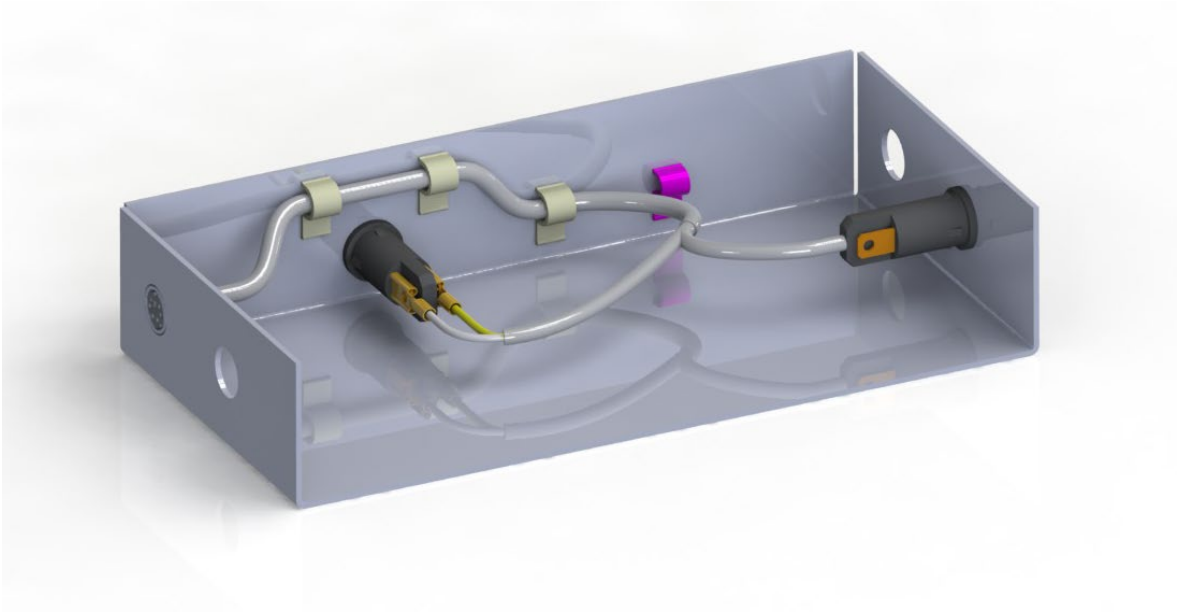
# SOLIDWORKS® tutorial 13

# Routing



Prepatory and Advanced Vocational Training

To be used with SOLIDWORKS® Educational Release 2018-2019

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Initiative: Kees Kloosterboer (SOLIDWORKS Benelux)
Educations advisor: Jack van den Broek
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 wersion 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 se the basics of 'Electrical': placing electrical components in your product and wireing it. After that you will see how you can create your own components.

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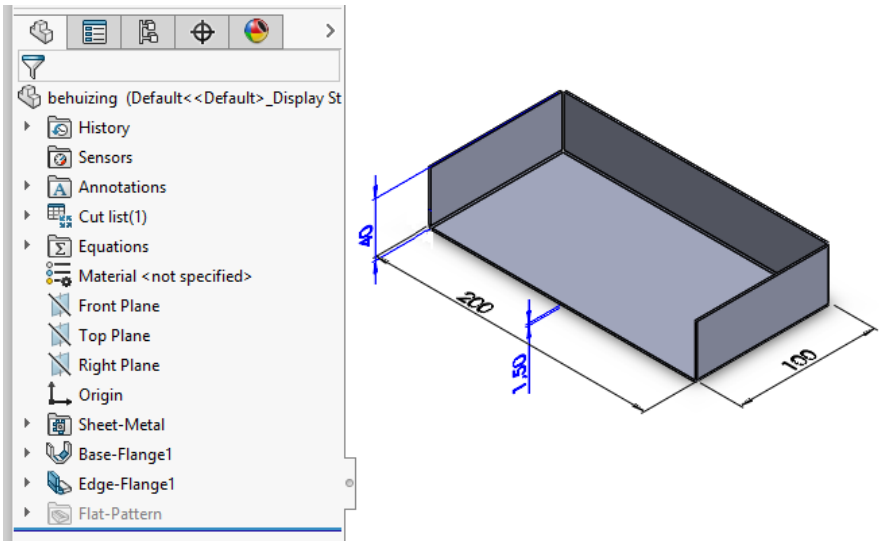
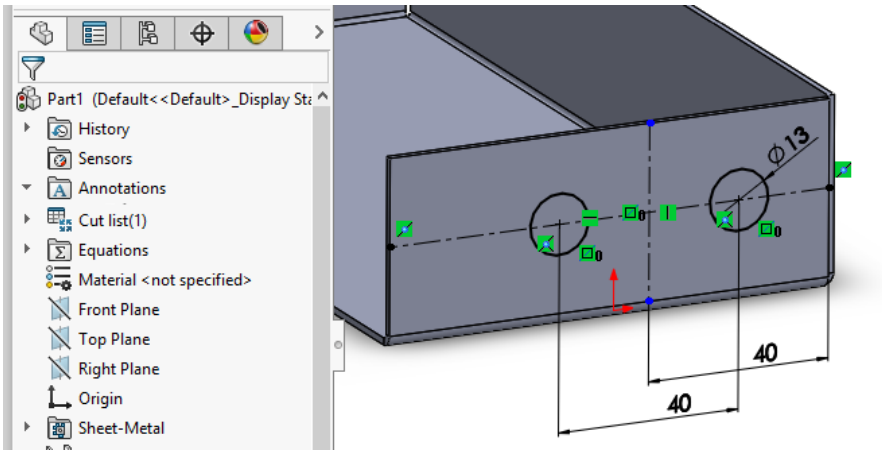
| **1** | Start SOLIDWORKS and open a new part. | |
| **2** | Use sheet metal to create the shape you see in the ilustartion. Have you got trouble doing so? Check tutorial 4, step 1 to 10.<br>The dimensions of this part are 200 x 100 x 40 mm. Thickness is 1.5 mm. |  |
| **3** | On the right side plane,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

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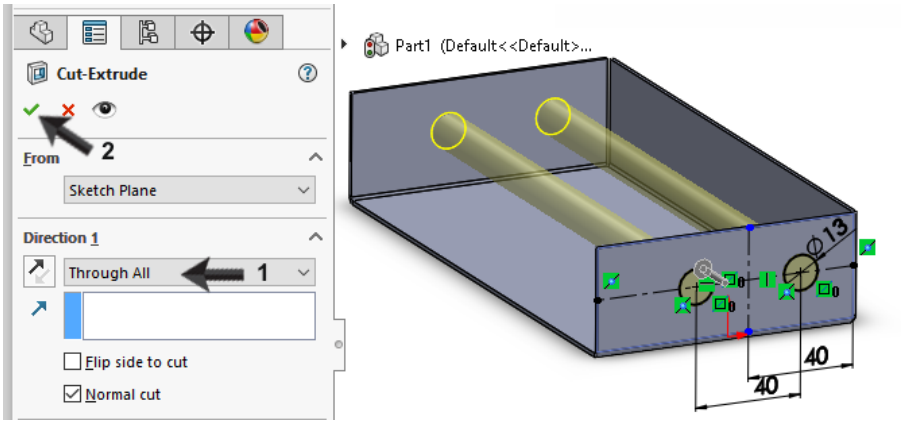
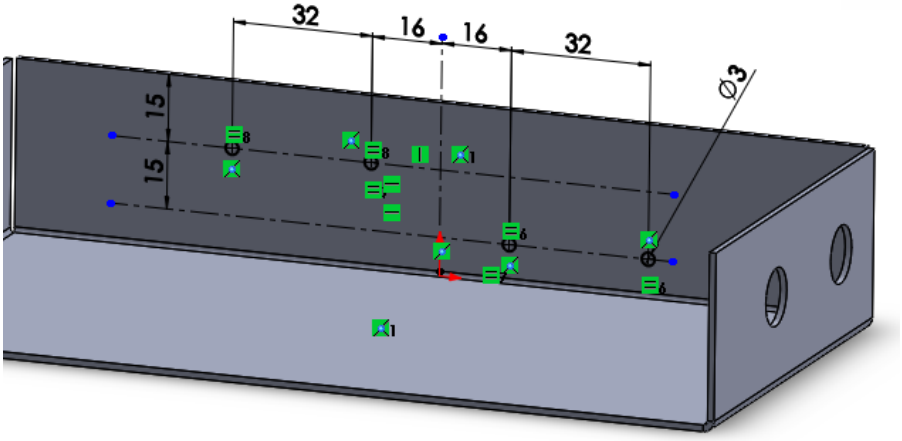
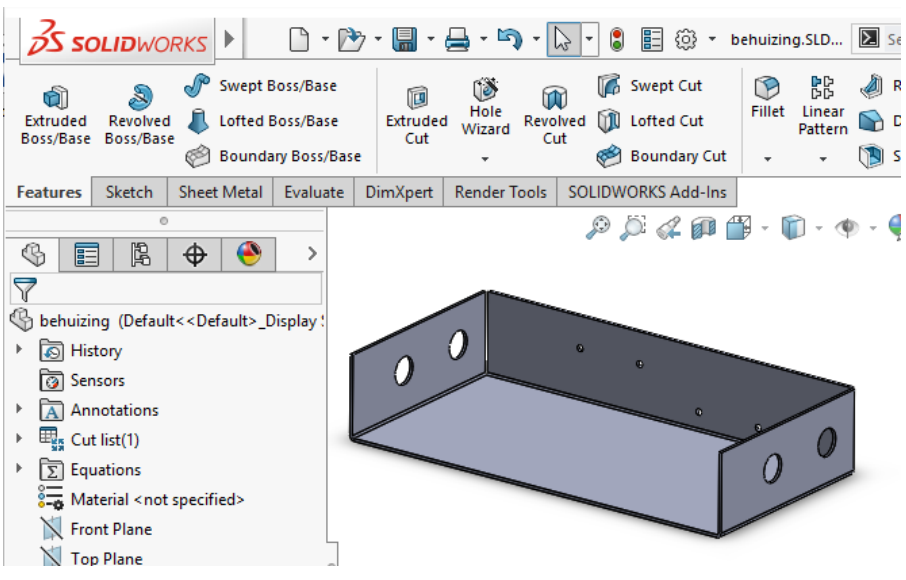
| **4** | Use the sketch to create an Extruded Cut through the whole part ('Through All') |  |
| **5** | On the plane in the back, create the sketch you see here. Make an Extruded Cut out of it, again Through All. |  |
| **6** | Save the part as housing.sldprt |  |

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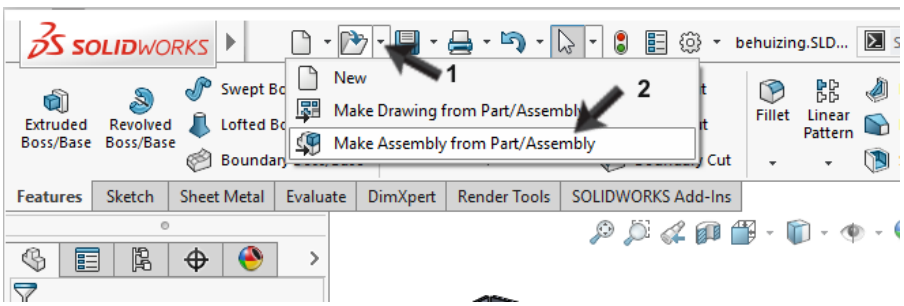
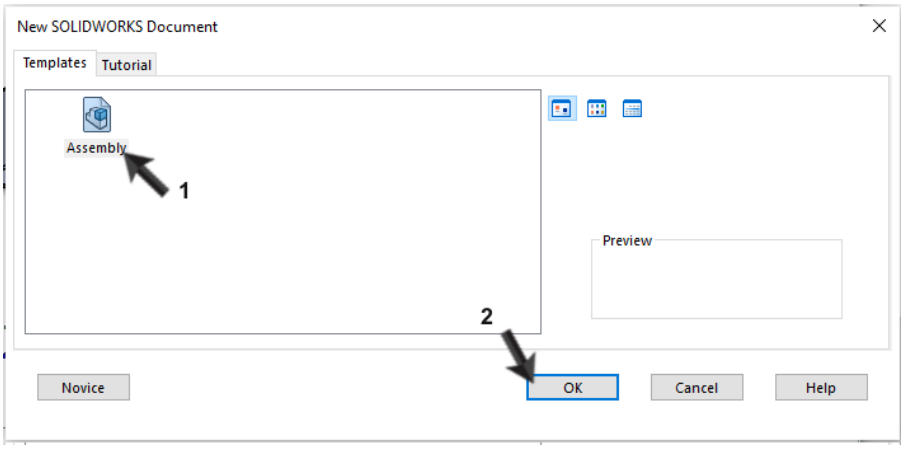
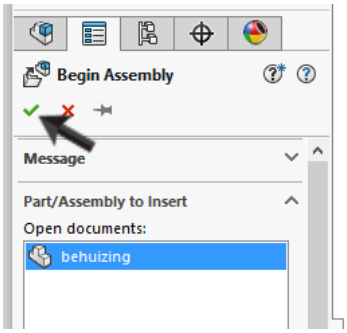
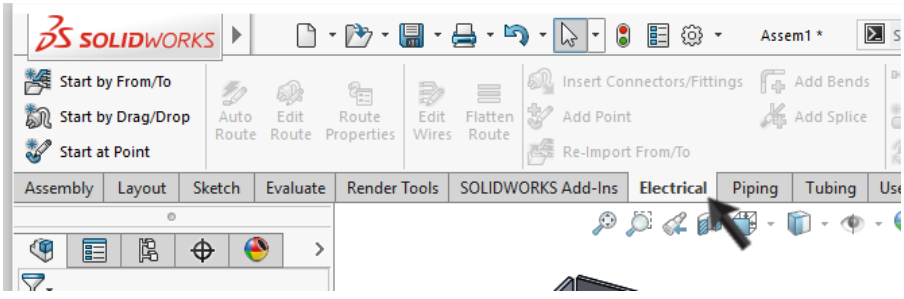
| **7** | Place the housing in an assembly<br>1. In the toolbar click on the arrow next to 'New'<br>2. Click on 'Make assembly from Part/Assembly' |  |
| **8** | Select the standard template for the assembly and Click OK. |  |
| **9** | In de PropertyManager click OK to place the housing. |  |
| **10** | Save the assembly with the name: housing-complete.sldasm | |
| **11** | In the CommandManager select the tab 'Electrical'.<br>If the tab 'Electrical' is not available, follow the next steps. Otherwise continue at step 14. |  |

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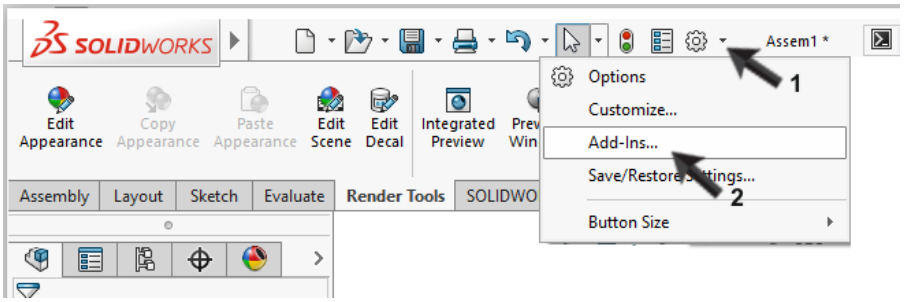
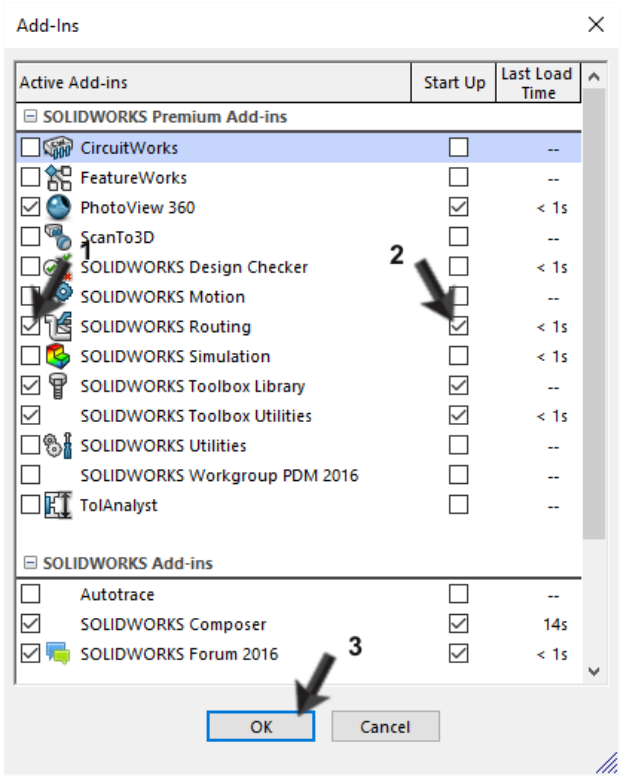
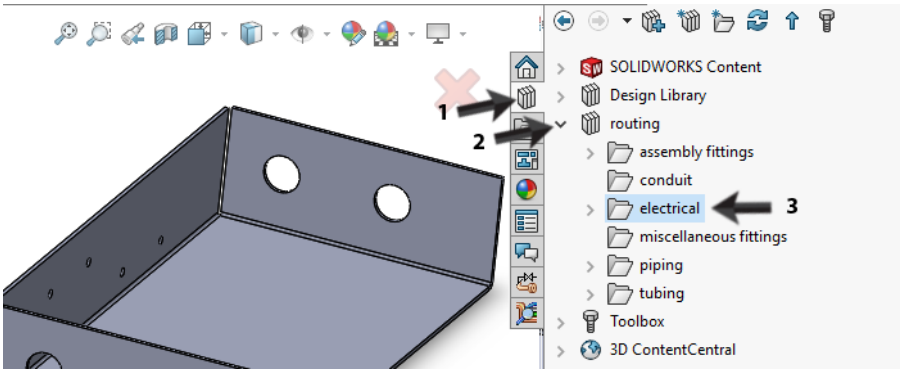
| **12** | If the Tab 'Electrical' is not available in the CommandManager, you'll have to activate the add-in.<br>1. In the toolbar, click on the arrow next to 'Options'.<br>2. Select 'Add-Ins…'. |  |
| **13** | In the menu mark the checkboxes in front and behind SOLIDWORKS Routing.<br>Click OK. |  |
| **14** | 1. At the right side of the screen, open the Design Library<br>2. Select Routing<br>3. Select Electrical<br>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 cab create your own components and add them to the Design Library. |  |

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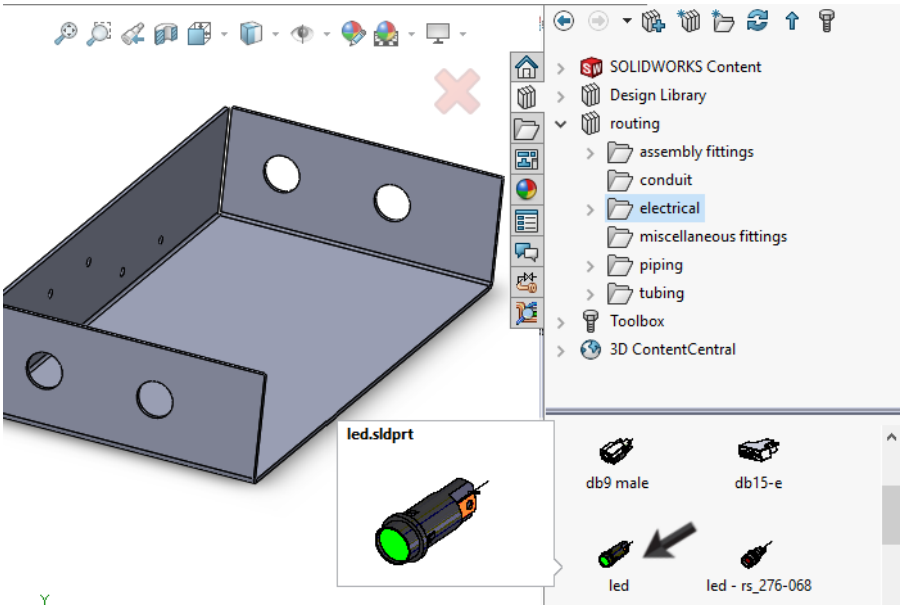
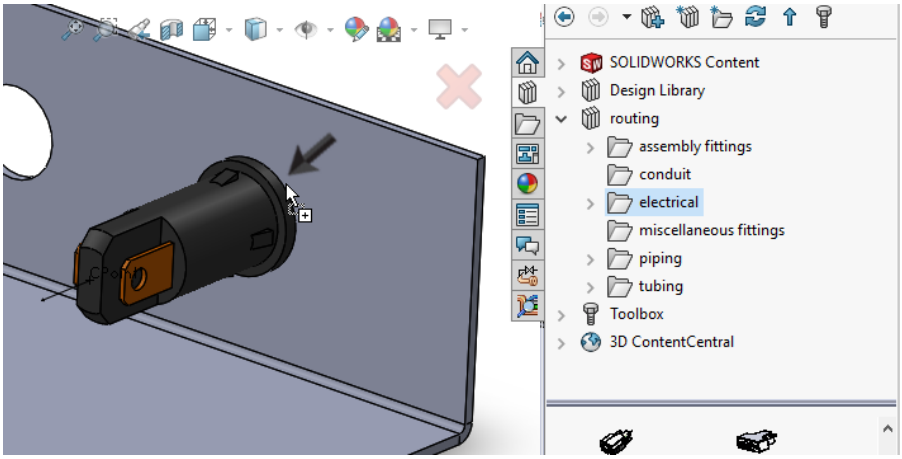
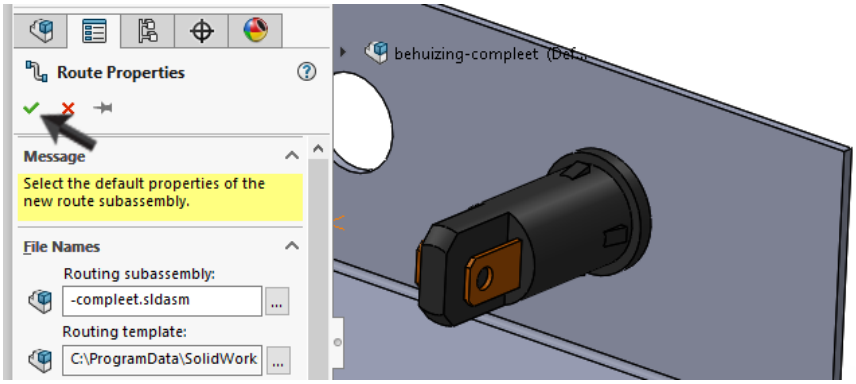
| **15** | Rotate the housing so that you can see the inside of the right wall.<br><br>In the Design Library select the component 'led'. |  |
| **16** | Drag the component 'led' to the hole on the right. Make sure the component snaps exactly in the right position before you release the mouse button. |  |
| **17** | In the PropertyManager click OK.<br><br>A new sub-assembly is created now, containing the electrical components and wiring.<br><br>SOLIDWORKS opens this assembly in-context, the housing might show trans-parent (depending on the settings).<br><br>The command Auto Route is activated now. |  |

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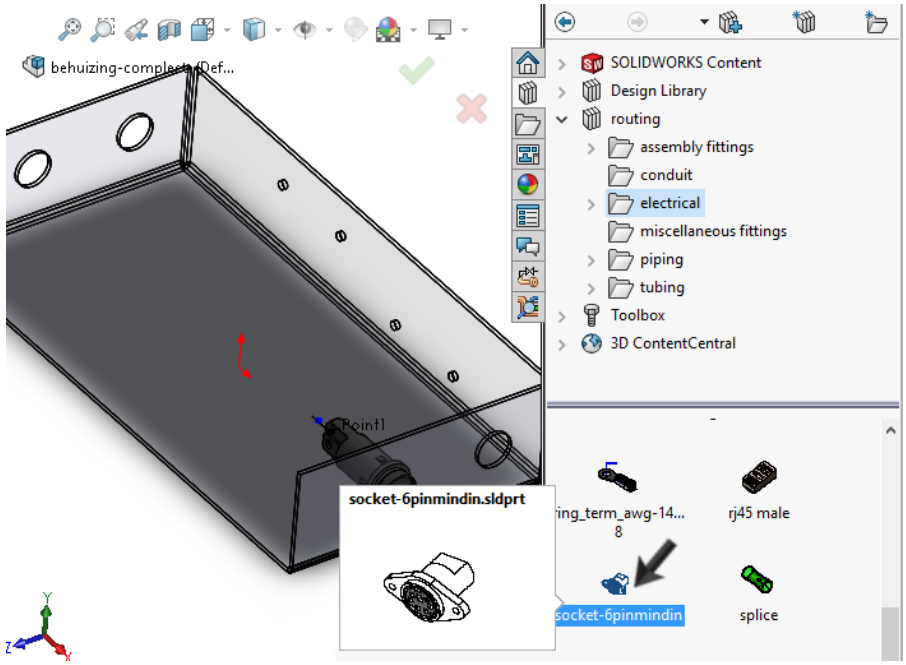
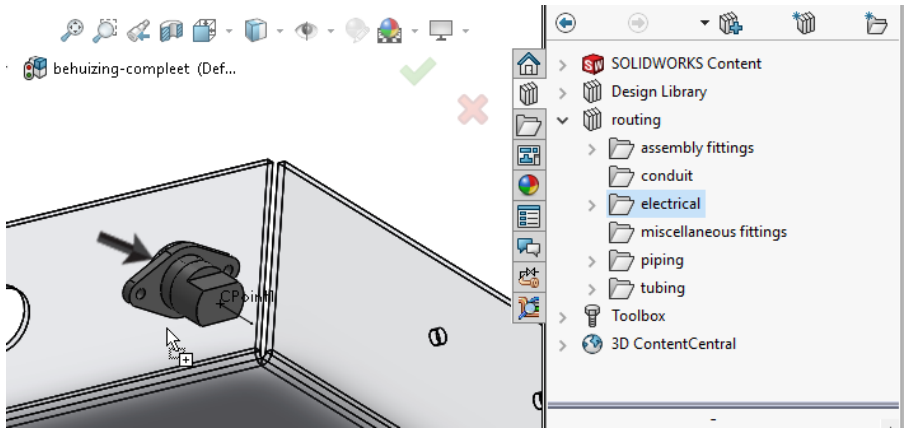
| **18** | Rotate the model so that you can see the inside of the left wall.<br><br>Lookup the component 'socket-6pinminidin' in de Design Library. |  |
| **19** | Drag this component to the right hole. Make sure it snaps to the right location. |  |
| | **Tip!** | Notice that both components have a 'CPoint' 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. |

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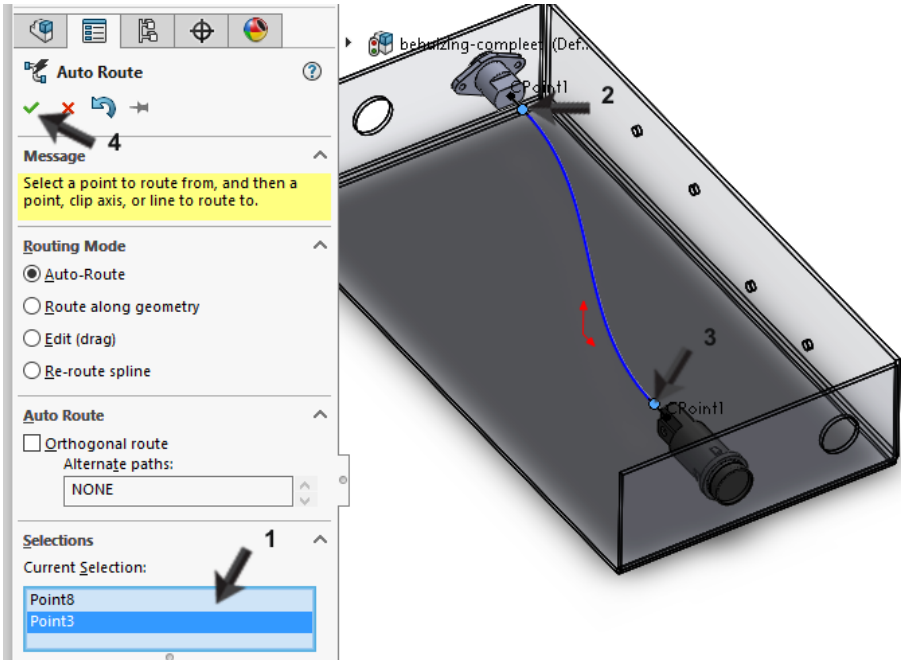
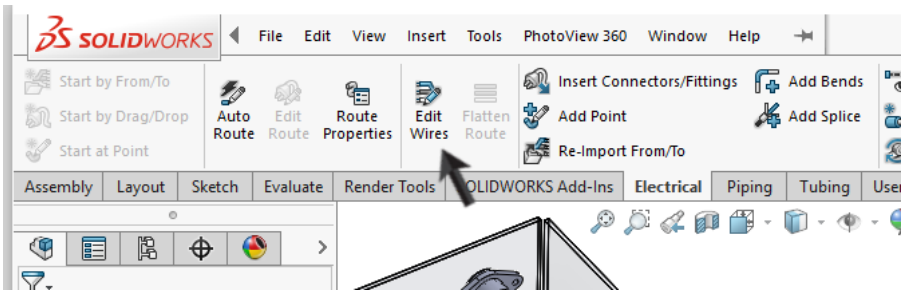
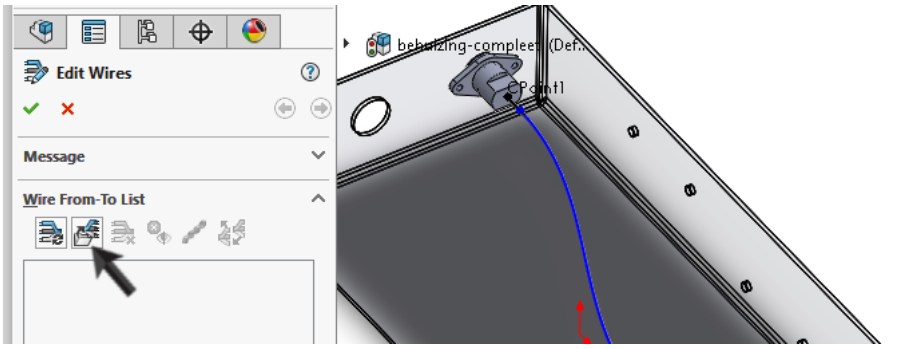
| **20** | Two components have been placed, the command Auto Route is still active.<br><br>Now we add a cable between these two components.<br><br>1. Make sure that in the PropertyManager the field 'current Selection' is active by clicking in it.<br><br>2. Click on the endpoint of the line near the connector.<br><br>3. Click on the endpoint of the line near the led.<br><br>The two components now are connectes bij a spline. We have created a 3D-Sketch.<br><br>4. Click OK. |  |
| **21** | The cable we just placed is actually an empty sleeve. We have to add wires to it.<br><br>In the CommandManager click om Edit Wires. |  |
| **22** | In the PropertyManager click on the button Add Wire |  |

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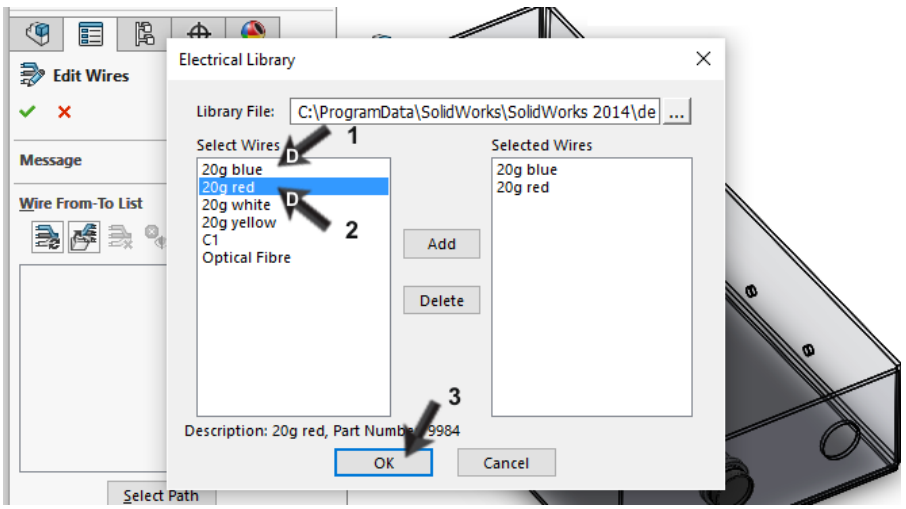
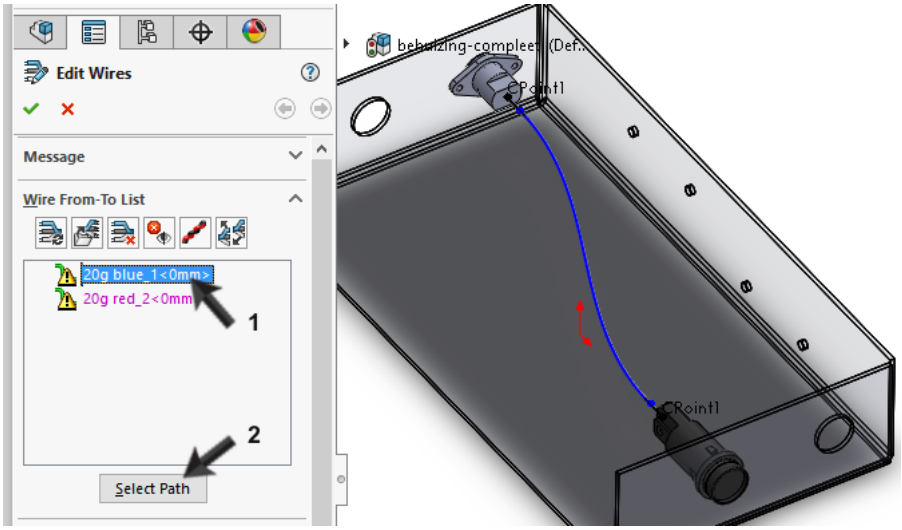
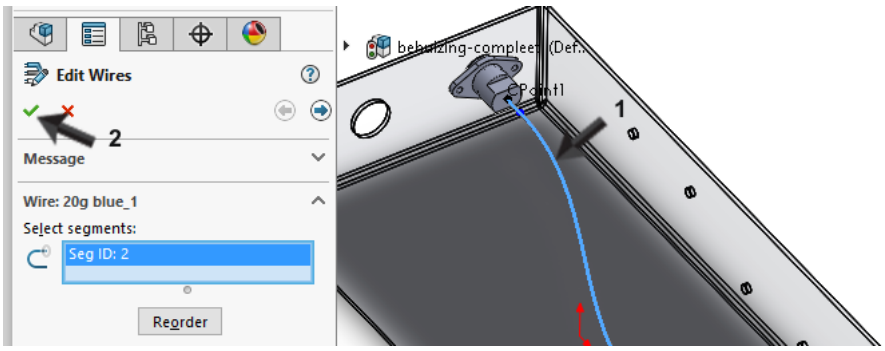
| **23** | In the menu that pops up, we add a blue and a red wire.<br>1. Double click on 20g blue.<br>2. Double click on 20g red.<br>Check if both wires are in the column 'Selected Wires'.<br>3. Click OK. |  |
| **24** | The wires now have tob e linked to the cable.<br>1. In the PropertyManager select the first wire (blue).<br>2. Click on Select Path. |  |
| **25** | 1. Select the cable we created earlier.<br>2. Click OK. |  |

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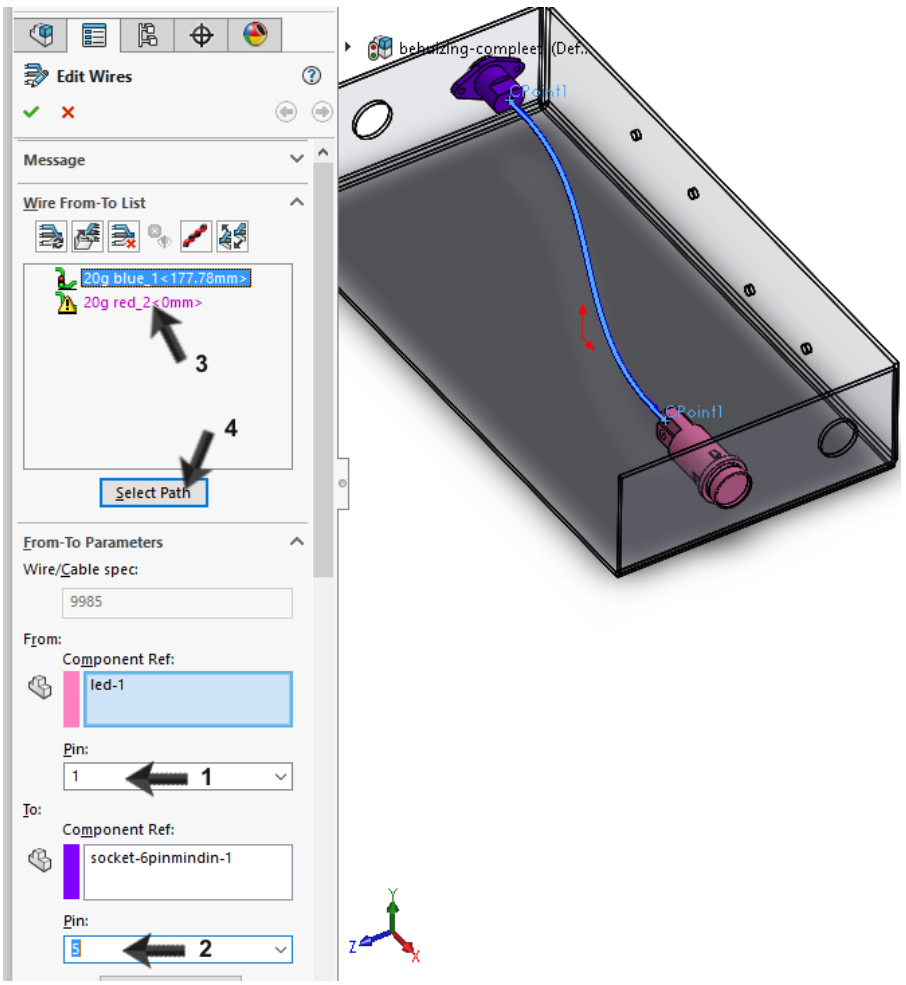
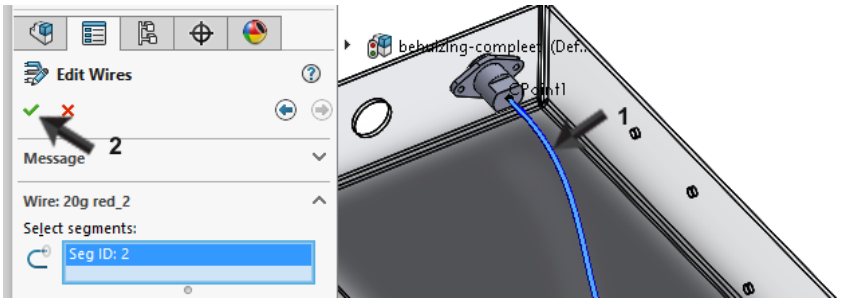
| **26** | In the PropertyManager, under From-To Parameters, you can connet the wires to a selected pin.<br>1. Select pin 1 of the LED.<br>2. Select pin 5 of the connector.<br>Now we connect the second wire.<br>3. Select the red wire.<br>4. Click on Select Path. |  |
| **27** | 1. Select the cable<br>2. Click OK |  |

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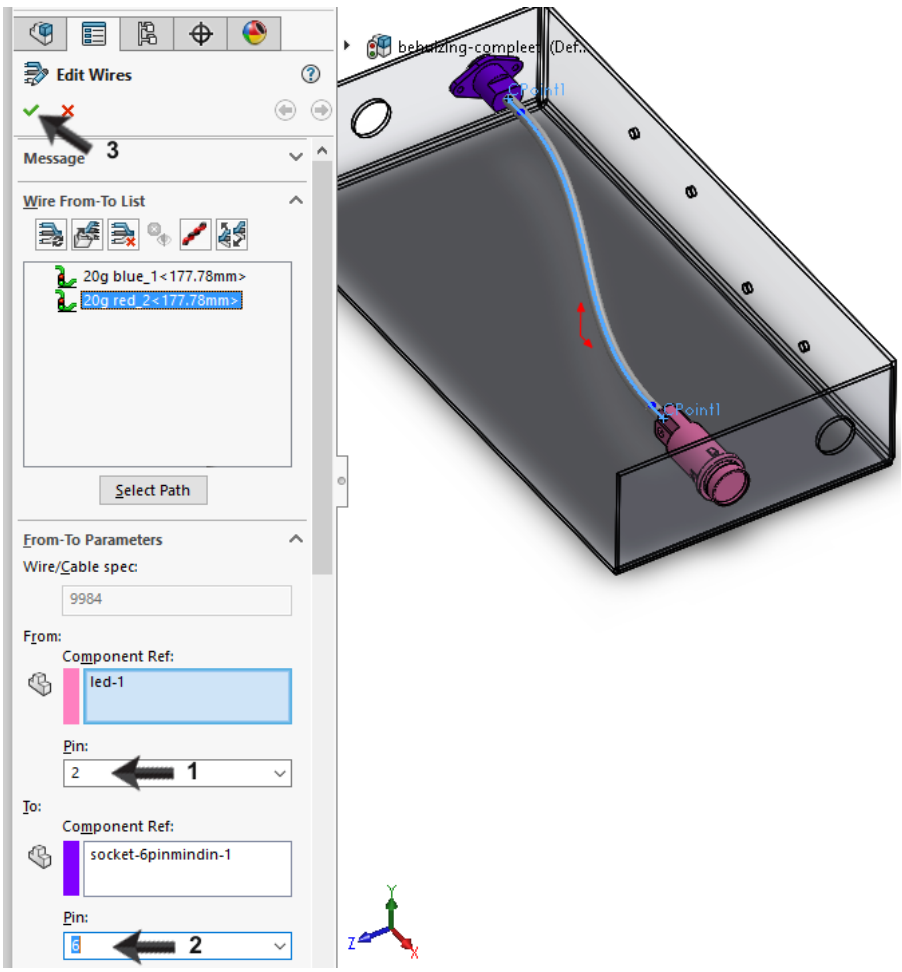
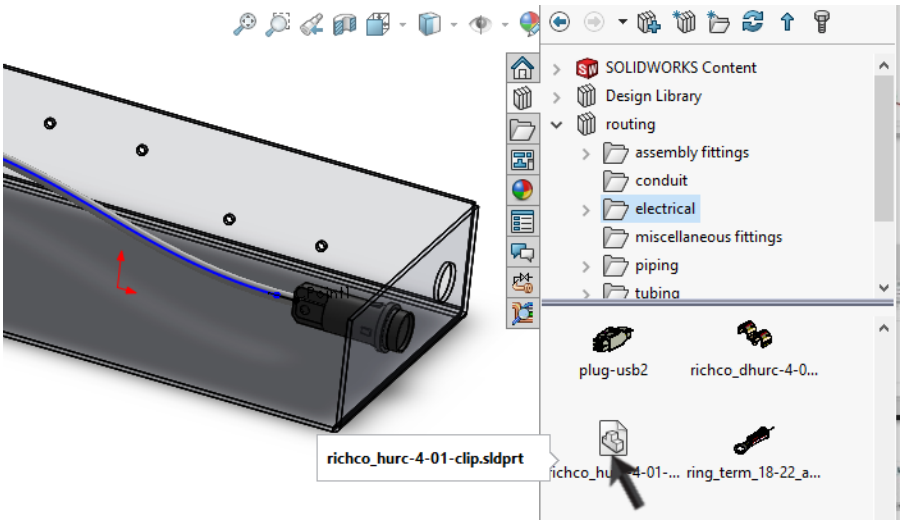
| **28** | Again the wire has to connect to the correct pins<br>1. Select pin 2 of the LED.<br>2. Select pin 6 of the connector.<br>3. Click OK. |  |
| **29** | Now we fix the cable with some clips.<br>In the Design Library select the clip named 'richco_hurc-4-01-clip' |  |

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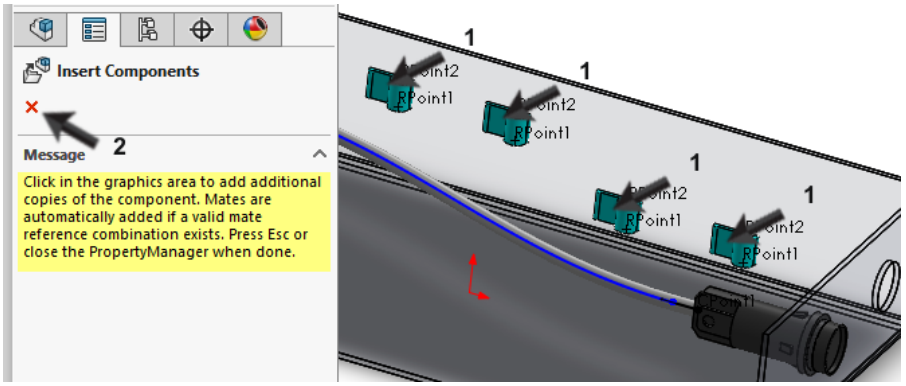
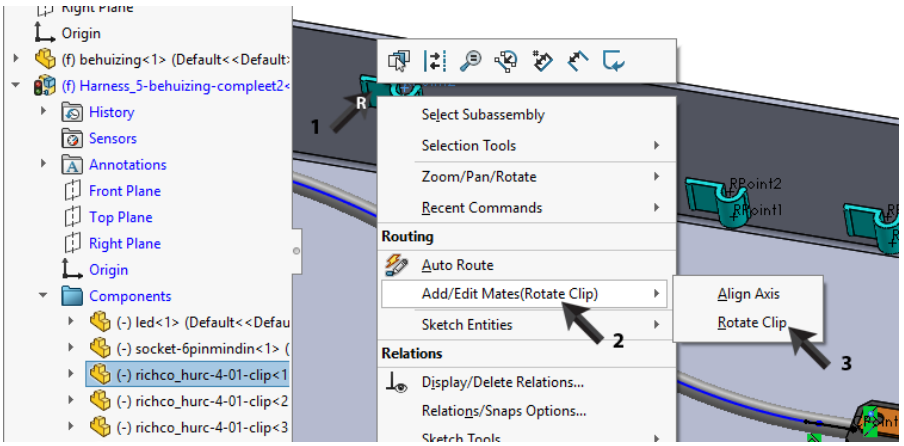
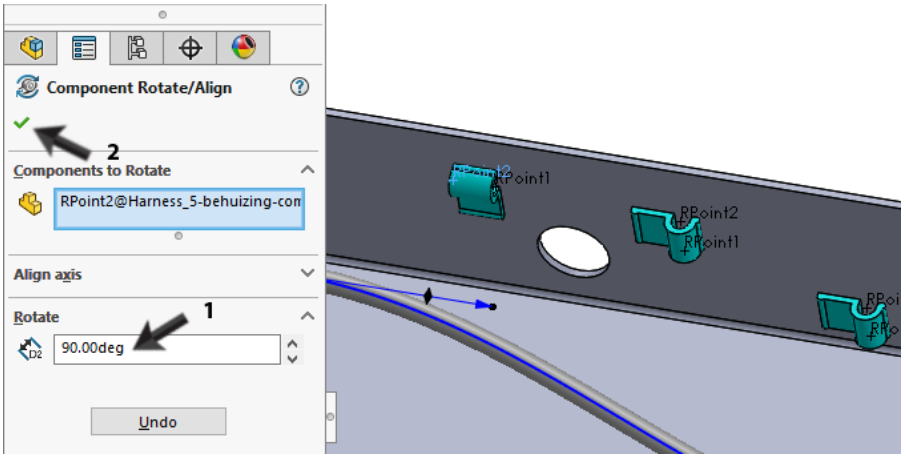
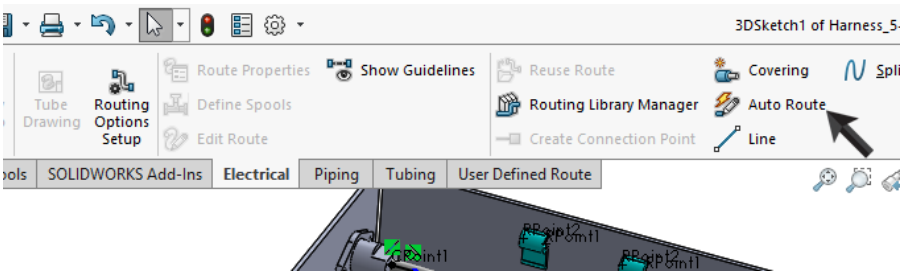
| **30** | 1. Drag the clip to the four holes<br>2. Click Cancel. |  |
| **31** | The clips now have tob e rotated upright.<br>1. Right click the first clip.<br>2. Pick Add/Edit Mates (Rotate Clip)<br>3. Click on Rotate Clip |  |
| **32** | 1. Set the angle at 90°<br>2. Click OK.<br>Repeat these steps for the other three clips. |  |
| **33** | In the CommandManager click on Auto Route. |  |

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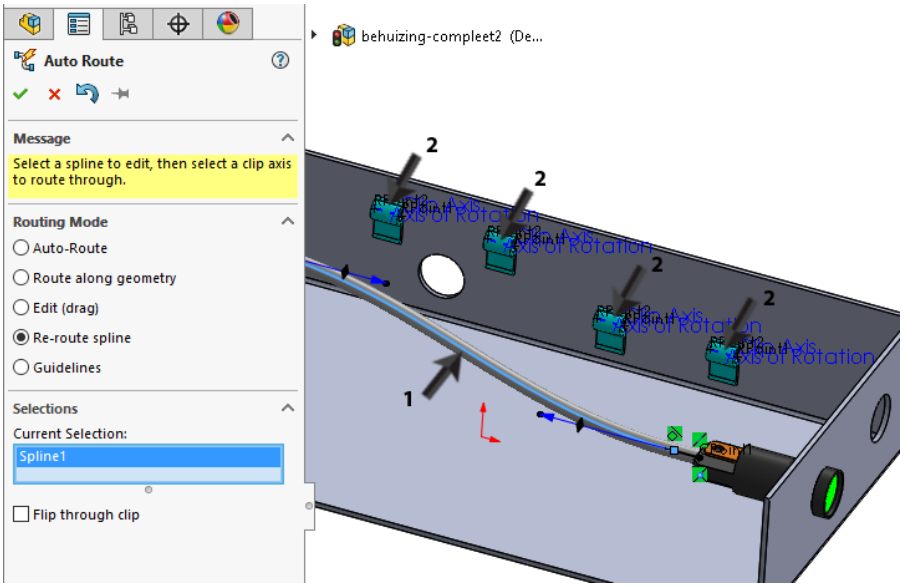
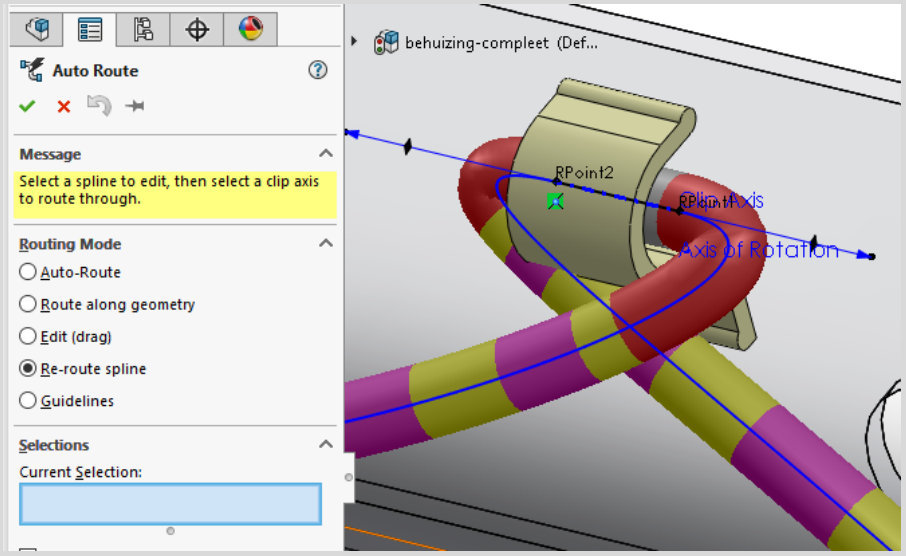
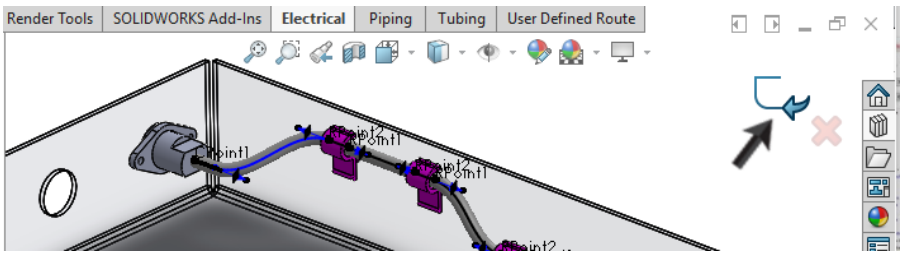
| **34** | 1. Select the cable<br>2. Select the four clips vier clips. It's best to select each clip by the center-line.<br>3. Click OK. |  |
| | **Tip!** | If something unintended happen when linking the cable to the clips, you'd best immediately click Undo in the PropertyManager and give it another try.<br>  <br>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. |
| **35** | Close the sketch of the cable |  |

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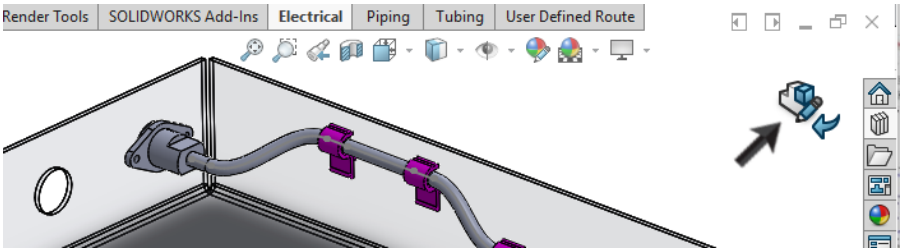
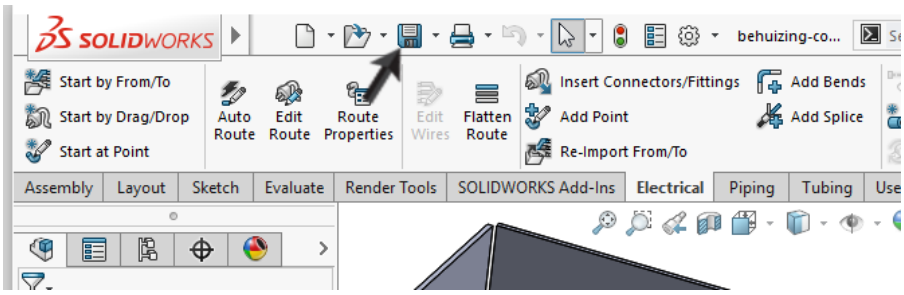
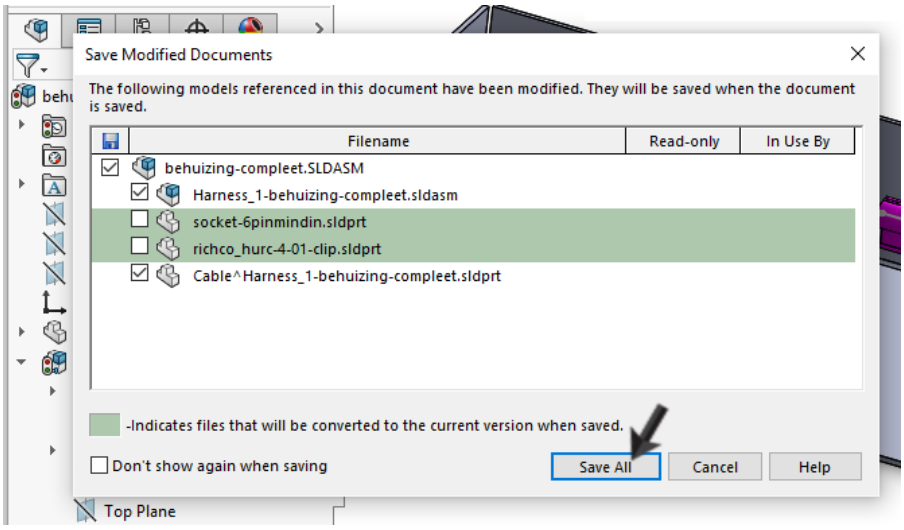
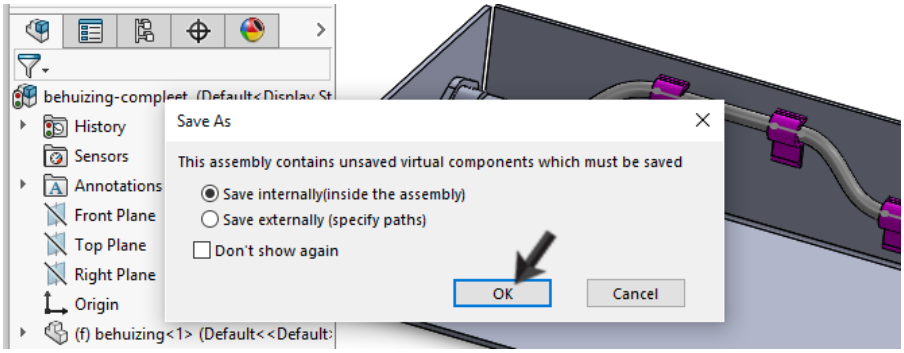
| **36** | Close the routing sub-assembly. |  |
| **37** | The first cable of the harness is ready now.<br>Save te file. |  |
| **38** | Click Save All-in the popup menu. |  |
| **39** | Click OK if you see the menu in the illustration. |  |
| | **Work plan** | At this point we have routed one cable in the housing, which directly connect the LED to the connector. Now we will add a second led this time with terminals. |

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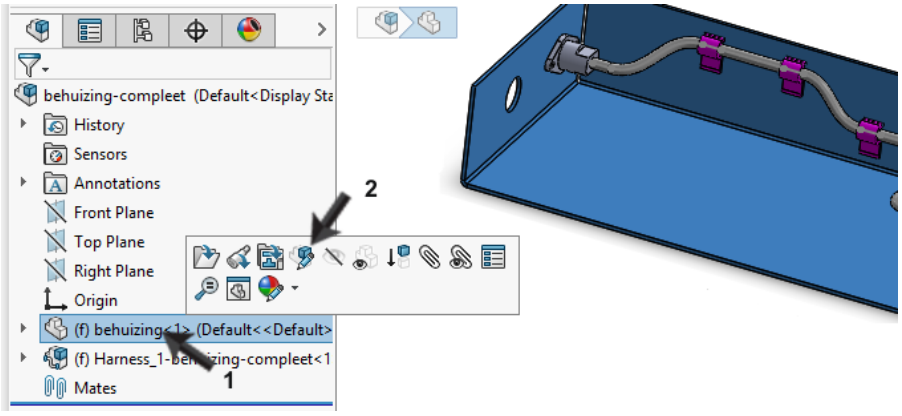
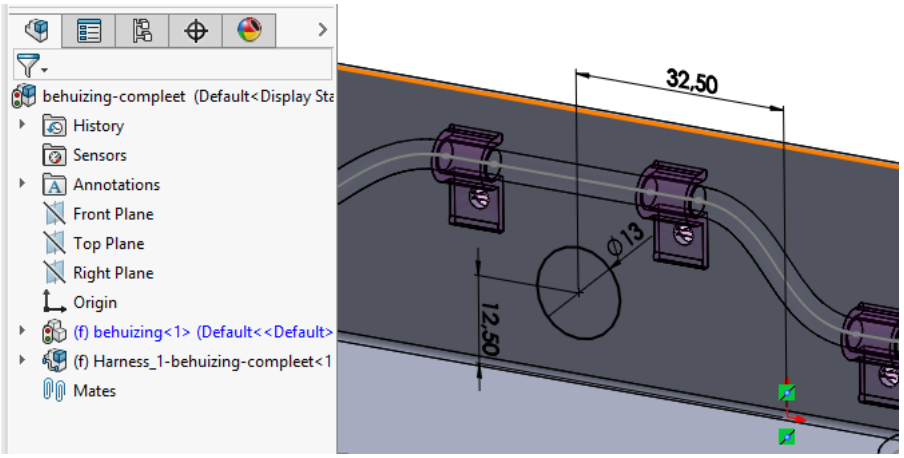
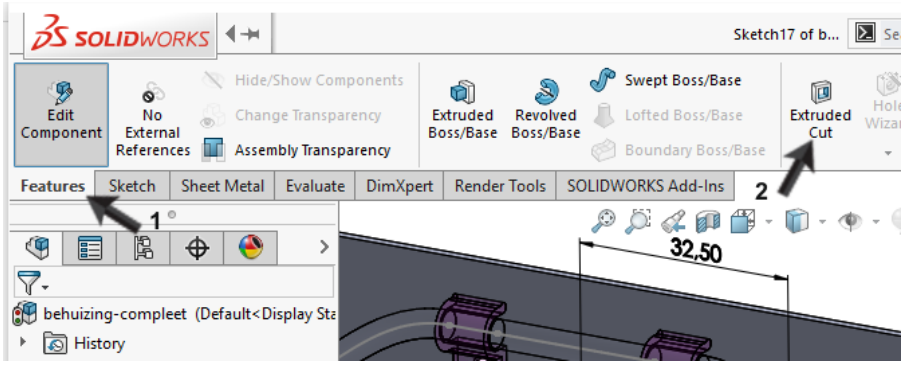
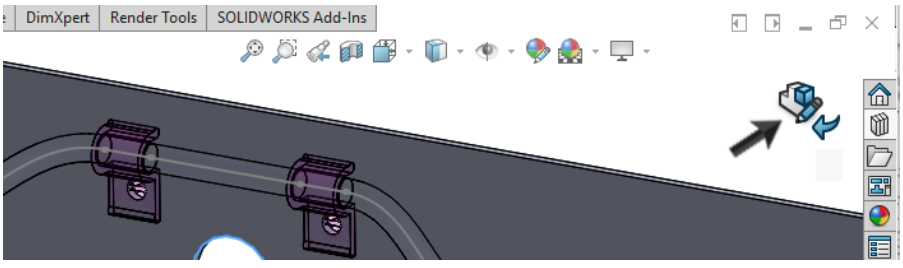
| **40** | First we make an extra hole in the housing.<br>1. Select the housing in the FeatureManager.<br>2. Click on Edit Part |  |
| **41** | Make a sketch as you can see in the illustration. |  |
| **42** | 1. In the CommandManager select the tab Features<br>2. Click on Extruded Cut.<br>Create the hole with depth Through All. |  |
| **43** | Exit In-context editing. |  |

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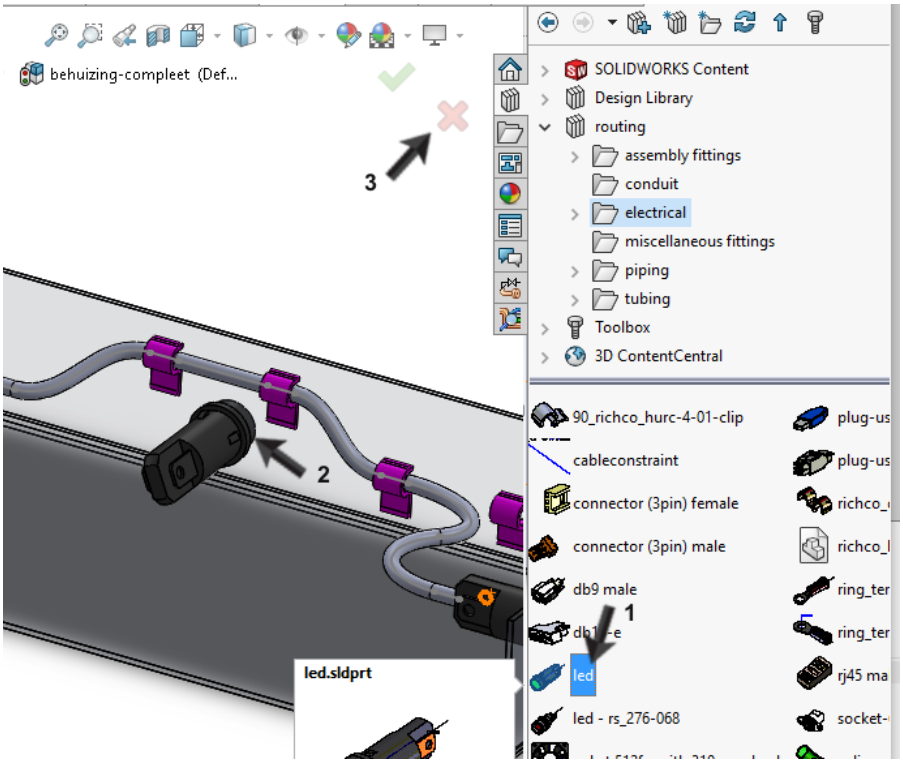
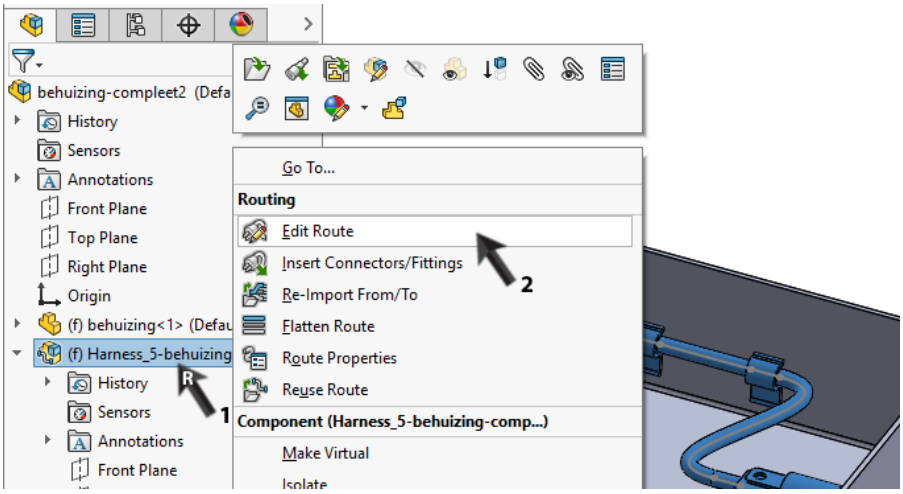
| **44** | 1. In the Design Library find the component 'led'.<br>2. Drag and drop this component to the hole you created.<br>3. In the PropertyManager click on **Cancel** |  |
| | **Tip!** | If at step 44 you would have clicked OK, a second harness would have been opened.<br>In this case we don't want the led to be a component of harness, but just a regular part in the assembly. We achieve this by clicking Cancel.<br>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. |
| **45** | 1. In the FeatureManager right click the subas-sembly of the harness.<br>2. Click on 'Edit Route'.<br>The existing harness, we created earlier, will be re-opened. |  |

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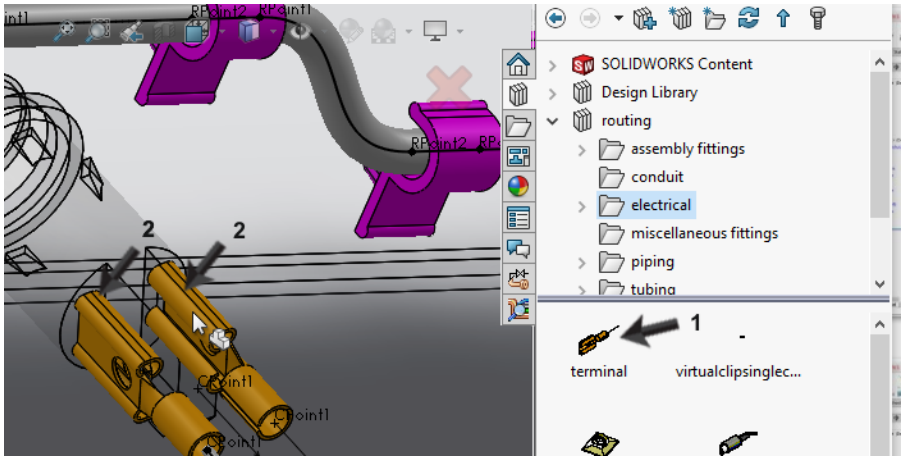
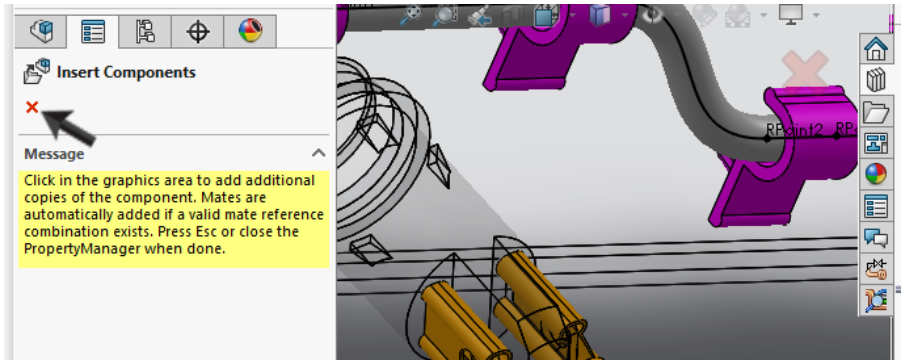
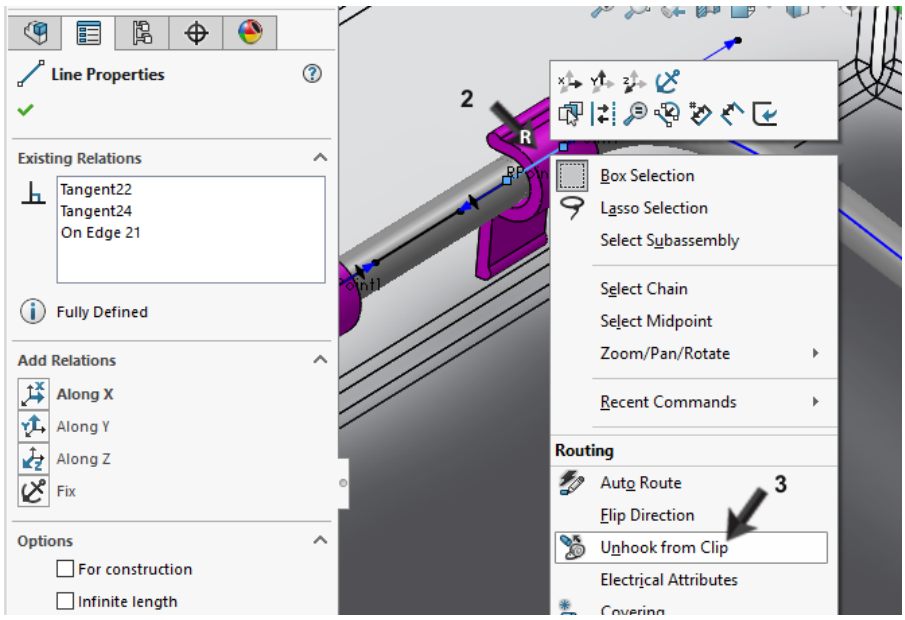
| **46** | 1. In the Design Library find the component 'terminal'.<br>2. Drag and drop this component twice to the LED. |  |
| **47** | Click Cancel to stop adding terminals. |  |
| **48** | Now we have to modify the cable. First we have to un-hook the cable from the rightmost clip. This gives us some room to branch-off the cable.<br>1. Zoom in to the right-most clip.<br>2. Richt click the line of the cable under the clip<br>3. Select Unhook from Clip |  |

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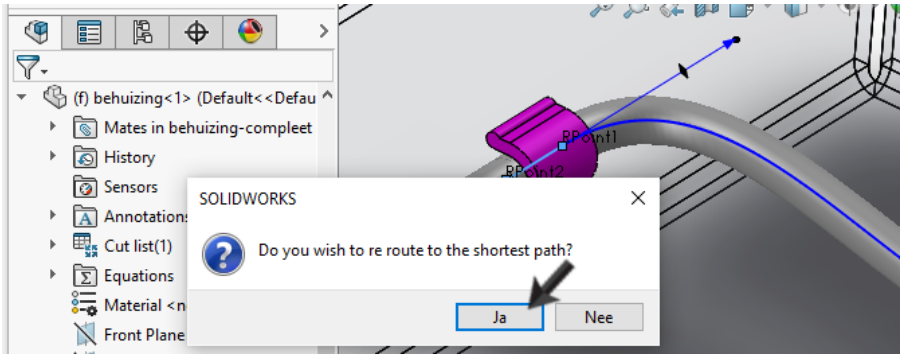
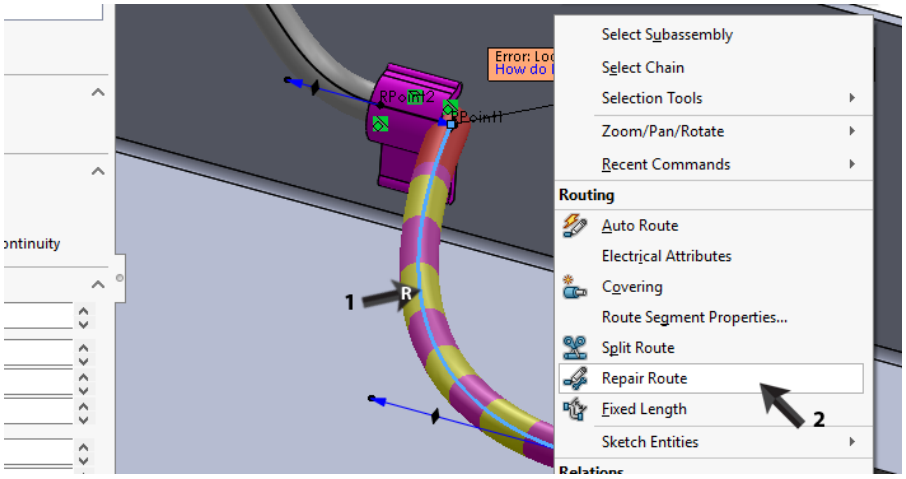
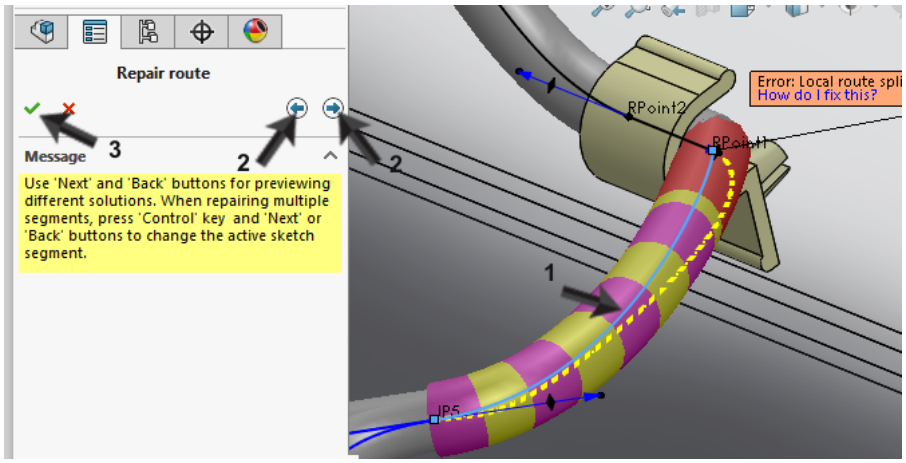
| **49** | Select Yes to recalculate the path of the cable. |  |
| | **Tip!** | In the CommandManager you'll also find a button 'Unhook from Clip'. It you use thatone, you will not get the option t oreroute the cable (step 49), and you will have to shape it manually (step 52 and further) |
| **50** | SOLIDWORKS now might show an errorif the cable is bent to shaply<br><br>Right click the cable and choose Repair Route |  |
| **51** | 1. Select the cable near the area that causes the problem.<br><br>With a yellow line, SOLIDWORKS will suggest a solution.<br><br>2. Would you like to see other solutions? Click Next or Previous in the PropertyManager until you see the desired curve.<br><br>3. Click OK. |  |
| | **Tip!** | 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 wil get some idea of how toe dit them. |

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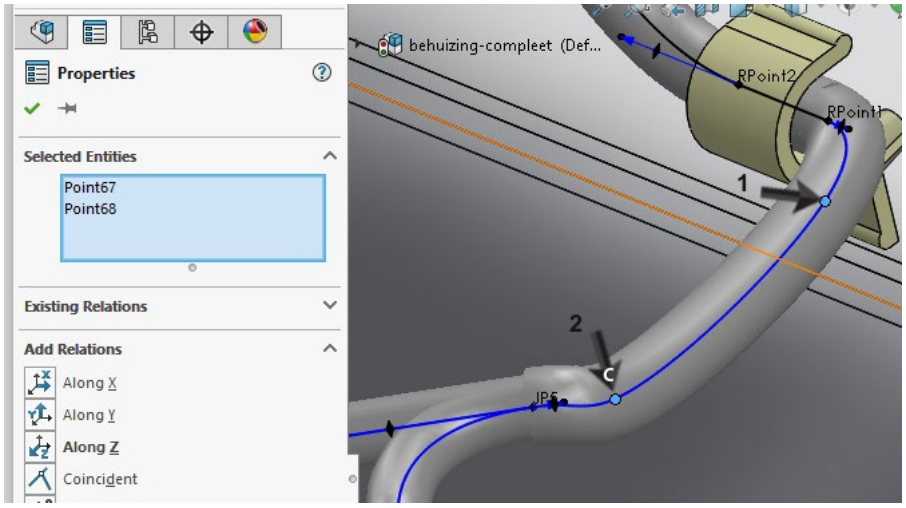
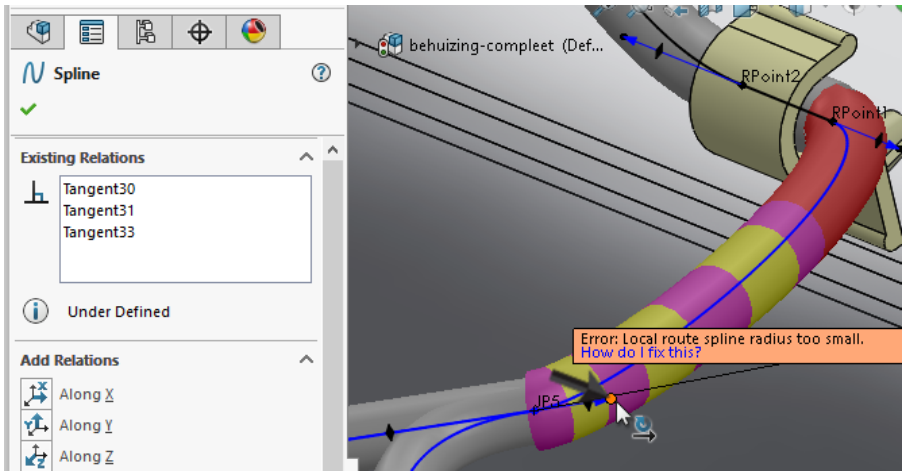
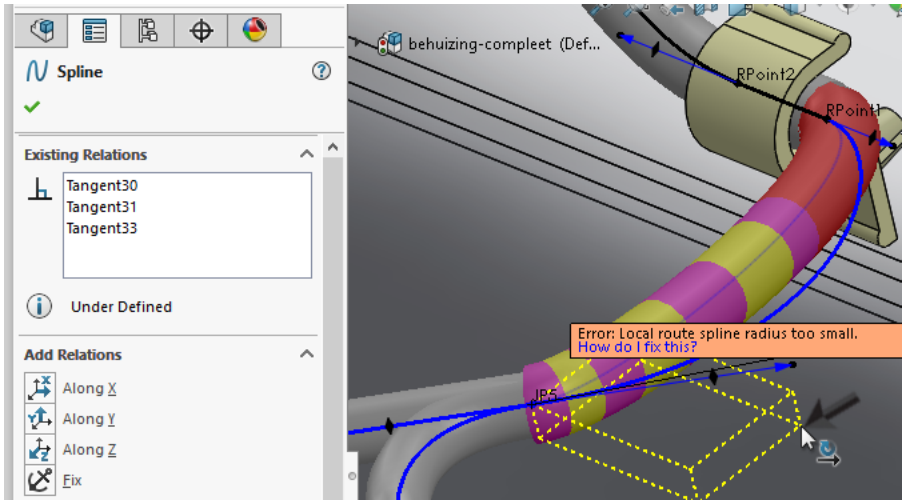
| **52** | We now will manually modify the shape of the cable (spline)<br><br>On the spline you can see control points. We don't need these anymore.<br><br>Select the points (use the <Ctrl>-key to select both).<br><br>Press <Delete><br><br>NOTE: the points can be in a completely different location than you see in the illustration! |  |
| **53** | Again the cable is bended to sharply. This time we will solve this manually.<br><br>At each control point you can see two arrows with a little circle at the and.<br><br>Select the circle as you can see here. |  |
| **54** | Dragt he circle tot he left.<br><br>Watch the shape of the spline change as you do so. Release the mouse button when the shape looks fine.<br><br>The error message now will have disapeared. |  |
| | **Tip!** | You'll get the smoothest curves when you use as few control points as possible. That is why we removed som of the control points in step 52.<br><br>With the arrows (step 53-54) you can control how sharp the curve is bended.<br><br>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 |

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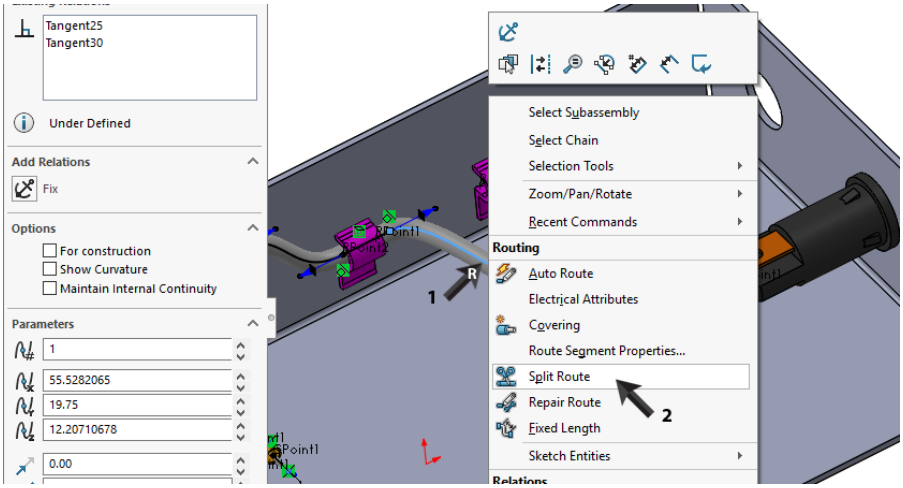
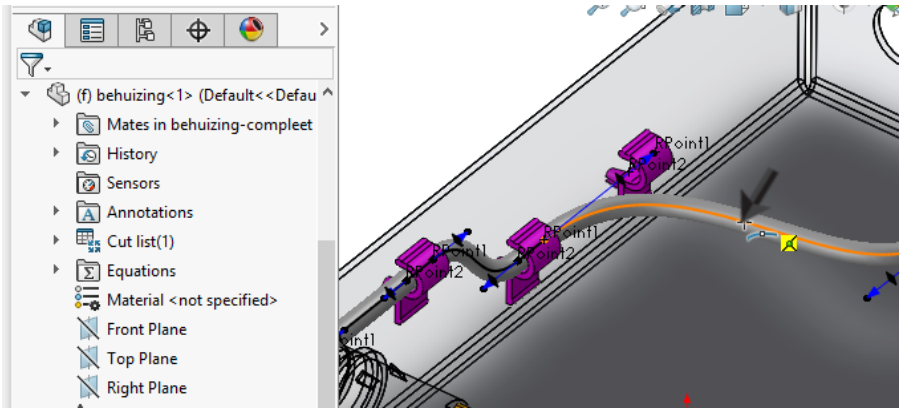
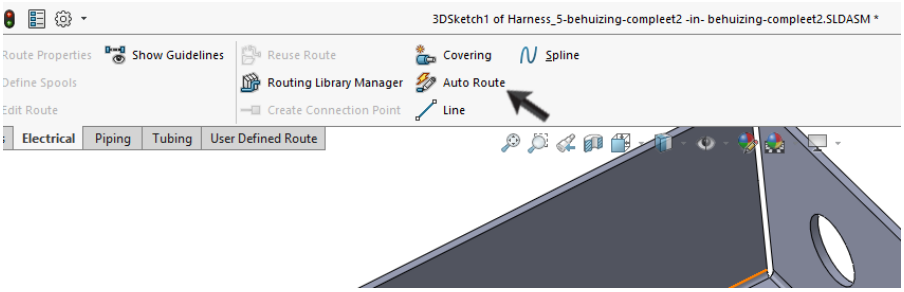
| | | 'Insert Spline Point'. Then you can drag that point to the right location, use relations, or use the arrows at the control point to modify the shape.<br>Give it a try! |
| **55** | Now we will connect the led.<br>First we branch-off the existing cable.<br>Right click the cable and choose Split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

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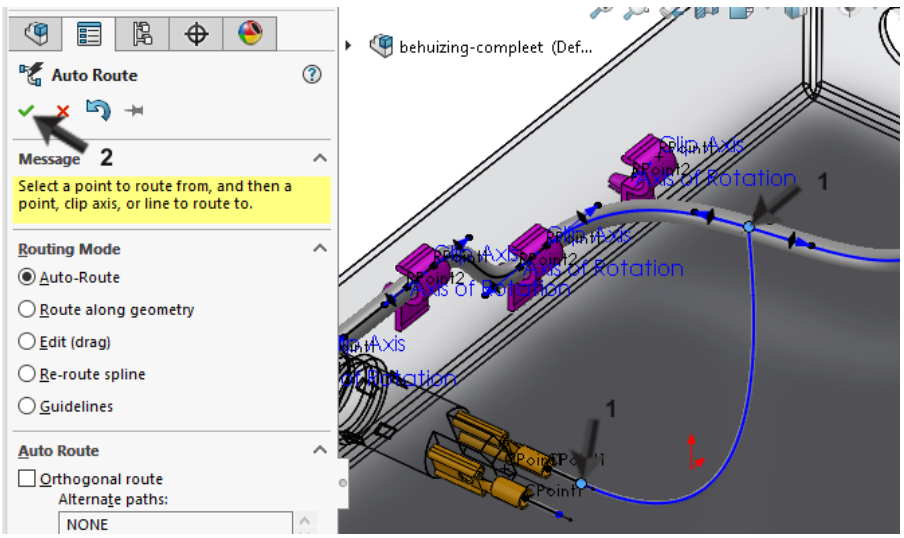
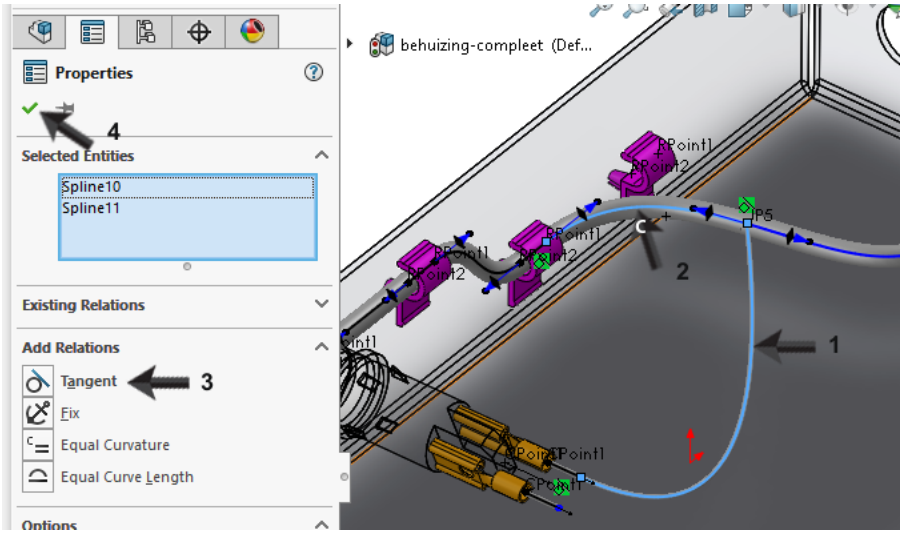
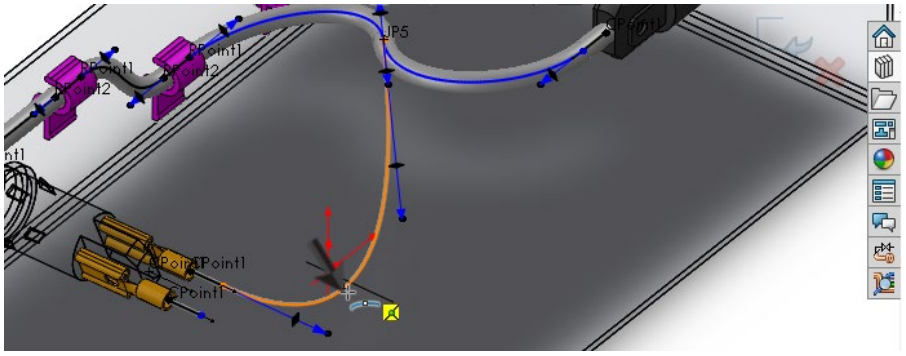
| **58** | 1. Select the endpoint of the first terminal and then the point where you have split the existing cable.<br>2. Click OK. |  |
| **59** | 1. Select the curve you've just drawn<br>2. Press and hold the <Ctrl>-key and select the curve as you can see in the illustration<br>3. In the PropertyManager select Tangent.<br>4. Cick OK<br>There is no sharp bend between the old and the new cable now. |  |
| **60** | Now we create a second branch-off<br>1. Right click on the cable and choose Split Route.<br>2. Click on the new cable as you can see in the illustration.<br>3. Press <Esc>. |  |

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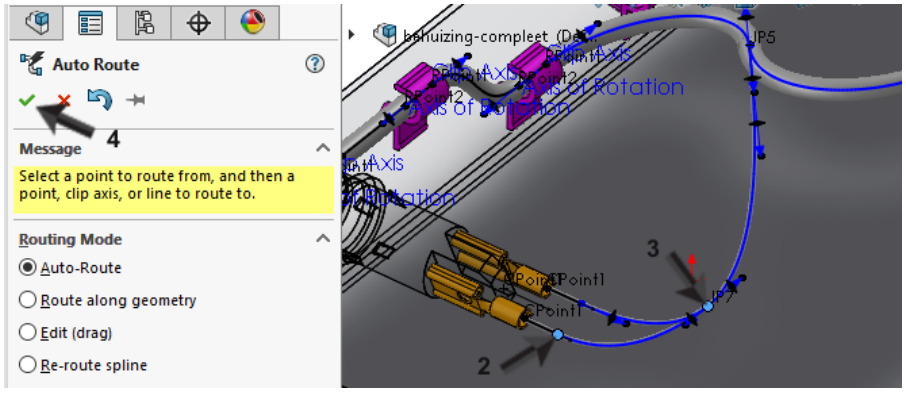
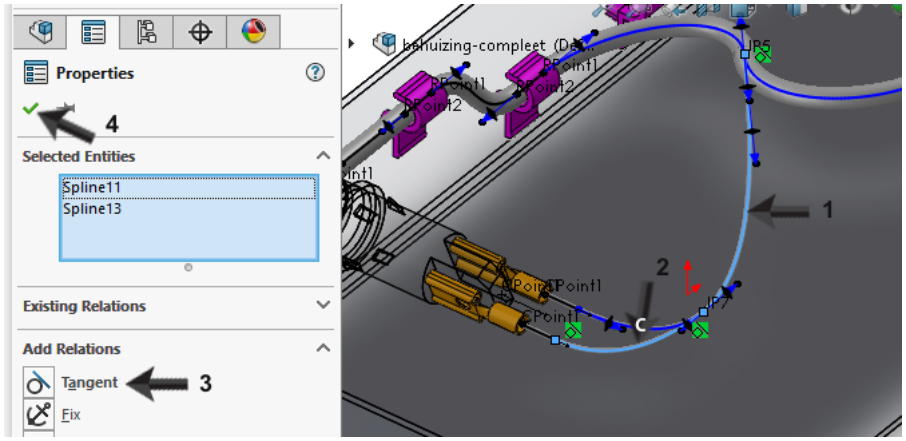
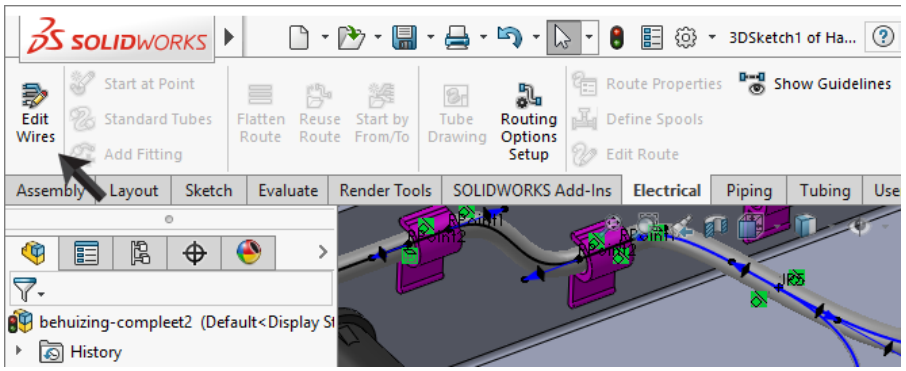
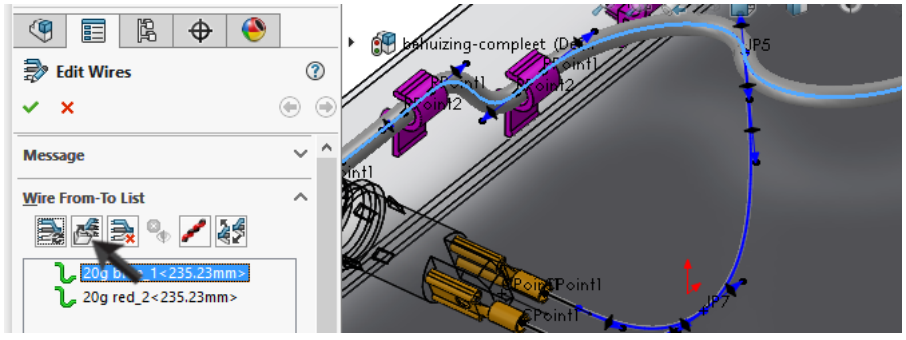
| **61** | 1. In the CommandManager, click Auto Route<br>2. Select the endpoint of the terminal<br>3. Select the point you just added to the cable.<br>4. Click OK. |  |
| **62** | 1. Select the newly created cable.<br>2. Select (hold down the <Ctrl>-key) the old cable.<br>3. In the CommandManager select Tangent.<br>4. Click OK. |  |
| **63** | All cables are now there, the next step is to add wires.<br>In the CommandManager click on Edit Wires.<br>In the PropertyManager you will see the two wires we created previously. |  |
| **64** | Click on Add Wire. |  |

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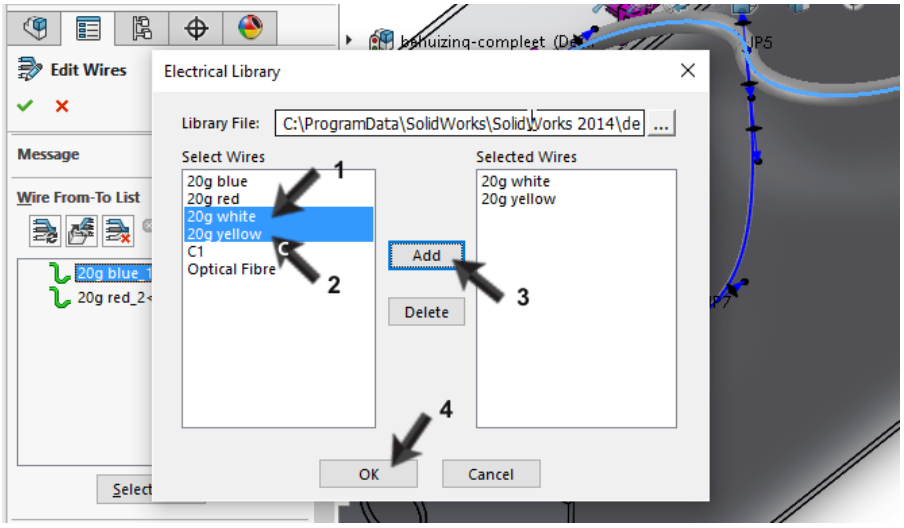
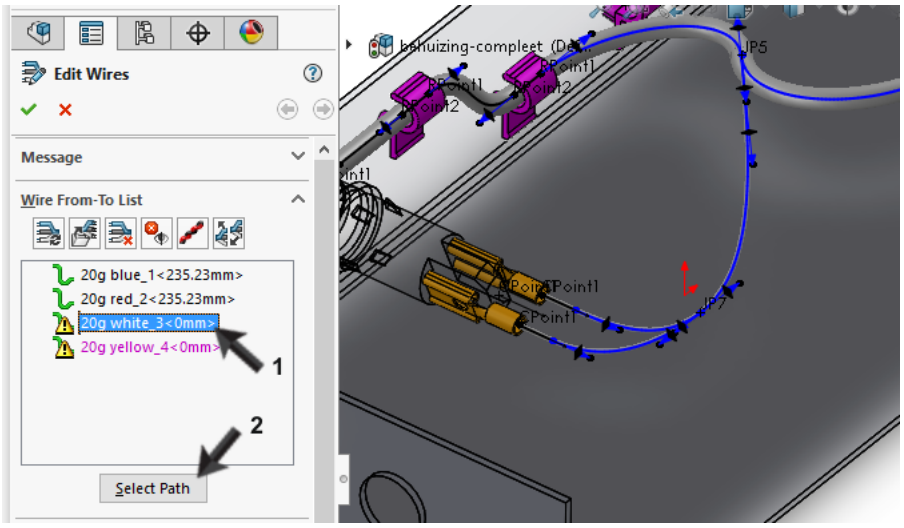
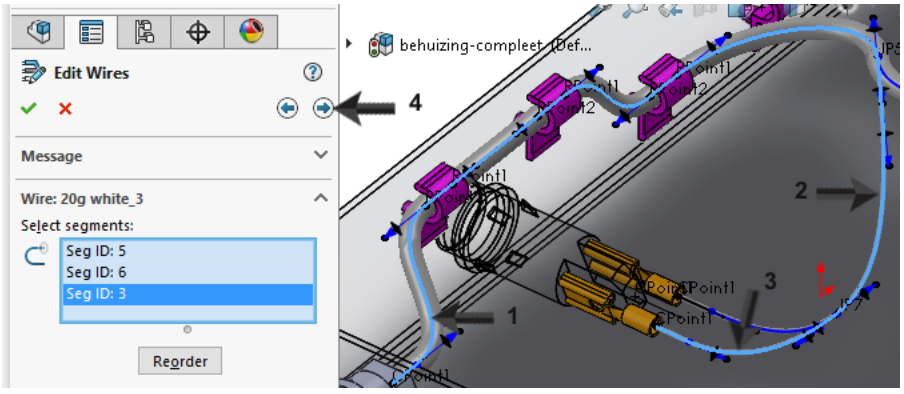
| **65** | 1. Select the white wire.<br>2. Select (hold down the <Ctrl>-key) the yellow wire.<br>3. Click on Add.<br>4. Click OK. |  |
| **66** | 1. Select the white wire in the PropertyManag-er<br>2. Click on Select Path. |  |
| **67** | 1-3 Select the parts of the cable, that connects the connector with the first terminal<br>4. Click on Next. |  |

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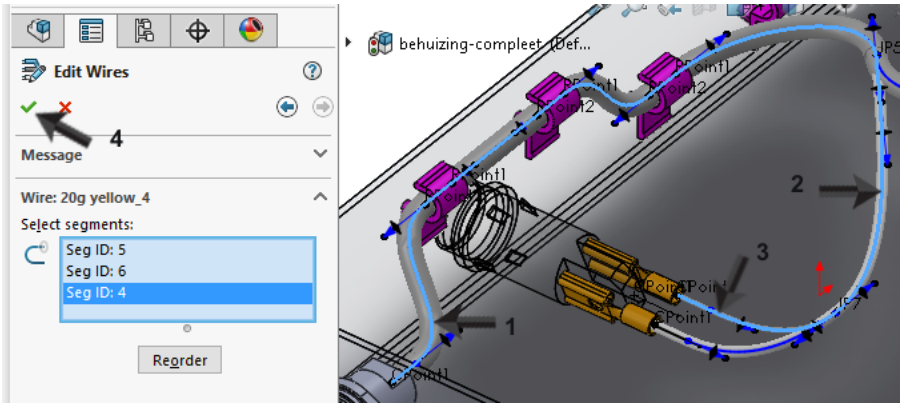
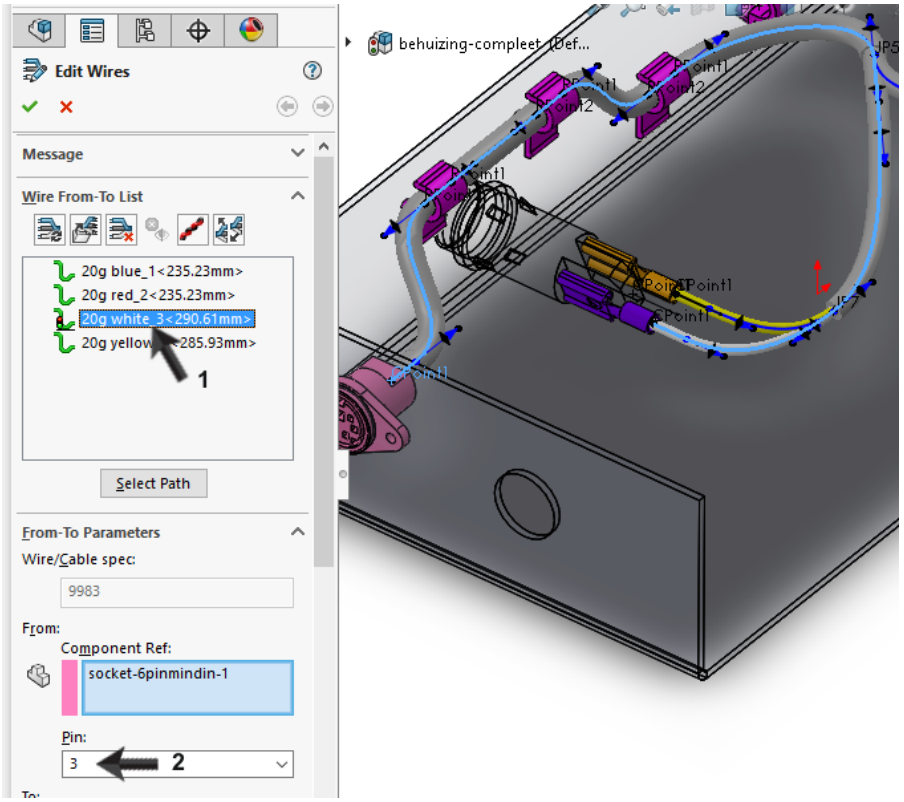
| **68** | 1-3 Select the parts of the cable, that connects the connector with the second terminal<br>4. Click OK. |  |
| **69** | 1. Select the white wire in the PropertyManag-er.<br>2. Link it to pin 3 of the connector. |  |

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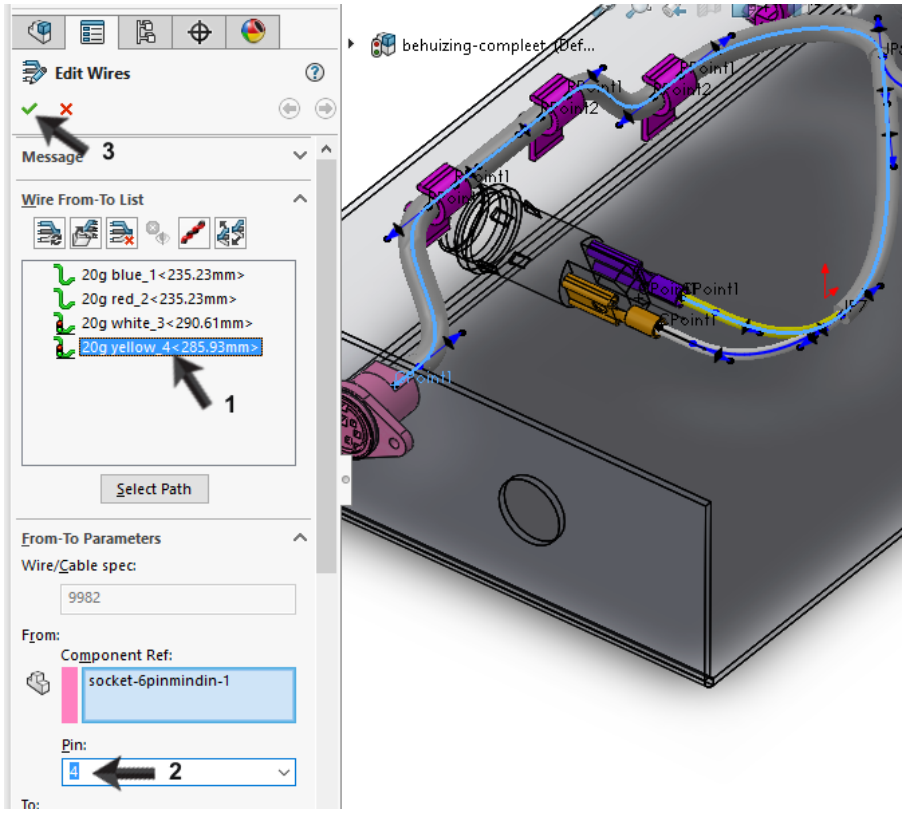
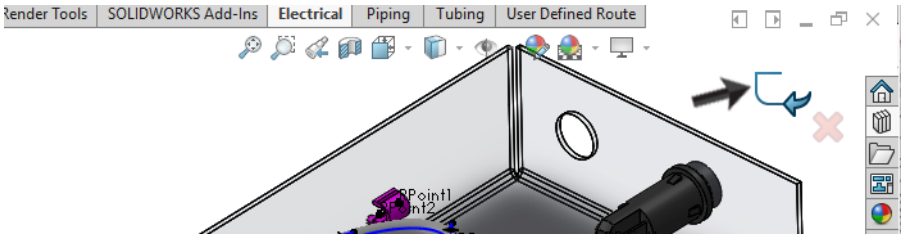
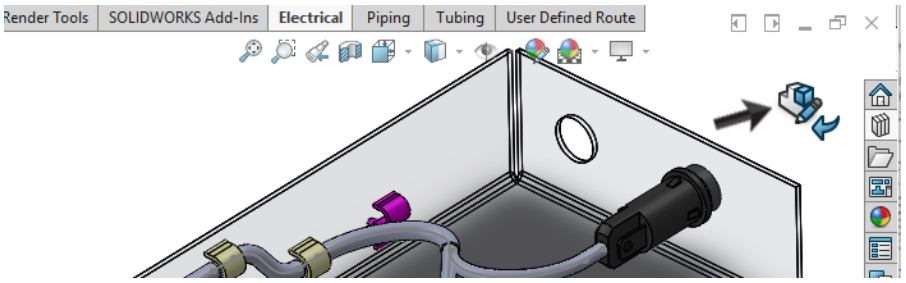
| **70** | 1. Select the yellow wire in the PropertyManager.<br>2. Link it to pin 4 of the connector.<br>3. Click OK. |  |
| **71** | Close the sketch of the cable. |  |
| **72** | Close the sub-assembly of the harness. |  |
| **73** | The harness is ready now. Save all files. | |

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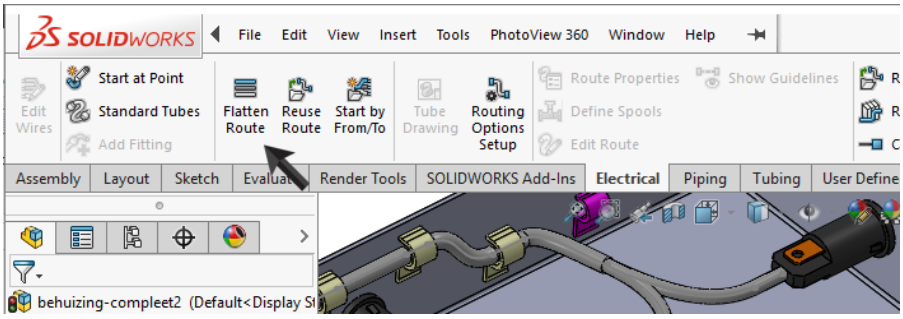
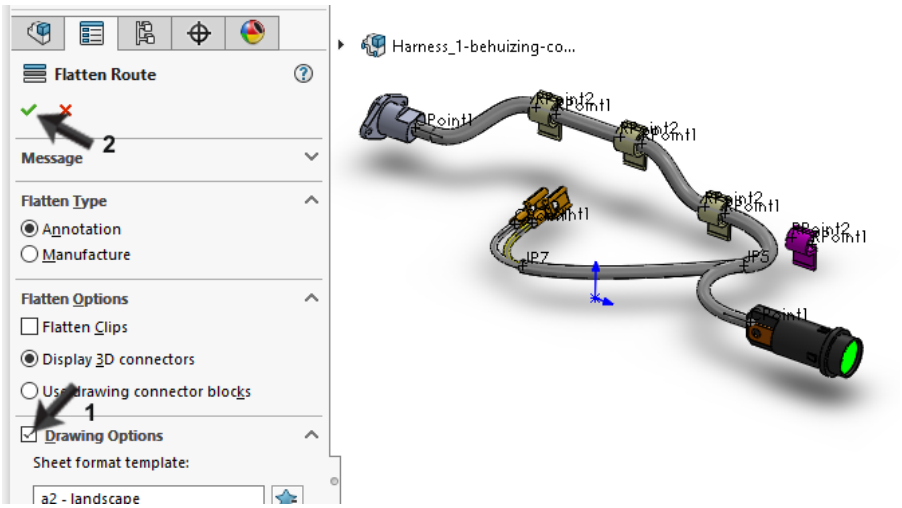
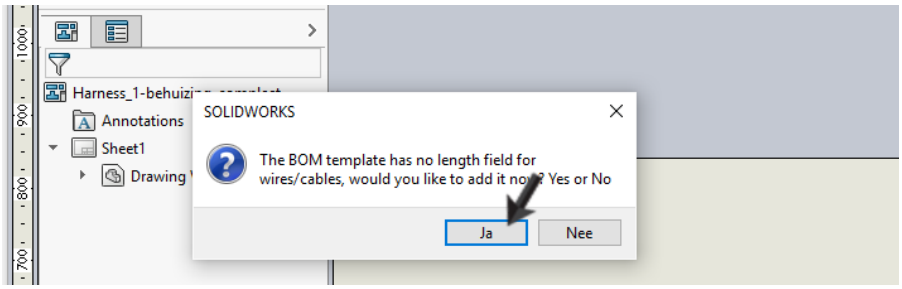
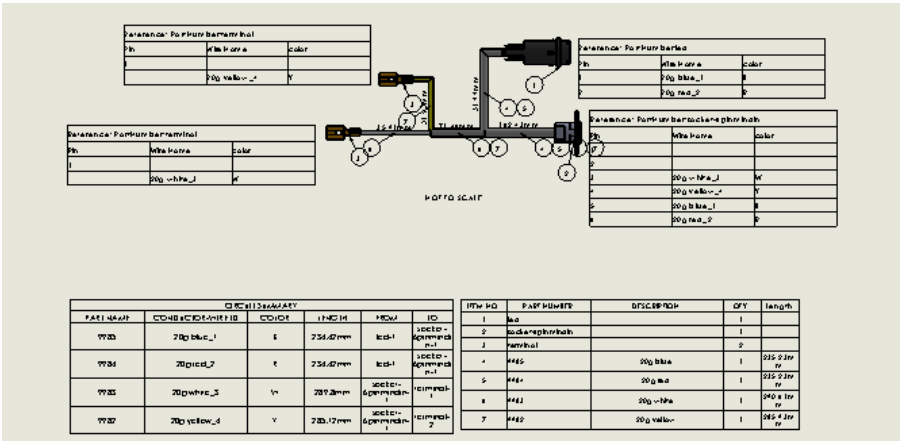
| **74** | Next we will make a schematic drawing of the harness.<br>In the CommandManager click on Flatten Route. |  |
| **75** | 1. In the PropertyManager check Drawing Options.<br>2. Click OK. |  |
| **76** | To add the length of the wires to the BOM, click Yes. |  |
| **77** | The drawing is now created, with the bill of materials and a wiring scheme<br>Save this drawing. |  |
| | **Tip!** | We have now seen how you can make a Harness. You coiuld also make a counduit, 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, |

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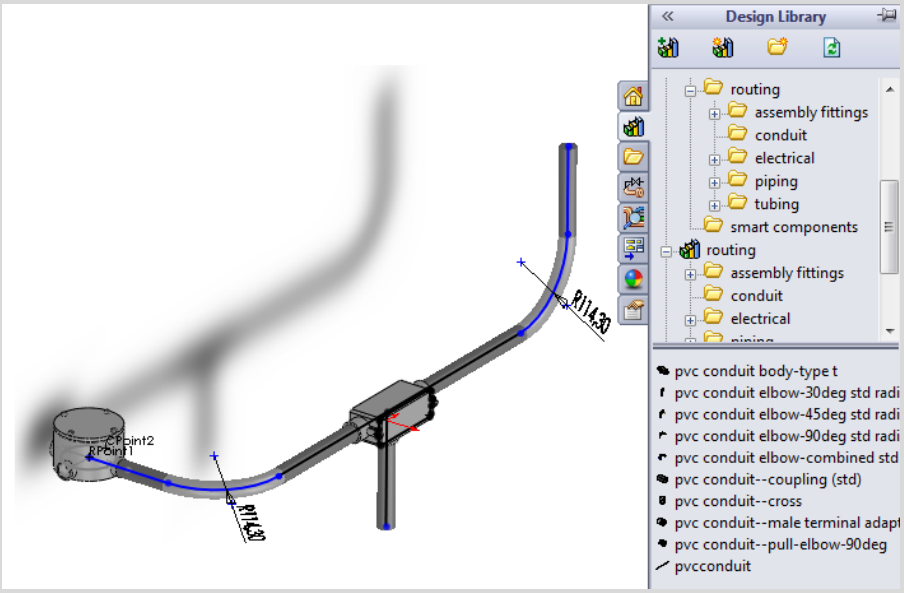
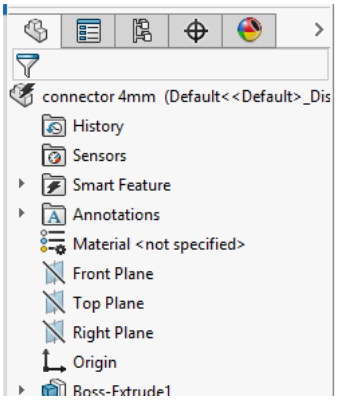
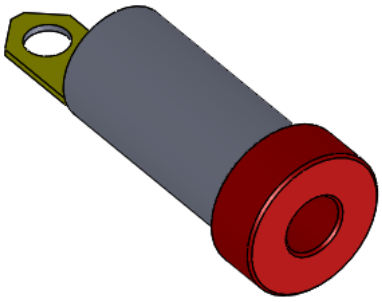
| | | as we did before, form the folder 'electrical'.<br>  |
| | **Tip!** | The number of components available in the Design Library is very limited. You can not do much more with it than a simple exercise like we did in this tutorial. There are two ways to get more components.<br>1. You can search for librarries on the internet, or 3D ContentCentral.<br>2. You could also define your own components. In the next steps we will show you how to add components to the Design Library. |
| **78** | First, model the component you want to use.<br>In this example we'll use the connector you see in the illustration. You can download this part from www.SOLIDWORKS.nl |  <br>  |

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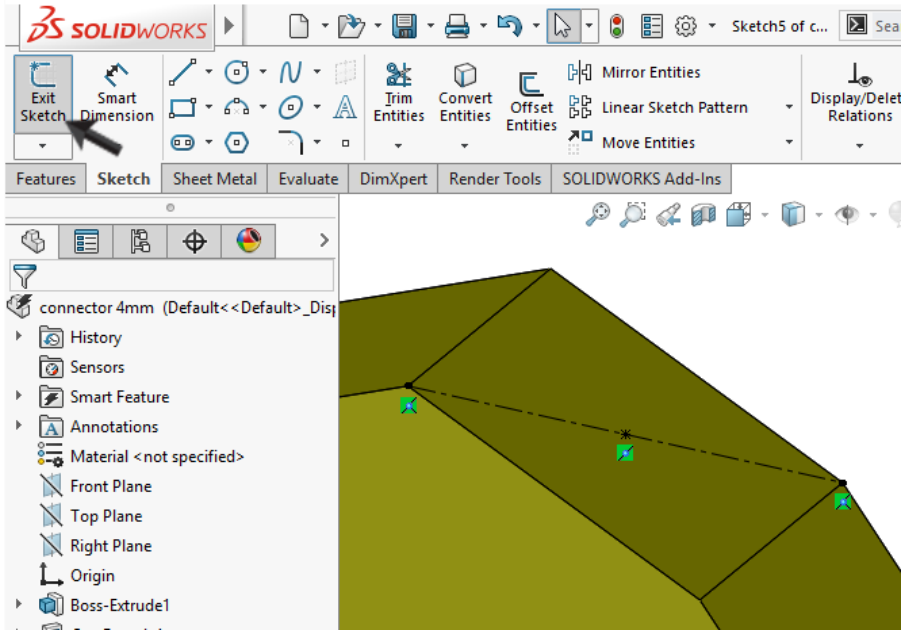
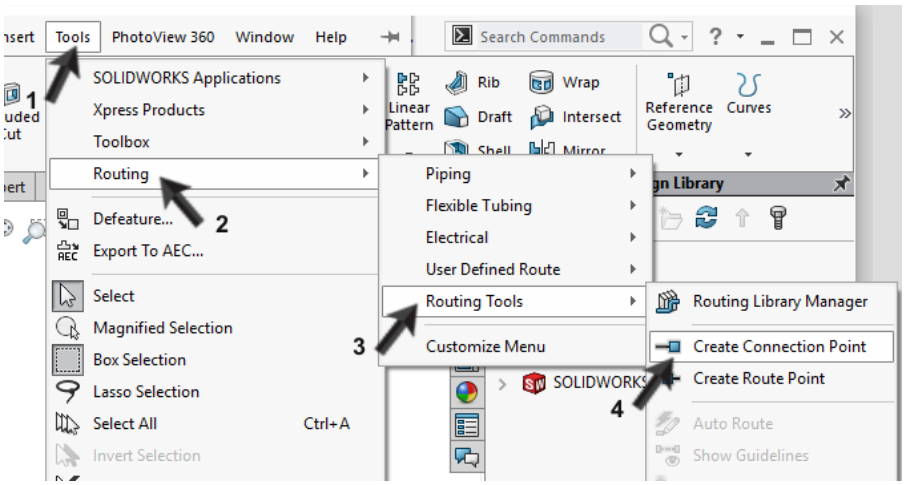
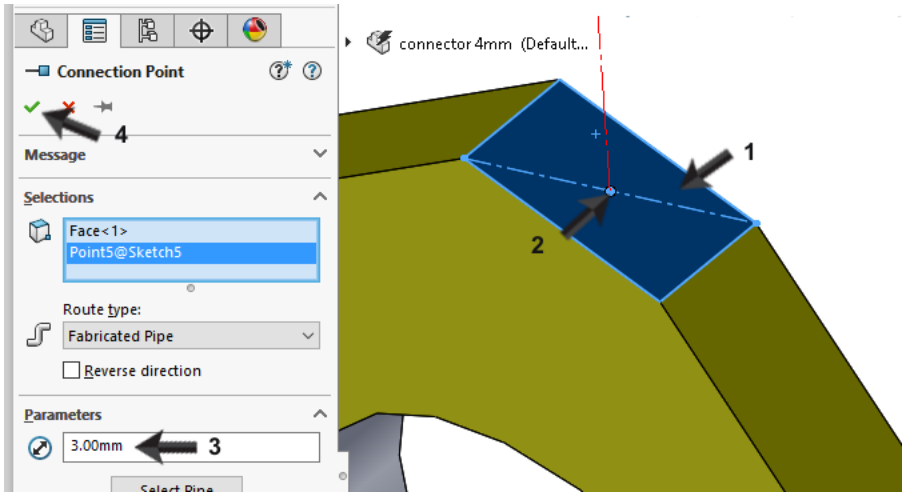
| **79** | In the model we have to define a connection point.<br><br>Make the sketch you see in the illustration, consisting of a construction line and a point on its midpoint.<br><br>Close de sketch. |  |
| **80** | 1. In the main menu select Tools<br>2. Click on Routing<br>3. Click on Routing Tools<br>4. Click on Create Connection Point |  |
| **81** | 1. Select the face on which the connection point is located<br>2. Select the point in the sketch you made earlier<br>3. Enter the maximum diameter of the cable that can be used for this part<br>4. Click OK. |  |

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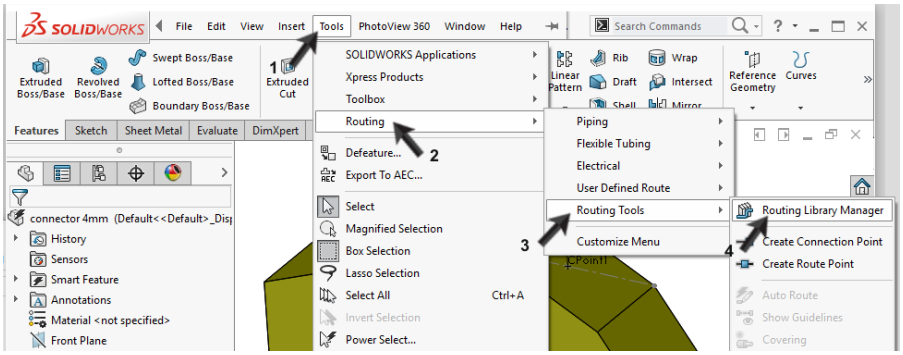
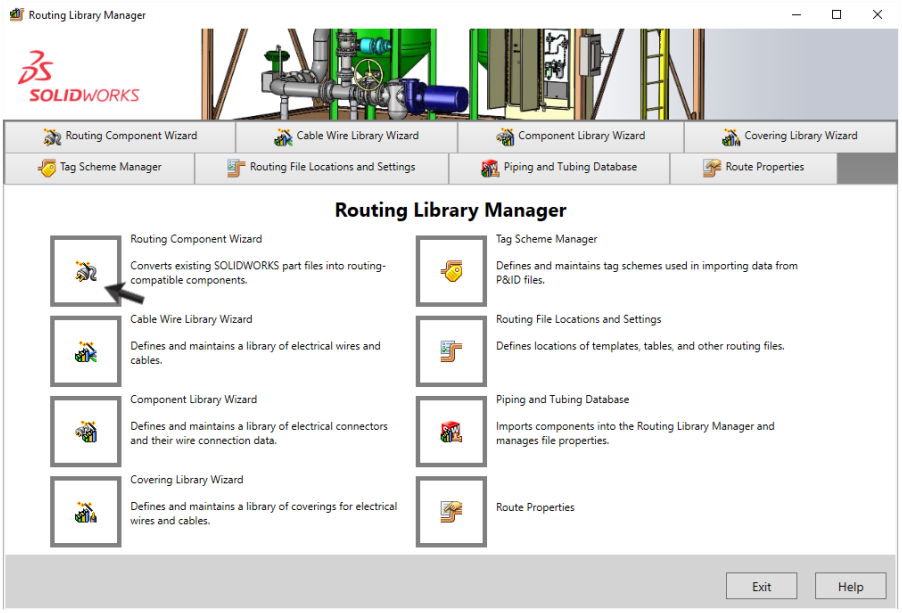
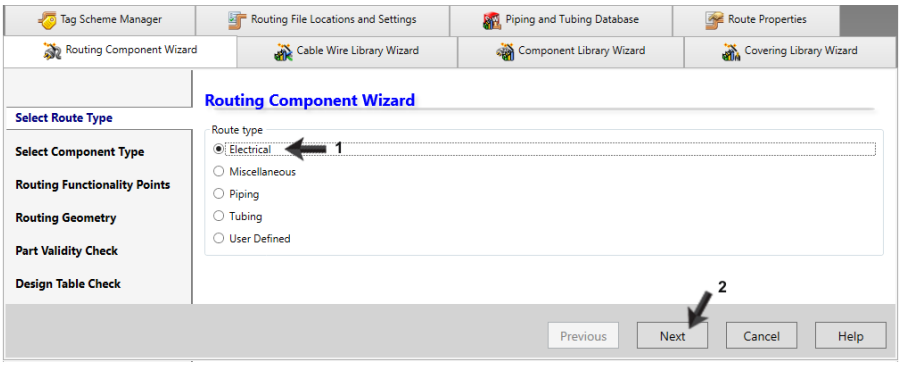
| **82** | To add this part to the Design Library we use a wizard.<br>1. In the mail menu select Tools<br>2. Click on Routing<br>3. Click on Routing Tools<br>4. Select Routing Library Manager |  |
| **83** | A new window pops up that will guide you through the process<br>Select the first option: Routing Component Wizard<br>We'll walk through a number of steps. |  |
| **84** | In the screen 'Select Route Type' click Next. |  |

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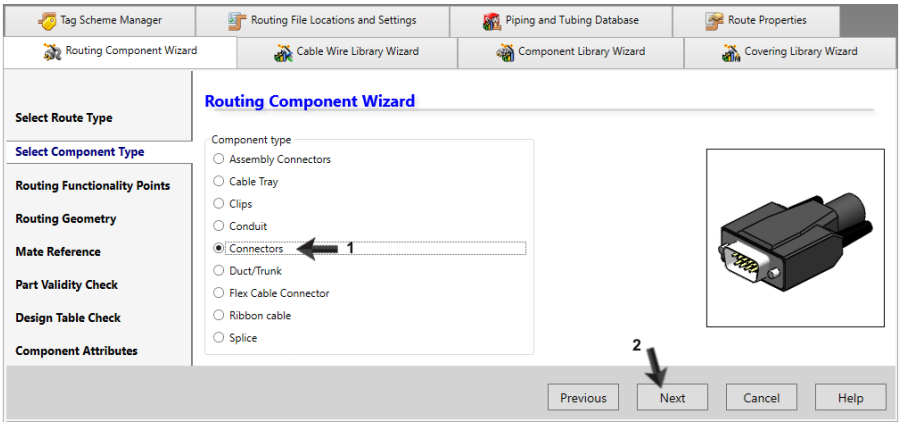
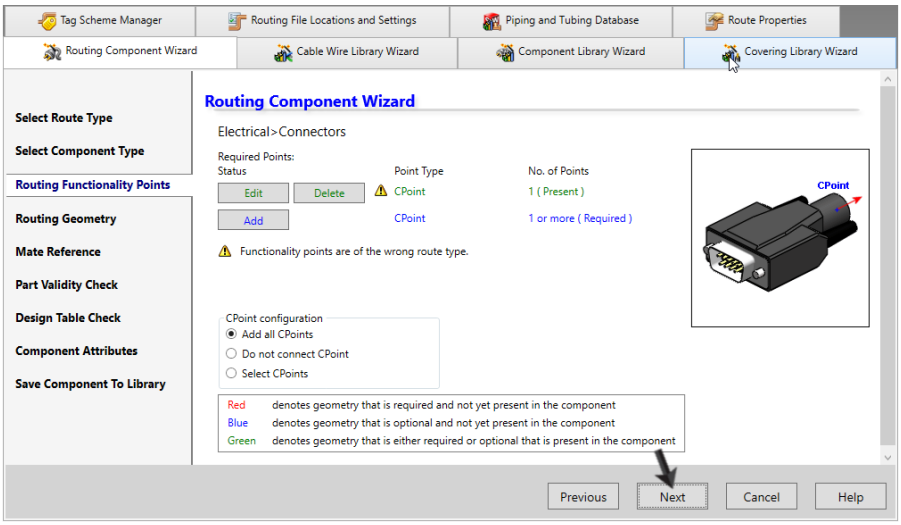
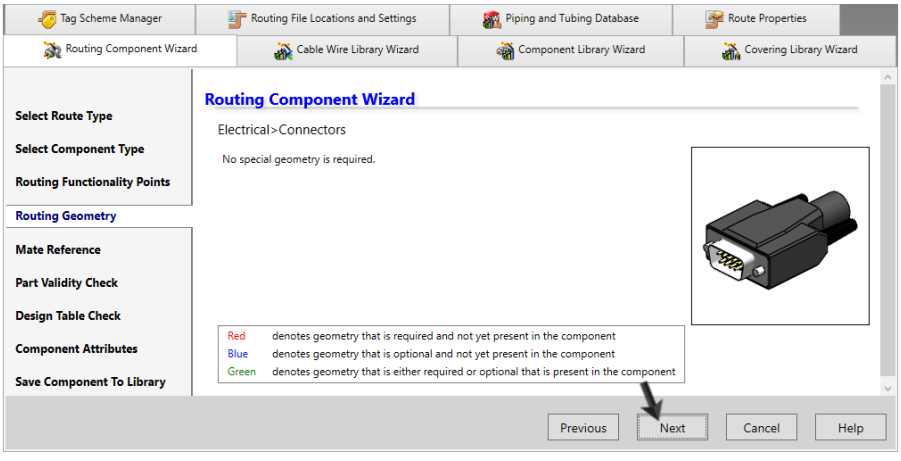
| **85** | 1. For component type chose: Connector.<br>2. Click Next. |  |
| **86** | We already defined the Connection Point.<br>Click Next |  |
| **87** | Click Next |  |

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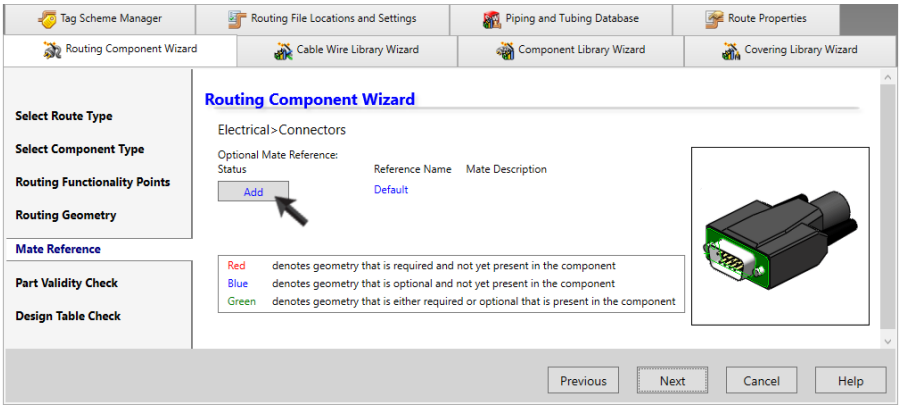
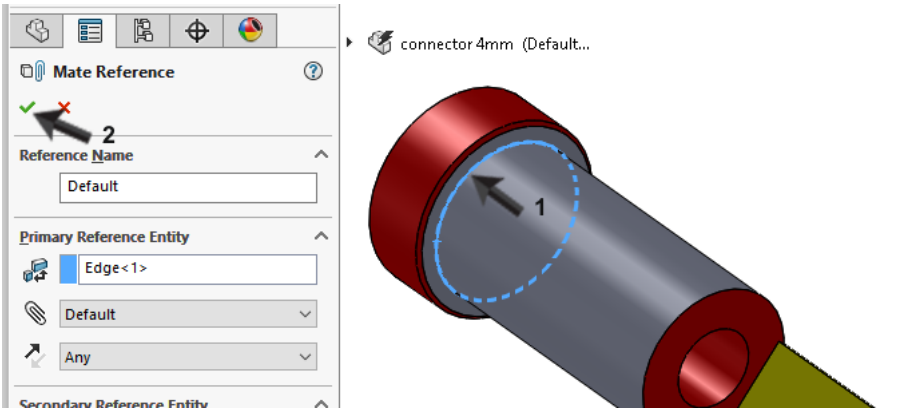
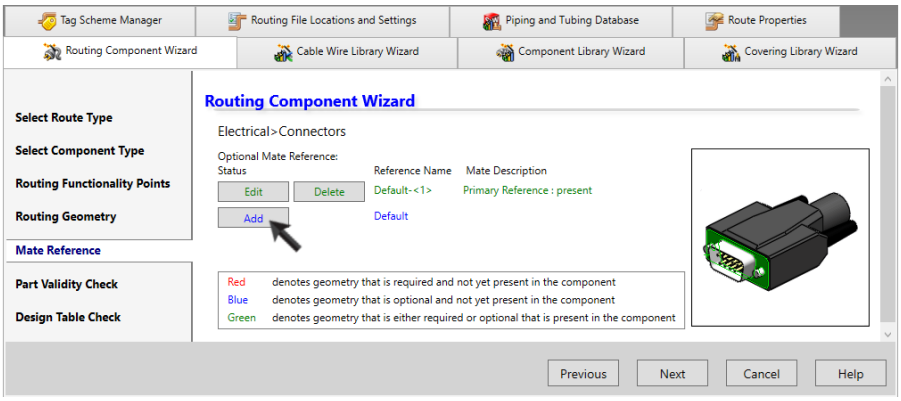
| **88** | Now we have to add mates, so that the component will snap into the right position<br><br>Click on Add. |  |
| **89** | The SOLIDWORKS window will re-appear now.<br><br>1. Select the edge that will align with the hole later on.<br>2. Click OK. |  |
| **90** | In theRouting component wizard click Add again. |  |

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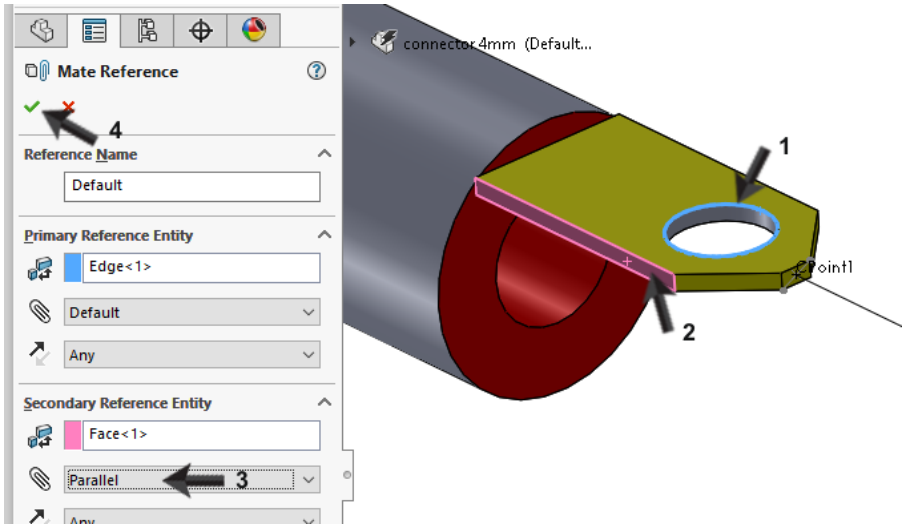
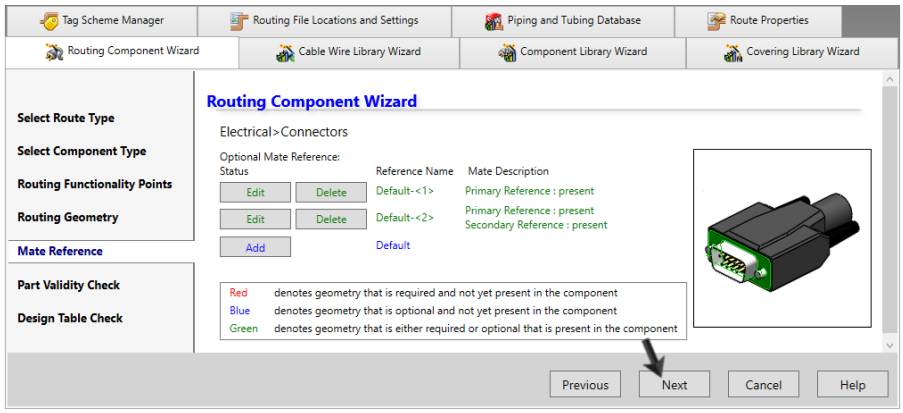
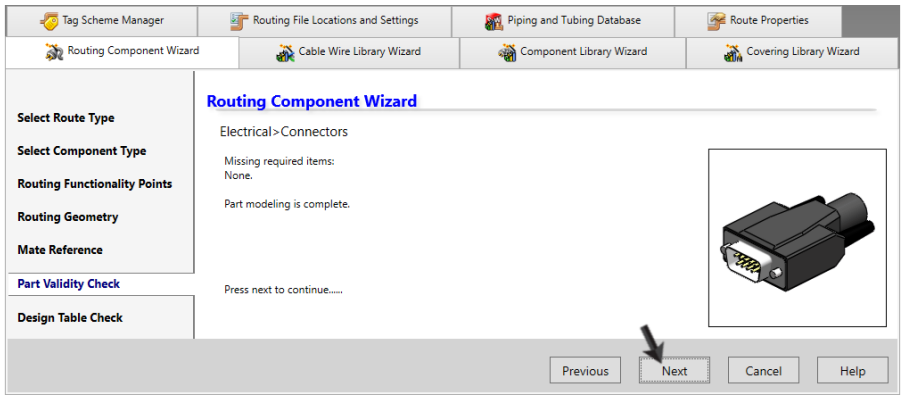
| **91** | Select the mates that will make the terminal snap into the right position:<br>1. Select the edge of the hole in the connection strip<br>2. Selecteer the side face of this strip. Make sure that this face is listed in the PropertyManager under Secundary Reference Entity<br>3. For this mate, select: parallel<br>4. Click OK. |  |
| **92** | In the Routing Component Wizard click Next. |  |
| **93** | All data will be checked.<br>Click Next. |  |

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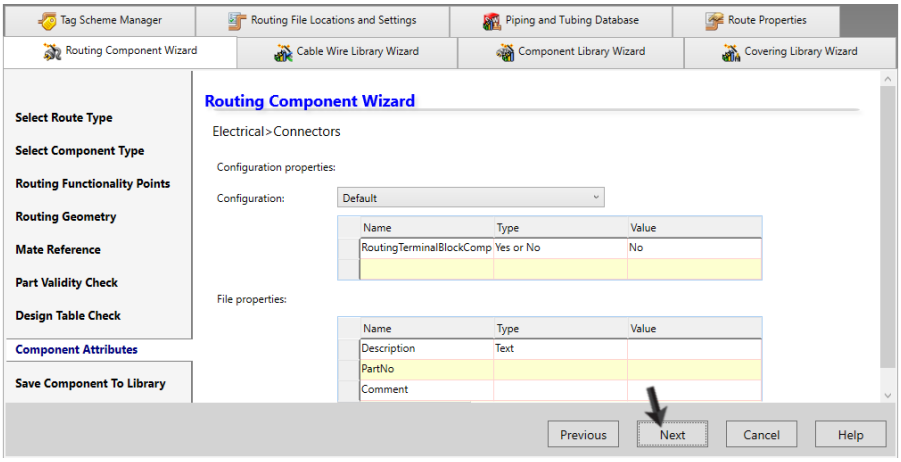
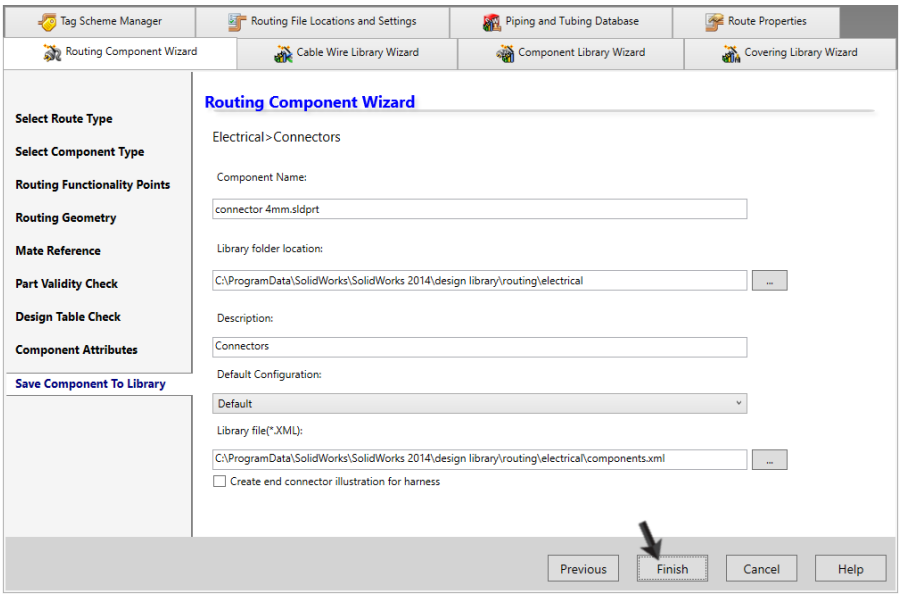
| **94** | If there are more configurations in your part you can set that here.<br><br>In our exaple this is not the case, so just click Next. |  |
| **95** | Finaly you can edit the name of the part and indicate to which library it should be added.<br><br>Normaly you will use the default values here.<br><br>Click on Finish |  |
| **96** | | The component now has been added tot he library. Use it in an assembly to check if everything works correct. |

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

# SOLIDWORKS works in education

You cannot imagine the modern technical world of today without 3D CAD. Whether your profession is in the Mechanical-, Electrical-, and Industrial Design- or Automotive industry: 3D CAD is THE tool of the designer or engineer from today.
SOLIDWORKS is the most used 3D CAD design software. Thanks to the unique combination of features: easy-to-use, widely applicable and with an excellent support. In the annual updates more and more customer wishes are implemented, which leads to an annual increase of the functionality, but also to optimization of functions already available in the software.

**Education**
A great number of educational institutes, in a variety from Technical Vocational Training to Universities already have chosen for SOLIDWORKS. Why?

For a **tutor** the choice for SOLIDWORKS is a choice for user-friendly software, easy to learn for pupils and students. SOLIDWORKS fits into the system of a problem-initiated training or a competence-related training. Tutorials are available for the different levels of training, like a series of tutorials for Technical Vocational level in which the scholar is lead through the software step-by-step. Also the higher levels, in which complex designing - for instance double curved planes - is needed, can work with SOLIDWORKS. All tutorials are in English and free-downloadable from www.SOLIDWORKS.com.

For a **scholar** or a **student**, learning to work with SOLIDWORKS is fun and defying. By using SOLIDWORKS, technique becomes more and more visible and tangible, which results in a more fun and realistic way of working on an assignment. Even better, every scholar of student knows that job-opportunities increase when SOLIDWORKS, the most used 3D-CAD software is on his or her resume. On many job sites you will find a great number of available jobs and internships that require SOLIDWORKS. This will increase the motivation to learn how to use SOLIDWORKS.
To make the use of SOLIDWORKS even easier, a Student Kit is available. If the school uses SOLIDWORKS, every scholar or student can get a **free download** of the Student Kit. It is a complete version of SOLIDWORKS, which is only allowed to be used for educational purposes. The data you need to download the Student Kit is available through your teacher or tutor.

The choice to work with SOLIDWORKS is an important issue for the ***ICT-department*** because the need to install new hardware can be postponed thanks to the fact that SOLIDWORKS has relatively low hardware demands. The installation and management of SOLIDWORKS in a network is very simple, amongst others because of the use of network licenses. And if a problem occurs after all, a qualified helpdesk is available, which will help you to get back on the right track again.

**Certification**
When you control SOLIDWORKS sufficiently you can join the CSWA-test. CSWA stands for Certified SOLIDWORKS Associate. After passing this exam, you will receive a certificate which can be used to prove that you are in control of SOLIDWORKS. This can be very useful when applying for a job or internship.
After finishing this series of tutorials, you will know enough to join the CSWA-test.

**Finally**
SOLIDWORKS has committed itself for an extended period to educational institutes and schools. By supporting teachers where possible, making tutorials available, adapting the software annually to the latest version and by supplying the Student Kit. The choice for SOLIDWORKS is a choice for the future. The future of education, which ensures itself of a wide support and a future of scholars and students, who want to have the best opportunities after their technical training.

**Contact**
Do you still have questions about SOLIDWORKS, please contact your local reseller.

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