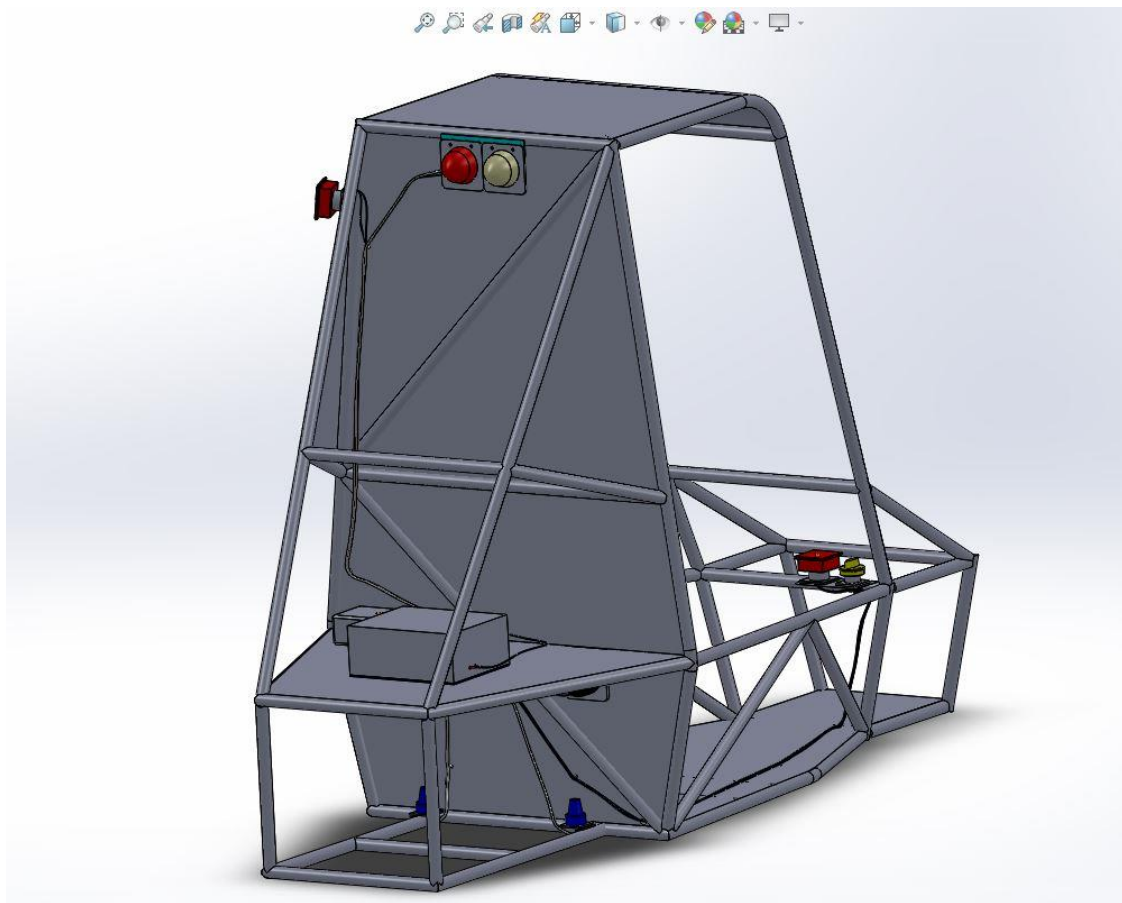


# **Baja SAE® Electrical Design and Analysis Project with SOLIDWORKS® Software**



Dassault Systèmes SolidWorks Corporation  
175 Wyman Street  
Waltham, MA 02451 USA

Outside the US: 1-781-810-5011  
Email: [solidworks.education@3ds.com](mailto:solidworks.education@3ds.com)

© 1995-2018, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

**The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systemes SolidWorks Corporation (DS SolidWorks).**

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

#### **Patent Notices**

**SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072; 9,262,863; 9,465,894; 9,646,412; 9,870,436; 10,055,083; 10,073,600 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).**

eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.

U.S. and foreign patents pending.

#### **Trademarks and Product Names for SOLIDWORKS Products and Services:**

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks. FeatureWorks is a registered trademark of HCL Technologies Ltd.

**SOLIDWORKS 2019, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, SOLIDWORKS Flow Simulation, SOLIDWORKS CAM, SOLIDWORKS Manage, eDrawings Viewer, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic Professional, SOLIDWORKS Electrical 3D, SOLIDWORKS Electrical Professional, CircuitWorks, SOLIDWORKS Composer, SOLIDWORKS Inspection, SOLIDWORKS MBD, SOLIDWORKS PCB powered by Altium, SOLIDWORKS PCB Connector powered by Altium, and SOLIDWORKS Visualize are product names of DS SolidWorks.**

Other brand or product names are trademarks or registered trademarks of their respective holders.

#### **COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY**

The Software is a "commercial item" as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of "commercial computer software" and "commercial software documentation" as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government

(a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R.

227.7202 -1 (JUN 1995) and 227.7202-4 (JUN 1995)

**In the event that you receive a request from any agency of the U.S. Government to provide Software with rights beyond those set forth above, you will notify DS SolidWorks of the scope of the request and DS SolidWorks will have five (5) business days to, in its sole discretion, accept or reject such request. Contractor/ Manufacturer: Dassault Systemes SolidWorks Corporation, 175 Wyman Street, Waltham, Massachusetts 02451 USA.**

#### **Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products:**

**Portions of this software © 1986-2018 Siemens Product Lifecycle Management Software Inc. All rights reserved.**

**This work contains the following software owned by Siemens Industry Software Limited: D-Cubed® 2D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.**

D-Cubed® 3D DCM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed® PGM © 2018. Siemens Industry Software Limited. All Rights Reserved. D-Cubed® CDM © 2018. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed® AEM © 2018. Siemens Industry Software Limited. All Rights Reserved.

Portions of this software © 1998-2018 HCL Technologies Ltd.

Portions of this software incorporate PhysX™ by NVIDIA 2006-2010.

Portions of this software © 2001-2018 Luxology, LLC. All rights reserved, patents pending.

Portions of this software © 2007-2018 DriveWorks Ltd.

© 2011, Microsoft Corporation. All rights reserved. Includes Adobe® PDF Library technology

**Copyright 1984 -2016 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.**

Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.

For more DS SolidWorks copyright information, see Help > About SOLIDWORKS.

#### **Copyright Notices for SOLIDWORKS Simulation Products:**

Portions of this software © 2008 Solversoft Corporation.

PCGLSS © 1992-2017 Computational Applications and System Integration, Inc. All rights reserved.

#### **Copyright Notices for SOLIDWORKS PDM Professional Product:**

**Outside In® Viewer Technology, © 1992-2012 Oracle ©**

**2011, Microsoft Corporation. All rights reserved.**

#### **Copyright Notices for eDrawings Products:**

Portions of this software © 2000-2014 Tech Soft 3D.

Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler. Portions of this software © 1998-2001 3Dconnexion.

Portions of this software © 1998-2014 Open Design Alliance. All rights reserved. Portions of this software © 1995 -2012 Spatial Corporation.

The eDrawings® for Windows® software is based in part on the work of the Independent JPEG Group.

Portions of eDrawings® for iPad® copyright © 1996-1999 Silicon Graphics Systems, Inc. Portions of eDrawings® for iPad® copyright © 2003 - 2005 Apple Computer Inc.

#### **Copyright Notices for SOLIDWORKS PCB Products:**

Portions of this software © 2017 - 2018 Altium Limited.

#### **Copyright Notices for SOLIDWORKS Visualize Products:**

NVIDIA GameWorks™ Technology provided under license from NVIDIA Corporation. Copyright © 2002 -2015 NVIDIA Corporation. All rights reserved.

Document Number: PMT19xx-E

## Contents:

### Lesson 1: Introduction

- Using this book.....3
- What is SOLIDWORKS Electrical (SWE) Software?.....3
- Prerequisites.....3
- Conventions used in this book.....3
- Getting started with SOLIDWORKS Electrical.....3
- Projects manager.....4

### Lesson 2: Draw a Line Diagram

- Insert a line diagram symbol.....6
- Create a line diagram symbol .....8
- Associate symbol to manufacturer part.....11
- Create a manufacturer part.....12
- Modify the existing Manufacturer parts and circuits.....13

### Lesson 3: Detailed Cabling

- Draw cables between the components/line diagram symbols.....16
- Create a new cable.....17
- Detailed cabling: Associate cable between the components.....19

### Lesson 4: SOLIDWORKS Parts –Add Electrical Connection points (CPoint)

- Create a 3D part to associate with the line diagram symbol.....22
- Create a connection point (CPoint) using Electrical Component Wizard.....23
- Create a cable connection point (Cable CPoint).....25

### Lesson 5: SOLIDWORKS Electrical 3D

- Open Electrical projects in SOLIDWORKS .....27
- Insert/ Associate the electrical components to 3D parts.....29

### Lesson 6: Routing Cables

- Routing cables.....31
- Create Routing Paths .....32
- Route cables .....33
- Flattening the route .....36
- Adding the Bill of Materials (BOM) to the drawing.....37

## Lesson 1: Introduction

### Using This Book

The SAE BAJA Electrical project helps you to create electrical schematics using SOLIDWORKS Electrical to integrate the schematics with your 3D model.

### What is SOLIDWORKS Electrical (SWE) Software?

SOLIDWORKS Electrical is a schematic software with an expandable set of libraries to design complex electrical systems. The automated design and management tools help to simplify the design tasks from terminal block to cross-reference assignments.

SOLIDWORKS Electrical 3D is an active add-in for SOLIDWORKS CAD which enables you to place electrical components and use the SOLIDWORKS routing technology to automatically interconnect the electrical schematic design within the 3D model.

Please note, that SOLIDWORKS Electrical Professional provides a license of both SWE Schematic Professional and SWE-3D.

### Prerequisites

Before you start the SAE BAJA Electrical project, you should install '**SOLIDWORKS Electrical**' and go through the tutorials that are integrated in SOLIDWORKS software.

### Conventions used in this book

Convention	Meaning
<b>Bold</b>	SOLIDWORKS Commands and options represented in this style
<b>Bold <i>Italic</i></b>	SOLIDWORKS CAD files are represented in this style

### Getting started with SOLIDWORKS Electrical

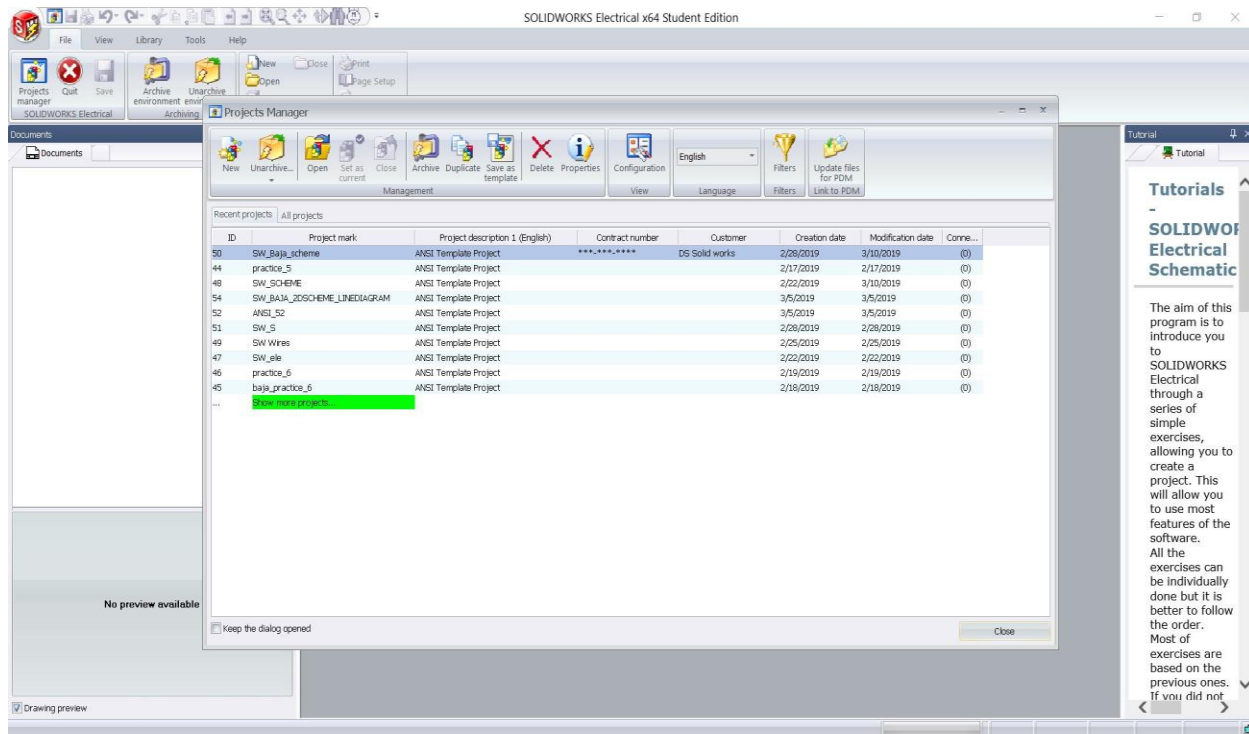
- SOLIDWORKS Electrical Schematic is a standalone program
- SOLIDWORKS Electrical 3D is an add-in for SOLIDWORKS

SOLIDWORKS Electrical Schematic is launched from the Windows start→ Programs→ SOLIDWORKS 2018→ SOLIDWORKS Electrical.

Download the zip files attached and extract the files.

## Projects Manager

The Projects manager will be automatically displayed when the software is started. Through this interface you can create a new project, open/duplicate, archive/unarchive and the change properties of an existing project.



To 'Start' a new project:

- Click 'New'
- Select the project template 'ANSI', click 'OK'
- Select language as 'English' click 'OK'
- Enter the name of the project as 'SAE BAJA ELECTRIC'
- Enter the required information in the properties dialog box

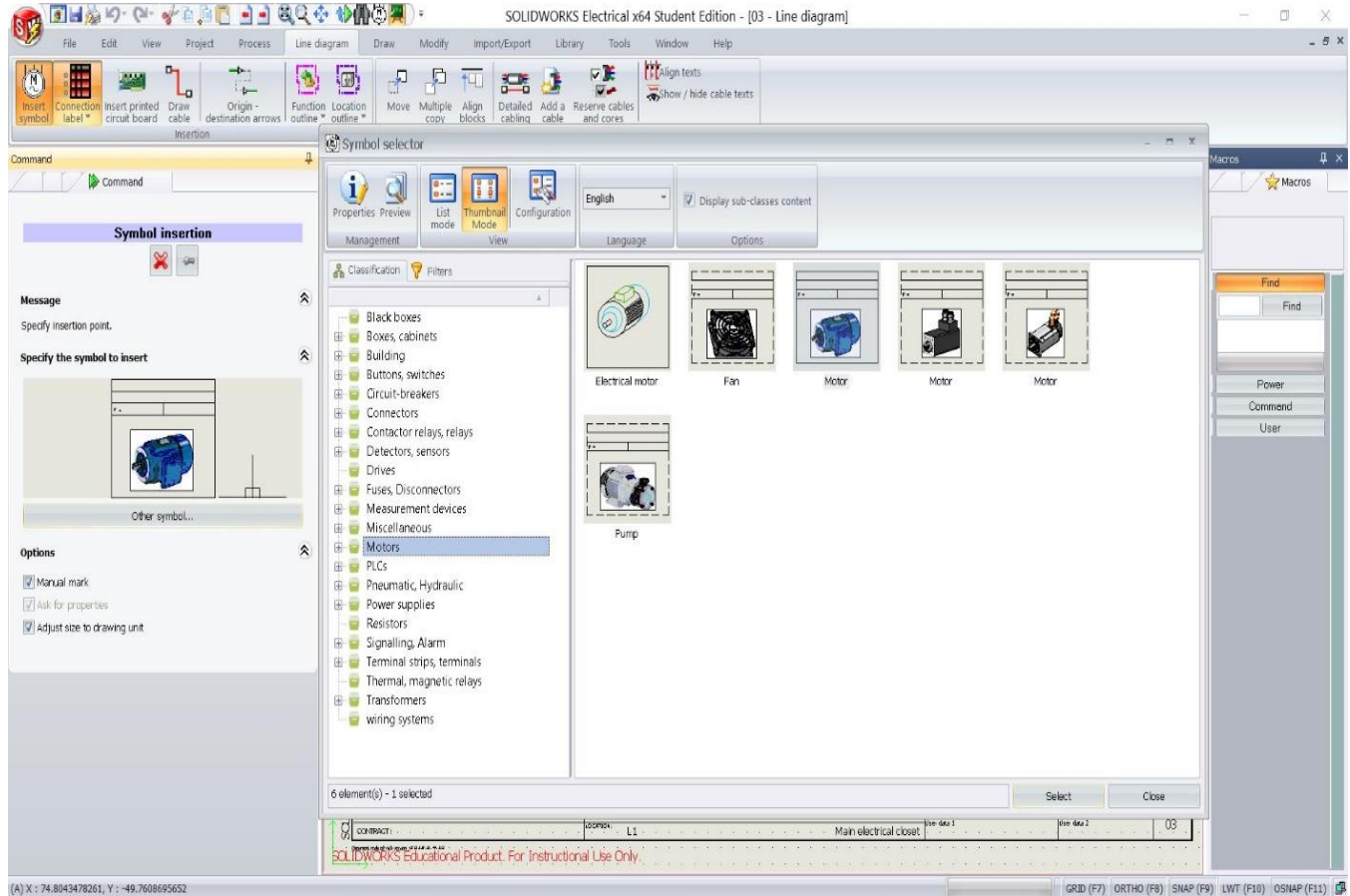
## Lesson 2: Draw a Line Diagram

- Insert a line diagram symbol
- Create a line diagram symbol
- Associate symbol to manufacturer part
- Create a manufacturer part
- Modifying existing manufacturer parts and circuits

### Insert a line diagram symbol:

Open '1-Document book' and double click on 'Line diagram' to open.

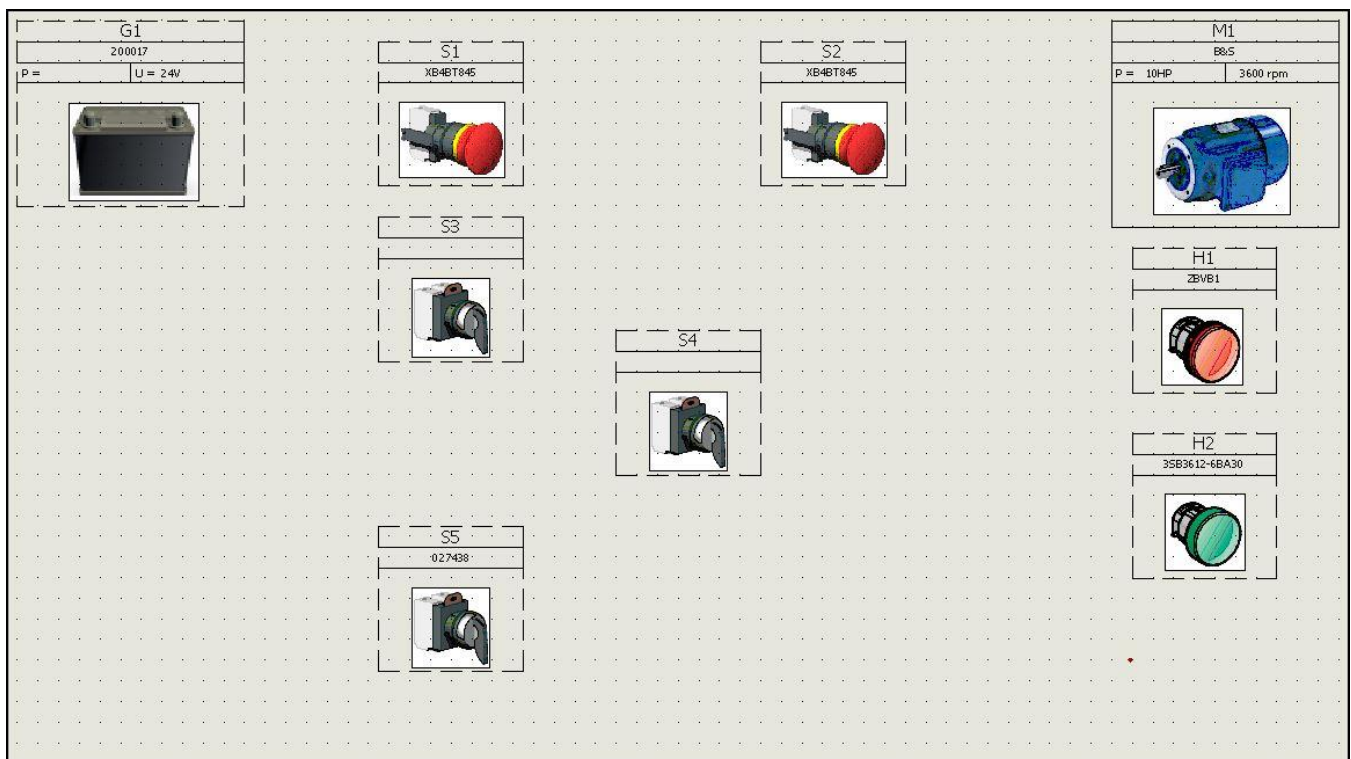
- The Line diagram page opens in the graphics window
- Click on 'line diagram' -> 'Insert symbol'
- A palette is displayed showing the last placed symbol. Click on 'other symbol' (If it is first time that SW Electrical is used on the system, 'symbol selector' opens automatically)



- In the symbol selector window, open the classification 'Motors'
- Highlight the Motor and click 'Select' to return to the drawing
- Click the left mouse button to insert the symbol on the extreme right corner as shown
- Click 'OK' to insert the symbol

Similar to Motor add the other symbols to the line diagram as shown.

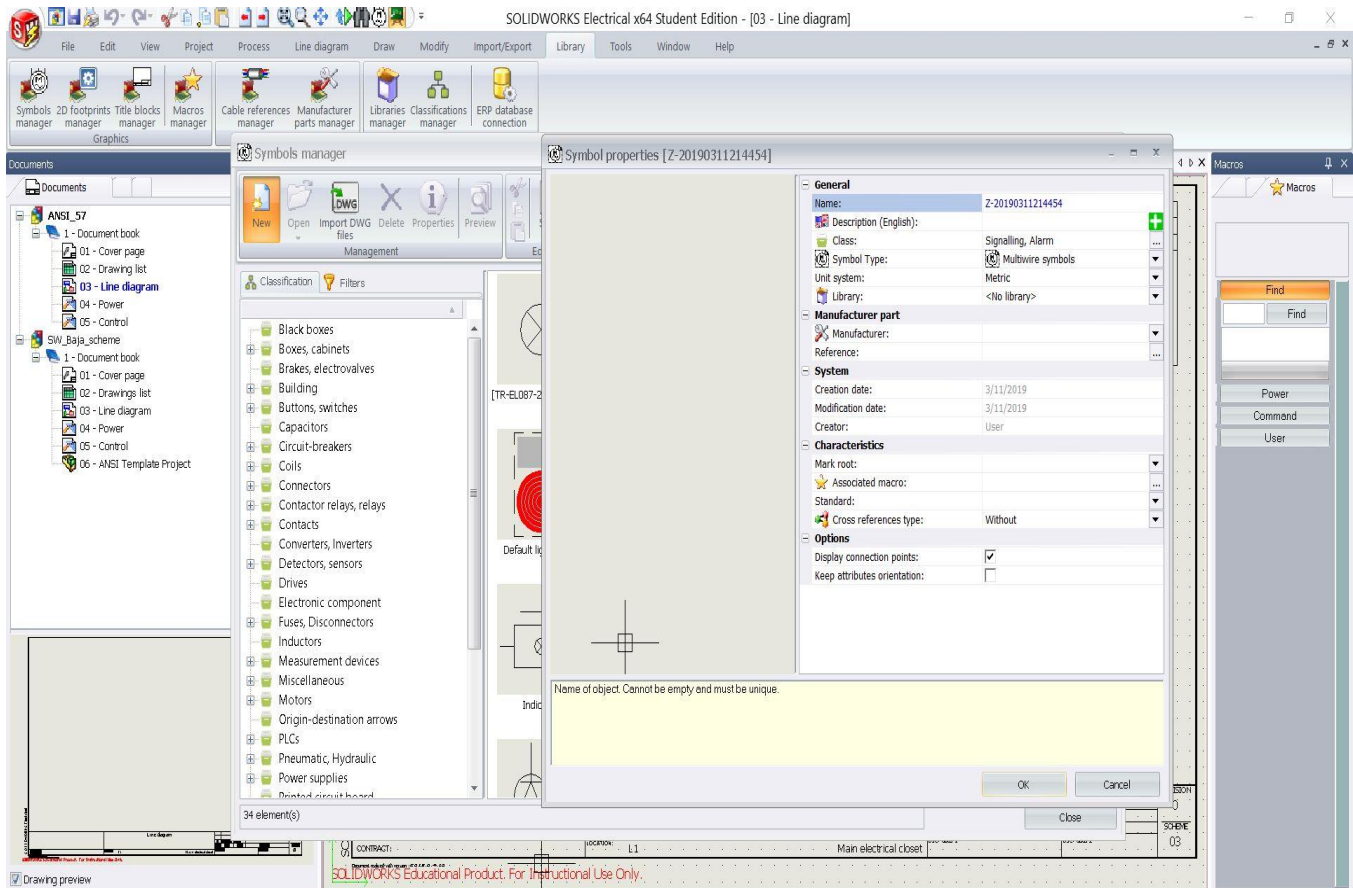
No.	Classification	Symbol
1	Power supplies	Cells Battery
2	Buttons, switches	Emergency stop button(2)
3	Signaling, alarm	Red/Green Indicator(1/1)
4	Buttons, switches	Switch(3)





### Create a line diagram symbol:

- Go to the 'Library' tab in SOLIDWORKS Electrical and select the 'Symbols manager' command. The Symbols manager dialog box appears as shown below



- Click on 'New'. This will allow you to create a brand new symbol (Note: you can copy and edit the symbol as required)
- Enter the relevant properties in the dialog box:
  - Name: 'Reverse alarm'
  - Description: 'Buzzer when reverse is activated'
  - Class: 'Signaling/ Alarm'
  - Symbol type: 'Line diagram symbols'
  - Unit system: 'Metric'
- Click 'OK' to add the symbol to the library
- Close the Library

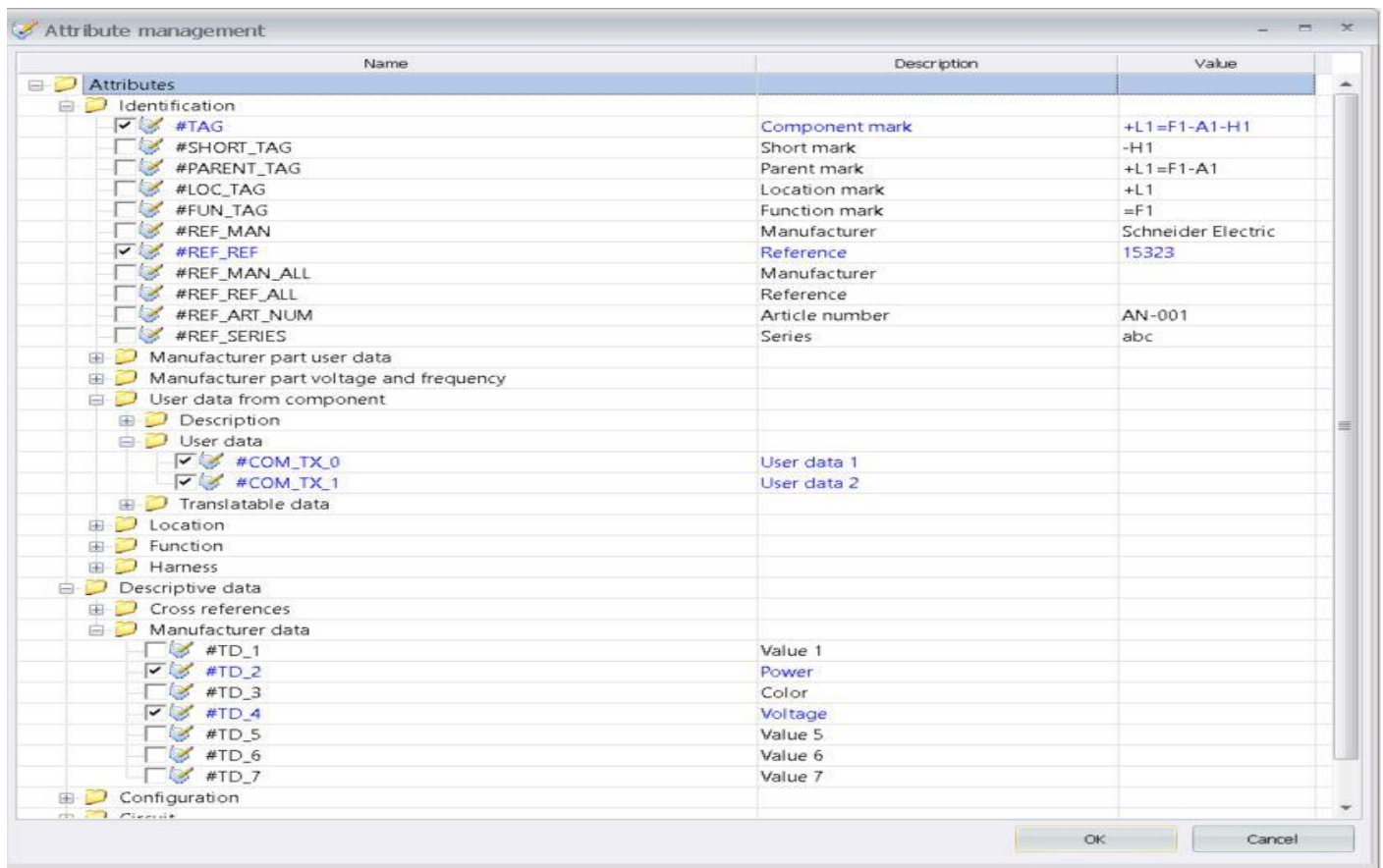
## Draw/ Update the symbol

- Locate the symbol '**Reverse alarm**' in the symbols manager. Open the symbol an empty drawing is opened with origin at (X: 0) ;( Y: 0)
- Click on the '**Draw**' tab and select '**Rectangle**'
- In the Command window enter the start point coordinates as (0, 0), click the '**green check mark**'
- Specify the opposite corner as (50, 50)
- Draw lines as shown in figure to display the name and properties of the symbol



- To change the line type, select the line. The '**graphical properties**' tab opens up on the right side of the window. Change the line type to '**CENTER AXES**'
- Go to the '**Edit symbol**' Tab located on the toolbar and select '**Insert attribute**'
- Select the following attributes as shown below to represent the data of the symbol
- Select the position to place the attributes. You can change the order or position of each attribute by drag and drop
- Go to the '**Draw**' tab. Click on '**Insert image**' select the image file that you need to insert
- The file format should be '**.bmp**'
- Open the '**SW\_CADPARTS.Zip**' file. Insert the '**Reverse alarm.bmp**' image

Close the symbol by clicking the '**X**' mark on the tab, SOLIDWORKS will automatically ask you to save the changes. Click on '**Yes**' to save the changes to the drawing.



Go to the Line Diagram Tab and Insert the symbol you created as shown.

Now you have successfully inserted all the line diagram symbols.

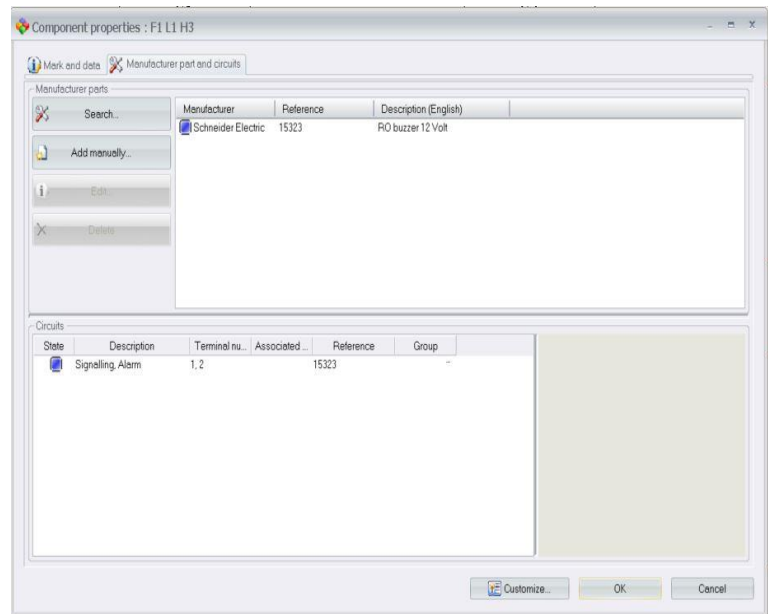
## Associating symbol to manufacturer parts

### Manufacturer Parts:

These are key to understanding the flow of information in SOLIDWORKS Electrical and are different from the SOLIDWORKS parts. Technically, these parts are ordered, purchased, installed and connected. Every manufacturer's part has different technical data related to its classification. By default, SOLIDWORKS Electrical only has a few manufacturer's parts that are added to the database after installation, it is possible to add new custom manufacturer parts or unarchive manufacturer's data from the online content portal.

### Adding manufacturer parts to the components:

- Right click on the reverse alarm symbol and go to 'Component properties'
- Go to 'Manufacturer part and circuits' and click on 'Search'
- It will open the 'Manufacturer part selection' window
- There are multiple ways you can add manufacturer part to the line diagram symbol
- You can go to the 'Classification' tab and go to 'Signaling, Alarm' and select option 'Sounding' and search for part '15323 RO BUZZER 12 VOLT' from 'Schneider electric'

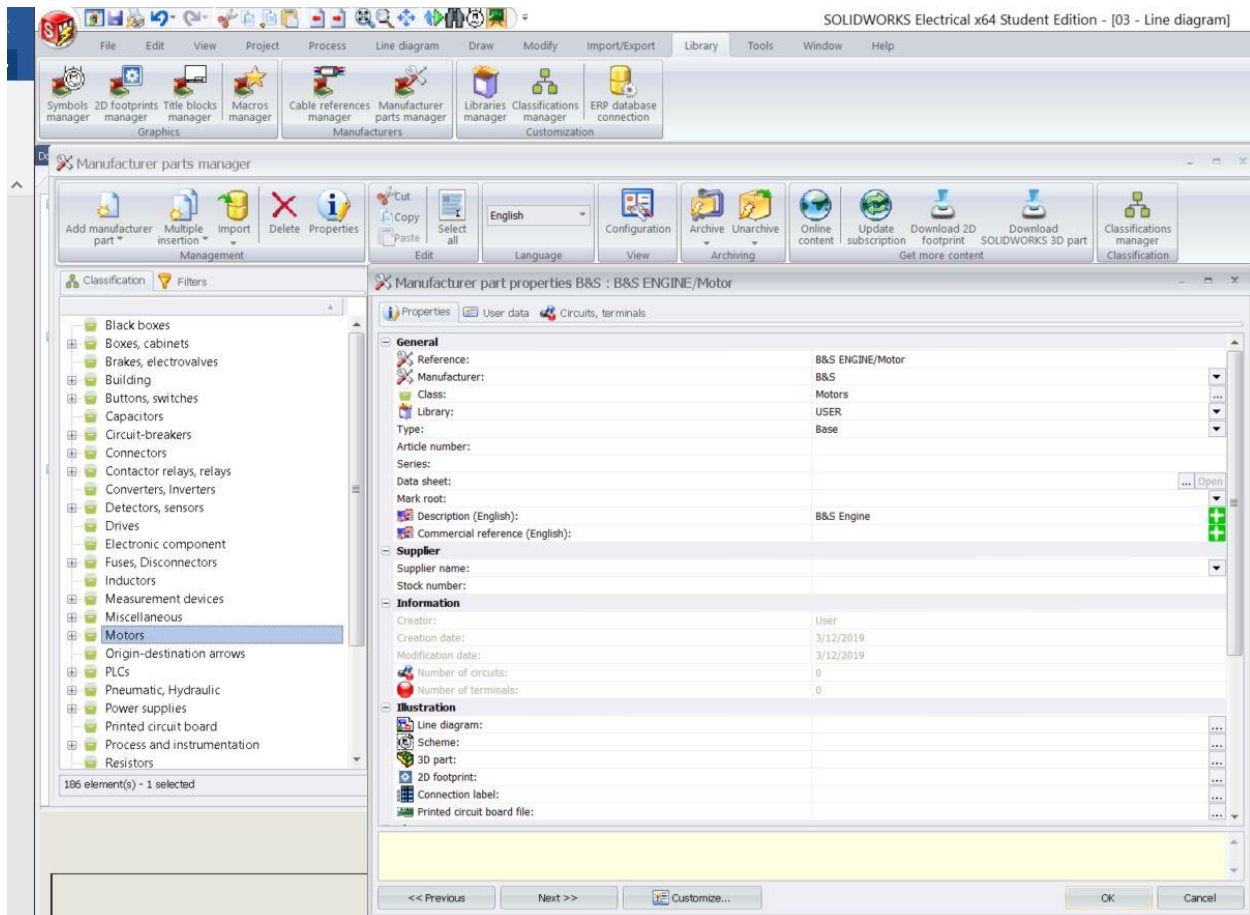


(Or)

- You can go to 'Filters' → 'Remove Filters' → enter '15323' in the 'Reference' section
- You can filter the manufacturer parts using different classifications such as 'class'; 'Library'; 'Type'; 'manufacturer'; 'Number of circuits and terminals'
- Select the manufacturer part and click on the '+' symbol to add the selected manufacturer part
- To remove a manufacturer's part, click on the manufacturer's part so it is highlighted. Then click on the red '-' symbol
- Click 'Select' to confirm the selection and click 'OK' to add the part to the symbol
- If you cannot find the part '15323,' add another part from the same classification

## Create a Manufacturer part

- Go to the 'Library' tab and select 'Manufacturer parts manager.' The Manufacturer parts manager window will open up, showing the list of available manufacturer parts
- Click on 'Add manufacturer part' and enter the part information



Reference: 'B&S ENGINE/Motor'

Manufacturer: 'B&S'

Class: 'Motors'

Library: 'User'

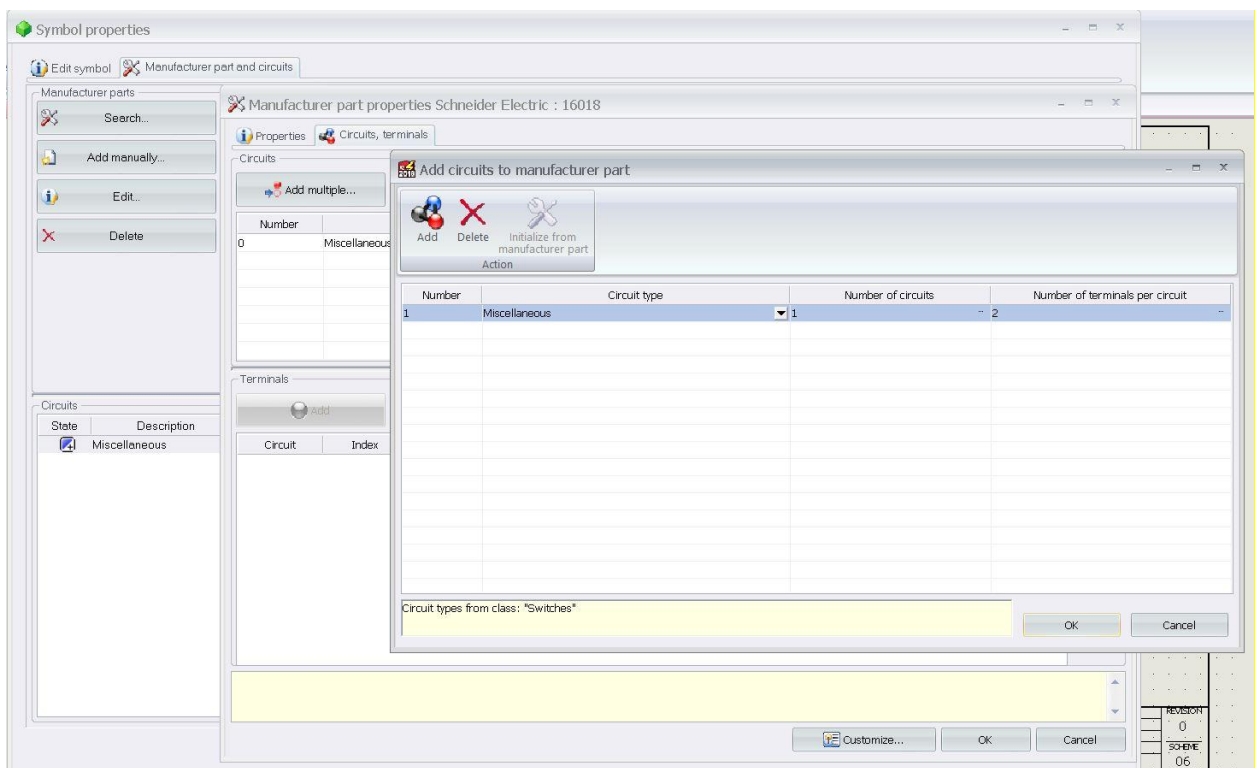
Description: 'B&S Engine'

Click on 'OK' to add the part to the library.

- Add the newly created manufacturer part to the '*B&S ENGINE/motor*' in your line diagram

### Modify the existing Manufacturer parts and Circuits:

- Select the switch 'S3' which connects the battery to the brake indicator (red light symbol)
- Right click the component and select '**component properties.**' Go to '**manufacturer parts and circuits.**' Remove the filters and "**search**" for reference number '**16018**'
- Add the manufacturer part and click '**select.**' It will redirect back to the component properties window
- Select the manufacturer part which was added and double click on it
- '**Manufacturer part properties...**' window will open for the selected manufacturer part
- **On the top tab, click on "Circuits, Terminals"**
- Click on '**Add multiple.**' '**Add circuits to manufacturer part**' window will open up. Click on '**Add**'
- It will add the circuit and terminals as required, by default it will add '**1**' circuit and '**2**' terminals



- Click on the drag down button in circuit type and select '**switch.**' Click '**OK**'
- Add terminal marks, which make the detailed cabling easy. To add the terminal marks, select the first circuit (with the number '**0**')
- In the terminals window located below, go to the "Mark" column and enter '**0**' in the first row and '**1**' in the second row
- Similarly, add the terminal numbers for second circuit as '**2**' and '**3**' consecutively
- '**Replace manufacture part**' window opens up and will display two options. Select option '**Modify this component only**'. Click '**OK**'
- The manufacturer part is updated and added to the symbol

- Similarly add the following manufacturer parts and update them as mentioned in the table with the correct number of circuits and terminals. This is critical for the next lesson “Detailed Cabling”.

Line diagram symbol	Reference number for manufacturer part	Is the part updated (Y/N)	Total Number of circuits (C) and terminals (T)
Kill switch	XB4BT845	N	2C 4T
Green Indicator	3SB3612-6BA30	N	1C 2T
Switch(S4)	16018	Y (Type: Switch)	2C 4T
Switch(S5)	027438	Y (Type: Switch)	3C 6T
Red Indicator	ZBVB1	Y (Type: Signaling, Alarm)	2C 4T
Battery	200017	Y (Type: Power supply)	4C 8T

Update the terminal numbers for all the manufacturer parts associated to the line diagram symbols. If you cannot find the specific manufacturer part, add another part from the same classification.

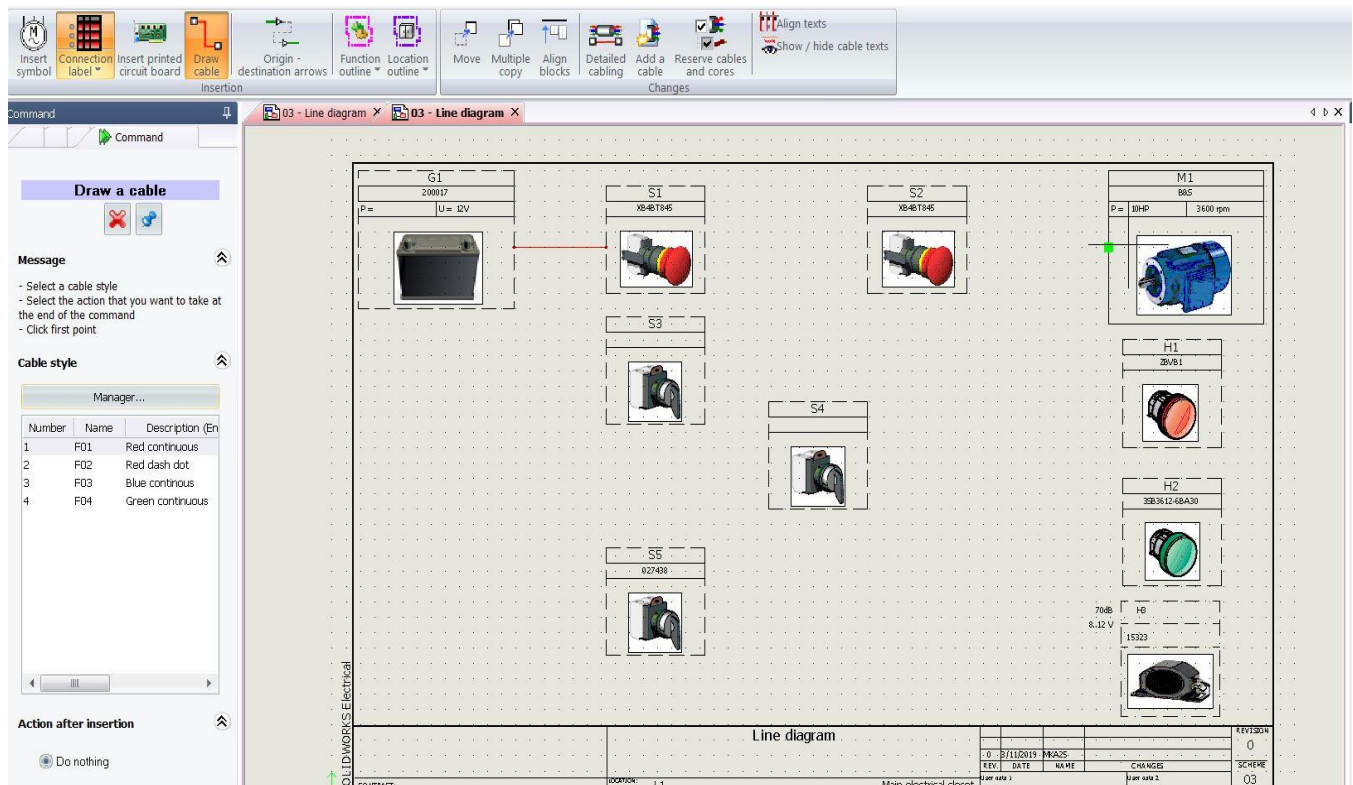
### Lesson 3: Detailed Cabling

- Draw cables between the components/line diagram symbols
- Create a new cable
- Detailed cabling: Associate cable between the components



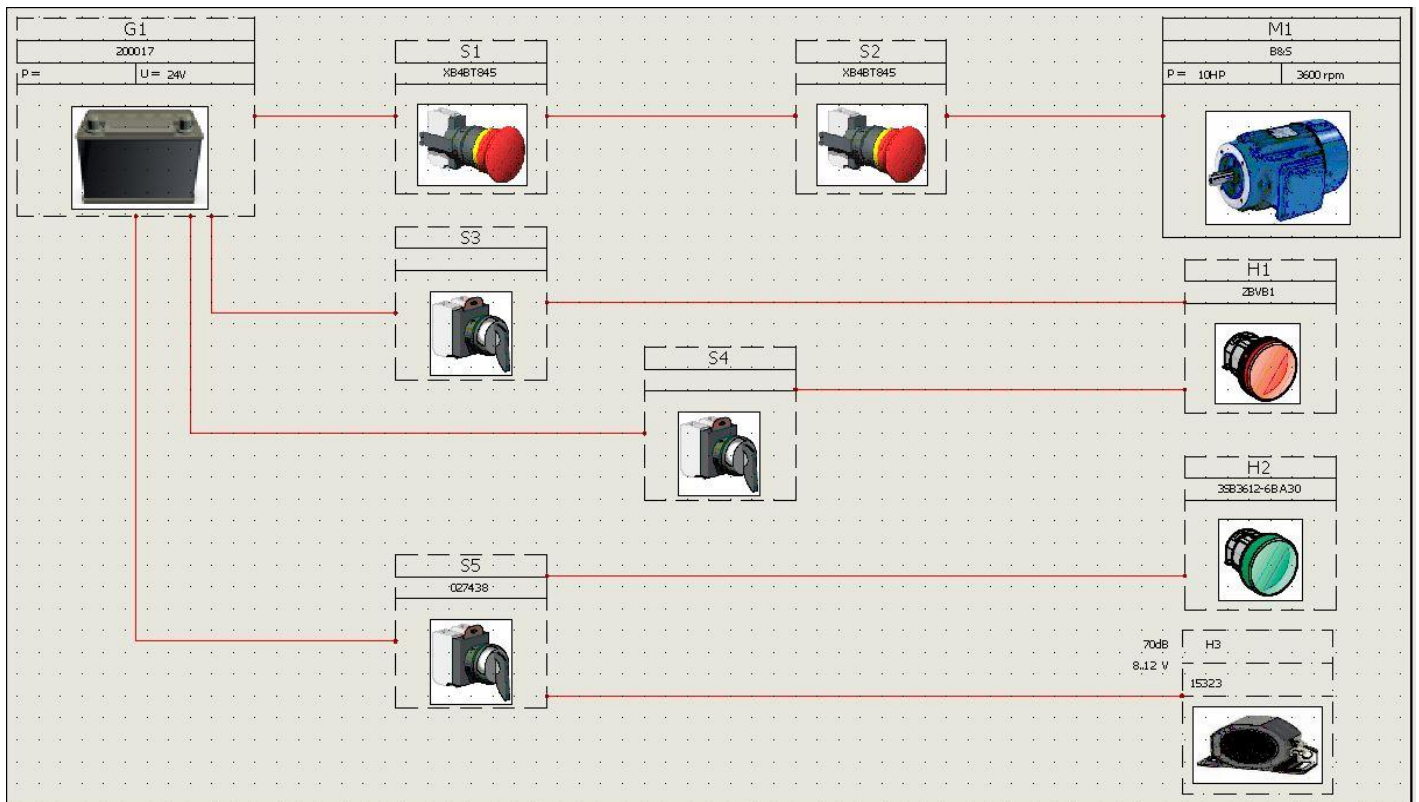
## Draw cables between components/line diagram symbols

- Go to the 'Line diagram' tab, select 'Draw cable'
- The cable style manager opens up. Select 'F01' (Red continuous line). You may need to manually change the color of the line. Click on "Color" and a Color palette will open up. Choose Red.
- You can see the pointer changes to green when you pass the mouse pointer on the top of a symbol



- When you connect two components the color of the pointer changes its color when an active connection is made between two components
- You will know if the connection is active by zooming in on the connection and noticing a dot on the outline of the symbols.

- Connect the line diagram symbols as shown
- Repeat the process by drawing the cables to inter connect all the components
- The 'Esc' key will stop creating the cables



### Create a new cable:

- The SOLIDWORKS library has a wide range of cables available
- If you cannot find the required cable, you can create your own cable with the desired properties and add it to the library
- Go to the 'Library' tab → 'cables reference manager' → "AWG" in Classifications → 'new reference' and enter the 'cable reference properties' as shown

**New cable reference**

Properties | User data | Cable cores

**General**

Reference: Project\_Username  
Manufacturer: Solidworks  
Class: \_AWG  
Article number:  
Library: USER  
Family:  
Standard:  
Series:  
Description (English):

**Supplier**

Supplier name:  
Stock number:

**Information**

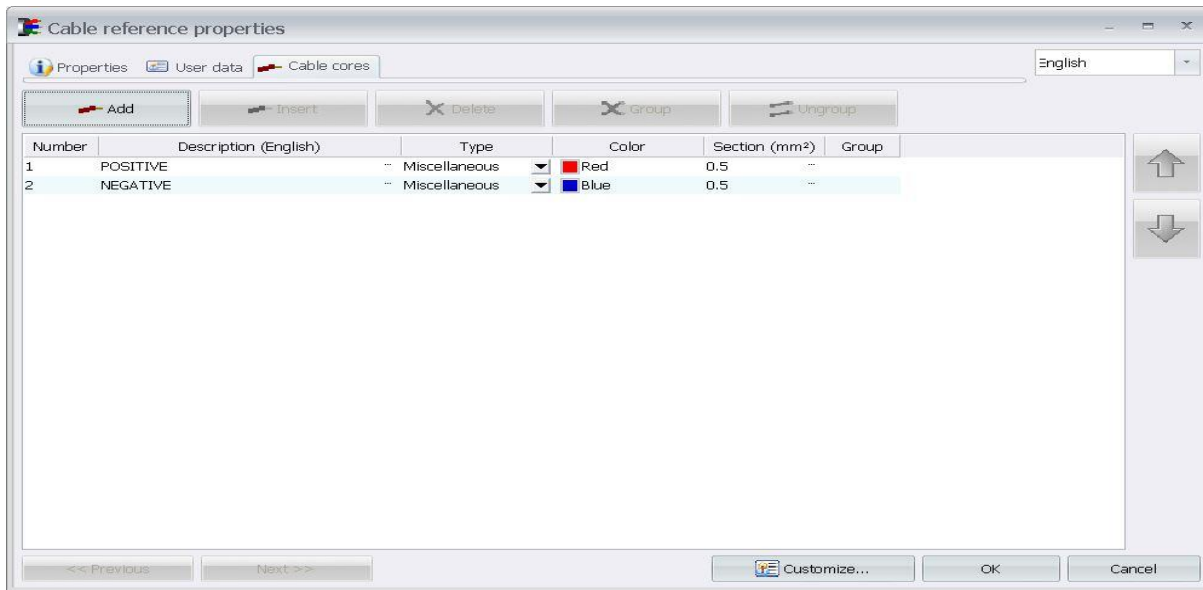
Creator: User  
Creation date: 3/14/2019  
Modification date: 3/14/2019  
Cable cores: 0

**Characteristics**

Type: Control  
Size standard: Section (mm<sup>2</sup>)  
Conductor section (mm<sup>2</sup>): 0.5  
Length (m): 0  
Diameter (mm): 2  
Color:  
Bend radius (x Diameter): 0  
Voltage drop (V/A/km): 0

Customize... OK Cancel

- Size standard: 'Section (mm<sup>2</sup>)'
- Conductor section: '0.5'
- Diameter of the cable: '2'
- Bend radius : '0'
- After entering the data go to the 'Cable cores' tab and click 'add' to add the cable cores to the new cable as shown



Click 'OK' to add the cable to the library.

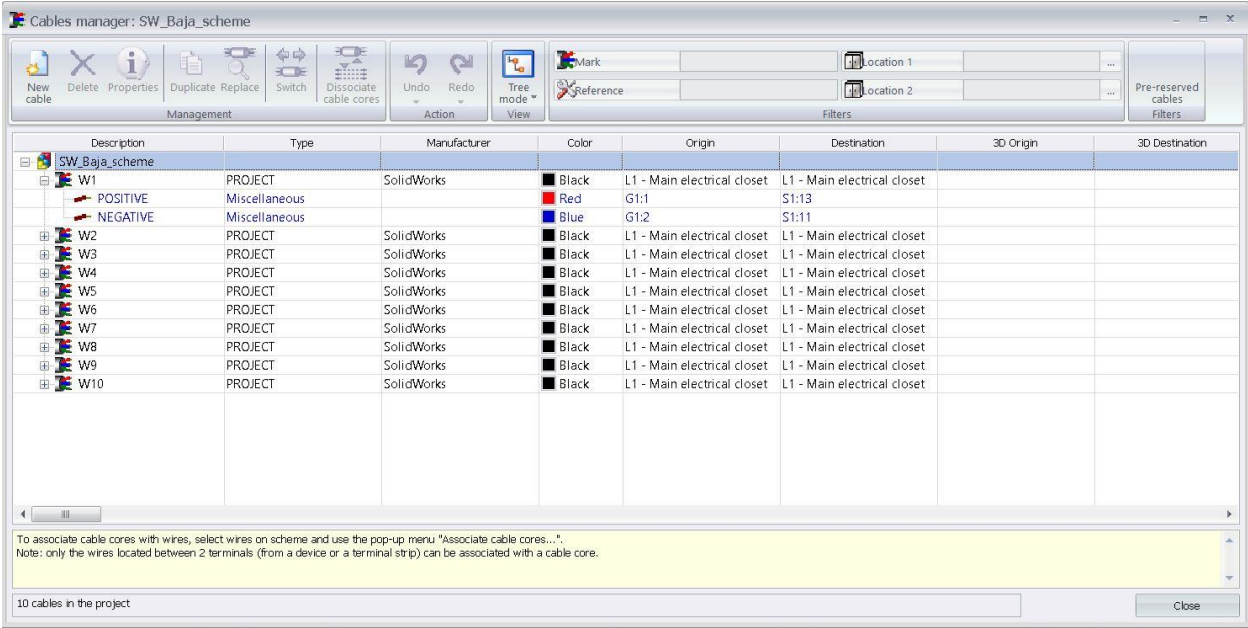
#### Detailed cabling: Associate cable between the components

- Select the cable passing through the '**Battery**' and '**Kill switch 1**'
- In the '**line diagram**' tab click '**Detailed cabling**'
- In the detailed cabling window, click on '**Add a cable.**' It will open the '**Cable references selection window**'
- Click on '**Remove filters.**' Enter '**Project**' in the reference section. The newly created cable will be displayed
- Select the cable and click on the '**+**' sign to add the cable and click '**select**'. This will add the cable to the detailed cabling window
- Add the **first circuit first terminal** of the battery to the **positive end** of the battery by drag and drop method. This is going to be a time taking process
- If errors are encountered here, please review the previous section - "**Modify the existing Manufacturer part Circuits and Terminals:**" on page 13
- Similarly add **first circuit second terminal** of the **battery** to **negative end** of the **battery**
- Similarly add the **first circuit first and second terminals** of the **Kill switch 1** to the **positive and negative ends** of the **cable**
- Click '**close**' to finish the process
- The wire number and reference name is displayed on the wire

You have successfully created a cable between two components '**Battery**' and '**Kill switch**'

Follow the same process and add a cable from detailed cabling for all the cables and assign terminals to the components.

Go to the 'Project' tab and click on cables and expand the project to see the cables connecting components, origin and destination of the cable, cable cores reserved to the link.



The window shows all the cables that are connected.

Save your project by clicking on 'File' → 'Save'.

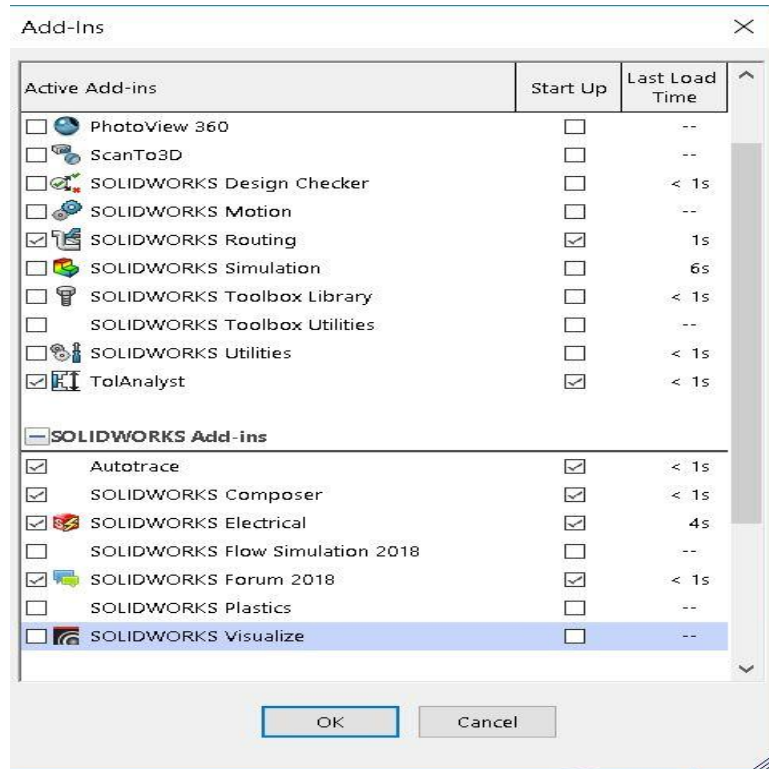
SOLIDWORKS Electrical saves the project in default location.

#### Lesson 4: SOLIDWORKS Parts – Add Electrical Connection points (CPoint)

- Create a 3D part to associate with the line diagram symbol
- Create a connection point (CPoint) using Electrical Component Wizard
- Create a cable connection point (Cable CPoint)

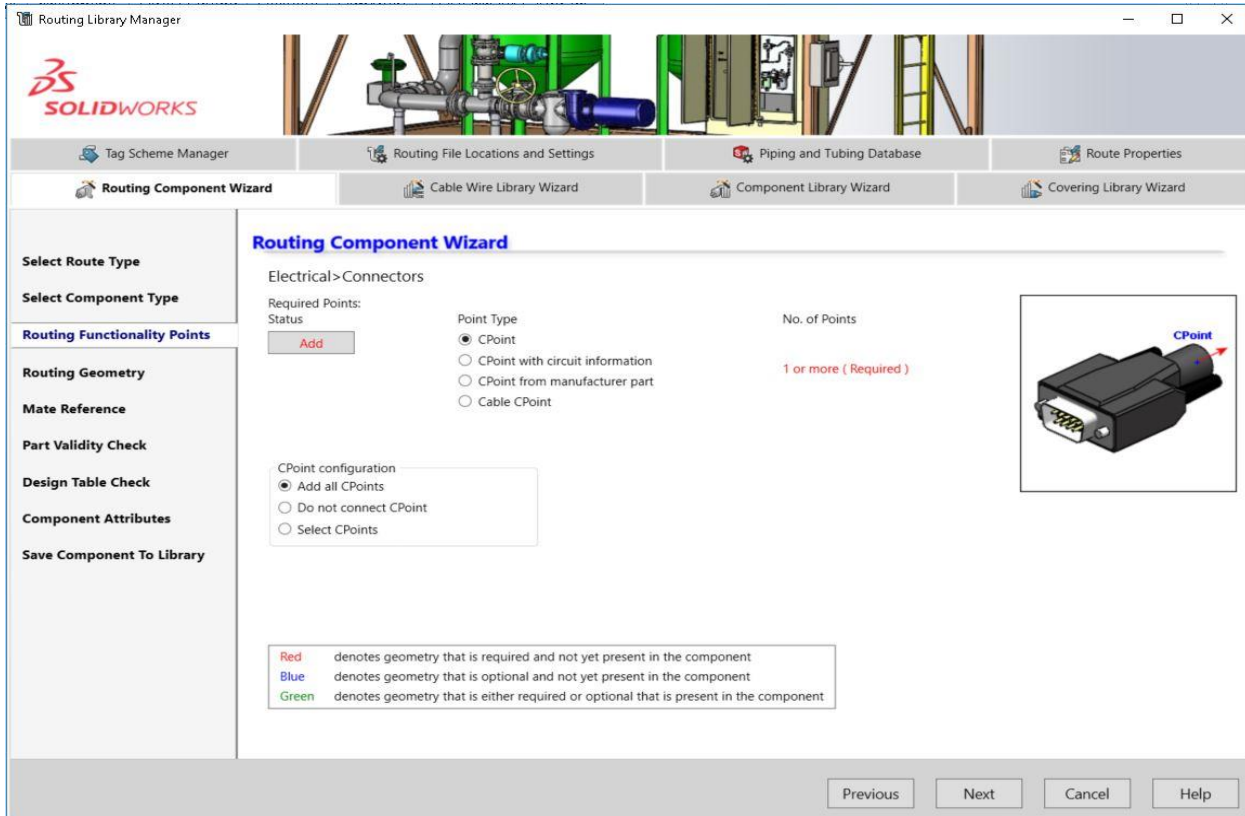
### Create a 3D part to associate with the line diagram symbol

- Download the 'SW\_CADPARTS.zip' file
- Open 'Reverse\_light.SLDPRT' from 'SW\_CADPARTS' folder
- Go to 'Tools' -> 'Add-Ins' -> add 'SOLIDWORKS Electrical' add-in and click 'OK'



## Create a connection point (CPoint) using Electrical Component Wizard

- Go to 'Tools' -> 'SOLIDWORKS Electrical' -> 'Electrical component Wizard'
- It will open the 'Routing Library Manager' (RLM) window as shown

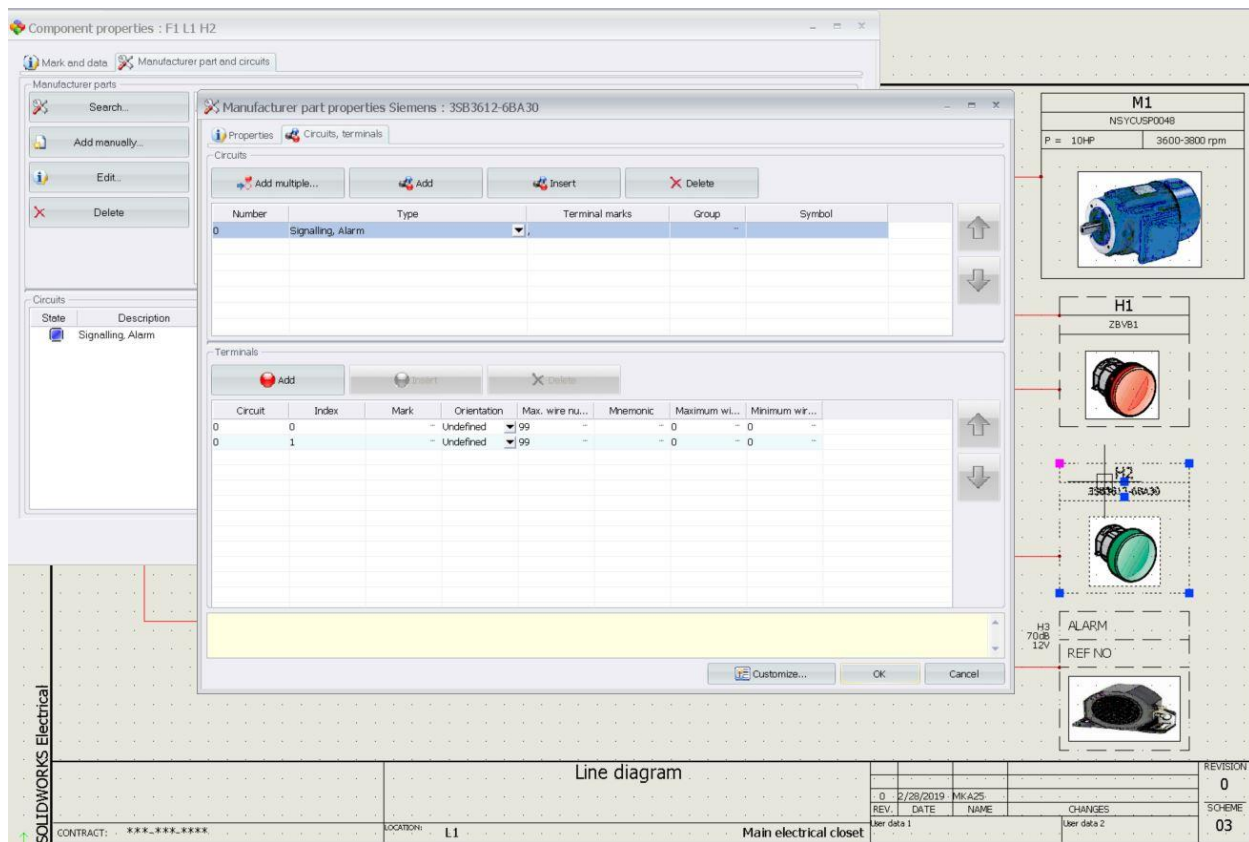


- Select the point type to 'CPoint with circuit information'. Click 'Add' to continue

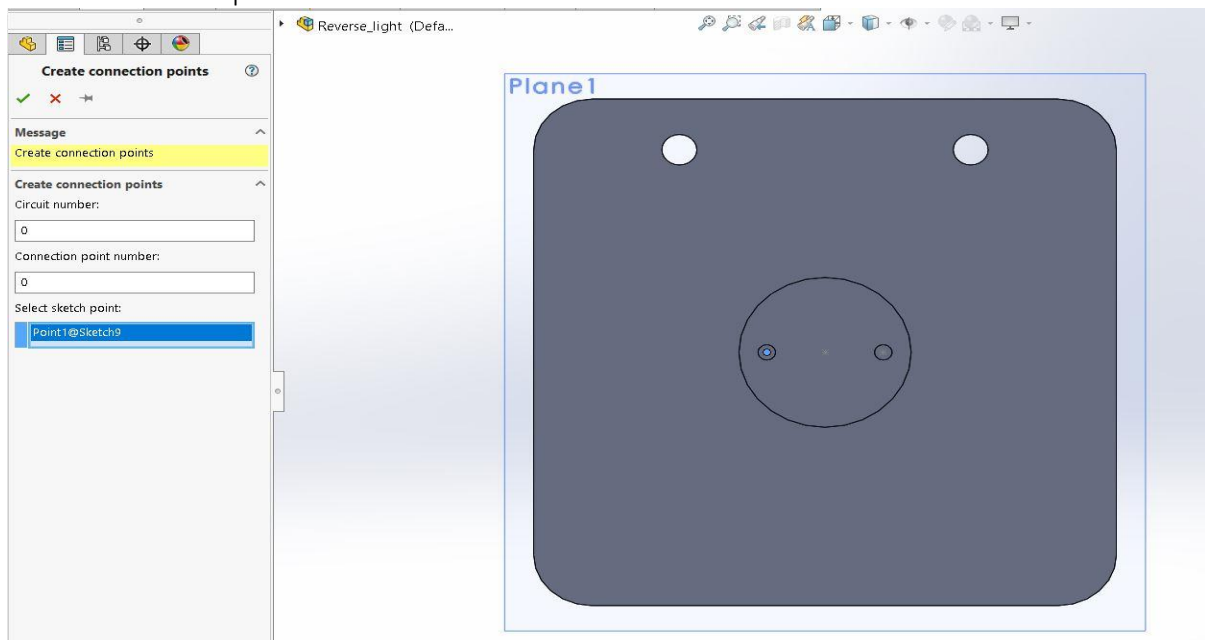
Note: The circuit number and connection point number represent the circuit number of the part which is going to be linked with the SOLIDWORKS Electrical part and the connection point number represents the index of the circuit.



- For 'Reverse\_light' the circuit and index numbers are shown below



- The circuit and index numbers should be entered correctly when you assign the CPoints to the 3D parts in order to successfully complete the next sections
- Enter the circuit number as '0' and the connection point number as '0'
- Select the sketch point as shown



- Click on **'OK'** to confirm the selection and to add the **'CPoint'** to the part
- The Routing Library Manager opens after adding the CPoint to the part. Select the CPoint with circuit information and click **'Add'**
- Repeat the same process for the second CPoint and enter circuit number as **'0'** and connection point number as **'1.'** Select the other point on sketch 9

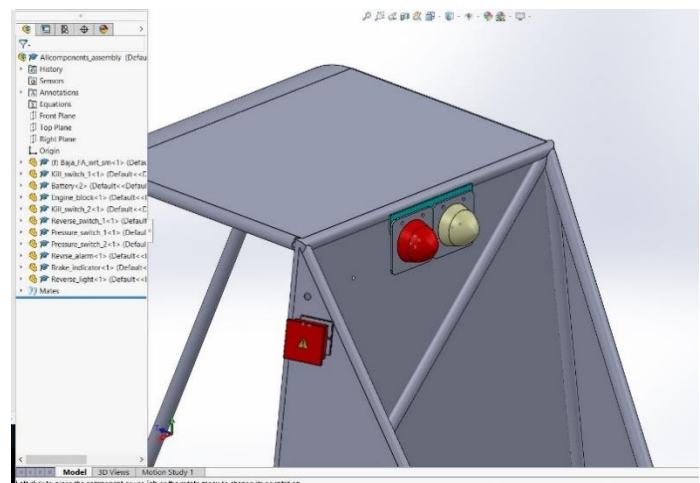
### Create a cable connection point (Cable CPoint)

Cable CPoint is the starting point for the cable. It is important to add the Cable CPoint to the 3D part to begin the route of the cables. After successfully adding the connection points to the part, the routing library manager will open automatically

- Select **'Cable CPoint'** and click **'Add'**
- Select the **'point'** on **'plane 2'**
- Click **'OK'** to confirm the selection and to add the Cable Connection Point to the part
- Close the **'Routing Library Manager'**
- Save the SOLIDWORKS file

You have successfully created the **'Connection points'** for the cable cores and **'Cable connection point'** for the cable.

- Open **'Allcomponents\_assembly.SLDASM'**
- Insert the **'Reverse\_light.SLDPRT'** into the assembly and mate it using coincident and concentric mates to fully define the assembly as shown
- The other parts have their respective CPoint and Cable CPoint included and are ready for use
- Save the assembly and close the assembly file



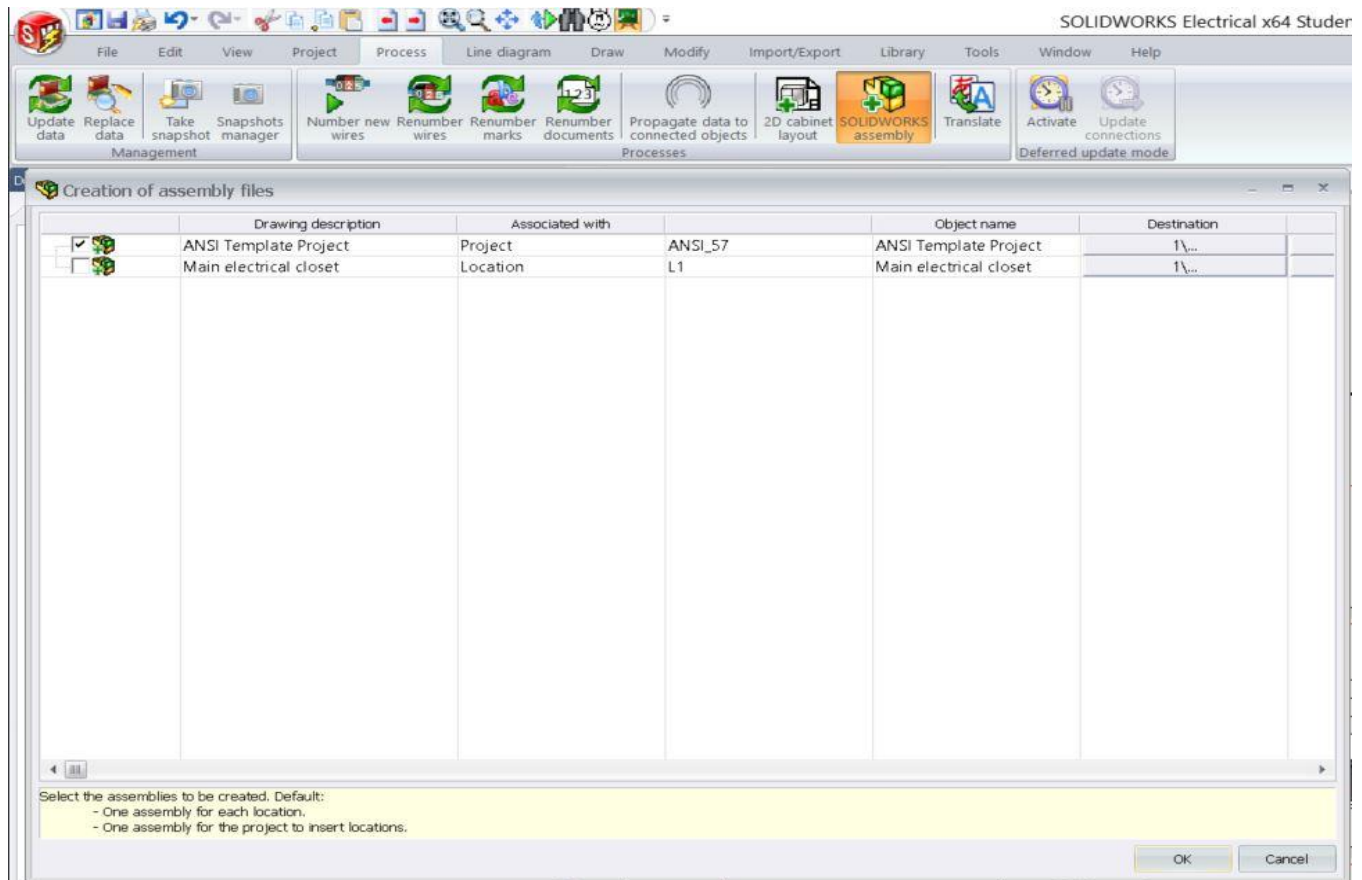
### Lesson 5: SOLIDWORKS Electrical 3D

- Open Electrical projects in SOLIDWORKS
- Insert/ Associate the electrical components to 3D parts

## Open Electrical projects in SOLIDWORKS

Open 'SOLIDWORKS Electrical' → open your project from the 'Projects manager' window.

- Go to 'Process' tab. Click on add 'SOLIDWORKS assembly'
- Since you have only one location associated with the project, (L1), you can add any one of those to the project

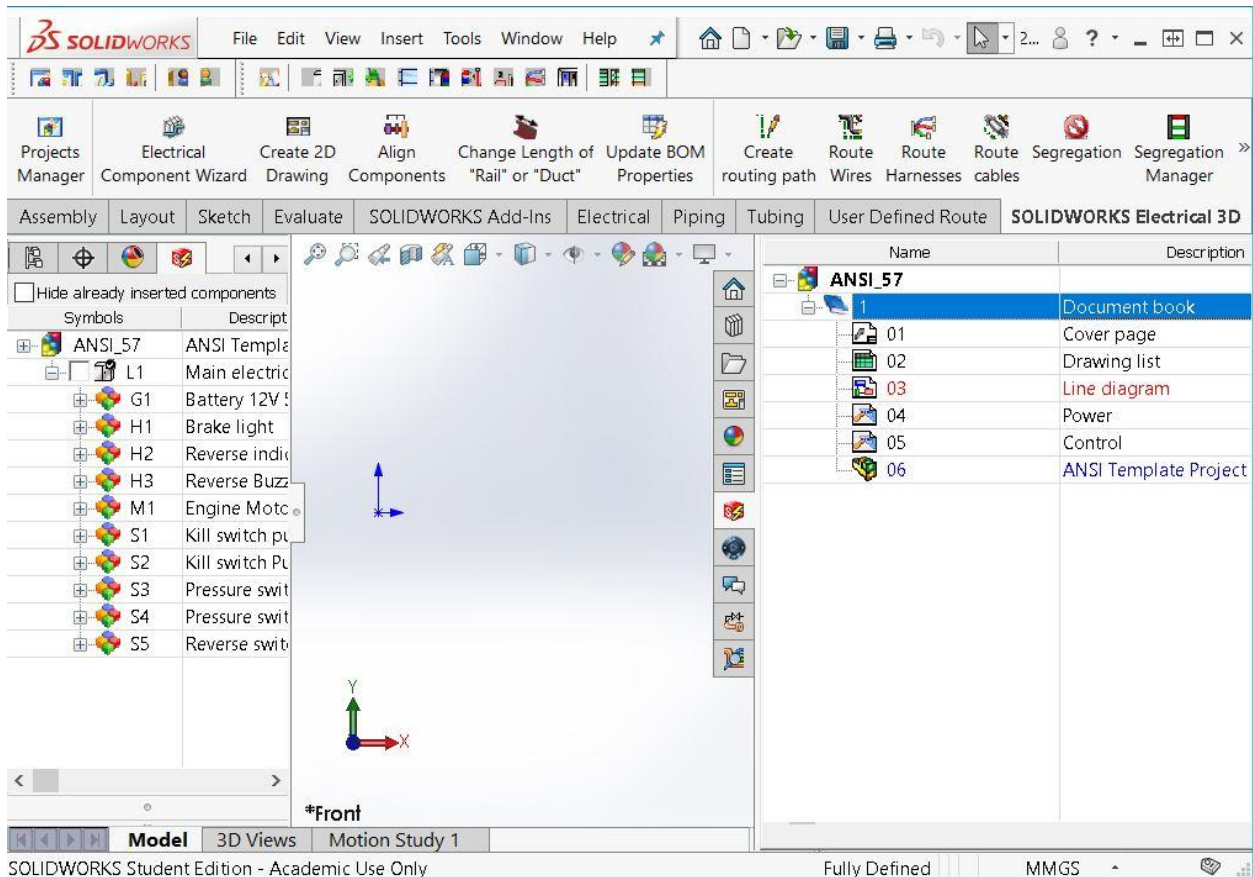


- Click 'OK' to generate SOLIDWORKS 3D assembly files
- 'ANSI Template project' appears on the document tree
- It opens the project as shown
- Double click on the 'ANSI Template project' to open SOLIDWORKS



You can open the electrical project from SOLIDWORKS

- Open 'SOLIDWORKS'
- Go to 'Tools' → 'SOLIDWORKS Electrical' → 'Projects manager' → 'SAE BAJA ELECTRIC' → 'Open'
- On the right side you can see your electrical project. Expand the document tree and open the 'ANSI Template project'



### Insert/ Associate the electrical components to 3D parts

- Open the Electrical manager and expand the project to see list of electrical components present in the project. You can see it on the left side on the above picture
- Note: If the electrical manager does not appear, close the assembly and open it back up from clicking on the “ANSI Template Project” from within SOLIDWORKS. This can be seen on the right in the picture above
- Go to ‘Assembly’ tab and click on ‘Insert components’ to insert the assembly file
- Open ‘All components\_assembly.SLDASM’ file from the ‘SW\_CADPARTS’ folder
- You can hide the origins of each part to make the view better
- You can Insert/Associate the electrical components in multiple ways
- Since we have the assembly ready we can associate the electrical components to the 3D parts by following process:
- In the ‘Electrical manager’, right click the ‘Battery 12V 5A’ and click on ‘Associate’
- ‘Associate component’ command window opens up. Expand the assembly tree and select ‘Battery’, click ‘Ok’ to associate battery to 3D part
- Similarly associate all the electrical components to their respective 3D parts
- You can check mark ‘Hide already inserted components’ to avoid confusion of adding the same component to multiple parts

## Lesson 6: Routing Cables

- Routing cables
- Create Routing Paths
- Route cables
- Flattening the route
- Adding the Bill Of Materials(BOM) to the drawing

## Routing Cables

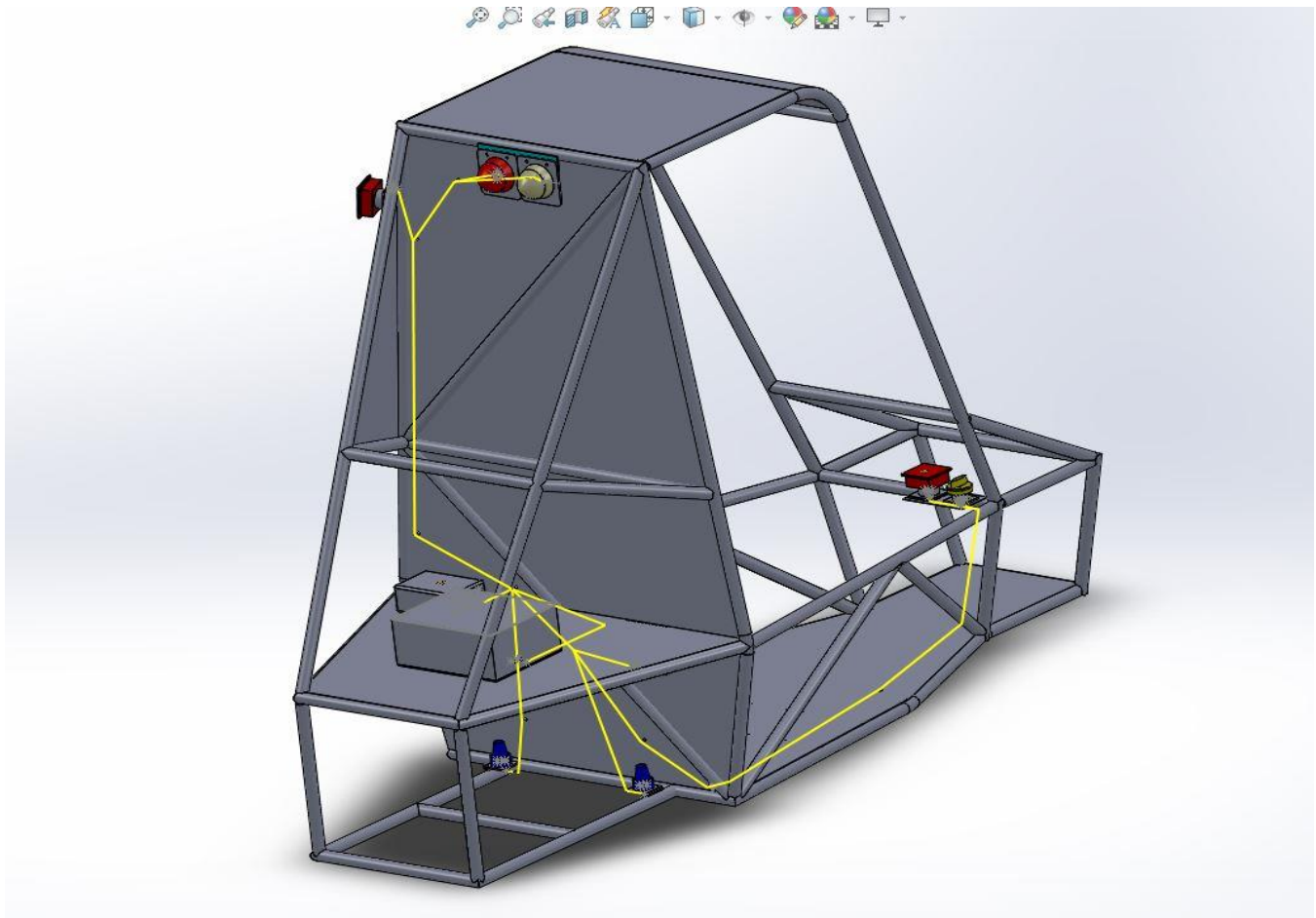
Cables can be automatically routed between parts in an assembly where specific criteria are met.

- 3D parts must be associated to SOLIDWORKS Electrical components
- The Electrical component must have detailed cabling connections in SOLIDWORKS Electrical Schematic
- The 3D part must have CPoint and Cable CPoint with a naming convention that matches the components circuit and terminal data
- Sketch paths with a specific naming convention must be used
- Routing parameters specified must allow the program to locate paths and component connection points



## Create Routing paths

- The Routing path is used to guide the route of the cables through the assembly.
- Sketch names must include 'EW\_PATH' to be recognized as a path when routing. Names like EW\_PATH1, EW\_PATH2 are valid
- Go to 'SOLIDWORKS Electrical 3D' → 'Create Routing Path' → 'Create sketch' → 'OK'
- Create a 3D sketch using the 'EW Cable\_Point' of the Electrical 3D parts, use lines or splines to create the paths of the cables
- **This is the most important part in the project.** Create the 3D sketch to route the cables
- If the sketch is not feasible ( Over defined ) or has any errors, SOLIDWORKS cannot generate the cables in that region(between those components)
- One of the ways is shown



## Route Cables:

Go to 'SOLIDWORKS Electrical 3D' tab, click on 'Route cables'

- Route cables property manager opens and shows the following options for routing cables.

### Routing analysis:

- Select the option 'Show errors', to show the errors while routing cables

### Select Route type

- **SOLIDWORKS Route:** This option is used to create the finished route including the routing sub-assemblies and physical cable parts
  - This will take a while depending upon the complexity of the electrical system
- **3D sketch Route:** This option is used to create a quick preview of the route using sketch geometry

### Renderer type:

- Splines and lines are the two main renderer types used in SOLIDWORKS

### Routing parameters:

- The Routing parameters are available regardless of the type of route being created. They allow you to define the distances the program will analyze to locate sketch paths and CPoint (0\_0)
- Routing parameters can mean a connection point cannot locate certain EW\_PATH sketches

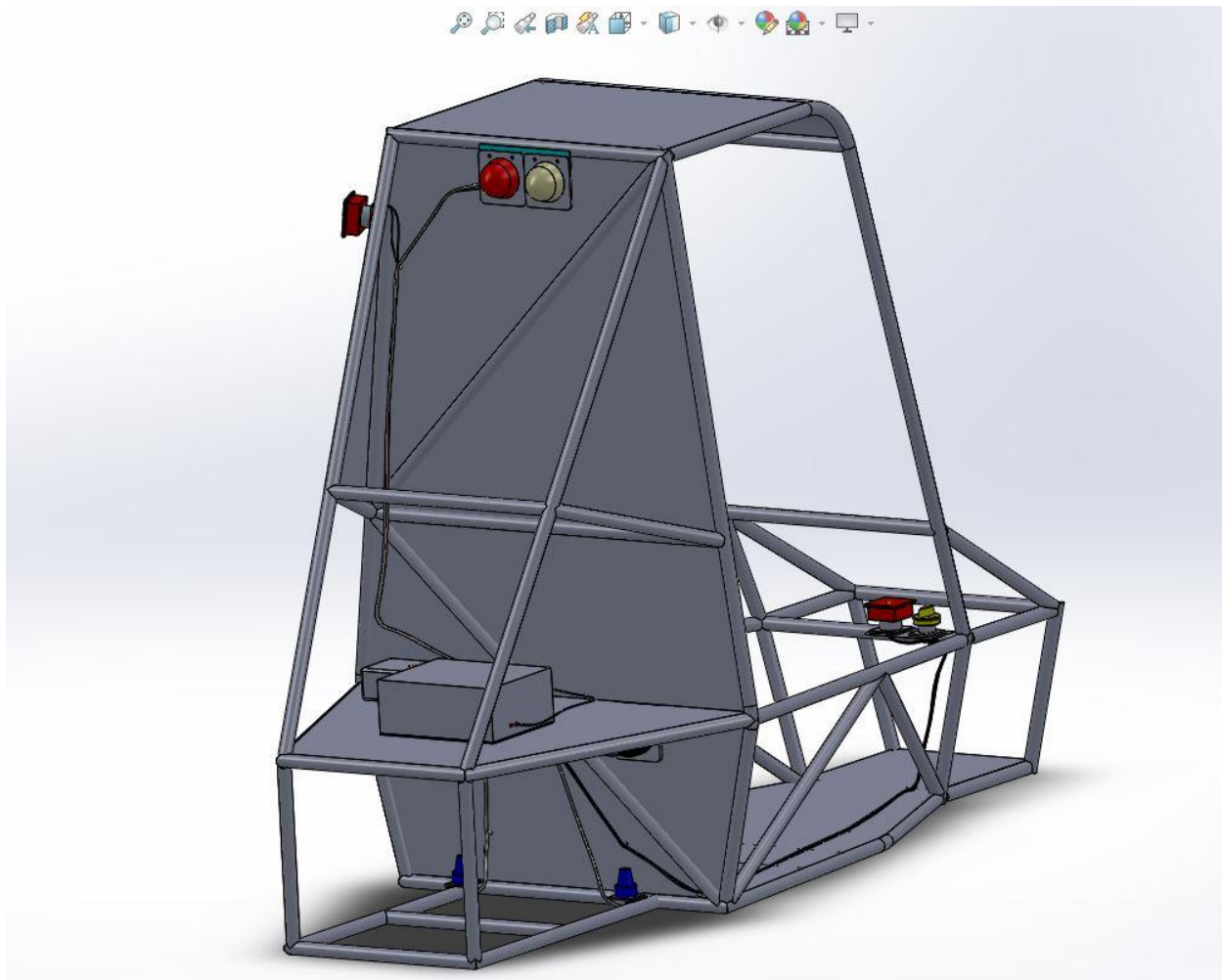
The screenshot shows the 'Route cables' property manager dialog box. It has a title bar with a green checkmark, a red X, and a help icon. The dialog is divided into several sections:

- Routing Analysis:** Contains a checked checkbox for 'Show errors'.
- Select route type:** Contains two radio buttons: 'SOLIDWORKS Route' (selected) and '3DSketch Route'. Below them are two unchecked checkboxes: 'Update origin / destination' and 'Cable core follow routing path'.
- Select renderer type:** Contains two radio buttons: 'Use splines' (selected) and 'Use lines'. Below them is a checked checkbox for 'Add Tangency'.
- Components to route:** Contains two radio buttons: 'All components' (selected) and 'Selected components'.
- Routing parameters:** Contains three input fields with icons to the left: '60.00mm', '120.00mm', and '0.50mm'. Each field has up and down arrow buttons. Below these fields are two buttons: 'Draw Graph' and 'Delete Graph'.

Route the cables by the following settings

- SOLIDWORKS Route
- Use splines
- Add tangency check
- All components
- Routing parameters 60mm—120mm—0.5mm
- Click 'OK' to generate the cables between components

The final assembly is shown below, the path of the cables changes depending upon your 3D sketch.



Go to SOLIDWORKS Electrical 3D tab, click on Update BOM (Bill of Materials) properties.

- This will automatically update the route lengths in SOLIDWORKS Electrical
- Go to **'Project'** tab. Select **'Reports'**
- Click on **'Add.'** Report configuration selector window opens, which shows the available default report templates available in SOLIDWORKS Electrical
- Select **'List of cables grouped by reference'**, **'Bill of materials grouped by manufacturer'**.
- The reports are generated as requested

Report manager: ANSI\_57

Report: Add Delete Properties Schedule Update Filter: Add filter Edit filter Generate Drawings Excel export Edit Text export XML export Book: 1 - Document book Parameter

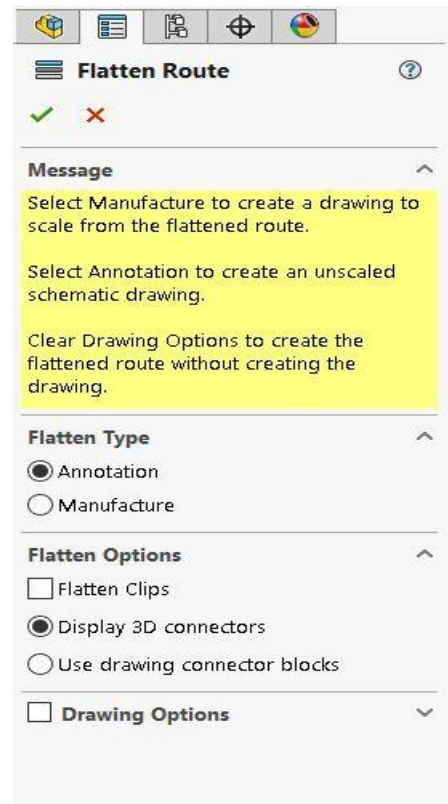
Order	Description	Filter description	Mark	Description	Path	Origin	Destination	Length (m)
1	Bill Of Materials grouped by manufac...	<no filter>	W12		L1<>L1	Main electric...	Main electric...	48.88
2			W13		L1<>L1	Main electric...	Main electric...	122.81
3			W14		L1<>L1	Main electric...	Main electric...	108.61
4			W15		L1<>L1	Main electric...	Main electric...	30.27
5			W16		L1<>L1	Main electric...	Main electric...	115.47
6			W17		L1<>L1	Main electric...	Main electric...	167.64
7			W18		L1<>L1	Main electric...	Main electric...	121.86
8			W19		L1<>L1	Main electric...	Main electric...	84.11
9			W20		L1<>L1	Main electric...	Main electric...	130.59
10			W21		L1<>L1	Main electric...	Main electric...	76.94

Close

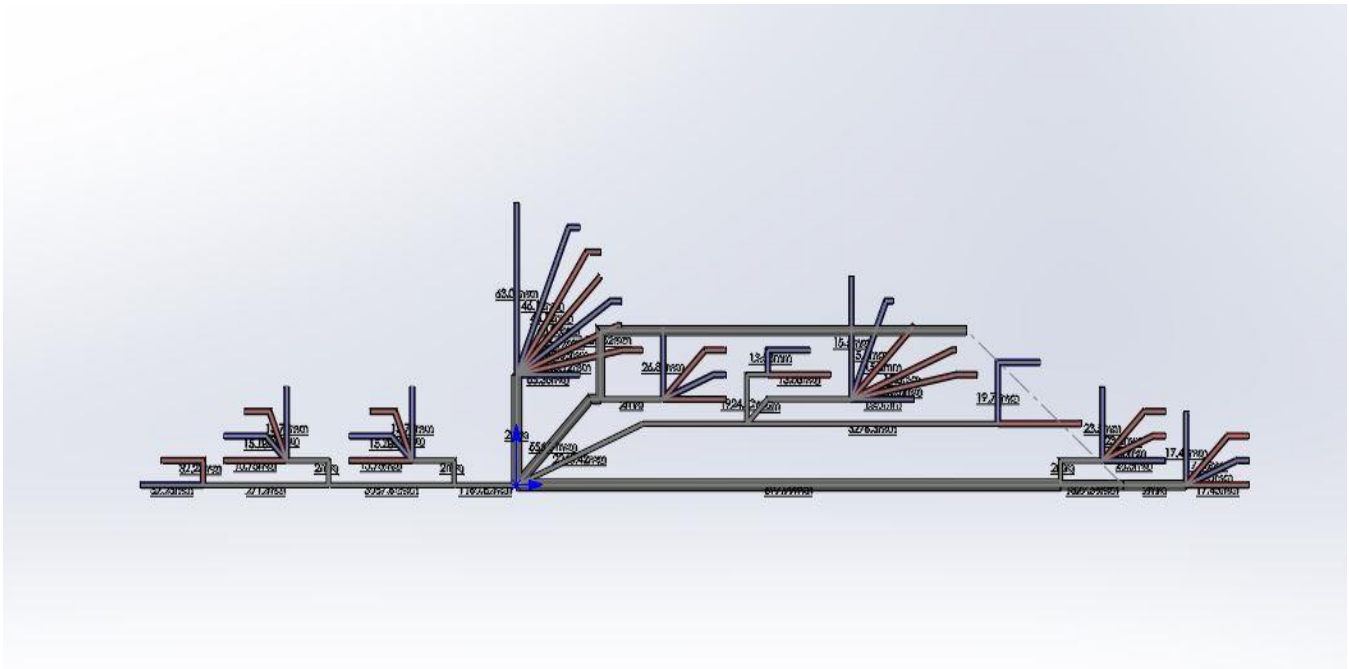
- The length of cables can be seen. Similarly, you can generate the required reports

### Flattening the Route: create a flattened (harness-board) drawing of the 3D harness model:

- The route can be flattened for use in manufacturing. Add the “SOLIDWORKS routing” add in feature to SOLIDWORKS.
- Go to ‘Tools’ -> ‘Add-Ins’ -> ‘SOLIDWORKS Routing’
- Go to ‘Electrical’ tab -> select ‘Flatten route’ option
- Flatten Route manager opens up showing the different options available
- Avoid collisions during user defined routing as it will cause errors during route flattening
- **Flatten type:**
  - **Annotation:** Simplified representation of the route as flattened configuration and drawing, lines are not true lengths but the annotations display the actual length.
  - **Manufacture:** Specifies scaled representation of the route as flatten configuration and drawing.
- Flatten options:
  - **Display 3D connectors:** Displays the 3D models of the connectors in the flattened route
  - **Use drawing connector blocks:** Displays the 2D connector blocks of the connectors in the drawing of the flattened route.
- Select the following options to create the flatten route
- Flatten type: ‘Annotation’
- Flatten option: ‘Display 3D connectors’



- Click **'OK'** (green check mark) to create the route

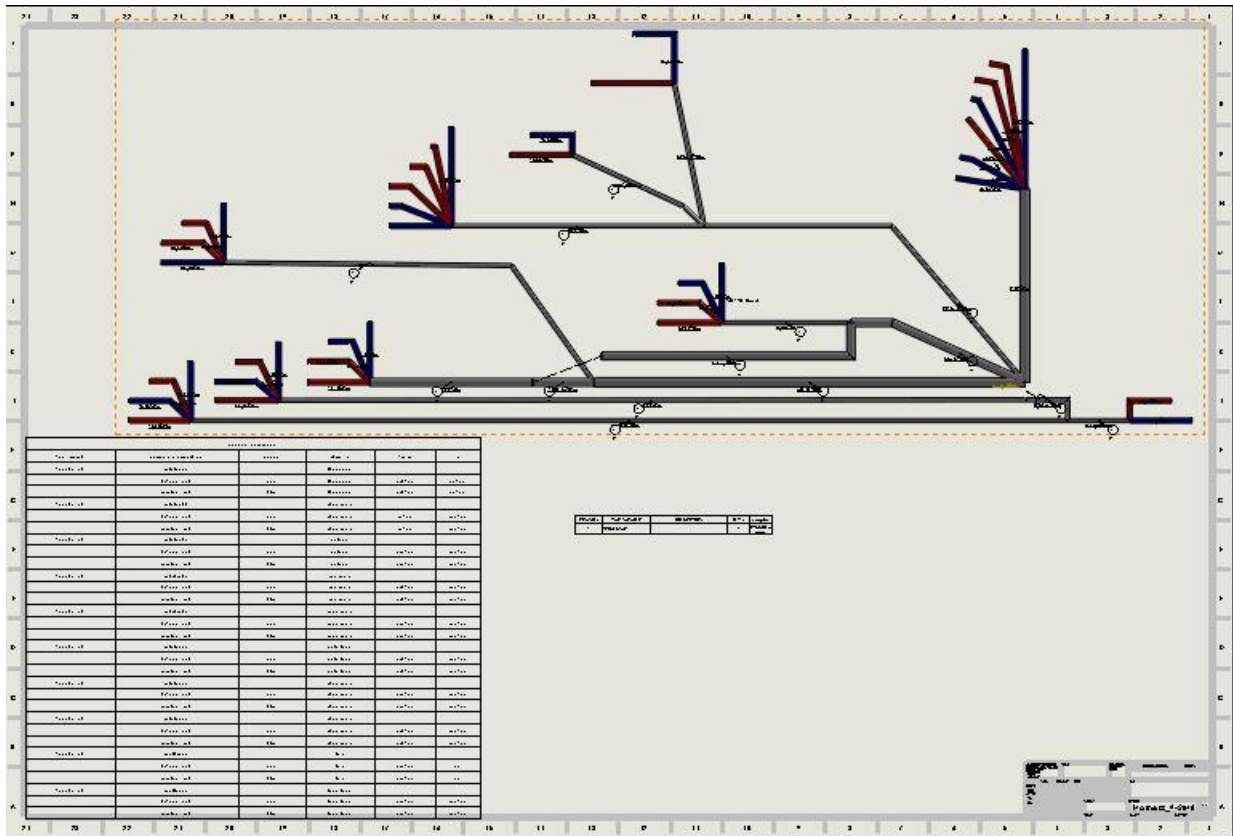


#### Edit the flatten route

- Select the route, right click and select **'Edit flattened route'**
- You can edit the route by default options like **'Drag'**, **'Horizontal'**, **'Vertical'**, **'Delete relations'**, **'Fan-out'**
- Edit the route as required

#### Adding the Bill of Materials (BOM) to the drawing

- Click on **'Flatten route'** one more time to create a drawing and add the BOM tables
- Select the following options
- Sheet format : **'a0 – landscape'**
- Select the **'Electrical BOM'**, **'Cut list'**, **'Connector table'** to create the BOM tables on the drawing
- Click **'OK'** to create the drawing
- Manually arrange the tables, annotations and harness views as desired. (tip: change to a larger drawing sheet format if needed)






CIRCUIT SUMMARY			
PART NAME	CONDUCTOR-WIRE ID	COLOR	LENGTH
PROJECT_0	W12 2597		1178.4mm
	1_POSITIVE	Red	1178.4mm
	2_NEGATIVE	Blue	1178.4mm
PROJECT_0	W13 2600		3105.42mm
	1_POSITIVE	Red	3105.42mm
	2_NEGATIVE	Blue	3105.42mm
PROJECT_0	W14 2608		2744.23mm
	1_POSITIVE	Red	2744.23mm
	2_NEGATIVE	Blue	2744.23mm
PROJECT_0	W15 2613		723.5mm
	1_POSITIVE	Red	723.5mm
	2_NEGATIVE	Blue	723.5mm
PROJECT_0	W16 2618		2960.38mm
	1_POSITIVE	Red	2960.38mm
	2_NEGATIVE	Blue	2960.38mm
PROJECT_0	W17 2623		4249.07mm
	1_POSITIVE	Red	4249.07mm
	2_NEGATIVE	Blue	4249.07mm
PROJECT_0	W18 2628		3103.32mm
	1_POSITIVE	Red	3103.32mm
	2_NEGATIVE	Blue	3103.32mm
PROJECT_0	W19 2633		2062.92mm
	1_POSITIVE	Red	2062.92mm
	2_NEGATIVE	Blue	2062.92mm
PROJECT_0	W20 2638		3296.07mm
	1_POSITIVE	Red	3296.07mm
	2_NEGATIVE	Blue	3296.07mm
PROJECT_0	W21 2643		1937.69mm
	1_POSITIVE	Red	1937.69mm
	2_NEGATIVE	Blue	1937.69mm

- Circuit summary table shows the lengths of the cables used and their respective lengths



- Go to 'SOLIDWORKS Electrical Drawing' -> select 'Create Project Drawing' to add the drawing sheet created to the SOLIDWORKS Electrical project

	1	Document book
	01	Cover page
	02	Drawing list
	03	Line diagram
	04	Power
	05	Control
	<b>06</b>	<b>ANSI Template Project</b>
	07	Main electrical closet
	<b>08</b>	<b>ANSI Template Project</b>

Congratulations! You have successfully completed SAE BAJA Electrical project using SOLIDWORKS Electrical 2D and SOLIDWORKS Electrical 3D