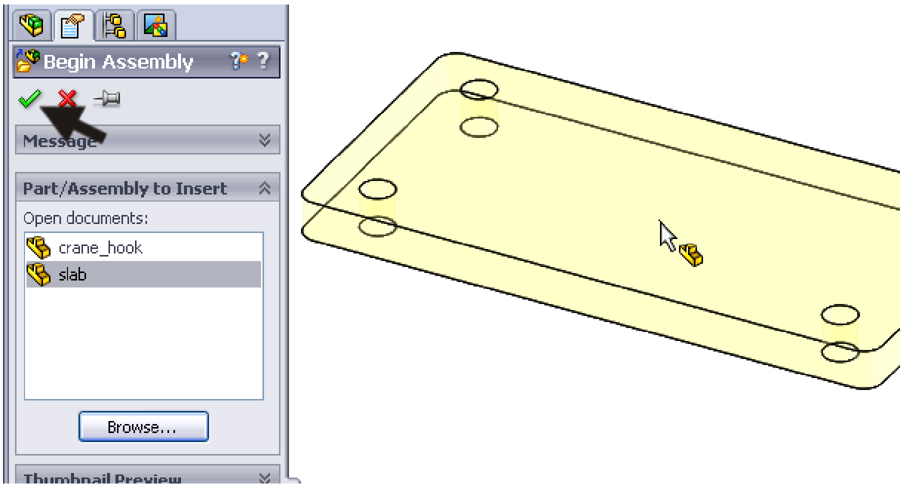
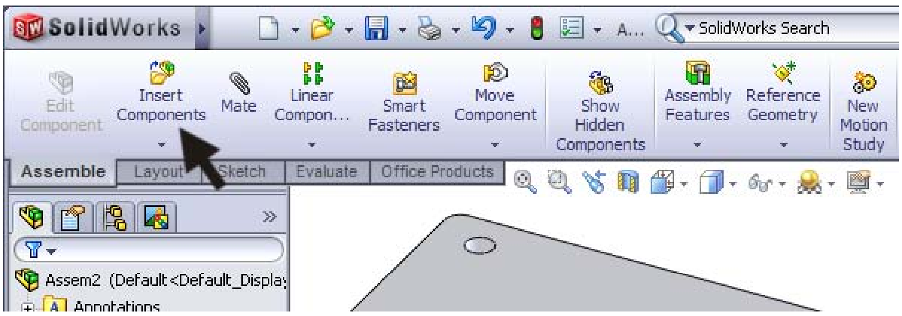
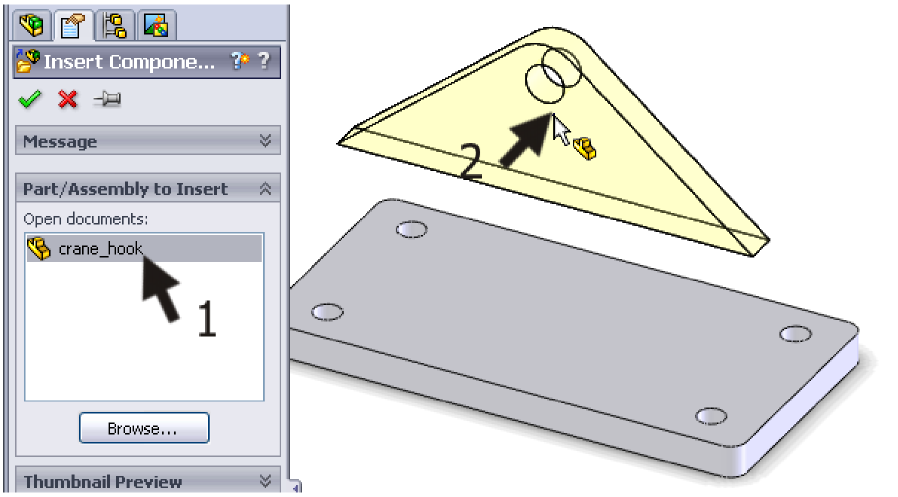
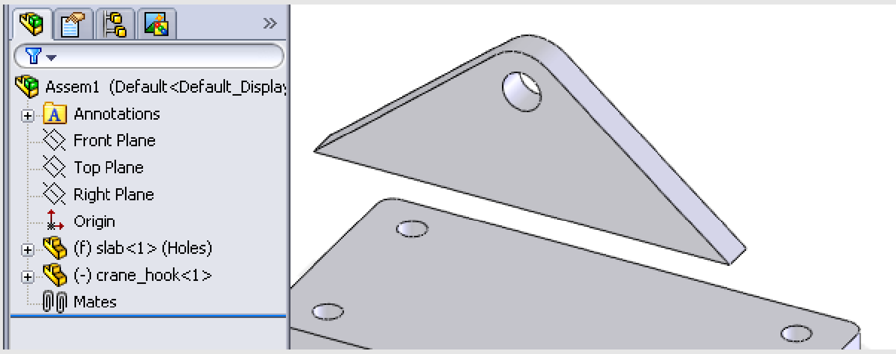
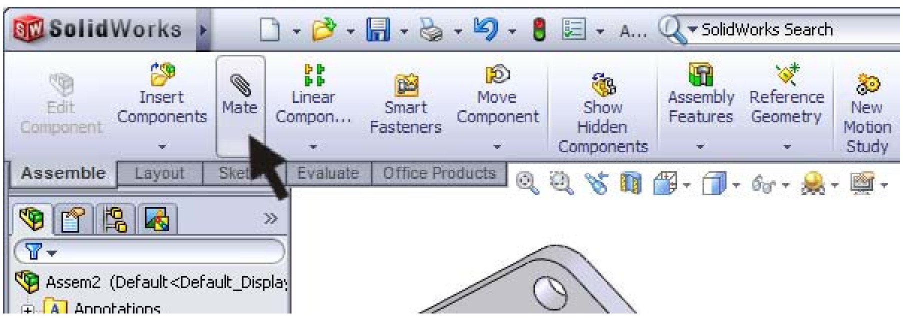
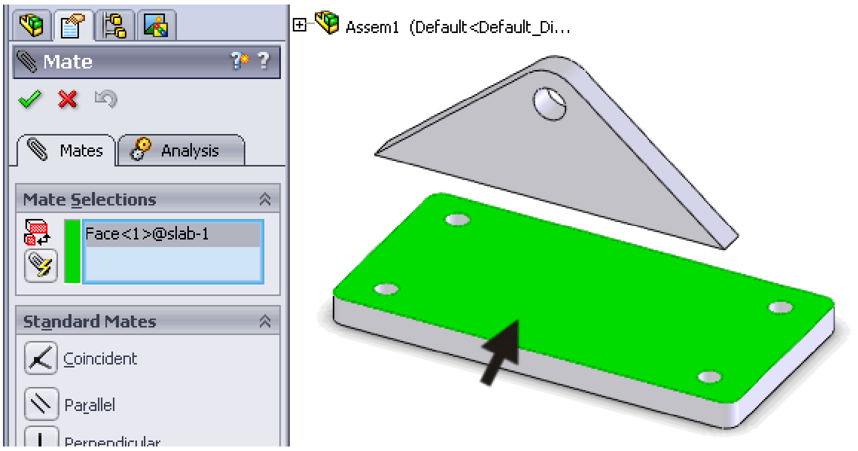
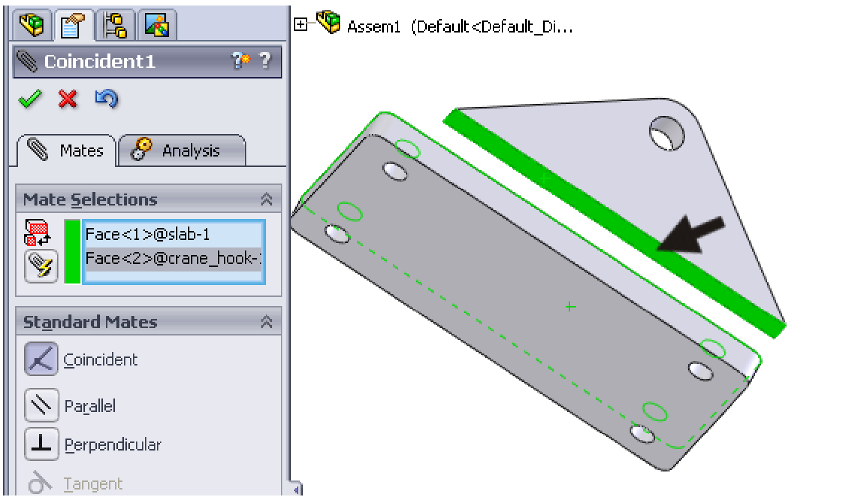
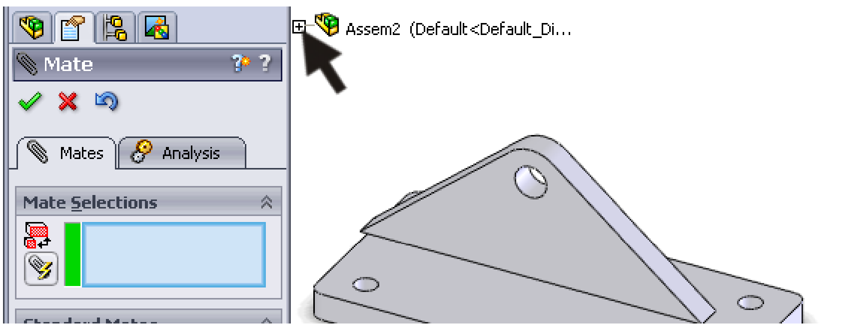
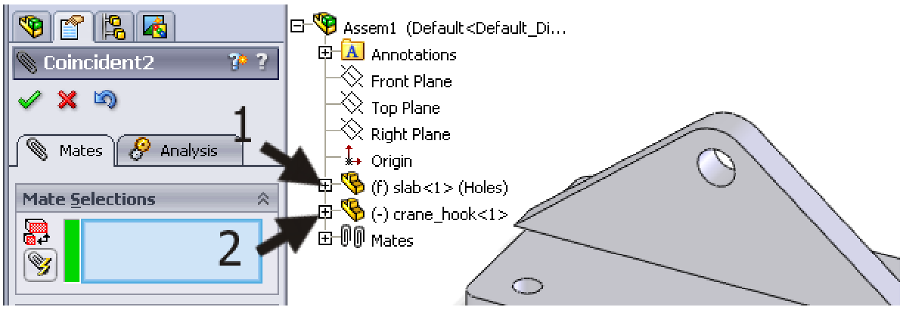
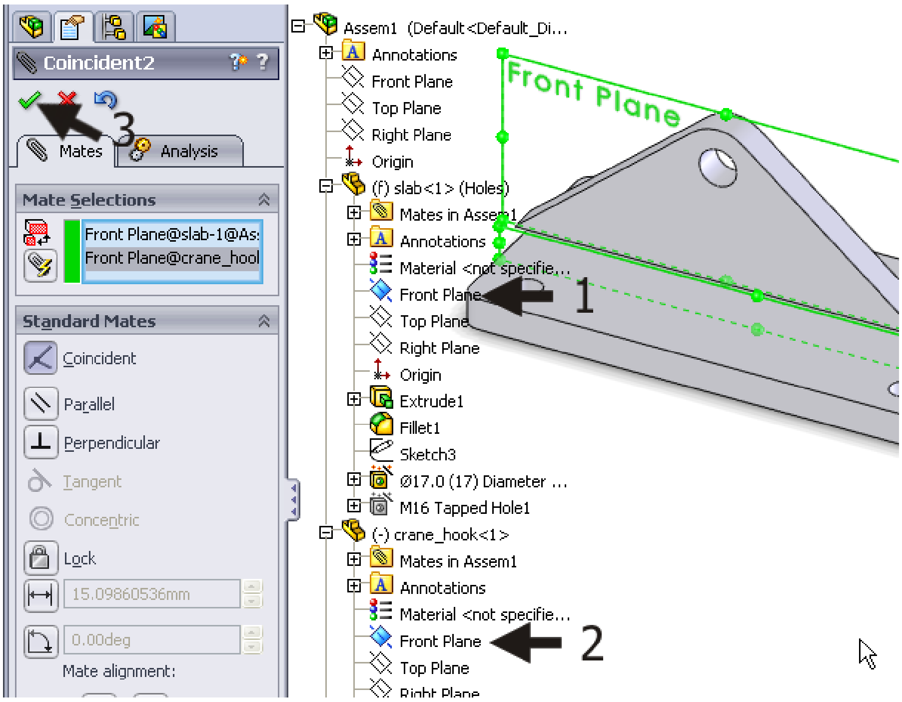
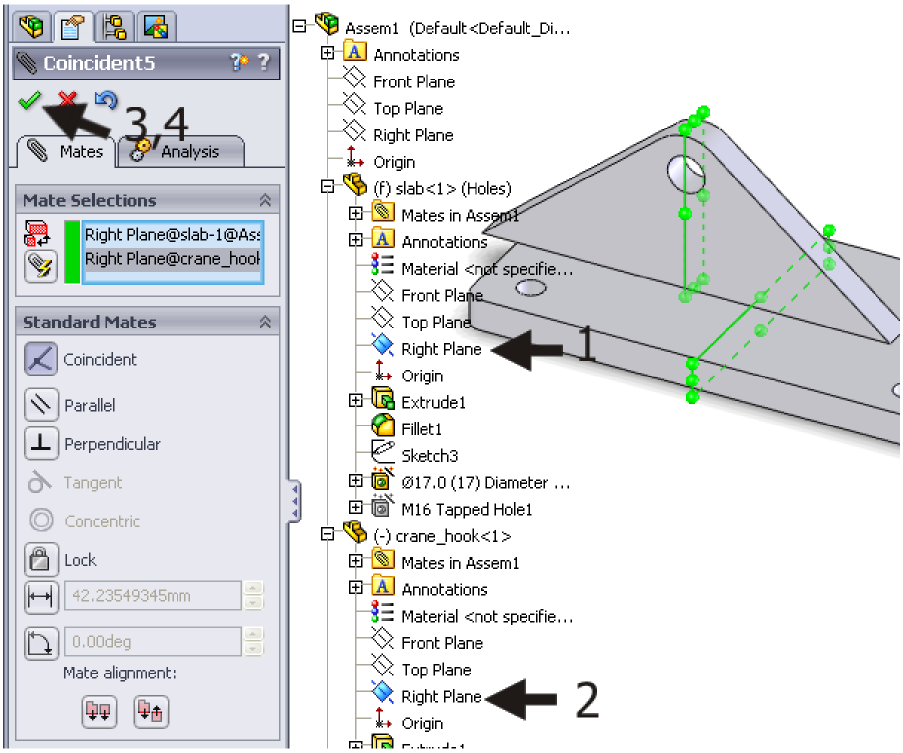
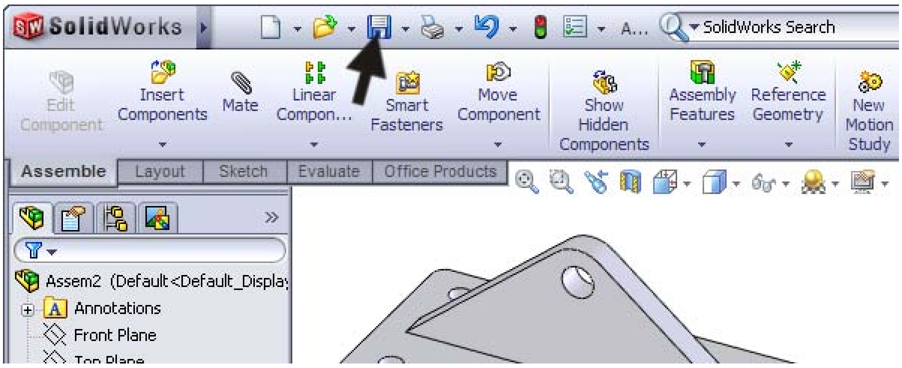
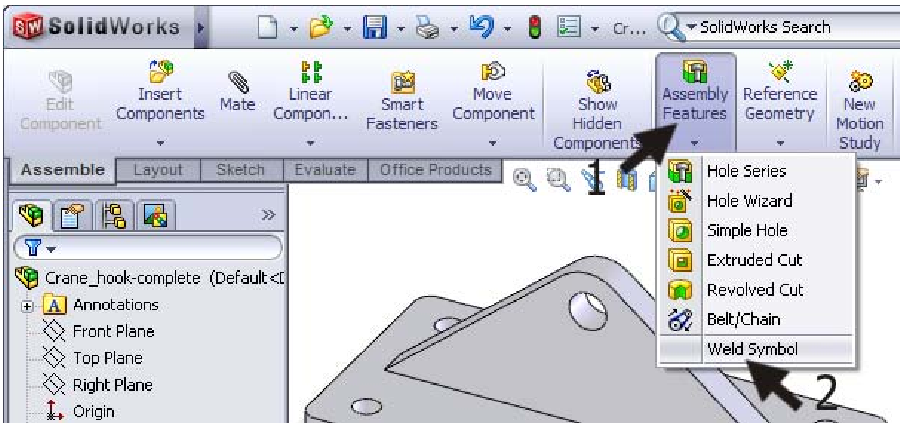
	Tip!	Normally, the Insert Components command starts automatically when a new assembly is opened. If this does not happen, click on 'Insert Components' in the CommandManager.
48	Find the file 'slab.SLDPRT', which we made earlier. 1 Select the file. 2 This file contains more than one configuration so you 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 PropertyManager.	
50	To add the next part, click on 'Insert Components' 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 previous steps correctly? You will notice that the base part cannot be moved, while 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 **'(-)'**. The part with an (f) is a floating part and can be moved around.  Be sure at all times that ONE part is Fixed; the other parts can be con-nected to this with the mate command. You can make any part Fixed or Floating by clicking on it with the right mouse buttons and choosing 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 of the bottom side of the crane hook. Push the scroll-wheel and move your mouse to rotate. 1 Click on the bottom of the crane hook. The parts now move toward each other. 2 Click on OK.	
55	The selection field in the PropertyManager is now empty, 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 the PropertyManager is now 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 'Front Plane' within the part 'Crane_hook'. 3 Next, click on OK.	

<table>
<tr>
<td>58</td>
<td>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.</td>
<td>
</td>
</tr>
<tr>
<td>59</td>
<td>Save the assembly as: crane_hook-complete.SLDASM.</td>
<td>
</td>
</tr>
<tr>
<td>60</td>
<td>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’.</td>
<td>
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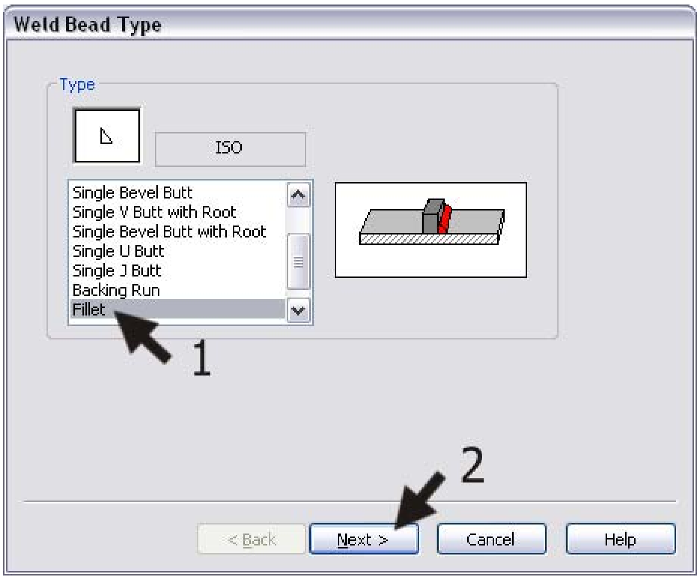
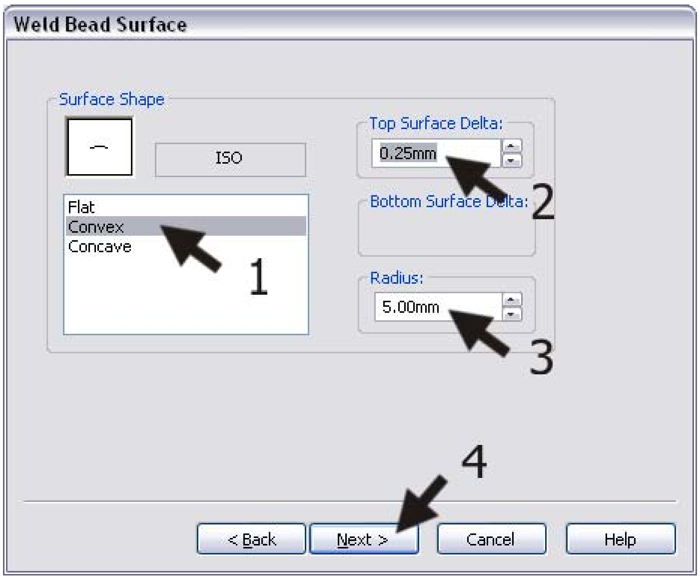
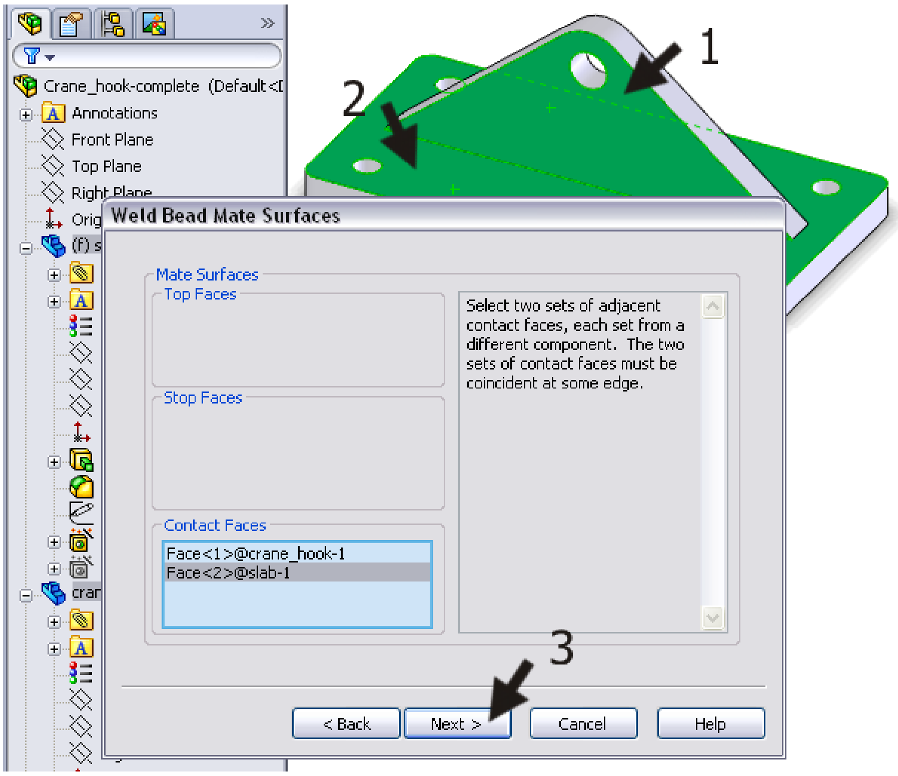
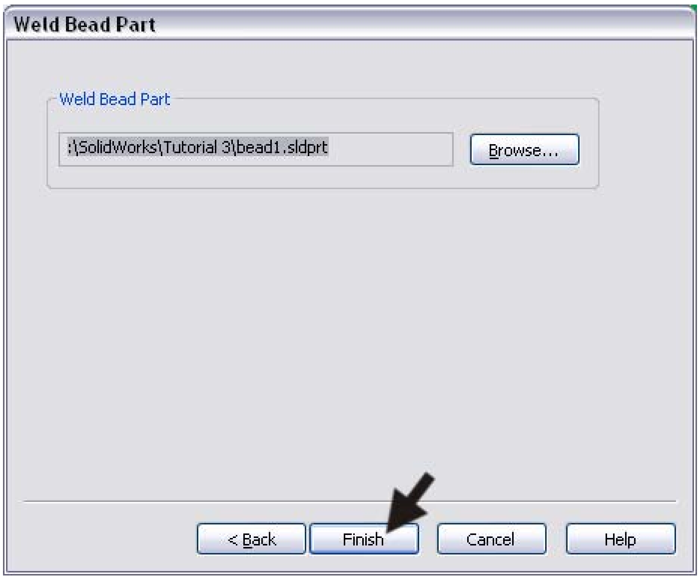
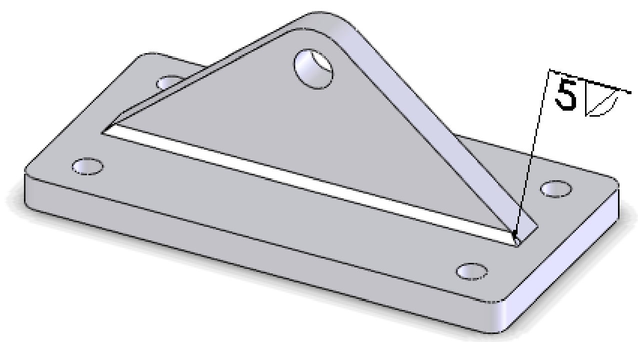
61	Select the 'Fillet' type in the menu that appears. This is a corner weld and the most simple to add. Then, click on 'Next'.	
62	We will make a curved weld. Set the features as in the illustration on the right, and click on 'Next'.	

<table>
<tr>
<td>63</td>
<td>Next, select the plane you want to weld: the upper plane of the base and the vertical plane of the crane hook.

Click on ‘Next’.</td>
<td>
</td>
</tr>
<tr>
<td>64</td>
<td>The weld will be a separate part in the assembly, and so it will be saved as a separate file. This time, SolidWorks determines the name of this ‘part’.

Click on ‘Finish’.</td>
<td>
</td>
</tr>
<tr>
<td>65</td>
<td>The weld is now made. SolidWorks automatically adds a weld symbol.

Drag the symbol to a position beside the model.

If you want to change the symbol, double-click on it. In one of the tutorials that follow we will get back to this.</td>
<td>
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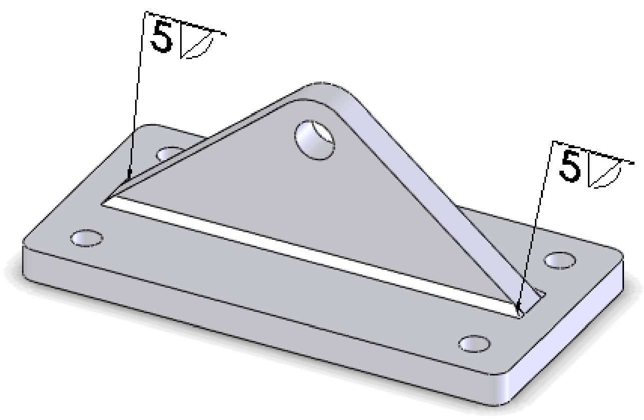
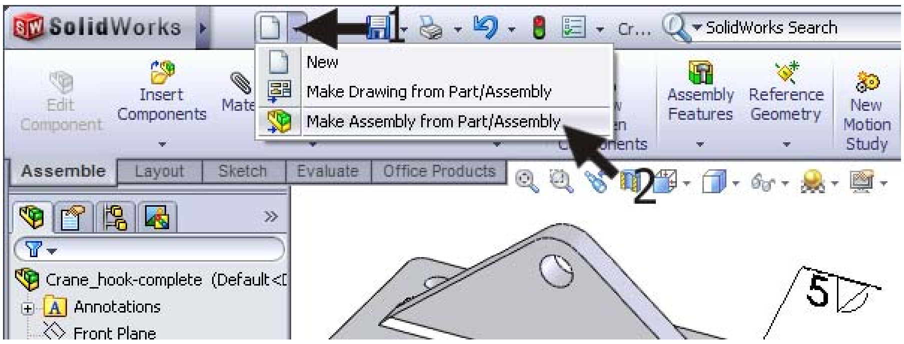
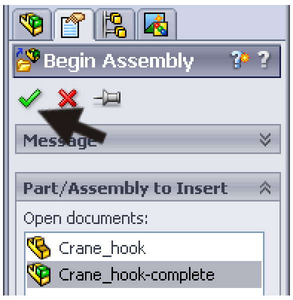
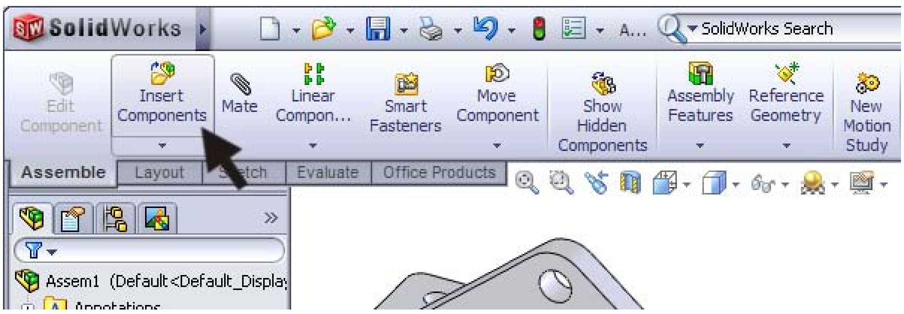
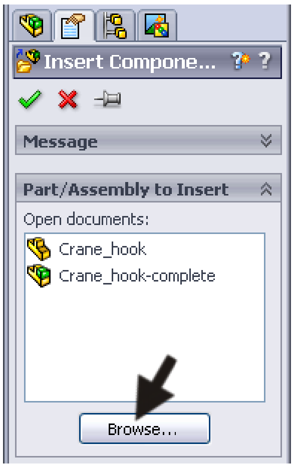
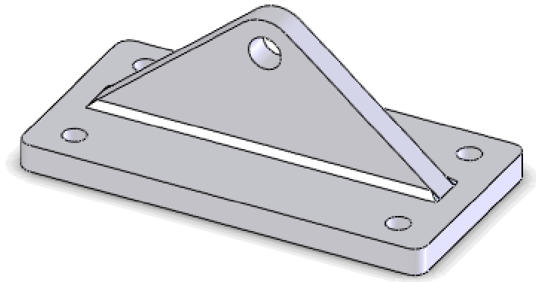
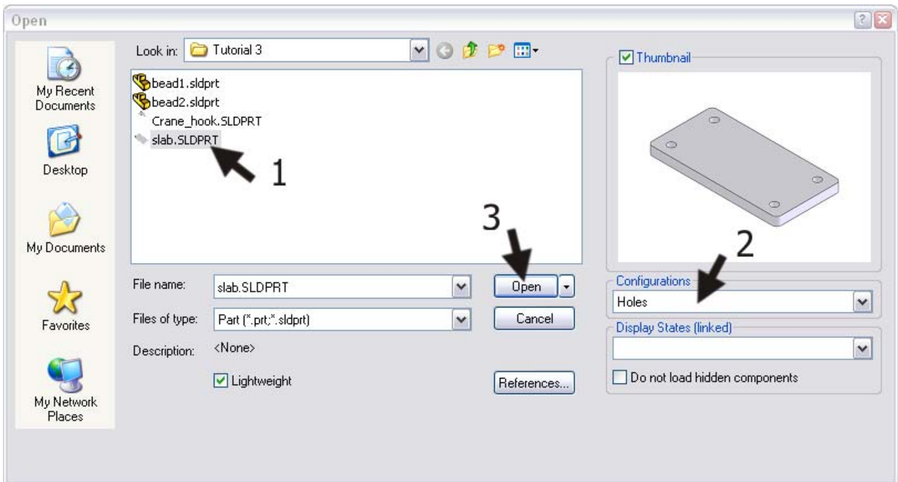
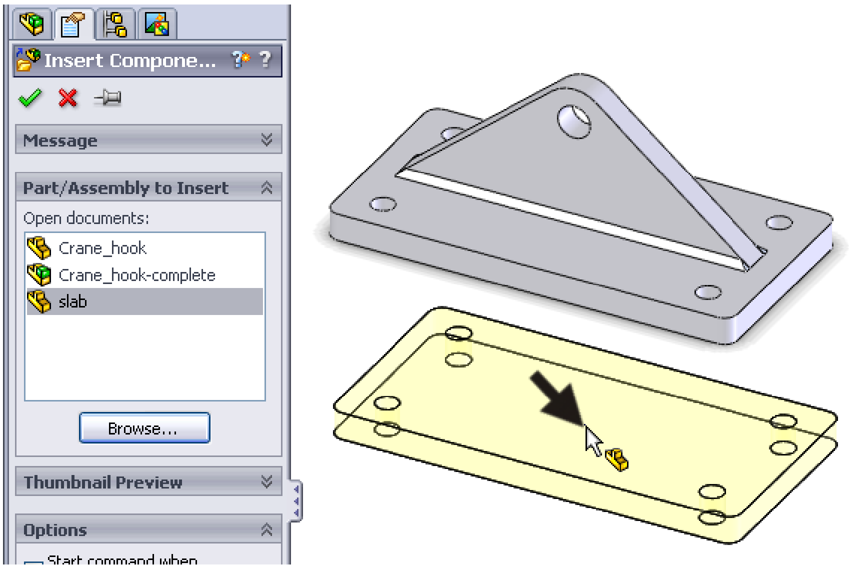
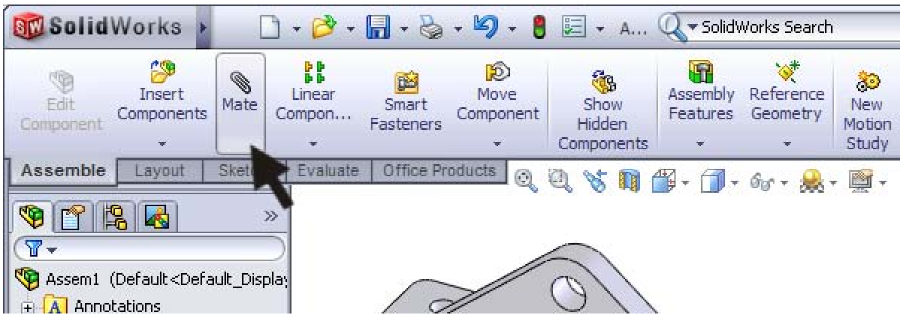
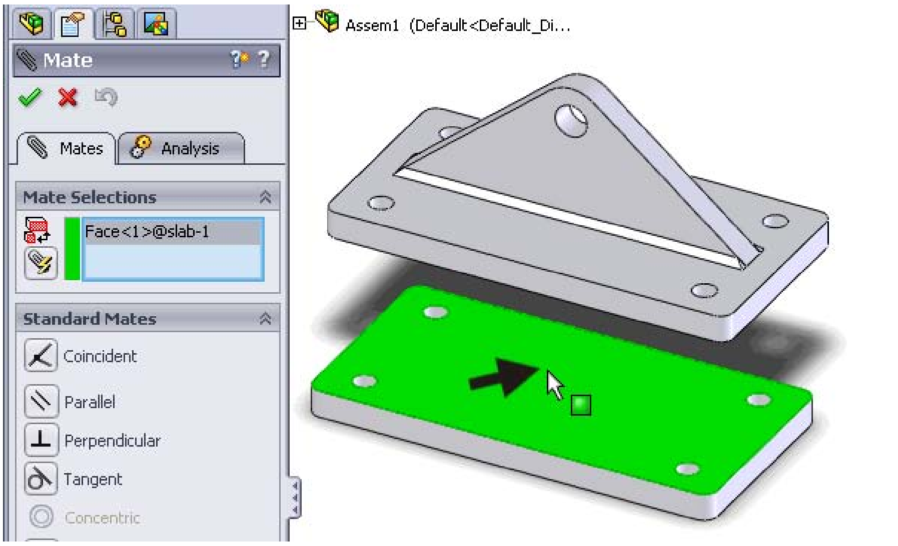
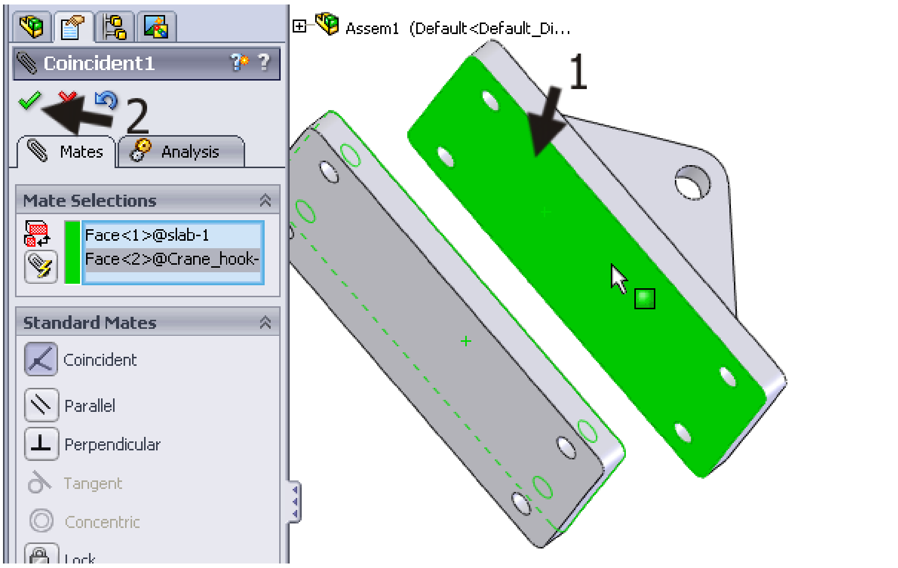
<tr><td>66</td><td>Repeat steps 60 to 65 to make a weld at the other side of the crane hook.</td><td>
</td></tr>
<tr><td>67</td><td>Save the assembly.</td><td></td></tr>
<tr><td>68</td><td>We are going to use the last assembly in the main assembly.

Click on ‘Make Assembly from Part/Assembly’ in the toolbar.</td><td>
</td></tr>
<tr><td>69</td><td>A new assembly appears in which the last assembly is added automatically.

Click on OK.</td><td>
</td></tr>
<tr><td>70</td><td>Click on ‘Insert Components’ in the CommandManager.</td><td>
</td></tr>
</table>

71	Click on 'Browse...' in the PropertyManager.	 
72	1. Select the file 'slab.SLDPRT'. 2. Select the configuration 'Holes'. 3. Click on 'Open'.	
73	Click at a random position to set the new base. Click on OK.	

<table>
<tr>
<td>74</td>
<td>Click on ‘Mate’ in the CommandManager.</td>
<td>
</td>
</tr>
<tr>
<td>75</td>
<td>Select the upper plane of the base first.</td>
<td>
</td>
</tr>
<tr>
<td>76</td>
<td>Next, rotate the model (by pushing the scroll-wheel of the mouse) and select the bottom plane of the crane hook.

Click on OK.</td>
<td>
</td>
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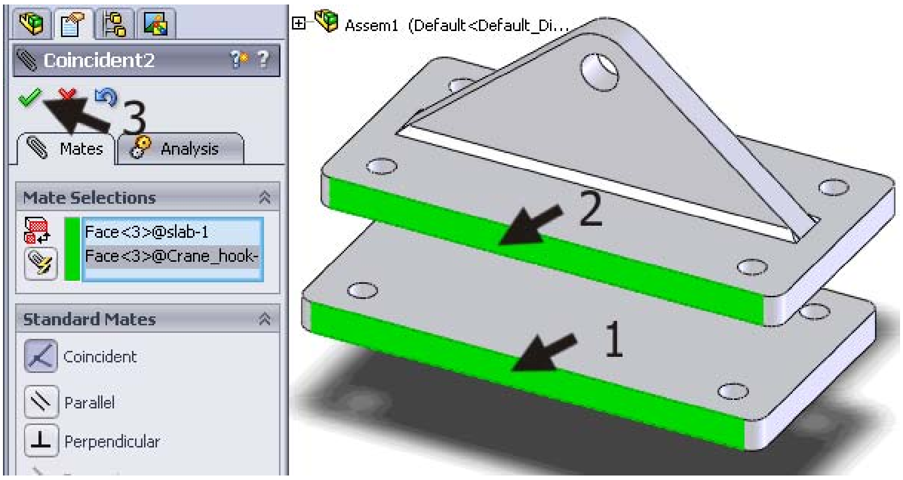
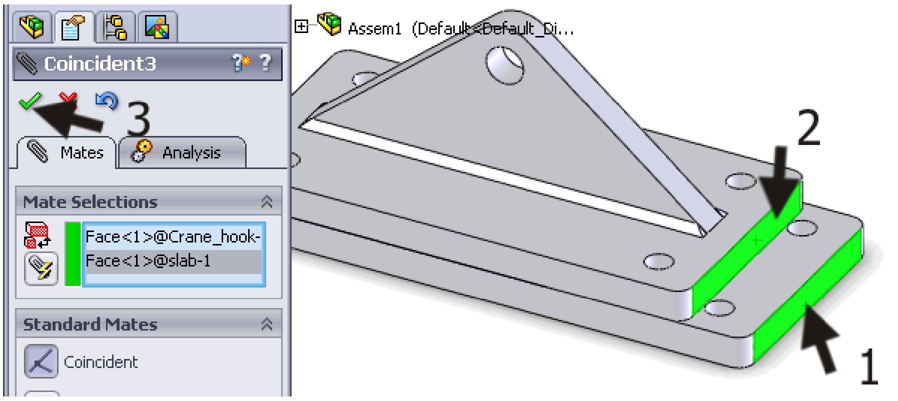
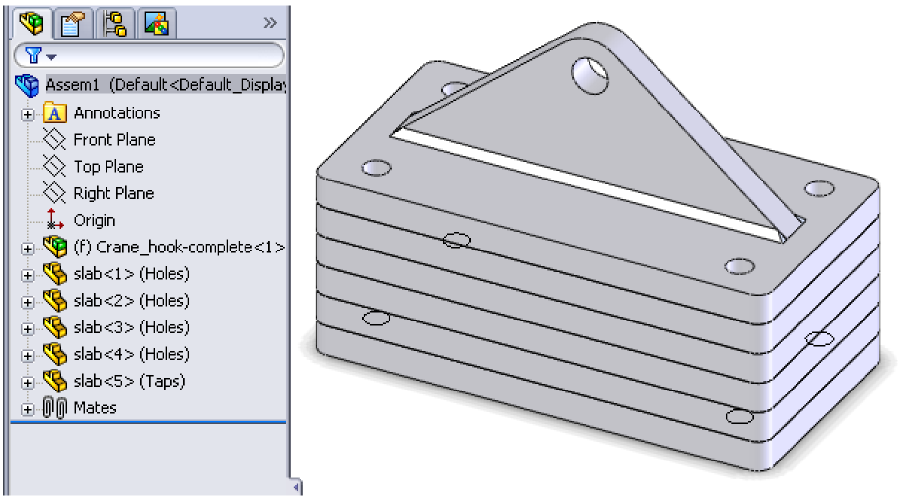
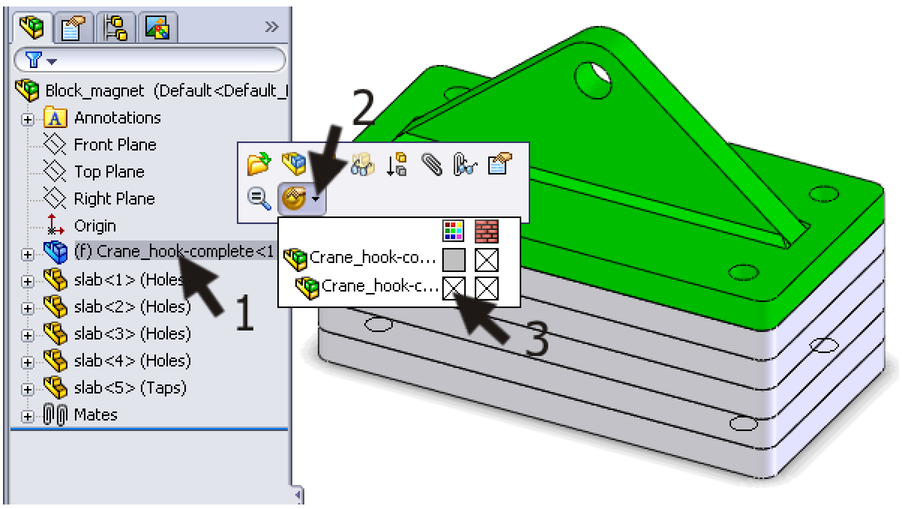
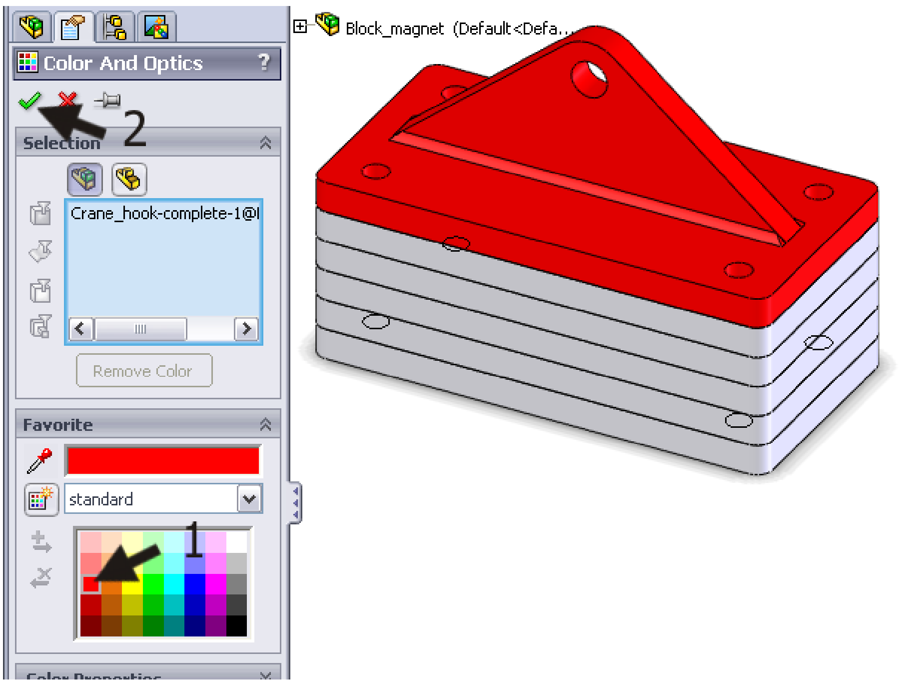
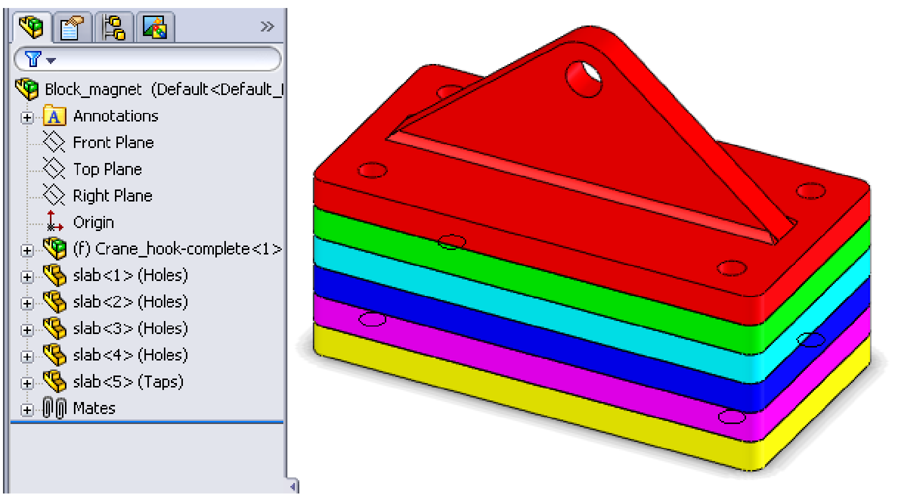
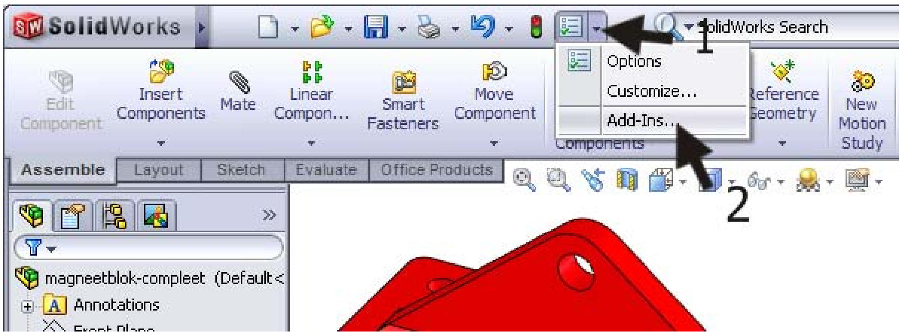
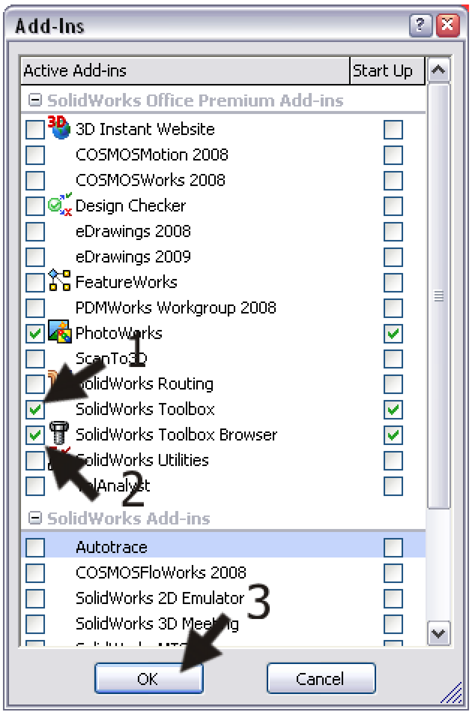
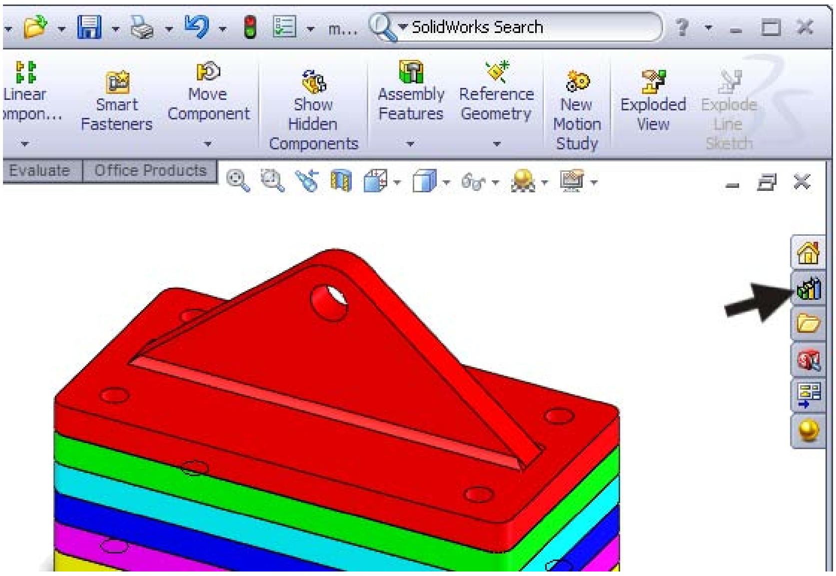
</table>

77	To make the next mate, you select the long sides of both parts and click on OK.	
78	To make the final mate, you select the short sides of both parts and click on OK. Click on OK again to end the Mate command	
79	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 thing again, only now you select the configuration 'Taps' when adding this part.	
80	Save the assembly as 'Block_magnet.SLDASM'.	

81	Next, we will add some colors to our model. 1 Click on the first part (Crane_hook-complete) in the FeatureManager. 2 Click on 'Appearance callout' in the menu that appears. 3 Click on Color in the bottom line.	
82	First click on 'Apply changes at assembly component level' in the PropertyManager. Select a color and click on OK. The whole part will be colored now.	
83	Select another color for each part of the magnetic block.	

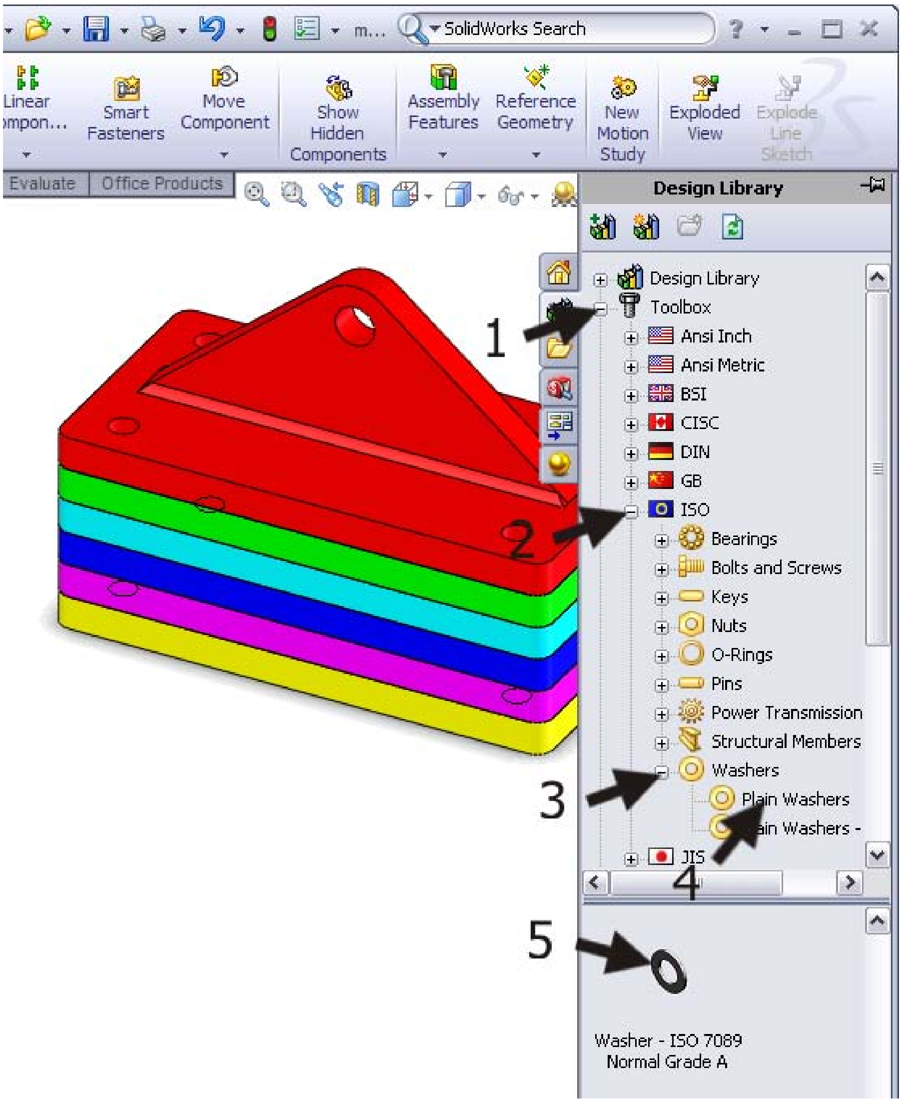
84	We will now add some washers and bolts. We will use a tool in SolidWorks that is called Toolbox. Before you can use this, you must first check if Toolbox is already installed AND activated on you computer. Click on 'Add-Ins' in the CommandManager.	
85	Be sure that the options 'SolidWorks Toolbox' and 'SolidWorks Toolbox Browser' are both selected with a 'check' symbol. If these options are 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 cannot finish this tutorial. If you still want to finish your model, you can download these parts (i.e., bolts and washers) from www.solidworks.nl. You do not use Toolbox to do this but put the bolts and washers in the assembly like you would with any other part.
	Tip!	By 'checking' the two options in step 85 (SolidWorks Toolbox and SolidWorks Toolbox Browser) these tools will be loaded automatically every time SolidWorks starts up. So you do not have to activate the Toolbox again.

<table>
<tr>
<td>86</td>
<td>Click on the symbol of the Design Library in the Task Pane (at the right of the screen).</td>
<td>
</td>
</tr>
<tr>
<td>87</td>
<td>The Task Pane unfolds itself and you can see the ‘Toolbox’ now. We are going to add some washers.

Double-click the following items one after another:

1. ‘Toolbox’.
2. ‘ISO’.
3. ‘Washers’.
4. ‘Plain Washers’.

The available washers appear in the lower part of the Task Pane.

5. Find the washer: ‘Washer – ISO 7089 Normal Grade A’.</td>
<td>
</td>
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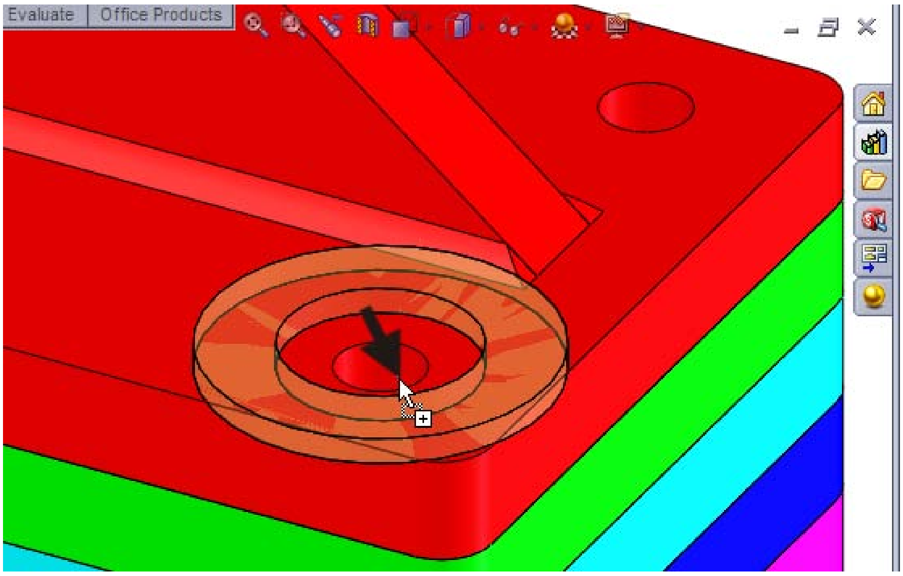
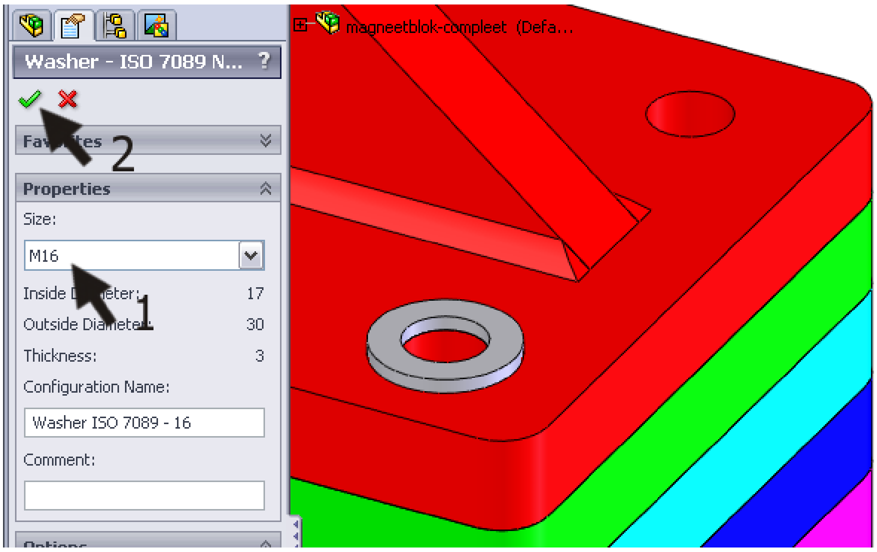
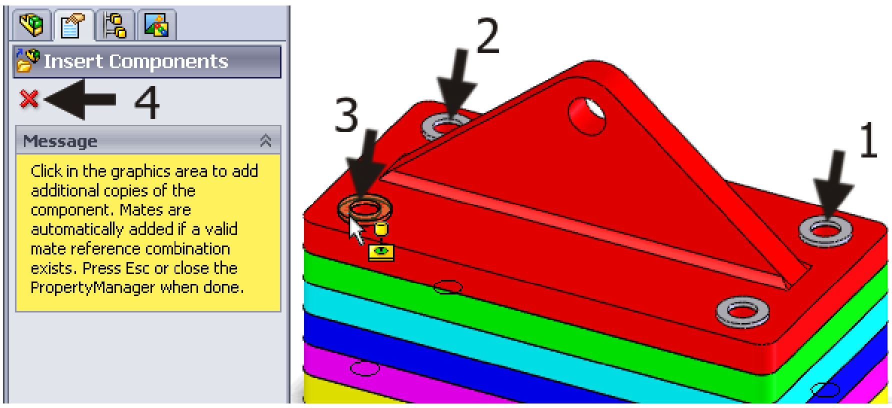
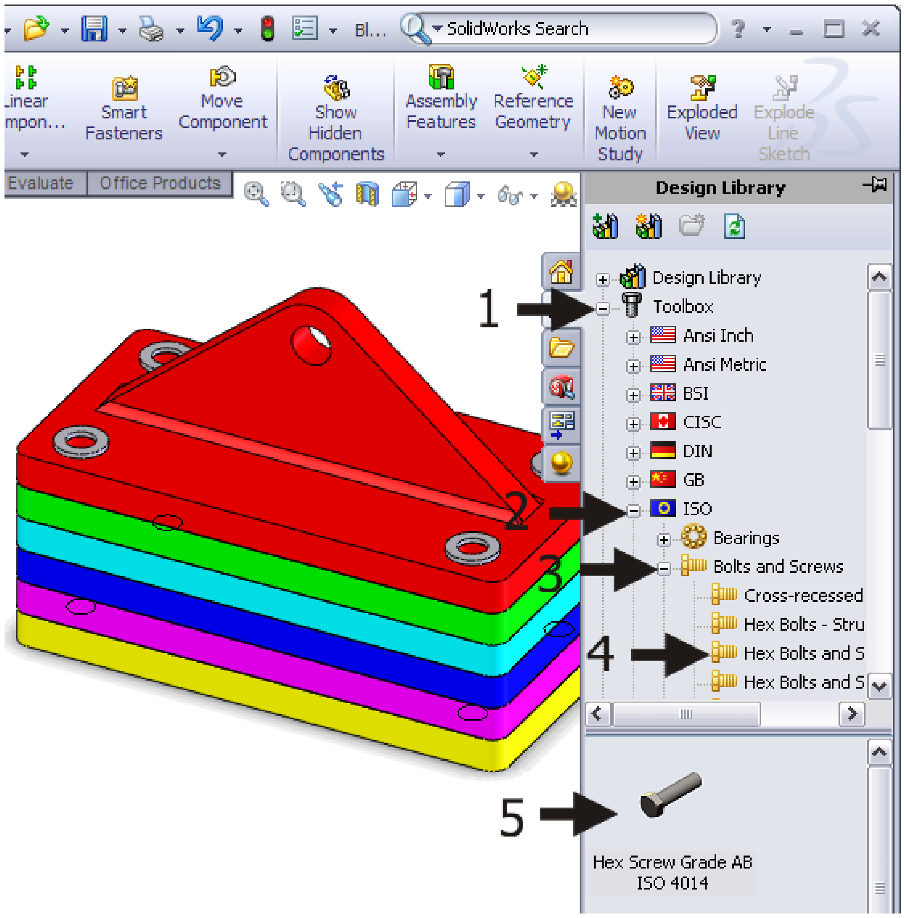
</table>

88	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 washer may appear too small or too big, but this does not matter at this point.	
89	Change the setting of the washer to ‘M16’ in the PropertyManager, and click on OK.	
90	The ring is now attached to your mouse and you can put it on the other holes. After you have finished placing all the washers, click on Cancel.	

<table>
<tr>
<td>91</td>
<td>Open the Task Pane again and go to:

1 ‘Toolbox’.
2 ‘ISO’.
3 ‘Bolts and Screws’.
4 ‘Hex Bolts and Screws’.

Select this bolt: ‘Hex Screw Grade AB ISO 4014’.</td>
<td>
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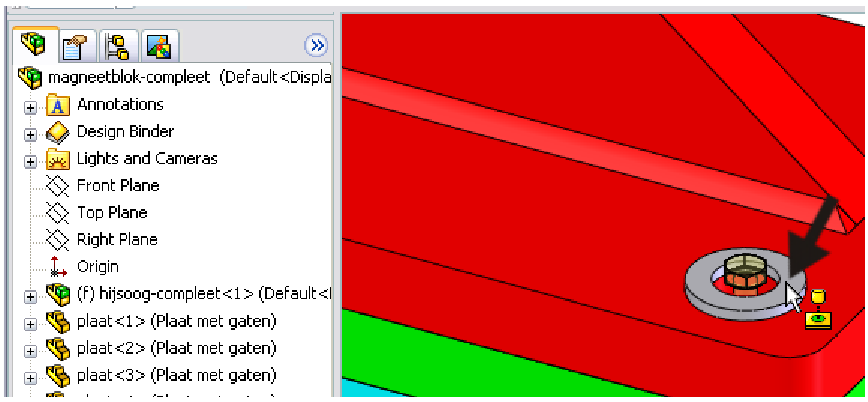
<tr>
<td>92</td>
<td>Again, drag this component to one of the holes.

Pay attention: release the mouse button when the cursor is above one of the holes.

This is important, because when the cursor is above the plane, the bolt will be positioned TOO LOW (at the surface of the plane and NOT on top of the washer).</td>
<td>
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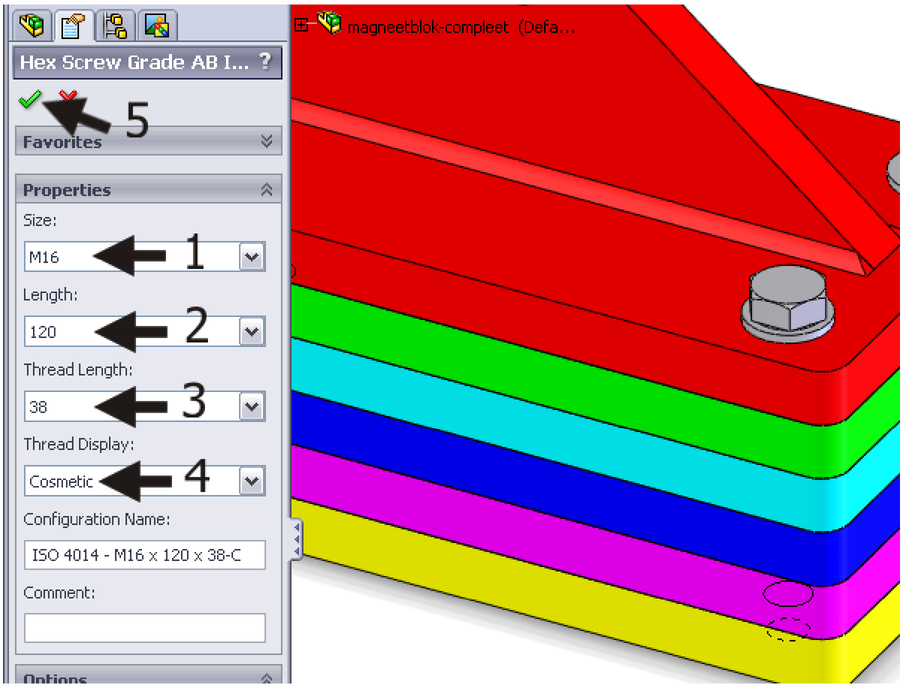
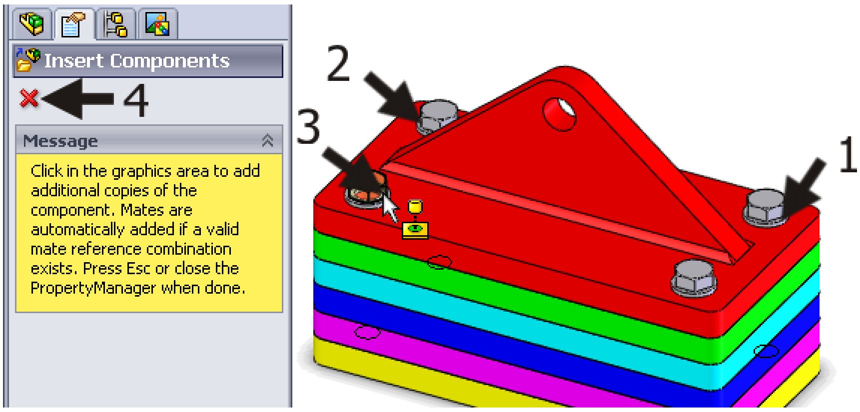
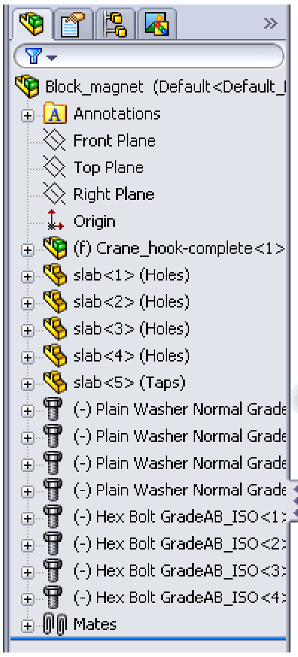
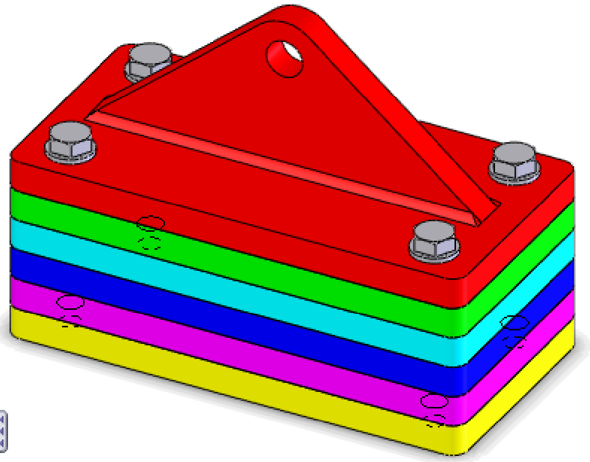
</table>

93	In the PropertyManager you can set the features of the bolt. 1. 'Size' (diameter) is 'M16'. 2. 'Length' of the bolt is '120mm'. 3. 'Thread Length' of the thread is '38mm'. 4. 'Thread Display' (the thread is displayed as) is 'Cosmetic'. 5. Click on OK.	
94	Now the bolt is attached to the cursor, so you can put in the other holes too. Pay attention to click on the washer and NOT in the hole!	

95	The magnetic block is ready now. Save the assembly.	 
	What are the main features you have learned in this tutorial?	In this exercise we have executed many new commands. • 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 learned some powerful tools.

SolidWorks works in education.

One cannot imagine the modern technical world without 3D CAD. Whether your profession is in the mechanical, electrical, or industrial design fields, or in the automotive industry, 3D CAD is THE tool used by designers and engineers today.

SolidWorks is the most widely used 3D CAD design software in Benelux. Thanks to its unique combination of features, its ease-of-use, its wide applicability, and its excellent support. In the software's annual improvements, more and more customer requests are implemented, which leads to an annual increase in functionality, as well as optimization of functions already available in the software.

Education

A great number and wide variety of educational institutions – ranging from technical vocational training schools to universities, including Delft en Twente, among others – have already chosen SolidWorks. Why?

For a **teacher** or **instructor,** SolidWorks provides user-friendly software that pupils and students find easy to learn and use. SolidWorks benefits all training programs, including those designed to solve problems as well as those designed to achieve competence. Tutorials are available for every level of training, beginning with a series of tutorials for technical vocational education that leads students through the software step-by-step. At higher levels involving complex design and engineering, such as double curved planes, more advanced tutorials are available. All tutorials are in English and free to download at www.solidworks.com.

For a **scholar** or a **student**, learning to work with SolidWorks is fun and edifying. By using SolidWorks, design technique becomes more and more visible and tangible, resulting in a more enjoyable and realistic way of working on an assignment. Even better, every scholar or student knows that job opportunities increase with SolidWorks because they have proficiency in the most widely used 3D CAD software in the Benelux on their resume. For example: at www.cadjobs.nl you will find a great number of available jobs and internships that require SolidWorks. These opportunities increase motivation to learn how to use SolidWorks.

To make the use of SolidWorks even easier, a Student Kit is available. If the school uses SolidWorks, every scholar or student can get a **free download** of the Student Kit. It is a complete version of SolidWorks, which is only allowed to be used for educational purposes. The data you need to download the Student Kit is available through your teacher or instructor.

The choice to work with SolidWorks is an important issue for ***ICT departments*** because they can postpone new hardware installation due to the fact that SolidWorks carries relatively low hardware demands. The installation and management of SolidWorks on a network is very simple, particularly with a network licenses. And if a problem does arise, access to a qualified helpdesk will help you to get back on the right track.

Certification

When you have sufficiently learned SolidWorks, you can obtain certification by taking the Certified SolidWorks Associate (CSWA) exam. By passing this test, you will receive a certificate that attests to your proficiency with SolidWorks. This can be very useful when applying for a job or internship. After completing this series of tutorials for VMBO and MBO, you will know enough to take the CSWA exam.

Finally

SolidWorks has committed itself to serving the needs of educational institutions and schools both now and in the future. By supporting teachers, making tutorials available, updating the software annually to the latest commercial version, and by supplying the Student Kit, SolidWorks continues its commitment to serve the educational community. The choice of SolidWorks is an investment in the future of education and ensures ongoing support and a strong foundation for scholars and students who want to have the best opportunities after their technical training.

Contact

If you still have questions about SolidWorks, please contact your local reseller.

You will find more information about SolidWorks at our website: http://www.solidworks.com

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