DASSAULT SYSTEMES

SOLIDWORKS

FUNDAMENTALS OF SOLIDWORKS ELECTRICAL

SOLIDWORKS EDUCATION EDITION 2018-2019

ENG

SOLIDWORKS® Education Edition 2018-2019

Fundamentals of SOLIDWORKS Electrical

Dassault Systèmes SolidWorks Corporation
175 Wyman Street
Waltham, MA 02451 U.S.A.

Document Number: PME-SWE103-ENG

Contents

Introduction

Lesson 1: Drawing Types

Lesson 2: Symbols and Components

Lesson 3: Manufacturers Parts

Lesson 4: Insert Components

Lesson 5: Routing Wires

Introduction

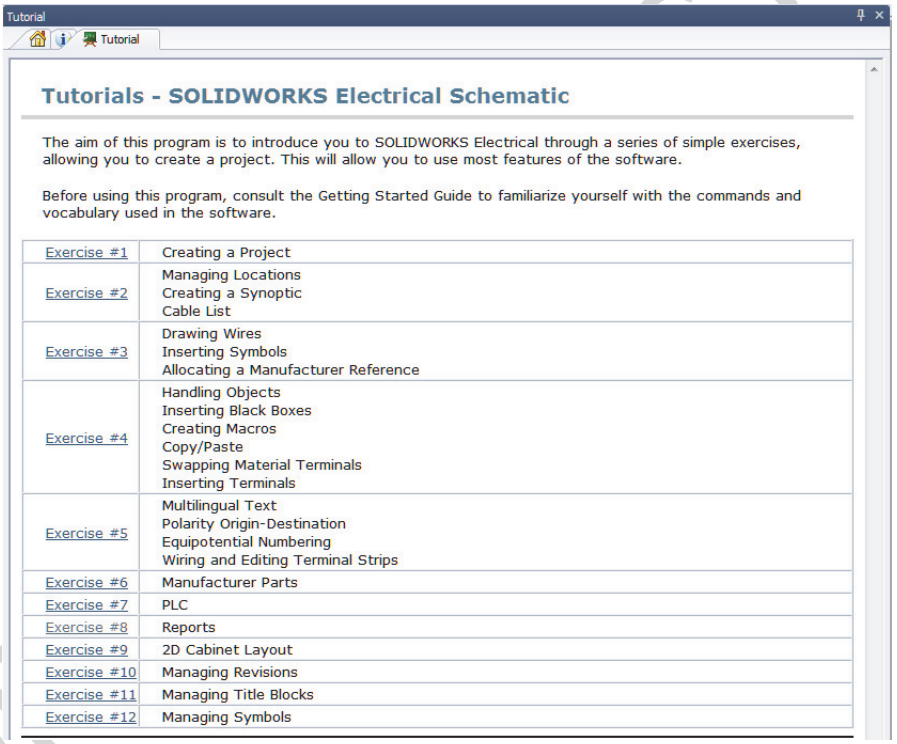
To the Teacher

The *SOLIDWORKS Education Edition - Fundamentals of SOLIDWORKS Electrical* manual is designed to assist you in teaching SOLIDWORKS Electrical in an academic setting. This guide offers a competency-based approach to teaching electrical design concepts and techniques.

Qualified schools on subscription have access to the eBook at no cost to students. Contact your SOLIDWORKS Value Added Reseller to obtain access.

SOLIDWORKS Electrical Tutorials

The *SOLIDWORKS Electrical Education Edition* manual also supplements the SOLIDWORKS Electrical Tutorials.



Accessing the SOLIDWORKS Electrical Tutorials

To start the SOLIDWORKS Electrical Tutorials, click **Help**, **Tutorial panel**. The Tutorials interface will be activated in the Resources side panel. There are 12 lessons in the SOLIDWORKS Electrical Tutorials, clicking on any of these will display steps and links to PDF and movies that can be reviewed while undertaking the exercises.

My SOLIDWORKS

My.SolidWorks.com is a community website to share, connect, and learn everything about SOLIDWORKS. My SOLIDWORKS learning contains additional video lessons and individual learning paths for your students.

Certification Exams

The Certified SOLIDWORKS Associate (CSWA) - Academic program provides free certification exams for you or your students in a proctored setting. Achieving CSWA proves the fundamentals of engineering design competency. Employers verify students job ready credentials through our online virtual tester. Schools that provide two or more courses in SOLIDWORKS-based instruction can also apply to be a Certified SOLIDWORKS Professional (CSWP) - Academic Provider.

More information and to apply can be found at www.solidworks.com/cswa-academic.

Training Files

A complete set of the various files used throughout the course can be downloaded from the following website: www.solidworks.com/EDU_FundamentalsSWElectrical

The files are organized by lesson number. The Case Study folder within each lesson contains the files you need when presenting the lessons. The Exercises folder contains any files that are required for doing the laboratory exercises.

Educator Resources link

The **Instructors Curriculum** link on the **SOLIDWORKS Resources** tab of the Task Pane includes substantial supporting materials to aid in your course presentation. Accessing this page requires a login account for the SOLIDWORKS Customer Portal. These supporting materials afford you flexibility in scope, depth, and presentation.

1. Start SOLIDWORKS.

   Using the **Start** menu, start the SOLIDWORKS application.

2. SOLIDWORKS Content.

   Click **SOLIDWORKS Resources** to open the SOLIDWORKS Resources Task Pane.

   Click on the **Instructors Curriculum** link which will take you to the SOLIDWORKS Customer Portal web page.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Experience with the Windows® operating system.
- Completed the online tutorials that are integrated in the SOLIDWORKS software. You can access the online tutorials by clicking **Help, Online Tutorial**.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. A process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a task.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that complements this objective.

Conventions Used in this Book

This manual uses the following typographic conventions:

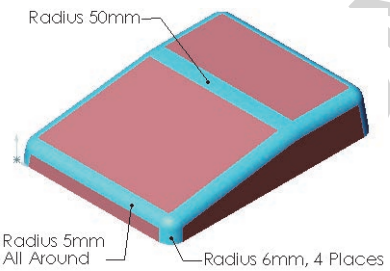
Convention	Meaning
Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, **Features > Extruded Cut** means click the **Extruded Cut** icon on the **Features** tab of the CommandManager.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Windows

The screen shots in this manual were made using the SOLIDWORKS software running a mixture of Windows® 7 and Windows 10. You may notice slight differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Use of Color

The SOLIDWORKS user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the SOLIDWORKS software. To take maximum advantage of this, the training manuals are printed in full color.



Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of a filleting operation with the fillets in a different color even though, by default, the SOLIDWORKS software would not display the results in that way.

Graphics and Graphics Cards

The SOLIDWORKS software sets a new standard with best-in-class graphics. The combination of a highly reflective material and the realism of **RealView Graphics** is an effective tool for evaluating the quality of advanced part models and surfaces.

RealView Graphics is hardware (graphics card) support of advanced shading in real time. For example, if you rotate a part, it retains its rendered appearance throughout the rotation.

Color Schemes

Out of the box, the SOLIDWORKS software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

We have not used the same color scheme for every case study and exercise because some colors are more visible and clear than others when used with different colored parts.

In addition, we have changed the viewport background to plain white so that the illustrations reproduce better on white paper.

As a result, because the color settings on your computer may be different than the ones used by the authors of this book, the images you see on your screen may not exactly match those in the book.

User Interface Appearance

Throughout the development of the software, there have been some cosmetic User Interface changes, intended to improve visibility, that do not affect the function of the software. As a policy, dialog images in the manuals which exhibit no functional change from the previous version are not replaced. As such, you may see a mixture of current and “old” UI dialogs and color schemes.

Lesson 1
Drawing Types

Upon successful completion of this lesson, you will be able to:

- Unarchive a project.
- Insert line diagram symbols.
- Associate symbols to components.
- Interconnect line diagram symbols.
- Draw multiple scheme wires.
- Insert scheme symbols.

What are Drawing Types?

There are multiple drawing types in SOLIDWORKS Electrical, many are automatically created by the program and populated with data applied to the project during its development, such as reports and terminal drawings.

The core design drawings allow for the creation of an overall interconnect synopsis, detailed schematic design, 2D general arrangement layouts and 3D assemblies.

This lesson will focus on two of the most common drawing types, wiring diagrams and schematics. Other drawing types will be introduced in later lessons.

Drawings

Drawings contain different information types that go to make up the complete project data set, a synopsis of the drawing types is as follows:

Cover Page

The Cover page contains a title block in which user can set attributes to display information such as the project name and description. The Draw tools may be used to add a company logo image.

Scheme

Scheme drawings contain circuit information related to the function of the electrical project.

Line Diagram

The Line diagram shows a simplified overview of project components interconnects and the cables used to form these connections.

Mixed Scheme

A Mixed scheme drawing allows users to mix the data style of scheme and line diagram type drawings to produce data rich hybrid designs. Using this drawing type can allow single line multi core cables and detailed scheme wiring to be shown between components.

Drawing Lists

The Drawing lists can contain a variety of project reports generated as drawings that can be updated automatically, on demand, or manually based on preference.

Terminal Drawings

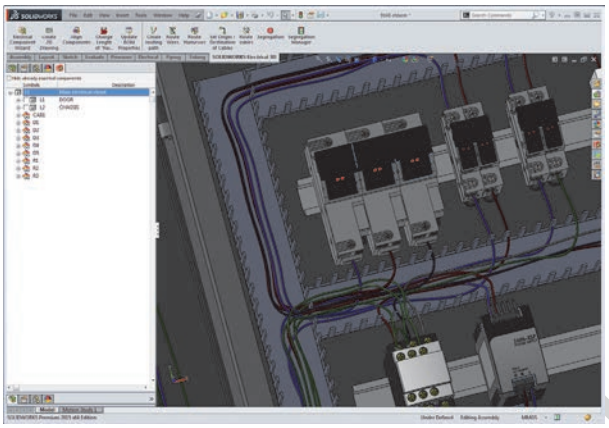
Terminals drawings are automatically created from data applied to terminals in the project drawing set and through the **Terminals editor**.

2D Assemblies

2D Assemblies 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 Assemblies

The SOLIDWORKS Assemblies allows for the development and prototyping of project data in a 3D environment.



Data Files

A Data file can be any type of file; XLS, PDF, CSV and provides a way to include supporting technical documentation related to the design.

Creating Drawings

Drawings can be created in a project from the **New** command, or from the book contextual menu, the latter is best employed where the project contains multiple books.

Stages in the Process

The major stages in the process are listed below:

- **Unarchiving a project.**
  To start the lesson a project must be unarchived from Lesson03\Case Study\Start_Lesson03.proj.tewzip see *Unarchiving a Project* on page 11.
- **Modify a line diagram symbol.**
  Open a line diagram drawing and use different methods to insert line diagram symbols.
- **Associate symbols to components.**
  Understand components and symbols and how to form associations between them.
- **Cable up components.**
  Interconnect components in the line diagram.
- **Draw wire interconnects.**
  Use multiple wire styles to interconnect schematic symbols.
- **Add a scheme symbol.**
  Open a scheme drawing and use different methods to insert scheme symbols.

Existing and Archived Projects

There are options in SOLIDWORKS Electrical for *opening* projects and *unarchiving* projects. These follow different procedures as one type uses compressed files and the other does not.

Opening an Existing Project

Existing projects are listed in the **Projects Manager** dialog. These projects have been created and opened at least once and appear with a unique **ID**, **Project name**, **Project description 1** and **Contract number**. By default, the files are stored in the `C:\ProgramData\SOLIDWORKS Electrical\Projects` folder and a sub-folder named for the project **ID**.

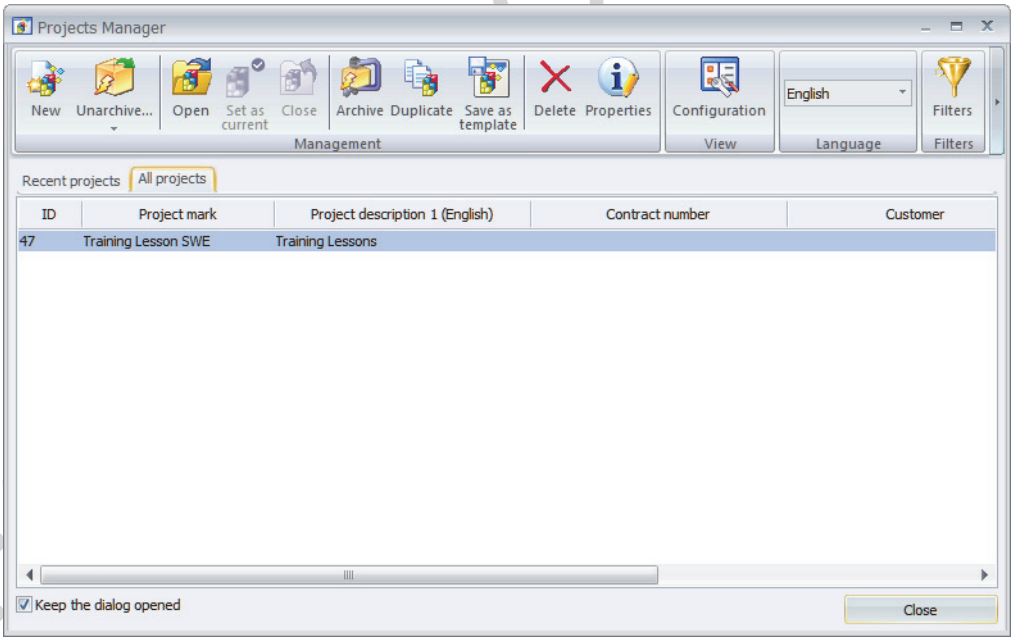
Note

Only the projects stored in that folder will appear in the **Projects Manager** dialog.

1. Click **Projects manager**.

   Click **Projects manager** . All the projects that have been opened or unarchived recently are listed.

2. Double-click a project by name.
   Double-click a project name under the `Recent projects` tab.



Where to Find It

- **Projects Manager** PropertyManager: Double-click a recent project from the list

Unarchiving a Project

Archived projects are stored in a compressed format that must be uncompressed before opening. The archive stores all the information required to open and edit the project.

Start Files

To begin a lesson or exercise with a completely up to date project, unarchive and open the appropriate project. For example, use the file Start_Lesson_03.proj at the start of this lesson and use Start_Exercise_03.proj at the start of the related exercise.

The start files are stored in the SOLIDWORKS Training Files\SOLIDWORKS Electrical Schematic\Lesson folders and the appropriate Case Study and Exercise sub-folders.

Note

Unarchiving a file does *not* open it unless you click **Yes** on the "open" message.

1. Click **Projects manager**.

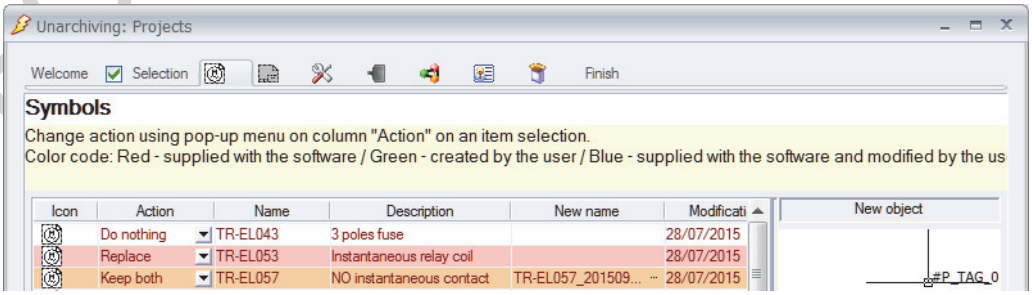
   Click **Projects manager** and click **Unarchive**. Browse to the folder C:\SOLIDWORKS Training Files\SOLIDWORKS Electrical Schematic\Lesson03\Case Study, click the file Start_Lesson_03.proj and click **Open**.

2. Project information.
   The project dialog includes text information about the project. Click **OK**.

3. Messages.
   At the message: Do you want to update your libraries? click **Update data**. At the message: Do you want to open this project? click **Yes**.

Using Update Data

If you click **Update data**, a wizard will appear to help you decide what action to take when newer files are available in different categories. At the message: Do you want to open this project? click **Yes**.



Where to Find It

- **Projects Manager** PropertyManager: **Unarchive**

Closing Projects

Open projects can be closed using the **Projects manager**. Select the project from the list (open projects will appear in blue text) and click **Close** .

Where to Find It

- **Projects Manager** PropertyManager: **Close**

Procedure

Unarchive a project, open a line diagram scheme and mixed scheme, and use different methods to insert symbols and interconnect them in the different drawing types.

Where to Find It

CommandManager: **Project >New>** ***Drawing type***

---

1 **Open a project.**

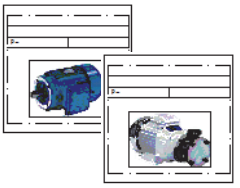
Click **Open** project Start Lesson 03.

2 **Open a line diagram.**

Expand the book and double click drawing 03 - Line diagram to open it.

Line Diagram Symbols

The **Symbols Manager** stores many symbolic representations for line diagrams broken in logical classification groupings. You can copy a symbol from the library to the drawing sheet by using a number of insert symbol options.

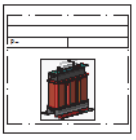
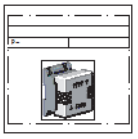


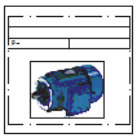
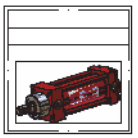
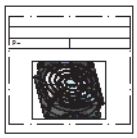
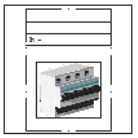
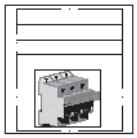
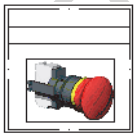
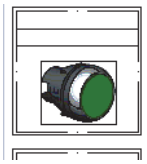
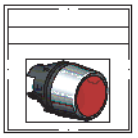
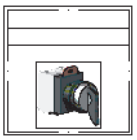
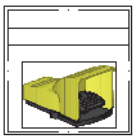
Adding Symbols

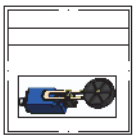
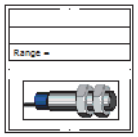
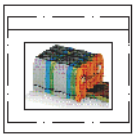
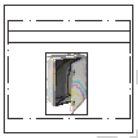
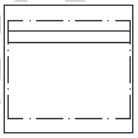
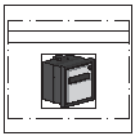
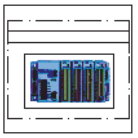
There are two main methods of adding symbols to a drawing; one uses a standard click (**Insert Symbol**) to place while the other uses the familiar drag drop method. Both methods can access the same symbol library.

Symbols Library

The side panel Symbols tab offers easier access to some of the more common symbols. These are shown in the following table.

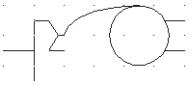
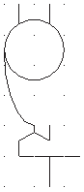
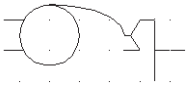
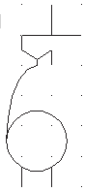
Supplies		
		
EW_SY_Battery Cells battery	EW_SY_TransformerHigh High Voltage Transformer	EW_SY_TransformerLow Low Voltage Transformer

Loads		
 EW_SY_Motor Motor	 EW_SY_Resistor Heat Heating resistor	 EW_SY_Jack Cylinder
 EW_SY_Fan Fan	 EW_SY_Pump Pump	
Breaking devices		
 EW_SY_Contactor Contactor relay	 EW_SY_CurcuitBreaker Circuit-breaker	 EW_SY_CurcuitBreakerMod Modular circuit-breaker
 EW_SY_FuseSwitch Fuse disconnector		
Command		
 EW_SY_EmergencyStop Emergency stop button	 EW_SY_PushButtonRun NO push button	 EW_SY_PushButtonStop NC push button
 EW_SY_Switch Switch	 EW_SY_PedalContact Foot contact	

Sensors		
 EW_SY_LimitSwitch Roller limit switch	 EW_SY_LimitSwitchLever Lever limit switch	 EW_SY_PressureSensor Pressure sensor
 EW_SY_ProximitySensor Proximity sensor	 EW_SY_TemperatureProbe Temperature probe	
Miscellaneous		
 EW_SY_Terminal Terminal strip	 EW_SY_Cabinet Electrical cabinet	 EW_SY_Ammeter Ammeter
 EW_SY_Voltmeter Voltmeter	 EW_SY_BlackBox Generic frame	 EW_SY_TimeCounter Time meter
 EW_SY_MotorDrive Drive	 EW_SY_Plc PLC	 EW_SY_Screen Screen

Symbol Orientation

The **Symbol Orientation** options control the rotation and possible mirroring of a symbol as it is added to the drawing.

Original orientation		**90 rotation**	
180 rotation		**270 rotation**	
90 rotation and mirror		**Mirror**	

Note The symbols shown in the symbol library will vary based on what type of drawing or scheme is open.

Tip Line diagram symbols do not require an image be inserted in them.

Note The Symbols tab on the resource side panel can be modified by adding selected symbols to any of the group such as **Command** and **Sensors**.

Where to Find It

- CommandManager: **Line diagram > Insert Symbol**
- Side Panel: Click **Symbols**
- Shortcut Menu: Right-click a component and click **Insert Symbol**

---

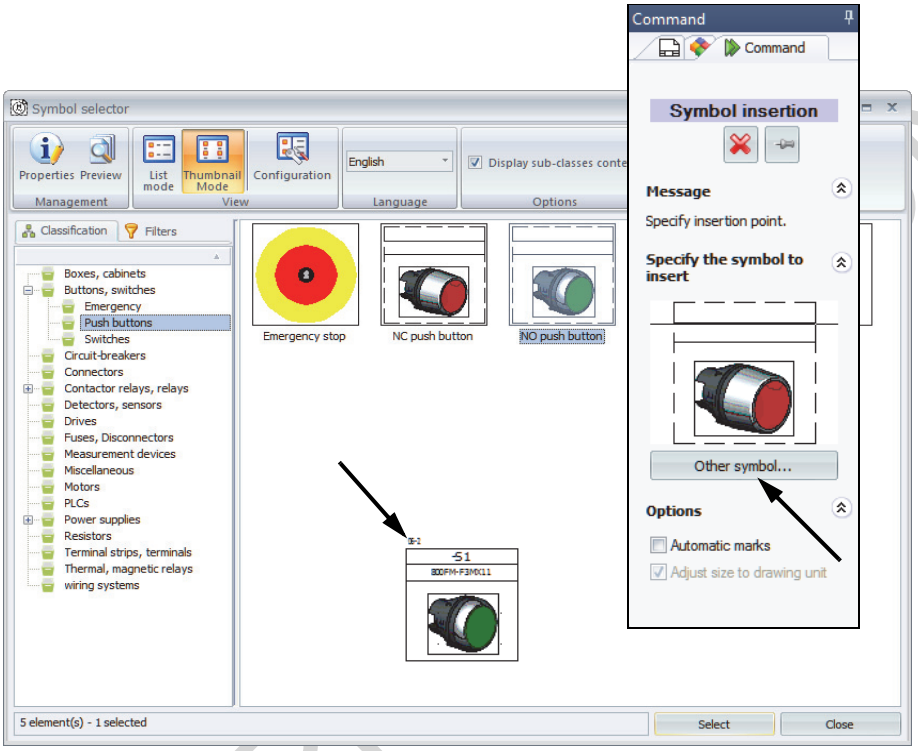
3 Select line diagram symbol.

Click **Insert symbol** .

Note If this is the first time you have inserted a symbol then the Symbols selector will be displayed automatically.

If the command has been used previously the last inserted symbol will be shown.

The **Other symbol** button can be used to access the Symbols selector.

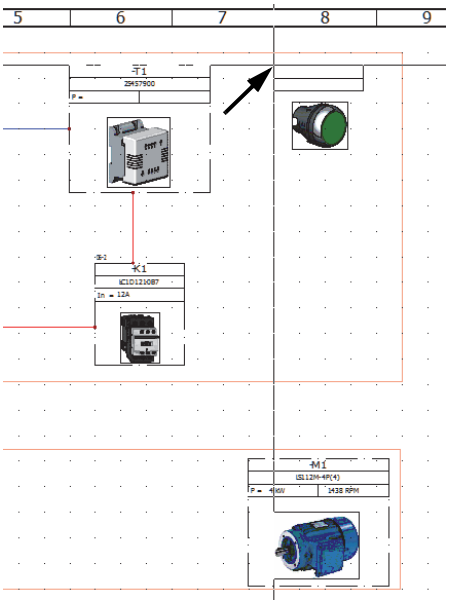


In the Symbols selector click the Buttons, switches classification.

Highlight the NO push button and click **Select** to return to the drawing.

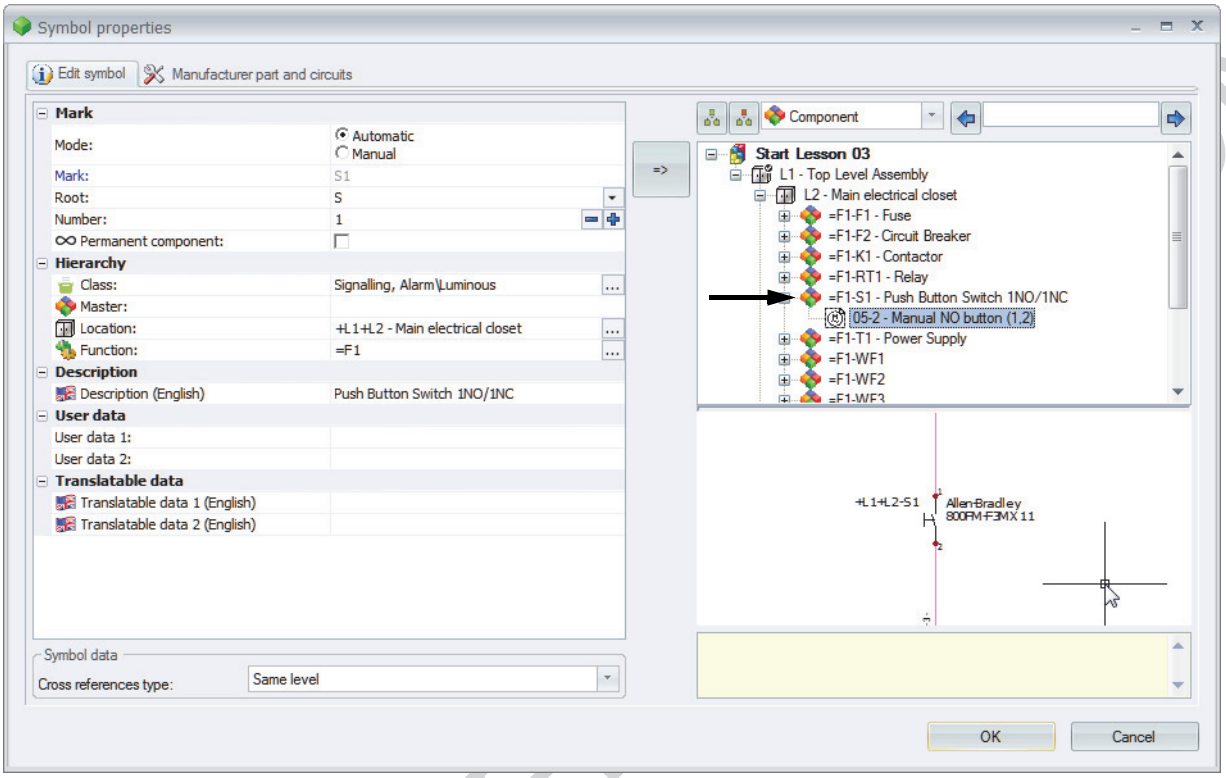
4 Insert symbol.

Click to insert the symbol to the right of -T1, in-line with -M1.



5 Symbol - component association.

Click =F1-S1 - Push Button Switch 1NO/1NC displayed in the components list.



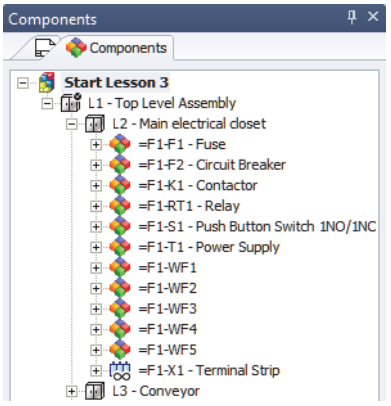
Click **OK** to create an association.

Note

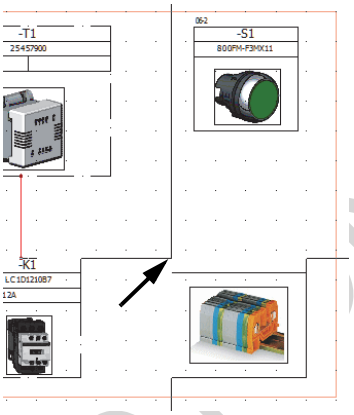
By selecting an existing project component the inserted symbol will be associated. There can be multiple symbolic representations of a component across different drawings. The component is the physical part that will be purchased and installed.

6 Components side panel.

On the Components side panel expand the location L1 - Top Level Assembly and sub location L2 - Main electrical closet.



7 Insert component symbol.
Right-click component =F1-X1 - Terminal Strip and click **Insert symbol** .



Using the same methods previously outlined select Terminal strip classification, highlight symbol name EW_SY_Terminal and **Select** to return to the drawing.

Place the symbol to the right of -K1 below -S1.

Note As the symbol is being inserted *from* a component it is automatically associated and the Symbols properties dialog is not displayed.

Adding Cables

Wiring diagrams indicate system level interconnects between components.

These interconnects are realized by a single line indicative of a cable.

A cable contains one or more cable cores or conductors that can be assigned to wires.

It may be used to simply indicate components connections, or to *reserve* a cable for use between two components, or define the detailed wiring between symbols in line diagrams.

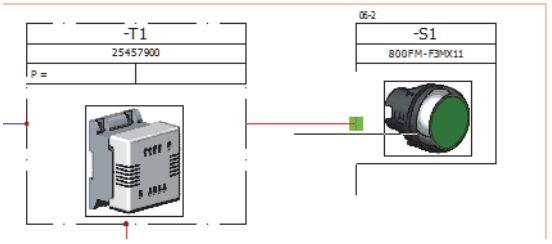
Tip This information bi-directionally updates in schemes and vise versa.

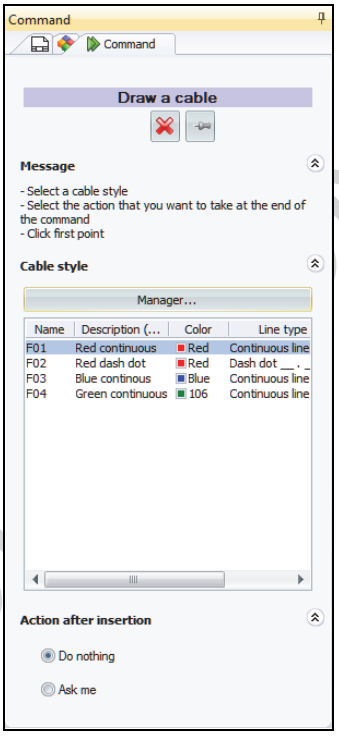
Note The cables are drawn to show connections between components. There is not yet any assignment of cable conductor cores to the cable.

Where to Find It CommandManager: **Line diagram > Draw cable**

8 Cables.

Click **Draw cable** and connect the symbols as illustrated.

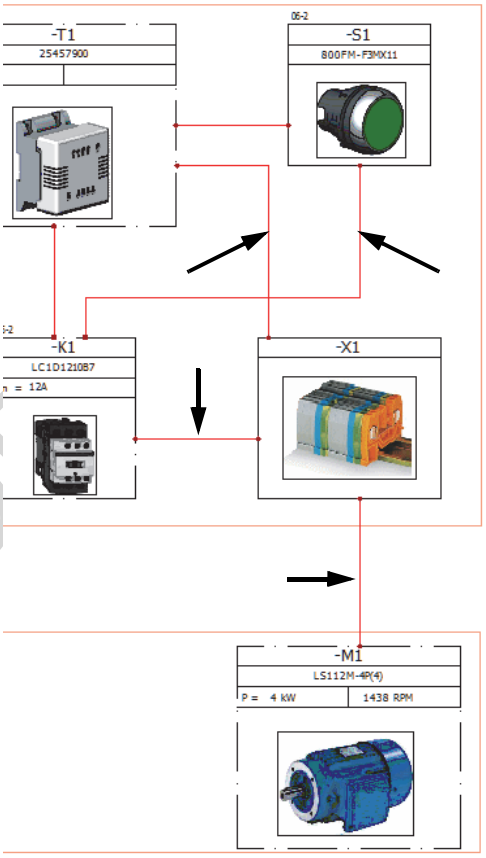




Tip

The **Esc** key will stop creating cables, when drawing a cable that does not terminate at another symbol.

Repeat the process drawing more cable interconnects as illustrated.



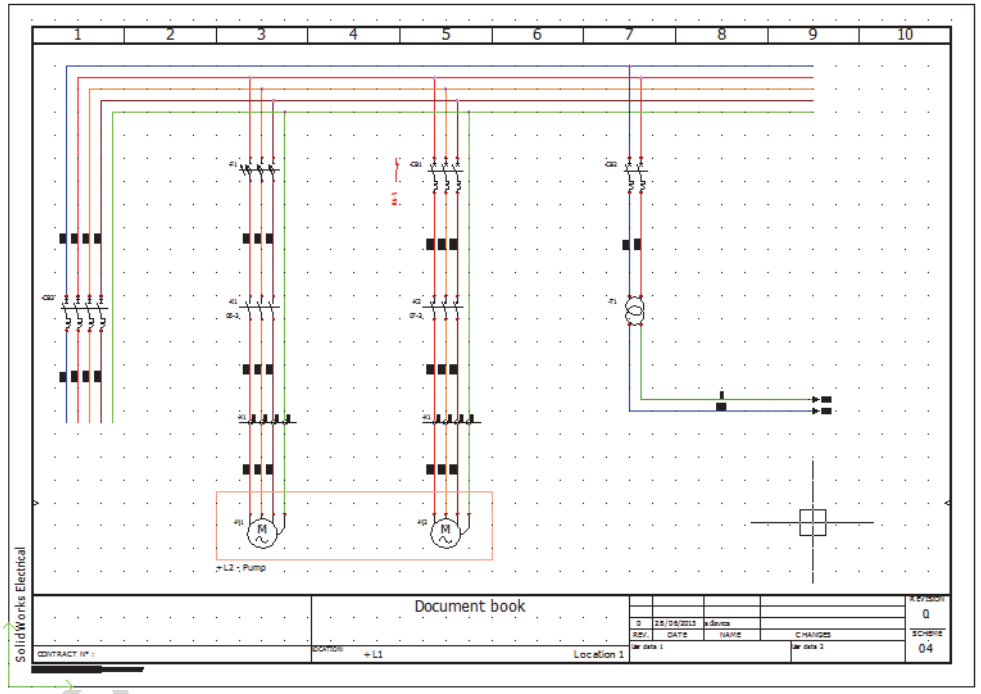
Schematic Drawing

A **Schematic** diagram is used to show the electrical components and the detailed electrical connections between them.

Schematics may appear in one or more project books.

When a schematic drawing is opened a unique set of commands are available for the design development.

Filters are used for the **Insert Symbols** command to ensure that superfluous data, such as line diagram, or footprint symbols, are unavailable.



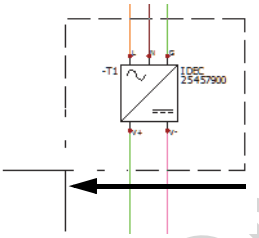
Note

Schemes such as the drawing `04 - Power` appear with the icon in the document list.

Scheme Best Practices

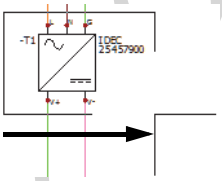
There are certain recommended best practices when working in scheme drawings that help ensure connectivity and make modifications easier.

- **Snaps**
  All scheme symbols are set up on a 5mm / 0.25inch grid system by maintaining **Snap** activate at these, or any divisible value, it ensures wires readily connect to symbols when inserting, moving, stretching.
- **Ortho**
  Keeping the **Ortho** active helps ensure wires drawn will be straight and improves the face of the drawing quality.
- **Selection windows**
  Dragging a rectangular window around geometry is a very efficient method of selecting multiple entities. It *does* make a difference whether the window is dragged left to right or from right to left.



Dragging a window *right to left* captures all the geometry in and crossing the window. In this example the symbol and all connected wires will be selected as they are within the crossing window.

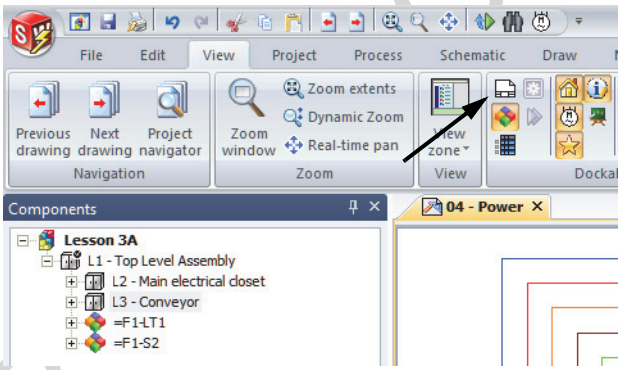
The selection will be shown as a dashed line when using this option.



Dragging a selection window *left to right* captures geometry within the window. In this example only the symbol will be selected, as the connected wires are not fully contained in the crossing window.

The selection will be shown as a solid line when using this option.

- **View tab**
  The View tab has a range of options including commands that allow for panels to be toggled on or off. If a panel, such as the Documents tab, is not shown then it will have been turned off.



In this image the documents dockable panel has been deactivated and only the Components tab is shown.

Stages in the Process

The major stages in the process are listed below:

- **Schematics**
  Identifying and opening a scheme drawing.
- **Draw Multiple Wires**
  Multiple wires are selected and drawn in a single operation.
- **Schematic Symbols**
  Schematic symbols are introduced and added to the drawing using a variety of methods.

Procedure

Complete a schematic power drawing using multiple wires and schematic symbols.

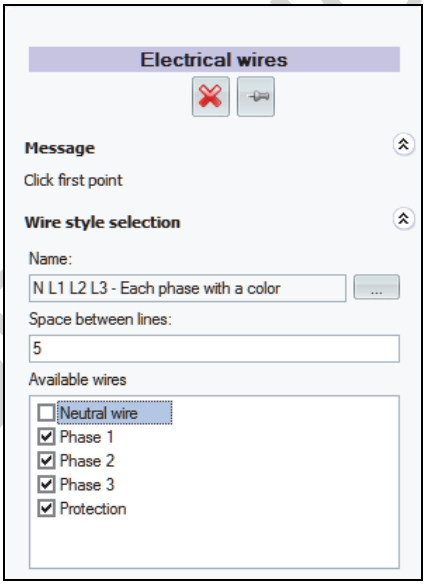
9 Scheme drawing.

Open drawing `04 - Power`.

10 Select multiple wires.

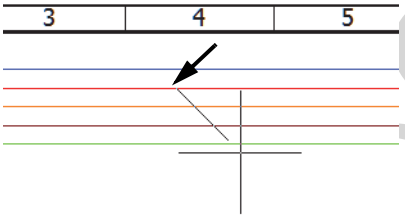
Click to **Draw multiple wires** .

Click to clear the `Neutral wire` and make sure that the four `available wires` are selected as illustrated.



11 Draw multiple wires.

Click 2nd wire, `Phase 1`, as illustrated, and move the cursor below the horizontal to select the right downward phase toggle.



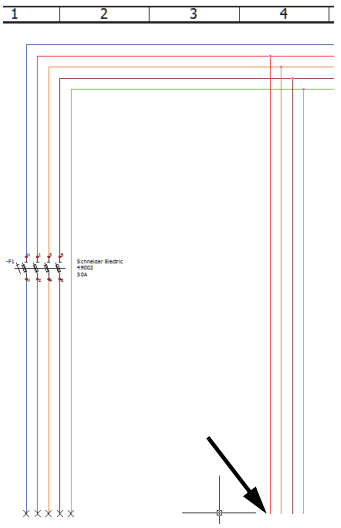
Note

On activating the command the incorrect wire style is active, by drawing off existing wires in a drawing, the program automatically adjusts the style to match the existing styles. This not only acts as a time saving feature but also helps resolve a common design error before it can occur.

12 Complete wires.

Click again on the lower part of the screen to complete the wires.

Click **OK** to end the command.



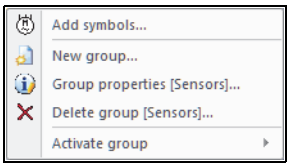
Symbols Panel

The Symbols panel is available in line diagram, schematic and mixed scheme drawings.

Depending on the drawing which is open it will only show line diagram, or schematic symbols.

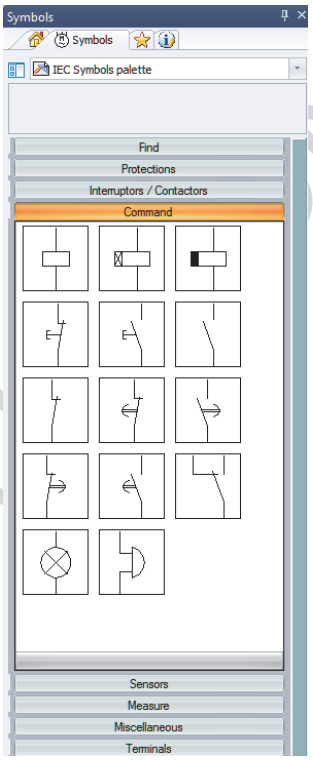
In a mixed scheme a drop down allows access to either line diagram or scheme symbols.

The symbols are grouped into generic defaults, but these can be removed, or new groups added via the contextual menu.



The groups and symbols contained in them are related to palettes that are stored in the application.

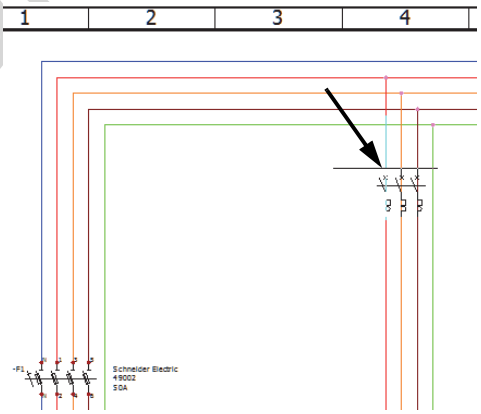
Palettes can also be added to individual projects and modified separately for each.

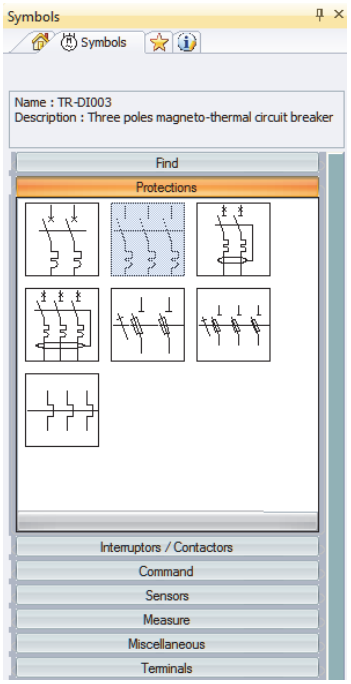


13 Symbols.

On the resource side panel Symbols tab select Protections group and double click the three pole circuit breaker TR-DI003.

Left-click to place the symbol as illustrated.





14 Symbol association.

The symbol being inserted is already represented in the wiring diagram, so a component exists to which it can be associated.

Highlight `=F1-F2 - Circuit Breaker` and click **OK**.

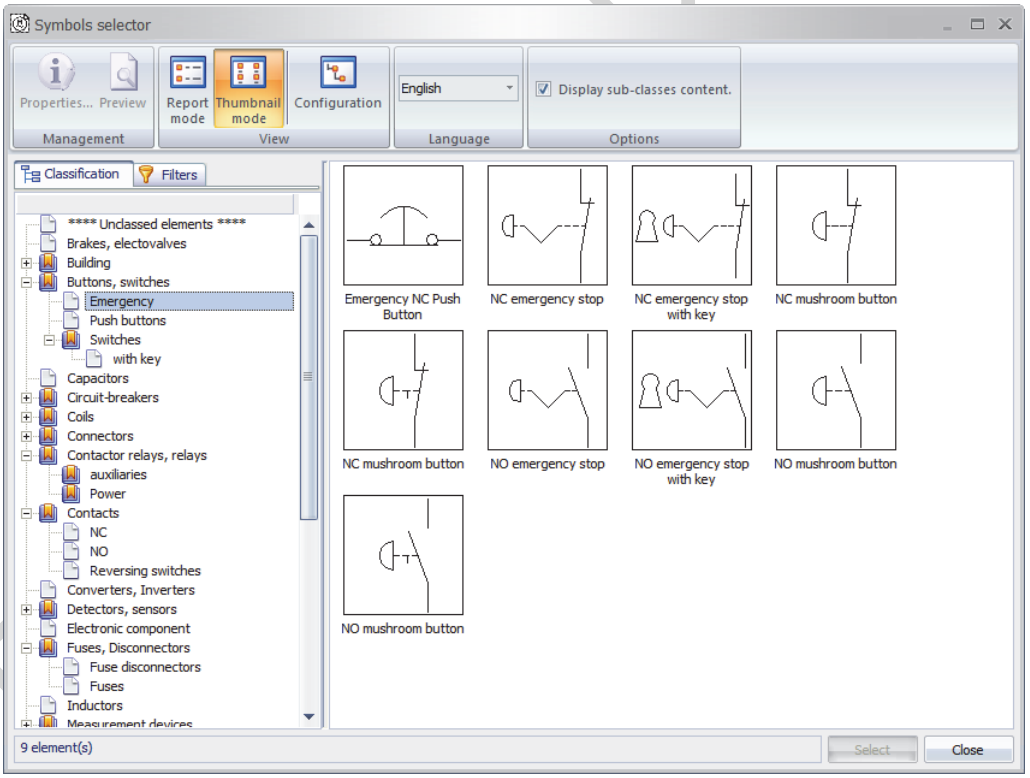
Schematic Symbols

The **Schematic Symbols** library stores many symbolic representations. The library contains different classification folders and sub-folders to suit a variety of needs.

The symbols themselves are traditional blocks, containing graphic entities and attributes, the content of the attributes is automatically populated based on command choices during the design process.

In addition symbols have properties held in the SQL database that define the symbol type, default part data.

If a required symbol is not available new symbols can be easily created.



Note

Scheme symbols are different from those for line diagrams introduced in *Line Diagram Symbols* on page 12 although they are all stored in the same general library. Those available are a filtered subset of the general set of blocks.

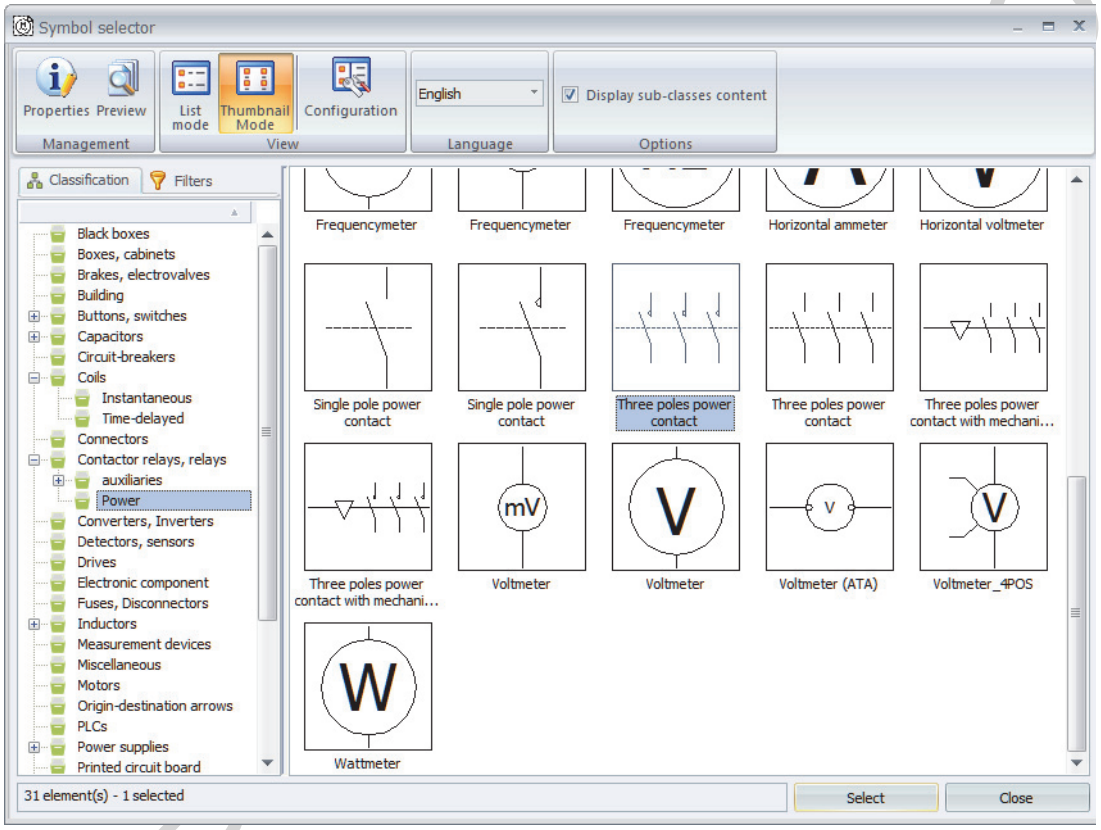
Where to Find It

- CommandManager: **Schematic > Insert Symbol**
- Side Panel: Click **Symbols**

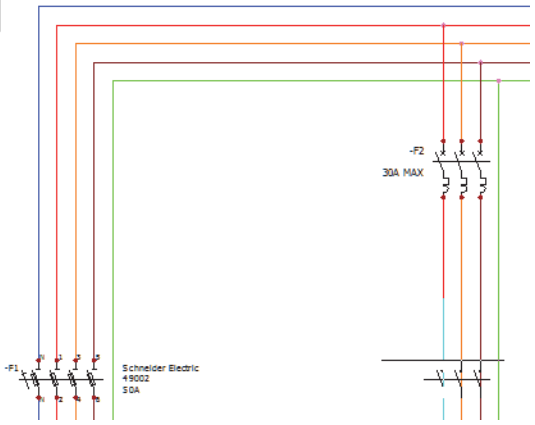
15 Insert scheme symbol.

Click to **Insert symbol** and **Other symbol** to access the Symbols selector dialog.

In the Contactor relays, relays classification click Three poles power contact TR-EL035.



Click to **Select** and position the contact below -F2 in line with -F1.



Symbol Properties

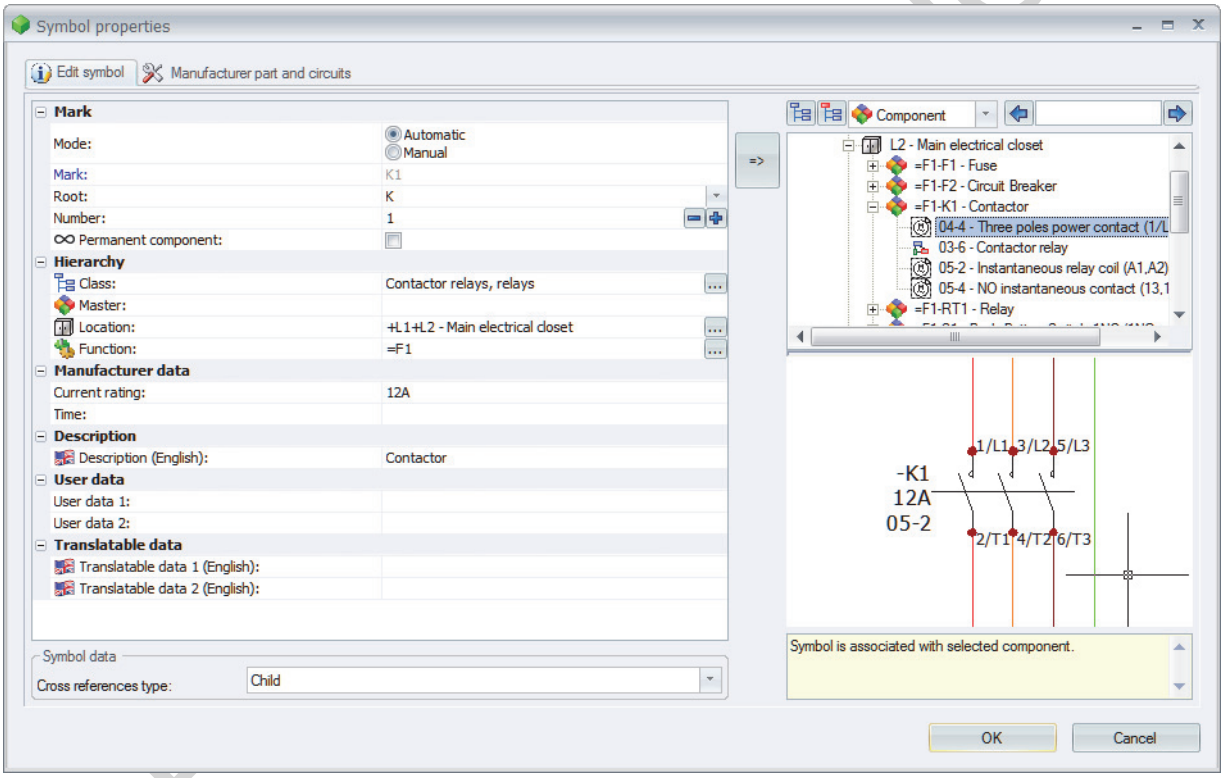
The **Symbol properties** dialog is used to set and control the attribute content of an individual symbol including the manufacturer part and the cross references to other symbols.

Types of Properties

For any symbol, there is an option to select either the **Symbol properties** or the **Component properties**. Both types include the same Manufacturer parts and circuits tab (*What are Manufacturers Parts?* on page 52) used to add manufacturer parts to the symbol.

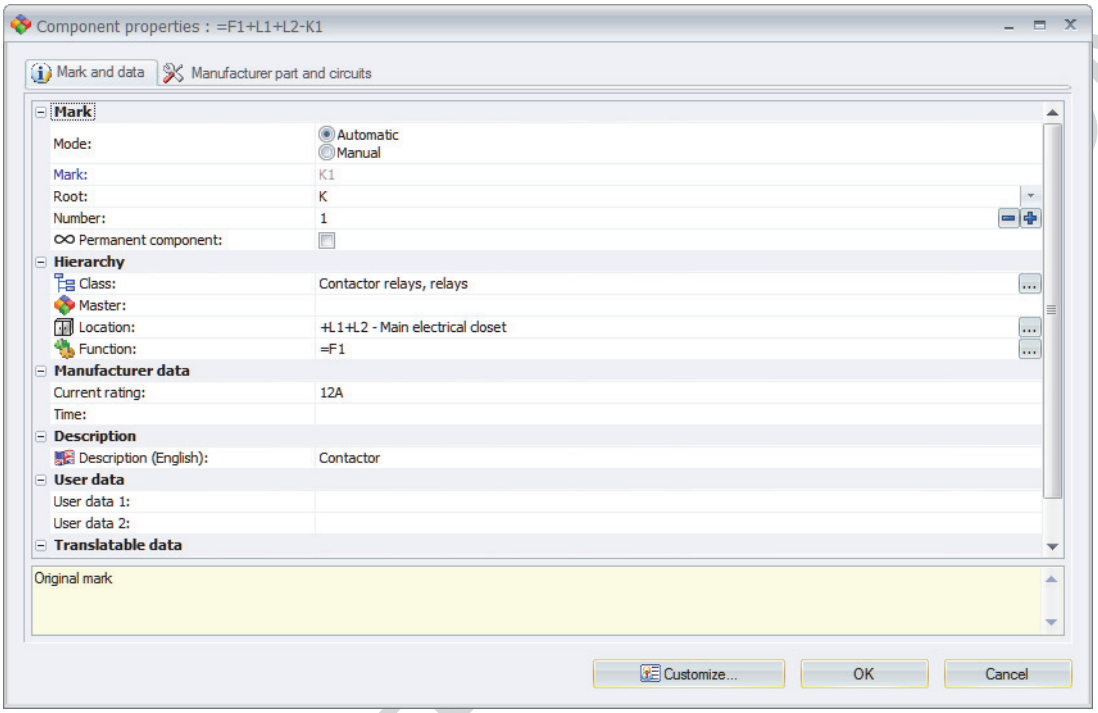
Symbol Properties

Symbol properties includes the Edit symbol and Manufacturer parts and circuits tabs. The Edit symbol tab includes textual property data as well as a listing of available components (right side column) that is useful when creating cross references.



Component Properties

Component properties includes the Mark and Data and Manufacturer parts and circuits tabs. The Mark and Data tab includes information (lower portion of dialog) about whether this symbol mark is original and unique.



Where to Find It

- Shortcut Menu: Right-click a symbol and click **Symbol properties**
- Shortcut Menu: Right-click a symbol and click **Component properties**

Note

Double-clicking a symbol accesses the **Component properties**.

The following section uses **Symbol properties**.

---

16 Coil-contact association.

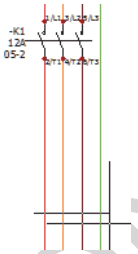
The contact is a part of the relay coil already represented in the wiring diagram and control scheme, associating the contact to this component exists to which the symbol can be associated.

=F1-K1 - Contactor and click **OK** to create the association.

17 Insert multiple terminals.

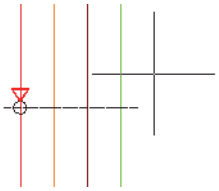
Click to **Insert 'n' terminals** and choose Terminal TR-BOO1 from the symbols selector using the processes outlined previously.

Click **Select** to return to the drawing.



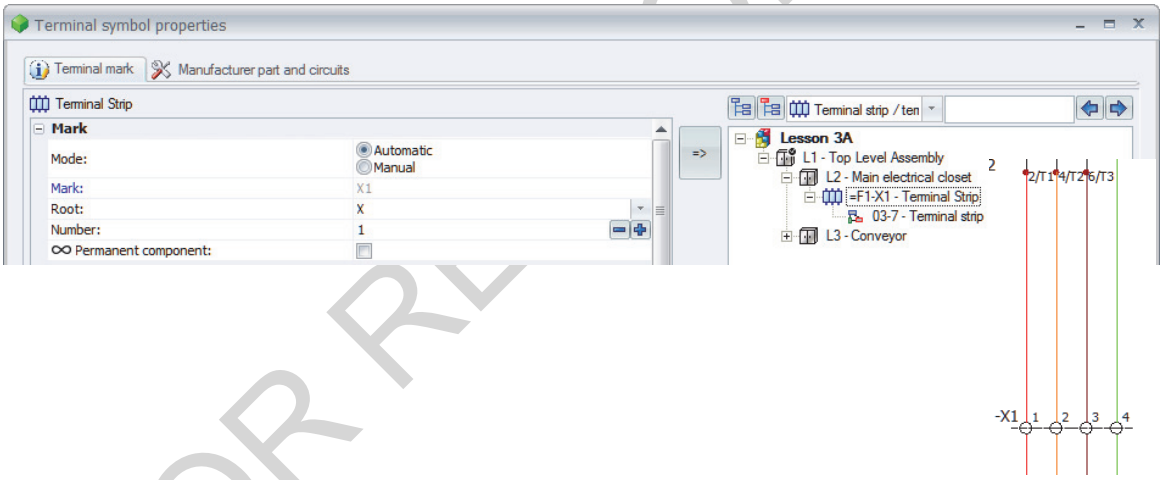
Draw a horizontal line from left to right across the wires below -K1 as illustrated.

Ensure the red triangle indicator is pointing toward the bottom of the page by moving your cursor above the axis line and left-click to place.



Question What does the red arrow do?

18 Associate multiple terminals.
the existing component =F1-X1 - Terminal strip and click **OK (all terminals)** to create multiple associations.



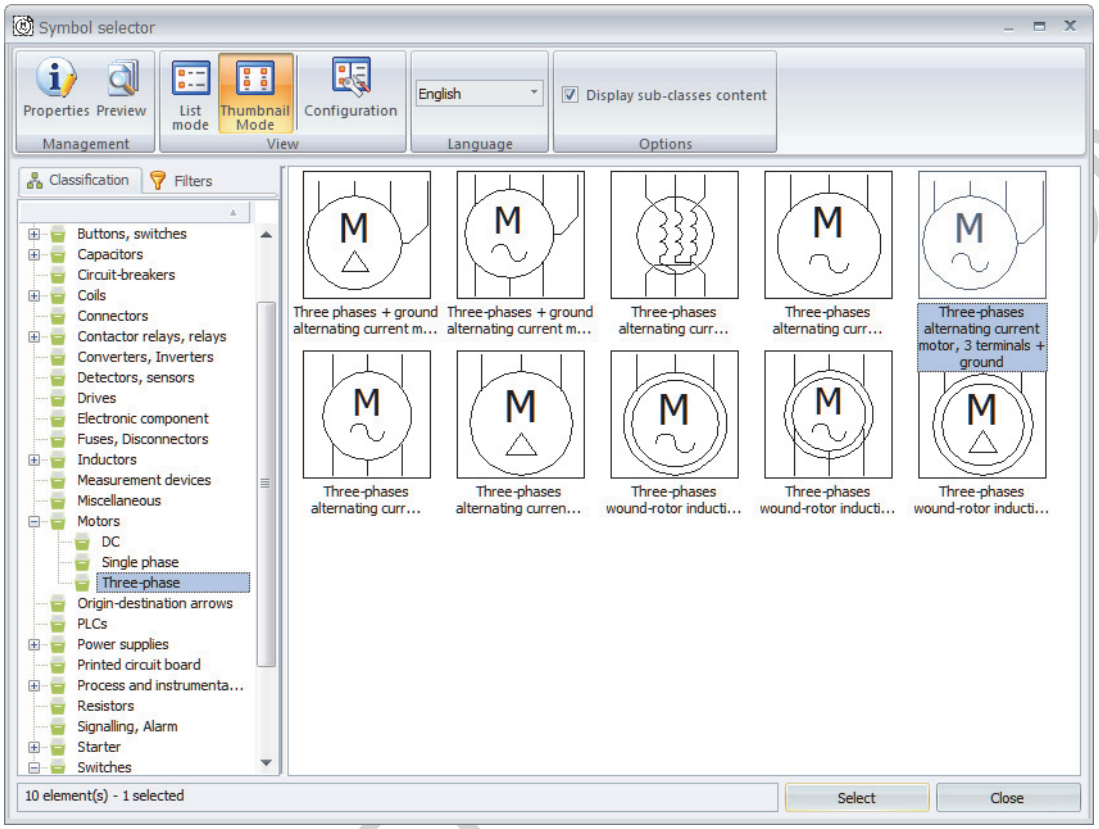
19 Scheme component symbol.
the Components side panel and expand location L3 - Conveyor.

Tip Right-clicking on the book in the Component tab shows a contextual menu where it is possible to toggle the component tree view between location and function group sorting.

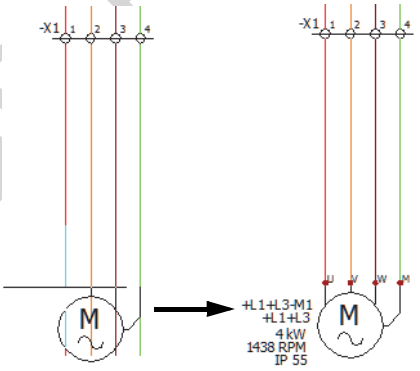
Right-click =F1-M1 - Motor component and **Insert symbol** use the processes outlined previously to locate the following symbol-

- Classification: Motors
- Description: Three-phases alternating current motor, 3 terminals + ground

Once located click **Select** to return to the drawing.



Position the motor at the end of the wires below the terminals and insert it as illustrated.



Note Due to the symbol being inserted from a component it automatically adopts all the assigned properties from the component without showing the Symbols properties dialog.

20 Close the project.

Right-click project name in the Documents side panel and **Close** the project.

Exercise 1: Drawing Types

Unarchive a project and create a mixed scheme, use different methods to insert symbols and interconnect the data with wires and cables.

This lab uses the following skills:

- *Unarchiving a Project* on page 11
- *Select line diagram symbol.* on page 15
- *Symbol - component association.* on page 17
- *Insert component symbol.* on page 18
- *Cables.* on page 19
- *Insert scheme symbol.* on page 25
- *Select multiple wires.* on page 22

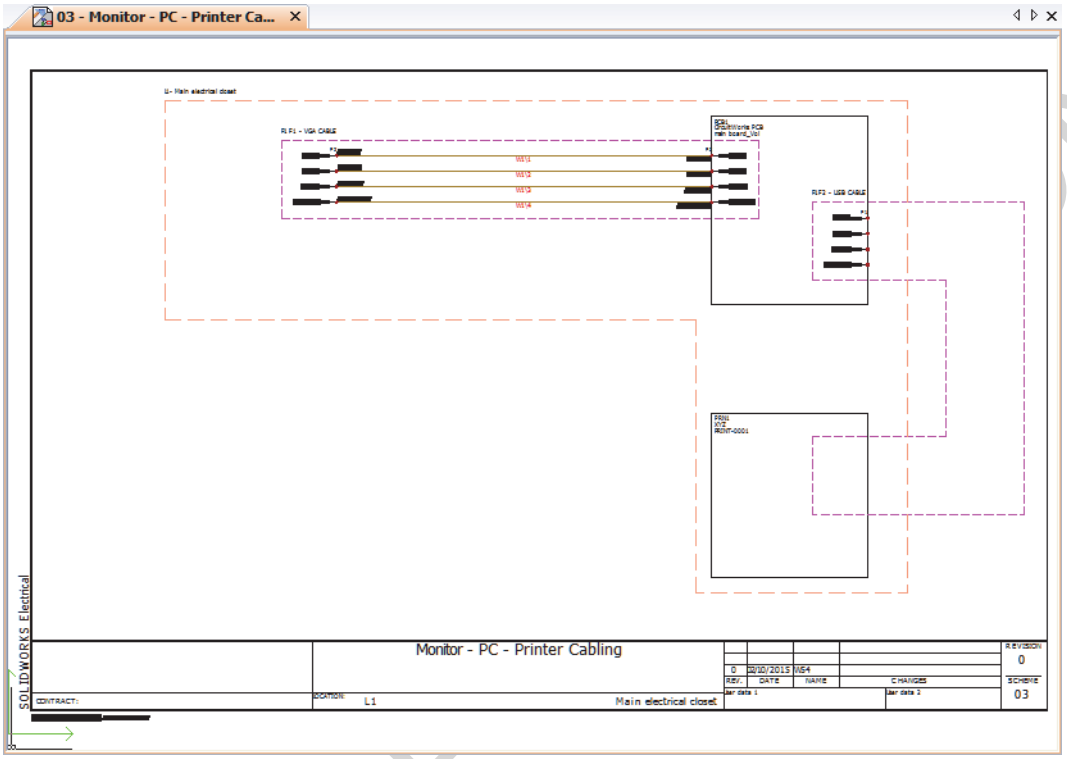
Procedure

Complete a mixed scheme drawing using both line diagram and schematic tools.

1. **Populate data to the application.**
   Unarchive the project located in `Lesson03\Exercises` folder.
2. **Data selection.**
   Click to **Update data** using the **Next** button to review data for processing.
3. **Complete the unarchive.**
   Finish the unarchive process leaving the settings as they were by default.
4. **Open the project.**
   Click **Yes** to open the project.

5 Open a mixed scheme.

Open mixed scheme drawing 03 - Monitor - PC - Printer Cabling.



6 Line diagram component symbol.

On the components panel use the contextual menu to **Insert symbol** for the monitor =F1-MON1 - MONITOR.

Tip

As this is a mixed scheme both line diagram and schematic tools and symbols are available.

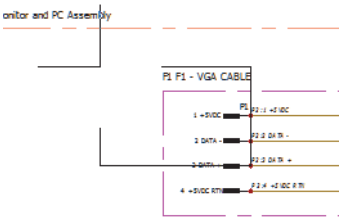
7 Select line diagram symbol.

Locate a line diagram symbol with the following properties.

- Classification: Black boxes
- Description: PCB WD
- Name: EW_BB_Blackbox_2+1

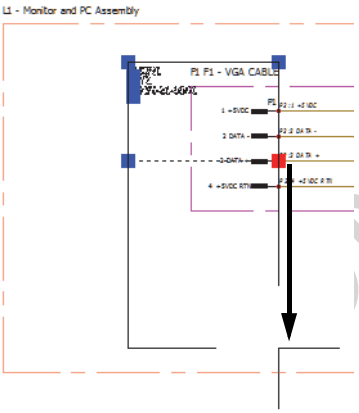
8 Insert symbol.

Place the symbol as illustrated.



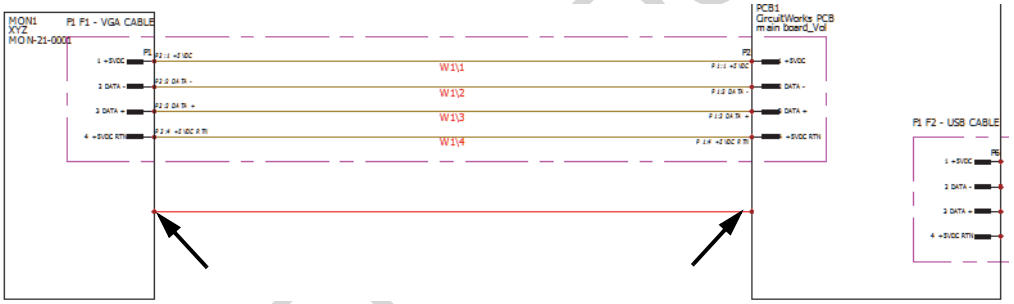
9 Resize the symbol.

Click the symbol and drag the lower right hand grip as illustrated to resize the symbol.



10 Draw a cable.

Use the line digram tools to draw a cable interconnecting MON1 and PCB1.



11 Insert scheme symbol.

Click to insert a schematic symbol from the contextual menu of component =F1=F2-P3 - PC TO PRINTER USB TYPE A.

12 Select scheme symbol.

Locate a scheme symbol with the following properties.

- Classification: Connectors
- Description: Male power pin (Training Exercise)
- Name: TR-PIN_M_02+1+1

13 Symbol orientation.

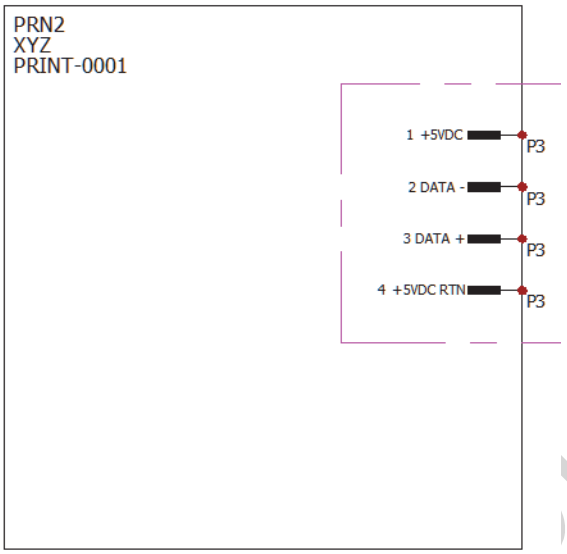
In the Symbol insertion side panel click to rotate the symbol **180°**

14 Pin the command.

On the Symbol insertion side panel click to **Pin** the command.

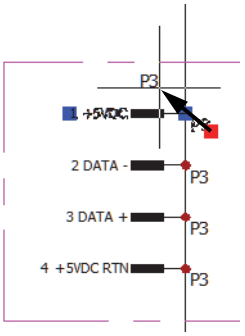
15 Position scheme symbols.

Insert 4 connector pins as illustrated.



16 Move an attribute.

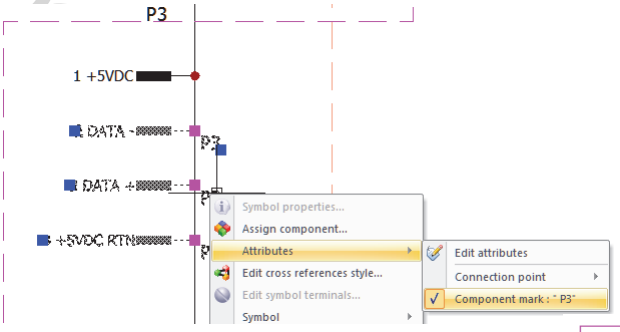
Click the connector pin 1 +5VDC and drag drop the mark P3 as illustrated.

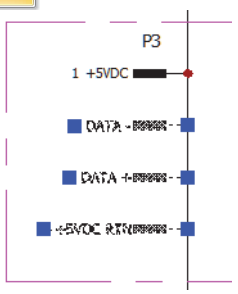


17 Attribute visibility.

Use a window selection to highlight pins 2, 3, and 4 and right-click one of them.

Click **Attributes > Component mark** to hide the attribute on all three symbols.



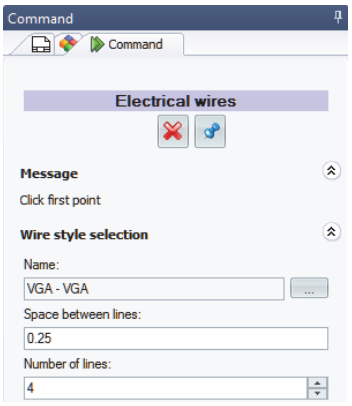


18 Select wires.

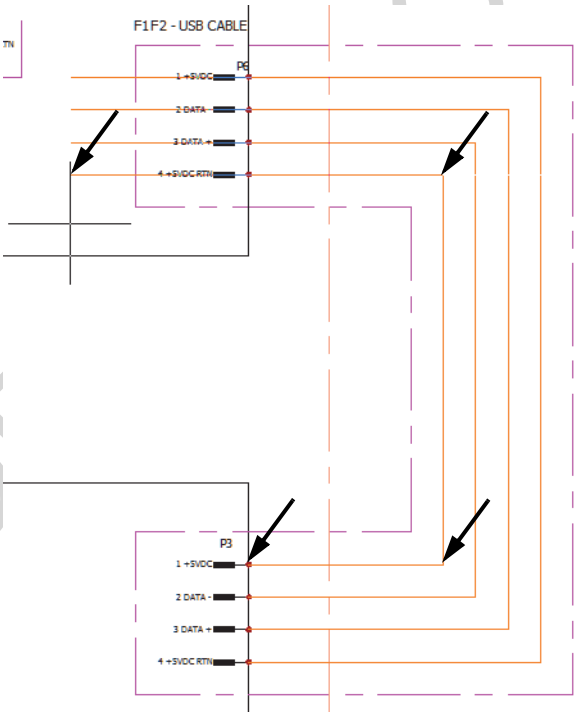
Click to **Draw single wire** selecting wire style VGA.

19 Draw multiple wires.

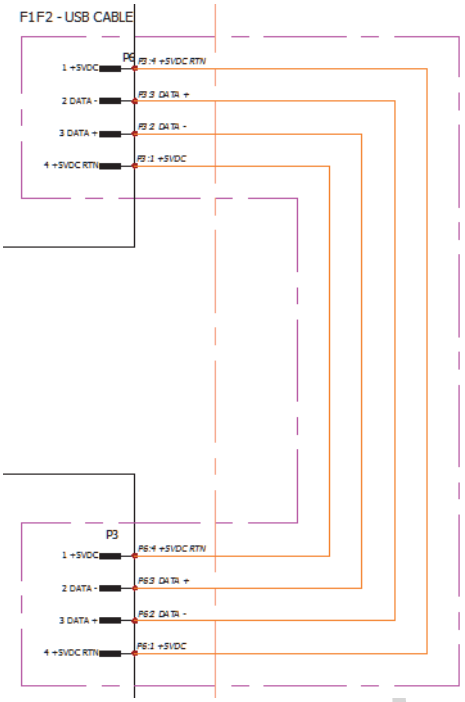
Set the electrical wires settings as illustrated.



Click at the points indicated by arrows to interconnect the pins starting on the connector P3 pin 1 +5VDC.



Then click to **Cancel** the command.



20 Close the project.

Right-click project name in the Documents side panel and **Close** the project.

Lesson 2
Symbols and Components

Upon successful completion of this lesson, you will be able to:

- Understand components.
- Create symbol components.
- Create a component.
- Insert a component symbol.
- Change component permanency.
- Assign component.

What is a component?

A component represents a uniquely identified assembly made up of a manufacturer part or parts, (for more information on manufacturers parts refer to *What are Manufacturers Parts?* on page 52).

Components can be represented by different symbols throughout the project documentation or exist purely as meta-data for inclusion in BOM and Parts Lists.

Altering component properties automatically populate changes to all associated symbols.

There are two distinct ways to manage components as follows:

- **Symbol > Component**
  Inserting a symbol into a drawing will automatically create a component. This is regarded as a symbol linked component, erasing or deleting the symbol automatically deletes the component.
- **Component > Symbol**
  Components can be created without having to insert any symbols or even have any drawings; for example costings can be checked by creating components and assigning parts, to obtain BOM and parts lists prior to starting any design work.

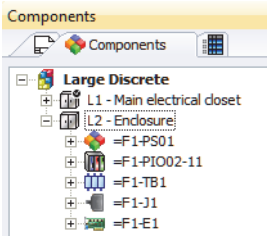
  Once a component is created symbol representation can be inserted, where this process is employed the component is regarded as *permanent* or *mandatory* to the design. These components are graphically identified with a continuum =F1-K3 icon on the component.

  Erasing related symbols will *not* delete the component.

  During this lesson the different ways of creating components will be explored.

Component Identification

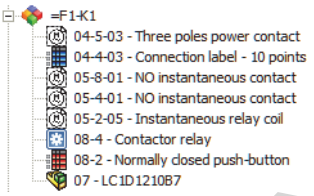
Components have their own side panel with different graphic representations denoting their type.



- Represents a standard component type. fuse, pushbutton, motor.
- Represents a PLC or programmable logic control.
- Represents a terminal strip, expanding the terminal strip will show different terminal types and states.
  - Standard terminal with associated scheme symbol.
  - Standard terminal with no scheme symbol.
  - Multi deck or multi level terminal.
- Represents a connector component.
- Represents a PCB or printed circuit board component.

Component Symbol Identification

The component content icons used in the tree are also meaningful illustrating the drawing and symbol type associated to them:



- means that the component has a representation in the form of a *scheme* symbol.
- means that the component has a representation in the form of a *wiring diagram* symbol.
- means that the component has a representation in the form of a *SOLIDWORKS* part.
- means that the component has a representation in the form of a *2D cabinet* symbol.
- means that the component has a representation in the form of a *Connection Label* symbol.
- means that the component's part(s) have representations in the form of *Connection Label* symbols.

Stages in the Process

The major stages in the process are listed below:

- **Create a symbol-component**
  Insert a symbol to create a component.
- **Delete a component automatically**
  Delete symbols to remove a component.
- **Create a component**
  Create a meta data component.
- **Insert a symbol from a component**
  Insert a symbol from a component and understand *permanency.*
- **Component association**
  Associate multiple symbols to an existing component.

Using an Archive File

To start the lesson, unarchive a project and open the file `Start_Lesson_04.proj` from the folder `Lesson04\Case Study`. For more information, see *Unarchiving a Project* on page 11.

Procedure

Different methods are used to create and remove components, create associations and insert symbols.

---

1 Create symbol-component.

Open drawing `03 - Electrical scheme` and **Insert symbol** .

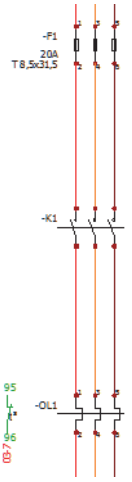
Locate and **Select** to insert the following symbol.

- Classification: `Contactor relays, relays`
- Description: `Three poles power contact`
- Name: `TR-EL035`

2 Position symbol.

Click to insert the symbol between -F1 and -OL1.

Click **OK** to accept the default Symbol properties settings.

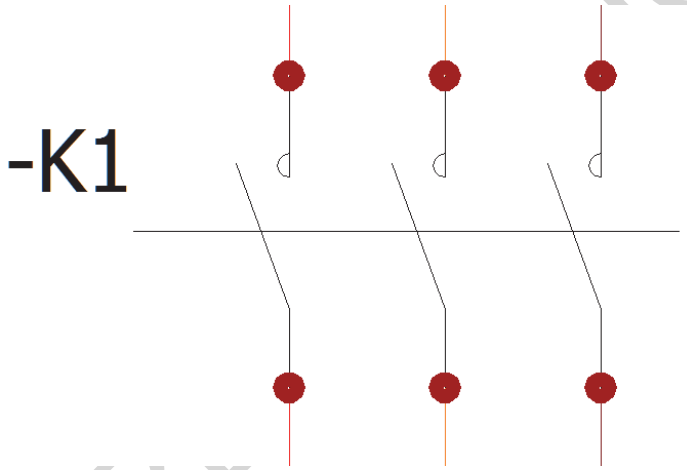


3 Find symbol-component.

Activate Components side panel and expand the location L1 - Electrical Enclosure and sub location L1 - Backplate to locate the component =F1-K1.

4 Go to symbol.

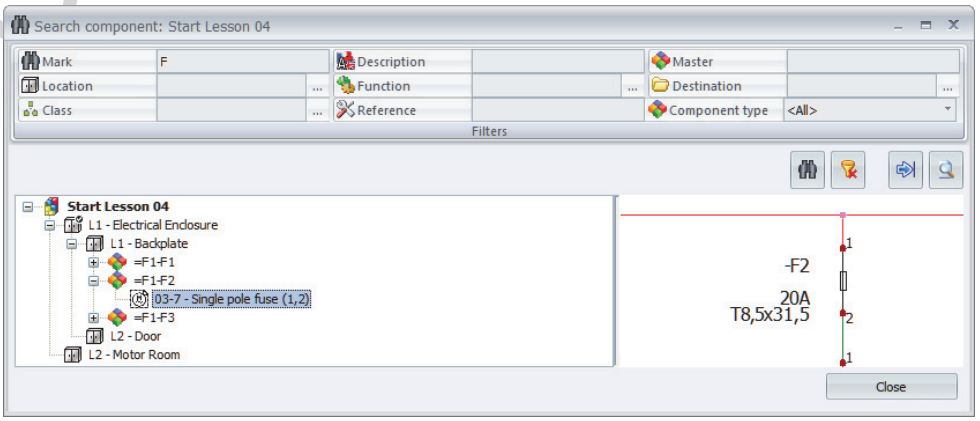
Expand component =F1-K1 and right-click components symbol and click **Go to** command to go directly to the symbol.



5 Find a component.

Right-click project name in the Components side panel, and click **Search component** .

In the Mark field type **F** to identify the fuses in the project.



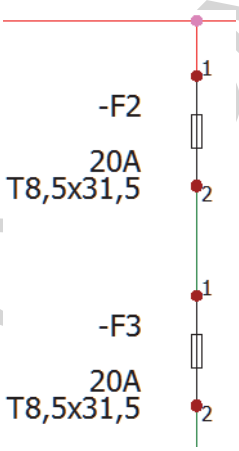
Highlight the scheme symbol for the second fuse, =F1-F2 and click **Go to drawing** .

Tip This will open any relevant drawing and zoom into the symbol.

Click **Close** to return to the drawing.

6 Component review.

Click **Pan** and move the drawing up to see the symbol below -F2.

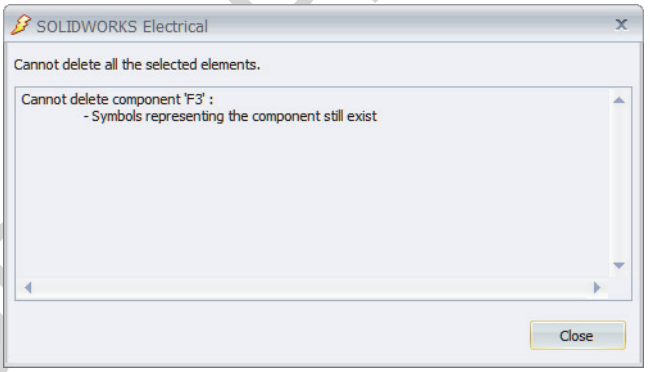


Right-click or press **Esc** to end the command.

On the Components panel expand location L1 - Electrical Enclosure and sub location L1 - Backplate.

Note The fuse -F3 is not required however the component cannot be deleted until all related drawing symbols are removed.

Right-click =F1-F3 component and **Delete component** click **OK** to confirm.



Click **Close** on the warning.

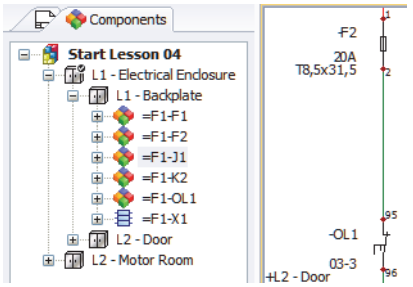
7 Delete symbol-component.

Click symbol -F3 in the graphic area and use one of the following methods to remove the symbol.

- Press the **Delete** key.
- Click **Erase** from the Modify tab.
- Right-click symbol and **Delete** .

Note By deleting the symbol three things occur simultaneously.

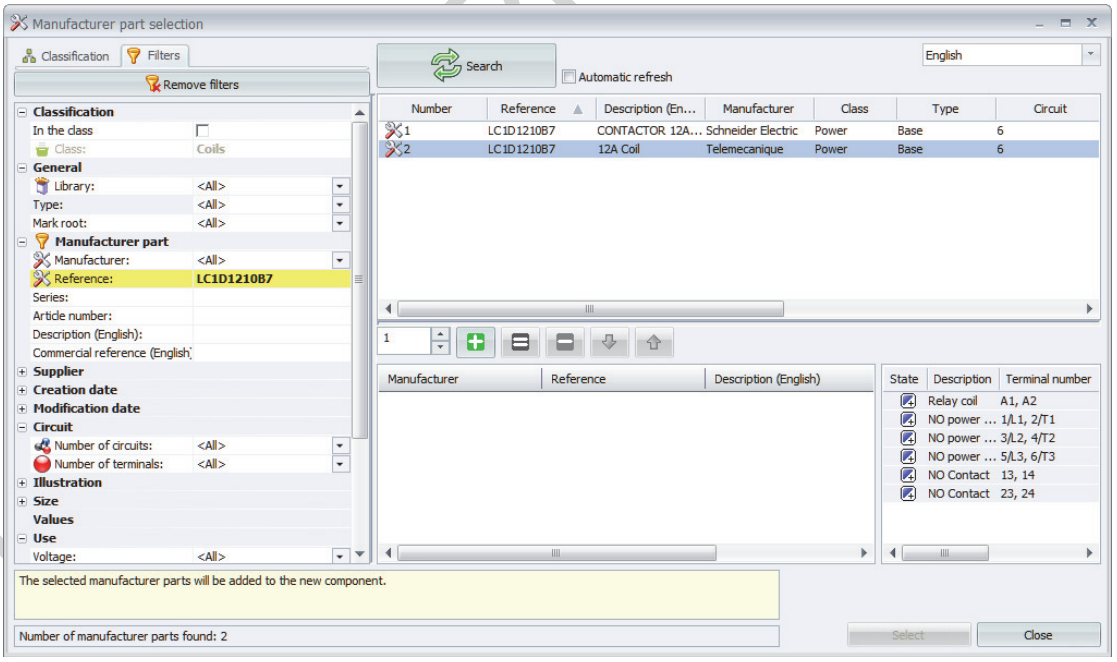
1. The symbol is deleted from the drawing.
2. The component is removed from the tree.
3. The wire is healed.



8 Create component from a part.
Right-click location `L1 - Electrical Enclosure` in the `Components` panel and click **New > Component manufacturers part**.

9 Locate a part.
On the `Filters` tab click the **Remove filters** button.

Type **LC1D1210B7** into the reference field and click the **Search** button.



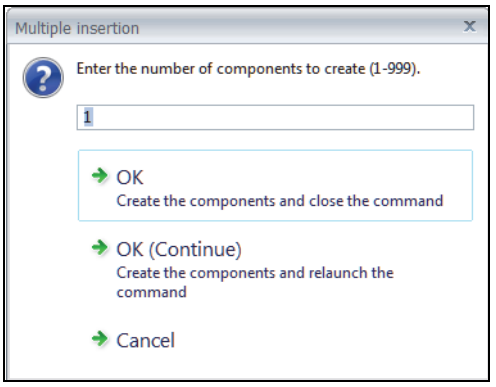
10 Add a part.
Highlight the listed part and click to **Add** the part.

Note Adding copies the part from the application to the project component.

Click **Select** to confirm.

11 Multiple components.

Leave the number of components as 1 and click **OK** to create a permanent or mandatory component.

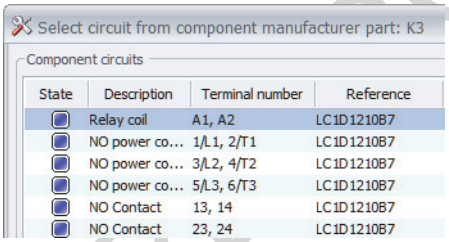


12 Insert component symbol.

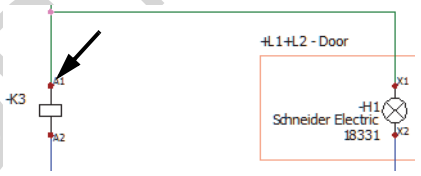
Right-click =F1-K3 component and click **Insert symbol** .

Click to **Insert symbol from manufacturer part circuits**.

Highlight Relay coil and **OK**.



Click to insert the symbol on the wires to the left of -H1.



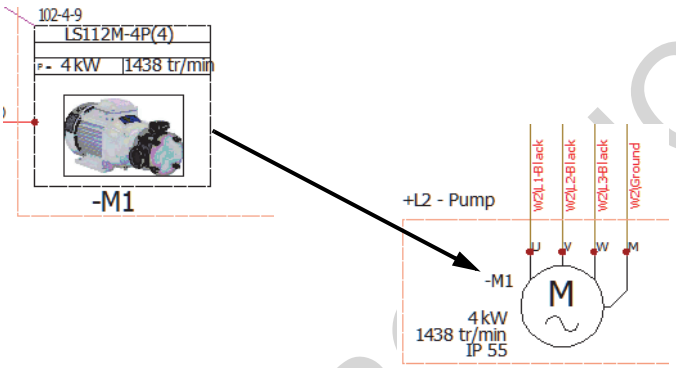
13 Change component permanency.

Right-click =F1-K3 component and **Set as non-permanent component** .

Symbol Component Association

Associations are made by applying the same *Mark* to different symbols, this process causes information from the component to automatically populate to associated symbols.

For example, a wiring diagram symbol was inserted which created a component `M1`. During the insertion process a manufacturers part was assigned in the `Symbol properties`. Later in the scheme design a motor symbol was inserted and was associated to the existing component `M1`. This automatically transferred all the technical data, and part information to the scheme symbol.



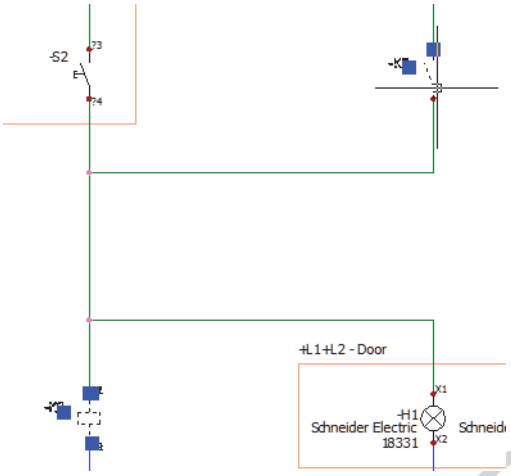
There are a number of ways to accomplish this association.

- Select the component from the tree in the `Symbols properties` and click **OK**.
- Set the `Root` and `Number` values so that they are equal to an existing component, click **Associate** and click **OK**.
- Select a symbol(s) and use the contextual menu select to **Assign components.** This activates the command side panel listing all the current components one of which can be selected and **OK** clicked to create the association.

The best method is dependent on the process being carried out.

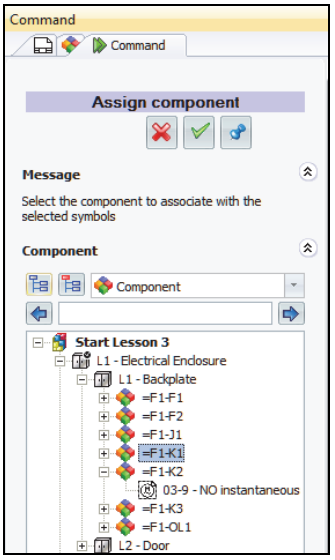
14 Assign symbols to component.

In the drawing use the **Ctrl** key to multi select symbols -K3 and -K2.



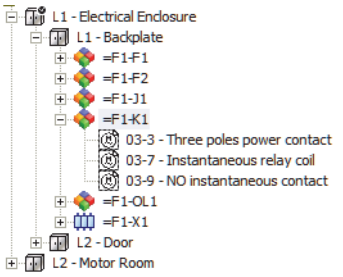
Right-click a highlighted symbol and select **Assign component**.

Highlight component =F1-K1 and click **OK**.



Note

Several changes occur when this operation is carried out. The non permanent components -K2 and -K3 are automatically removed as they no longer have any symbols associated to them, and -K1 is shown with three associated scheme symbols.



Another change is that the manufacturers parts that were applied to components -K2 and -K3 are removed. The *parent* component the symbols were assigned to is -K1 which had no part.

If the process had assigned to -K2 or -K3 then the part would have been retained and applied to all the associated symbols.

Click to **Cancel** the command.

15 Close the project.
Right-click project name in the Documents side panel and click **Close** .

Exercise 2: Symbols and Components

Unarchive a project and use different methods to create components and insert a symbol from an existing component.

This lab uses the following skills:

- *Unarchiving a Project* on page 11
- *Insert component symbol.* on page 43
- *Create symbol-component.* on page 39
- *Attribute visibility.* on page 33

Procedure

Complete a mixed scheme drawing using different methods to display components in the design.

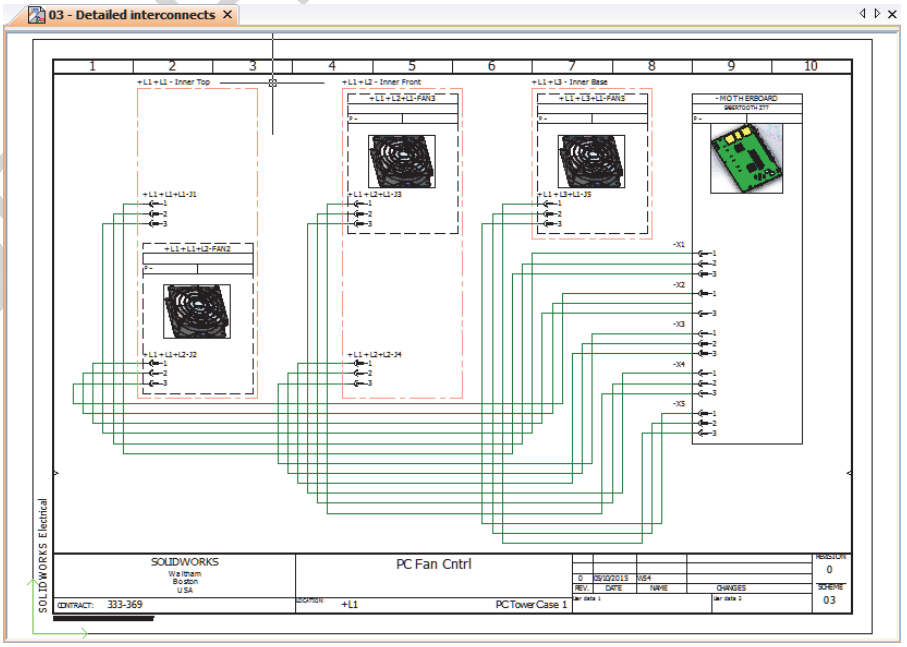
1. **Populate data to the application.**
   Unarchive the project located in `Lesson04\Exercises` folder.

2. **Data selection.**
   Click to **Update data** using the **Next** button to review data for processing.

3. **Complete the unarchive.**
   Finish the unarchive process leaving the settings as they were by default.

4. **Open the project.**
   Click **Yes** to open the project.

5. **Open a mixed scheme.**
   Open mixed scheme drawing `03 - Detailed interconnects`.



6 Line diagram component symbol.

On the components panel use the contextual menu to **Insert symbol**

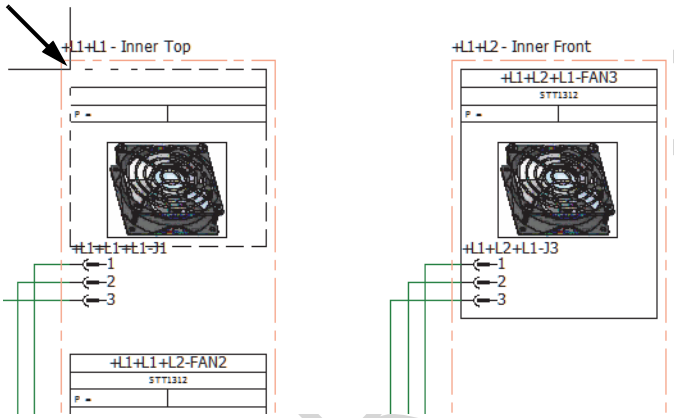
 for the fan `=F1-FAN1`.

Tip

As this is a mixed scheme both line diagram and schematic tools and symbols are available.

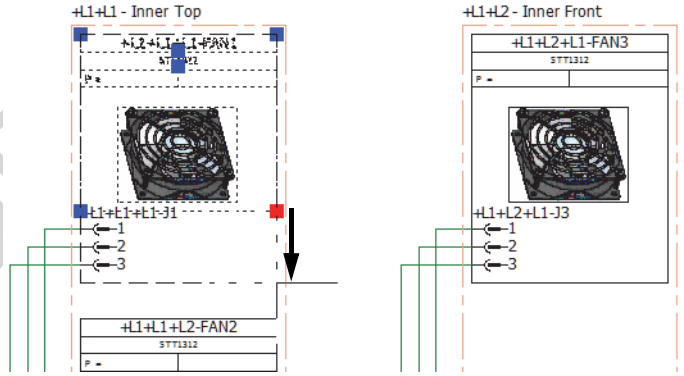
7 Insert symbol.

Click to insert a Line diagram symbol and place the symbol as illustrated.



8 Resize the symbol.

Click the symbol and drag the lower right hand grip as illustrated to resize the symbol.

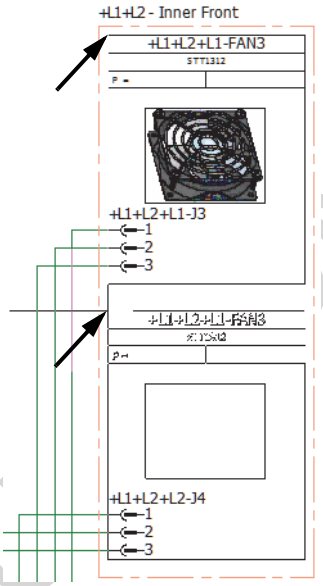


9 Create component from Copy.

Click the line diagram **Multiple copy** command.

Click and confirm the selection of +L1+L2+L1-FAN3 only.

Specify a base point on the upper left of Fan3 drag the cursor down and left-click again to position the copy as illustrated.



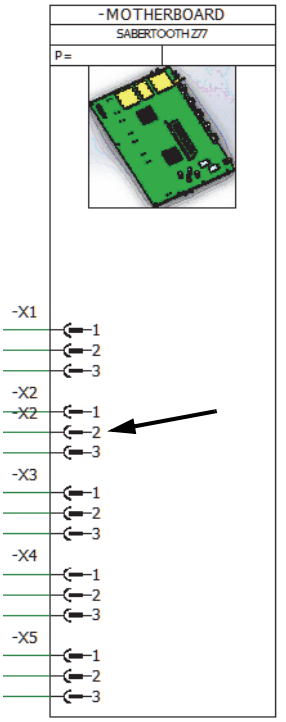
10 Insert scheme symbol.

On the Components panel right-click =F1-X2 and click **Insert symbol**.

Select to insert a schematic symbol.

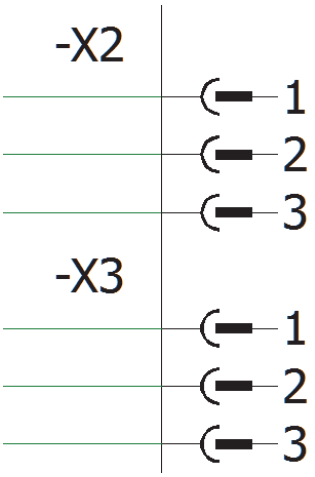
- Classification: Connectors
- Description: Female pin Training
- Name: TR-PIN_F_02+1

Click to place between pins 1 and 3.



11 Turn off attribute.

Right-click the newly inserted symbol and turn off the `Symbol mark` attribute.



12 Close the project.

Right-click project name in the Documents side panel and **Close** the project.

Lesson 3
Manufacturers Parts

Upon successful completion of this lesson, you will be able to:

- Understand manufacturers parts.
- Obtain parts on-line.
- Locate manufacturer parts.
- Modify a project level component part.
- Change circuit association.
- Employ circuit symbols.

What are Manufacturers Parts?

Manufacturers parts are key to understanding the flow of information in SOLIDWORKS Electrical and should not be confused with SOLIDWORKS parts, (*.SLDPRT files).

Manufacturers parts represent the physical part(s) that will be ordered, purchased, then installed and wired up.

Parts contain a range of different technical data related to classification, each class having different technical characteristics. For example motors have different fields compared to pushbuttons.

Default symbols can be associated to parts in the form of scheme, line diagram symbols, 3D *SLDPRT* files, 2D footprints, connection label, and PCB *EMN* files and terminal strip symbols. This acts as a time saver when inserting symbols from components.

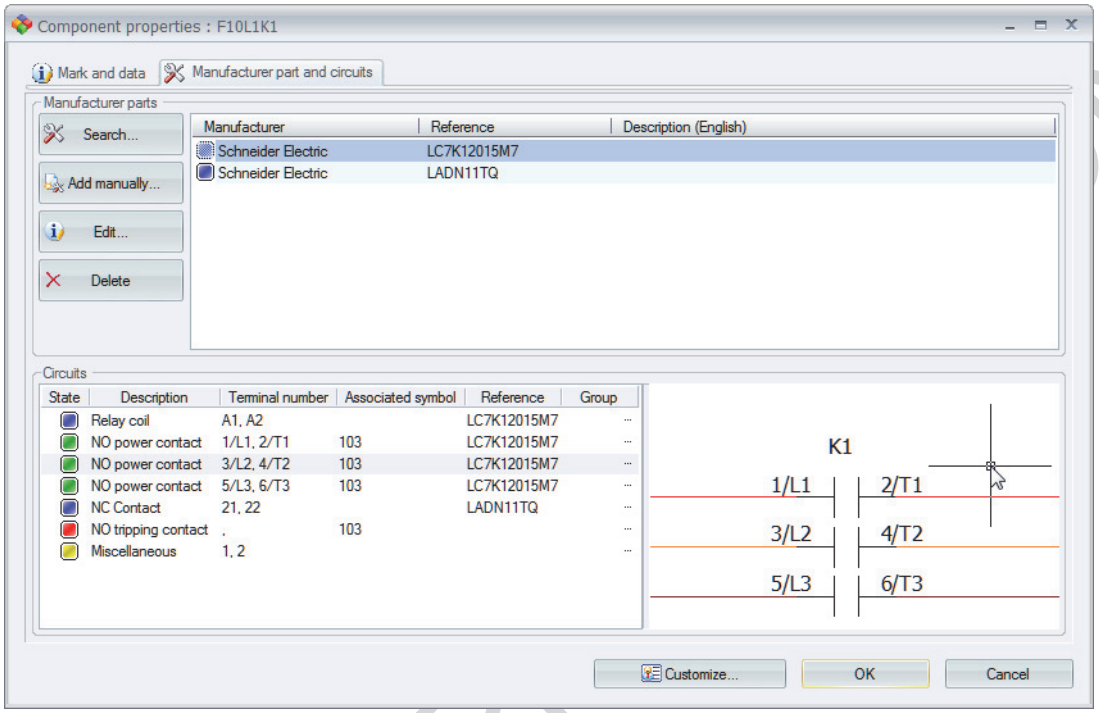
By default only a few manufacturers parts are added to the database after installation, this reduces unwanted system data and processing time when first launching the program.

It is possible to selectively download and **Unarchive** manufacturers parts data using the **Online content** from the Electrical content portal.

Circuits and Terminals

Parts include the number and type of circuits available as well as terminal marks, (pin numbers). These are compared against scheme symbol circuits they are applied to providing valuable design and ordering viability information.

Circuit associations are most commonly seen in symbol and component properties in the form of graphically displayed State information.



The different state colors are as follows:

Question

Does anyone in the class have difficulty differentiating colors green, red, blue? Can the colors be changed if that is the case?

- **Available, spare circuit**
  A blue state indicates a part applied to a component or symbol, but no graphical representation exists in the scheme's that matches the circuit type.
- **Assigned, correctly matched circuit**
  A green state indicates a part circuit has been applied to a component or symbol, and there is a scheme symbol with a matching circuit.
- **Unavailable, design issue circuit**
  A red state indicates a scheme symbol has been used but either no part has been applied, or there is no matching part circuit type.

Note

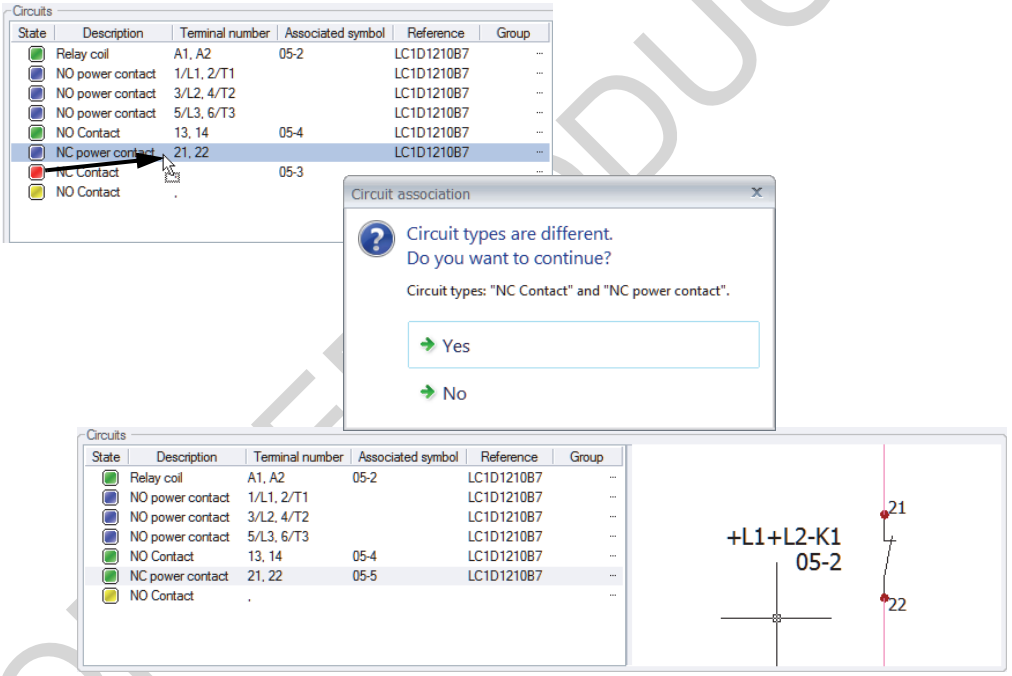
Where a part has been applied but the red circuit type is displayed it means that the part being ordered will not accommodate the design and production delays may occur.

- **Virtual circuit**
  A yellow state indicates a virtual circuit. These can only be added manually and are not directly related to either scheme symbols or manufacturers parts. Virtual circuits can be added to components where a known connection is required, but the design is not sufficiently advanced to know the required parts, and the scheme design is yet to be completed.

Circuit Association

Matching scheme symbol and part circuits automatically associate to one another.

Where no match is found circuits can be manually forced to create an association. This is done using drag drop from a red state circuit onto a blue or green part circuit.



Only virtual circuits cannot be associated this way. Virtual circuits must correctly associate to a manufacturers parts circuits to attain a match.

Tip

This limitation allows the lead engineer to limit parts assigned in other areas of the design to *only* those with matching circuits.

Stages in the Process

The major stages in the process are listed below:

- **Download on line parts**
  Through the parts manager download a manufacturers parts file and unarchive it.
- **Find parts.**
  Locate specific parts using classifications and filters.
- **Edit a part.**
  Modify a part applied to a component and learn the update processes available.
- **Circuit association.**
  Manually modify the parts circuit-symbol association.
- **Circuit symbols.**
  Insert a symbol that has been applied to a manufacturers part circuit.

Using an Archive File

To start the lesson with an up to date project, unarchive and open the file `Start_Lesson_05.proj` from the folder `Lesson05\Case Study`. For more information, see *Unarchiving a Project* on page 11.

Procedure

Employ the parts manager to obtain manufacturer data on line, learn to apply parts to symbols using filters and modify parts *on-the-fly* without design interruptions. Manually alter circuit-symbol associations and insert symbols that are associated to parts circuits.

Where to Find It

- CommandManager: **Library > Manufacturer parts / Cable references Manager** 

Tip

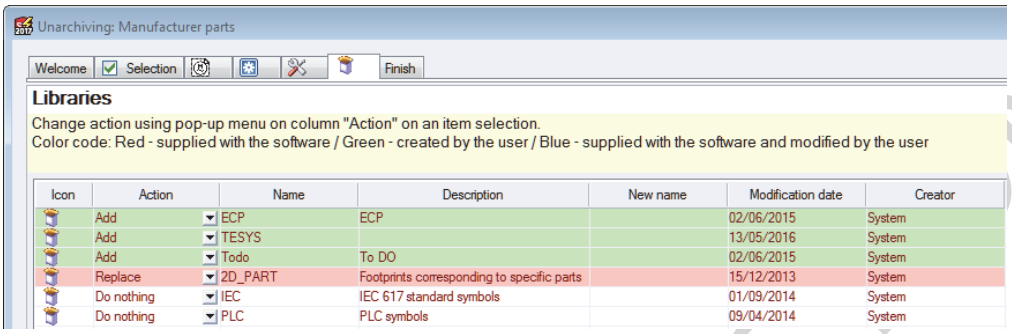
When **Unarchiving** it is possible to select multiple *.tewzip files.

---

1 **Unarchive parts.**
In the `Manufacturers parts manager` click to **Unarchive manufacturers parts**  .

Browse to the download location of the `Lesson05\Case Study` folder and select `Schneider_electric.part.tewzip` file and click **Open**.

2 Unarchive wizard.
Click **Next** twice and ensure the Add option is set to **Update**.

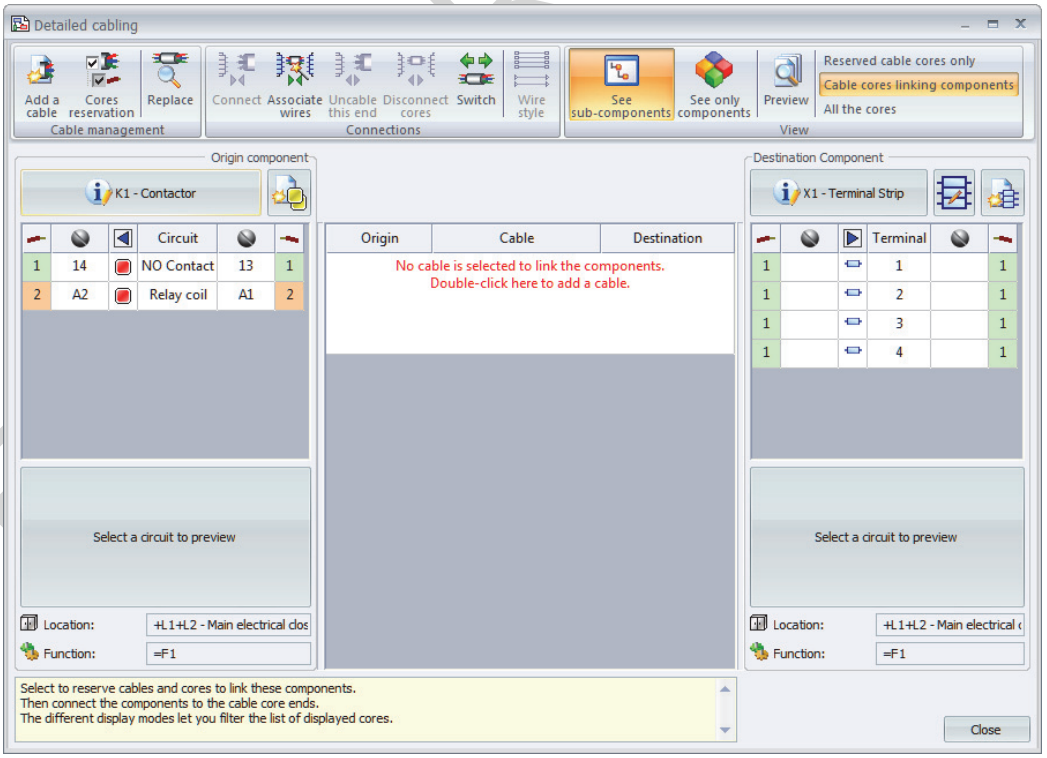


Click **Next**, **Finish** and **Finish** to end the unarchive process.

Click **Close** to exit the manager.

3 Detailed cabling.
Open drawing 03 - Line diagram.

Right-click cable interconnecting -X1 and -K1 and click **Detailed cabling**.



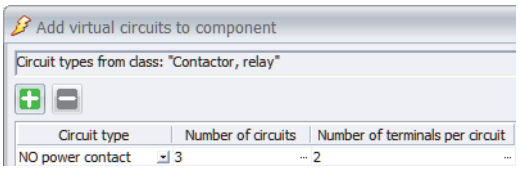
4 Assigning virtual circuits.

On the origin component K1 - Contactor click

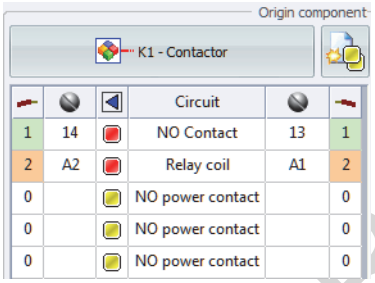
Add virtual circuits .

Click **Add** and change the Circuit type to **NO power contact**.

Change the Number of circuits to **3**.



Click **OK** to create the virtual circuits.



Click **Close** to return to the drawing.

Finding Manufacturer Parts

Parts can be applied to symbols or components through their respective properties dialogs or by accessing a symbol or components contextual menu and selecting **Assign manufacturers parts**.

The latter option reduces the number of steps required to complete the procedure as no properties dialog is accessed however limits options to make modifications, or apply other parts.

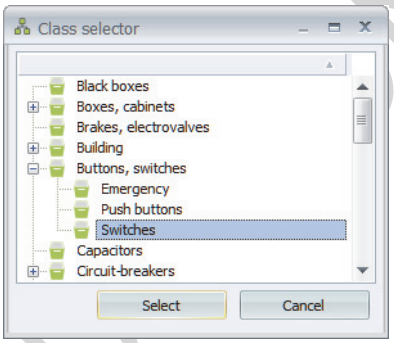
Note

Applying parts to a component or symbol achieve the same results as a symbol is a graphic representation of a component.

Search Options

Searching for manufacturer parts can be made easier by making use of the search Filters tab. Filtering will generally reduce the volume of results and make it easier to select the proper part. The following is a list of the filters available. Cables are also parts and have some slightly different options.

- **In the class**
  Click the **In the class** option to limit the search to a specific class of part such as Push buttons or Motors. The classes are listed in the Class selector which can be browsed when the option is selected.



Note The In the class option also limits the search. Try clearing the check box if the search results are not satisfactory.

- **Library**
  Selects the library to search. Unless there is a user library, All is used.
- **Manufacturer**
  Limits the search to parts by manufacturer name such as GE or Square D.
- **Type**
  Selects the type: Base, Auxiliary or Accessory for example to filter the search.

  Base: Base parts are used for components that have a single reference or are the main reference in components with multiple references.

  Auxiliary: Auxiliary parts are parts used only used in conjunction with a Base reference because they are connected to the base. They also perform an electrical function.

  Accessory: Accessory parts are (like Auxiliary parts) used in conjunction with a Base part, but do not have an electrical function. An accessory can be something like screws to fix the part to the cabinet or a text plate explaining the function.

Note PLC components have their own unique set of types.

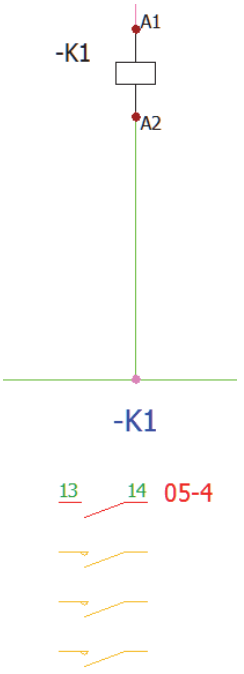
- **Reference**
  Text typed in the Reference field searches the Part name for a match. For example, the part name of a luminous signaling alarm is XACV06. The partial name XACV can be used to narrow the search.
- **Description**
  Text typed in Description field searches the parts for a match. For example, the description of a luminous signaling alarm is P LIGHT FOR XAC-B DIRECT. The partial description LIGHT can be used to narrow the search.
- **Article**
  A secondary reference or name for the part, often one that is used internally.
- **Circuit**
  Select the Number of circuits or Terminal number from a pull down list.
- **Use**
  Select the Use Voltage or Frequency from a pull down list.
- **Control**
  Select the Control Voltage or Frequency from a pull down list.

---

5 Zoom to symbol.

Close the wiring diagram and open the scheme drawing 05 - Control.

Click to **Zoom** in on the relay -K1.



Note The component currently has no part applied to it, but it does contain virtual circuits and has an associated contact, these are shown below the relay coil with the same color coding as those used for circuit states.

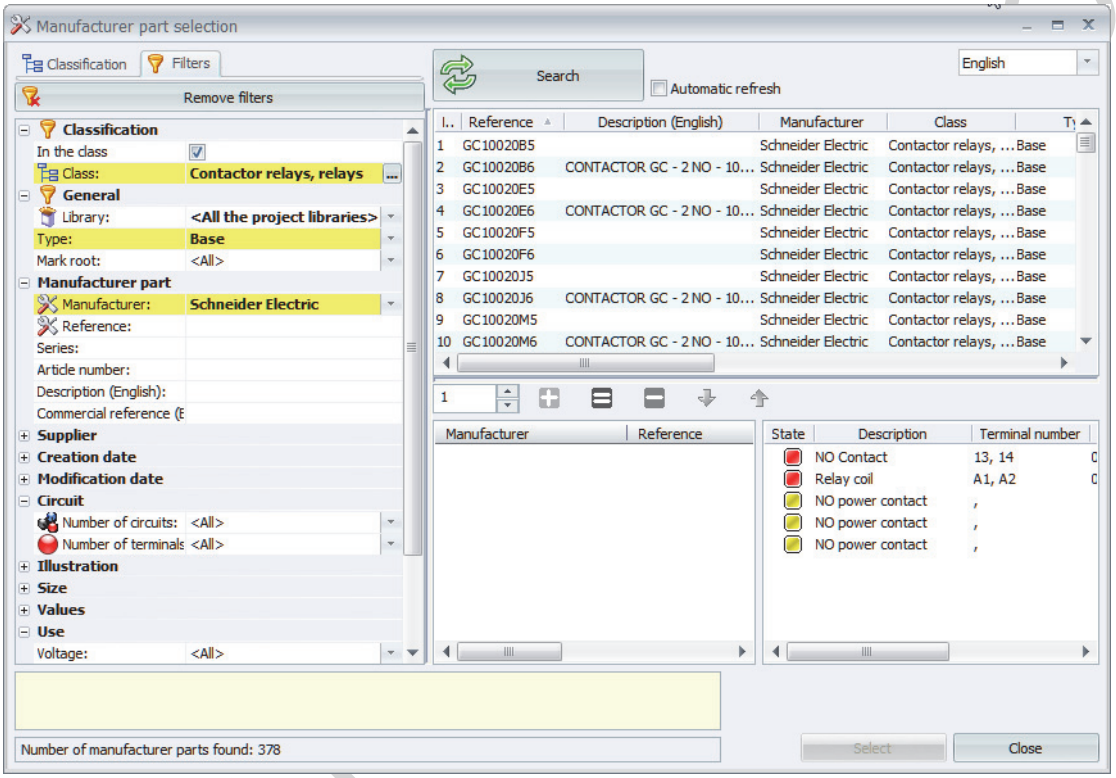
Question Are there any other ways to zoom direct to a symbol?

6 Searching for a part.

Right-click -K1 and click **Symbol properties** on the Manufacturer part and circuits tab click **Search** to access Manufacturers part selection.

7 Filter parts.

On the Filters tab use the following criteria.

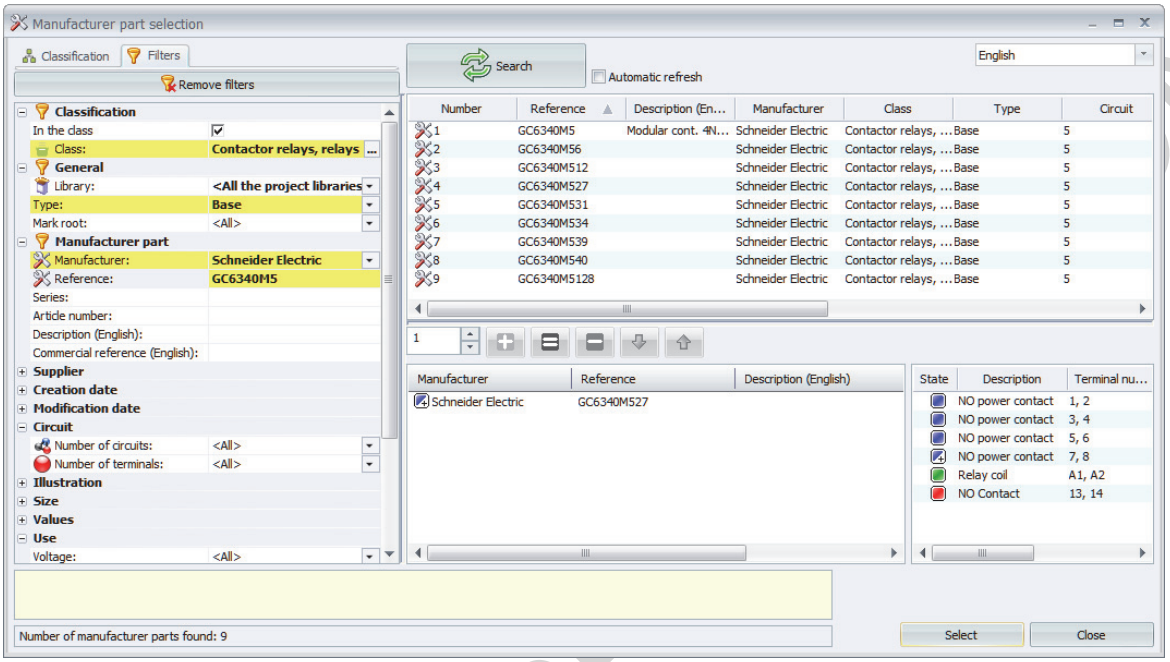


Click **Search** to update the listed parts.

Tip Check on the Automatic refresh option to have the parts list dynamically update without having to **Search**.

8 Refined filter.

The list includes several hundred returns, in order to reduce the number of parts type GC6340M5 into the Reference field and click **Search**.



Highlight the part GC6340M527 and click to **Add** it to the symbol.

Click **Select** to confirm the changes.

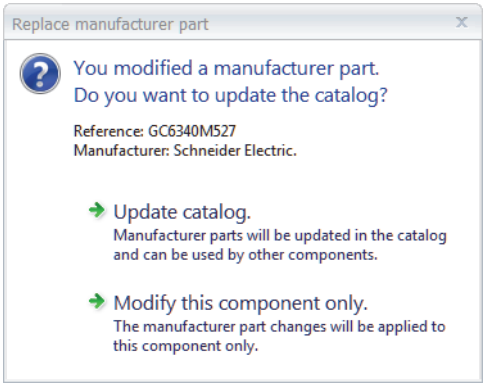
Note

Circuit states showing that are only half filled with + indicates a pending confirmation, until *selected* and *OK* is clicked it is possible to *cancel* the command.

Editing Parts

Parts can be edited at an application level through the Manufacturers parts manager, or from the Symbol or Component properties dialog.

When a change is made to a project level part, via the Symbol or Component properties dialogs, modifications can be made to either the component, or to update the application level catalog.



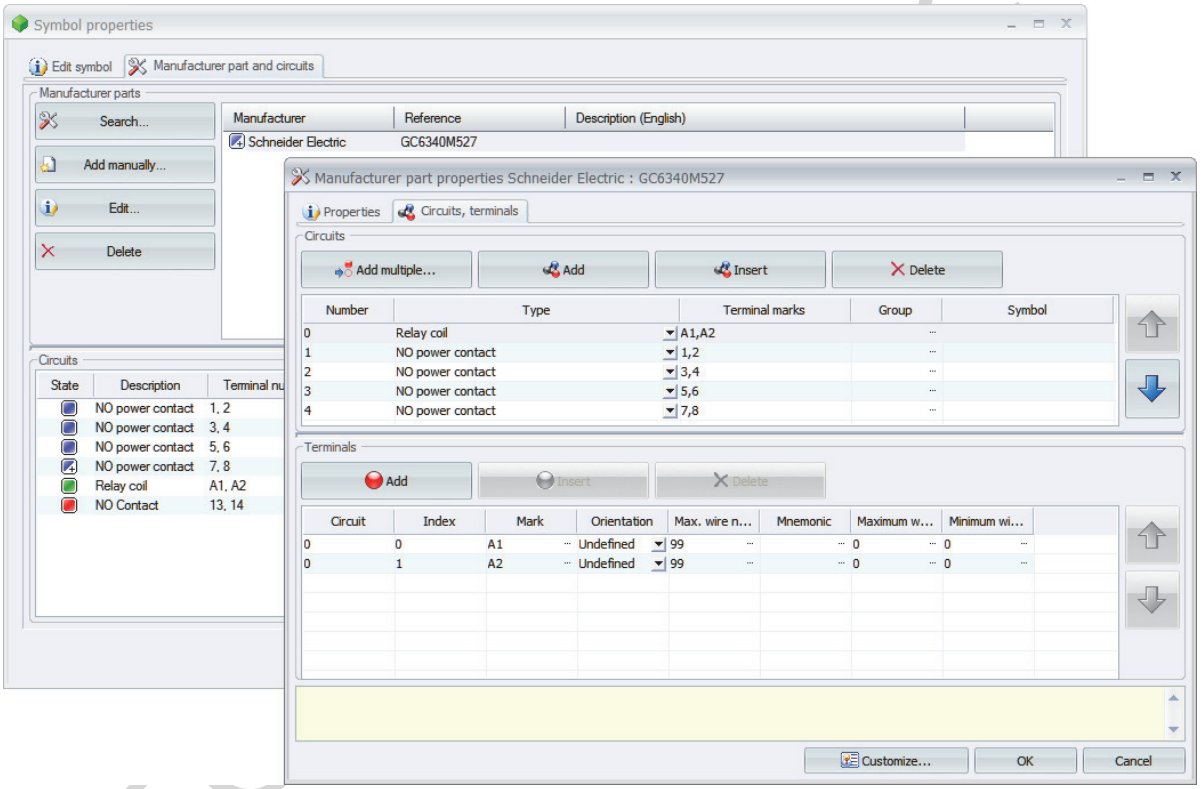
Updating the catalog writes the changes to the application level which will be seen any time the part is used in the future, regardless of project.

Modifying only the component part will alter the part applied to a single component, it will not change the part anywhere else.

9 Change circuit order.

Highlight manufacturers part Schneider Electric GC6340M527 and **Edit**.

On the Circuits, terminals tab move the Relay coil circuit to the top of the list using the arrow button.



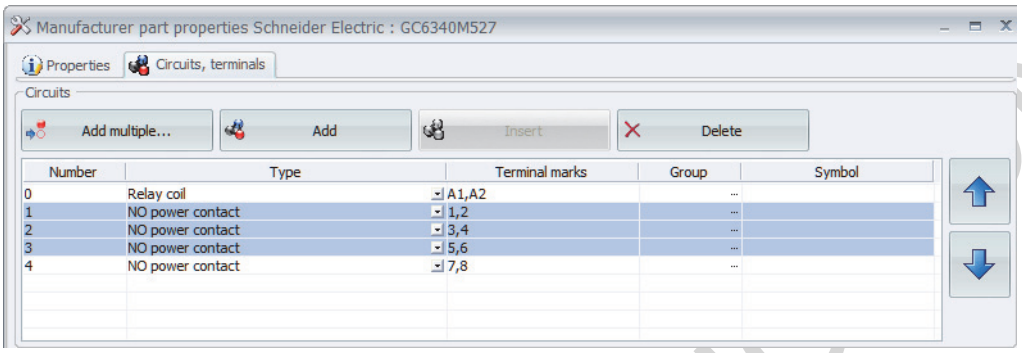
Circuit Symbols

Symbols can be applied to the different part circuits, these symbols can then be used when inserting components as symbols in schemes. This can be useful when regularly creating components for costing purposes before carrying out the detailed design.

Another advantage of associating symbols to part circuits is that it can reduce the chance of erroneous association of unrequired contactors to a relay coil, for example.

10 Part circuit symbols.

Select NO power contact circuits 1, 2 and 3.

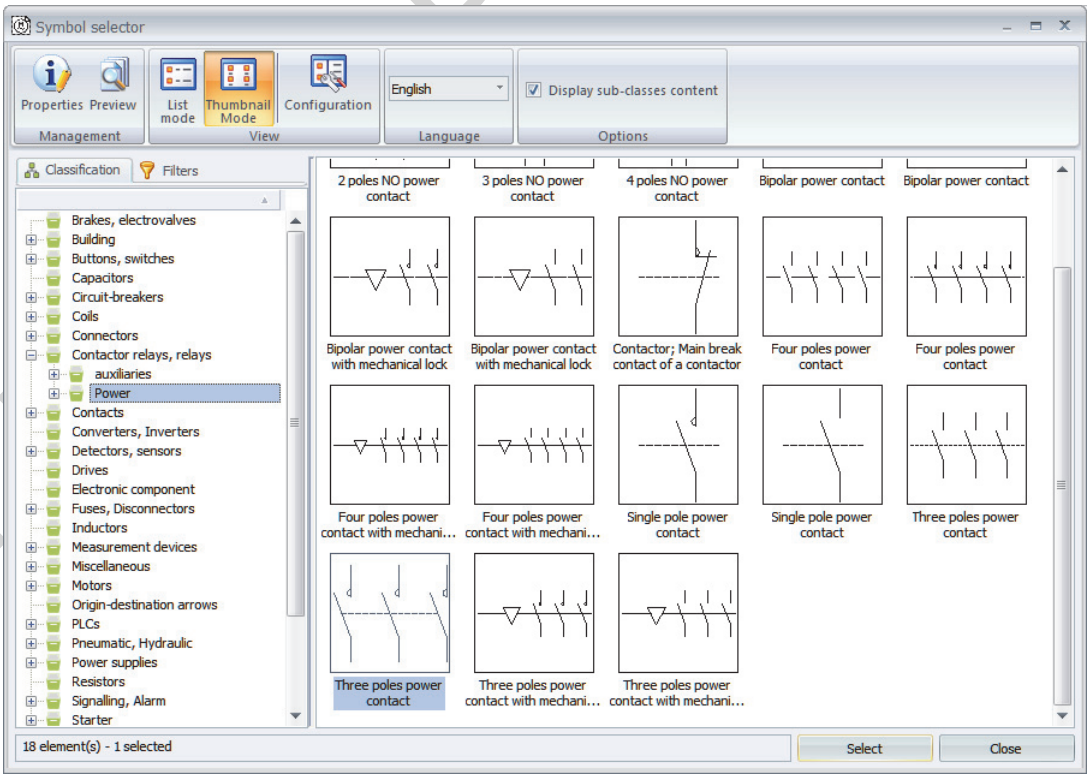


Right-click Symbol column and click **Assign symbol** to access the Symbol selector.

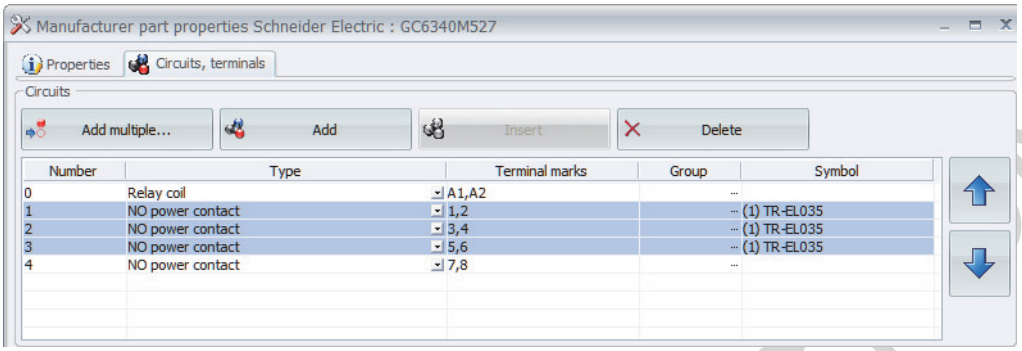
11 Locate symbol.

Locate the following symbol.

- Classification: Contactor relays, relays > Power
- Description: Three poles power contact
- Name: TR-EL035



Click **Select** to assign the symbols to the circuits.

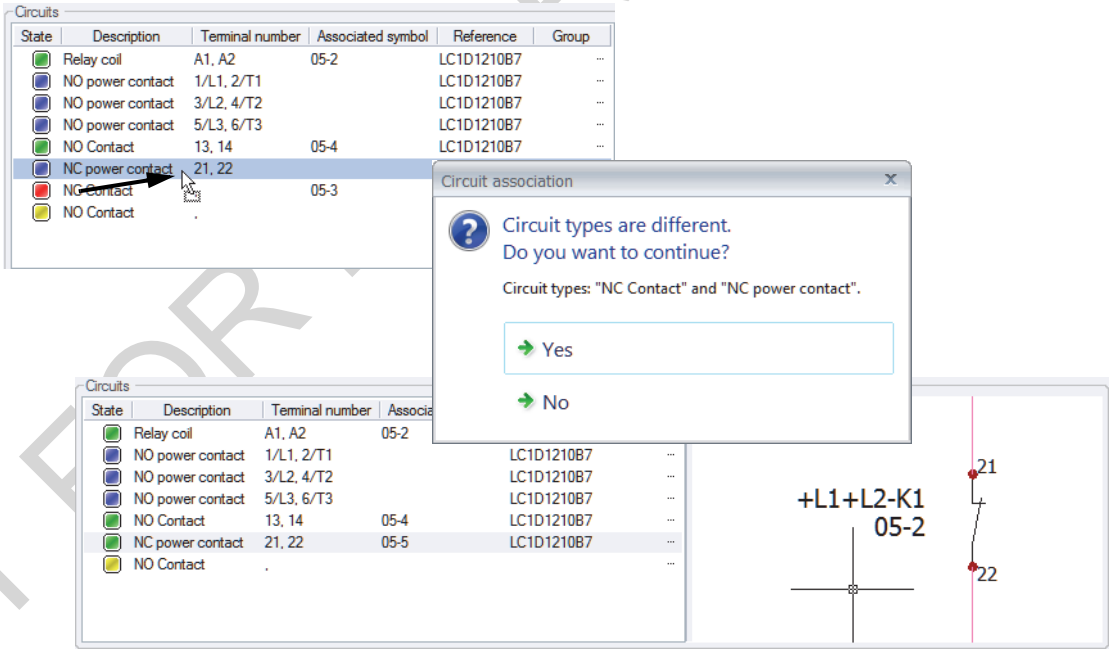


Click **OK** to confirm all the changes and select to **Modify this component only**.

Circuit Association

Circuits will automatically associate to one another where a match is found between a scheme symbol and a parts circuit.

Where a match is not found it is possible to override the program by using drag drop between a symbols red state circuit onto a blue or green part circuit.



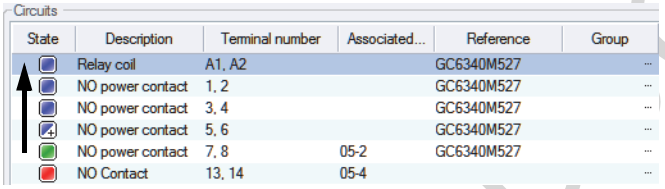
The only circuit type which cannot be associated by dragging and dropping in this way are virtual circuits. The virtual circuit must be correctly associated to a manufacturers part circuit in order to attain a match.

This limitation allows the lead engineer to limit the parts being assigned in other area's of the design to only those that have matching circuits.

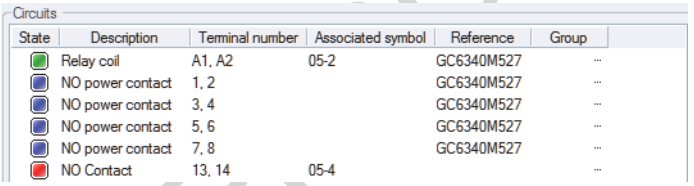
---

12 Symbol circuit association.

Drag drop the green circuit state NO power contact onto the Relay coil circuit.



Click **Yes** to change the circuit type.



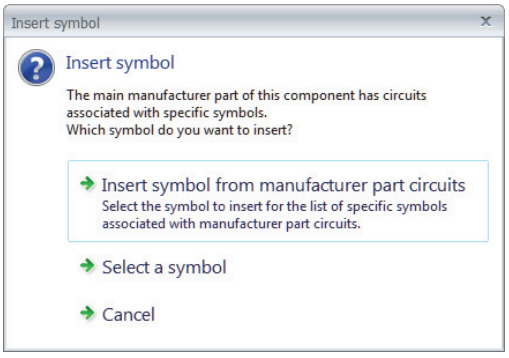
Click **OK** again to confirm the changes.

13 Insert component circuit symbol.

Open drawing 04 - Power.

Expand location L2 - Main electrical closet and right-click =F1-K1 - Contactor component.

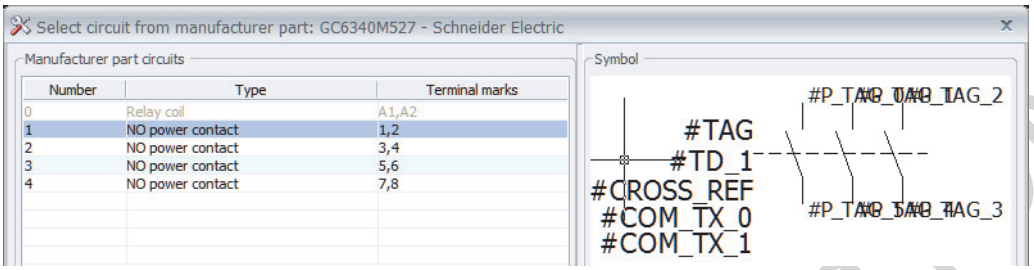
Click **Insert symbol** .



Click **Insert symbol from manufacturer part circuits**.

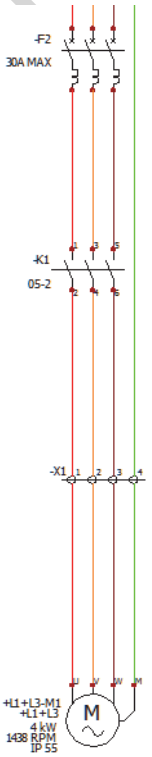
14 Select circuit symbol.

Highlight line Number 1 - NO power contact.



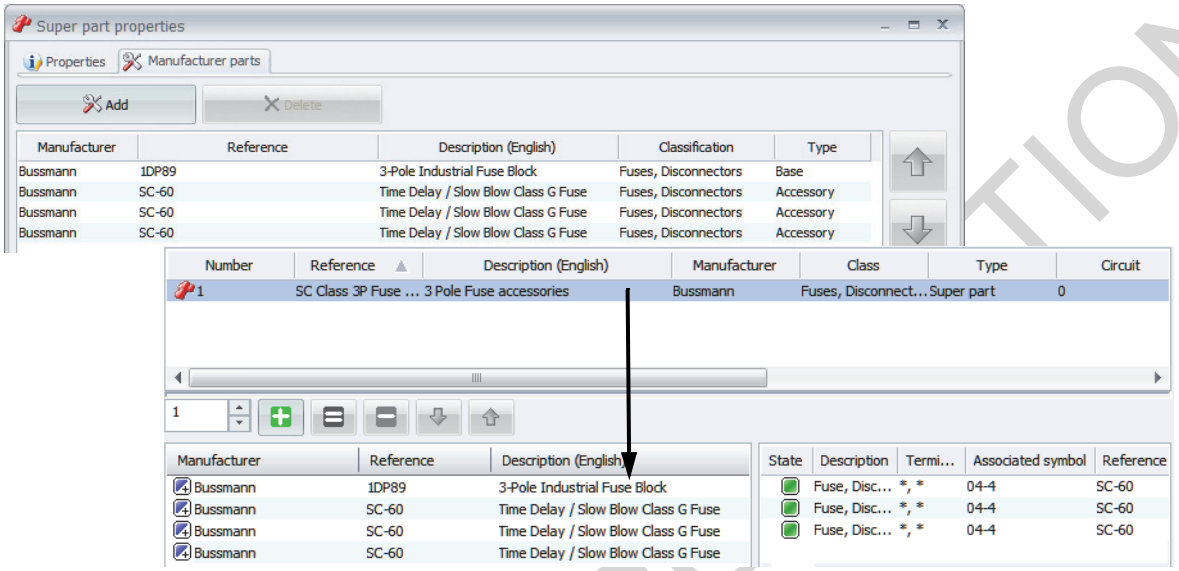
Click **OK** to return to the drawing.

Position the symbol between -F2 and the terminal strip -X1 and left-click to insert.



Super Parts

Super parts are theoretical parts that can be made up of multiple individual manufacturers parts. When assigning a super part to a project component, only the parts which make up the super part will be applied.



This provides a quick way to save and apply parts made up of accessories.

Using an Archive File

To start this section of the lesson unarchive and open the file `Bussman Fuse Parts.part.tewzip` from the folder `Lesson05\Case Study`. For more information, see *Unarchive parts.* on page 55.

Procedure

Employ the parts manager to create a super part and apply it to a project component.

Where to Find It

- CommandManager: **Library > Manufacturer parts** > **Add manufacturers part** > **Add Super part**

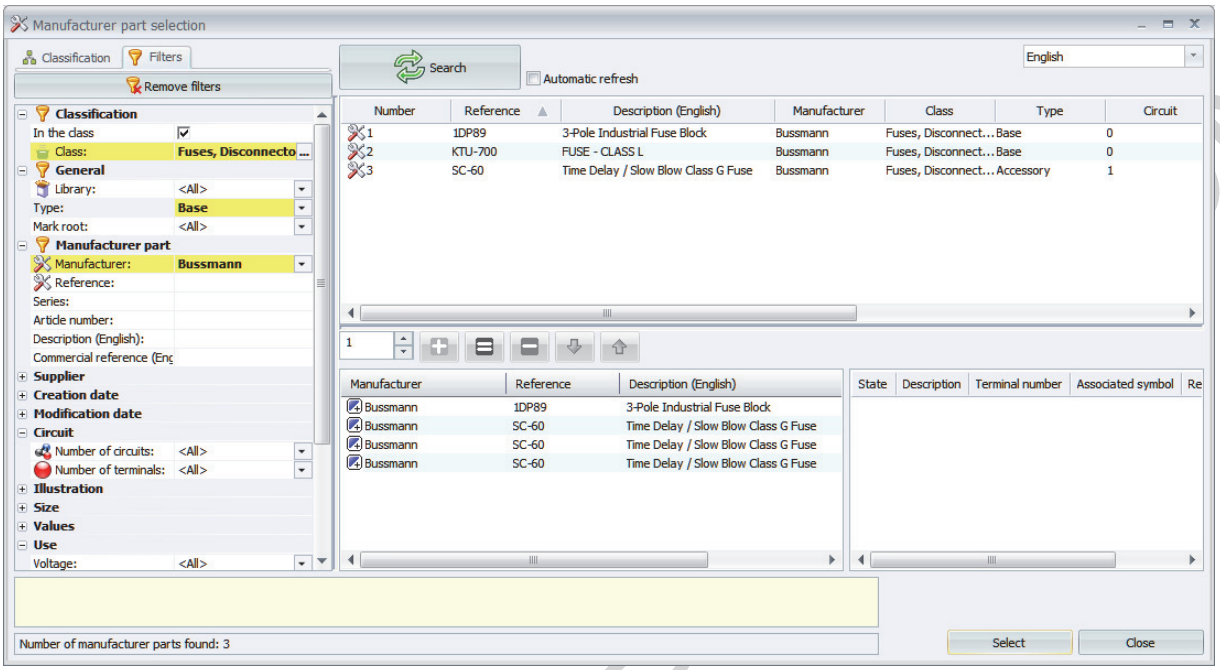
---

15 Create super part.

Click **Add Super part** and then click Add to define the parts that make up the super part.

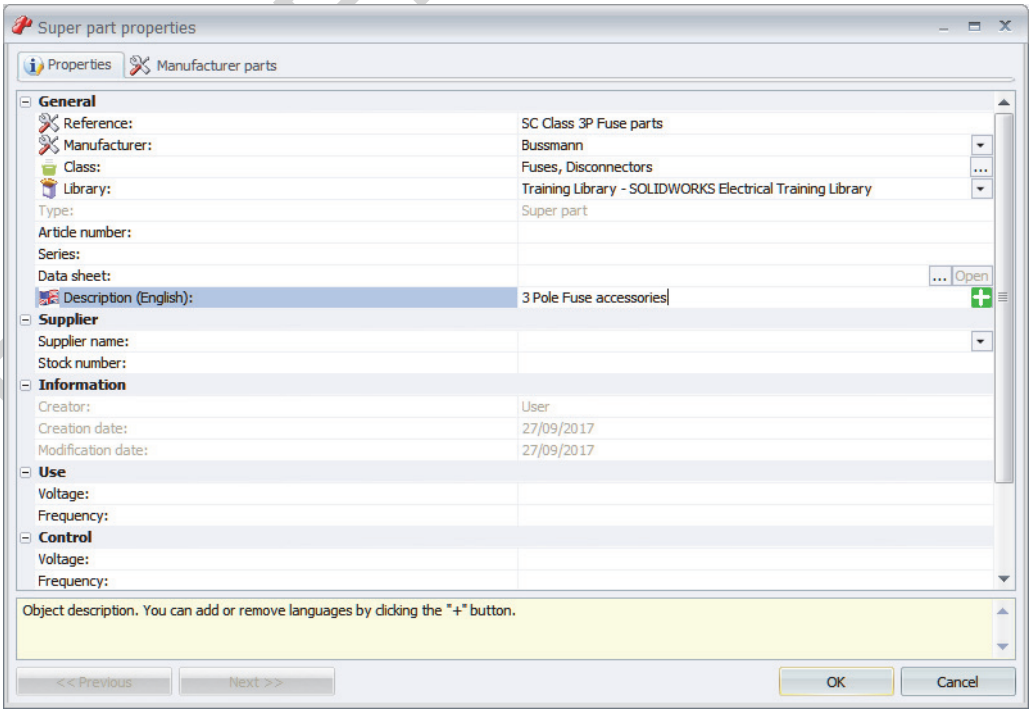
16 Add parts.

Search for and then add the following parts and click **Select**.



17 Super part data.

Click **Properties** and fill in the properties as follows.



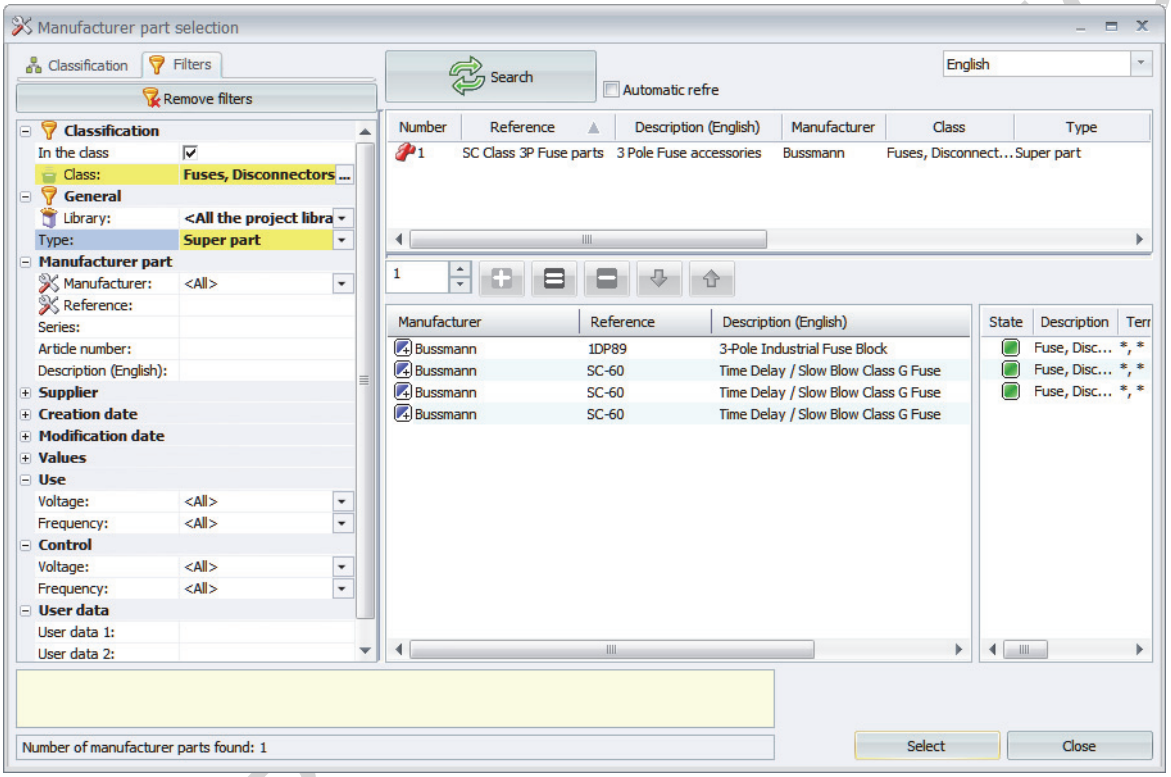
Click **OK**.

18 Locate symbol.

Click **Close** to exit the Manufacturers parts manager and **Open** drawing 04-Power and zoom into fuse -F2.

19 Apply Super part.

Right-click on -F2 and click **Symbol properties**, click Manufacturer parts and circuits tab and search for and add the newly created super part as follows.



Click **Select** and **OK** to confirm and return to the drawing.

20 Close the project.

Right-click project name in the Documents side panel and click **Close** .

Exercise 3: Manufacturers Parts

Unarchive a project and catalogs, find and apply parts to a component, manually override circuit association.

This lab uses the following skills:

- *Unarchive parts.* on page 55
- *Unarchive wizard.* on page 56
- *Find a component.* on page 40
- *Finding Manufacturer Parts* on page 57
- *Filter parts.* on page 60
- *Refined filter.* on page 61
- *Symbol circuit association.* on page 65

Procedure

Apply matching parts to a component.

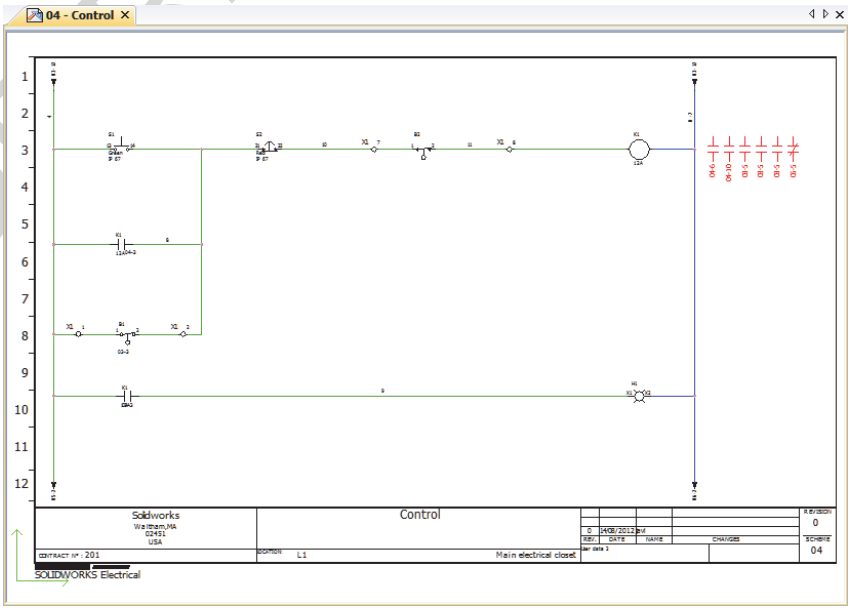
1 **Populate data to the application.**
Unarchive the project located in `Lesson05\Exercises` folder.

2 **Data selection.**
Click to **Update data** using the **Next** button to review data for processing.

3 **Complete the unarchive.**
Finish the unarchive process leaving the settings as they were by default.

4 **Open the project.**
Click **Yes** to open the project.

5 **Open a scheme.**
Open scheme drawing `04 - Control`.



6 Unarchive parts.

Open the Manufacturers parts manager and **Unarchive manufacturers parts** .

Browse to the Lesson05\Exercises folder and select and open all the part archives.

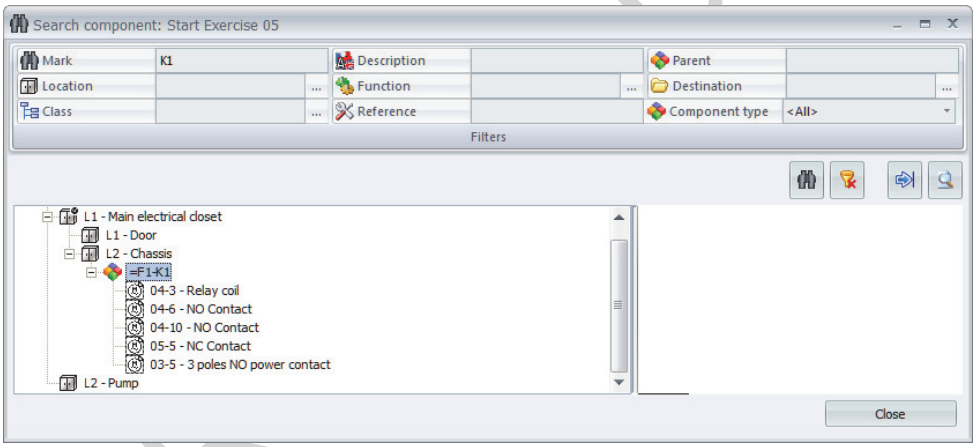
7 Unarchive wizard.

Run through the wizard selecting to update for each archive.

When all three archives are processed close the manager to return to the drawing.

8 Search for component.

On the components panel use the **Search component** command to locate component K1.



9 Component properties.

Right-click the component and open its **Properties** .

10 Find and apply parts.

Search for a part that match the following criteria.

- Classification: None
- Type: Base
- Manufacturer: Schneider Electric
- Reference: GC2530B

Select and add the part that has 4 matching circuits with no spares.

Search for another part that matches the following criteria.

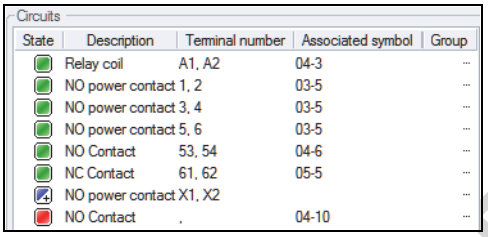
- Classification: None
- Type: Auxiliary
- Manufacturer: Schneider Electric
- Reference: LA1DN

Select and add the first listed part that has two matching circuits with no spares.

Search for another part that matches the following criteria.

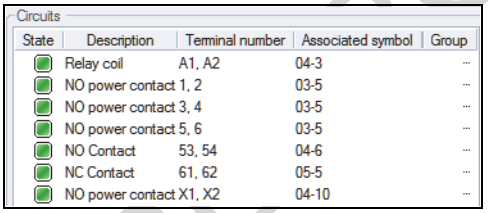
- Classification: None
- Type: Auxiliary
- Manufacturer: Schneider Electric
- Reference: LA1LC080

Select and add the first listed part, and confirm the selections to return to the component.



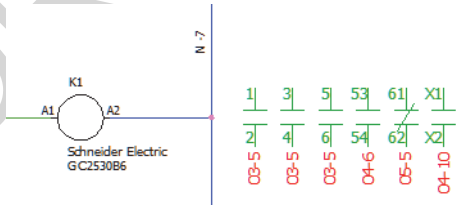
11 Force circuit association.

Force the NO Contact to associate with the NO power contact.



12 Results.

Confirm the changes and return to the drawing to review the cross reference information on relay coil -K1.



13 Close the project.

Right-click project name in the Documents side panel and **Close** the project.

Lesson 4
Insert Components

Upon successful completion of this lesson, you will be able to:

- Insert components into an assembly.
- Create a component and insert it into an assembly.
- Associate components to existing parts.
- Replace a component part with another.

Insert Components

Any component that exists within a project and has a manufacturers part applied can be inserted into an assembly drawing.

Components can be created in scheme drawings by inserting a symbol and applying a part, by applying parts to locations in the project, or by creating a component manually in either SOLIDWORKS Schematic or 3D.

Individual or multiple components can be inserted into an assembly, in the case of terminal strips, that are composed of multiple components, multiple insertion is used by default.

Components can be added to assemblies in a number of ways as follows:

- **Insert**
  This option will look for a 3D part associated to a manufacturers part, if one does not exist a default part related to the classification will be used.
- **Insert from file**
  Allow you to browse and select a SOLIDWORKS part from a local or network location.
- **Associate**
  Will link a component to a SOLIDWORKS part that is already inserted in an assembly and has no other component association link.

Stages in the Process

The major stages in the process are listed below:

- **Insert a component**
  Insert a component with an associate manufacturers part association to a 3D part.
- **Insert component from a file**
  Browse to locate and insert a part.
- **Associate a component to a part**
  Associate a component to an existing part in the assembly.
- **Replace a component**
  Replace a linked 3D part with another SOLIDWORKS part.
- **Insert terminals**
  Insert all the terminals associated to a terminal strip.

Using an Archive File

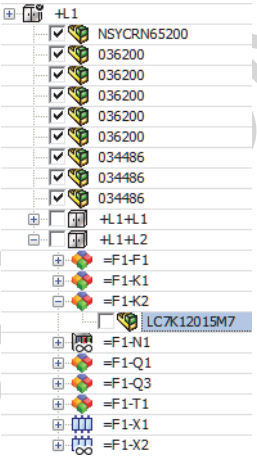
To start the lesson unarchive and open the file `Start_Lesson04.proj` from the folder `Lesson04\Case Study`.

Procedure

Utilize different tools to insert and create associations between electrical components and SOLIDWORKS 3D parts in an assembly.

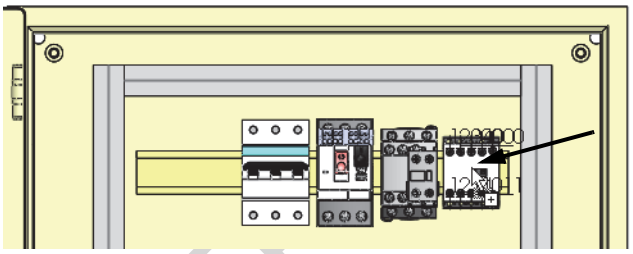
1 **Open assembly.**
Right-click drawing `107 - Main electrical closet` and click **Open**.

2 **Insert a part from a file.**
Expand location **L2 - Chassis** and component **=F1-K2** then right-click on part number **LC7K12015M7** and click **Insert from file**.



Browse to the `Lesson04\Case Study` folder, highlight `LC7K12015M7.SLDPRT` and click **Open**.

3 **Place component.**
Click to place the component on the rail as shown. The component retains a single degree of freedom that allows it to slide freely along the rail.



4 **Create component.**
Right-click on location **L2 - Chassis** and click **Add, Component manufacturer part**.

Define the following filter criteria:

- **In the class** = `cleared`
- **Manufacturer** = `Legrand`
- **Type** = `Base`
- **Reference** = `006468`

Click **Search** highlighting part **006468** and click **Add** then click **Select**.

Note

If the part is not present an archive can be found in the `Lesson04\Case Study` folder, and can be unarchived from **Tools, SOLIDWORKS Electrical, Manufacturer parts manager, Unarchive**.

Click **OK** leaving the number of components as **1** to create the component and close the command.

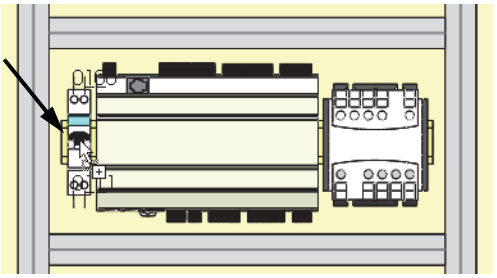
Question

Does this component appear anywhere else?

5 Insert created component.

Expand component **=F1-Q2** and right-click on part **006468** then click **Insert** .

Click to place the component on the rail as shown.



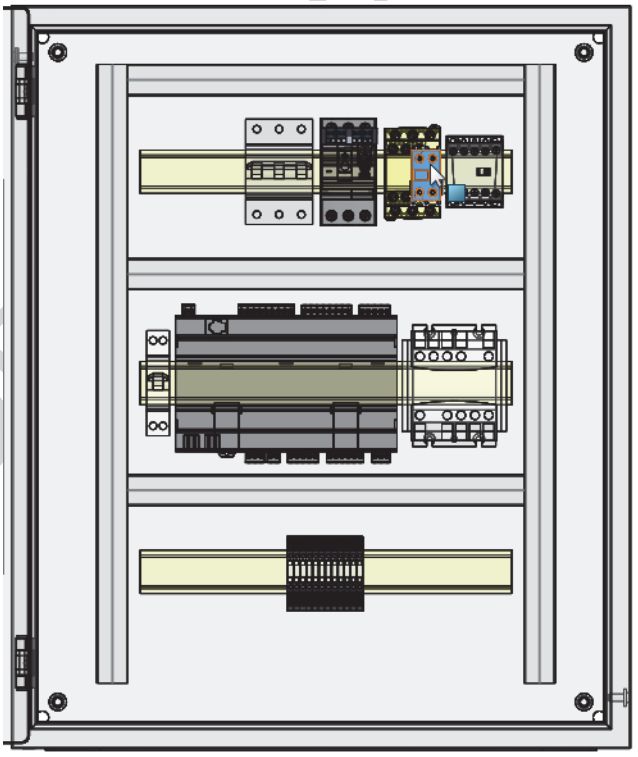
6 Associate component part.

Expand component **=F1-K1** and right-click on part **LADN11TQ** then click **Associate** .

Question

Why is everything transparent?

Select the clip on contact as illustrated and click **OK** to create an association.



Align Components

Components inserted on a DIN rail can be aligned or spaced based on requirements.

When employed to space components faces applied to the components are used as the setting out points for the defined offset. Two components with a spacing of 20mm will analyze the left, right, top or bottom face and offset the next component to the opposing face by 20mm.

Component parts that do not have faces applied will be excluded from the spacing. Using the **List invalid components** option will identify parts that require amendment prior to finalizing the process.

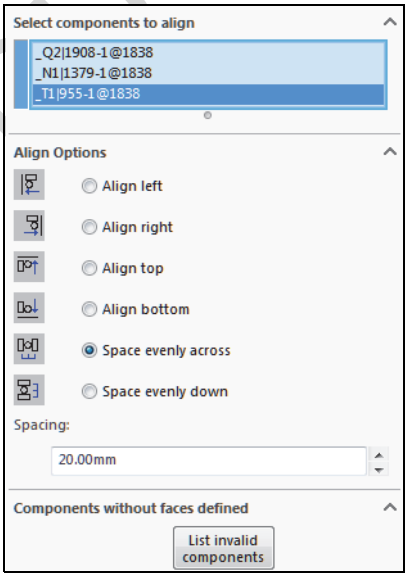
Where to Find It

- CommandMenu: **SOLIDWORKS Electrical 3D > Align components**

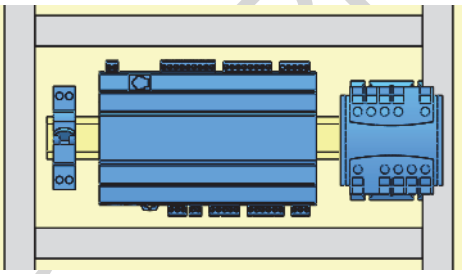
---

7 Space components.

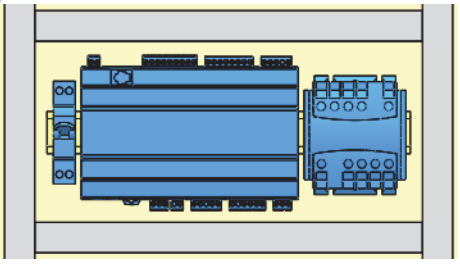
Click **Align components** changing the settings as illustrated and select the three parts on the middle rail.



Click **List invalid components** to ensure the selected parts have faces then click **OK**.



With the command still active change the spacing to **5mm** and click **OK**.

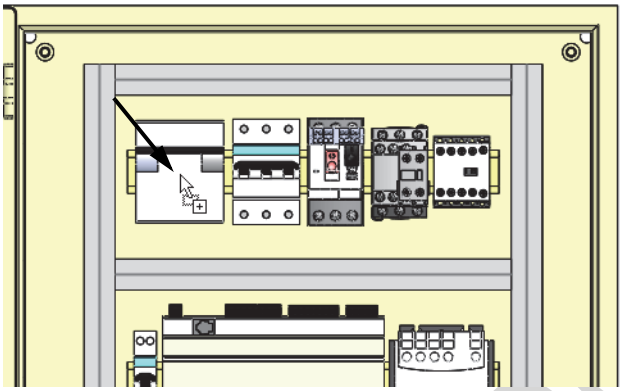


Click **Cancel** to end the command.

8 Insert a component.

Expand component **=F1-Q1** and right-click on part **006557** then click **Insert** .

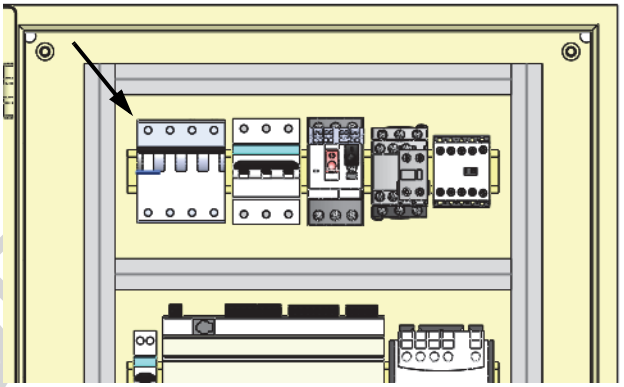
Click to place the component on the rail as shown.



9 Replace 3D part.

Right-click on previously inserted part number **006557** and click **Replace** .

Browse to the `Lesson04\Case Study` folder, highlight `EW_C_BREAKER_4P_35.SLDPRT` and click **Open**.



Inserting Terminals

Terminals are electrical routing components that contain mate references, CPoints and other routing attributes. They differ only in how they are placed in stacks using a set spacing.

Tip

Multi components can also be selected for insertion, and will prompt for automatic spacing after the first component position is indicated. The program employs the Faces, (Left, Right, Top, Bottom), in order to space the components.

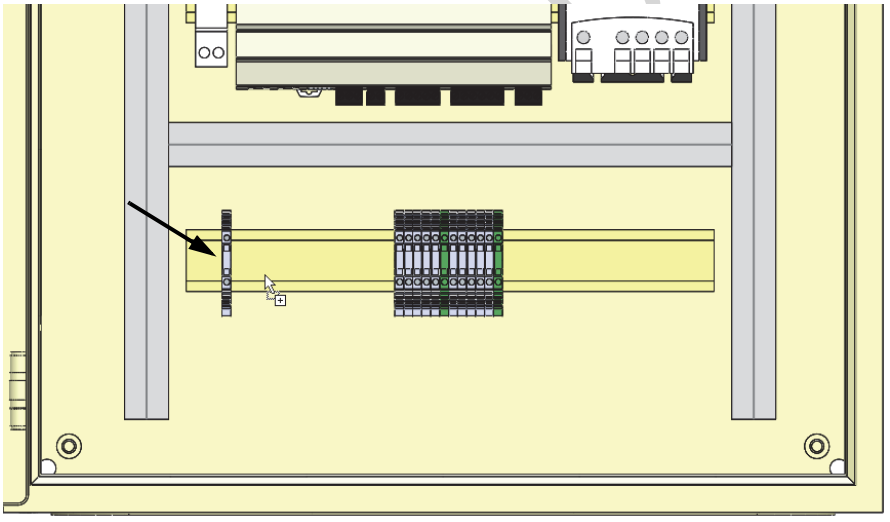
Where to Find It

- Shortcut Menu: Right-click a terminal and click **Insert terminals**

1 Insert terminals.

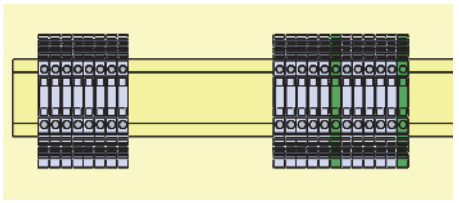
Right-click terminal strip **=F1-X2** and click **Insert terminals**.

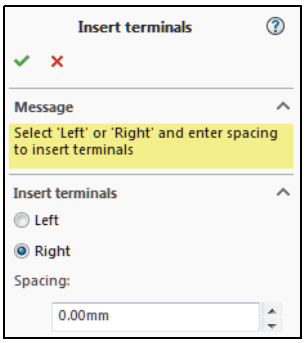
Click to place the terminal on the lower rail as shown.



2 Position setting.

Select **Right** and set the **Spacing** to 0. Click **OK** to create a total of 12 terminals as shown.



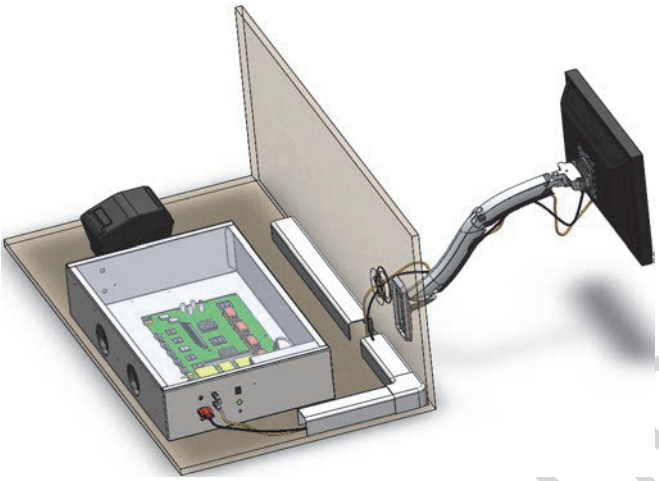


3 Close the project.

On the **Electrical Project Document** panel right-click the project name and click **Close project** selecting to **Save** when prompted.

Exercise 4: Insert Components

Insert and associate components in an assembly.



This lab uses the following skills:

- *Insert a component.* on page 78
- *Associate component part.* on page 76

Using an Archive File

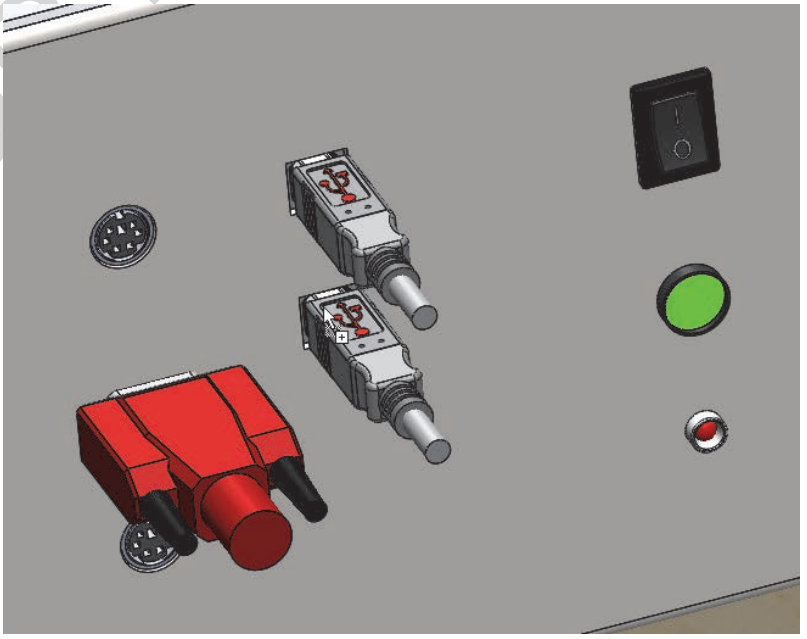
To start the exercise unarchive and open the file `Start_Exercise04.proj` from the folder `Lesson04\Exercises`.

Procedure

Insert a component and associate a component to an existing SOLIDWORKS part.

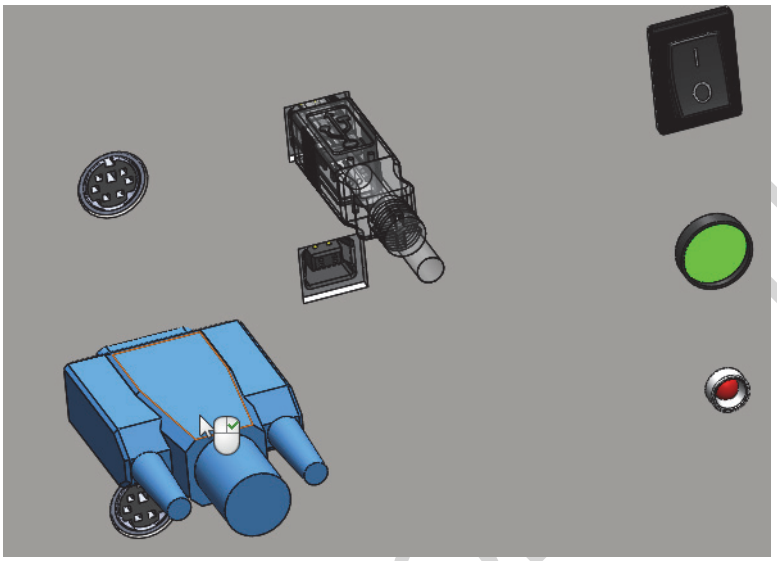
1 Open assembly.
Open drawing `04 - Monitor and PC Assembly`.

2 Insert component.
Insert connector `=F1-X2` at the USB connection as illustrated.



3 Associate part.

Associate component =F1-X5 part CON45612 to the VGA connector as illustrated.



4 Close and Save.

Close the SOLIDWORKS assembly selecting to **Save all**.

5 Close the project.

On the **Electrical Project Document** panel right-click the project name and click **Close project** .

Lesson 5
Routing Wires

Upon successful completion of this lesson, you will be able to:

- Create a routing path sketch.
- Generate routes using route wires.

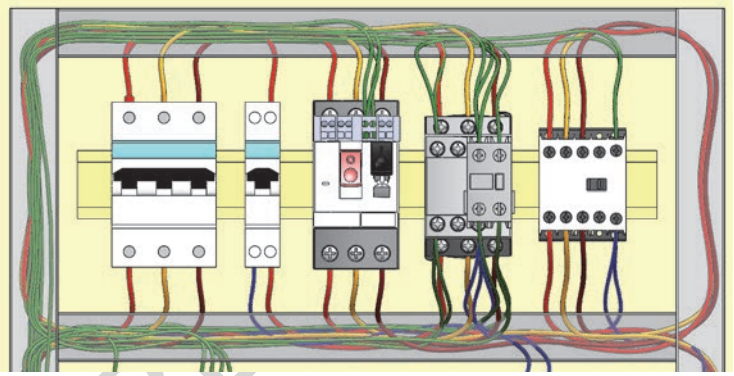
Routing Wires

Wires can be automatically routed between parts in an assembly where specific criteria are met as follows:

- 3D parts must be related to SOLIDWORKS Electrical components.
- The component must have detailed wiring connections in SOLIDWORKS Electrical Schematic.
- The 3D part must have CPoints with a naming convention that matches the components circuits and terminals.
- Sketch paths with a specific naming convention must be used.
- The routing parameters specified must allow the program to locate paths and component connection points.

If anyone of these requirements not be fulfilled wires may not produced the expected results.

During the course of this lesson wires will be routed in different ways to highlight potential issues and illustrate how they can be resolved.



Stages in the Process

The major stages in the process are listed below:

- **Highlight on the importance of Paths**
  Routing wires where paths have not been defined, illustrates their importance for achieving relevant results
- **Routing Path**
  Routing paths are sketches used to shape a group of routed wires.
- **Route Wires**
  The route wires options previews or routes a group of wires.

Using an Archive File

To start the lesson unarchive and open the file `Start_Lesson05.proj` from the folder `Lesson05\Case Study`.

Procedure

Create 3D sketch geometry and use it to guide previews and a route.

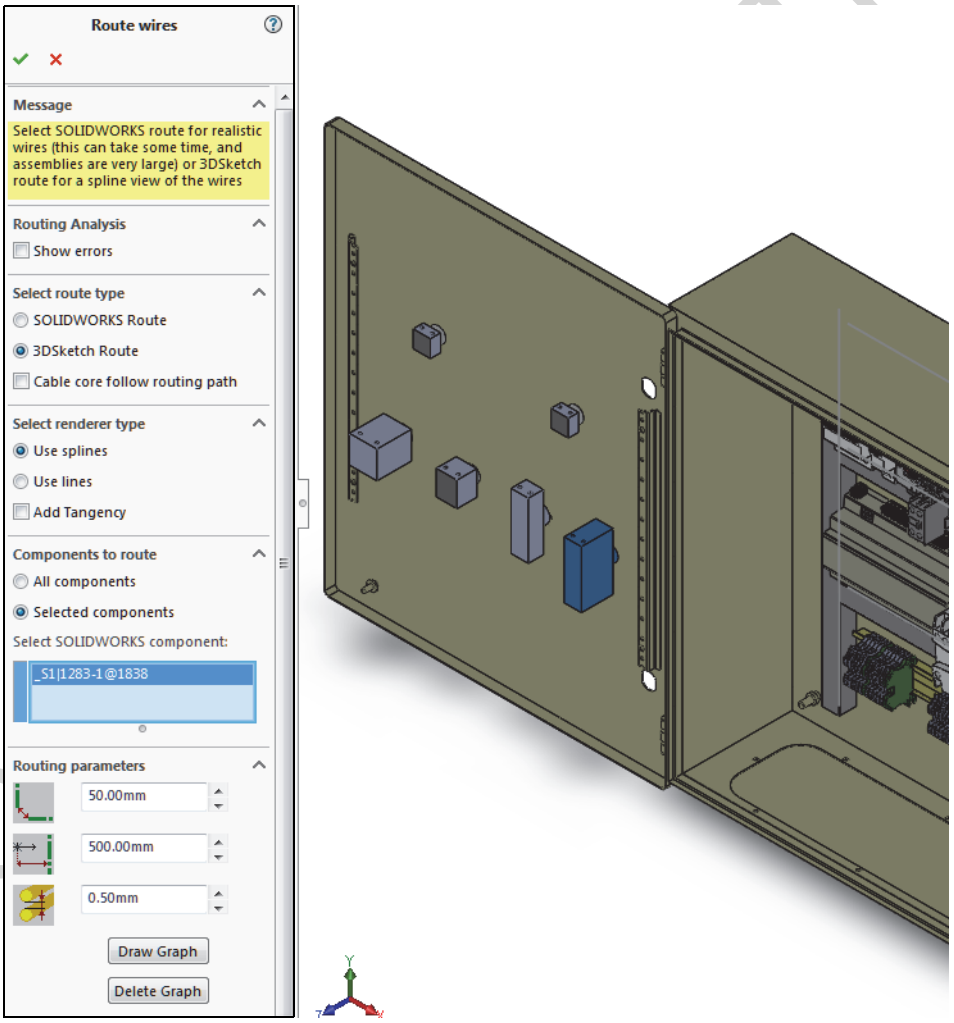
1 Open assembly.

Right-click drawing `107 - Main electrical closet` and click **Open**.

2 Route wires.

Click **Isometric orientation** to better view the cabinet.

On the **SOLIDWORKS Electrical 3D** tab click **Route Wires** command and define the settings as illustrated.



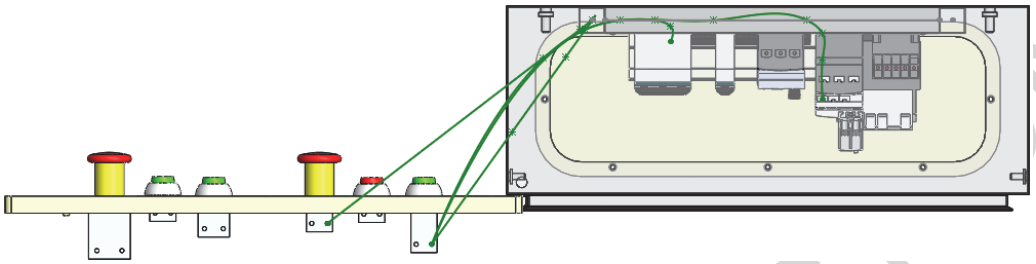
Select the pushbutton as illustrated and click **OK** to route the wires.

Tip

Utilizing **Selected components** is a quick wiring check as only specified components will be routed rather than the entire machine or installation.

3 Results without Paths.

Click **Top orientation** then select the top of the cabinet and click **Change transparency** .



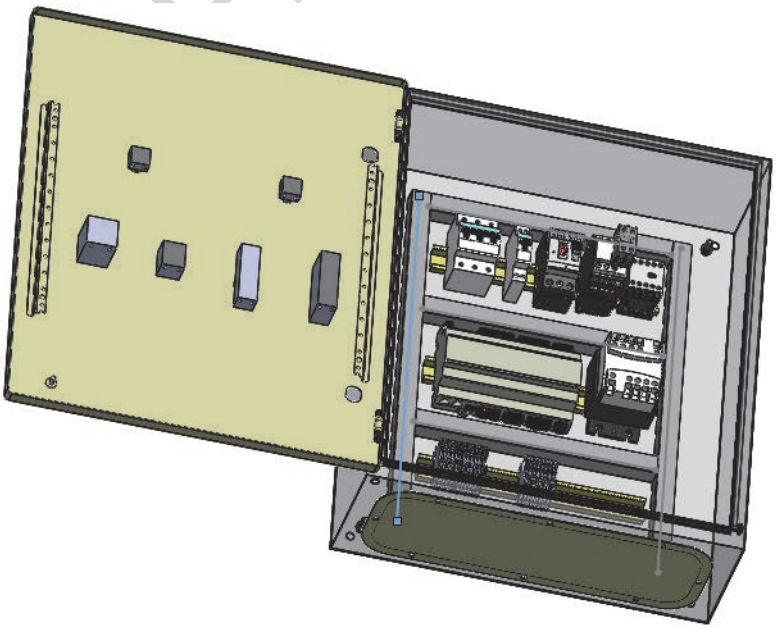
The wires have routed to the nearest sketch path contained within the ducts, this results in wires passing through the door and side of the cabinet. In order to achieve the correct routing results more sketch paths are required.

4 Manually delete wire assembly.

Click **SOLIDWORKS Feature Manager Design Tree** scroll to the bottom of the list. Right-click `EWS [~ 24V _ Control]20` and click **Delete** to remove the sketched wires.

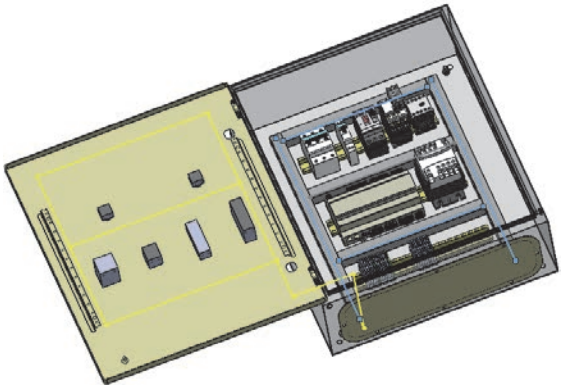
5 Sketches.

Ensure that **View sketches** is active in order to see the `EW_PATH` sketch in the existing `EW_DUCT` components.



Routing Path

3D sketch geometry is used as a routing path to guide the routing of wires (see *Route Wires* on page 89).



Sketch Name

The sketch name must include `EW_PATH` in it in order for it to be recognized as a path when routing. Names like `EW_PATH1` and `EW_PATH2` are valid.

Note

The SOLIDWORKS **3D Sketch** option can also be used and then converted using the same command.

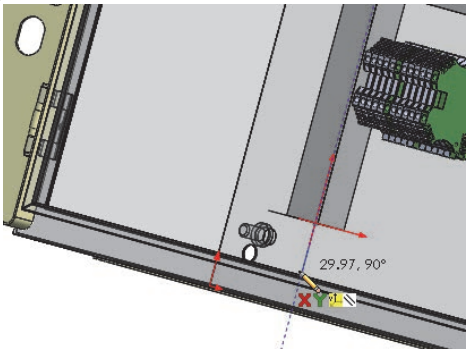
Where to Find It

- Menu: **SOLIDWORKS Electrical 3D, Create routing path**

---

6 **New 3D sketch.**
Click **Create routing path** and click **Create sketch** action then click **OK**. At the message: `A new 3D sketch has been created; you can use standard SOLIDWORKS commands to create your routing path. Only lines and sketch points are used by routing algorithm.` Click **OK**.

7 **Line.**
The sketch tab is automatically activated, a **Sketch**, (`EW_PATH1`), is created and opened. Click **Line** command to create a line connected to the end point of the sketch shown.



Tip

Ducts that are used to hold wire bundles can be created with an EW_PATH sketch already included.

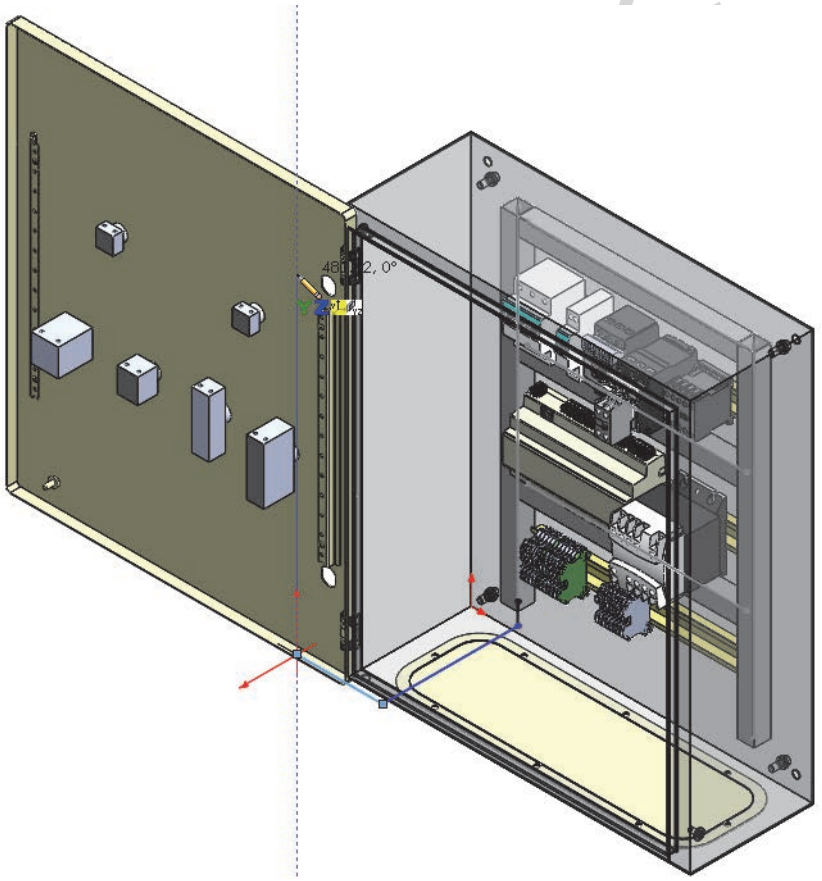
8 Isometric.

Using different orientation views, sketch the following lines, these run from the base of the duct, to just before the base of the cabinet, up the inside of the cabinet to slightly above door level and finally up the side of the door beside the lights and pushbuttons as shown.

Tip

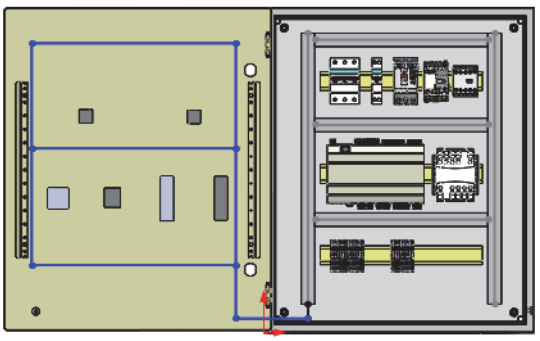
The placement of the sketch path line, must always be considered in relation to where the user wants wires to run when interconnecting components. If a sketch path runs through the side of a machine the wires will also run through the sides.

The following image shows three drawn sketch line positions, the fourth is about to be defined running up the inside of the door.



9 Front.

Click **Front** orientation view and complete the line sketching as illustrated.

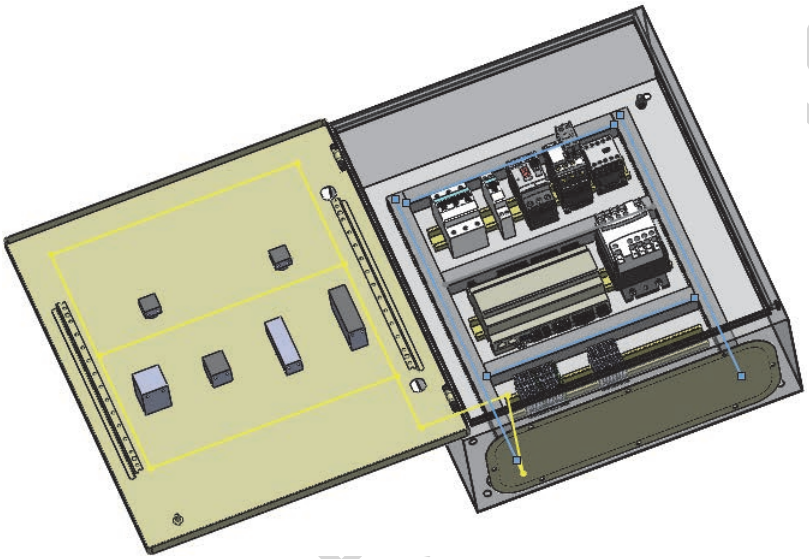


10 Exit Hide Sketch.

Click **Exit** to leave the sketch, by default the newly created path will be displayed in yellow.

Tip

Selecting **View sketches** to hide the sketch lines in the assembly; this will have no impact when routing wires.



Route Wires

There are numerous options available when routing wires that produce different preview results. All **Route Wires** options create SOLIDWORKS route geometry based on the shortest possible path to interconnect components, this is based on the detailed wiring defined in the schematic project drawings.

3DSketch Route

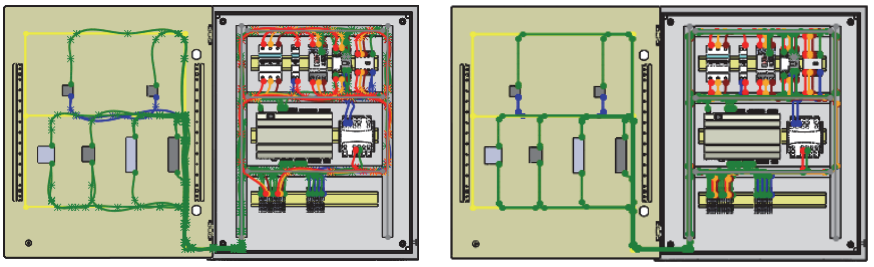
The **3DSketch Route** option is used to create a quick preview of the route using sketch geometry.

Note

The **3DSketch Route** option does not create actual geometry. See *SOLIDWORKS Route* on page 92 for the creation of geometry.

Renderer Type

Splines (left) and **Lines** (right) are the two main renderer types used to shape the route.



For information about the SOLIDWORKS Route option, see *SOLIDWORKS Route* on page 92.

Routing Parameters

The **Routing parameters** are available regardless of the type of routing being carried out, they allow the user to define distances the program will analyze to locate sketch paths and 0_0 CPoints.

Reducing routing parameters can mean a connection point cannot locate certain `EW_PATH` sketches and less paths will need to be analyzed by the program to locate the optimized route.

Lowering the parameters too much, so that CPoints and sketch paths are not found, will negatively affect routing.

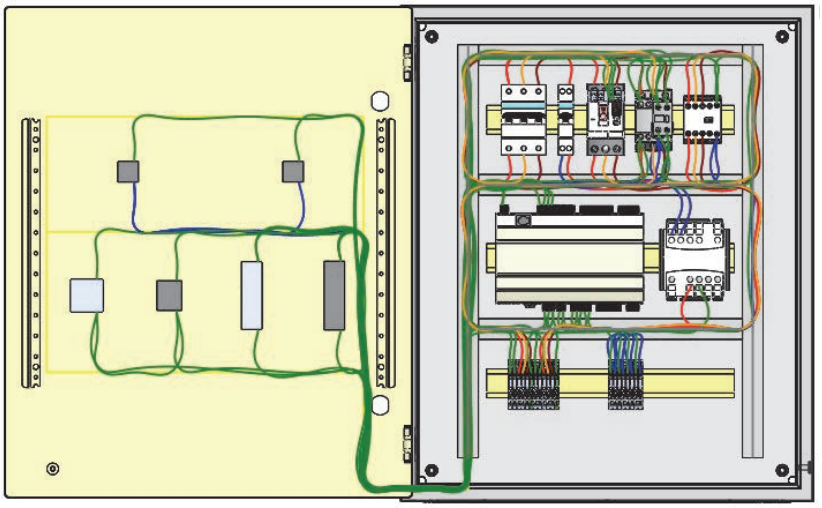
Where to Find It

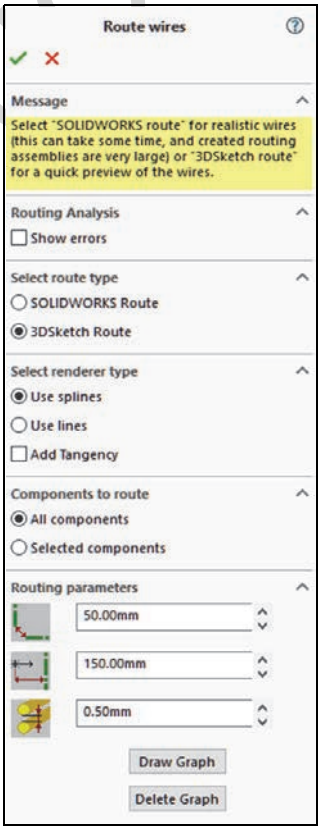
- CommandManager: **SOLIDWORKS Electrical 3D > Route wires**

---

11 Route wires using splines.

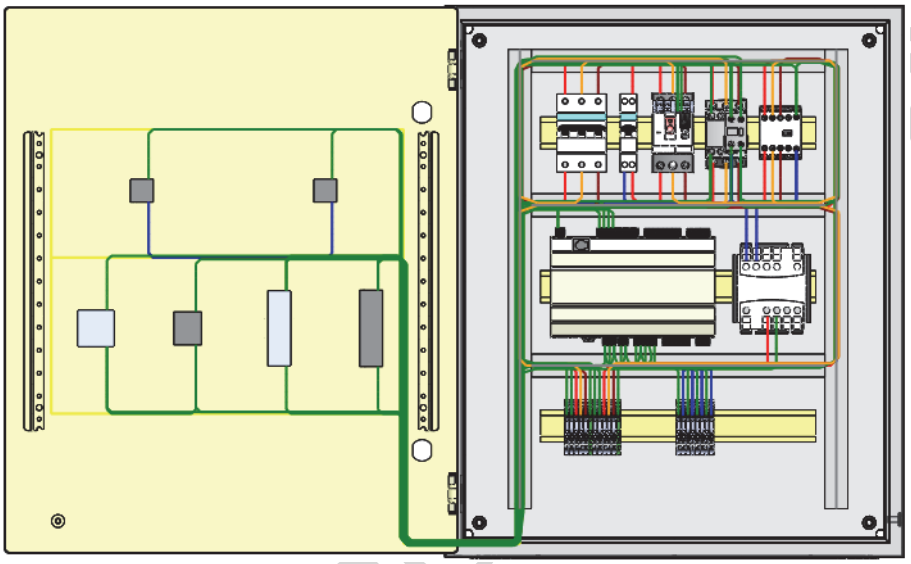
Click **Route wires** and set the options as illustrated then click **OK** to route the wires.





12 Route wires using lines.

Click **Route wires** and click **Use lines** to change the render type and click **OK**. At the message: 3DSketch already exists, click **Delete existing route**.



Wire Sketches

The **Wire Sketches** are created by the **3DSketch Route** option.

They show individual wires in separate sketches, (a single sketch for each wire style), using different colors, (the colors used are taken from the wire style defined in SOLIDWORKS Electrical: Schematics). The sketches can be hidden or shown to see specific wire styles.

A set of 3D sketches is created by the command. They are listed at the end of the FeatureManager design tree. In this example they are:

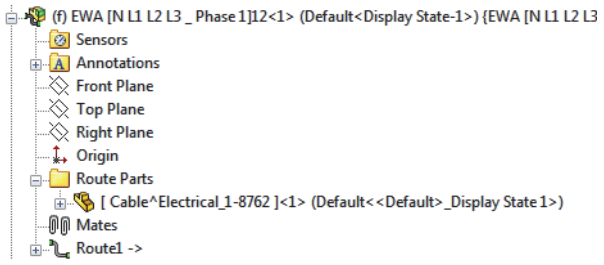
- EWS [N L1 L2 L3 _ Phase 1]1
- EWS [N L1 L2 L3 _ Phase 2]2
- EWS [N L1 L2 L3 _ Phase 3]3
- EWS [N L1 L2 L3 _ Neutral wire]4
- EWS [N L1 L2 L3 _ Protection]5
- EWS [~ 24V _ Control]6

Note

These are not completed routes but previews using route center lines. The same names will be used for the routing sub-assemblies when they are created with **SOLIDWORKS Route**.

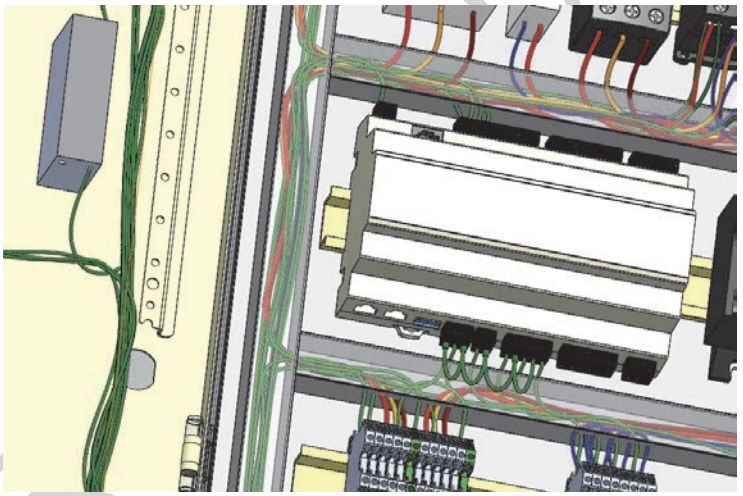
SOLIDWORKS Route

The **SOLIDWORKS Route** option is used to create the finished, (photo realistic), route including the routing sub-assemblies and physical wire parts. The sketches (see *Wire Sketches* on page 91) will map to a route sub-assembly that includes a cable part.



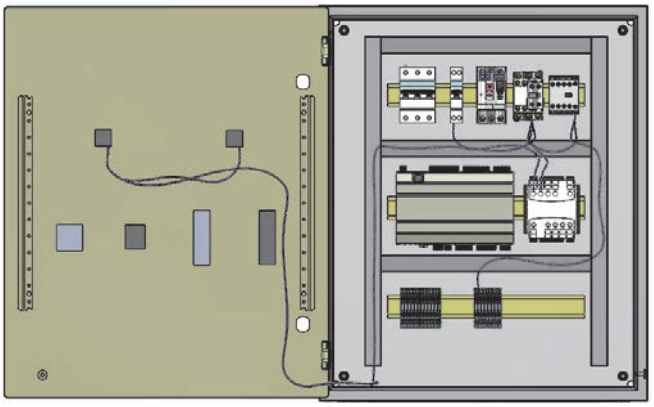
13 SOLIDWORKS route.

Click **Route wires** and the options **SOLIDWORKS Route**, **Use Splines** and **All components**. Click **OK**.



Note

Sketches and parts have been hidden in this image. To see only single routes or groups of routes, use **Display States** within a **Configuration**.



Wire Segregation

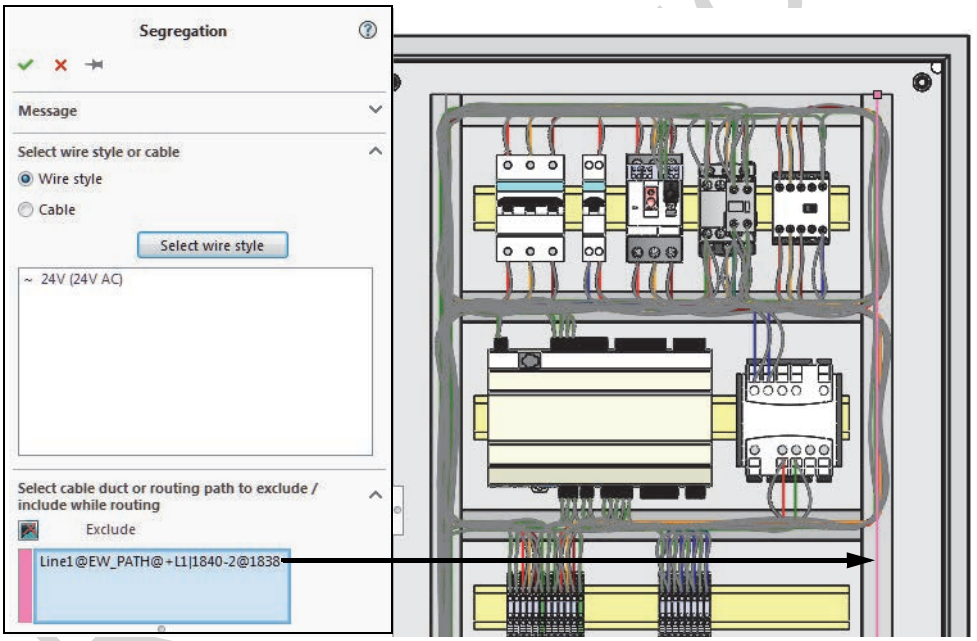
It is possible to segregate wires so that specific wire styles will not route along selected sketch paths. This provides a quick way to reduce noise in a machine, or to reduce the packing density in duct.

Where to Find It

- CommandManager: **SOLIDWORKS Electrical 3D > Segregation**

14 Segregate wire style.

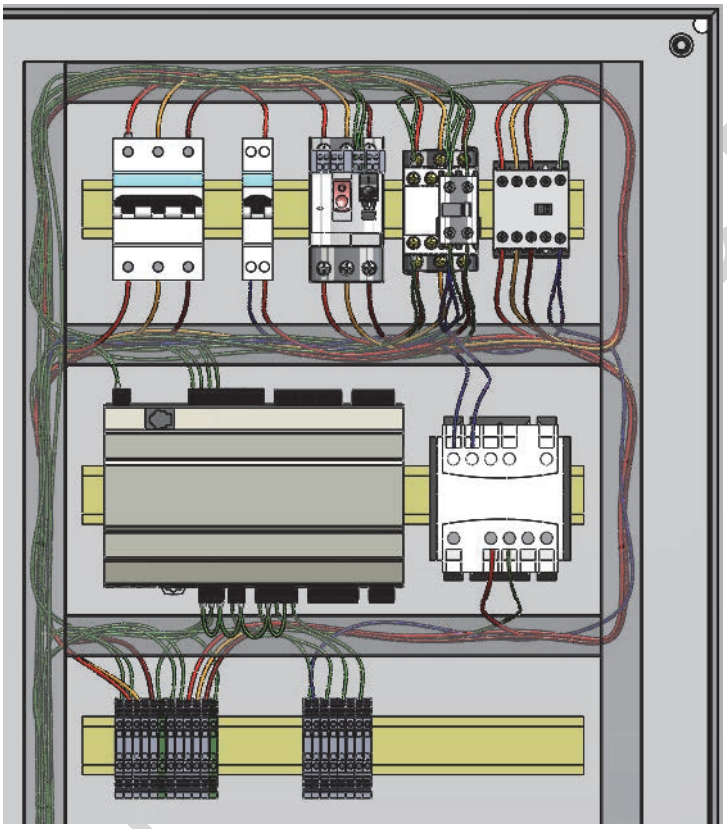
Click **Segregation** command ensure Wire style is active and click **Select wire style**. In the dialog expand `Group : 0 - Electrical` and click **~24V AC** to check it on and click **Select**. Click **Exclude** and select the duct on the far right of the cabinet then click **OK**.



15 Route Wires.

Click **Route wires** leaving the settings as previously defined and click **OK** and click **Delete existing route** when prompted.

The green, 24V AC wires now interconnect devices using all paths with the exception of the duct on the right of the cabinet.

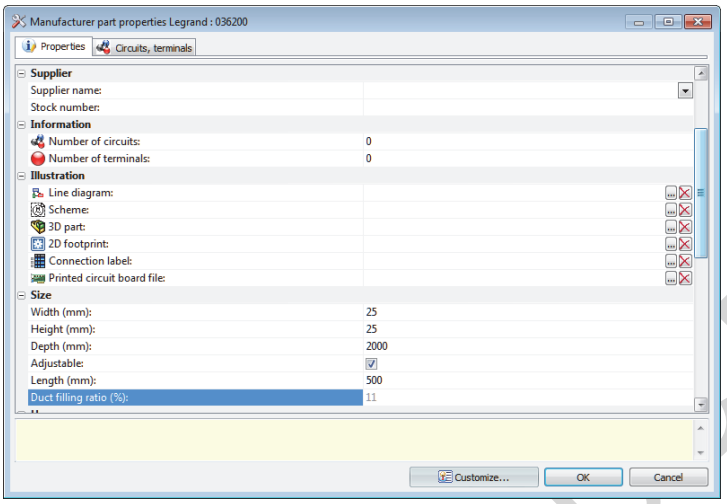


16 Duct Fill Factor.

Click **Tools, SOLIDWORKS Electrical, Calculate Cable Duct Filling Ratio** and click **Calculation of cable duct filling ratio** when prompted.

Right-click on the first listed duct reference **036200** on the Electrical Manager and click **Properties** .

Scroll down the Parts properties to review the **Duct filling ratio (%)**.

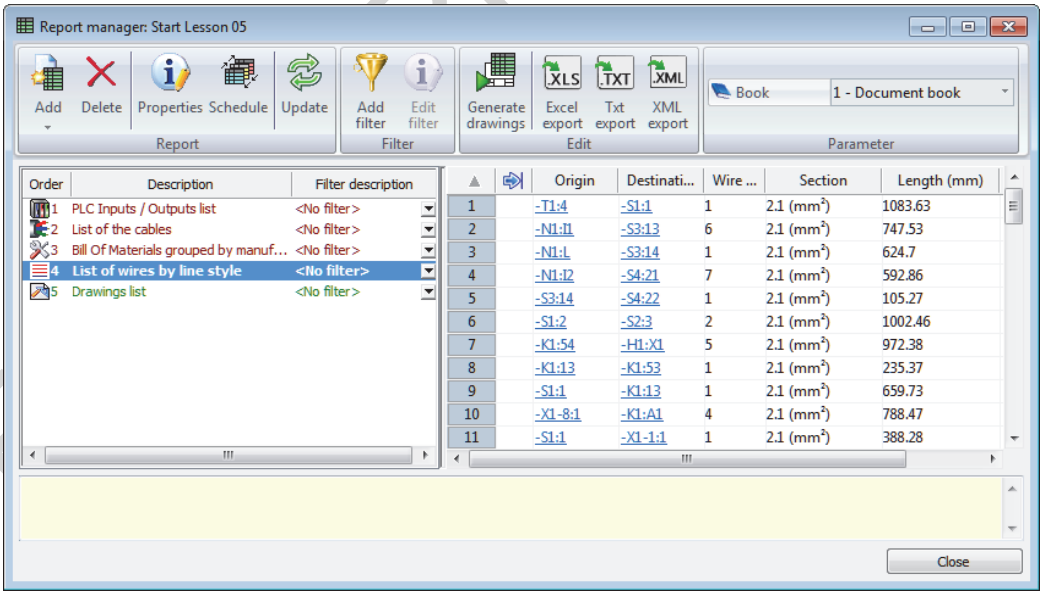


Click **OK**.

17 Report lengths.

Click **Tools**, **SOLIDWORKS Electrical, Project, Reports** and click the **List of wires by line style** tab.

The wire lengths are listed by wire style.



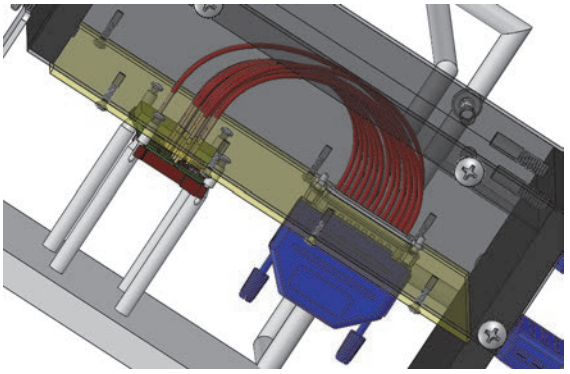
Note Lengths will vary due to the position of the EW_PATH created.

18 Close the project.

On the **Electrical Project Document** panel right-click the project name and click **Close project** selecting to **Save** when prompted.

Exercise 5: Routing Wires

Route wires using the information provided.



This lab uses the following skills:

- *Routing Path* on page 87
- *Route Wires* on page 89

Using an Archive File

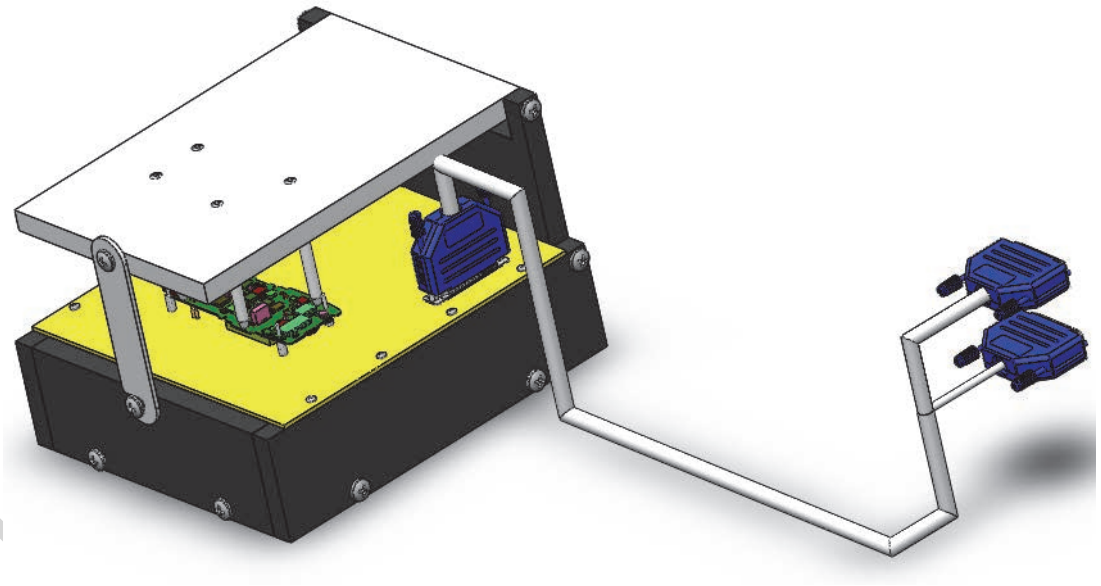
To start the exercise unarchive and open the file `Start_Exercise05.proj` from the folder `Lesson05\Exercises`.

Procedure

Route the wires as shown.

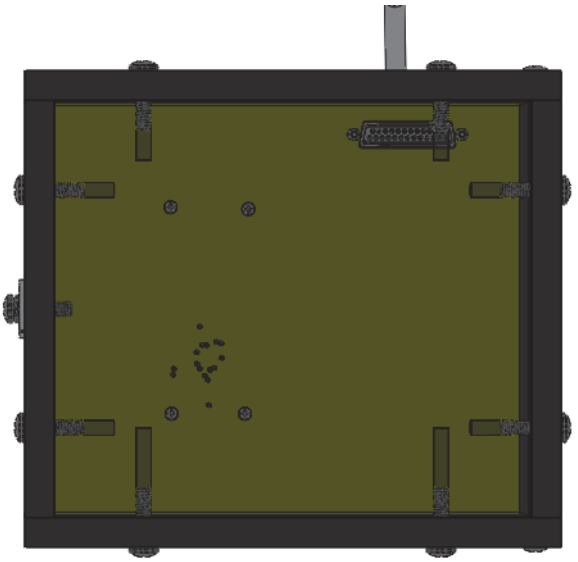
1 Open assembly.

Open drawing `04 - Route Wires`.



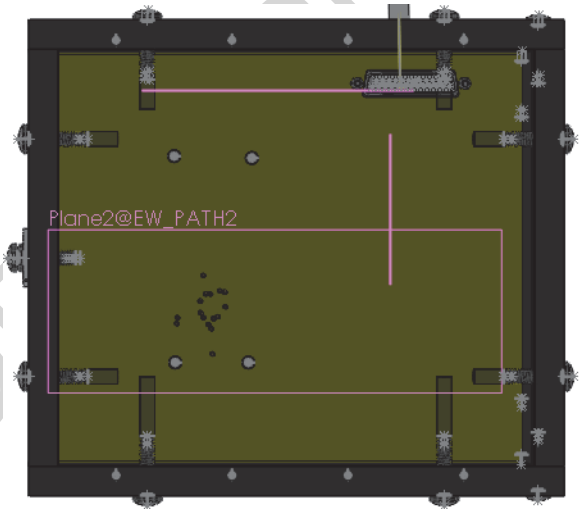
2 Orientation.

Change the view orientation to Bottom and change the transparency of the base as illustrated.



3 Routing path.

Unsuppress EW_PATH2 and set visibility to enable sketches.

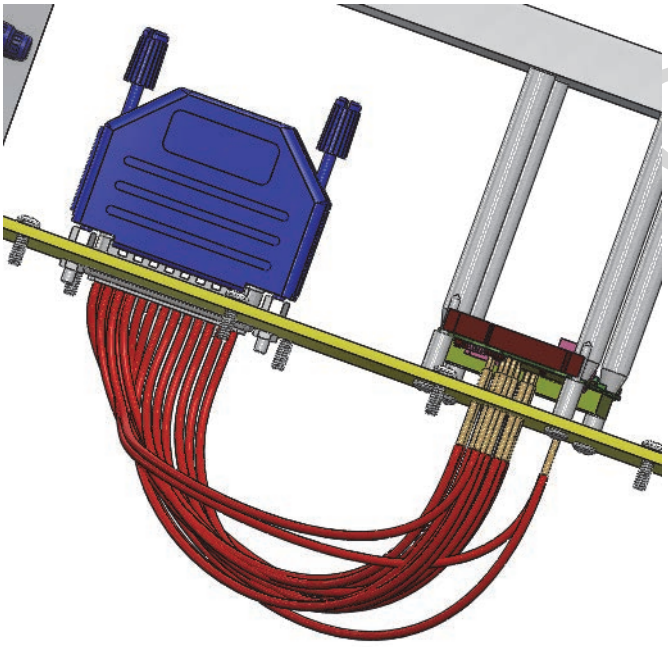


4 Route wires.

Route wires applying the following settings:

- SOLIDWORKS Route
- Use Splines
- Add Tangency checked
- All components
- Routing parameters - 5.00in - 5.00in - 0.02in

Turn off sketches and rotate the assembly to review the results.



5 Close the project.

On the **Electrical Project Document** panel right-click the project name and click **Close project** selecting to **Save** when prompted.