

SOLIDWORKS



Getting Started Guide

SOLIDWORKS Electrical FIRST Robotics Edition

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1/2/2015

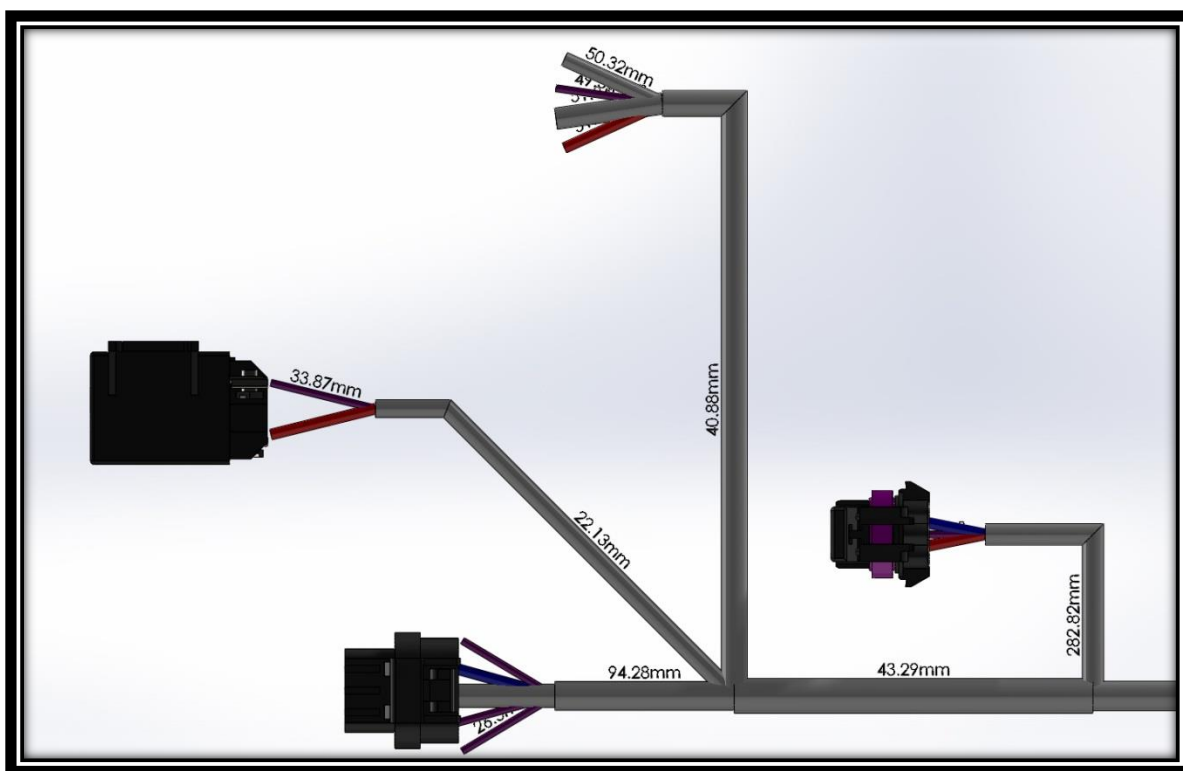
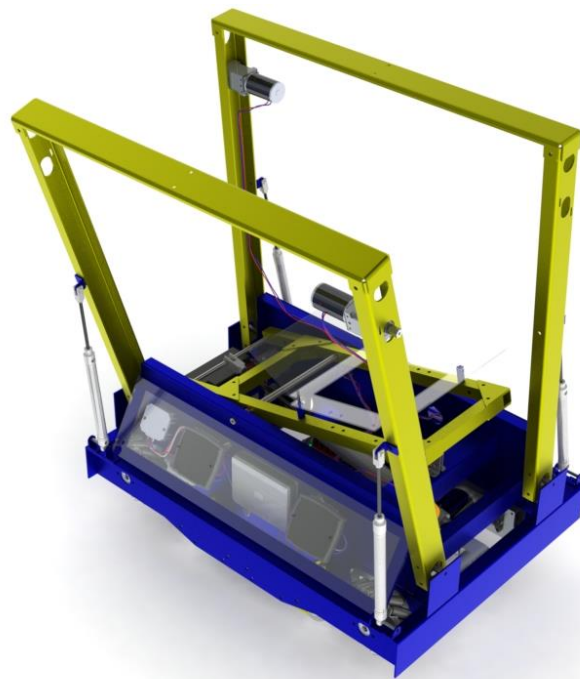


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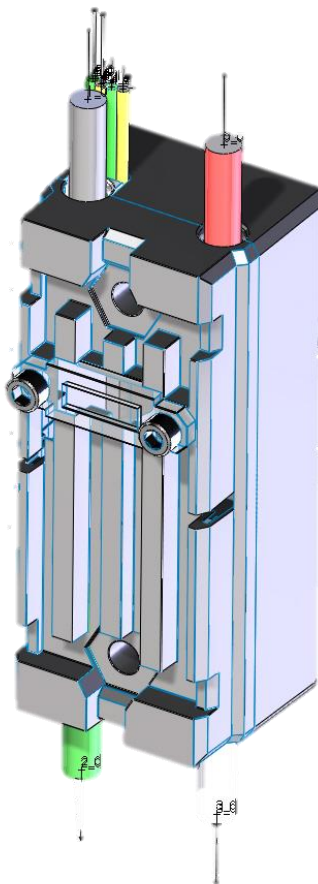


INTRODUCTION

The purpose of this document is to teach new users the fundamentals of SOLIDWORKS Electrical and assist in the creation of new projects using SOLIDWORKS Electrical.

What is SOLIDWORKS Electrical?

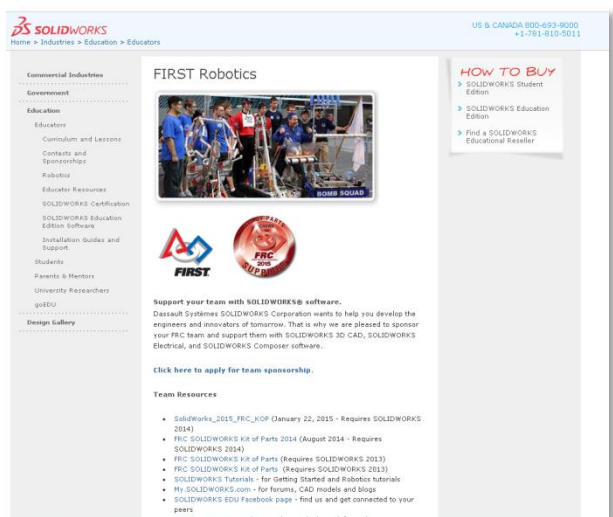
SOLIDWORKS Electrical is an add-on program which allows you to design and create documentation for your electrical schematics which can then be routed in SolidWorks3D CAD. This allows for added intelligence and design intent in the creation of new systems and assemblies.



Learning Resources

Online Help and Support

You can learn more about SOLIDWORKS Electrical from our [YouTube Page](#) and from our [Facebook Page](#). You can also visit the [SOLIDWORKS Electrical Website](#). If you have any question you can leave a post on our [SOLIDWORKS Teacher Blog](#). You can also use the SolidWorks FIRST Tutorials



Setting Up SOLIDWORKS Electrical

To get SOLIDWORKS Electrical for your team, you first need to apply to SolidWorks for sponsorship. Once you apply, SolidWorks will let you have access to the following programs:

- SOLIDWORKS 3D CAD (SOLIDWORKS)
- SOLIDWORKS Electrical
- SOLIDWORKS Simulation
- SOLIDWORKS Composer

FIRST Robotics



Support your team with SOLIDWORKS® software.

Dassault Systèmes SOLIDWORKS Corporation wants to help you develop the engineers and innovators of tomorrow. That is why we are pleased to sponsor your FRC team and support them with SOLIDWORKS 3D CAD, SOLIDWORKS Electrical, and SOLIDWORKS Composer software.

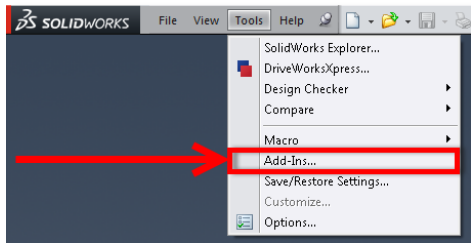
[Click here to apply for team sponsorship.](#)

How do you Apply for SolidWorks Sponsorship?

To apply for SolidWorks Sponsorship:

1. Click the **link** to the [SolidWorks FIRST Robotics Website](#).
2. Click the **SOLIDWORKS SPONSORSHIP** link and fill out the survey.
3. Once you've completed the survey, you will have to wait between 3 to 5 business days in order for SolidWorks to approve you or not.
4. Once APPROVED SolidWorks will send you codes and instruction on how to get your SolidWorks products by the email you provided.

Enabling SOLIDWORKS Electrical in SOLIDWORKS

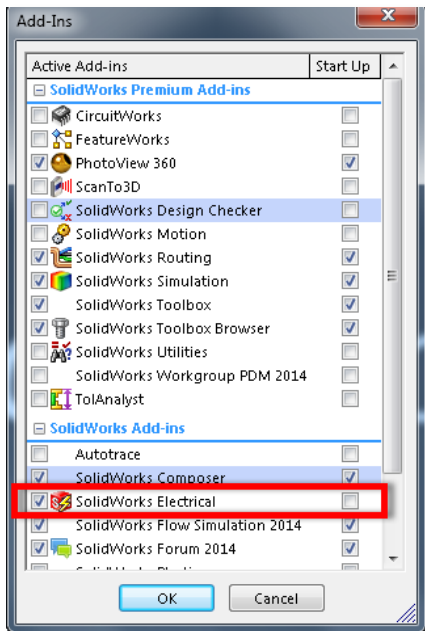


SOLIDWORKS Electrical has two parts to it:

- SOLIDWORKS Electrical 2D, which is a standalone program.
- SOLIWORKS Electrical 3D, which is an add-in in SOLIDWORKS.

To enable SOLIDWORKS Electrical 3D in SOLIDWORKS:

1. Run **SOLIDWORKS**.
2. Click **Tools > Add Ins**.
3. Check the **ACTIVE ADD-INS** check box next to **SOLIDWORKS Electrical** to active SOLIDWORKS Electrical in SOLIDWORKS.
4. Click **OK** to exit the menu and save your changes.



NOTE: You can set SOLIDWORKS Electrical to automatically run once SOLIDWORKS 3D CAD runs by checking the START UP check box next to SOLIDWORKS ELECTRICAL.

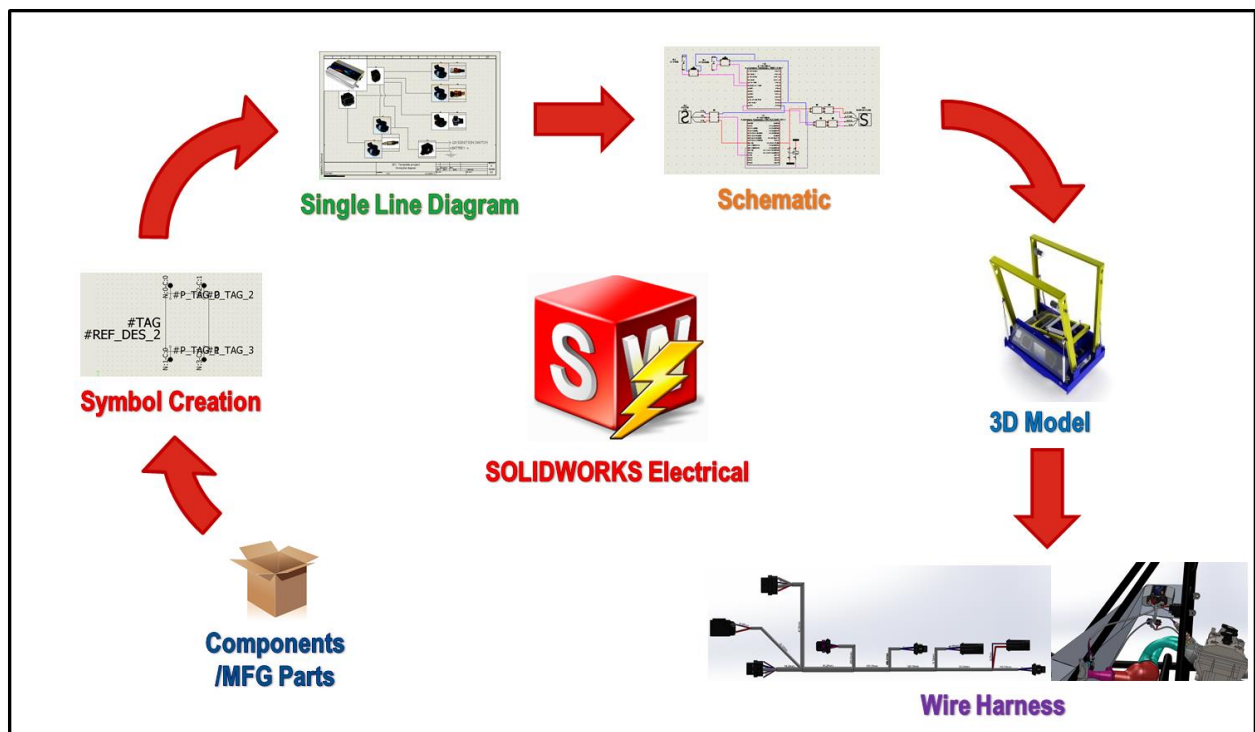
SOLIDWORKS ELECTRICAL DESIGN CYCLE

Overview

In SOLIDWORKS Electrical consists of seven main design steps, assuming you have already acquired your components and created 3D models for them:

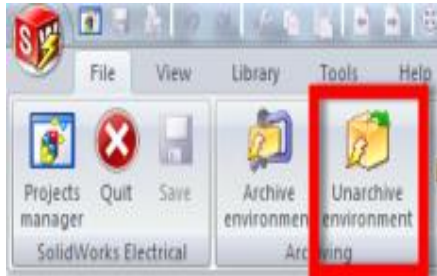
1. Creating a Library
2. Creating Manufactures Parts
3. Creating Symbols
4. Line Diagram
5. Schematics Diagram
6. 3D Representation
7. Wire Harness

By going through these steps SOLIDWORKS Electrical will help you bring your electrical design ideas to life.



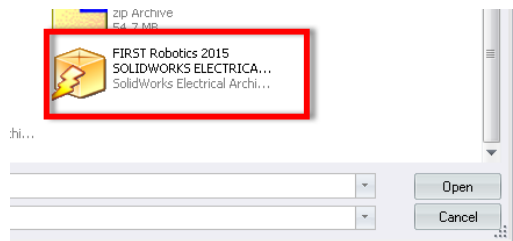
Using 2015 FIRST Robotics Electrical Environment

Importing the Environment



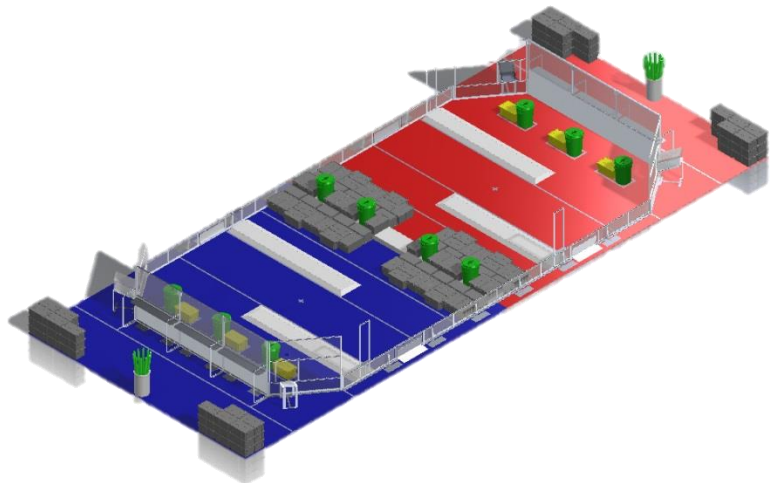
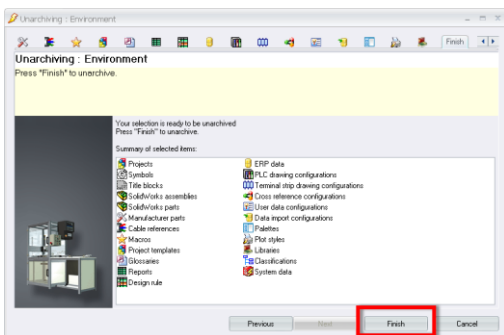
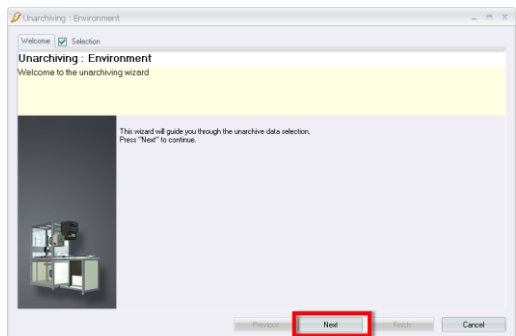
Designing a robot using SOLIDWORKS Electrical requires the proper libraries and settings. This information is contained within the FIRST Robotics 2015 SOLIDWORKS Electrical Environment.

To Import the environment, open SOLIDWORKS Electrical and select **Unarchive Environment** from the Archiving tab under the file menu.



When prompted, select **"FIRST Robotics 2015 SOLIDWORKS Electrical Environment.twezip"**

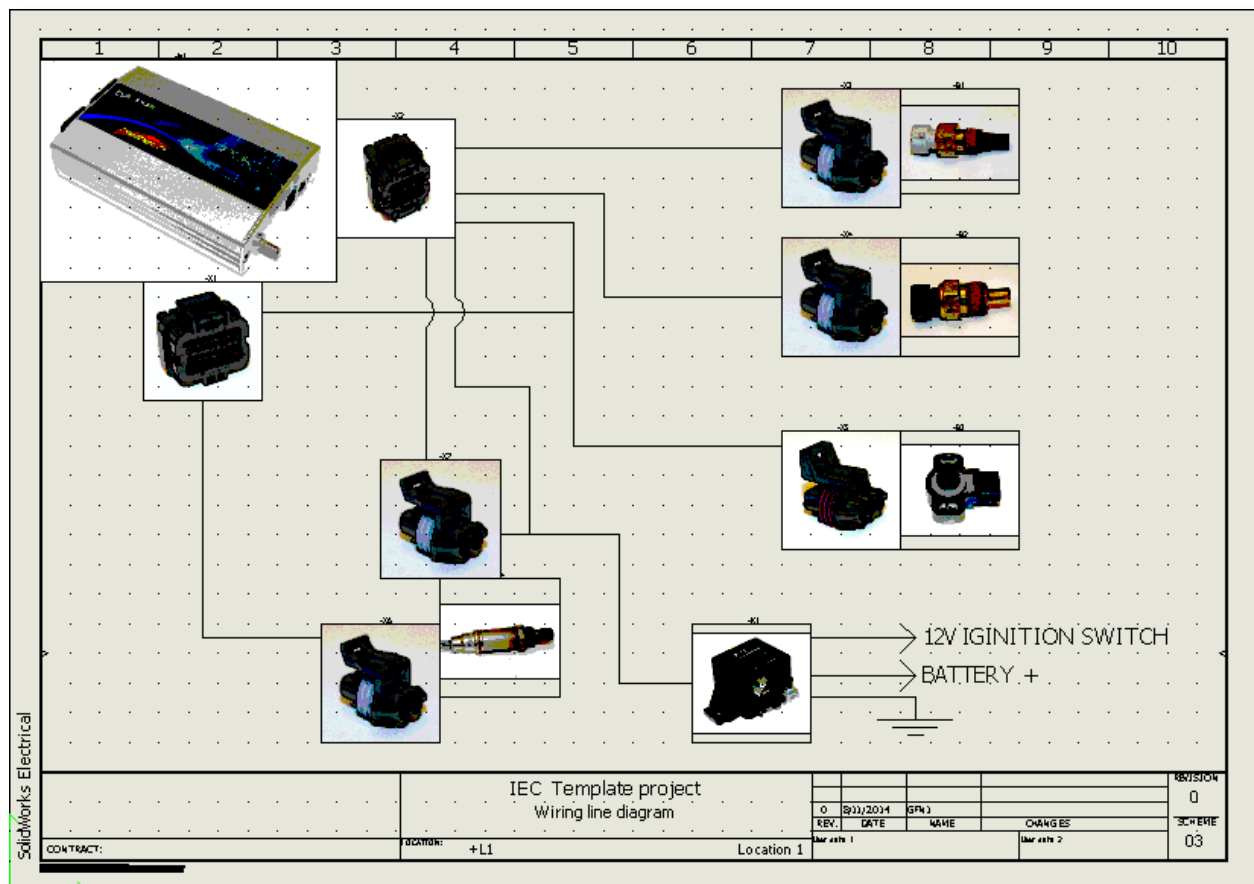
Continue through the menu and click okay at each prompt until the finish option appears. Click finish and the environment has been fully imported.



UNDERSTANDING THE WORKFLOW

Single Line Diagram

Single line diagrams provide an excellent project overview along with a quick means to create basic cable connections. Symbols are first placed that represent each major electrical component. These components can be associated to a manufacturer's part at any time. Cables are then drawn to visually connect components and optionally connect circuits.



Schematics are built for power and control circuits using symbols tied to components. Single or multiple wires can be drawn between symbols and wires are numbered and trimmed automatically. Origin and destination arrows are created easily to ensure multi-page schematics are wired correctly. SOLIDWORKS Electrical also prevents origin – destination arrows from connecting to the wrong wire type to avoid connecting ground to power. Additional schematic tools automate the placement and organization of terminals, PLCs and black box components.

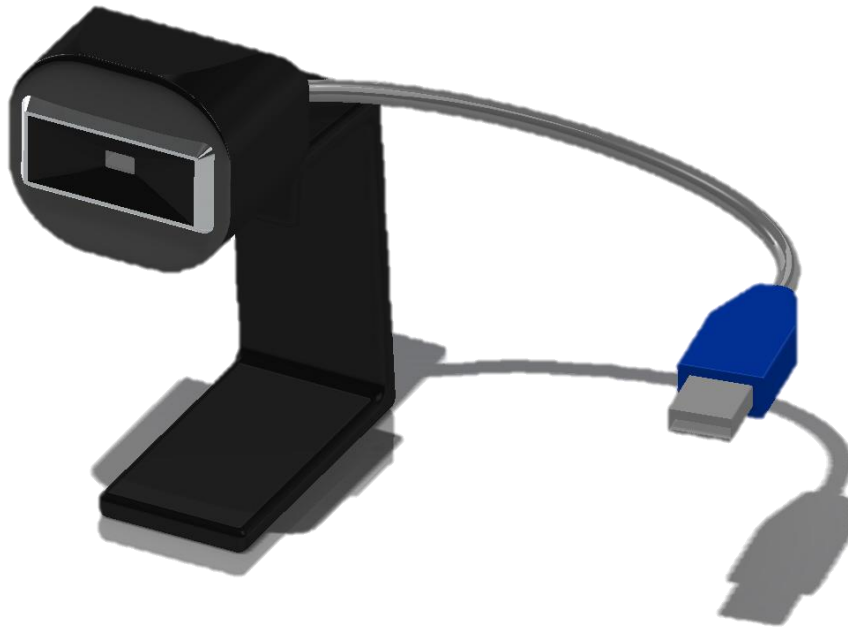
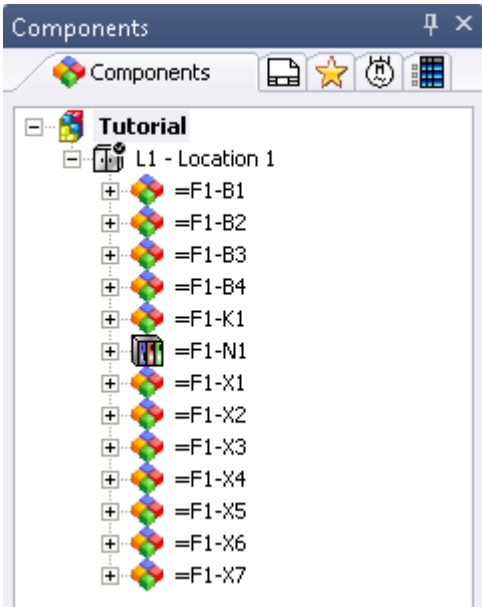


Component Tree

As symbols are added, a Component **tree** is built dynamically. Locations can be defined for the project so that components can be identified with them. This helps with documentation as well as organization. The component tree can be used in schematics and 3D to insert their corresponding symbols and 3D models.

SOLIDWORKS Electrical 3D can be used to assemble components along with their mechanical counterparts to create complete 3D models. Finally, wires and cables are routed automatically along guide lines to layout route locations as well as calculate lengths.

The symbol and manufacturer part libraries can be tailored to your needs by removing or importing symbols and parts. Parts can be added individually as they are used and are shared with all Electrical users through SQL database.



The User Interface

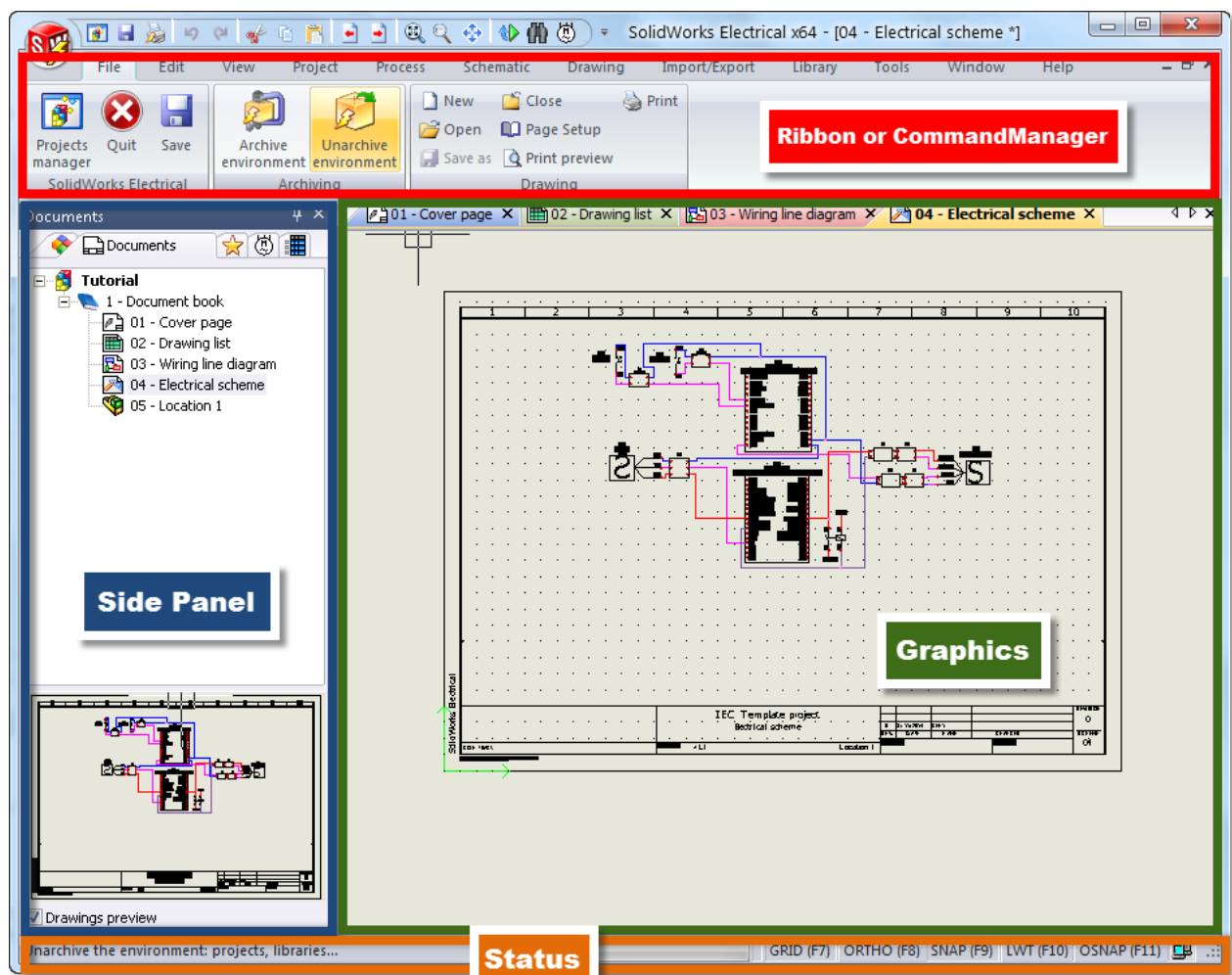
The SOLIDWORKS Electrical user interface is divided up into four main sections.

The **Side Panel** provides access to project documents as well as components, macros and symbols. It also includes a command tab and options when a command is active.

The **Ribbon**, or **Command Manager** menu, contains input commands sorted into multiple groupings (tabs).

The **Graphics Zone** is limited to drawing access and editing.

The **Status Bar** shows the cursor position and allows you to toggle modes like SNAP on and off.



PROJECTS

What are Projects?

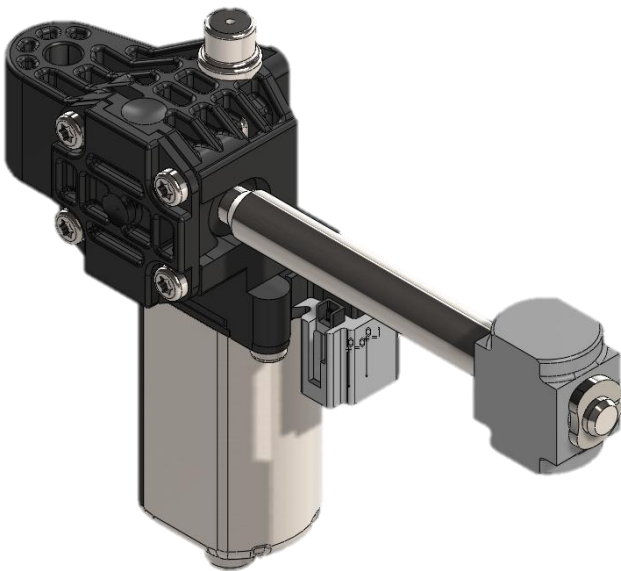
A Project is the collection of the many different types of files used to create the reports, data and other files that together fully defines the system.

A Project contains one or more document books and each book can contain multiple folders to manage the different files related to that book.

Project Elements

Some important elements of projects to know about when using SOLIDWORKS Electrical are:



- **Project:** is the name assigned to the entire project including all the books and drawing files.
- **Book:** contains all the documentation that is used to create the project including folders, drawings and associated documents. Each project can contain one or more books.
- **Folders:** contain data within the Book, and can be used to categorize data, for greater manageability.
- **Drawings:** contain different information types that go to make up the complete project data set, drawings can contain, schemes, lists, 2D cabinet layouts...
- **Cover Page:** contains a title block in which user can set attributes to display information such as the project name and description, contents of a book. The drawing tools may be used to add an image.
- **Scheme:** Scheme drawings contain circuit information related to the function of the electrical project.
- **Line diagram:** shows a simplified overview of project components interconnects and cables used to form these connections.
- **Mixed schema:** drawing allows users to mix the data style of scheme and line diagram type drawings to produce data rich hybrid designs.
- **Drawing lists:** contain a variety of project reports generated as drawings.

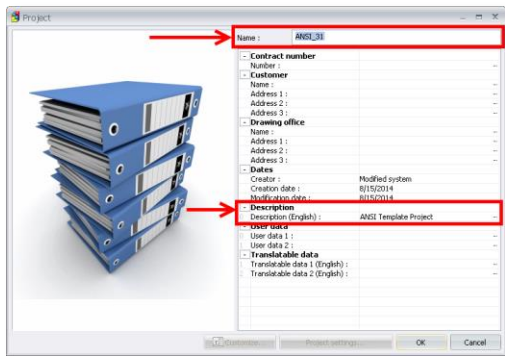
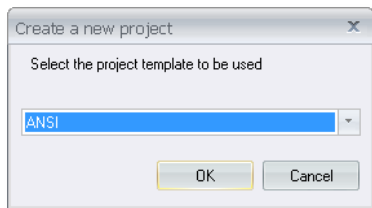


- **Terminal drawings:** are automatically created from data applied to terminals in the project drawing set and through the **Terminals editor**.
- **2D Cabinet Layouts:** are general arrangement layouts that can be created within SolidWorks Electrical Schematic to aid in the positioning of components within a machine, cabinet, installation...
- **SolidWorks Cabinet Layouts:** allows for the development of project data in a 3D environment.

Creating a New Project

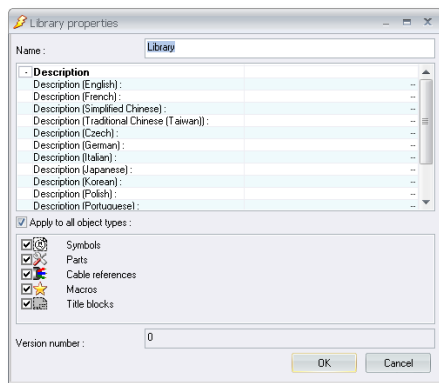
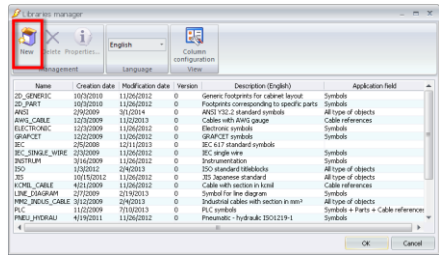
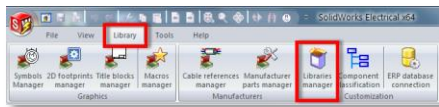
To create a New Project:

1. Start **SOLIDWORKS Electrical 2D**.
2. Click **Project Manager**  > **New Project** .
3. Select the **standard** you want to use using the dropdown menu (e.g. ANSI). Click **OK** to confirm your selection.
4. Click the **English** (or the language you want to use) and click **OK**.
5. Type in your project information. Give your project a name and a description by filling out the **Name** and **Description (English)** sections.
6. Click **OK** when done and a new project will be created.



***NOTE:** Creating a new project automatically creates multiple files of several different types. Additional drawings and other files can be added manually. The structure of a new project is based on a template. You only need to create an initial project structure and configure its settings once. It can then be saved as a template to keep all projects consistent and to speed up initial design.*

Creating a new Library



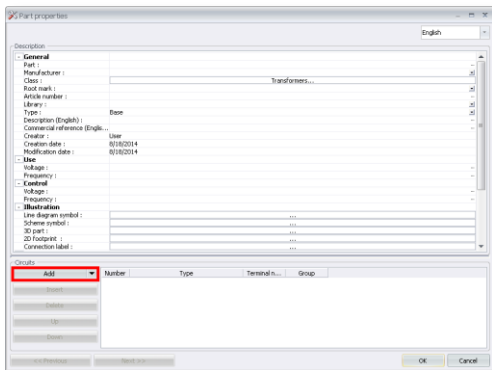
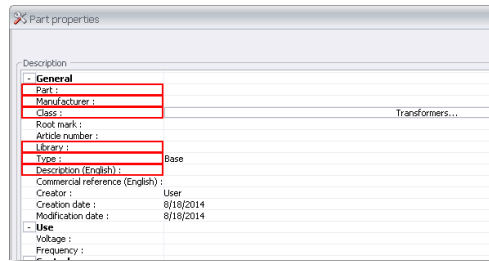
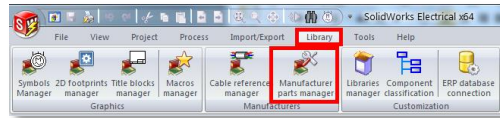
To create a new Library:


1. Run the **SOLIDWORKS Electrical 2D** standalone program.
2. The Project Manager will automatically open every time you launch SOLIDWORKS Electrical 2D. Close the **Project Manager**.
3. Click the **Library Tab > Libraries Manager**.
4. In the Libraries Manager click **New** to create a new library.
5. The Library Property Manager should open. Give your library a name and description. For this guide we will call the library **Custom**.
6. Click **OK** to save you data and create a new library.



Creating a New Manufacturer Part

To create a new manufacturer part that does not yet exist in the catalog:



1. Click the **Library** Tab > **Manufacturer parts manager**.
2. Click **Add part**  in the **Manufacturer parts manager** window.
3. Fill in the following sections with information about your component:
 - **Part:** For the part number.
 - **Manufacturer:** For the name of the manufacturer
 - **Class:** What type of component you have (Connector, Relay, Transformer...)
 - **Library:** Which library your component will be in.
 - **Type:** This should be set as **BASE**.
 - **Description (English):** The name of the part (e.g. 4 Pin Connector)
4. Click the **Add** button to start adding circuits to your component.

Circuits are what give your component life. It defines how many pins or terminals your component has and what each of those pins or terminals do.

There are four fields related to circuits:

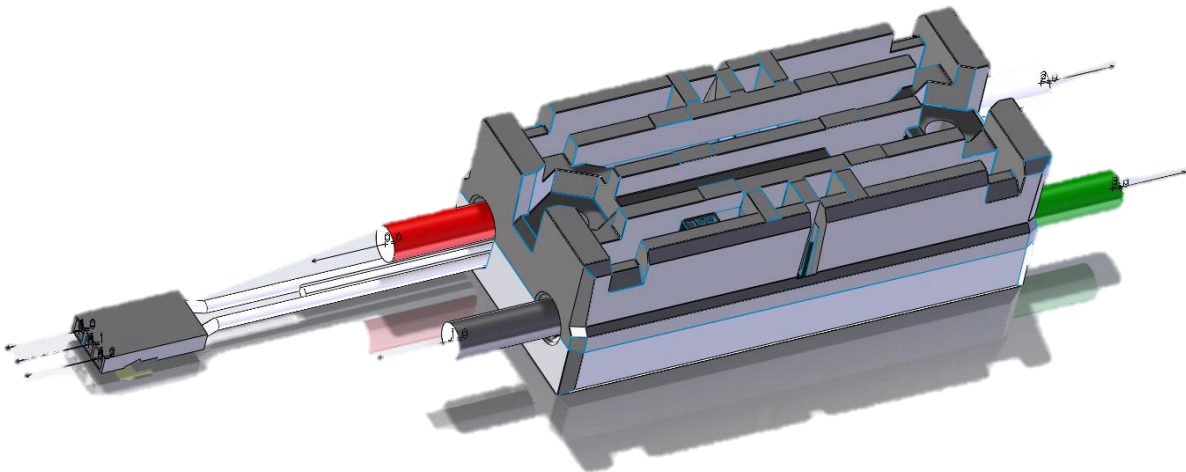
- **Number:** This is the number of circuits you have. As you add more circuits the number increases starting from zero.
- **Type:** This is the type of circuit you have. They range from a variety of type ranging from Ground, Power supply, Terminals, etc....
- **Terminal Number:** This is the number or terminals in the circuit which share the same type. You can either give it a name such as GND for or if you have multiple

Number	Type	Terminal number	Group
0	777 - Ground	GND	Ground

terminals of the same type, you number them starting from one with commas after each number (e.g. 1, 2, 3, 4...).

- **Group:** This helps you to give more information about the terminal. It is not necessary to be filled out but will help you document your project.

-
5. Fill out the information for your circuits and add more circuits as needed. Click **OK** to save your work.



SINGLE LINE DIAGRAMS

What are Single Line Diagrams?

A Line Diagram, or Single Line Diagram, is a simplified representation of the cabling that will be used in the project. It shows the original concept of the project, optionally with drawing graphics, rather than the final result. The single line diagram is a useful tool for managing components and their relationships, functionality and connectivity. Cables can be defined between components within the line diagram.

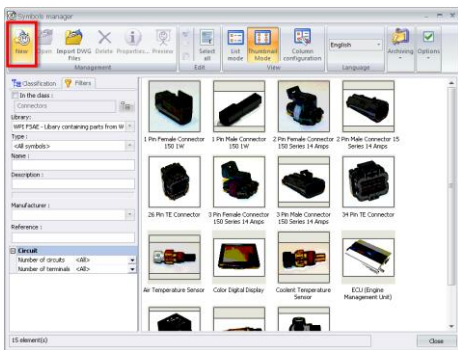
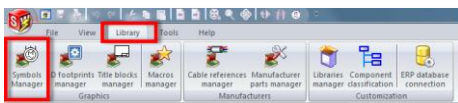
In a line diagram you use symbolic representation of your components which most often are images of the component being used. SOLIDWORKS Electrical comes with an extensive library of line diagram symbols which you can use. You can also make your own symbols for components to use in the line diagram.

Creating a Single Line Diagram Symbol

Creating the Symbol

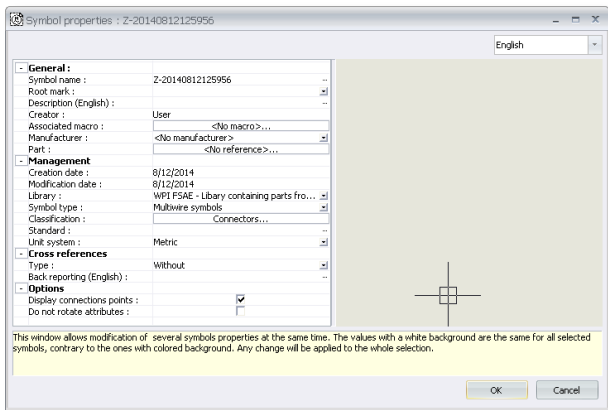
To create a Line Diagram Symbol:

1. Click the **Library Tab > Symbols Manager**.
2. Once the Symbols Manager opens, click **New** to create a new symbol.



Once you create a new symbol, the symbol properties window will open. This is where you can enter all the data about your component. The data entered has a direct impact on how the symbol will be used and how it will interact with other symbols.

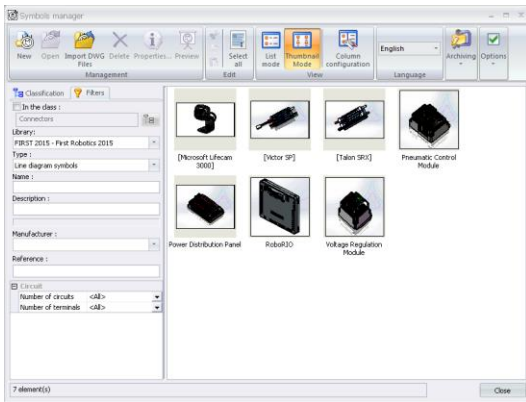
Some of the main fields available are:



- **Symbol name:** The name of the symbols DWG file.
- **Description:** This will be displayed when viewing the symbols through the Thumbnail Mode, if left blank the Symbol name will be shown.
- **Manufacturer/Part:** A manufacturer and part that will be automatically applied to the symbol on insertion.
- **Library:** The library the symbol will be associated to can be filtered at a project level to limit different symbol standards availability in project drawings.
- **Symbol type:** Defines that command that will allow access to the symbol, Multi wire symbols are available when using the **Insert Symbol** command from the **Schematic** tab, for example.
- **Classification:** Allows association of the symbol to specific classification, manufacturers parts also have classification, matching symbol and part classification assists in part assignment in scheme drawings.
- **Unit system:** Metric or Imperial, sets the symbols drawing unit system, this can affect symbol size during insertion of a mm symbol into a scheme drawing in inches.

-
3. Fill in the information associated with the component you want to use. Make sure to select **line diagram symbols** for the Symbol type and **Custom** (the library you created earlier) for the Library.
 4. Click **OK** so save the data and create a new symbol. The symbol should now appear in the Symbol Manager under the **Custom** library.

NOTE: You can search for any library by going to the Filters tab in the Symbols manager and selecting your library under the Library drop box. Make sure that the 'In the class:' box is unchecked when doing this.



Drafting the Symbol

Now that you have created the symbol you now need to create a visual representation so you know what component the symbol is representing.

To Draft your symbol:

1. Double click **your symbol** in the Symbols Manager to open the drawing window.

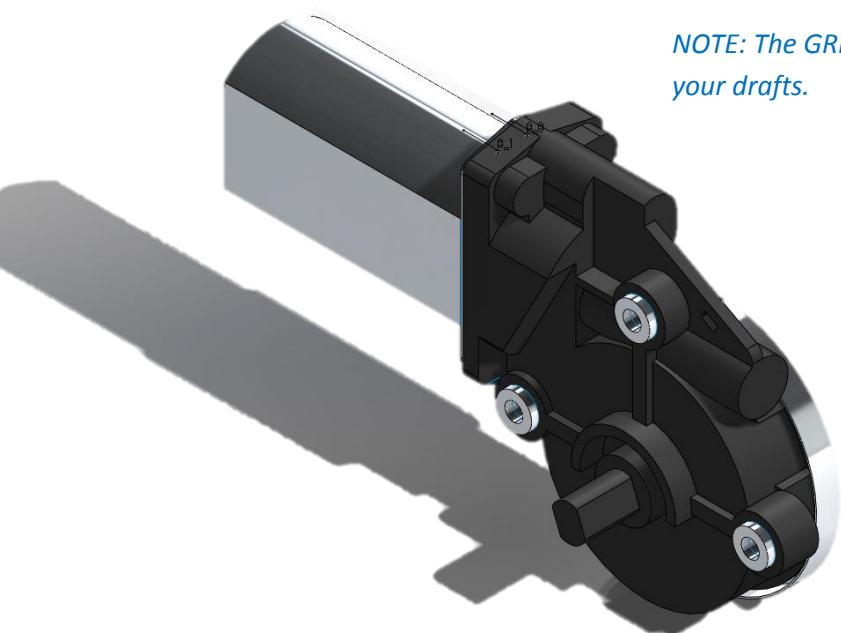
When you are in the drawing window, a new tab called **Drawing** will appear. This tab will house most of the tools you will need to make your draft.

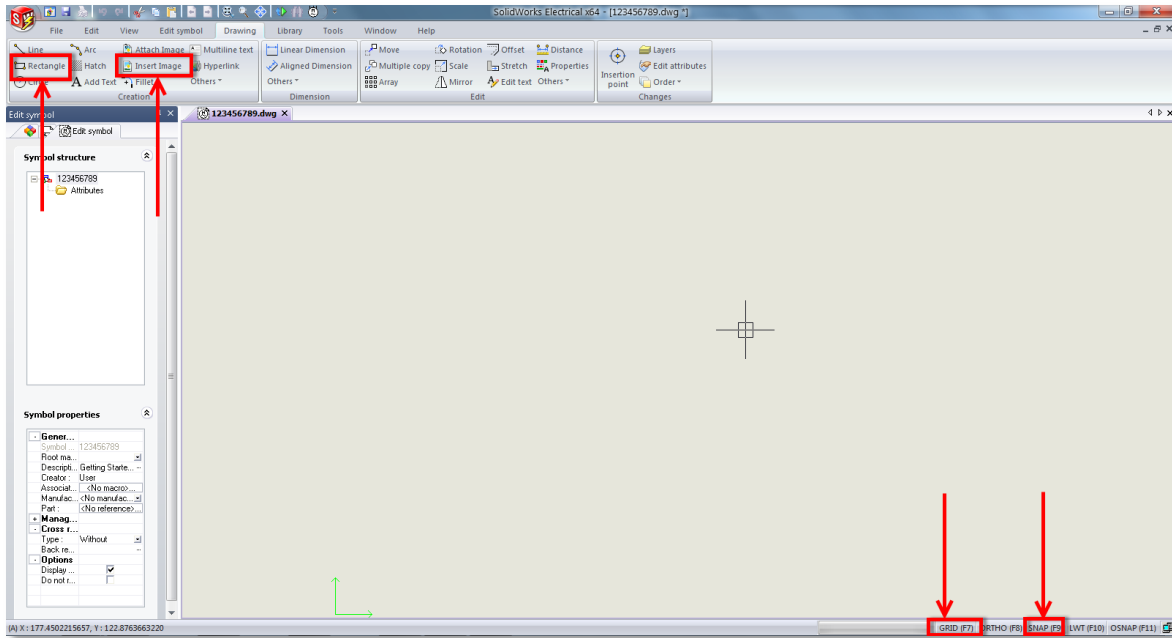
2. Click the **Drawing** tab to bring up the drawing tools.

NOTE: Normally a line diagram symbol draft consists of an image of the component you are using in real life which is inside a box. The box is needed so that program knows the boundaries of the draft.

3. Click on the **GRID** button to show the grid.
4. Click the **SNAP** button to turn snap on.

NOTE: The GRID and SNAP tools will make it easier to create your drafts.

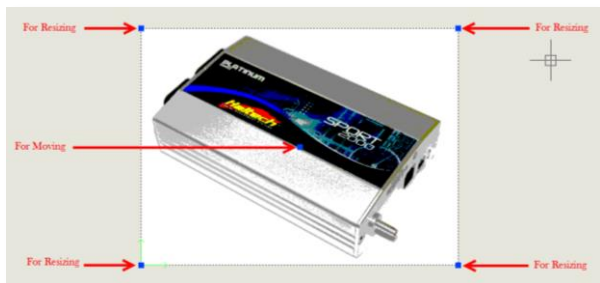




TIP: You can right click on either the GRID or SNAP button to adjust the grid, grid spacing and snap distance.

- Click **Insert Image** and browse for the image you are going to use. Select the image click **Open** to confirm your selection. The image you select must be in bitmap form. The Insert image window should appear. Click **OK** in insert your image.

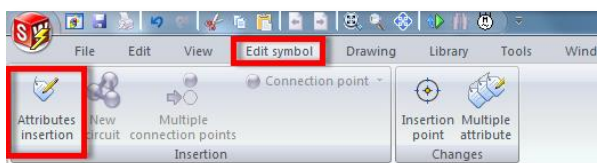
NOTE: The bitmap image should be a 16 color bitmap.

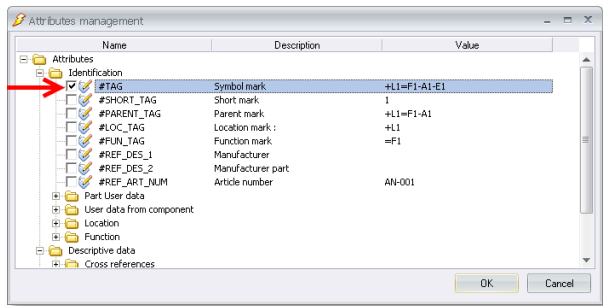


- To adjust the size and location of the image, click on **one of the corners** of the image. Five blue squares will appear on the image, four are for resizing the image and one for moving the image.
- Draw a rectangle around your image making sure that your image is within the boundary. To do this click the **Rectangle** tool. The first click is where the **upper left corner** will be and the second click will be where the **bottom right corner** will be.

Next you need to properly label your draft. To do this we have to add an attribute. Attributes label your draft with specific information such as:

- Location of component
- Manufacture's part number





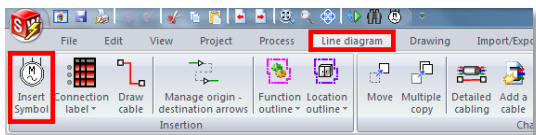
- Manufacture

8. Add an attribute by clicking **Edit symbol > Attributes insertion**.
9. Check the check box new to the **#TAG** attribute in the **Identification** folder. Click **OK** to confirm your selection.
10. Click a **location** where you want the attribute to appear.

NOTE: The location doesn't have to be inside the rectangle that you drew earlier.

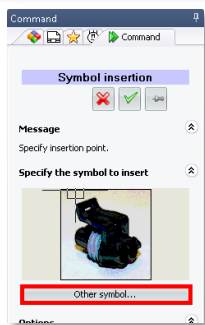
Creating a Single Line Diagram

Inserting a Symbol

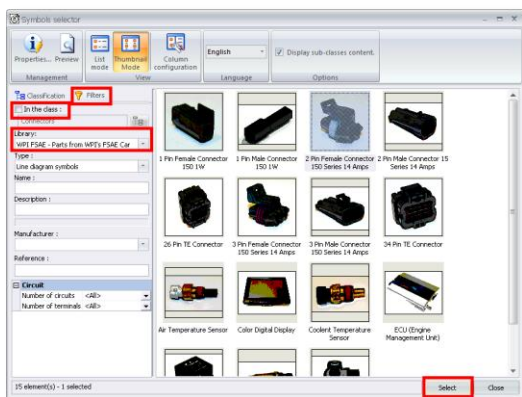


To insert a symbol into your Single Line Diagram:

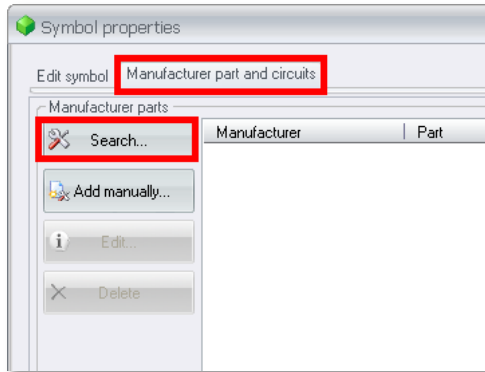
1. Open **03 – Wiring line diagram** from the project drawing list.
2. Click the **Line diagram** tab > **Insert Symbol**.
3. The **Side Panel** will change to a **Symbol insertion** menu. To pick a symbol to insert click the **Other symbol...** button which will bring up the **Symbol selector**.



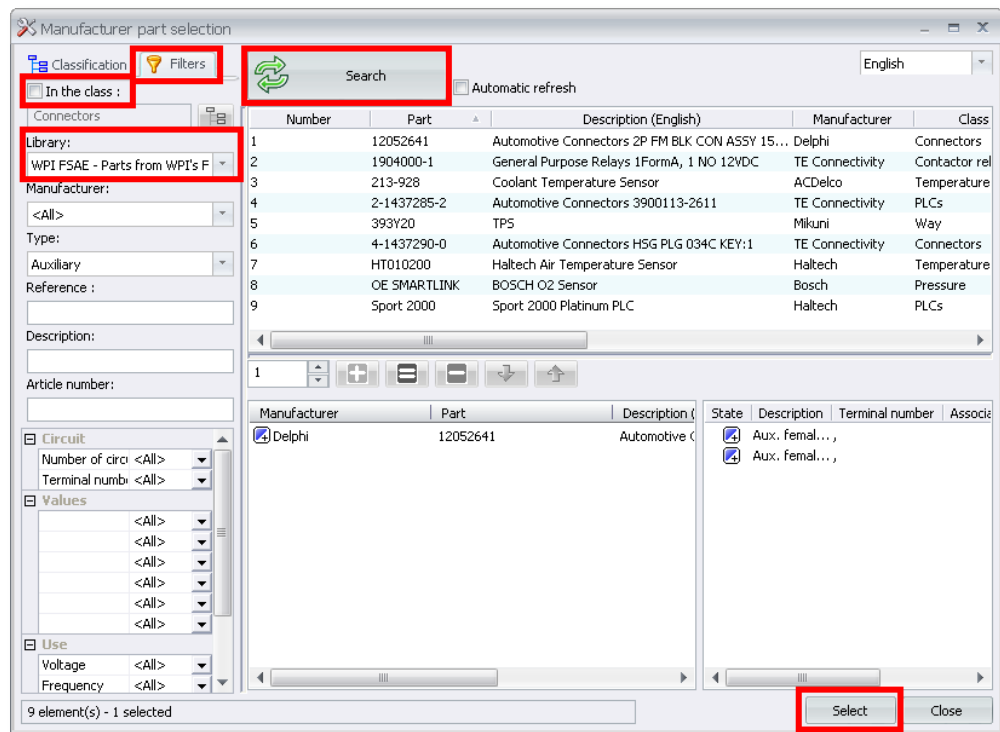
NOTE: The Symbol selector is where you can find any symbol that SOLIDWORKS Electrical has in its database or any symbols you add to the database to use in your single line diagrams.



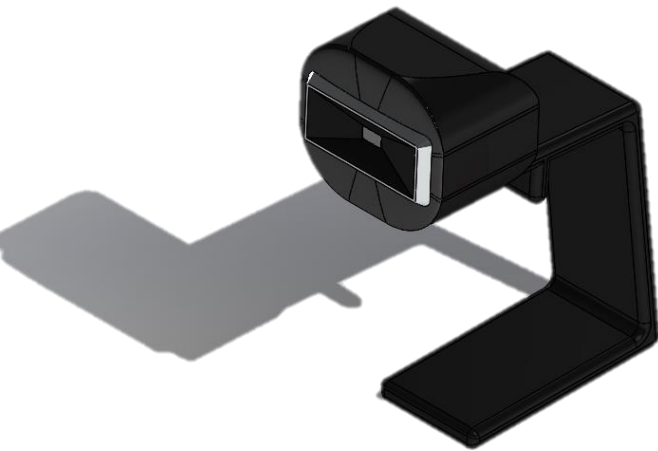
4. Click the **Filters** tab in the **Symbols selector**. Make sure that **In the class** is unchecked.
5. Select **your library** from the **Library** list to bring up all your components in your library, click **the component you want to insert**.
6. Click **Select**.
7. Place **your component** symbol where you want it on the sheet.



8. After placing a symbol, the **Symbol properties** dialog will be displayed.
9. Leave the default location and root mark.
10. Select the **Manufacturer part and circuits** tab.
11. Click **Search**.
12. In the **Filters** tab select **your library** from the **Library** list.
13. Click **Search** or enable **Automatic refresh**
14. Double-click the **component** that corresponds to **your symbol**.
15. Click **Select** to associate the manufacturer part data to the symbol.
16. Click **OK**.



Sketching Wires



Since we are using the Single Line Diagram as a representation of our wire connections, we can sketch simple lines to illustrate the connections.

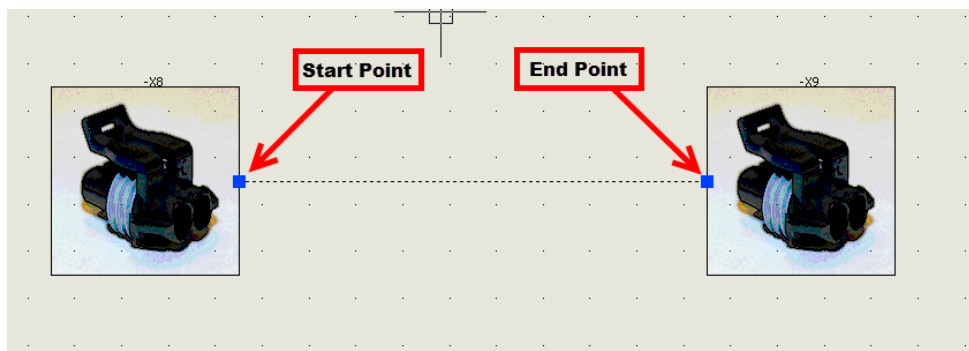
The two sketch tools you will mostly need the most to draw your wire representations are:

- **Line Tool** .
- **Arc Tool** .

Sketching a Line

To sketch a Line:

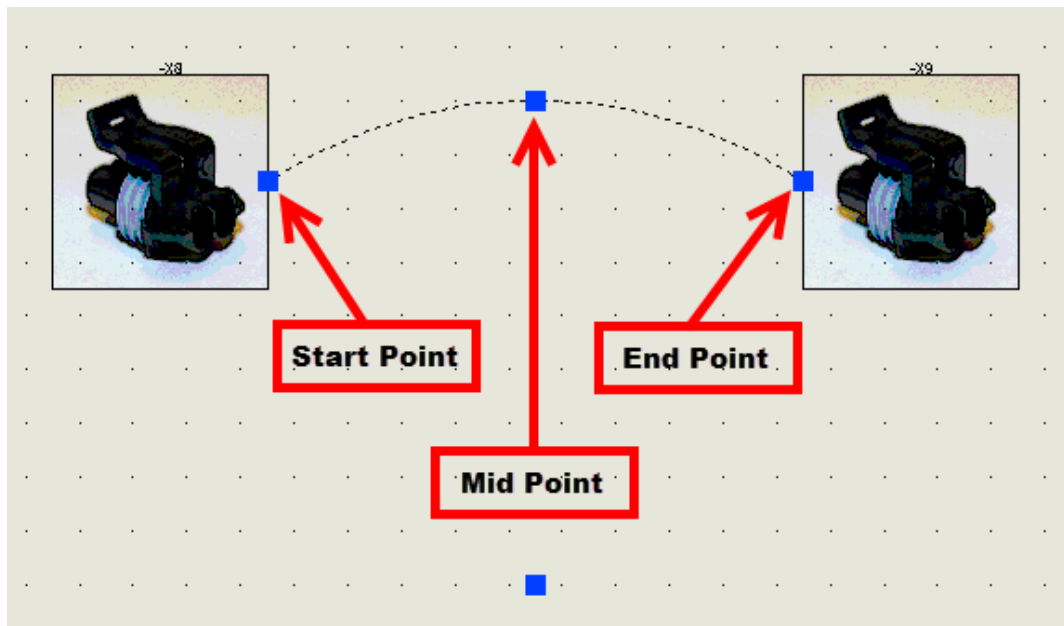
1. From the **Drawing** tab, click the **Line** tool.
2. Click where you would like the line to **start**.
3. Move your **mouse** and click where you would like the line to **end**.
4. You can keep clicking to **continue the line**. Once you are finished creating your line click the **Esc** key to stop drawing lines.



Sketching an Arc

To sketch an Arc:

1. From the **Drawing** tab, click the **Arc** tool.
2. Click where you would like the arc to **start**.
3. Move your **mouse** and click where you would like the **midpoint of the arc** to be.
4. Click a **third time** where you would like the **end point of the arc** to be.



SCHEMATICS

What are Schematics?

A **Schematic** diagram is used to show the electrical components and their electrical connections. Schematics may appear in one or more of the drawings of the book.

Like Single Line Diagrams, SOLIDWORKS Electrical comes with an extensive library of schematic symbols which you can use and you can also make your own symbols to use.

Creating a Schematics Symbol

Creating the Symbol

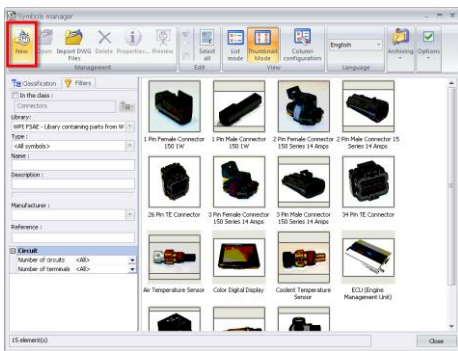
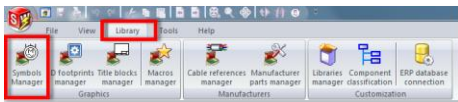
To create a Schematic Symbol:

1. Click the **Library Tab > Symbols Manager**.
2. Once the Symbols Manager opens, click **New** to create a new symbol. The Symbol properties window will open.

NOTE: See page 15 to view a description of the different data properties for the symbols.

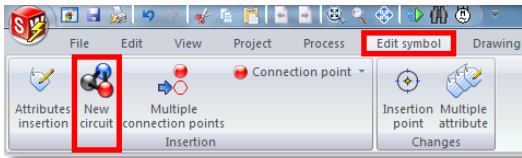
NOTE: For this section we will be creating a simple 2 Pin Female Connector.

3. Fill in the information associated with the component you want to use. Make sure to select **Multiwire symbols** for the Symbol type and **Custom** (the library you created earlier) for the Library.
4. Click **<No reference>** under the **Part** field.
5. The Manufacturer part selection window will appear. Make sure **In the class** is **unchecked**. Select **Custom** for **Library**. Click **Search**.



- Double-click **the component** (the 2 Pin Female Connector) you are creating the Symbol for.
- Click **OK** to create the symbol.

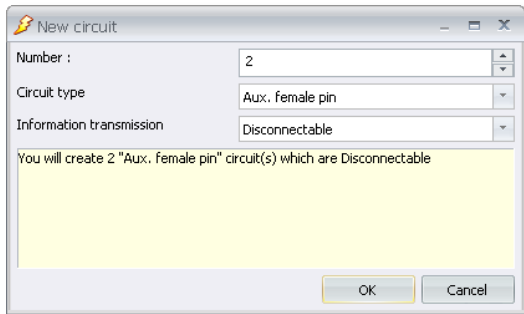
Drafting a Symbols



To Draft your symbol:

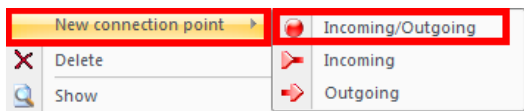
- Double click **your symbol** in the Symbols Manager to open the drawing window.
- Click on the **GRID** button to show the grid.
- Click the **SNAP** button to turn snap on.
- Set your **SNAP** and **GRID** to **1.00 mm** spacing.

TIP: You can right click on either the GRID or SNAP button to adjust the grid, grid spacing and snap distance.



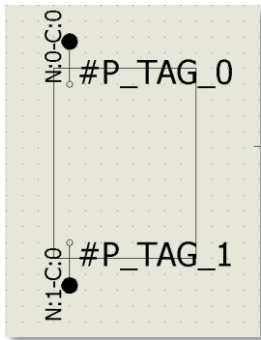
- Click the **Drawing** tab to bring up the drawing tools. Click the **Rectangle** tool.
- Draw a **12 mm X 9 mm** rectangle.
- Click **New circuit** under the **Edit Symbol** tab to add terminals to your connector.
- The **New circuit** window will open. Enter **2** for the **Number**. Select **Aux. female pin** for **Circuit type**. Make sure that you have **Disconnectable** selected for **Information transmission**.
- Click **OK**.

Since we are making a 2 Pin Connector that means we will have two different circuits attached to the connector. Each circuit will have two nodes to connect the two ends through the connector.



- Right Click **Circuit: 0 – Aux. female pin** in the **Side Panel**, **New connection point > Incoming/Outgoing**.
- Click a location in **the upper left corner** of the rectangle so that the **bigger black circle** is **outside** the rectangle while the **smaller white circle** is **inside** the rectangle.

TIP: You can right-click to rotate the node.



12. Right Click **Circuit: 0 – Aux. female pin** in the **Side Panel**, **New connection point > Incoming/Outgoing**.
13. Click a location in the **lower left corner** of the rectangle so that the **bigger black circle** is **outside** the rectangle while the **smaller white circle** is **inside** the rectangle.

Now when you use the symbol in your schematic anything attached to these two nodes will be connected to each other.

14. Repeat **steps 10-13** but making connections for **Circuit: 1 – Aux. female pin**.
15. Click **Attributes insertion** in the **Edit symbol** tab.
16. Check the boxes next to **#TAG** and **#REF_DES_2** in the **Identification** folder. Click **OK**.
17. Click a location **next to your symbol**.
18. Save and close **your symbol**.

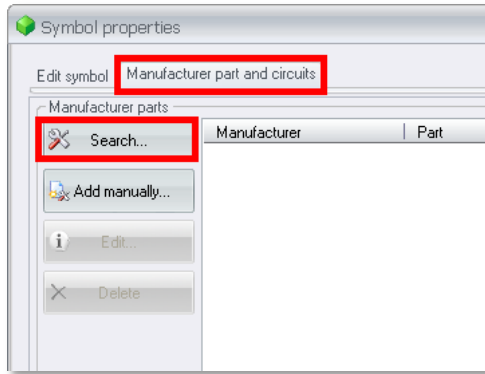
Creating a Schematic Diagram

Inserting a Symbol

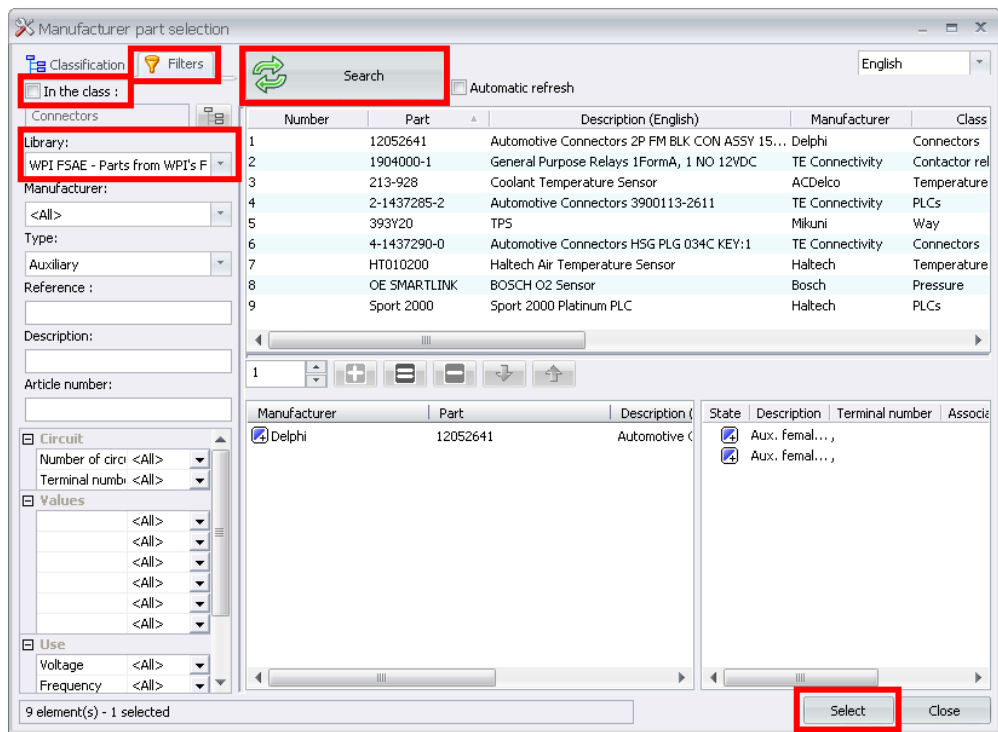


To insert a symbol into your Schematic:

1. Open **04 – Electrical scheme** from the project drawing list.
2. Click the **Schematic tab > Insert Symbol**.
3. The **Side Panel** will change to a **Symbol insertion** menu. To pick a symbol to insert click the **Other symbol...** button which will bring up the **Symbol selector**.
4. Click the **Filters** tab in the **Symbols selector**. Make sure that **In the class** is unchecked.
5. Select **your library (Custom)** from the **Library** list to bring up all your components in your library, click **the component you want to insert (2 Pin Female Connector)**.
6. Click **Select**.
7. Place **your component** symbol where you want it on the sheet.

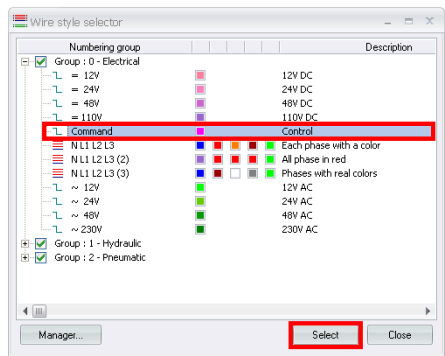
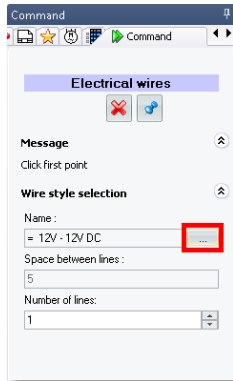
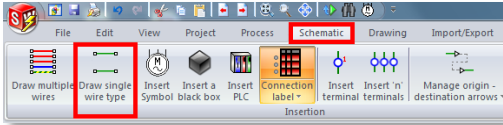


8. After placing a symbol, the **Symbol properties** dialog will be displayed.
9. Leave the default location and root mark.
10. Select the **Manufacturer part and circuits** tab.
11. Click **Search**.
12. In the **Filters** tab select **your library** from the **Library** list.
13. Click **Search** or enable **Automatic refresh**
14. Double-click the **component** that corresponds to **your symbol**.
15. Click **Select** to associate the manufacturer part data to the symbol.
16. Click **OK**.



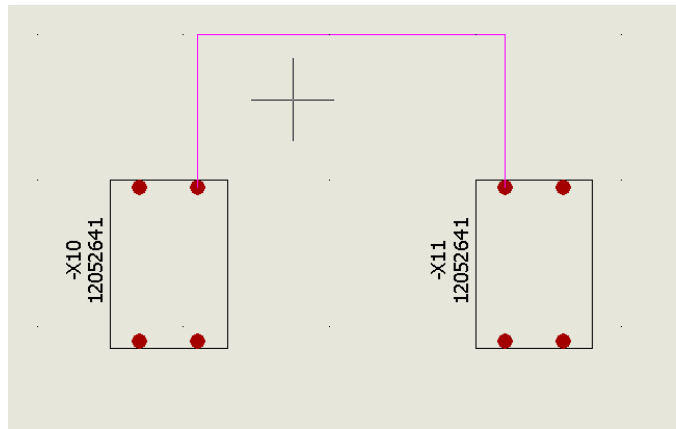
Inserting a Wire

Now that you have placed your components in your schematic, it's time to add wire connection between your components. To add wires:



Note: The wire connection will automatically end when a positive connection is made

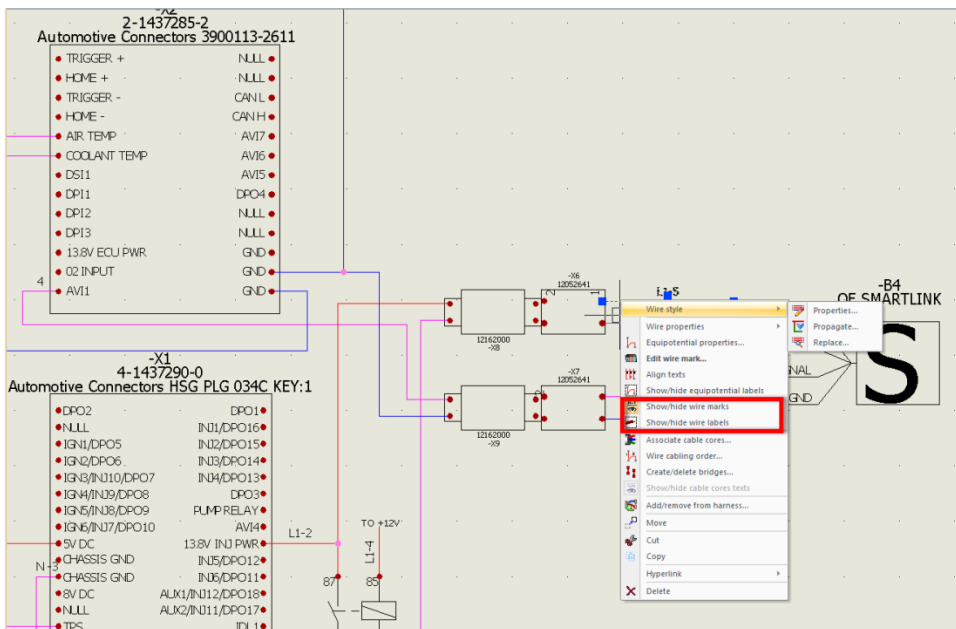
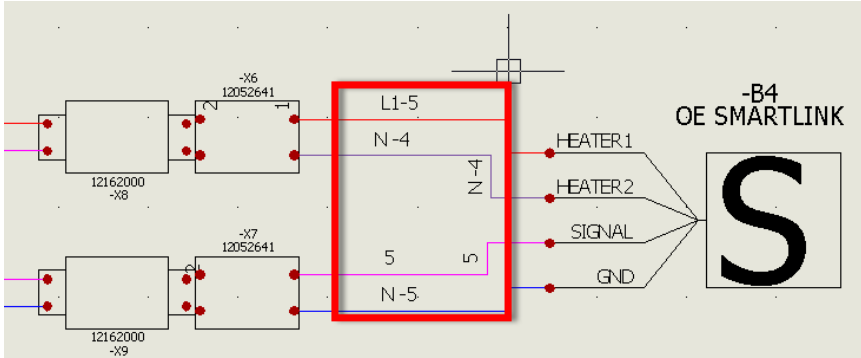
1. Click **Draw single wire type** in the **Schematic** tab.
2. The **Side Panel** should show the **Electrical wire** manager. Click the **... Browse** button.
3. The Wire style selector will open. Here you can select what type of wire you would like to use. Click the **Command** wire and then click **Select**.
4. Click on a **node of a component** to **start** your wire. You can click to **add bends** to the wire. Click on a **second node** to **end** your wire and create a connection.
5. When you are done drawing wires, press the **Esc** key on your keyboard to **stop using** the wire tool.



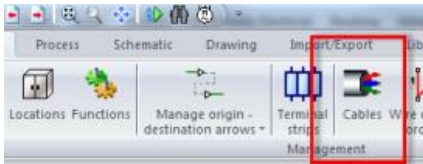


Adding Wire Numbers

SolidWorks Electrical can be configured to a variety of wire numbering conventions. These can include sheet, row and column numbers, static text, origin and destination component information among other data. These options are selected via the Wire Style Manager.

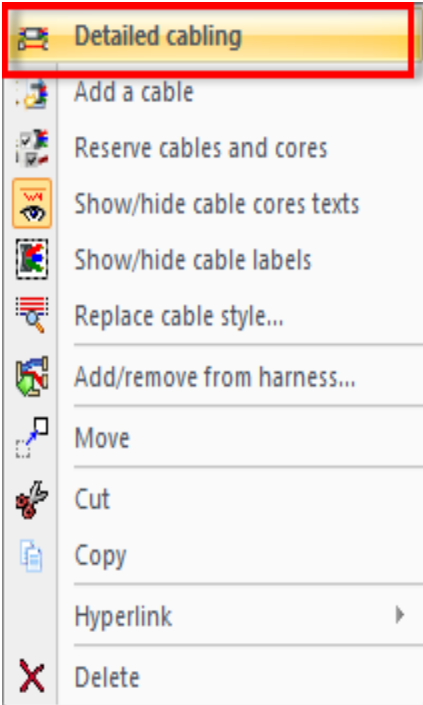


Note: You can hide and show wire numbers on any wire line by right-clicking and toggling Show/hide wire marks.



Assigning Cables and Harnesses

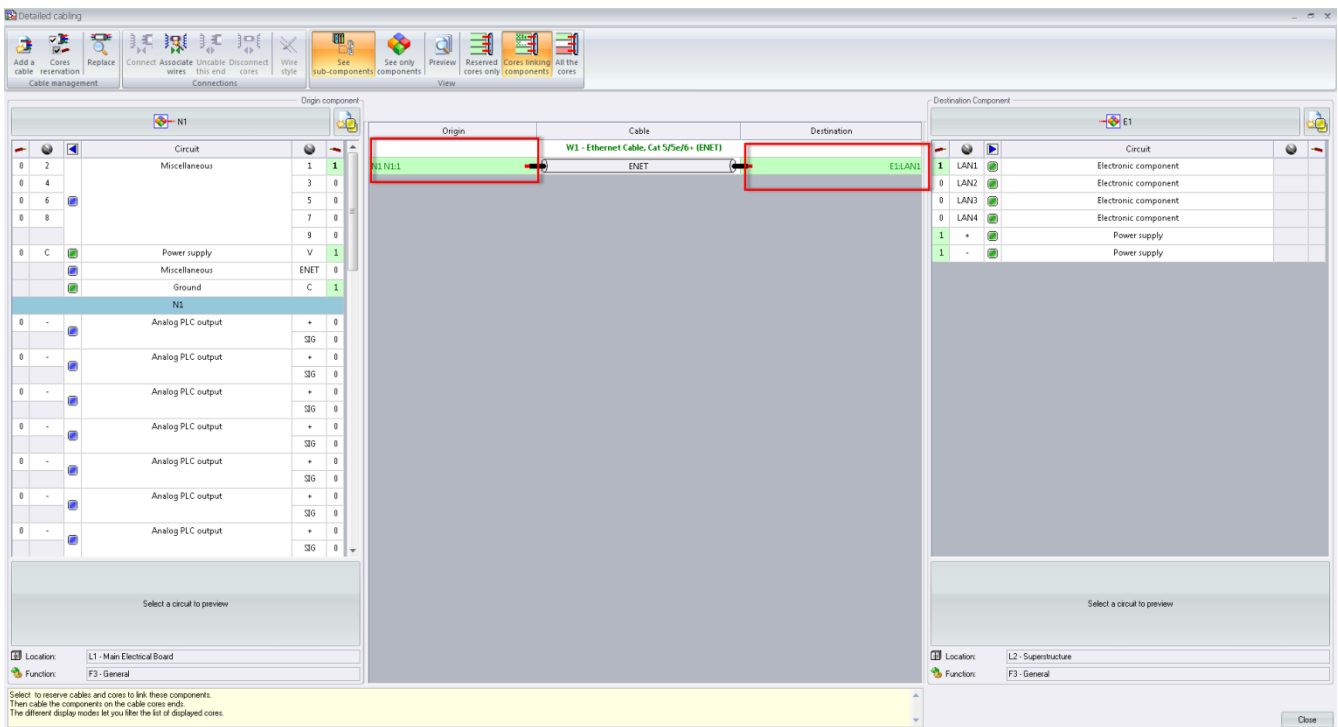
As you learned earlier cables are used in single line diagrams to connect various components to one another. These cables have the additional ability to connect circuits aside from simply creating visual connections. Harnesses are groupings of wires and or cables which are routed together to create simplified designs.



Once components are connected, a purchasable cable with predefined cores can be associated to the connection from the existing part library. As components are assigned on each end of the cable the connectivity between the components and the cable will be created as a cabling detail

To assign a cable:

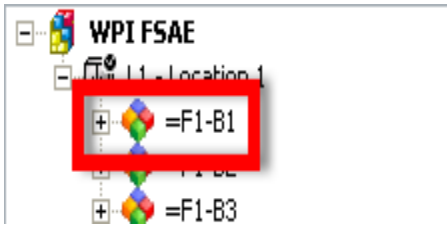
1. Right click the cable
2. Select **Detailed Cabling** to open the **Detailed Cabling** dialog
3. Select the wires with “?” assigned in the origin column
4. Drag the selected wires to the terminal column for the desired component
5. Close the dialog
6. From the **File** tab, click **Save** to ensure changes are saved to the project



SolidWorks Electrical 3D

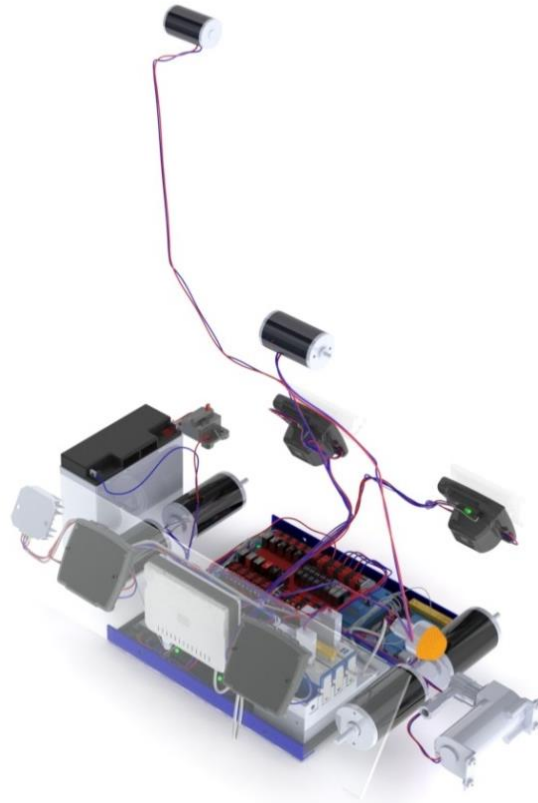
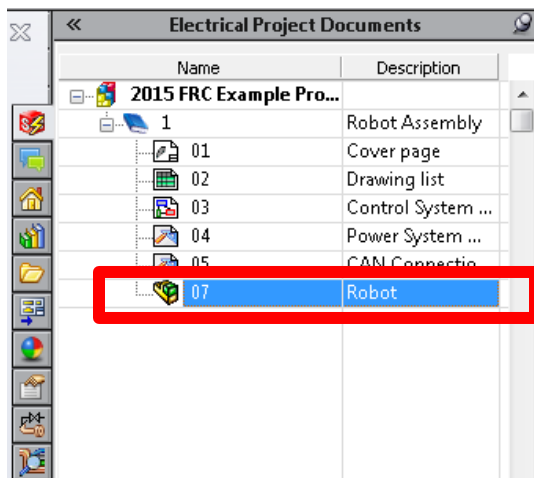
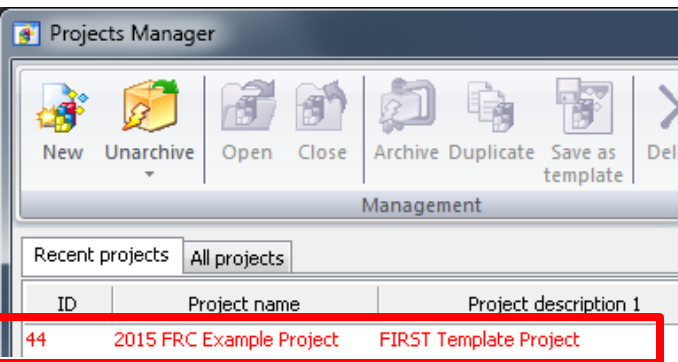
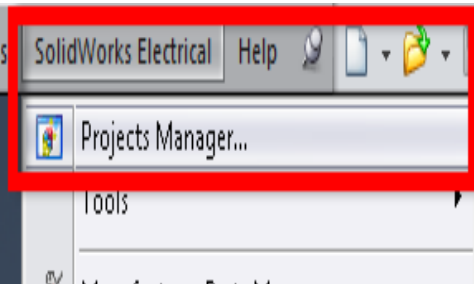
Inserting 3D Components:

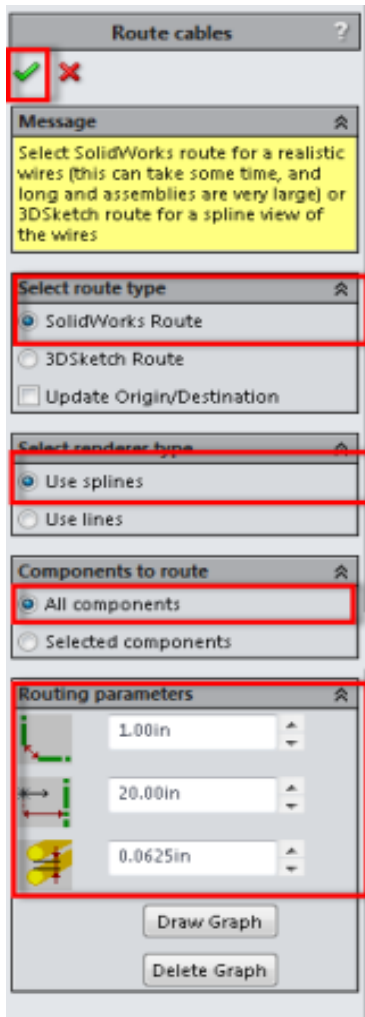
As soon as an electrical assembly is created in SolidWorks, the Electrical Component Tree displays in the Property Manager pane on the left. Components that already have 3D models will display those parts under the component mark.



Opening a part in SolidWorks:

1. From the SolidWorks Electrical Menu, select the **Projects Manager**.
2. Select the project highlighted in red and select open
3. Expand the feature tree on the left and double click the 3D model icon.

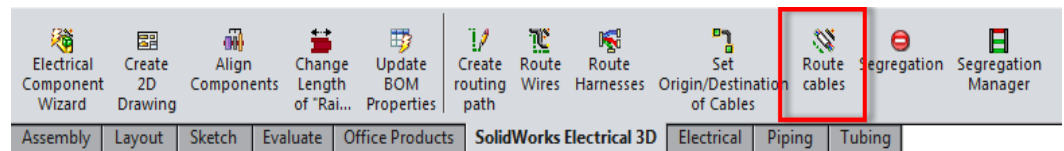




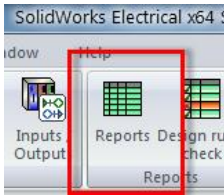
Routing Cables:

After inserting components into your SOLIDWORKS assembly, the associated cable and wire routes can be created. SOLIDWORKS Electrical provides an Electrical Component Wizard to walk through the required setup of parts to include wire and cable points and mate references to help you use your own parts or downloaded components.

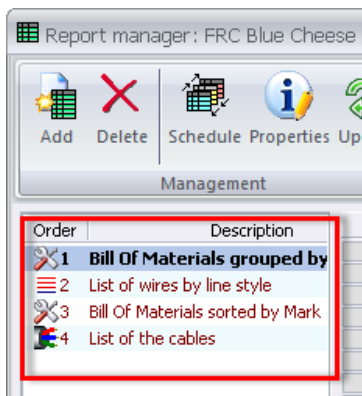
1. Select the **Route Cables** command from the SolidWorks Electrical 3D Command Manager tab
2. Select **SolidWorks Route** as the route type
3. Select **Use Splines**
4. Select **All Components**
5. Set **Routing parameters**
6. Click the check mark



Generating Reports



SolidWorks Electrical maintains a database of information about components, wires, cables and assemblies within a project. A variety of reports are available to share and organize this data. Drawings with this data can be easily shared and edited.



To generate a report:

1. Select the **Project Tab** in the SolidWorks Electrical Application
2. Click **Reports** to open the **Reports Manager**
3. Click the reports to review the data
4. To generate drawings, click **Generate Drawings**
5. Select each report listed and click **OK**
6. Click **Close**

