



FUNDAMENTALS OF SOLIDWORKS ELECTRICAL SOLIDWORKS EDUCATION EDITION 2016-2017



ENG

This is a preview of the Fundamentals of SOLIDWORKS Electrical. 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 2016-2017

Fundamentals of SOLIDWORKS Electrical

Dassault Systèmes SolidWorks Corporation 175 Wyman Street Waltham, MA 02451 U.S.A. © 1995-2015, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

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

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

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

Patent Notices

SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028, 8,910,078, 9,129,083, 9,153,072 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).

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

U.S. and foreign patents pending.

Trademarks and Product Names for SOLIDWORKS Products and Services

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

CircuitWorks, FloXpress, PhotoView 360, and TolAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of Geometric Ltd.

SOLIDWORKS 2016, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Workgroup PDM, SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, eDrawings, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical, SOLIDWORKS Composer, and SOLIDWORKS MBD are product names of DS SolidWorks.

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

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

The Software is a "commercial item" as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of "commercial computer software" and "commercial software documentation" as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government (a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R. 227.7202-1 (JUN 1995) and 227.7202-4 (JUN 1995).

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

Copyright Notices for SOLIDWORKS Standard, Premium, Professional, and Education Products

Portions of this software © 1986-2015 Siemens Product Lifecycle Management Software Inc. All rights reserved.

This work contains the following software owned by Siemens Industry Software Limited:

D-Cubed[™] 2D DCM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed^TM 3D DCM @ 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ PGM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed™ CDM © 2015. Siemens Industry Software Limited. All Rights Reserved.

D-CubedTM AEM \otimes 2015. Siemens Industry Software Limited. All Rights Reserved.

Portions of this software © 1998-2015 Geometric Ltd.

Portions of this software incorporate PhysXTM by NVIDIA 2006-2010.

Portions of this software @ 2001-2015 Luxology, LLC. All rights reserved, patents pending.

Portions of this software © 2007-2015 DriveWorks Ltd.

Copyright 1984-2010 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.

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

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

Copyright Notices for SOLIDWORKS Simulation Products

Portions of this software © 2008 Solversoft Corporation.

PCGLSS @ 1992-2014 Computational Applications and System Integration, Inc. All rights reserved.

Copyright Notices for SOLIDWORKS Standard Product

© 2011, Microsoft Corporation. All rights reserved.

Copyright Notices for SOLIDWORKS PDM Professional Product

Outside In® Viewer Technology, © 1992-2012 Oracle © 2011, Microsoft Corporation. All rights reserved.

Copyright Notices for eDrawings Products

Portions of this software © 2000-2014 Tech Soft 3D. Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.

Portions of this software © 1998-2001 3D connexion.

Portions of this software $\ensuremath{\mathbb{C}}$ 1998-2014 Open Design Alliance. All rights reserved.

Portions of this software © 1995-2012 Spatial Corporation.

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

Portions of eDrawings $\$ for iPad $\$ copyright $\$ 1996-1999 Silicon Graphics Systems, Inc.

Portions of eDrawings $\ensuremath{\mathbb{R}}$ for iPad $\ensuremath{\mathbb{R}}$ copyright $\ensuremath{\mathbb{C}}$ 2003 - 2005 Apple Computer Inc.

Document Number: PME-SWE101-ENG

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 7	4
Use of Color	5
Graphics and Graphics Cards	5
Color Schemes	5

Lesson 1:	
Drawing Types	
	What are Drawing Types?
	Drawings
	Scheme
	Creating Drawings
	Stages in the Process
	Existing and Archived Projects
	Opening an Existing Project 10
	Unarchiving a Project
	Closing Projects
	Line Diagram Symbols 12
	Adding Symbols 12
	Symbols Library 12
	Symbol Orientation15
	Adding Cables
	Schematic Drawing
	Scheme Best Practices
	Stages in the Process
	Symbols Panel
	Schematic Symbols
	Symbol Properties
	Types of Properties
	Exercise 1: Drawing Types
Lesson 2:	

Symbols and Components

What is a component?
Component Identification
Component Symbol Identification
Stages in the Process
Symbol Component Association
Exercise 2: Symbols and Components

Lesson 3:	
Manufacturers Parts	
	What are Manufacturers Parts?50
	Circuits and Terminals
	Circuit Association
	Stages in the Process
	Finding Manufacturer Parts
	Search Options
	Editing Parts
	Circuit Symbols
	Circuit Association
	Exercise 3: Manufacturers Parts
Lesson 4:	
Insert Components	
• • • •	Insert Components
	Stages in the Process
	Align Components
	Inserting Terminals
	Exercise 4: Insert Components
Lesson 5: Routing Wires	
	Routing Wires
	Stages in the Process
	Routing Path
	Sketch Name
	Route Wires
	3DSketch Route
	Routing Parameters
	Wire Sketches
	SOLIDWORKS Route
	Wire Segregation
	Exercise 5: Routing Wires

Introduction

SOLIDWORKS

Electrical

Tutorials

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.

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

	_
SOLIDWORKS Electrical Schematic	
program is to introduce you to SOLIDWORKS Electrical through a series of simple exercises, create a project. This will allow you to use most features of the software.	
program, consult the Getting Started Guide to familiarize yourself with the commands and in the software.	
Creating a Project	
Managing Locations Creating a Synoptic Cable List	
Drawing Wires Inserting Symbols Allocating a Manufacturer Reference	
Handling Objects Inserting Black Boxes Creating Macros Copy/Paste Swapping Material Terminals Inserting Terminals	
Multilingual Text Polarity Origin-Destination Equipotential Numbering Winng and Editing Terminal Strips	
Manufacturer Parts	
PLC	
Reports	
2D Cabinet Layout	
Managing Revisions	
Managing Title Blocks	
	SOLIDWORKS Electrical Schematic rogram is to introduce you to SOLIDWORKS Electrical through a series of simple exercises, reate a project. This will allow you to use most features of the software. program, consult the Getting Started Guide to familiarize yourself with the commands and in the software. Creating a Project Managing Locations Creating a Synoptic Cable List Drawing Wires Inserting Symbols Allocating a Manufacturer Reference Handling Objects Inserting Black Boxes Creating Macros Copy/Paste Swapping Material Terminals Inserting Terminals Multilingual Text Polarity Origin-Destination Equipotential Numbering Wiring and Editing Terminal Strips Manufacturer Parts PLC Reports 2D Cabinet Layout Managing Title Blocks Managing Symbols

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 at 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 (a) 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 with th Completed the onl SOLIDWORKS so clicking Help, Onl 	experience. Ne Windows [®] operating system. Ine tutorials that are integrated in the oftware. You can access the online tutorials by line Tutorial .
Course Design Philosophy	This course is designed training. A process-bas procedures you follow studies to illustrate the options and menus in	d around a process- or task-based approach to sed training course emphasizes the processes and to complete a particular task. By utilizing case se processes, you learn the necessary commands, 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 f	following typographic conventions:
	Convention	Meaning
	Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, Features >
		Extruded Cut image means click the Extruded Cut icon on the Features tab of the CommandManager.

Typewriter

17 Do this step

Windows 7

The screen shots in this manual were made using the SOLIDWORKS software running on Windows 7. If you are not using Windows 7, or if you have a different theme setting, you may notice slight differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Feature names and file names appear in this

Double lines precede and follow sections of the procedures. This provides separation

between the steps of the procedure and large

themselves are numbered in sans serif bold.

blocks of explanatory text. The steps

style. For example, Sketch1.

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.
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, CSVand 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 LessonO1\ Case Study\Start_LessonO1.proj.tewzip see <i>Unarchiving a</i> <i>Project</i> 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.

Close

Existing and Archived Projects	There are options in SOLIDWORKS Electrical for <i>opening</i> projects and <i>unarchiving</i> 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.	
	 Click Projects manager. Click Projects manager . opened or unarchived recently are listed. Double-click a project by name. Double-click a project name under the Recent projects tab. 	
	😰 Projects Manager 📃 🗖 🗴	
	Image: Second	

Where to Find It

4

Close window at project opening

■ **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_O1.proj at the start of this lesson and use Start_Exercise_O1.proj at the start of the related exercise.
	The start files are stored in the SOLIDWORKS Training Files\ FundamentalsElectrical\Lesson folders and the appropriate Case Study and Exercise sub-folders.
Note	Unarchiving a file does <i>not</i> open it unless you click Yes on the "open" message.
	1. Click Projects manager.
	Click Projects manager and click Unarchive C . Browse to the folder C:\SOLIDWORKS Training Files\ FundamentalsElectrical\LessonO1\Case Study, click the file
	Start_Lesson_01.proj 😥 and click Open .
	 Project information. The project dialog includes text information about the project. Click OK.
	 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 .
💋 Unarchivi	ng: Projects _ 🗆 🗙



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 <a>2	
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 01.	
	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	

	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	



SymbolThe Symbol Orientation options control the rotation and possibleOrientationmirroring of a symbol as it is added to the drawing.

Original orientation		90 rotation 📤	
180 rotation 🔄		270 rotation 🖅	
90 rotation and mirror 🛋		Mirror ►	
Nata	The graphele cherry in	the symbol library will	very besed on what type
Note	of drawing or scheme	is open.	vary based on what type
Тір	of drawing or scheme Line diagram symbols	do not require an image	be inserted in them.
Tip Note	of drawing or scheme Line diagram symbols The Symbols tab on th selected symbols to an	is open. do not require an image ne resource side panel ca by of the group such as C	be inserted in them. n be modified by adding command and Sensors.
Tip Note Where to Find It	 of drawing or scheme Line diagram symbols The Symbols tab on the selected symbols to an CommandManager Side Panel: Click S Shortcut Menu: Right Symbol (5) 	the symbol holary will is open. do not require an image ne resource side panel ca by of the group such as C r: Line diagram > Inser Symbols (3) ght-click a component a	t Symbol 🕅 no click Insert
Note Tip Note Where to Find It	 The symbols shown in of drawing or scheme Line diagram symbols The Symbols tab on the selected symbols to an CommandManager Side Panel: Click \$ Shortcut Menu: Right Symbol (5) Select line diagram series	is open. do not require an image ne resource side panel ca by of the group such as C r: Line diagram > Inser Symbols (2) ght-click a component a Symbol. (2)	t Symbol 🕅 normal type
Note Tip Note Where to Find It 3 Note	 The symbols shown in of drawing or scheme Line diagram symbols The Symbols tab on the selected symbols to an CommandManager Side Panel: Click S Shortcut Menu: Rither Symbol Select line diagram service Click Insert symbol If this is the first time selector will be display 	the symbol horary will is open. do not require an image ne resource side panel ca by of the group such as C r: Line diagram > Inser Symbols (3) ght-click a component a symbol. (3) you have inserted a symbol ayed automatically.	be inserted in them. n be modified by adding command and Sensors. t Symbol 😥 nd click Insert bol then the Symbols



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.

- Mark			Es Es 🚸 Component 🔹 🗇	
Mode:	Automatic Manual		Eeson 3A	
Mark:	S1		L1 - Top Level Assembly	
Root:	S	-	El El El El	
Number:	1		=FI-FI-FUSE	
∞ Permanent component:			=F1+F2 - Circuit bleaker	
- Hierarchy			-F1-RT1 - Relay	
Class:	Signalling, Alarm\Luminous		=F1-S1 - Push Button Switch 1NO/1NO	
📀 Master:			(C) 06-2 - Manual NO button (1.2)	
Location:	+L1+L2 - Main electrical doset]		
Sunction:	=F1]		_
Manufacturer data				
Power:				
Color:				
Voltage:				
Description			1	
Description (English):	Push Button Switch 1NO/1NC		+L1+L2-S1 Alen-Bradley	
User data			A GOOLAN SHALL	
User data 1:			•2 ⁰³⁻⁷	
User data 2:				
 Translatable data 				
🎆 Translatable data 1 (English):				
🎆 Translatable data 2 (English):				
			¥	-
Symbol data				
Same level		-		

Click **OK** to create an association.

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.



Note

	7 Insert component symbol. Right-click component =F1-X1 - Terminal Strip and click Insert symbol (1)				
	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				
Note	As the symbol is being inserted <i>from</i> 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 <i>reserve</i> a cable for use between two components, or define the detailed wiring between sumbols in line diagrams				
	witting between symbols in fine diagrams.				
Тір	This information bi-directionally updates in schemes and vise versa.				
Tip Note	This information bi-directionally updates in schemes and vise versa. The cables are drawn to show connections between components. There is not yet any assignment of cable conductor cores to the cable.				

8 Cables.

Click **Draw cable** 1 and connect the symbols as illustrated.



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.





Тір

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** (1) command to ensure that superfluous data, such as line diagram, or footprint symbols, are unavailable.



Note

Scheme Best Practices **Schemes** such as the drawing O4 - Power appear with the \bowtie icon in the document list.

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.

The major stages in the process are listed below:

Stages in the Process

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.



Electrical wires	
Message	۲
Click first point	
Wire style selection	۲
Name:	
N L1 L2 L3 - Each phase with a color	
Space between lines:	
5	
Available wires	
Neutral wire	
Phase 1	
Phase 2	
Phase 3	
Protection	

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 $\mathbf{OK} \lor \mathbf{i}$ 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.

Ð	Add symbols	
ø	New group	
(\mathbf{i})	Group properties [Sensors]	
X	Delete group [Sensors]	
	Activate group	+

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-DIO03.

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.



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.

- CommandManager: Schematic > Insert Symbol 🕅
- Side Panel: Click Symbols 🕲

Note

Where to Find It

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 (<i>What are</i> <i>Manufacturers Parts?</i> on page 50) 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.

- Mark				E E Scomponent 🗸 🗇
Mode:		Automatic Manual	[E L2 - Main electrical closet
Mark:		K1		= F1-F1 - Fuse
Root:		к	-	+ + + + + - + Circuit Breaker
Number:		1	- 4	
∞ Permanent component	:			B 036 - Contactor relay
- Hierarchy				(1) 05-2 - Instantaneous relay coil (A1
Class:		Contactor relays, relays		(t) 05-4 - NO instantaneous contact (1
Naster:				= + - + = F1-RT1 - Relay
Location:		+L1+L2 - Main electrical doset		
Sunction:		=F1		
Manufacturer data				
Current rating:		12A		
Time:				
Description				1/11 0/10 5/10
E Description (English):		Contactor		41/L143/L243/L3
User data				-K1 \d \d \d
User data 1:				12A
User data 2:				05-2
Translatable data				2/T1T4/T2T6/T3
🎆 Translatable data 1 (E	nglish):			
Translatable data 2 (E	nglish):			
Combol data				Symbol is associated with selected component.
Symbol data	0.41			
Cross references type:	Child		Ŧ	

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.

😵 Component proper	Component properties : =F1+L1+L2-K1 – E			
(j) Mark and data	Manufacturer part and circuits			
- Mark				
Mode:	Automatic			
Mark-	K1			
Boot:	K	-		
Number:	1			
Number:				
CO Permanent con	mponent:			
- Hierarchy				
E Class:	Contactor relays, relays			
Master:				
Location:	+L1+L2 - Main electrical doset			
Sunction:	=F1			
Manufacturer da	ata			
Current rating:	12A			
Time:				
Description				
Description (En	nglish): Contactor			
- User data				
User data 1				
User data 1:				
User data 2:	• 22			
- I ranslatable dat	ta	T		
Original mark				
	 properties (1) Shortcut Menu: Right-click a symbol and click Comp properties (1) 	onent		
	Double-clicking a symbol accesses the Component prop	erties.		
	The following section uses Symbol properties .			
16	Coil-contact association. The contact is a part of the relay coil already represented diagram and control scheme, associating the contact to the exists to which the symbol can be associated.	in the wiring is component		
	=F1-K1 - Contactor and click OK to create the associat	tion.		
17	Insert multiple terminals.			
	Click to Insert 'n' terminals bbb and choose Terminal			

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.

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

Tip



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** is 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 LessonO1\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.

