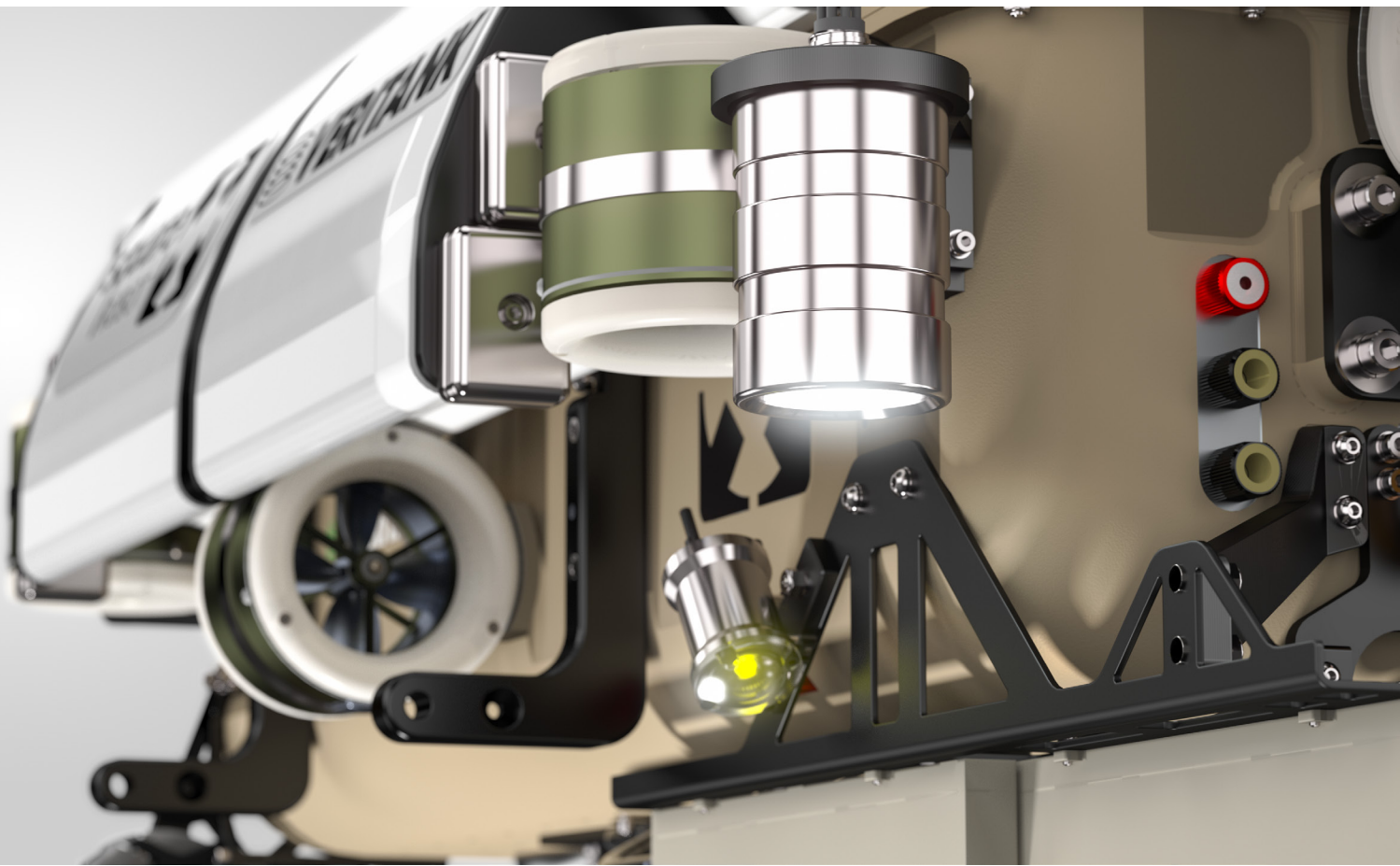


FUNDAMENTALS OF SOLIDWORKS ELECTRICAL

SOLIDWORKS EDUCATION EDITION 2025



ENG

This is a preview of the Fundamentals of 3D Design and Simulation.

Customers on active subscription have access to the full content located on the Customer Portal, under Downloads. If you are not on active subscription and would like to get access to this content, please contact your local reseller at: www.solidworks.com/edureseller.

SOLIDWORKS® Education Edition 2025

Fundamentals of SOLIDWORKS Electrical

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

© 1995-2025, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

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

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

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

For a full list of the patents, trademarks, and third-party software contained in this release, please go to the Legal Notices in the SOLIDWORKS documentation.

Restricted Rights

This clause applies to all acquisitions of Dassault Systèmes Offerings by or for the United States federal government, or by any prime contractor or subcontractor (at any tier) under any contract, grant, cooperative agreement or other activity with the federal government. The software, documentation and any other technical data provided hereunder is commercial in nature and developed solely at private expense. The Software is delivered as "Commercial Computer Software" as defined in DFARS 252.227-7014 (June 1995) or as a "Commercial Item" as defined in FAR 2.101(a) and as such is provided with only such rights as are provided in Dassault Systèmes standard commercial end user license agreement. Technical data is provided with limited rights only as provided in DFAR 252.227-7015 (Nov. 1995) or FAR 52.227-14 (June 1987), whichever is applicable. The terms and conditions of the Dassault Systèmes standard commercial end user license agreement shall pertain to the United States government's use and disclosure of this software, and shall supersede any conflicting contractual terms and conditions. If the DS standard commercial license fails to meet the United States government's needs or is inconsistent in any respect with United States Federal law, the United States government agrees to return this software, unused, to DS. The following additional statement applies only to acquisitions governed by DFARS Subpart 227.4 (October 1988): "Restricted Rights - use, duplication and disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252-227-7013 (Oct. 1988)."

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

Contents

Introduction

To the Teacher	2
SOLIDWORKS Electrical Tutorials	2
Accessing the SOLIDWORKS Electrical Tutorials	2
My SOLIDWORKS	3
Certification Exams	3
Training Files	3
Educator Resources link	3
Prerequisites	4
Course Design Philosophy	4
Conventions Used in this Book	4
Windows	4
Use of Color	5
Graphics and Graphics Cards	5
Color Schemes	5

Lesson 1: Drawing Types

What are Drawing Types?	8
Drawings	8
Scheme	8
Creating Drawings	9
Stages in the Process	9
Existing and Archived Projects	10
Opening an Existing Project	10
Unarchiving a Project	10
Closing Projects	12
Line Diagram Symbols	12
Adding Symbols	12
Symbols Library	12
Symbol Orientation	15
Adding Cables	18
Schematic Drawing	20
Scheme Best Practices	20
Stages in the Process	21
Symbols Panel	23
Schematic Symbols	25
Symbol Properties	27
Types of Properties	27
Exercise 1: Drawing Types	32

Lesson 2: Symbols and Components

What is a component?	40
Component Identification	40
Component Symbol Identification	41
Stages in the Process	41
Deleting Components	43
Description Columns	46
Symbol Component Association	49
Exercise 2: Symbols and Components	52

Lesson 3: Manufacturers Parts

What are Manufacturers Parts?	58
Circuits and Terminals	58
Circuit Association	60
Stages in the Process	61
Finding Manufacturer Parts	63
Search Options	63
Editing Parts	68
Circuit Symbols	69
Circuit Association	71
Electrical Assemblies	73
Exercise 3: Manufacturers Parts	77

Lesson 4: Insert Components

Insert Components	82
Stages in the Process	82
Align Components	85
Inserting Terminals	87
Exercise 4: Insert Components	88

Lesson 5: Routing Wires

Routing Wires	92
Stages in the Process	92
Routing Path	95
Sketch Name	95
Route Wires	97
3DSketch Route	97
Routing Parameters	98
Wire Sketches	99
SOLIDWORKS Route	100
Wire Segregation	101
Exercise 5: Routing Wires	105

NOT FOR PRODUCTION

Introduction

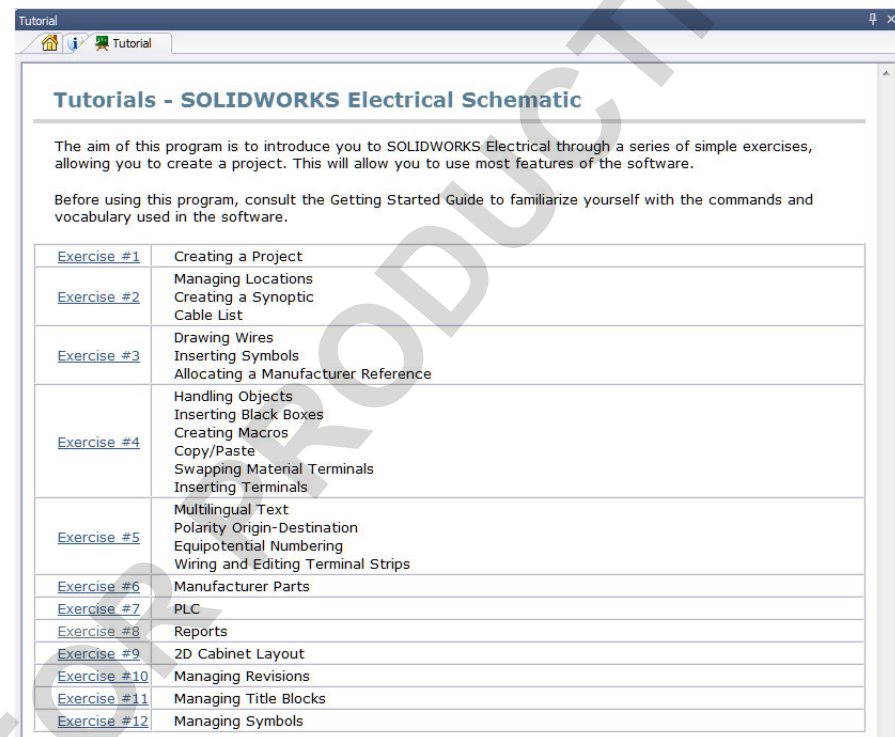
To the Teacher

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/certifications/solidworks-cad-design-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 CaseStudy 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.

Additional Educator Resources

Additional curriculum resources for educators to use directly in their classrooms can be found at:

www.solidworks.com/curriculum.

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, <code>Sketch1</code> .
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.

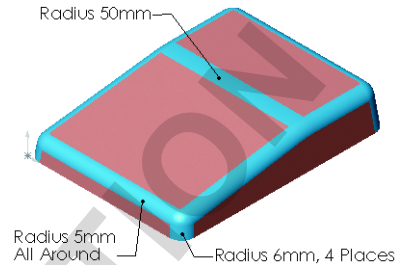
Windows

The screen shots in this manual were made using the SOLIDWORKS software running mostly on 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.

NOT FOR PRODUCTION

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

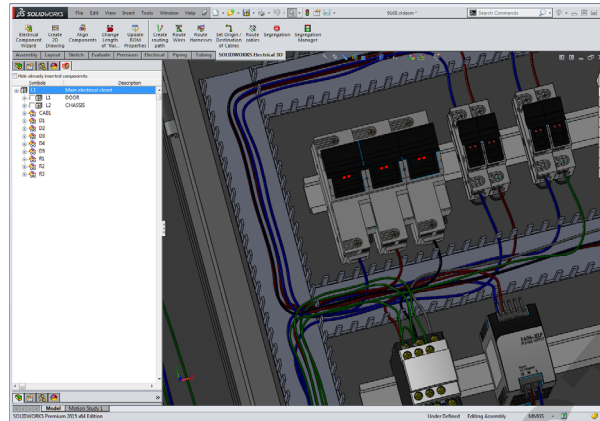
Terminals drawings are automatically created from data applied to terminals in the project drawing set and through the **Terminals editor**.

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 10.
- **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

Opening an Existing Project


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.

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

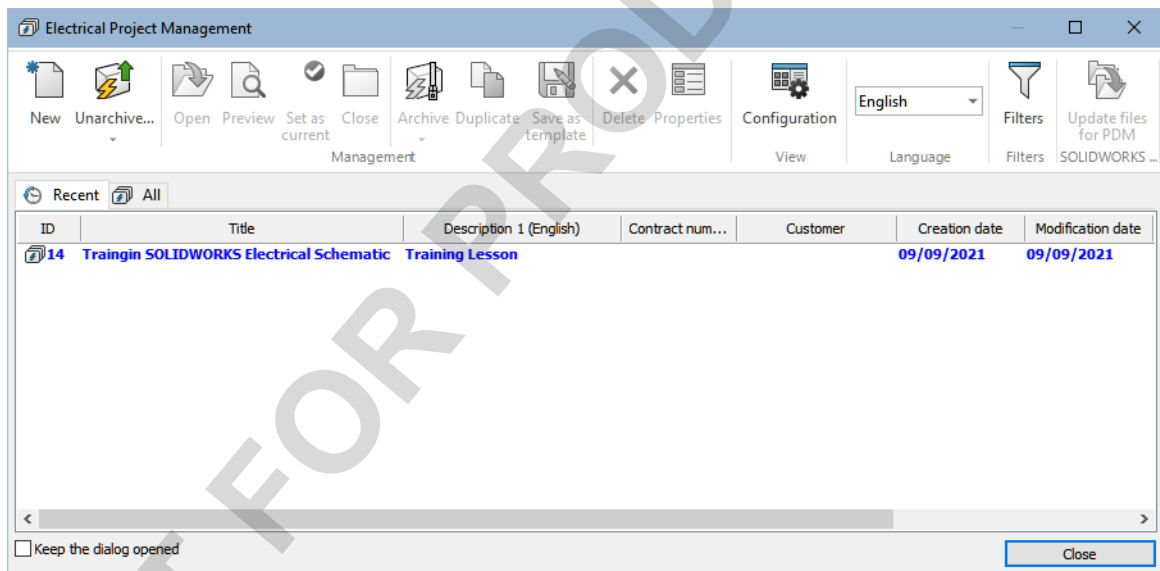
Only the projects stored in that folder will appear in the **Projects Manager** dialog.

1. Click **Electrical Project**.

Click **Electrical Project** . 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

- PropertyManager: **Electrical Project Management** > 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?

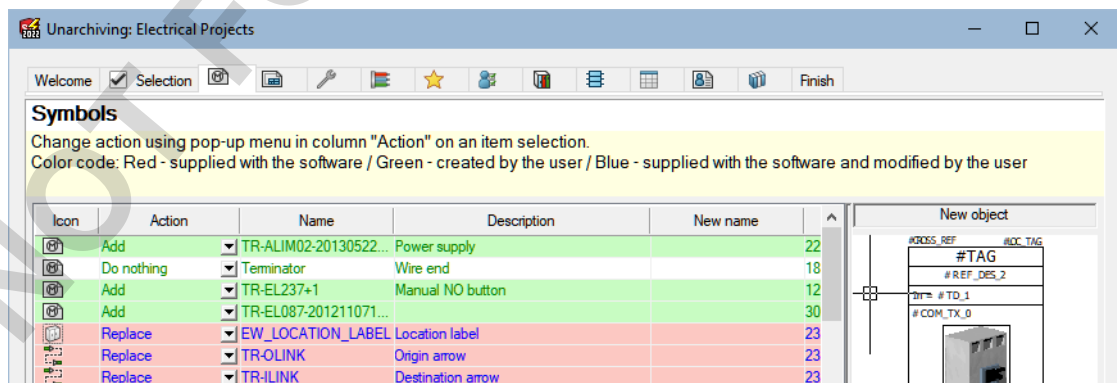
Click **Yes**.

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 **Electrical Project Management**. Select the project from the list (open projects will appear in blue text) and click **Close** .

Where to Find It

PropertyManager: **Electrical Project Management > 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: **Electrical Project > New > Drawing type**

1 Open a project.

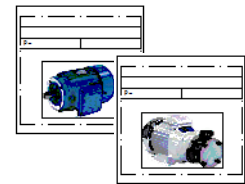
Click **Open**  project Start Lesson 03.

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.


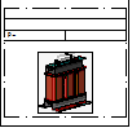
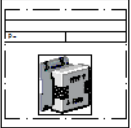


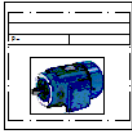
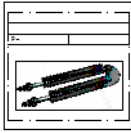





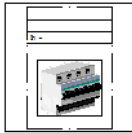
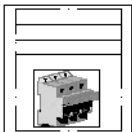
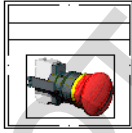
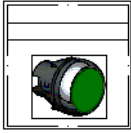
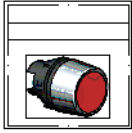
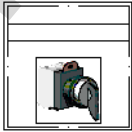

Adding Symbols

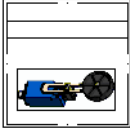

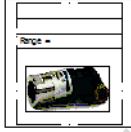
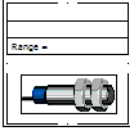

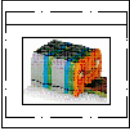
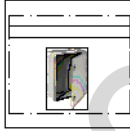


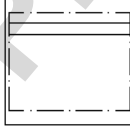
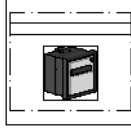

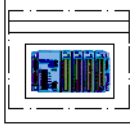

There are two main methods of adding symbols to a drawing; one uses a standard click (**Insert Symbol**) to place while the other uses the familiar drag drop method. Both methods can access the same symbol library.

Symbols Library

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




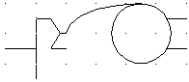

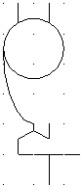



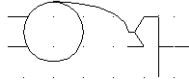

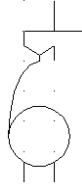
Supplies		
		
EW_SY_Battery Cells battery	EW_SY_TransformerHigh High Voltage Transformer	EW_SY_TransformerLow Low Voltage Transformer

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_CircuitBreaker Circuit-breaker	 EW_SY_CircuitBreakerMod Modular circuit-breaker
 EW_SY_FuseSwitch Fuse disconnecter		
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		
 <p>EW_SY_LimitSwitch Roller limit switch</p>	 <p>EW_SY_LimitSwitchLever Lever limit switch</p>	 <p>EW_SY_PressureSensor Pressure sensor</p>
 <p>EW_SY_ProximitySensor Proximity sensor</p>	 <p>EW_SY_TemperatureProbe Temperature probe</p>	
Miscellaneous		
 <p>EW_SY_Terminal Terminal strip</p>	 <p>EW_SY_Cabinet Electrical cabinet</p>	 <p>EW_SY_Ammeter Ammeter</p>
 <p>EW_SY_Voltmeter Voltmeter</p>	 <p>EW_SY_BlackBox Generic frame</p>	 <p>EW_SY_TimeCounter Time meter</p>
 <p>EW_SY_MotorDrive Drive</p>	 <p>EW_SY_Plc PLC</p>	 <p>EW_SY_Screen Screen</p>

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 X or Y 		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** 

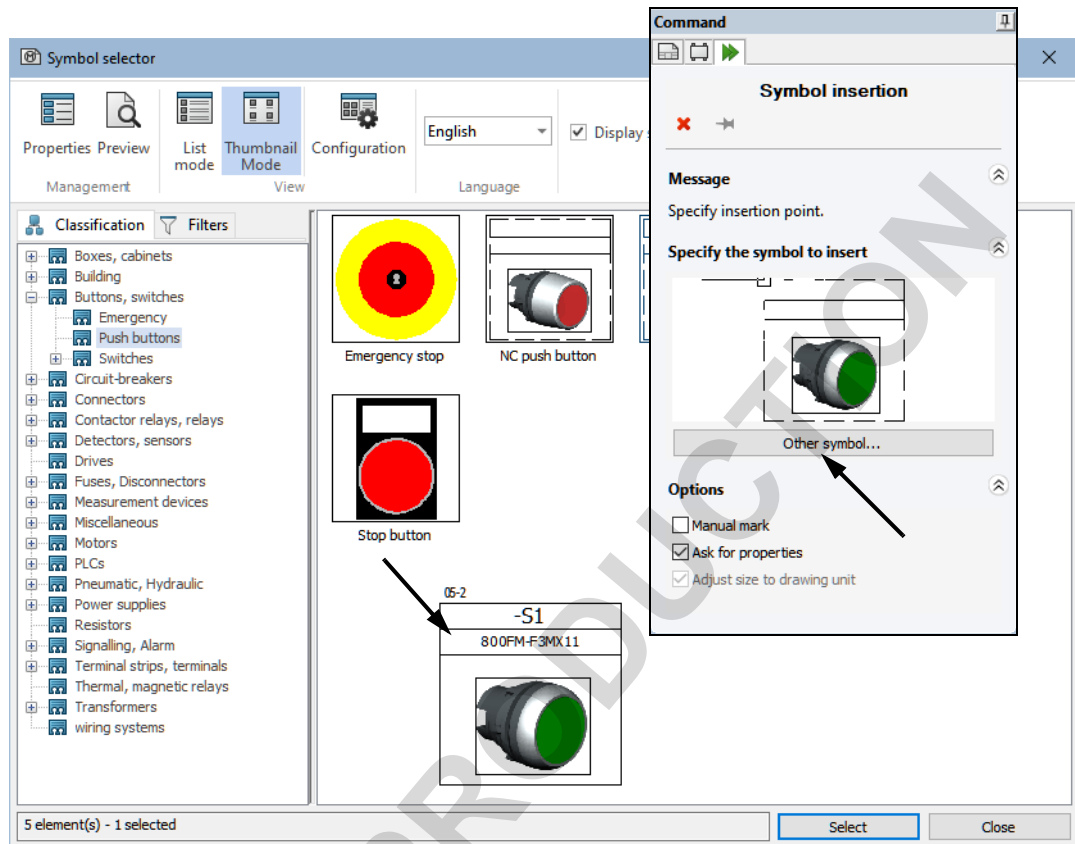
3 **Select line diagram symbol.**

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**.

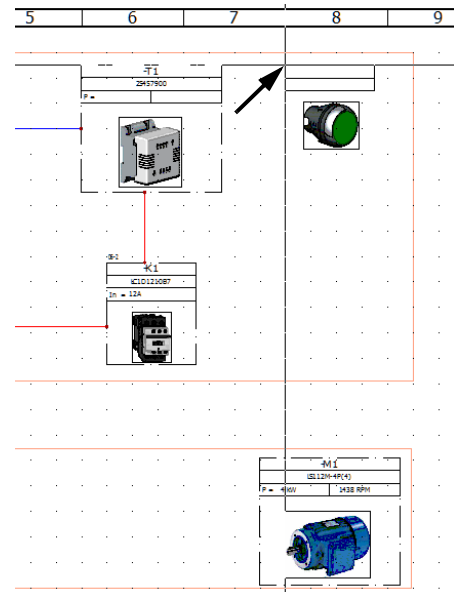


In the **Symbols selector** click the **Buttons, switches** classification.

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

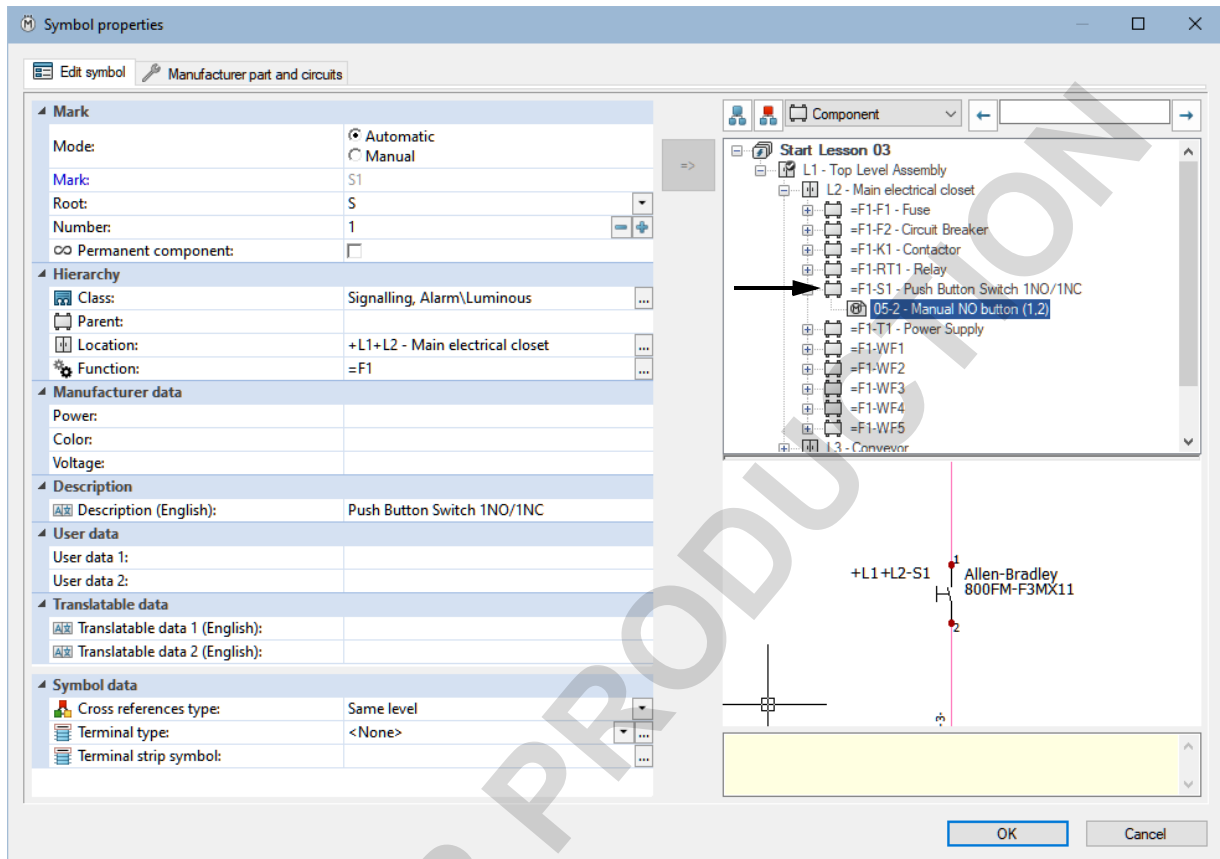
4 Insert symbol.

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.



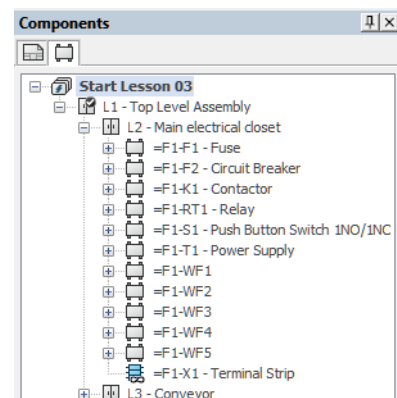
Click **OK** to create an association.

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.



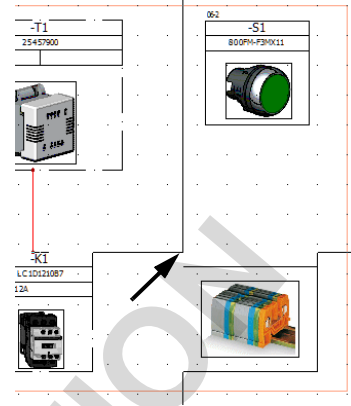
7 Insert component symbol.

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 vice versa.


Note

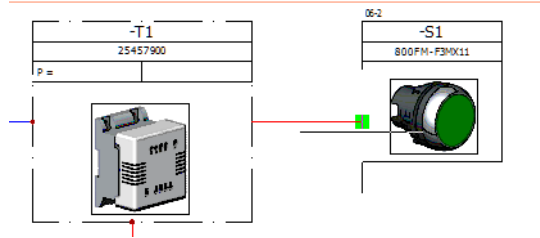
The cables are drawn to show connections between components. There
is not yet any assignment of cable conductor cores to the cable.

Where to Find It

CommandManager: **Line diagram** > **Draw cable** .

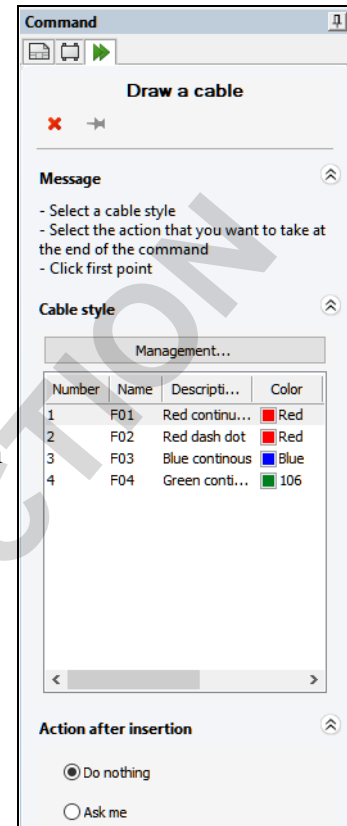
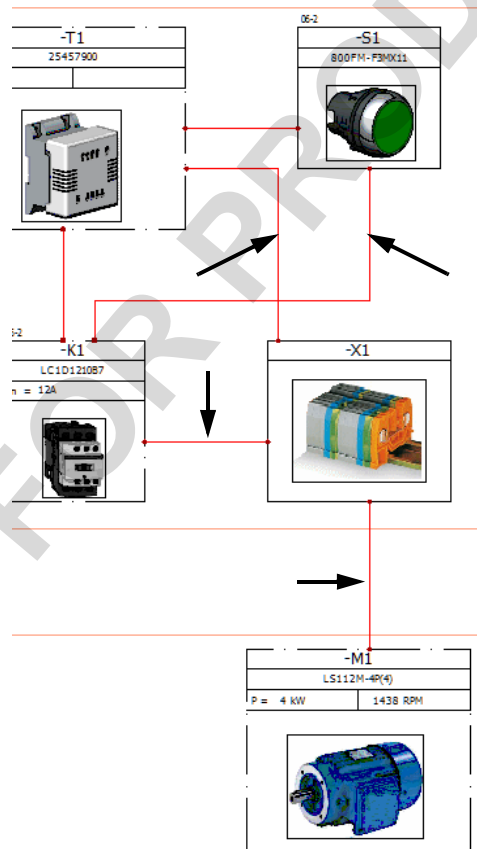
8 Cables.

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.




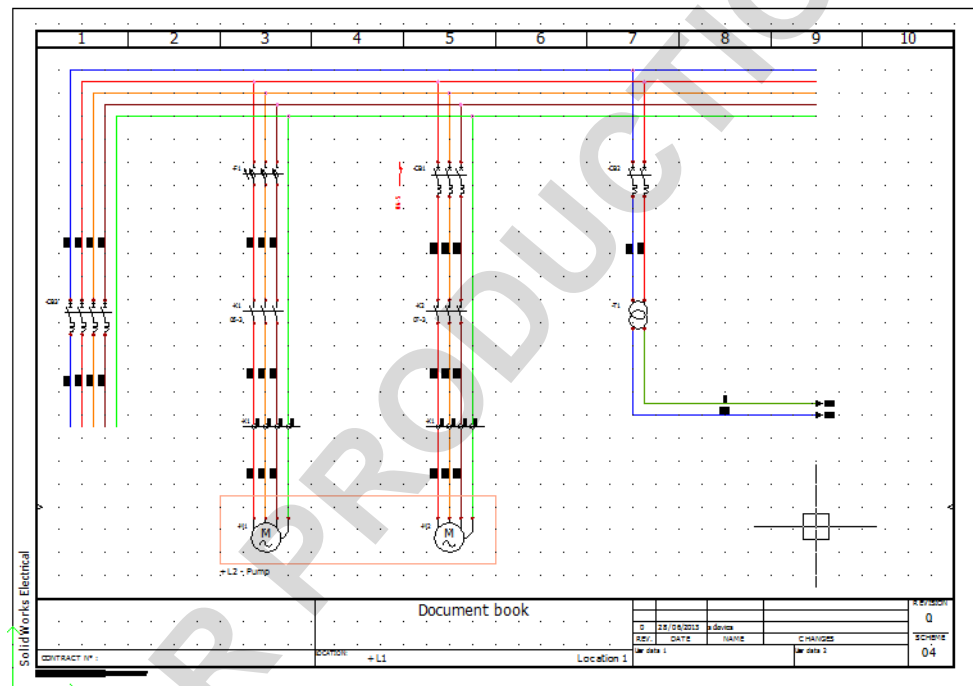
Schematic Drawing

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

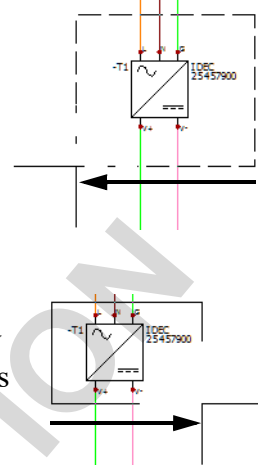
Dragging a rectangular window around geometry is a very efficient method of selecting multiple entities. It *does* make a difference whether the window is dragged left to right or from right to left.

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.

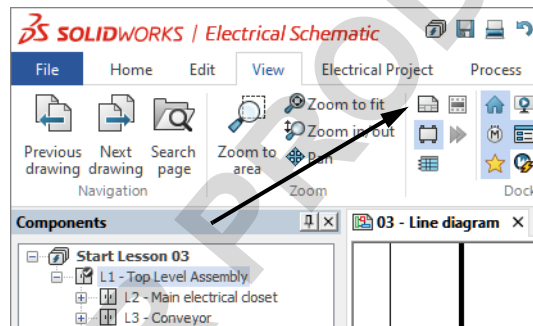
Dragging a selection window *left to right* captures geometry within the window. In this example only the symbol will be selected, as the connected wires are not fully contained in the crossing window.

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 Pages tab, is not shown then it will have been turned off.



In this image the Pages 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.

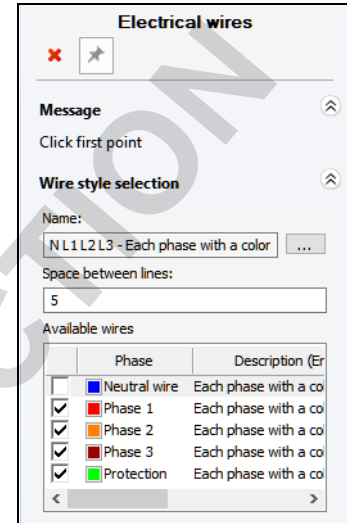
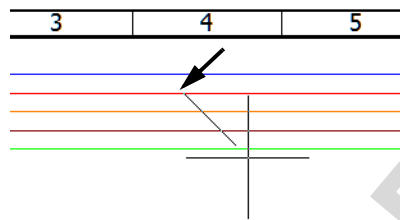
10 Select multiple wires.

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




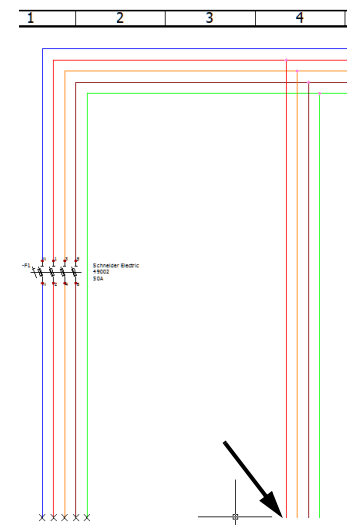
Note

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.



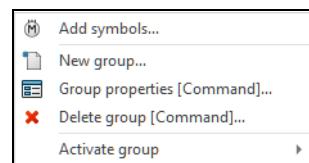
Symbols Panel

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.

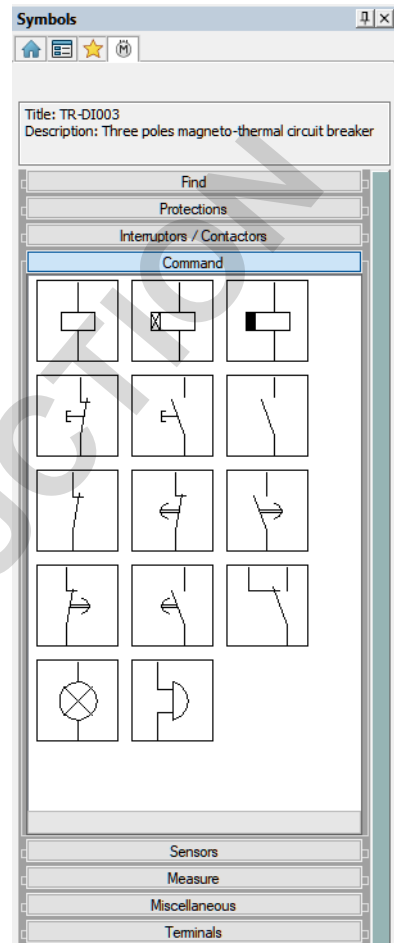
In a mixed scheme a drop down allows access to either line diagram or scheme symbols.

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.

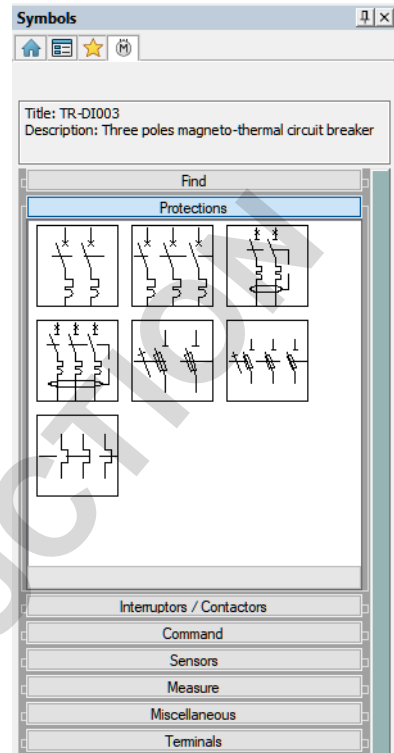
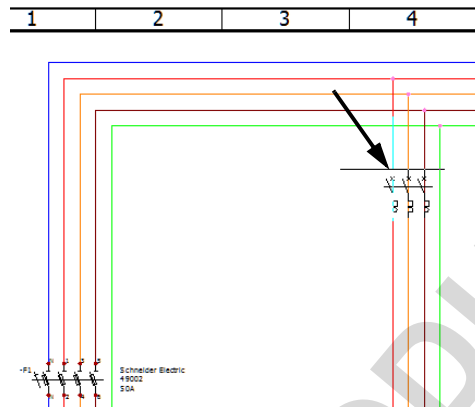
Palettes can also be added to individual projects and modified separately for each.



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**.

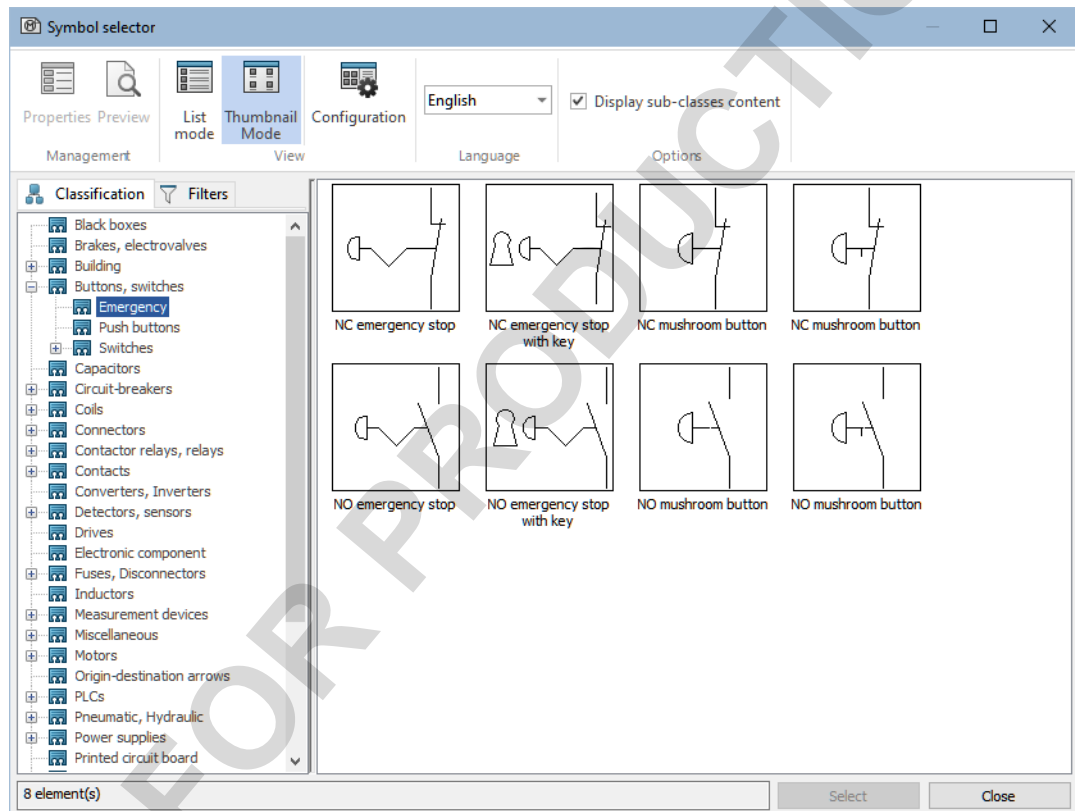
Schematic Symbols

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.

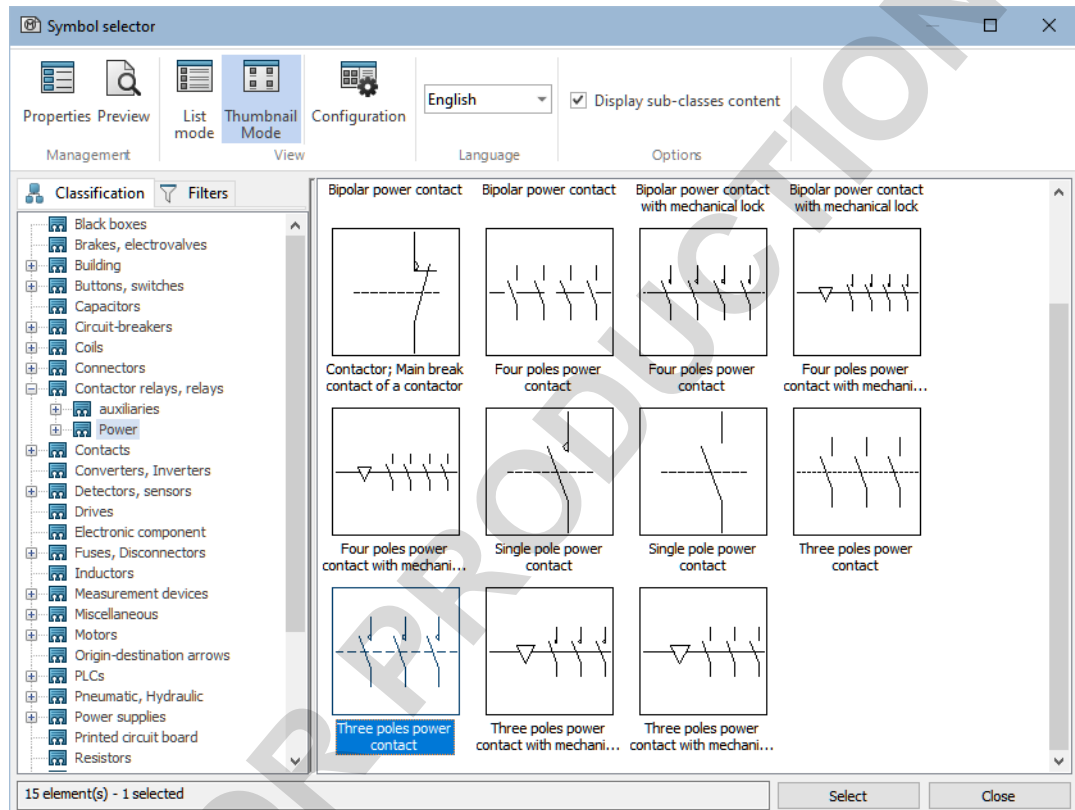
Where to Find It

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

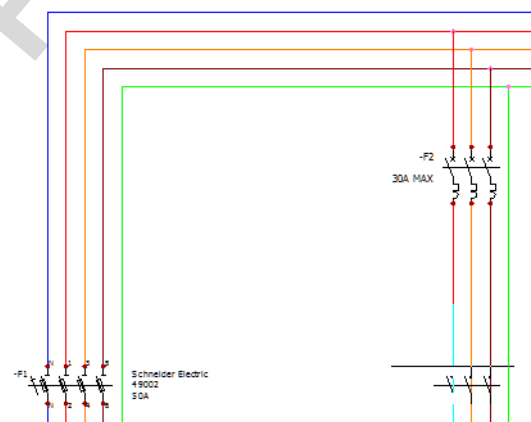
15 Insert scheme symbol.

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

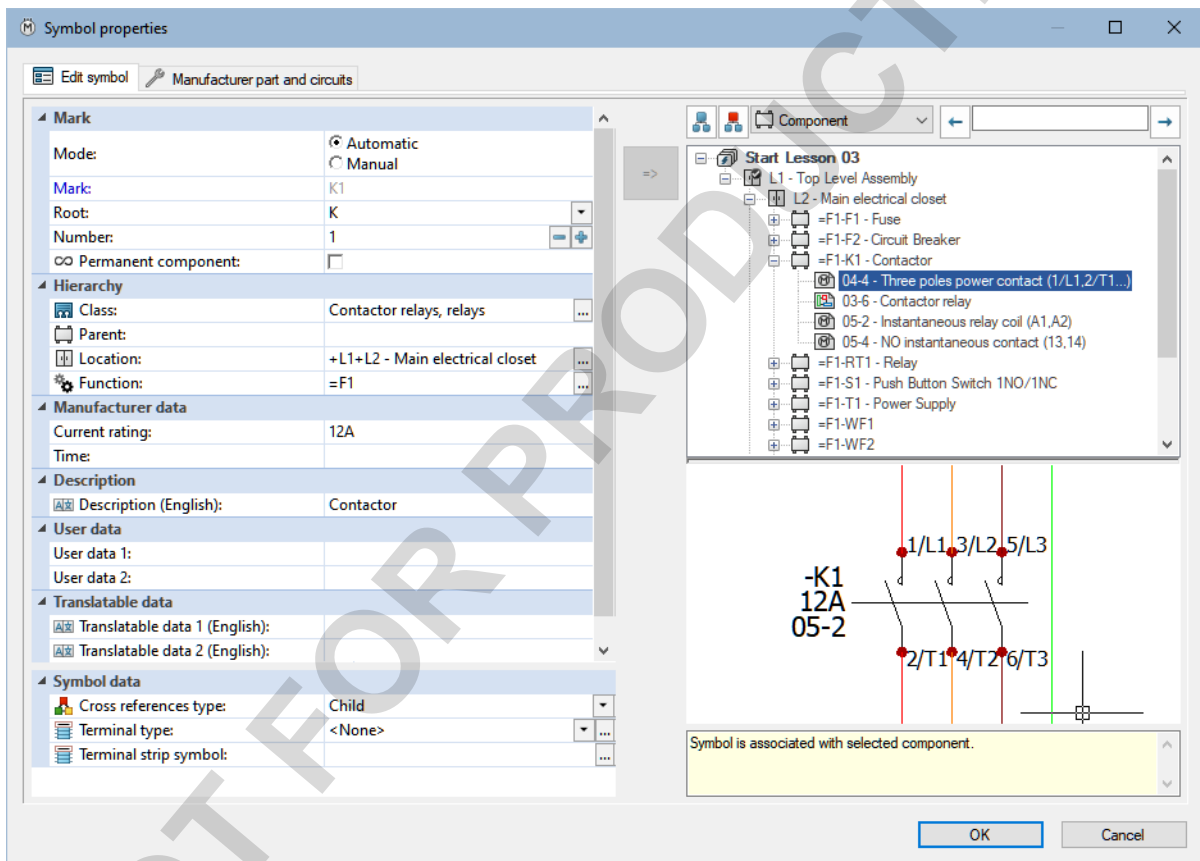
Types of Properties

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.

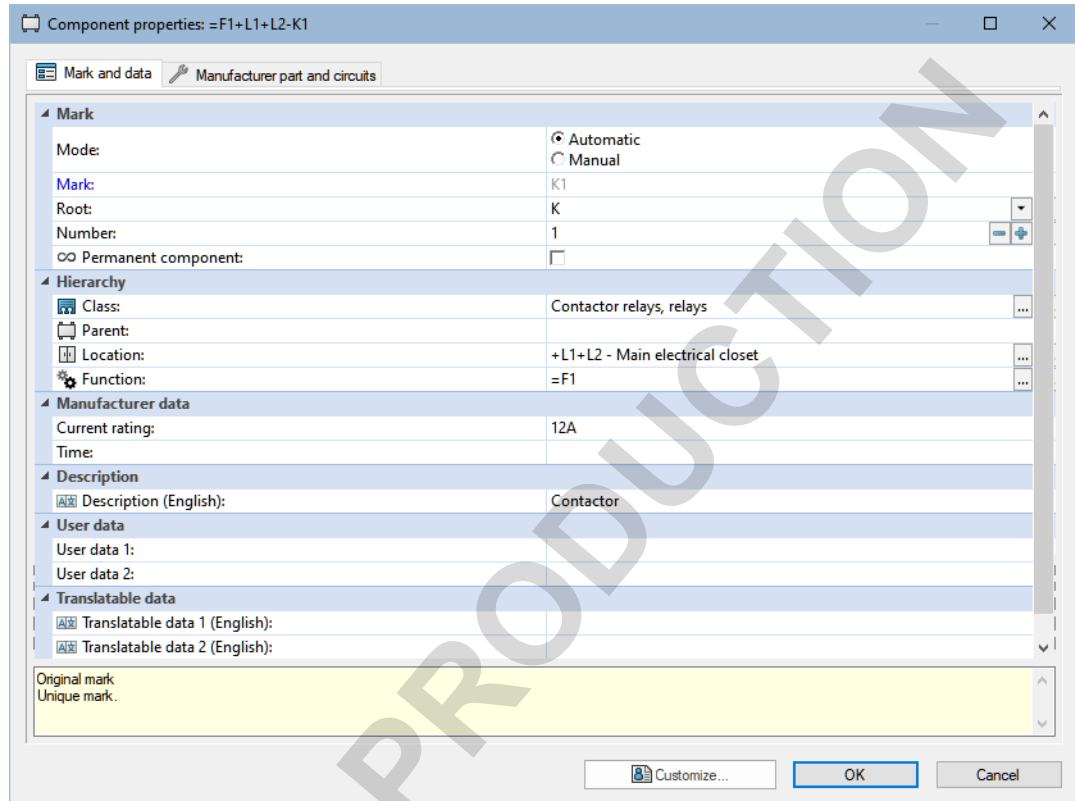
For any symbol, there is an option to select either the **Symbol properties** or the **Component properties**. Both types include the same Manufacturer parts and circuits tab (*What are Manufacturers Parts?* on page 58) used to add manufacturer parts to the symbol.

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.

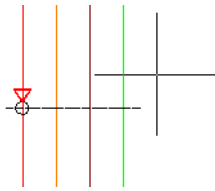
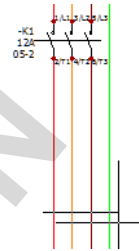
17 Insert multiple terminals.

Click to **Insert 'n' terminals**  and choose Terminal TR-B001 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.

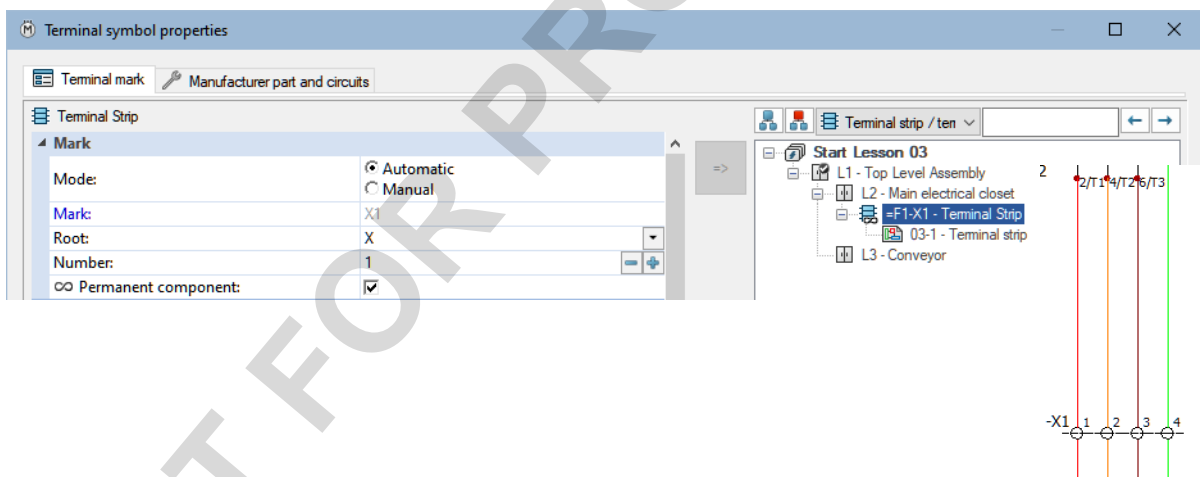
Ensure the red triangle indicator is pointing toward the bottom of the page by moving your cursor above the axis line and left-click to place.

**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.

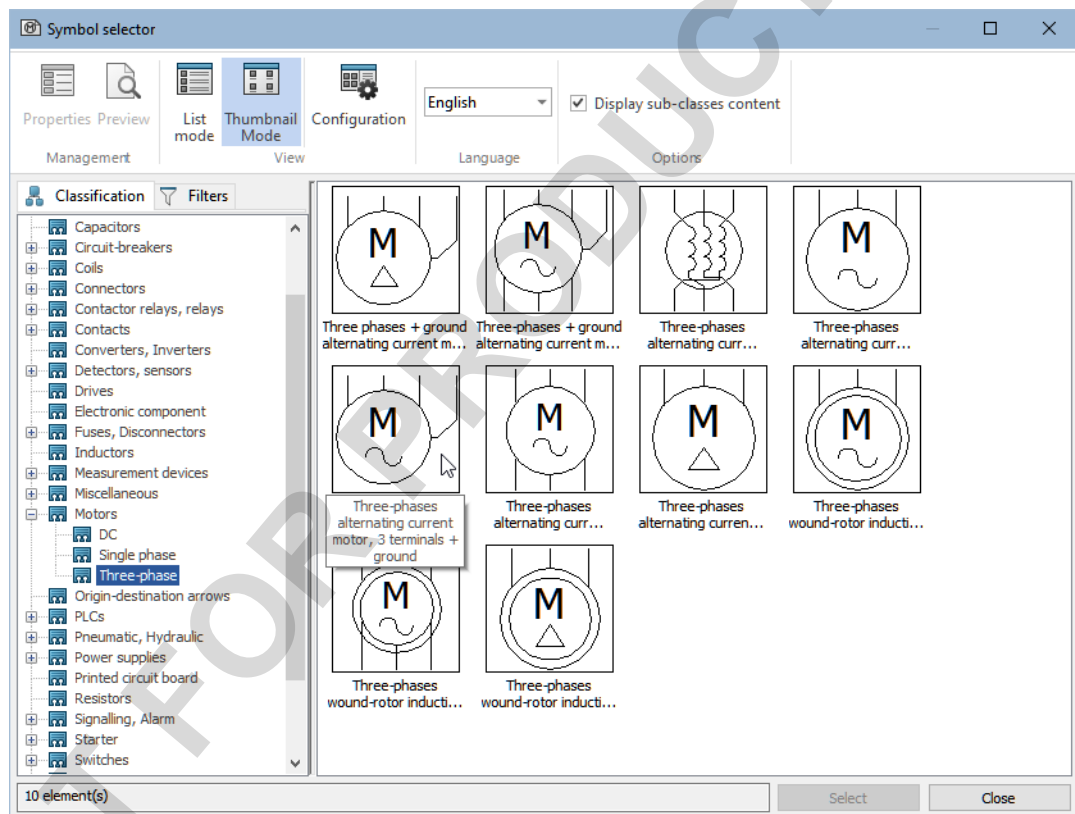
Tip

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.

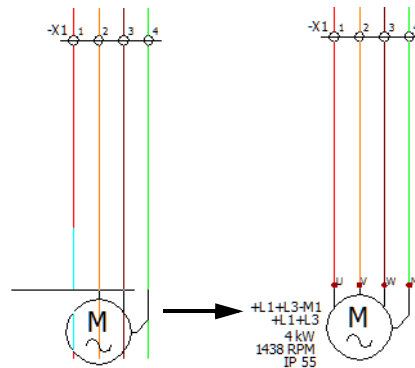
Right-click =F1-M1 - Motor component and **Insert symbol**  use the processes outlined previously to locate the following symbol:

- 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 Pages 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 10
- *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 26
- *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.

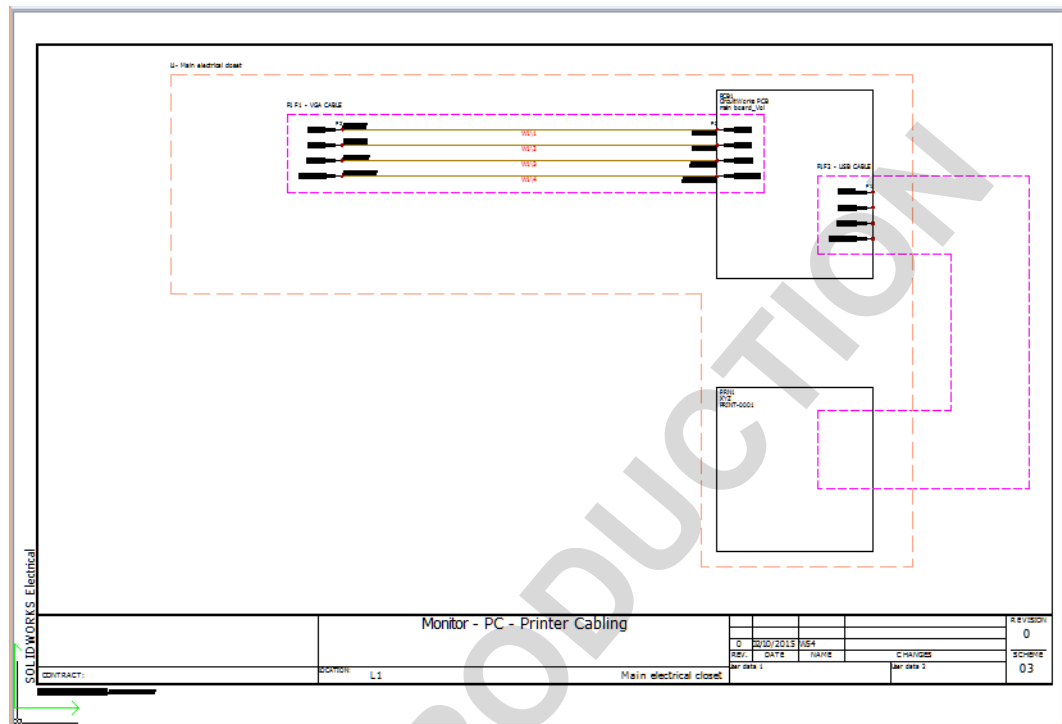
Finish the unarchive process leaving the settings as they were by default.

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.

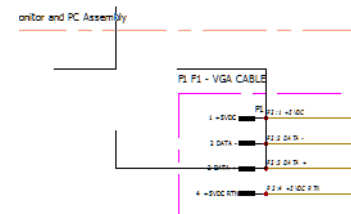
7 Select line diagram symbol.

Locate a line diagram symbol with the following properties.

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

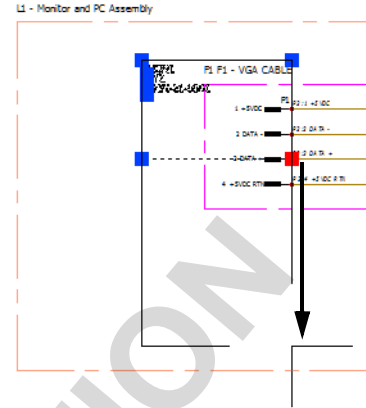
8 Insert symbol.

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_O2+1+1

13 Symbol orientation.

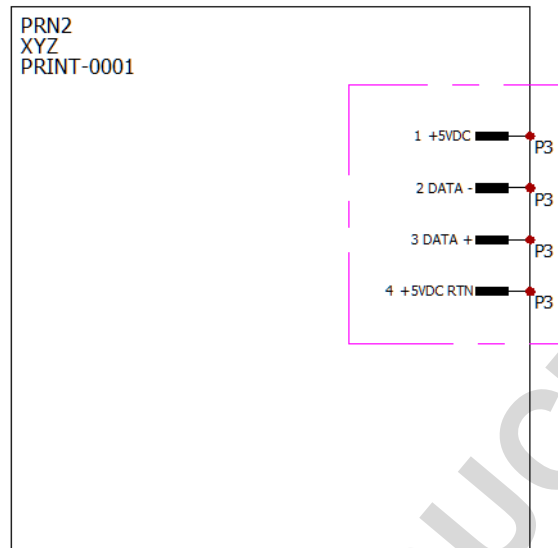
In the Symbol insertion side panel click to rotate the symbol **180°** 

14 Pin the command.

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

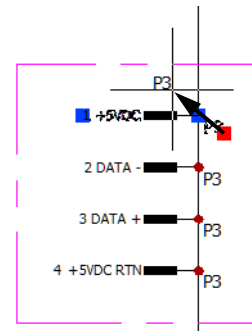
15 Position scheme symbols.

Insert 4 connector pins as illustrated.



16 Move an attribute.

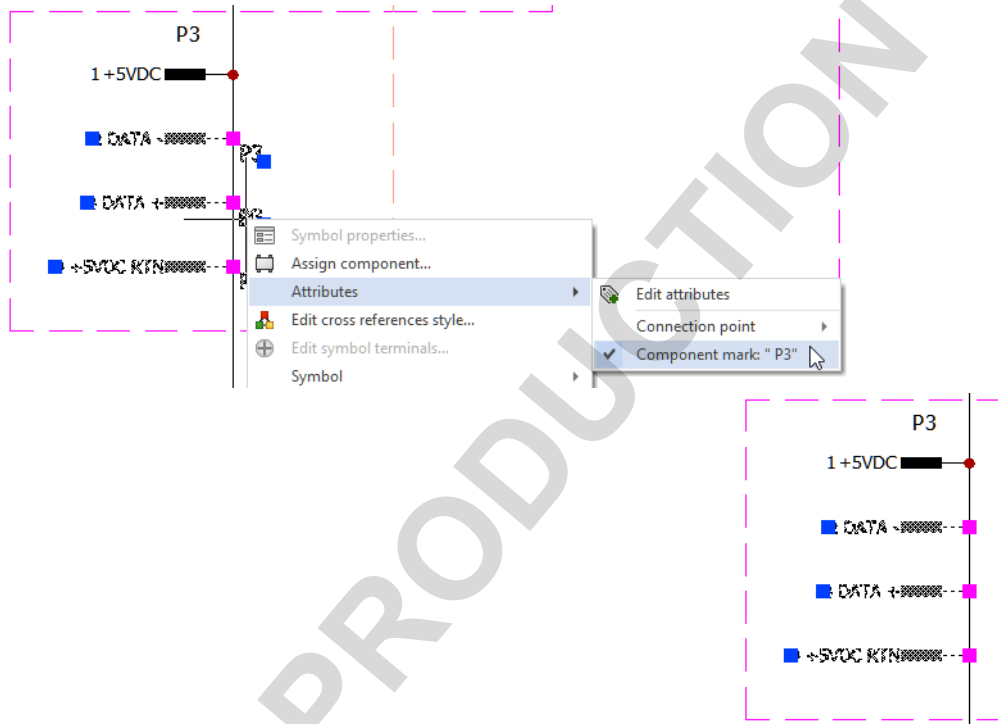
Click the connector pin 1 +5VDC and drag drop the mark P3 as illustrated.



17 Attribute visibility.

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

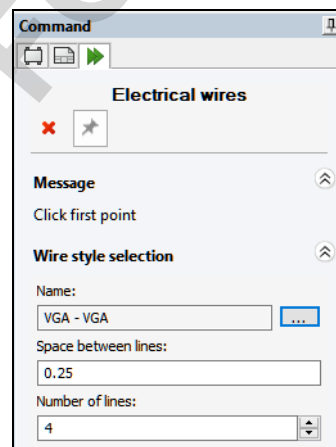
Click **Attributes > Component mark** to hide the attribute on all three symbols.

**18 Select wires.**

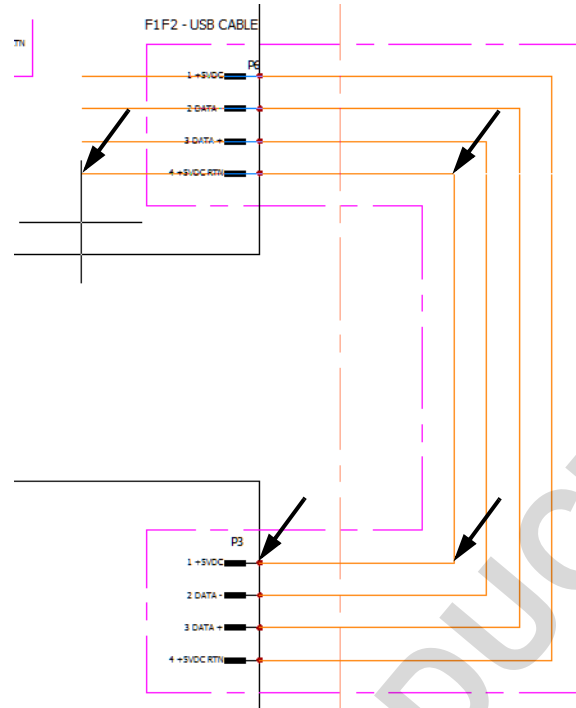
Click to **Draw single wire** selecting wire style VGA.

19 Draw multiple wires.

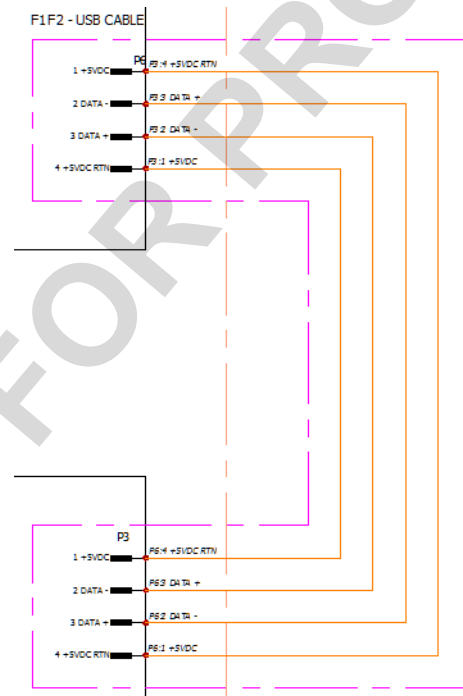
Set the electrical wires settings as illustrated.




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 Pages side panel and **Close**  the project.