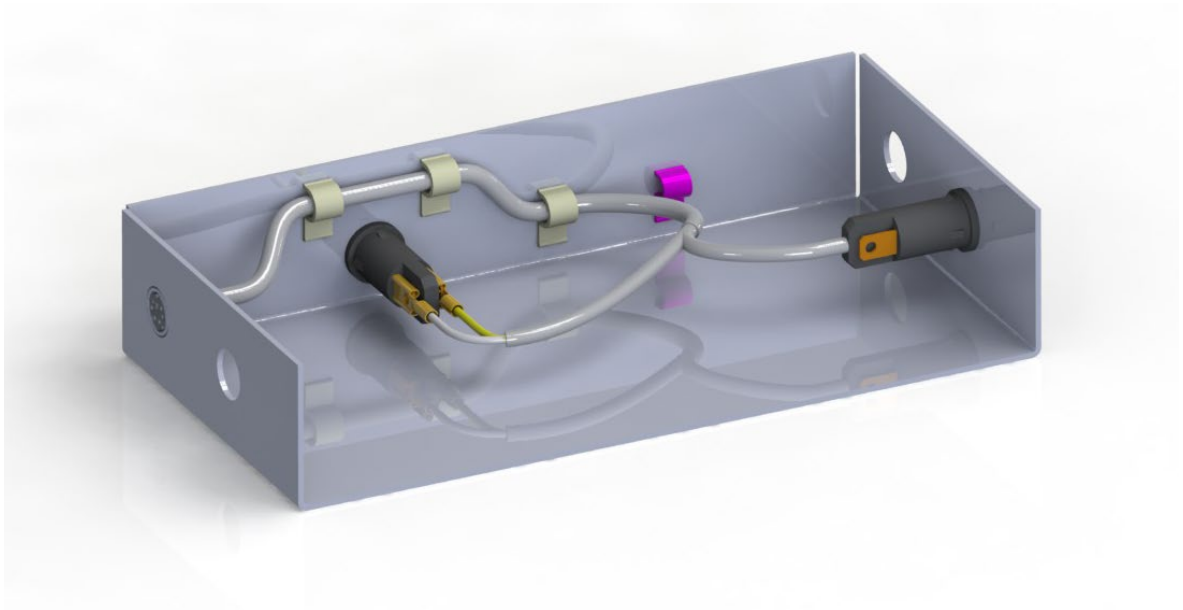


SOLIDWORKS® tutorial 13

Routing



Preparatory and Advanced Vocational Training



To be used with SOLIDWORKS® Educational Release 2018-2019

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Initiative: Kees Kloosterboer (SOLIDWORKS Benelux)

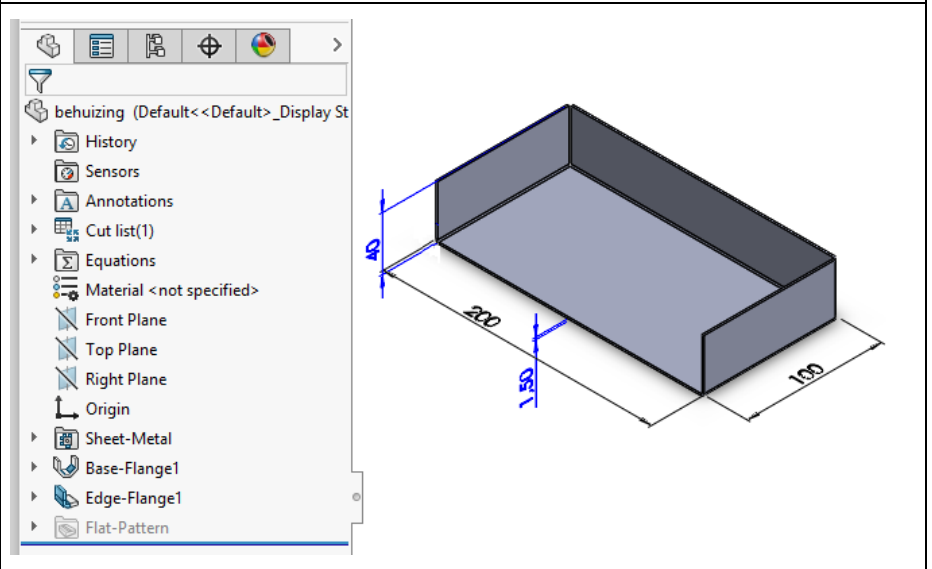
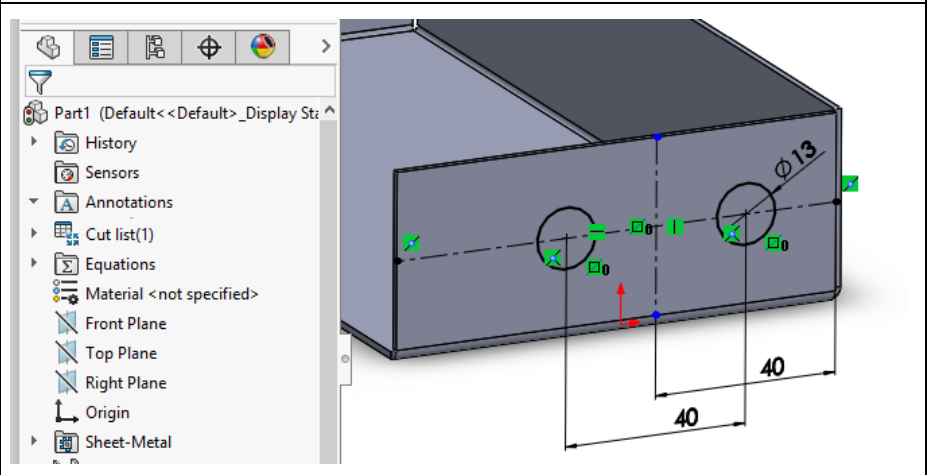
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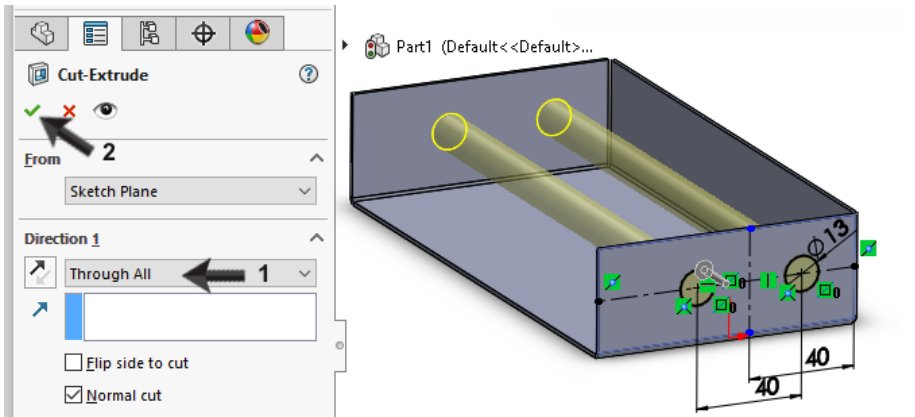
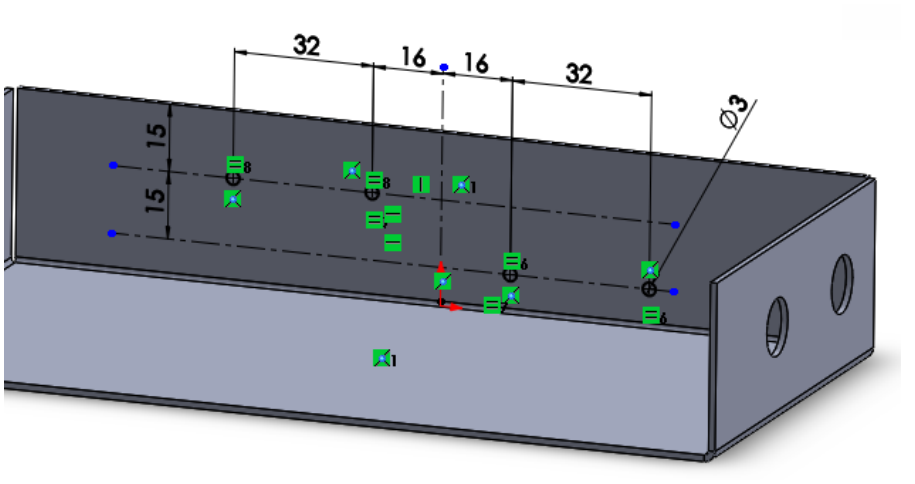
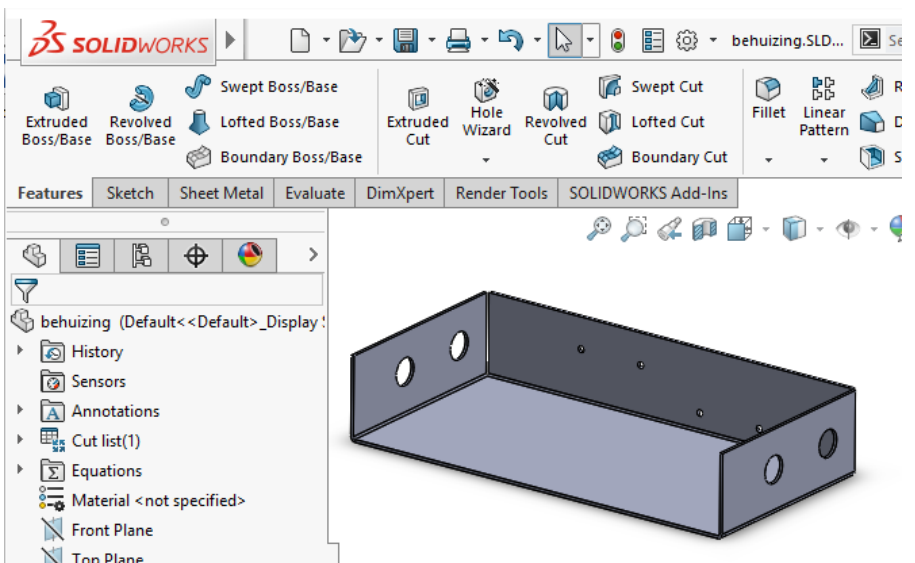
Realisation: Arnoud Breedveld (PAZ Computerworks)

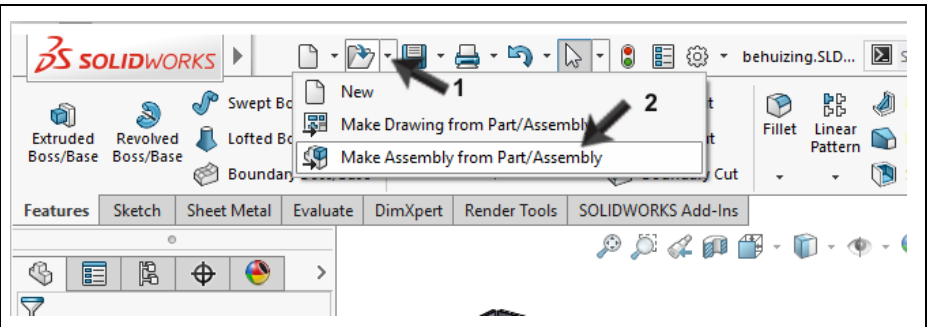
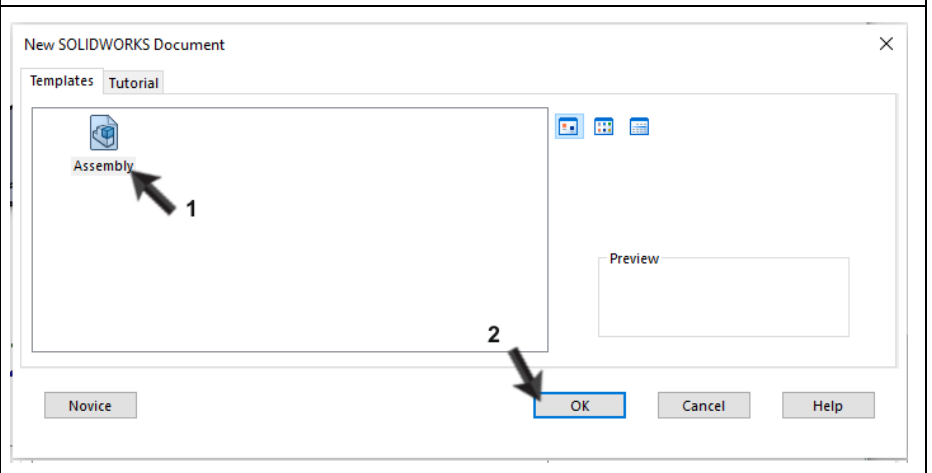
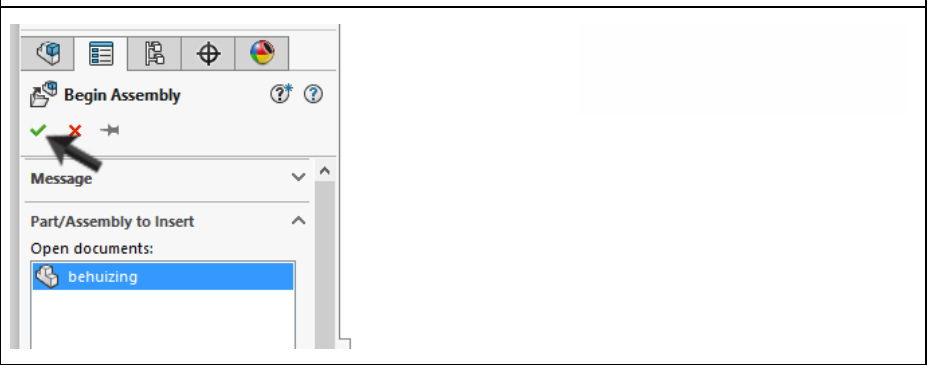
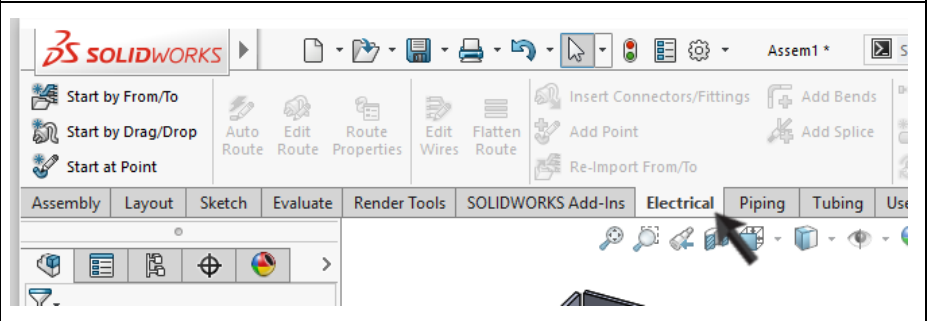
Routing

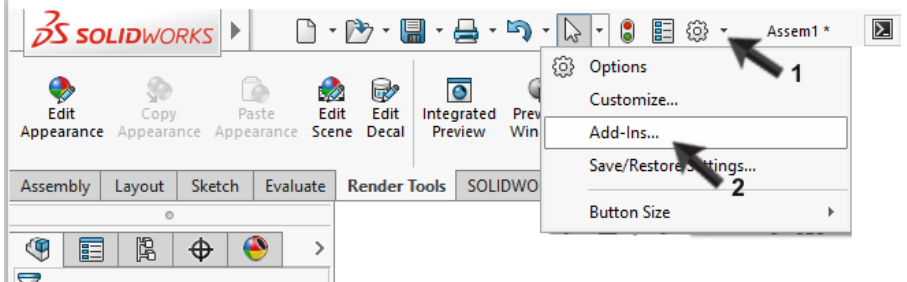
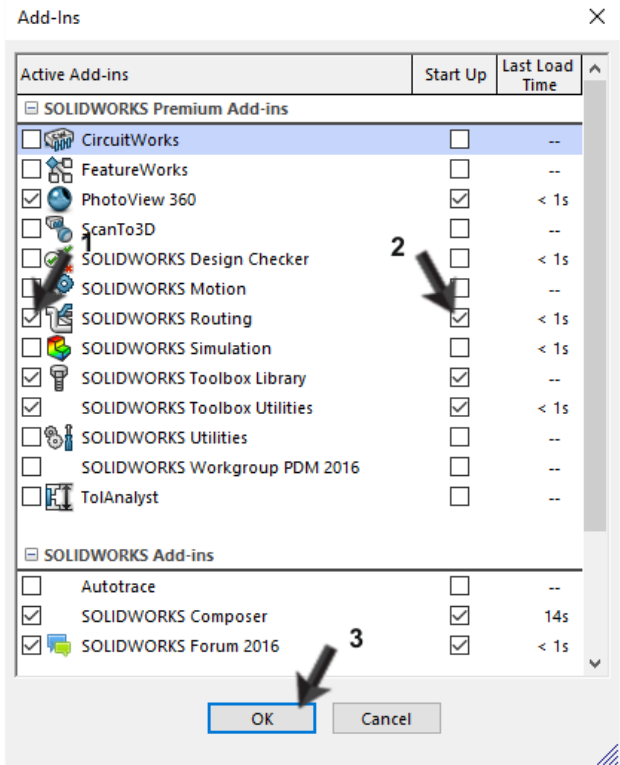
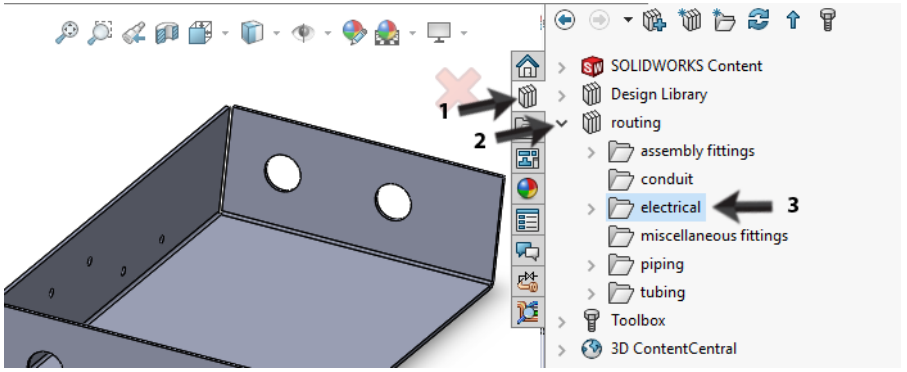
Routing is the section of SOLIDWORKS, with which you can add tubes, wires and components to your product. Routing is not a part of the standard version of SOLIDWORKS. If you are using the Student Design Kit of SOLIDWORKS, you cannot do this tutorial. In the Student Edition, Routing is available as an add-in.

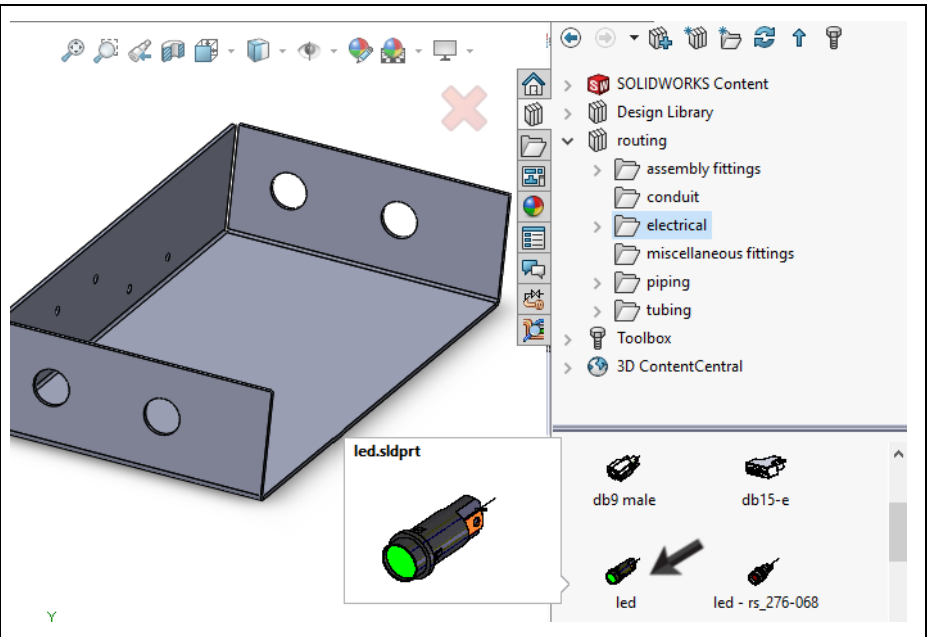
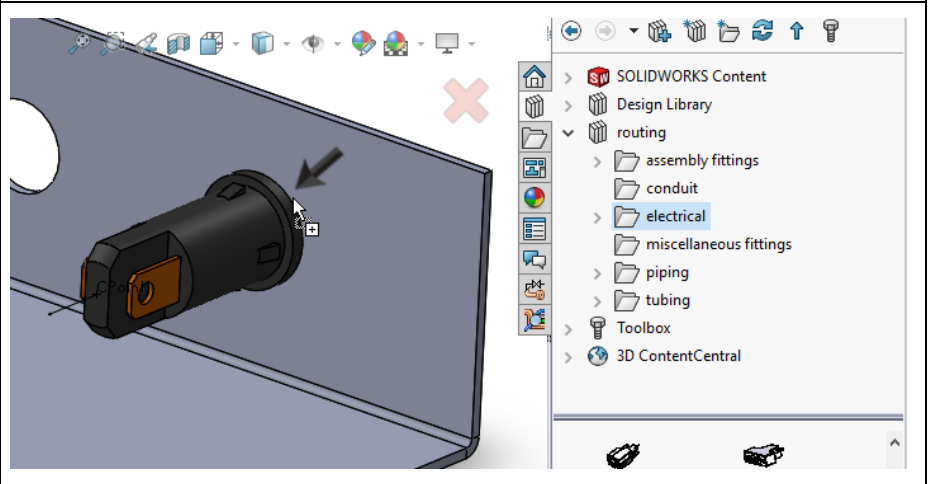
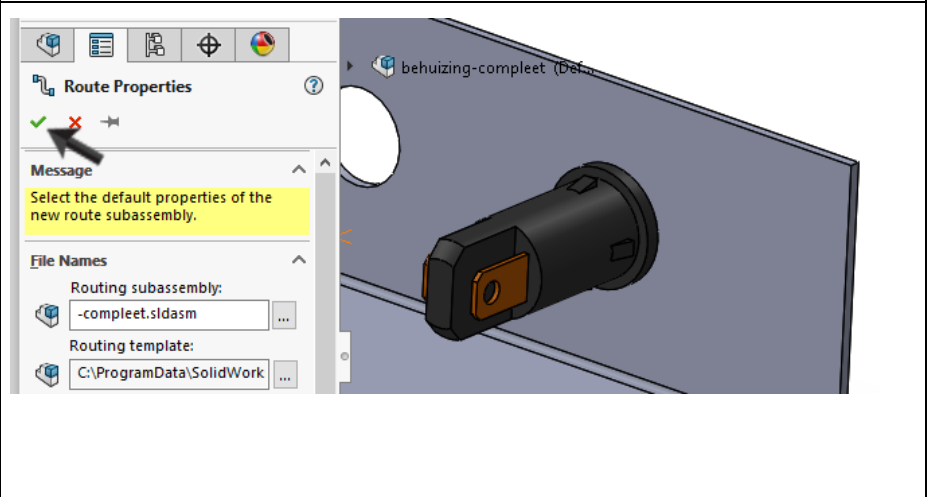
Routing consists of three parts, that are completely apart from each other: Electrical, Piping and Tubing. In this tutorial you will see the basics of 'Electrical': placing electrical components in your product and wiring it. After that you will see how you can create your own components.

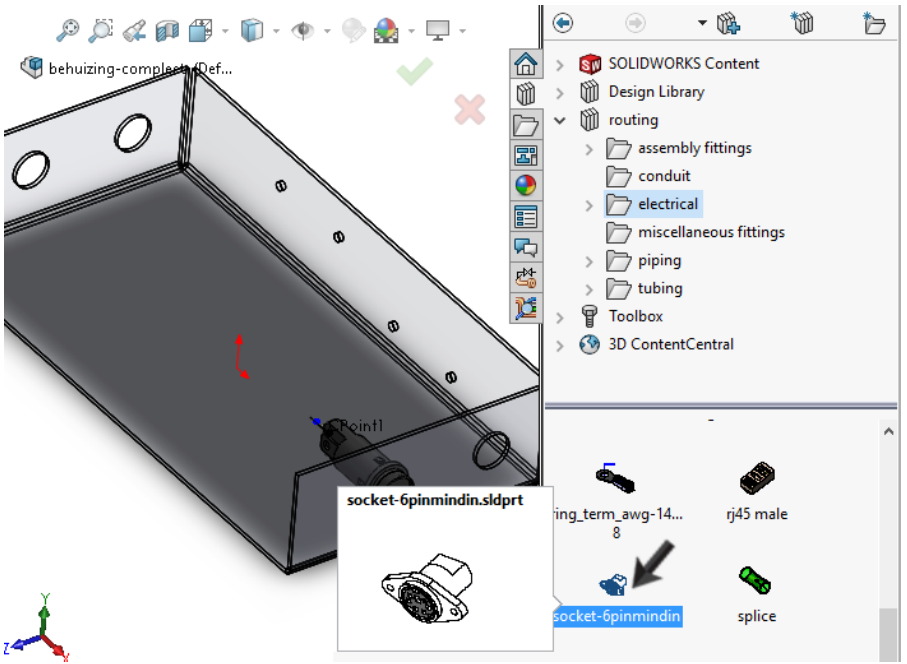
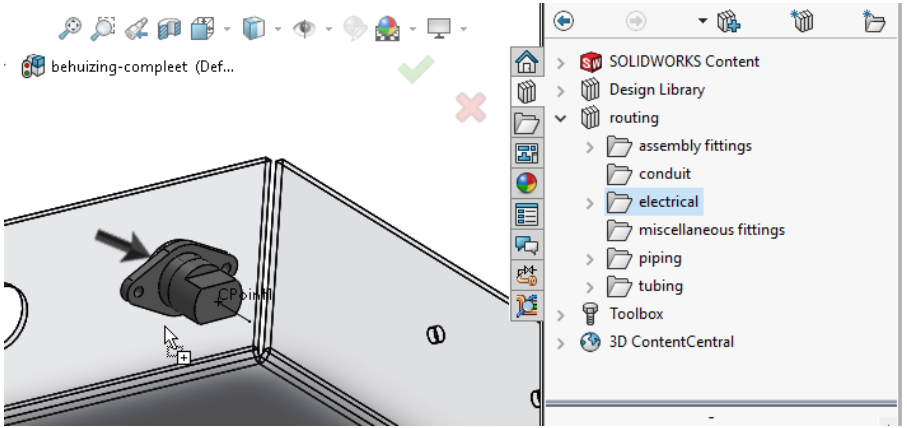
<p>1</p>	<p>Start SOLIDWORKS and open a new part.</p>	
<p>2</p>	<p>Use sheet metal to create the shape you see in the illustration. Have you got trouble doing so? Check tutorial 4, step 1 to 10.</p> <p>The dimensions of this part are 200 x 100 x 40 mm. Thickness is 1.5 mm.</p>	
<p>3</p>	<p>On the right side plane, create the sketch you see in the illustration.</p>	

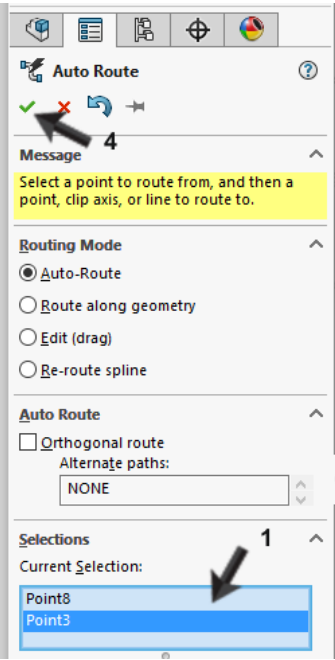
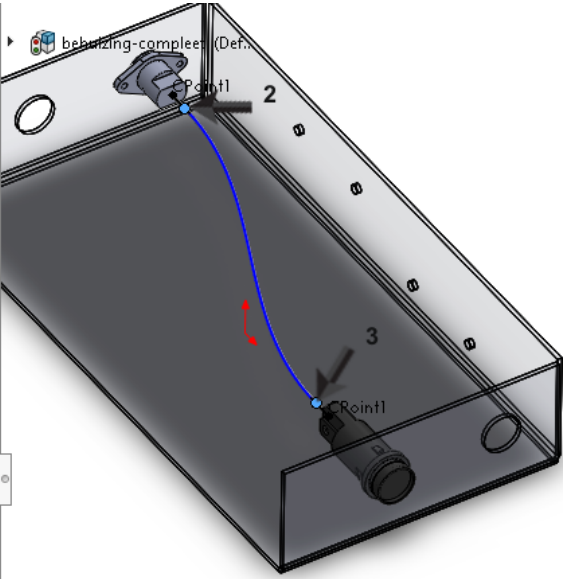
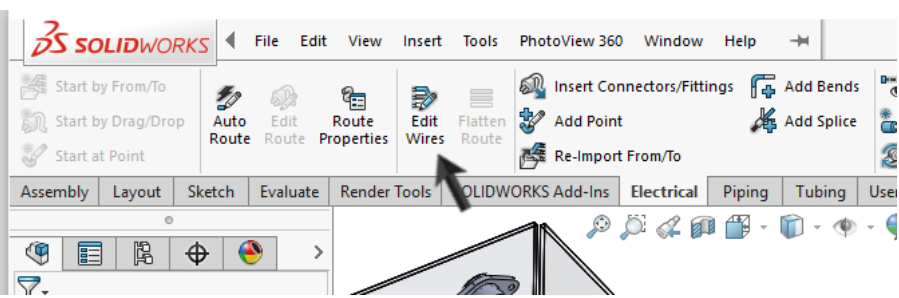

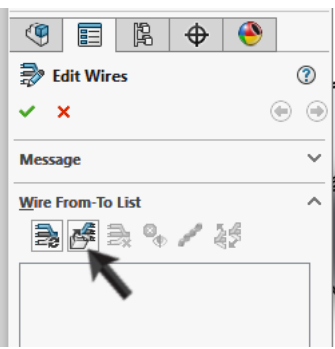
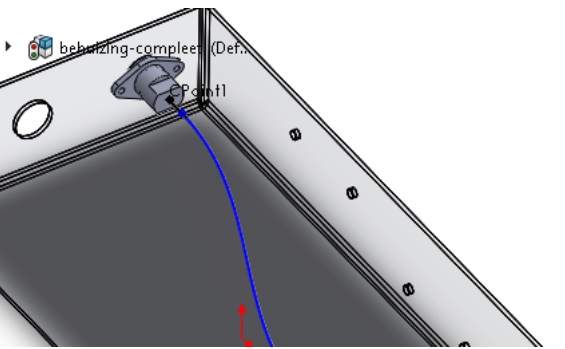
<p>4</p>	<p>Use the sketch to create an Extruded Cut through the whole part ('Through All')</p>	
<p>5</p>	<p>On the plane in the back, create the sketch you see here. Make an Extruded Cut out of it, again Through All.</p>	
<p>6</p>	<p>Save the part as housing.sldprt</p>	

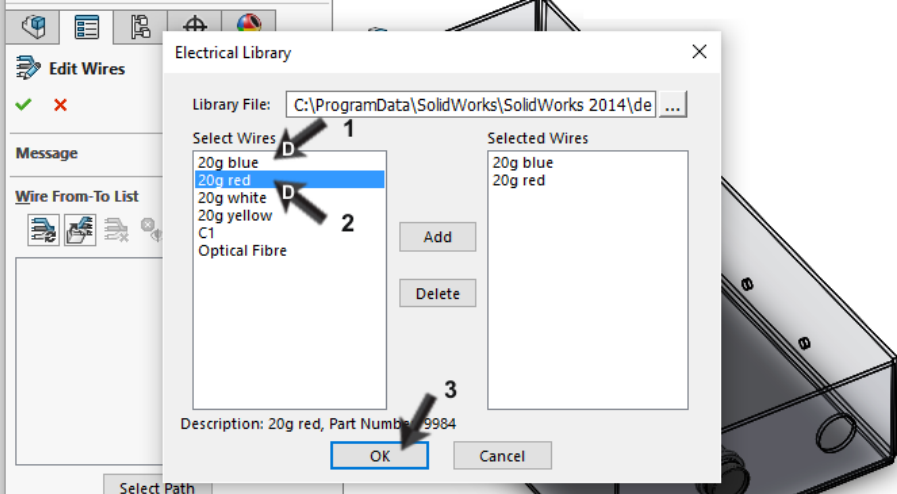
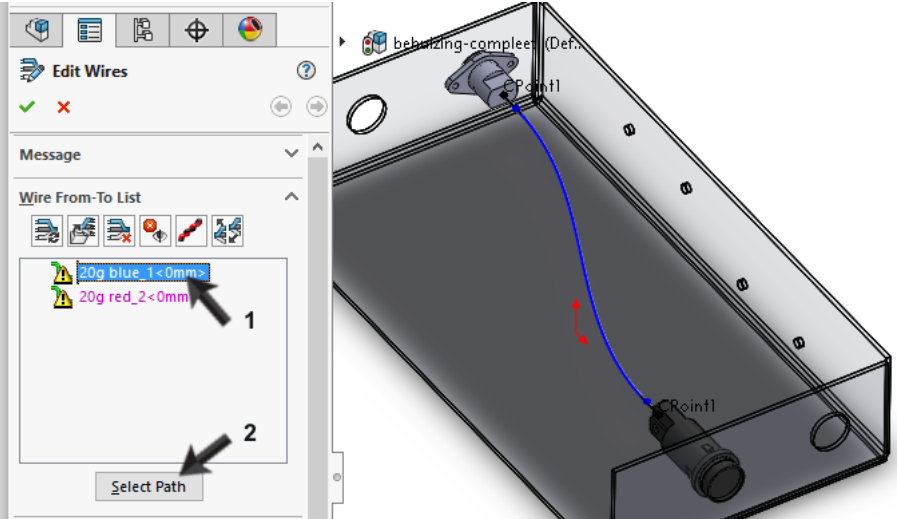
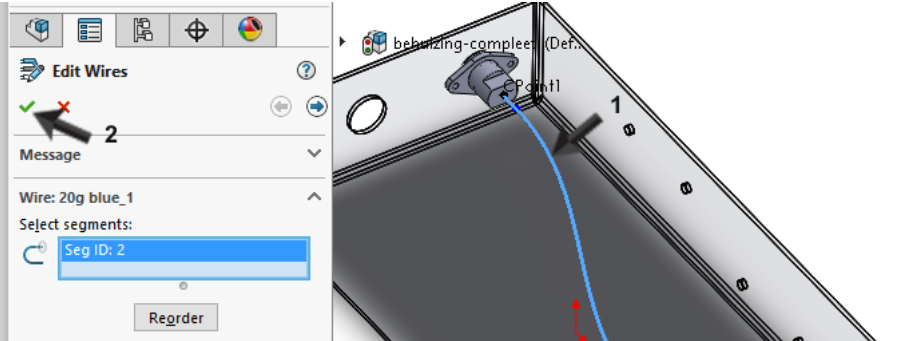
<p>7</p> <p>Place the housing in an assembly</p> <ol style="list-style-type: none"> 1. In the toolbar click on the arrow next to 'New' 2. Click on 'Make assembly from Part/Assembly' 		
<p>8</p> <p>Select the standard template for the assembly and Click OK.</p>		
<p>9</p> <p>In de PropertyManager click OK to place the housing.</p>		
<p>10</p> <p>Save the assembly with the name: housing-complete.sldasm</p>		
<p>11</p> <p>In the CommandManager select the tab 'Electrical'.</p> <p>If the tab 'Electrical' is not available, follow the next steps. Otherwise continue at step 14.</p>		

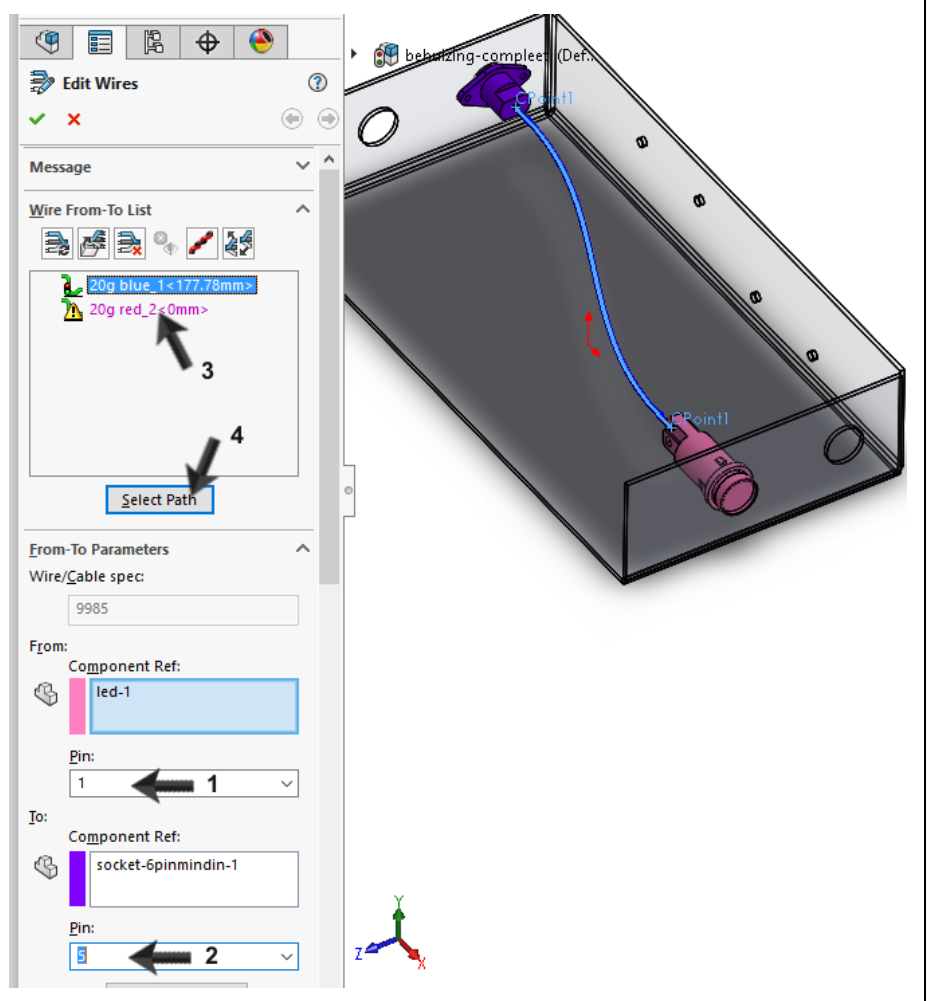
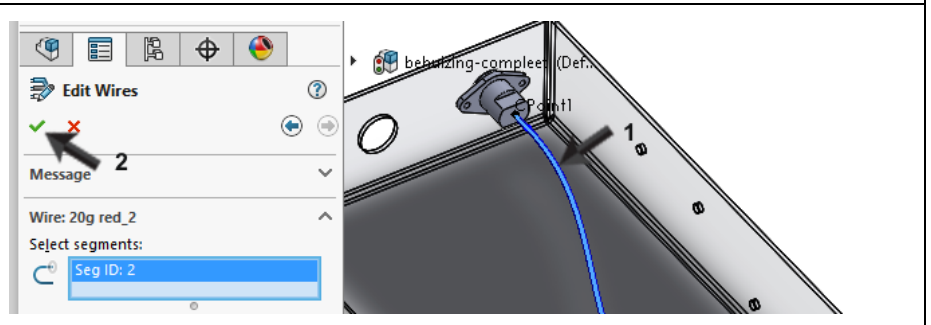
<p>12 If the Tab 'Electrical' is not available in the CommandManager, you'll have to activate the add-in.</p> <ol style="list-style-type: none"> 1. In the toolbar, click on the arrow next to 'Options'. 2. Select 'Add-Ins...'. 	
<p>13 In the menu mark the checkboxes in front and behind SOLIDWORKS Routing. Click OK.</p>	
<p>14</p> <ol style="list-style-type: none"> 1. At the right side of the screen, open the Design Library 2. Select Routing 3. Select Electrical <p>The number of components available in the standard version of SOLIDWORKS is quite limited. At the end of this tutorial you'll see how you can create your own components and add them to the Design Library.</p>	

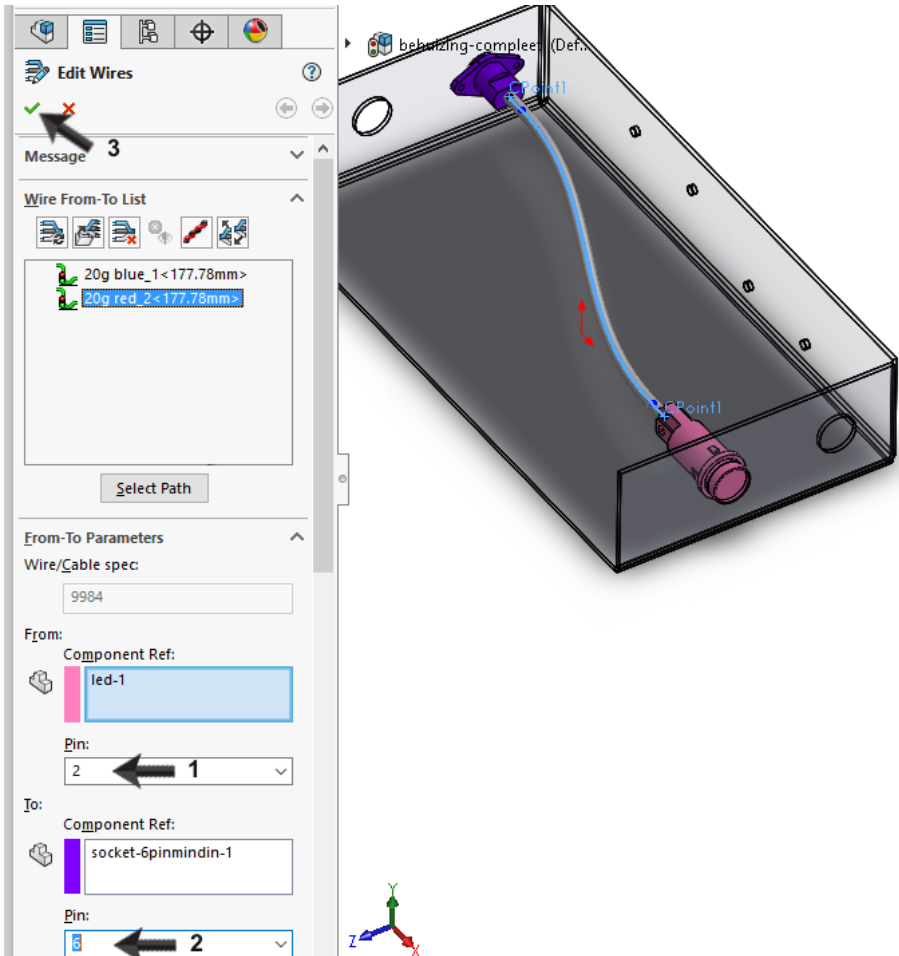
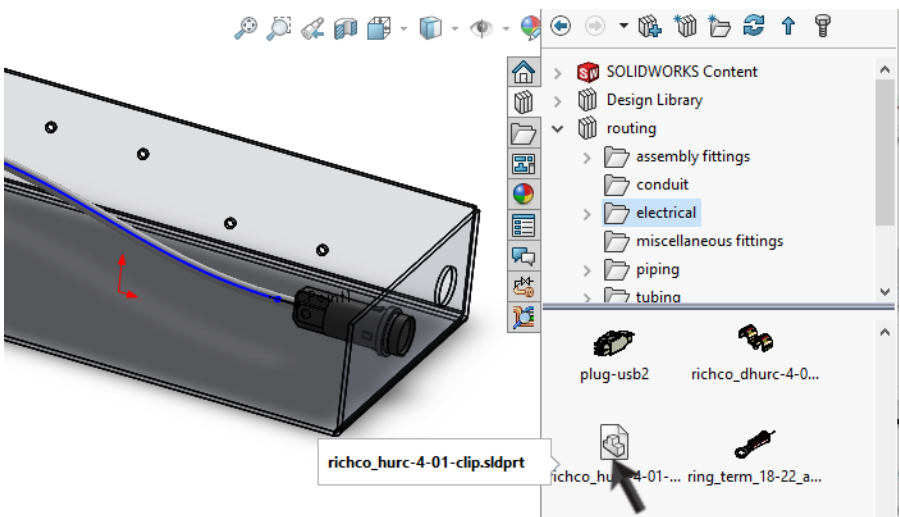
<p>15 Rotate the housing so that you can see the inside of the right wall.</p> <p>In the Design Library select the component 'led'.</p>		 <p>The screenshot shows the SolidWorks interface. On the left, a 3D model of a grey housing is shown from an isometric perspective. A red 'X' is placed over the top of the housing. On the right, the Design Library is open, showing a tree view with folders for 'routing', 'assembly fittings', 'conduit', 'electrical', 'miscellaneous fittings', 'piping', and 'tubing'. The 'electrical' folder is expanded, and the 'led' component is selected. Below the Design Library, a preview of the 'led' component is shown, along with other components like 'db9 male', 'db15-e', and 'led - rs_276-068'.</p>
<p>16 Drag the component 'led' to the hole on the right. Make sure the component snaps exactly in the right position before you release the mouse button.</p>		 <p>The screenshot shows the SolidWorks interface with the 'led' component being dragged into a hole in the housing. A black arrow points to the component, and a red 'X' is placed over the top of the housing. The Design Library is visible on the right, showing the 'led' component selected.</p>
<p>17 In the PropertyManager click OK.</p> <p>A new sub-assembly is created now, containing the electrical components and wiring.</p> <p>SOLIDWORKS opens this assembly in-context, the housing might show transparent (depending on the settings).</p> <p>The command Auto Route is activated now.</p>		 <p>The screenshot shows the SolidWorks PropertyManager dialog box for 'Route Properties'. The 'Message' section is highlighted, containing the text: 'Select the default properties of the new route subassembly.' Below the message, there are fields for 'Routing subassembly:' and 'Routing template:'. The 'Routing subassembly:' field contains '-complete.sldasm' and the 'Routing template:' field contains 'C:\ProgramData\SolidWork...'. A red 'X' is placed over the 'OK' button in the dialog box. The background shows the 3D model of the housing with the 'led' component inserted into the hole.</p>

<p>18</p> <p>Rotate the model so that you can see the inside of the left wall.</p> <p>Lookup the component 'socket-6pinmindin' in de Design Library.</p>		
<p>19</p> <p>Drag this component to the right hole. Make sure it snaps to the right location.</p>		
<p>Tip!</p>		<p>Notice that both components have a 'CPoint' hebben. A CPoint (or Connection Point) is the pont where a cable is connected to the component. Next to the CPoint you see a short line, which is actually the beginning of the cable. Usaaly a component has only one CPoint, because most components connect to only one cable (with multiple wires). Lateron we will se how to connect the wires to the right pin.</p>

<p>20</p> <p>Two components have been placed, the command Auto Route is still active.</p> <p>Now we add a cable between these two components.</p> <ol style="list-style-type: none"> 1. Make sure that in the PropertyManager the field 'current Selection' is active by clicking in it. 2. Click on the endpoint of the line near the connector. 3. Click on the endpoint of the line near the led. <p>The two components now are connects bij a spline. We have created a 3D-Sketch.</p> <ol style="list-style-type: none"> 4. Click OK. 		
<p>21</p> <p>The cable we just placed is actually an empty sleeve. We have to add wires to it.</p> <p>In the CommandManager click on Edit Wires.</p>		
<p>22</p> <p>In the PropertyManager click on the button Add Wire</p>		

<p>23</p> <p>In the menu that pops up, we add a blue and a red wire.</p> <ol style="list-style-type: none"> 1. Double click on 20g blue. 2. Double click on 20g red. <p>Check if both wires are in the column 'Selected Wires'.</p> <ol style="list-style-type: none"> 3. Click OK. 	
<p>24</p> <p>The wires now have to be linked to the cable.</p> <ol style="list-style-type: none"> 1. In the PropertyManager select the first wire (blue). 2. Click on Select Path. 	
<p>25</p> <ol style="list-style-type: none"> 1. Select the cable we created earlier. 2. Click OK. 	

<p>26</p> <p>In the PropertyManager, under From-To Parameters, you can connect the wires to a selected pin.</p> <ol style="list-style-type: none"> 1. Select pin 1 of the LED. 2. Select pin 5 of the connector. <p>Now we connect the second wire.</p> <ol style="list-style-type: none"> 3. Select the red wire. 4. Click on Select Path. 	
<p>27</p> <ol style="list-style-type: none"> 1. Select the cable 2. Click OK 	

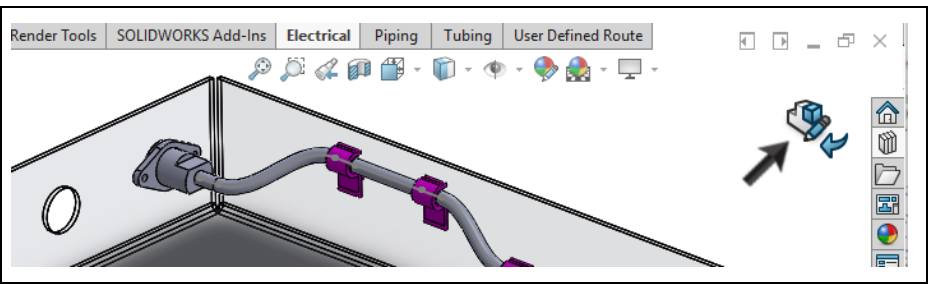
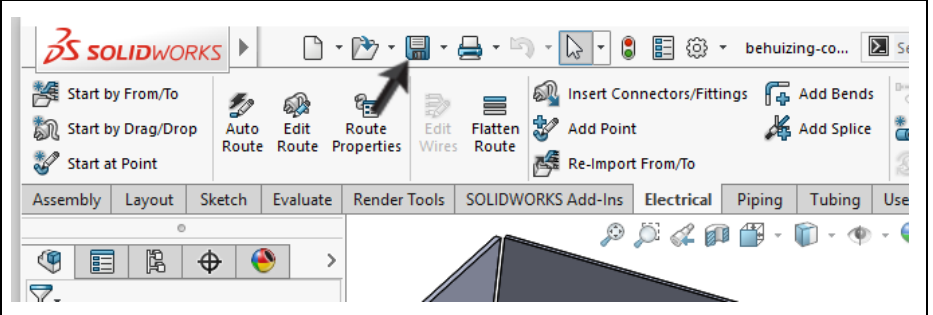
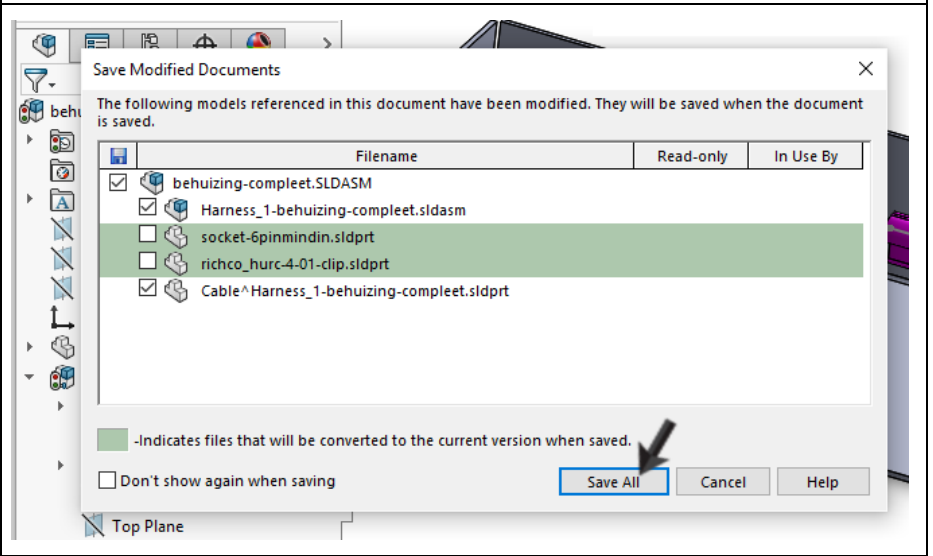
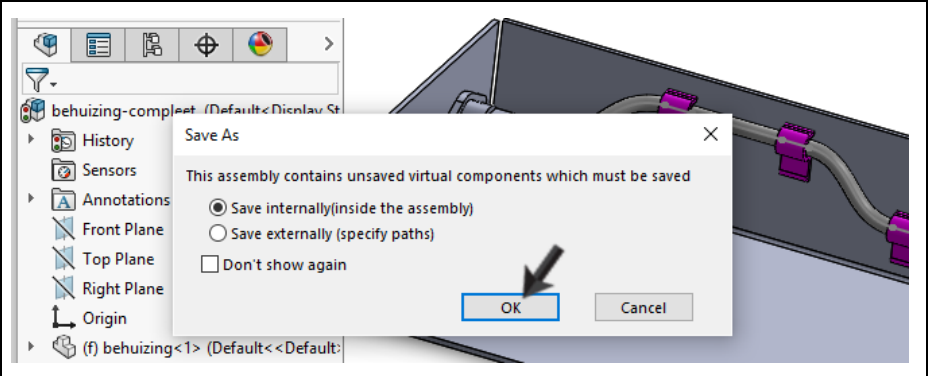
<p>28</p> <p>Again the wire has to connect to the correct pins</p> <ol style="list-style-type: none"> 1. Select pin 2 of the LED. 2. Select pin 6 of the connector. 3. Click OK. 	
<p>29</p> <p>Now we fix the cable with some clips.</p> <p>In the Design Library select the clip named 'rich-co_hurc-4-01-clip'</p>	

<p>30</p>	<ol style="list-style-type: none"> 1. Drag the clip to the four holes 2. Click Cancel. 	
<p>31</p>	<p>The clips now have to be rotated upright.</p> <ol style="list-style-type: none"> 1. Right click the first clip. 2. Pick Add/Edit Mates (Rotate Clip) 3. Click on Rotate Clip 	
<p>32</p>	<ol style="list-style-type: none"> 1. Set the angle at 90° 2. Click OK. <p>Repeat these steps for the other three clips.</p>	
<p>33</p>	<p>In the CommandManager click on Auto Route.</p>	

<p>34</p>	<ol style="list-style-type: none"> 1. Select the cable 2. Select the four clips vier clips. It's best to select each clip by the center-line. 3. Click OK. 	
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<p>Tip!</p>	<p>If something unintended happen when linking the cable to the clips, you'd best immediately click Undo in the PropertyManager and give it another try.</p>	<p>The linking of the cable to the clips does not result in another feature you can easily edit afterwards. It's rather a relation in a 3D sketch.</p>
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<p>35</p>	<p>Close the sketch of the cable</p>	
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<p>36</p>	<p>Close the routing sub-assembly.</p>	
<p>37</p>	<p>The first cable of the harness is ready now. Save to file.</p>	
<p>38</p>	<p>Click Save All-in the popup menu.</p>	
<p>39</p>	<p>Click OK if you see the menu in the illustration.</p>	
<p>Work plan</p>	<p>At this point we have routed one cable in the housing, which directly connect the LED to the connector. Now we will add a second led this time with terminals.</p>	

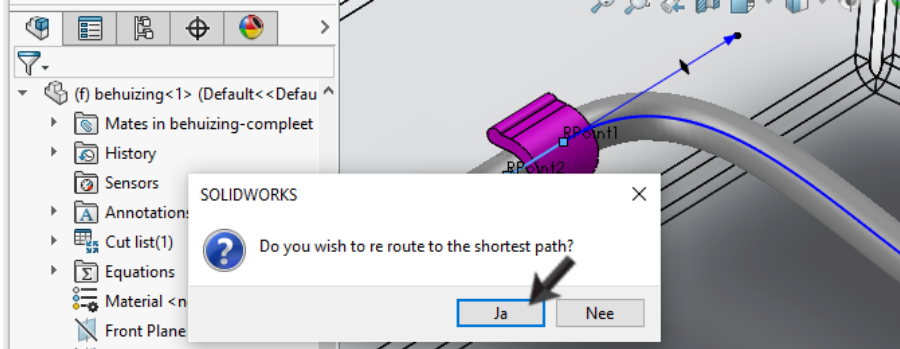
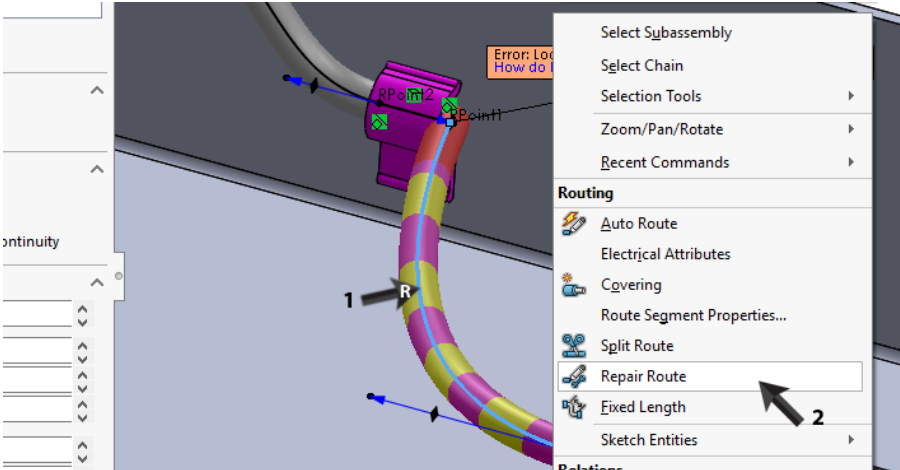
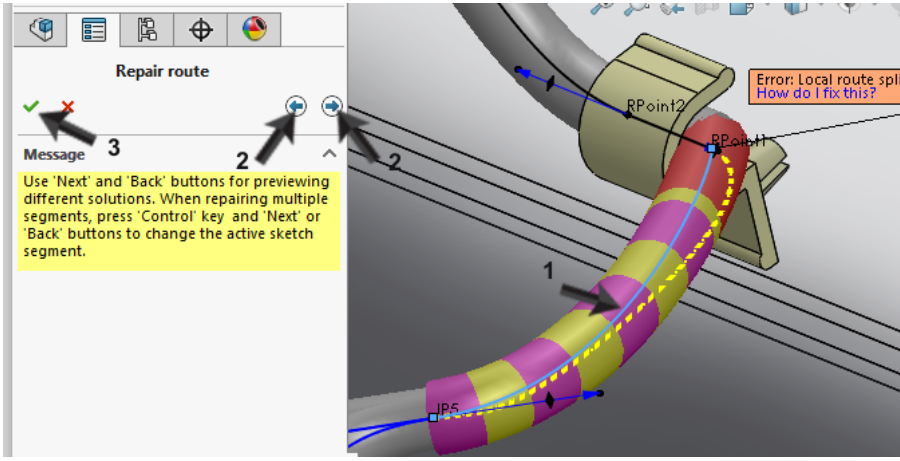
<p>40</p>	<p>First we make an extra hole in the housing.</p> <ol style="list-style-type: none"> 1. Select the housing in the FeatureManager. 2. Click on Edit Part 	<p>The screenshot shows the FeatureManager tree on the left with 'behuizing-compleet' selected. An arrow labeled '1' points to the 'behuizing-compleet' folder. Another arrow labeled '2' points to the 'Edit Part' button in the top right corner of the software interface. To the right, a 3D model of a blue housing with a cable is shown.</p>
<p>41</p>	<p>Make a sketch as you can see in the illustration.</p>	<p>The screenshot shows the sketch environment with a grey housing part. A sketch of a hole is being created. Dimensions are shown: a horizontal distance of 32,50 and a vertical distance of 12,50. A circular hole with a diameter of $\varnothing 13$ is also visible. The FeatureManager tree on the left shows 'behuizing-compleet' selected.</p>
<p>42</p>	<ol style="list-style-type: none"> 1. In the CommandManager select the tab Features 2. Click on Extruded Cut. <p>Create the hole with depth Through All.</p>	<p>The screenshot shows the CommandManager with the 'Features' tab selected. An arrow labeled '1' points to the 'Features' tab. Another arrow labeled '2' points to the 'Extruded Cut' button. The background shows the sketch from the previous step.</p>
<p>43</p>	<p>Exit In-context editing.</p>	<p>The screenshot shows the software interface with the 'Exit In-context editing' button highlighted by an arrow. The background shows the housing part with the hole.</p>

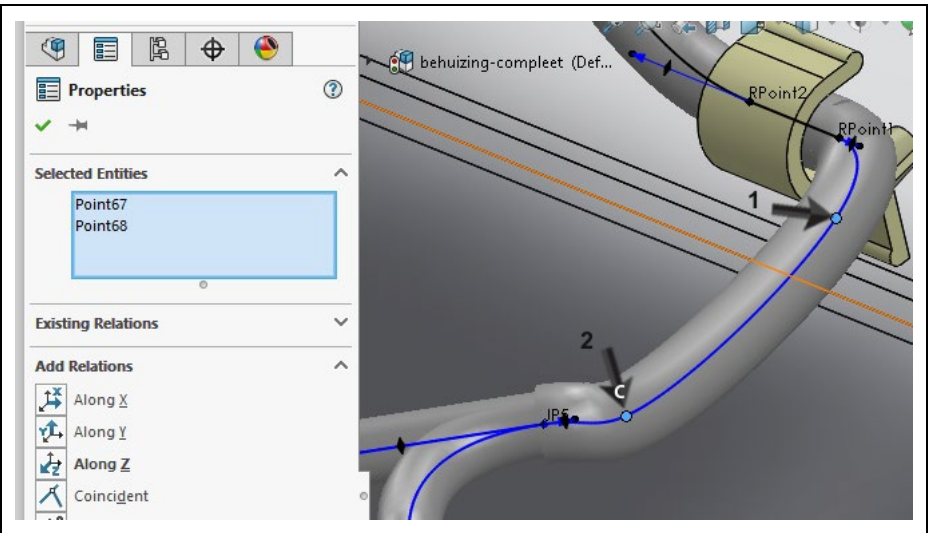
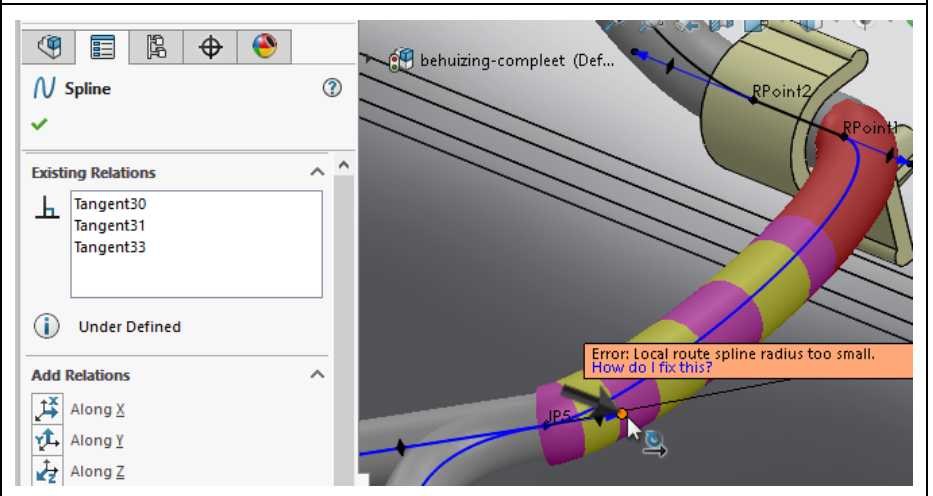
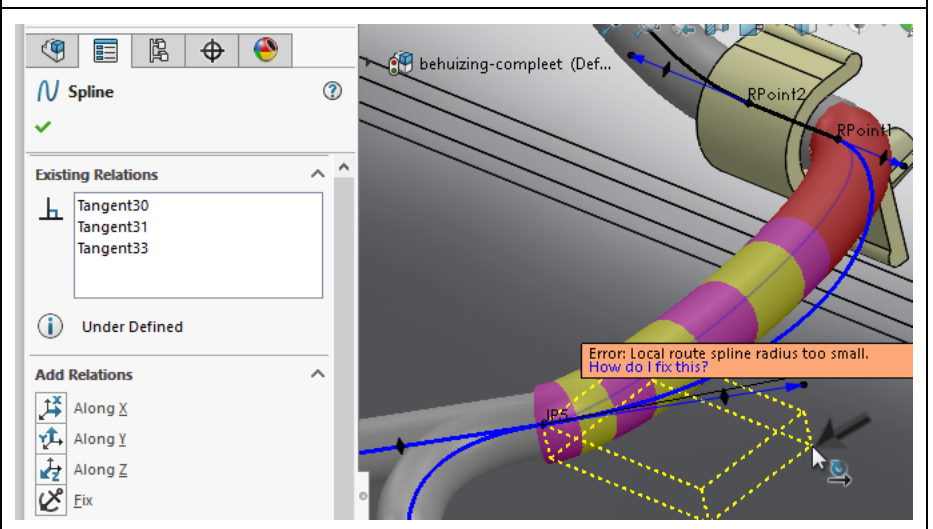
<p>44</p>	<ol style="list-style-type: none"> 1. In the Design Library find the component 'led'. 2. Drag and drop this component to the hole you created. 3. In the PropertyManager click on Cancel 	
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<p>Tip!</p>	<p>If at step 44 you would have clicked OK, a second harness would have been opened.</p> <p>In this case we don't want the led to be a component of harness, but just a regular part in the assembly. We achieve this by clicking Cancel.</p> <p>The two terminals we are going to use to connect the led should become part of the existing (not a new) harness. You will see now how we can do this.</p>
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<p>45</p>	<ol style="list-style-type: none"> 1. In the FeatureManager right click the sub-assembly of the harness. 2. Click on 'Edit Route'. <p>The existing harness, we created earlier, will be re-opened.</p>	
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<p>46</p>	<ol style="list-style-type: none"> 1. In the Design Library find the component 'terminal'. 2. Drag and drop this component twice to the LED. 	
<p>47</p>	<p>Click Cancel to stop adding terminals.</p>	
<p>48</p>	<p>Now we have to modify the cable. First we have to unhook the cable from the rightmost clip. This gives us some room to branch-off the cable.</p> <ol style="list-style-type: none"> 1. Zoom in to the rightmost clip. 2. Right click the line of the cable under the clip 3. Select Unhook from Clip 	

<p>49</p>	<p>Select Yes to recalculate the path of the cable.</p>	
<p>Tip!</p>		<p>In the CommandManager you'll also find a button 'Unhook from Clip'. If you use that one, you will not get the option to reroute the cable (step 49), and you will have to shape it manually (step 52 and further)</p>
<p>50</p>	<p>SOLIDWORKS now might show an error if the cable is bent to sharply</p> <p>Right click the cable and choose Repair Route</p>	
<p>51</p>	<ol style="list-style-type: none"> 1. Select the cable near the area that causes the problem. <p>With a yellow line, SOLIDWORKS will suggest a solution.</p> <ol style="list-style-type: none"> 2. Would you like to see other solutions? Click Next or Previous in the PropertyManager until you see the desired curve. 3. Click OK. 	
<p>Tip!</p>		<p>The curve of the cable is defined by a spline in a 3D-sketch. Splines are not covered in these tutorials. In the next steps you will get some idea of how to do them.</p>

<p>52 We now will manually modify the shape of the cable (spline)</p> <p>On the spline you can see control points. We don't need these anymore.</p> <p>Select the points (use the <Ctrl>-key to select both).</p> <p>Press <Delete></p> <p>NOTE: the points can be in a completely different location than you see in the illustration!</p>	
<p>53 Again the cable is bended to sharply. This time we will solve this manually.</p> <p>At each control point you can see two arrows with a little circle at the end.</p> <p>Select the circle as you can see here.</p>	
<p>54 Drag the circle to the left.</p> <p>Watch the shape of the spline change as you do so. Release the mouse button when the shape looks fine.</p> <p>The error message now will have disappeared.</p>	
<p>Tip!</p>	<p>You'll get the smoothest curves when you use as few control points as possible. That is why we removed some of the control points in step 52.</p> <p>With the arrows (step 53-54) you can control how sharp the curve is bended.</p> <p>If the curve has to go through a specific point, you could add control points. Right click the location of the spline you want to change and choose</p>

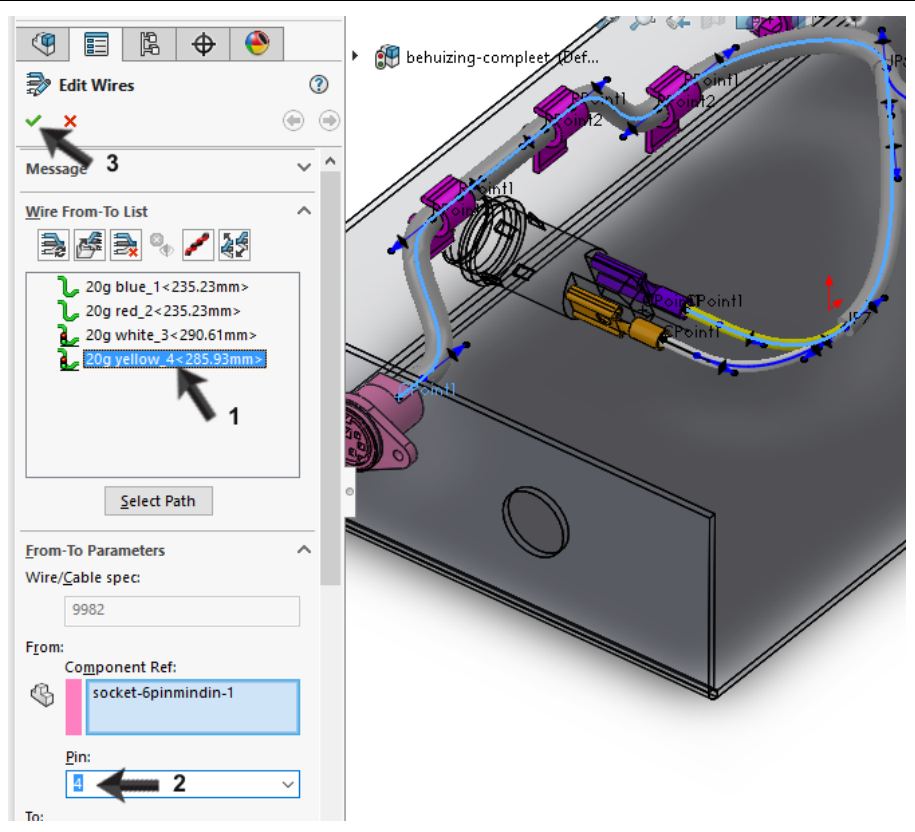
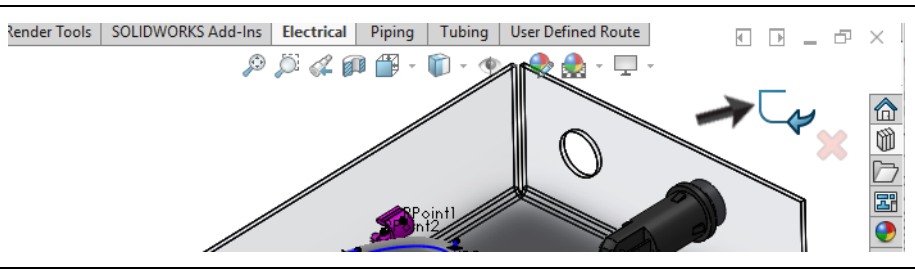
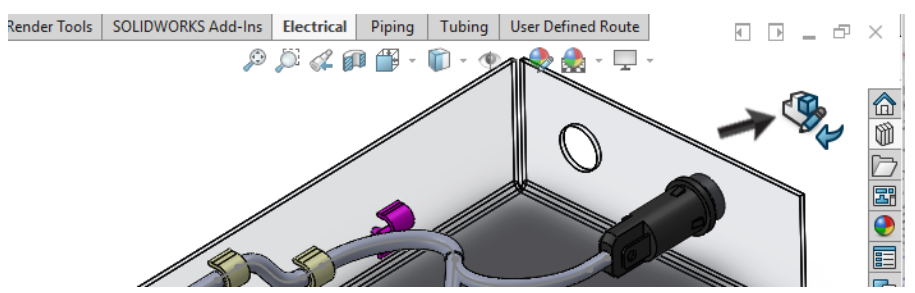
		<p>'Insert Spline Point'. Then you can drag that point to the right location, use relations, or use the arrows at the control point to modify the shape.</p> <p>Give it a try!</p>
<p>55</p>	<p>Now we will connect the led.</p> <p>First we branch-off the existing cable.</p> <p>Right click the cable and choose Split Route.</p>	
<p>56</p>	<p>Click on the cable as you can see in the illustration.</p> <p>Press <Esc>.</p>	
<p>57</p>	<p>In the CommandManager click on Auto Route to create the cable.</p>	

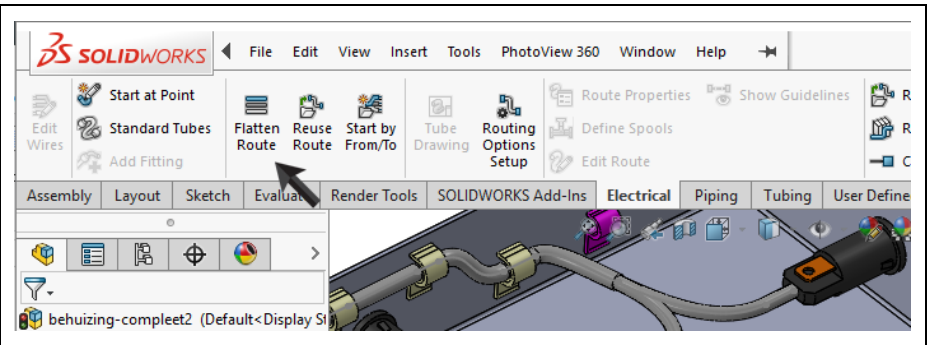
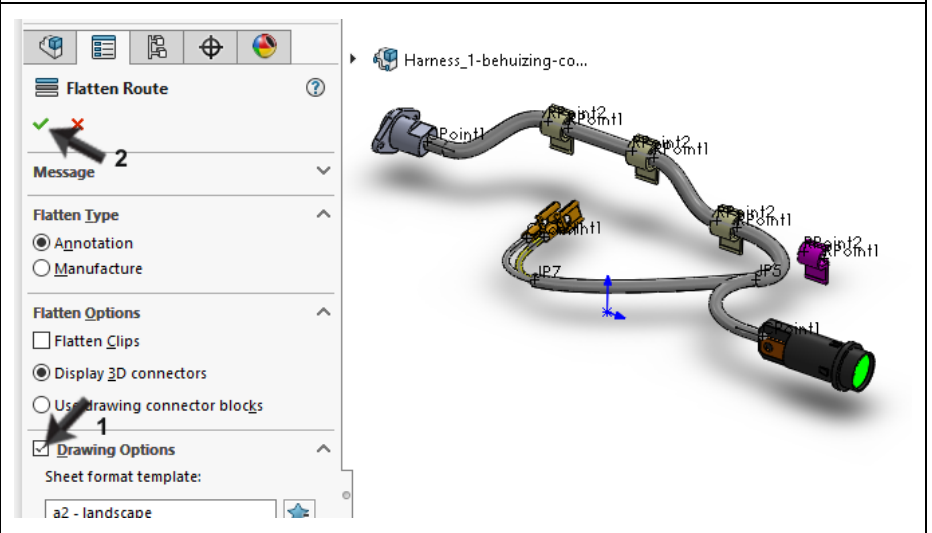
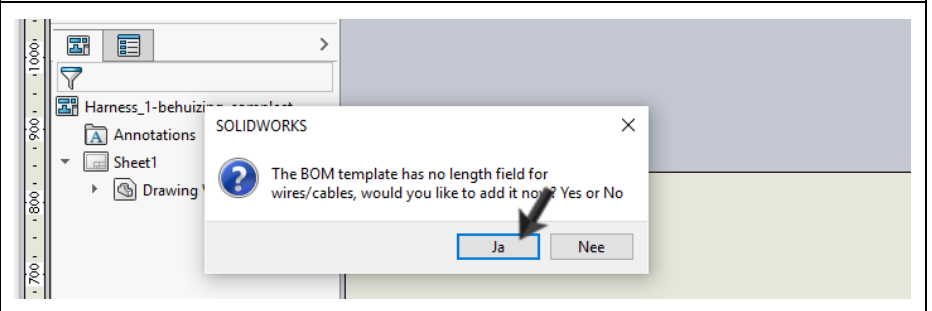
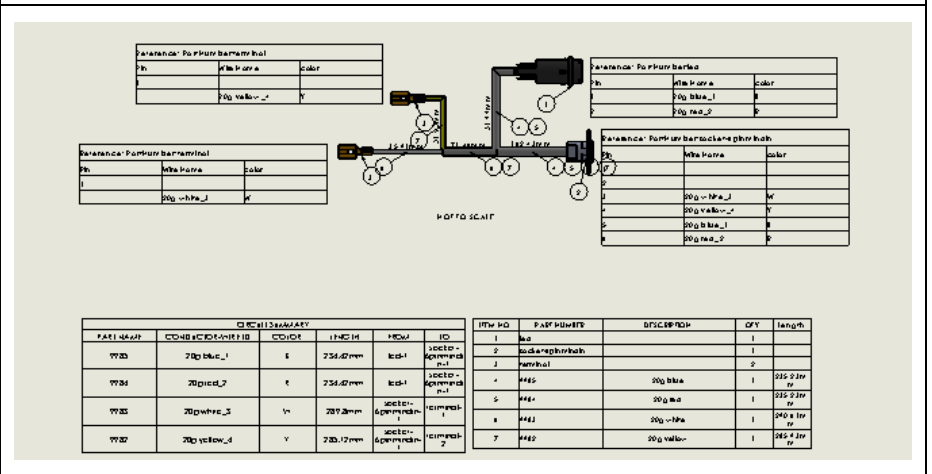
<p>58</p>	<ol style="list-style-type: none"> 1. Select the endpoint of the first terminal and then the point where you have split the existing cable. 2. Click OK. 	
<p>59</p>	<ol style="list-style-type: none"> 1. Select the curve you've just drawn 2. Press and hold the <Ctrl>-key and select the curve as you can see in the illustration 3. In the PropertyManager select Tangent. 4. Click OK <p>There is no sharp bend between the old and the new cable now.</p>	
<p>60</p>	<p>Now we create a second branch-off</p> <ol style="list-style-type: none"> 1. Right click on the cable and choose Split Route. 2. Click on the new cable as you can see in the illustration. 3. Press <Esc>. 	

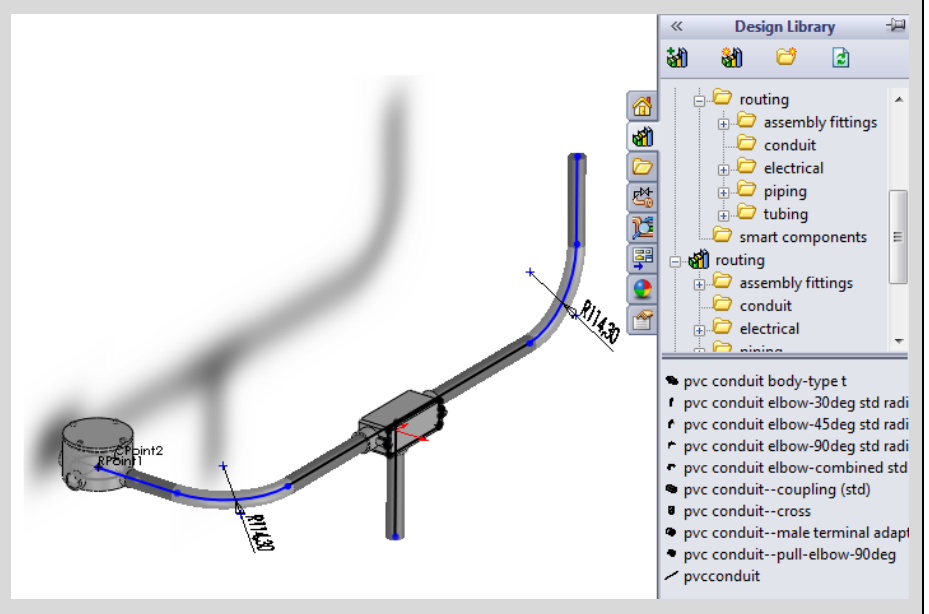
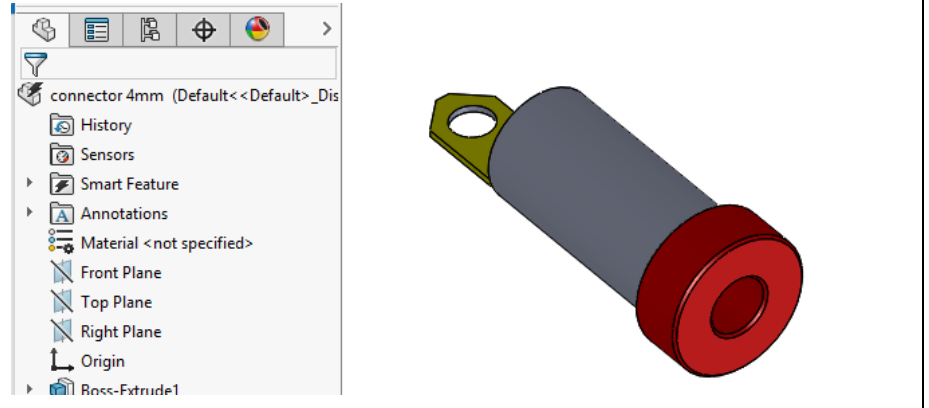
<p>61</p>	<ol style="list-style-type: none"> 1. In the CommandManager, click Auto Route 2. Select the endpoint of the terminal 3. Select the point you just added to the cable. 4. Click OK. 	
<p>62</p>	<ol style="list-style-type: none"> 1. Select the newly created cable. 2. Select (hold down the <Ctrl>-key) the old cable. 3. In the CommandManager select Tangent. 4. Click OK. 	
<p>63</p>	<p>All cables are now there, the next step is to add wires.</p> <p>In the CommandManager click on Edit Wires.</p> <p>In the PropertyManager you will see the two wires we created previously.</p>	
<p>64</p>	<p>Click on Add Wire.</p>	

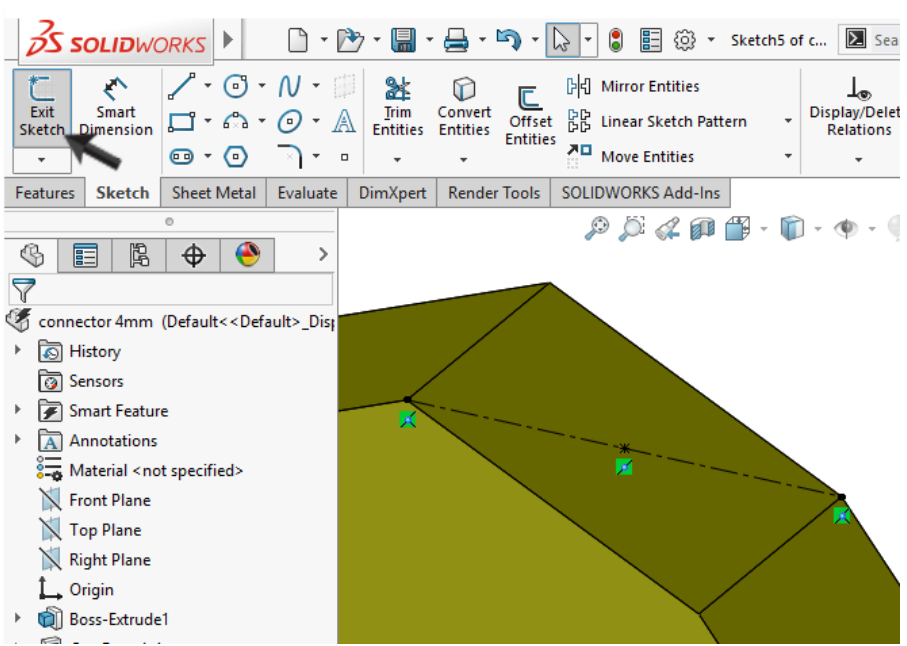
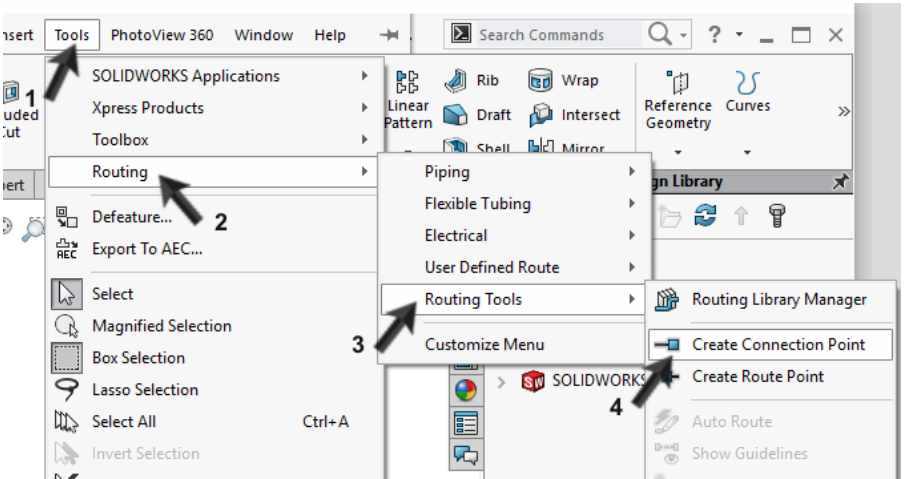
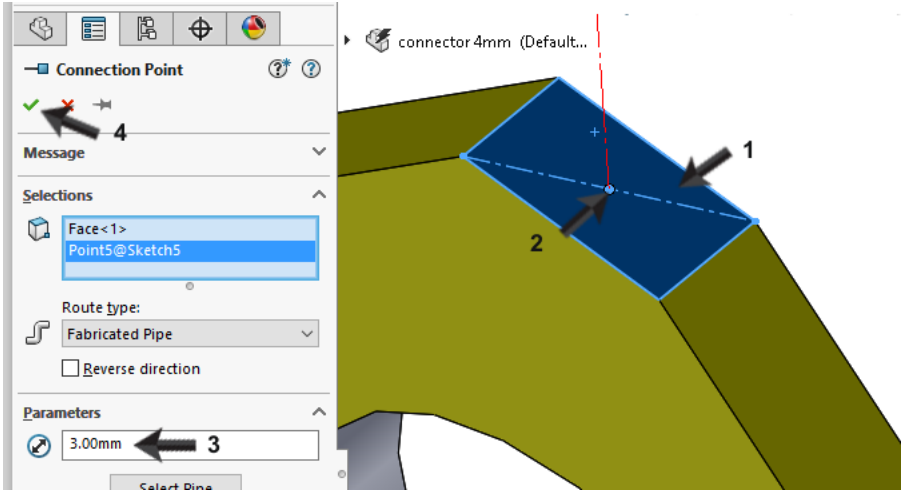
<p>65</p>	<ol style="list-style-type: none"> 1. Select the white wire. 2. Select (hold down the <Ctrl>-key) the yellow wire. 3. Click on Add. 4. Click OK. 	
<p>66</p>	<ol style="list-style-type: none"> 1. Select the white wire in the PropertyManager 2. Click on Select Path. 	
<p>67</p>	<ol style="list-style-type: none"> 1-3 Select the parts of the cable, that connects the connector with the first terminal 4. Click on Next. 	

<p>68</p>	<p>1-3 Select the parts of the cable, that connects the connector with the second terminal</p> <p>4. Click OK.</p>	
<p>69</p>	<p>1. Select the white wire in the PropertyManager.</p> <p>2. Link it to pin 3 of the connector.</p>	

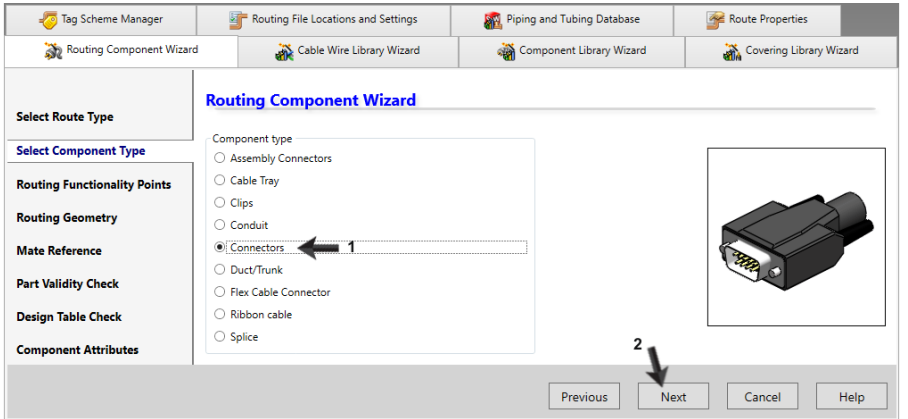
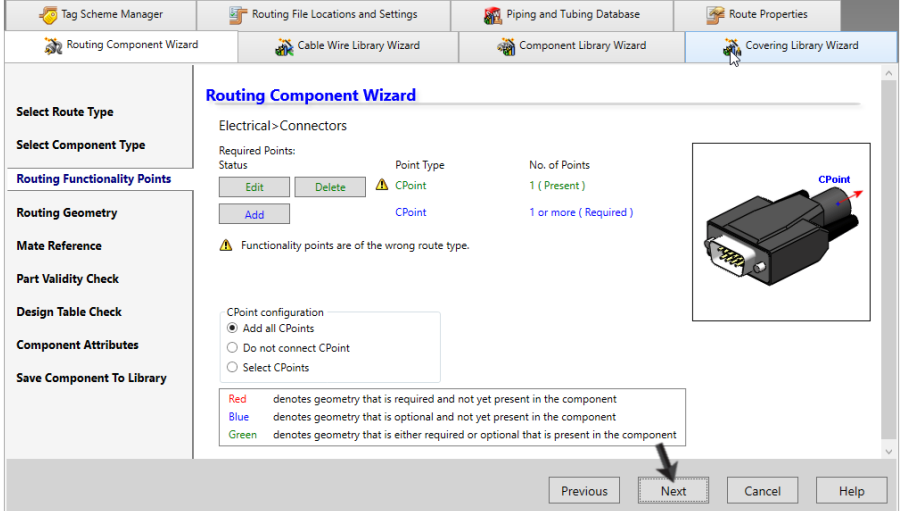
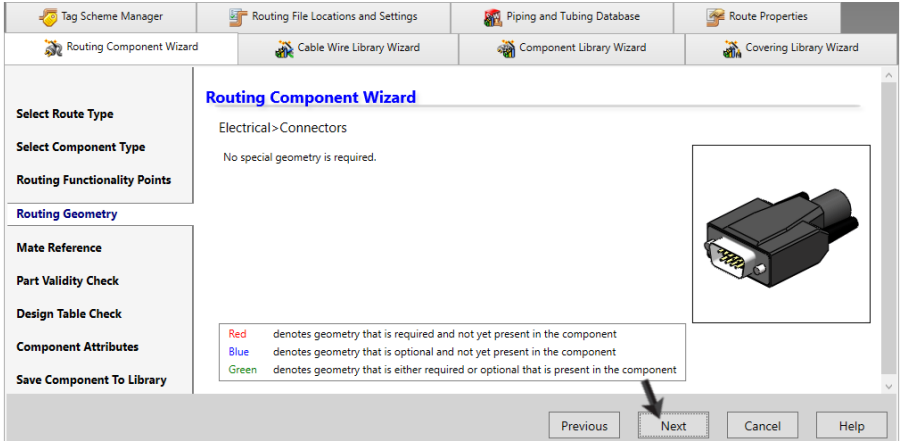
<p>70</p> <ol style="list-style-type: none"> 1. Select the yellow wire in the PropertyManager. 2. Link it to pin 4 of the connector. 3. Click OK. 	
<p>71</p> <p>Close the sketch of the cable.</p>	
<p>72</p> <p>Close the sub-assembly of the harness.</p>	
<p>73</p> <p>The harness is ready now. Save all files.</p>	

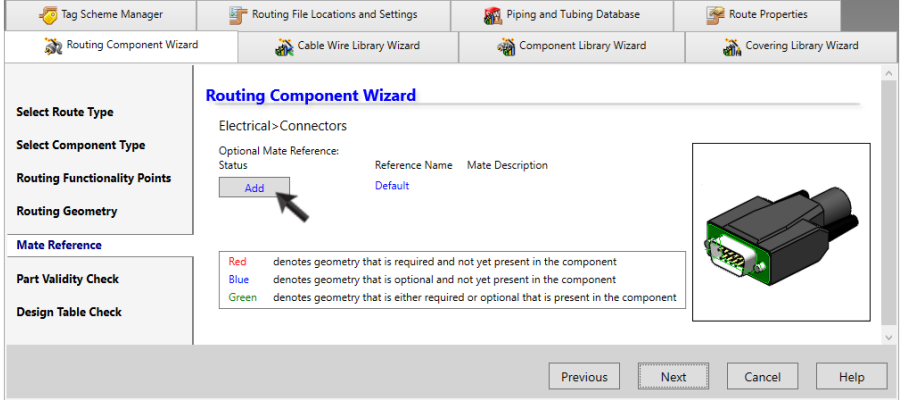
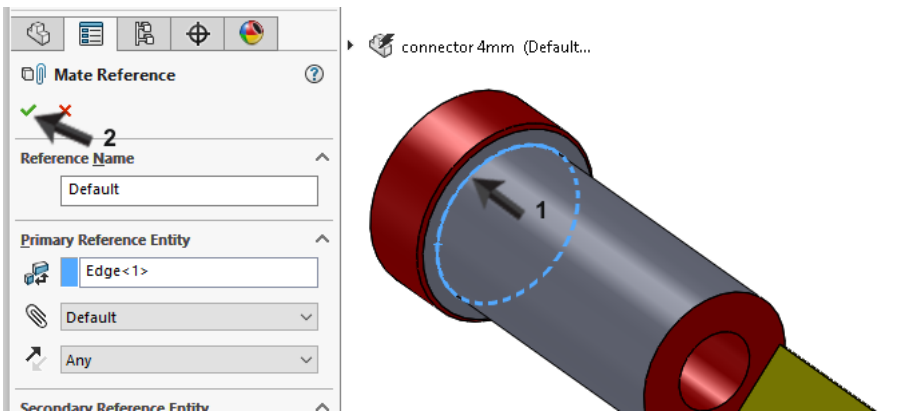
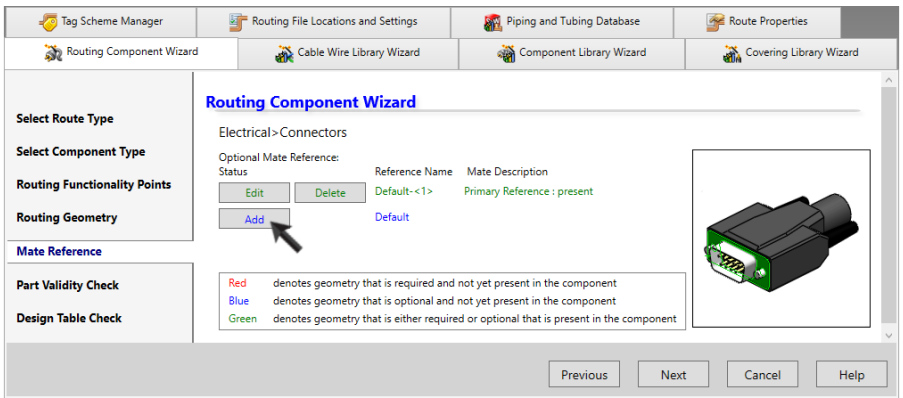
<p>74</p>	<p>Next we will make a schematic drawing of the harness. In the CommandManager click on Flatten Route.</p>																																					
<p>75</p>	<ol style="list-style-type: none"> In the PropertyManager check Drawing Options. Click OK. 																																					
<p>76</p>	<p>To add the length of the wires to the BOM, click Yes.</p>																																					
<p>77</p>	<p>The drawing is now created, with the bill of materials and a wiring scheme Save this drawing.</p>	 <table border="1" data-bbox="670 1680 1053 1814"> <thead> <tr> <th colspan="6">BOM SUMMARY</th> </tr> <tr> <th>PART NAME</th> <th>CODE</th> <th>UNIT</th> <th>QTY</th> <th>UNIT PRICE</th> <th>TOTAL</th> </tr> </thead> <tbody> <tr> <td>W03</td> <td>20g black_1</td> <td>1</td> <td>1</td> <td>234.07mm</td> <td>234.07mm</td> </tr> <tr> <td>W04</td> <td>20g black_2</td> <td>1</td> <td>1</td> <td>234.07mm</td> <td>234.07mm</td> </tr> <tr> <td>W05</td> <td>20g black_3</td> <td>1</td> <td>1</td> <td>207.2mm</td> <td>207.2mm</td> </tr> <tr> <td>W06</td> <td>20g black_4</td> <td>1</td> <td>1</td> <td>234.07mm</td> <td>234.07mm</td> </tr> </tbody> </table>	BOM SUMMARY						PART NAME	CODE	UNIT	QTY	UNIT PRICE	TOTAL	W03	20g black_1	1	1	234.07mm	234.07mm	W04	20g black_2	1	1	234.07mm	234.07mm	W05	20g black_3	1	1	207.2mm	207.2mm	W06	20g black_4	1	1	234.07mm	234.07mm
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<p>Tip!</p>	<p>We have now seen how you can make a Harness. You could also make a conduit, consisting of tubes. To do so, make sure that the first routing part you add to the assembly is selected from the folder 'conduit' and not,</p>																																					

		<p>as we did before, form the folder 'electrical'.</p> 
	<p>Tip!</p>	<p>The number of components available in the Design Library is very limited. You can not do much more with it than a simple exercise like we did in this tutorial. There are two ways to get more components.</p> <ol style="list-style-type: none"> 1. You can search for libraries on the internet, or 3D ContentCentral. 2. You could also define your own components. In the next steps we will show you how to add components to the Design Library.
<p>78</p>	<p>First, model the component you want to use.</p> <p>In this example we'll use the connector you see in the illustration. You can download this part from www.SOLIDWORKS.nl</p>	

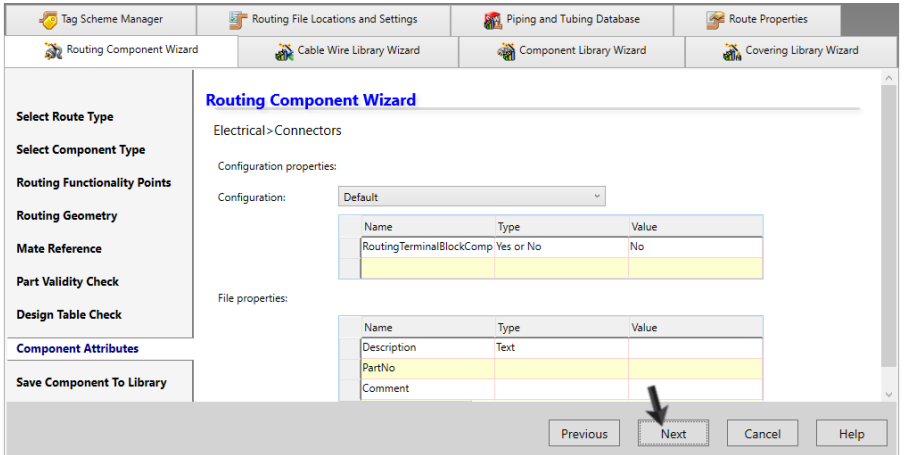
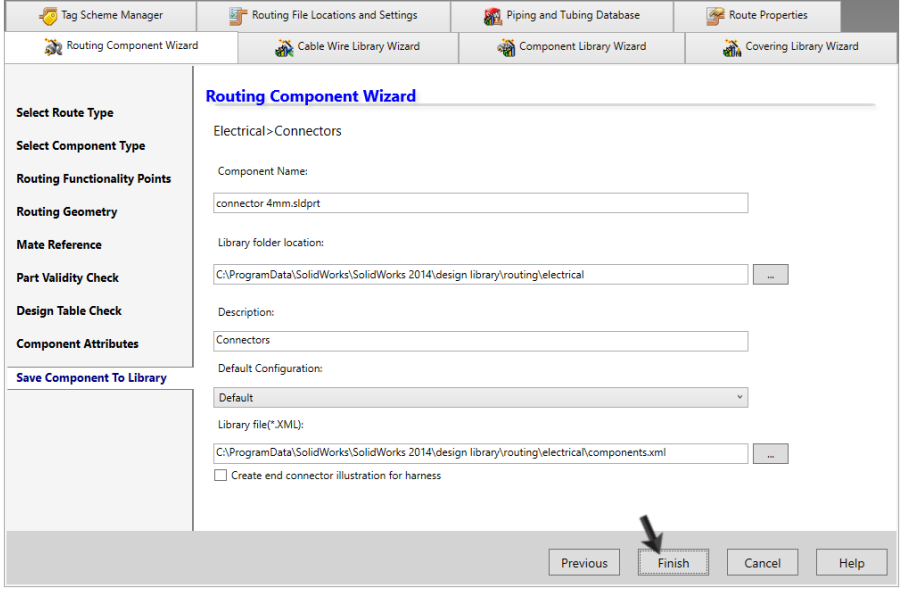
<p>79</p> <p>In the model we have to define a connection point.</p> <p>Make the sketch you see in the illustration, consisting of a construction line and a point on its midpoint.</p> <p>Close de sketch.</p>	
<p>80</p> <ol style="list-style-type: none"> 1. In the main menu select Tools 2. Click on Routing 3. Click on Routing Tools 4. Click on Create Connection Point 	
<p>81</p> <ol style="list-style-type: none"> 1. Select the face on which the connection point is located 2. Select the point in the sketch you made earlier 3. Enter the maximum diameter of the cable that can be used for this part 4. Click OK. 	

<p>82</p> <p>To add this part to the Design Library we use a wizard.</p> <ol style="list-style-type: none"> 1. In the mail menu select Tools 2. Click on Routing 3. Click on Routing Tools 4. Select Routing Library Manager 		
<p>83</p> <p>A new window pops up that will guide you through the process</p> <p>Select the first option: Routing Component Wizard</p> <p>We'll walk through a number of steps.</p>		
<p>84</p> <p>In the screen 'Select Route Type' click Next.</p>		

<p>85</p>	<p>1. For component type chose: Connector. 2. Click Next.</p>	
<p>86</p>	<p>We already defined the Connection Point. Click Next</p>	
<p>87</p>	<p>Click Next</p>	

<p>88</p>	<p>Now we have to add mates, so that the component will snap into the right position</p> <p>Click on Add.</p>	
<p>89</p>	<p>The SOLIDWORKS window will re-appear now.</p> <ol style="list-style-type: none"> 1. Select the edge that will align with the hole later on. 2. Click OK. 	
<p>90</p>	<p>In the Routing component wizard click Add again.</p>	

<p>91</p> <p>Select the mates that will make the terminal snap into the right position:</p> <ol style="list-style-type: none"> 1. Select the edge of the hole in the connection strip 2. Select the side face of this strip. Make sure that this face is listed in the PropertyManager under Secondary Reference Entity 3. For this mate, select: parallel 4. Click OK. 	
<p>92</p> <p>In the Routing Component Wizard click Next.</p>	
<p>93</p> <p>All data will be checked. Click Next.</p>	

<p>94</p>	<p>If there are more configurations in your part you can set that here.</p> <p>In our exaple this is not the case, so just click Next.</p>	 <p>The screenshot shows the 'Routing Component Wizard' dialog box. On the left is a tree view with options like 'Select Route Type', 'Select Component Type', 'Routing Functionality Points', 'Routing Geometry', 'Mate Reference', 'Part Validity Check', 'Design Table Check', 'Component Attributes', and 'Save Component To Library'. The main area is titled 'Routing Component Wizard' and shows 'Electrical > Connectors'. It has 'Configuration properties' and 'File properties' sections, each with a table. The 'Next' button at the bottom right is highlighted with a black arrow.</p>
<p>95</p>	<p>Finally you can edit the name of the part and indicate to which library it should be added.</p> <p>Normally you will use the default values here.</p> <p>Click on Finish</p>	 <p>The screenshot shows the 'Routing Component Wizard' dialog box. The 'Component Name' field contains 'connector 4mm.sldprt'. The 'Library folder location' is 'C:\ProgramData\SolidWorks\SolidWorks 2014\design library\routing\electrical'. The 'Description' is 'Connectors'. The 'Default Configuration' is 'Default'. The 'Library file (*.XML)' field contains 'C:\ProgramData\SolidWorks\SolidWorks 2014\design library\routing\electrical\components.xml'. The 'Finish' button at the bottom right is highlighted with a black arrow.</p>
<p>96</p>		<p>The component now has been added to the library. Use it in an assembly to check if everything works correct.</p>

	What are the main features you have learned in this tutorial?	<p>In this tutorial you've seen the basics of routing. You've seen how you can add electrical components and cables between the components.</p> <p>With clips you can route the cables.</p> <p>Finally you created your own component and made it fit to be used in an electrical installation.</p>
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