SolidWorks® Tutorial 8

Bearing Puller

Preparatory Vocational Training
and Advanced Vocational Training

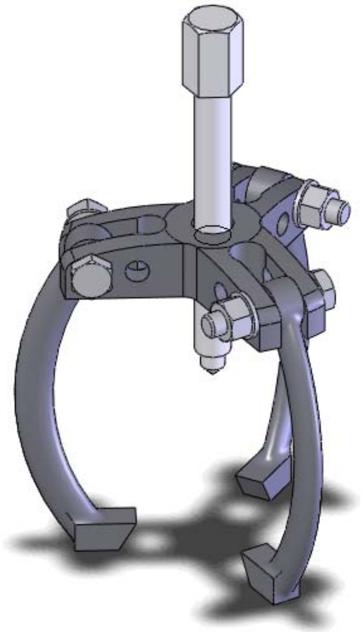
To be used with SolidWorks® Educational Edition Release 2008-2009

SolidWorks Benelux developed this tutorial for self-training with the SolidWorks 3D CAD program. **Any other use of this tutorial or parts of it is prohibited.** For questions, please contact SolidWorks Benelux. Contact information is printed on the last page of this tutorial.

Initiative: Kees Kloosterboer (SolidWorks Benelux)
Educational Advisor: Jack van den Broek (Vakcollege Dr. Knippenberg)
Realization: Arnoud Breedveld (PAZ Computerworks)

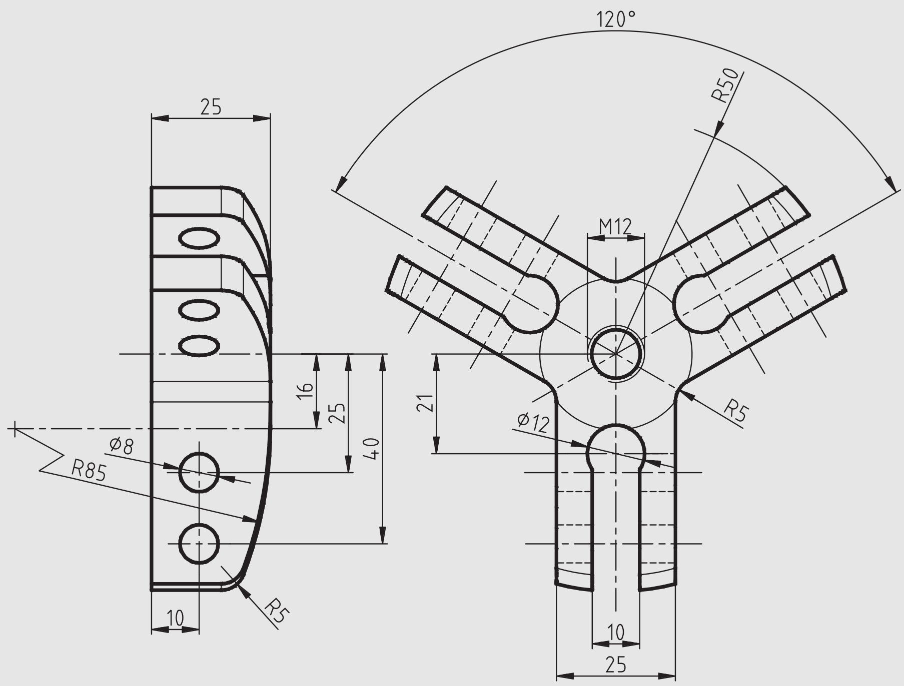
Bearing Puller

In this tutorial, we will build a bearing puller. This product consists of three parts. We will learn a few new functions in this tutorial. We will also perform a simple analysis on some of the parts.

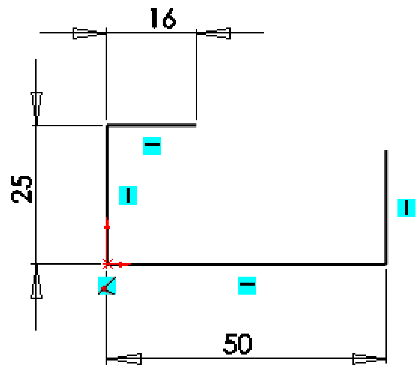
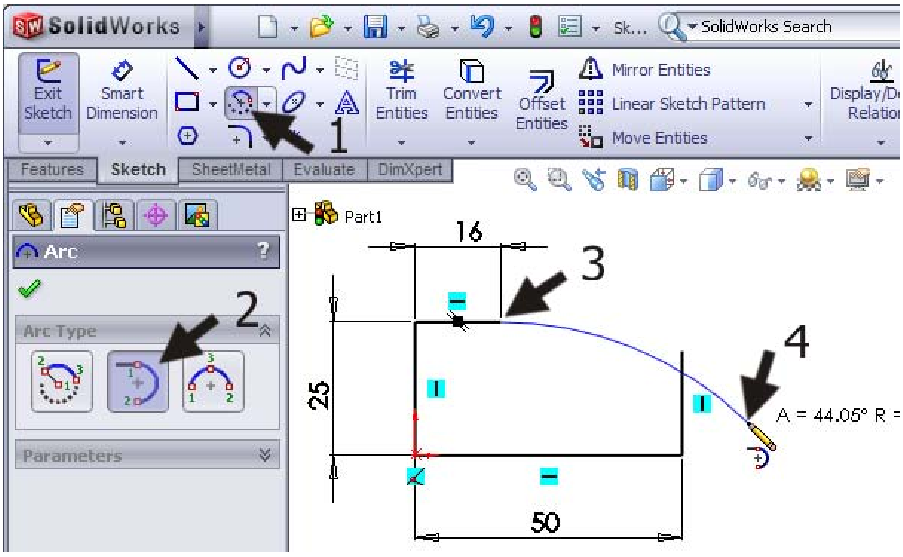
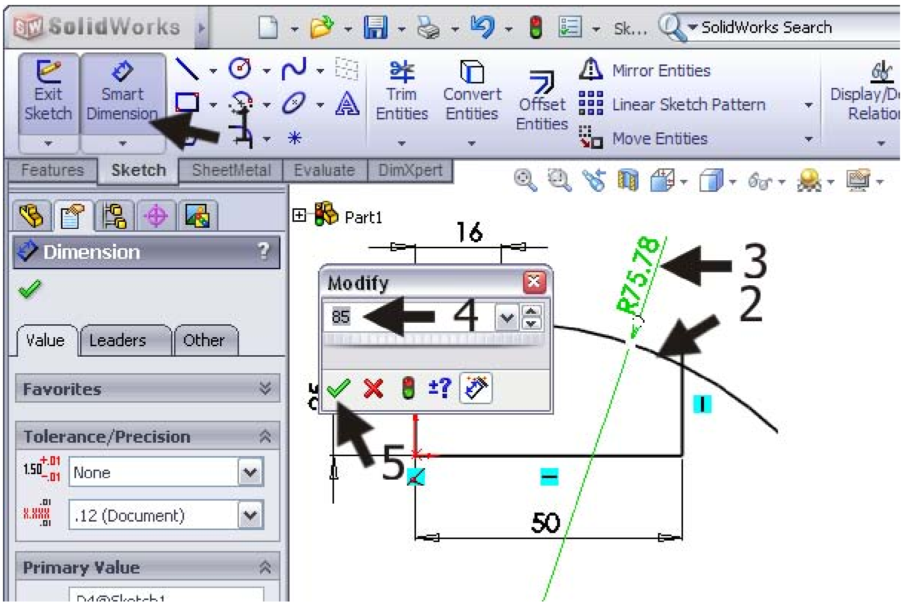


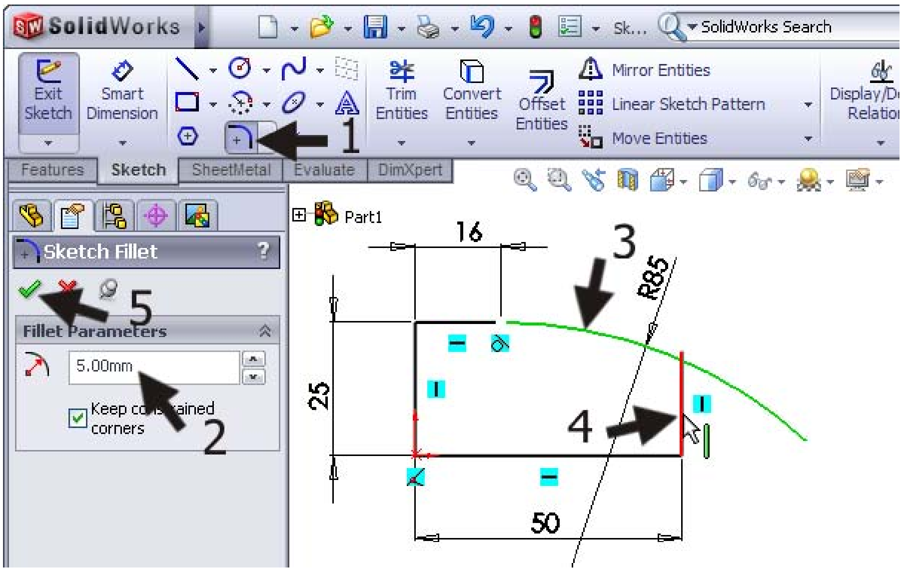
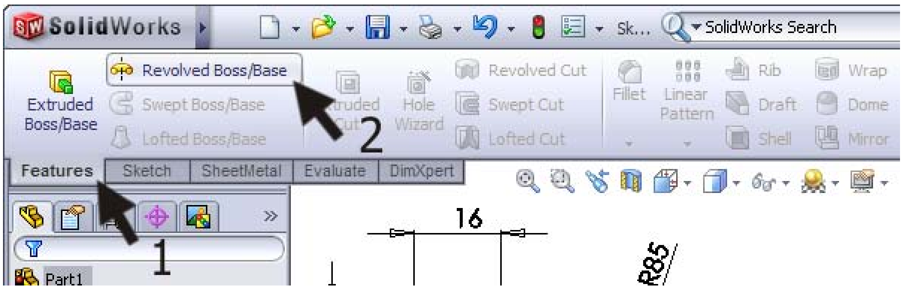
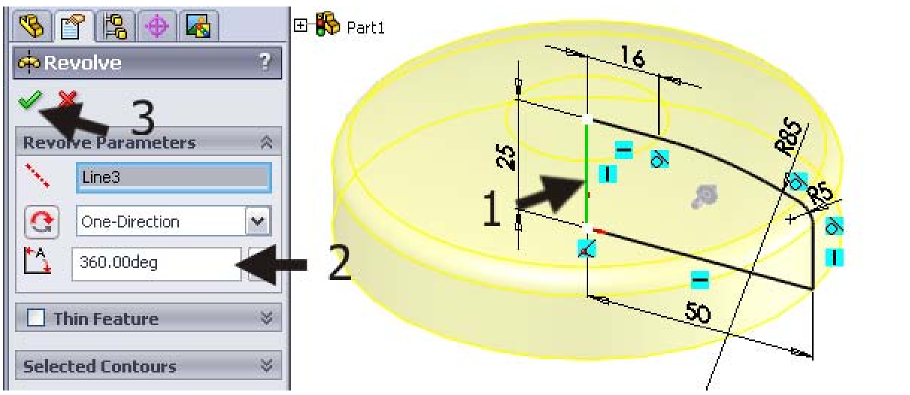
Work plan

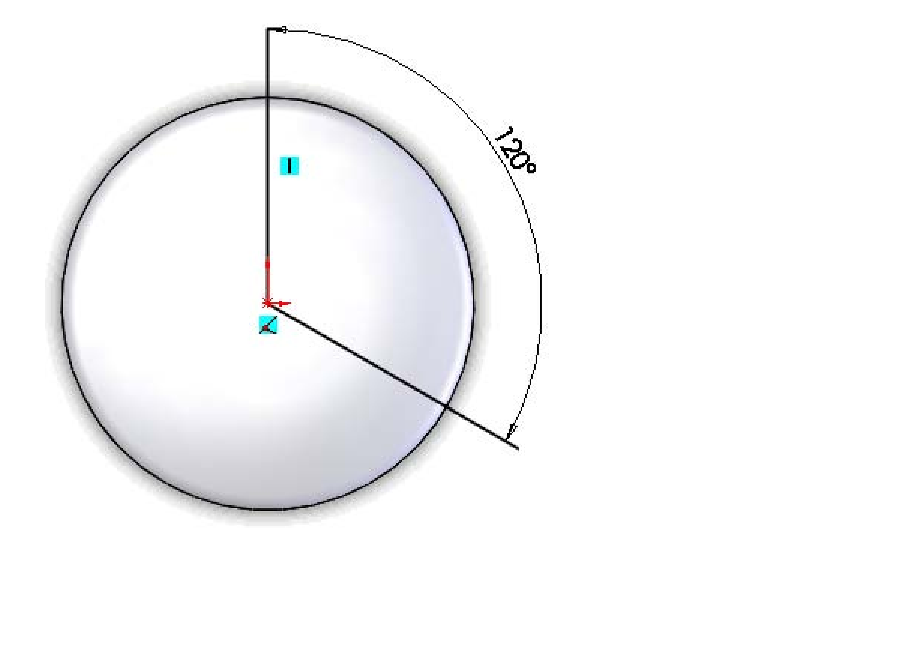
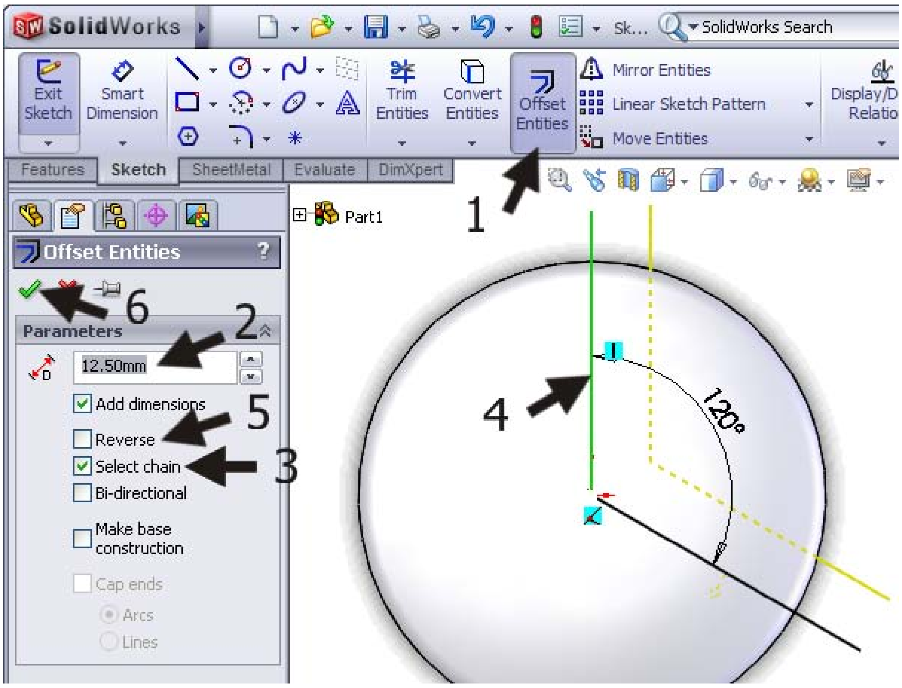
The first part we will make is the main bridge. We will make this according to the drawing below.

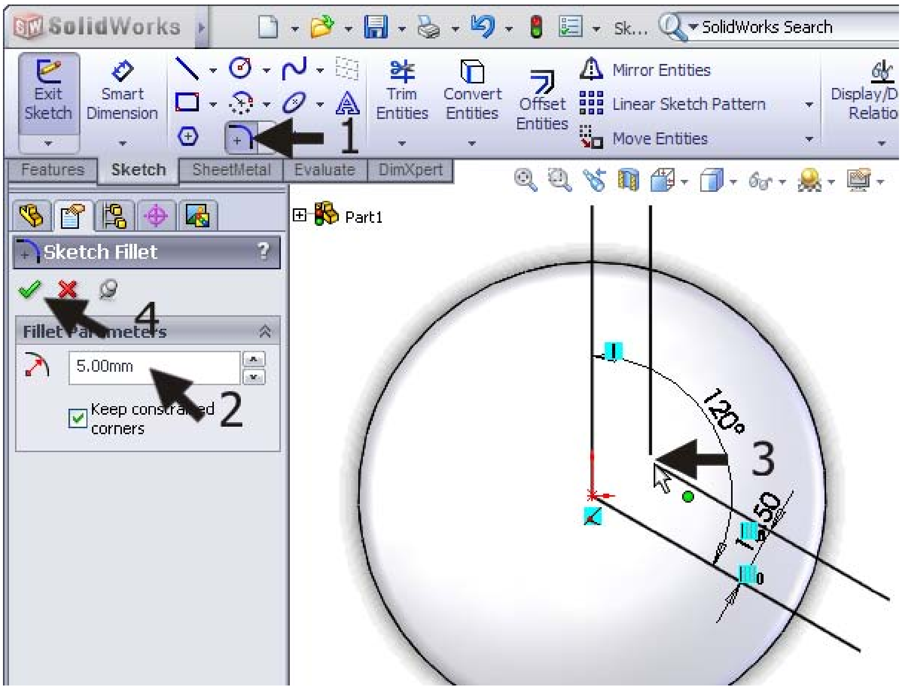
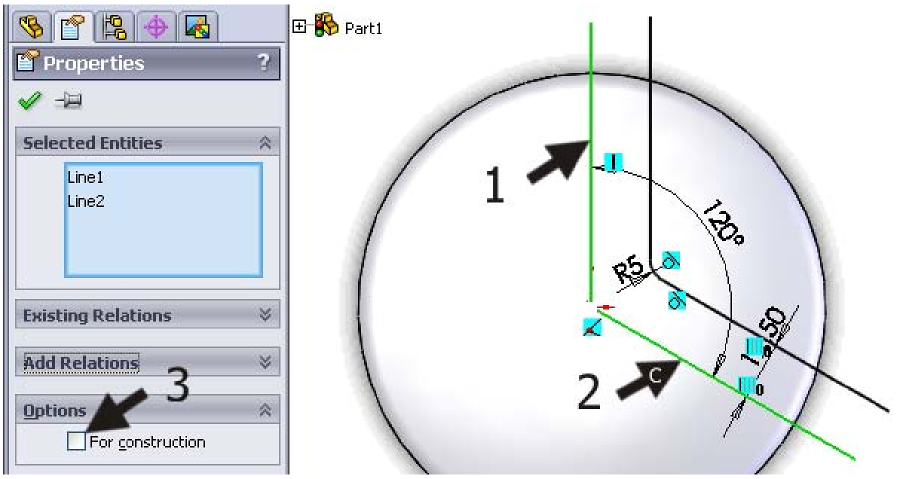
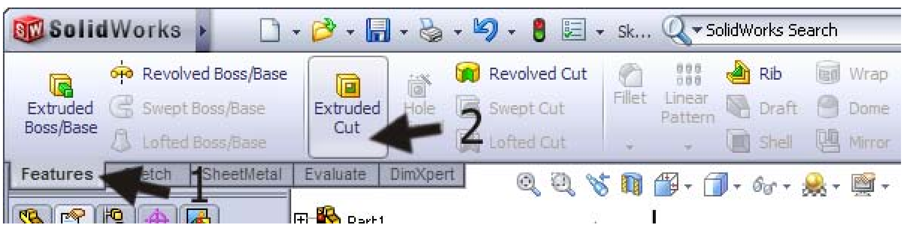


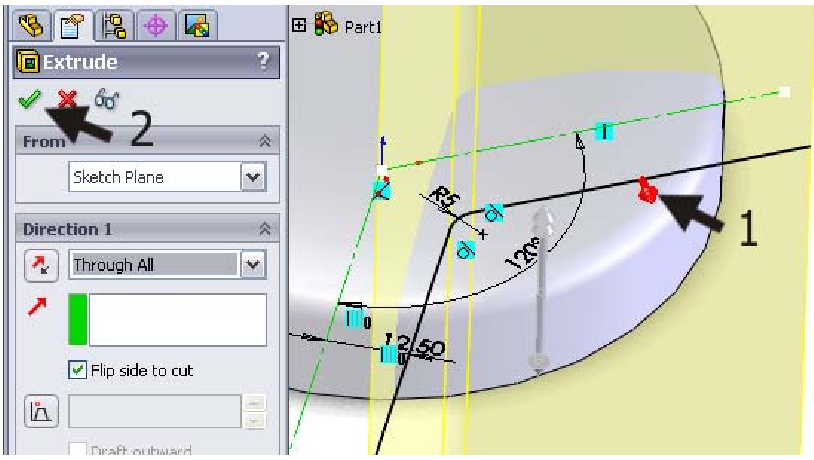
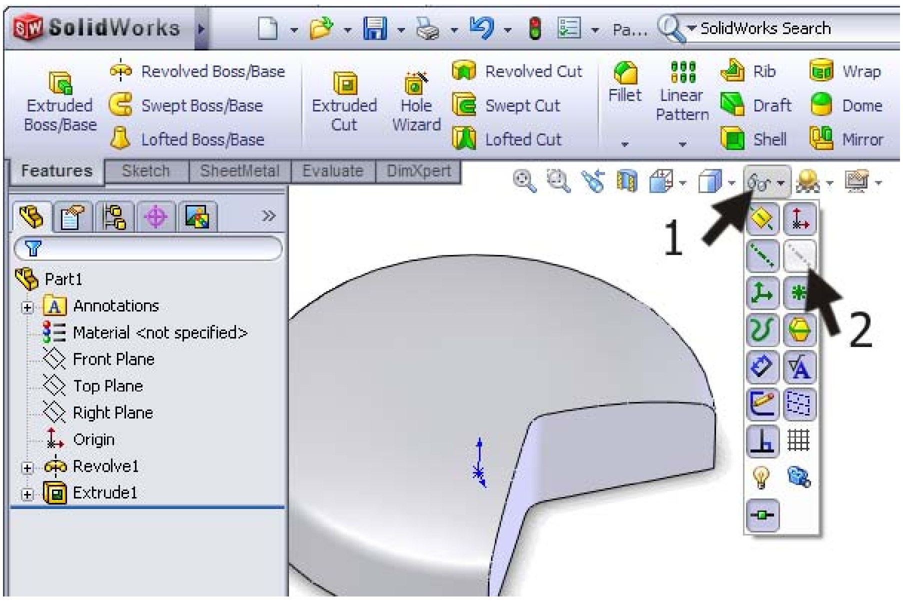
Make a plan! How would you build this part? Make a plan for yourself and compare it with the plan we have developed for this tutorial.

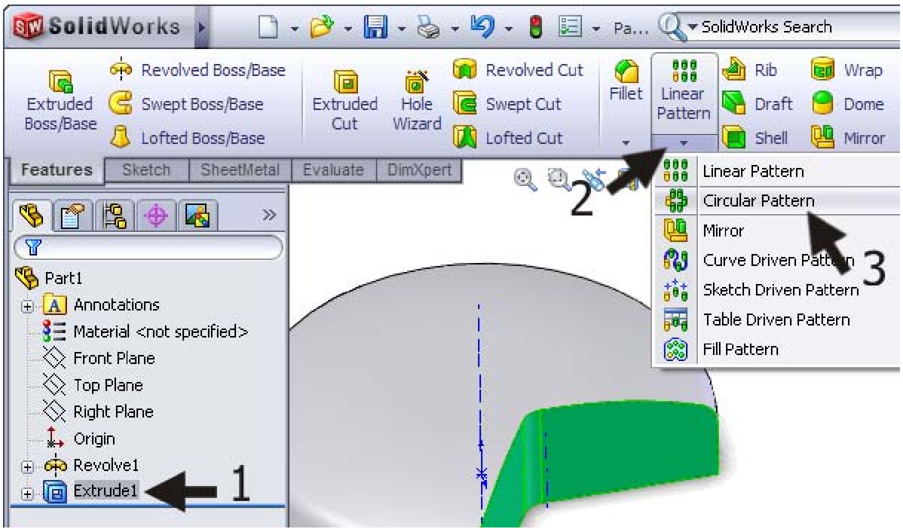
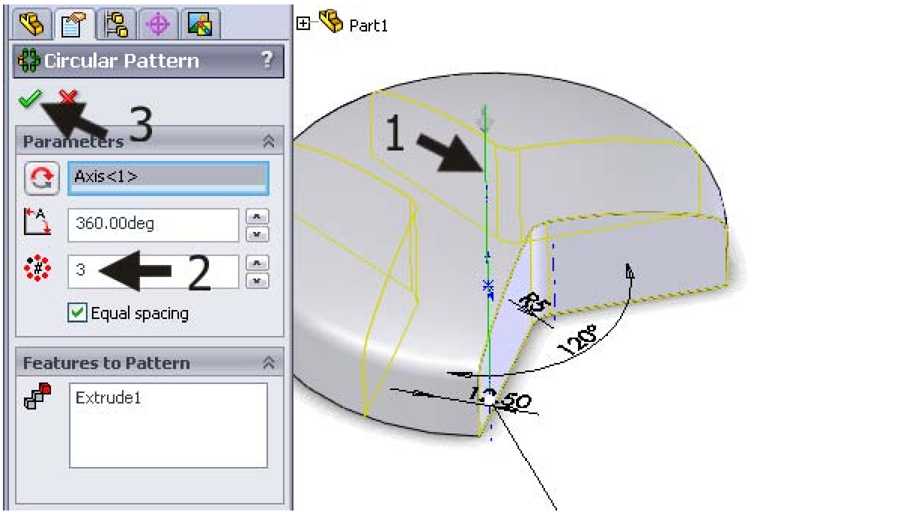
1	Start SolidWorks and open a new part.	
2	Select the Front Plane and make a sketch like in the illustration on the right. The sketch consists of four lines and three dimensions. Make sure the left bottom corner of the sketch is at the origin.	
3	1. Click on Arc in the CommandManager. 2. Click on Tangent Arc in the PropertyManager. 3. Click on the right end of the upper horizontal line. 4. Put the end of the arc at about the same location as in the drawing. The exact spot is not relevant at this point. 5. Push the <Esc> key to end the line command.	
4	Set dimensions for the arc you have just drawn: 1. Click on 'Smart Dimension' in the CommandManager. 2. Click on the arc. 3. Set the dimension. 4. Change the radius of the arc to '85'. 5. Click on OK.	

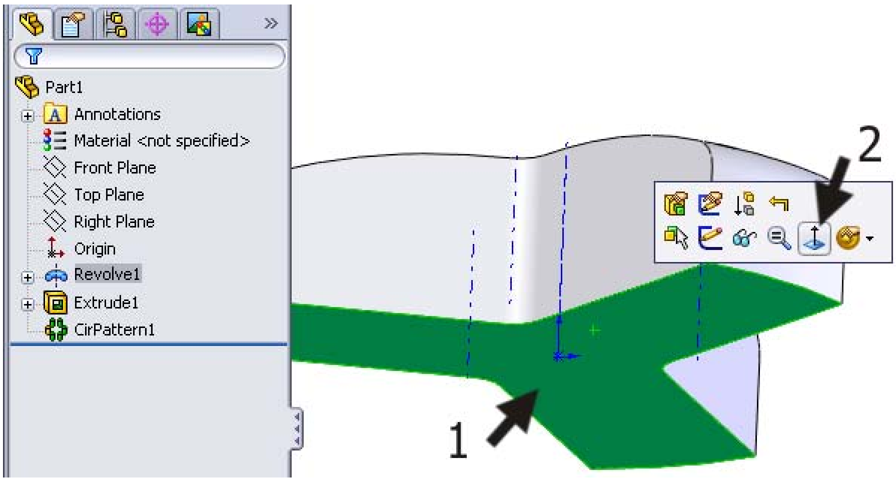
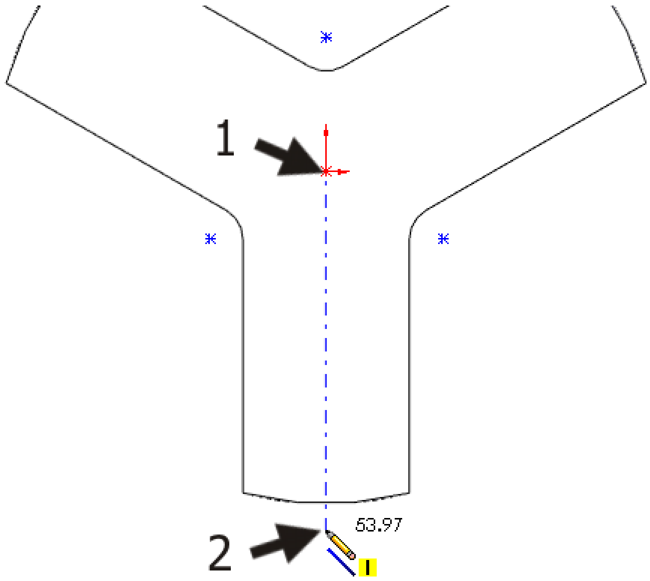
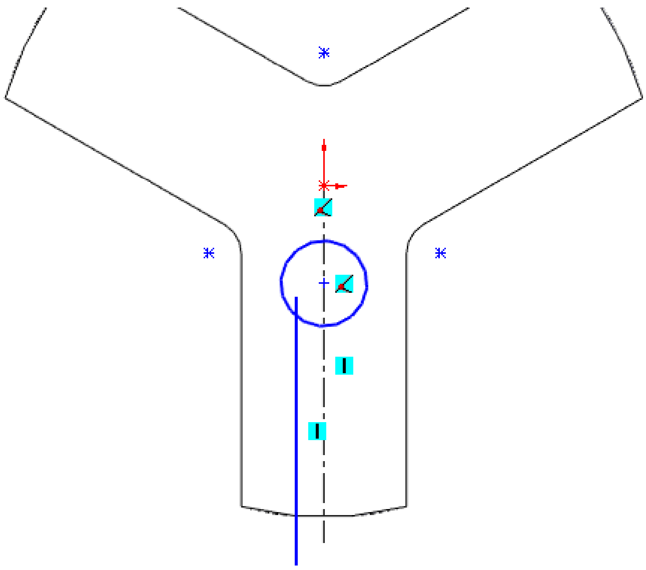
5	Make a curved edge between the arc and the vertical line. 1. Click on Sketch Fillet in the CommandManager. 2. Change the radius to '5mm' in the PropertyManager. 3. Click on the arc, to the left of the vertical line. 4. Click on the vertical line, just below the arc. 5. Click on OK.	
6	Click on 'Features' in the CommandManager and next on 'Revolved Boss/Base'.	
7	Next, you have to set the rotation axis: 1. Click on the left vertical line in the sketch. 2. Make sure the rotation angle in the PropertyManager is set to '360 degrees' (a complete circle). 3. Click on OK.	

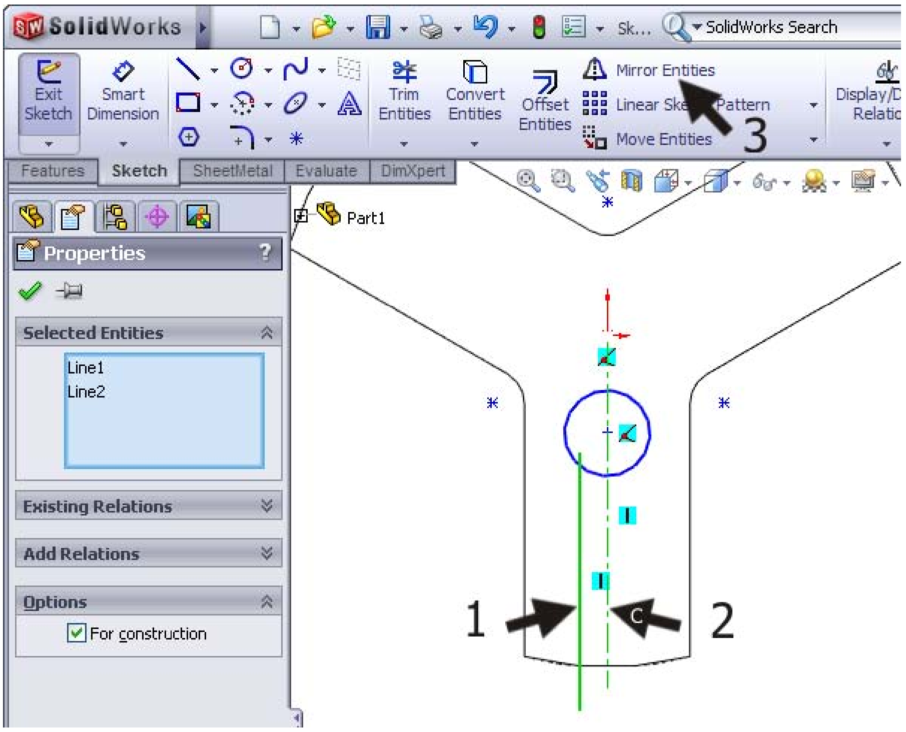
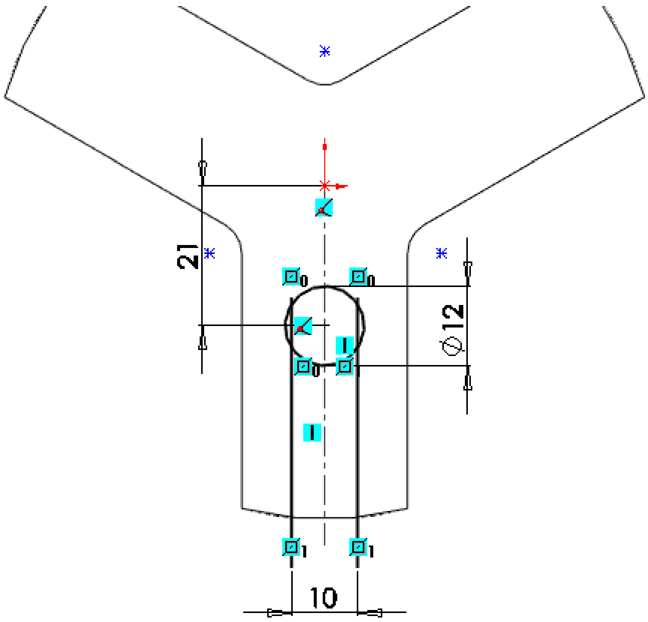
8	The basic form is ready. We will now remove three triangles from this body. Select the Top Plane and create a sketch like in the illustration on the right. The sketch consists of two lines emanating from the origin: one line goes straight up and the other runs downwards under an angle of about 120 degrees to the first line. Both lines cross the outside edge of the part. Set the dimension of '120 degrees' between the two lines.	
9	Make a parallel copy of the two lines. 1. Click on 'Offset Entities' in the CommandManager. 2. Change the distance in the PropertyManager to '12.5mm'. 3. Make sure the option 'Select chain' is selected. 4. Click on one of two lines in the sketch. You can now see a preview. Both lines from the sketch are copied. 5. When the lines are copied in the wrong direction, click on 'Reverse' in the Property-Manager. 6. Click on OK.	

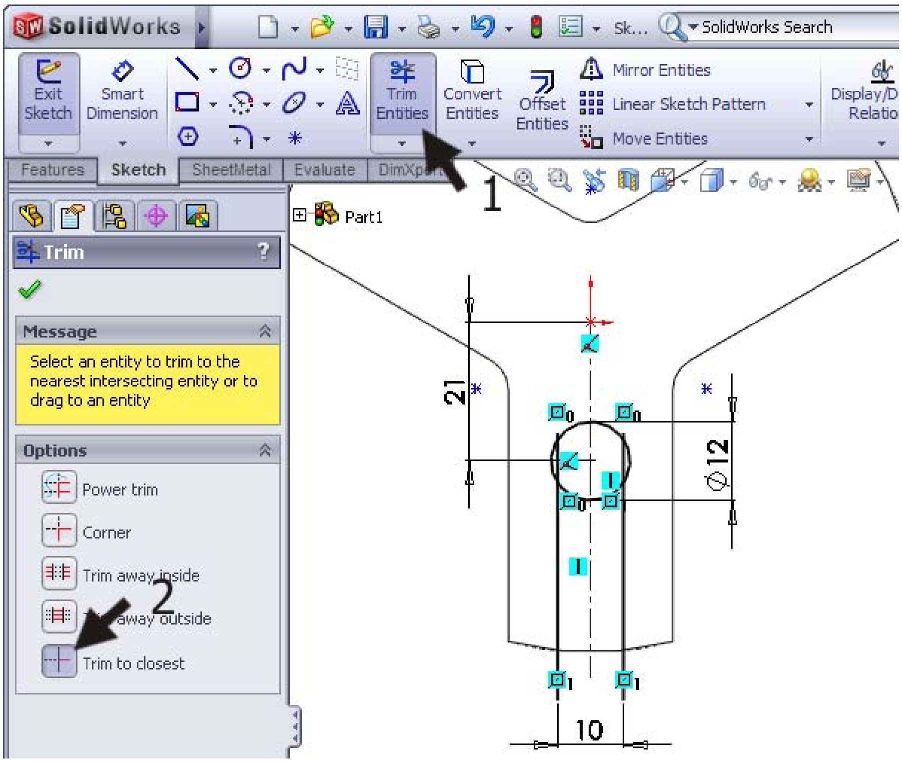
10	Round of the corners between the two lines. 1. Click on Sketch Fillet in the CommandManager. 2. Check to make sure that the radius is still 5mm (you set this in step 6 already, and it should have remained in SolidWorks). 3. Click on the corners of both copied lines 4. Click on OK.	
11	Next, we will make construction lines from the first two lines we have drawn. 1. Select the first line. 2. Hold the <Ctrl> key on your keyboard and select the second line. 3. Check the option 'For construction' in the PropertyManager. The two lines will now be displayed as centerlines.	
	Tip!	We have also used centerlines in other tutorials. These lines are actually auxiliary lines. When you use a sketch to make an extrusion, for example, SolidWorks only uses the 'real' lines and not the auxiliary lines. In step 13 you have seen that you can easily change a 'real line' (or circle of arc) into an auxiliary line and vice versa. For this the option, the 'For construction' box in the PropertyManager must be checked.
12	Next, we will cut a corner from the model: 1. Click on 'Features' in the CommandManager. 2. Click on 'Extruded Cut'.	

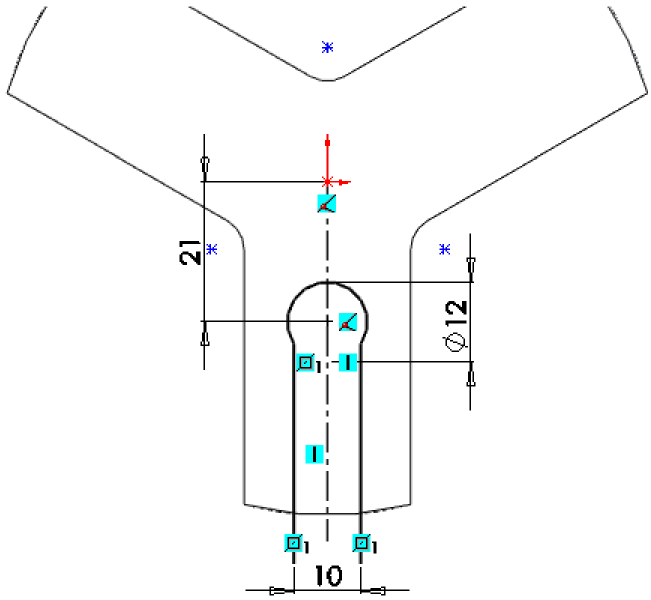
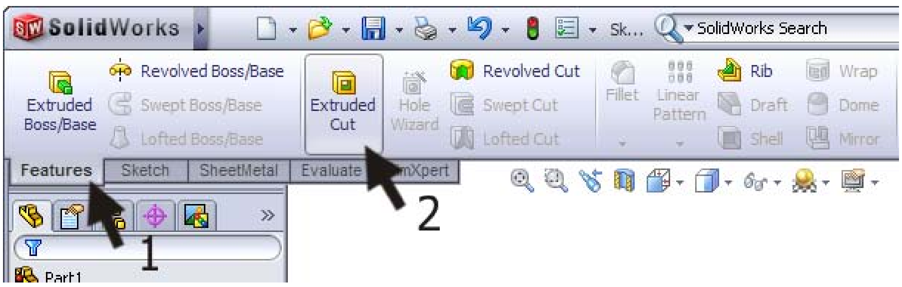
13	You can see a small arrow In the model that indicates from which side of the sketch the material will be removed. 1. Make sure these arrows point outwards. Click on it when you need to change the direction. 2. Click on OK.	
	Tip!	In most cases you will use a closed sketch for an 'Extruded Cut'. In the case of a circle or a square you will only make a hole in the shape of that sketch. In the last step, we used an open sketch to make an 'Extruded Cut'. It is handled in the same way except for two differences: 1. An 'Extruded Cut' with an open sketch will always go through the entire depth of the model ('Through all'). You cannot set a depth. 2. SolidWorks needs to know from which side the material has to be cut away. You must pay attention to the little arrow, which indicates the cutting side. By the way, you can also change this direction in a closed sketch and cut away the material from the inside or outside of the sketch boundaries.
14	For the next features we need an auxiliary line that runs through the middle of the model. This axis consists in the model already but is not visible with the standard (default) settings. 1. Click on the Hide/Show Items icon. 2. Make sure the button View Temporary Axes is set.	

15	Next, we can copy the part with the cut three times around the axis. 1. Select the last feature: 'Extrude1' in the FeatureManager. 2. Click on the arrow below 'Linear Pattern' in the CommandManager. 3. Click on 'Circular Pattern'.	
16	1. Select the centerline that runs through the middle of the model. 2. Change the number of copies in the PropertyManager to '3'. 3. Click on OK.	
	Tip!	Notice that in the three last steps we first selected a feature in the FeatureManager and then selected the 'Circular Pattern' command. At this point, SolidWorks 'understands' that you want to use this command for the selected items and automatically adjusts the settings in the PropertyManager. You can also do this in the reverse order by giving the command first and then selecting the elements in the PropertyManager. SolidWorks does not have a preference for how you do it. You will have to find out for yourself the approach that works best for you.

17	We will now make a sketch on the lower surface of the model. Rotate the model so you can see the bottom plane of the part. 1. Click on the surface to select it. 2. Click on Normal To in the menu that appears.	
18	Draw a Centerline. 1. Put the first point right on the origin. 2. Set a second point at a random distance directly below the origin.	
19	Draw a circle and a line at the locations indicated on the right. The midpoint of the circle must be on top of the centerline.	

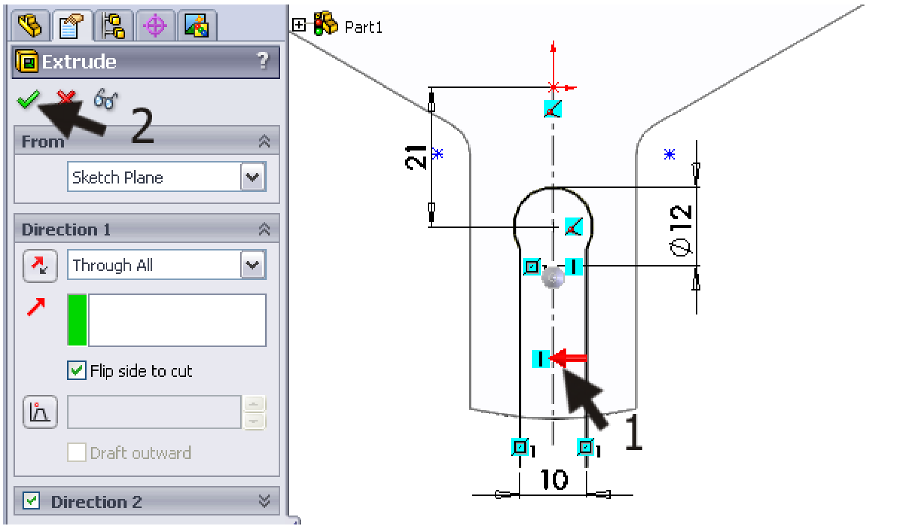
20	Make a mirrored image of this line at the other side of the centerline. 1. Select the centerline (hold the <Ctrl>-key). 2. Click on 'Mirror Entities' in the CommandMa-nager.	
21	Now, set the three dimen-sions you see in the illu-stration on the right. Do this using Smart Dimension and change the values.	

<table>
<tr>
<td>22</td>
<td>1. Click on ‘Trim Entities’ in the CommandManager.
2. Select the option ‘Trim to closest’ in the PropertyManager.</td>
<td>
</td>
</tr>
<tr>
<td>23</td>
<td>Next, click on the parts of the sketch that must be removed. Make sure you end up with a sketch similar to the one on the right.

Should the dimension of 10mm disappear as a result of the trimming command, resize that item by using Smart Dimension again in the sketch.</td>
<td>
</td>
</tr>
<tr>
<td>24</td>
<td>Click on ‘Features’ in the CommandManager and then on ‘Extruded Cut’.</td>
<td>
</td>
</tr>
</table>

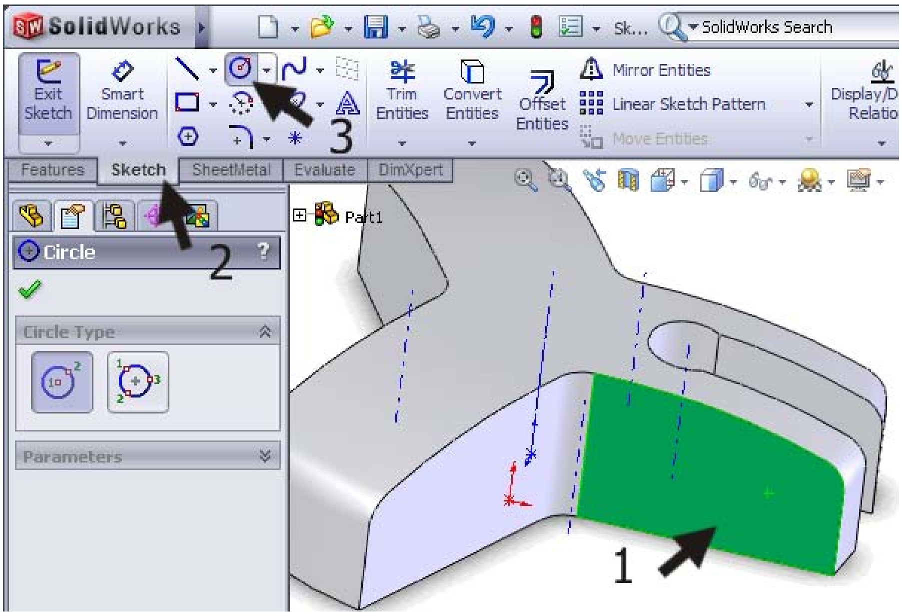
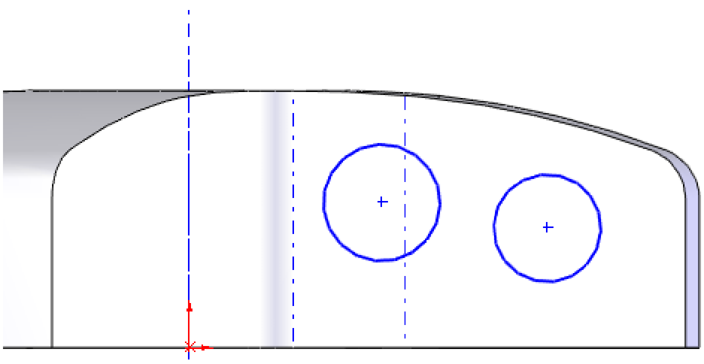
<table>
<tr>
<td>25</td>
<td>You must pay attention to which direction the material is removed from because the sketch is not entirely closed.

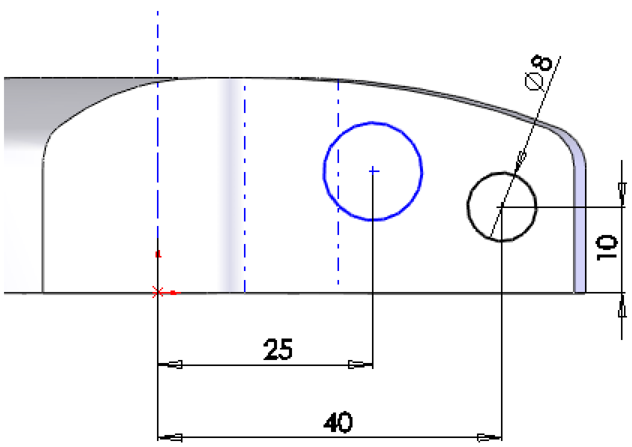
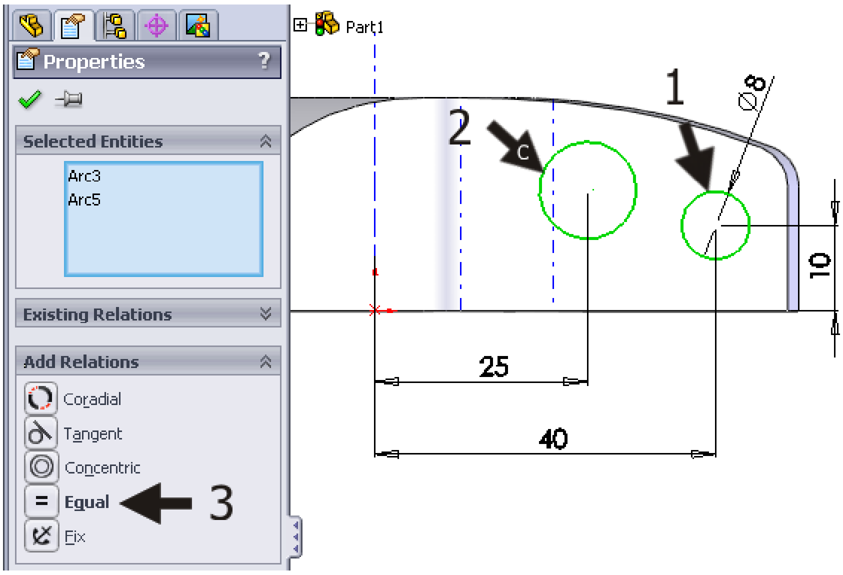
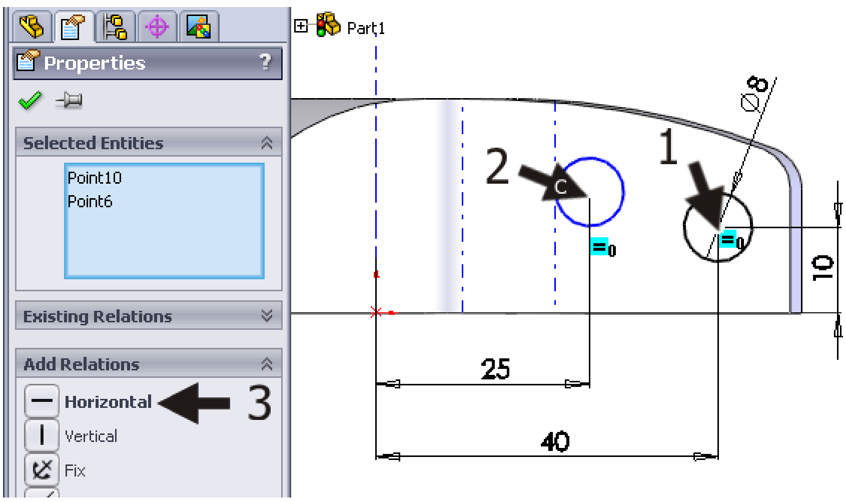
1. Make sure the little arrow that sets the direction is pointing inward.

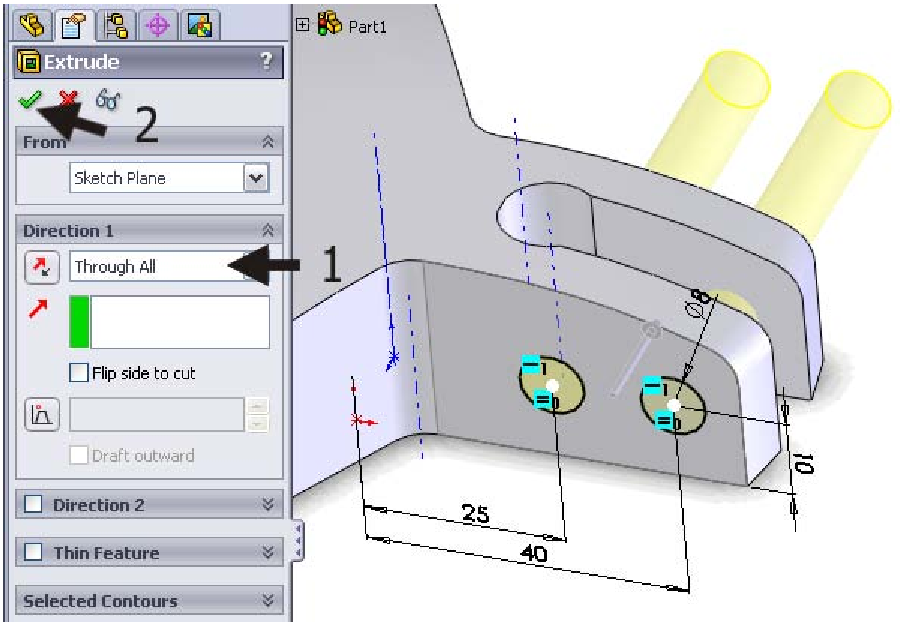
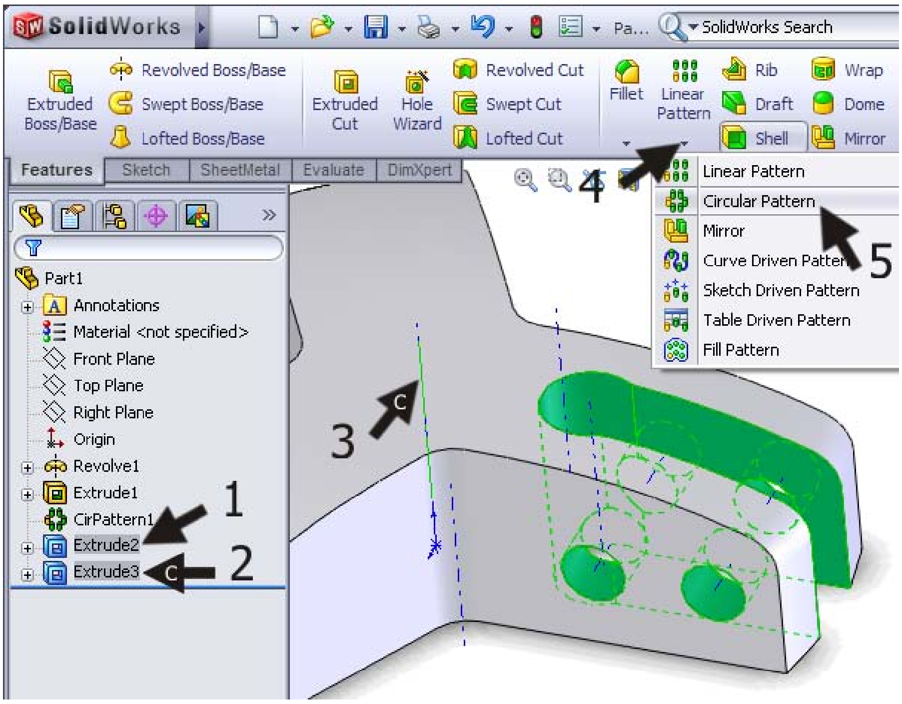
2. Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>26</td>
<td>Next, we have to make some holes.

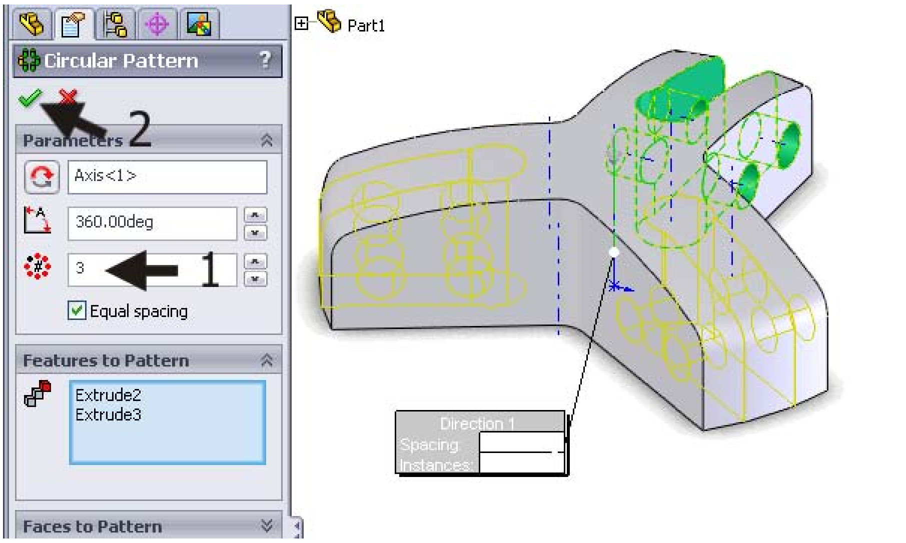
1. Select the plane as indicated in the illustration.

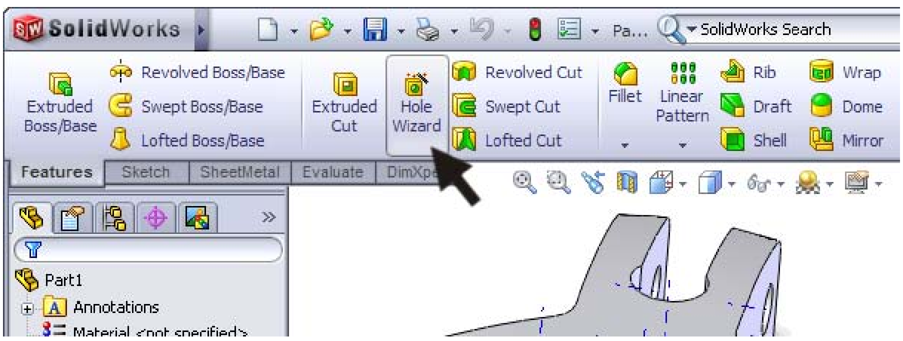
2. Click on 'Sketch' in the CommandManager.

3. Click on Circle.</td>
<td>
</td>
</tr>
<tr>
<td>27</td>
<td>Rotate the model with Normal To, and draw two circles at random positions like in the drawing on the right.</td>
<td></td>
</tr>
</table>

28	Use Smart Dimension to set four dimensions in the sketch, and change their values as indicated on the right. Push the <Esc> key to close the Smart Dimension command.	
29	Next, set the circles to the same size: 1. Select one of the circles. 2. Hold the <Ctrl> key and select the other circle. 3. Click on 'Equal' in the PropertyManager.	
30	Next, set the circles to the same height: 1. Select the midpoint of one of the circles. 2. Hold the <Ctrl> key and select the midpoint of the other circle. 3. Click on 'Horizontal' in the PropertyManager.	

31	Click on 'Features' in the CommandManager, and after that on 'Extruded Cut'. 1. Set the depth to 'Through All' in the PropertyManager. 2. Click on OK.	
32	We must now copy the holes we just made to the other 'legs'. 1,2 Select the last two features in the FeatureManager. 3. Select (holding the <Ctrl> key) the axis that runs through the middle of the model. 4. Click on the arrow below 'Linear Pattern' in the CommandManager. 5. Click on 'Circular Pattern'.	

<table>
<tr>
<td>33</td>
<td>1. Set the number of copies in the PropertyManager to ‘3’.
2. Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>34</td>
<td>Finally, we have to make the metric thread in the hole:

Click on ‘Hole Wizard’ in the CommandManager.</td>
<td>
</td>
</tr>
</table>

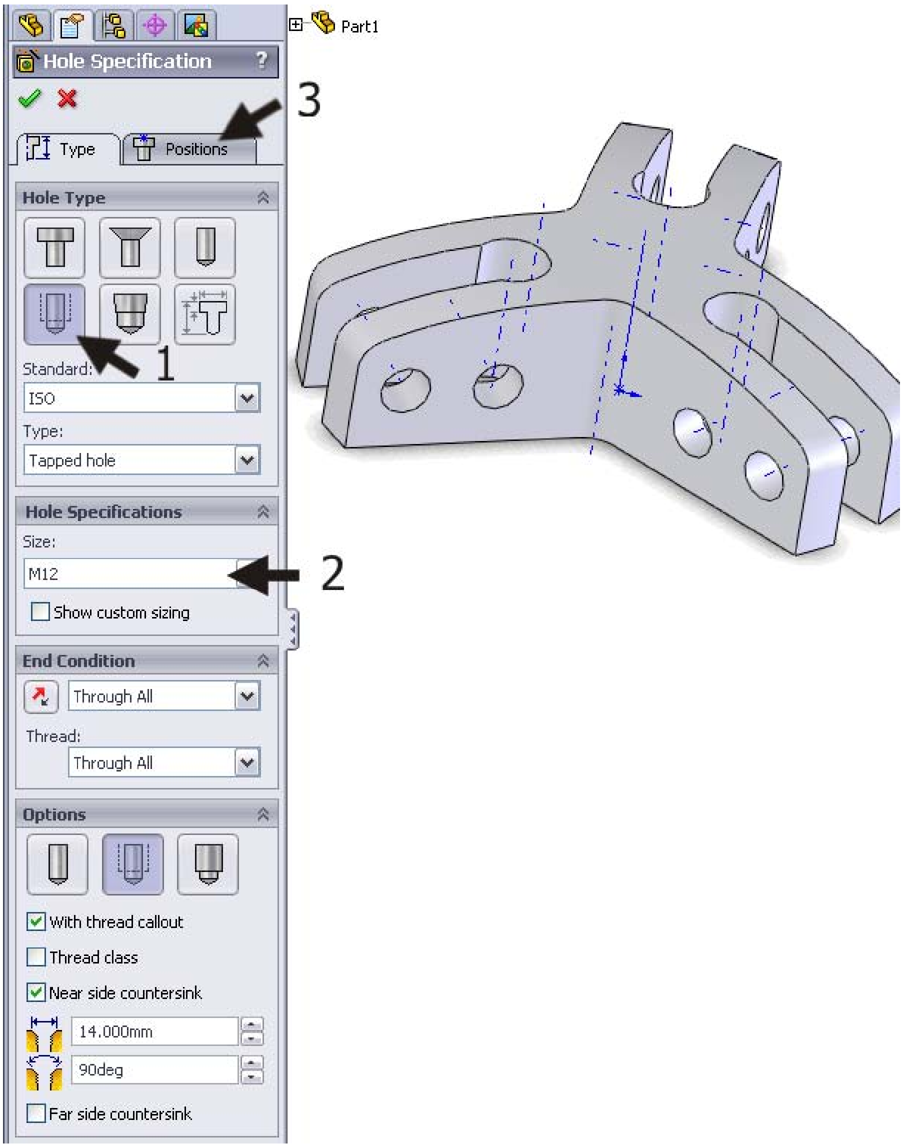
35

Set the following features in the PropertyManager:

1. The 'Hole Type' is Tap.
2. The 'Size' is 'M12'.

Check the other settings to make sure they concur with the illustration on the right.

3. When everything is set properly, click on 'Positions' to place the hole.

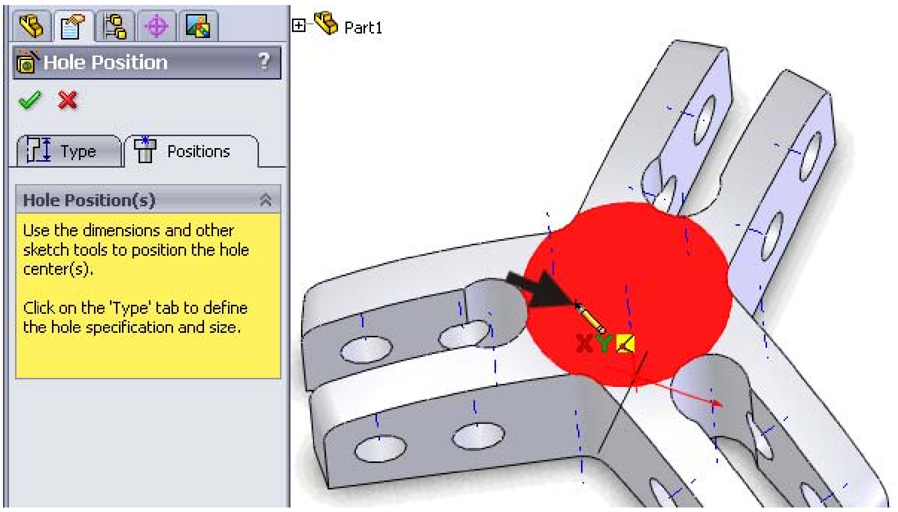


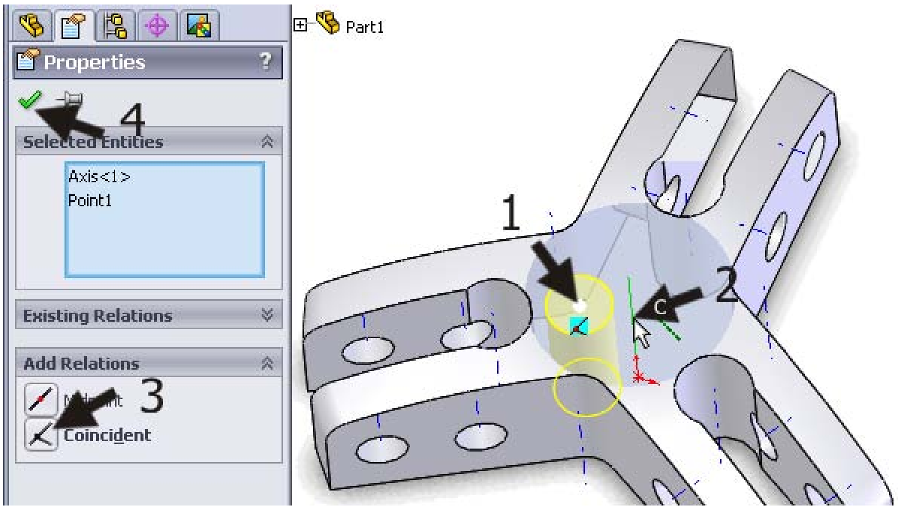
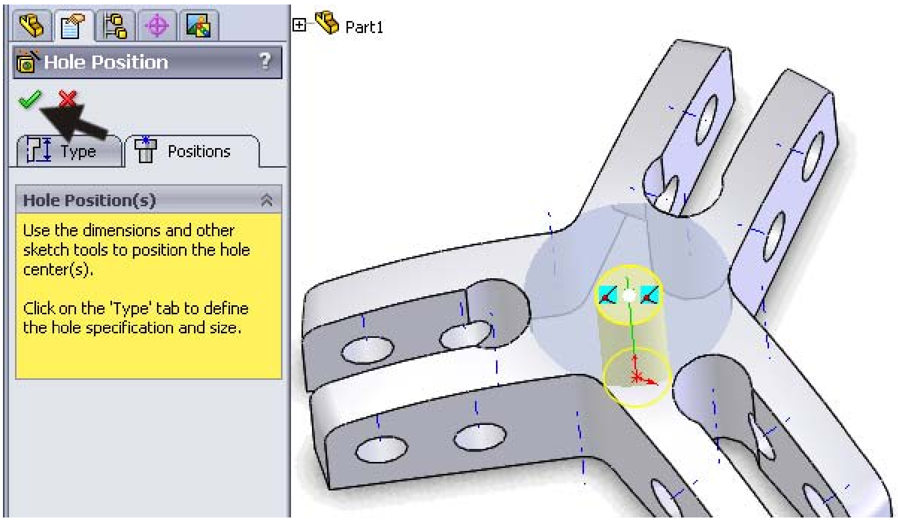
36

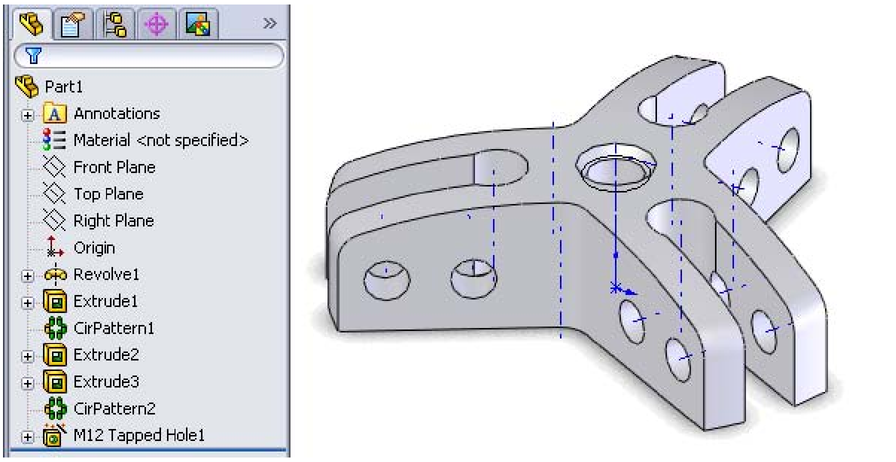
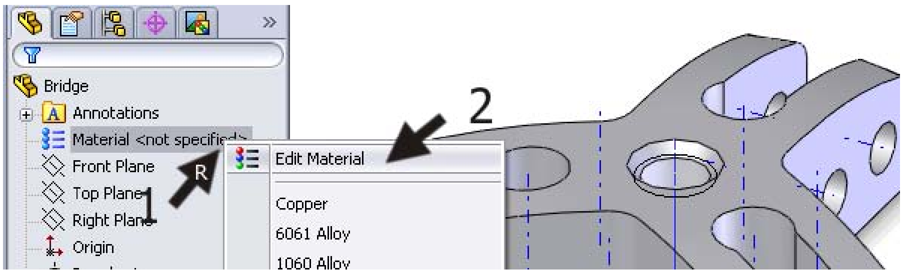
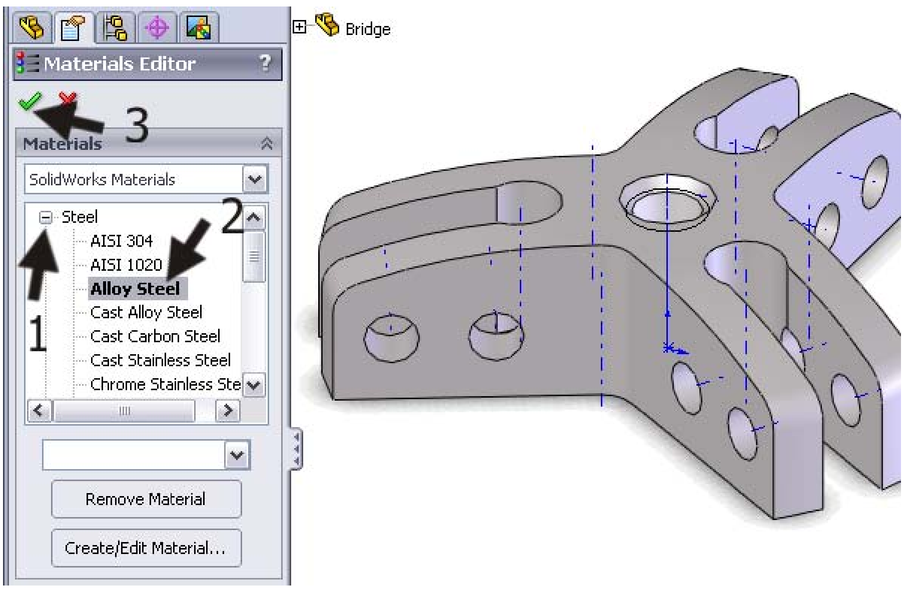
Set the hole on the top plane of the bridge at a random position.

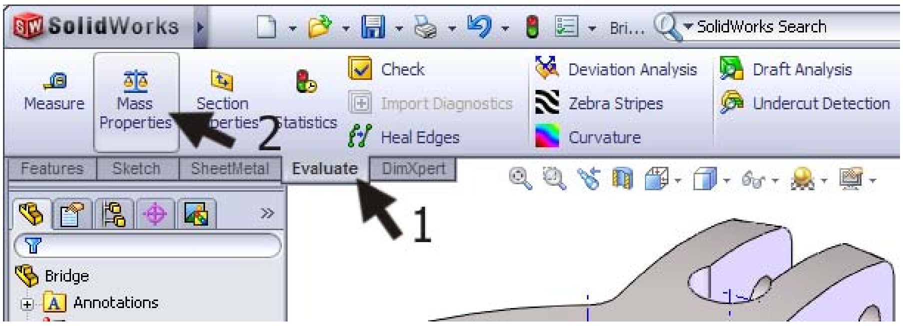
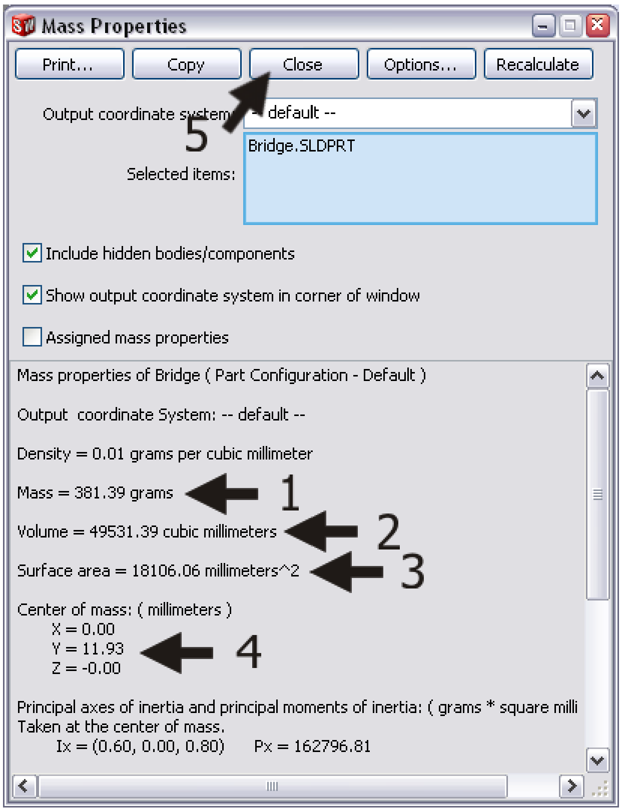
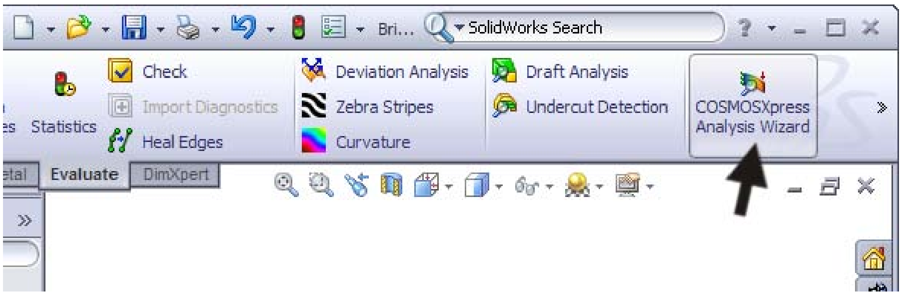
Actually, you are setting a point now, which will determine the position of the hole.

The point is on the plane, but unfortunately it is not possible to put this point in the midpoint of the plane. To do this, we conduct an additional step.

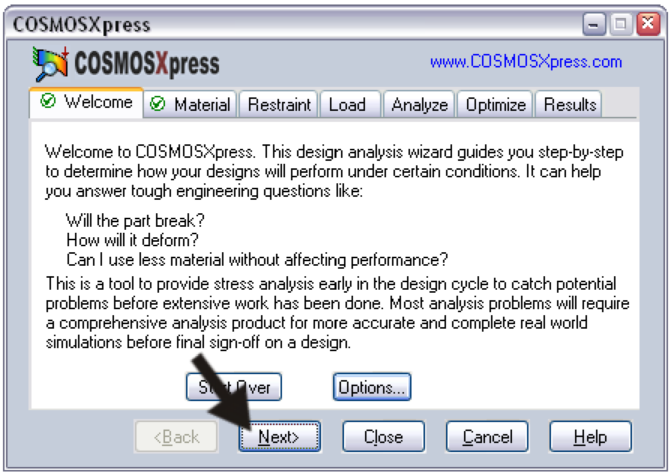
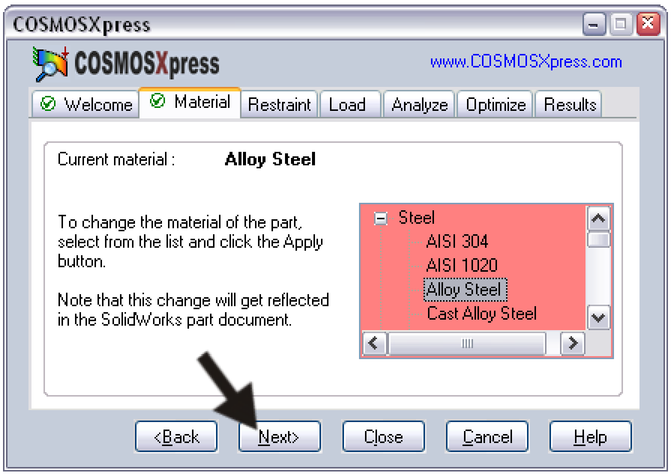


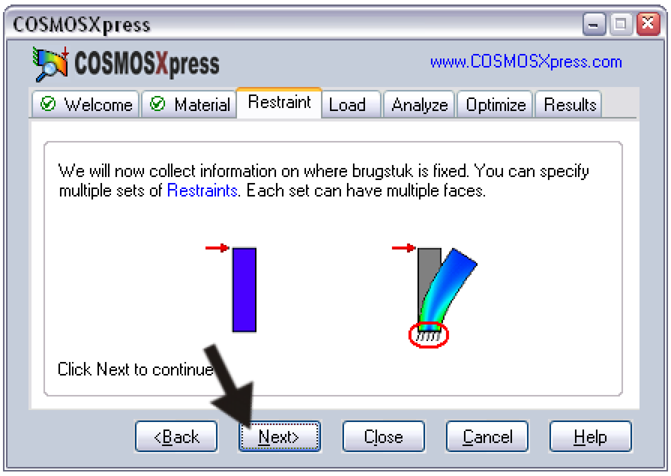
<table>
<tr>
<td>37</td>
<td>Push the <Esc> key first.
1. Select the point that you positioned in the last step.
2. Push the <Ctrl> key and select the axis we used before for circular patterns.
3. Click on 'Coincident' in the PropertyManager.
4. Click on OK.
The hole will now shift to the middle of the plane.</td>
<td>
</td>
</tr>
<tr>
<td>38</td>
<td>You can now return to the 'Hole Wizard'.
Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>When you have to place a hole using the Hole Wizard (steps 36-37), you are actually making a sketch. By putting a point in that sketch, you are positioning the hole.
The sketch you are making at this point is not an ordinary sketch, but a 3D sketch. In a 3D sketch you do not work in a plane (like in a regular sketch) but in a 3D environment. These 3D sketches will only occur in special applications in SolidWorks.</td>
</tr>
</table>

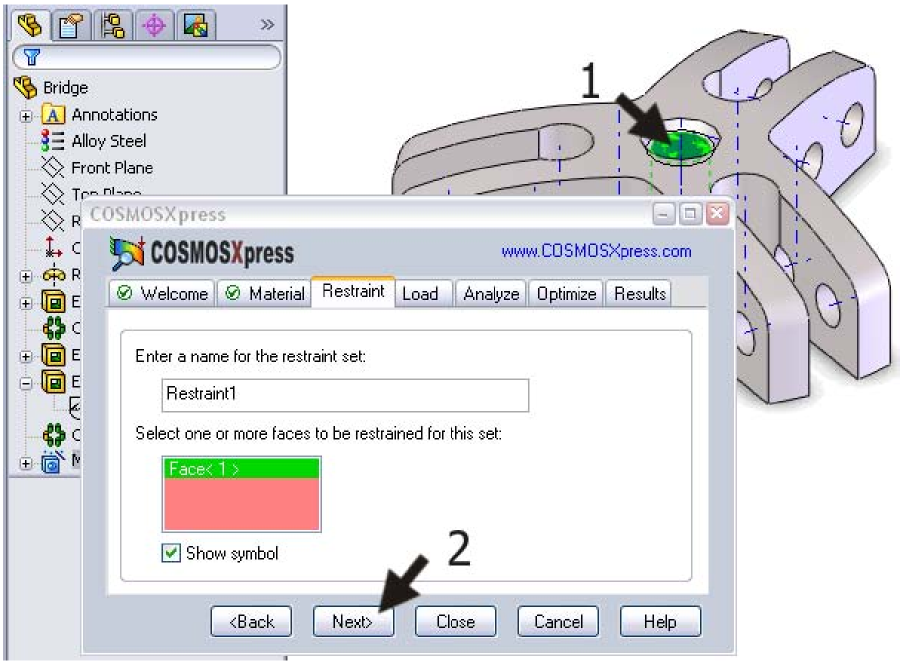
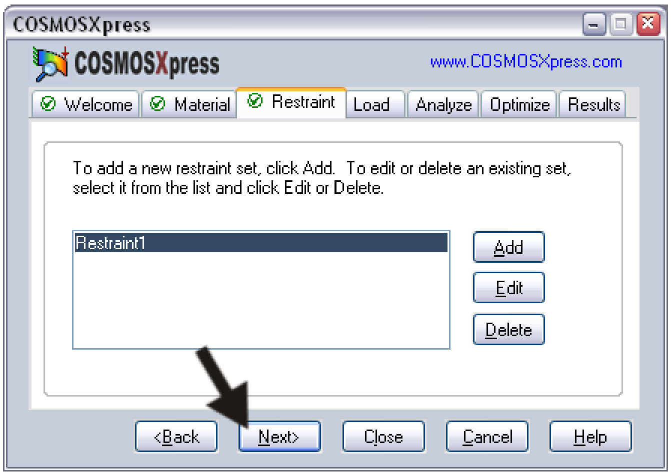
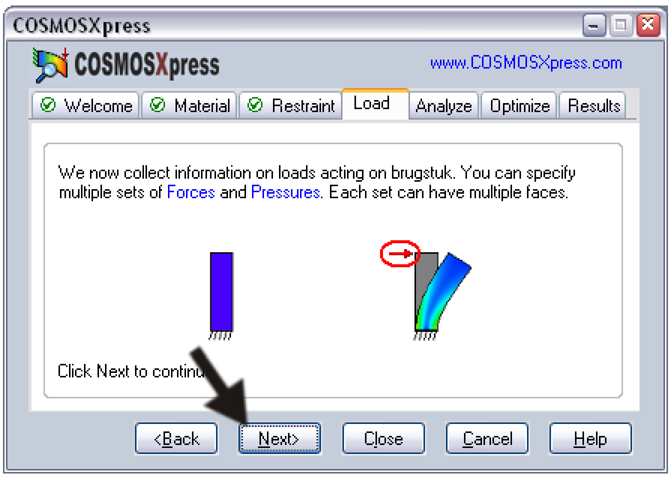
39	The model is now ready. Save it as: bridge.SLDPRT. First, create a new folder, so you can keep all files together.	
40	We would like to have more information about this model. What does is weigh? Where is the center of gravity? Is it strong enough? To be able to answer these kinds of questions, we must first determine the kind of material to use to make the part. 1. Right-click on 'Material' in the FeatureManager. 2. Select 'Edit Material' in the menu.	
41	1. Open the main group 'Steel' by clicking on the '+' symbol. 2. Select 'Alloy Steel' as the desired material. 3. Click on OK.	

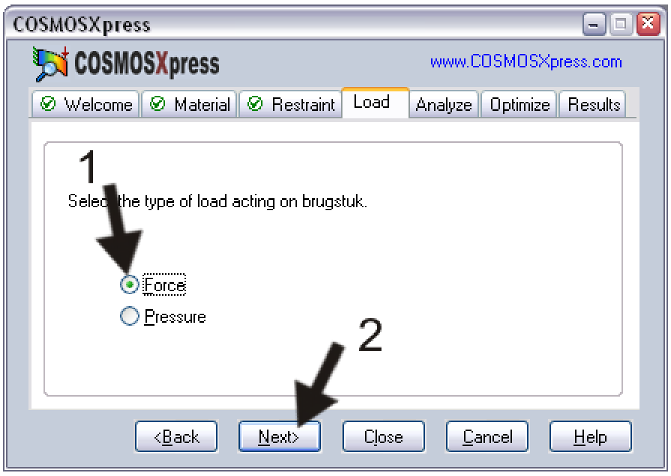
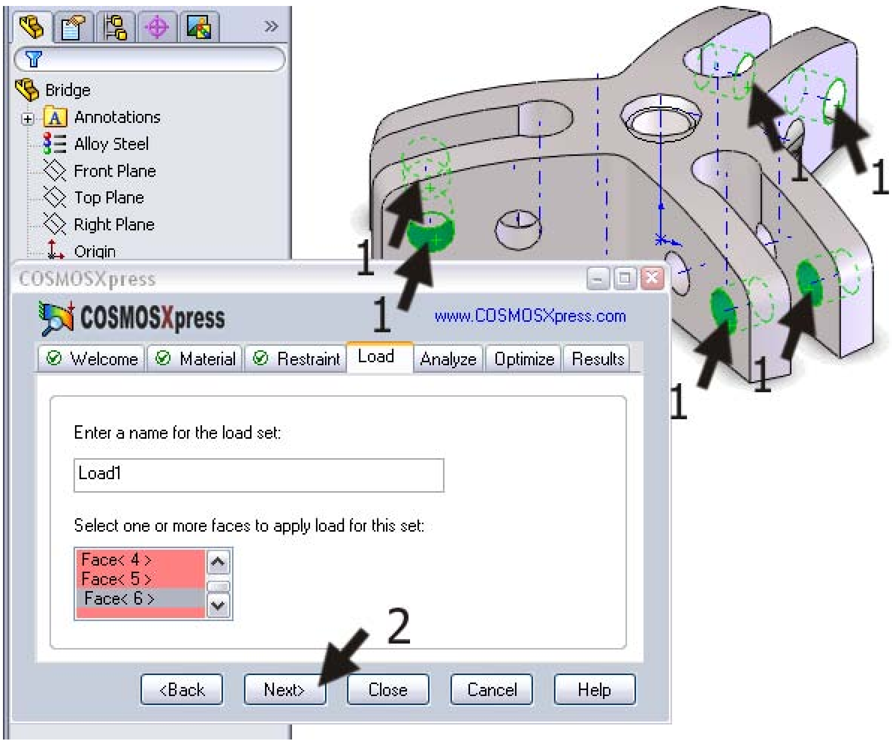
42	We can evaluate the data now. 1. Click on the tab 'Evaluate' in the CommandManager. 2. Click on 'Mass Properties'.	
43	A menu appears, in which you can read the data, including: 1. The weight of the part. 2. The volume. 3. The total surface of the part. This could be important when a part has to be painted. 4. The coordinates of the point of gravity. This is also displayed as a coordinate. 5. When you have finished reading the data, click on Close to close the window.	
44	Next we want to know if the part is strong enough for our purpose. We want to be able to pull 600kg (=6000N). To find out if our part is strong enough for this, we will use COSMOSXpress. Click on the 'COSMOSXpress Analysis Wizard' in the CommandManager.	

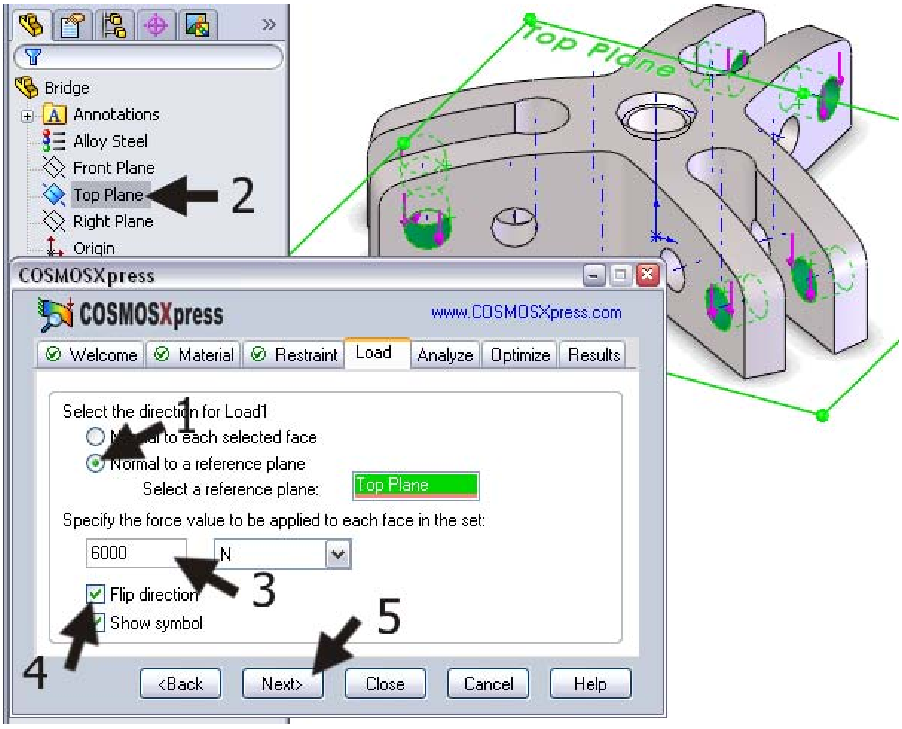
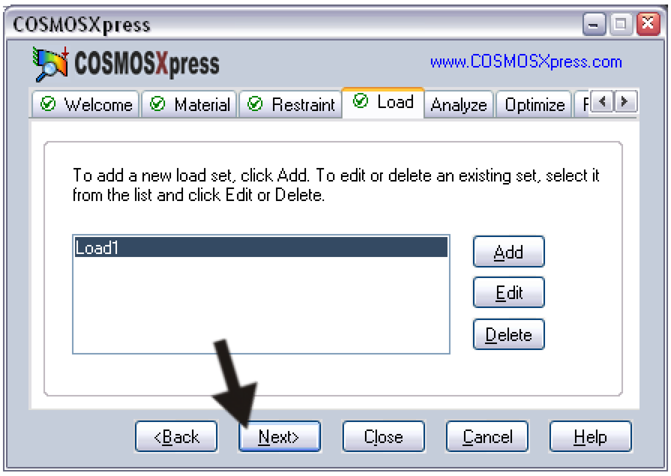
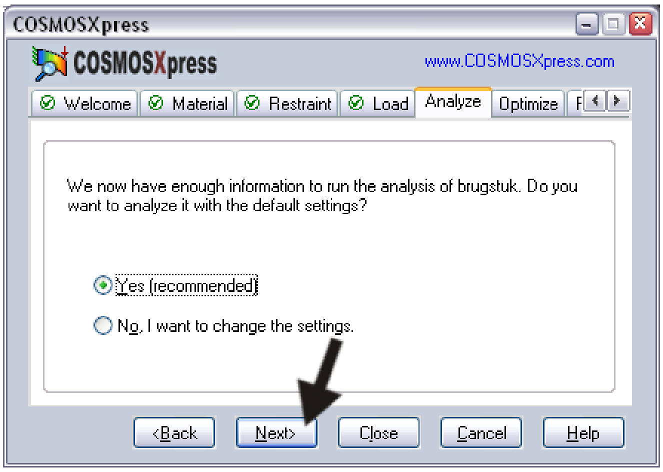
<table>
<tr>
<td>45</td>
<td>COSMOSXpress starts as a wizard. You will be led through a number of steps and will get a result at the end.

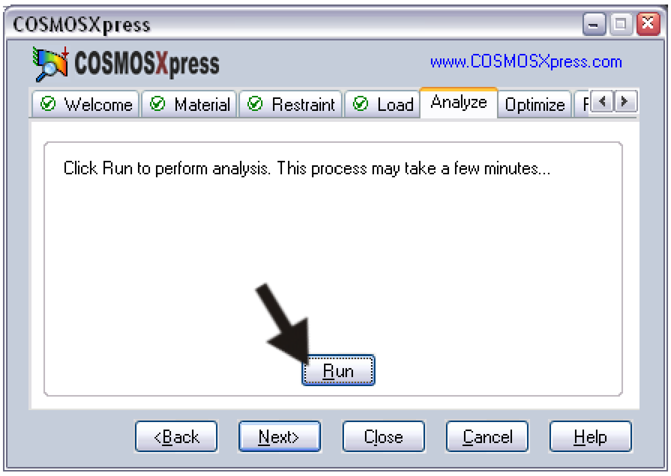
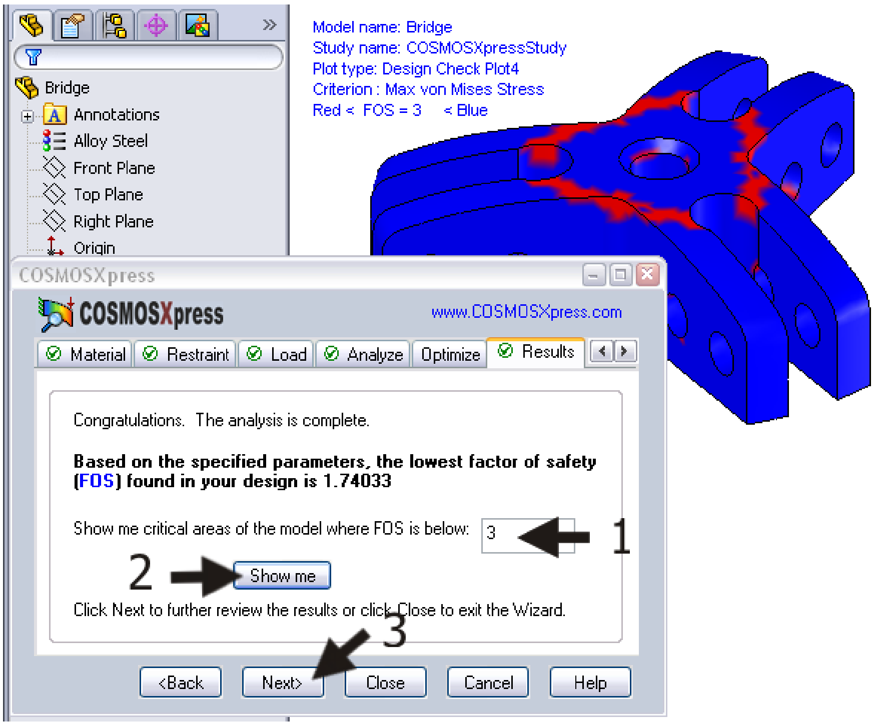
Click on next in the startup screen.</td>
<td>
</td>
</tr>
<tr>
<td>46</td>
<td>First, you must select the ‘Material’. We already did this so click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>47</td>
<td>We then establish the ‘Re-straint’: the fixed part of the bridge.

Click on Next.</td>
<td>
</td>
</tr>
</table>

<table>
<tr>
<td>48</td>
<td>1. Select the inside of the threaded hole in the model. In this calculation we assume that this is the plane that is fixed and cannot move.
2. Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>49</td>
<td>When desired, you can add more fixed planes. In this example we will not do so, so click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>50</td>
<td>We have now reached the tab where we can set the ‘Load’.
Click on Next.</td>
<td>
</td>
</tr>
</table>

51	You can set the load as a pressure or as a force. 1. Select 'Force'. 2. Click on Next.	
52	1. Select the six holes in which the arms will be mounted. 2. Click on Next.	

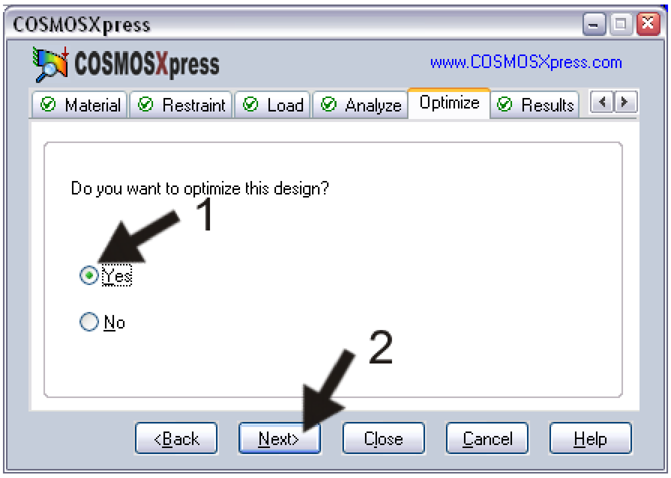
<table>
<tr>
<td>53</td>
<td>You must now set the direction of the force.
1. Check the option ‘Normal to a reference plane’. You will set the force in one direction with this command.
2. Click on ‘Top Plane’ in the FeatureManager.
3. Set the force to ‘6000 N’ (Newton).
4. Check ‘Flip Direction’ in order to let the pink arrows point downward.
5. Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>54</td>
<td>You can add more forces in you like, but we will not do so in this example. Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>55</td>
<td>The calculation can now be made.
Click on Next.</td>
<td>
</td>
</tr>
</table>

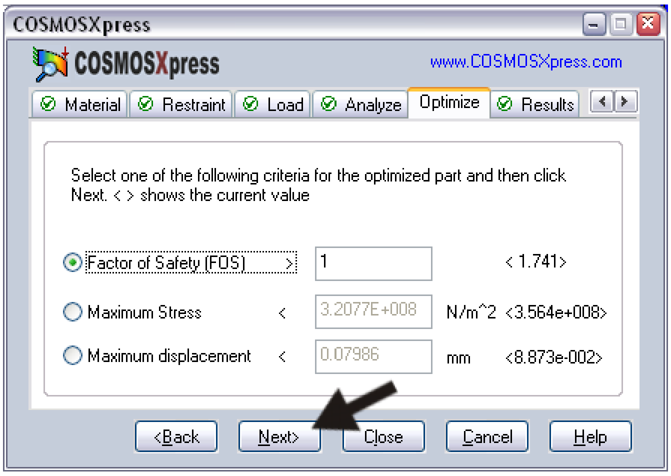
56	Click on 'Run'.	
57	The result of the analysis is that the lowest factor of safety is 1.7. The part is strong enough (read the tip below). Do you want to see the weak spots? 1. Set the FOS value to '3' (as an example). 2. Click on 'Show me'. You will see the weak spots in red now.	
	Tip!	The factor of safety (FOS) is a number calculated by COSMOS. When the FOS value is less than 1, the part will collapse when the given forces are applied. When the FOS value is more than 1, the model is strong enough, maybe even too strong.

<table>
<tr>
<td>58</td>
<td>Because the calculated FOS value is 1.7, the construction of the model is obviously too heavy.

You can now decide to optimize the design by setting the FOS value to exactly '1'.

1. Click on Yes.

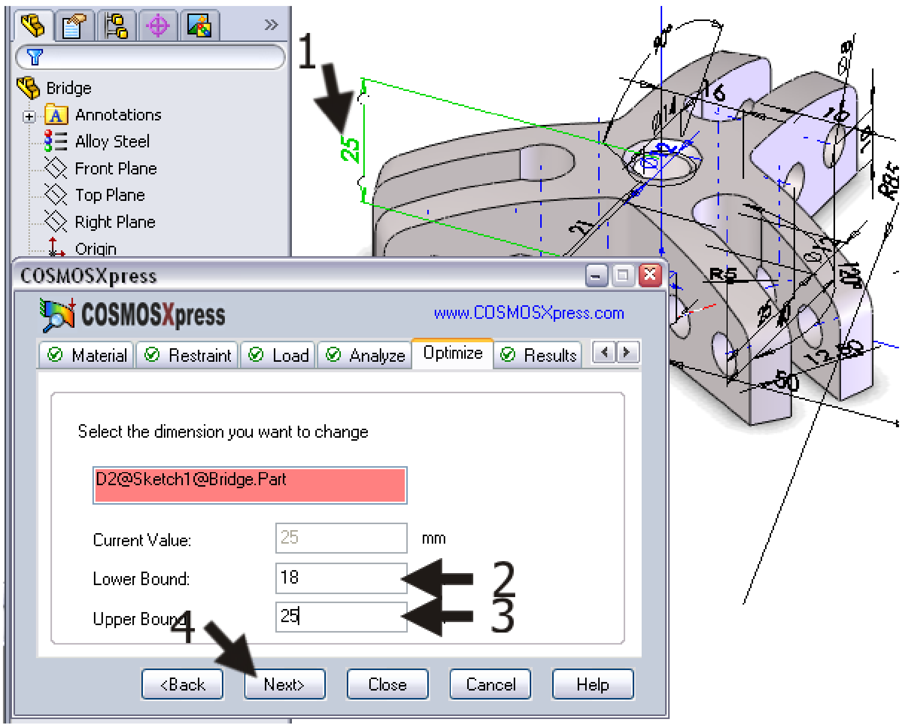
2. Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>59</td>
<td>We will alter a dimension, so the FOS value will decrease to 1.

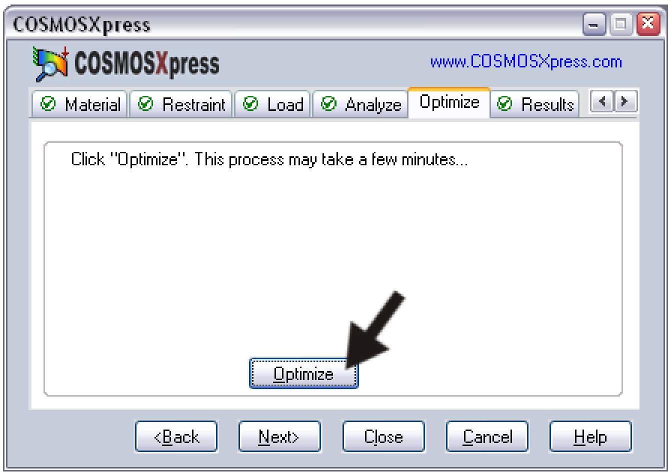
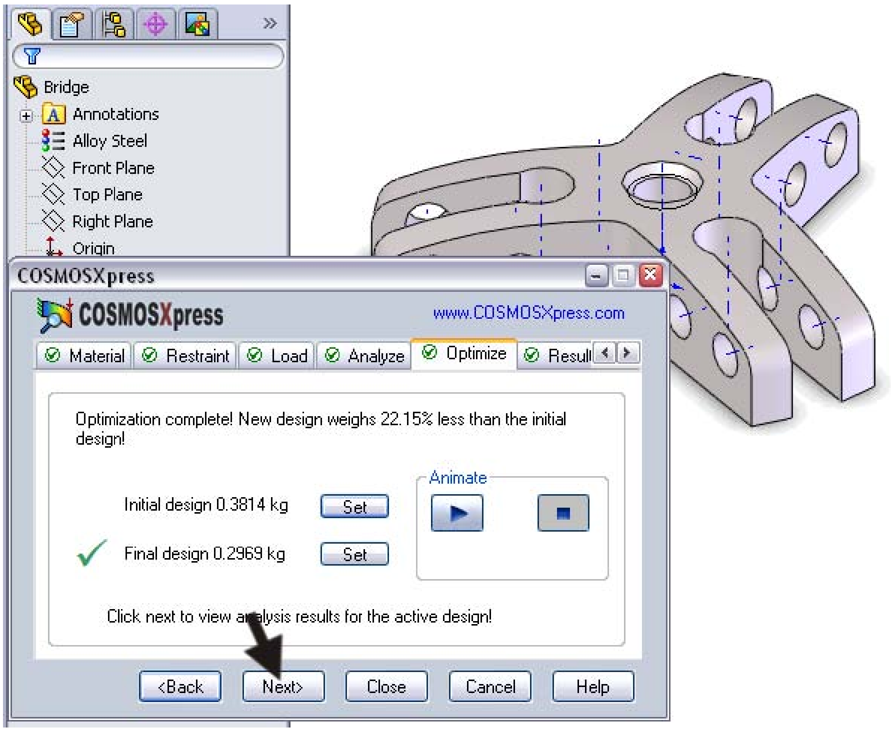
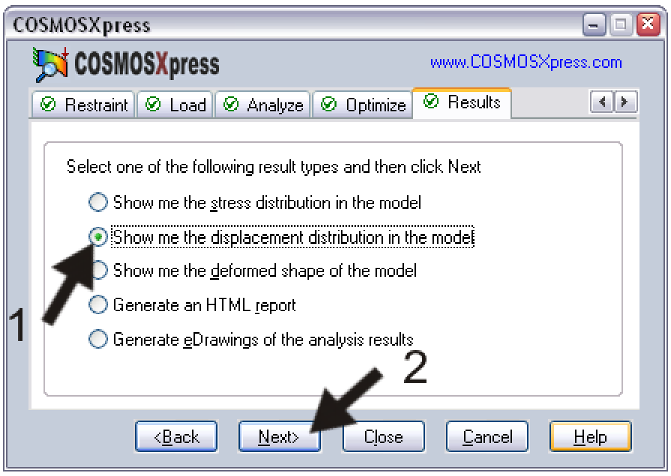
Click on Next.</td>
<td>
</td>
</tr>
<tr>
<td>60</td>
<td>All dimensions are visible now.

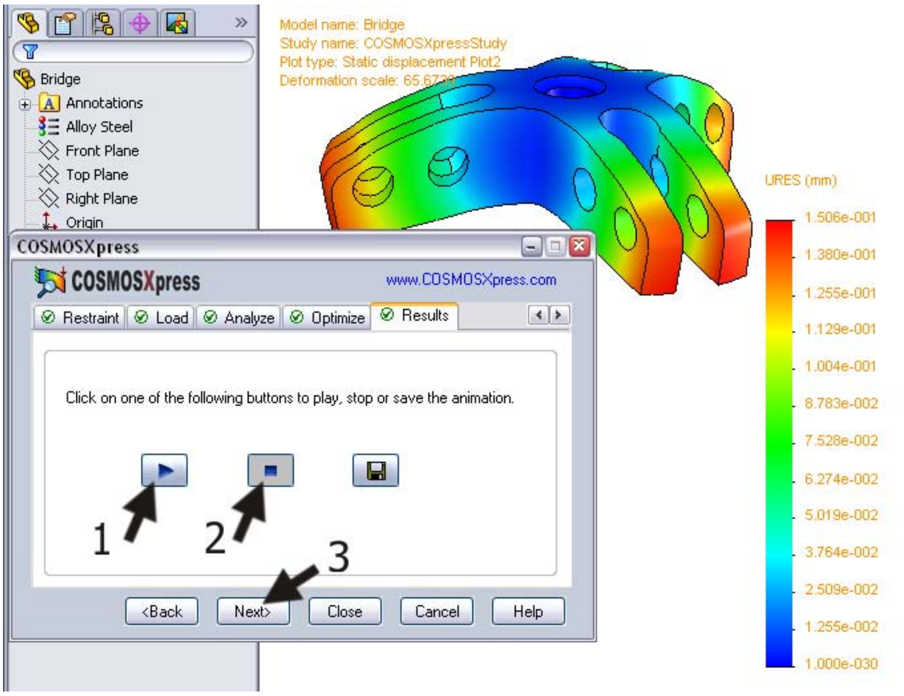
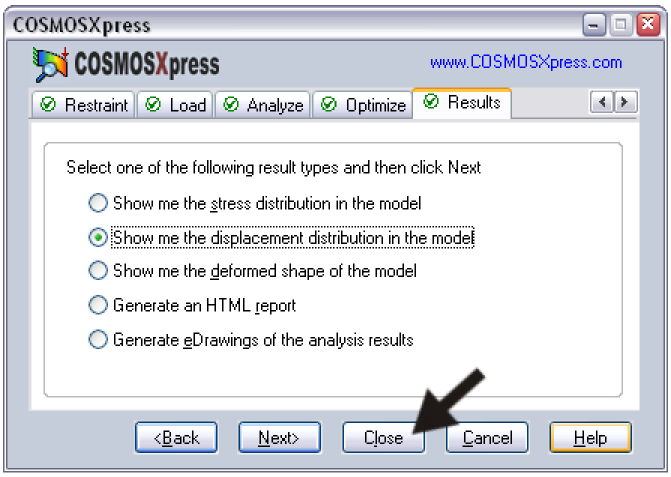
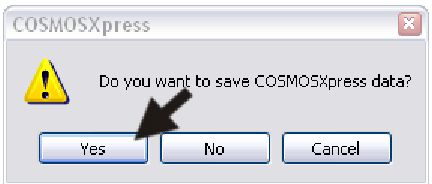
1. Select the dimension of 25mm that indicates the height of the model. Make sure to select the right dimension! In the pink selection field in COSMOSXpress you can see the selected dimension is extracted from 'sketch1' (the first sketch you have made in this part).

2. Set the minimal height to '18mm'.

3. Set the maximum height to '25mm'.

4. Click on Next.</td>
<td>
</td>
</tr>
</table>

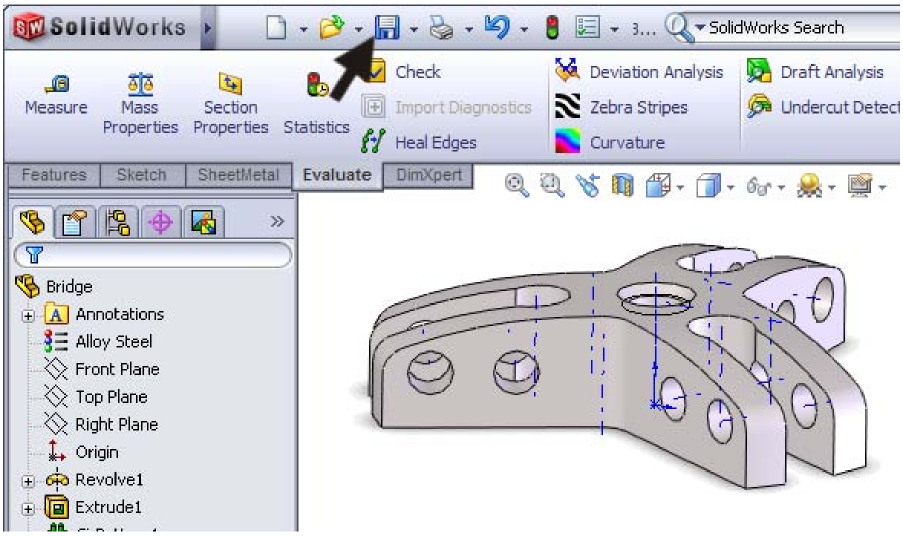
61	Click on 'Optimize'.	
62	COSMOSXpress has calculated that the model can be reduced in height. The weight has reduced by 22%, from 381 grams to 297 grams. Click on Next.	
63	You can now see the results of the calculation. The distortion during the application of the force is clear now. 1. Click on 'Show me the displacement distribution in the model'. 2. Click on Next.	

64	You can now see how the model distorts (exaggerated display) under the influence of the force. 1. Click on Play to see an animation of the distortion. 2. Click on Stop to stop the animation. You can save the animation in a separate file if you like. 3. Click on Next to go on.	
65	You will now return to the screen from step 68. You can try other options if you like. Click on Close when ready.	
	You can now save the data that was generated by COSMOSXpress.	

67

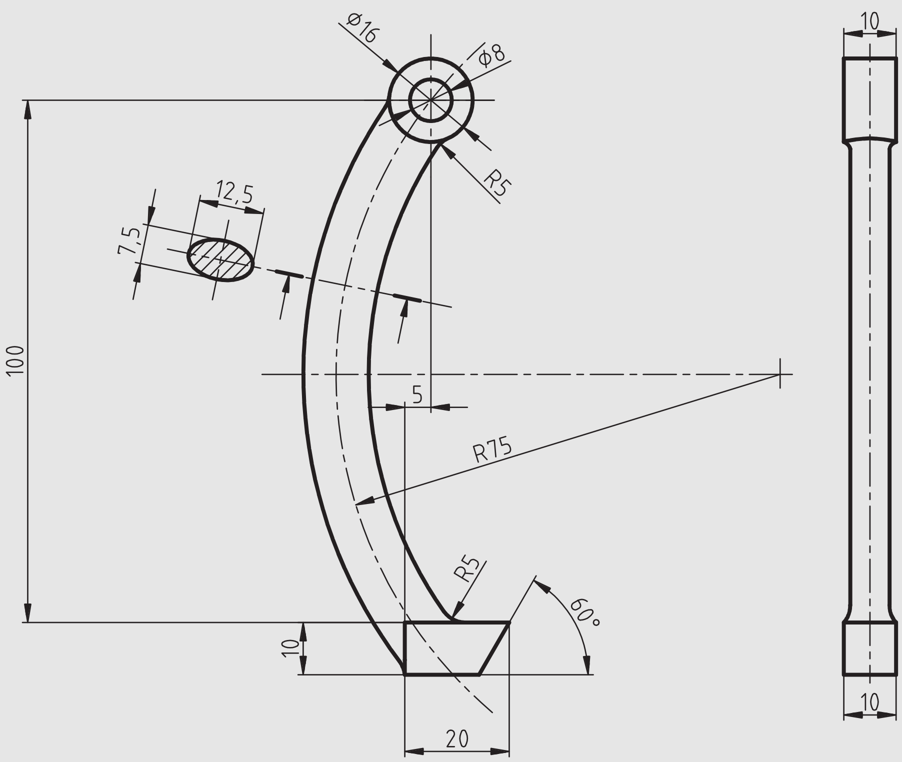
Save the changes to the file.

Click on Save in the Standard toolbar.



Work plan

The next part we will make is one of the arms. In the drawing below the part is already completed.



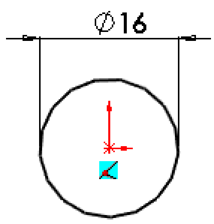
We will build this model by shaping the upper circle and lower part of the finger and will add the arm as a sweep later.

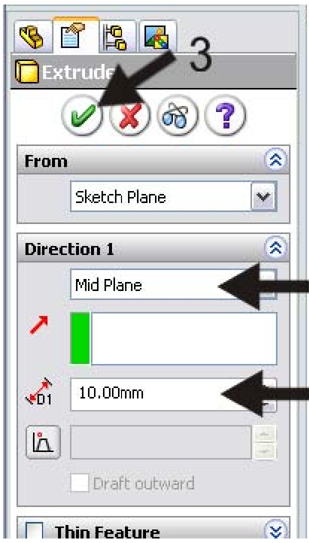
68

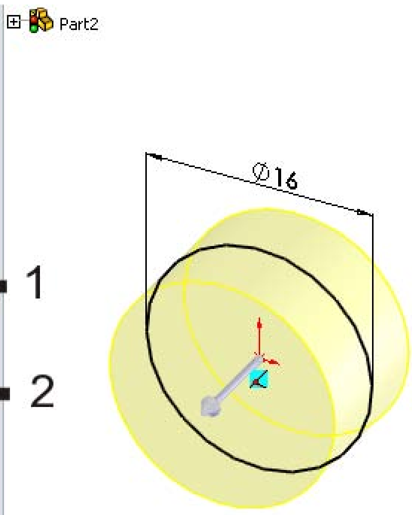
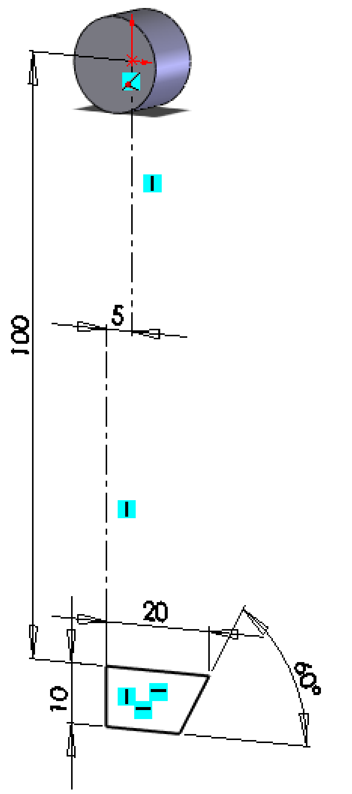
Open a new part.

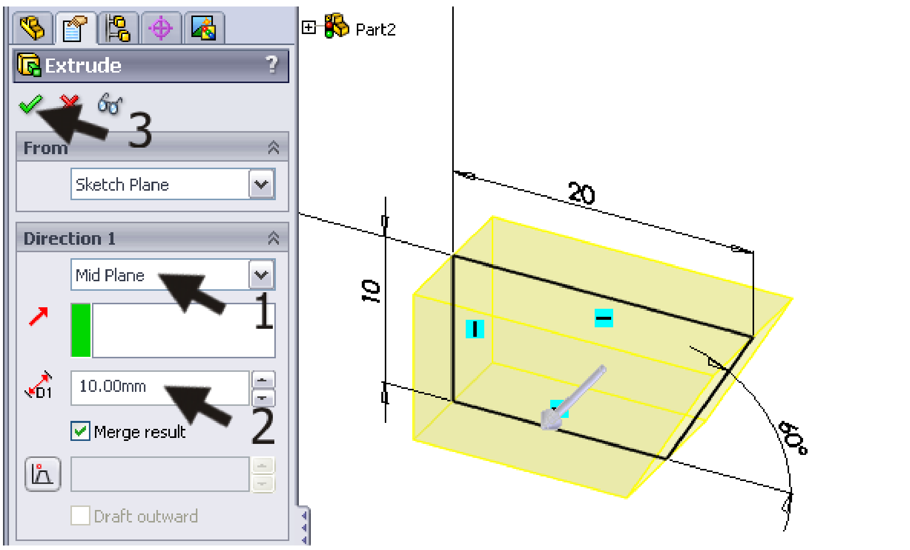
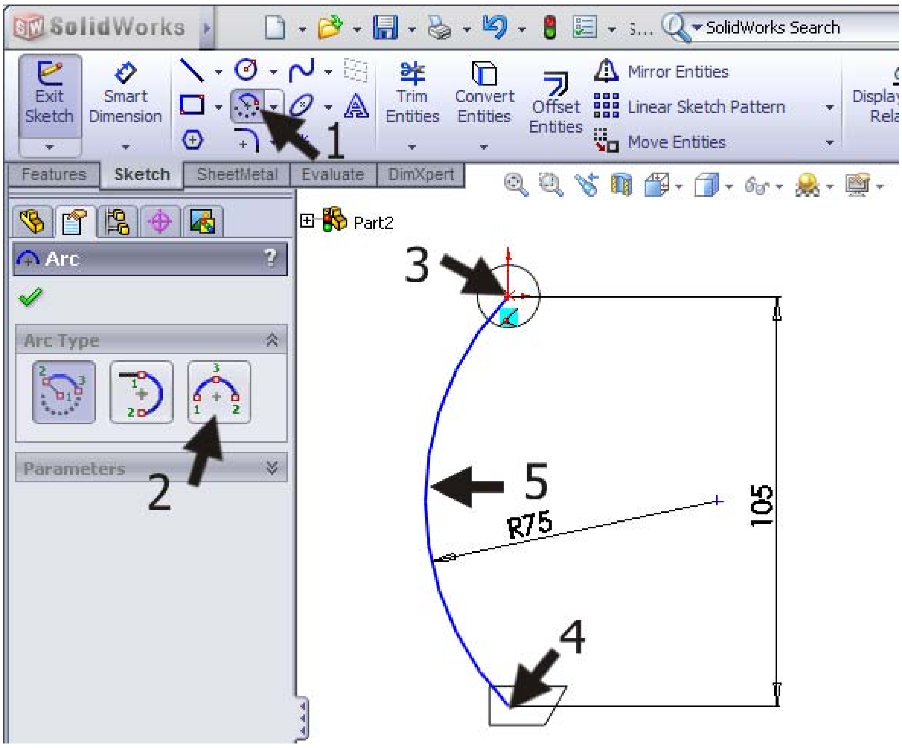
Start a sketch on the Front Plane.

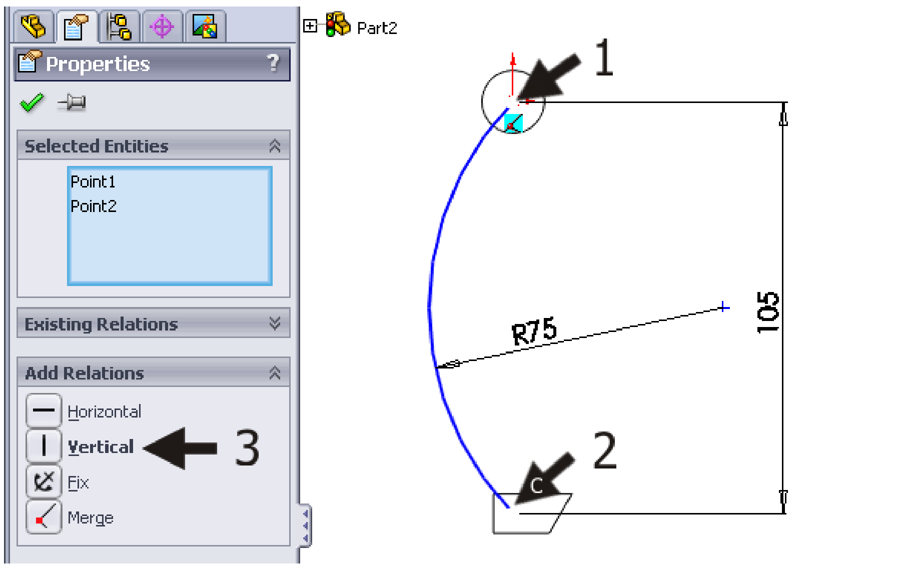
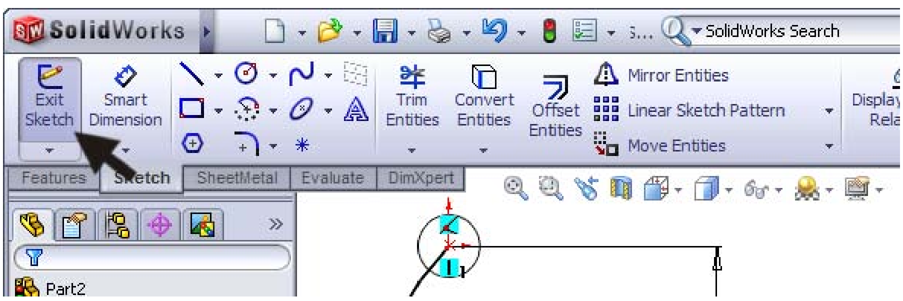
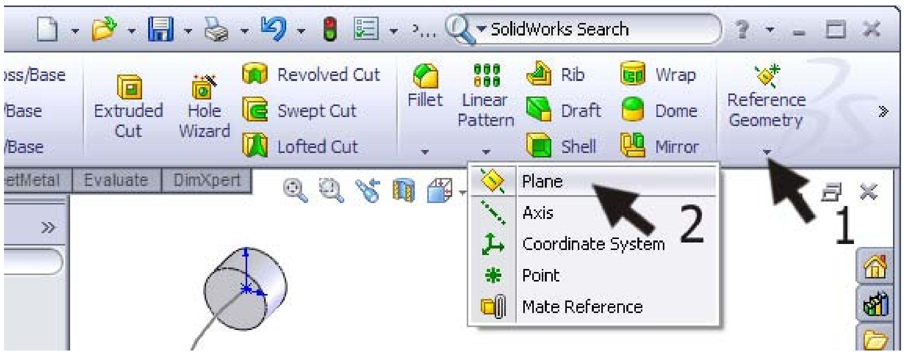
Draw a circle with a diameter of 16mm, with the midpoint above the origin.

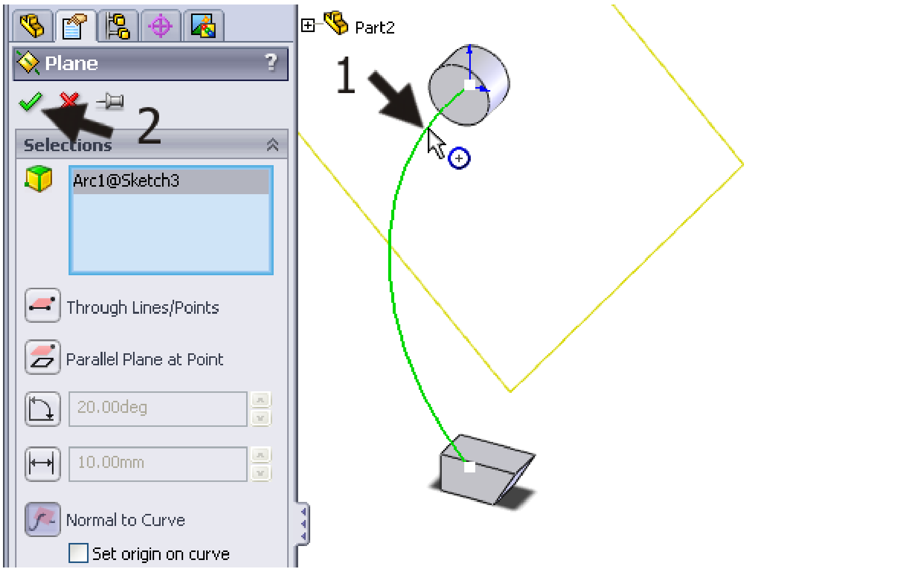
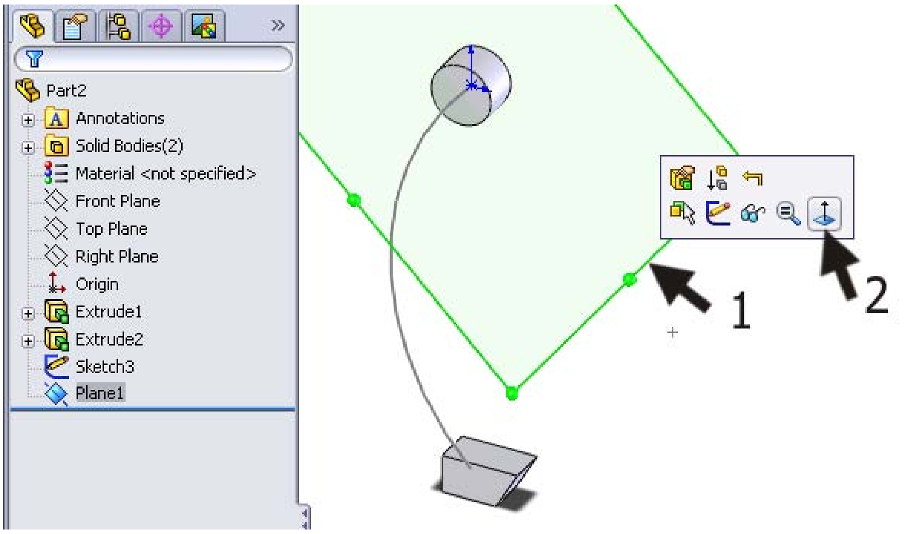
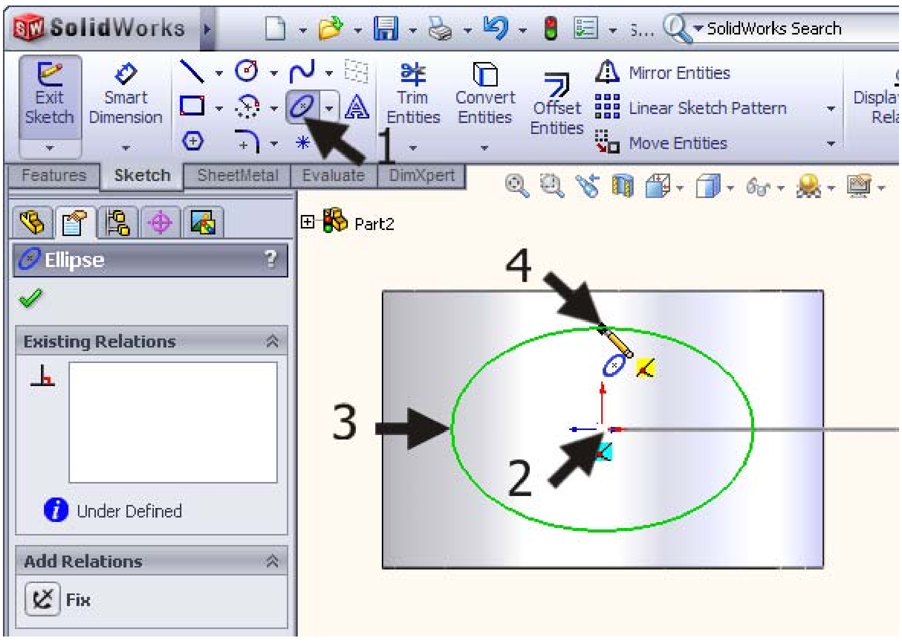


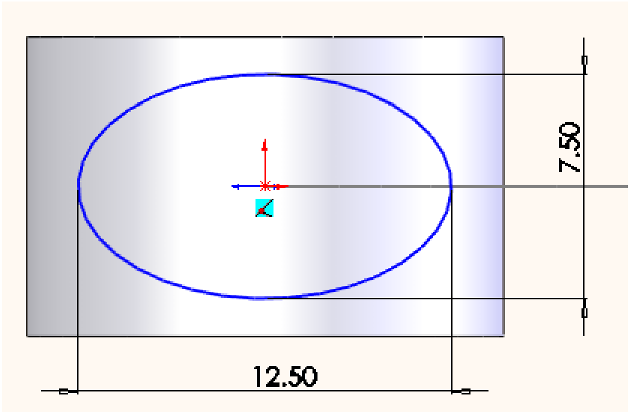
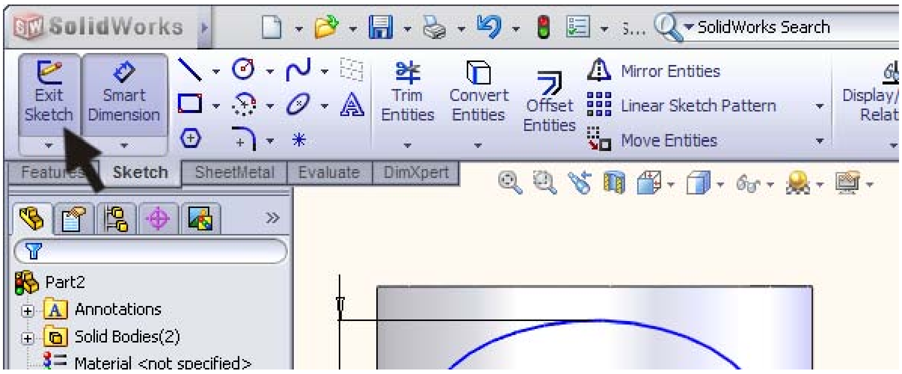
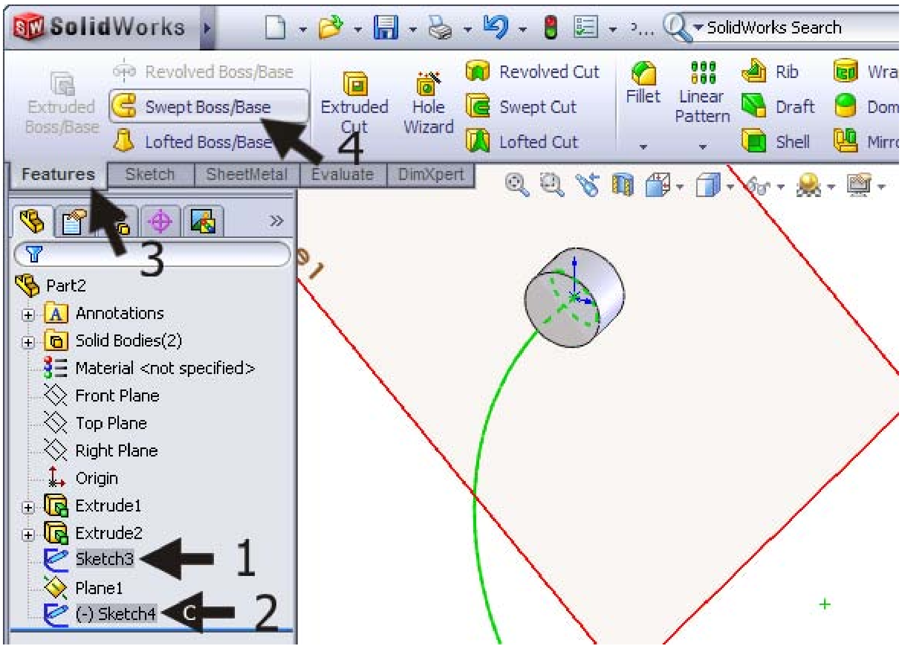
<table>
<tr>
<td>69</td>
<td>Make an extrusion from this circle:
1. Select the option ‘Mid Plane’ in the Property-Manager.
2. Set the thickness to ‘10mm’.
3. Click on OK.</td>
<td>

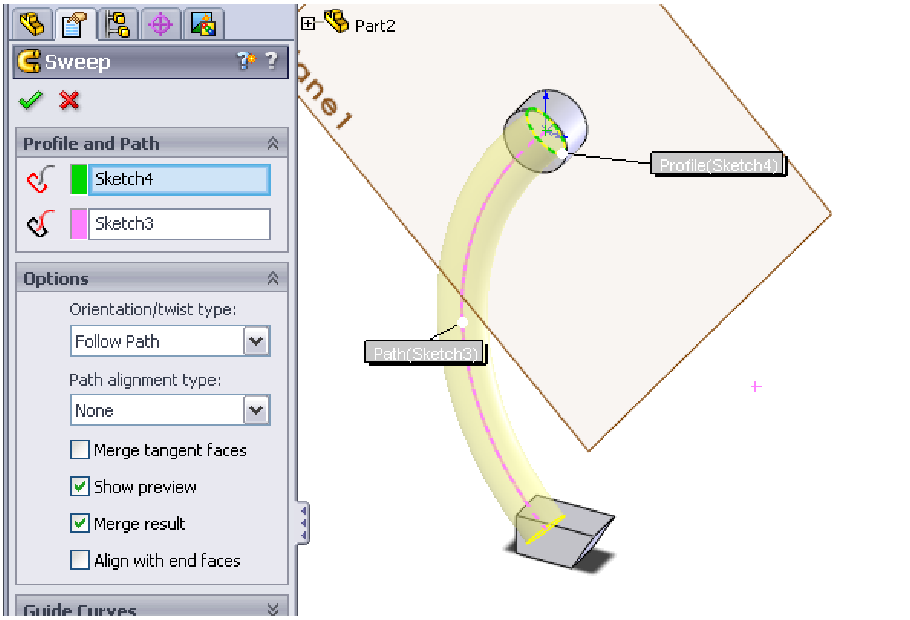
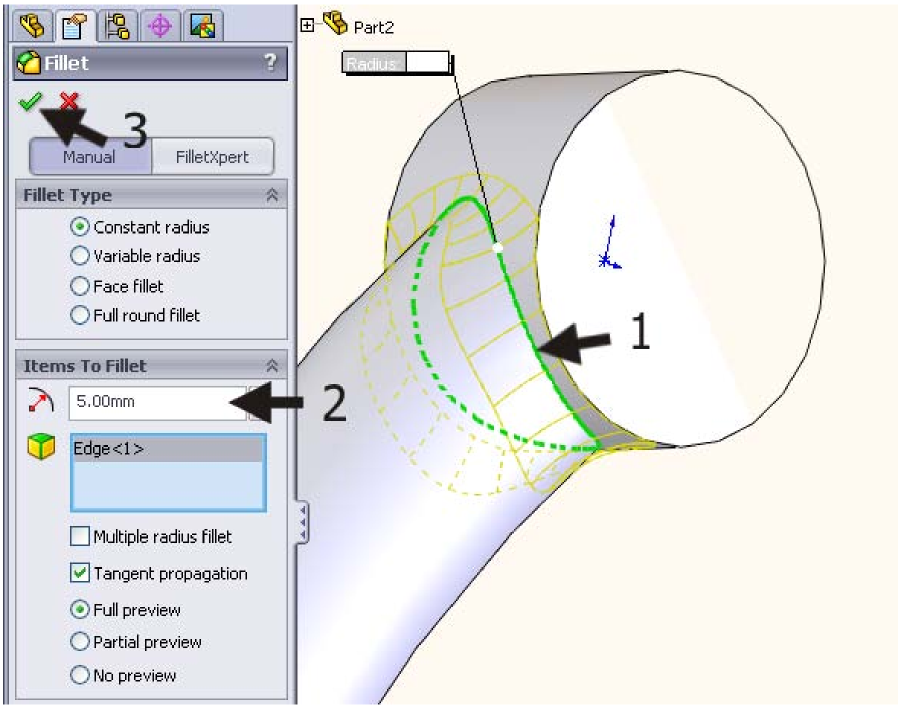

</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>We have not used the Mid Plane option before. This tool is very convenient when you want to build a symmetrical model. The sketch will extruded equally wide in two directions.</td>
</tr>
<tr>
<td>70</td>
<td>Select the Front Plane again and make the sketch similar to the drawing on the right.</td>
<td>
</td>
</tr>
</table>

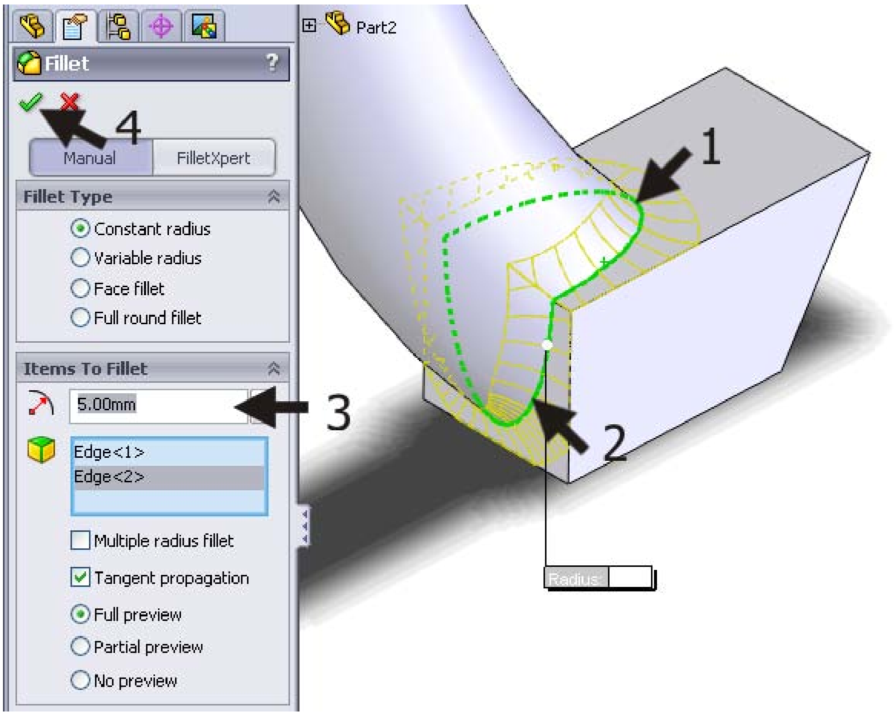
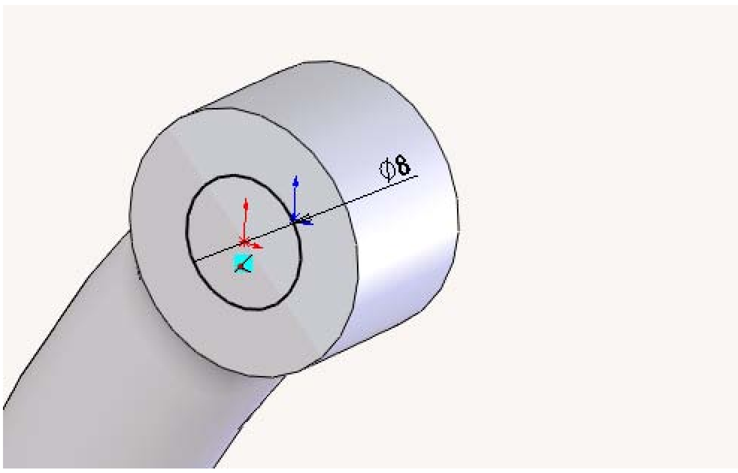
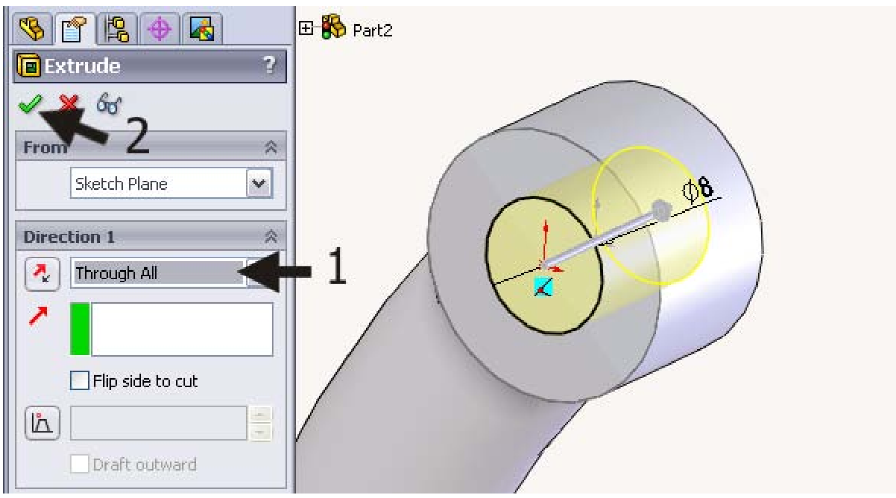
71	Make an extrusion from this sketch. 1. Use the option 'Mid Plane' again. 2. Set the thickness to '10mm'. 3. Click on OK.	
72	We will create a sweep now. A sweep is a feature in which you extrude a sketch next to another sketch. So, we have to make two sketches first. Select the Front Plane and make a new sketch on it. 1. Click on Arc in the CommandManager. 2. Select 3-Point Arc in the PropertyManager. 3. Click on the origin to set the starting point. 4. Click at the point as illustrated here to set the end of the arc. Its position does not have to be accurate at this point. 5. Click at the third point as illustrated here. Again, accuracy is not required. Add two sizes as illustrated. It does not matter if the arc is not properly aligned at this point.	

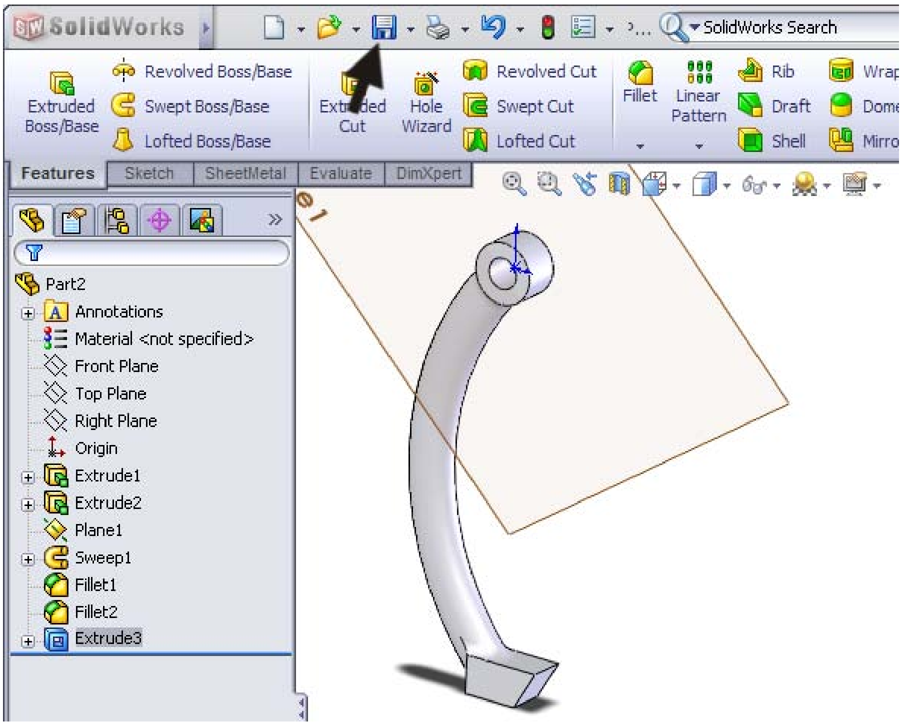
<table>
<tr>
<td>73</td>
<td>1. Select the upper end of the arc.
2. Select the bottom end of the arc too (use the <Ctrl> key).
3. Click on ‘Vertical’ in the PropertyManager.</td>
<td>
</td>
</tr>
<tr>
<td>74</td>
<td>We will use this sketch later on.
Click on ‘Exit Sketch’ in the CommandManager to close the sketch.</td>
<td>
</td>
</tr>
<tr>
<td>75</td>
<td>The second sketch is made at a right angle to the end of the first sketch. For this we need to create an auxiliary plane first.
1. Click on the ‘Features’ tab in the CommandManager.
2. Click on ‘Reference Geometry’.
3. Click on ‘Plane’.</td>
<td>
</td>
</tr>
</table>

76	1. Click on the upper end of the arc that you drew before. The auxiliary plane will be positioned at a right angle to the end of the arc. 2. Click on OK.	
77	Rotate the model so you will have a clear view of the plane you just created. 1. Click on the last mentioned plane. 2. Click on Normal To in the menu that appears.	
78	Zoom in on the origin, and draw an ellipse: 1. Click on Ellipse in the CommandManager. 2. Click on the origin. 3. Click on a horizontal position besides the origin to set the long axis of the ellipse. 4. Click straight above the origin to set the short axis. The exact dimensions do not matter yet.	

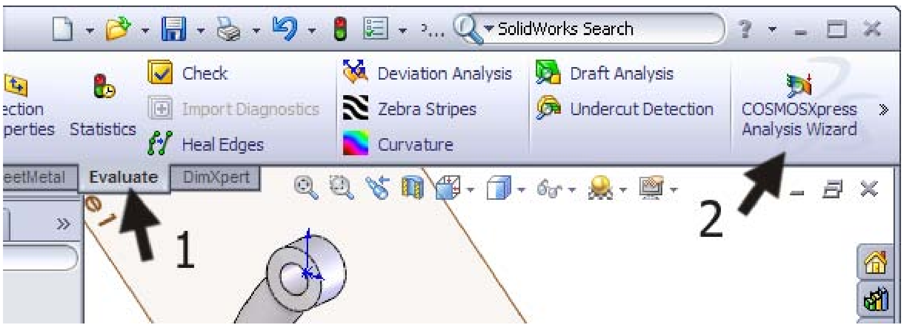
79	Set the dimensions of the two axes as illustrated on the right with Smart Dimension.	
80	This sketch is now done, so Click on 'Exit Sketch' in the CommandManager.	
81	We will combine the two sketches to a sweep. 1. Select the sketch with the arc in the FeatureManager. 2. Select the sketch with the ellipse too (use the <Ctrl> key) 3. Click on 'Features' in the CommandManager. 4. Click on 'Swept Boss/Base'.	

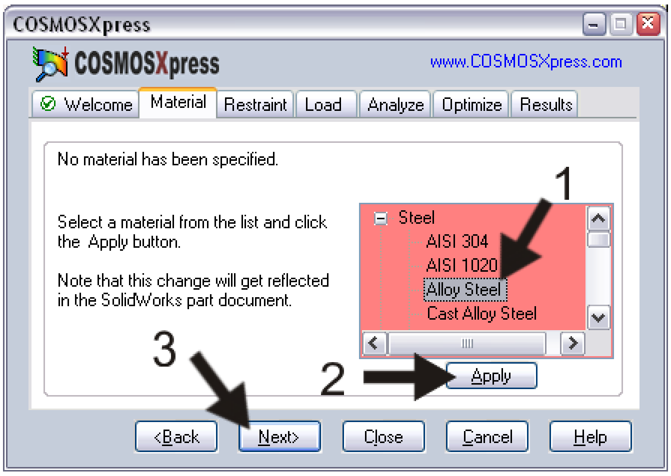
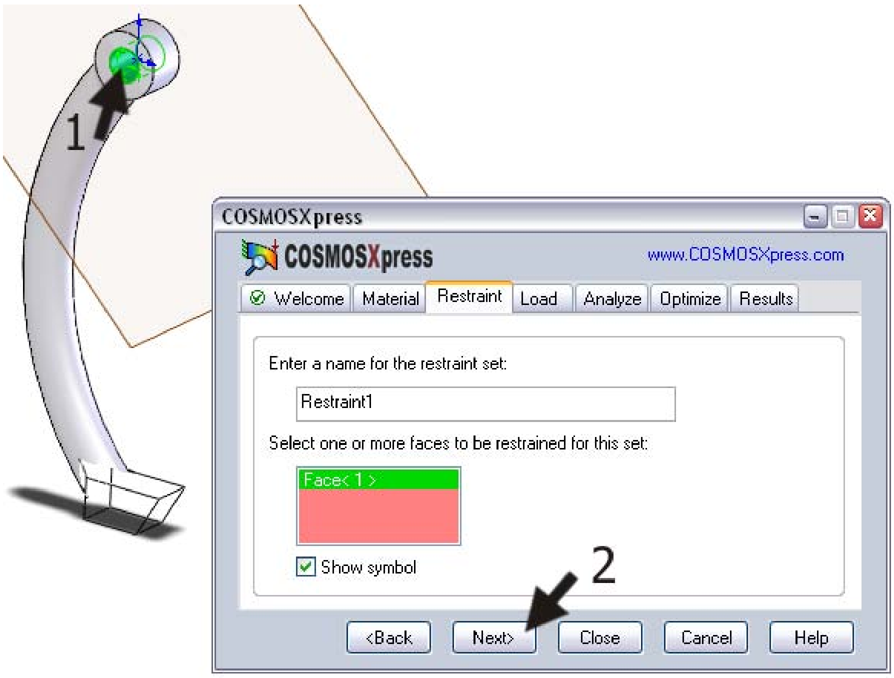
82	You do not have to set any other features in the PropertyManager. Click on OK.	
83	The connection between the arm and the top and bottom parts has to be finished. Click on 'Fillet' in the CommandManager. 1. Select the cutting edge between the arm and the upper circle. 2. Set the radius to '5 mm' in the PropertyManager. 3. Click on OK.	

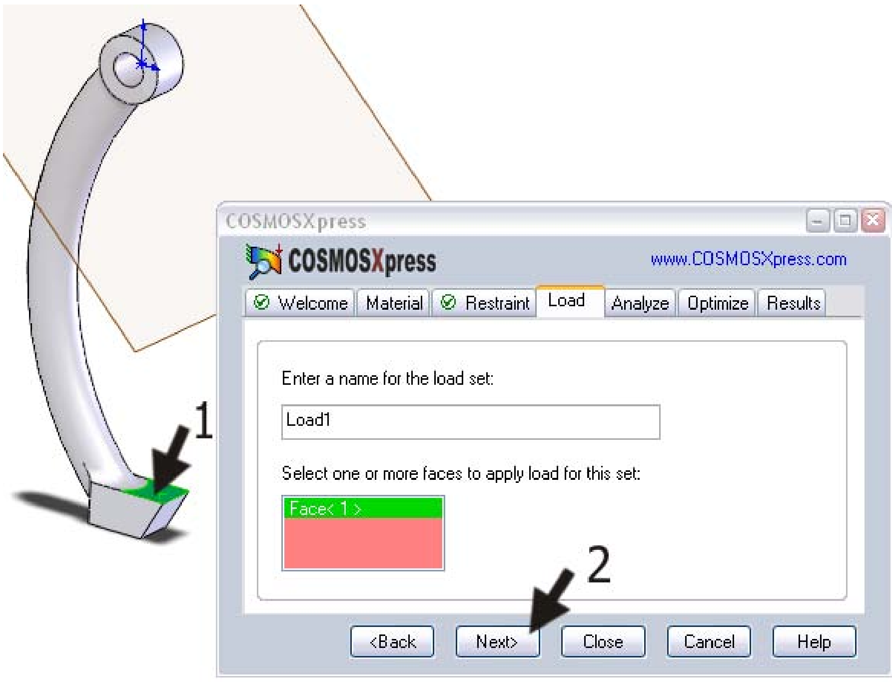
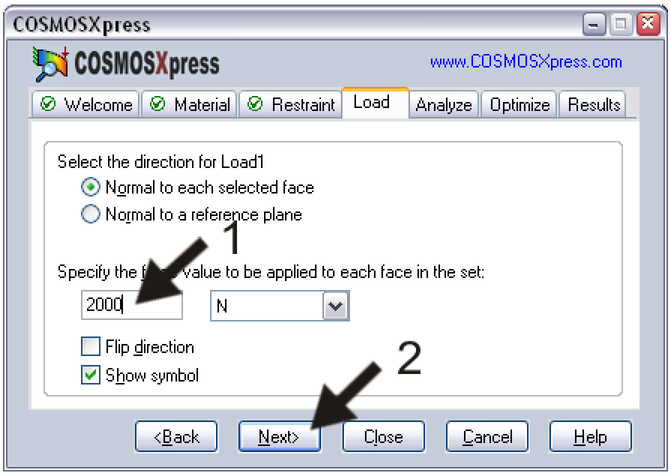
84	Next, round off the connection at the bottom. Click on 'Fillet' in the CommandManager. Select both cutting lines now. The radius is also set to '5mm'.	
86	Finally, we have to put a hole in the upper circle to accommodate a bolt. Make the sketch as shown on the right.	
87	Make an 'Extruded Cut' from this sketch. 1. Set the option 'Through All' to go all the way through the material. 2. Click on OK.	

<table>
<tr>
<td>88</td>
<td>Save the file as:
Arm.SLDPRT.</td>
<td>
</td>
</tr>
<tr>
<td>89</td>
<td>Of course, we also want to know if the arm is strong enough for our purpose. The complete tool should be able to pull 600kg, or about 200kg (=2000N) per arm.

Click on the tab ‘Evaluate’ in the CommandManager.
Click on ‘COSMOSXpress Analysis Wizard’.

Run the wizard by clicking Next every time. We will only display and describe the steps that need input.</td>
<td>
</td>
</tr>
</table>

90	Define the desired materi-al: 1. Select 'Alloy Steel'. 1. Click on 'Apply' (do not forget!). 2. Click on Next.	
91	Define the 'Restraint' (this is the fixed plane): 1. Select the hole where the bolt goes through. 2. Click on Next.	

92	Set the 'Load', 1. Select the plane in the model as illustrated on the right. 2. Click on Next.	
93	Set the force to '2000N'. The pink arrows in the model must point downward. When they do not, click on 'Flip direction'.	

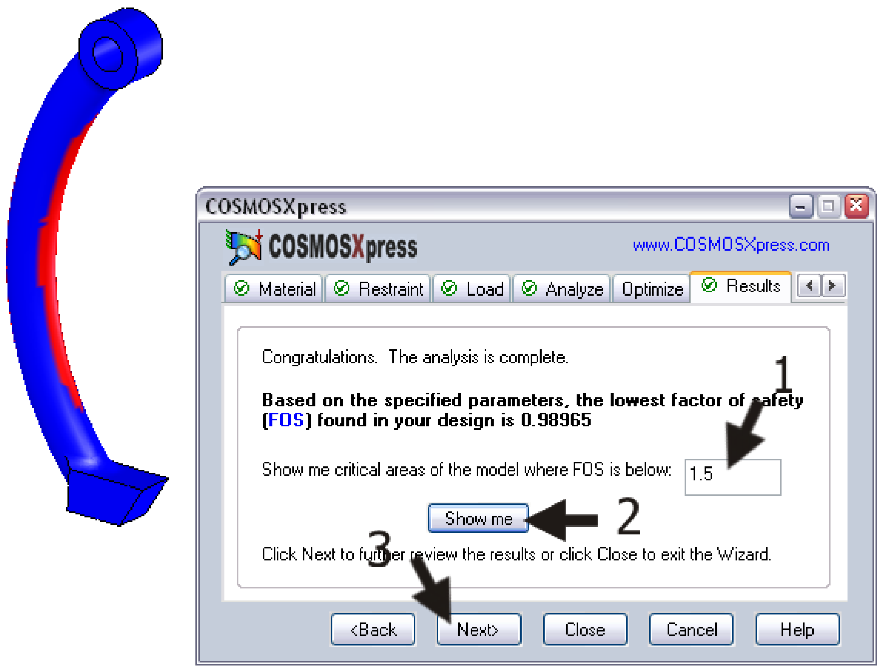
94

After the analysis is done, the FOS value turns out to be 0.98. So this is just not enough!

1. Fill in '1.5' in the menu.
2. Click on 'Show me'.

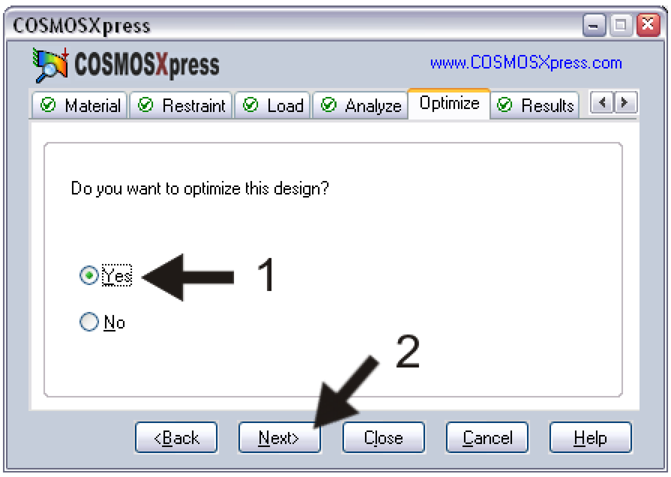
You can now see clearly where the strain is the highest: on the inside of the arm.

3. Click on Next.



95

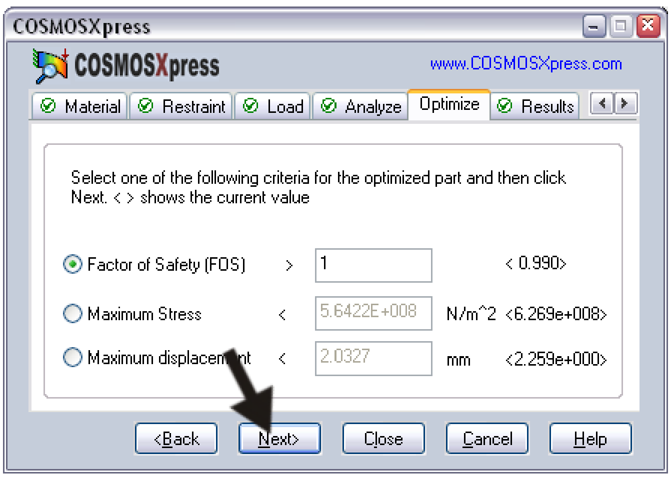
We can strengthen the part by decreasing the curve of the arm, so the radius will increase.

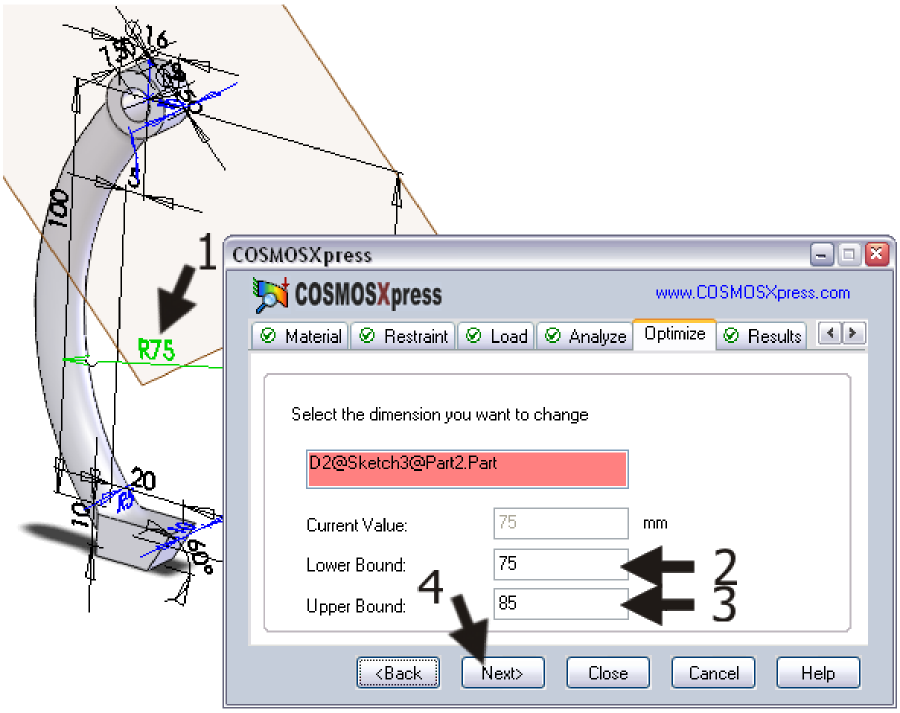


96

We improve the model to get a FOS value of 1.

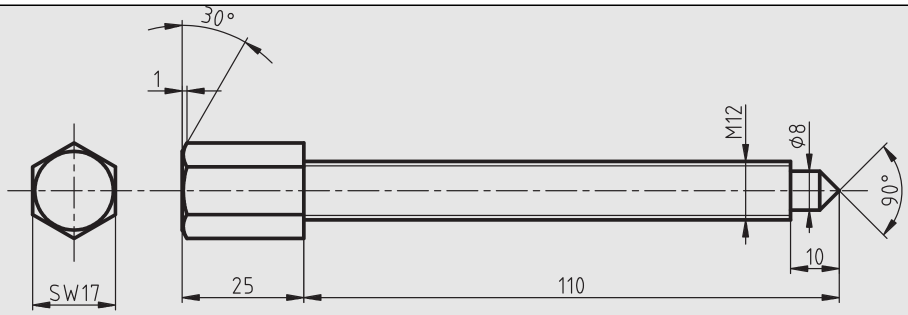
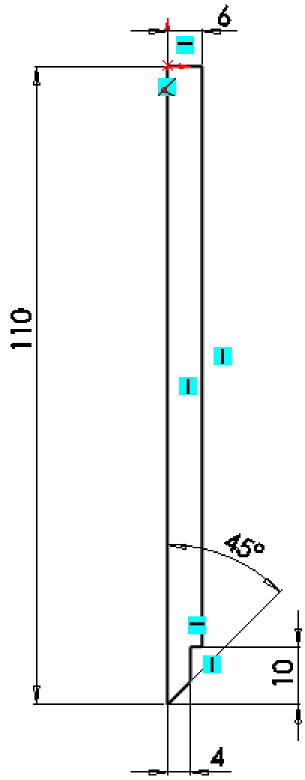
Click on Next.

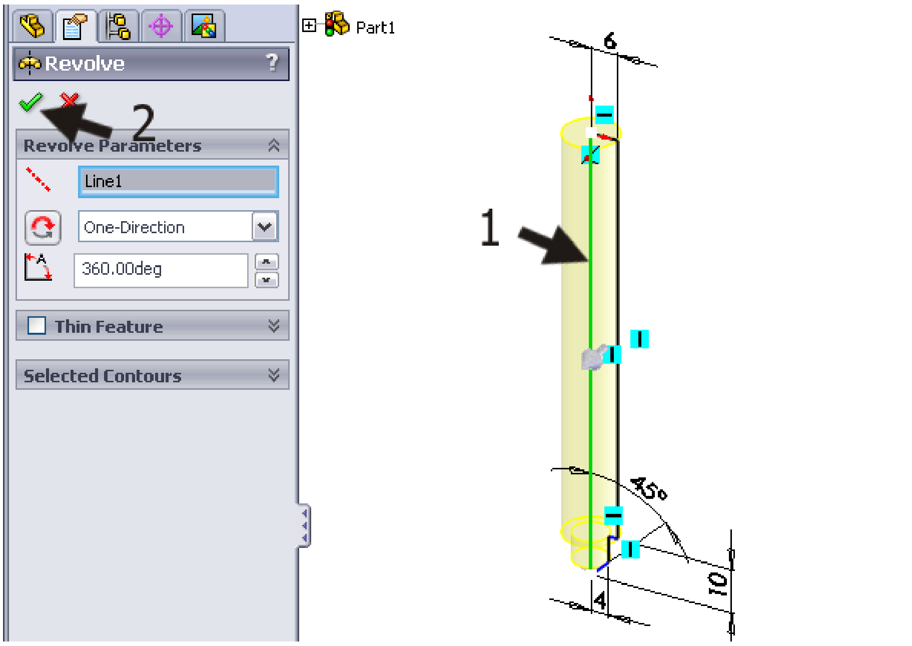
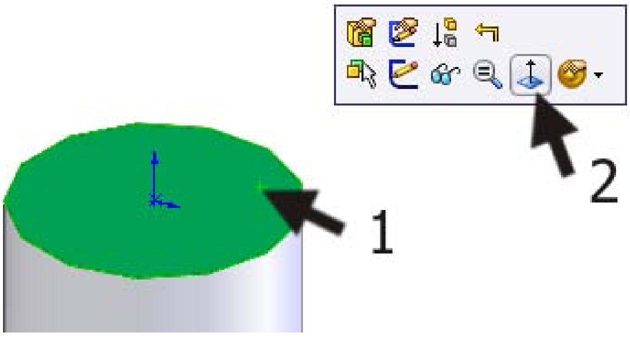


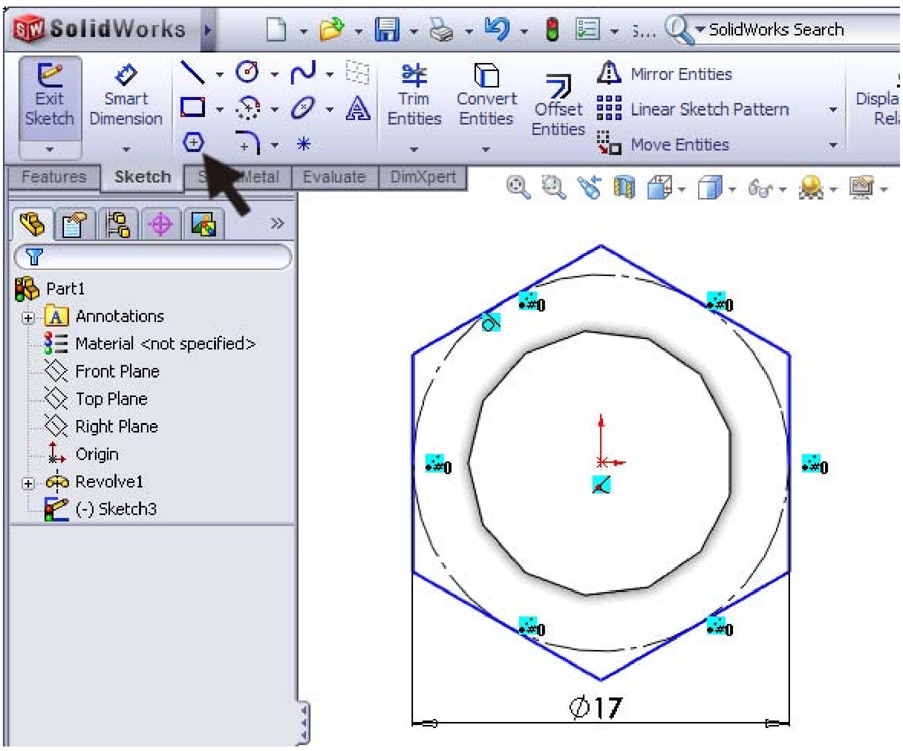
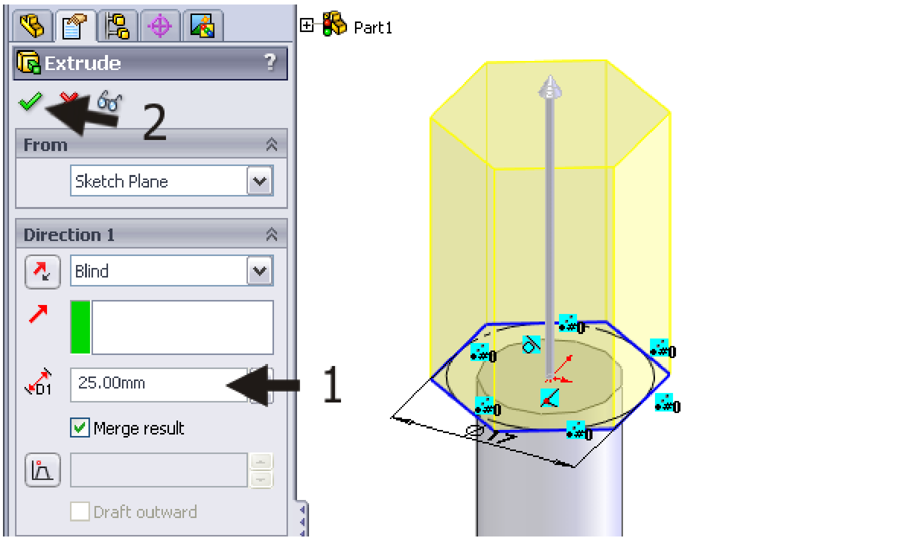
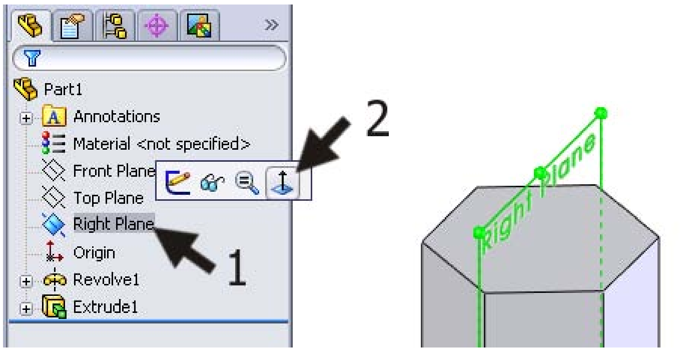
<table>
<tr>
<td>97</td>
<td>
1. Select the dimension 'R75' in the model. We will change this radius to optimize the model
2. Set a minimum value of '75'.
3. Set a maximum value of '85'.
4. Click on Next.
Pay attention: the minimum and maximum values are values that should be within a certain range. When you change a value that leads to an error, COSMOSXpress cannot use that value.
</td>
<td>


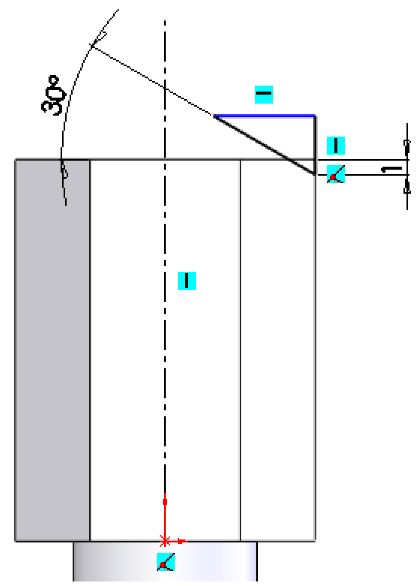
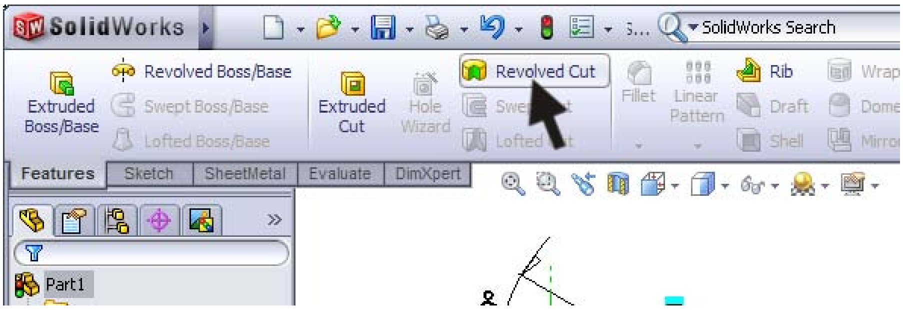
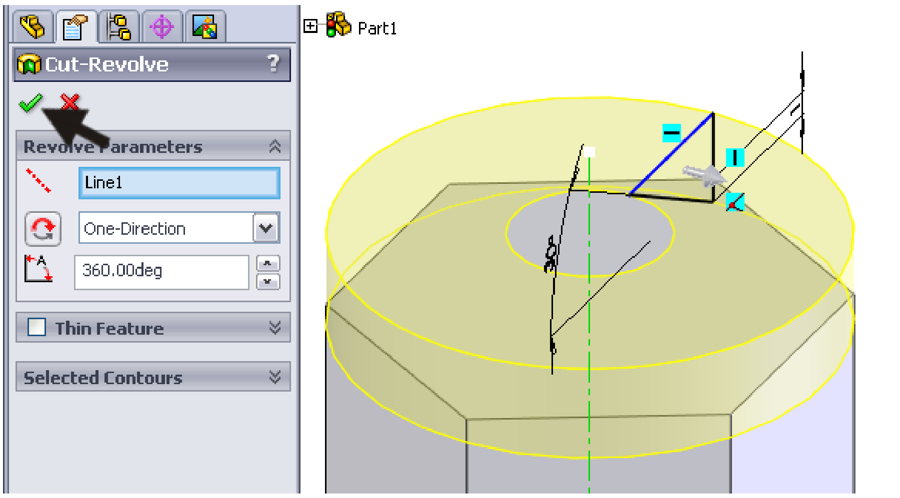
</td>
</tr>
<tr>
<td>98</td>
<td>
COSMOSXpress has now changed the dimension.
If you would like to see more data (e.g., the distortion), click on Next.
If not, end COSMOSXpress by clicking on Close.
</td>
<td>


</td>
</tr>
<tr>
<td>99</td>
<td>Save the changes to the file.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Work plan</td>
<td>The third and last part of this product is relatively simple: an extended bolt with an M12 thread. In the drawing below you can see how this part looks.</td>
</tr>
</table>

		 We will create the rod with the thread and the pointed end as a rotation form. The hexagonal part will be added to this as an extrusion.
100	Open a new part. Make the sketch as you can see on the right on the Front Plane.	

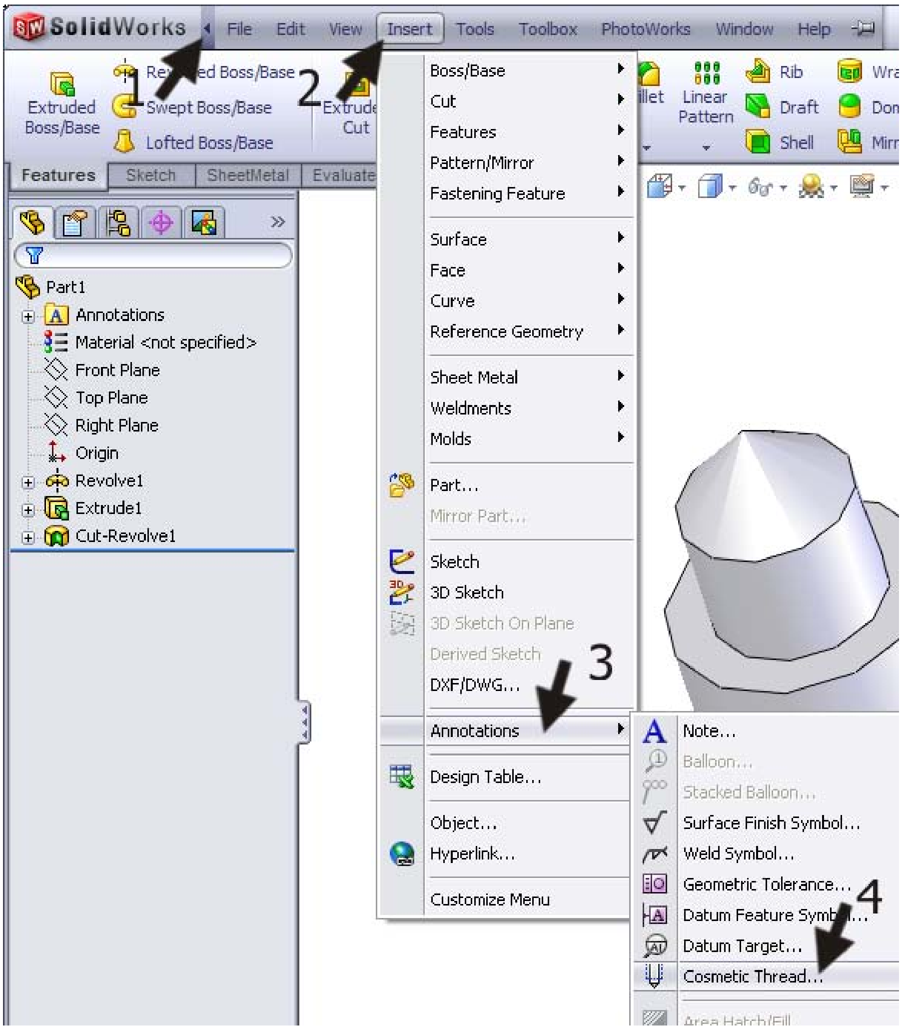
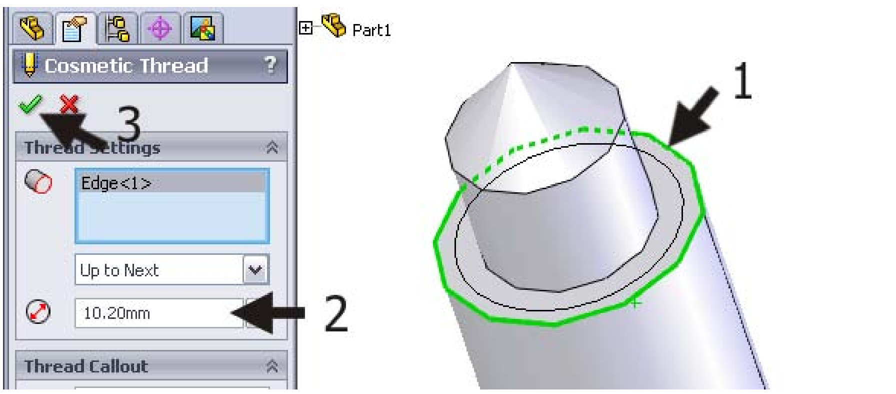
101	Make a Revolved Boss/Base from this sketch. 1. Select the line which you want to use as a rotation axis. 2. Click on OK.	
102	Select the top plane to the model. We will make the next sketch on this. Rotate the model to Normal To.	

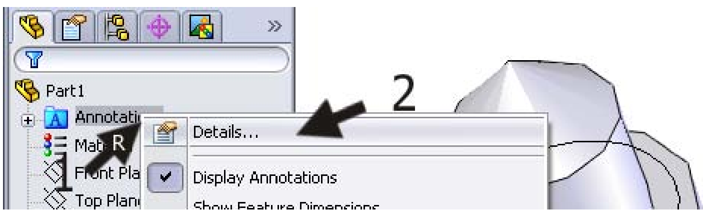
103	Click on Polygon in the CommandManager. Draw a hexagon, and set the dimensions according to the illustration on the right. Make sure that one of the vertices of the hexagon is vertically aligned directly above the origin.	
104	Make an extrusion from this sketch. 1. Set the height to '25mm'. 2. Click on OK.	
105	We have to create a sloped edge at the top of the hex-agon head. Select the 'Right Plane' in the FeatureManager, and rotate the model Normal To.	

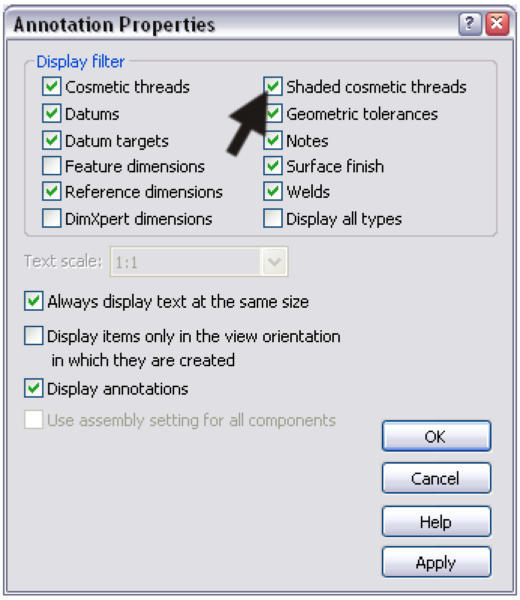
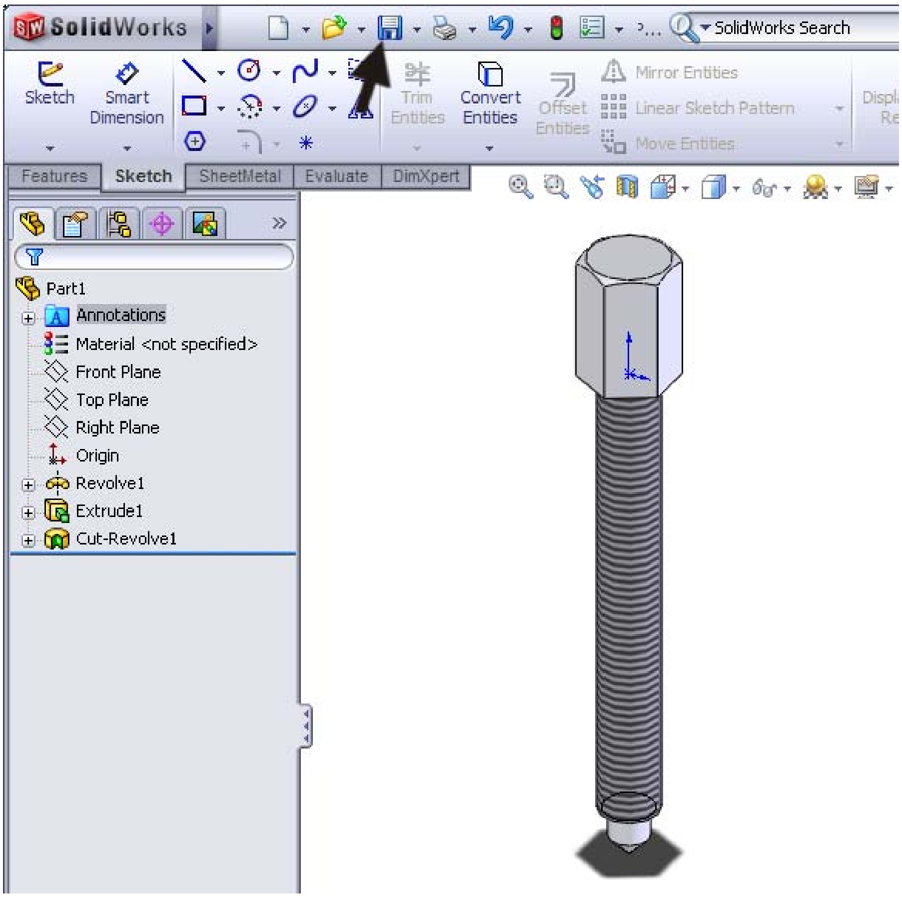
106	Make the sketch as in the illustration: Draw the centerline from the origin vertically upward. Next, draw a triangle. Add two dimensions to finish it.	
107	1. Click on the tab 'Features' in the FeatureManager. 2. Click on 'Revolved Cut'.	
108	Click on OK in the PropertyManager.	

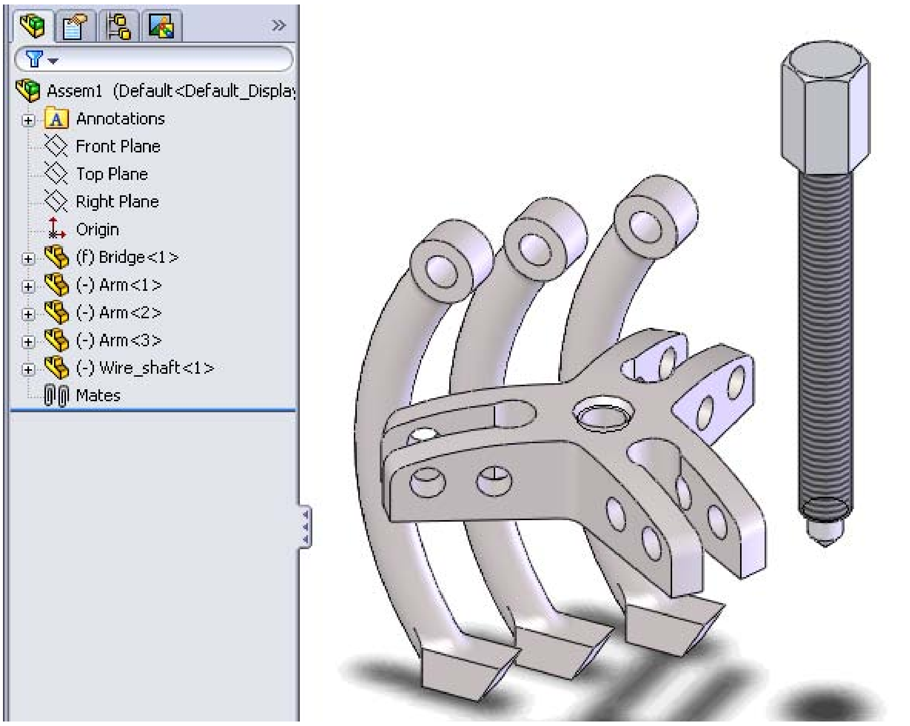
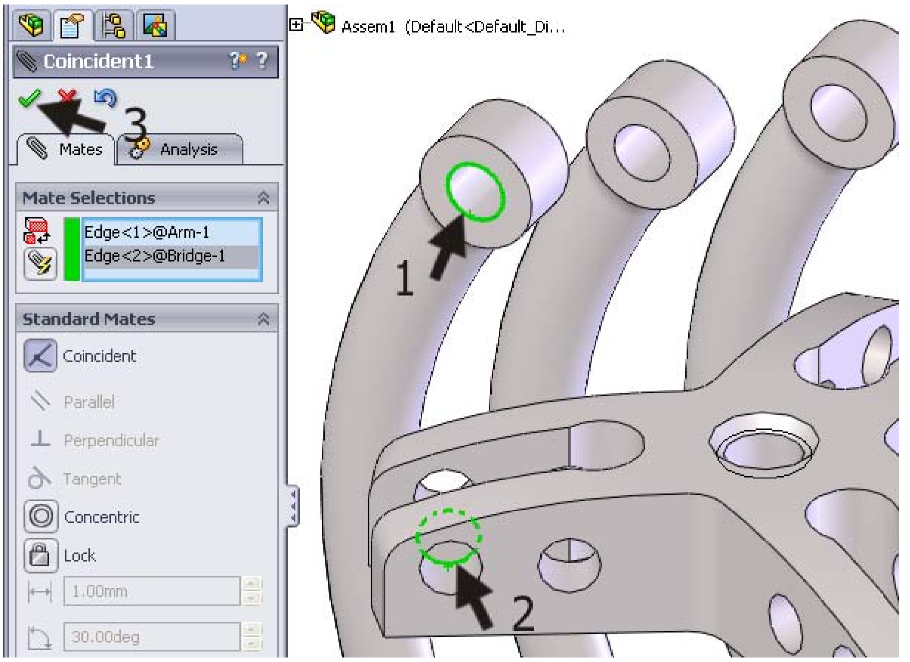
<table>
<tr>
<td>109</td>
<td>Finally, we will cut thread son the bolt.

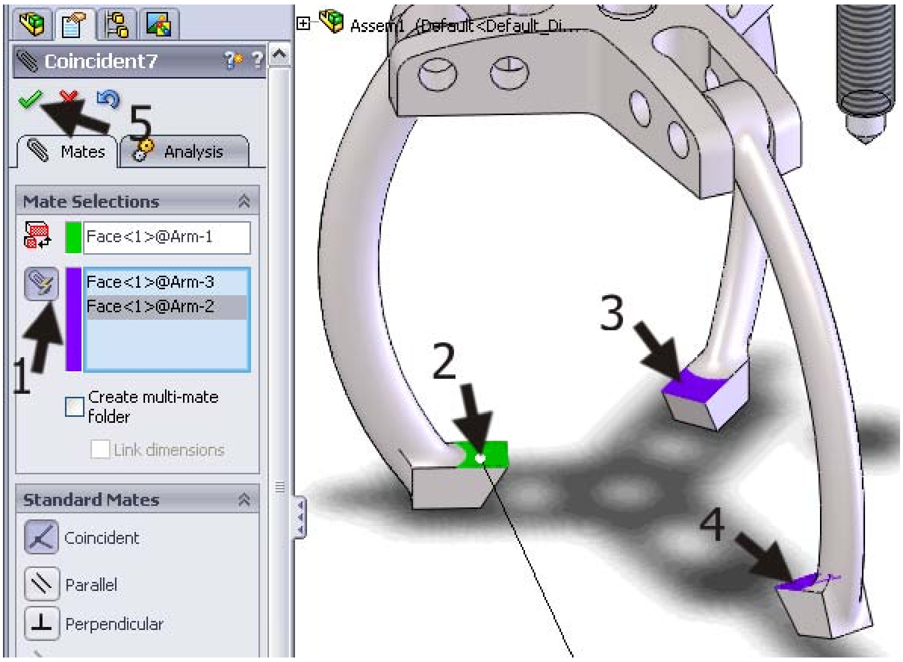
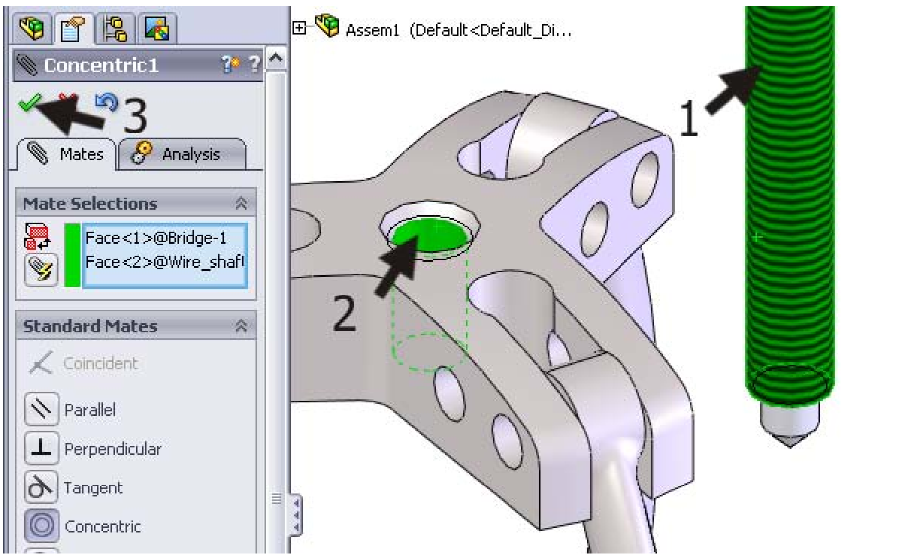
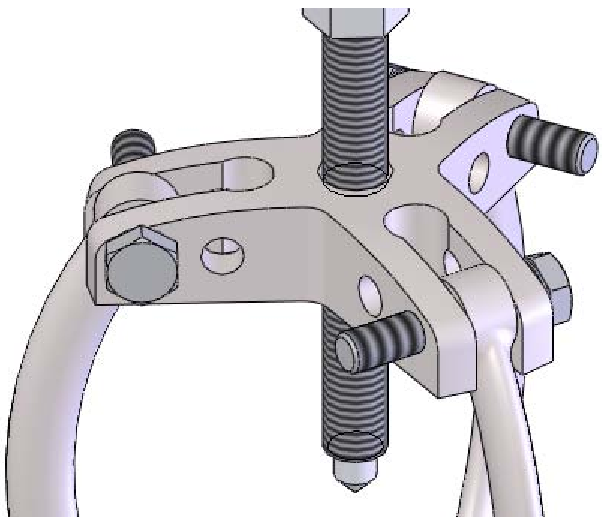
You will find the command for this in the Pull-down menus:

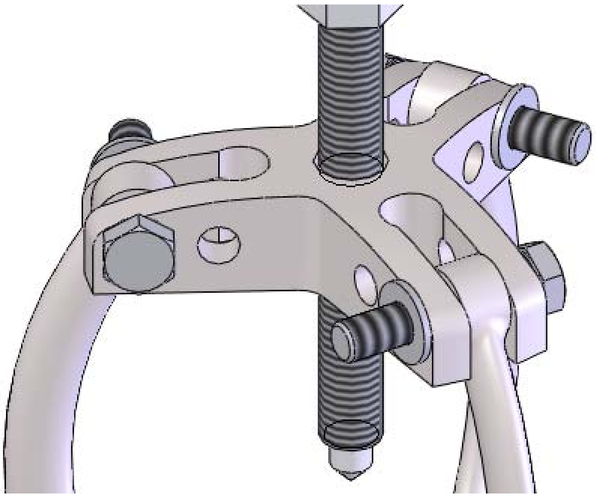
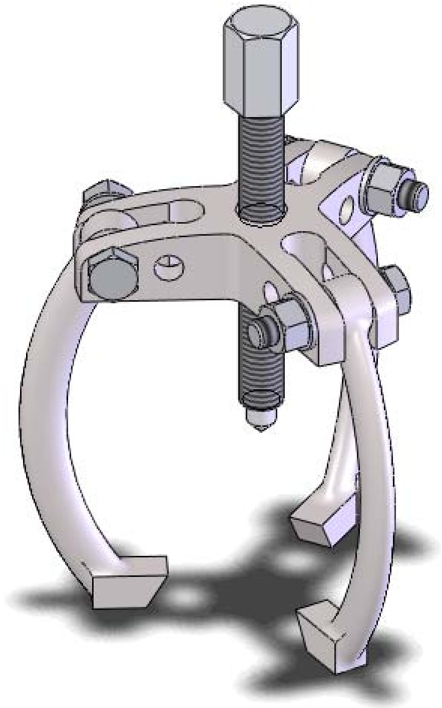
1. Open the Pull-down menus.
2. ‘Insert’.
3. ‘Annotations’.
4. ‘Cosmetic Thread’.</td>
<td>
</td>
</tr>
<tr>
<td>110</td>
<td>1. Select the edge of the plane you want to convert into thread.
2. Set the diameter to ‘10.2mm’.
3. Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>111</td>
<td>To display the thread you can:

1. Right-click on ‘Annotations’ in the FeatureManager.
2. Click on ‘Details’.</td>
<td>
</td>
</tr>
</table>

112	1. Check the option 'Shaded cosmetic threads' in the menu that appears. 2. Click on OK.	
113	This part is also now done. Save it as: wire_shaft.SLDPRT.	

114	We will assemble all parts to build a bearing puller. Open a new assembly. Put the bridge in the assembly first. Next, add the arm three times and add the wire-shaft once. Place them at random positions in the assembly.	
115	First, put the arms in the bridge. Click on 'Mates' in the CommandManager. Select the two edges as illustrated to put the first arm in its place. Next, set the two other arms in their positions in the same way. Pay attention: use the Mate alignment command ('aligned' or 'anti-aligned') to turn an arm around when necessary.	

116	To set the arms straight, we will add a few extra mates. 1. Click on Multiple Mate Mode in the Property-Manager. 2-4 Select the three top planes at the end of each arm one by one. 5 Click on OK.	
117	Finally, we have to put the bolt in position. Create a mate between the surfaces as illustrated on the right. How far to insert the shaft in the bridge is up to you.	
118	Add bolts, washers, and nuts to the assembly from the Toolbox. Find the bolts in the Tool-box by looking for 'Din > Bolts and Screws > Hex Bolts and Screws'. Select 'Hex Screw Grade AB – DIN and 24014'. Set the size: 'M8' with a length of '40'. Add this bolt to the assem-bly three times.	

119	For the washers, find 'Din > Washers > Plain Washers' in the Toolbox. Select 'Washer – Grade A – DIN125 Part1'. Select size: '8.4' (for thread 'M8'). Add this washer to the assembly three times too.	
120	Finally, we need to place the nuts. Use 'DIN > Nuts > Hex Nuts' from the Toolbox. Select 'Hex Nut Grade C – DIN and 24034'. Select size: 'M8'. Again, add this nut three times to the assembly.	
121	We have finished the assembly. Save the file as Bearing_puller.SLDASM.	
	What are the main features you have learned in this tutorial?	The most important item you have seen in this tutorial is how to use COSMOSXpress to find out if a model is strong enough to perform its designed purpose. A number of other new items include: • Creating a more complex model (the bridge) and using the 'circular pattern' command. • Using an Axis and learning another way to define an auxiliary plane. • Creating a model using a 'real' material.

		• Determining the weight and volume from a part or from the model. • Using the sweep feature • Learning it is very convenient to create outer parts first and building up the middle sections later, as in the modeling of the arm. • Working with Cosmetic Thread. After finishing this tutorial, you have learned a lot about using SolidWorks. You probably understand much more about using the program now and are building real expertise in the use of SolidWorks. You can continue to grow your SolidWorks skills and learn even more by discovering the purpose of additional functions yourself. If you get stranded at any point, use the Help functions or refer to a book on SolidWorks where all of the functions are explained.

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

SolidWorks Benelux
RTC Building
Jan Ligthartstraat 1
1800 GH Alkmaar, Netherlands
Tel: +31 (0)72 514 3550