SOLIDWORKS® tutorial 7

GARDEN LIGHT

Prepatory and Advanced Vocational Training

To be used with SOLIDWORKS® Educational Release 2018-2019

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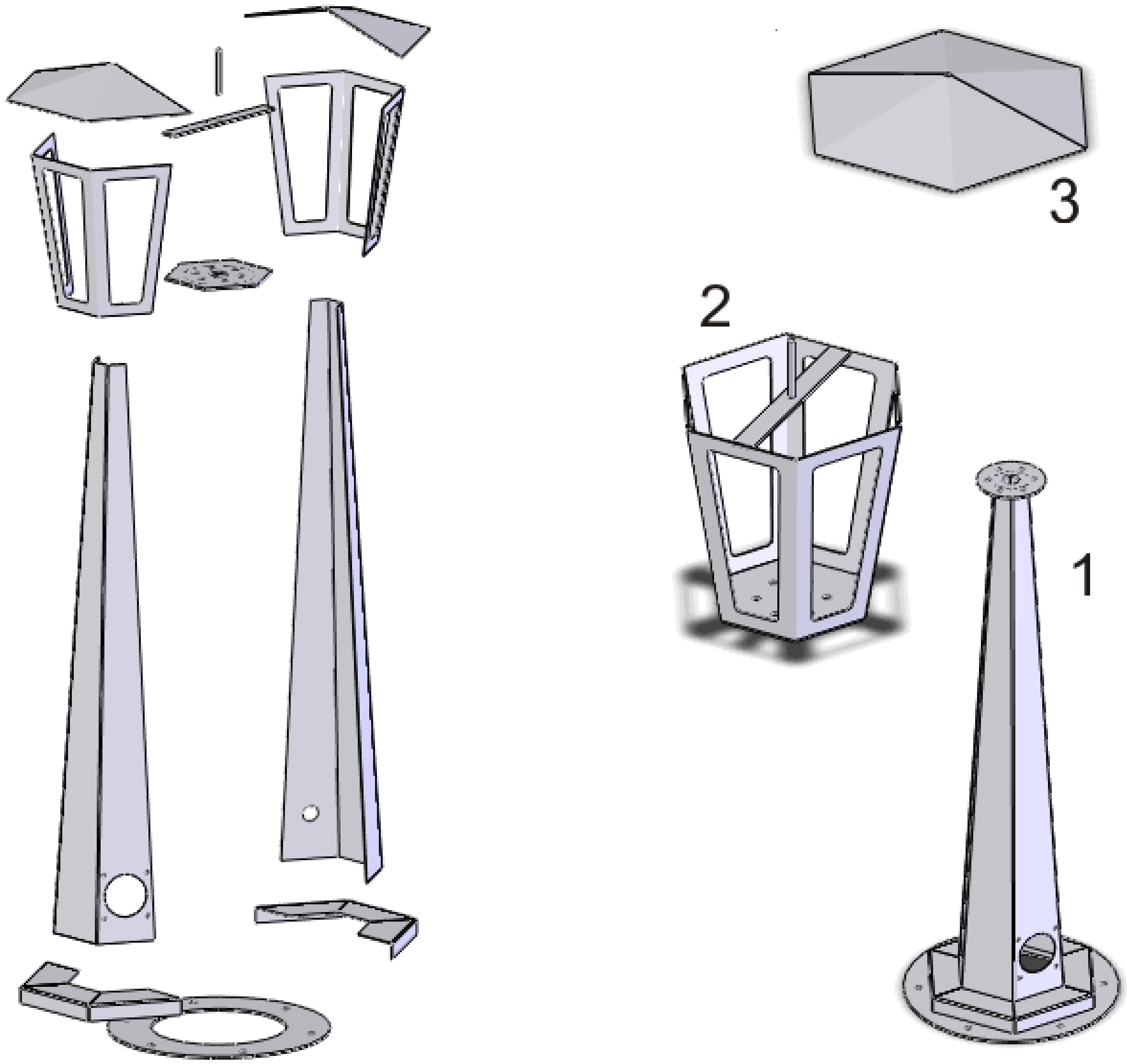
Educations advisor: Jack van den Broek
Realisation: Arnoud Breedveld (PAZ Computerworks)

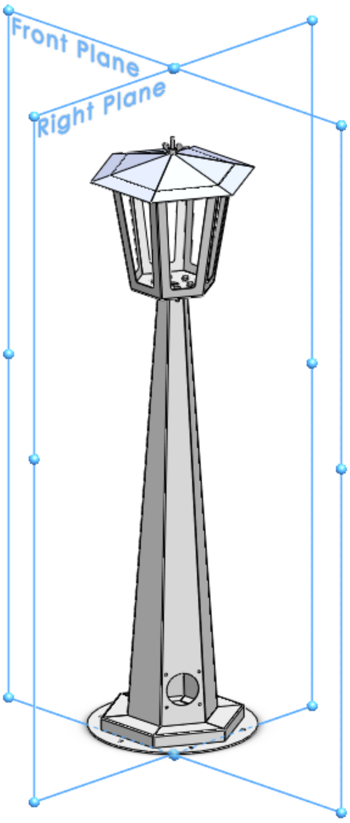
Garden light

In this tutorial we will create a garden light. It is completely build from sheet metal. In tutorial # 4 (candle light) you have seen how you can shape sheet metal in SOLIDWORKS. In this tutorial we will go further in this technique. We will create several parts from sheet metal.

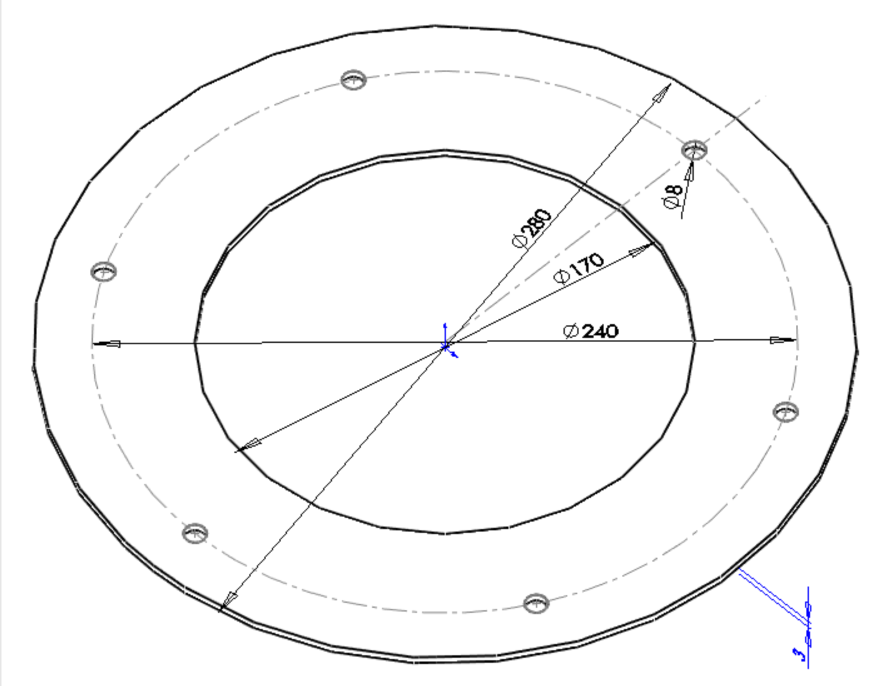
The garden light is a pretty complicated product and you will learn a lot from this tutorial. For instance, how to make a copy of a part and how to change it afterwards. How do you solve problems which are reported and how to build a model from sub-assemblies?

Down below you can see 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.

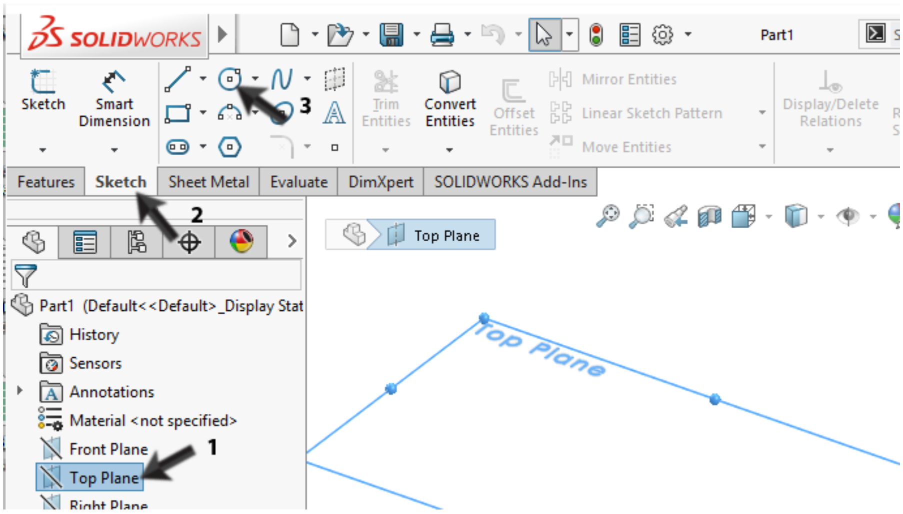


	With every part we create, we make sure the origin is exactly in the center of the model. The Front planes and Right planes of all parts will fit exactly afterwards if we do so. This will make it a lot easier to make and assemble all the different parts at the end. 

	Work plan	Let's get started. We create a base first and will end 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.

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

How would you handle this part? We will build it from two features:

At first we make the ring with a hole in the center. We will use Boss-Extrude for this.
After that we will position the six holes with Circular pattern.
</td>
</tr>
<tr>
<td>1</td>
<td>Start SOLIDWORKS and open a new part.</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>
Select the Top-plane in the FeatureManager.
Click on Sketch in the CommandManager,
Click on Circle.
</td>
<td>
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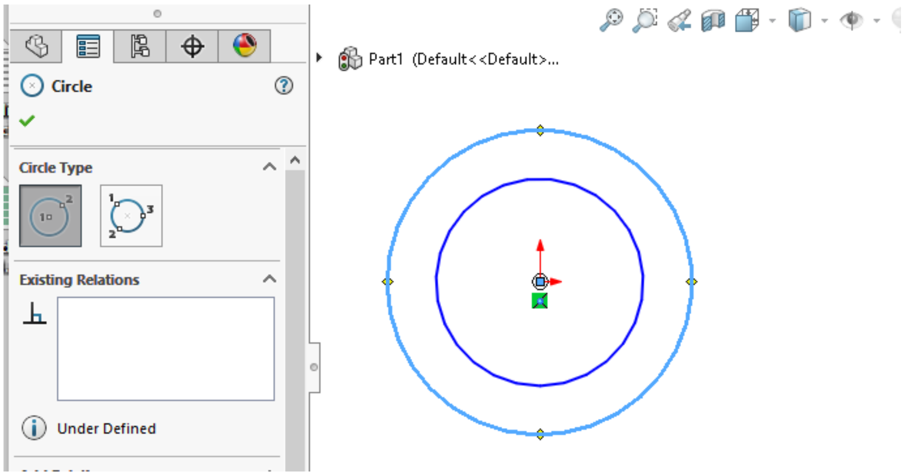
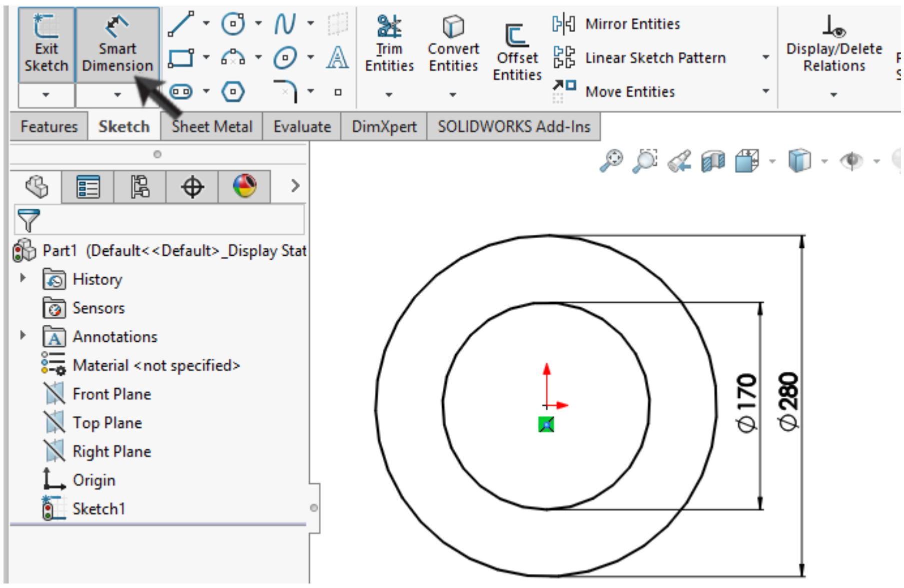
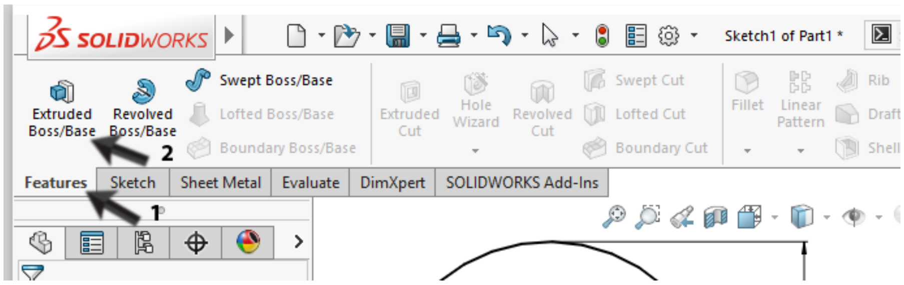
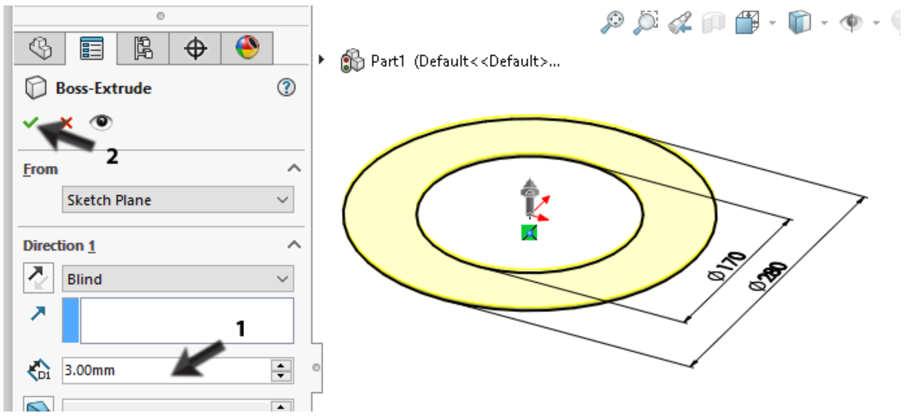
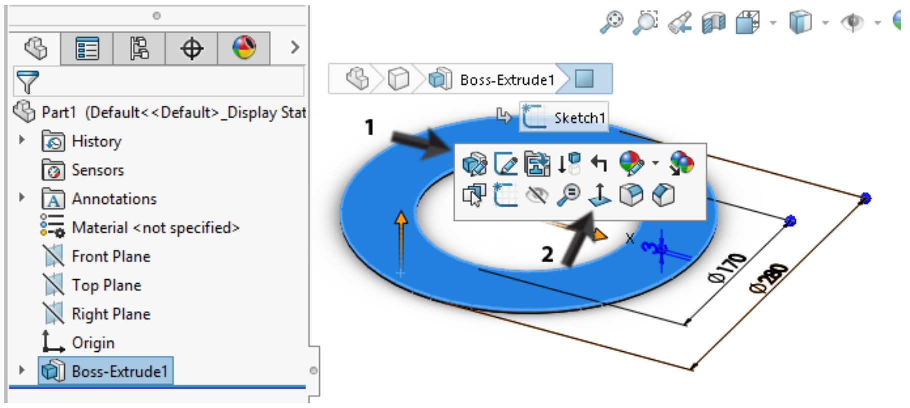
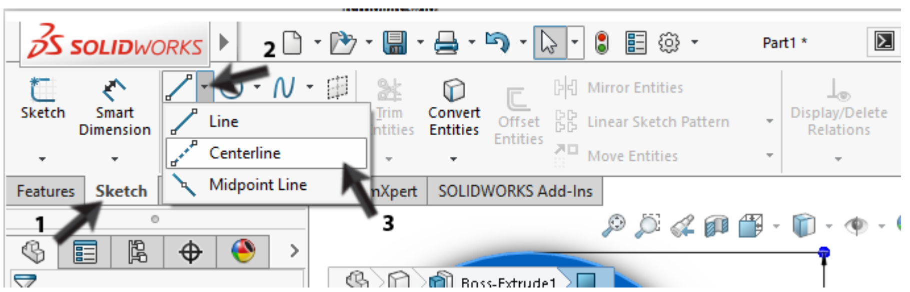
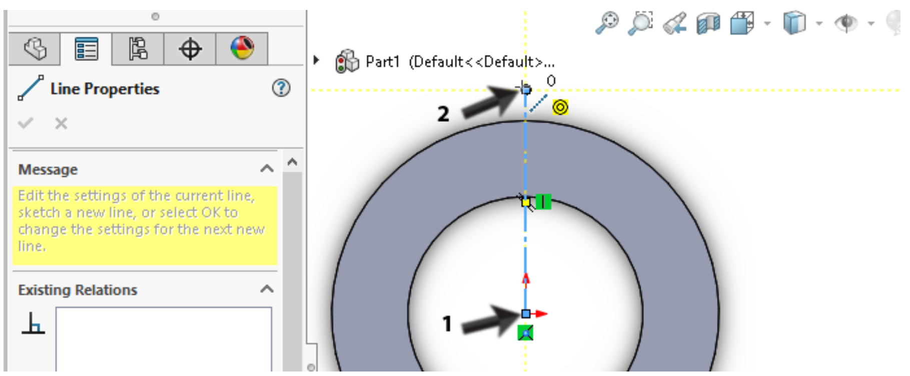
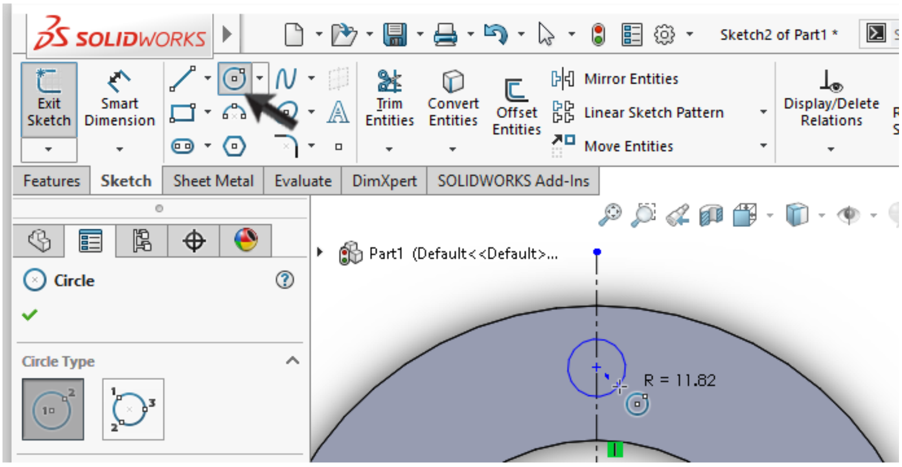
<table>
<tr>
<td>3</td>
<td>Draw two circles and make sure the center of both cir-cles is at the Origin.</td>
<td>
</td>
</tr>
<tr>
<td>4</td>
<td>Click on Smart Dimensions in the CommandManager and add dimensions for both cicles.

After this you can change the value of the dimen-sions.

Make sure the outer circle has a diameter of 280mm and the inner one a diame-ter of 170mm.</td>
<td>
</td>
</tr>
<tr>
<td>5</td>
<td>Click on Features in the CommandManager and next on Extruded Boss.</td>
<td>
</td>
</tr>
</table>

6	Set the thickness in the PropertyManager to 3 mm and Click OK.	
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 following commands: 1. Click on the top plane 2. Select the option Normal To.	
8	First draw a construction line: 1. Click on Sketch in the CommandManager 2. Open (when necessary) the extended menu 3. Click on Centerline.	
9	Draw the centerline from the origin vertically upwards. Push <esc> on the keyboard to end the centerline command.	

<table>
<tr>
<td>10</td>
<td>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)</td>
<td>
</td>
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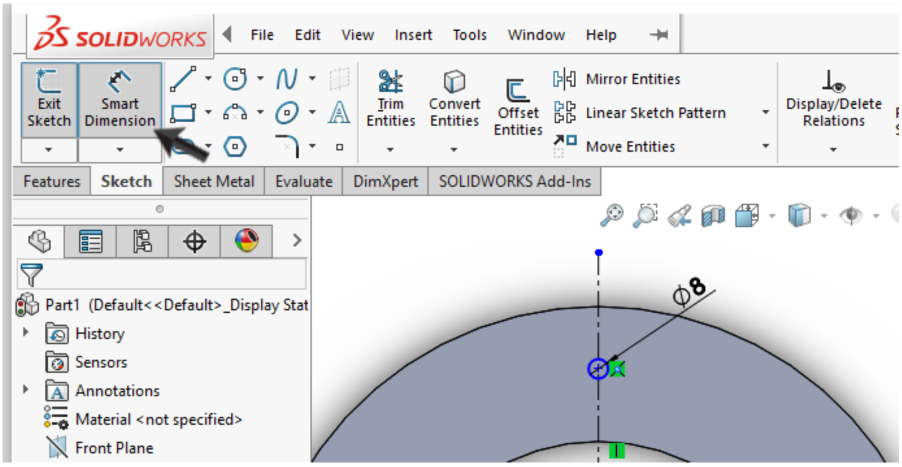
<tr>
<td>11</td>
<td>Click on Smart Dimensions in the CommandManager and add a dimension of Ø8mm for the circle.</td>
<td>
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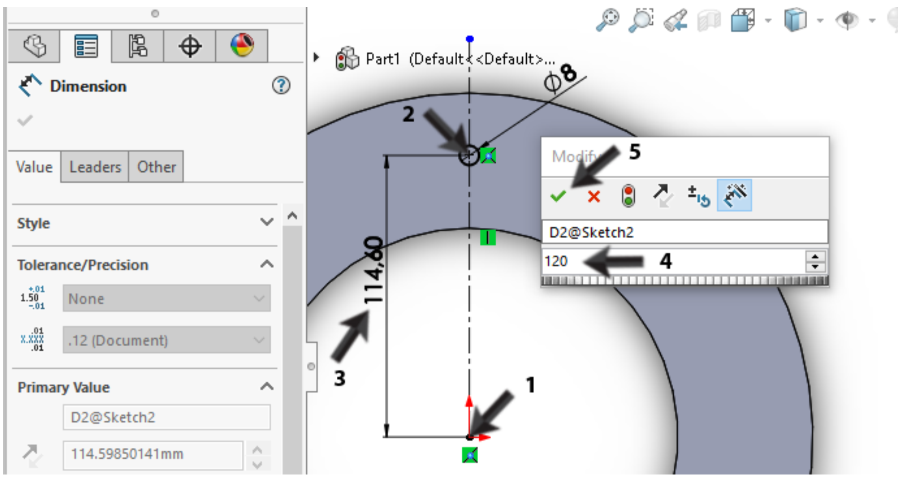
</tr>
<tr>
<td>12</td>
<td>Add a dimension for the distance between the circles to the origin, like shown in the illustration.

With the Smart Dimensions-command still active, click on:

1. The center of the circle
2. The origin
3. The point where you want the dimension to be placed.
4. Change this size to 120mm
5. Click OK.</td>
<td>
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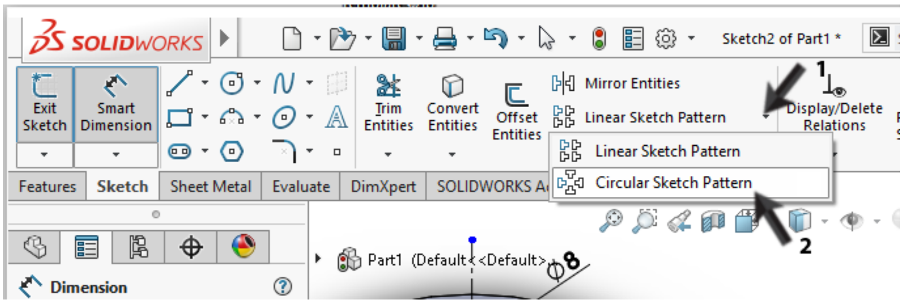
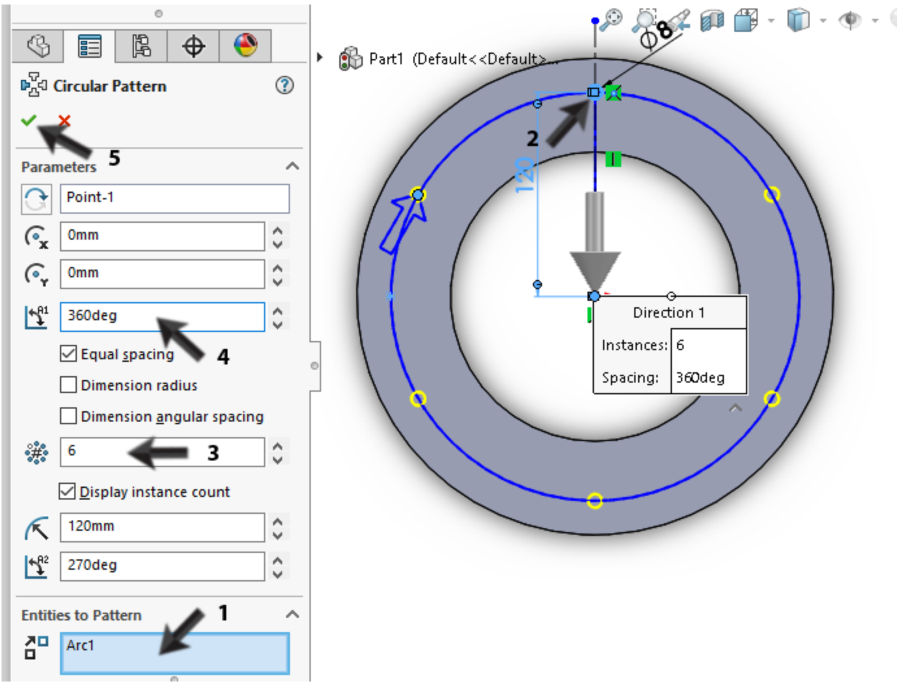
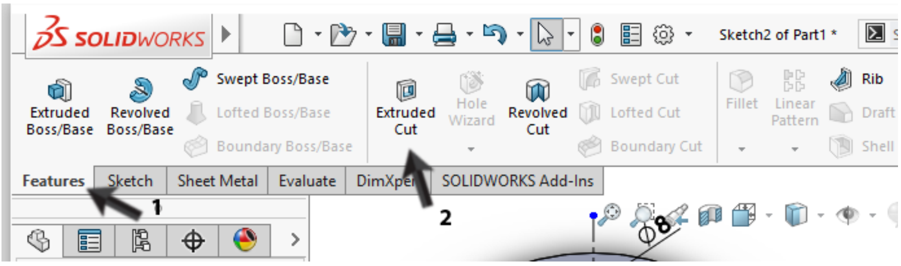
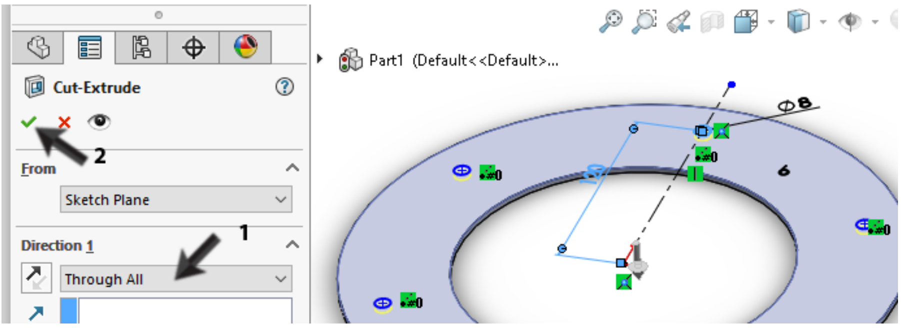
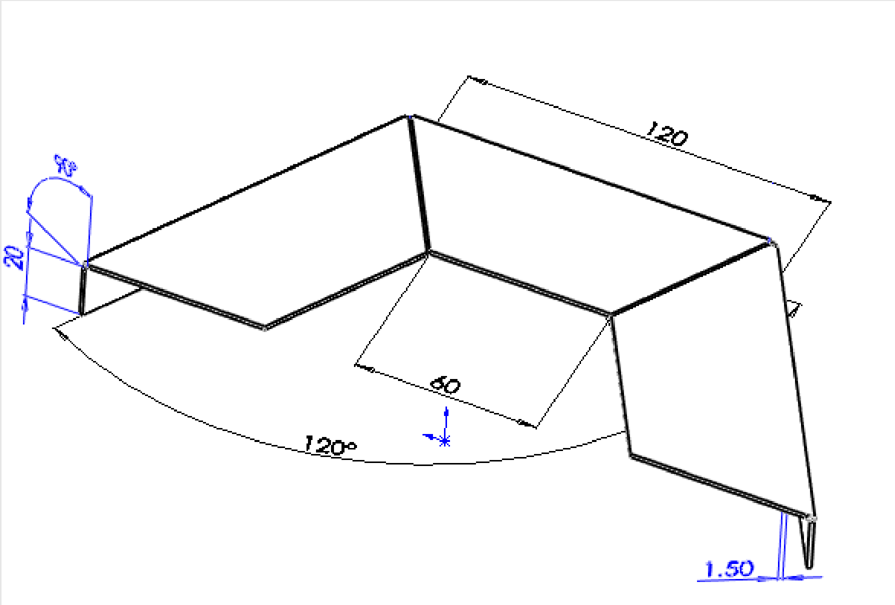
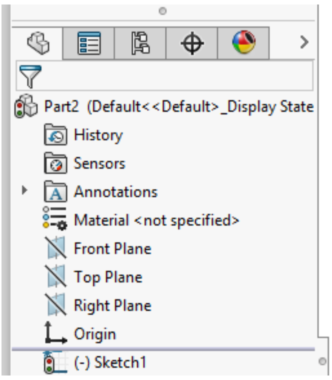
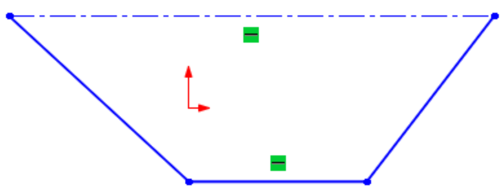
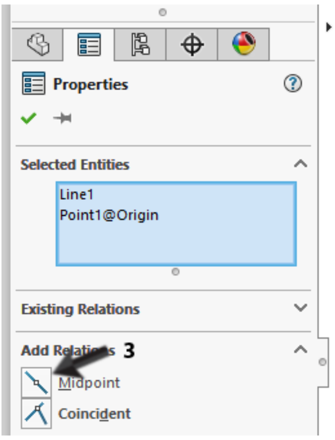
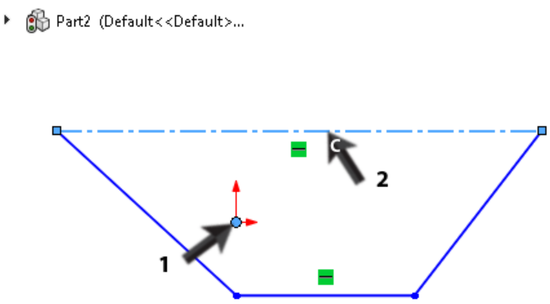
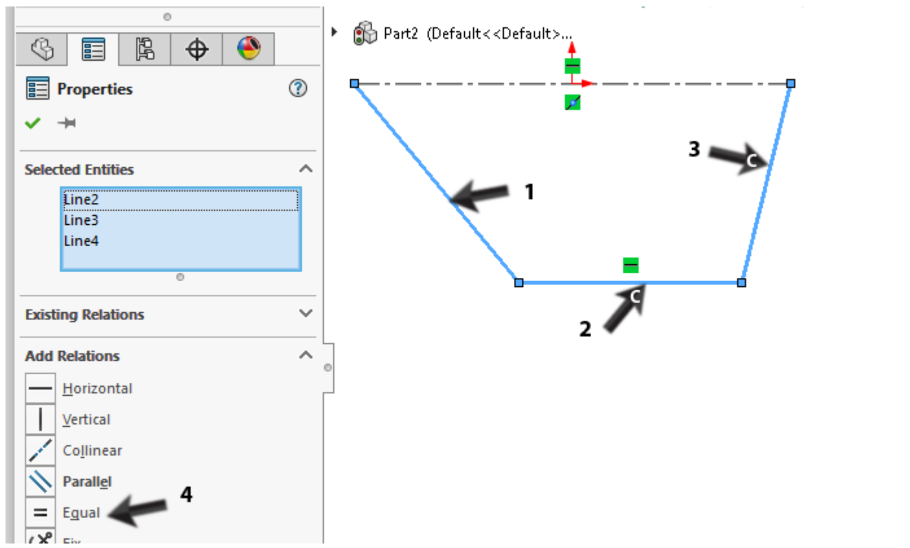
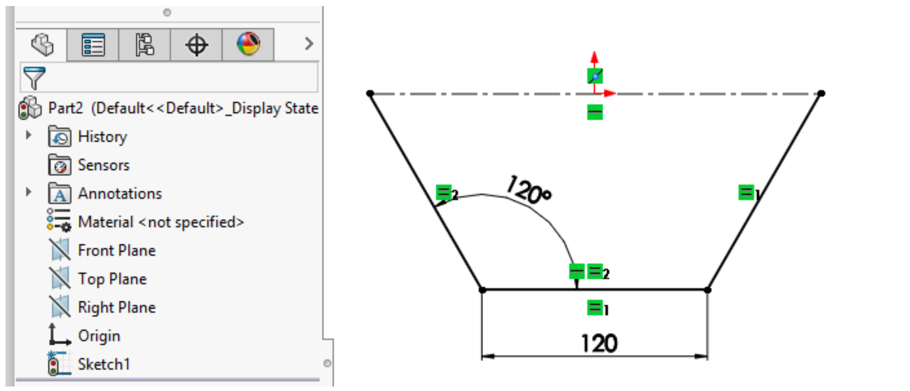
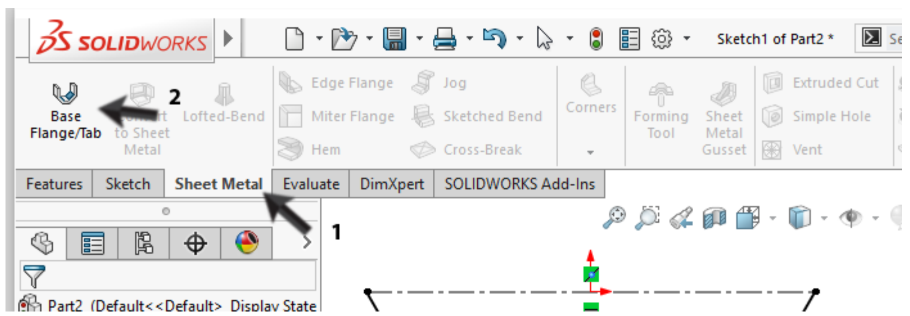
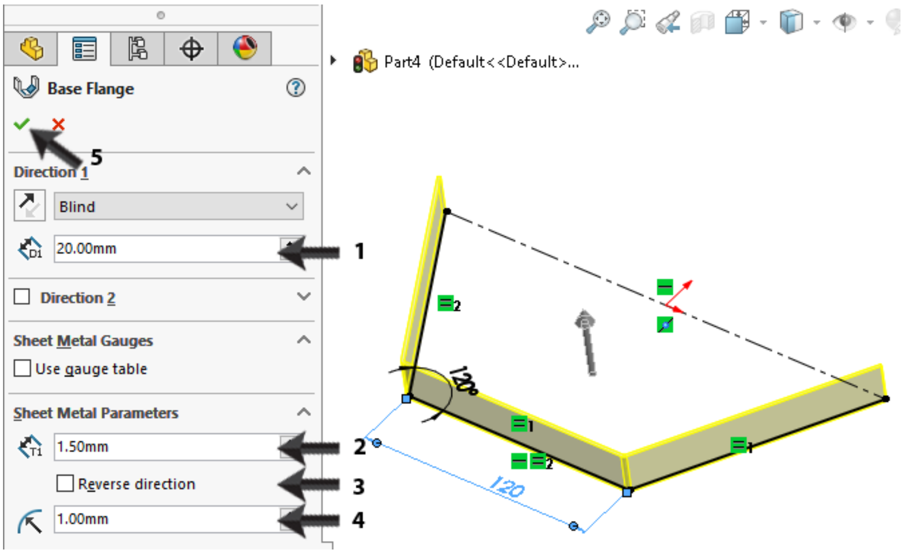
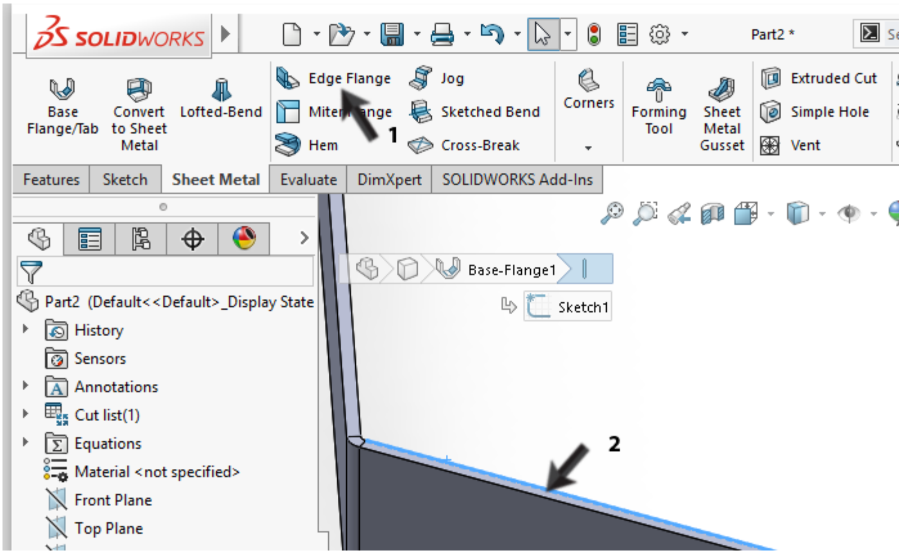
</table>

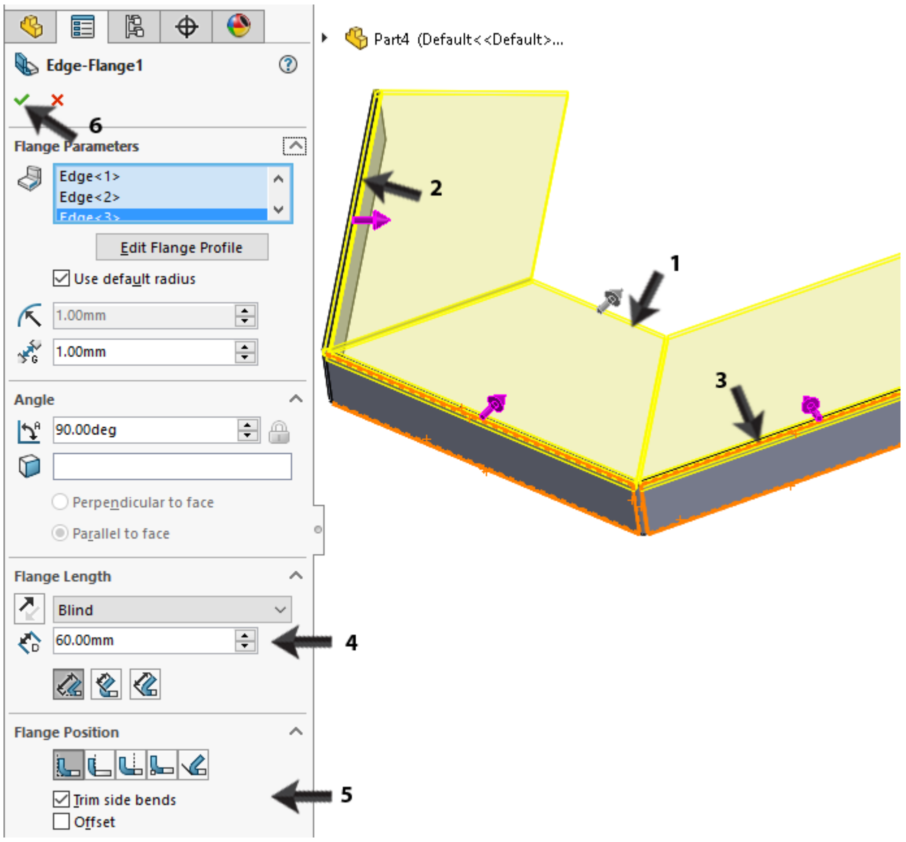
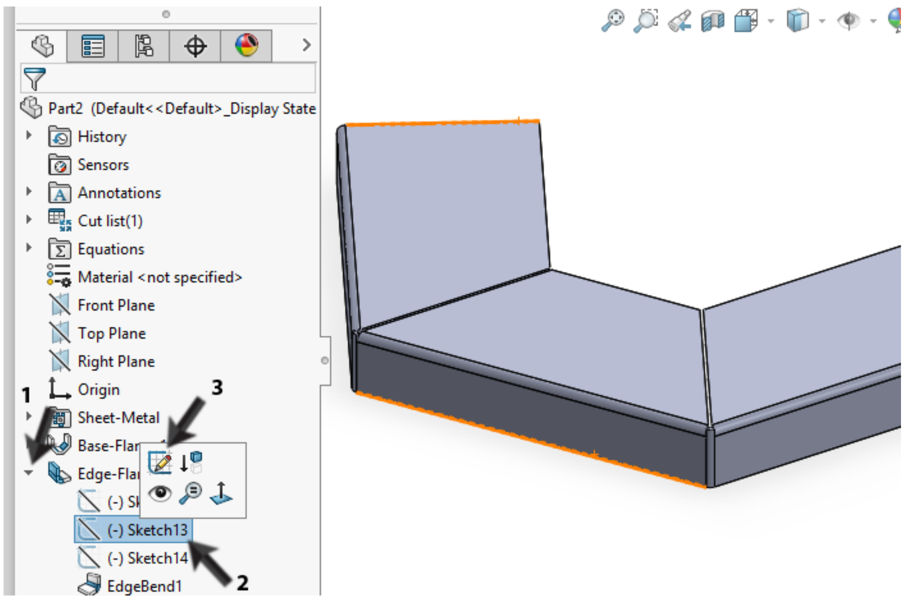
13	1. Click on the arrows next to the Linear Sketch Pattern in the CommandManager 2. Click on Circular Sketch Pattern	
14	1. Click on Entities to Pattern in the PropertyManager. The selection field turns blue 2. Select the circle you want to copy 3. Change the number of copies to 6 4. Check if the corner is at a complete 360°. 5. Click 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 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 making is the base. It looks a bit like a part of a hexagonal container. See the drawing below.  We will create this part from sheet metal.
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 position.	 
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.	 

<table>
<tr>
<td>21</td>
<td>Make the length of the three lines equally long:
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.</td>
<td>
</td>
</tr>
<tr>
<td>22</td>
<td>Click on Smart Dimensions in the CommandManager. Add the dimensions as in the illustration on the right.</td>
<td>
</td>
</tr>
<tr>
<td>23</td>
<td>1. Click on Sheet Metal in the CommandManager.
2. Click on Base-Flange</td>
<td>
</td>
</tr>
<tr>
<td></td>
<td>Tip!</td>
<td>When the button Sheet Metal is not visible in the CommandManager, right click on one of the tabs of the CommandManager. A list will appear and you can turn Sheet Metal on.

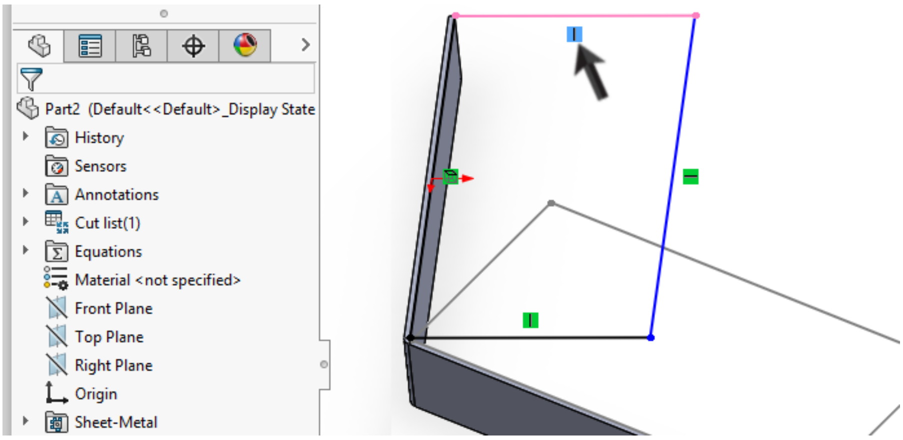
In tutorial 4 (candle light) this is described extensively.</td>
</tr>
</table>

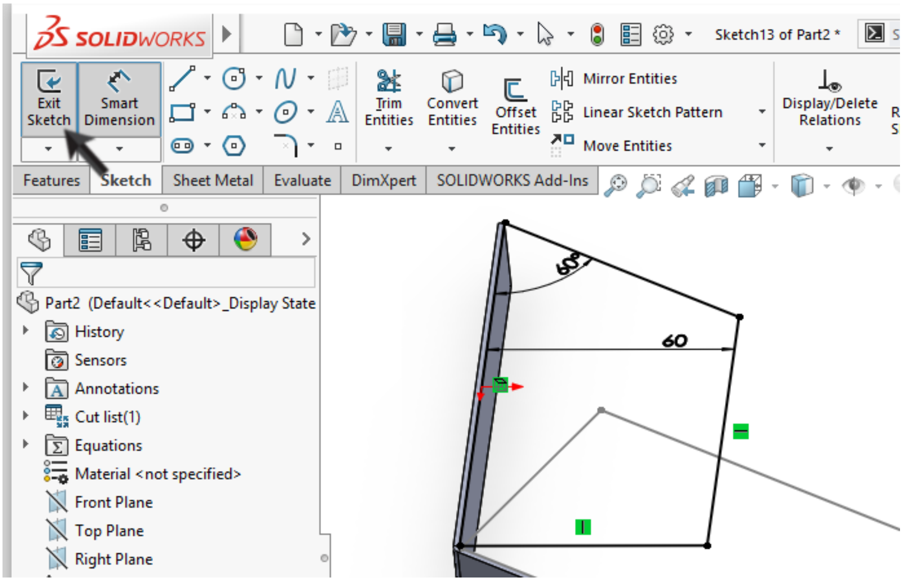
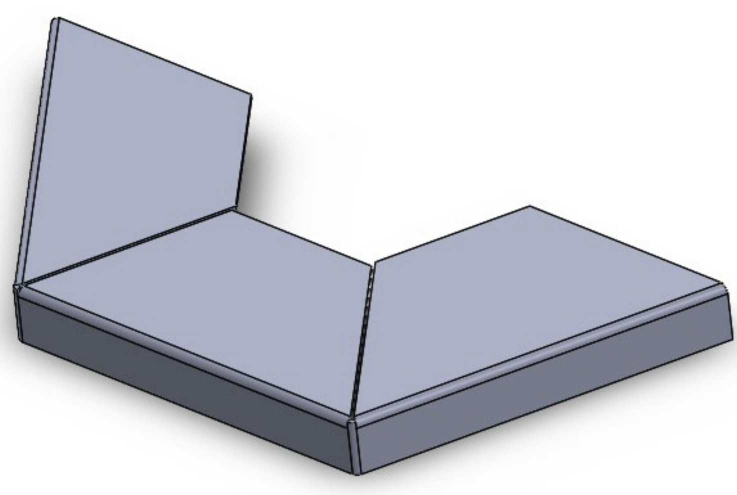
<table>
<tr>
<td>24</td>
<td>Set the following features in the PropertyManager:
1. The height of the part is 20mm.
2. The thickness is 1.5mm.
3. Make sure the material is added at the outside of the sketch, if necessary click Reverse Direction
4. The bending radius is 1 mm.
5. Click OK.</td>
<td>
</td>
</tr>
<tr>
<td>25</td>
<td>Next we will create the bended surface:
1. Select the edge you want to bend
2. Click on Edge Flange in the CommandManager.</td>
<td>
</td>
</tr>
</table>

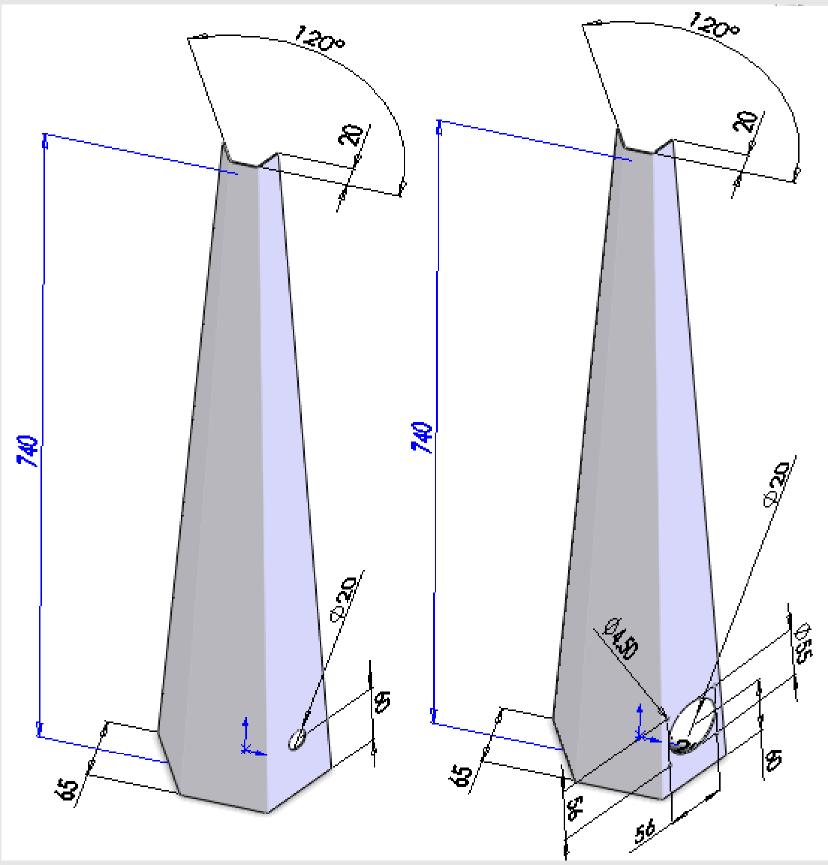
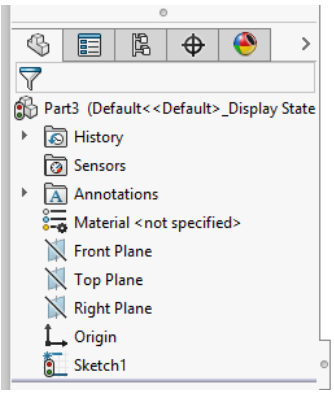
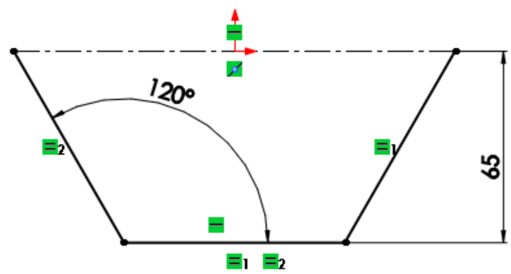
26	1. Click at a random point to set the first plane. 2,3 Click on both other edges in order to make planes there as well. 4. Set the length of the planes to 60mm 5. Select the option Trim side bends 6. Click OK.	
27	The shape of the planes is determined by the sketch. The sketches have to be altered 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.	

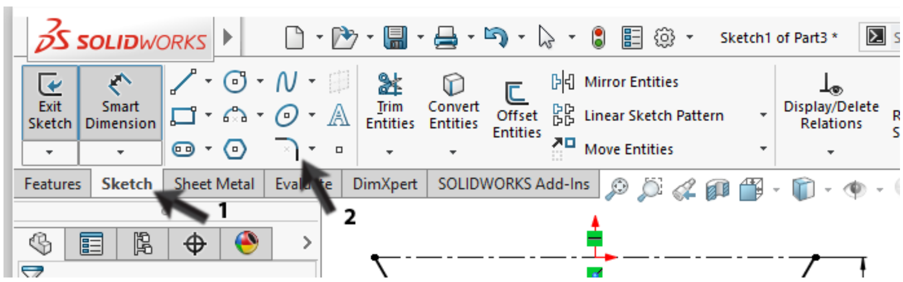
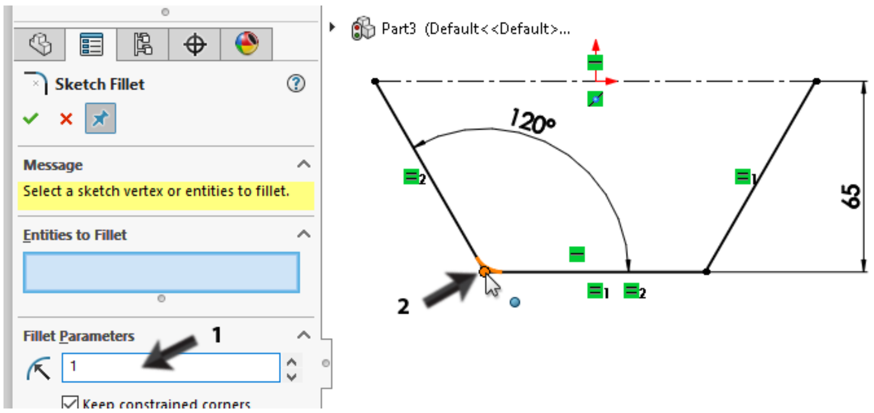
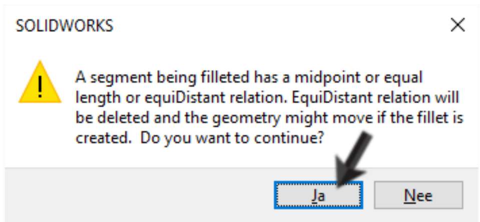
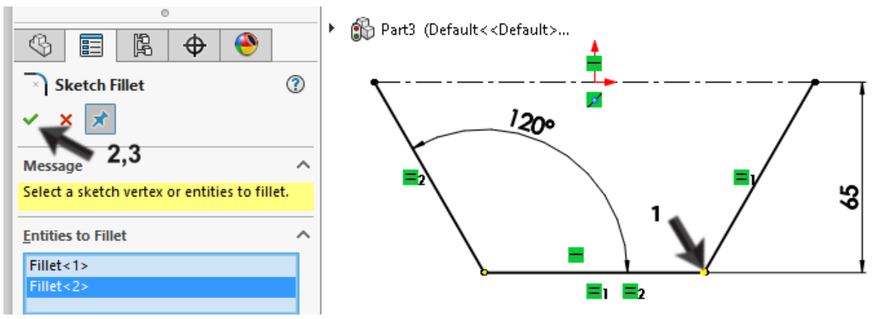
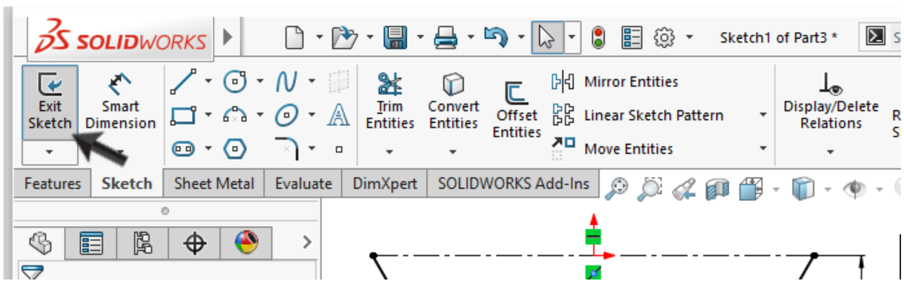
<table>
<tr>
<td>28</td>
<td>Now we can change the sketch.

Select the relation ‘Vertical’ (look at the drawing on the right)

Push <delete> at the key-board.</td>
<td>
</td>
</tr>
<tr>
<td>29</td>
<td>Add the dimensions with the Smart Dimensions command like in the illus-tration.

Click on Exit Sketch in the CommandManager.</td>
<td>
</td>
</tr>
<tr>
<td>30</td>
<td>Repeat steps 27 to 29 for the plane at the other side. The end result will look like the image on the right.</td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>Save the file as: base.sldprt.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Work plan</td>
<td>The next part we will make is the standard. We will make two varieties (configurations)</td>
</tr>
</table>

		1. one with a hole of Ø20 as a cable transit 2. one with a larger hole (Ø55) and four smaller holes (Ø4.5) to mount a wall socket. The sheet metal 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 cannot build it like we did before. Therefore we choose another method: we 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 like in the illustration. If you have a problem with this, look at steps 19 to 22. You did exactly the same there (only with other dimensions).	

<table>
<tr><td>33</td><td>We will round the corners now. Click on Sketch Fillet in the CommandManager.</td><td>
</td></tr>
<tr><td>34</td><td>1. Change the radius to 1 mm in the Property-Manager
2. Click on the first corner in the sketch.</td><td>
</td></tr>
<tr><td>35</td><td>Click Yes in the message that appears.</td><td>
</td></tr>
<tr><td>36</td><td>Next click on the second corner. The message from step 35 appears again. Then, click Yes twice.</td><td>
</td></tr>
<tr><td>37</td><td>Click on Exit Sketch in the CommandManager.</td><td>
</td></tr>
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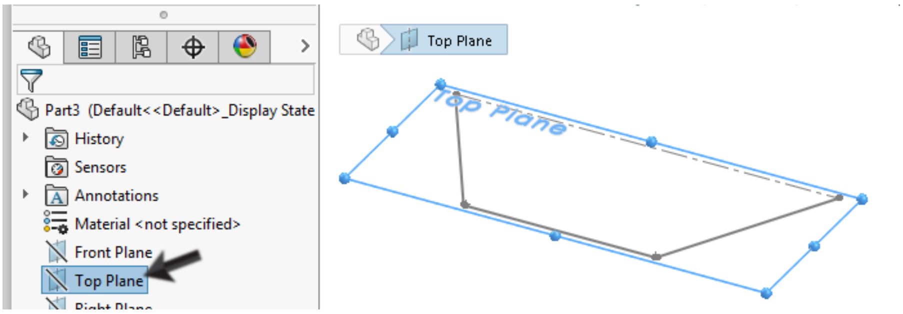
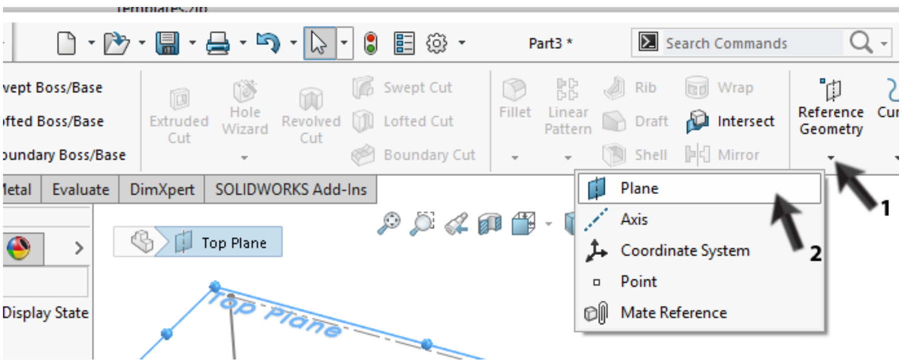
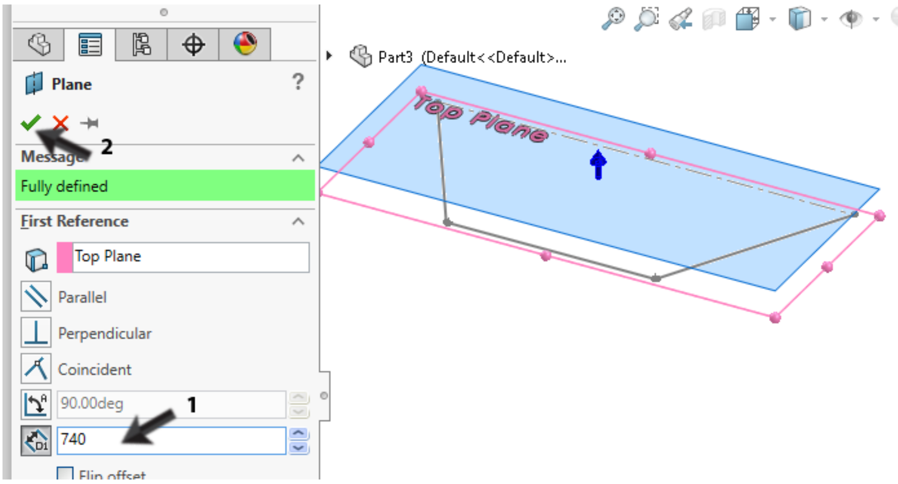
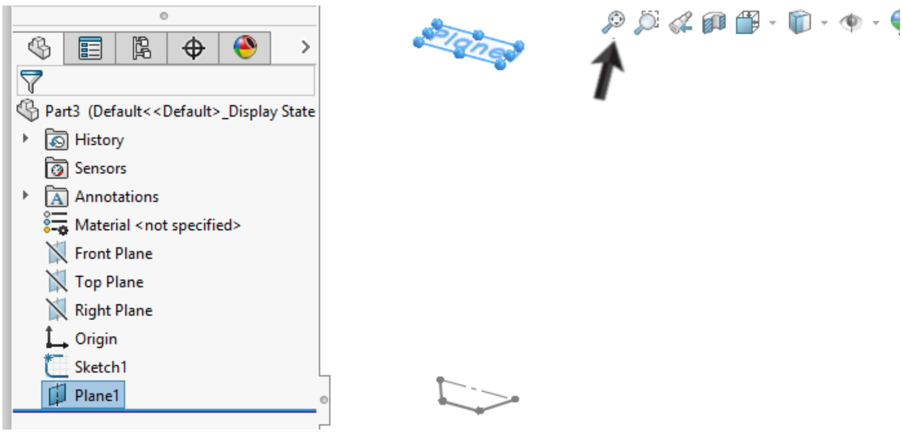
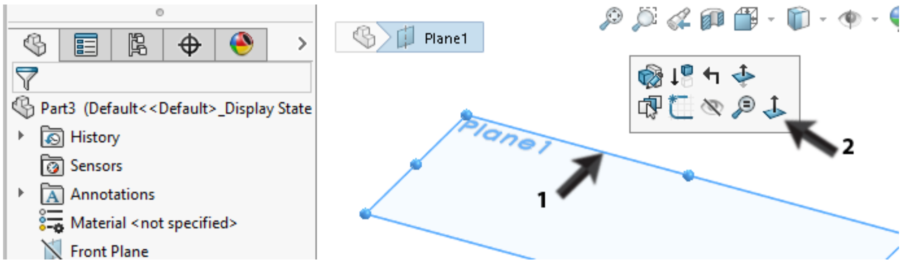
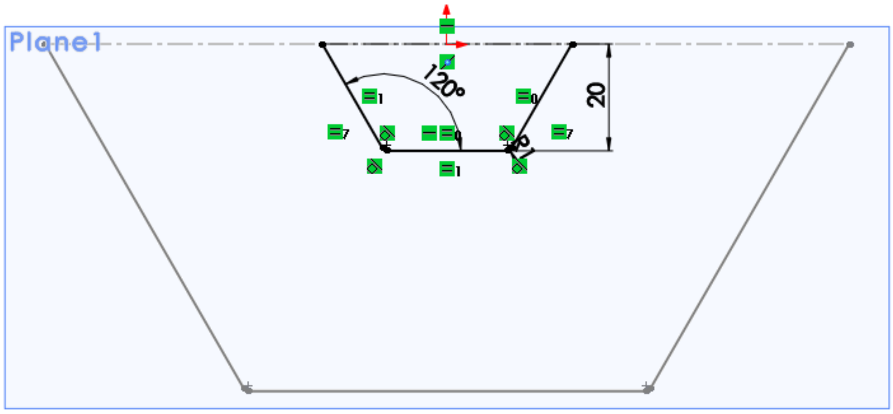
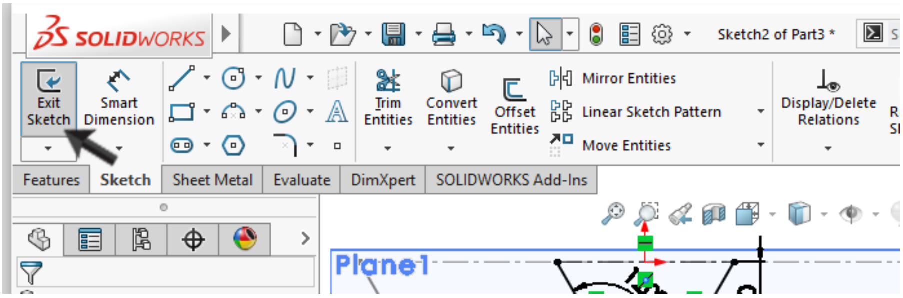
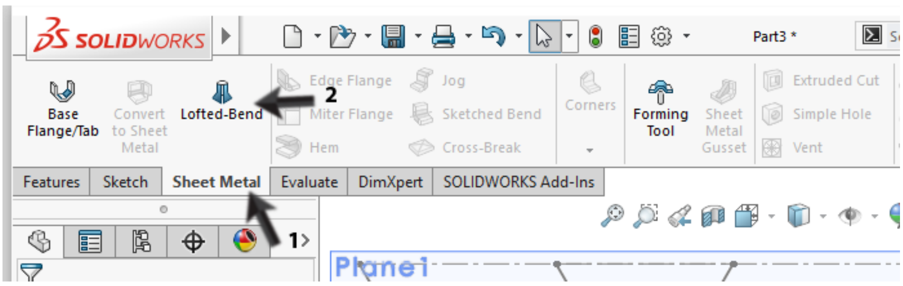
38	Click on the Top plane in the FeatureManager.	
39	1. Click on Reference Geometry in the CommandManager. 2. Click on Plane.	
40	1. Set a distance of 740mm in the PropertyManager. 2. Click OK.	
41	Click on Zoom to fit in the View-toolbar. Notice that a plane called Plane1 is floating above the sketch you have just made.	
	Tip!	We have seen before that you can draw a sketch on every plane in SOLIDWORKS. 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.

<table>
<tr><td></td><td></td><td>If is also possible of course, that you want to make a sketch at a point, and no plane is available. In such a case you create a plane yourself (Plane). You can define it in every spot and with every angle in relation to the standard planes.

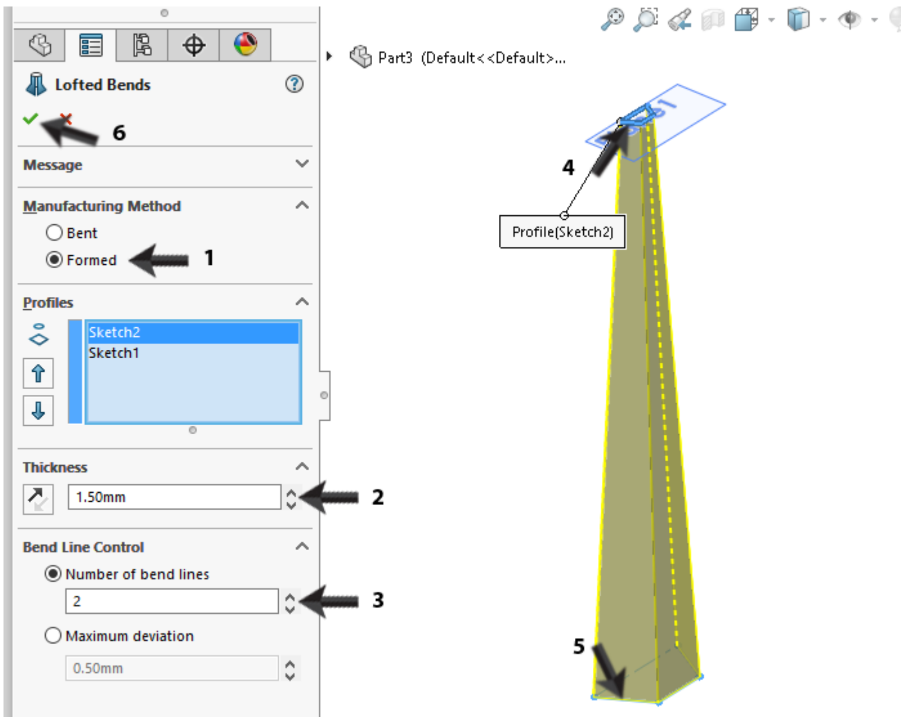
This is what you have done in step 40: you have created a construction plane 740mm above the Top plane. Here we can draw our next sketch.</td></tr>
<tr><td>42</td><td>1. Select Plane1
2. Click on Normal To.</td><td>
</td></tr>
<tr><td>43</td><td>Now make exactly the same sketch as you did before. The only difference is that the height is now 20 mm instead of 65mm.

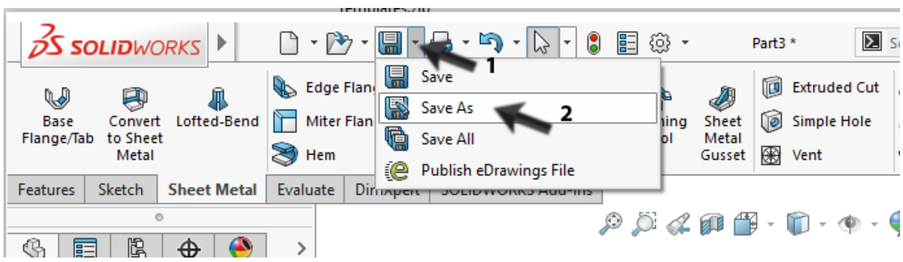
Follow steps 34 to 39 to do so.

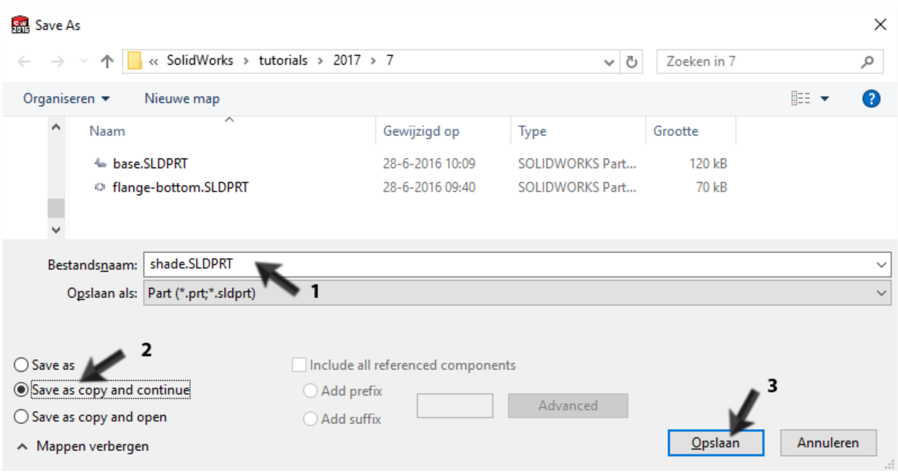
When the sketch is done, it should look like the illustration on the right.

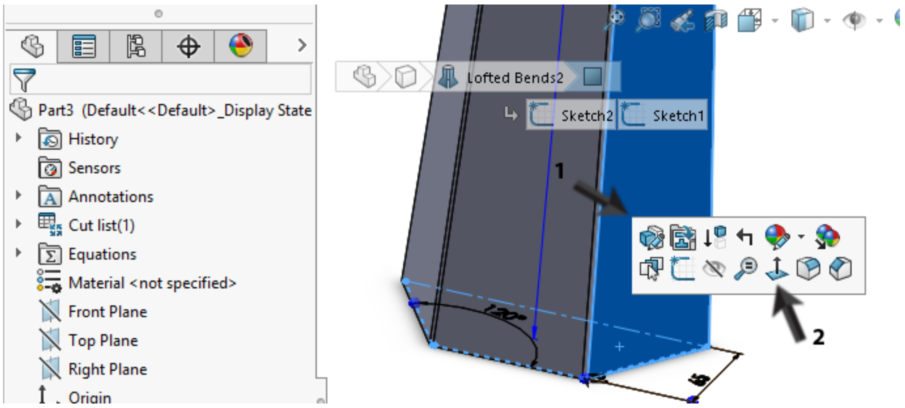
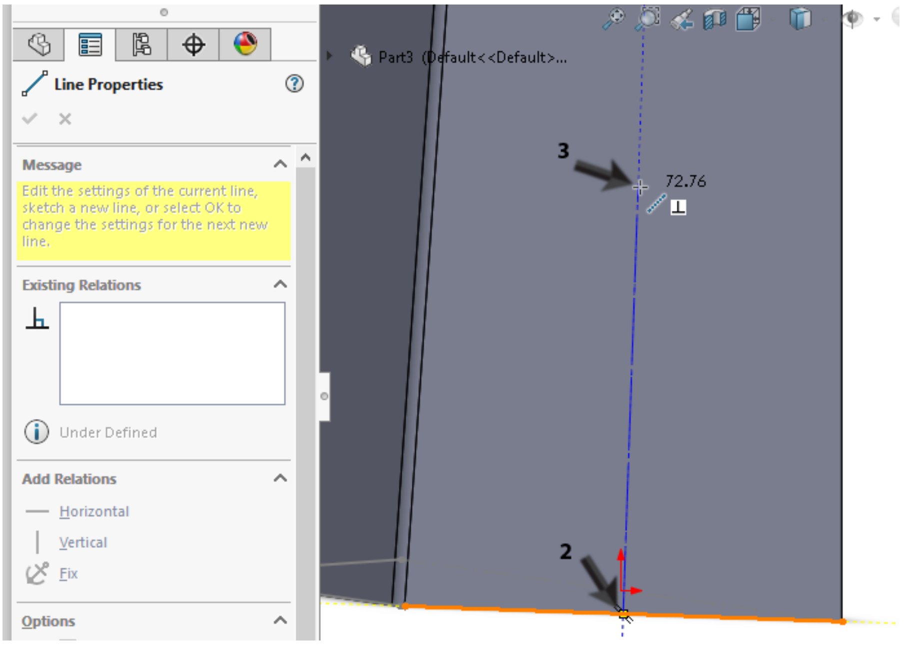
Notice that the big sketch in grey is the first sketch as created op the Top-plane.</td><td>
</td></tr>
<tr><td>44</td><td>Click on Exit Sketch in the CommandManager to close the sketch.</td><td>
</td></tr>
<tr><td>45</td><td>1. Click on Sheet Metal in the CommandManager
2. Click on Lofted Bends</td><td>
</td></tr>
</table>

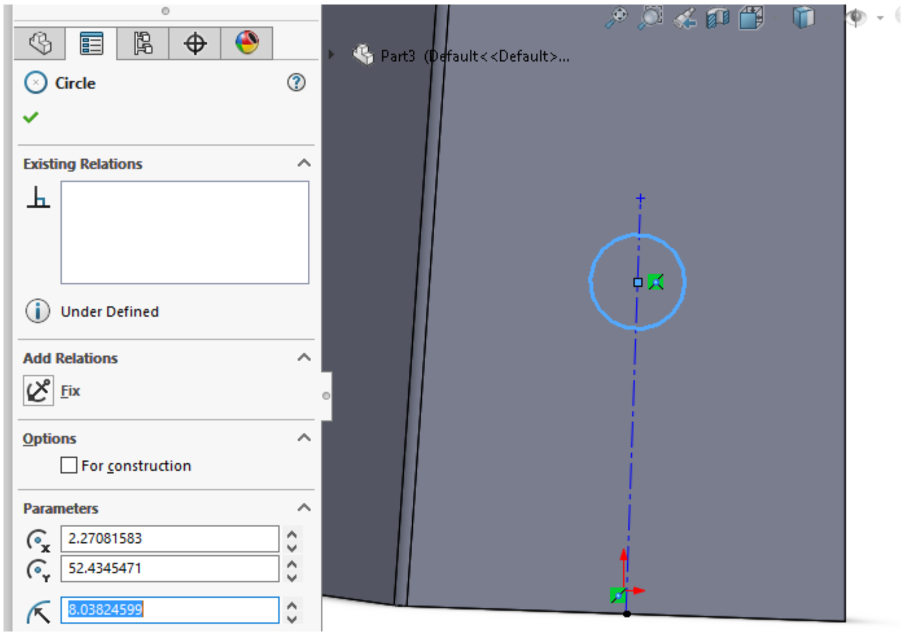
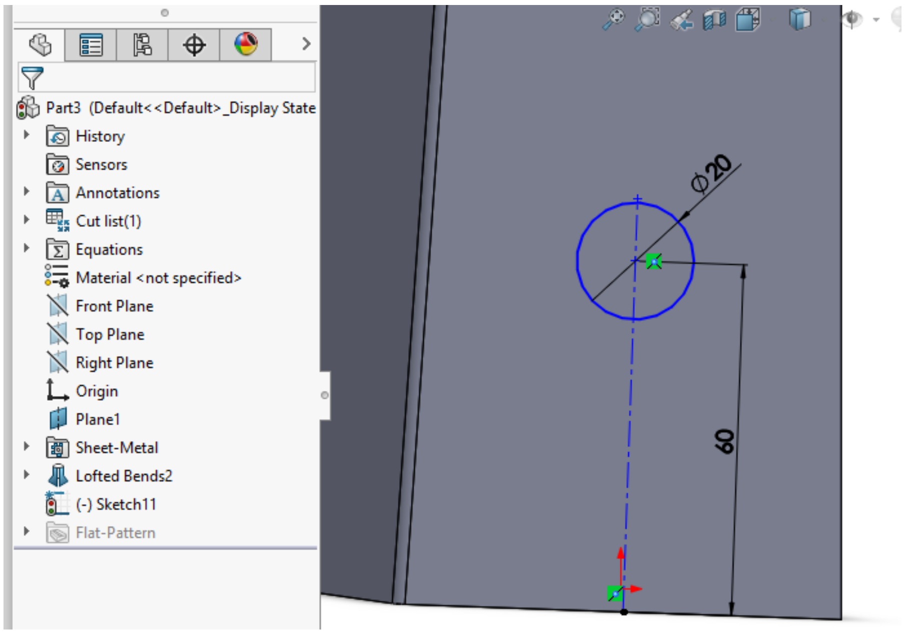
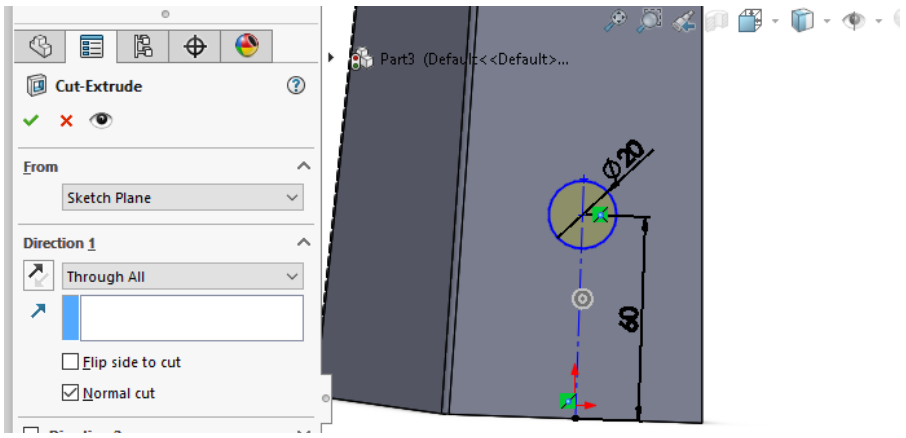
<table>
<tr>
<td>46</td>
<td>Set following features:
1. Select the option 'formed'.
2. Thickness of the material is 1.5mm.
3. The number of bending lines is 2.
4. Select the upper sketch on the right side.
5. Also select the lower sketch on the right side.
6. When the preview looks OK, Click OK.

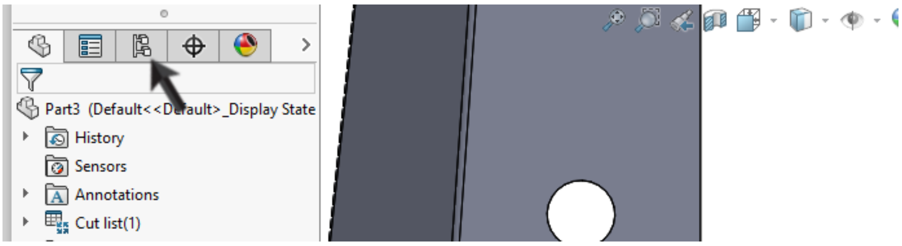
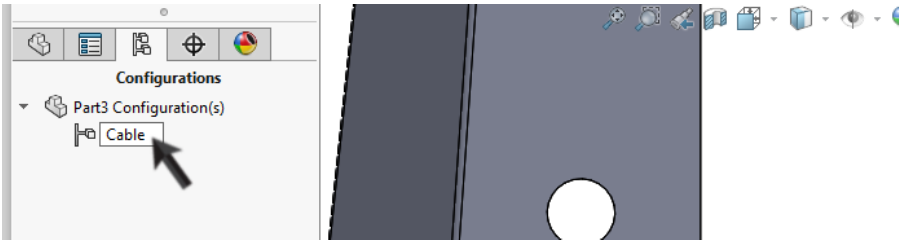
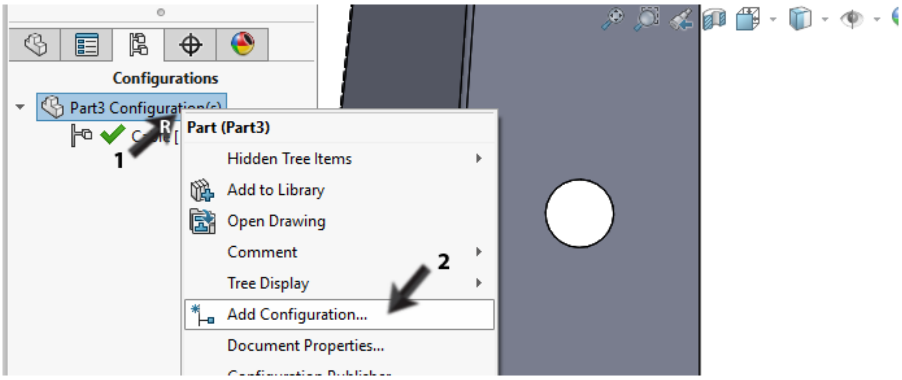
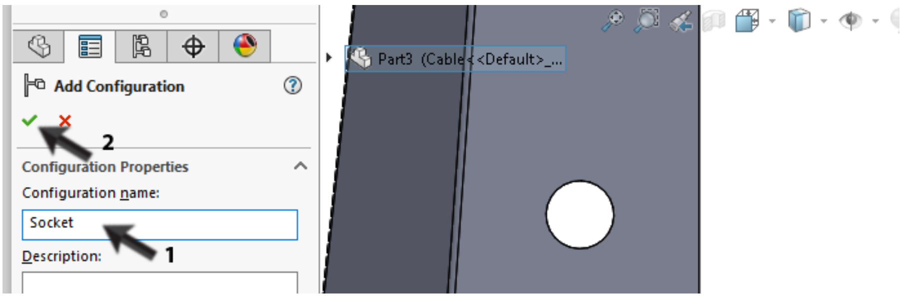
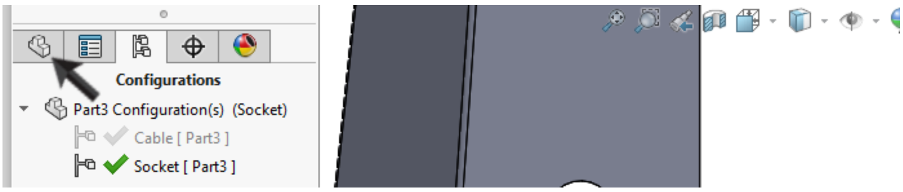
If an error message appears, the material is probably added at the wrong side of the sketch. Click the button next to the material thickness to change the direction.</td>
<td>
</td>
</tr>
<tr>
<td>47</td>
<td>The basic shape is ready now. We need this shape once more for the lampshade. That is why we will make a copy of this file at this point already and use it later.

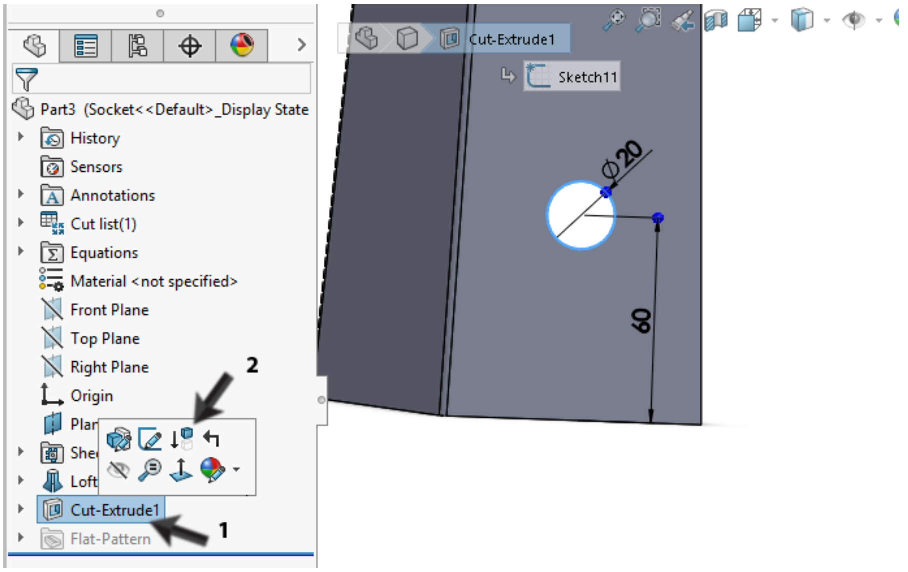
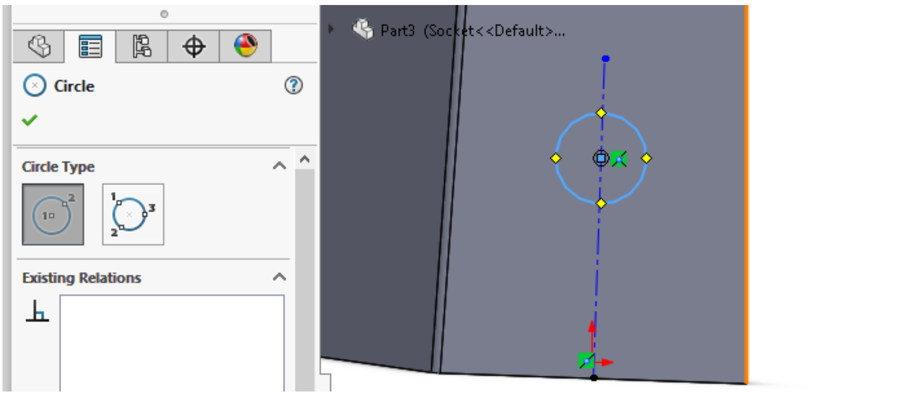
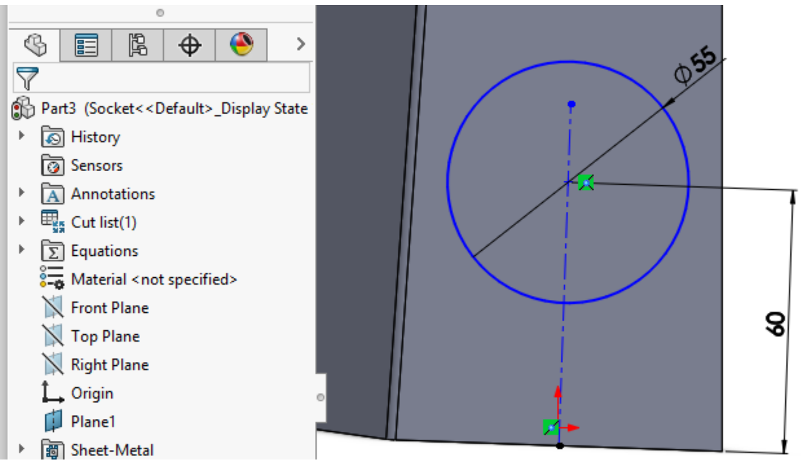
Click on the arrow next to Save in the Toolbar and click on Save as ….</td>
<td>
</td>
</tr>
<tr>
<td>48</td>
<td>1. Name the copy: shade.SLDPRT
2. IMPORTANT: Check the option 'save as copy'.
3. Click on Save.

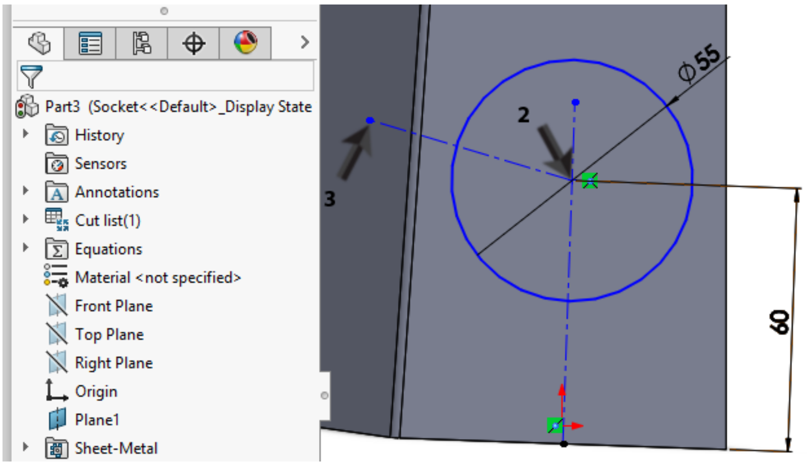
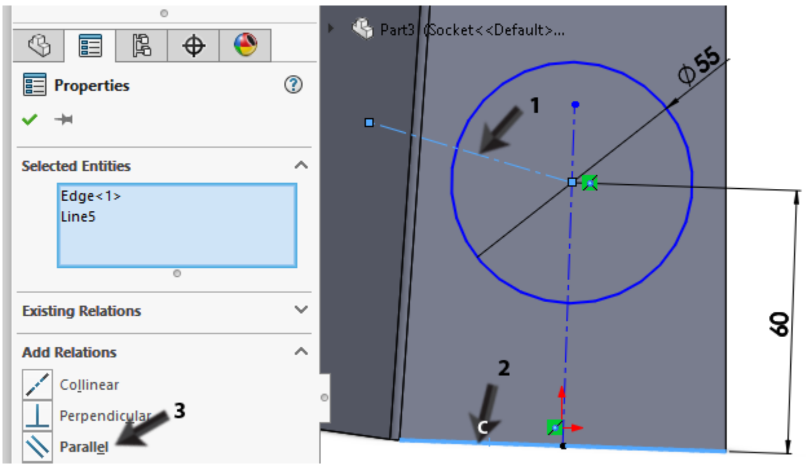
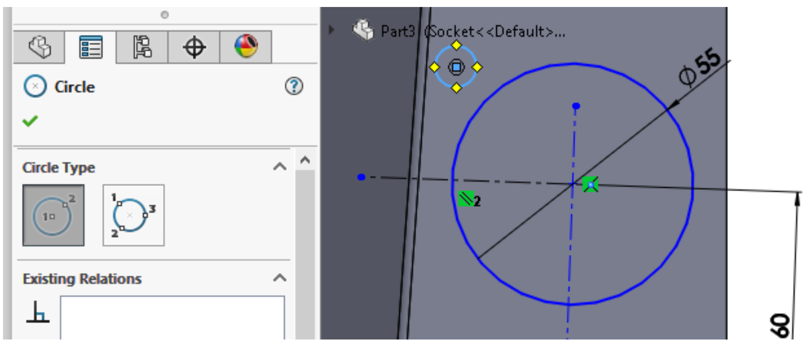
A new file has just been created (shade.SLDPRT). The name of the model we were working in has not changed.</td>
<td>
</td>
</tr>
</table>

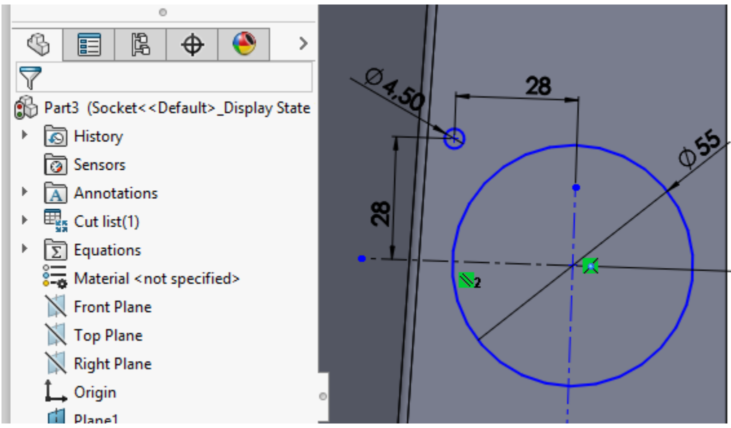
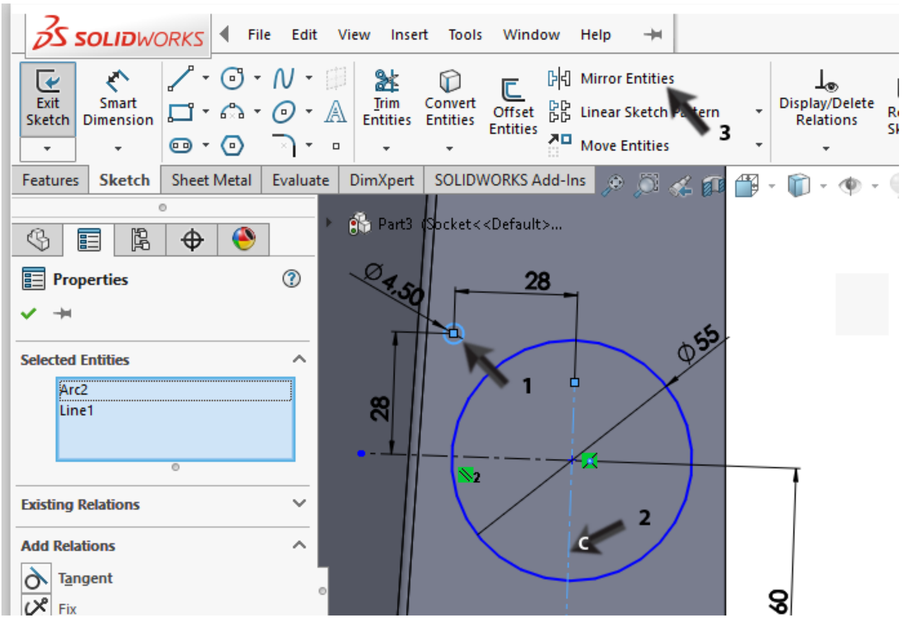
49	Next we will make a hole for the cable feed. 1. Select the plane to make a sketch 2. Click on Normal To in the menu that appears.	
50	First draw a centerline straight across the plane in which we want to draw the hole 1. Click on Centerline in the CommandManager. 2. For the first point, click on the middle of the lower edge of the plane. Notice: 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. Notice: we must draw a line which 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 <Esc>.	

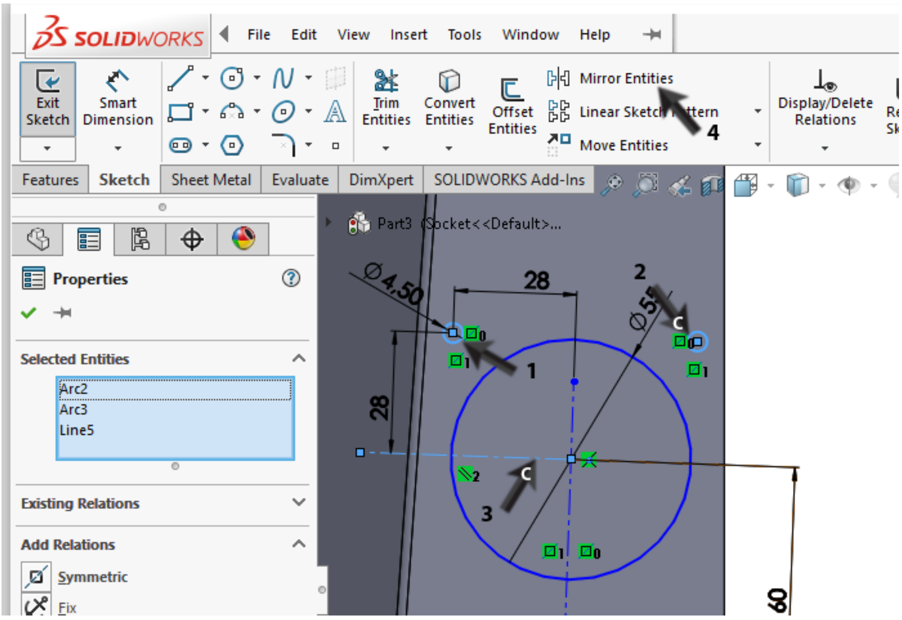
<table>
<tr>
<td>51</td>
<td>Draw a circle. Make sure the center of the circle is on the centerline.</td>
<td>
</td>
</tr>
<tr>
<td>52</td>
<td>Add two dimensions like in the illustration.</td>
<td>
</td>
</tr>
<tr>
<td>53</td>
<td>Create a Cut-Extrude from this sketch. Set the depth to Through All.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

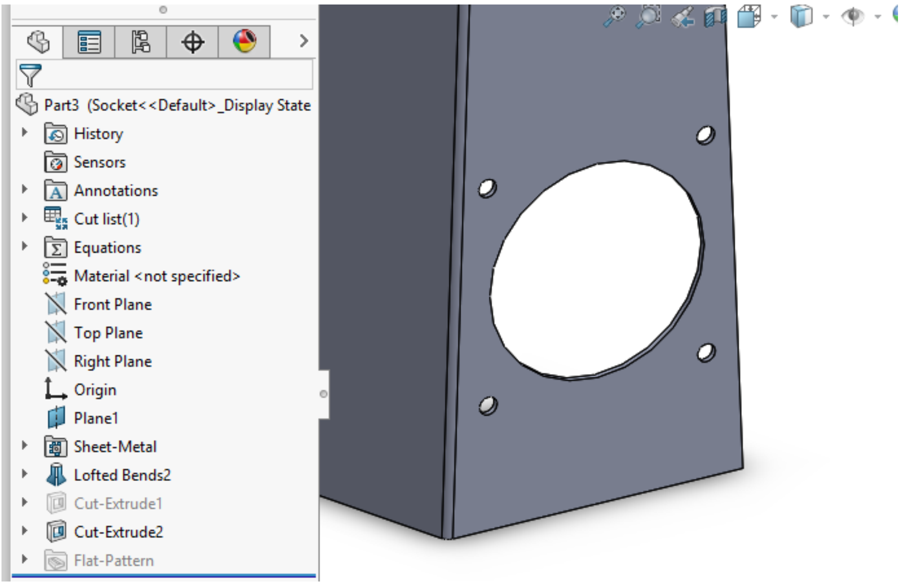
54	We will now make a second configuration of this part. Click on the Configuration Manager tab.	
55	The current configuration is called Default. Click twice (slowly) on that name and change it to Cable.	
56	1. Right-click on the upper line in the ConfigurationManager. 2. Click on Add Configuration.	
57	1. Enter the name of the new configuration in the PropertyManager: Socket. 2. Click OK.	
58	Return to the FeatureManager.	

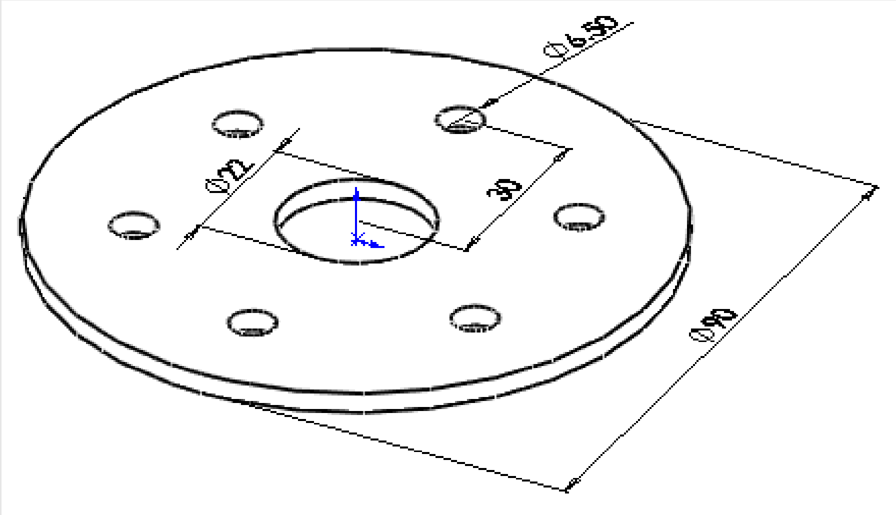
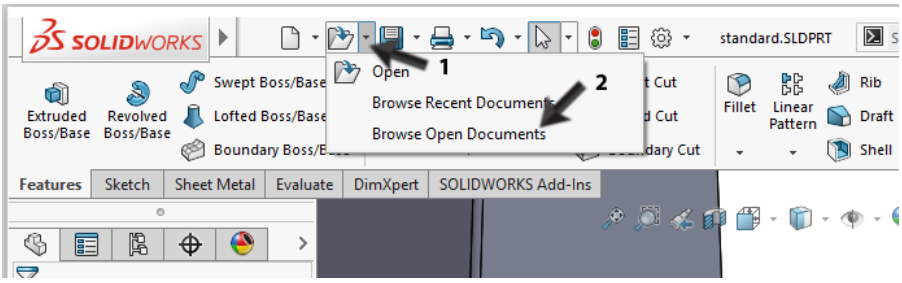
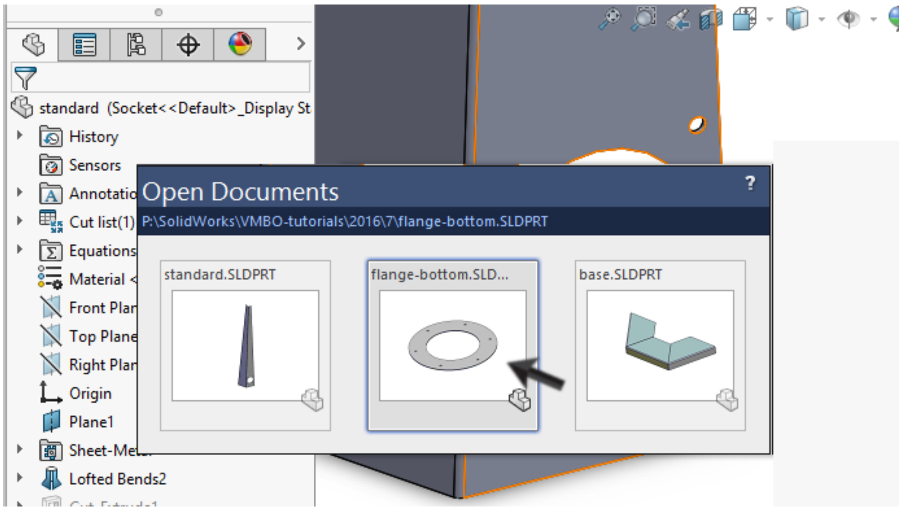
59	The configuration 'Socket' is active now. In this configuration we will suppress the cable feed hole. 1. Right-click on the feature of the hole (Cut-Extrude1) in the FeatureManager 2. Click on Suppress in the menu that appears.	
60	Next we will make a hole for the power socket. Start again with a sketch on the right plane. Draw a centerline and draw a circle, like you did in steps 49 to 52.	
61	Add the dimensions as shown in the drawing on the right.	

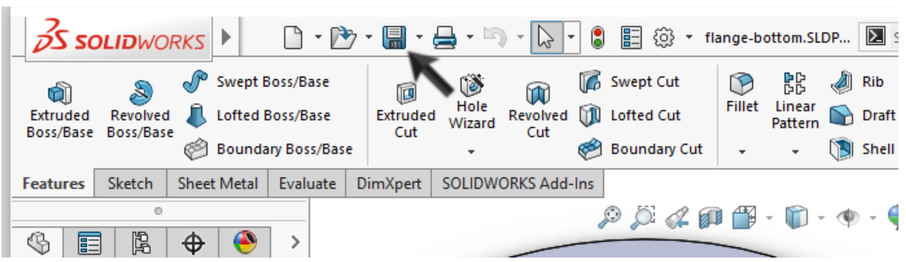
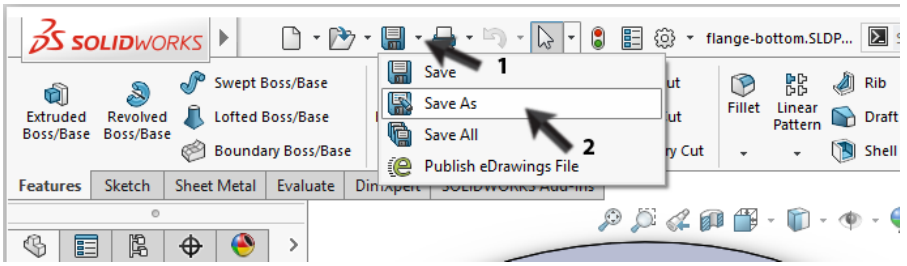
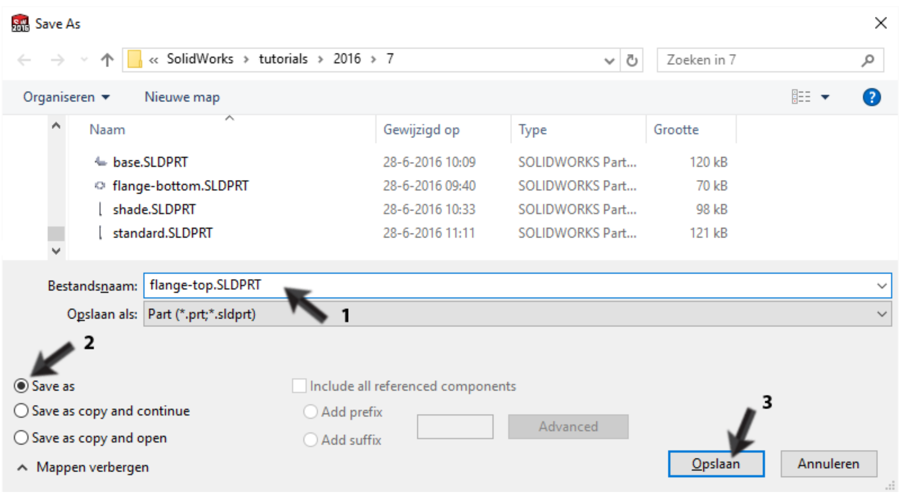
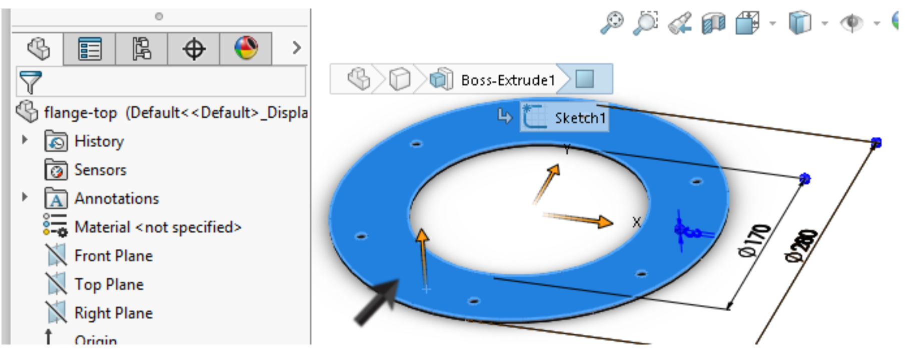
<table>
<tr>
<td>62</td>
<td>We have to create four mounting holes now. First we draw a horizontal centerline.
1. Click on Centerline in the CommandManager.
2. Click on the midpoint of the circle to set the first point.
3. Click outside the circle to get the second point. NOTICE: this is not a horizontal line. Therefore, you can even better draw it under an angle, in order to avoid any unwanted relations.
4. Push <esc> to close the Centerline-command.</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>

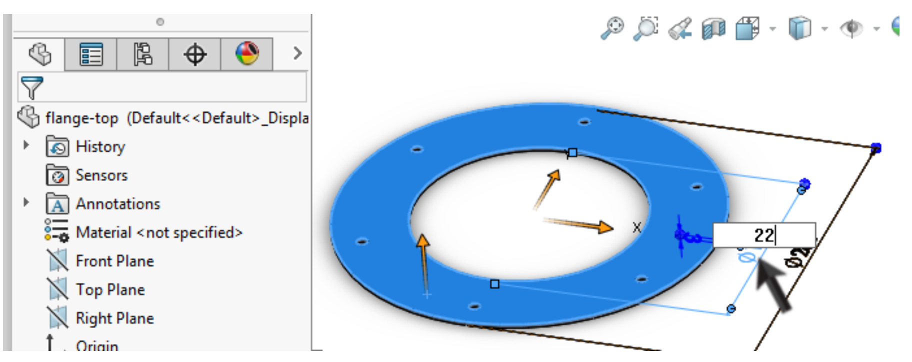
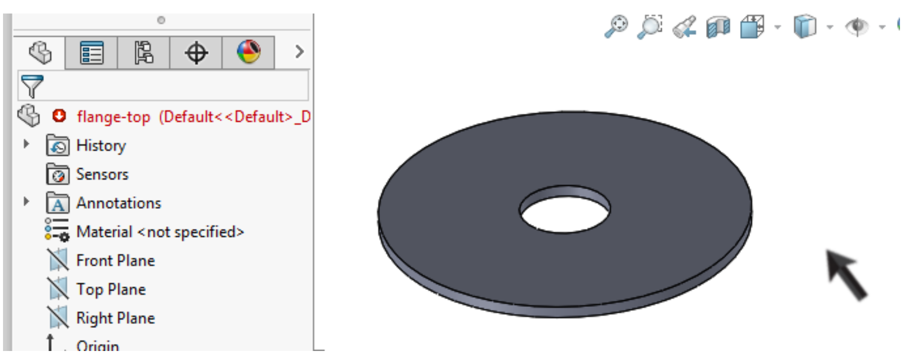
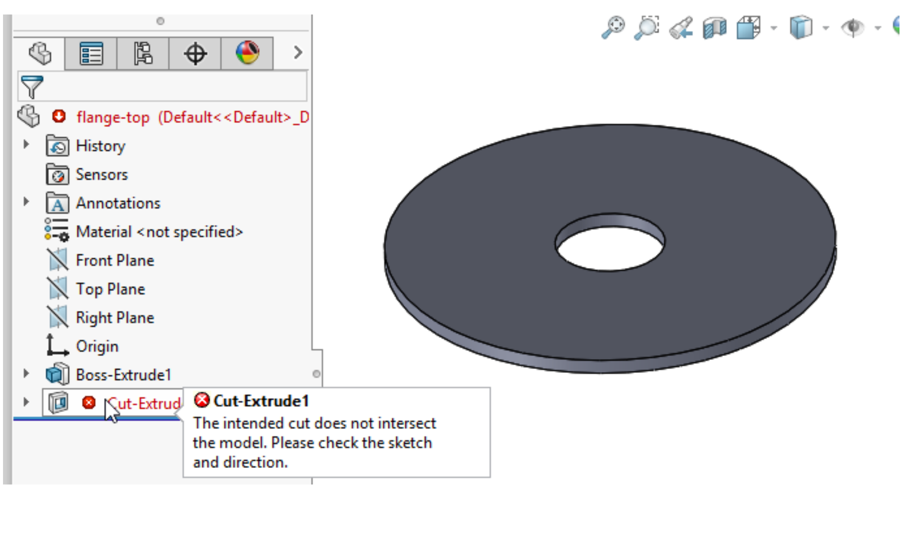
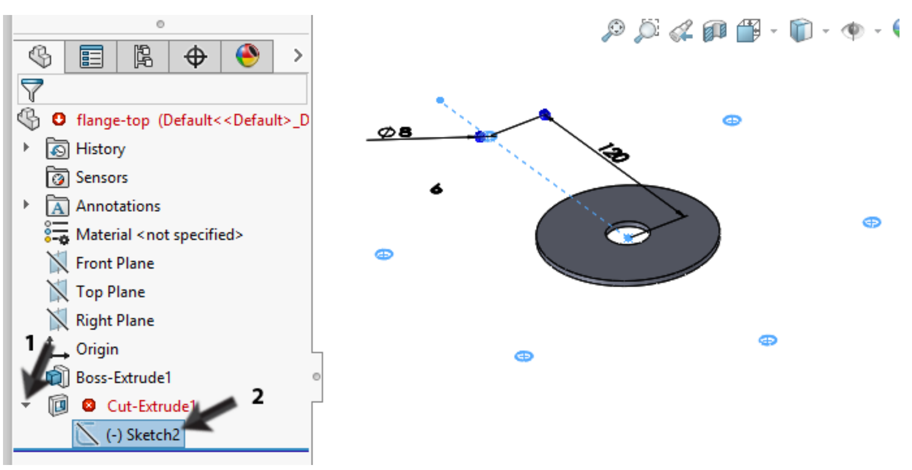
<table>
<tr>
<td>65</td>
<td>Give the circle a dimension: look at the illustration.</td>
<td>
</td>
</tr>
<tr>
<td>66</td>
<td>1. Select the small circle
2. Push the <ctrl>-key and select the vertical centerline
3. Open (when necessary) the extended menu in the CommandManager
4. Click on Mirror.</td>
<td>
</td>
</tr>
<tr>
<td>67</td>
<td>Select both circles AND the horizontal centerline.

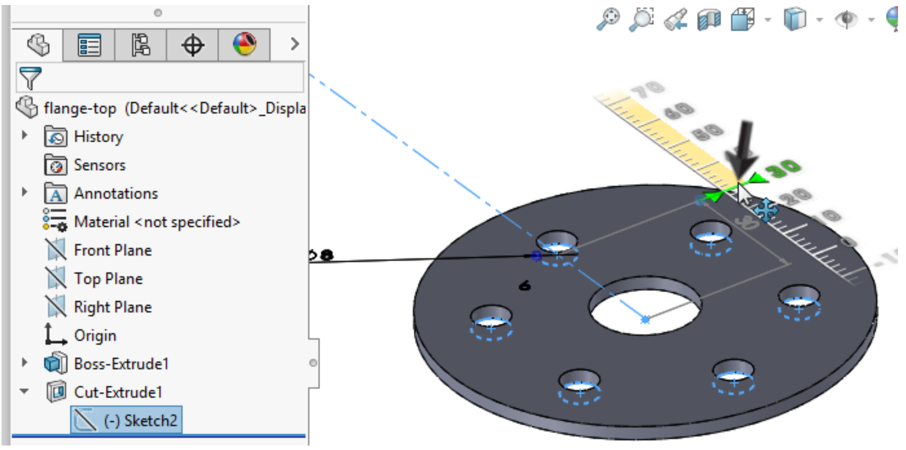
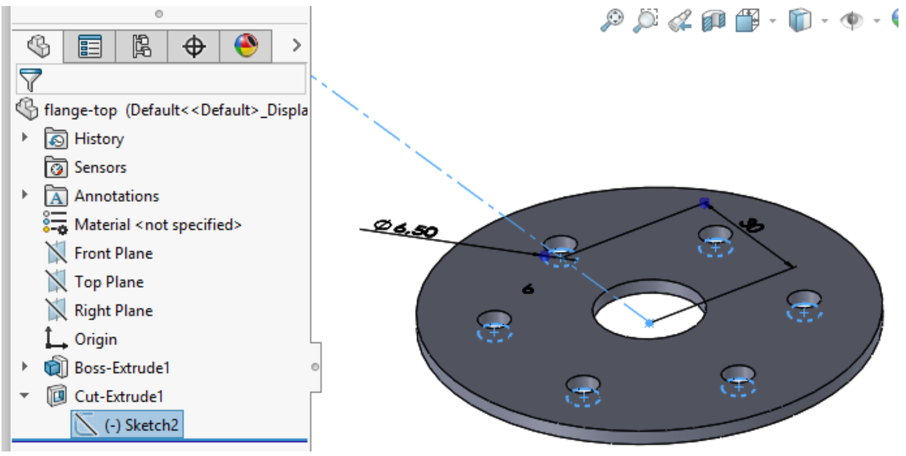
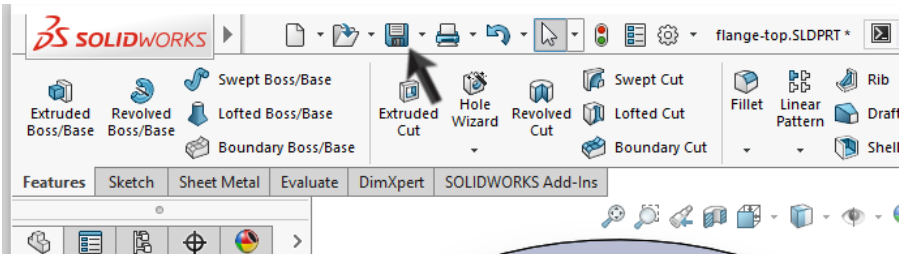
Click on Mirror in the CommandManager again. Now you will have four mounting holes.</td>
<td>
</td>
</tr>
</table>

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 (use Save As…) standard.sldprt.	

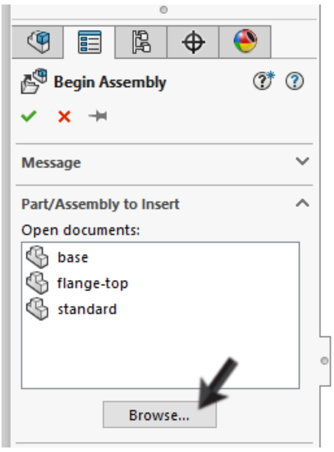
	Work plan	The next part will be the top plate. This part looks very much the same as the flange-bottom plate, which we have made in the first place. Only the dimensions are different. For this reason, we will not make an new part. We will make a copy of the first part instead and will adapt it afterwards. 
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.	

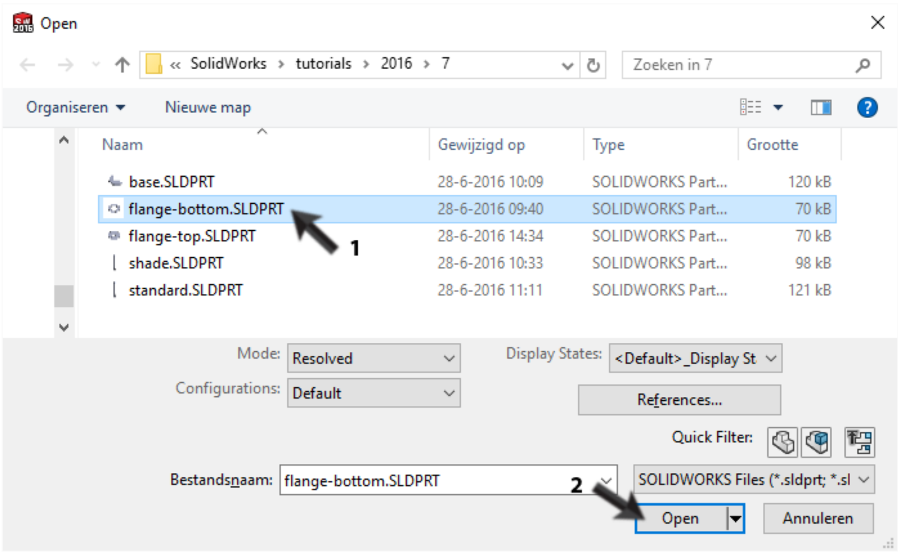
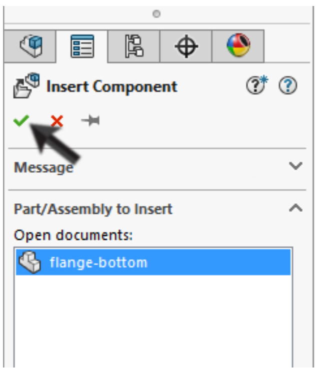
72	Are you sure you have already saved the changes in this model? Just to be sure, do it now by clicking Save in the Toolbar.	
73	Make a copy now: 1. Click on the arrow next to Save in the Toolbar. 2. Click on Save As…	
74	1. Change the name of the file to flange-top.sldprt 2. Make sure the option ‘Save as’ has been selected. 3. Click on Save. You have renamed the file now and we will continue to work in this one.	
	Tip!	Configuration of Copy? While making the standard you have seen that 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. For example the standard: should you decide to change the height, it must be done in both parts. A configuration is very convenient then. 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.
75	Click somewhere on the plate. You see that the dimensions appear.	

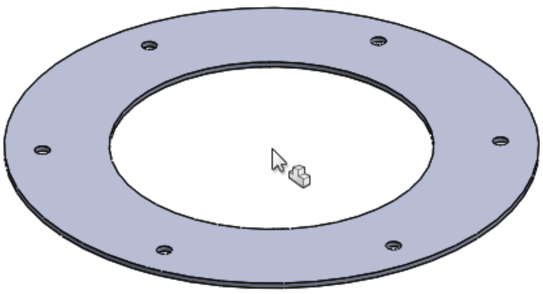
76	Click on the smallest dimension (Ø170). A small menu appears. Change the size to 22mm and push <Enter>.	
77	Change the size from 280 to 90mm similarly. Click somewhere besides the model to end the command.	
78	In the FeatureManager you will see a red x next to the last feature: this means an error has occured. 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 therefore useless.	
79	1. Click on the + symbol before the hole feature (Extrude2) in the FeatureManager 2. Click on the sketch that appears. In the model you can see the holes now, which are outside the flange very clearly.	

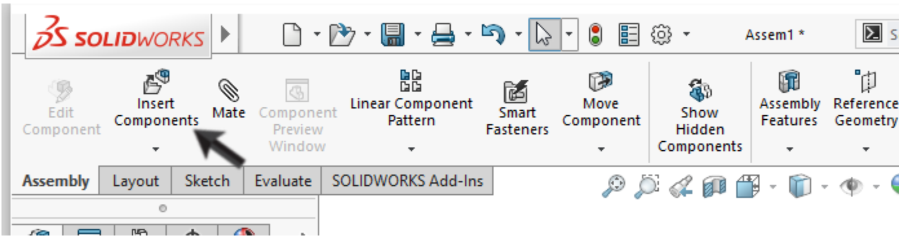
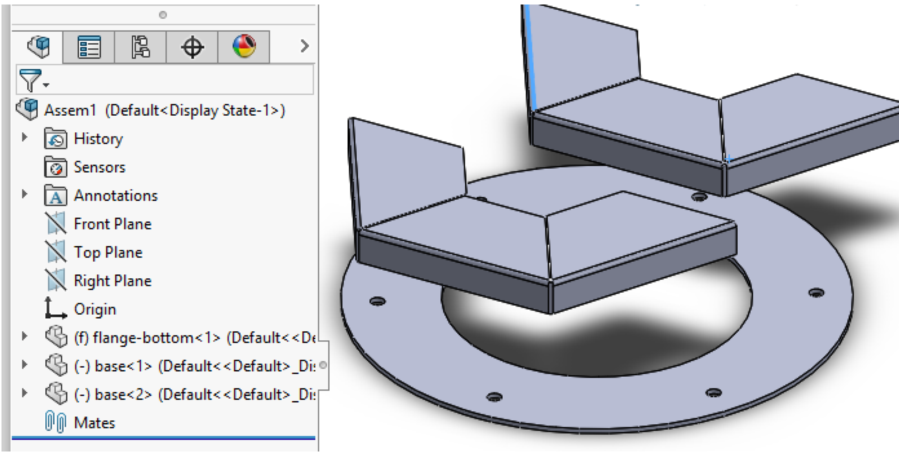
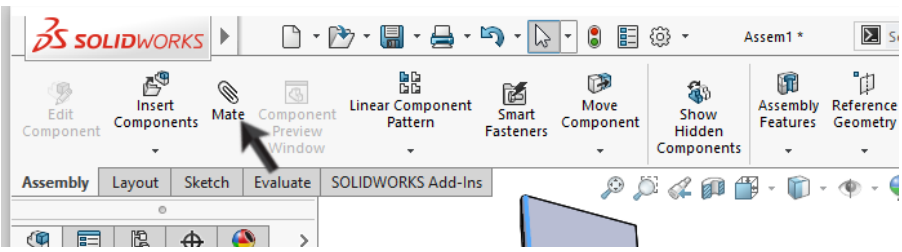
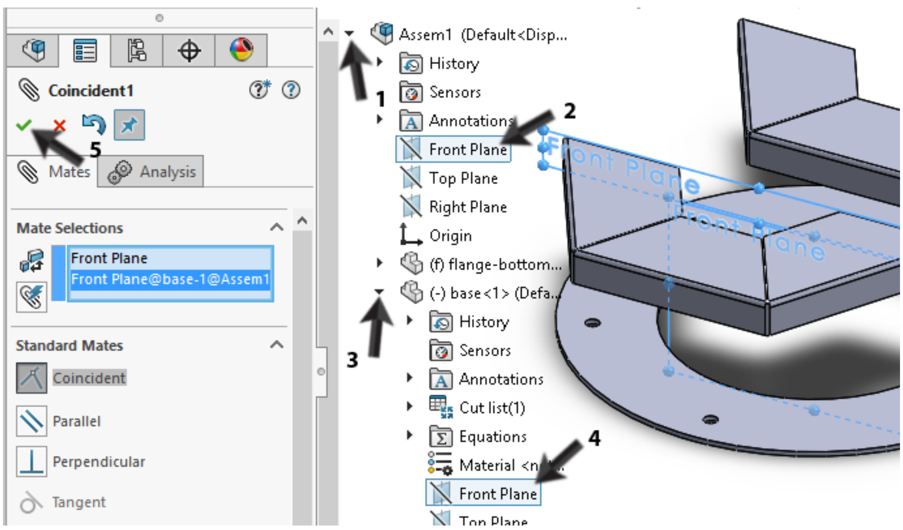
	Tip!	Sooner or later you will get errors in SOLIDWORKS. Every change you make will mean that SOLIDWORKS recalculates the entire model and looks if everything is still 'logical'. If not, an error occurs. What can go wrong? You have just seen an example: by changing the size of the ring, the holes 'drop out'. This is something that SOLIDWORKS 'does not understand'. Another very frequent problem: you have just made a sketch on a plane in a feature and threw away 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, you can imagine. When you see an error, try to solve the problem. You first reaction may be: 'I would better draw this part again' but it saves you a lot of time if you get smarter in 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 a red text you can 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 fill 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.	

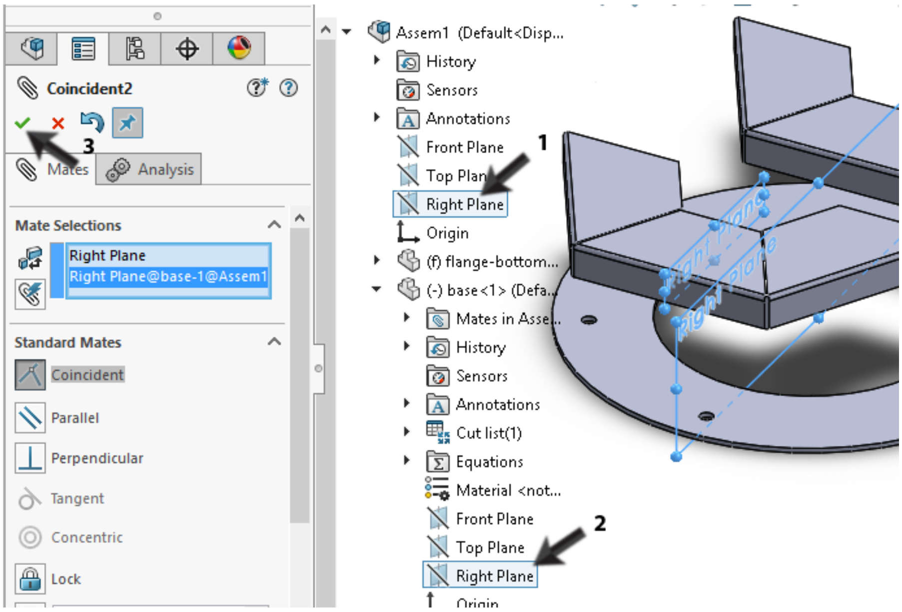
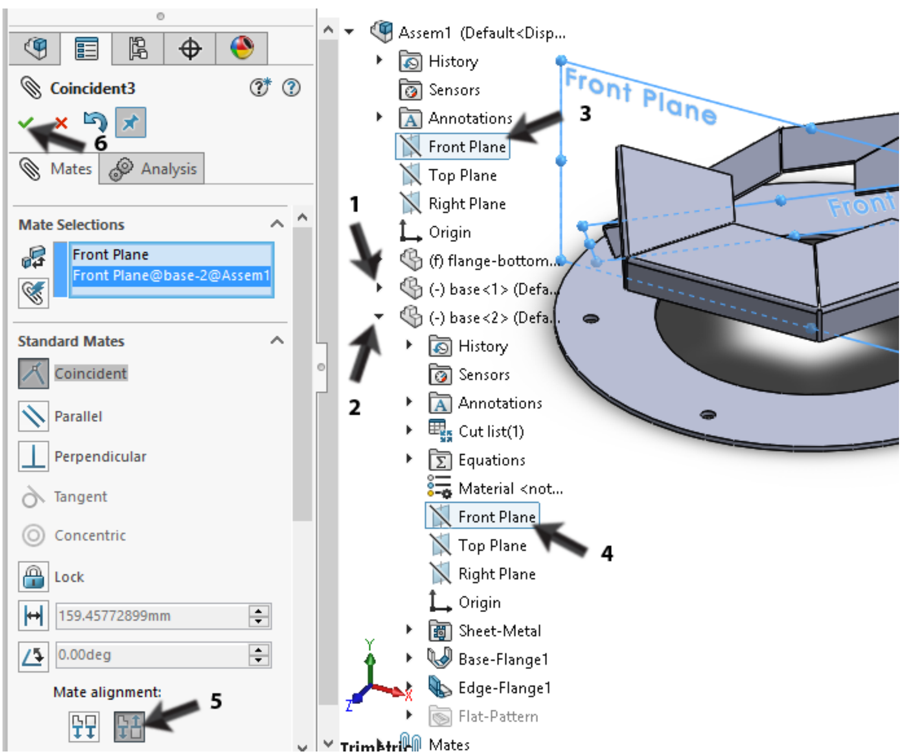
<table>
<tr><td></td><td>Work plan</td><td>All parts of the base of the garden light are ready now. We can make an assembly of them.

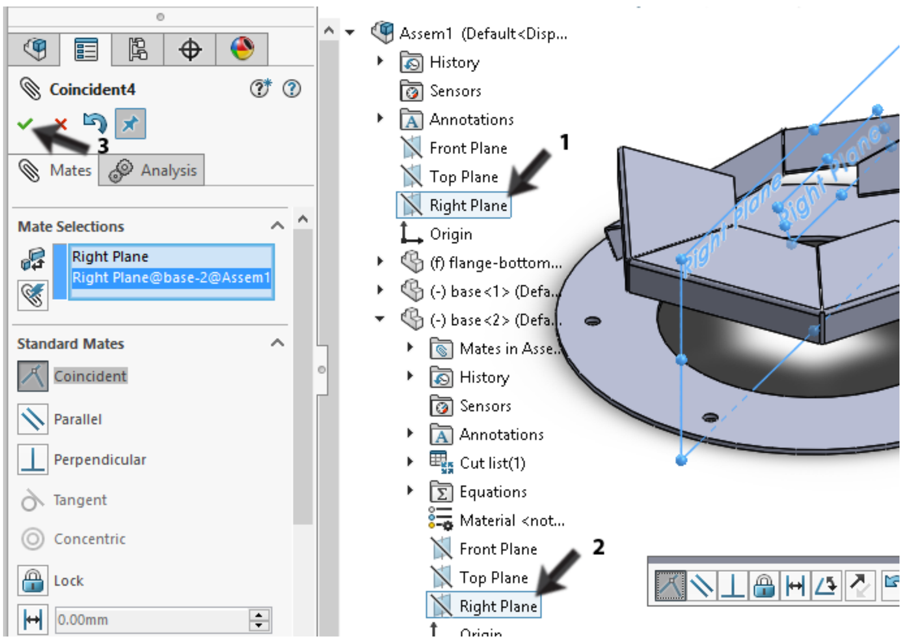
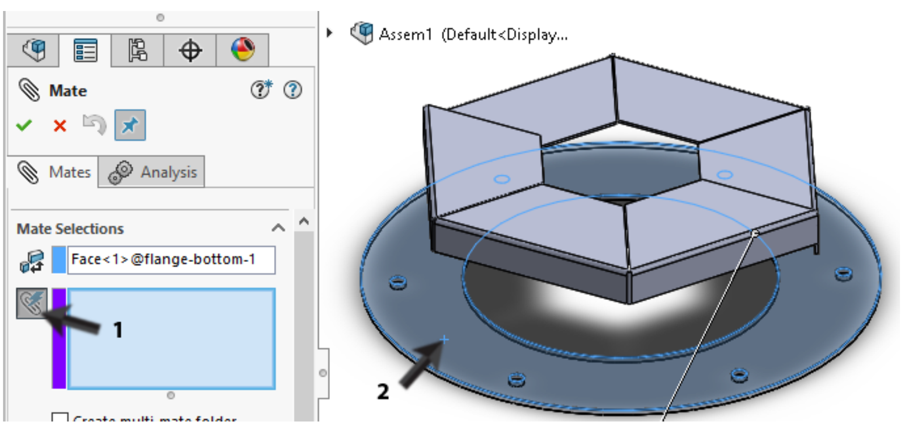
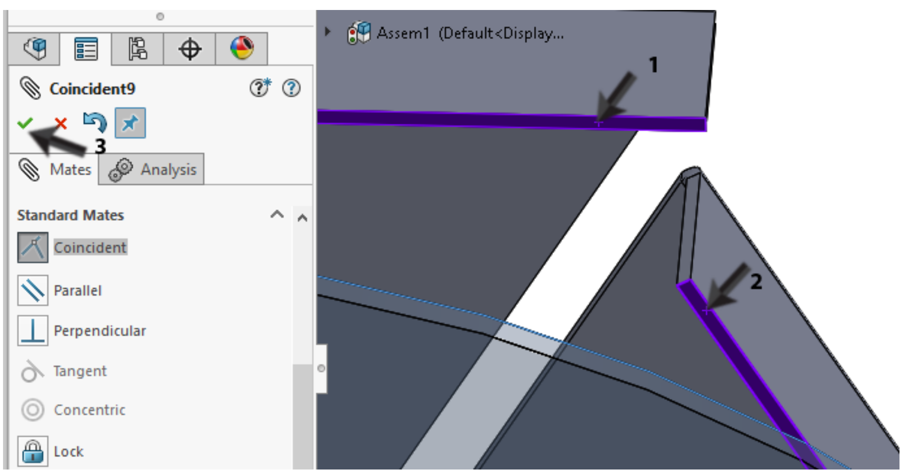
Because all parts have their midpoint in the origin, we can use the Front- en Right-planes for a lot of mates. By combining these planes of all the parts, their position is already determined. We only have to set the height then.</td></tr>
<tr><td>83</td><td>Open a new assembly.</td><td></td></tr>
<tr><td>84</td><td>First we must choose the part ‘flange-bottom’. This is probably not open at this point. Therefore, click on Browse.</td><td>
</td></tr>
<tr><td>85</td><td>1. Select the file ‘flange-bottom’

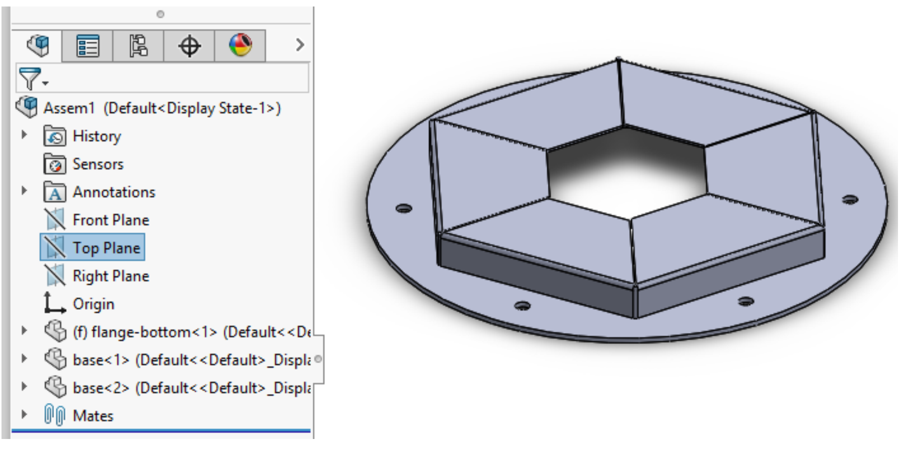
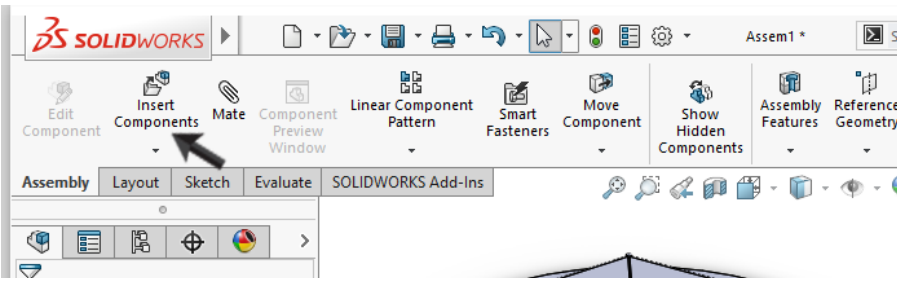
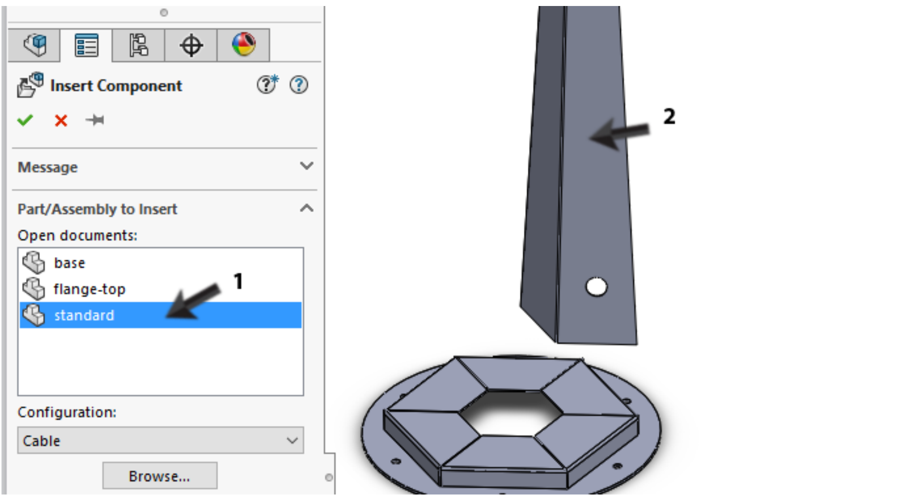
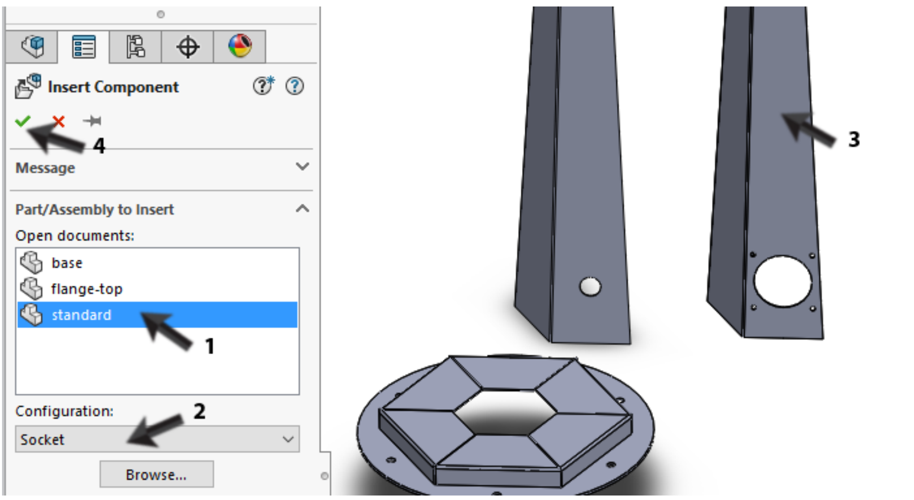
2. Click on Open.</td><td>
</td></tr>
<tr><td>86</td><td>Do NOT click randomly to place the part, but Click OK in the PropertyManager. The part will be placed exactly on the origin.</td><td>

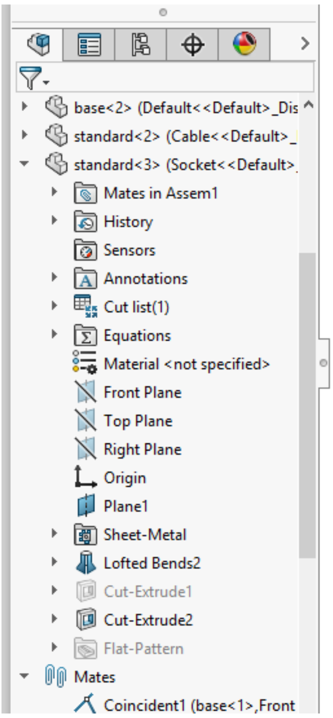
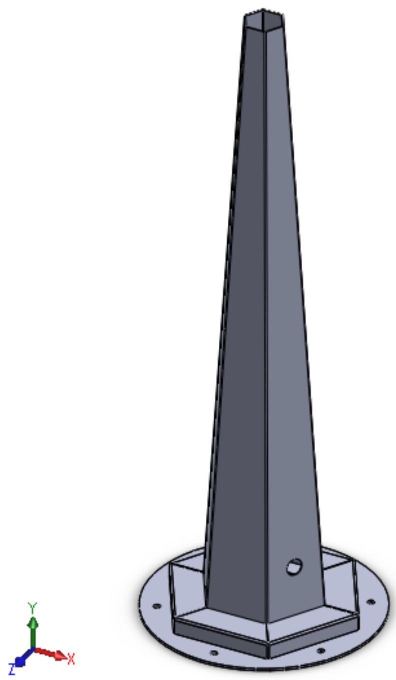
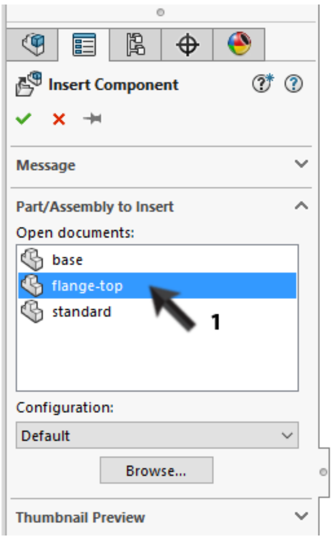
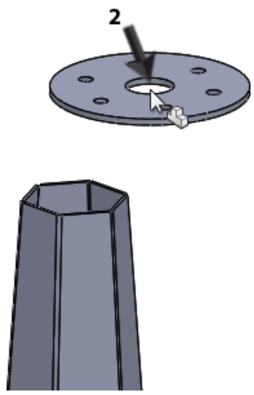
</td></tr>
</table>

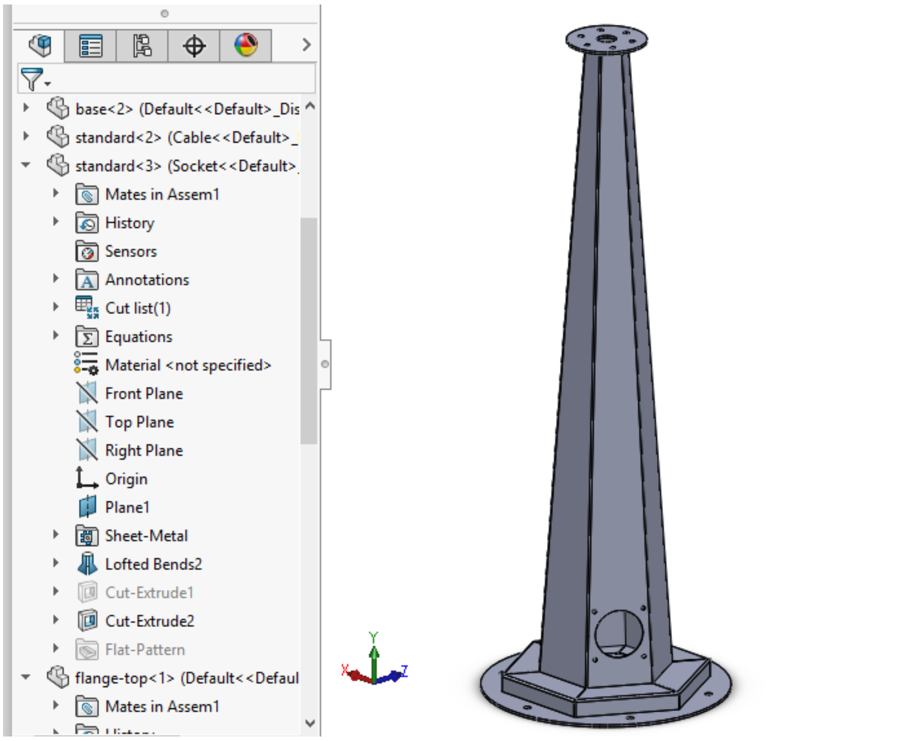
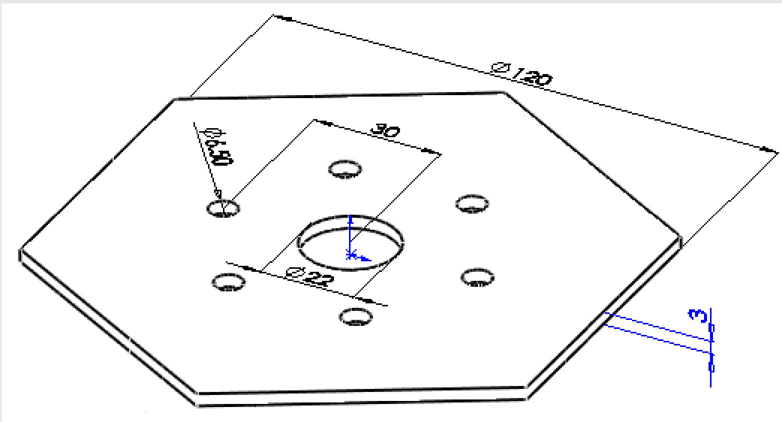
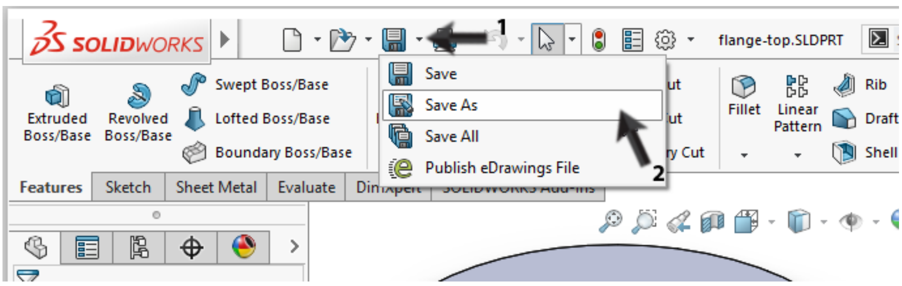
87	Click on Insert Component in the CommandManager to place the next part in the assembly.	
88	Add the file ‘base.sldprt’ twice. Put these parts at a random position in the drawing.	
89	We will add mates now. Click on Mate in the Com-mandManager.	
90	Because all parts are built around the origin, we can use the Front and Right plane to set the mates. You can select these planes in the FeatureManager, which is shown next to the model. 1. Open the FeatureM-anager 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> SOLIDWORKS chooses the mate Coincident automati-cally	

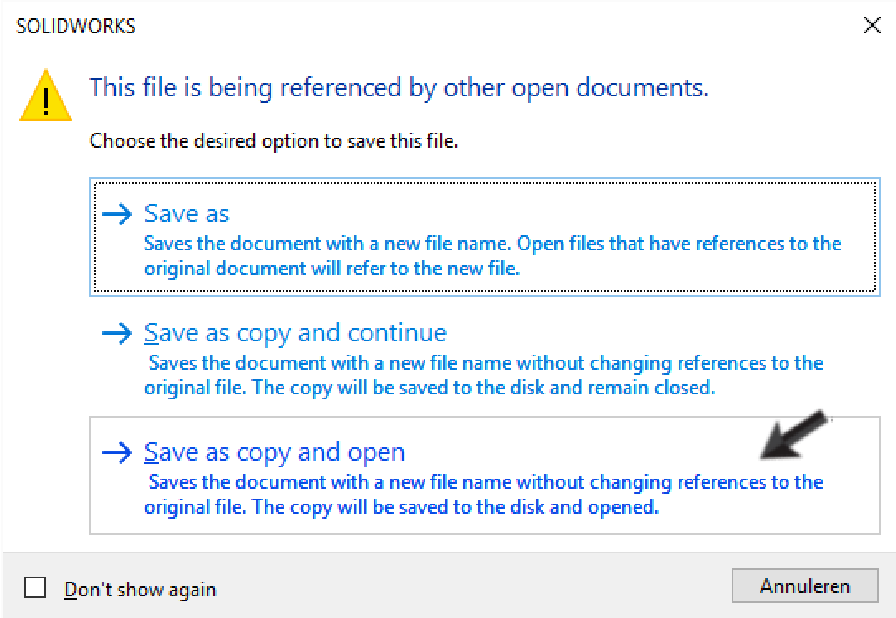
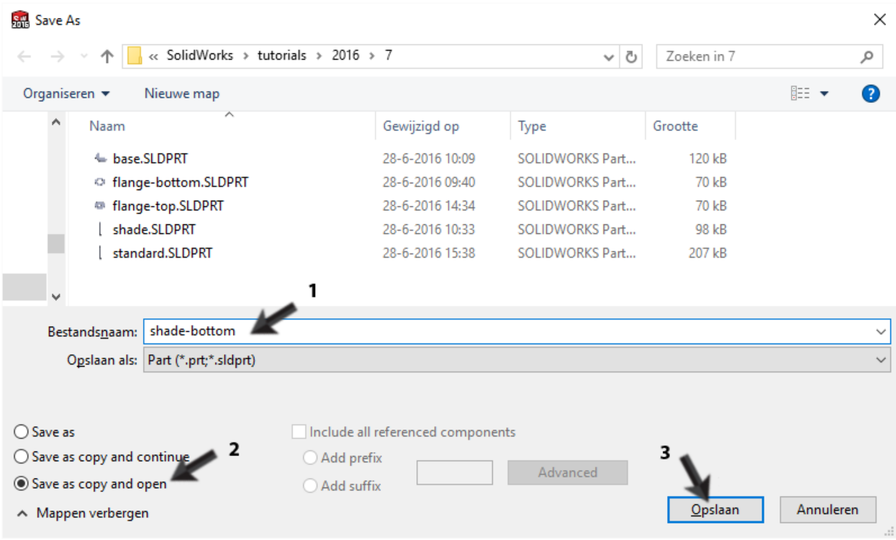
<table>
<tr><td></td><td>5. Click OK.</td><td></td></tr>
<tr><td>91</td><td>Repeat step 90, but use the Right-plane from the assembly and from base<1>.</td><td>
</td></tr>
<tr><td>92</td><td>We will do the same with base <2>:
1. Close the base<1> command tree, or else the list will be very extended: 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 has to be turned around:
5. Click on anti-aligned in the PropertyManager
6. Click OK.</td><td>
</td></tr>
</table>

<table>
<tr>
<td>93</td>
<td>Next mate the Right-planes:
1. Select the Right-plane from the assembly
2. Select the Right-plane from base<2>
3. Click OK.</td>
<td>
</td>
</tr>
<tr>
<td>94</td>
<td>Next we have to mate the parts to get them at the same height:
1. Click on Multiple Mate Mode in the Property-Manager
2. Select the top from the bottom plate.</td>
<td>
</td>
</tr>
<tr>
<td>95</td>
<td>Rotate the model and zoom in.
1,2 Select an edge from the bottom of base<1> and base<2>.
3. Click OK.
4. Click OK again to close the Mate-command.</td>
<td>
</td>
</tr>
</table>

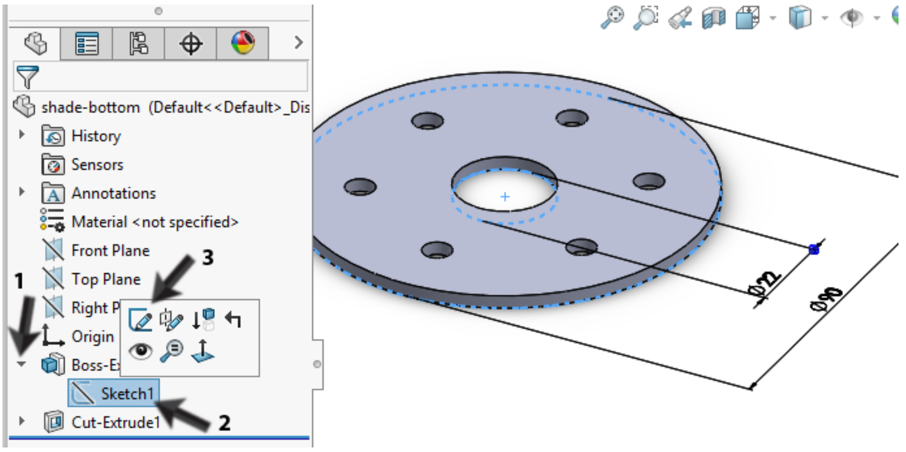
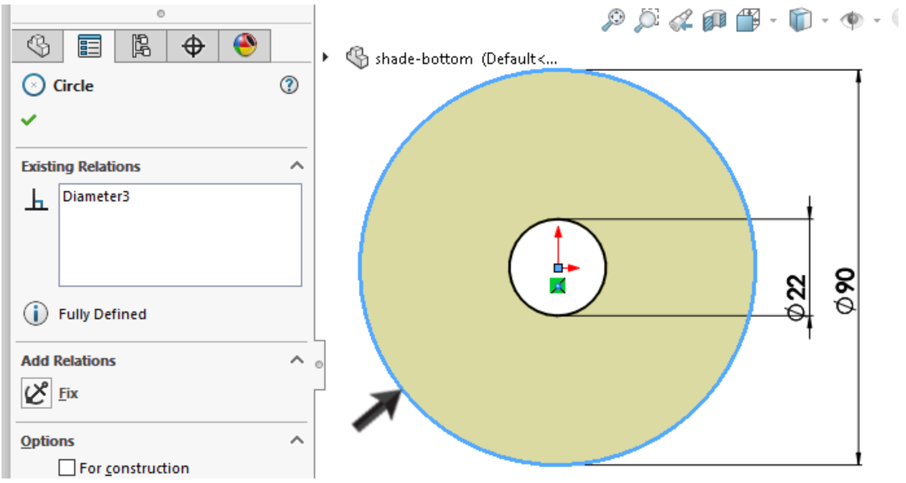
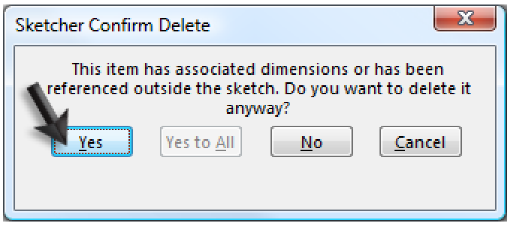
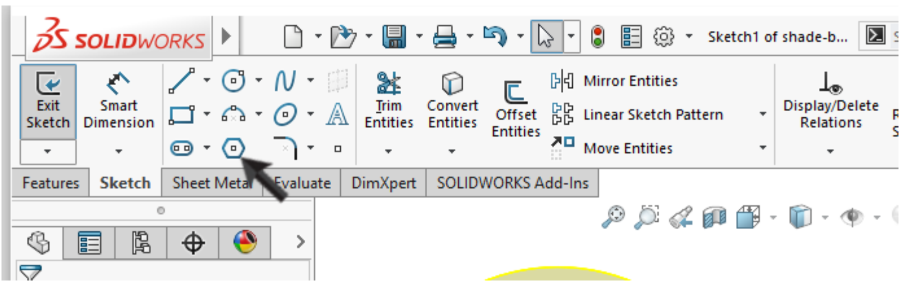
96	These three parts are now fixed.	
97	We will add the standard to the assembly too. Click on Insert Component in the CommandManager	
98	When the part standard.sldprt is still open, you can see it in the list in the PropertyManager. 1. Click on the part called standard.sldprt. 2. Put it at a random position in the model. When you did close the file before; find it by using Browse….	
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. Again, start the command Insert Component in the CommandManager. 1. Select the file standard.sldprt in the menu	

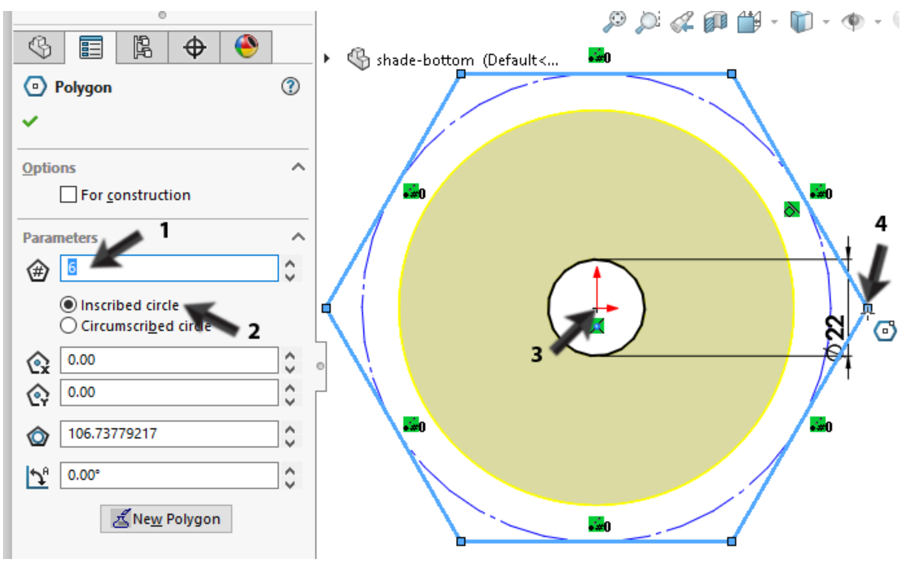
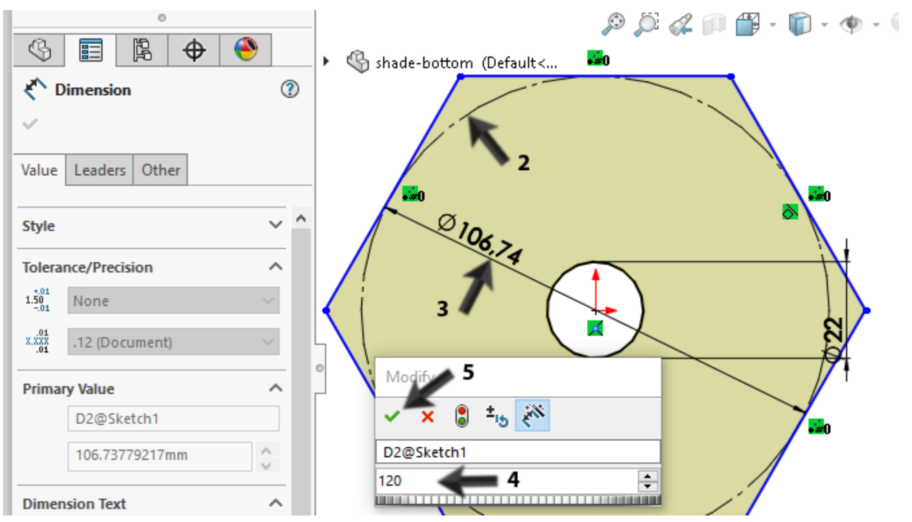
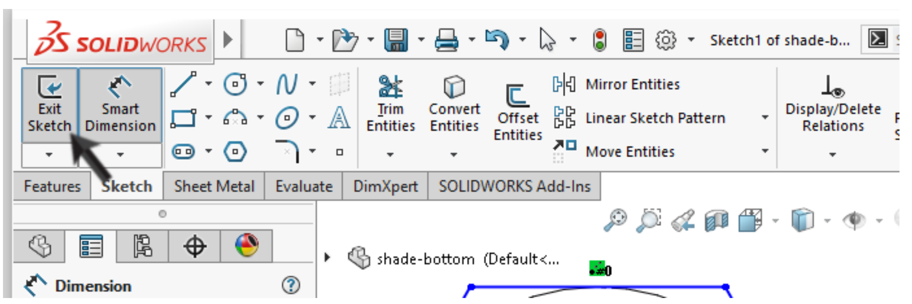
	that appears. 2. Select the configuration Cable. 3. Place the part randomly in the assembly 4. Click OK	
100	Add mates in exactly the same way as you did before. Follow steps 89 to 96.	 
101	Add the part Flange-top to the assembly	 

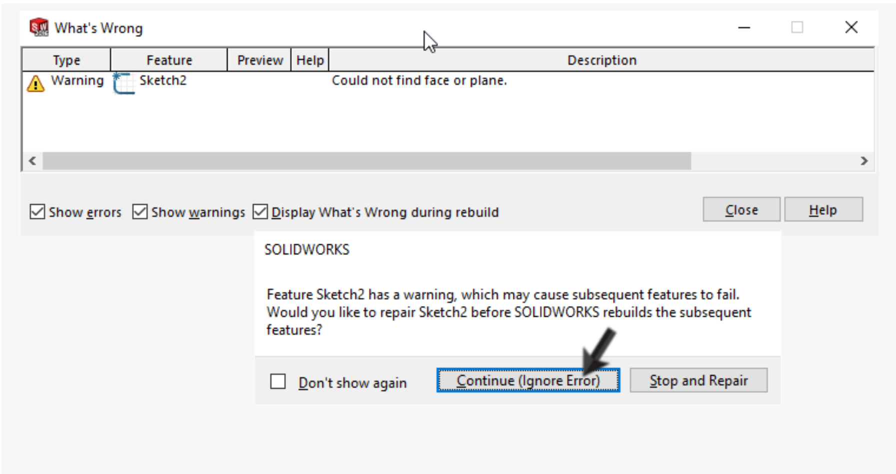
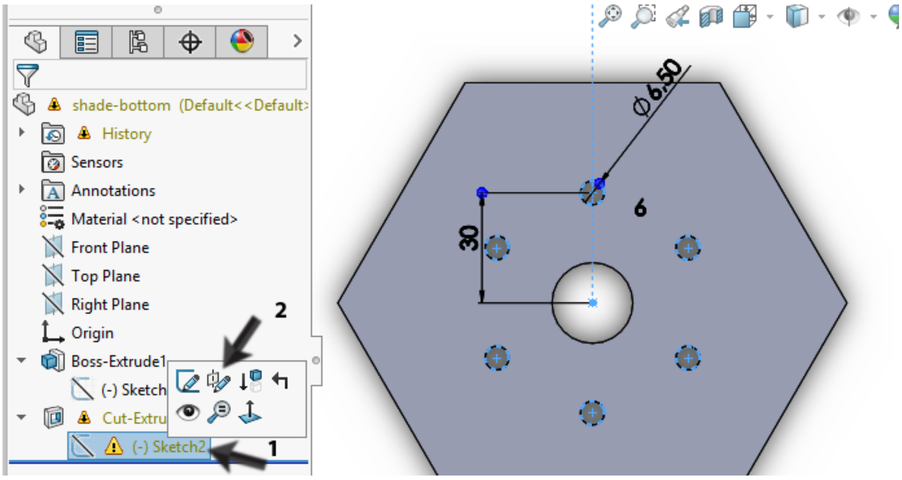
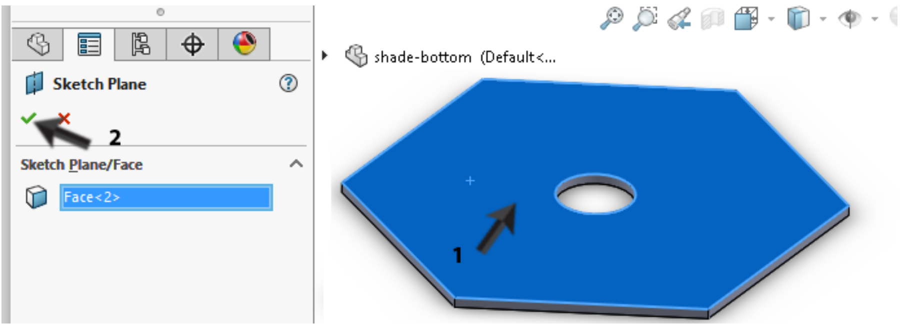
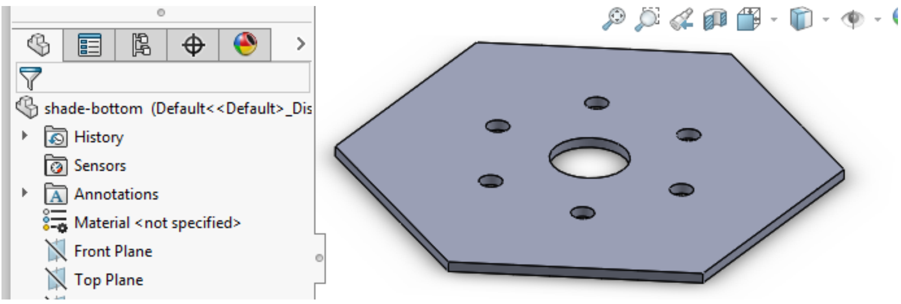
102	Add mates (use front and right planes) to position this component correctly.	
103	Save the assembly as standard-complete.sldasm	
	Work plan	We will get started with the lamp shade. We create the base plate first. As you can see in the illustration it looks a lot like the upper plate at the base of the light. Therefore, we can decide to make a copy of this part and change it. 
104	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	

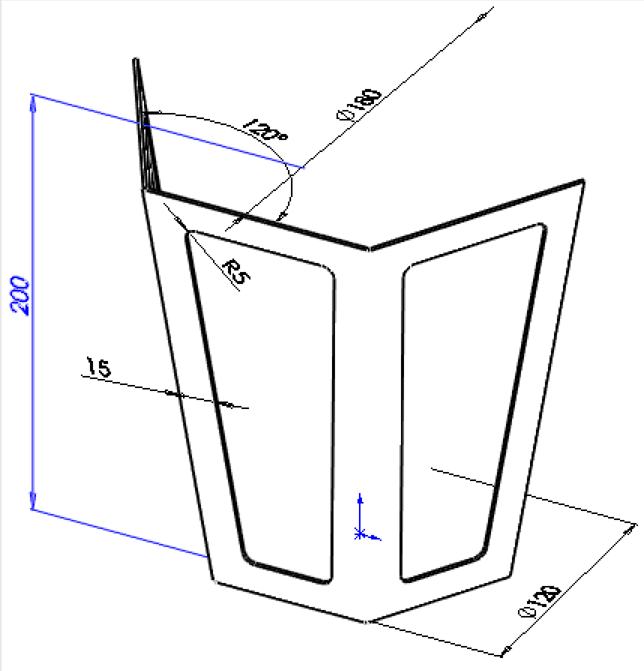
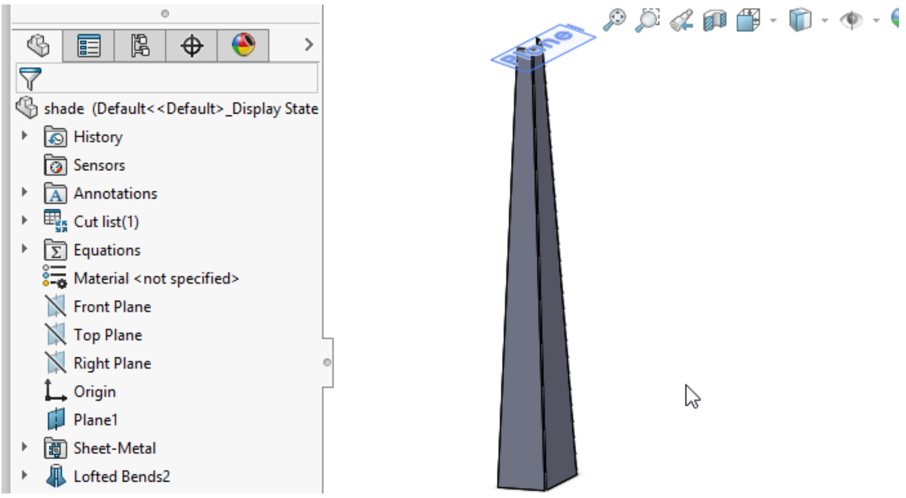
	2. Click on Save As…	
105	When this message appears, Click OK.	
106	1. Rename as: shade-bottom 2. **IMPORTANT**: check the option Save as copy. 3. Click on Save.	
	Tip!	What does the option 'Save as copy' mean? The file flange-top is used in the assembly that we have made before. If you would change the name of this part with 'Save as' the name in the assembly will also change. But we do not want this to happen in this case: this would mean that the flange-top in the assembly would be replaced by the part named shade-bottom that we have just made. By using 'Save as copy' the assembly stays the same. The new file has absolutely nothing to do with it.
	Tip!	When this all seems to be a bit too complicated for you, you could also use the Windows explorer to copy the file and rename it. To be able to do so, 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 find this part again and you will get multi-

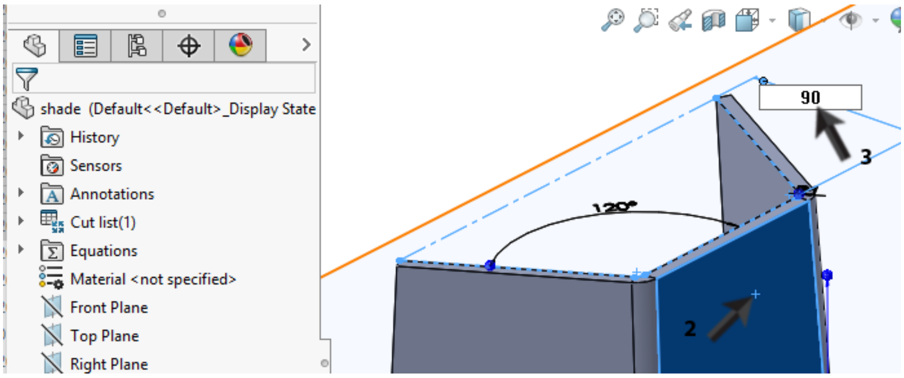
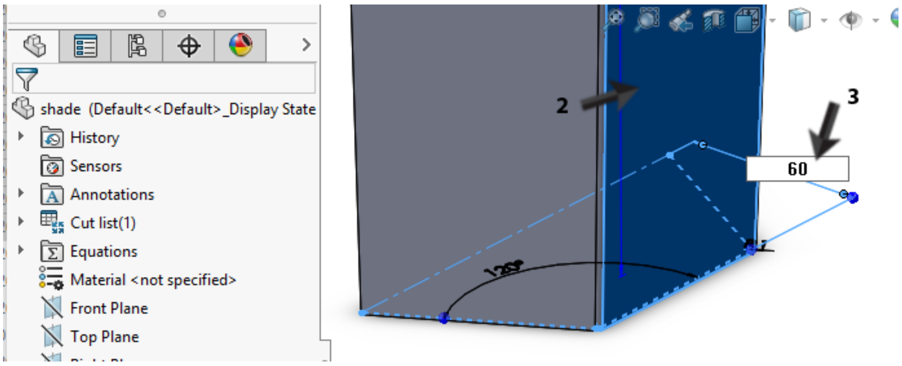
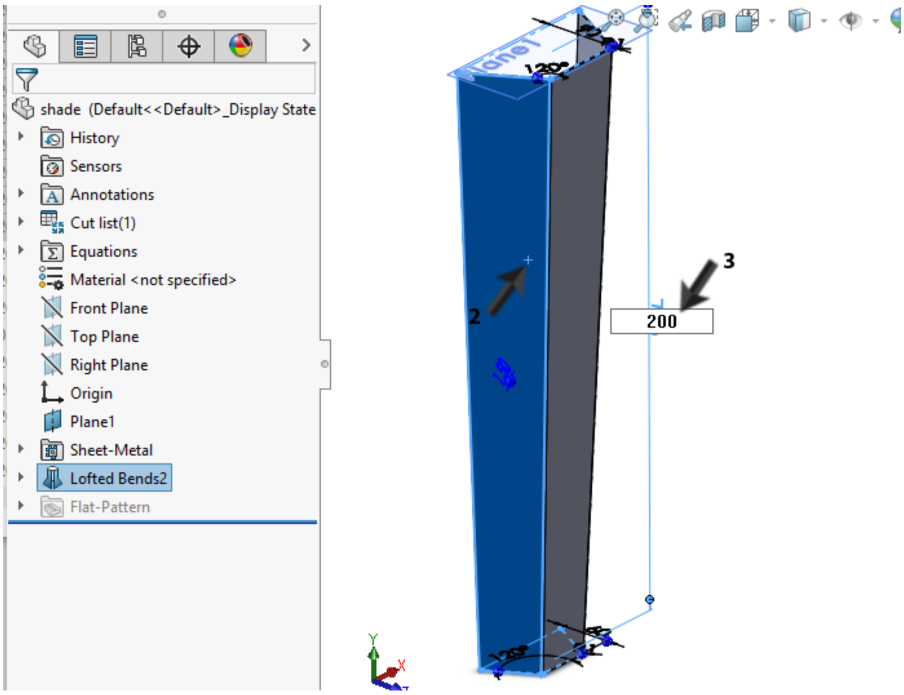
<table>
<tr><td></td><td></td><td>ple, unsolvable errors.</td></tr>
<tr><td>107</td><td>1. Click on the arrow in front of the first feature (Extrude1)
2. Right-click on Sketch1
3. Select Edit Sketch in the menu.

Rotate the sketch with Normal To.</td><td>
</td></tr>
<tr><td>108</td><td>Click on the outer circle of the sketch and push <delete>.</td><td>
</td></tr>
<tr><td>109</td><td>Click Yes in the message that appears.</td><td>
</td></tr>
<tr><td>110</td><td>Click on Polygon in the CommandManager</td><td>
</td></tr>
</table>

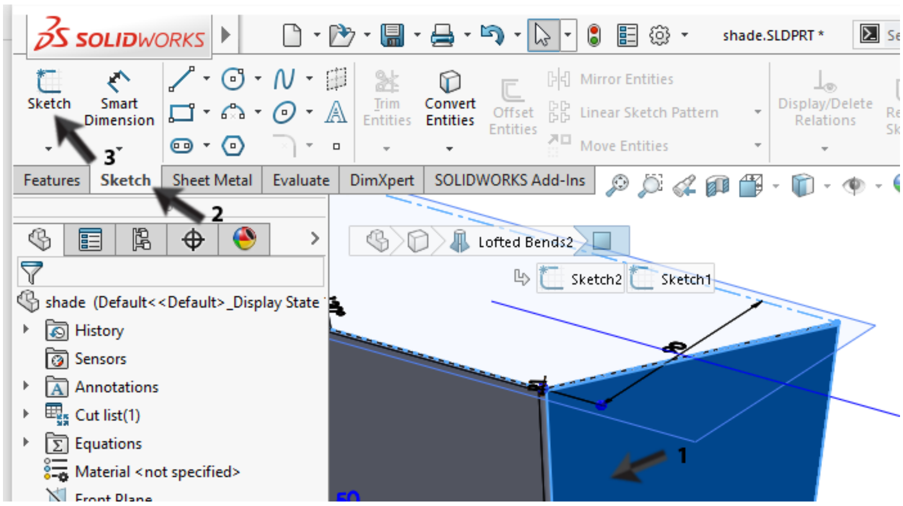
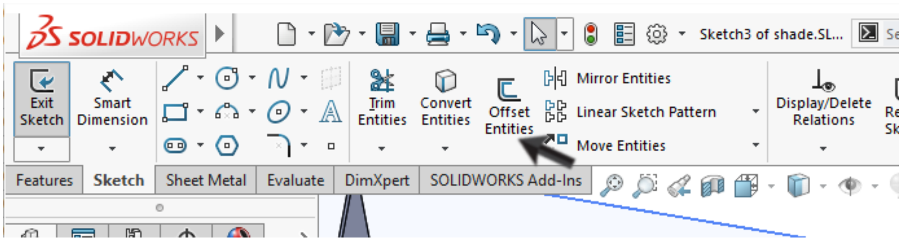
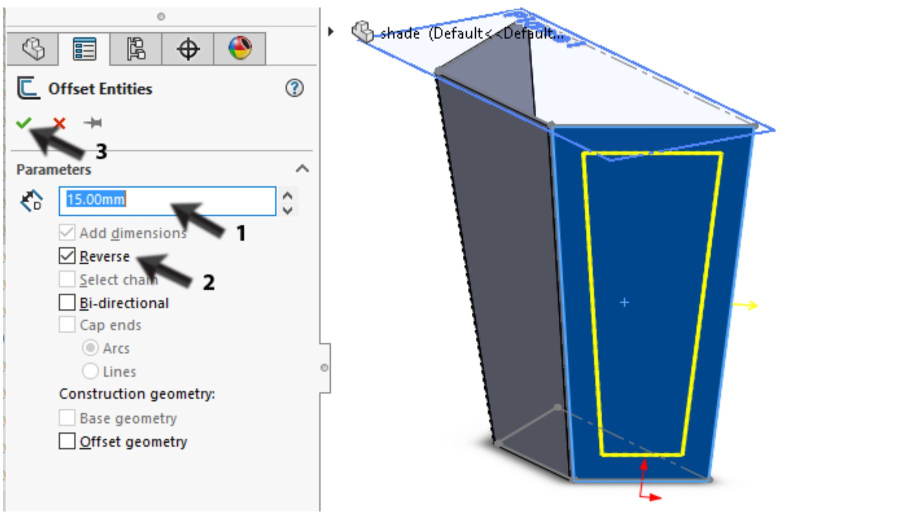
111	1. Set the number of sides to 6. 2. Make sure the option Inscribed circle is selected. 3. Click on the origin 4. Click besides the origin, horizontally to the origin. The distance does not matter.	
112	Set the size of the inside circle with Smart Dimensions. 1. Click on Smart Dimensions in the CommandManager 2. Click on the inner circle 3. Place the dimension 4. Change the value to 120mm 5. Click OK.	
113	The sketch is done now. Click on Exit Sketch in the CommandManager.	

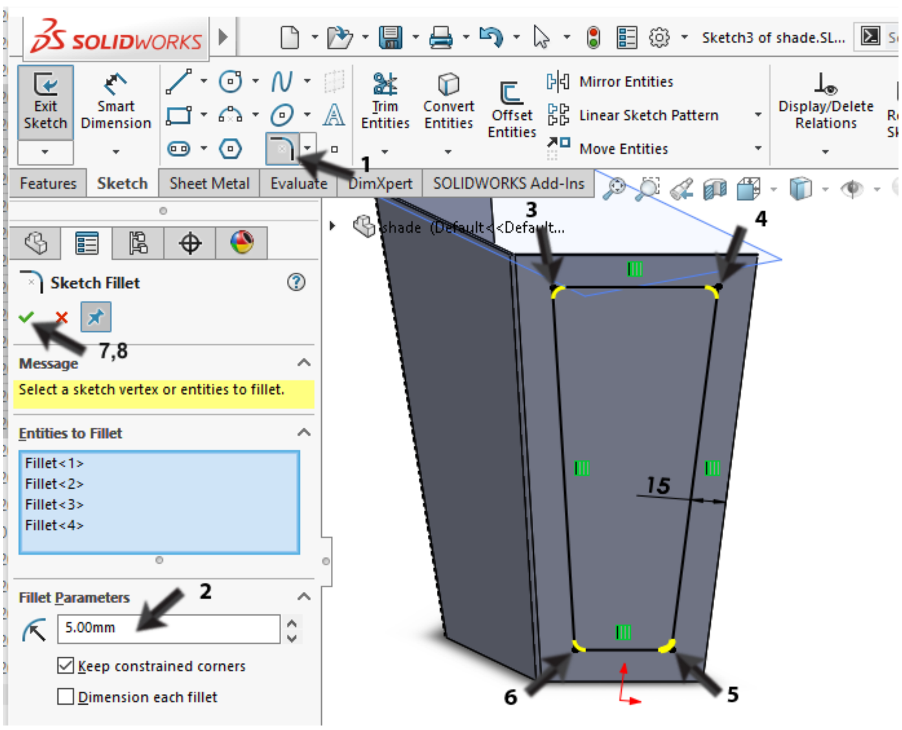
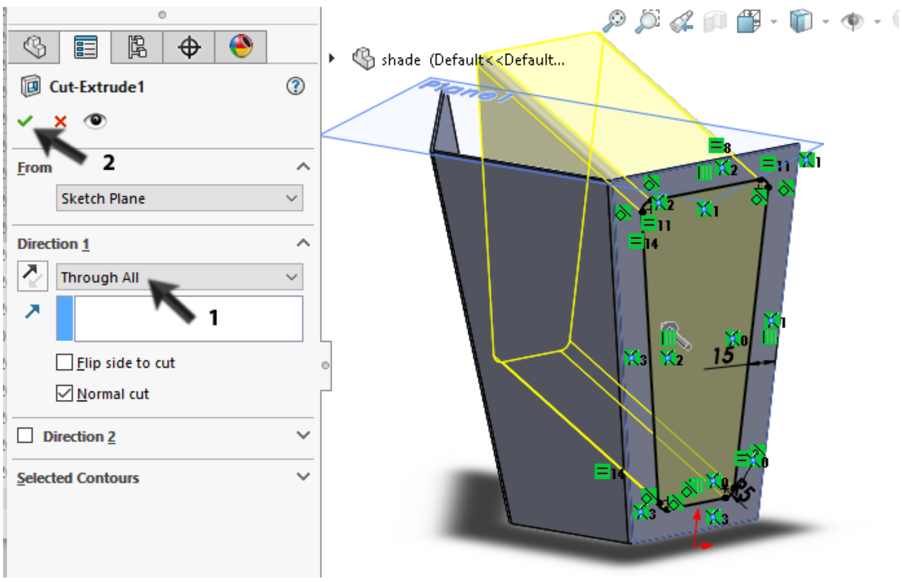
114	At this point, an error occurs! **Why?** You have just changed the first feature from this part (the plate). In this part there were six mounting holes. By changing the first feature, SOLIDWORKS does not know in which plane the sketch of the holes was drawn. Click on Close.	
115	We are going to determine 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.	
116	1. Click somewhere on the top plane of the model 2. Click OK in the PropertyManager.	
117	The error has disappeared, the part is ready. Save the file by using the Save button in the toolbar.	

	Work plan	We will start modeling 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 our needs. From that file we have to remove a few items, for instance the holes we have made at the bottom and the configurations. After that we can resize the part and open the sidewalls. 
118	Open the file shade.sldprt. This file is saved in step 47.	

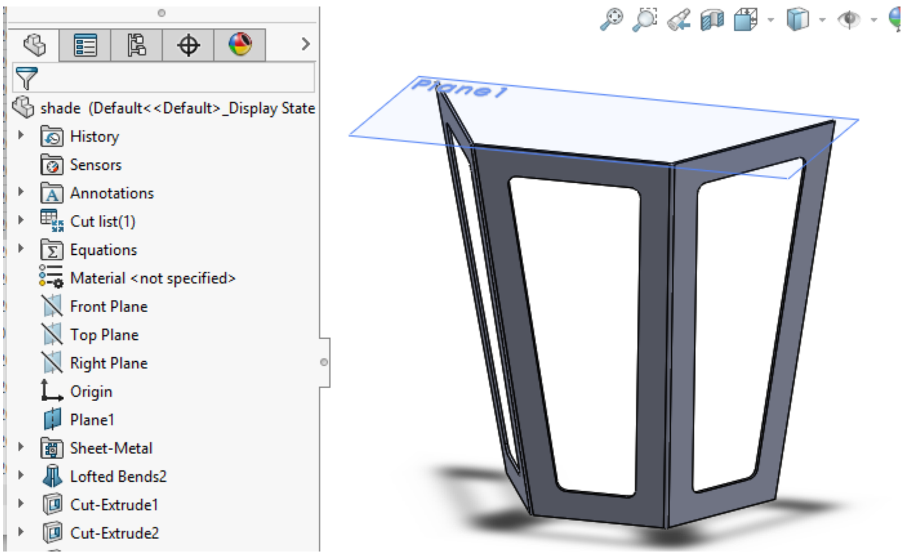
119	We have to change a number of dimensions in the model. 1. Zoom in at the top of the model. 2. Click at a random point. 3. Click on the size of 20mm and change it to 90mm.	
120	1. Zoom in at the bottom of the model. 2. Click on the model again. 3. Click on the size of 65mm and change this to 60mm.	
121	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 indi-cates the height. Change it to 200mm.	

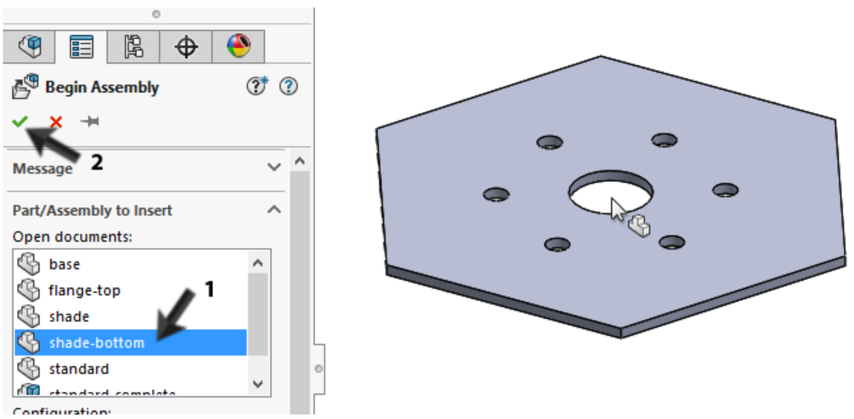
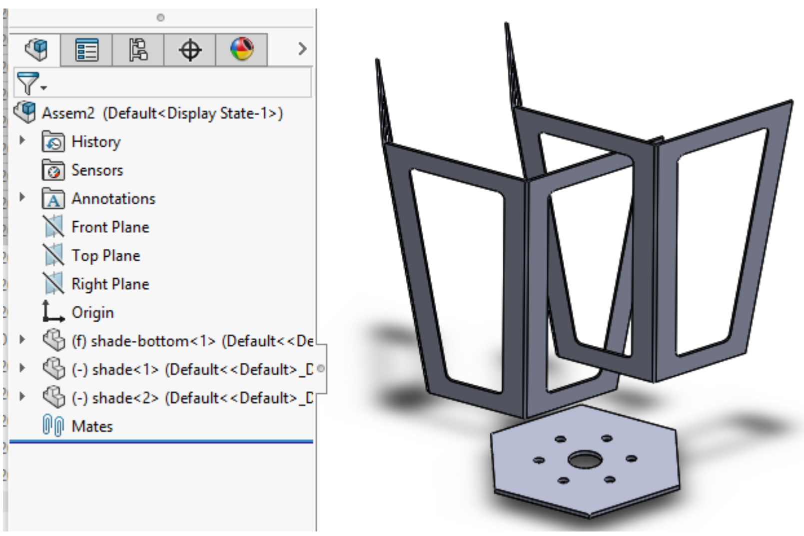
<table>
<tr>
<td>122</td>
<td>We will now make the openings in the sidewalls.

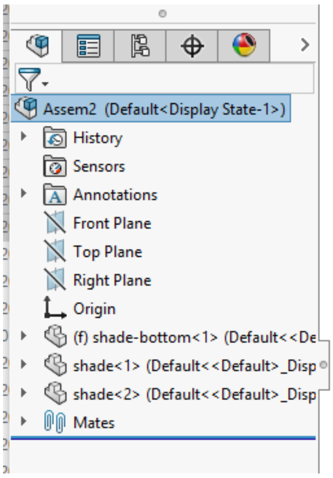
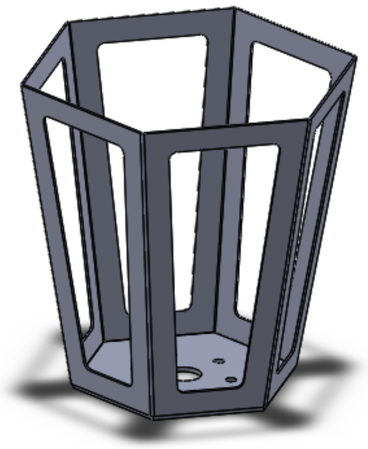
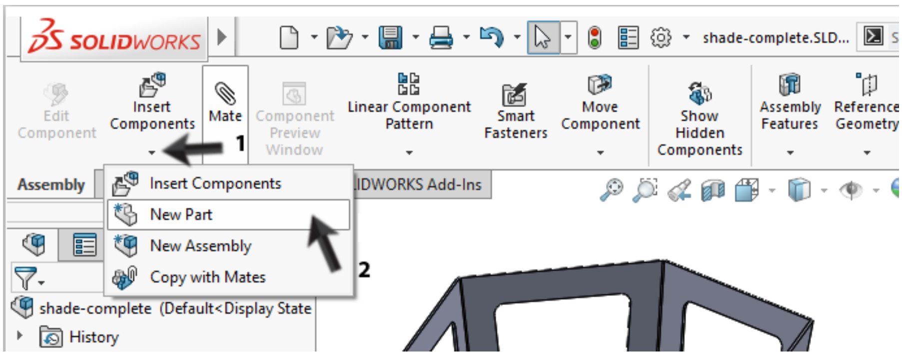
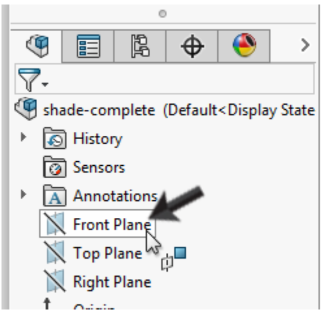
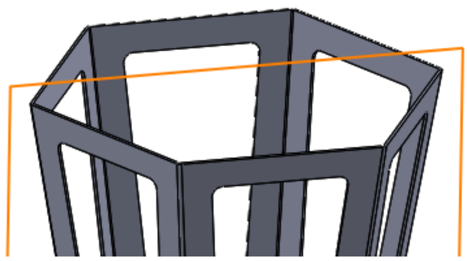
Select one of the side-walls.
Click on Sketch in the CommandManager.
Open the sketch.
</td>
<td>
</td>
</tr>
<tr>
<td>123</td>
<td>Click on Offset in the CommandManager.</td>
<td>
</td>
</tr>
<tr>
<td>124</td>
<td>
Set the distance for the offset to 15mm.
Click on the option Re-verse (when neces-sary), in order to show the yellow line at the inside of the plane.
Click OK.
</td>
<td>
</td>
</tr>
</table>

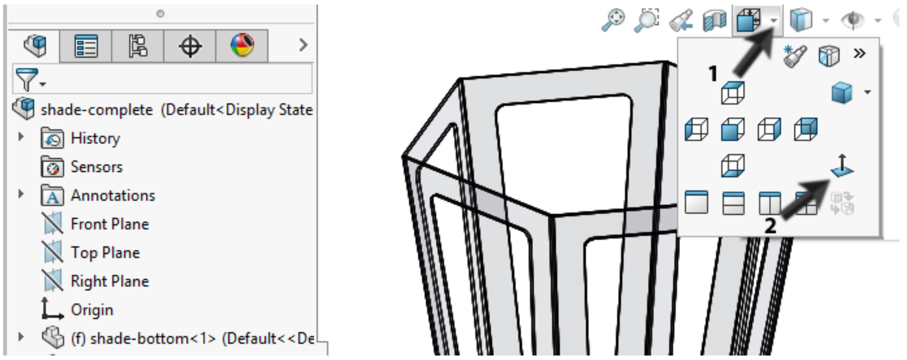
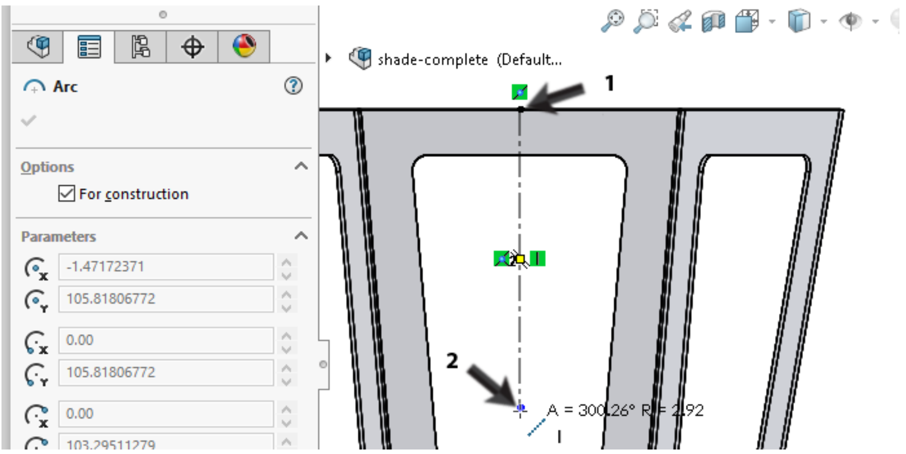
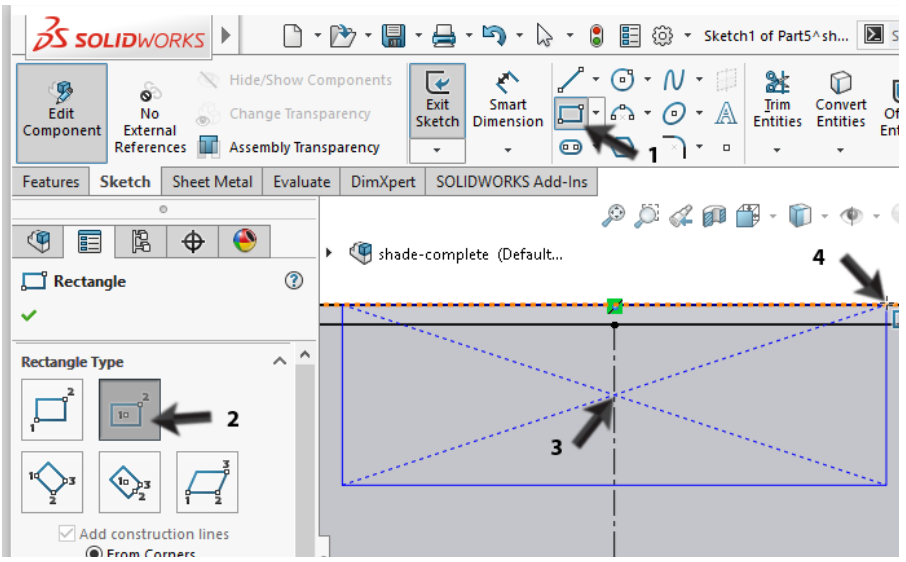
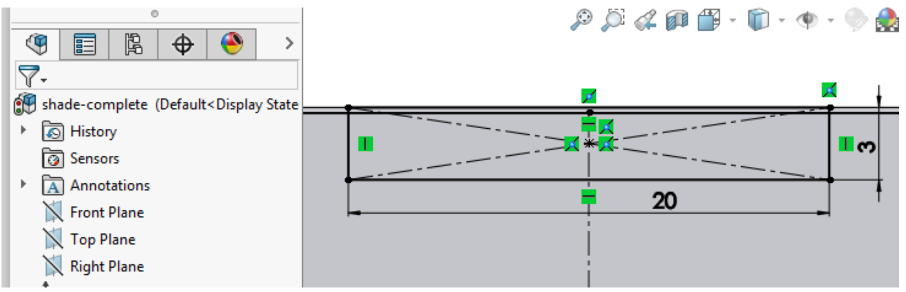
125	1. Click on Sketch Fillet in the CommandManager 2. Set the radius to 5mm in the PropertyManager 3-6.Click on the four corners of the sketch. 7. Click OK twice.	
126	Make a Cut-Extrude from this sketch. Set the depth to Through All. Repeat steps 122 to 125 in the two other planes of the model.	

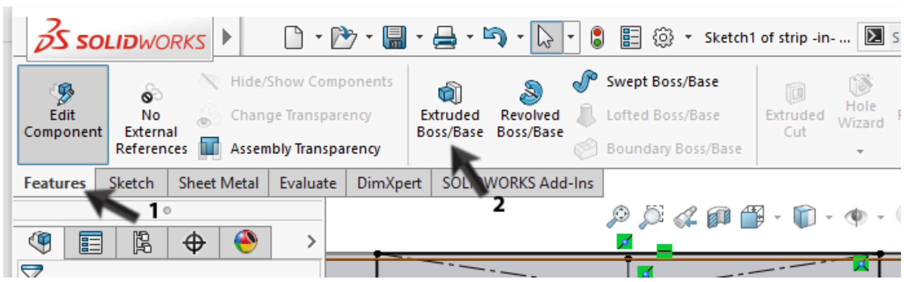
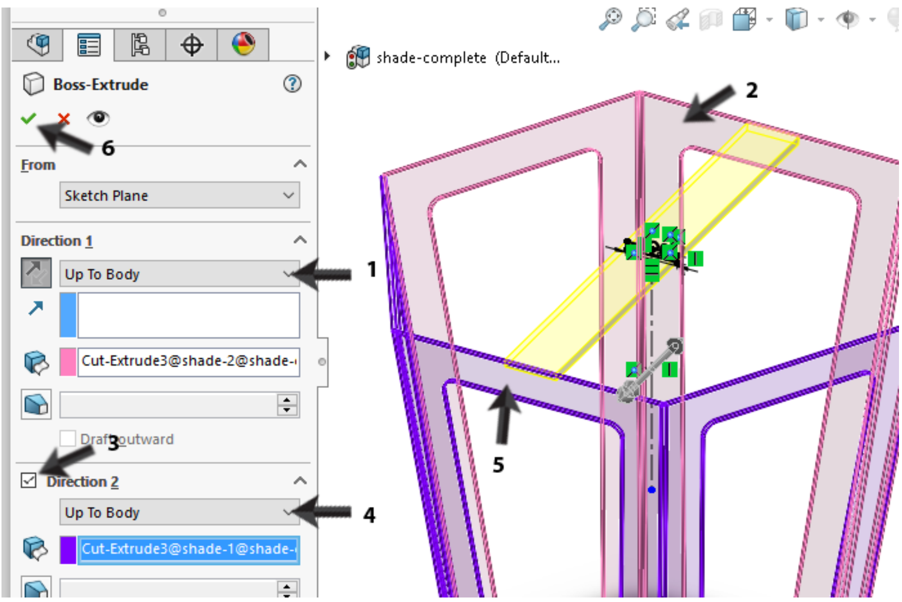
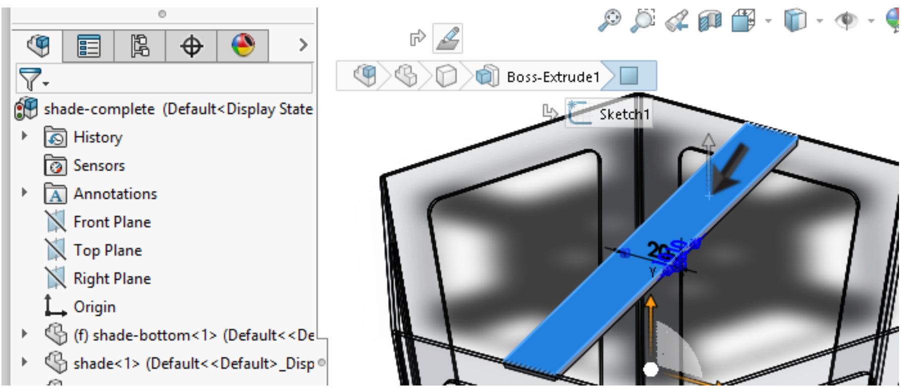
<table>
<tr>
<td>127</td>
<td>This part of the shade is ready now.

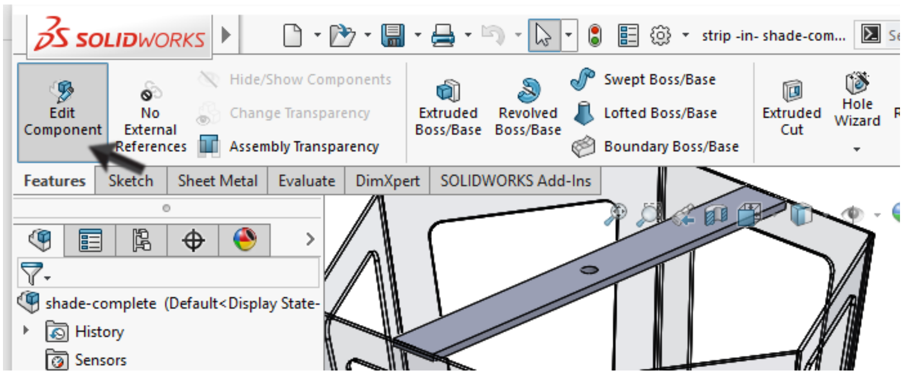
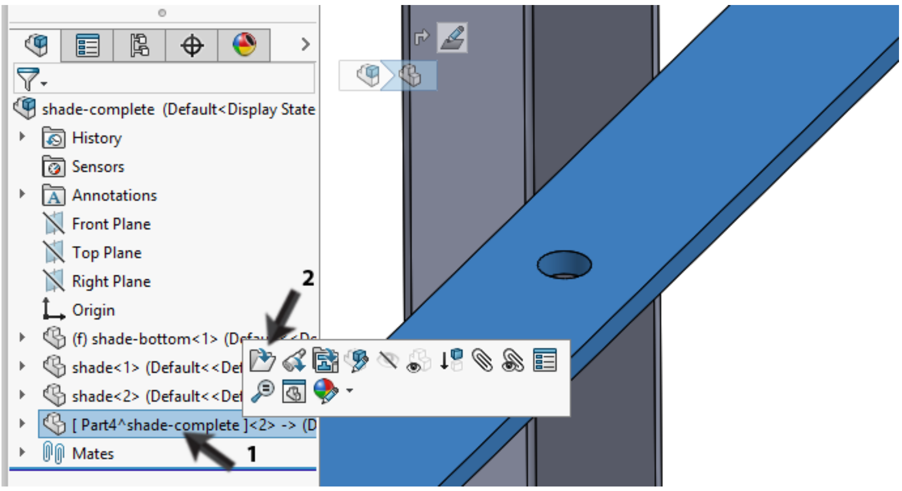
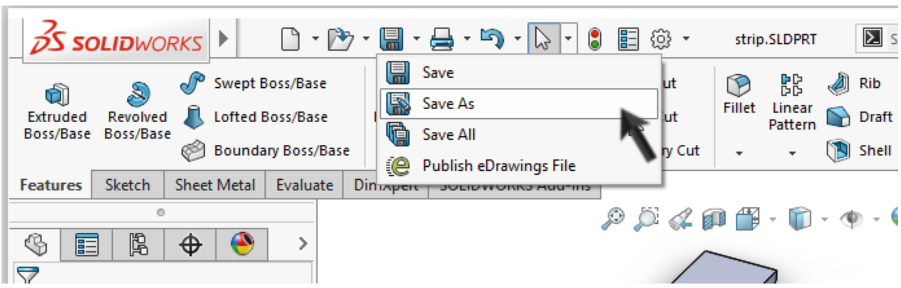
Save the file.</td>
<td>
</td>
</tr>
<tr>
<td></td>
<td>Werkplan</td>
<td>Although not all parts of the shade are ready yet, we will already make and assembly. We can create the rest of the parts in the assembly itself more easily.</td>
</tr>
<tr>
<td>128</td>
<td>Open a new assembly.

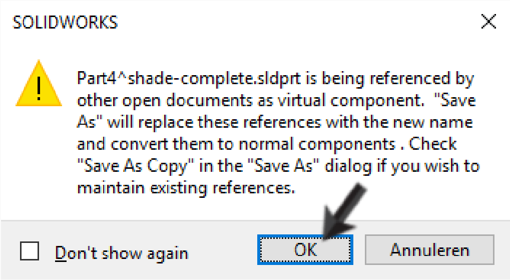
Add the flange-bottom file first. Do not put it at a random position, but by clicking OK, the part will be positioned directly at the origin.</td>
<td>
</td>
</tr>
<tr>
<td>129</td>
<td>Insert the part shade.SLDPRT twice. Place these in random positions.</td>
<td>
</td>
</tr>
</table>

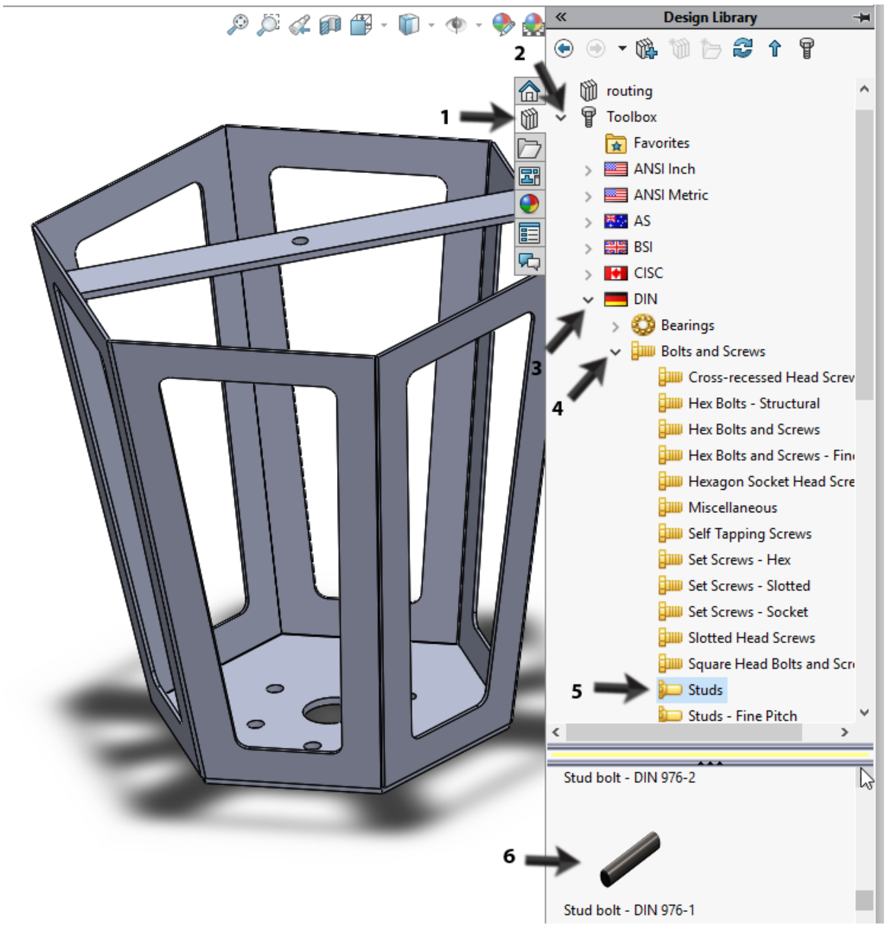
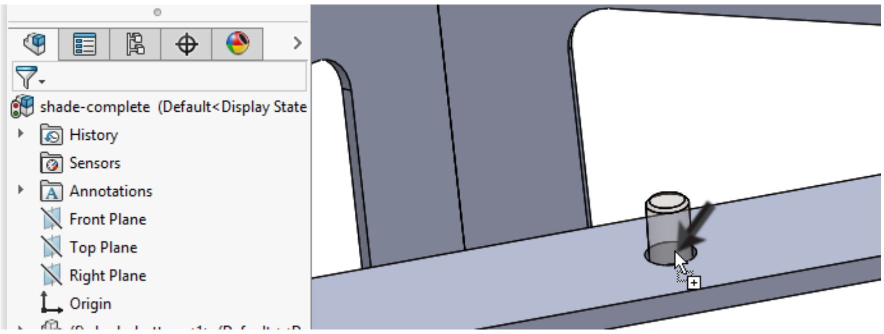
130	Add mates by using the Front- and Right-planes. You have done this before in steps 87 to 93.	 
131	Save the assembly as: shade-complete	
	Werkplan	At the top of the hood a metal strip has to be welded in. The problem is, that the size and the angled ends of the strip are very hard to calculate or 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 can not change the assembly, but you can use it to add relations.

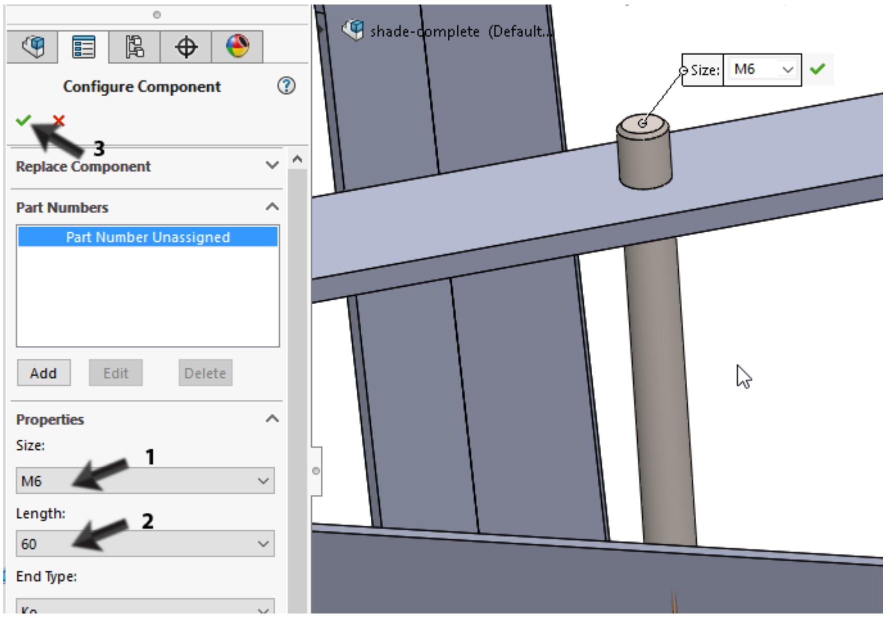
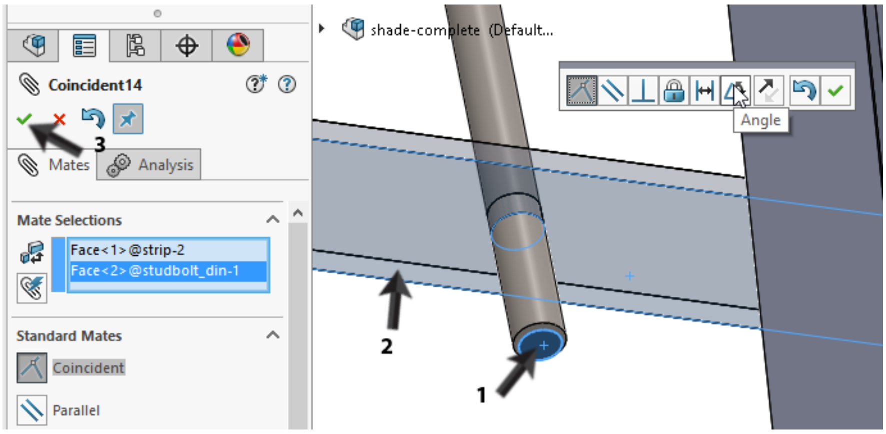
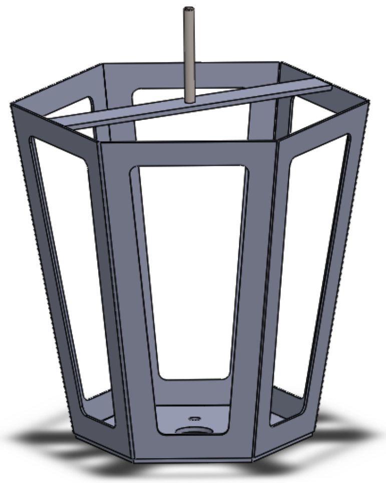
<table>
<tr><td>134</td><td>Rotate the model so you get a clear view at the sketch.
1. Open the rotate menu.
2. Click on Normal To.</td><td>
</td></tr>
<tr><td>135</td><td>Next draw a centerline.
1. Click on the middle of the upper edge to set the first point. Be sure to find the midpoint, check the symbols for this.
2. Click on a second point vertically underneath the first one.
3. Press <esc></td><td>
</td></tr>
<tr><td>136</td><td>Draw a rectangle. First zoom in so that you can see that there are two top edges. This is, because the faces are not vertical, but at an angle.
1. Click Rectangle in the CommandManager
2. Select the option to draw the rectangle from the midpoint.
3. Place the midpoint on the centerline.
4. Place the corner of the rectangle on the upper edge.</td><td>
</td></tr>
<tr><td>137</td><td>Add the dimensions by using Smart Dimensions as shown in the illustration.</td><td>
</td></tr>
</table>

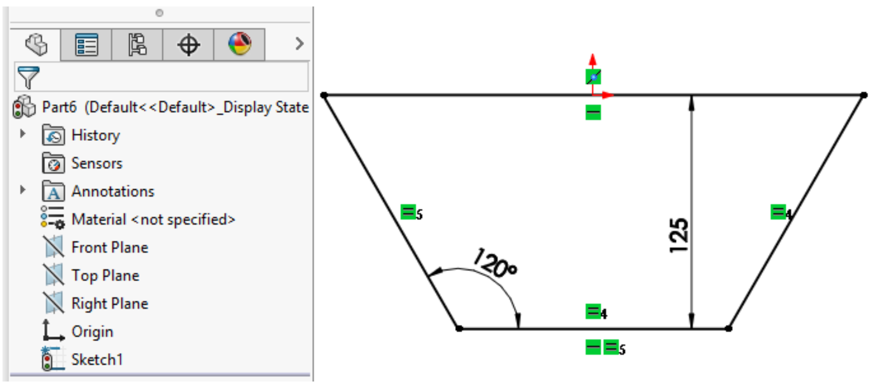
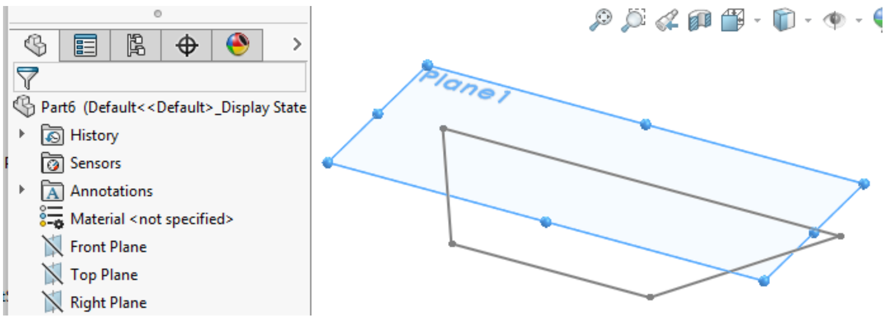
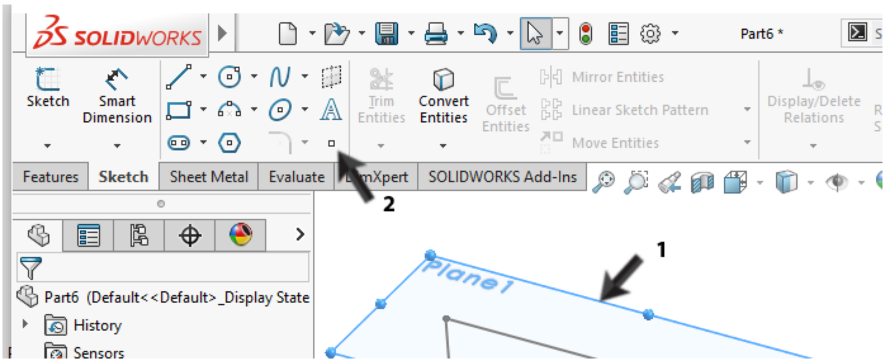
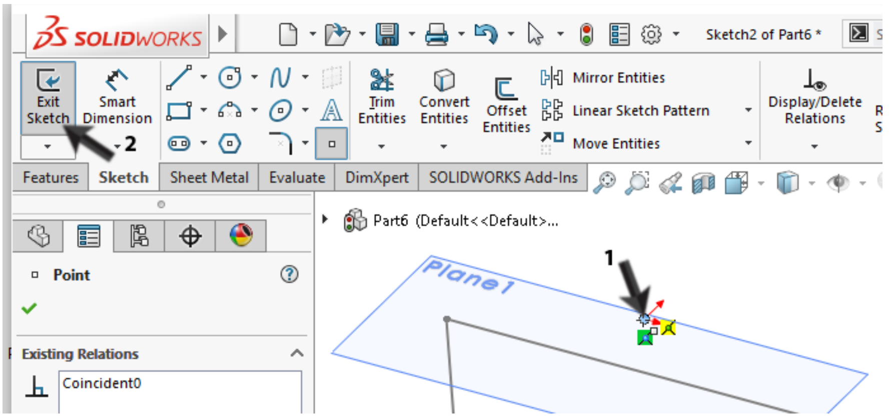
138	Click on Features in the FeatureManager. Click on Extruded Boss.	
139	To make the extrusion set the following features: 1. Select Up to Body for Direction1 2. Click on one side of the shade. 3. Check Direction2 in the PropertyManager, this is to expand the sketch in two directions. 4. Select Up to Body for Direction2 also. 5. Click on the other side of the shade. 6. When it looks OK to you, Click OK.	
140	1. Select the upper side of the strip 2. Open the extended menu from the Com-mandManager when needed. 3. Click on Circle.	
141	Draw a circle, with its mid-point at the origin. Set the size of the circle with Smart Dimensions. The diameter has to be Ø6. Make a Cut-Extrude from this circle and set the depth to Through All.	

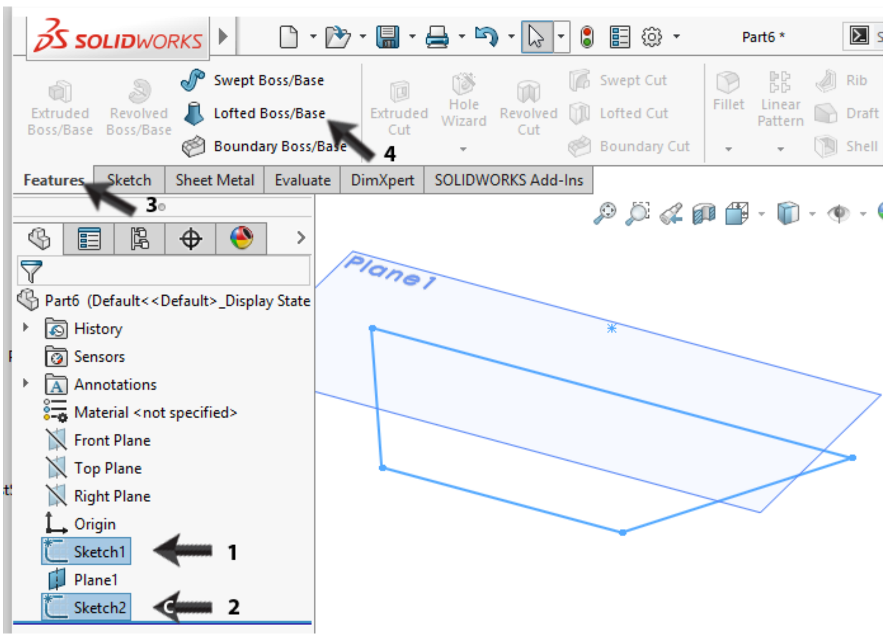
142	Click on Edit Component in the CommandManager to switch off this function. You are no longer working in-context. The assembly turns back to ‘normal’ again (it is no longer transparent).	
	Tip!	The strip is ready now and is directly fixed at the right place. You must have noticed that modeling In-context works fast and is very easy to do. There is another important advantage: when you change items later, for instance 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.
143	First open the strip. 1. Select the part in the FeatureManager. 2. In the pop-up menu select Open part.	
144	Click on Save As… and name the part: Strip.sldprt	

145	If the message in the illustration appears, click OK.	

	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.
146	1. Open the Design Library 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	
147	Release the stud bolt in the hole in the strip	

148	1. Set the diameter to M6 in the PropertyManager 2. Set the length to 60mm 3. Click OK. 4. Push <Esc> to end this command.	
149	Next add a mate: it has to be between the bottom of the stud bolt and the bottom of the strip.	
150	The assembly of the shade is ready now. Save the assembly.	
	Work plan	We need one more part: the roof of the shade. Because this is a pointed sheet metal part, we cannot create it in the same way. We can however use a third method to create sheet-metal by using a solid part.

151	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 centerline now, but a normal edge. Close the sketch by clicking on Exit Sketch in the CommandManager.	
152	Add a construction plane at a height of 40mm above the Top-plane. You have done this before in steps 39 to 41.	
153	Make a sketch on Plane1. 1. Select Plane1 2. Click on Point	
154	1. Place one point, directly in the origin of the sketch. 2. Click on Exit Sketch in the CommandManager	

<table>
<tr>
<td>155</td>
<td>
1. Select the first sketch in the FeatureManager.
2. Hold the <ctrl>-key and select the second sketch.
3. Click on Features in the CommandManager
4. Click on Lofted Boss/Base
</td>
<td>


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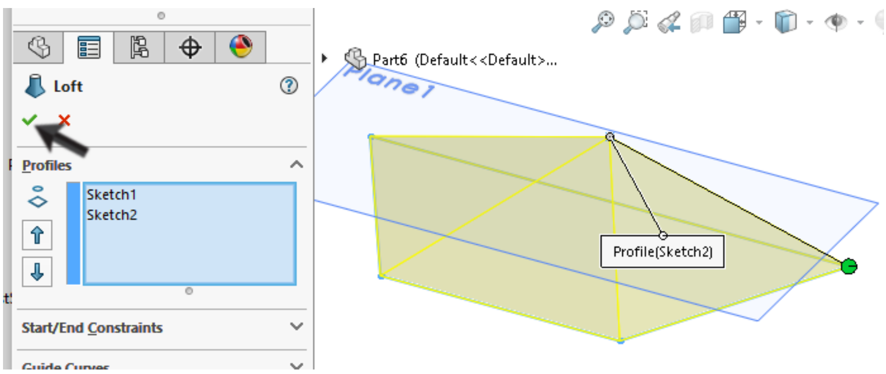
<td>156</td>
<td>Click OK in the Property-Manager.</td>
<td>


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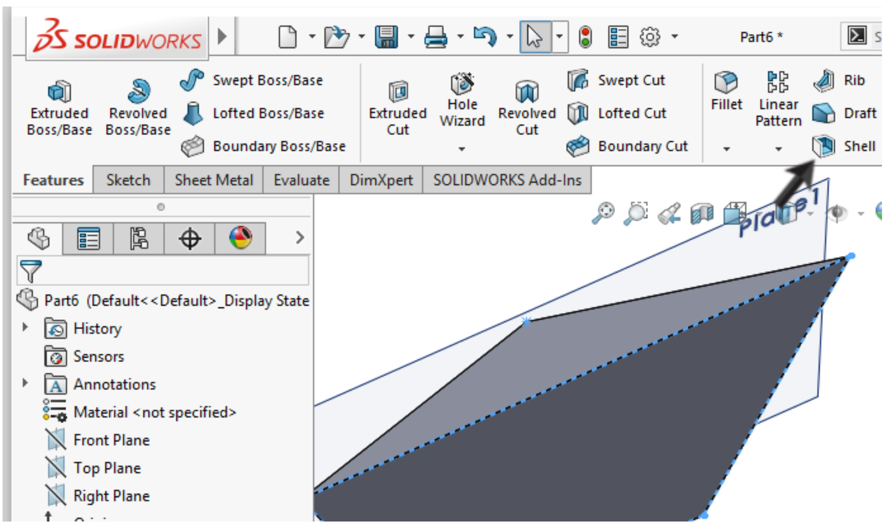
<tr>
<td>157</td>
<td>
We have a solid part now. We will make this hollow.

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

Click on Shell in the Com-mandManager.
</td>
<td>


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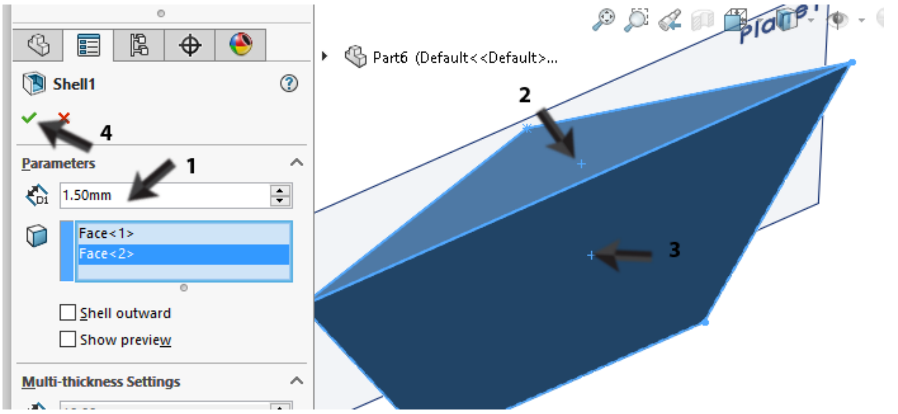
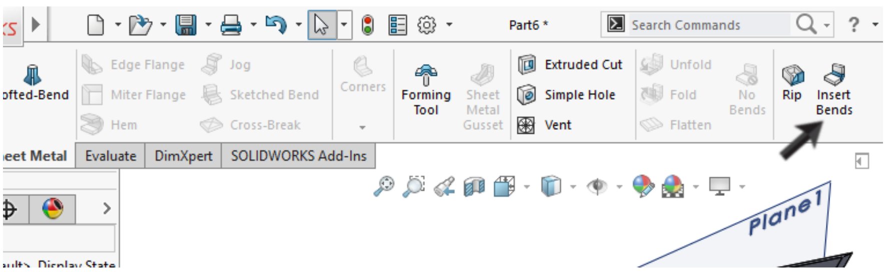
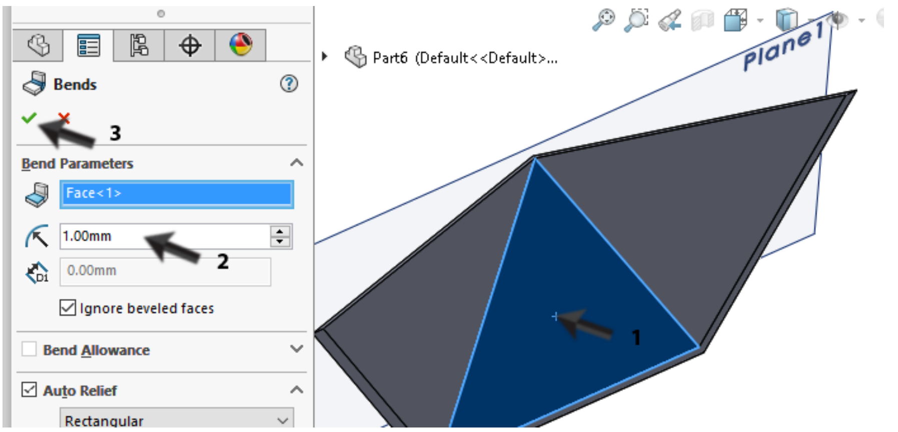
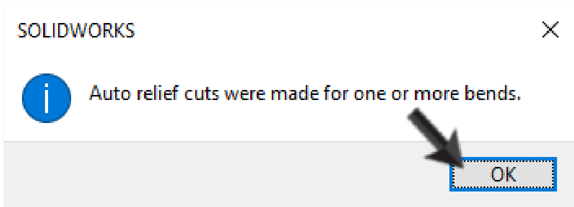
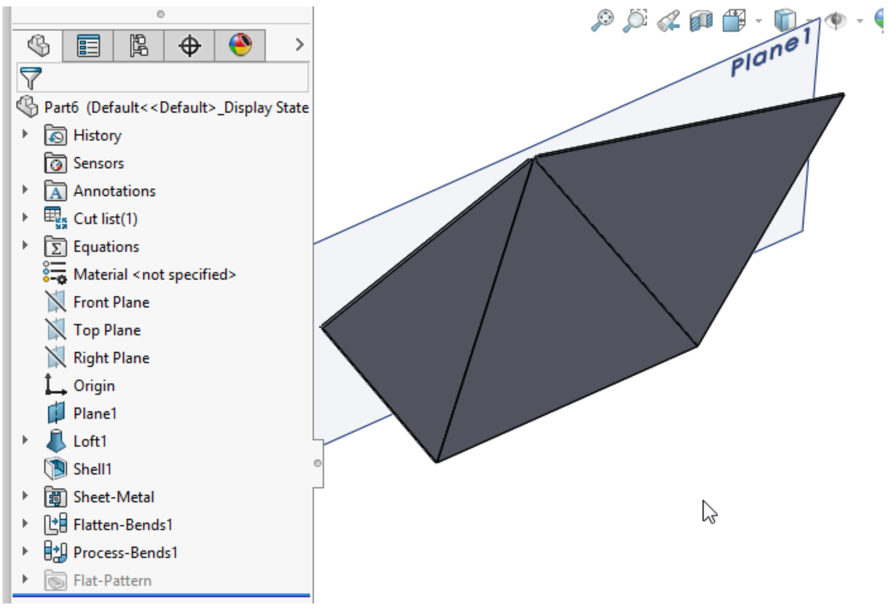
158	1. Set the thickness to 1.5mm. 2. Select the back plane 3. Select the bottom plane 4. Click OK.	
159	We will change this part in-to a Sheet Metal part. 1. Click on Sheet Metal in the CommandManager 2. Click on Insert Bends	
160	1. Click on the middle plane of the model. When making a flat drawing this plane will hold its position. 2. Set the bending radius to 1mm. 3. Click OK.	
161	A message appears, saying that SOLIDWORKS has modified the geometry to be able to bend the materi-al. Click OK.	

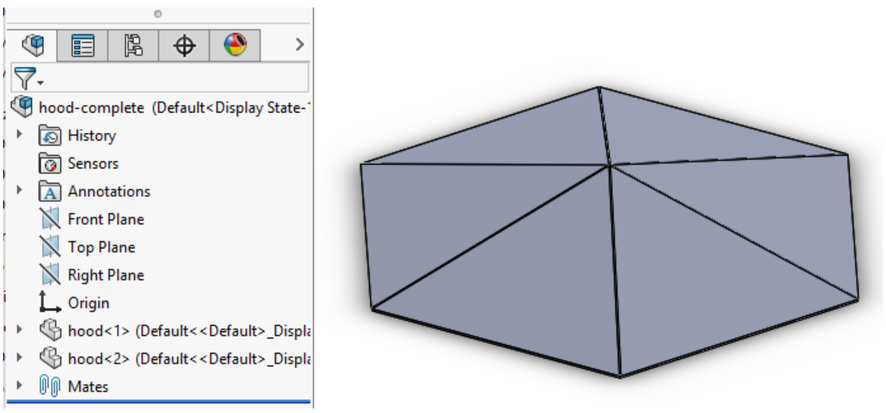
<table>
<tr>
<td>162</td>
<td>A few features have been added to the FeatureManager now, which indicates clearly that you are dealing with a sheet metal part.

One half of the roof is ready now.

Save this as: hood.sldprt</td>
<td>
</td>
</tr>
<tr>
<td>163</td>
<td>Next we will make an assembly 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</td>
<td>
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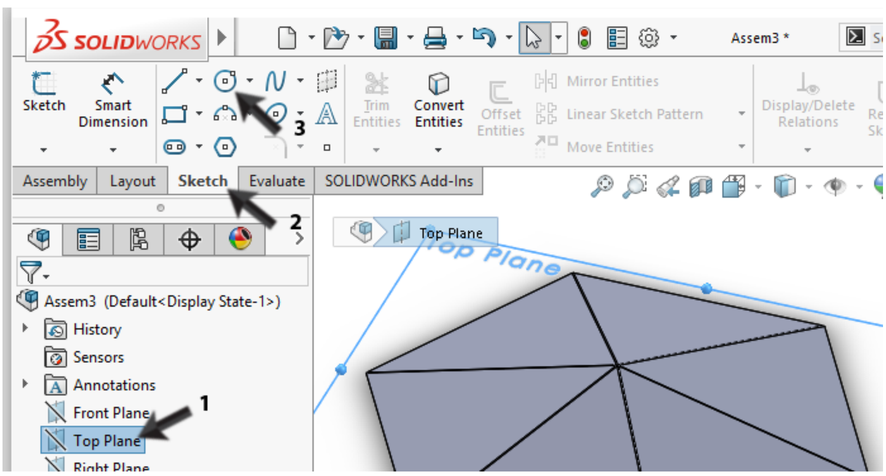
<tr>
<td>164</td>
<td>We have to make a mounting hole in the roof to fix it.</td>
<td>
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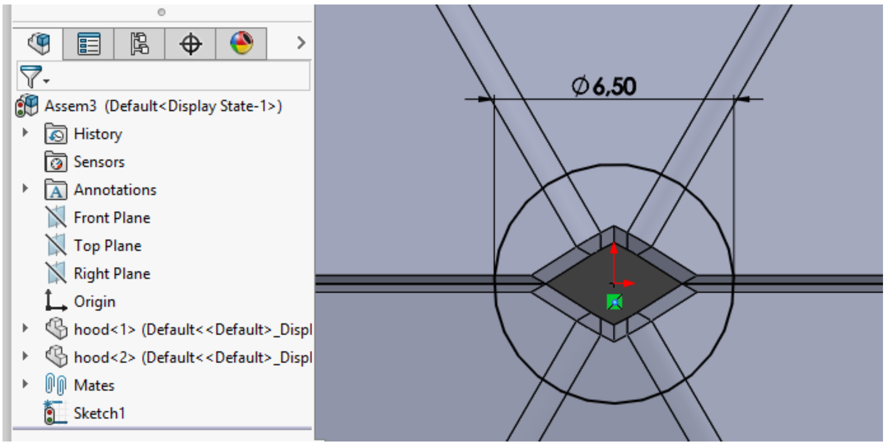
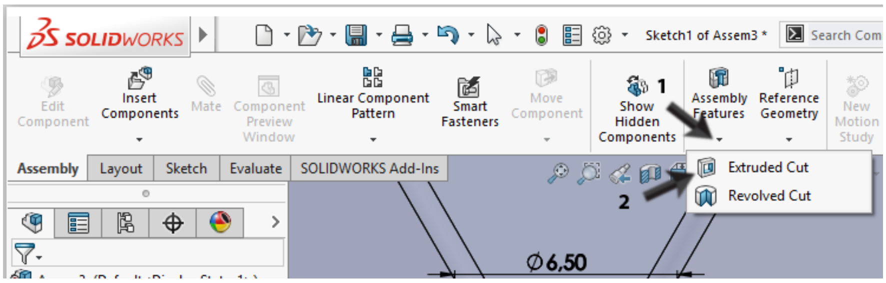
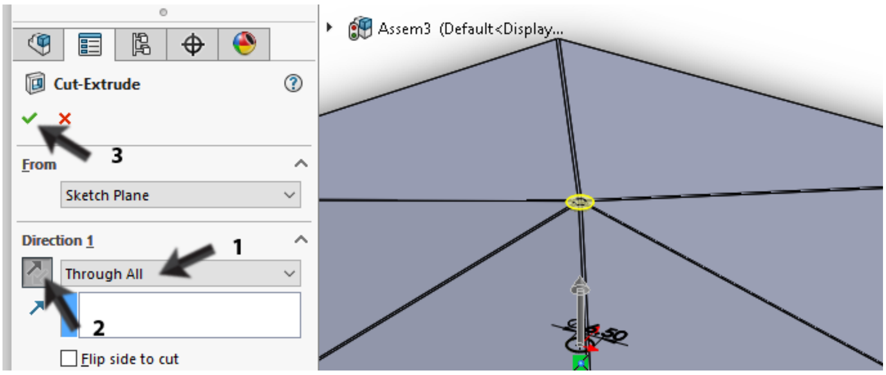
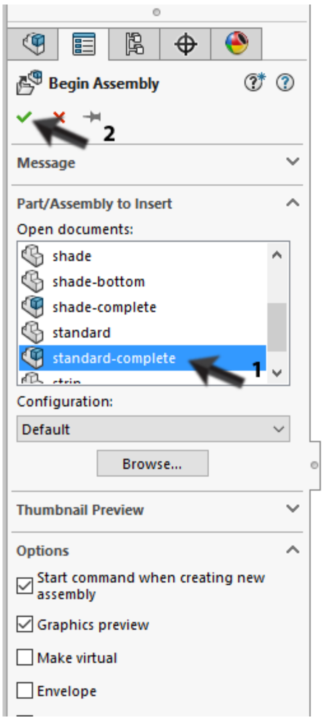
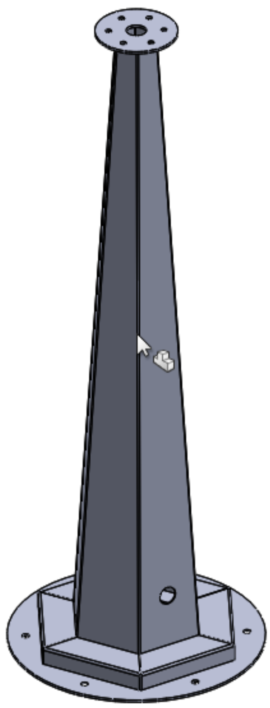
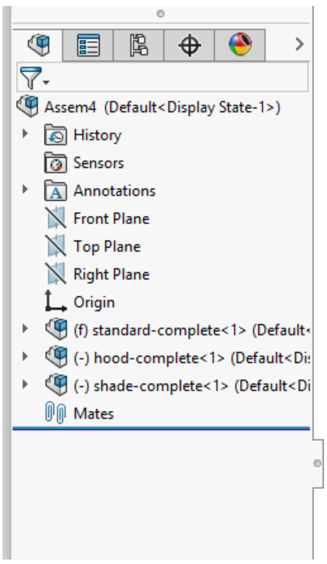
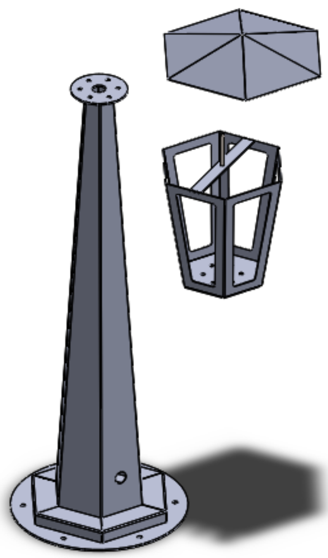
165	Draw a circle with the center on the origin. Add a dimension for the circle with Smart Dimensions Change it to 6.5mm.	
166	1. Click on Assembly Features in the CommandManager 2. Click on Extruded Cut	
167	1. Set the depth of the hole to Through All in the PropertyManager. 2. Change the direction of the hole when needed, in order to lead it through the model 3. Click 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 a so called assembly feature. We did nothing else than 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 in real live.
168	The hood is ready now. Save it as hood-complete.sldasm	
169	All parts are ready now and we have created three sub-assemblies:	

	- standard-complete - shade-complete - hood-complete These three can be assembled to get the end product Open a new assembly.	
170	1. Select the file Standard-complete in the PropertyManager 2. Click OK.	 
171	Add the two other assemblies now. Put them at a random position.	 

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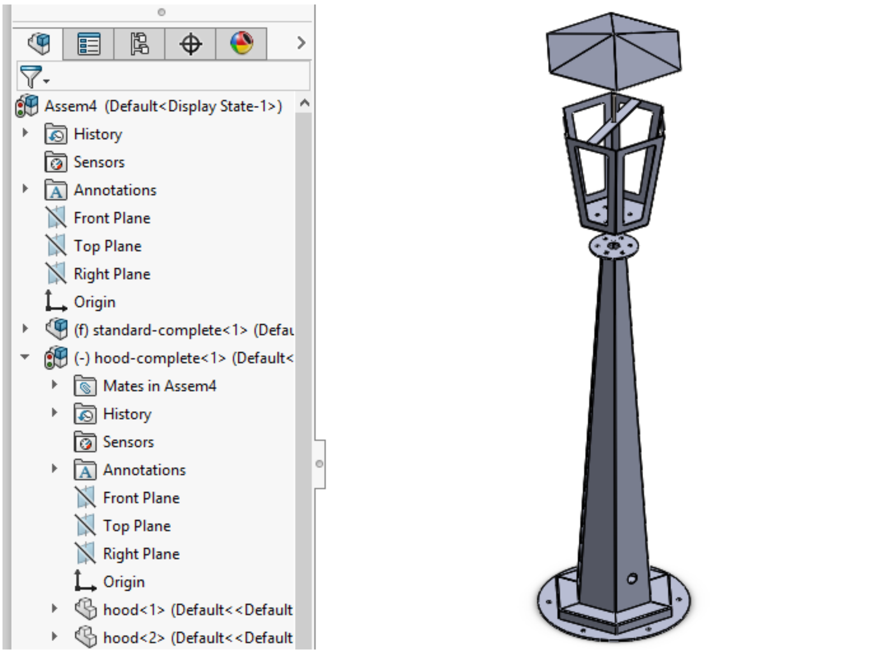
<tr>
<td>172</td>
<td>Add mates now.

Again, use the Front- and Right-planes to put the parts above each other. You have done this before in steps 89 to 93.</td>
<td>
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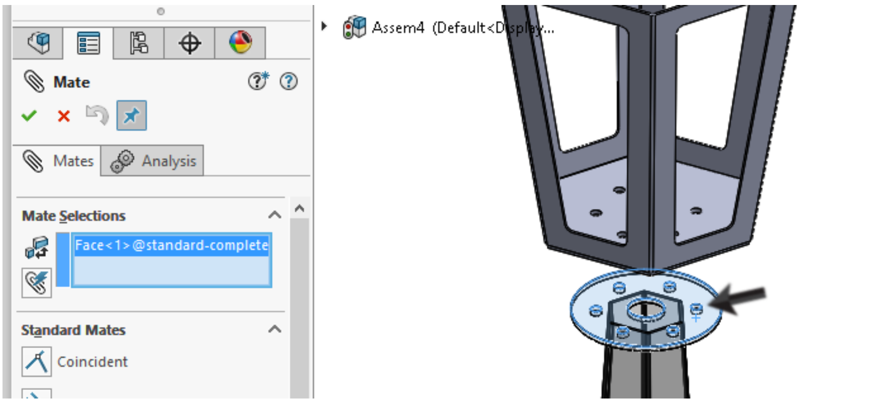
<td>173</td>
<td>To put the shade onto the standard, first select the top plane of the standard.</td>
<td>
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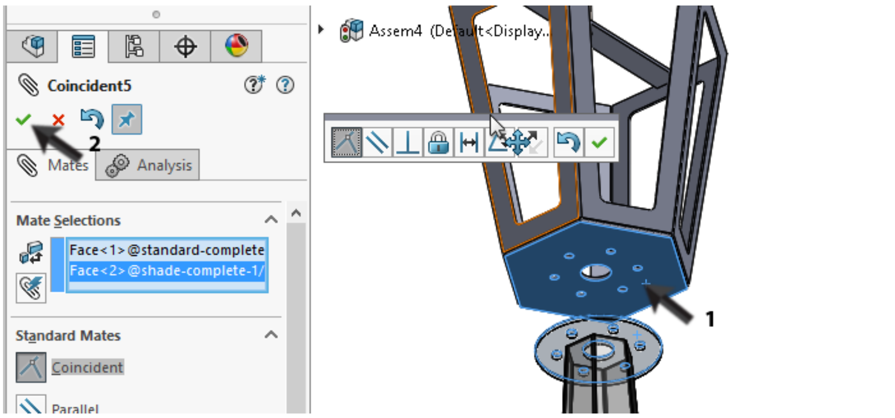
<tr>
<td>174</td>
<td>Rotate the model and se-lect the bottom plane of the shade.</td>
<td>
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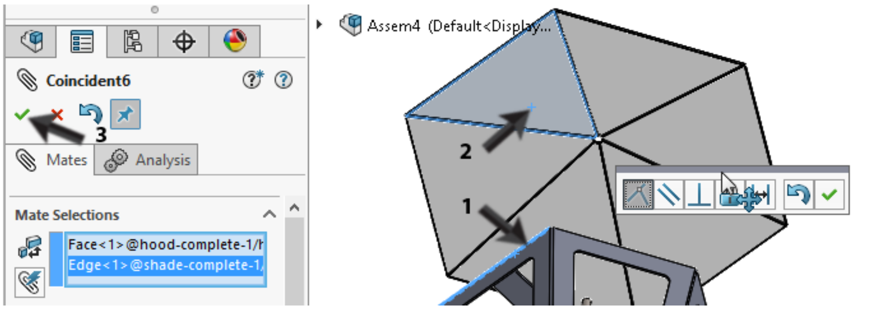
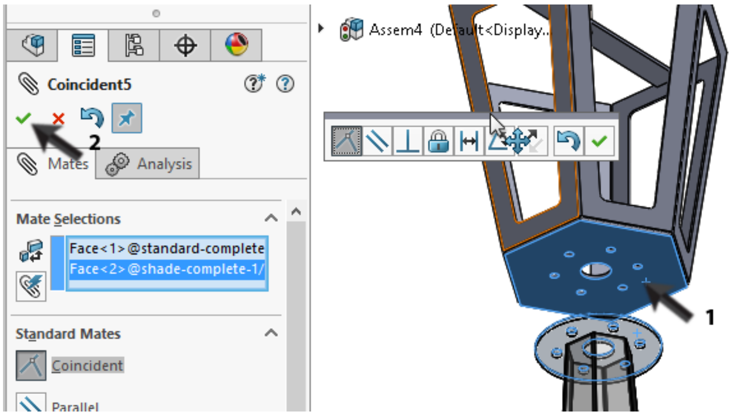
175	We will put the roof onto the shade. 1. Select an edge at the bottom side of the roof. (be sure to select the **outside of the wall**) 2. Select the correspond-ing bottom plane of the roof. 3. Click OK.	
174	The garden light is ready now. Save it as: garden-light.sldasm	
	And now …	There are a couple of features which we have not done in this tutorial. You could try this yourself: 1. We did not weld the subassemblies. We have done 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 (Candle light). 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 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 fea-tures you have learned in this tutorial?	In this tutorial you have seen a lot of items: • You have seen three ways to create a part from sheet metal: 1. Starting with a base flange and add planes to it. We did this while creating the base of the standard 2. Starting from a Loft: use two sketches, and shape the metal sheet in between them. This is what we have done while creating 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 solve errors • You have created a part 'in-context' in an assembly. • Finally you have used an assembly feature.

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