SolidWorks® Tutorial 7

GARDEN LIGHT

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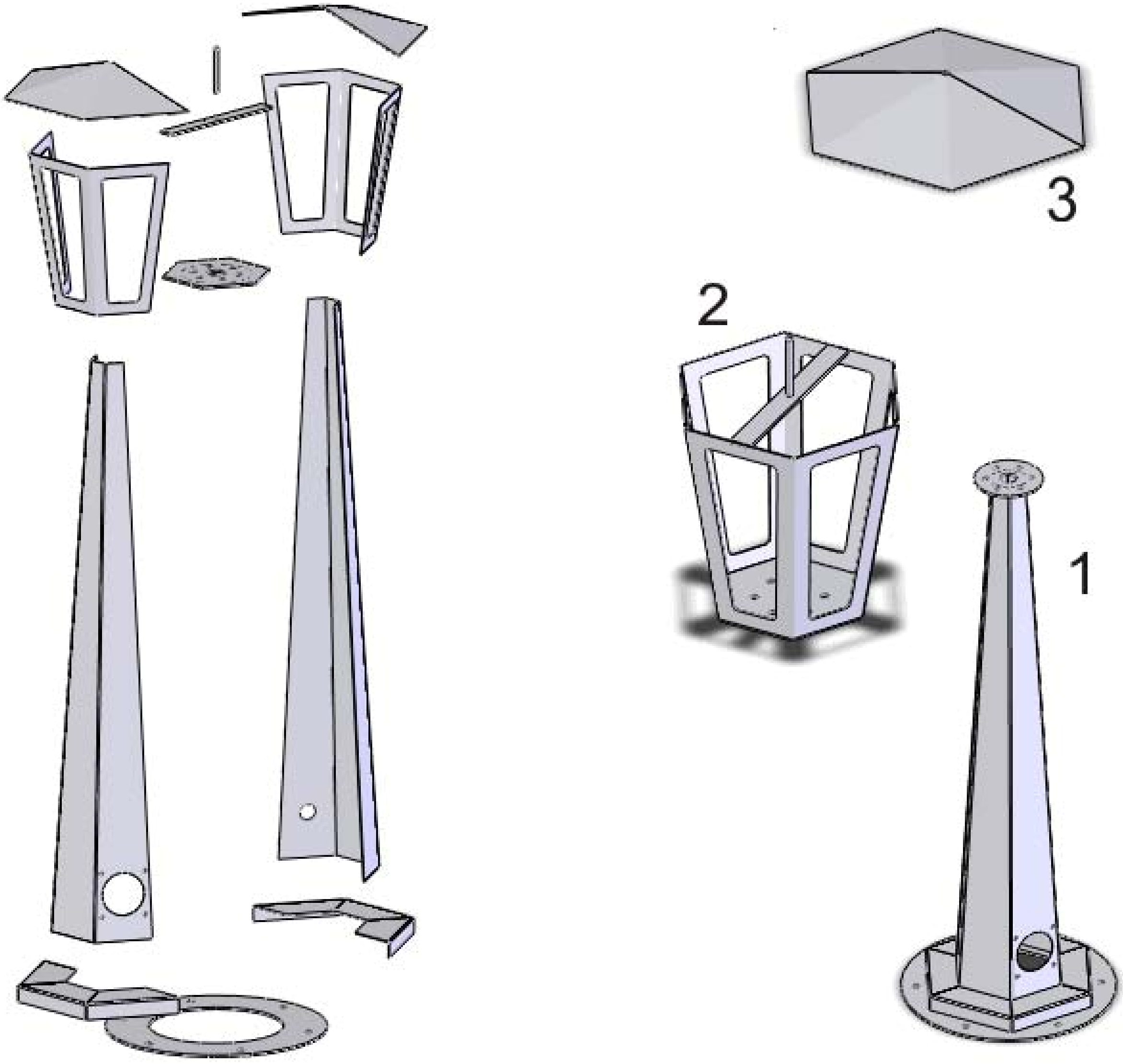
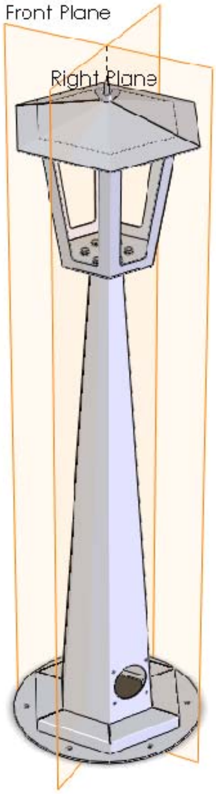
In this tutorial we will create a garden light. It is completely built from sheetmetal. In Tutorial 4 (candlestick) you learned how to shape sheetmetal in SolidWorks. In this tutorial we will go further using these techniques. We will create several parts from sheetmetal.

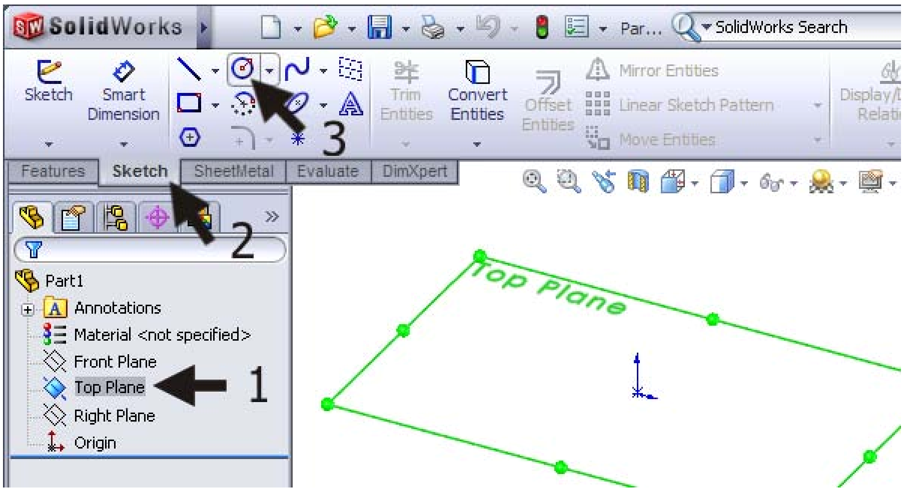
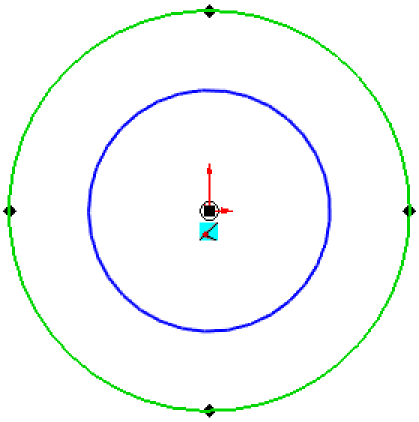
The garden light is a fairly complicated product and you will learn a lot from this tutorial. For instance, how to make a copy of a part and how to change it afterwards. How to you solve problems that are reported back and how to build a model from sub-assemblies?

Below you will find the exploded view with all parts of the light. We will build the whole product from three sub-assemblies (or welding assemblies). These are also visible in the illustration (numbers 1, 2 and 3). The welded parts or assemblies are bolted together with nuts and bolts.



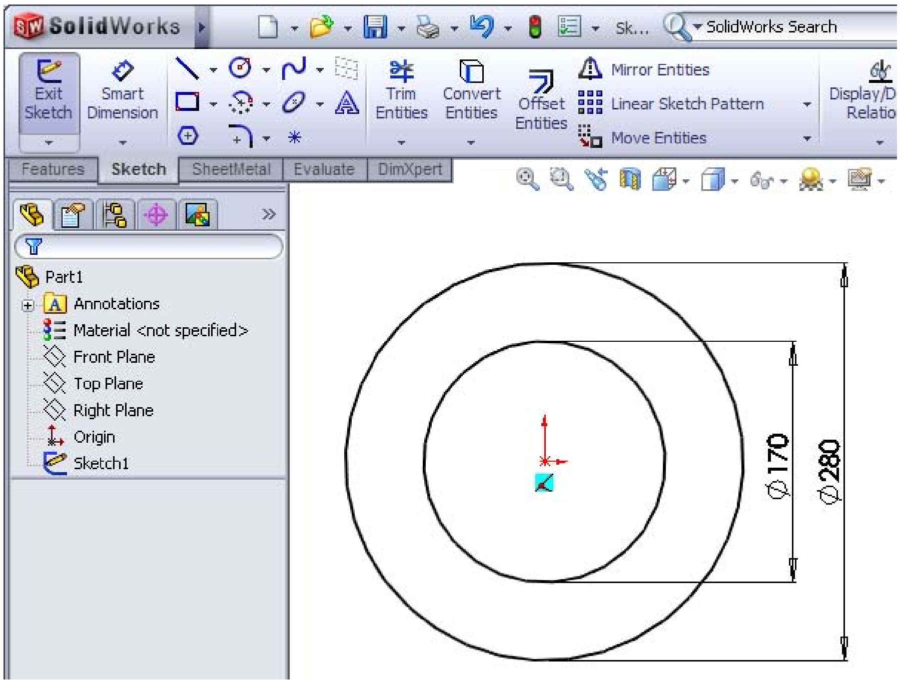
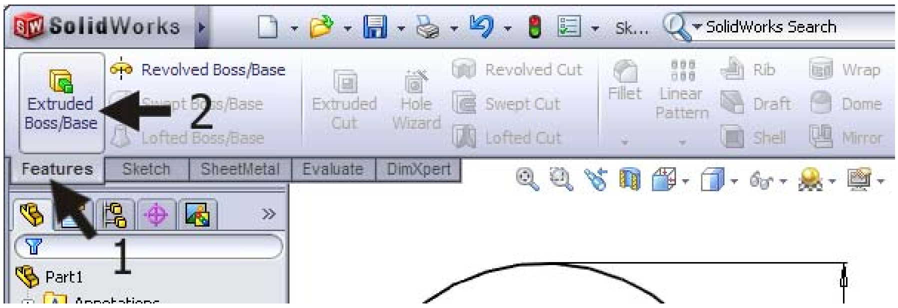
	With every part we create, we make sure that the origin is exactly in the center of the model. If we do so, the Front planes and Right planes of all parts will fit exactly. This will make it a lot easier to create and assemble all of the different parts at the end. 

	Work plan	Let's get started. First, we create a base that will end up at the top. The first part is the base flange. This is a simple round part with a number of holes according to the illustration below.

		How would you handle this part? We will built it from two features: 1. First, we will make a ring with a hole in the center. We will use Ex- truded Boss/Base for this. 2. After that we will position the six holes with Circular pattern.
1	Start SolidWorks and open a new part.	
2	1. Select the ‘Top Plane’ in the FeatureManager. 2. Click on ‘Sketch’ in the CommandManager. 3. Click on Circle.	
3	Draw two circles and make sure the center of both cir- cles is at the origin (the ze- ro point of the drawing field).	

<table>
<tr>
<td>4</td>
<td>Click on ‘Smart Dimension’ in the CommandManager and give every circle a dimension.

After this you can change the dimension of the circles.

Make sure the outer circle has a diameter of 280mm and the inner one has a diameter of 170mm.</td>
<td>
</td>
</tr>
<tr>
<td>5</td>
<td>Click on ‘Features’ in the CommandManager and then on ‘Extruded Boss/Base’.</td>
<td>
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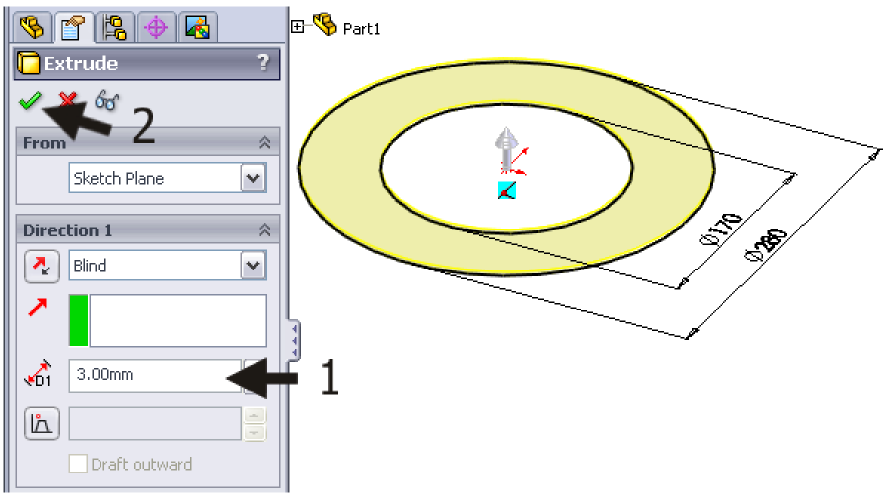
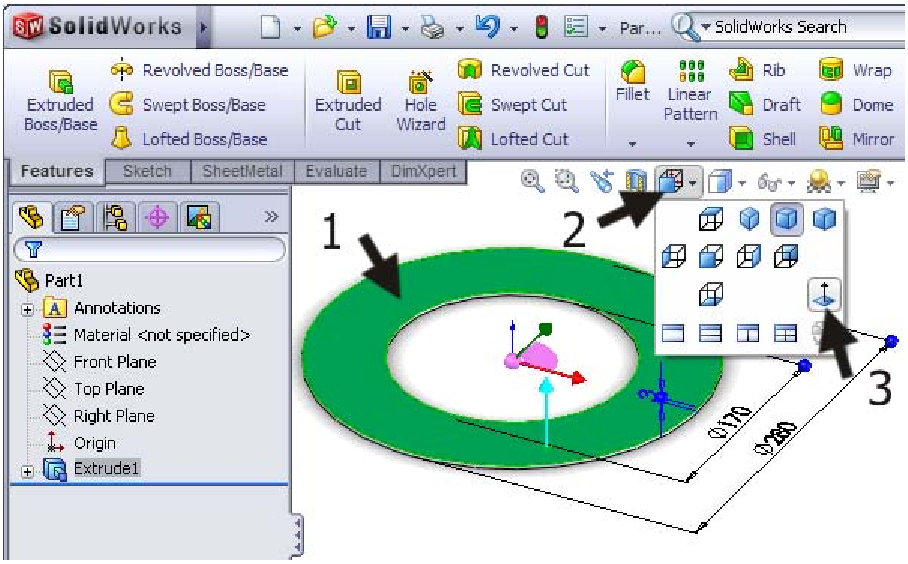
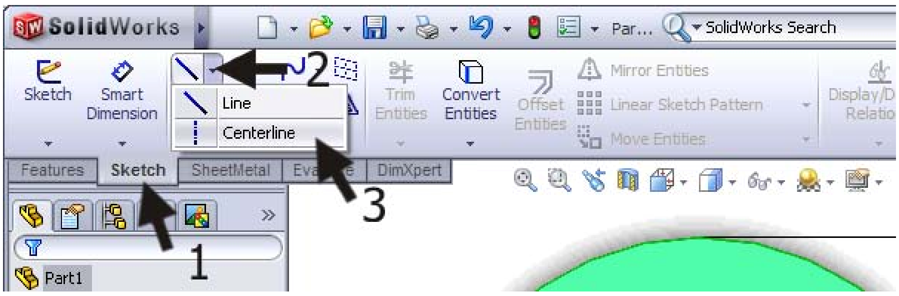
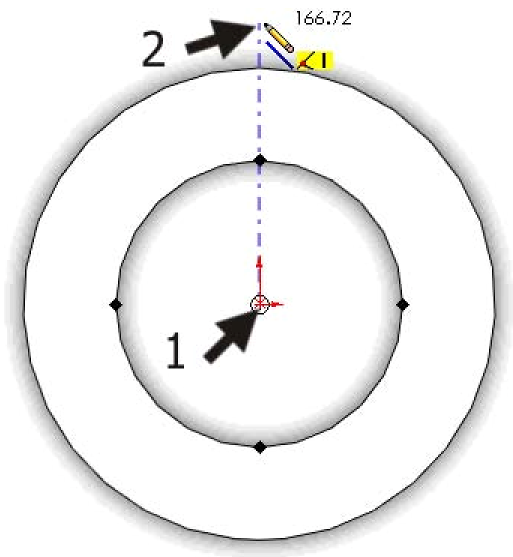
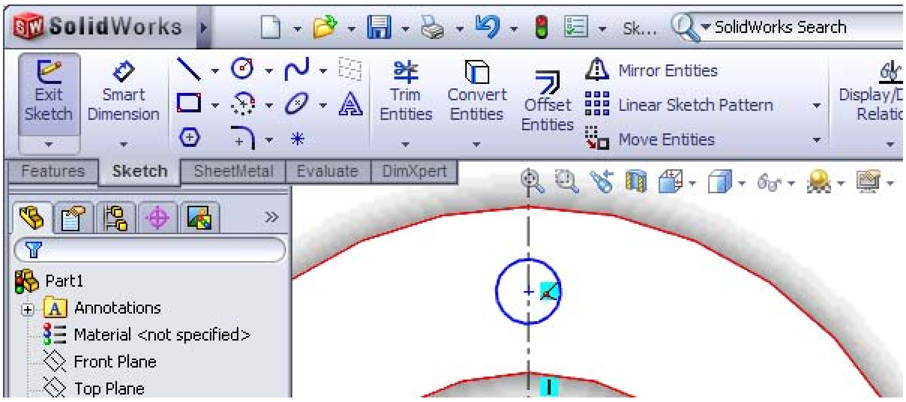
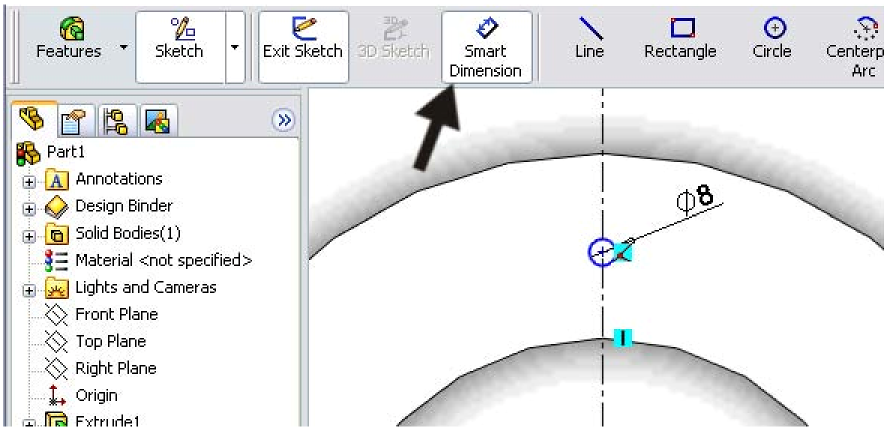
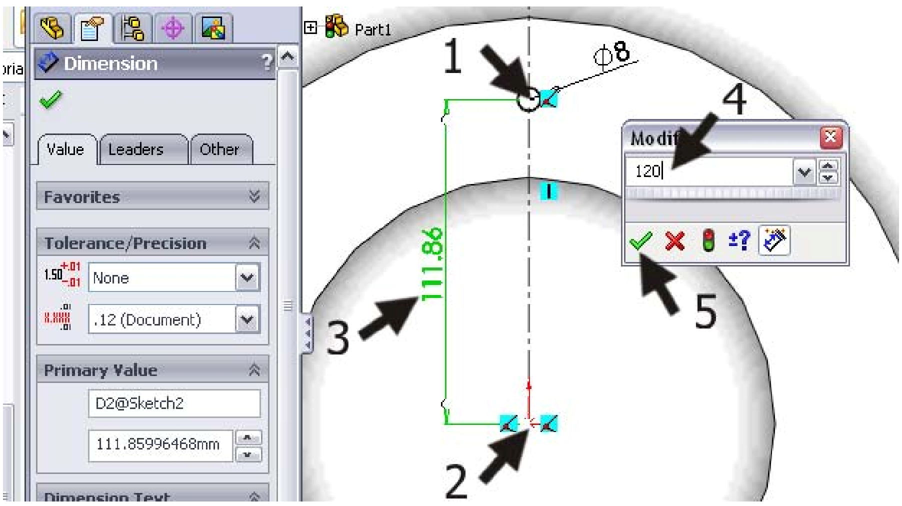
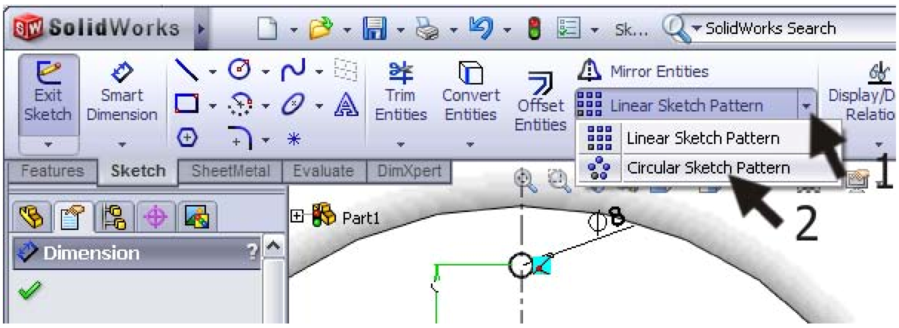
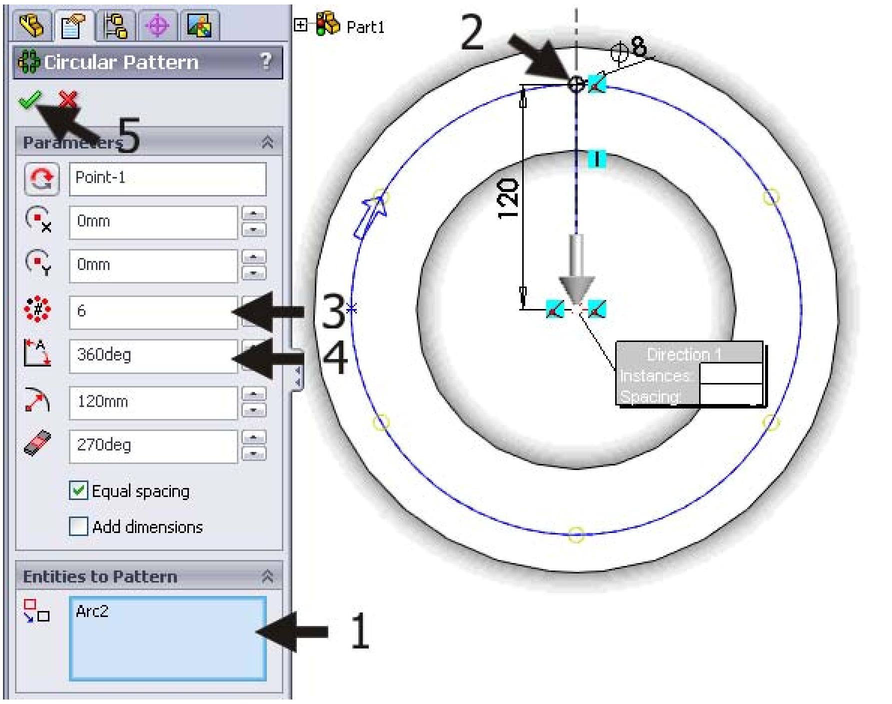
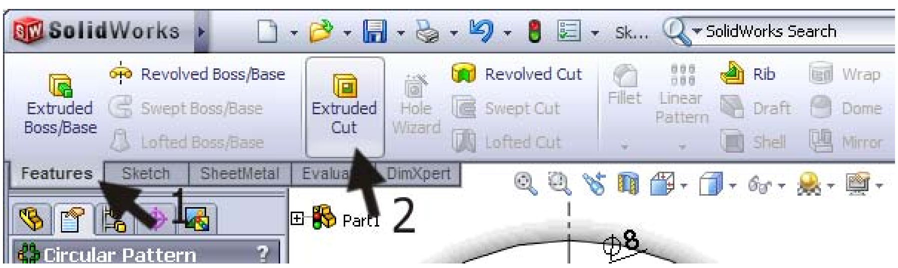
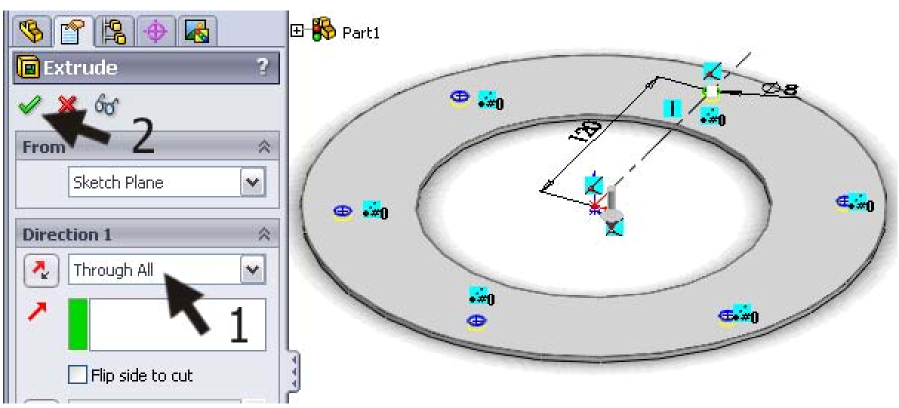
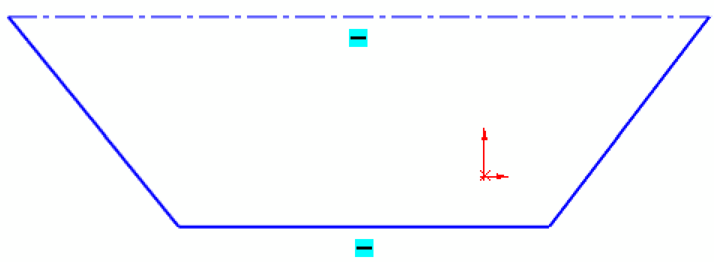
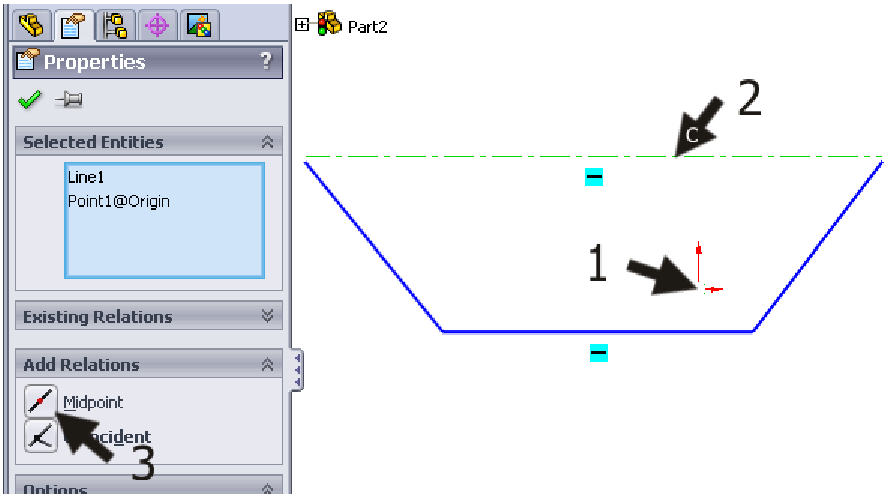
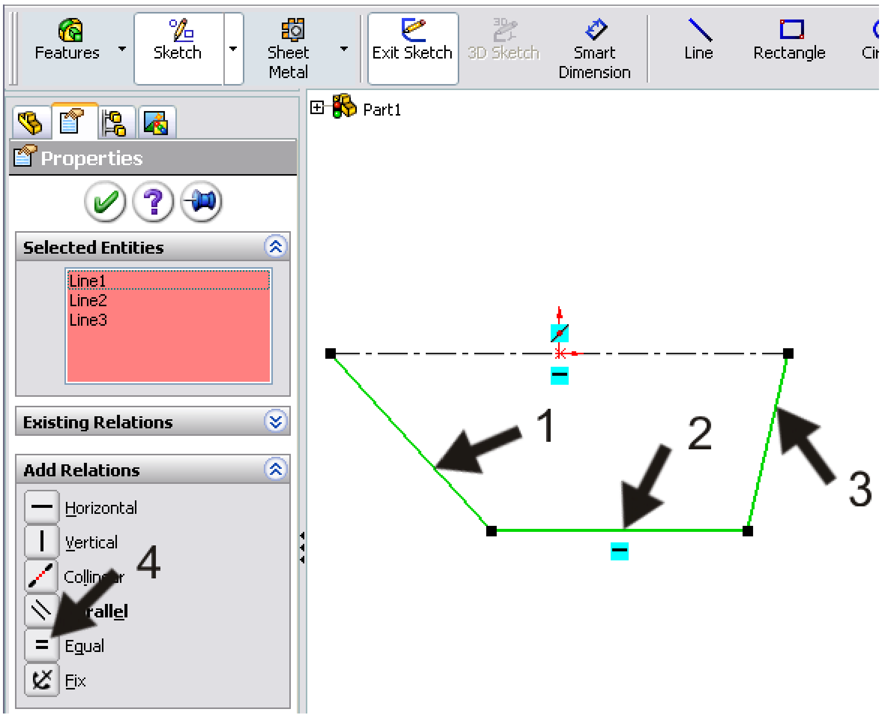
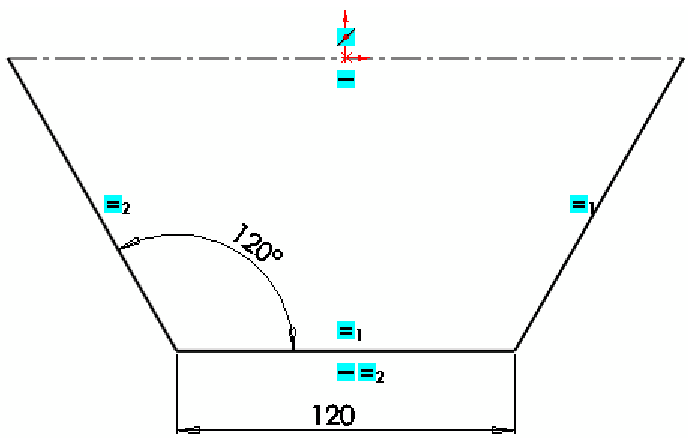
<tr>
<td>6</td>
<td>Set the thickness in the PropertyManager to 3mm and click on OK.</td>
<td>
</td>
</tr>
</table>

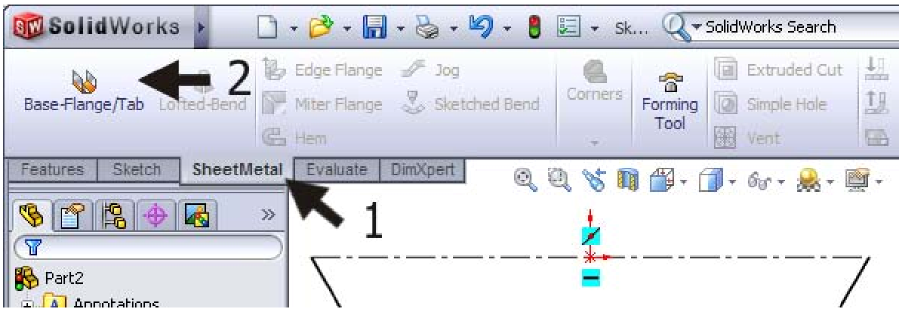
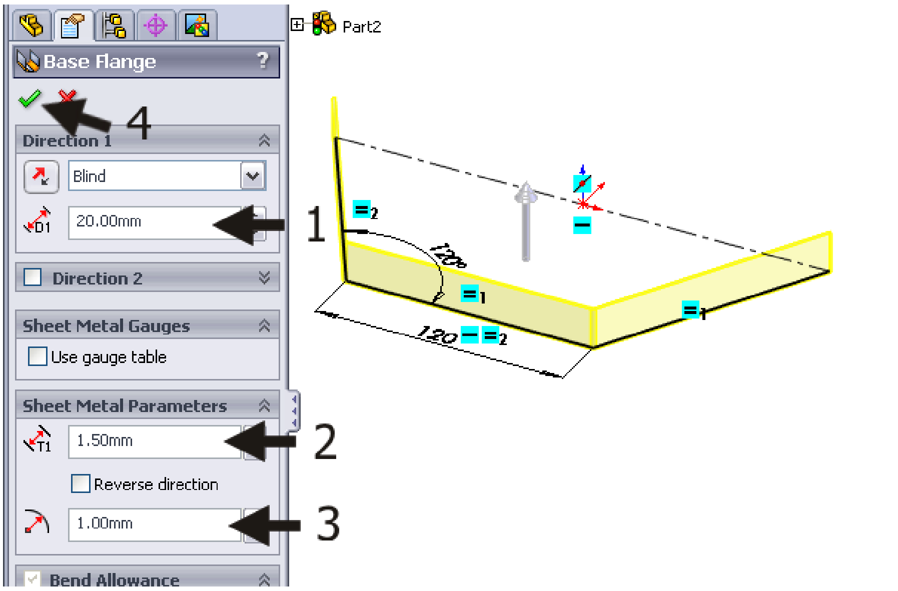
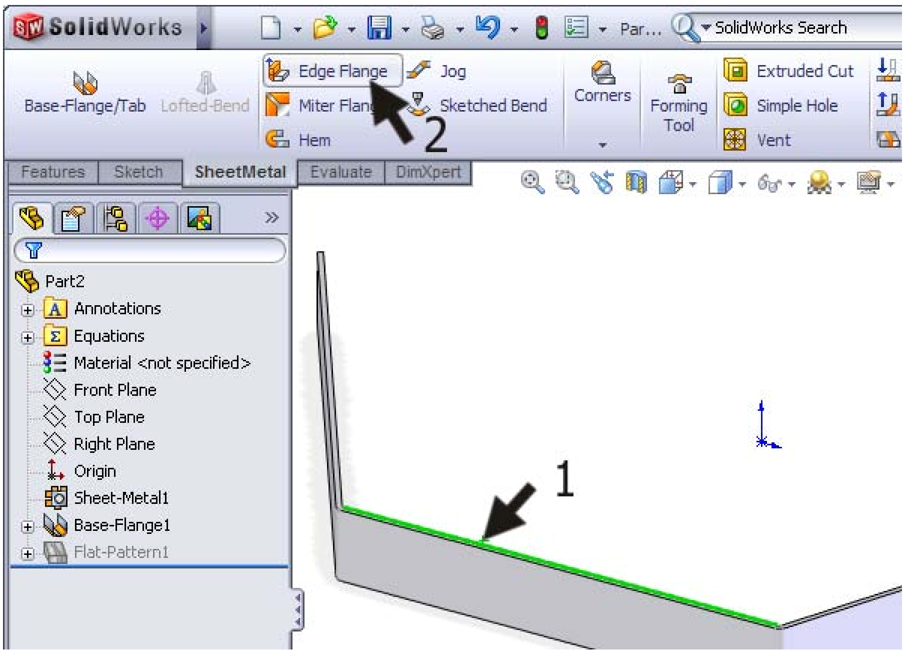
7	Next, we will make a sketch of the six mounting holes in the Top Plane. Be sure to have a straight view at this plane by using the following commands: 1. Click on the Top Plane. 2. Click on the Rotate button. 3. Select the option Nor- mal To.	
8	First, draw an auxiliary line: 1. Click on 'Sketch' in the CommandManag er. 2. Open (whenneces- sary) the extended menu.	
9	Draw the centerline from the origin vertically up- wards. Push the <Esc> key on the keyboard to end the cen- terline command.	

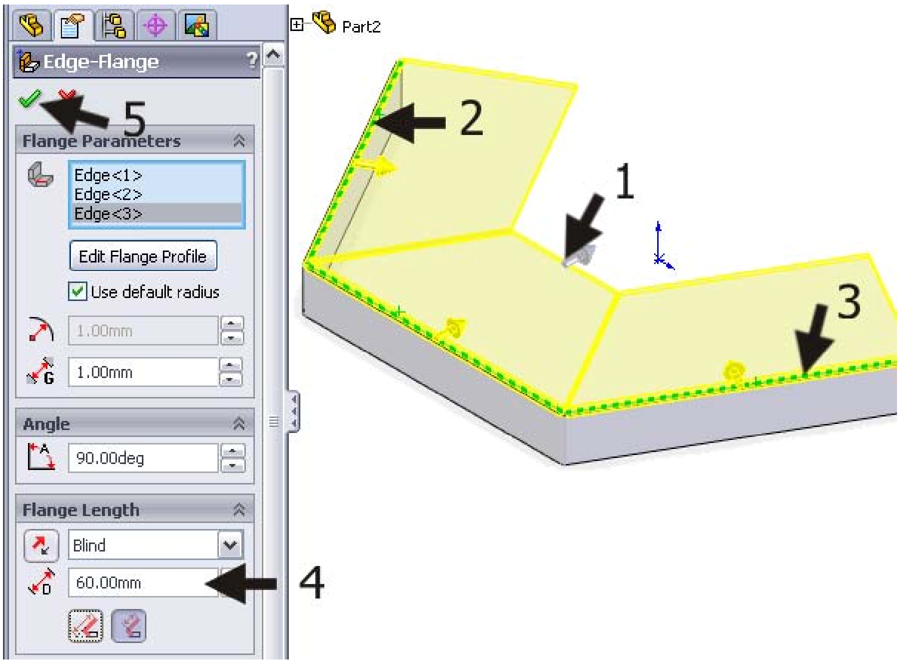
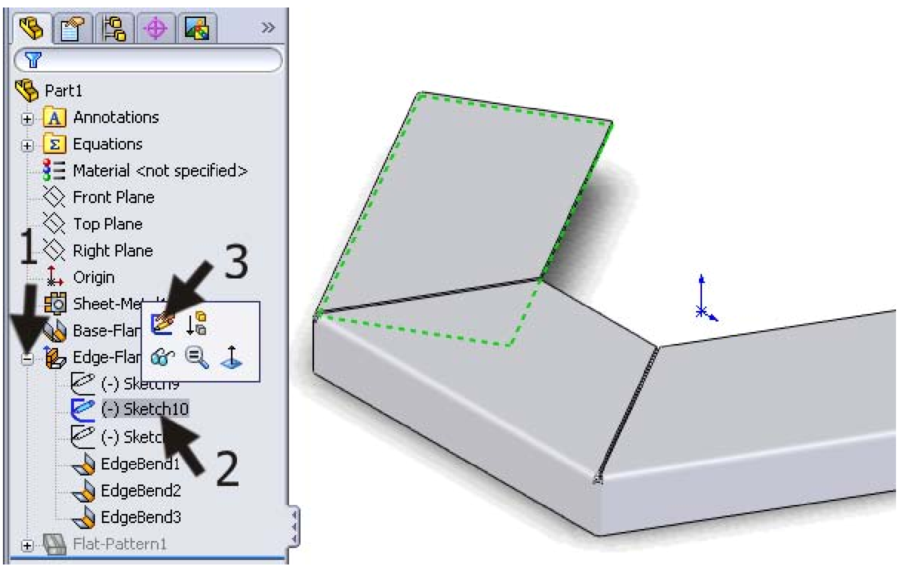
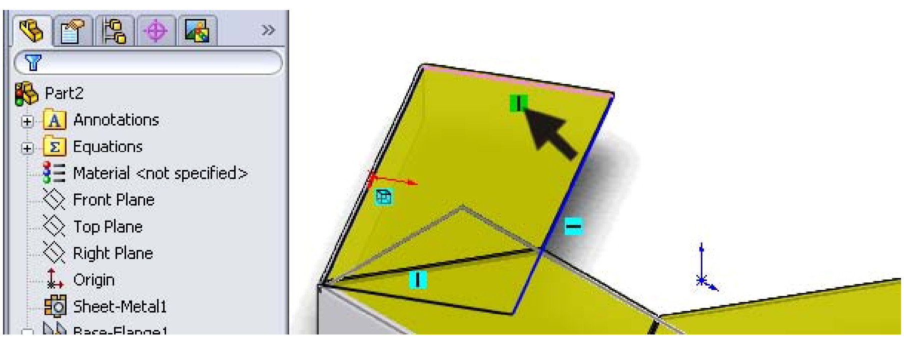
10	Click on Circle in the CommandManager, and draw a small circle like in the illustration on the right. Make cure the center of the circle is directly above the centerline (check the blue symbol).	
11	Click on 'Smart Dimension' in the CommandManager and set a dimension of Ø8mm for the circle.	
12	Set a dimension for the distance between the circles to the origin, as shown in the illustration. With the Smart Dimension command still active, click on: 1. The center of the circle. 2. The origin. 3. The point where you want the dimension to be. 4. Change this size to '120mm' . 5. Click on OK.	

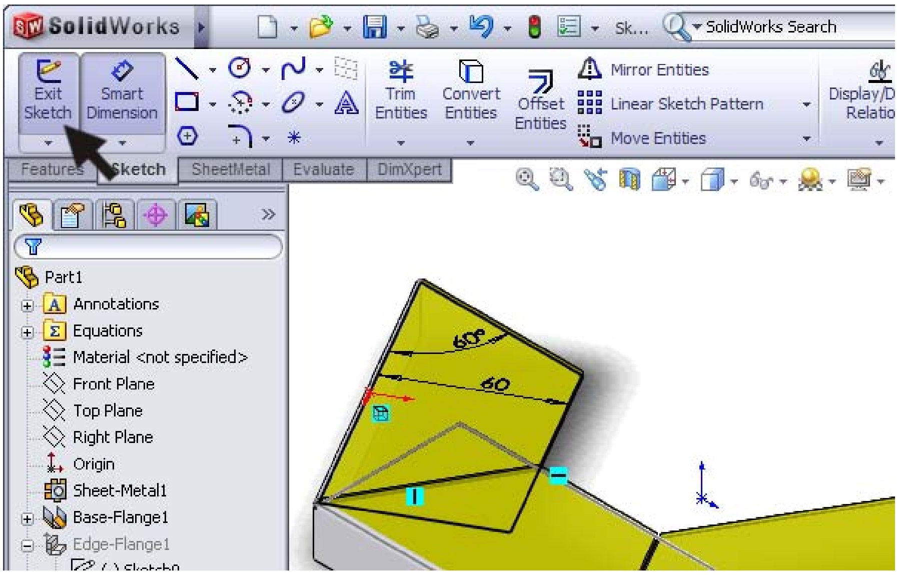
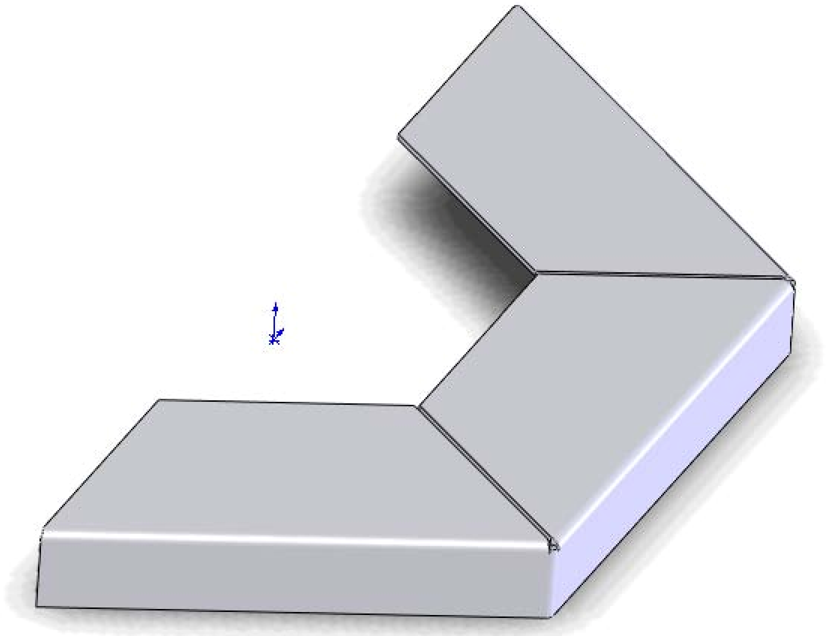
13	1. Click on the arrows next to the 'Linear Sketch Pattern' in the CommandManager. 2. Click on 'Circular Sketch Pattern'.	
14	1. Click on 'Entities to Pattern' in the Proper- tyManager. The selec- tion field turns blue 2. Select the circle you want to copy 3. Change the number of copies to '6'. 4. Check that the corner is at a complete 360°. 5. Click on OK.	
15	Click on 'Features' in the PropertyManager and next on 'Extruded Cut'.	
16	1. Set the depth of the hole to 'Through All' (through the entire model). 2. Click on OK.	

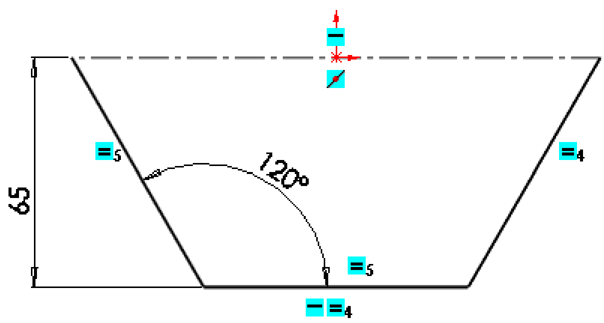
17	The first part is ready now. Create a new folder for the garden light, and save this part as: flange-bottom.SLDPRT.	
	Work plan	The second part we will be make is the base. It looks a bit like a part of a hexagonal container. See the drawing below. We will create this part from sheetmetal.
18	Open a new part.	
19	Select the 'Top Plane' in the PropertyManager. Draw a horizontal center-line at a random point first. The length is about 250mm. After that, draw three lines like in the illustration on the right. Make sure the middle one is also in a horizontal posi-	

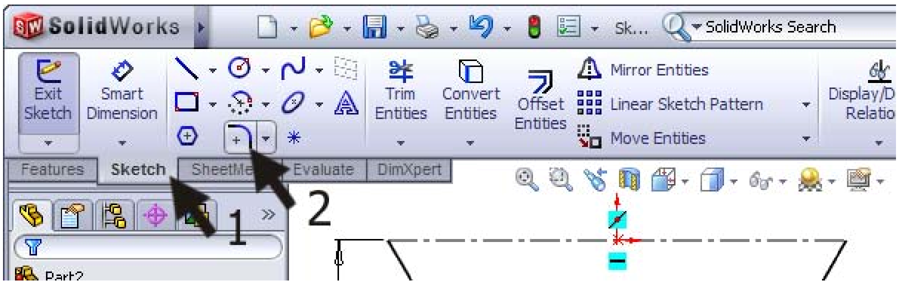
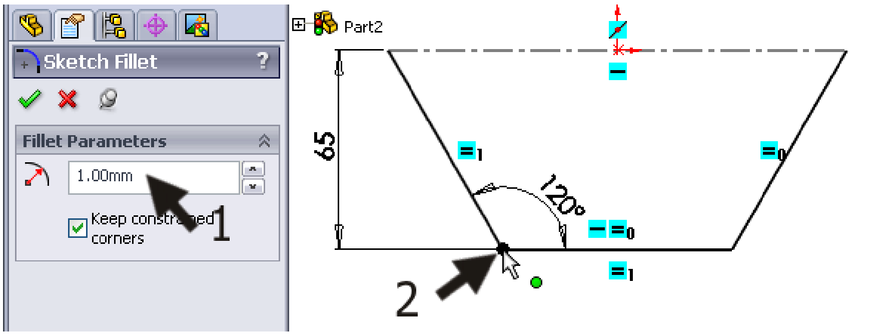
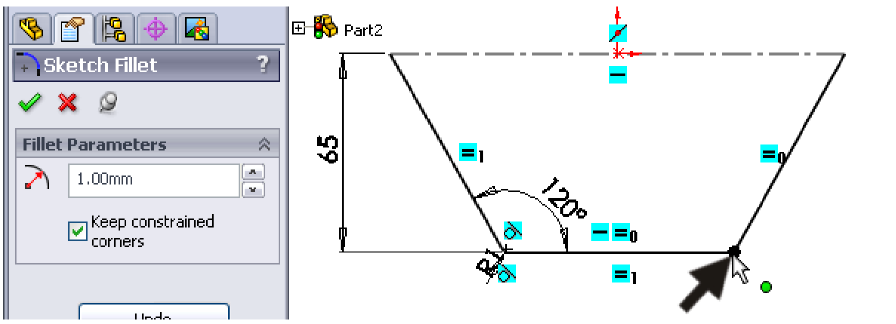
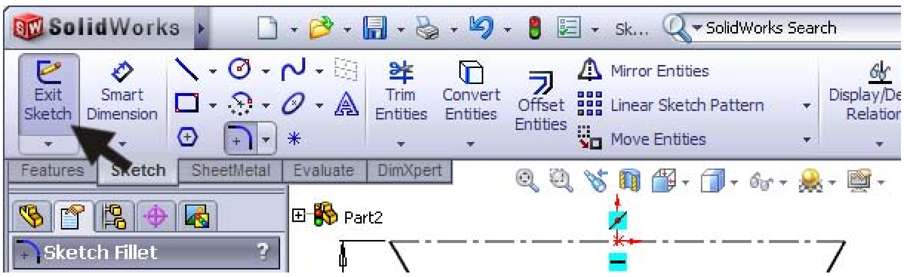
20	Next, move the middle of the centerline towards the origin. 1. Click on the origin 2. Hold the <Ctrl> key at the keyboard and click on the centerline. 3. Click on ‘Midpoint’ in the PropertyManager.	
21	Make the length of the three lines equal: 1. Click on the first line. 2. Hold the <Ctrl> key and select the second one. 3. Select the third one, still holding the <Ctrl> key. 4. Click on ‘Equal’ in the CommandManager.	
22	Click on ‘Smart Dimension’ in the CommandManager. Set the dimensions as in the illustration on the right.	

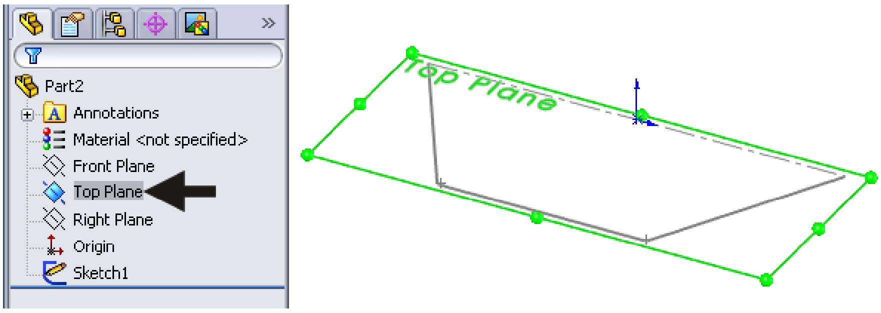
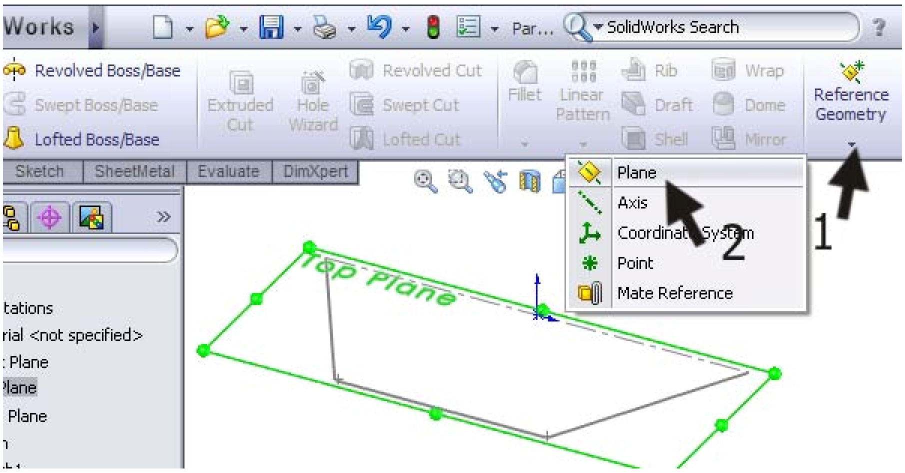
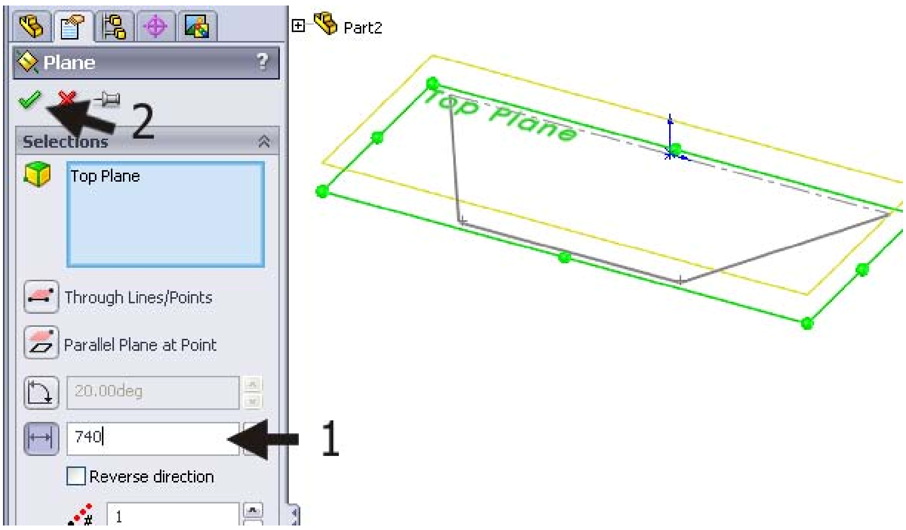
23	1. Click on ‘SheetMetal’ in the CommandManager. 2. Click on ‘Base-Flange/Tab’.	
	Tip!	When the SheetMetal button is not visible in the CommandManager, click on one of the tabs of the CommandManager. A list will appear and you can turn SheetMetal on. This is described extensively in Tutorial 4 (candlestick).
24	Set the following features in the PropertyManager: 1. The height of the part is ‘20mm’. 2. The thickness is ‘1.5mm’. 3. The bending radius is ‘1 mm’. 4. Click on OK.	
25	Next, we will create the bended surface: 1. Select the edge you want to bend. 2. Click on ‘Edge Flange’ in the CommandManager.	

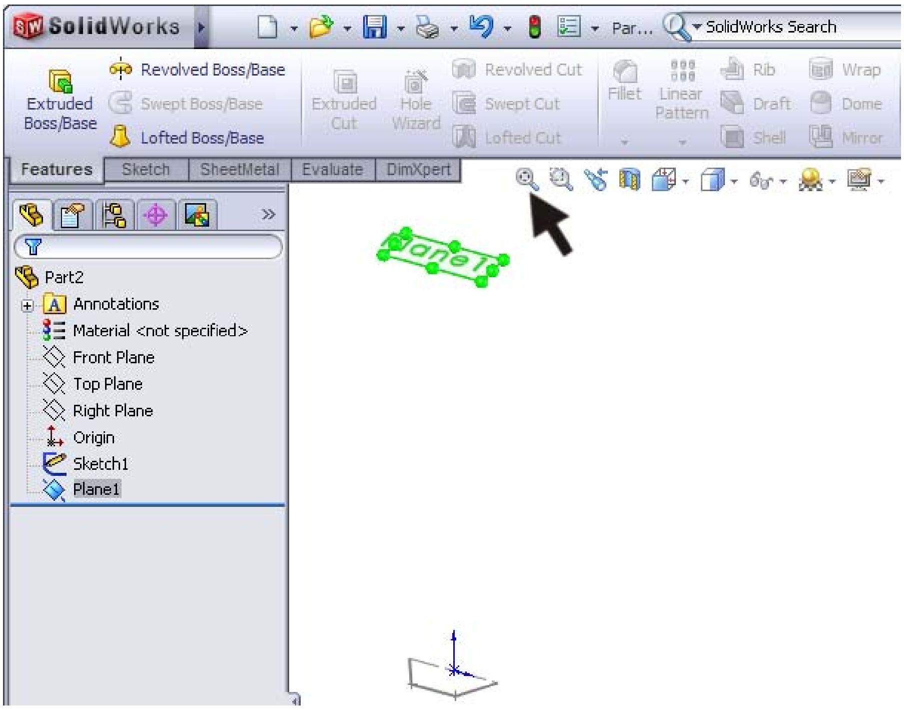
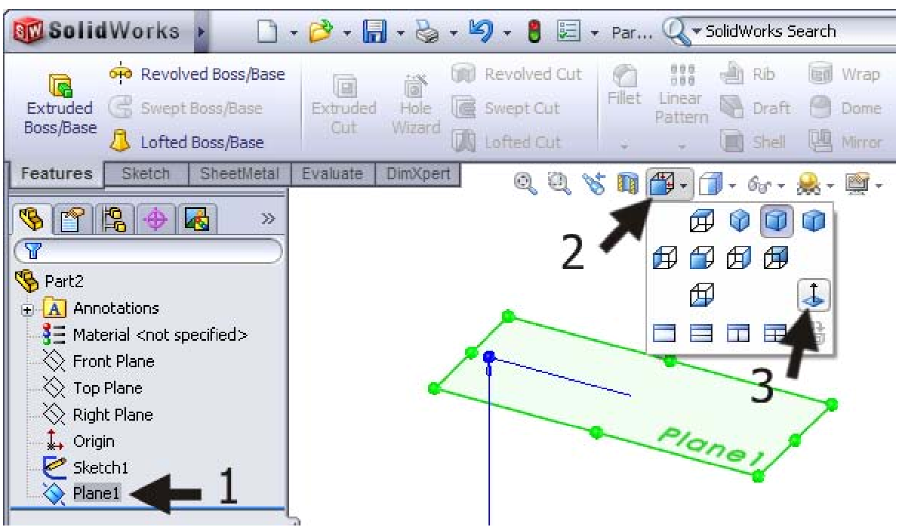
26	1. Click at a random point to set the first plane. 2,3 Click on both other edges in order to make planes there as well. 4. Set the length of the planes to ‘60mm’. 5. Click on OK.	
27	The shape of the planes is determined by the sketch. The sketches have to be altered now. 1. Click on the ‘+’ symbol before ‘Edge Flange’ in the FeatureManager. 2. Three sketches will appear: click on the sketch of one of the outer planes. 3. Click on Edit Sketch in the menu that appears.	
28	Now, we can change the sketch. Select the relation ‘Vertical’ (look at the drawing on the right). Push <Del> (delete) key on the keyboard.	

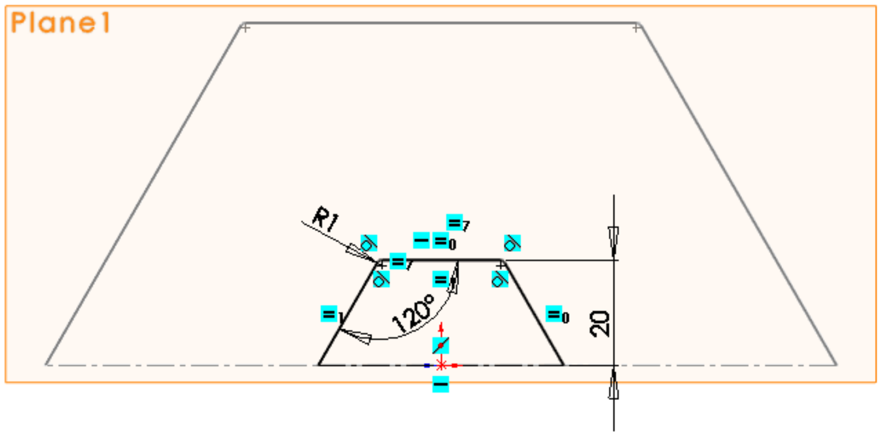
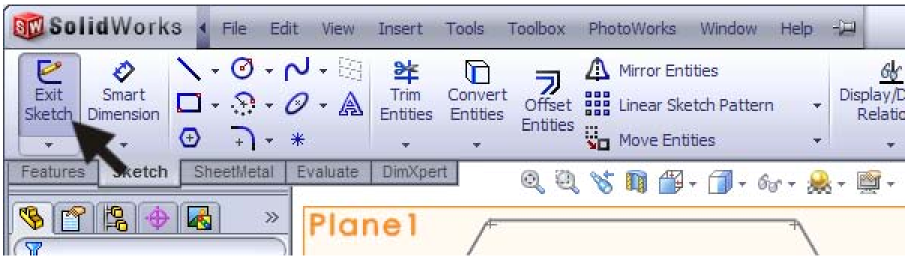
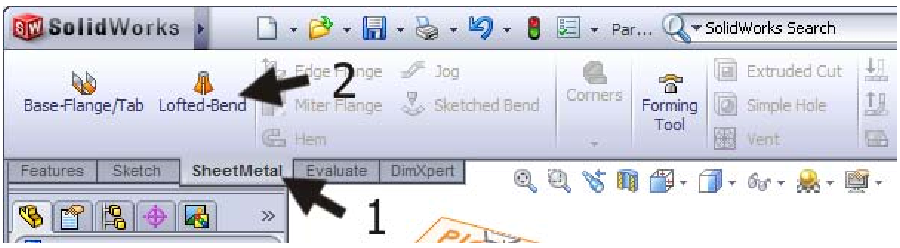
29	Set the dimensions with 'Smart Dimension' like in the illustration. Click on 'Exit Sketch' in the CommandManager.	
30	Repeat steps 27 to 29 for the plane on the other side. The end result will look like the image on the right.	
31	Save the file as: base.SLDPRT.	

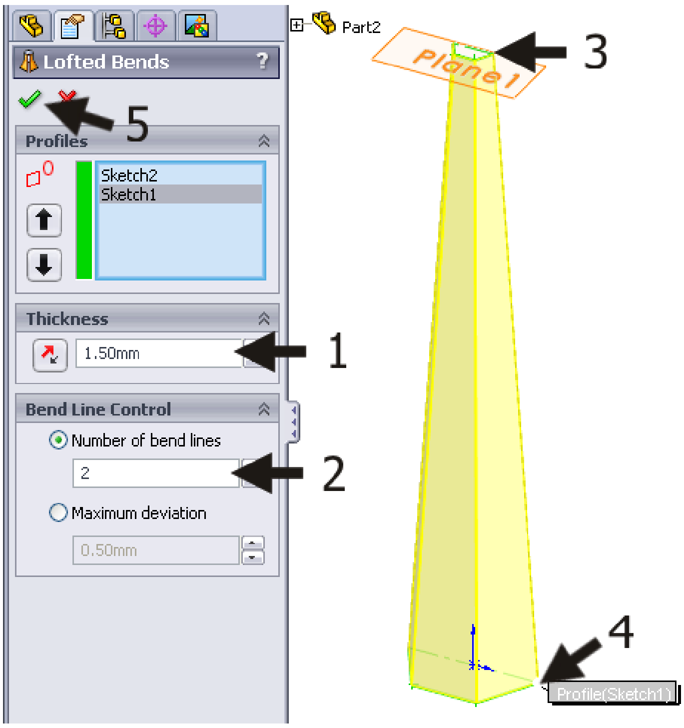
	Work plan	The next part we will make is the light stand. We will make two varieties (configuration s). 1. One version has a hole of Ø20 as a cable transit. 2. The other version has a larger hole (Ø55) and four smaller holes (Ø4.5) for mounting a wall socket. The sheetmetal shape is the same for both configurations, so we will start with those. Because all planes of this part are in an angled position, we can not build it like we have built parts previously. Therefore, we will use another method. W will draw the base flange and SolidWorks will calculate the shape of the sheet in between.
32	Open a new part. Select the ‘Top Plane’, and draw the sketch as in the illustration. If you have a problem with this, look at steps 19 to 22. You did exactly the same thing there (only with oth-er dimensions).	

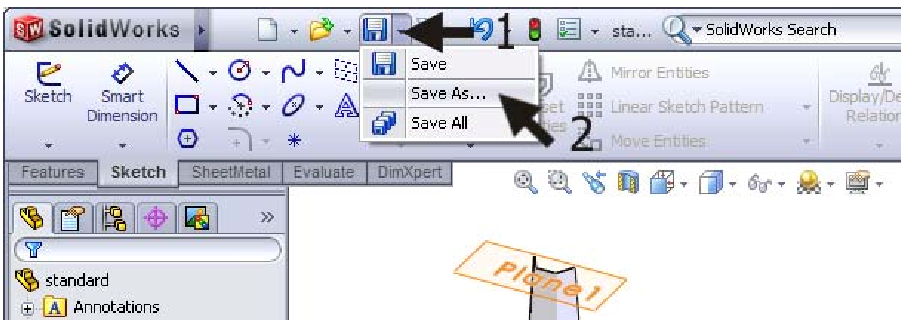
33	We will round the corners now. Click on 'Sketch' and then Fillet in the CommandManager.	
34	1. Change the radius to '1 mm' in the PropertyManager. 2. Click on the first corner in the sketch.	
35	Click 'Yes' in the message that appears.	
36	Next, click on the second corner. The message from step 35 appears again. Again, click 'Yes'.	
37	Click on 'Exit Sketch' in the CommandManager.	

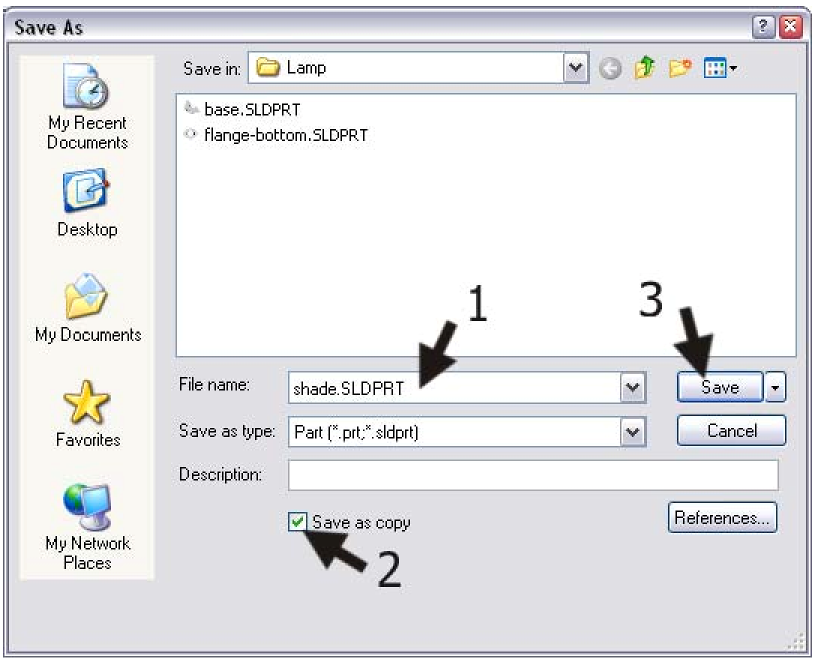
38	Click on the ‘Top Plane’ in the FeatureManager.	
39	1. Click on Reference Geometry in the CommandManager. 2. Click on ‘Plane’.	
40	1. Set a distance of ‘740mm’ in the Proper- tyManager. 2. Click on OK.	

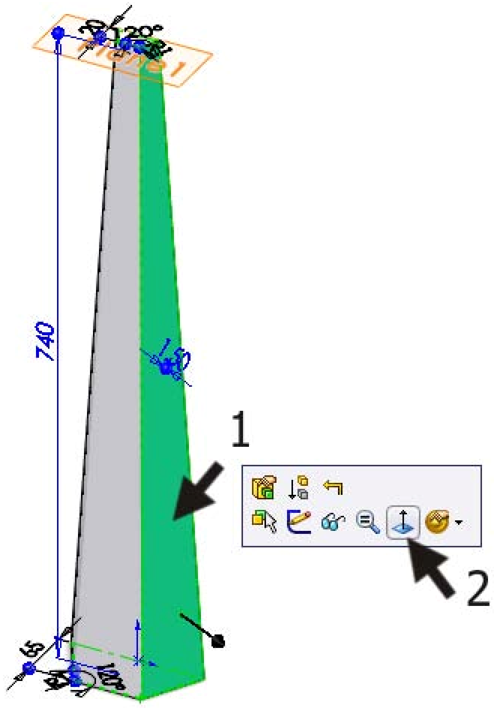
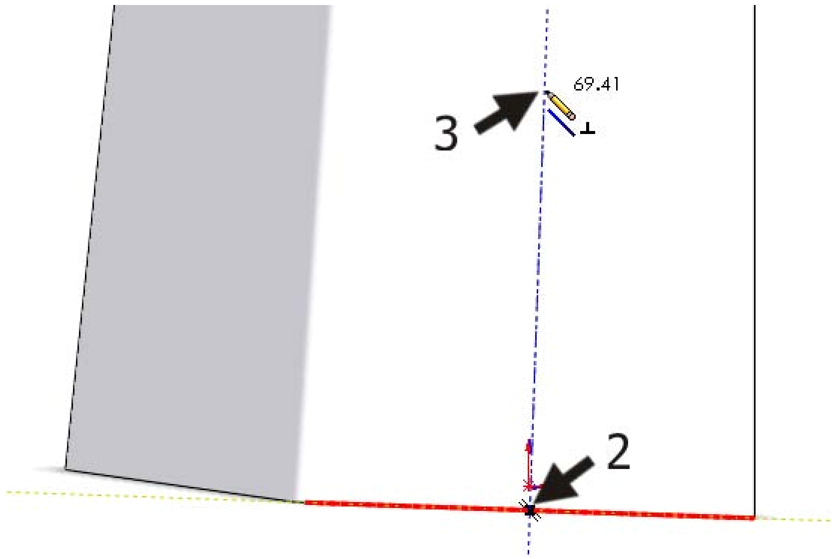
41	Click on Zoom to fit in the View Toolbar. Notice that a plane called 'Plane1' is floating above the sketch you have just made.	
	Tip!	We have seen before that you can draw a sketch on every plane in Solid-Works. This is normally one of the planes Top, Front or Right, which are always available, but it can also be a plane from your model. If is also possible to make a sketch at a point, when no plane is available. In such a case you can create a plane yourself (Plane). You can define it in every spot and with every angle in relation to the standard planes. This is what you have done in step 40. You have created an auxiliary plane 740mm above the Top Plane. Here we can draw our next sketch
42	1. Make sure 'Plane1' is still selected. If not, click on it in the Fea-tureManager. 2. Click on View Orienta- tion. 3. Click on Normal To.	

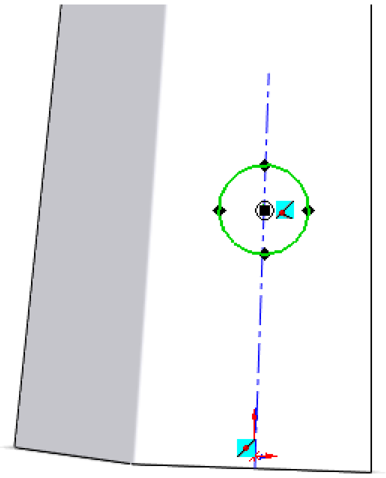
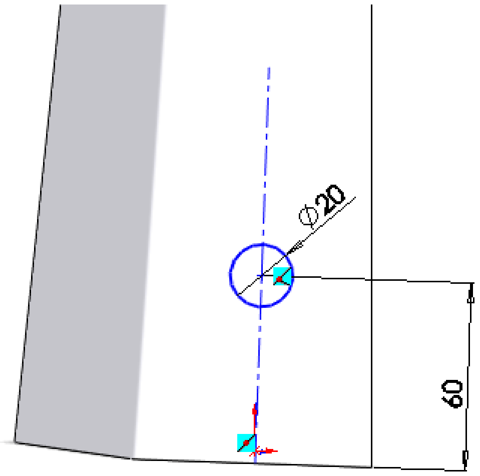
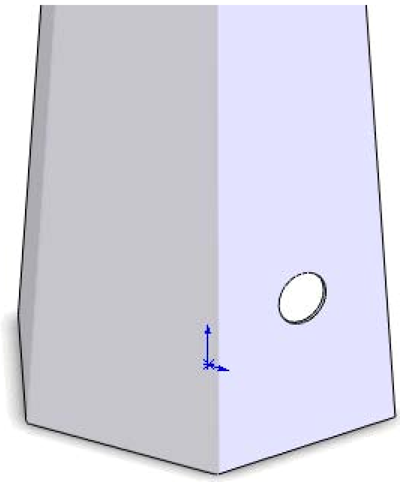
43	Now make exactly the same sketch as you did before. The only difference is that the height is now 20 mm instead of 65mm. Follow steps 34 to 39 to do so. When the sketch is done, it should look like the illustration on the right. Notice that the big sketch in gray is the first sketch you created of the Top-plane.	
44	Click on 'Exit Sketch' in the CommandManager to close the sketch.	
45	1. Click on 'SheetMetal' in the CommandManager. 2. Click on 'Lofted-Bend'.	

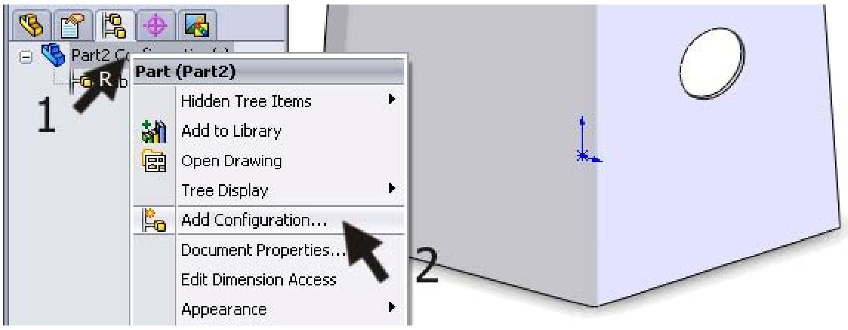
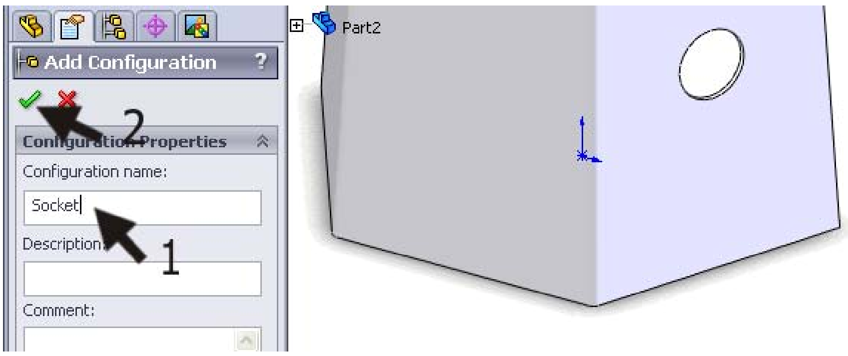
<table>
<tr><td>46</td><td>Set the following features:
1. ‘Thickness’ of the ma- terial is ‘1.5mm’.
2. The number of bending lines is ‘2’.
3. Select the upper sketch on the right side.
4. Also select the lower sketch on the right side.
5. When the preview looks OK, click on OK.</td><td>
</td></tr>
<tr><td>47</td><td>The basic shape is ready now. We need this shape once more for the lampshade. That is why we will make a copy of this file at this point and use it later.

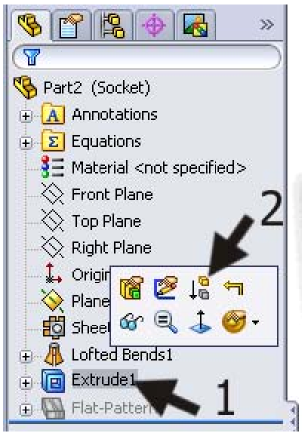
Click on the arrow next to Save in the Toolbar and click on ‘Save As…’.</td><td>
</td></tr>
<tr><td>48</td><td>1. Name the copy: ‘shade.SLDPRT’.
2. IMPORTANT: Check the option ‘Save as copy’.
3. Click on ‘Save’.

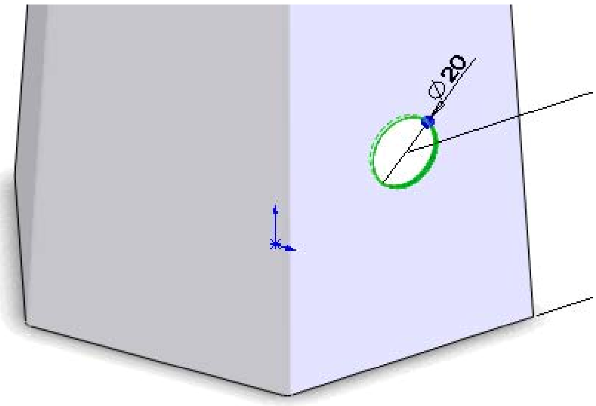
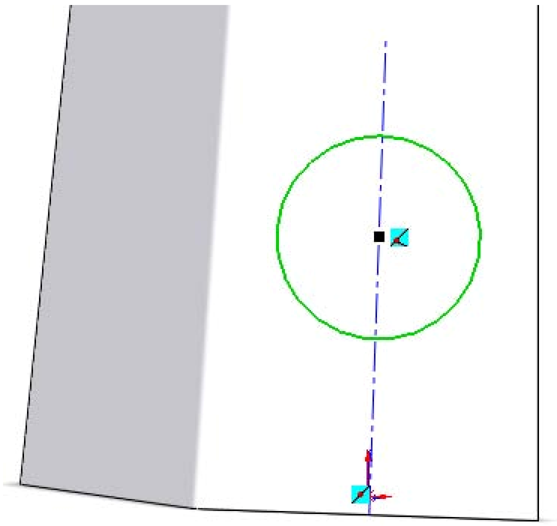
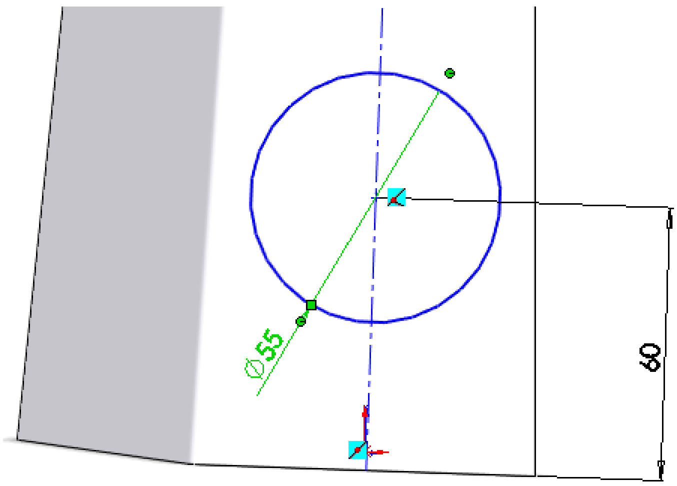
A new file has just been made (shade.SLDPRT). The name of the model we were working on has not changed.</td><td>
</td></tr>
</table>

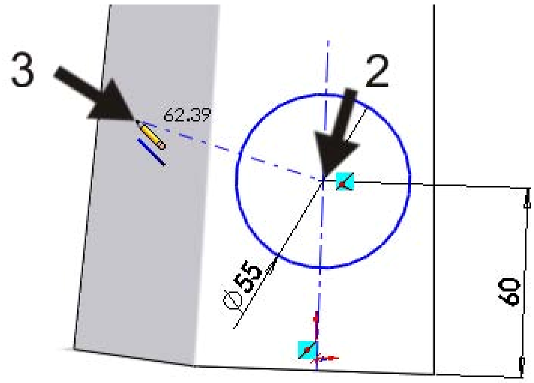
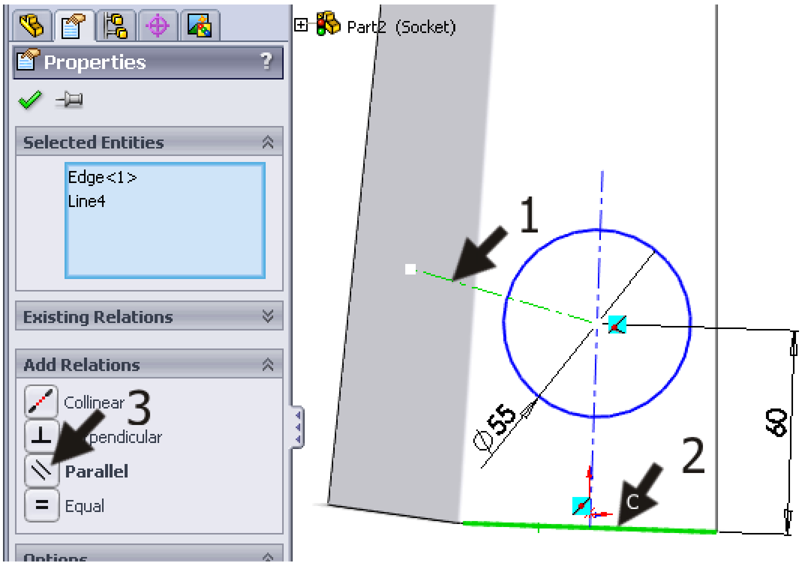
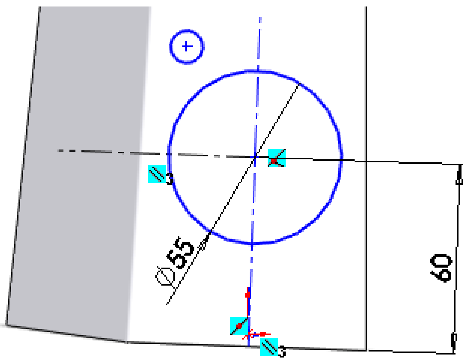
49	Next, we will make a hole for the cable feed. 1. Select the plane to make a sketch. 2. Click on Normal To in the menu that appears.	
50	First, draw a centerline straight across the plane in which we want to draw the hole 1. Click on 'Centerline' in the CommandManager. 2. For the first point, click on the middle of the lower edge of the plane. Note that this is not the origin. Zoom in so you will get a close view! 3. Next, click about 100mm above the lower side of the plane. Note that we must draw a line that is vertical on the plane (it has an angle of 90 degrees to the lower line and is NOT a vertical line!). Pay attention to the symbol that occurs during the drawing action: it tells you if you have indeed a vertical line in relation to the base line. 4. Push the <Esc> key.	

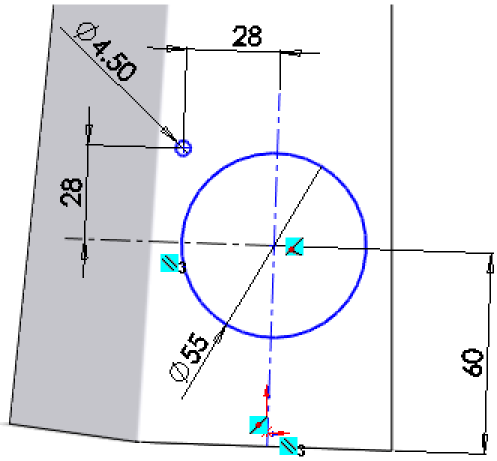
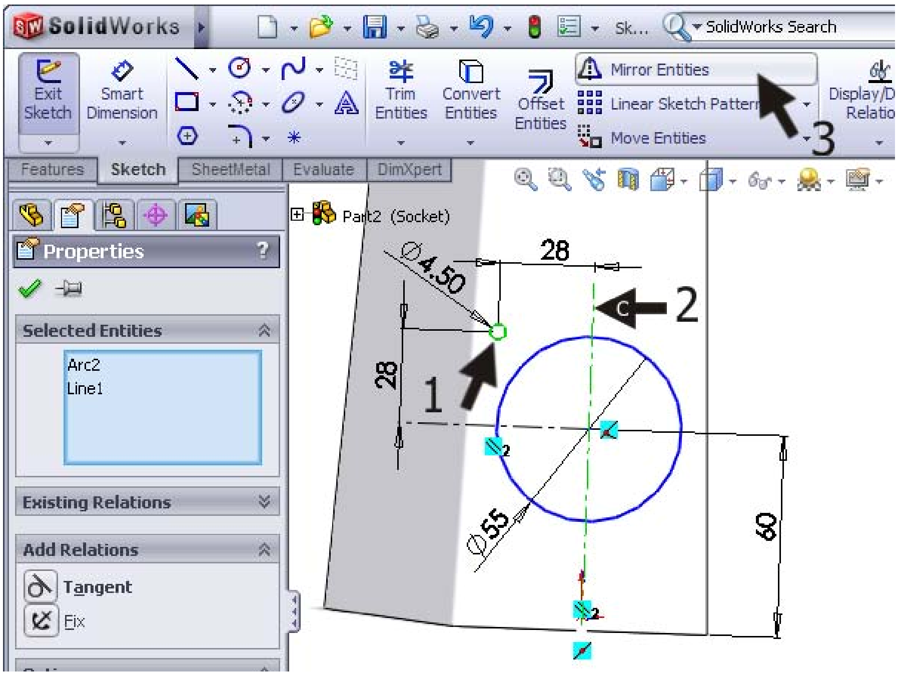
51	Draw a circle. Make sure the center of the circle is on the centerline.	
52	Add two dimensions like in the illustration.	
53	Create a Cut-Extrude from this sketch. Set the depth to Through All.	

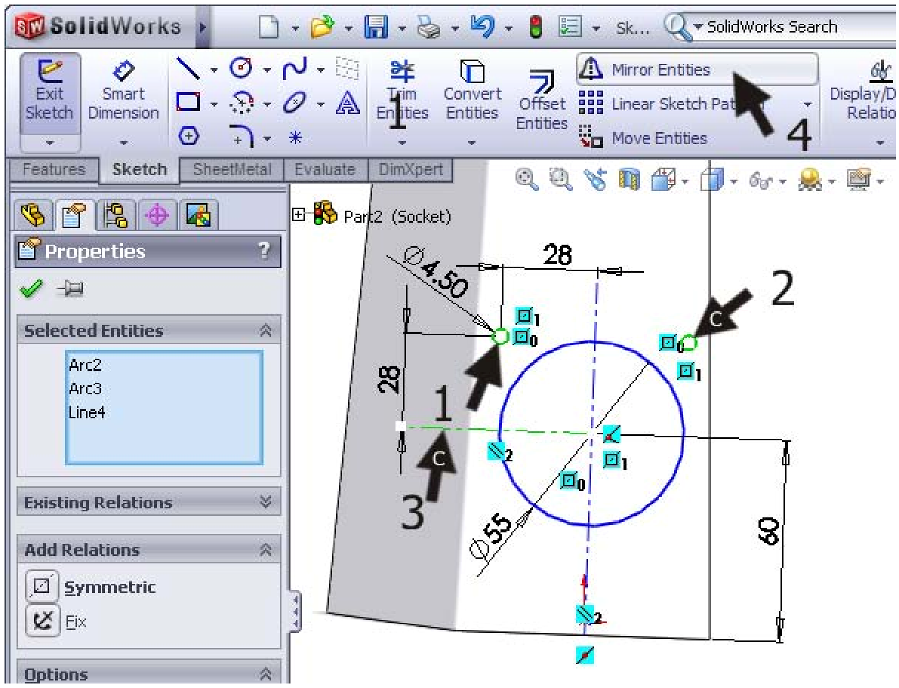
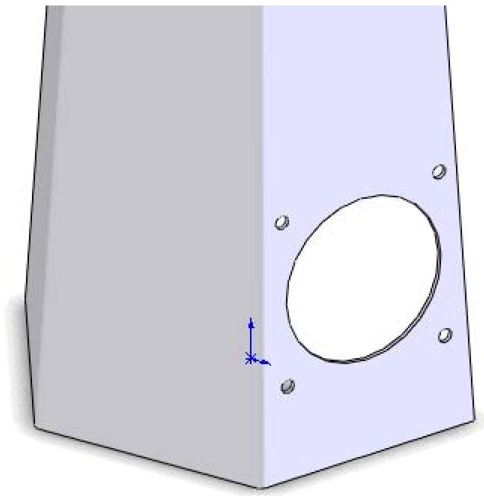
54	We will now make a second configuration of this part. Click on the Configuration- Manager tab.	
55	The current configuration is called ‘Default’. Click twice (slowly) on that name and change it to ‘Ca- ble’.	
56	1. Right-click on the up- per line in the Configu- rationManager. 2. Click on ‘Add Configu- ration...’.	
57	1. Fill in the name of the configuration in the PropertyManager: ‘Socket’ . 2. Click on OK.	
58	Return to the FeatureMa- nager.	

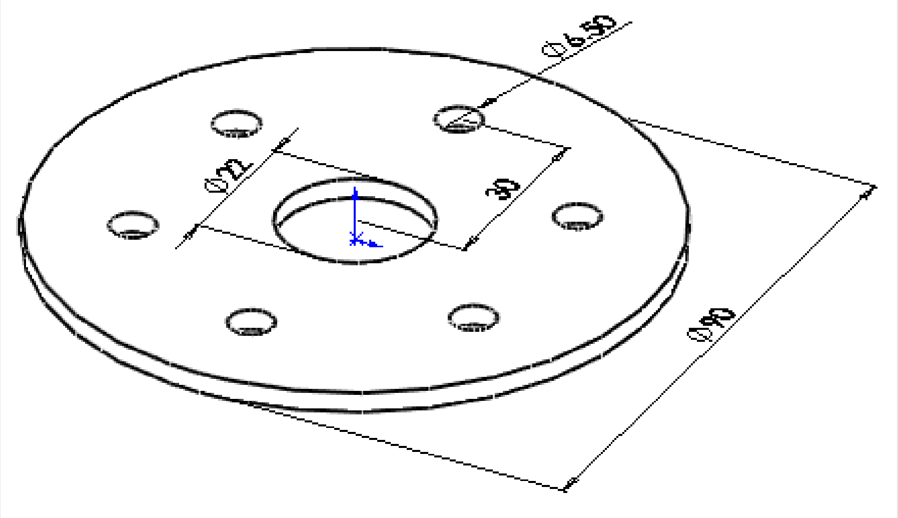
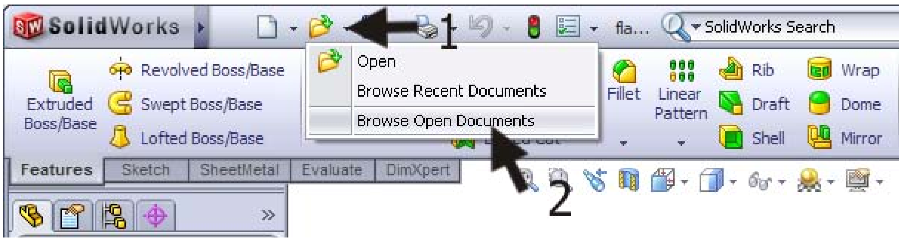
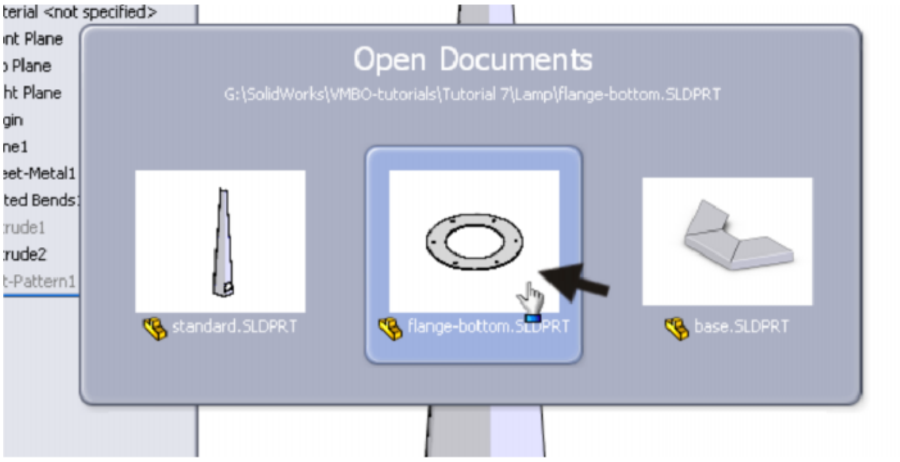
<table>
<tr><td>59</td><td>The configuration 'Socket' is active now. In this confi- guration we will suppress the cable feed hole.
1. Right-click on the fea- ture of the hole (Cut 'Extrude1') in the FeatureManager.
2. Click on Suppress in the menu that appears.</td><td>

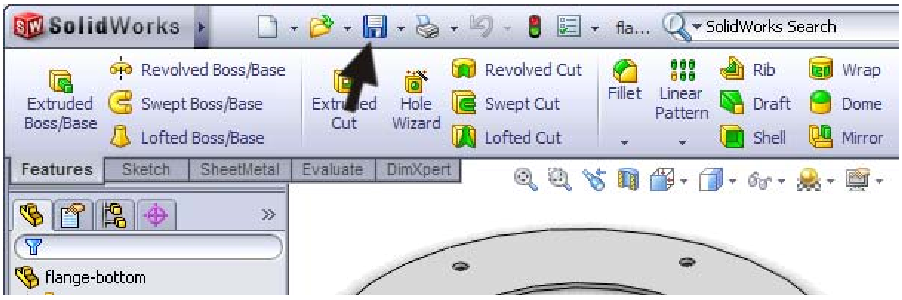
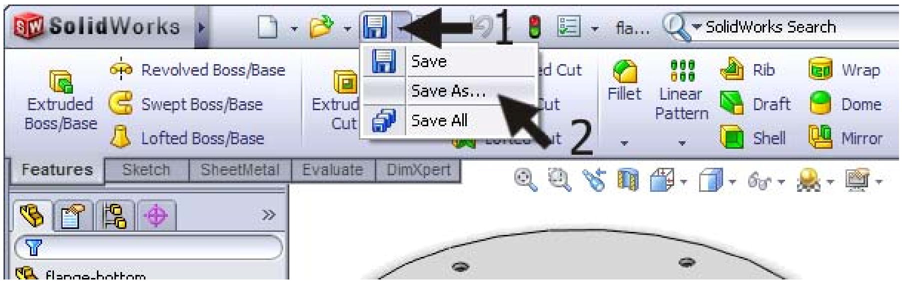
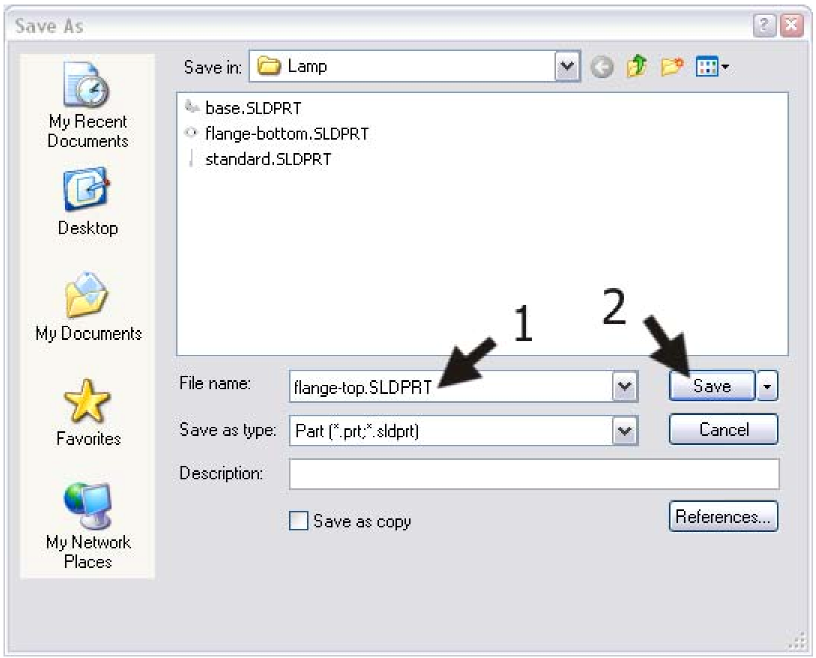

</td></tr>
<tr><td>60</td><td>Next we will make a hole for the power socket.
Start again with a sketch on the right plane. Draw a centerline and draw a cir- cle, like you did in steps 50 to 52.</td><td></td></tr>
<tr><td>61</td><td>Set the dimensions as shown in the drawing on the right.</td><td>
</td></tr>
</table>

<table>
<tr><td>62</td><td>Now, we have to create four mounting holes. First, we draw a horizontal centerline.
1. Click on ‘Centerline’ in the CommandManager.
2. Click on the midpoint of the circle to set the first point.
3. Click outside the circle to get the second point. NOTE that this is not a horizontal line. Therefore, you can better draw under it at an angle in order to avoid any unwanted relations.
4. Push the <Esc> key to close the Centerline</td><td></td></tr>
<tr><td>63</td><td>1. Select the centerline you have just made.
2. Push the <Ctrl> key and select the lower edge of the plane.
3. Click on ‘Parallel’ in the PropertyManager.</td><td></td></tr>
<tr><td>64</td><td>Draw a small circle, just about the same size and position as in the illustration on the right.</td><td></td></tr>
</table>

65	Give the circle a dimension: look at the illustration.	
66	1. Select the small circle. 2. Push the <Ctrl> key and select the vertical centerline. 3. Open (when necessary) the extended menu in the CommandManager. 4. Click on 'Mirror Enti- ties'.	

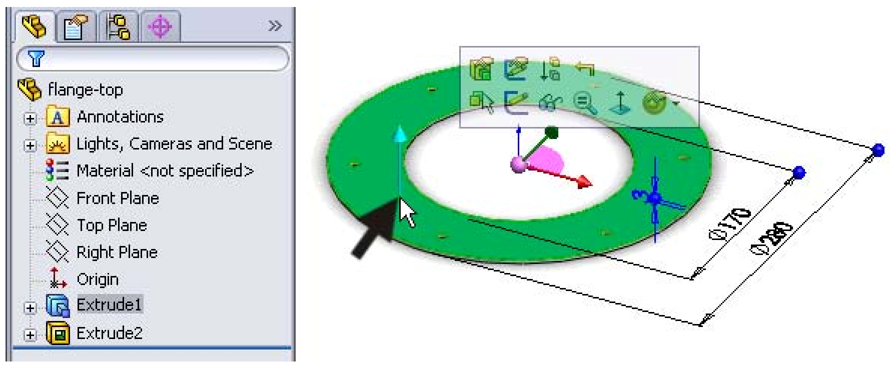
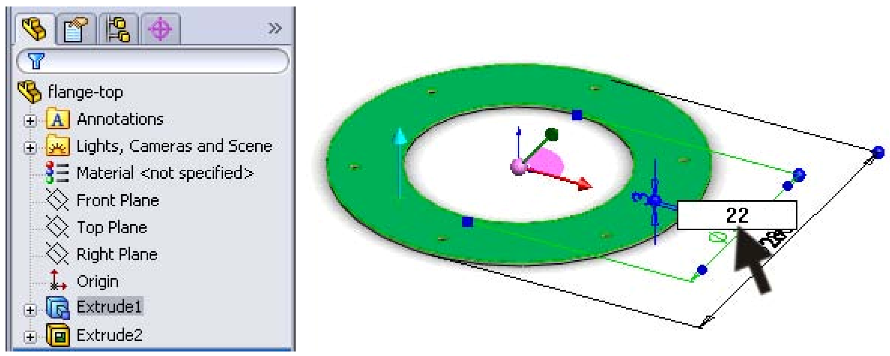
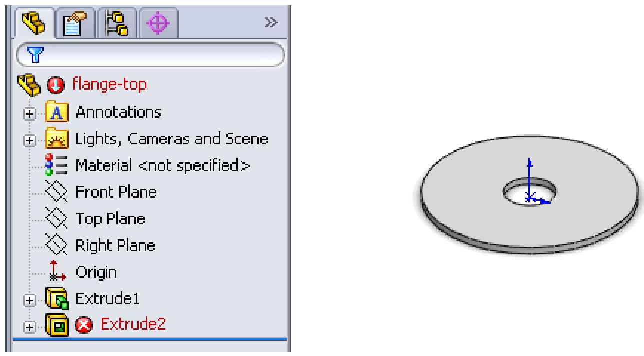
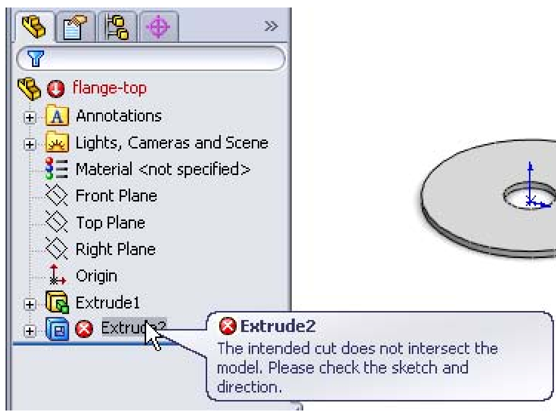
67	Select both circles AND the horizontal centerline. Click on 'Mirror Entities' in the CommandManager again. Now, you will have four mounting holes.	
68	Make a Cut-Extrude from this sketch. Set the depth to Through All.	
69	The part is ready now, with two configurations. Save the file as standard.SLDPRT.	

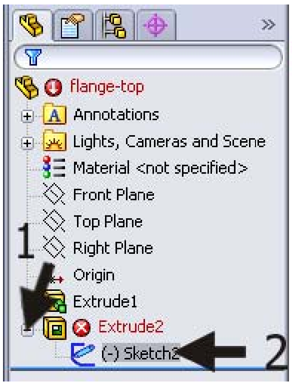
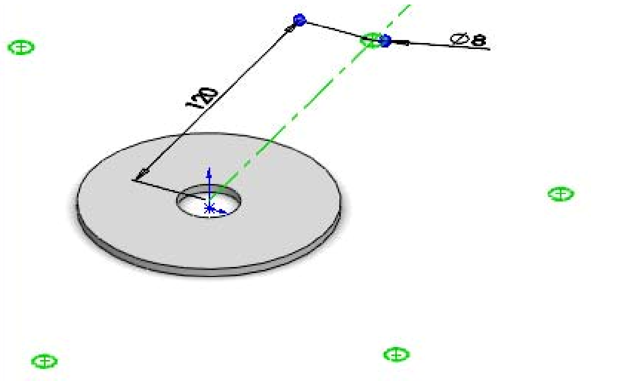
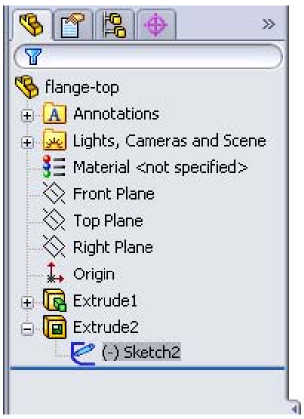
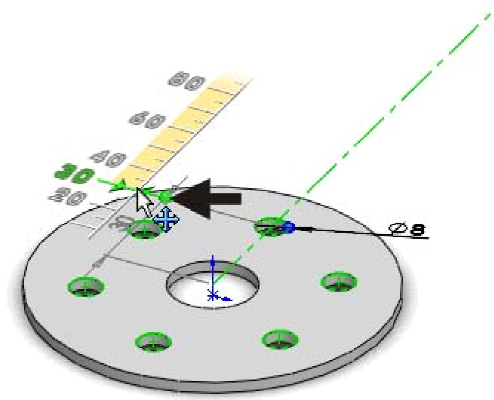
	Work plan	The next part will be the top plate. This part looks very much the same as the flange-bottom plate, which we made first: only the dimensions are different. For this reason, we will not make a new part. We will make a copy of the first part and will adapt it instead. 
70	Find the part flange-bottom.SLDPRT. It should still be open. 1. Click on the arrow next to Open in the Toolbar. 2. Click on 'Browse Open Documents'.	
71	Select the file 'flange- bottom.SLDPRT' in the menu that appears.	

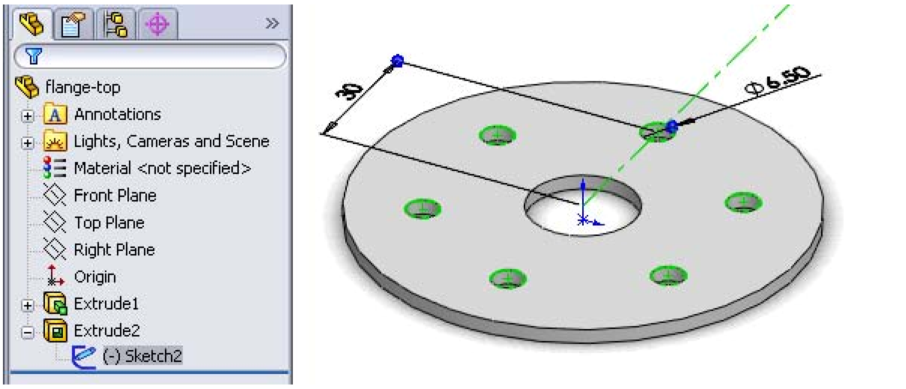
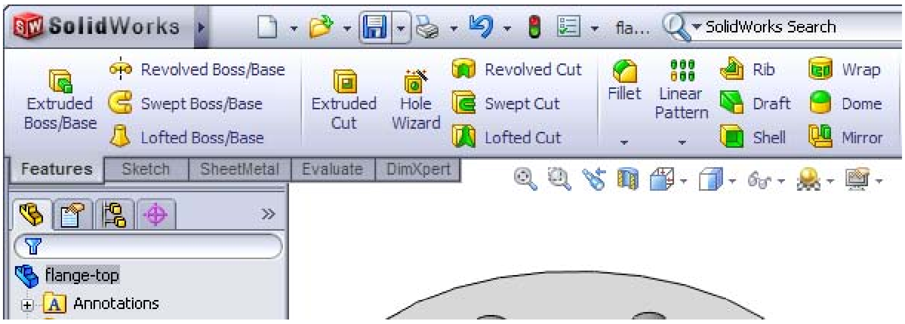
<table>
<tr>
<td>72</td>
<td>Are you sure you have already saved the changes in this model? Just to be sure, do it now by clicking Save in the Toolbar.</td>
<td>
</td>
</tr>
<tr>
<td>73</td>
<td>Make a copy now:
1. Click on the arrow next to Save in the Toolbar.
2. Click on ‘Save As…’.</td>
<td>
</td>
</tr>
<tr>
<td>74</td>
<td>1. Change the name of the file to ‘flange-top.SLDPRT’.
2. Click on ‘Save’.
You have renamed the file now and we will continue to work in it.</td>
<td>
</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>Configuration of Copy? While making the standard we used two configurations, and now we are making a copy. Why?

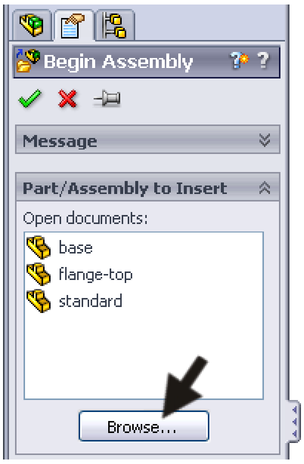
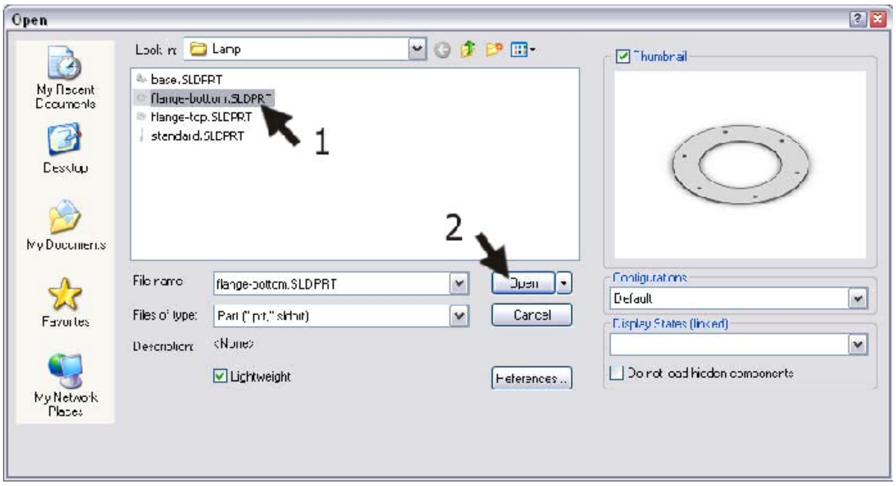
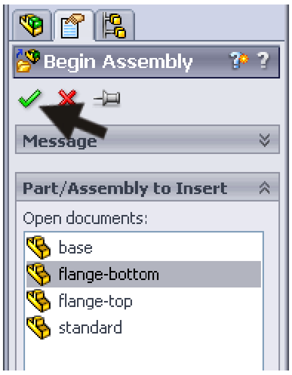
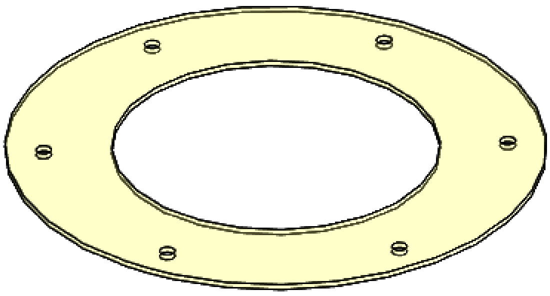
A configuration is especially useful for parts that are mainly the same AND must stay that way. The standard is a good example. Should you decide to change the height, it must be done in both parts. A configuration is a very convenient way to do this.

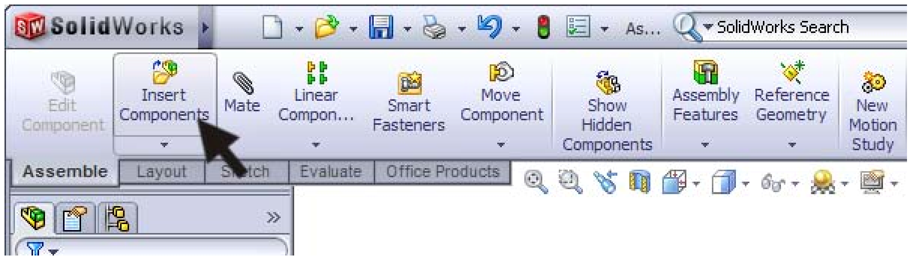
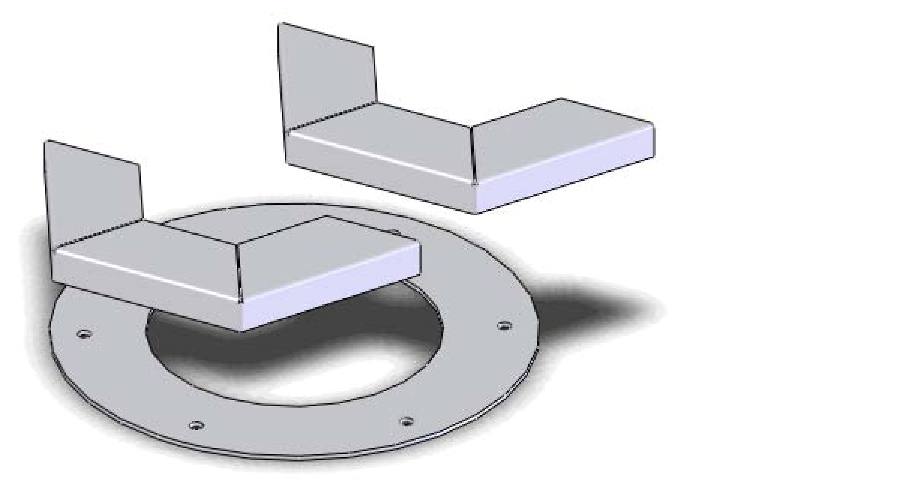
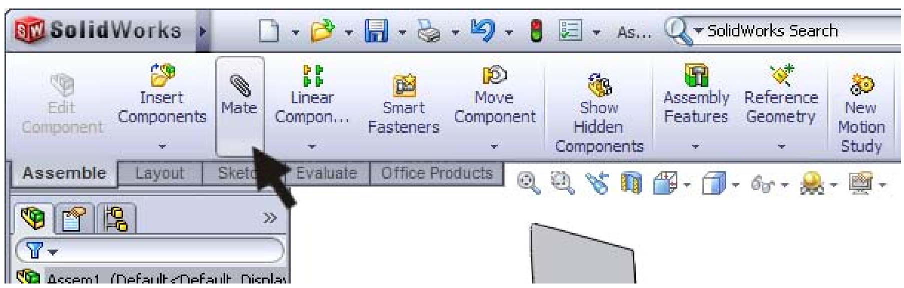
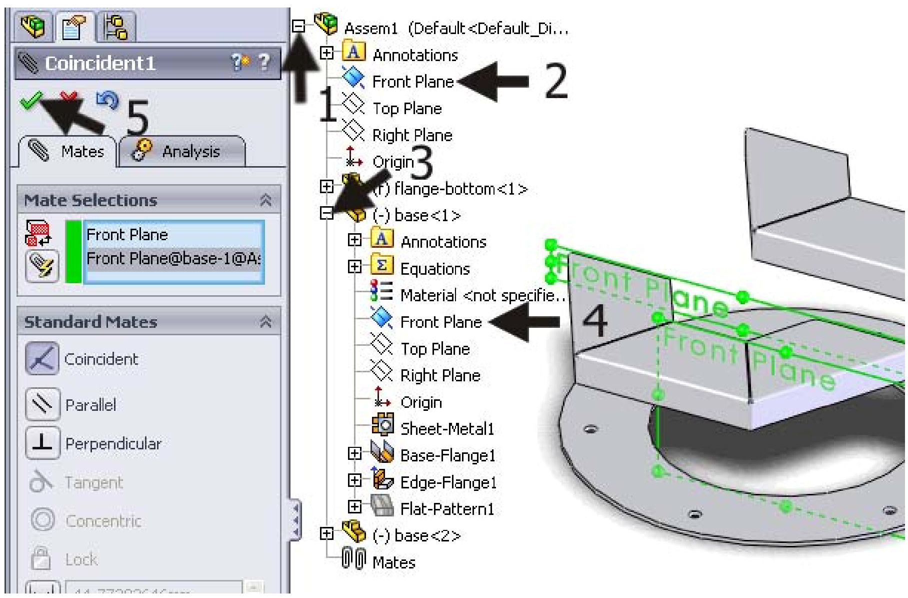
The upper- and lower flange have no relation to each other. That is why it is more convenient to make separate files by copying the first one.</td>
</tr>
</table>

75	Click somewhere on the plate. You will see the dimensions appear.	
76	Click on the smallest dimension (Ø170). A small menu appears. Change the size to ‘22mm’ and push the <Enter> key.	
77	Similarly, change the size from 280 to 90mm. Click somewhere beside the model to end the command.	
78	In the FeatureManager you will see a red ‘x’ next to the last feature: this means an error has occurred. Move the cursor to the feature. You will see a short explanation of the error. In this case it says: “The intended cut does not intersect the model.” Why? By changing the size of the ring, the six mounting holes are now outside the perimeter of the ring and are therefore useless.	

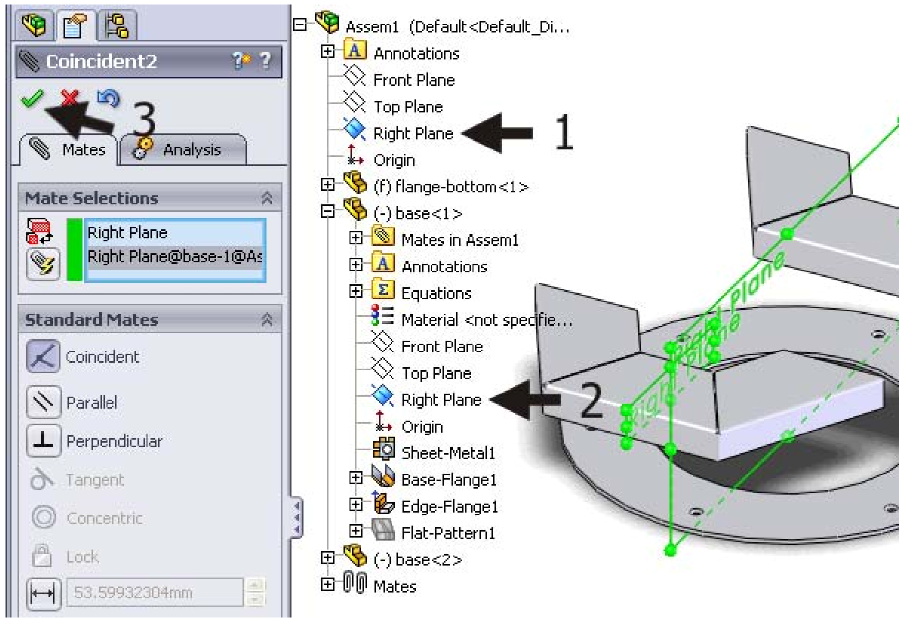
79	1. Click on the ‘+’ symbol before the hole feature (‘Extrude2’) in the Fea- tureManager. 2. Click on the sketch that appears. In the model you can see the holes now, which are very clearly outside the flange.	 
	Tip!	Sooner or later you will receive errors in SolidWorks. Every change you make will mean that SolidWorks recalculates the entire model and looks to see if everything is still ‘logical’. If not, an error occurs. What can go wrong? You have just seen an example: by changing the size of the ring, the holes ‘drop out’. This is something that SolidWorks ‘does not understand’. Another very frequent problem involves making a sketch on a plane in a feature and then discarding the feature afterwards. SolidWorks will not know on which plane the sketch should be positioned. There are a number of other reasons why errors occur, as you most likely can imagine. When you see an error, try to solve the problem. Your first reaction may be: ‘I better draw this part again,’ but it saves you a lot of time if you become smarter at solving problems and deleting errors. In the FeatureManager you can always see exactly where the problem is. In step 79 you can see this too: marked with a red x and red text. You can easily see in which feature or sketch the error is.
80	Change the size from 120mm to 30mm. You can do this by clicking on the dimension and filling in the new value OR by dragging the blue sphere at the end of the ruler (set to 120 mm).	 

81	Also, change the hole sizes from Ø8 to Ø6.5mm.	
82	The model has now been changed, and the error has disappeared from the FeatureManager. Save the file. Use the Save command in the standard Toolbar.	

	Work plan	All parts of the base of the garden light are ready. We can now make an as- sembly of them. Because all parts have their midpoint at the origin, we can use the Front and Right planes for mating a lot of the parts. By combining these planes for all of the parts, their positions are already determined. We only have to set the height.
83	Open a new assembly.	
84	First, we must choose the part ‘flange-bottom’. This is probably not open at this point. Therefore, click on ‘Browse...’.	
85	1. Select the file ‘flange-bottom.SLDPRT’. 2. Click on ‘Open’.	
86	Do NOT click randomly to place the part, but click on OK in the PropertyManager. The part will be placed ex- actly on the origin.	 

87	Click on ‘Insert Compo- nents’ in the CommandMa- nager to place the next part in the assembly.	
88	Add the file ‘base.SLDPRT’ twice. Put these parts at a random position in the drawing.	
89	We will add mates now. Click on ‘Mate’ in the Com- mandManager.	
90	Because all parts are built around the origin, we can use the Front and Right planes to set the mates. You can select these planes in the FeatureManager, which is shown next to the model. 1. Open the FeatureMa- nager. 2. Select ‘Front Plane’ from the assembly. 3. Click on the ‘+’ symbol in front of part ‘base<1>’. 4. Select the ‘Front Plane’ from ‘base<1>’.	

<table>
<tr><td></td><td>SolidWorks chooses the mate ‘Coincident’ automati- cally

5. Click on OK.</td><td></td></tr>
<tr><td>91</td><td>Repeat step 90, but use the ‘Right Plane’ from the as- sembly and from ‘base<1>’.</td><td>
</td></tr>
<tr><td>92</td><td>We will do the same with ‘base <2>’:

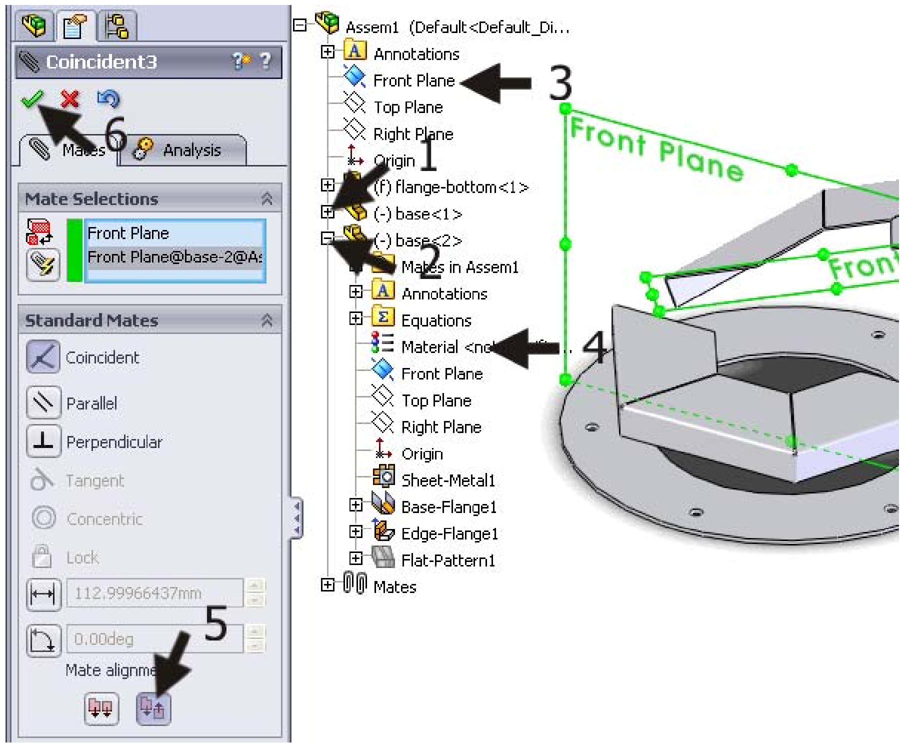
1. Close the ‘base<1>’ command tree, or else the list will be very long. Click on the minus symbol in front of ‘base<1>’.

2. Open the command tree from ‘base<2>’. Click on the ‘+’ symbol in front of ‘base<2>’.

3. Select the ‘Front Plane’ from the assembly.

4. Select the ‘Front Plane’ from ‘base<2>’.

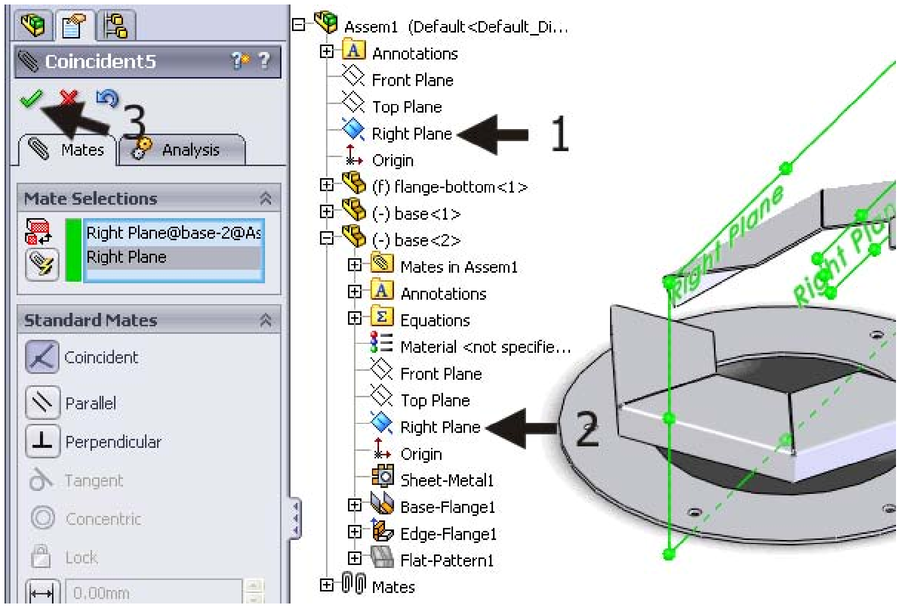
The part now has to be turned around:

5. Click on anti-aligned in the PropertyManager.</td><td>
</td></tr>
</table>

93

Next, mate the Right planes:

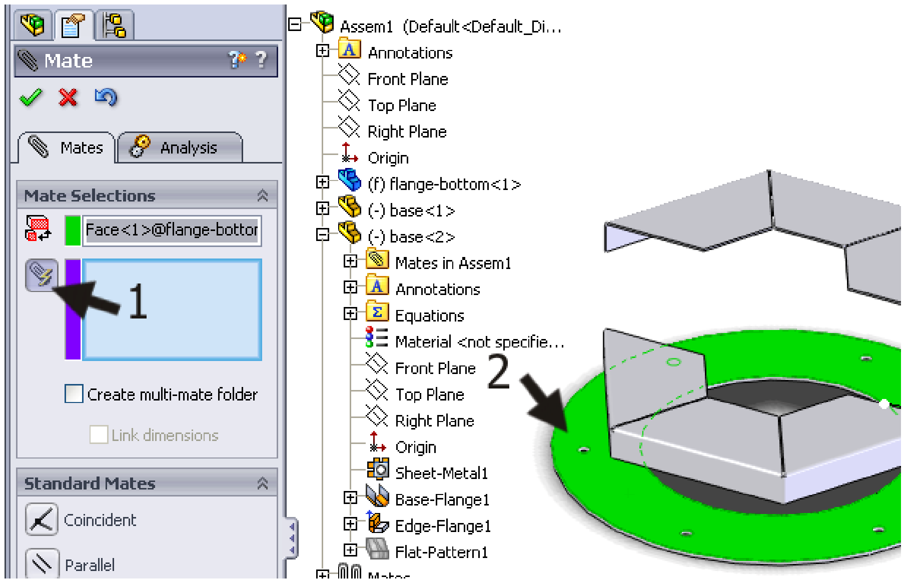
1. Select the ‘Right Plane’ from the assembly.
2. Select the ‘Right Plane’ from ‘base<2>’.
3. Click on OK.



94

Next, we have to mate the parts to place them at the same height:

1. Click on Multiple Mate Mode in the Property- Manager.
2. Select the top from the bottom plate.

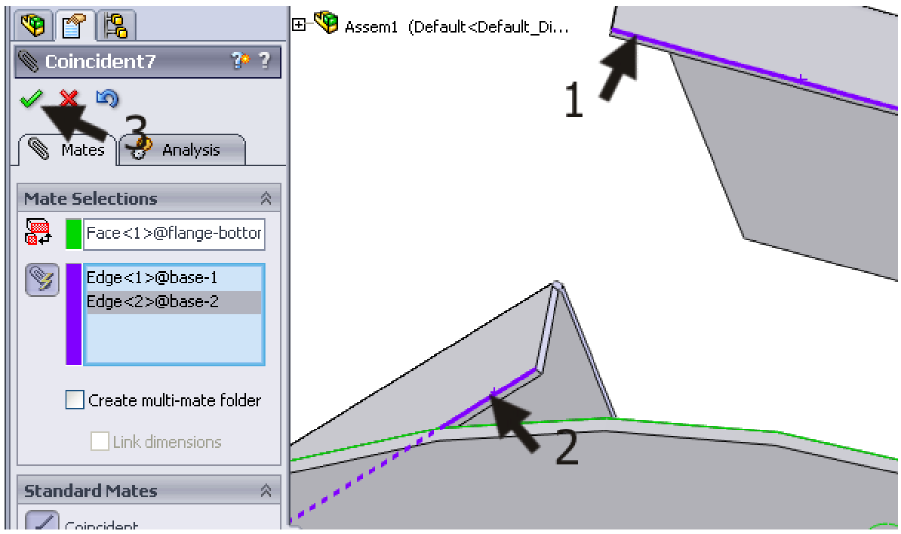


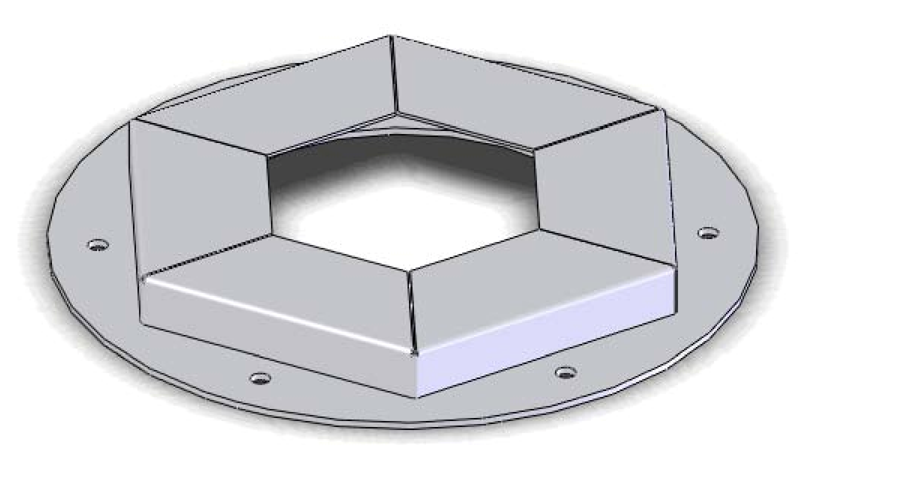
95

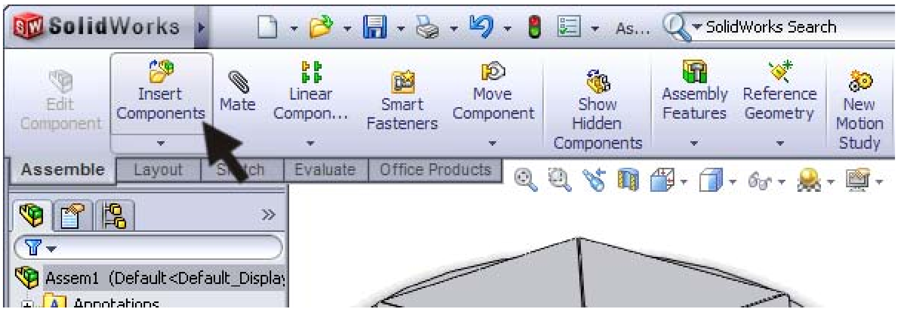
Rotate the model and zoom in.

1,2 Select an edge from the bottom of ‘base<1>’ and ‘base<2>’.

3. Click on OK.
4. Click OK again to close the Mate command.

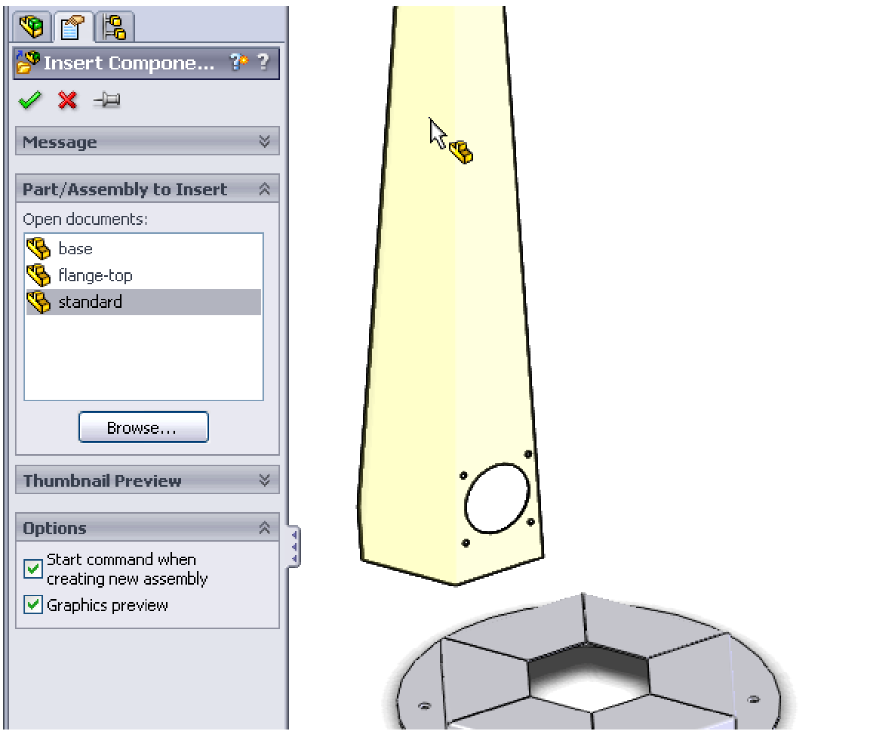


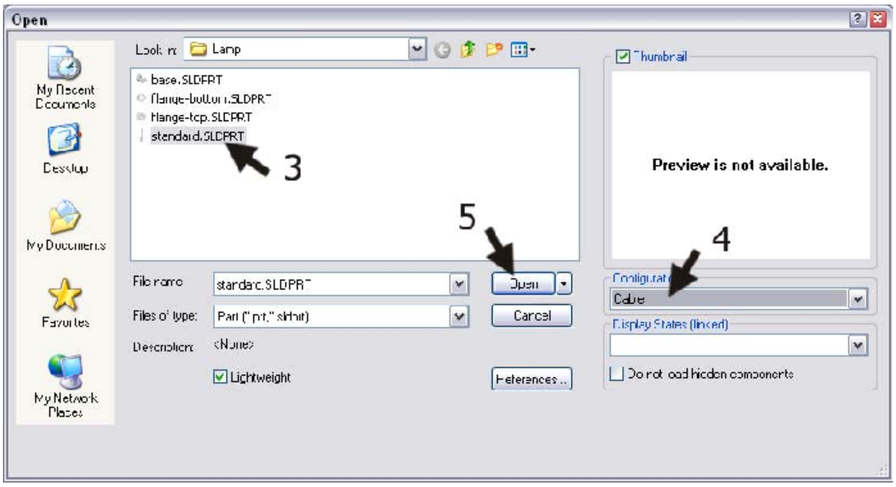
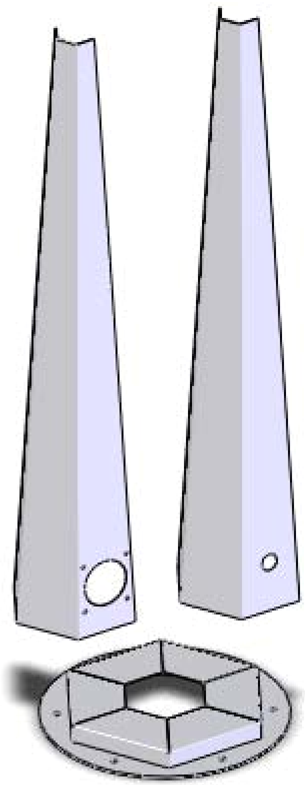
<table>
<tr>
<td>96</td>
<td>These three parts are now fixed.</td>
<td></td>
</tr>
<tr>
<td>97</td>
<td>We will add the standard to the assembly too.

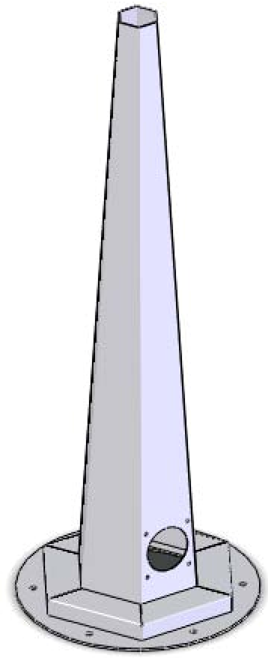
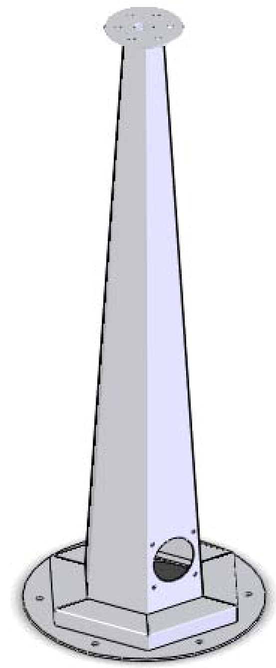
Click on ‘Insert Compo- nents’ in the CommandMa- nager.</td>
<td>
</td>
</tr>
<tr>
<td>98</td>
<td>When the part stan- dard.SLDPRT is still open, you can see it in the list in the PropertyManager.

1. Click on the part called standard.SLDPRT.

2. Put it at a random posi- tion in the model.

If you closed the file before, find it by using ‘Browse…’.</td>
<td>
</td>
</tr>
</table>

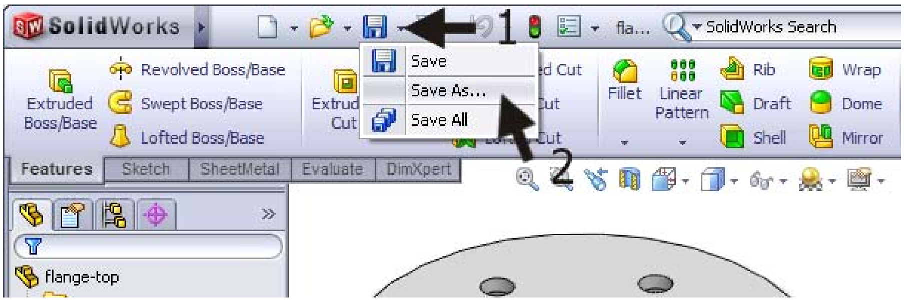
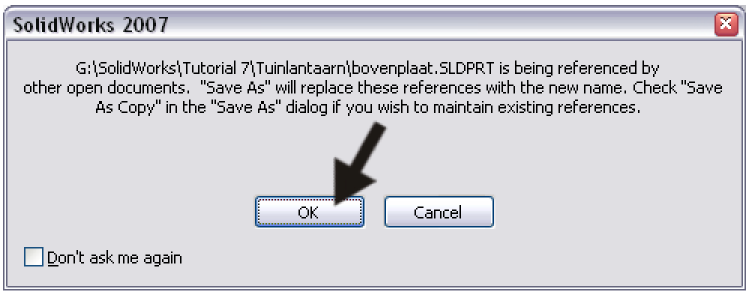
99	From this part we have made two configurations: ‘Cable’ and ‘Socket’. Most likely you have used the configuration ‘Socket’ in step 98 (the one with the big hole and four small holes). We have to put in the other configuration as well. 1. Click on ‘Insert Com- ponents’ in the Com- mandManager again. 2. Click on ‘Browse...’ in the PropertyManager. 3. Select the file ‘stan- dard.SLDPRT’ in the menu that appears	
100	Put this part in the assem- bly as well.	

101	Add mates in exactly the same way as you did be- fore. Follow steps 89 to 96. On the right you see the result.	
102	Finally, the flange-top must be added. For this you create mates using the Front and Right planes.	
103	Save the assembly as stan- dard- complete.SLDASM	
	Work plan	We will get started with the lamp shade. We will create the base plate first. As you can see in the illustration it looks a lot like the upper plate of the base of the light. Therefore, we can make a copy of this part and change it.

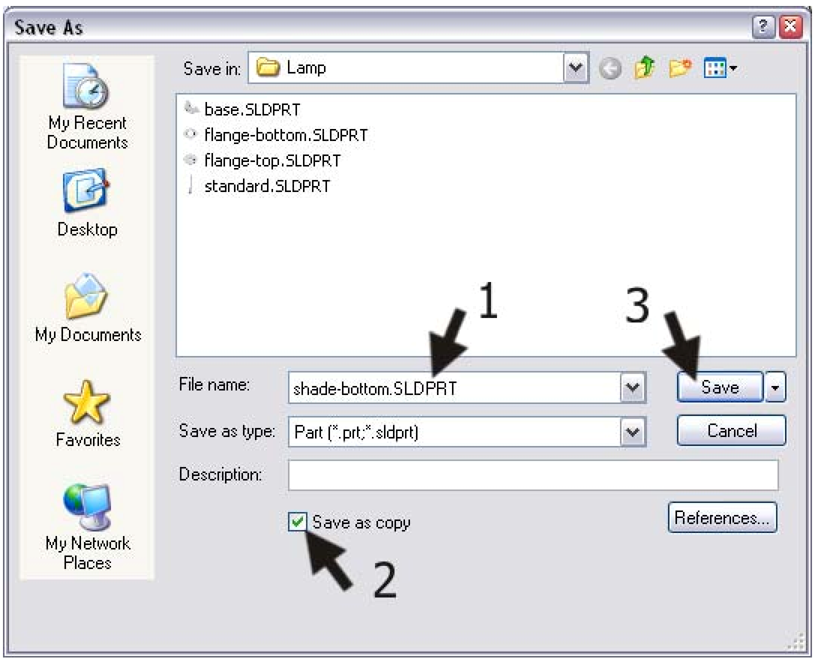
<table>
<tr><td></td><td></td><td></td></tr>
<tr><td>104</td><td>Open the file ‘flange-top’. Are you sure you have saved all changes? Just to be sure, click on ‘Save’ in the Toolbar first.

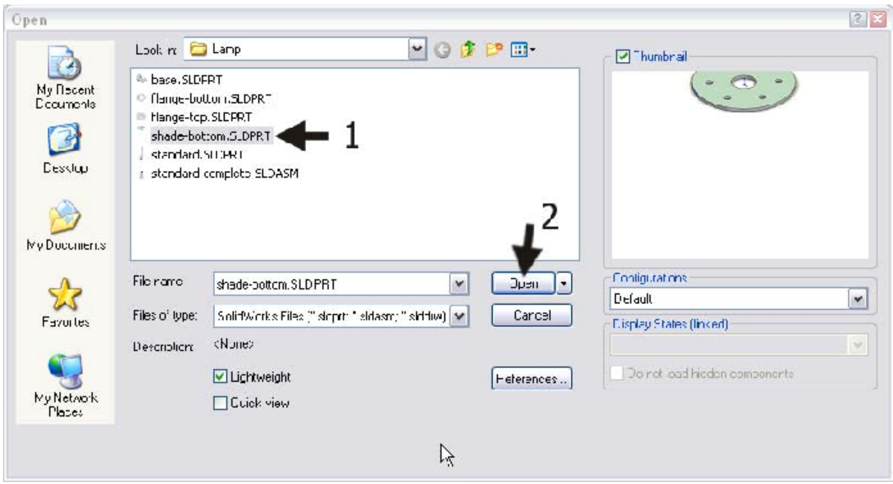
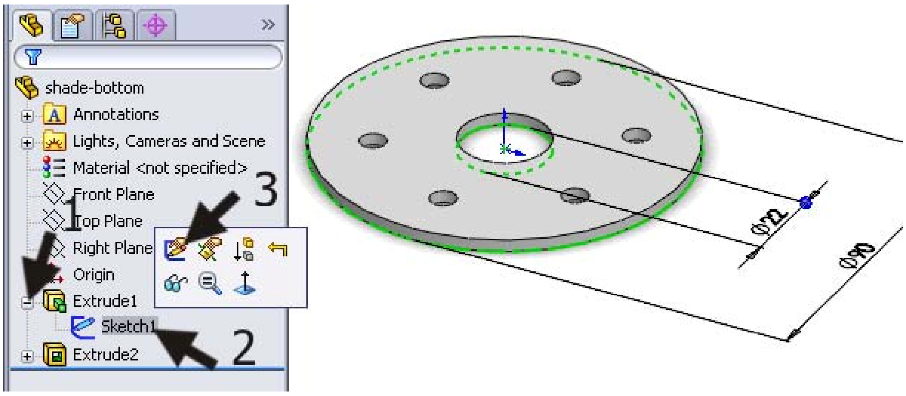
Let’s make a copy now:

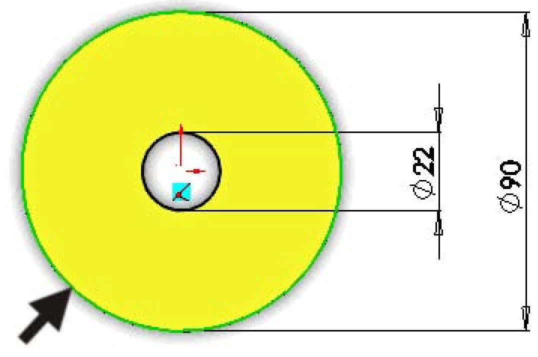
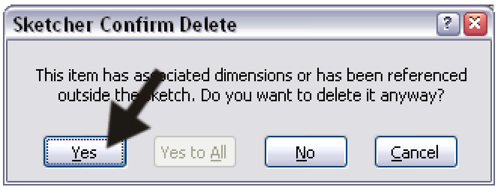
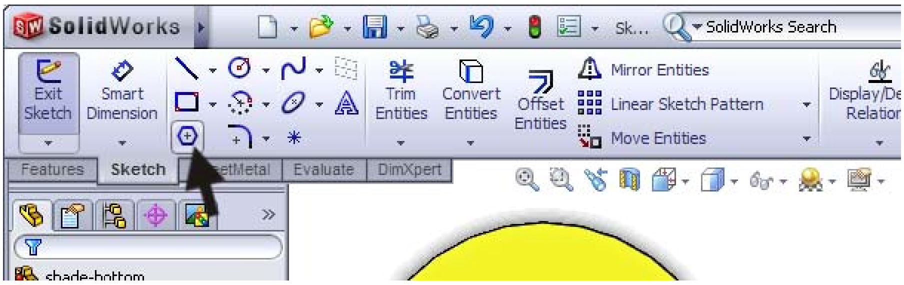
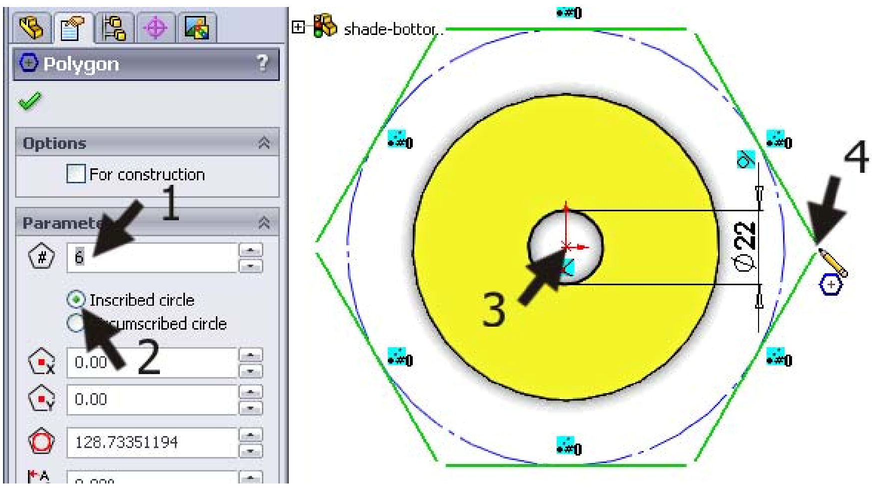
1. Click on the arrow next to ‘Save’.

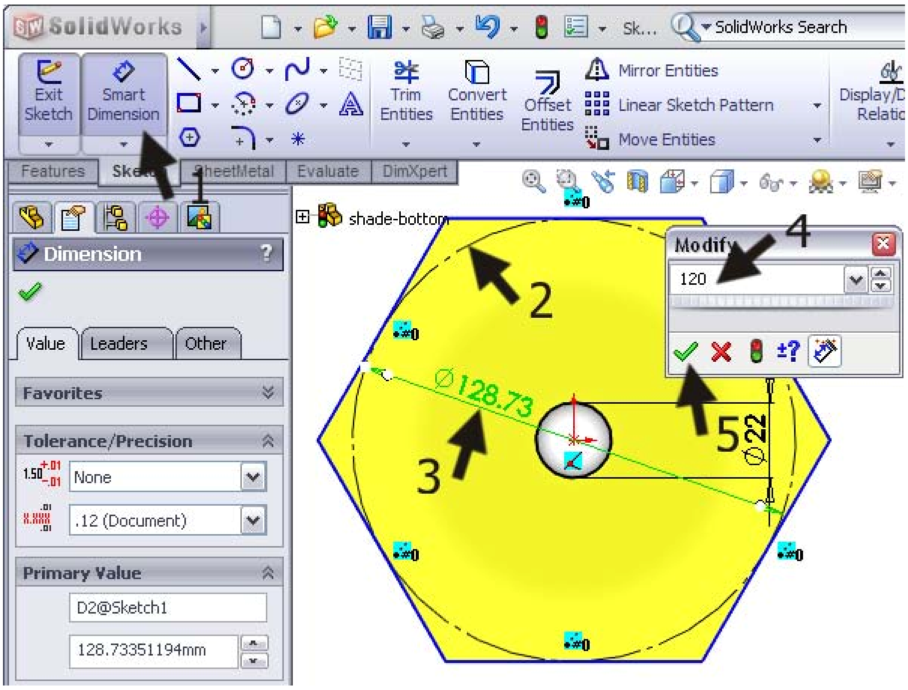
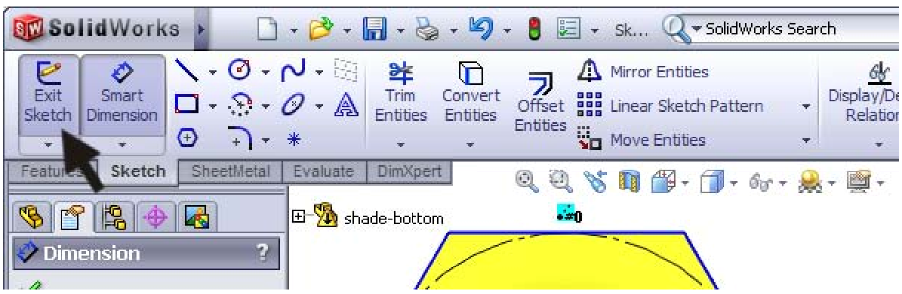
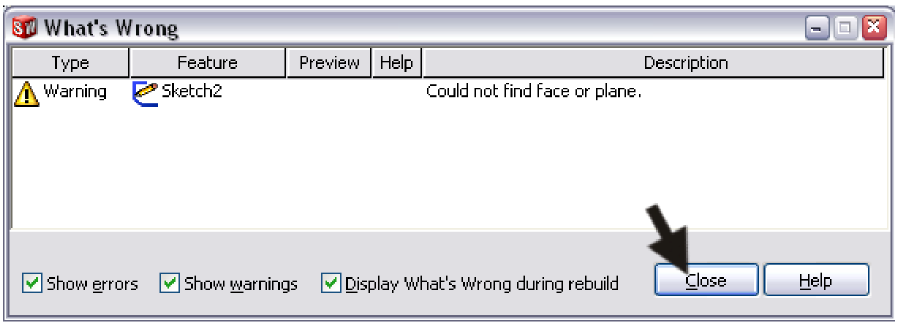
2. Click on ‘Save As...’.</td><td>
</td></tr>
<tr><td>105</td><td>When this message appears, click on OK.</td><td>
</td></tr>
<tr><td>106</td><td>1. Rename the file as ‘shade-bottom’.

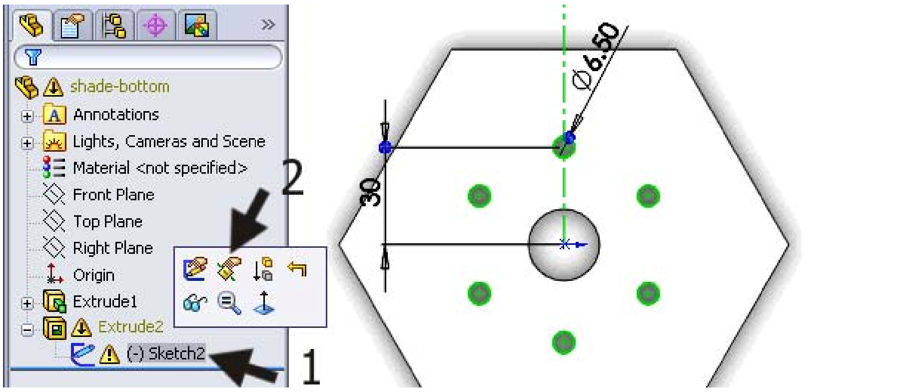
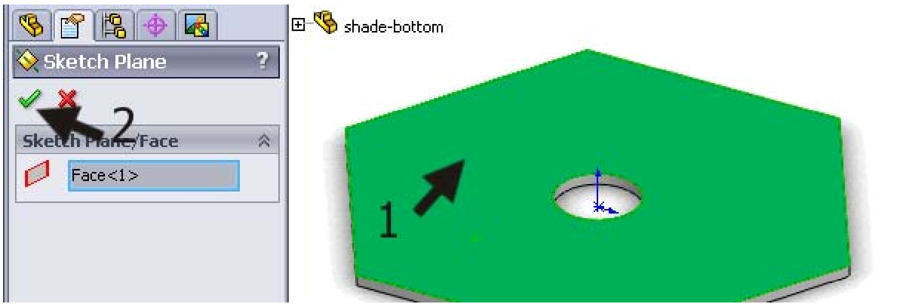
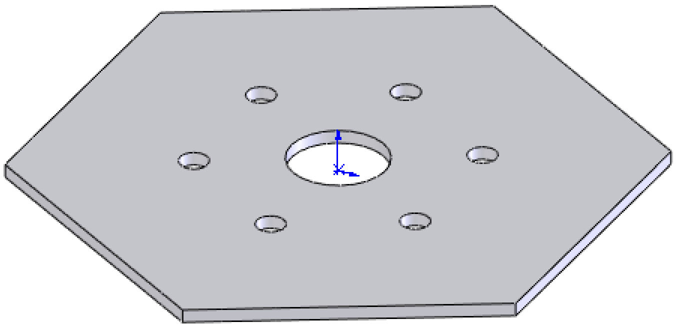
2. IMPORTANT: check the option ‘Save as copy’.

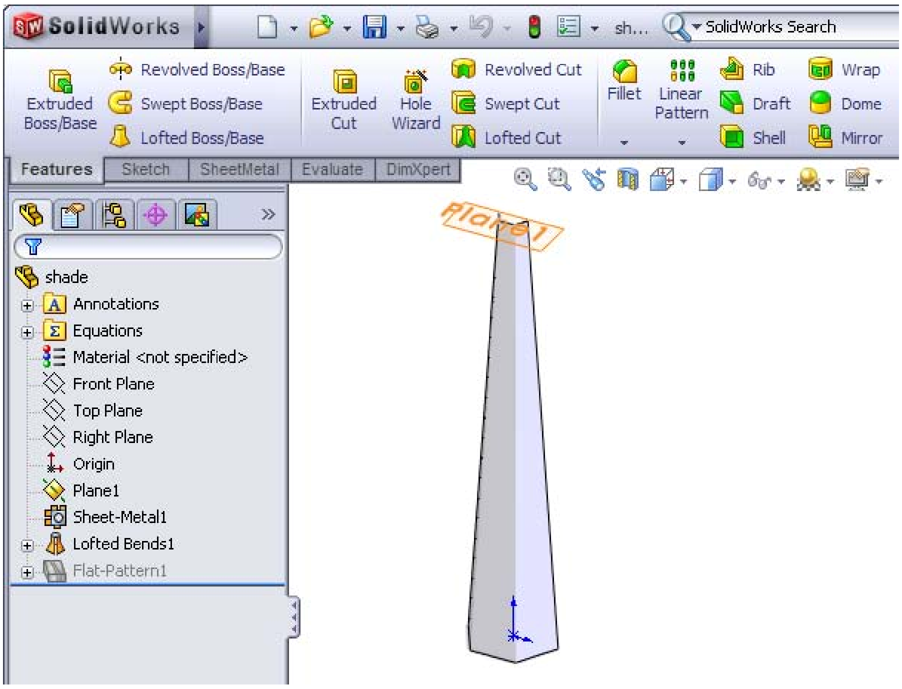
3. Click on ‘Save’.</td><td>
</td></tr>
</table>

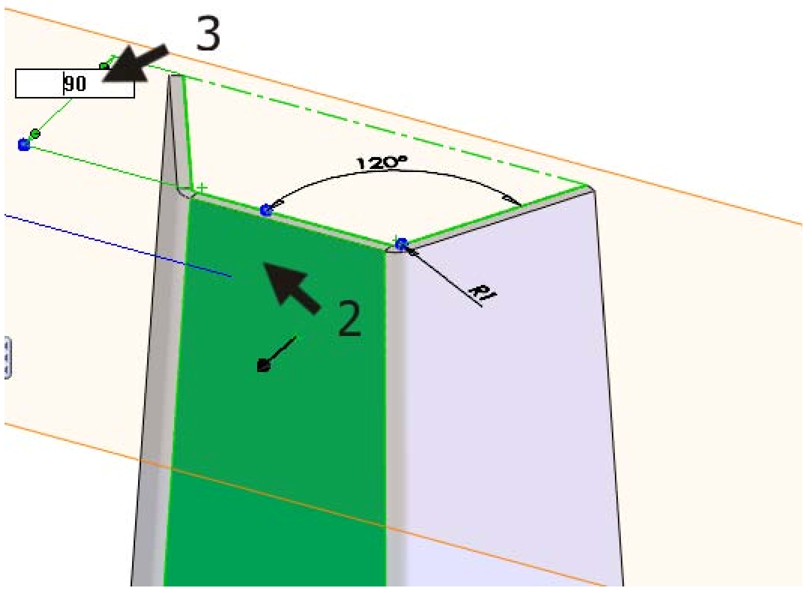
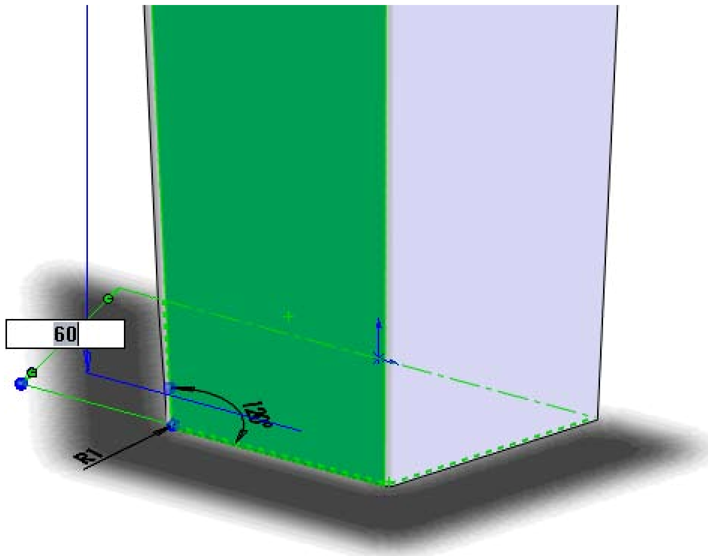
	Tip!	What does the option ‘Save as copy’ mean? The file ‘flange-top’ is used in the assembly that we previously. If you would change the name of this part with ‘Save As…’ the name in the assembly would also change. In this case, we do not want that to happen because it would mean that the ‘flange-top’ in the assembly would be replaced by the part we just made named ‘shade- bottom’. By using ‘Save as copy’ the assembly stays the same. The new file has ab- solutely nothing to do with it.
	Tip!	If this seems too complicated for you, you can also use the Windows Ex- plorer to copy the file and rename it. To do so, however, you have to close the file in SolidWorks first. Pay attention: NEVER rename a part that is used in an assembly in Windows Explorer. The assembly will not be able to find this part again and you will get multiple, unsolvable errors.
107	The file ‘shade-bottom’ has been made but has not been opened yet. Do this now before you continue!	
108	1. Click on the ‘+’ symbol in front of the first fea- ture (‘Extrude1’). 2. Right-click on ‘Sketch1’. 3. Select Edit Sketch in the menu. Rotate the sketch with Normal To.	

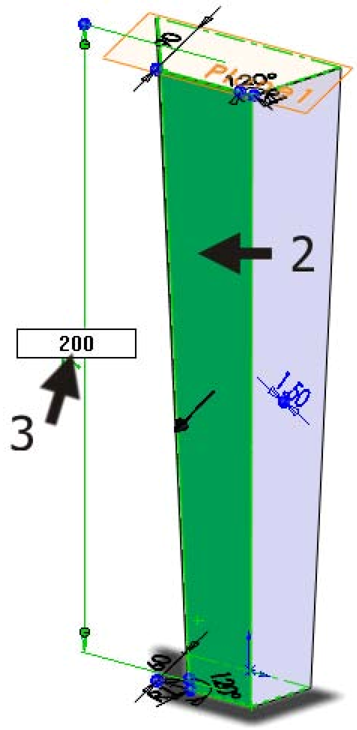
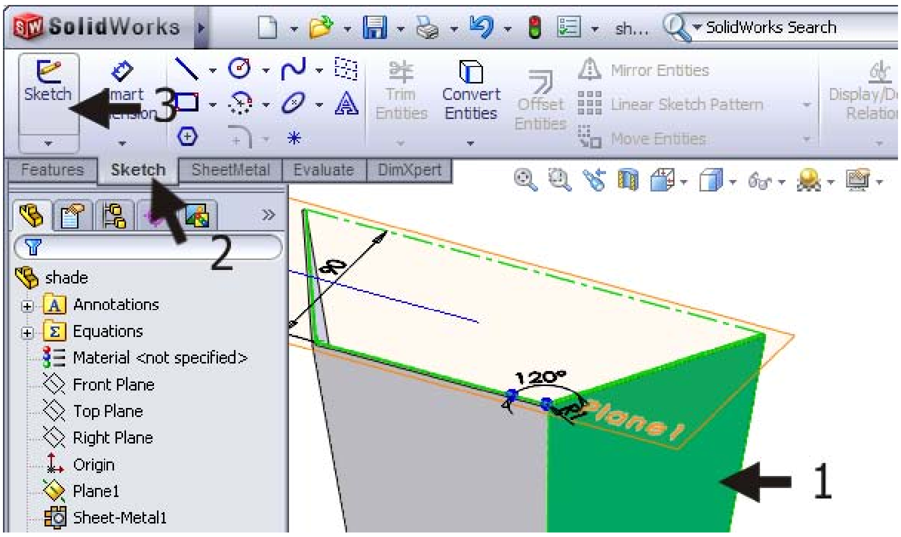
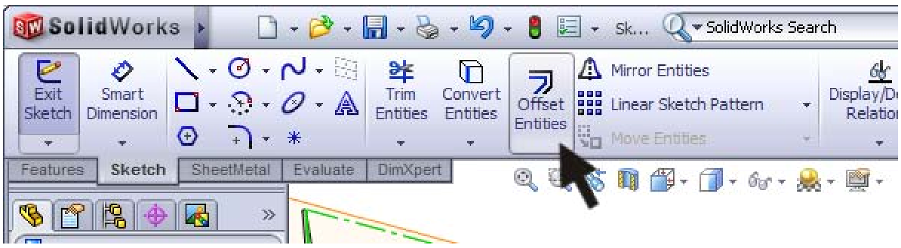
109	Click on the outer circle of the sketch and push the <Del> (delete) key.	
110	Click 'Yes' in the message that appears.	
111	Click on Polygon in the CommandManager.	
112	1. Set the number of sides to '6'. 2. Make sure the option 'Inscribed circle' is se- lected. 3. Click on the origin. 4. Click beside the origin, horizontally to the ori- gin. The distance does not matter.	

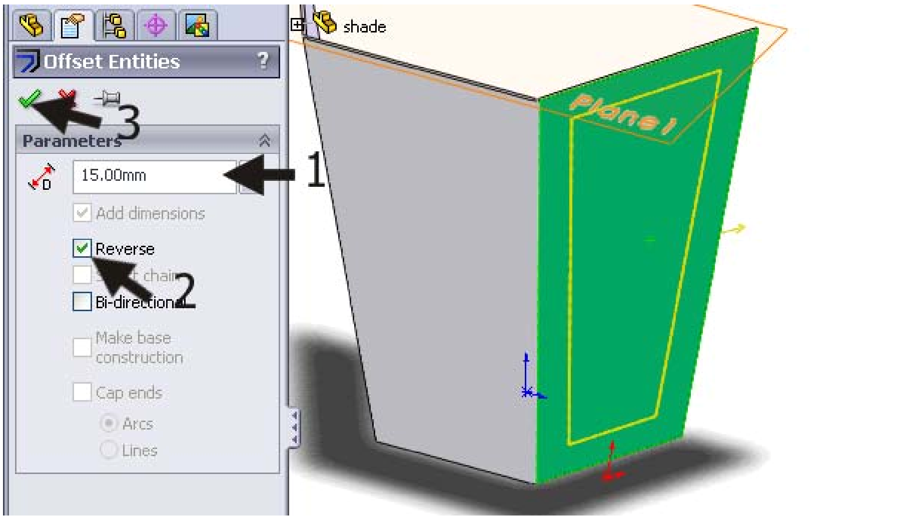
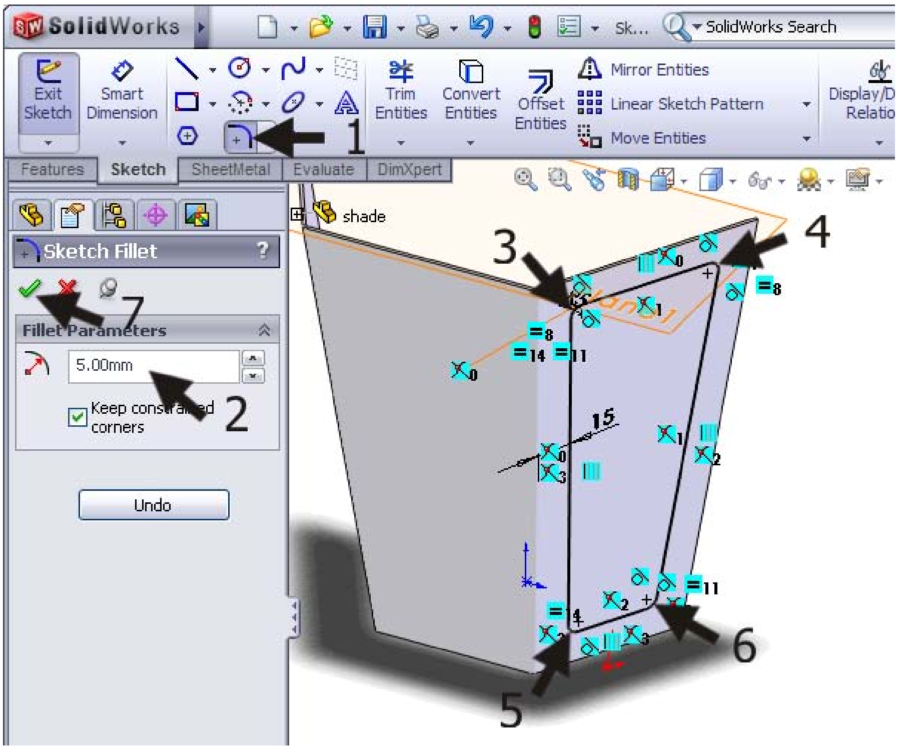
113	Set the size of the inside circle with Smart Dimen- sion. 1. Click on ‘Smart Dimen- sion’ in the Command- Manager. 2. Click on the inner circle. 3. Set the dimension. 4. Change the value to ‘120mm’ . 5. Click on OK.	
114	The sketch is now done. Click on ‘Exit Sketch’ in the CommandManager.	
115	At this point, an error oc- curs! Why? You have just changed the first feature from this part (the plate). In this part there were six mounting holes. By changing the first feature, SolidWorks does not know in which plane the sketch of the holes was drawn. Click on ‘Close’.	

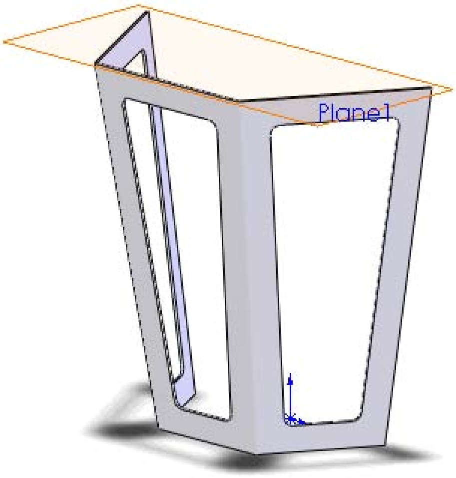
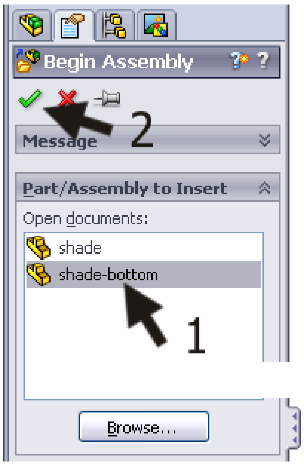
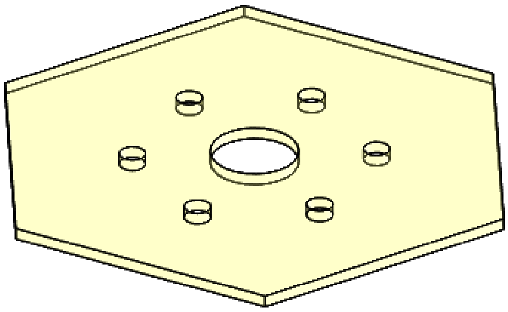
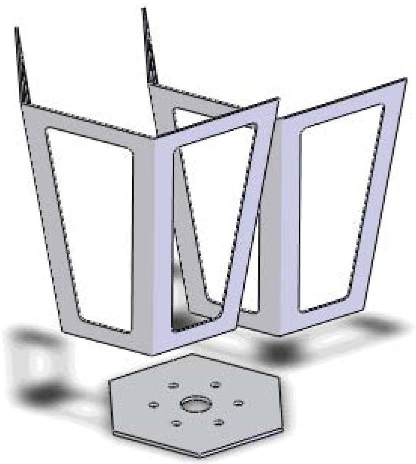
116	We are going to determine a new plane, on which the sketch of the holes has to be placed. Right-click on the sketch of the six holes. Select Edit Sketch Plane in the menu that appears.	
117	1. Click somewhere on the top plane of the model. 2. Click on OK in the Pro- pertyManager.	
118	The error has disappeared, and the part is ready. Save the file by using the Save button in the Toolbar.	

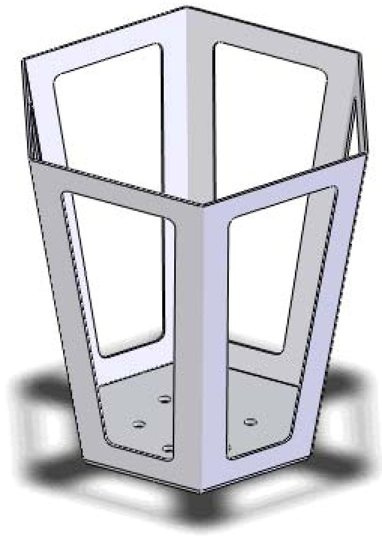
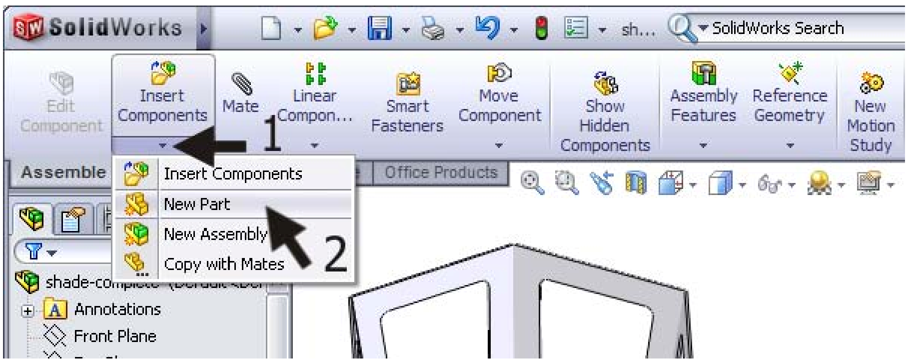
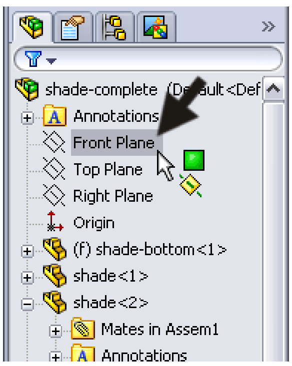
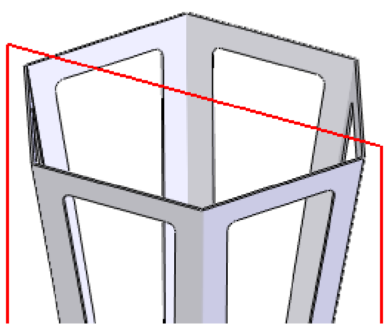
	Work plan	We will start drawing the side wall of the shade now. The construction is identical to the standard. This part must also be made with the Lofted-Bend command. To save us a lot of work we will use a copy of the standard and change this to fit our needs. We have to remove a few items from that file, however, such as the holes we made at the bottom and the configurations. After that we can resize the part and open the sidewalls.
119	Open the file shade.SLDPRT. This file is saved in step 47.	

120	We have to change a number of dimensions in the model. 1. Zoom in at the top of the model. 2. Click at a random point. 3. Click on the size of 20mm and change it to 90mm.	
121	1. Zoom in at the bottom of the model. 2. Click on the model again. 3. Click on the size of 65mm and change this to 60mm.	

122	1. Zoom out, in order to get a clear view at the whole model. 2. Click on the model. 3. Click on the dimension 740mm, which indicates the height. Change it to 200mm.	
123	We will now make the openings in the sidewalls. 1. Select one of the sidewalls. 2. Click on 'Sketch' in the CommandManager. 3. Open the sketch.	
124	Click on 'Offset Entities' in the CommandManager.	

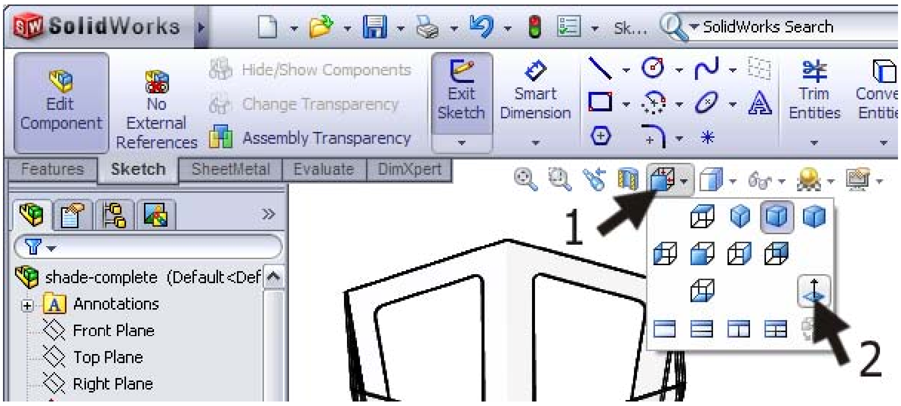
125	1. Set the distance for the offset to '15mm'. 2. Click on the option 'Reverse' (when neces- sary), in order to show the yellow line at the inside of the plane. 3. Click on OK.	
126	1. Click on Sketch Fillet in the CommandManager. 2. Set the radius to '5mm' in the PropertyManag- er. 3-6.Click on the four cor- ners of the sketch. 7. Click on OK.	

127	Make a Cut-Extrude from this sketch. Set the depth to Through All. Repeat steps 123 to 126 in the two other planes of the model. This part of the shade is ready now. Save the file.	
	Work plan	Although not all parts of the shade are ready yet, we are ready to make the assembly because we can create the rest of the parts in the assembly itself more easily.
128	Open a new assembly. Add the flange-bottom file first. Do not put it at a random position, but by clicking OK, the part will be positioned directly at the origin.	 
129	Add the part shade.SLDPRT twice. Put these in random positions.	

130	Add mates by using the Front and Right planes. You have done this before in steps 87 to 93.	
131	Save the assembly as: shade-complete.	
	Work plan	At the top of the hood a metal strip has to be welded in. The problem is, that the size and the angled ends of the strip are very hard to calculate or determine. For this reason we will create the strip directly in the assembly.
132	1. Click on the arrow underneath 'Insert Components' in the CommandManager. 2. Click on 'New Part'.	
133	Click on the 'Front Plane' in the FeatureManager. In this plane you will make a first sketch of the strip.	 
	Tip!	You are modeling 'in-context' now: you are creating a part, which will be colored blue, while the assembly is transparent. You cannot change the assembly, but you can use it to add relations.

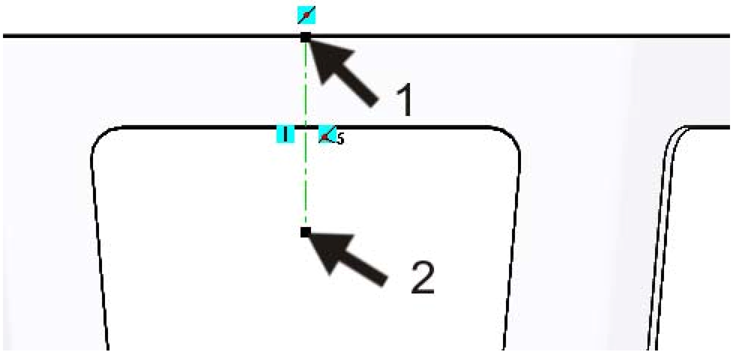
<table>
<tr>
<td>134</td>
<td>Rotate the model so you get a clear view at the sketch.

1. Open the rotate menu.

2. Click on Normal To.</td>
<td>
</td>
</tr>
<tr>
<td>135</td>
<td>Next draw a centerline.

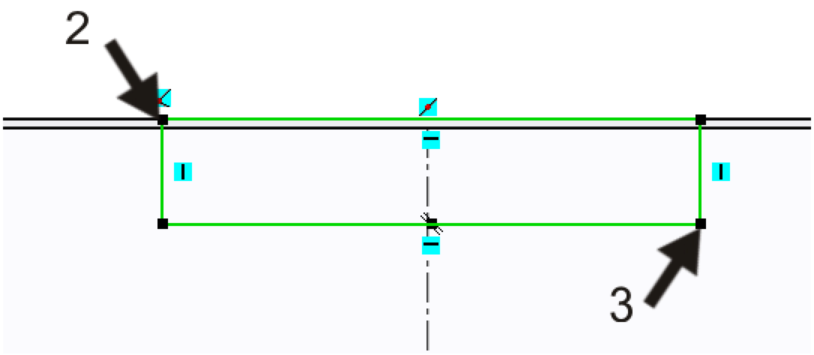
1. Click on the middle of the upper edge to set the first point. Be sure to find the midpoint, and check the symbols for this.

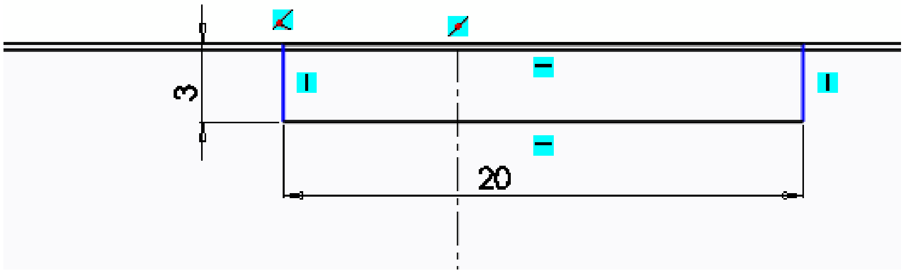
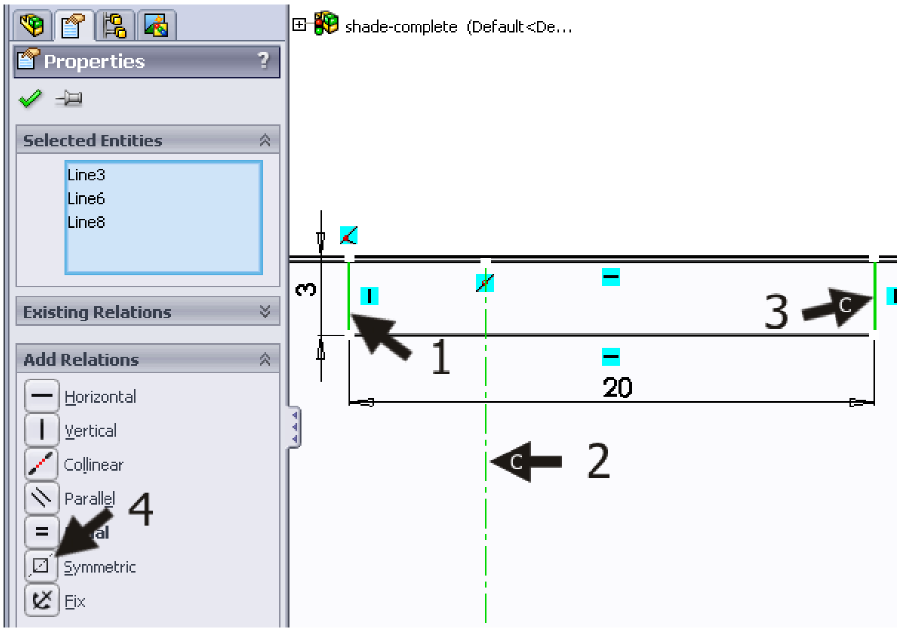
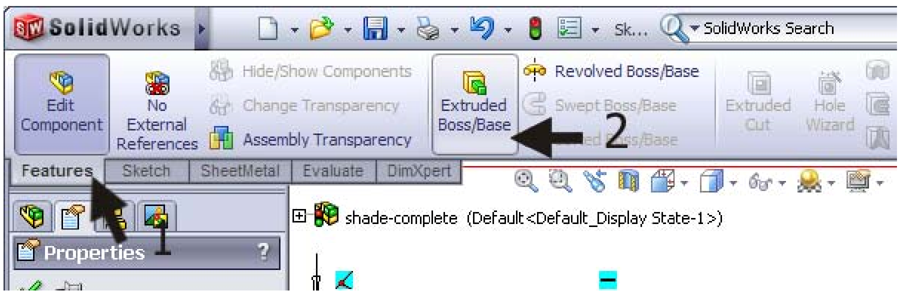
2. Click on a second point vertically underneath the first one.

3. Push the <Esc> key.</td>
<td>
</td>
</tr>
<tr>
<td>136</td>
<td>Draw a rectangle:

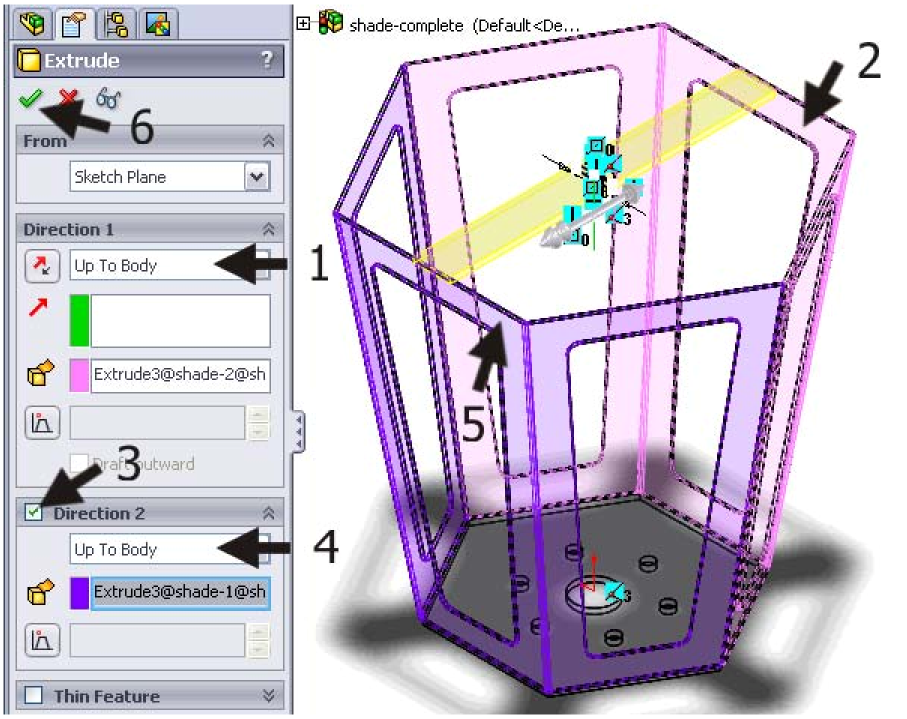
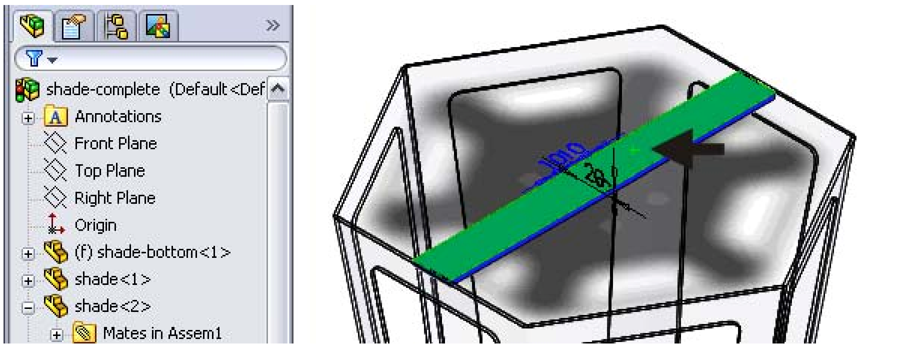
1. Zoom in as far as you can to see the two top edges because the planes are at a certain angle to the horizon (you are looking at the top side of the sheet-metal now).

2. Click at the upper line to set the first corner of the rectangle.

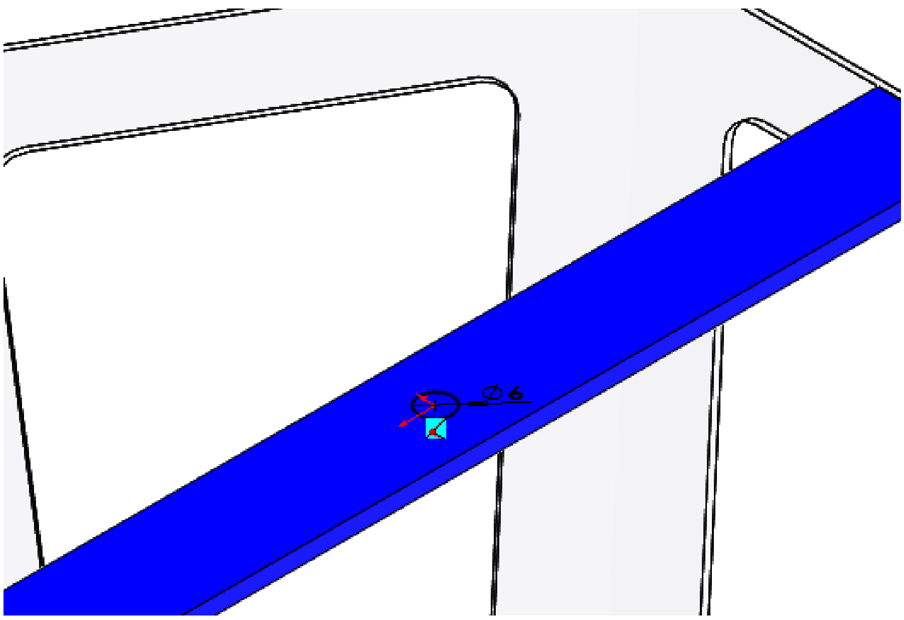
3. Click at a second point as indicated in the drawing to get the second corner.</td>
<td>
</td>
</tr>
</table>

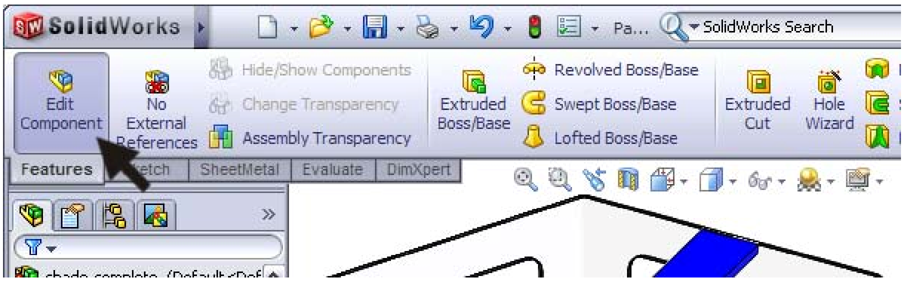
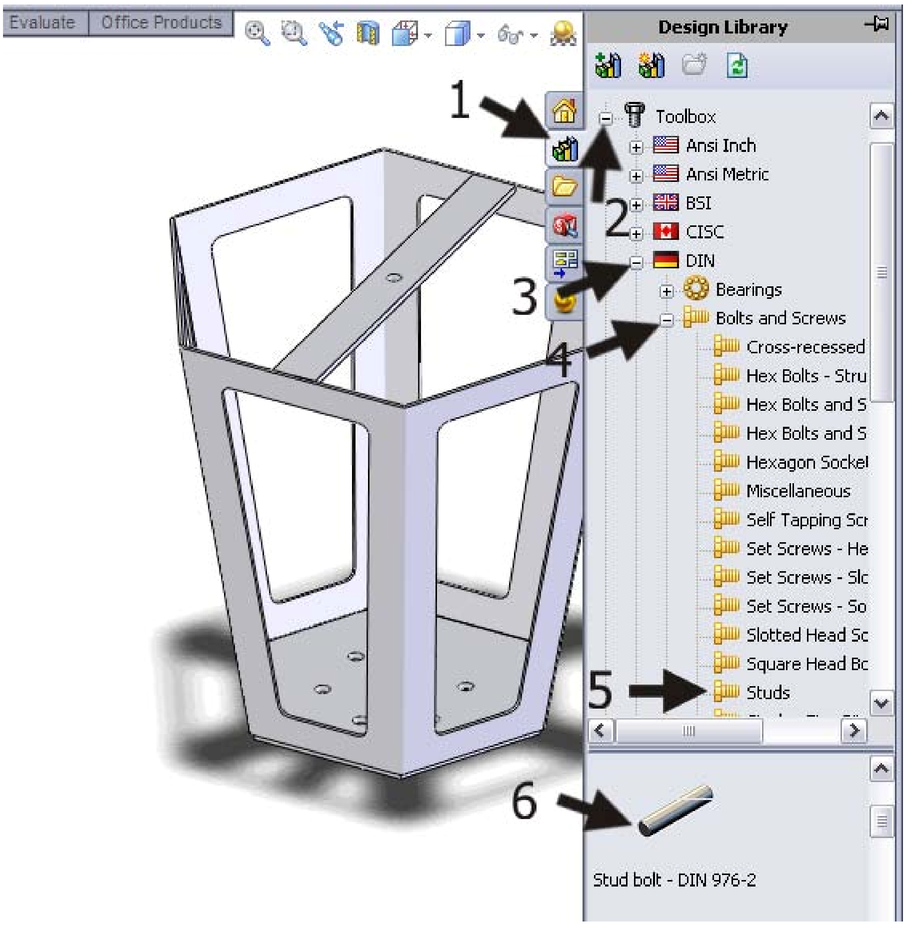
137	Set the dimensions by using Smart Dimension as shown in the illustration.	
138	Next, we will make the rectangle symmetrical to the centerline. 1. Select the left vertical side of the rectangle. 2. Push the <Ctrl> key and select the centerline. 3. Hold the <Ctrl> key and select the right side of the rectangle. 4. Click on 'Symmetric' in the PropertyManager.	
139	Click on 'Features' in the FeatureManager. Click on 'Extruded Boss/Base'.	

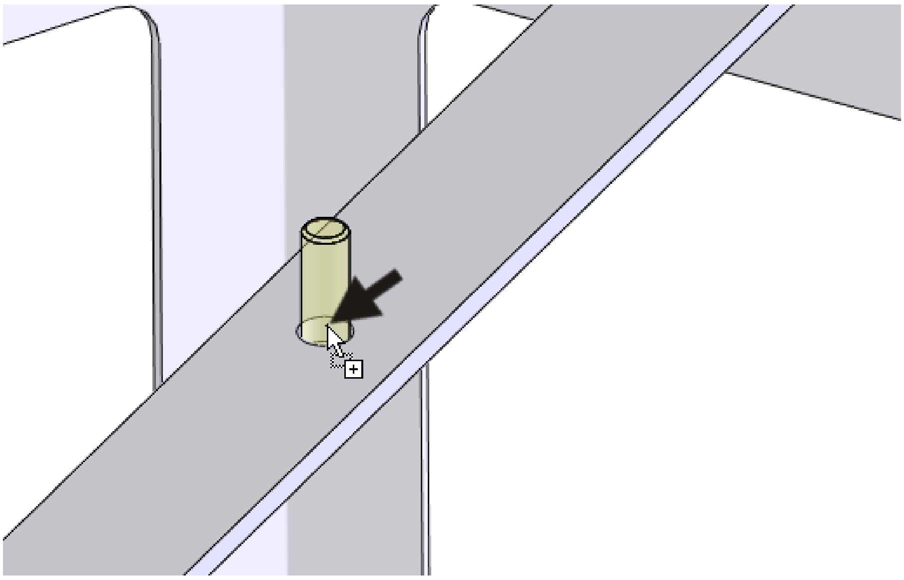
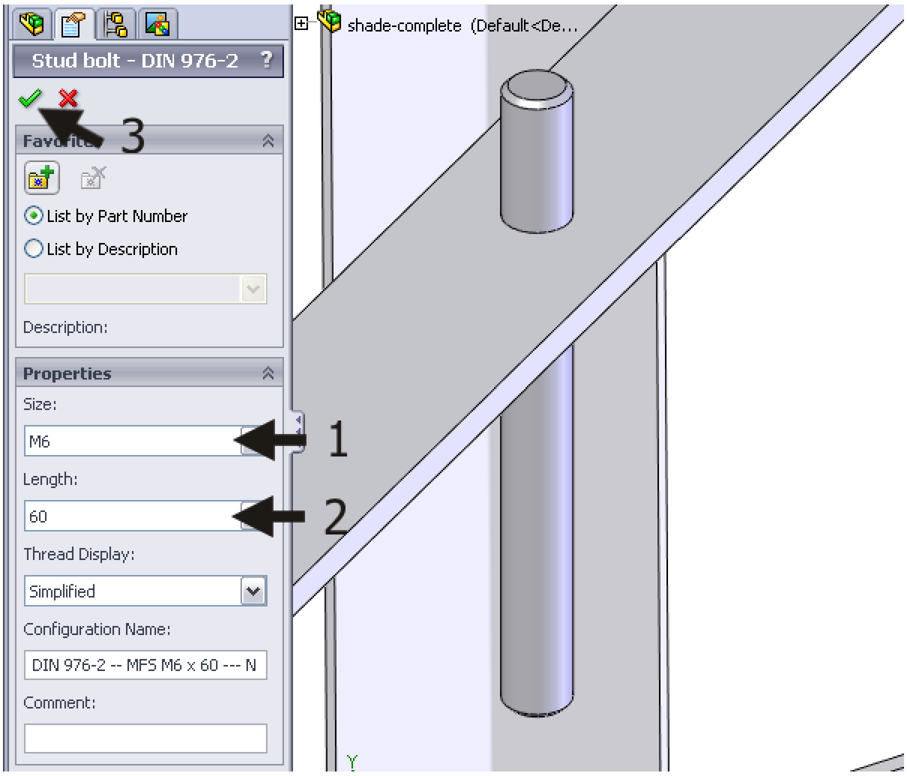
<table>
<tr>
<td>140</td>
<td>To make the extrusion set the following features:

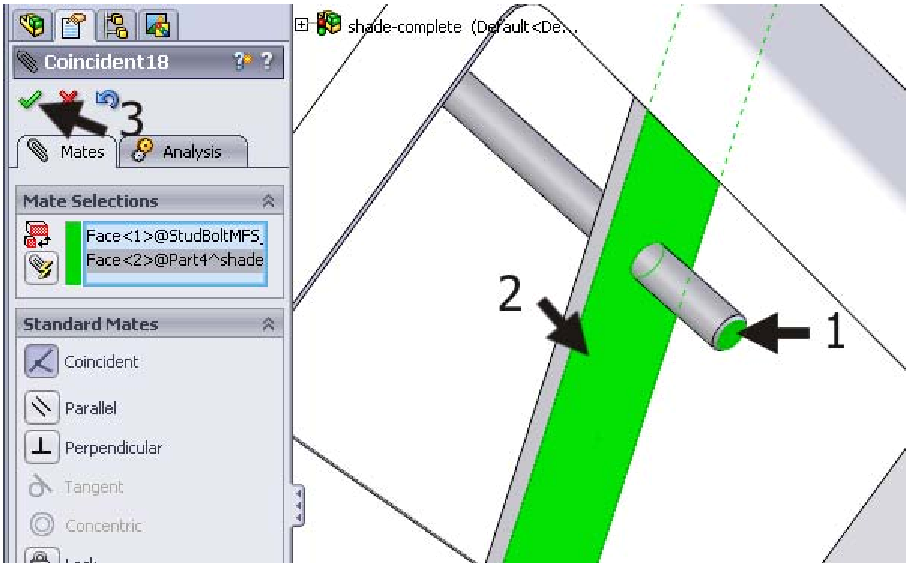
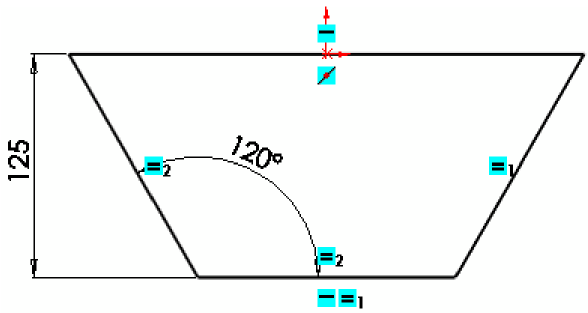
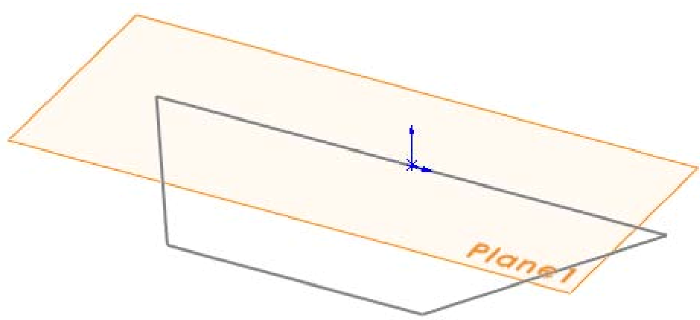
Select ‘Up to Body’ for ‘Direction1’.
Click on one side of the shade.
Check ‘Direction2’ in the PropertyManager to expand the sketch in two directions.
Select ‘Up to Body’ for ‘Direction2’ also.
Click on the other side of the shade.
When it looks OK to you, click on OK.
</td>
<td>
</td>
</tr>
<tr>
<td>141</td>
<td>
Select the upper side of the strip
Open the extended menu from the Com-mandManager when needed.
Click on Circle.
</td>
<td>
</td>
</tr>
<tr>
<td>142</td>
<td>Draw a circle, with its mid-point at the origin.

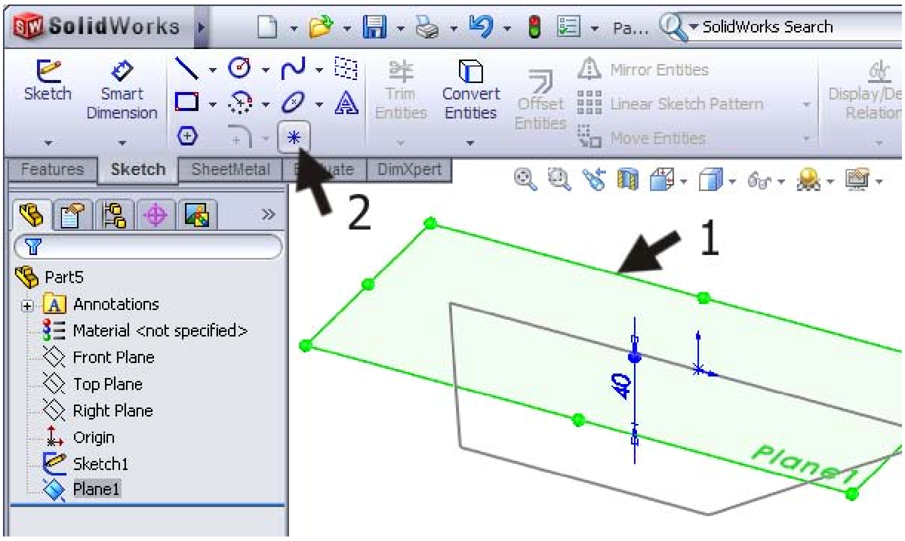
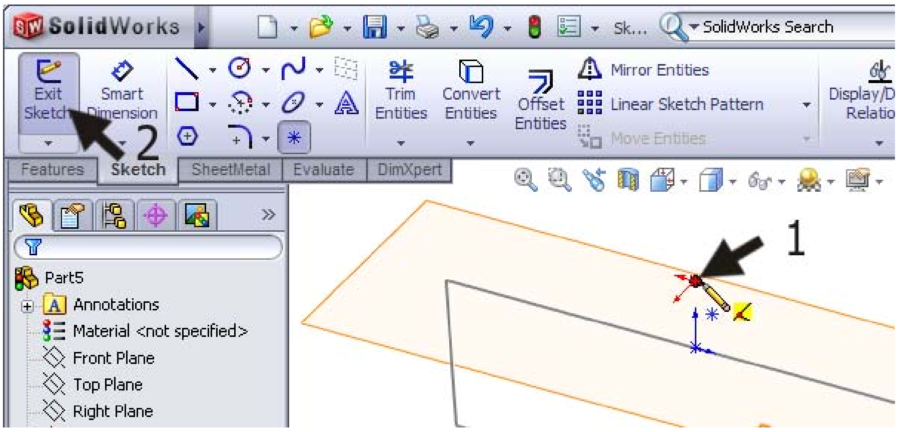
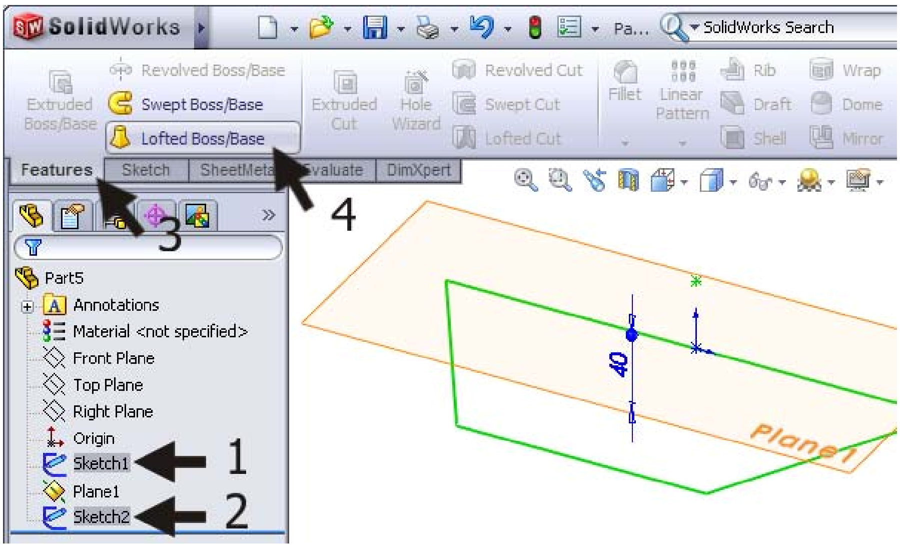
Set the size of the circle with Smart Dimension. The diameter has to be Ø6.

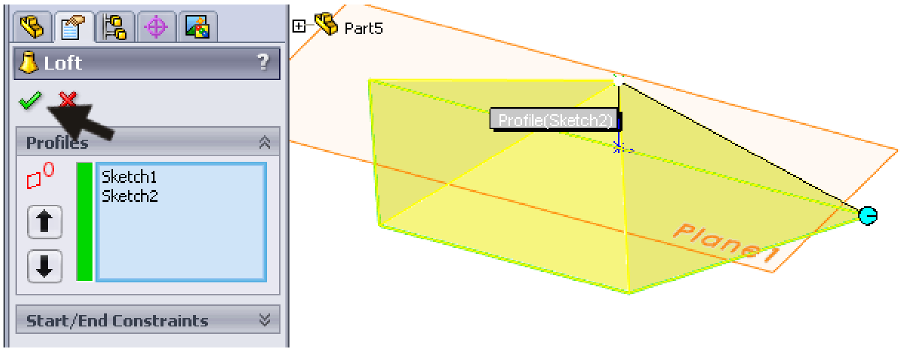
Make a Cut-Extrude from this circle and set the depth to Through All.</td>
<td>
</td>
</tr>
</table>

143	Click on ‘Edit Component’ in the CommandManager to switch off this function. You are no longer working in-context. The assembly turns back to ‘normal’ again (it is no longer transparent).	
	Tip!	The strip is ready now and is directly fixed at the correct position. You may have noticed that modeling in-context is fast and very easy to do. There is another important advantage. When you change items later – for example, the size of the shade – the size of the strip will change automatically too. We did not save the strip and did not name it. SolidWorks does this automatically and saves the part within the assembly.
	Work plan	On top of the strip we need a piece of thread M6, which is welded to the strip. We will select this from the Toolbox, and put it through the hole in the strip.
144	1. Open the ‘Design Li- brary’. 2. Click on ‘Toolbox’. 3. Click on ‘DIN’. 4. Click on ‘Bolts and Screws’ . 5. Click on ‘Studs’. 6. Select the ‘Stud bolt – DIN 976-1’, and drag it to the model.	

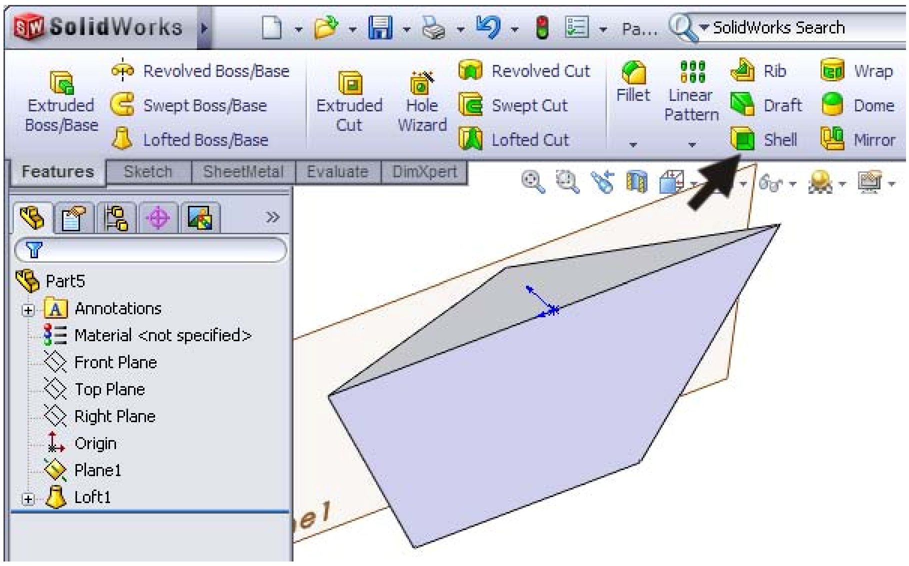
145	Release the stud bolt in the hole in the strip.	
146	1. Set the diameter to 'M6' in the Property-Manager. 2. Set the length to '60mm'. 3. Click on OK. 4. Push the <Esc> key to end this command.	

147	Next, add a mate: it has to be between the bottom of the stud bolt and the bot- tom of the strip.	
149	The assembly of the shade is now ready. Save the assembly.	
	Work plan	We need one more part: the roof of the shade. Because this is a pointed sheetmetal part, we cannot create it in the same way. We can, however, use a third method to create sheetmetal by using a solid part.
150	Open a new part. Select the ‘Top Plane’ and create a sketch, similar to the one on the right. You have done this before in steps 19 to 24. Pay attention: the upper horizontal line is not a cen- terline now, but a normal edge. Close the sketch by clicking on ‘Exit Sketch’ in the CommandManager.	
151	Add an auxiliary plane at a height of 40mm above the Top Plane. You have done this before in steps 39 to 41.	

152	Make a sketch on ‘Plane1’. 1. Select ‘Plane1’. 2. Click on a point.	
153	1. Set one point directly in the origin of the sketch. 2. Click on ‘Exit Sketch’ in the CommandManager.	
154	1. Select the ‘Sketch1’ in the FeatureManager. 2. Hold the <Ctrl> key and select ‘Sketch2’. 3. Click on ‘Features’ in the CommandManager. 4. Click on ‘Lofted Boss/Base’.	

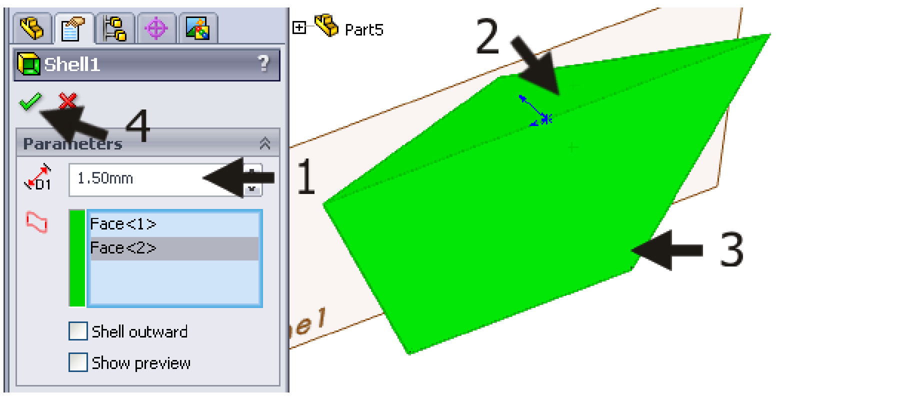
<table>
<tr><td>155</td><td>Click on OK in the Proper- tyManager.</td><td>
</td></tr>
<tr><td>156</td><td>We have a solid part now. We will make this hollow.

Rotate the model around until you see it like in the illustration.

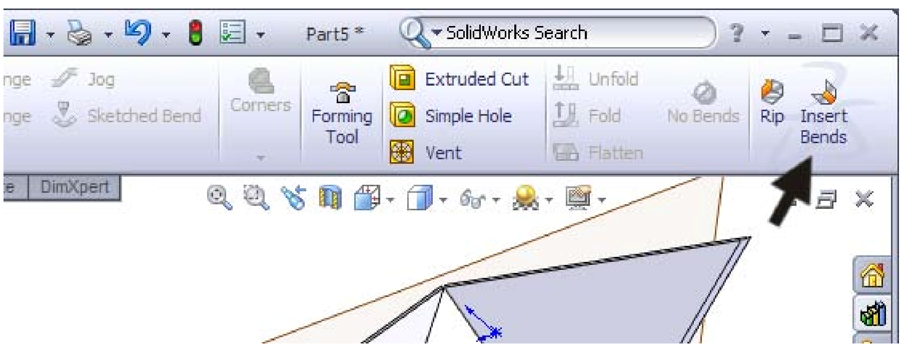
Click on 'Shell' in the CommandManag er.</td><td>
</td></tr>
<tr><td>157</td><td>1. Set the thickness to '1.5mm'
.

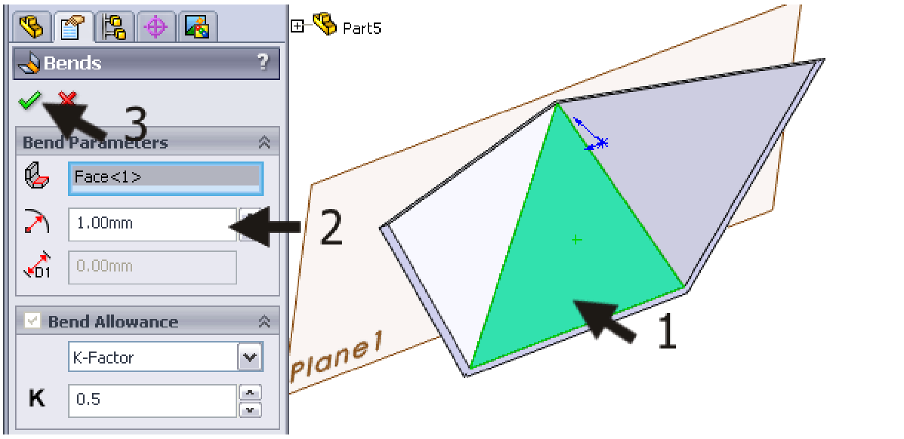
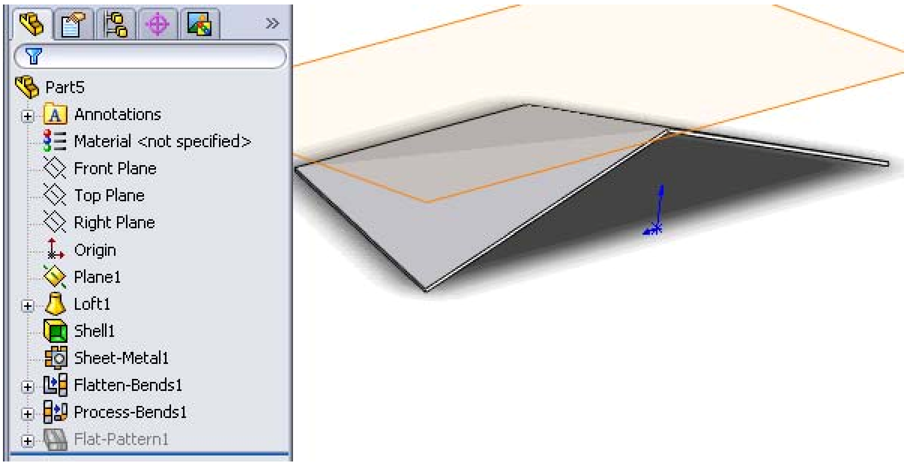
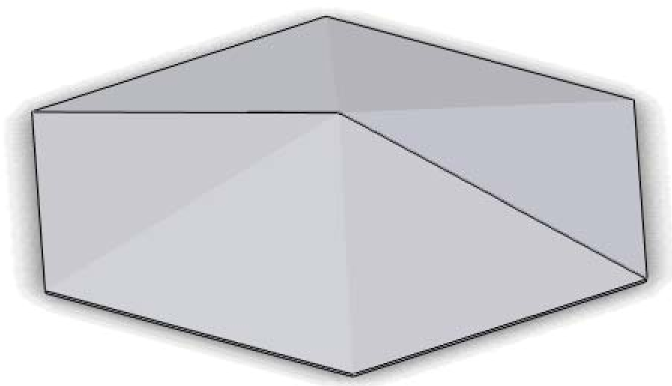
2. Select the back plane.

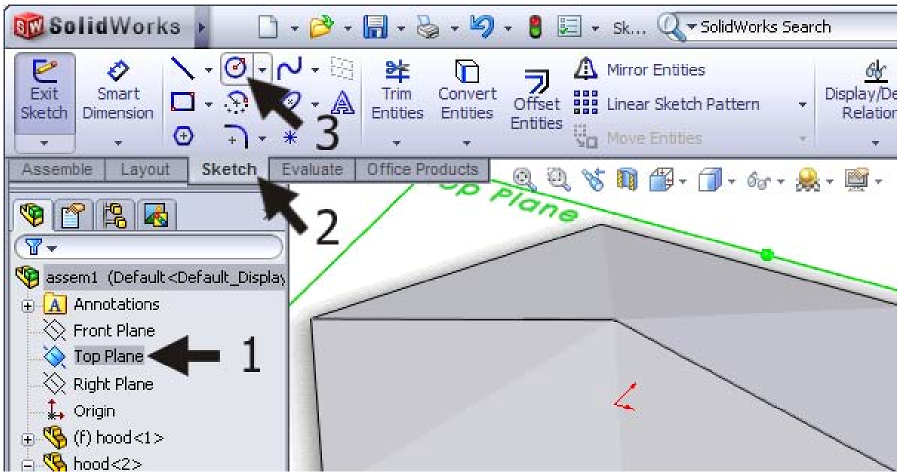
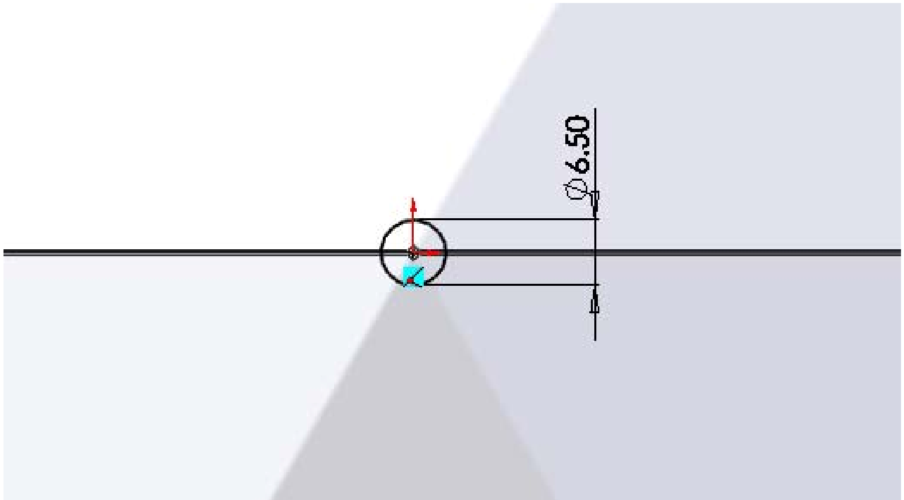
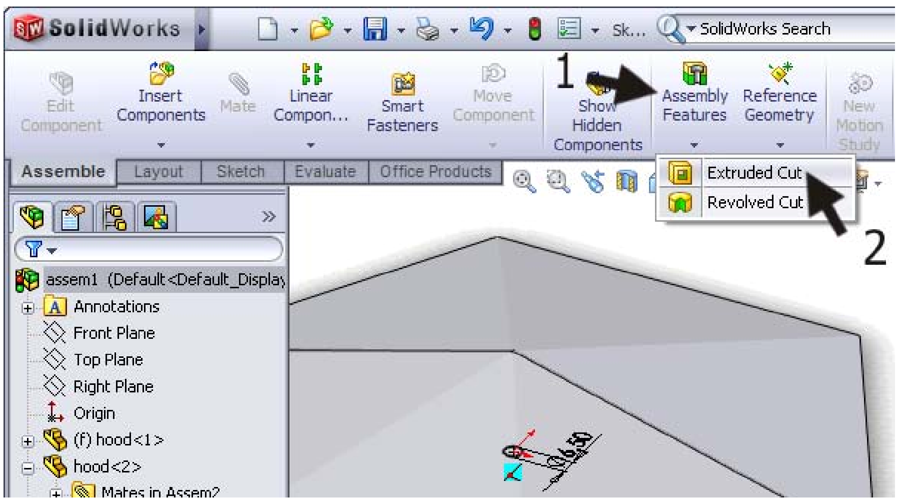
3. Select the bottom plane.

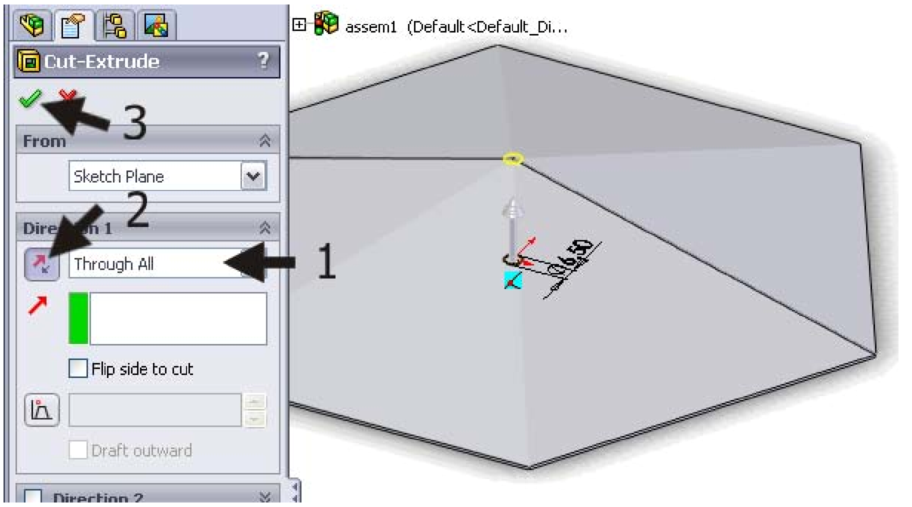
4. Click on OK.</td><td>
</td></tr>
<tr><td>158</td><td>We will change this part in- to a sheetmetal part.

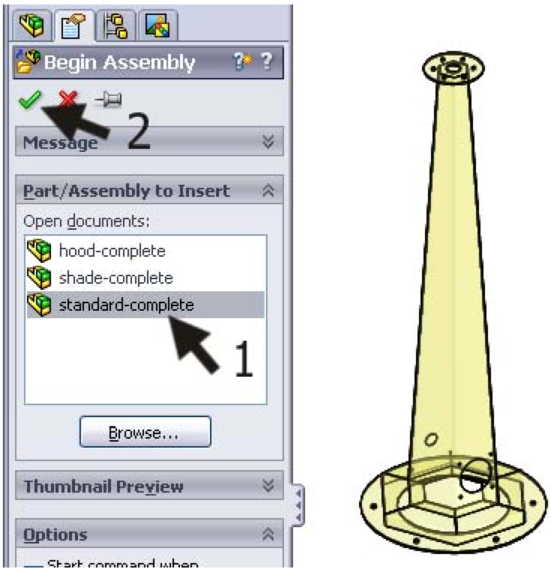
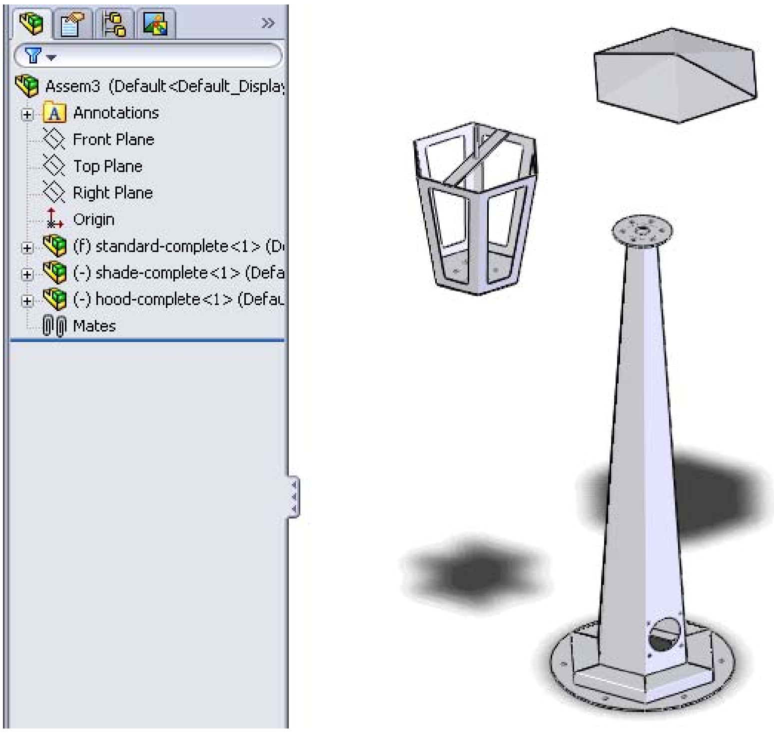
1. Click on 'SheetMetal' in the CommandManager.

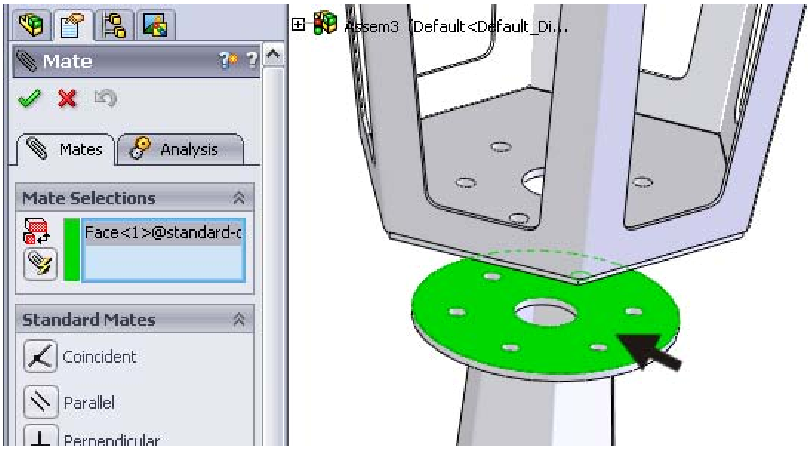
2. Click on 'Insert Bends'.</td><td>
</td></tr>
</table>

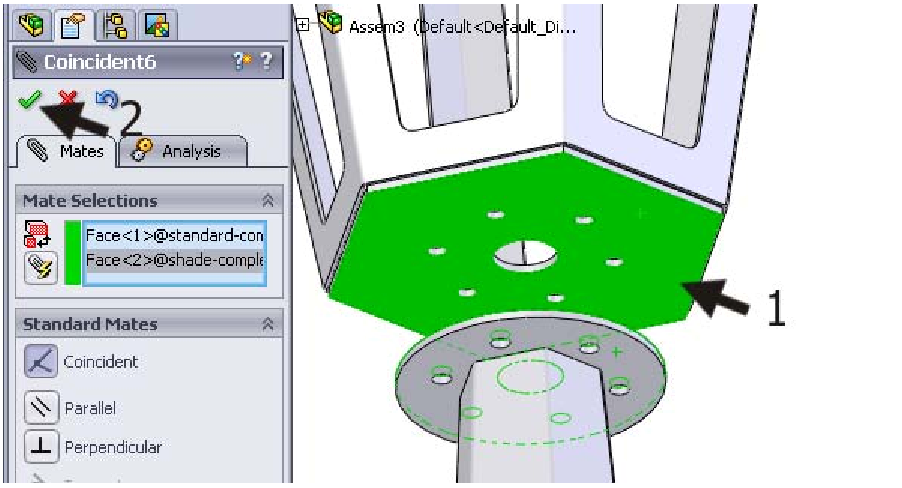
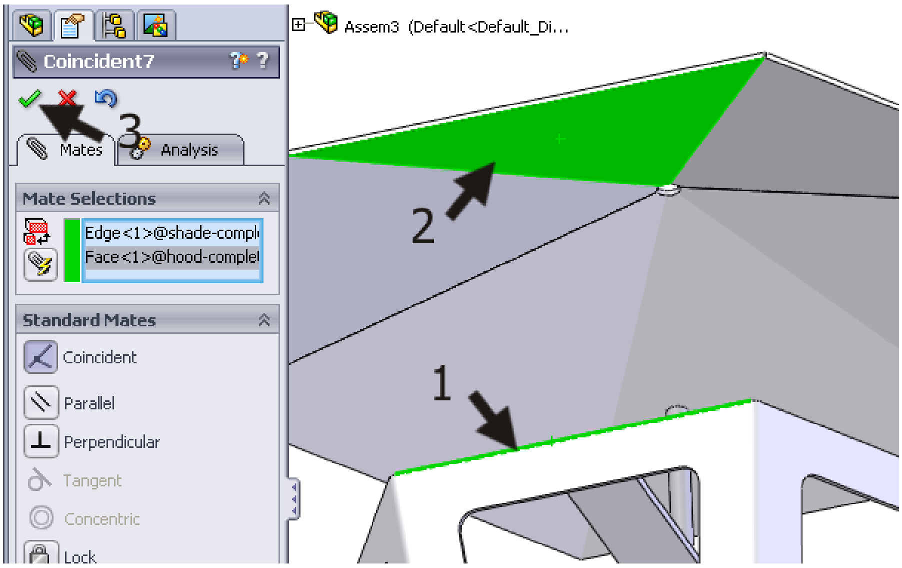
159	1. Click on the middle plane of the model. When making a flat drawing this plane will hold its position. 2. Set the bending radius to ‘1mm’. 3. Click on OK.	
160	A few features have been added to the FeatureManager now, which indicates clearly that you are dealing with a sheetmetal part. One half of the roof is ready now. Save this as: hood.SLDPRT.	
161	Next, we will make an as- sembly of the roof. Open a new assembly. Add the part hood.SLDPRT twice. Make mates to set the parts to the right position. Use the method we have used before in this tutorial: make mates between the Front and Right planes. You can set the height by mating the Top Planes. Check steps 89 to 95 on how to make these mates	

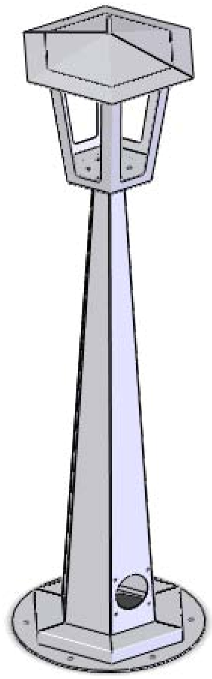
162	We have to make a mounting hole in the roof to fix it.	
163	Draw a circle with the mid-point on the origin. Set a dimension at the circle with Smart Dimension. Change it to 6.5mm.	
164	1. Click on 'Assembly Fea- tures' in the Com- mandManager. 2. Click on 'Extruded Cut'.	

165	1. Set the depth of the hole to ‘Through All’ in the PropertyManager. 2. Change the direction of the hole when neces- sary in order to lead it through the model. 3. Click on OK.	
	Tip!	Until now we have only added parts together in an assembly, but in the last step we have made a hole in the assembly. This is called an assembly fea- ture. We did nothing other than what we would have done to create this part for real: - First weld the pieces together (= make an assembly). - After that, drill a hole through the top. While making a Work plan to create a part in SolidWorks, think about how you would make the part for real.
166	The hood is ready now. Save it as hood-complete.SLDASM.	
167	All parts are now ready, and we have created three sub-assemblies: - standard-complete - shade-complete - hood-complete These three can be assem- bled to get the end product Open a new assembly.	

<table>
<tr>
<td>168</td>
<td>1. Select the file ‘standard-complete’ sub- assembly in the PropertyManager.
2. Click on OK.</td>
<td>
</td>
</tr>
<tr>
<td>169</td>
<td>Add the two other sub-assemblies now. Put them at a random position.</td>
<td>
</td>
</tr>
</table>

170	Add mates now. Again, use the Front and Right planes to put the parts above each other. You have done this before in steps 89 to 93.	
171	To put the shade onto the standard, first select the top plane of the standard.	

172	Rotate the model and select the bottom plane of the shade.	
173	We will now put the roof onto the shade. 1. Select an edge at the bottom side of the roof (be sure to select the outside of the wall). 2. Select the corresponding bottom plane of the roof. 3. Click on OK.	

174	The garden light is ready now. Save it as: garden-light.SLDASM.	

	And now ...	There are a couple of features that we have not used in this tutorial. You could try this yourself:
		1. We did not weld the sub-assemblies. We did this in Tutorial 3 (Magnetic Block).
		2. We did not create a 2D drawing from the several sheet metal parts. We have done this before in tutorial 4 (Candlestick).
		3. We have not bolted together the three parts with nuts and bolts. You could do this by using the parts from the Toolbox. We did this before in Tutorial 3 (Magnetic Block) and Tutorial 5 (Tic-Tac-Toe). For mounting the shade to the standard, use the following parts 6 times. All parts can be found in the Toolbox using the DIN menu.
		  1. Washer (Washer grade A – DIN 125 part1).
		  2. Hex Bolt (Hex screw grade AB - DIN EN 24017) M6x20.
		  3. Curved spring washer (Washer curved spring - DIN128).
		  4. Nut (Hex nut grade C – DIN EN 24034) M6.
		Use a wing nut to fix the roof. (Wing nut – DIN 315).

	What are the main features you have learned in this tutorial?	In this tutorial you have learned a lot:
		• You have seen three ways to create a part from sheetmetal:
		  1. Starting with a base flange and adding planes to it. We did this while creating the base of the standard.
		  2. Starting from a loft: use two sketches, and shape the sheetmetal in between them. This is what we did to create the standard and the shade.
		  3. Starting from a solid part. This was what we did while creating the roof.

		• You have seen how to continue with a copy of an existing part. • You have seen how to build a bigger product from sub-assemblies and assemblies. • You have seen how convenient it is to use the origin as a reference point. You can simply add mates by using the Front and Right planes. • You have seen how to change sketches. • You have seen how to resolve errors. • You have created a part 'in-context' in an assembly. • Finally you have used an assembly feature.