SolidWorks® Tutorial 9

AXLE SUPPORT

Preparatory Vocational Training
and Advanced Vocational Training

To be used with SolidWorks® Educational Edition Release 2008-2009

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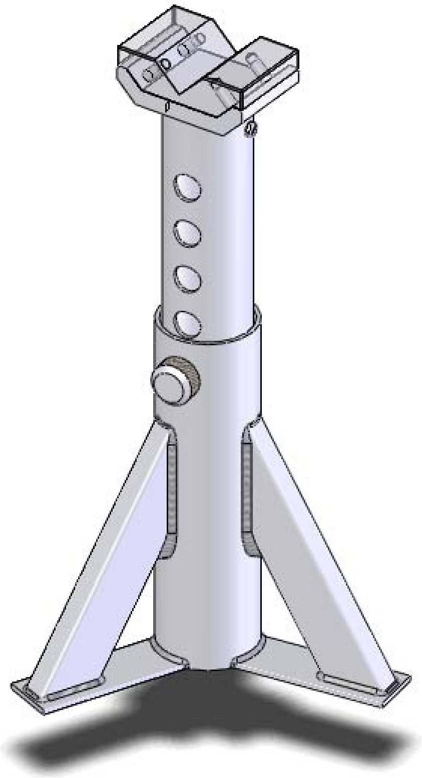
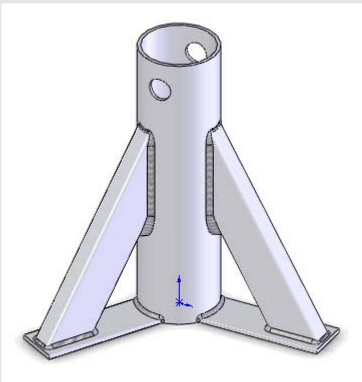
Axle Support

In this tutorial, we will build an axle support. It is a rather complex product, with several different parts. We will repeat a lot of the functions that you have already learned, but we will also introduce some new topics with SolidWorks. We will show you how to build simple constructions from tubes and profiles using weldments. We will also utilize patterns for the first time.



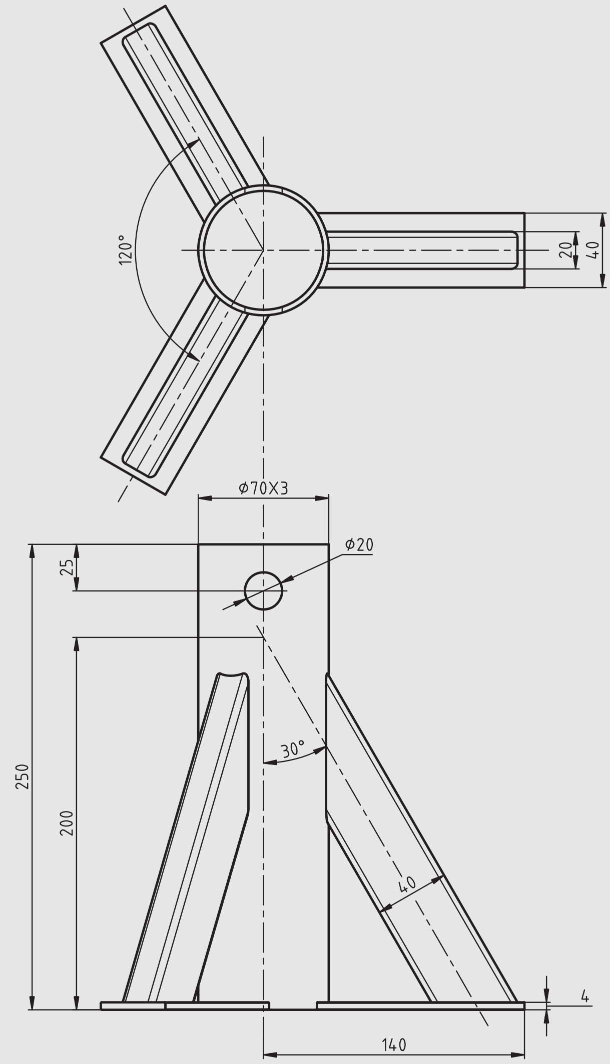
Work plan

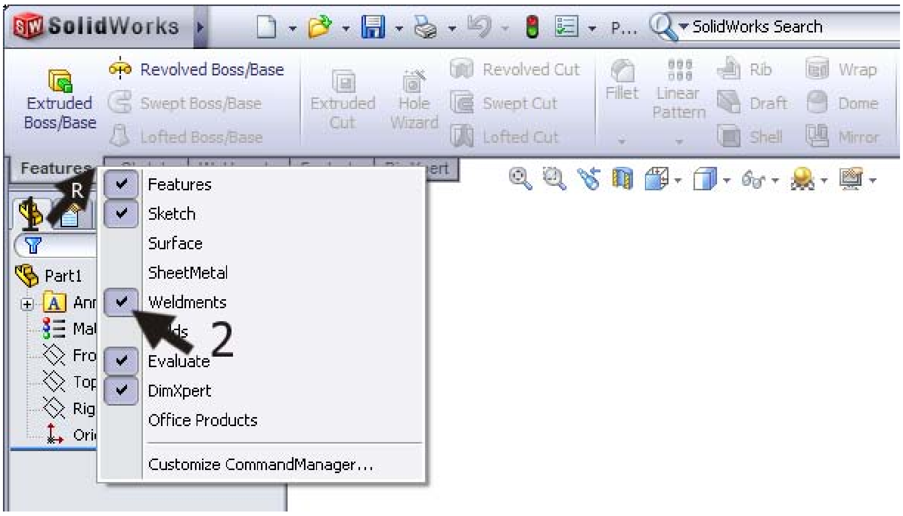
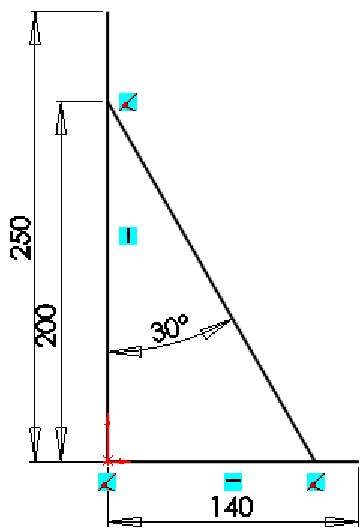
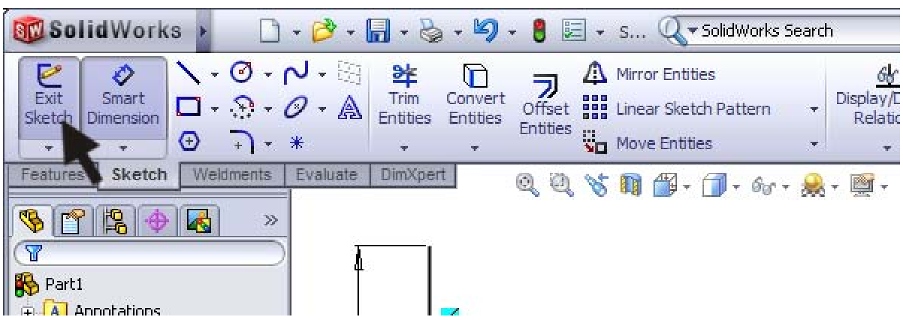
We will create the base of the support first. As you can see in the illustration below, the base consists of 7 parts that are welded together.

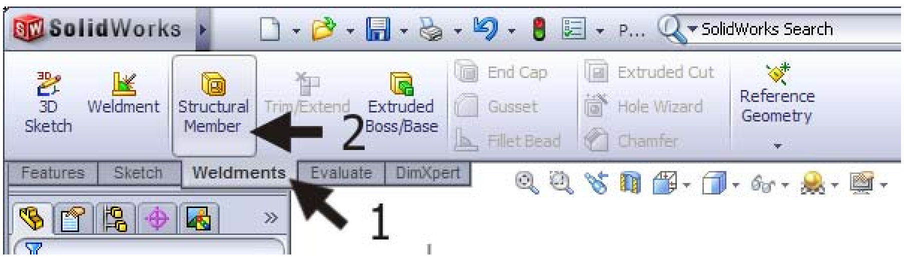
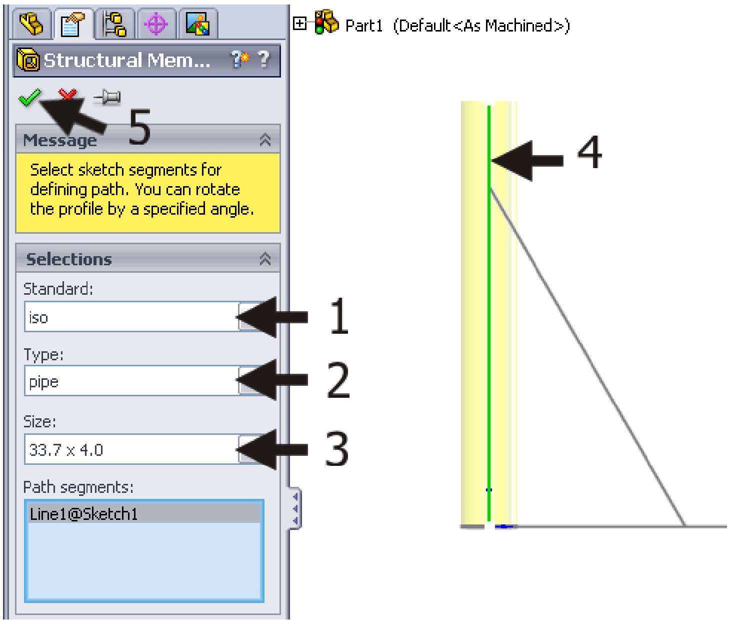


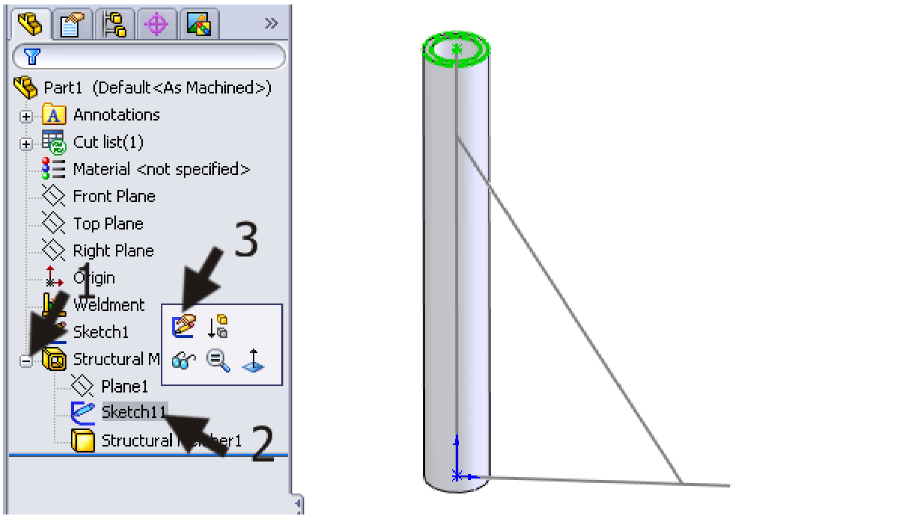
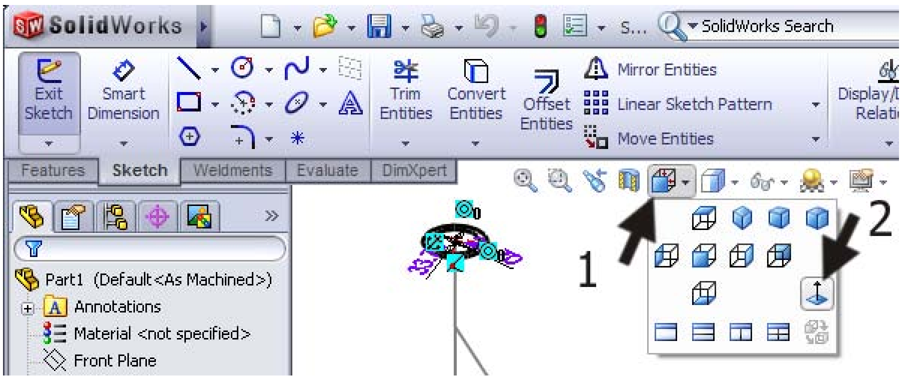
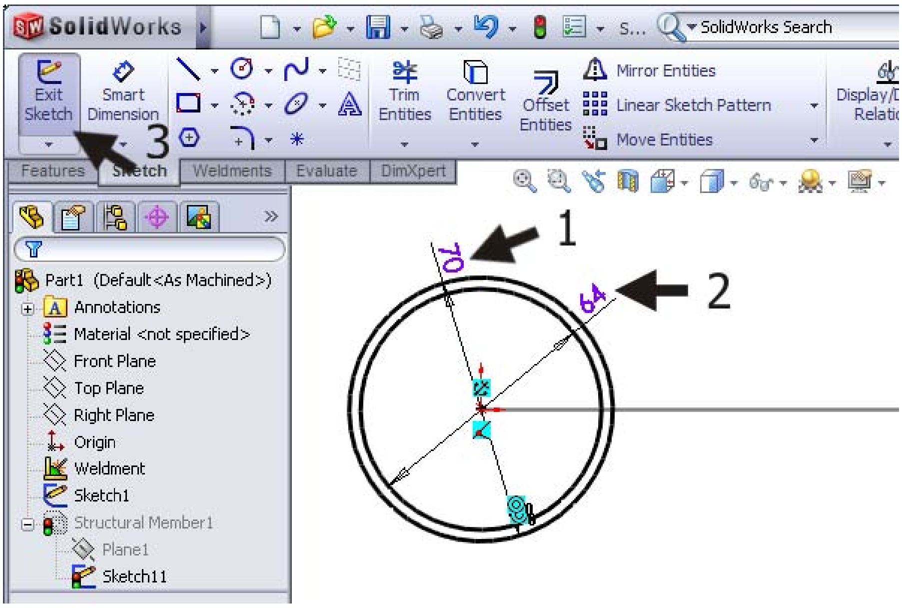
You could build this in the same manner we have worked in up until now: create the parts first and then assemble them with the assembly command. However, in this case that approach would be overly time-intensive and laborious. Just think about how you would shape the sloped supports, including the dimensions. That approach would not be easy.

Fortunately, we have another option for modeling this design SolidWorks: weldments. With the weldments command you can build standard tubes

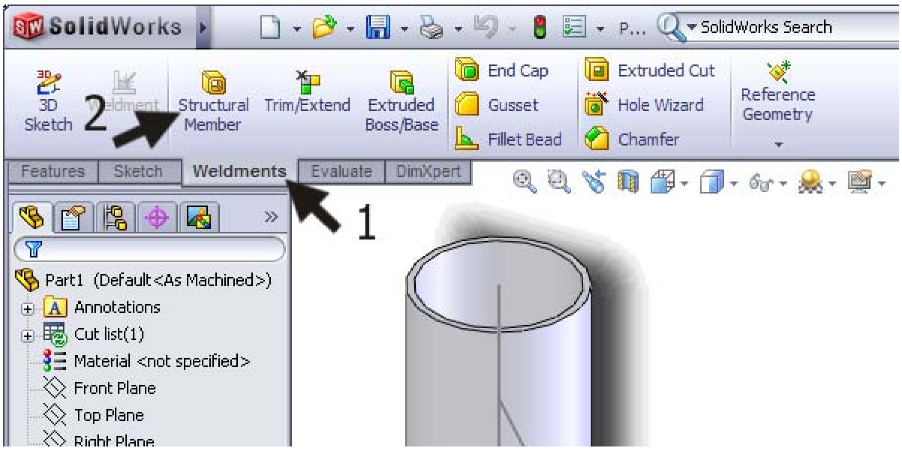
		and profiles within a single part. You can also save each part as a separate file, if you want. We will perform the next few steps: 1. First, we will create a round vertical tube, one of the bottom strips and one of the diagonal square-shaped tubes. 2. After that step, we will add the weldments. 3. Next, we will copy the parts around the vertical tube, so there will be three supports connected to the central tube. 4. Finally, we will make a hole at the top of the round tube. 

1	Start SolidWorks and open a new part.	
2	Make sure the 'Weldments' function is available. As we did when we worked with SheetMetal in Tutorial 4, we will now add the 'Weldments' keys to the CommandManager. 1. Right-click on a tab in the CommandManager. 2. Check the option 'Weldments'.	
3	Select the Front Plane, and create a sketch as shown on the right. 1 Draw a vertical line from the origin. 2 Draw a horizontal line from the origin. 3 Draw a diagonal line beginning and ending on the first two lines. 4 Set the dimensions in the sketch.	
4	Click on 'Exit Sketch' in the CommandManager to end the 'Sketch' command.	

5	1. Click on ‘Weldments’ in the CommandManager 2. Click on ‘Structural Member’. With this command you can add tubes and profiles to a construction.	
6	Set the following features: 1 Select ‘ISO’ as the ‘Standard’. 2 Select ‘Pipe’ as the profile ‘Type’. 3 Set the dimension to ‘33.7 x 4.0’. 4 Select the vertical line in the sketch. 5 Click on OK.	
	Tip!	There are a small number of pre-defined tubes and profiles in SolidWorks. To be able to use exactly the right tube, there are two possibilities: 1. Create a new tube and add it to the library. You do this once and then you can use this part every time you need it. Adding the part is not difficult, but you will not have the access rights to do so in a school environment. For this reason, we will not explain this procedure as part of this tutorial. 2. The second option is to use an existing tube from the library, which looks similar to the one you need. You can then adapt or alter the dimensions to use it every time you need this part. In this tutorial we will use the second method.

7	Find the feature (the tube) you have just made in the FeatureManager. This is called 'Structural Member1' (the number can vary). 1 Click on the '+' symbol in front of the name of the feature. 2 Right-click on the sketch in this feature. 3 Click on Edit Sketch.	
8	Click on Standard Views in the View Orientation, and then on Normal To.	
9	Change the two dimensions in the sketch: 1 The inside diameter must be set to '64'. 2 The outside diameter must be set to '70'. 3 Click on 'Exit Sketch'.	

<table>
<tr>
<td>10</td>
<td>Rotate the model so you can get a clear view.

Click on ‘Weldments’ in the CommandManager and next on ‘Structural Member’.</td>
<td>
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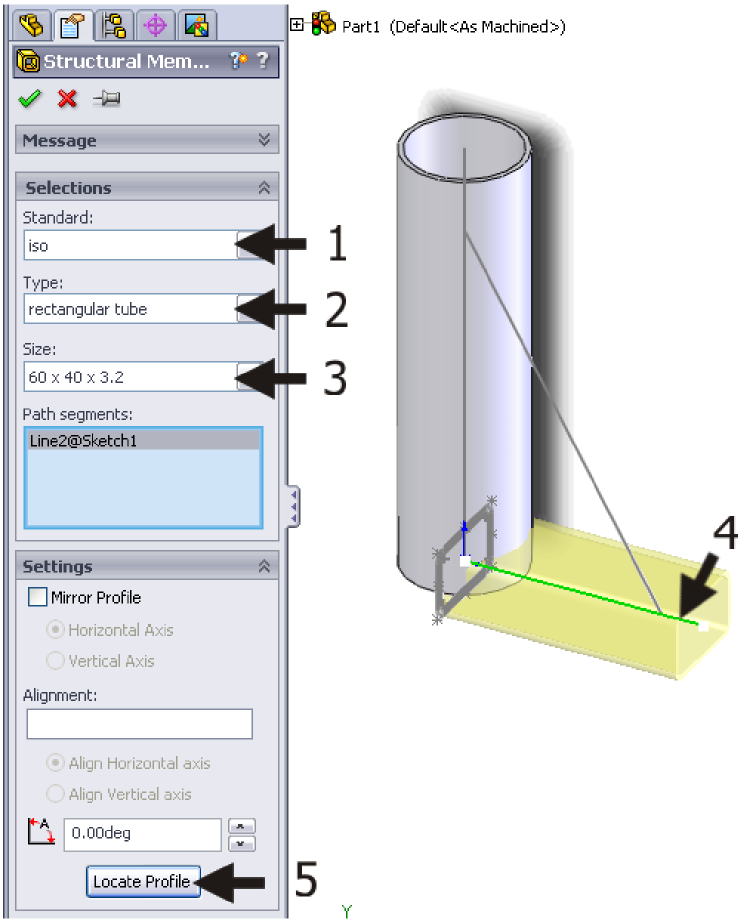
<tr>
<td>11</td>
<td>Set the following items in the PropertyManager:

1 Select ‘ISO’ as the ‘Standard’.

2 Select the ‘rectangular tube’ as the profile ‘Type’.

3 Select a size of ‘60 x 40 x 3.2’.

4 Select the horizontal line in the sketch.

5 Click on ‘Locate Profile’.</td>
<td>
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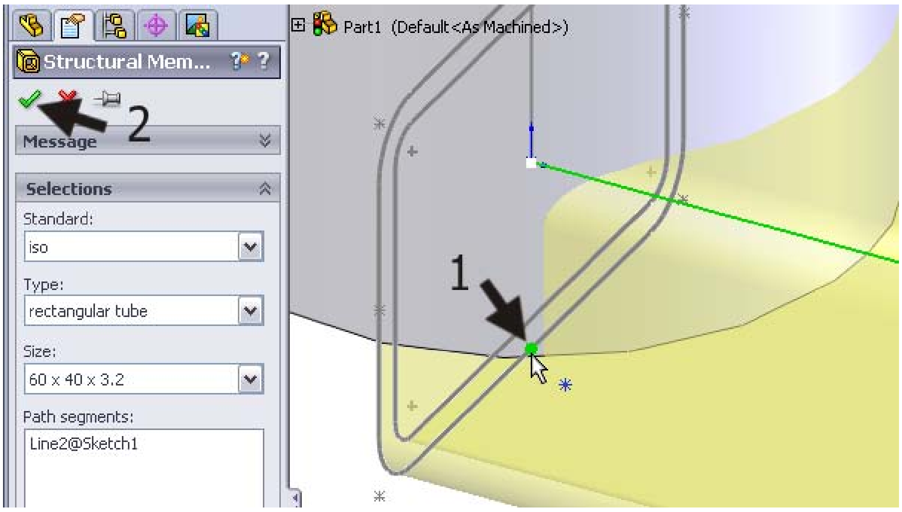
<td>12</td>
<td>SolidWorks will automatically zoom in on the profile now.

1 Click in the middle of the bottom line of the profile. The profile will move upward.

2 Click on OK.</td>
<td>
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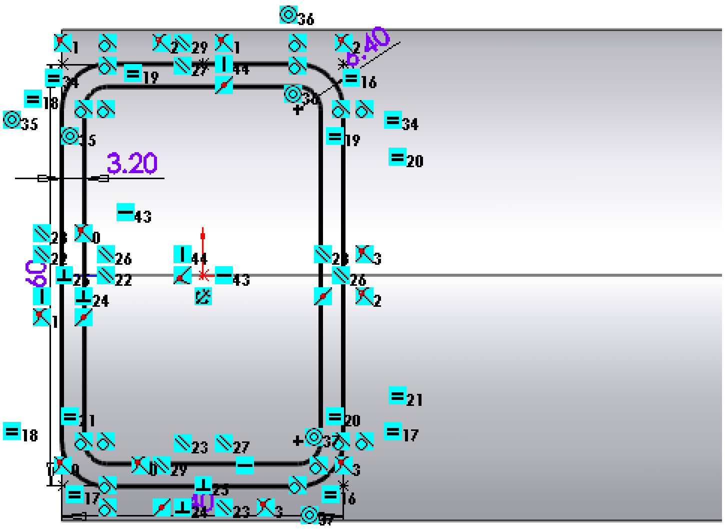
<tr>
<td>13</td>
<td>Open the sketch from this rectangular tube, just as you did previously (steps 7, 8 and 9).

This sketch looks pretty complicated because of the presence of a great number of relations.

We will convert the tube into a strip.</td>
<td>
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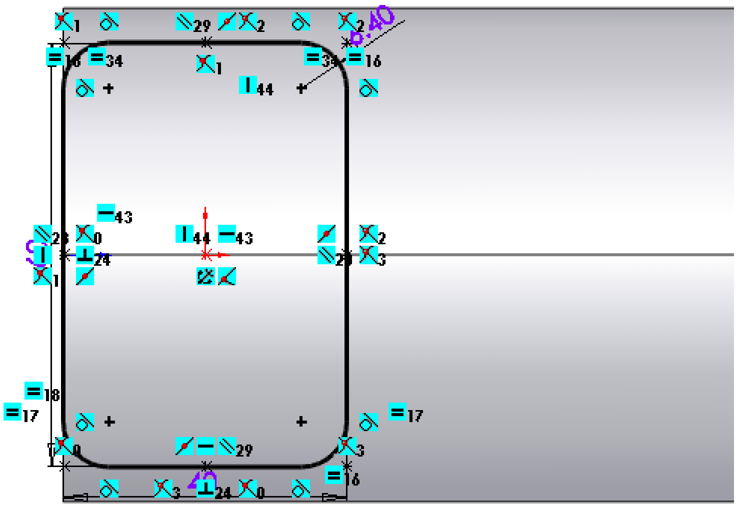
<td>14</td>
<td>Remove the inner contour of the tube: click on a line or bend and push the <Del> delete key on the keyboard.</td>
<td></td>
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15

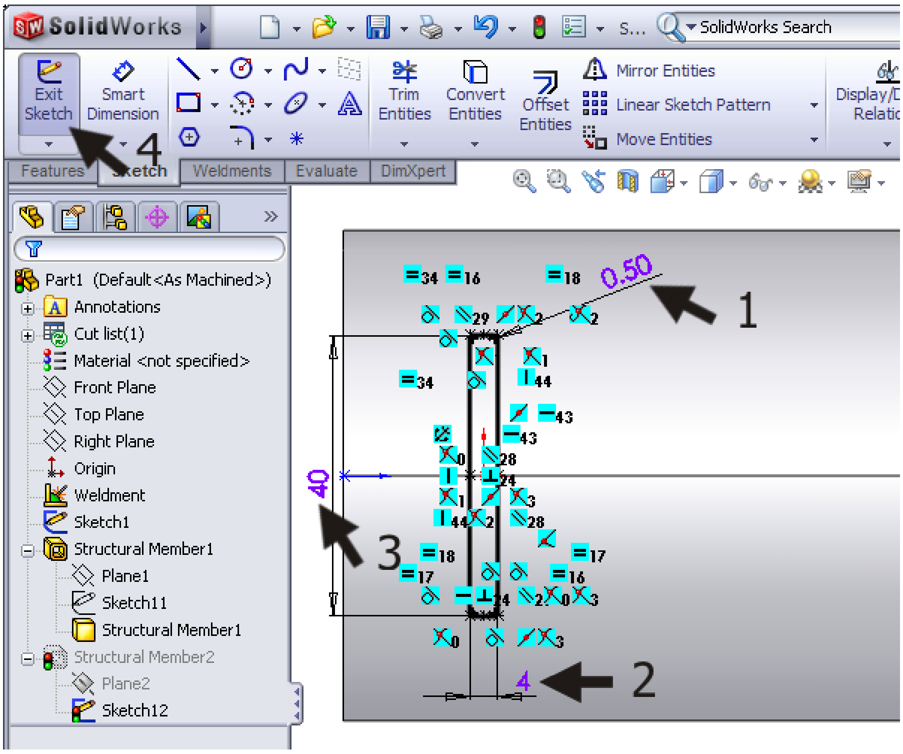
Next, change the dimensions:

1 The radius is set to '0.5'.

2 The height will be '4mm'. The profile is no longer the same height as the bottom of the tube. This is 0, because after you have clicked on Exit Sketch, as in step 4, everything will be all right again.

3 Change the width to '40mm'.

4 Click on 'Exit Sketch'.



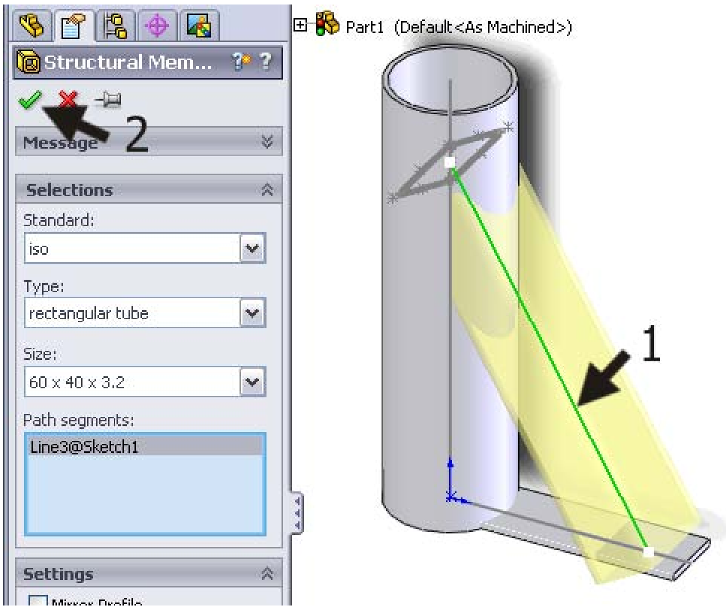
16

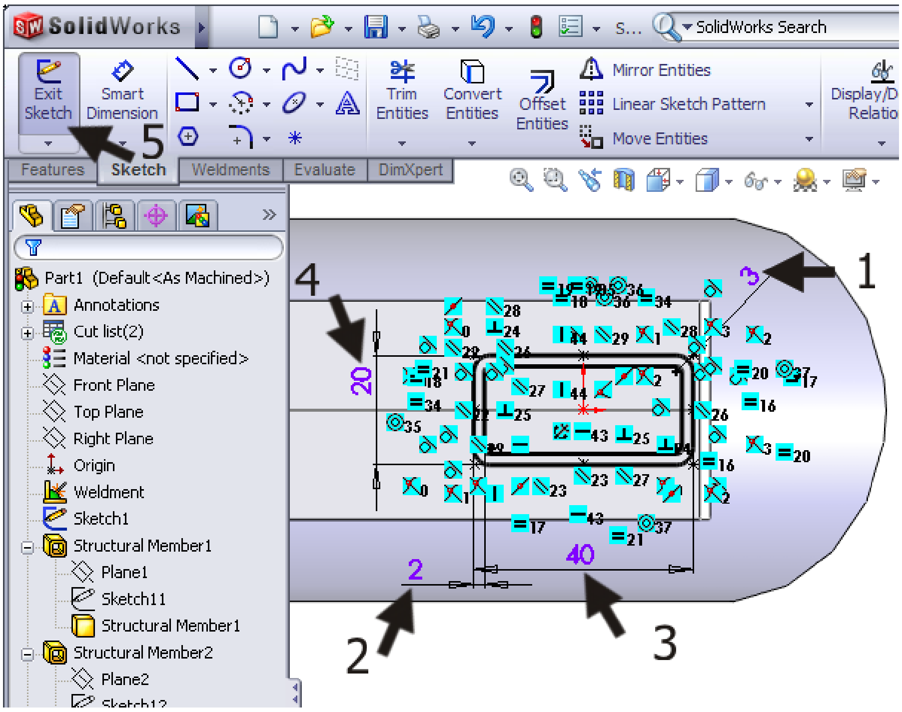
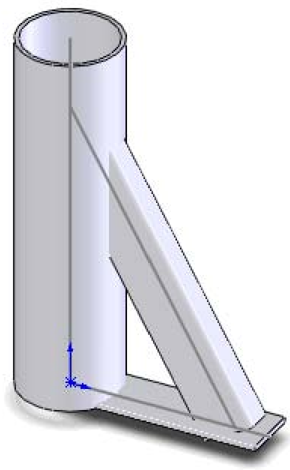
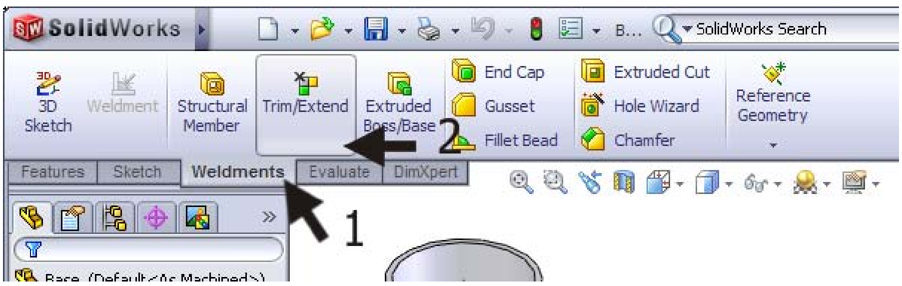
Now, we will create the last tube. Click on 'Weldments' in the CommandManager again and after that on 'Structural Member'.

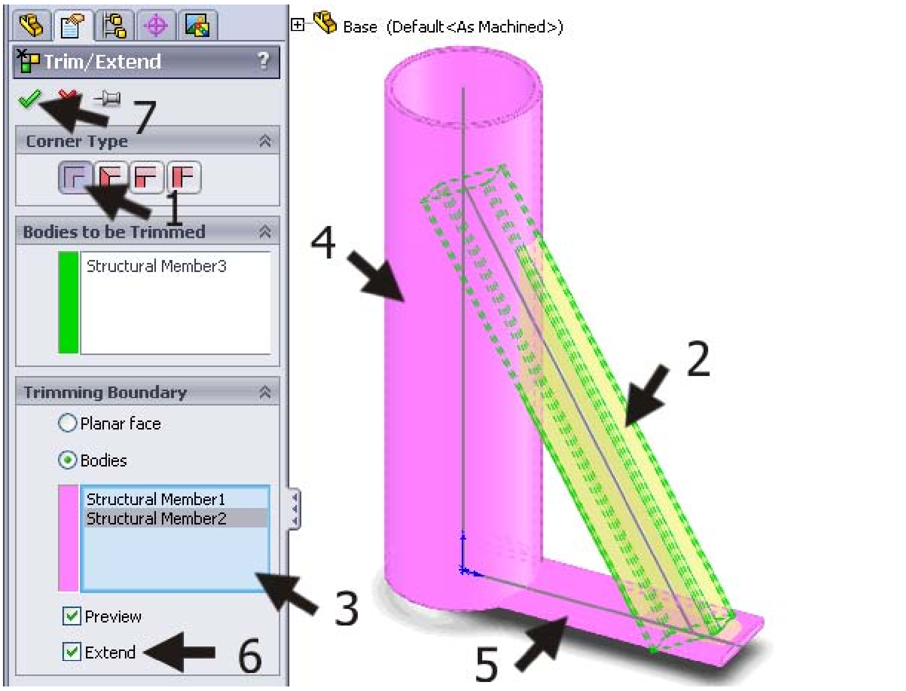
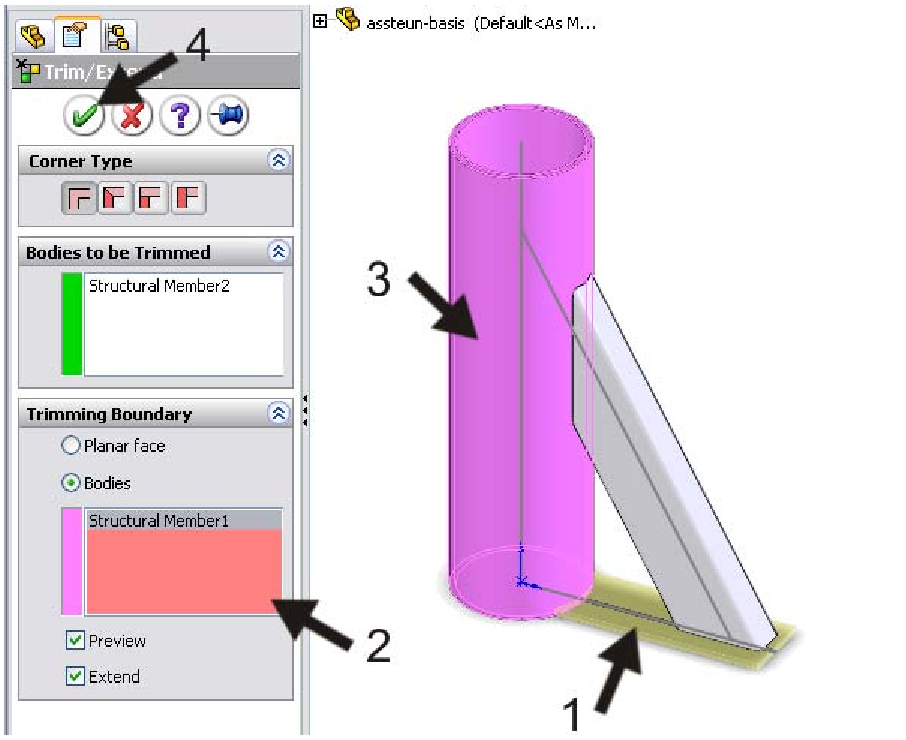
Use the same settings for the tube. You do not have to change any of them

1 Select the diagonal line.

2 Click on OK.



17	Open the sketch of the tube to alter the dimensions: 1 The radius of the tube is set to ‘3mm’. 2 The thickness must be set to ‘2mm’. 3 The height is ‘40mm’. 4 The width is ‘20mm’. 5 Click on ‘Exit Sketch’.	
18	Save this file as: base.SLDPRT.	
19	Click on ‘Weldments’ in the CommandManager and next on ‘Trim/Extend’. With this command we will make sure that the tubes will fit together (and do not intersect each other any more).	

20	Set following items: 1 Make sure that the first option End Trim is selected in the 'Corner Type' tab field : 2 Select the diagonal tube. It will be mentioned in the 'Bodies to be Trimmed' field. 3 Click on the selection field next to 'Trimming Boundary'. This will turn active now (it will turn blue). 4 Select the round tube. 5 Select the strip. 6 Make sure the option 'Extend' is checked. 7 When the model looks OK, click on OK.	
21	We still have to shorten the bottom strip. Select 'Trim/Extend' in the CommandManager again. Most of the settings will be still there from the last time we did this. 1 Select the bottom strip. 2 Click on the selection field next to 'Trimming Boundary'. 3 Select the vertical tube. 4 Click on OK.	

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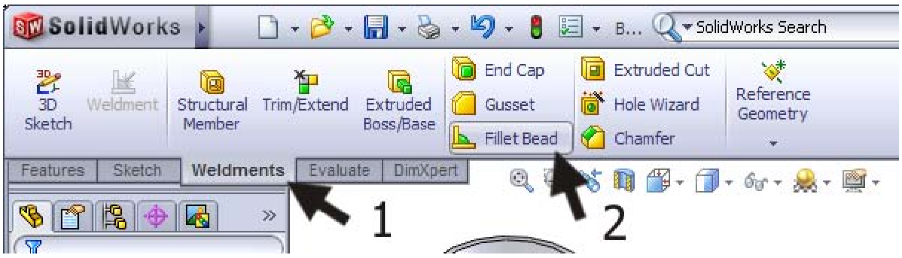
<tr>
<td>22</td>
<td>To make the weldments, click on ‘Weldments’ in the CommandManager and next on ‘Fillet Bead’.</td>
<td>
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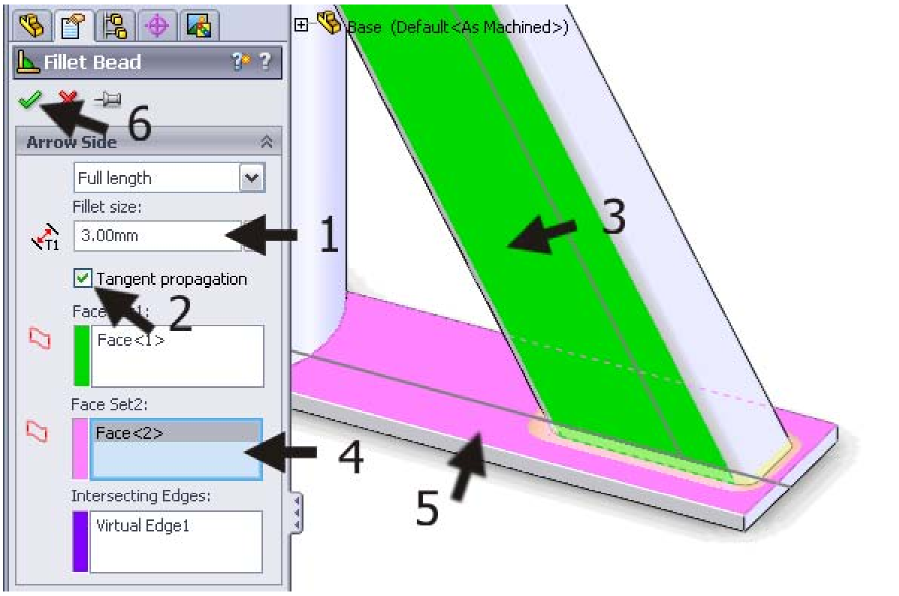
<tr>
<td>23</td>
<td>Set the following items:
1 Set the weld dimension to ‘3mm’.
2 Check the option ‘Tangent propagation’: this will make sure the weld is made around the tube.
3 Select a plane from the rectangular tube.
4 Click in the ‘Face Set2’ area to activate it (it will turn blue).
5 Select a plane from the strip.
6 Click on OK.</td>
<td>
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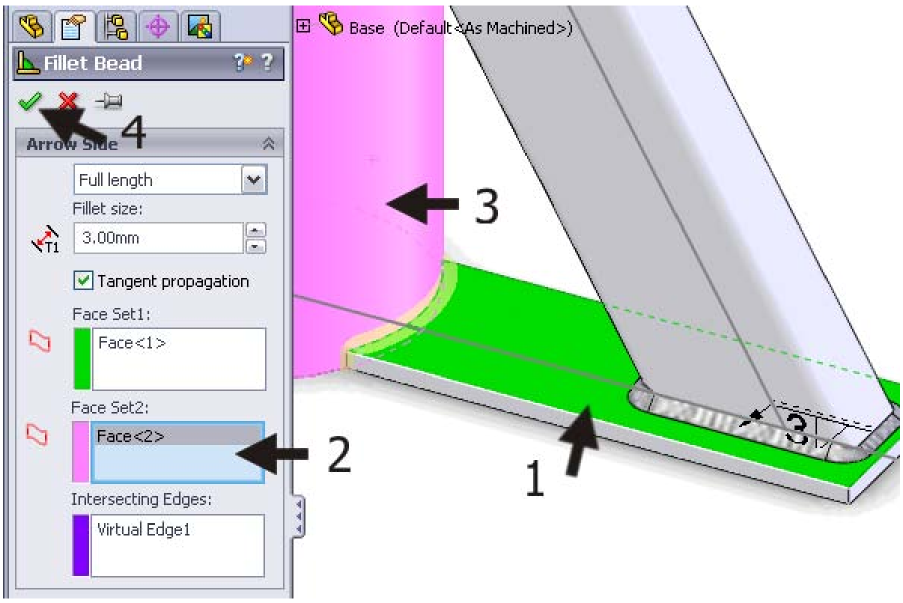
<td>24</td>
<td>We will now weld the section between the strip and the tube. Click on ‘Fillet Bead’ in the CommandManager. Most settings will remain the same as in the last weld we made.
1. Select the top plane of the strip.
2. Click in the ‘Face Set2’ selection field to activate it (it will turn blue).
3. Select the tube.
4. Click on OK.</td>
<td>
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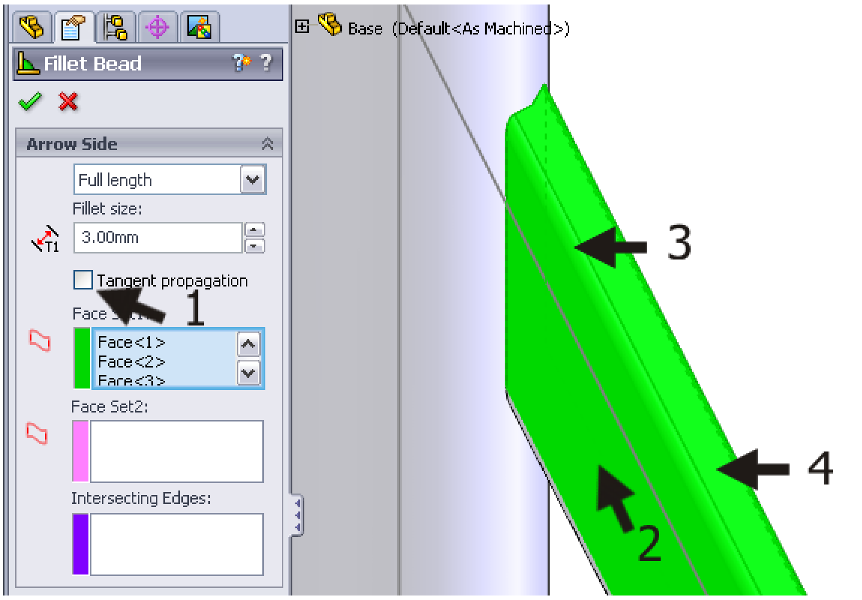
<td>25</td>
<td>We will now make the final weld between the diagonal tube and the round vertical tube. We will not weld the bottom section of this connection.
1. Uncheck the option ‘Tangent propagation’.
2. Select the side plane from the rectangular tube.
3. Select the rounded edge from the tube.
4. Select the top surface plane from the tube.</td>
<td>
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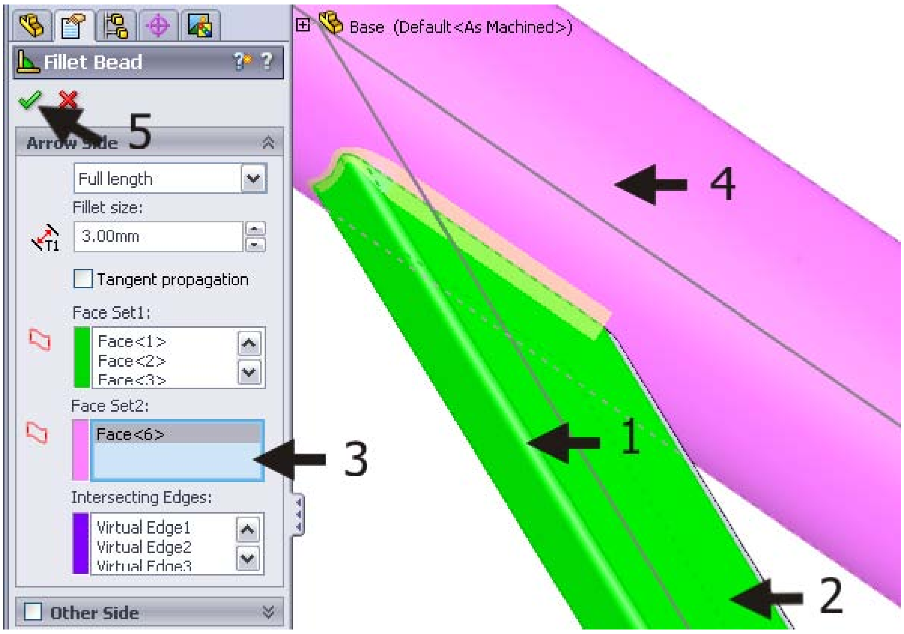
<tr>
<td>26</td>
<td>Rotate the model so you see the other side of this part.
1. Select the rounded edge.
2. Select the side plane.
3. Click on the ‘Face Set2’ selection field to activate it (it will turn blue).
4. Select the vertical tube.
5. Click on OK.</td>
<td>
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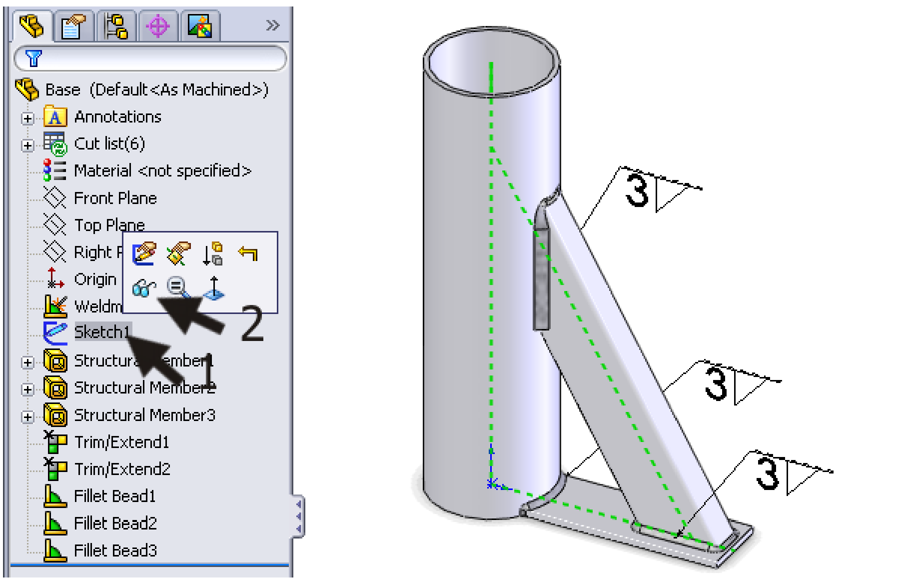
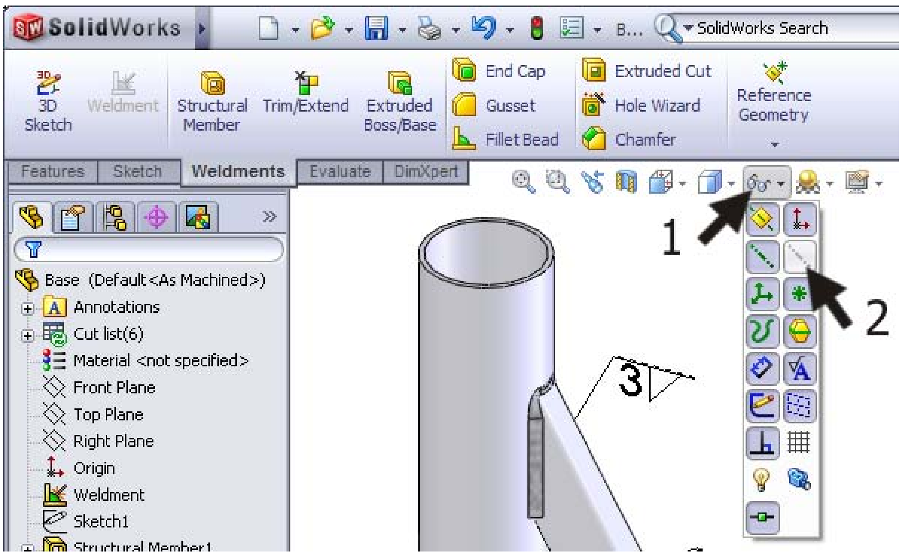
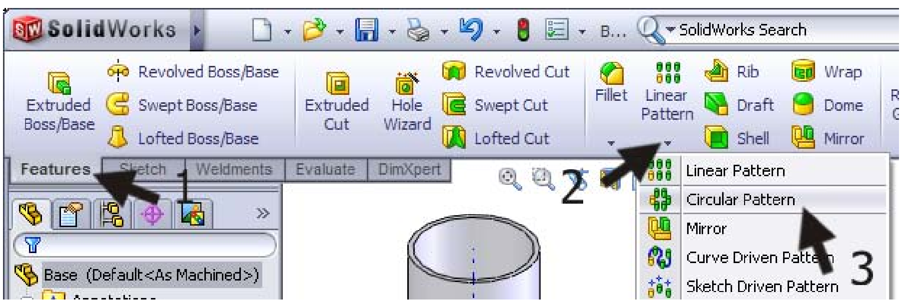
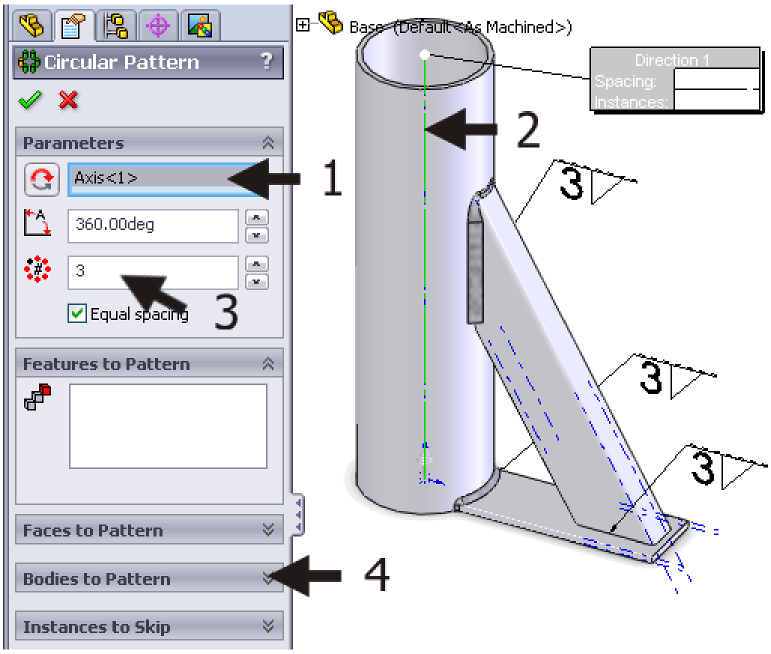
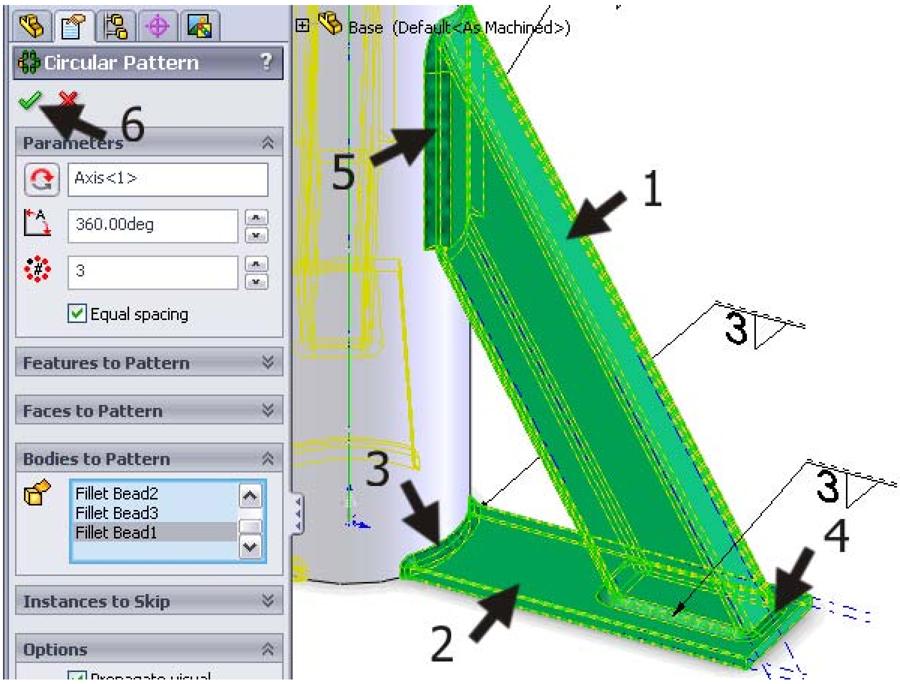
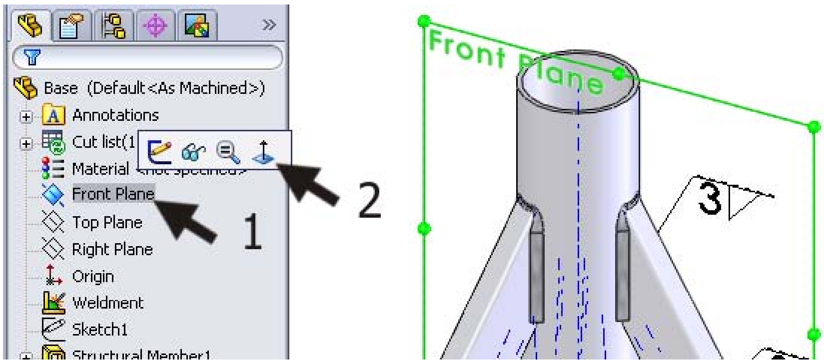
27	We can also hide the original sketch that we used before. 1 Click on the first sketch in the FeatureManager. 2 Select Hide in the pop-up menu.	
28	One of the supports of the product is now ready and we will copy it twice around the vertical tube. We will use the centerline from the tube to do so, but first we have to show it. 1 Click on Hide/Show Items. 2 Set the option Temporary Axes.	
29	Click on 'Features' in the CommandManager and select 'Circular Pattern'. You may have to open the extended menu first.	

30	Set the next items in the PropertyManager: 1 Click in the selection area of the 'Axis' pattern. 2 Select the centerline from the vertical tube as a rotation axis. 3 Set the number of items in the pattern to '3'. 4 Open the menu 'Bodies to Pattern'.	
31	Select all of the parts that you want to rotate: 1 The rectangular tube. 2 The strip. 3 The weldment between the strip and the tube. 4 The weldment between the strip and the diagonal tube. 5 The weldment between the vertical and diagonal tube. 6 When all parts are selected, click on OK.	
32	Finally, we have to create a hole in the support. 1 Select 'Front Plane' in the FeatureManager. 2 Click on Normal To in the pop-up menu.	

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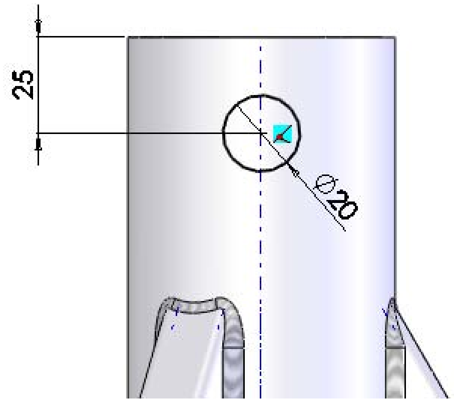
<tr>
<td>33</td>
<td>Make a sketch as in the illustration on the right.

Draw a circle and put the midpoint on the centerline of the tube.

Set the two dimensions as shown.</td>
<td>
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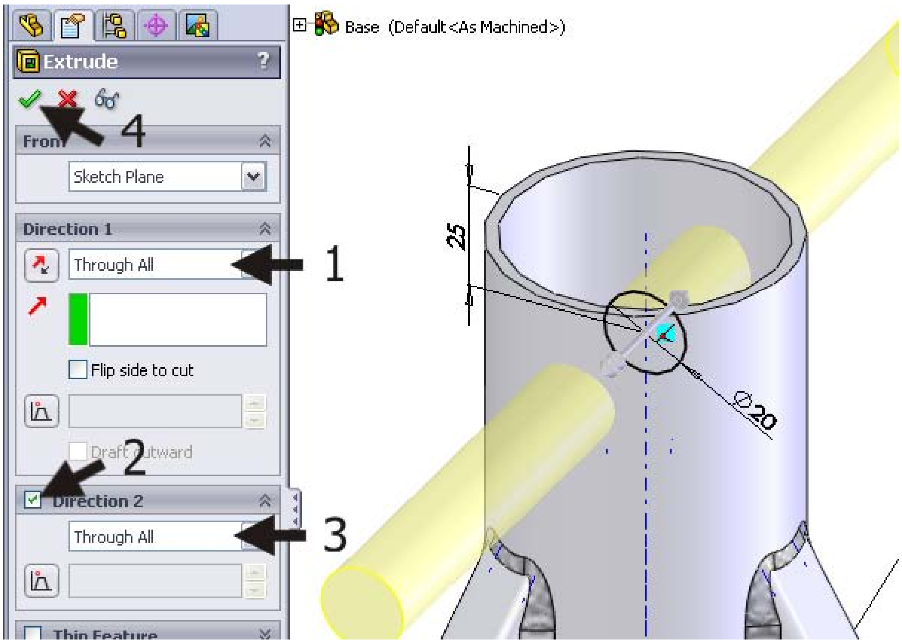
</tr>
<tr>
<td>34</td>
<td>Make an Extruded Cut from the sketch. Set the following items in the Property-Manager:

1 Set the option ‘Through All’ in the ‘Direction1’ field (through the entire model).

2 Activate menu ‘Direction2’ also, because the hole has to be through both sides.

3 Set the depth to ‘Through All’.

4 Click on OK.</td>
<td>
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<td>35</td>
<td>This part is now ready. Hide the Temporary Axes.</td>
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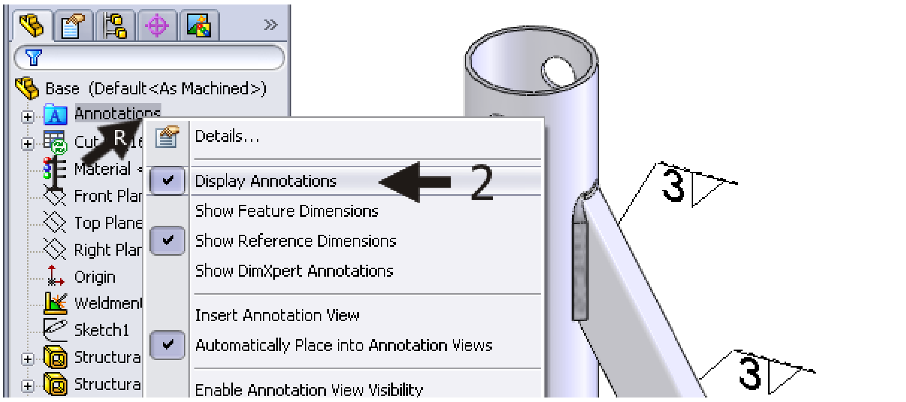
<td>36</td>
<td>To hide the welding icons, follow the next few steps:
1. Right-click on the map ‘Annotations’ in the FeatureManager.
2. Uncheck the option ‘Display Annotations’.</td>
<td>
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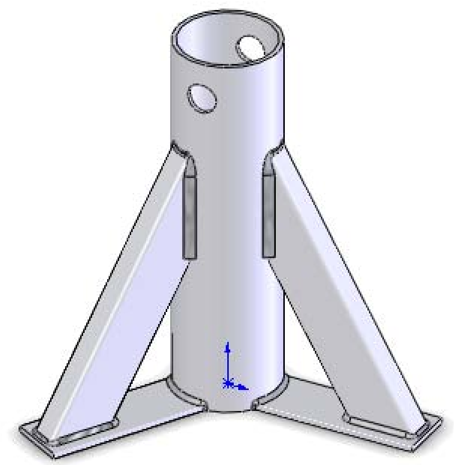
<td>37</td>
<td>Save the file.</td>
<td></td>
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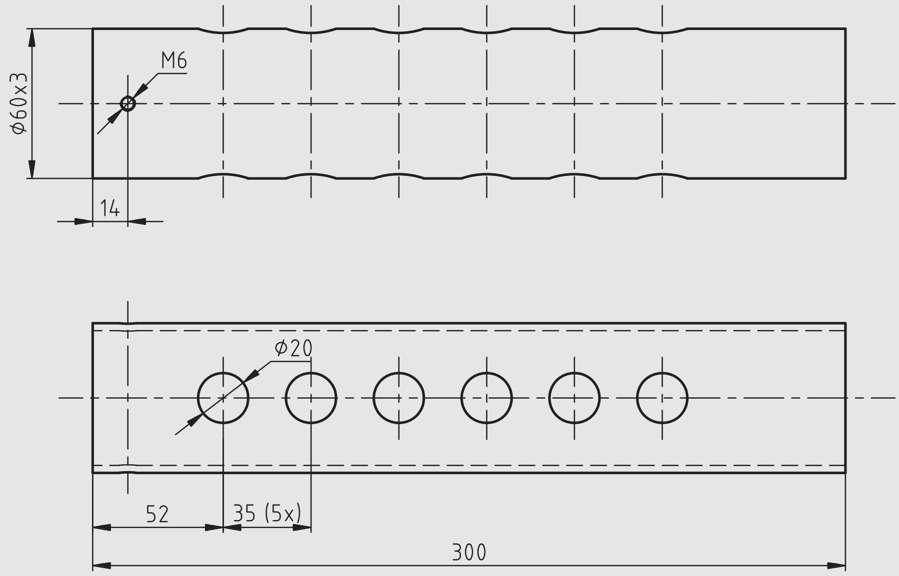
<td></td>
<td>Work plan</td>
<td>The second part will be the expandable inner tube based on the drawing below.


This part is not as complicated. We will build it following these steps:
1. Make the tube.
2. Make only one of the bigger holes.</td>
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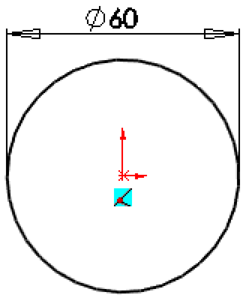
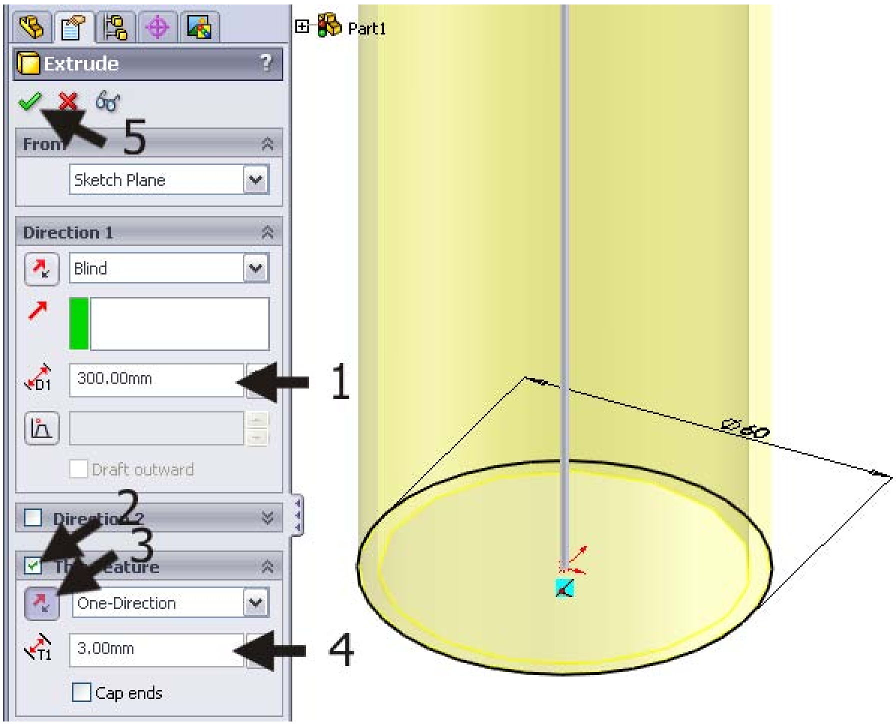
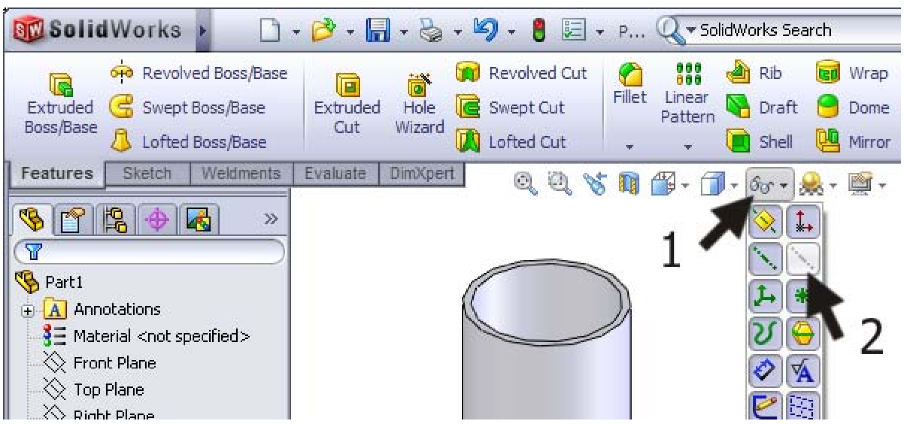
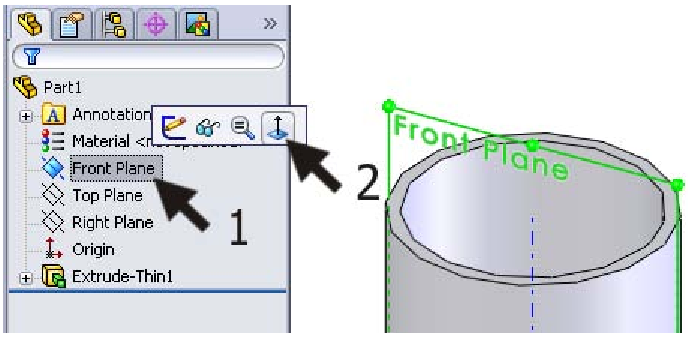
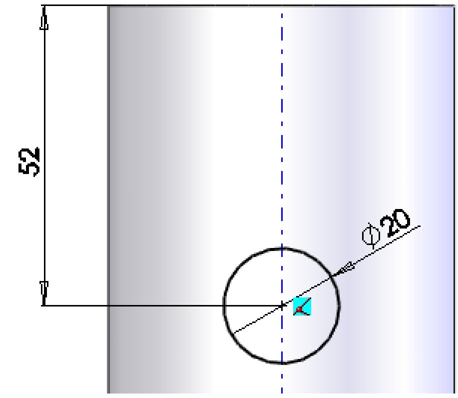
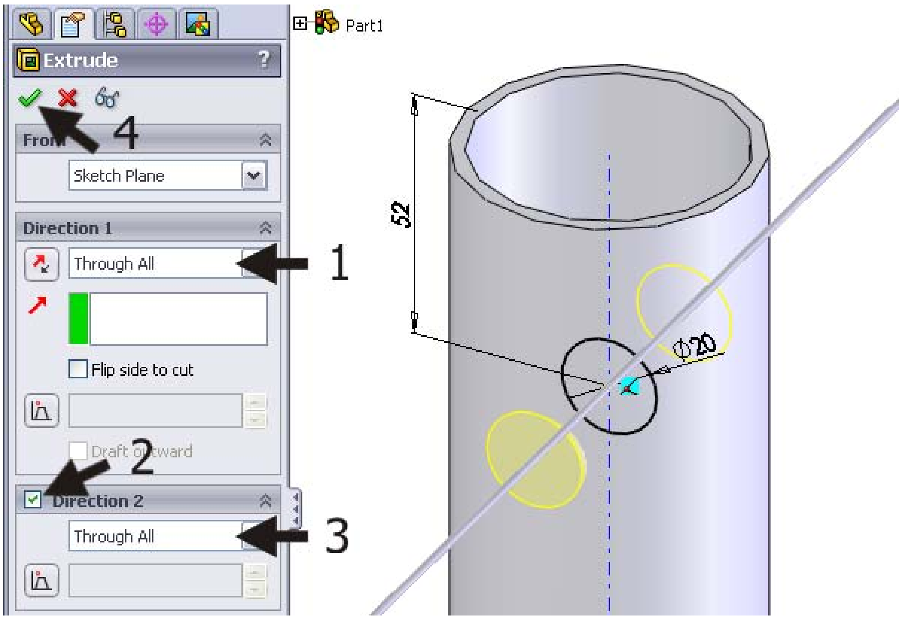
</table>

		3. Copy the holes. 4. Make the small hole.
38	Open a new part and start sketching on the Top Plane. The sketch consists of one circle with the mid-point at the origin.	
39	Go to 'Features' and make an 'Extruded Boss/Base'. Set following items in the PropertyManager: 1 Set the length to '300mm'. 2 Activate the menu 'Thin Feature'. By doing so, you will create a hollow tube instead of a massive part. 3 By clicking 'Reverse Direction' you can determine if the material is added to the inside or the outside of the circle. Watch the model closely. Make sure the material is added at the inside of the circle. 4 Set a thickness T1 of '3mm'. 5 Click on OK.	
40	Display the centerline of the tube: make sure the view Temporary Axes is selected.	

<table>
<tr><td>41</td><td>Select the 'Front Plane' to make a sketch on it and make sure you have a clear view of it.</td><td>
</td></tr>
<tr><td>42</td><td>Make a sketch as shown in the illustration. Make sure the midpoint of the circle is on the centerline of the tube.</td><td>
</td></tr>
<tr><td>43</td><td>Make an Extruded Cut from this sketch. Set the following items in the PropertyManager:
1 Select the depth: 'Through All'.
2 Activate the 'Direction2' menu.
3 Set depth 'Through All'.
4 Click on OK.</td><td>
</td></tr>
<tr><td>44</td><td>Click on 'Linear Pattern' in the CommandManager.
With this feature we will copy the hole several times.</td><td>
</td></tr>
</table>

<table>
<tr>
<td>45</td>
<td>
1 First, you have to set the direction in which the elements should be copied. For this, you have to select the centerline of the tube.

2 Set the distance between two holes to '35mm'.

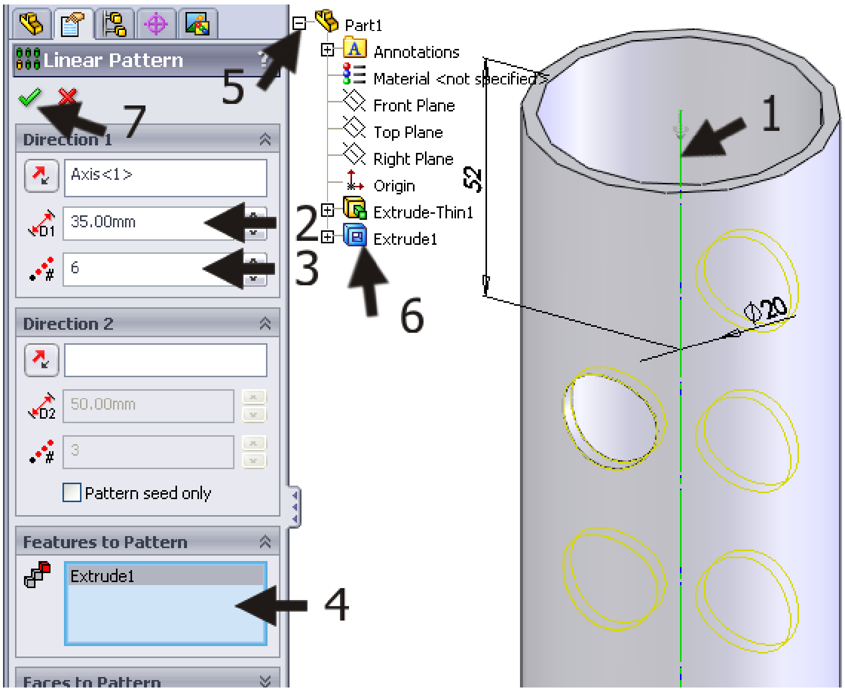
3 Set the number to '6'.

4 Click on the 'Features to Pattern' selection field.

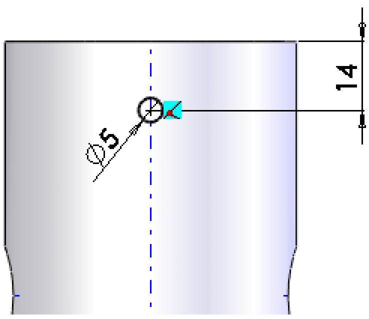
Next, you have to select the hole. You can do it in the model, but it is easier to do so in the FeatureManager.

5 Open the FeatureManager tree next to the model.

6 Select the last feature in the list.

7 When the preview looks ok to you, click on OK.
</td>
<td>


</td>
</tr>
<tr>
<td>46</td>
<td>
Next, make the small hole at the top. Select the Right Plane and make the sketch as shown.

Make an Extruded Cut in two directions 'Through All', like you did in Step 43.
</td>
<td>


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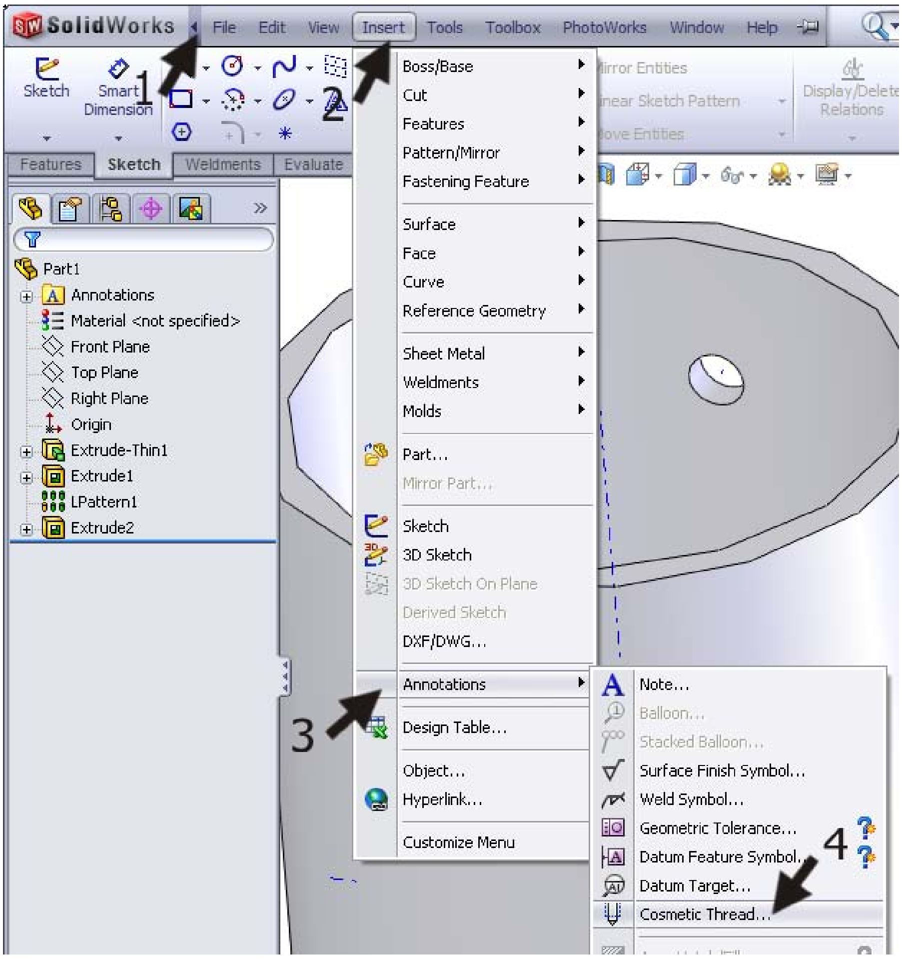
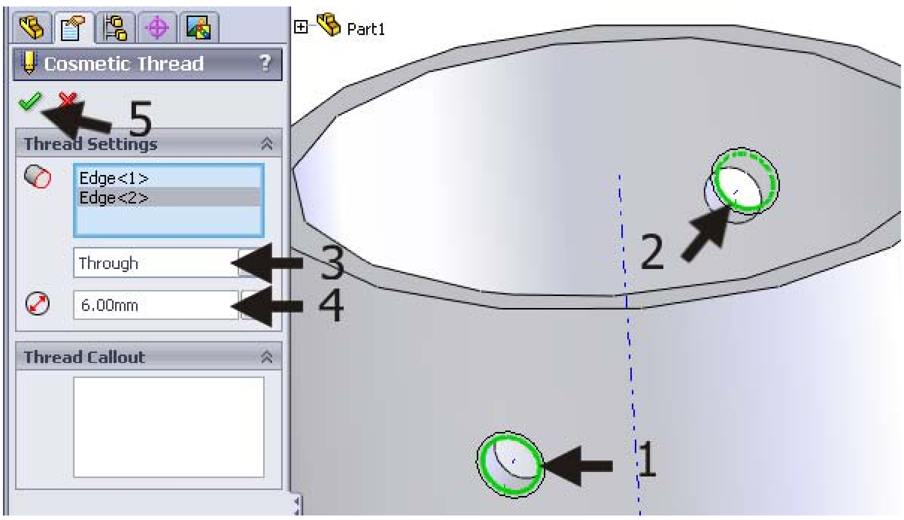
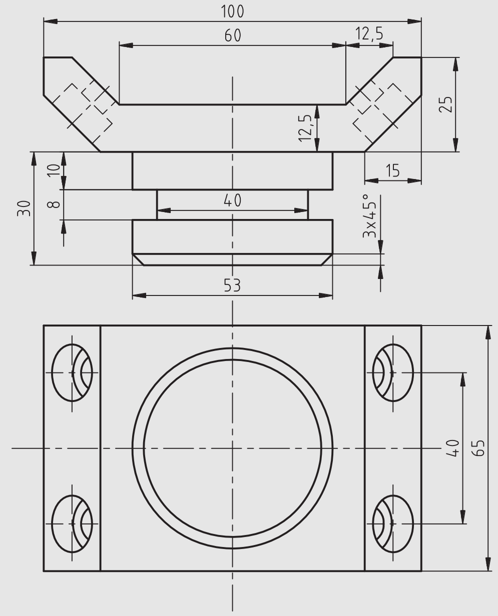
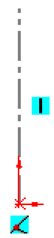
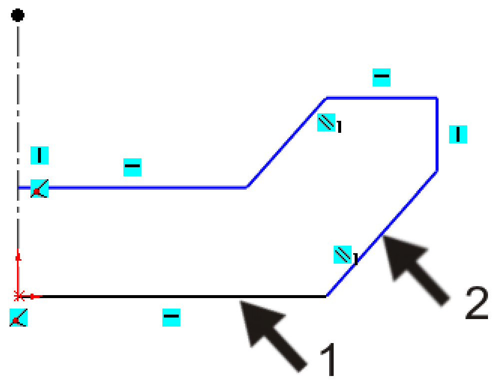
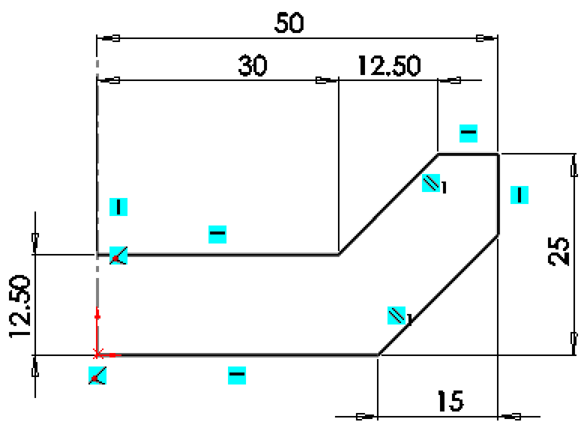
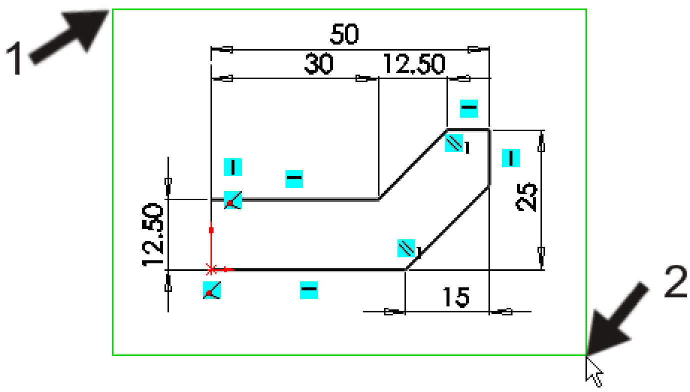
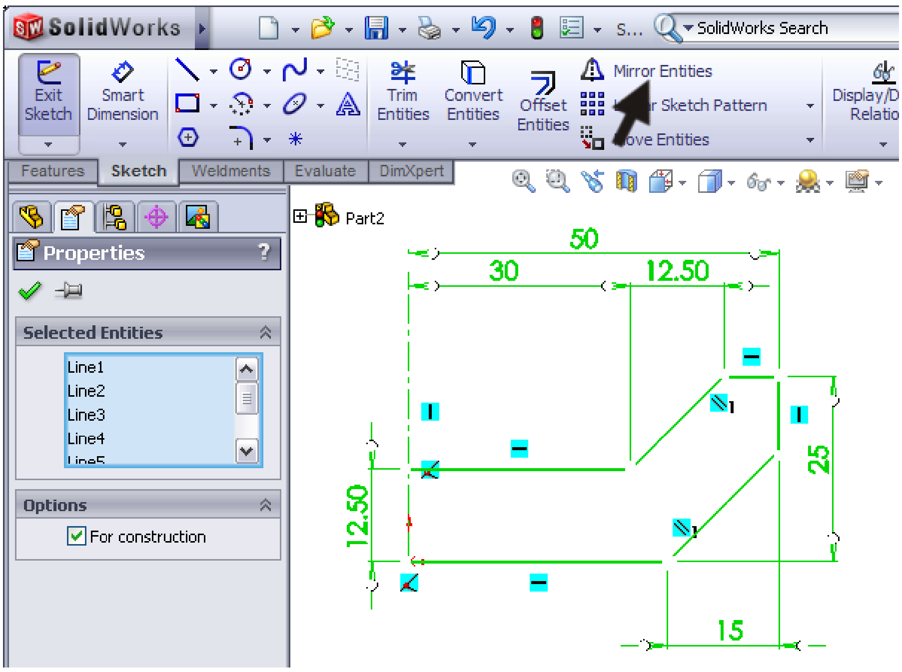
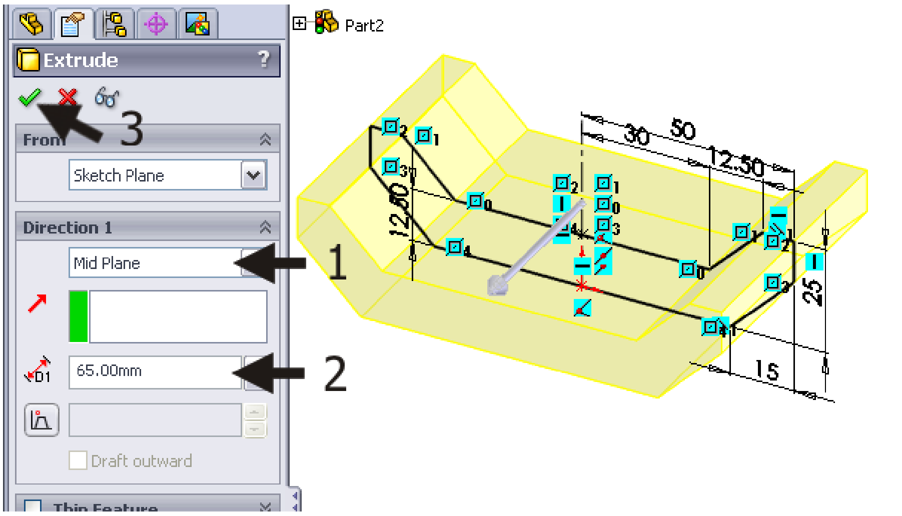
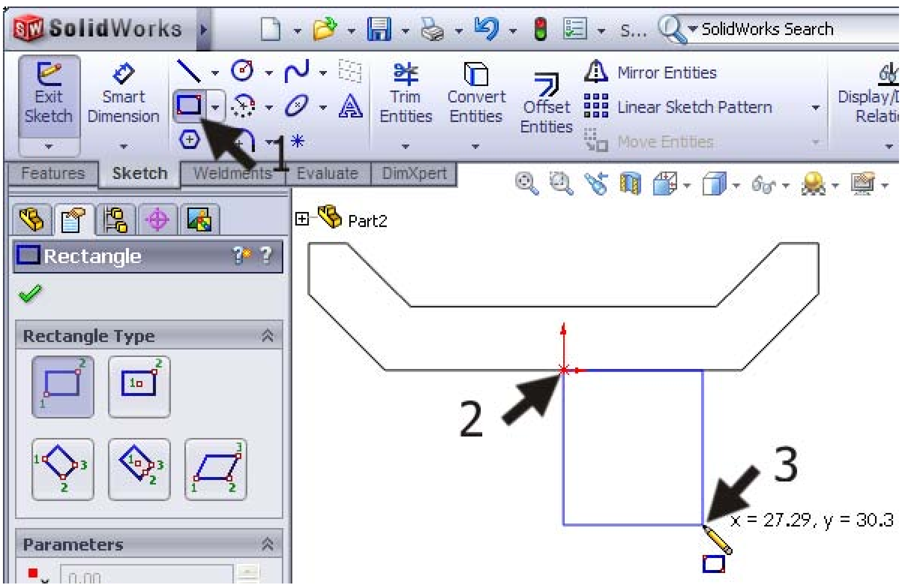
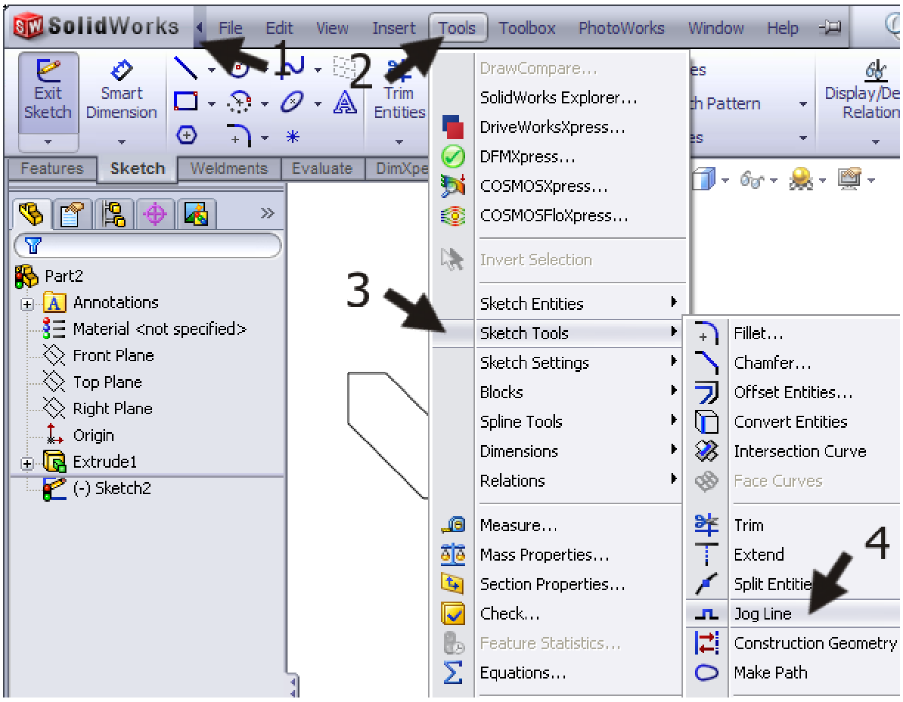
</table>

<table>
<tr>
<td>47</td>
<td>To add a screw thread to the hole, select the following items in the pull-down menu:
1. Open the pull-down menu.
2. Click on ‘Insert’.
3. ‘Annotations’.
4. ‘Cosmetic Thread’.</td>
<td>
</td>
</tr>
<tr>
<td>48</td>
<td>Select the edges of the holes in which you want to put the thread.
3. Set the depth to ‘Through’.
4. Set the diameter to ‘6’.
5. Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>49</td>
<td>Hide the Temporary Axes again and save the model as: pipe.SLDPRT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Work plan</td>
<td>The next part we will build is the support block on top. We will create this part from two features: an extrusion and a rotation. After that, we will make the countersink holes with the Hole Wizard. The difficulty with this part is that you have to draw two different sketches and join them togeth-</td>
</tr>
</table>

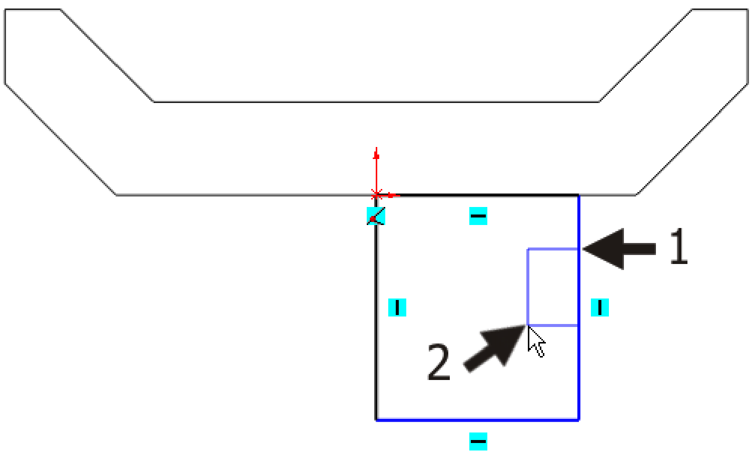
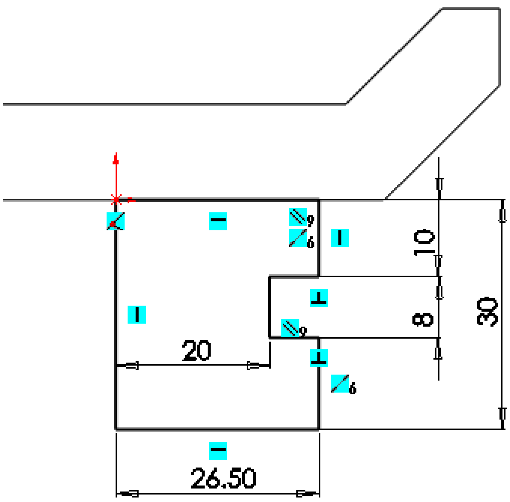
		er. 
50	Open a new part, select the Front Plane and make a sketch. Draw a vertical centerline from the origin up (length of about 40mm).	
51	Next, make a horizontal line (not a centerline) according to the sketch as shown on the right. 1. The first line is a horizontal line from the origin with a length of approximately 40mm. 2. Draw the rest of the sketch from this point on. The sizes are not important yet. Only make sure that the end of the last line is on the centerline again.	

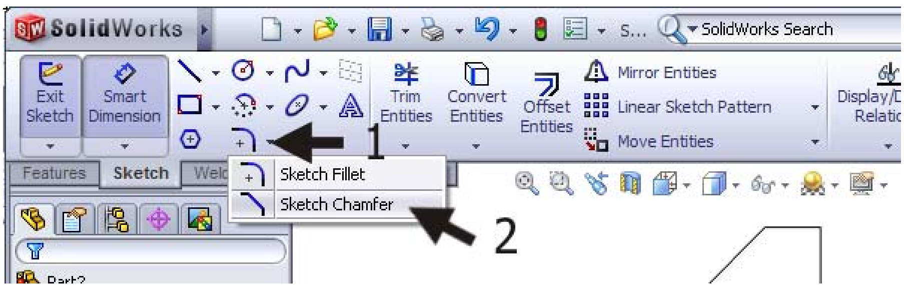
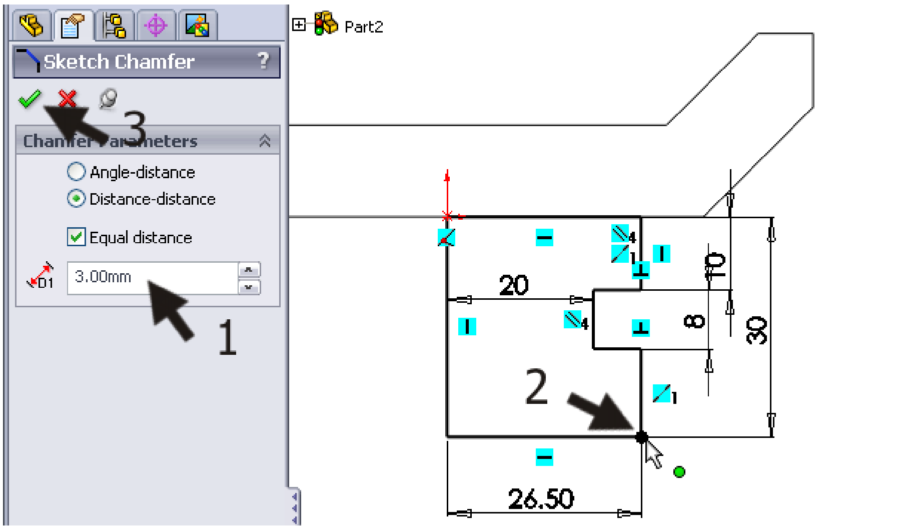
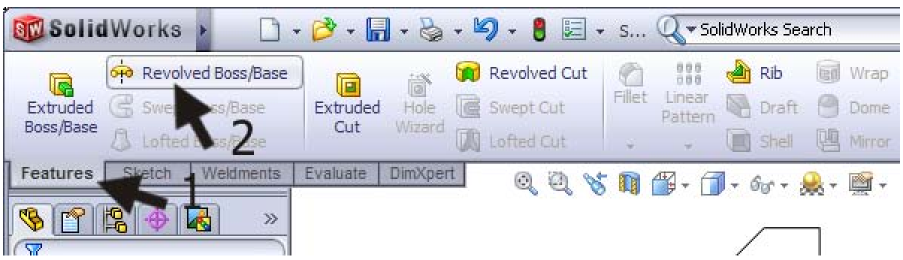
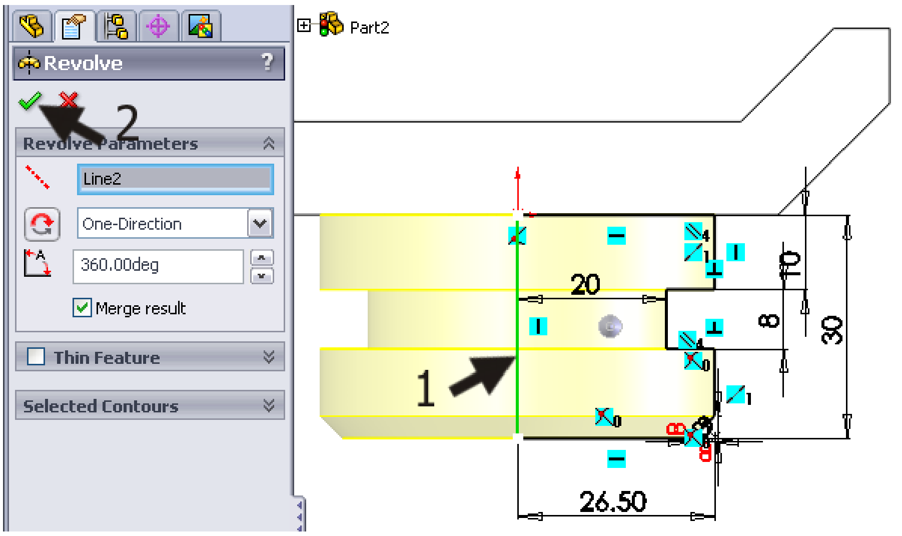
52	Add the exact dimensions with Smart Dimension. Look at the illustration. If the sketch from the previous step was not drawn very accurately, it is possible that you will see strange things happen. The best you can do is throw away (<Del> delete the sketch) and start again at Step 50. The most important part of this drawing is the first horizontal line: this should be about 40mm long.	
53	Next, select the entire sketch: click at a point on the left top and hold the mouse button while dragging the cursor to the bottom right. You will draw a frame around the sketch; notice that all parts should be included in this frame.	
54	Click on 'Mirror Entities' in the CommandManager. When you follow the correct steps, the sketch will be mirrored around the centerline from Step 52. Did you select more or less than one centerline? The sketch will not be mirrored immediately. You will have to select one line in the PropertyManager to use as a mirror axis.	

55	Make an Extruded Boss/Base from this sketch. Set the following features in the Property-Manager: 1. Select 'Mid Plane' for 'Direction1'. 2. Set the length to '65'. 3. Click on OK.	
	Tip!	Using the Mid Plane option, the sketch will be extruded in two directions with equal length. This is very convenient when creating symmetrical products (like this one) because the origin will remain in the middle of the product. This again is very convenient if you want to mirror parts later on. You could also get the same results by setting 'Direction2' in the Property-Manager. You will get more options that way, so it is less applicable to this situation.
56	Start a new sketch on the Front Plane. Draw a rectangle first, as shown in the drawing on the right. The left top corner is at the origin.	

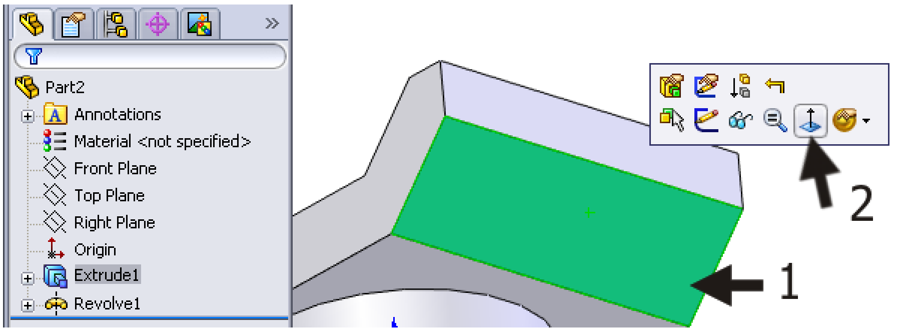
<table>
<tr>
<td>57</td>
<td>1. Open the pull-down menus
2. Click on: ‘Tools’.
3. ‘Sketch Tools’.
4. ‘Jog Line’.</td>
<td>
</td>
</tr>
<tr>
<td>58</td>
<td>Draw a smaller rectangle as shown.

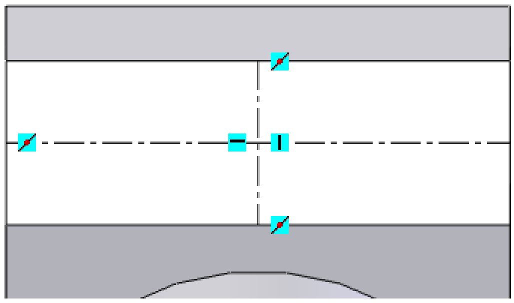
The first point of the small rectangle should be on the vertical line of the large rectangle.

The small rectangle will now be removed from the big one.</td>
<td>
</td>
</tr>
<tr>
<td>59</td>
<td>Set the dimensions as shown with Smart Dimension.</td>
<td>
</td>
</tr>
</table>

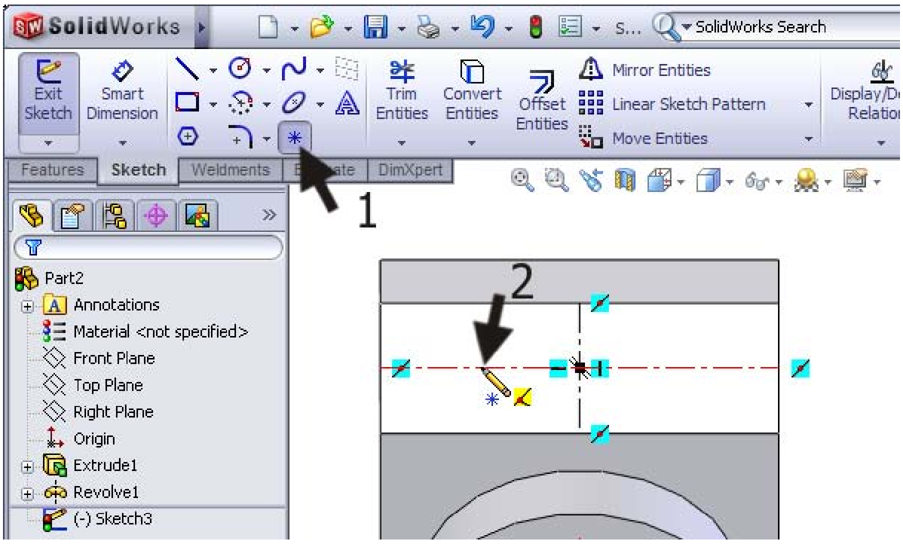
60	We will now make the sloped end at the bottom of the shape. 1. Click on the arrow next to 'Sketch Fillet'. 2. Click on 'Sketch Chamfer'.	
61	1. Set the distance for the chamfer to '3mm' in the PropertyManager. 2. Click on the corner point at the right bottom side. 3. Click on OK.	
62	The sketch is now ready, and we will make a rotation shape from it. Click on 'Features' in the CommandManager and next on 'Revolved Boss/Base'.	
63	1. Select the rotation axis first. This is the left vertical line in the sketch. 2. Click on OK.	

<table>
<tr><td>64</td><td>In the sloped surface we will make countersink holes with the Hole Wizard.

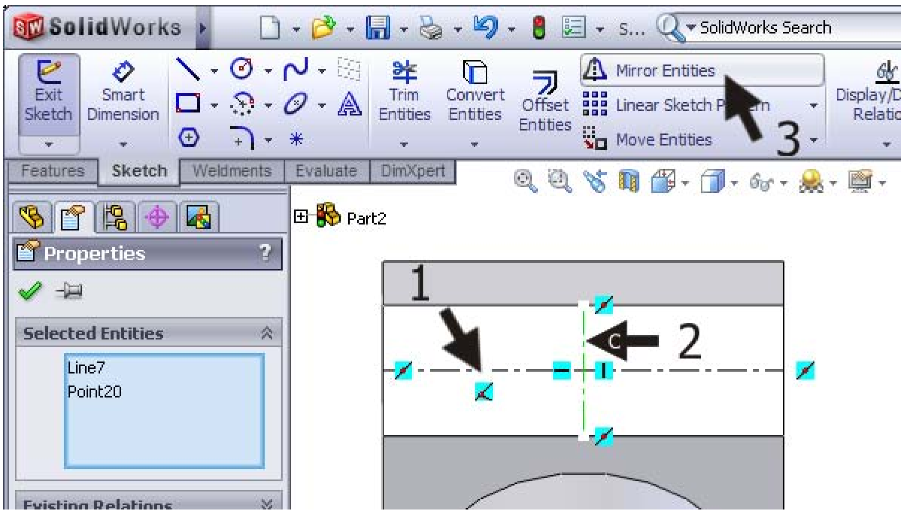
Click on the surface of the model and use the Normal To command to get a good view at this plane.</td><td>
</td></tr>
<tr><td>65</td><td>Draw a horizontal and a vertical centerline on the plane.

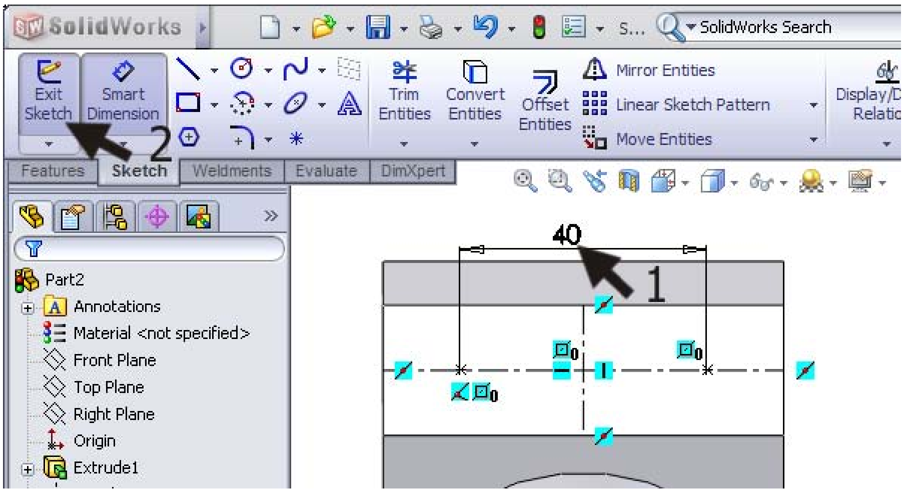
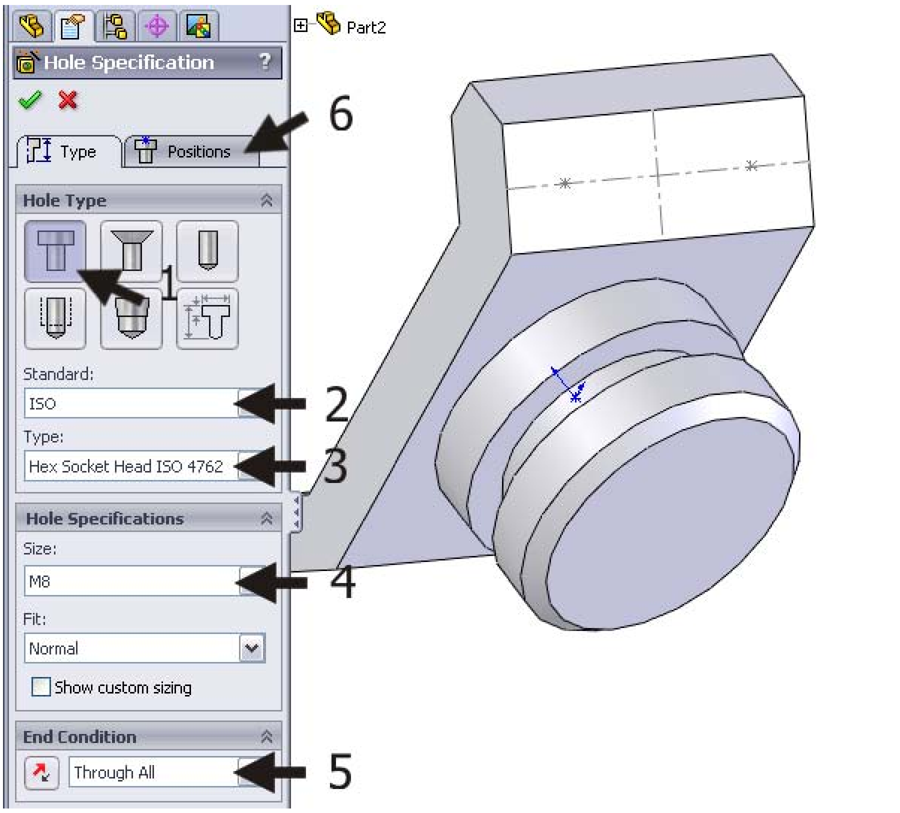
Make sure to use the center of the borderlines to begin and end the centerlines.</td><td></td></tr>
<tr><td>66</td><td>1 Click on Point in the CommandManager.

2 Set a point somewhere on the horizontal centerline, as shown in the drawing.

Push the <Esc> key to end the Point command.</td><td>
</td></tr>
<tr><td>67</td><td>Select the point and the vertical centerline (use the <Ctrl> key to select more than one element).

Click on ‘Mirror Entities’ in the CommandManager.

The point will be mirrored to the other side of the centerline.</td><td>
</td></tr>
</table>

<table>
<tr>
<td>68</td>
<td>1. Use Smart Dimension to set a dimension of ‘40mm’ between the two points.
2. Close the sketch with ‘Exit Sketch’.</td>
<td>
</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>We have just fixed the position of the countersink holes that we will make in the next step. You can also do this directly with the Hole Wizard (without making the sketch first), but often is it much easier to make the sketch first.</td>
</tr>
<tr>
<td>69</td>
<td>Click on ‘Hole Wizard’ in the CommandManager and set following items:
1. Choose Counterbore as the ‘Hole Type’.
2. ‘Standard’ is ‘ISO’.
3. ‘Type’ is ‘Hex Socket Head ISO 4762’.
4. ‘Size’ is ‘M8’.
5. The depth is ‘Through All’.
6. Click on the second tab: ‘Positions’.</td>
<td>
</td>
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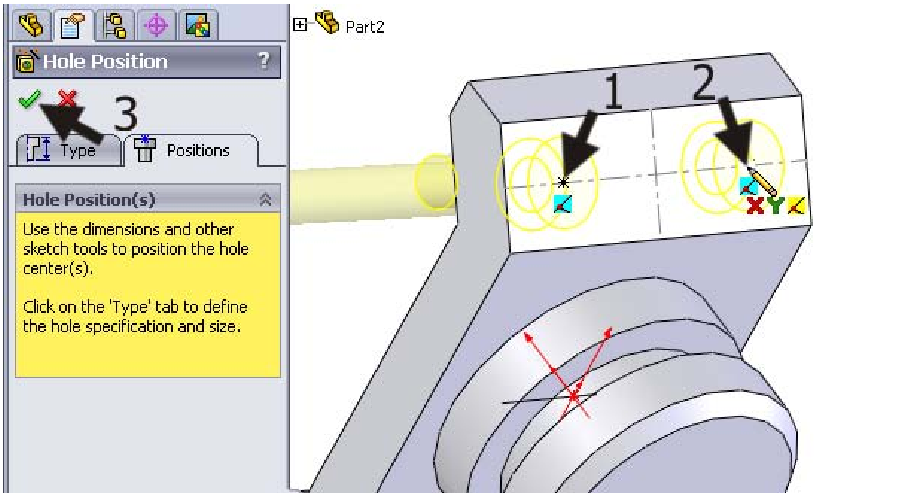
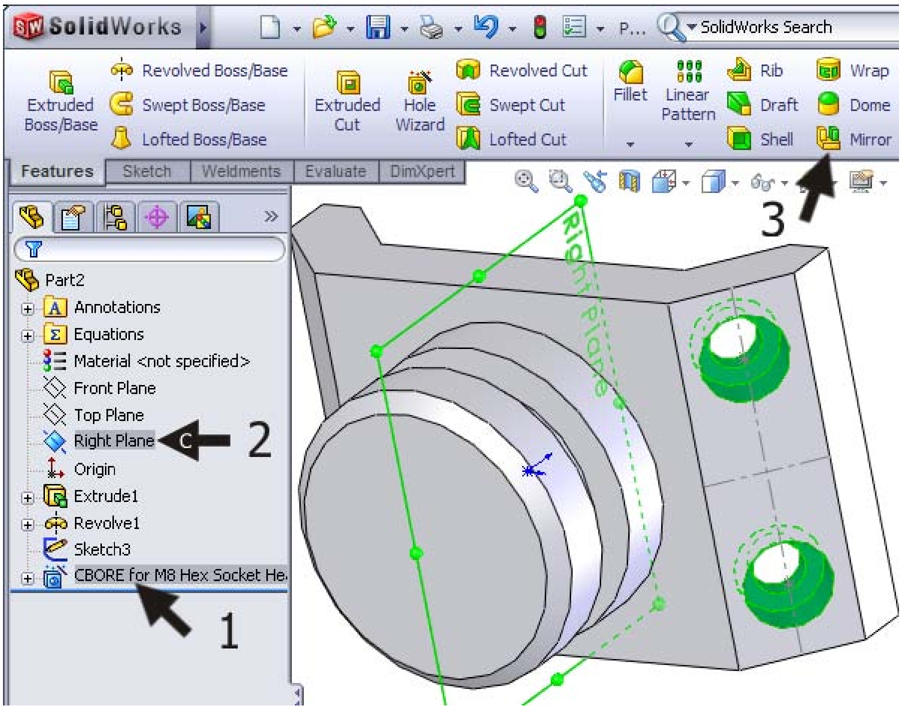
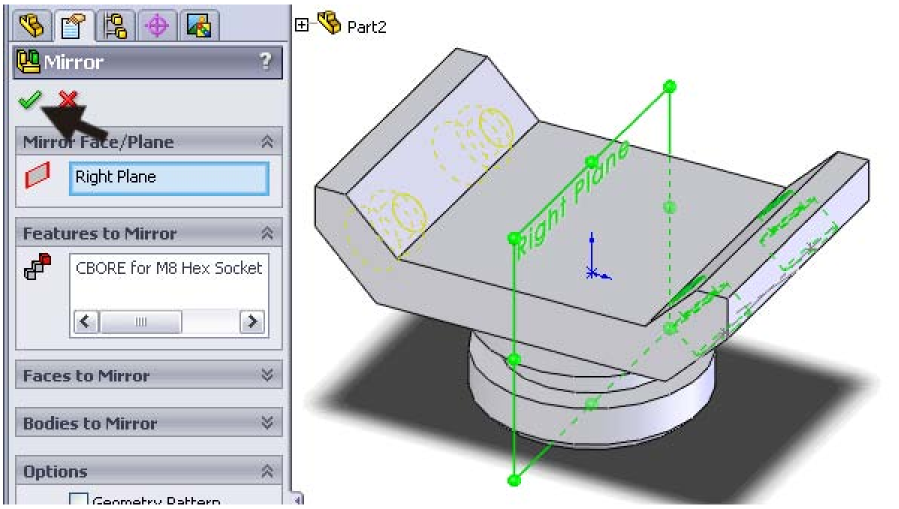
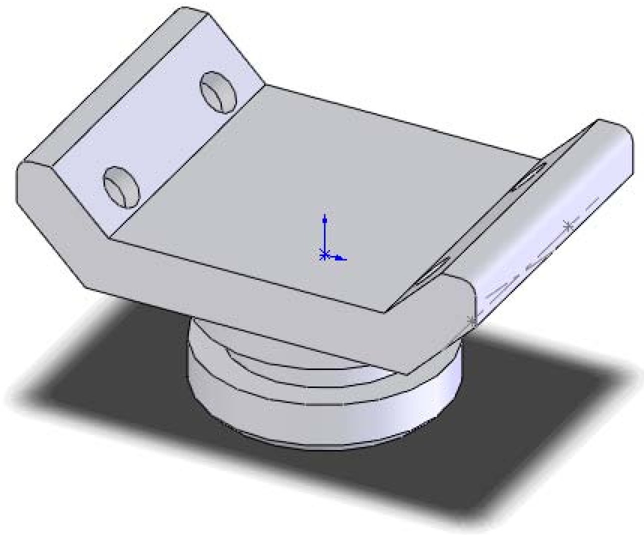
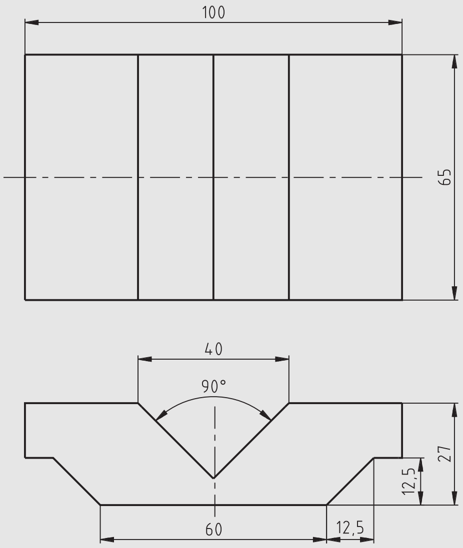
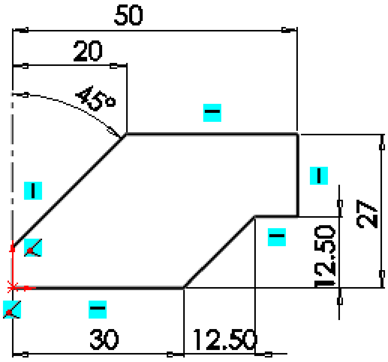
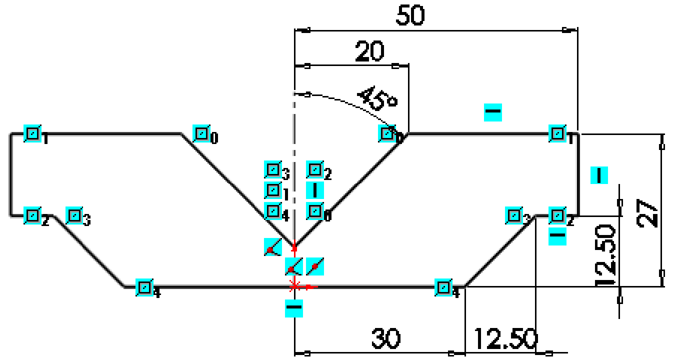
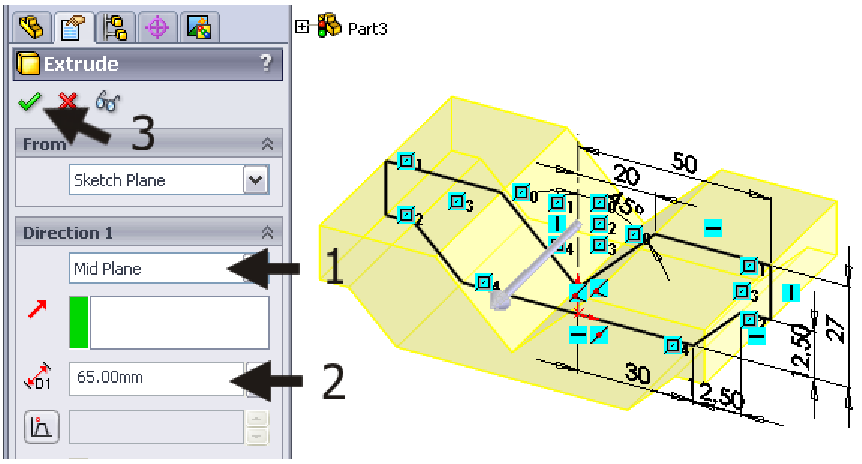
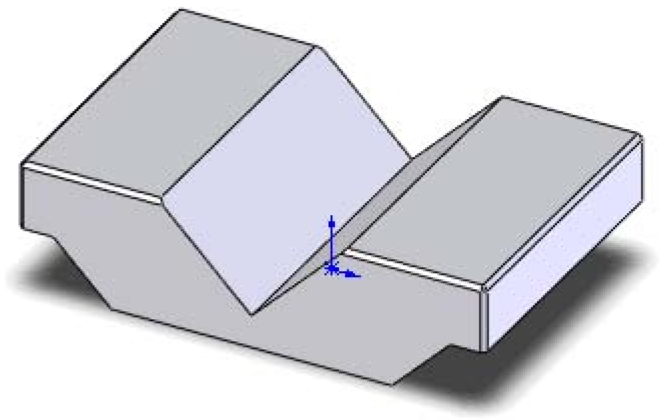
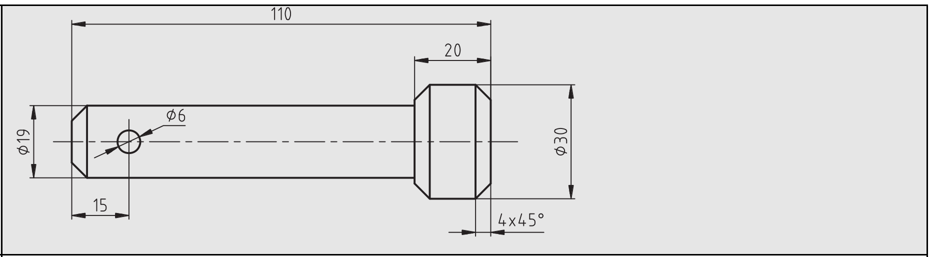
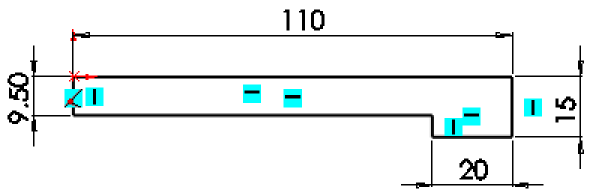
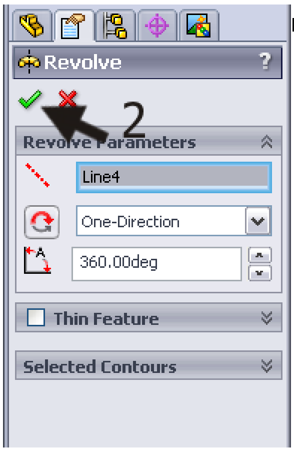
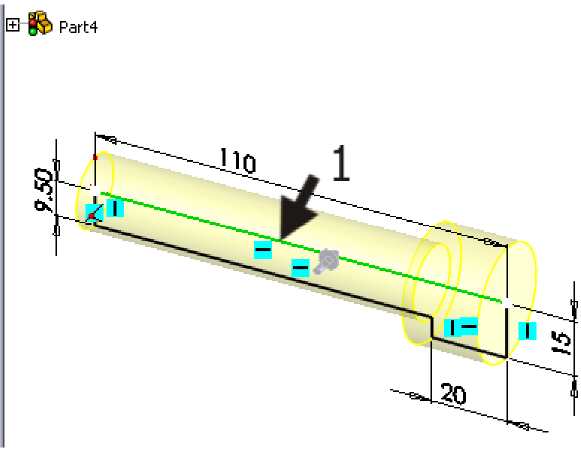
</table>

70	Next, click on the two points from the sketch to make the holes. Click on OK.	
71	Select the holes you have just made in the Feature-Manager and select the 'Right Plane' (use the <Ctrl> key). Click on 'Mirror' in the CommandManager.	
72	Everything is already set in the PropertyManager. Click on OK.	

73	This part is now ready. If you want, you can round some edges with Fillet or Chamfer. Save the model as: Support.SLDPRT.	
	Work plan	The next part is the insert. We will create only the main shape, not the screw holes. We will make these later after we have finished the assembly. The position of the holes will be fixed to the position of the support that we did earlier in this tutorial. The main shape is made from only one extrusion. The sketch is similar to the sketch we made for the support. 
74	Open a new part and make the sketch as shown on the Front Plane. The structure of this part is the same as the one from the last part (Steps 51 to 54). First, draw the vertical centerline from the origin. Next, draw a horizontal line with a length of about	

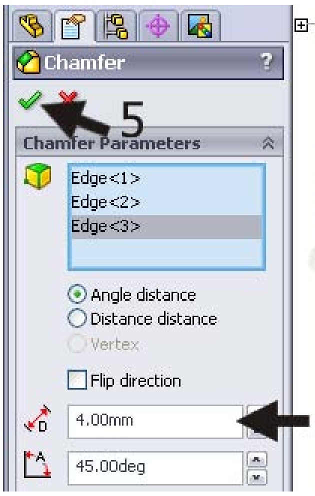
	'30mm' from the origin. Draw a raw shape to get the rest of the sketch. Make sure the sizes and proportions are about right. Finally, add the dimensions.	
75	Make a mirrored copy from the sketch around the centerline using the Mirror command.	
76	Next make an extrusion. Use the option 'Mid Plane' as you did before with the support and set the length to '65 mm'.	
77	Use the Chamfer feature to shape a number of corners as desired. Save the model as Insert.SLDPRT.	
	Work plan	Finally, we will make the last part of the axle support: the pin that is used to fix the tubes at a certain height. This part is mainly made as a rotation shape.

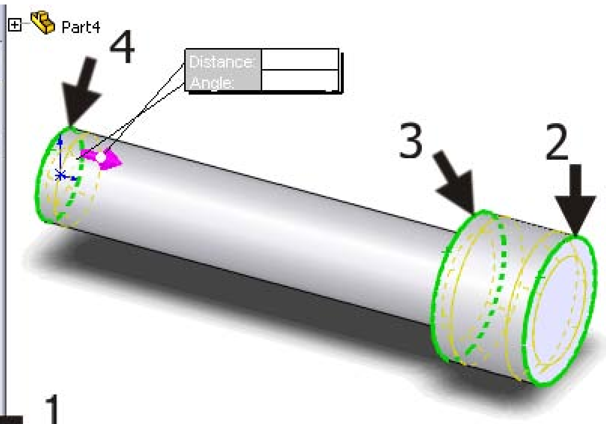
<table>
<tr><td></td><td></td><td>
</td></tr>
<tr><td>78</td><td>Open a new part and make a sketch on the Front Plane as shown on the right.</td><td>
</td></tr>
<tr><td>79</td><td>Make a Revolved Boss/Base from the sketch. Select the upper horizontal line in the sketch to be used as a rotation axis.</td><td>

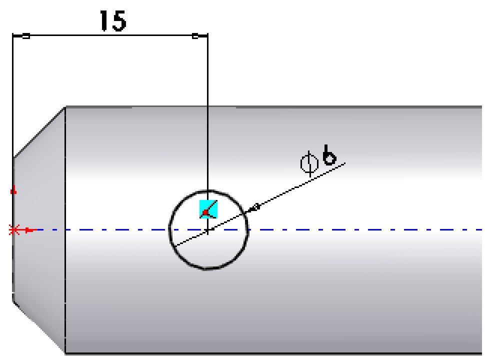
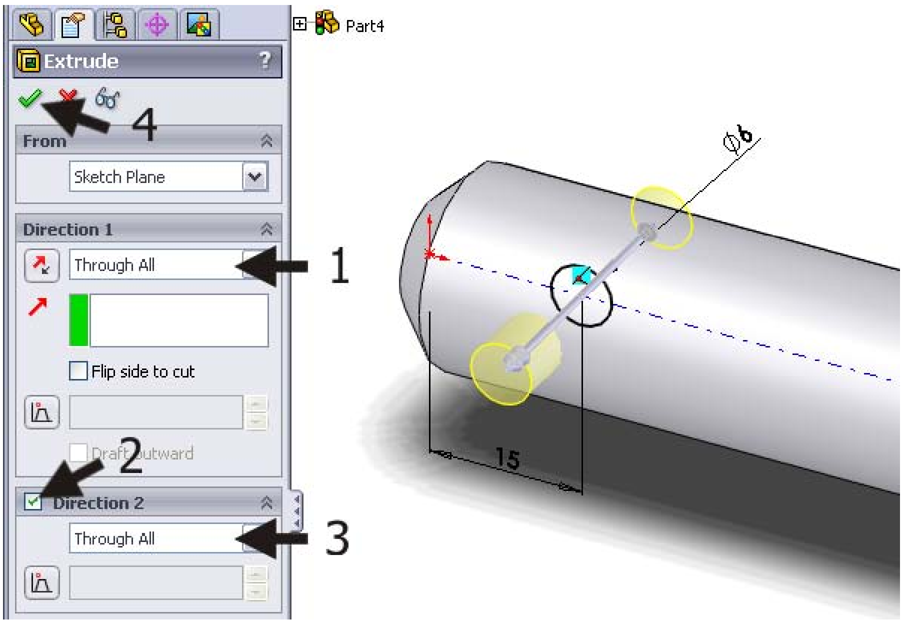
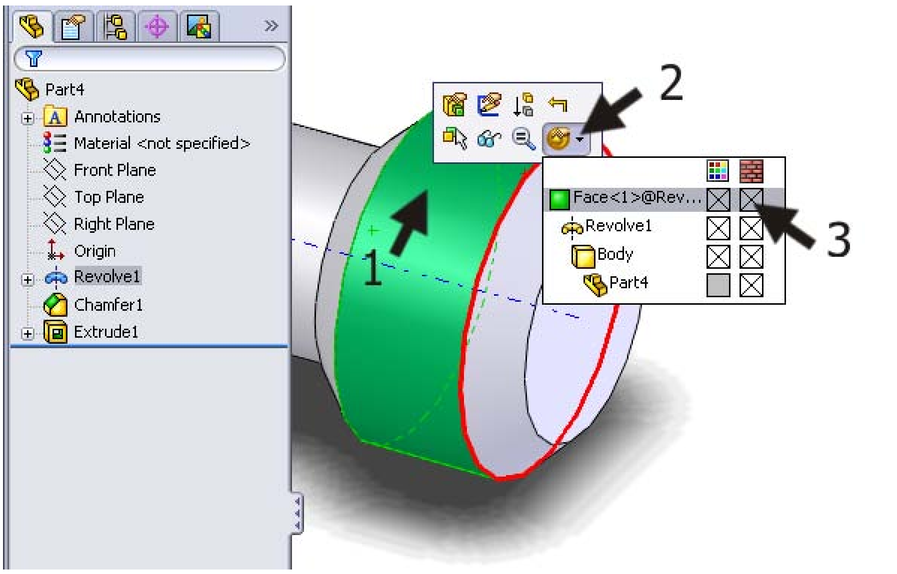

</td></tr>
<tr><td>80</td><td>We will chamfer a number of corners. Click on ‘Chamfer’ in the CommandManager.

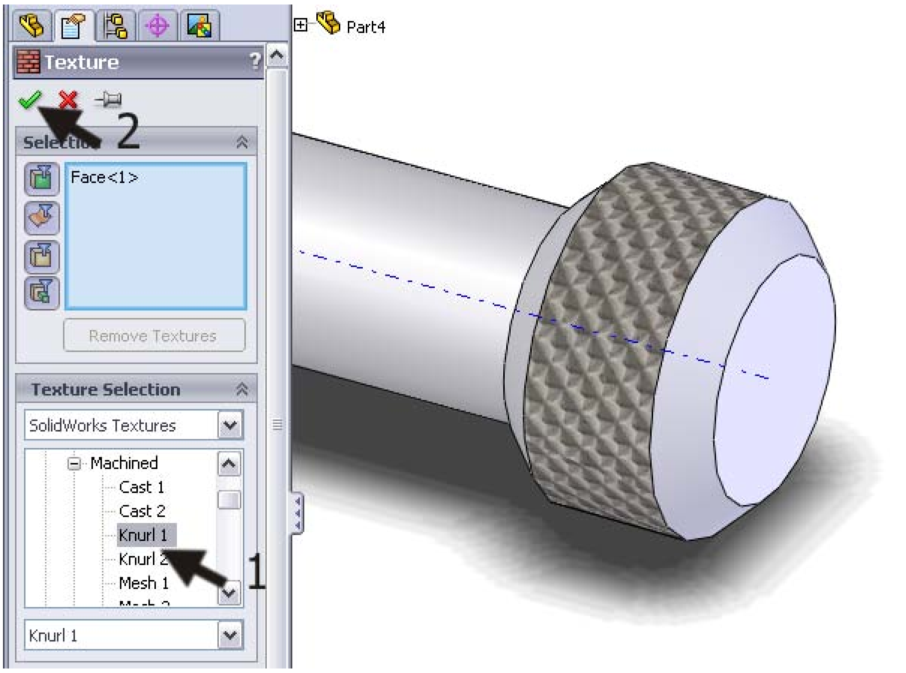
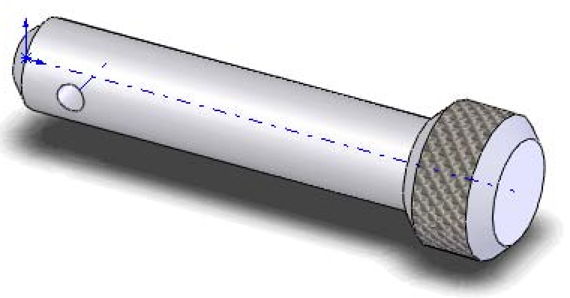
Set the dimension of the slope to ‘4mm’.

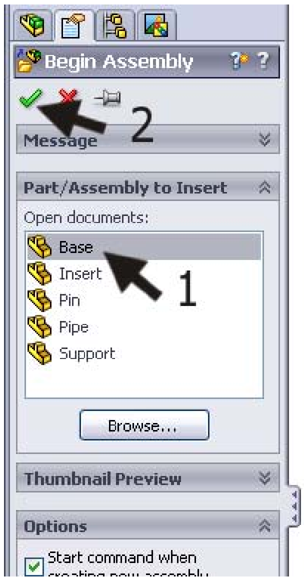
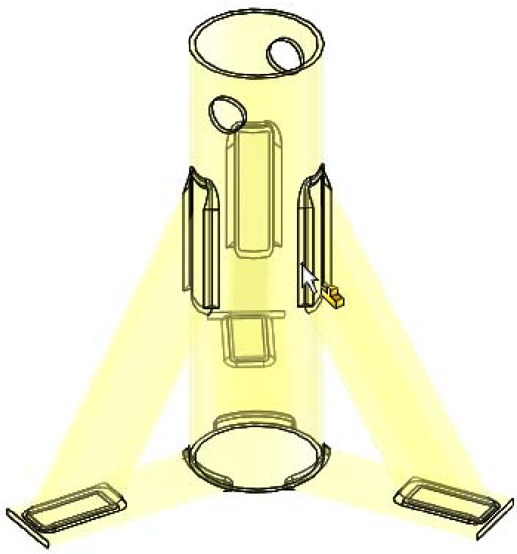
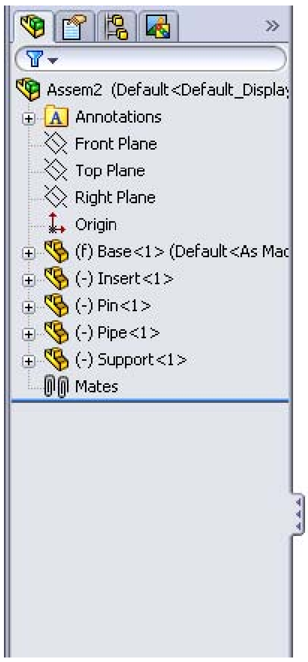
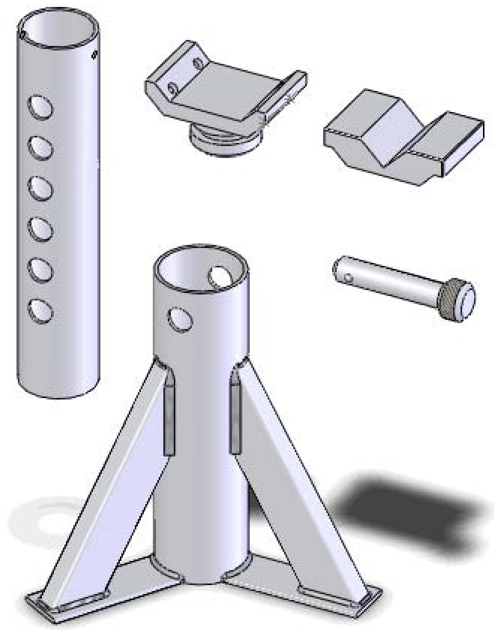
Select the three edges (do NOT select planes!) as shown on the right.

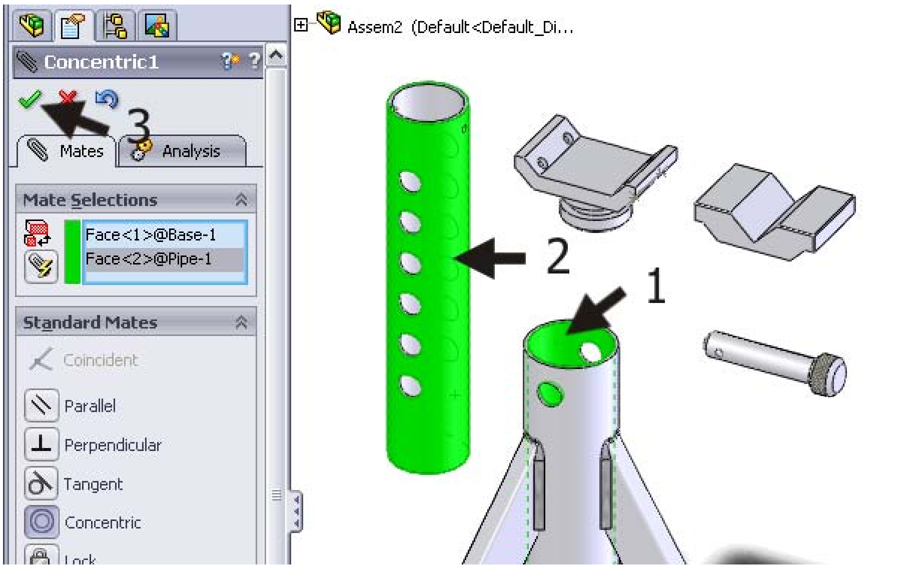
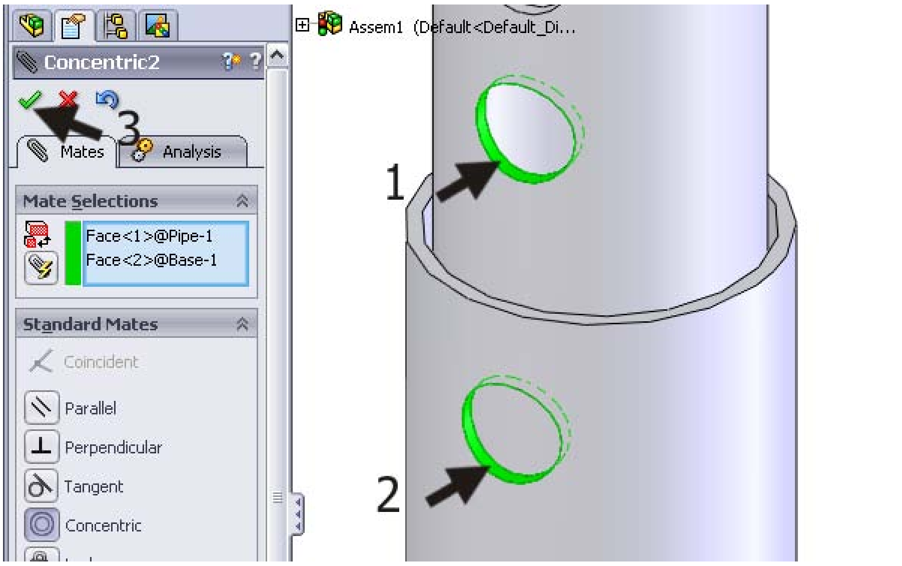
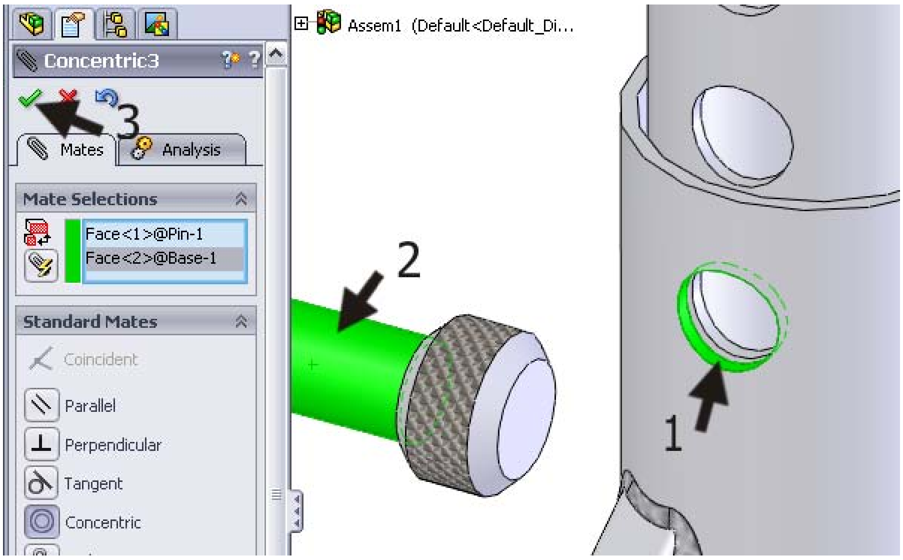
Click on OK.</td><td>

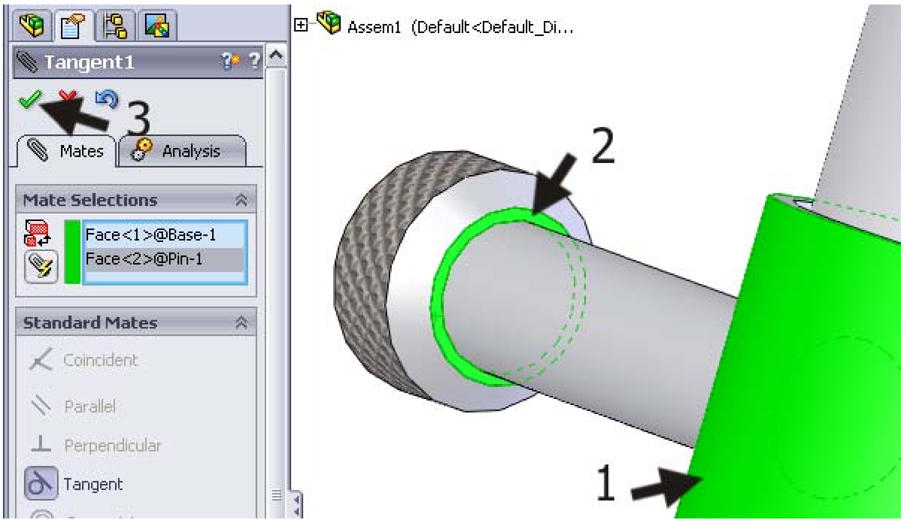

</td></tr>
</table>

81	Select the Front Plane and make sure that you have a straight view at it by using the Normal To command. Make sure the Temporary Axes are visible. Make the sketch as shown in the illustration.	
82	Make an Extruded Cut from this sketch. Select the option 'Through All' in the PropertyManager to set both directions.	
83	Finally, we will give the outside plane of the pin a new texture. 1 Click on the surface. 2 Click on Appearance in the pop-up menu. 3 Click on Texture in the 'Face<1>' line.	

	Tip!	Using the menu from the last step you can add a color or texture to different parts of a model: a Face, a Feature, a Body or a Part. Do you want the whole model to get the same color or texture? Then, click on the check-box next to Body or Part. We will handle only one surface now, so click on the check-box behind Face. In the next step you will get the opportunity to change your selection.
84	Select 'Knurl1' in the list of materials. You will find this under 'Metal > Machined'. Click on OK.	
85	The part is ready. Save it as: Pin.SLDPRT.	
	Assembly	At the end of this tutorial we will make the assembly. All parts will be joined together as one product. After that is done, we will make the holes in the insert on the support. Finally, we will add some screws from the Toolbox.

86	Open a new assembly. The Insert Component command will start automatically. 1. Click on ‘Base’ in the list of open files. 2. Click on OK. If you have closed the file base.SLDPRT before, click on the ‘Browse...’ key and find the file.	 
87	Add all of other parts to the assembly. The exact location is irrelevant at this point.	 

88	First, make a 'Concentric' mate between the outside of the tube and the inside of the axle support base.	
89	Select the hole in the base and the hole in the tube. These will be 'Concentric'. Make sure to select the in-side planes in the holes and not the edges.	
90	Select the plane on the in-side of the hole in the base again and also select the surface of the pin that goes through it. This mate will be set to be 'Concen-tric' too.	

<table>
<tr>
<td>91</td>
<td>Select the plane of the pin and the outside surface of the base tube as shown on the right. These two must only touch each other so the mate must be set to ‘Tangent’.</td>
<td>


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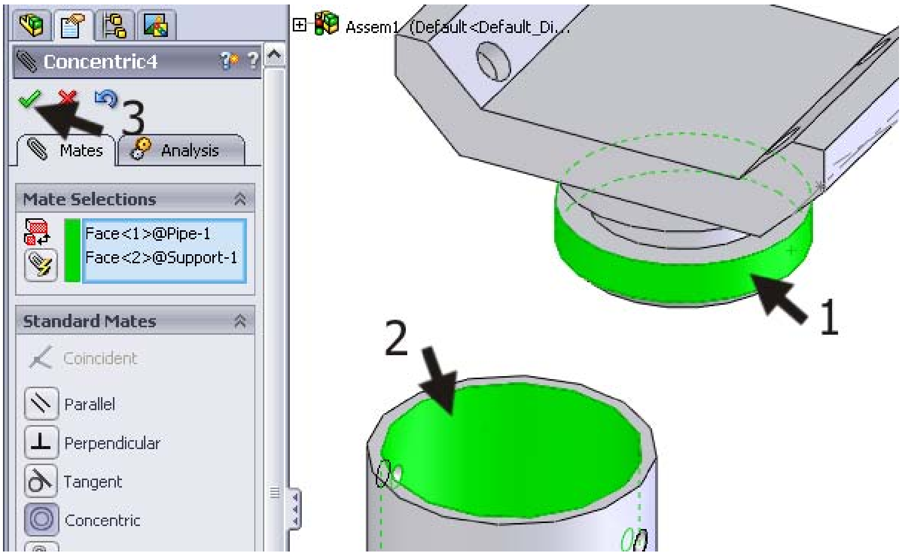
<tr>
<td>92</td>
<td>The next mate is the mate between the cylindrical plane of the support and the inside of the inner tube. This mate will be ‘Concentric’ again.</td>
<td>


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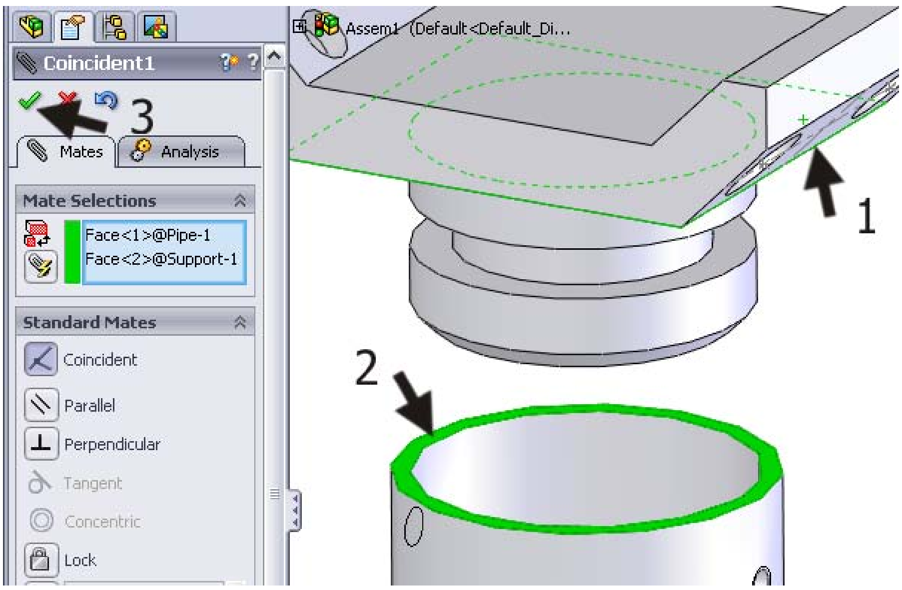
<tr>
<td>93</td>
<td>To create the next mate, you must select the top of the tube and the bottom of the support block. These planes must coincide, so select the mate type ‘Coin-cident’.</td>
<td>


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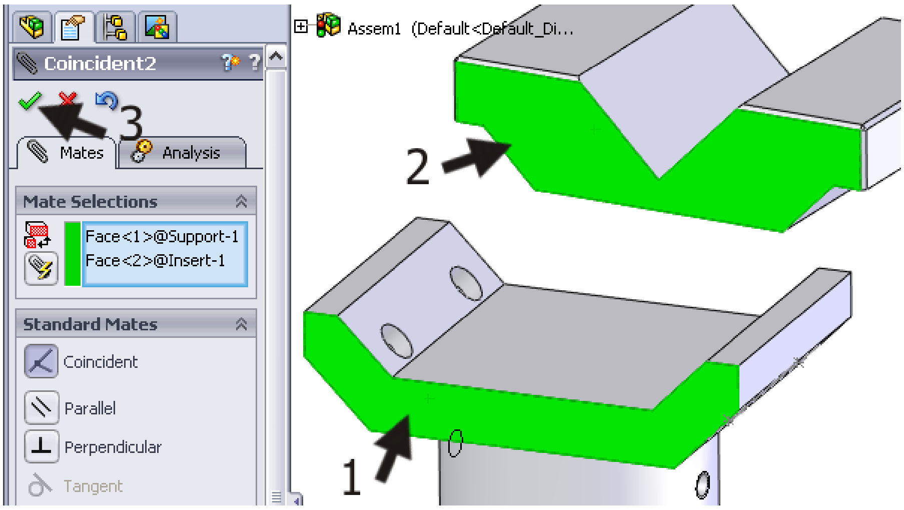
<td>94</td>
<td>Finally, the insert must be put in its position.

Select the front plane of the support block first and after that the front plane of the insert. Make these planes ‘Coincident’.</td>
<td>
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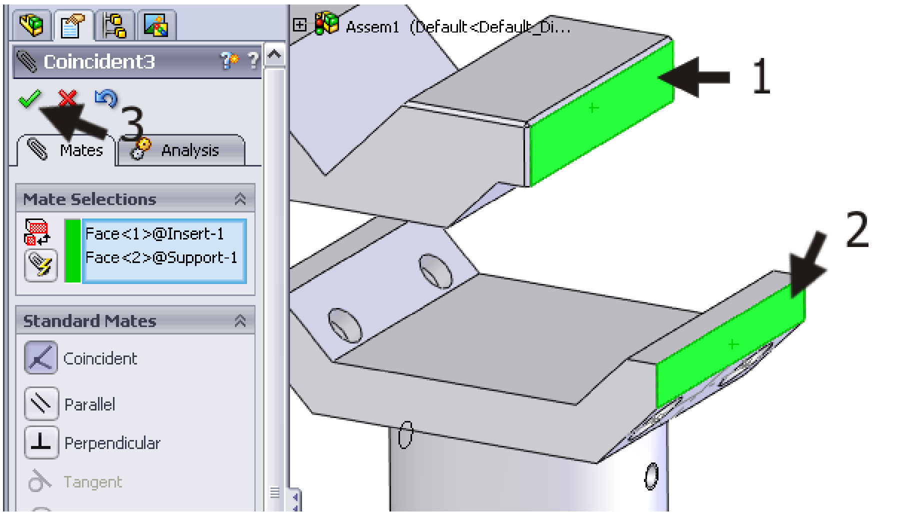
<td>95</td>
<td>The next mate will be between the sides of both parts. Make these ‘Coincident’ too.</td>
<td>
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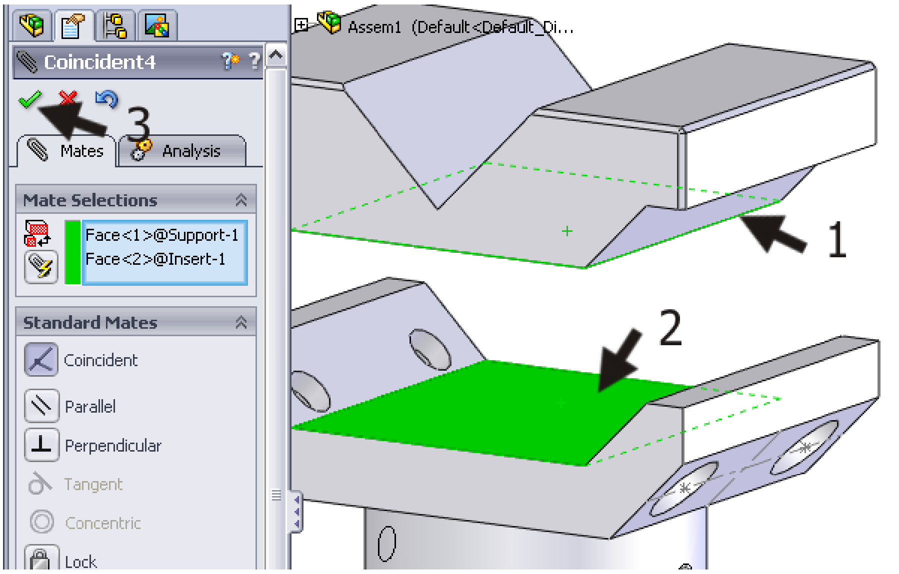
<tr>
<td>96</td>
<td>The final mate is between the bottom of the insert and the top of the support block.</td>
<td>
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</tr>
<tr>
<td>97</td>
<td>Save the assembly as Axle_stand.SLDASM.</td>
<td></td>
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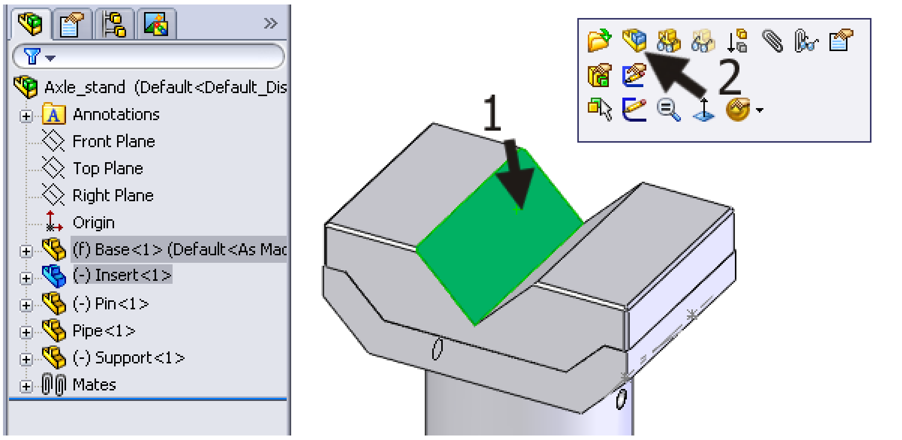
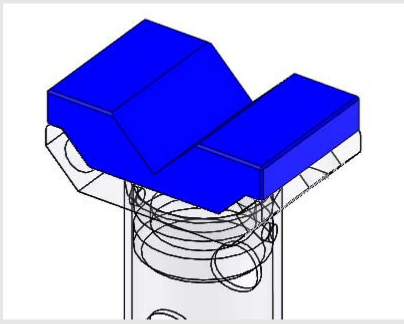
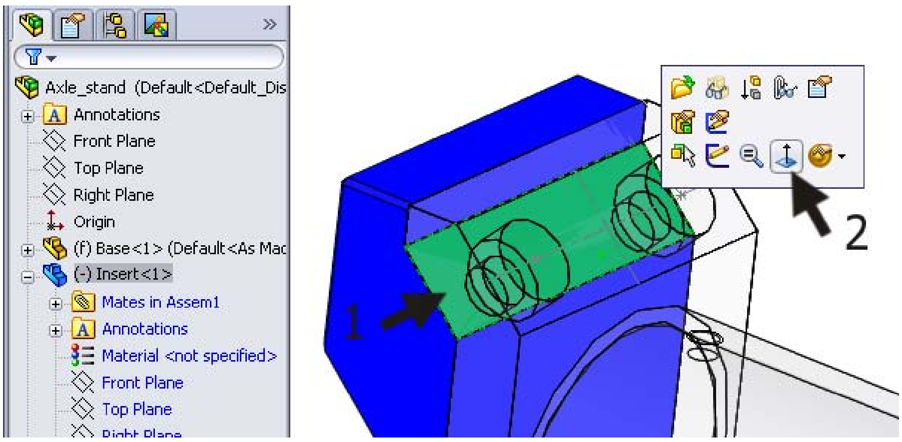
98	We have to make a couple of tapped holes in the insert, and the holes have to be aligned with the holes in the support block. We will do this by changing the part 'In Context'. Click at a random point on the insert and select Edit Part (second icon) to change the part.	
	Tip!	You can now see the whole assembly turning transparent/gray – only the insert turns blue. You can work on this part as you can with any other part; the only difference is that the assembly remains visible. The advantage is that you can see directly how the part fits in the product. You can use this while modeling to link items together. We call this type of modeling 'In Context'. 
99	Rotate the model so you can see the bottom of the insert. Select the sloped plane and click on Normal to for a straight-on view.	

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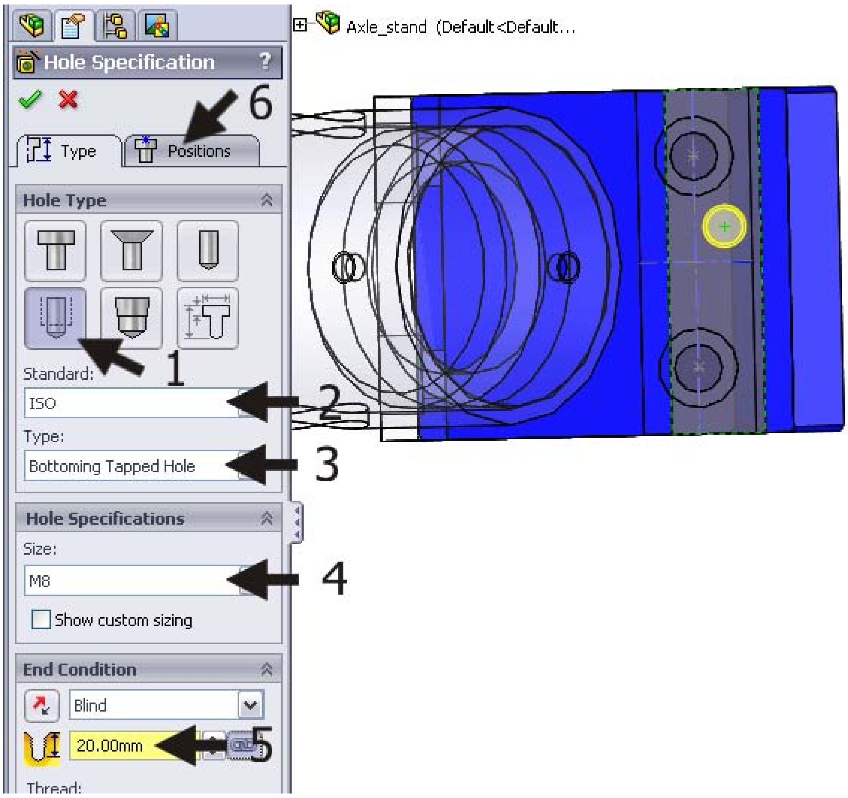
<td>100</td>
<td>Click on 'Features' in the CommandManager and then on 'Hole Wizard'. Use the following settings:
1. Select 'Tap' as the 'Hole Type'.
2. 'Standard' is 'ISO'.
3. 'Type' is 'Bottoming Tapped Hole'.
4. 'Size' is 'M8'.
5. Depth is '20mm'.
6. Click on the tab 'Positions'.

Notice that one hole is already positioned at the exact spot where you have selected the plane. We will delete this later.</td>
<td>
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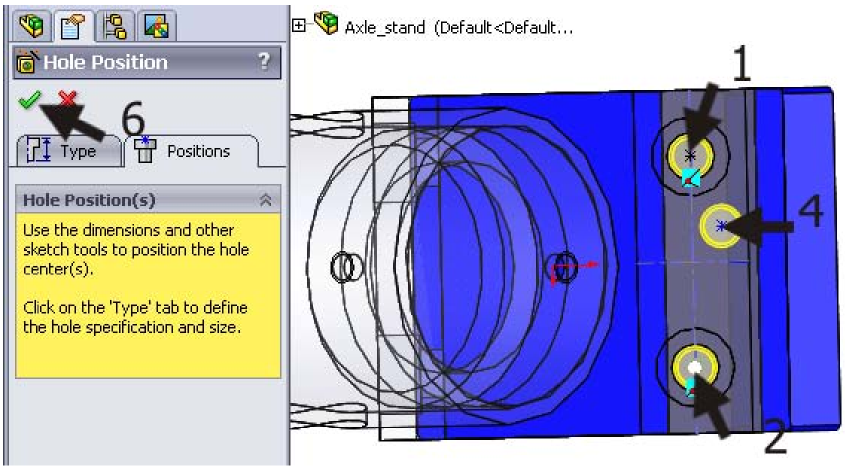
<td>101</td>
<td>1,2 Click on the midpoints of the existing holes to align the tapped holes at exactly the same position.
3. Push the <Esc> key.
4. Select the midpoint from the first hole (this was set by the software automatically, remember?).
5. Push <Del> delete to delete the hole.
6. Click on OK.</td>
<td>
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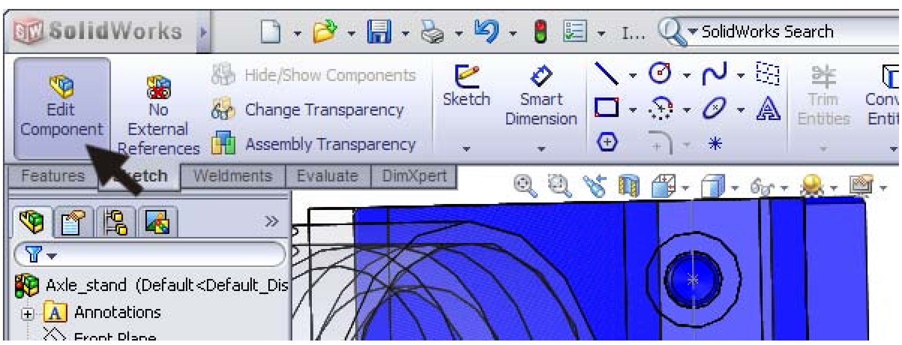
<td>102</td>
<td>You have now finished the necessary actions.

Click on 'Edit Component' in the CommandManager (you actually switch it off now) and you will return to the 'normal' assembly.</td>
<td>
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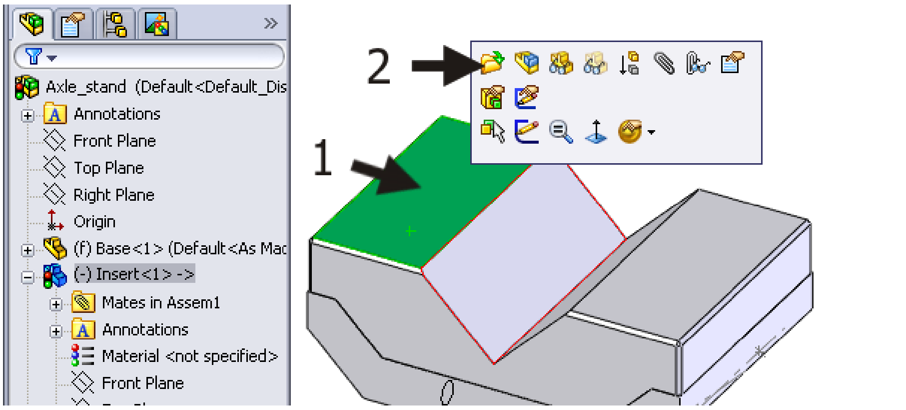
<td>103</td>
<td>Click on the insert again. Select the Open Part icon (first icon). The part is now open.</td>
<td>
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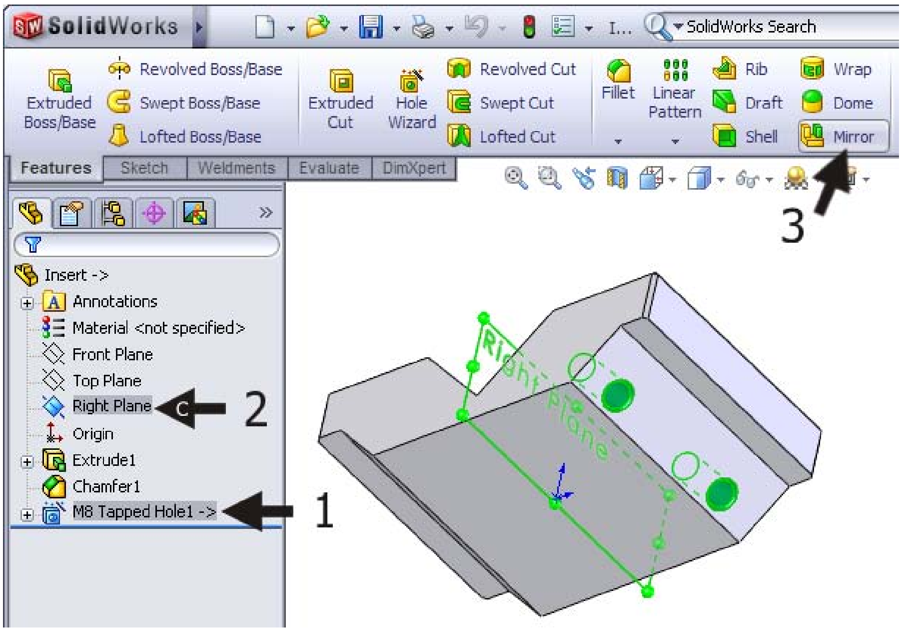
<tr>
<td>104</td>
<td>Notice that the holes you have just made in the assembly are also visible in the part now.

1. Select the holes in the FeatureManager (it is the last feature in the list).

2. Also select the 'Right Plane'. Use the <Ctrl> key to select both items simultaneously.

3. Click on 'Mirror' in the CommandManager.</td>
<td>
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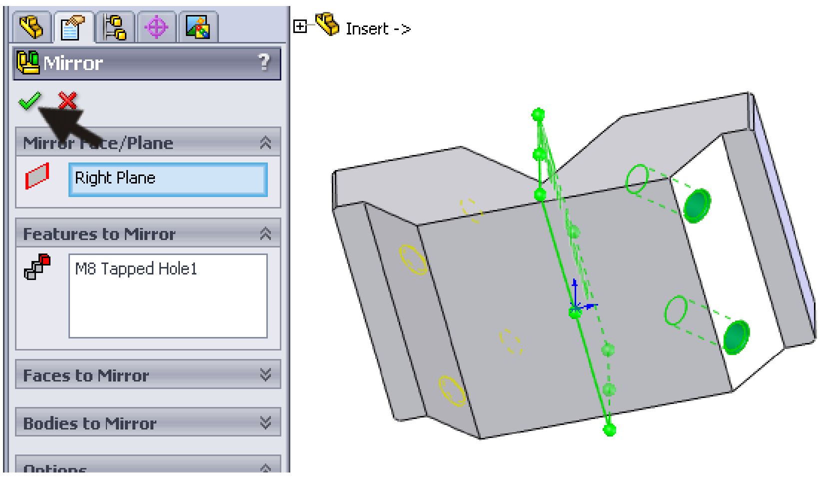
<td>105</td>
<td>All settings are already present in the PropertyManager.

Click on OK.</td>
<td>
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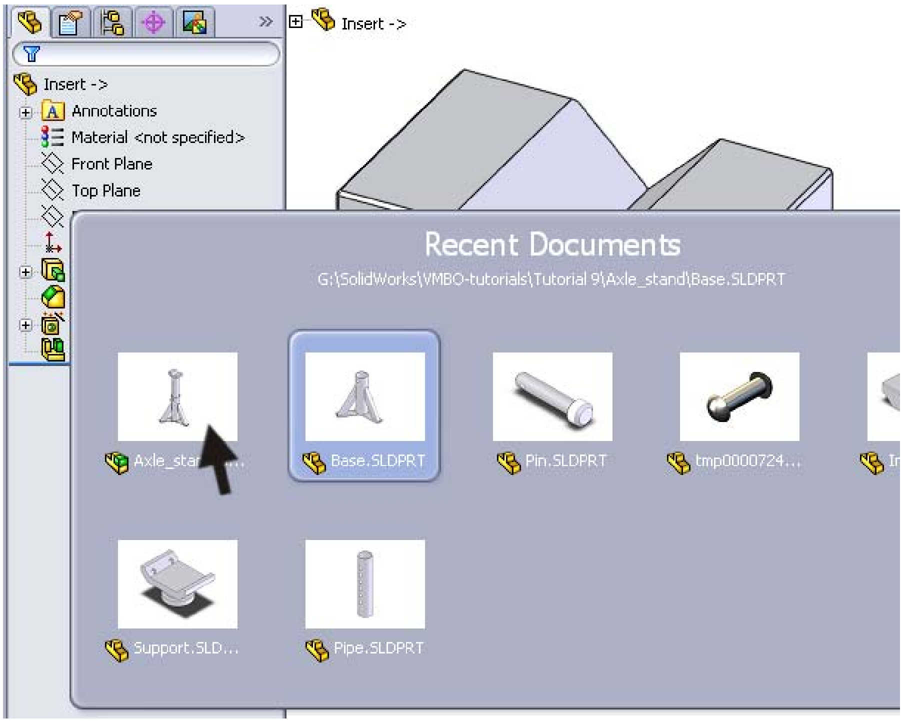
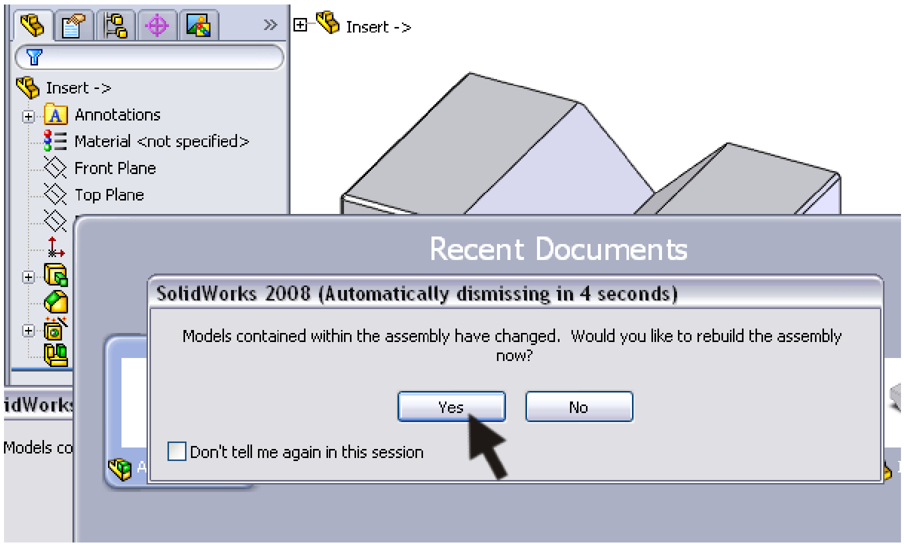
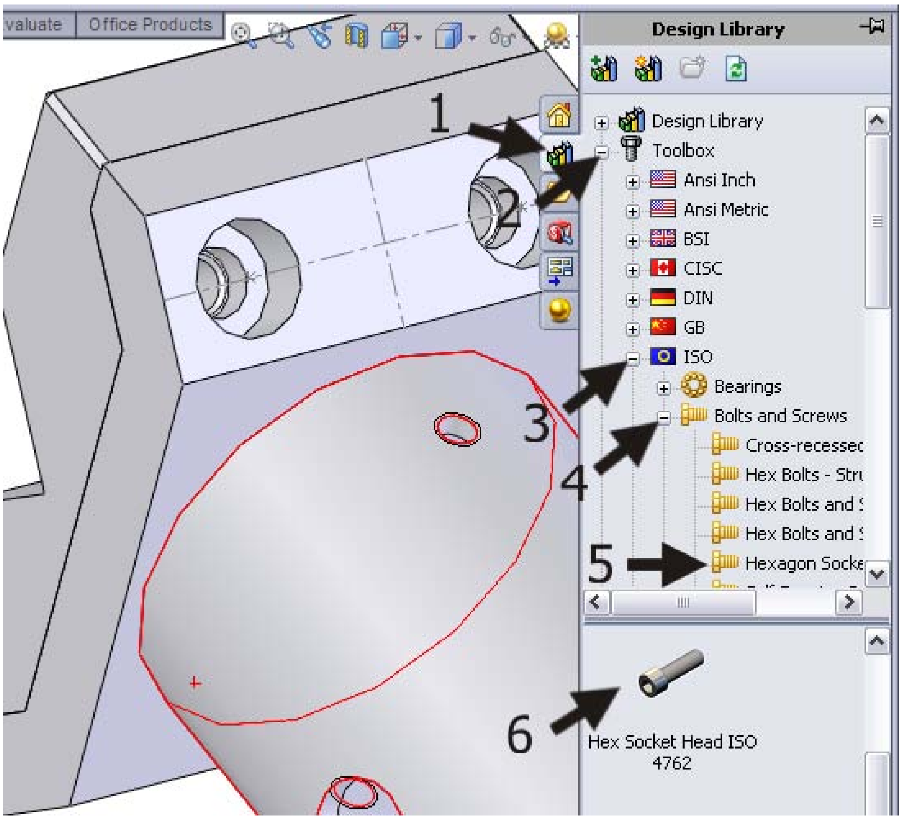
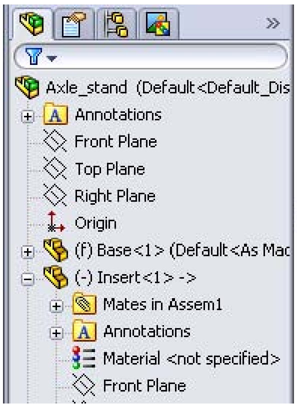
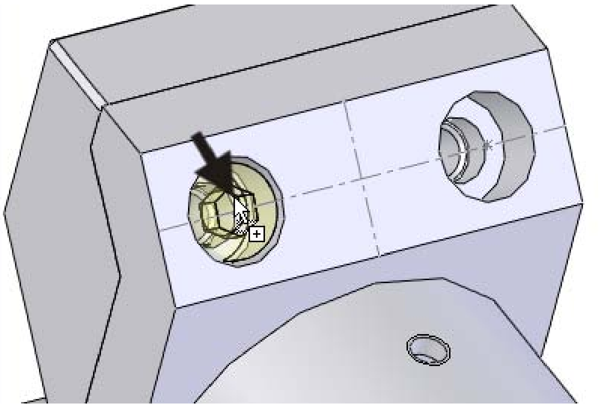
106	Return to the assembly. 1. Push the ‘R’ key on your keyboard 2. Click on the assembly Axle_stand in the pop-up menu.	
107	The assembly ‘knows’ a part has been changed and asks if the assembly should be rebuilt. Click on ‘Yes’ (or wait for about 10 seconds).	

108	Rotate the model so you can see at least two of the holes in the support block. 1. Open the ‘Design Library’. 2. Go to ‘Toolbox’. 3. ‘ISO’. 4. ‘Bolts and Screws’. 5. ‘Hexagon Socket Head Screws’. 6. ‘Hex Socket Head ISO 4762’.	
109	Drag the screw to the assembly. Release it on the deeper surface in one of the holes. The screw size may be wrong, but this does not matter.	 

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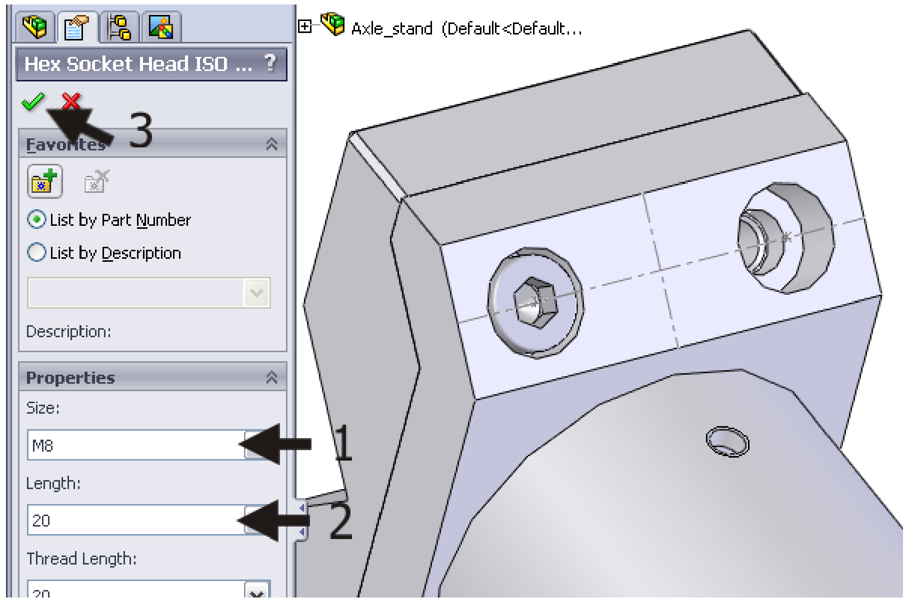
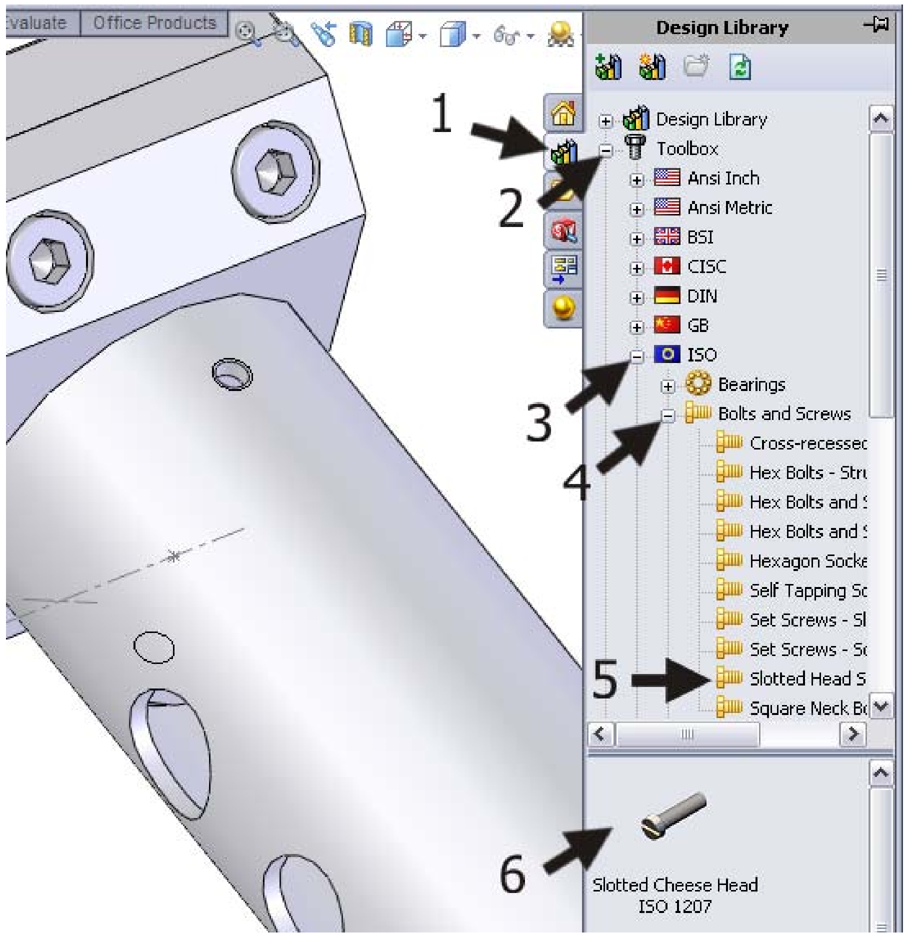
<tr>
<td>110</td>
<td>1. Set the screw thread ‘Size’ to ‘M8’.
2. Set the ‘Length’ to ‘20mm’.
3. Click on OK,

After this, you can also put the screw into the three other holes.</td>
<td>
</td>
</tr>
<tr>
<td>111</td>
<td>Finally, we need a screw to fasten the support block into the tube.
1. Open the ‘Design Library’.
2. Go to ‘Toolbox’.
3. ‘ISO’.
4. ‘Bolts and Screws’.
5. ‘Slotted Head Screws’.
6. Select the next screw: ‘Slotted Cheese Head ISO 1207’ and drag this to the model.</td>
<td>
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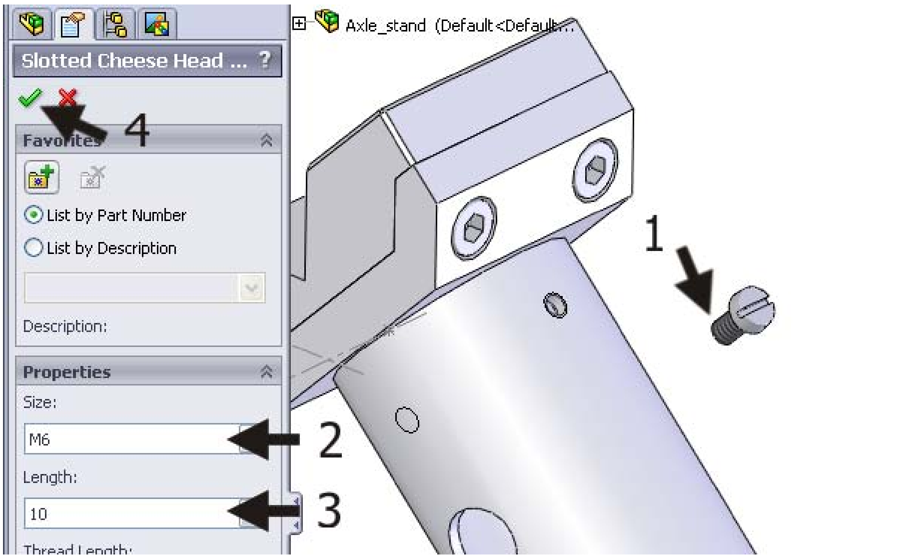
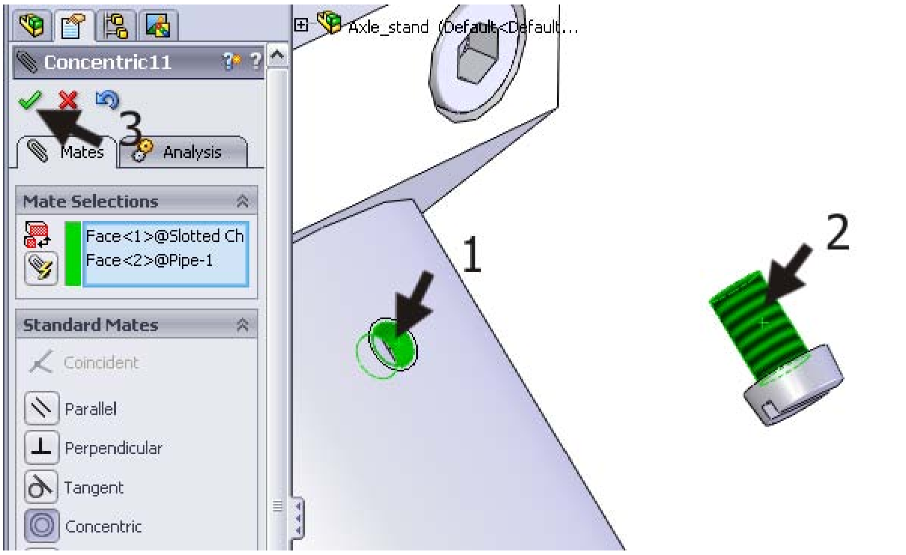
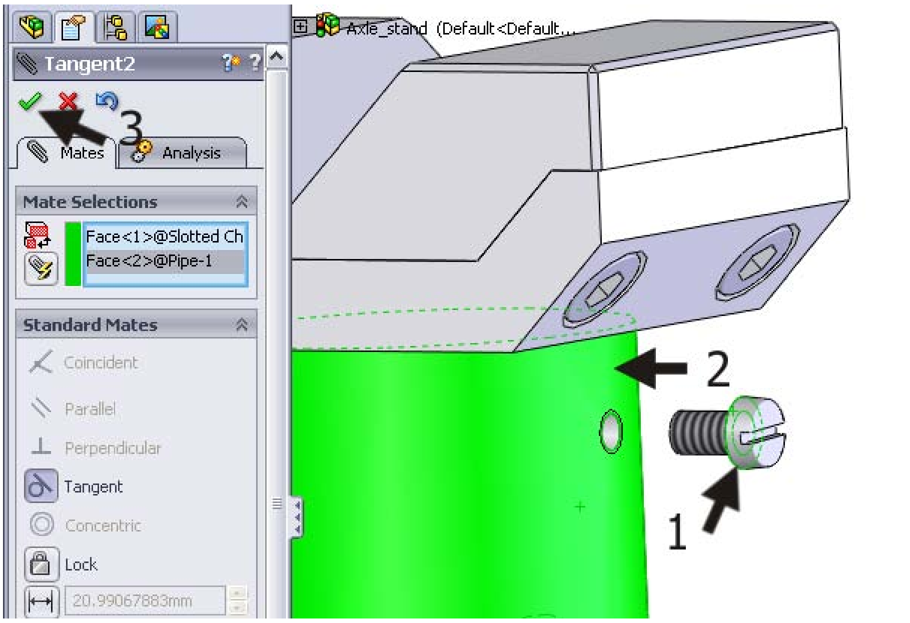
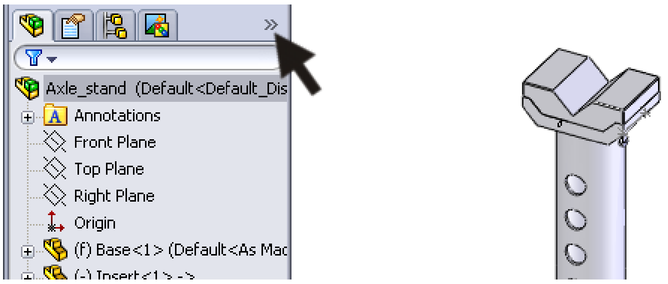
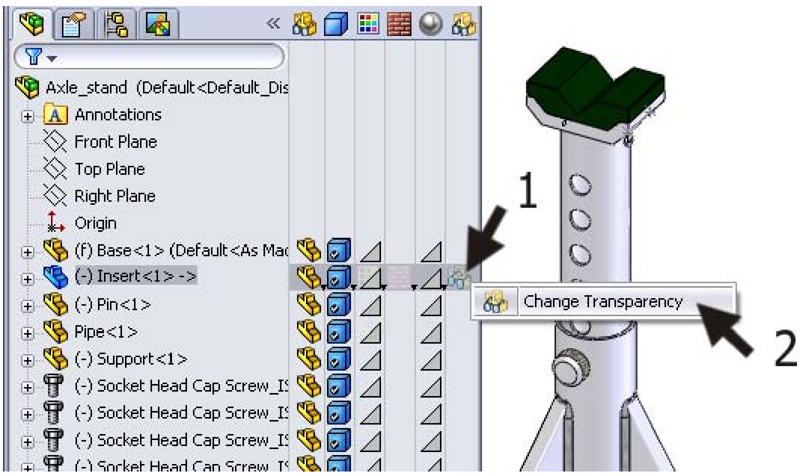
112	1. Place the screw at a random position beside the model. Because the hole is in a round tube, it is not possible to put the screw directly in the right position. 2. Set the size to 'M6' in the PropertyManager. 3. The 'Length' must be set to '10mm'. 4. Click on OK. 5. Push the <Esc> key on the keyboard to end the screw selection command.	
113	Next set a mate between the screw and the hole: select the planes as shown.	

<table>
<tr><td>114</td><td>Finally, make a mate between the bottom of the screw head and the outer surface of the tube.</td><td>
</td></tr>
<tr><td>115</td><td>Put a screw in the other hole as well. Use the same method again.</td><td></td></tr>
<tr><td>116</td><td>To clarify the model we will make the insert transparent.

Open the Display Pane menu in the FeatureManager as shown in the illustration on the right (use the double arrow icon).</td><td>
</td></tr>
<tr><td>117</td><td>1 Click on the last column, behind the part ‘Insert’.

2 Click on ‘Change Transparency’.

The part turns transparent now.</td><td>
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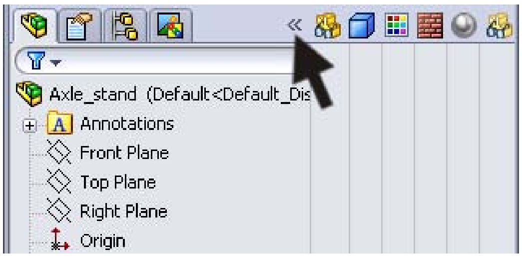
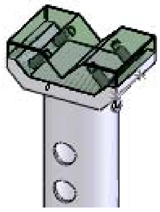
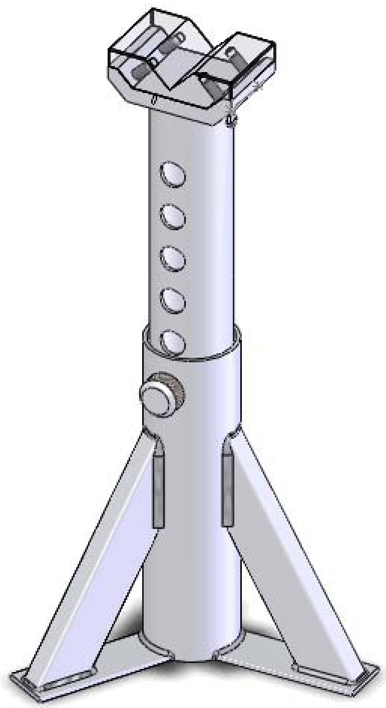
118	To close the Display Pane menu, click on the double arrow that you used before to open it.	 
	Tip!	Using the Display Pane menu gives you a quick method for to setting the way each part is shown. Try the different settings yourself.
119	The model is now ready. Save it.	
	What are the main features you have learned in this tutorial?	Congratulations! You have created a fairly complex model in SolidWorks. You have used many of the tools that you have already learned but have also been introduced to a number of new subjects. • You have learned how to make a rotation shape with Rotated Boss/Base. • You have created patterns by using Linear Pattern and Circular Pattern • You have copied features using the Mirror command and you have mirrored parts in the sketch with the Mirror command. It showed you how to build symmetrical products. • The last and maybe the most important new features were the weldments. You have built a construction using tubes and profiles. So you have learned a lot of new items again. We have practiced making mates in an assembly for the second time now and have used the Toolbox again. This was designed to improve your knowledge of these functions. You have again reached an even higher level of SolidWorks usage!

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.

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