



FUNDAMENTALS OF SOLIDWORKS ELECTRICAL

SOLIDWORKS EDUCATION EDITION 2021-2022



SOLIDWORKS[®] Education Edition

Fundamentals of SOLIDWORKS Electrical

To be used with SOLIDWORKS Education Edition 2020-2021 or 2021-2022

Dassault Systèmes SolidWorks Corporation 175 Wyman Street Waltham, MA 02451 U.S.A. © 1995-2019, Dassault Systemes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

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

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

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

Patent Notices

SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072; 9,262,863; 9,465,894; 9,646,412; 9,870,436; 10,055,083; 10,073,600; 10,235,493 and foreign patents, (e.g., EP 1,116,190 B1 and JP 3,517,643).

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

U.S. and foreign patents pending.

Trademarks and Product Names for SOLIDWORKS Products and Services

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

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

FeatureWorks is a registered trademark of HCL Technologies Ltd.

SOLIDWORKS 2020, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, SOLIDWORKS Flow Simulation, SOLIDWORKS CAM, SOLIDWORKS Manage, eDrawings Viewer, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic Standard, SOLIDWORKS Electrical Schematic SOLIDWORKS Electrical 3D, SOLIDWORKS Electrical Professional, CircuitWorks, SOLIDWORKS Composer, SOLIDWORKS Inspection, SOLIDWORKS MBD, SOLIDWORKS PCB powered by Altium, SOLIDWORKS Visualize are product names of DS SolidWorks.

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

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

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

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

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

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

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

D-Cubed® 2D DCM © 2019. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed @ 3D DCM @ 2019. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed @ PGM @ 2019. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed® CDM @ 2019. Siemens Industry Software Limited. All Rights Reserved.

D-Cubed® AEM © 2019. Siemens Industry Software Limited. All Rights Reserved.

Portions of this software © 1998-2019 HCL Technologies Ltd.

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

Portions of this software @ 2001-2019 Luxology, LLC. All rights reserved, patents pending.

Portions of this software © 2007-2019 DriveWorks Ltd. © 2012, Microsoft Corporation. All rights reserved.

Includes Adobe® PDF Library technology.

Copyright 1984-2016 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 6,563,502; 6,639,593; 6,754,382; Patents Pending.

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

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

Copyright Notices for SOLIDWORKS Simulation Products

Portions of this software © 2008 Solversoft Corporation.

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

Copyright Notices for SOLIDWORKS PDM Professional Product

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

Copyright Notices for eDrawings Products

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

Portions of this software $\ensuremath{\mathbb{C}}$ 1995-1998 Jean-Loup Gailly and Mark Adler.

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

Portions of this software @ 1998-2017 Open Design Alliance. All rights reserved.

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 $\mbox{\ensuremath{\mathbb{B}}}$ for iPad $\mbox{\ensuremath{\mathbb{C}}}$ copyright $\mbox{\ensuremath{\mathbb{C}}}$ 2003 – 2005 Apple Computer Inc.

Copyright Notices for SOLIDWORKS PCB Products

Portions of this software © 2017-2018 Altium Limited.

Copyright Notices for SOLIDWORKS Visualize Products

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

Document Number: PME-SWE106-ENG

Contents

Introduction

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

Lesson 1:	
Drawing Types	
	What are Drawing Types?
	Drawings
	Scheme
	Creating Drawings9
	Stages in the Process
	Existing and Archived Projects
	Opening an Existing Project
	Unarchiving a Project
	Closing Projects
	Line Diagram Symbols 12
	Adding Symbols
	Symbols Library
	Symbol Orientation
	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?	38
Component Identification	38
Component Symbol Identification	39
Stages in the Process	39
Description Columns	43
Symbol Component Association.	46
Exercise 2: Symbols and Components	49

Lesson 3: Manufacturers Parts	
	What are Manufacturers Parts?
	Circuits and Terminals
	Circuit Association
	Stages in the Process
	Finding Manufacturer Parts
	Search Options
	Editing Parts
	Circuit Symbols
	Circuit Association
	Electrical Assemblies
	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

To the TeacherThe 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.

👔 🐺 Tutorial	
Tutorials	- SOLIDWORKS Electrical Schematic
The aim of this allowing you to	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.
Before using th vocabulary use	is program, consult the Getting Started Guide to familiarize yourself with the commands and d in the software.
Exercise #1	Creating a Project
Exercise #2	Managing Locations Creating a Synoptic Cable List
Exercise #3	Drawing Wires Inserting Symbols Allocating a Manufacturer Reference
Exercise #4	Handling Objects Inserting Black Boxes Creating Macros Copy/Paste Swapping Material Terminals Inserting Terminals
Exercise #5	Multilingual Text Polarity Origin-Destination Equipotential Numbering Wiring and Editing Terminal Strips
Exercise #6	Manufacturer Parts
Exercise #7	PLC
Exercise #B	Reports
Exercise #9	2D Cabinet Layout
Exercise #10	Managing Revisions
Exercise #11	Managing Title Blocks
Exercise #12	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 progra 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 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 f	collowing typographic conventions:								
	Convention	Meaning								
	Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, Features >								
	Extruded Cut icon on the Features tab of the CommandManager									

Typewriter

17 Do this step

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.

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 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 \swarrow 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 LessonO3\ Case Study\Start_LessonO3.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	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 : All the projects that have been

opened or unarchived recently are listed.

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

🍯 Proj	jects Manager														_		×
New	Unarchive	Jnarchive Open Set as Close current			Archive Duplicate Save as template			Configuration Englis		English	nglish 🔹		Filters	Update files for PDM			
				Mana	agement					View		Lar	anguage		Filters	ters Link to	
🕖 Re	cent projects	🥞 Allı	projects														
ID		Project n	nark		Project description 1 (English) Contract					number Customer			Creation dat		te Modificatio		n date
6	Training Le	sson SW	E		Training Lesson						(02/10/20	18	02/	10/201	8	
<																	>
Keep	the dialog open	ed														Close	

Where to Find It
Projects Manager PropertyManager: Double-click a recent project from the list

Unarchiving aArchived 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 FilesTo 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	Un me	exactiving a file does <i>not</i> open it unless you click Yes on the "open" essage.
	1.	Click Projects manager.
		Click Projects manager is and click Unarchive in Browse to the folder C:\SOLIDWORKS Training Files\ SOLIDWORKS Electrical Schematic\Lesson03\Case Study,
		click the file Start_Lesson_03.proj ቓ 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 Closing Projects Projects Manager PropertyManager: Unarchive

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	EW_SY_TransformerHigh	EW_SY_TransformerLow
Cells battery	High Voltage Transformer	Low Voltage Transformer
Loads		
EW_SY_Motor	EW_SY_Resistor Heat	EW_SY_Jack
Motor	Heating resistor	Cylinder





Where to Find It

Note

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

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

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

- 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 (28).

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.

	Command 4
🕲 Symbol selector	Symbol insertion ×
Image: Second	
Management View Language Options	Message 🙁
A Classification Y Filters	Specify insertion point.
Boxes, cabinets Building Buttons, switches Emergency stop Push buttons Contactor relays, relays Contactor relays, relays Fuses, Disconnectors Stop button	Specify the symbol to insert
Measurement devices Miscellaneous	Options
Motors PLCs Somer supplies Resistors	☐ Manual mark ☑ Ask for properties ☑ Adjust size to drawing unit
Grandling, Alarm Grandling, Alarm Grandling, Alarm Terminal strips, terminals Transformers Transformers	
5 element(s) - 1 selected	Select Close

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				🚓 🎝 🚸 Component 🗸 🧹	-
Mode:	 O Automatic ○ Manual 		- `	Start Lesson 03	^
Mark:	S1			L1 - Top Level Assembly	
Root:	S	-		= = F1-F1 - Fuse	
Number:	1	- +		F1-F2 - Circuit Breaker	
∞ Permanent component:				🖶 🚸 =F1-K1 - Contactor	
Hierarchy				🕀 🚸 =F1-RT1 - Relay	
Class:	Signalling, Alarm\Luminous			= 🔷 = F1-S1 - Push Button Switch 1NO/1NC	
🚸 Master:				-F1-T1 - Power Supply	
Location:	+L1+L2 - Main electrical closet				
🐀 Function:	=F1				- 1
 Manufacturer data 				🖅 🚸 =F1-WF3	
Power:				€ \$\$	
Color:				±	
Voltage:					
Description					
secription (English):	Push Button Switch 1NO/1NC				
User data				.1	
User data 1:				+L1+L2-S1 Allen-Bradley	
User data 2:					
 Translatable data 				2	
🌆 Translatable data 1 (English):					
🔜 Translatable data 2 (English):					
Symbol data				ń	
Cross references type:	Same level	*			

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.



	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 <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.
Adding Cables	Wiring diagrams indicate system level interconnects between components. These interconnects are realized by a single line indicative of a cable.
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.
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 symbols in line diagrams.
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 symbols in line diagrams. This information bi-directionally updates in schemes and vise versa.
Adding Cables Tip Note	 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 symbols in line diagrams. 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.



Command				ф.
	Com	nmand		
	"			1
	_			
	D)raw a cab	le	
		×	5	
Message				*
- Select a - Select th of the con - Click firs	cable styl e action t nmand t point	e that you want	to take at th	ie end
Cable style	e			۲
		Manager		
Number	Name	Descripti	Color	
1	F01	Red continu	. 📕 Red	Cont
2	F02	Red dash dot	Red	Dash
3	F03	Blue continous	s Blue	Cont
<				>
Action aft	ter insert	tion		۲
O Do r	nothing			
() Ask	me			

Tip

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

Scheme Best Practices **Schemes** such as the drawing O4 - Power appear with the \square 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.

Lesson 1 Drawing Types

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 the 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	
M 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 **OK** \checkmark 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.

ē	Add symbols	
	New group	
6	Group properties [Command]	
×	Delete group [Command]	
	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.

SchematicThe Schematic Symbols library stores many symbolicSymbolsrepresentations. 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 (<i>What are Manufacturers Parts?</i> on page 54) 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 Mode: Mark: Root: Number: ∞ Permanent component: Hierarchy Class: Master: Location: Function: Manufacturer data Current rating: 	© Automatic © Manual K1 K 1 Contactor relays, relays +L1+L2 - Main electrical closet =F1 12A		ΞŅ	Start Lesson 03 L1 - Top Level Assembly L2 - Main electrical closet F1-F1 - Fuse F1-F2 - Circuit Breaker F1-F2 - Circuit Breaker F1-F2 - Circuit Breaker F1-F2 - Circuit Breaker F1-F1 - Fuse F1-F1 - Suse F1-F1 - Circuit Breaker F1-F1 - Suse F1-F1 - Fuse F1-F1 - Power Supply
Time: Description Description (English): User data User data 1: User data 2: Translatable data Symbol data Cross references type:	Contactor	~		-K1 12A 05-2 2/T1 ⁰ 4/T2 ⁶ /T3

Component **Component properties** includes the Mark and Data and Manufacturer **Properties** 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 properties : =F1+L1+L2-K1	- D X					
	😚 Mark and data 🥕 Manufacturer part and circuits						
	🖃 Mark	^					
	Mode:	 O Automatic ○ Manual 					
	Mark:	K1					
	Root:	К 🗸					
	Number:	1 - +					
	○ Permanent component:						
	Hierarchy						
	Class:	Contactor relays, relays					
	Master:						
	Location:	+L1+L2 - Main electrical closet					
	% Function:	=F1					
	Manufacturer data						
	Current rating:	12A					
	Time:						
	Description	A					
	Description (English):	Contactor					
	- User data						
	User data 1:						
□ Transla Original ma Unique ma							
	Original mark Unique mark.						
		Customize OK Cancel					
Where to Fir	nd It Shortcut Menu	a: Right-click a symbol and click					
	Symbol prope ■ Shortcut Menu	erties 6 1: Right-click a symbol and click					
	0	roportios 💰					
	Component p						
Note	Component p Double-clicking a	symbol accesses the Component 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.

Terminal Strip				🔏 🛃 🚍 Terminal strip / ten 🗸			+
Mark Mode:	© Automatic © Manual	^	=>	Start Lesson 03 L1 - Top Level Assembly	/ 2	2/T	4/T2
Mark: Root:	X1 X	•		= = = = = = = = = = = = = = = = = = =	al Strip nal strip		
Number:	1	-+		i L3 - Conveyor			
Permanent component:	V						

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** (2) 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.



Tip
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 \square 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 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.

U- Hain electrical doest					
			Section of a		
	RI FI - VISA CABLE		min board_Vol		
		W1/1			
		W1/2			
		WE(2		RLF2 - USB CABLE	
		W1/#		r	
	L				
			PR04		
			N/Z #077-0001		
			i l		
				il	
	M	Ionitor - PC - Printer Cab	ling		REV
			0 800/2	015 454	(
	1		REV. DAT	E NAME CHANGES	50

6 Line diagram component symbol.

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

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.



Тір

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 $Pin \rightleftharpoons$ 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.

ommand	
Command	
Electrical wires	;
×	
Message	é
Click first point	
Wiss stale coloration	8
wire style selection	
Name:	
Name: VGA - VGA	
Name: VGA - VGA Space between lines:	
Name: VGA - VGA Space between lines: 0.25	
Name:	

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



Then click to **Cancel** \mathbf{x} the command.



20 Close the project.

Right-click project name in the Documents side panel and Close \square 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 54).

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 $\bigotimes = 1.43$ 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.
 - \Rightarrow 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: 0.5+01-NO instantaneous contact 0.5+01-NO instantaneous contact 0.5+01-NO instantaneous contact
	 Means that the component has a representation in the form of a <i>scheme</i> symbol. means that the component has a representation in the form of a <i>wiring diagram</i> symbol. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>SOLIDWORKS</i> part. means that the component has a representation in the form of a <i>Connection Label</i> symbol. means that the component has a representation in the form of a <i>Connection Label</i> symbol.
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 <i>permanency</i> .
	 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 <i>Unarchiving a Project</i> on page 10.
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** \rightarrow command to go directly to the symbol.



-F1 20A U U U T8,5x31.5 + + +

5 Find a component.

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

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

M Search component: St	art Lesson 04					-		×
Mark	F	the Description		_	Master]		
Location		Supervision			Destination			
Å Class		🗡 Manufacturer	part		Component type	<all></all>		•
			Filters					
						*)	\$
🖃 🥞 Start Lesson 04				[b	_
E L1 - Electrical En	ndosure					I.		
======================================	ue -				-			
🖃 🚸 =F1-F2					-	2		
	7 - Single pole fuse (1,2)			L	20	ΑÏ		
				L	18,5x31,	5 2		
						1		
						-		
							Close	

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

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

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

6 Component review.

Click **Pan** 3 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

Tip

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** ★ and click **OK** to confirm.

SOLIDWORKS Electrical	×
Cannot delete all the selected elements.	
Cannot delete component 'F3' : - Symbols representing the component still exist	
	Close

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**

🔏 Classification 🌱 Filter	s		\rightarrow	arch						English		~
🖗 Ren	nove filters		~		Automatic refresh							
Classification		^	Number	Reference	Description (En	Manufacturer	Class	.	Type	Circu	it	
In the class	 Any class Same class Same base class 		2 1 2	LC1D1210B7 LC1D1210B7	CONTACTOR 12A 12A Coil	. Schneider Electric Telemecanique	Power Power		Base Base	6 6		12 12
Class:	**** Unclassed elemen	ts ***										
🗉 🏹 General												
🧧 Library:	<all></all>	•										
Туре:	Base	•										
Mark root:	<all></all>	•										
🗉 🌱 Manufacturer part												
🔑 Manufacturer:	<all></all>	•	<									>
🖋 Reference:	LC1D1210B7		1		+ +							
Series:												
Article number:			Manufactu	rer	Reference		Descri	State	Description	Terminal number	Associ	iated sy
Description (English):								4	Relay coil	A1, A2		
Commercial reference (Eng	gi 🛛							4	NO power	1/L1, 2/T1		
Supplier								4	NO power	3/L2, 4/T2		
E Creation date								4	NO power	5/L3, 6/T3		
 Modification date 								4	NO Contact	13, 14		
Circuit								+	NO CONTACT	23, 24		
🖧 Number of circuits:	<all></all>	*										
Number of terminals:	<all></all>	- v	<				>	<)
The selected manufacturer part	s will be added to the new o	omponer	nt.									
Number of manufacturer parts f	ound: 2								Sele	rt	Close	

Description Columns

Descriptions fields are found throughout the program interfaces. These are linked to a Translatable Data fields which has multiple languages available.

This allows users to populate a description in multiple languages; by selecting preferred Project languages the description fields will be shown Main language field will display the matching description languages.



Changing the Main language in the Project configuration will switch the language descriptions in multiple locations throughout the program, affecting everything project related from drawing titleblocks to report content.

Interfaces, such as the Manufacturer Parts selection are regarded as application level dialogs as the information is consistent and may be accessed and used in any project. Due to this the description language is user definable in application dialogs.

The language drop downs allow for selection of a preferred language, that sets the description listings.

Manufacturer part selection				_		×
Classification Remove filters	C Sear	ch 🗌 Auto	matic refresh	English		~
- Vassification	Number	Reference 🔻	Description (English)	1	Manufacturer	^
C Any class	/ 15330 / 15331	3SB3651-0AA51 C 3SB3214-0AA41 C	OMPLETE UNIT ROUND ILLUMINATE	D PUSH Sier TED PUSSier	mens mens	
✗ Manufacturer part selection					- 🗆	×
Classification Filters Remove filters	2	Search	Automatic refresh	French		
□ ♥ Classification	Number	Reference	Description (Frend	h)	Manufact	urer /
C Any class In the class C Same class C Same base class	/15330 /15331 /15332	3SB3651-0AA51 3SB3214-0AA41 3SB3203-1HA20	APPAREIL COMPLET ROND BOUT AUXIL.DE COMMANDE COMPLET AUXIL.DE COMMANDE COMPLET	FON-POUSSO E,ROND BOU E,ROND BOU	. Siemens . Siemens . Siemens	
Class: Buttons, switches	15333	3SB3203-0AA21	AUXIL.DE COMMANDE COMPLET	E,ROND BOU	. Siemens	

	10 Change Languages.
	Click the language drop down and click French .
Note	The descriptions will retain the original language unless the
	automatic refresh check box is active, or until the Search 💈 button is clicked.
	Click Search 🔁.
	Click the language drop down and click English then click Search.
	11 Add a part.
	Highlight the listed part LC1D1210B7 and click to Add 🖬 the part.
Note	Adding copies the part from the application to the project component.
	Click Select to confirm.
	12 Multiple components.Leave the number of components as 1 and click OK to create a permanent or mandatory component.
	Multiple insertion



13 Insert component symbol.

Right-click =F1-K3 component and click **Insert symbol** (1).

Click to Insert symbol from manufacturer part circuits.

Highlight Relay coil and OK.

Select circuit from component manufacturer part: K3										
Compone	ent circuits									
State	Description	Terminal number	Reference	Default symbol						
	Relay coil	A1, A2	LC1D1210B7	TR-EL053						
	NO power co	1/L1, 2/T1	LC1D1210B7	TR-EL035						
	NO power co	3/L2, 4/T2	LC1D1210B7	TR-EL035						
	NO power co	5/L3, 6/T3	LC1D1210B7	TR-EL035						
	NO Contact	13, 14	LC1D1210B7	TR-EL057						
	NO Contact	23, 24	LC1D1210B7	TR-EL057						
	Select Compone State	 Select circuit from cc Component circuits State Description Relay coll NO power co NO power co NO power co NO power co NO contact 	 Select circuit from component manufa Component circuits State Description Terminal number Relay coil A1, A2 NO power co 1/, 1, 2/T1 NO power co 3/, 2, 4/T2 NO power co 5/, 3, 6/T3 NO Contact 13, 14 NO Contact 23, 24 	 ✓ Select circuit from component manufacturer part: K2 Component circuits State Description Terminal number Reference Relay coil A1, A2 LC1D121087 NO power co 1/L1, 2/T1 LC1D121087 NO power co 3/L2, 4/T2 LC1D121087 NO power co 3/L3, 6/T3 LC1D121087 NO contact 13, 14 LC1D121087 NO Contact 23, 24 LC1D121087 						

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



14 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.

15 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.

16 Component part.

Double-click on relay coil symbol -K1 symbol and click on the

Manufacturer parts and circuits tab then click Search 🗡.

On the Filters tab click the **Remove filters** \mathbf{k} button.

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

button. Highlight the Telemecanique part LC1D1210B7 click

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

Double clicking on a symbol accesses the Component properties, any changes made to this component automatically affect and update all symbols associated to it regardless of where they are held within the project documents.

17 Close the project.

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

48

Note

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 10
- *Insert component symbol.* on page 45
- *Create symbol-component*. on page 39
- *Attribute visibility*. on page 34

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.

Тір

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 \square 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 <i>SLDPRT</i> files, 2D footprints, connection label, and PCB <i>EMN</i> 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.

arch /	Add manufacturer A part	dd electrical assembly Manufacturer part	Proper	ties Move up	Move down	it			
anufac	turer parts Manufacturer	Reference			Desci	iption (English)			_
	Schneider Electric	LADN01	1	FRONT CONT	ACTS BLOCK 1	NC			
rcuits State	Description	Terminal numbe	er Asso	ciated symbol	Reference	Group		 	

The different state colors are as follows:

Question

Note

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.

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. Circuit

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.

Matching scheme symbol and part circuits automatically associate to Association 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.

ircuits										
State	Descrip	tion Terminal r	number Associated	symbol Reference	e Group					
	Relay coil	A1, A2	05-2	LC1D1210						
	NO power co	ontact 1/L1, 2/T1	l	LC1D1210						
	NO power co	ontact 3/L2, 4/T2	2	LC1D1210						
	NO power co	ontact 5/L3, 6/T3	3	LC1D1210						
	NO Contact	13, 14	05-4	LC1D1210						
	NC power co	ontact N 21, 22		LC1D1210	37					
	NC Contact	N.	05-3							
	NO Contact			Circuit association	on		х			
				Circuit	types are d	ifferent.				
					want to co	ntinue?				
				20,00						
				Circuit t	/pes: "NC Cont	act" and "NC	C power contact".			
				A 14						
				🤝 Y e	es					
				N	0					
	Circuits		l					1		
	State	Description	Terminal number	Associated symbol	Reference	Group				
		Relay coil	A1, A2	05-2	LC1D1210B7					
	- i i i i i i i i i i i i i i i i i i i	NO power contact	1/L1, 2/T1		LC1D1210B7				24	
		NO power contact	3/L2, 4/T2		LC1D1210B7					
	1 0	NO power contact	5/L3, 6/T3		LC1D1210B7		+L1-	+L2-K1	Ļ	
	1 👼	NO Contact	13, 14	05-4	LC1D1210B7			05.2	1	
	i iii	NC power contact	21, 22	05-5	LC1D1210B7			03-2		
	ē	NO Contact							•	
									22	

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

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

Tip

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 <i>Unarchiving a Project</i> on page 10.
Procedure	Employ the parts manager to obtain manufacturer data on line, learn to apply parts to symbols using filters and modify parts <i>on-the-fly</i> 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 🗦 እ
Тір	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.

😸 Unarchiv	ving: Manufacturer pa	arts				
Welcome	Selection	🛛 🔀 🏂	Finish			
Librari	ies					
Change Color co	action using pop-u de: Red - supplied	p menu on column "A I with the software / Gi	Action" on an item selection. reen - created by the user / Blue - si	upplied with the soft	ware and modified b	ov the user
						,
Icon	Action	Name	Description	New name	Modification date	Creator
Icon	Action	Name ECP	Description ECP	New name	Modification date 02/06/2015	Creator
Icon	Action Add Add	Name ECP TESYS	Description ECP	New name	Modification date 02/06/2015 13/05/2016	Creator System System
Icon	Action Add Add Add	Name ECP TESYS Todo	Description ECP To DO	New name	Modification date 02/06/2015 13/05/2016 02/06/2015	Creator System System System
Icon	Action Add Add Add Replace	Name ECP TESYS Todo 2D_PART	Description ECP To DO Footprints corresponding to specific parts	New name	Modification date 02/06/2015 13/05/2016 02/06/2015 15/12/2013	Creator System System System System
	Action Add • Add • Add • Replace • Do nothing •	Name ECP TESYS Todo 2D_PART IEC	Description ECP To DO Footprints corresponding to specific parts IEC 617 standard symbols	New name	Modification date 02/06/2015 13/05/2016 02/06/2015 15/12/2013 01/09/2014	Creator System System System System System

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**.

🚮 Add	l virtual ci	rcuits to compone
4	X	S
Add	Delete r	Initialize from nanufacturer part
	Act	tion

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

Origin componen								
	💕 K1 - Contactor							
-		←	Circuit		-			
1	14		NO Contact	13	1			
2	A2		Relay coil	A1	2			
0			NO power contact		0			
0			NO power contact		0			
0			NO power contact		0			

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 relates 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.

🔏 Class s	elector		\times
🕀 🚽 🖶	oxes, cabinets		
Br	akes, electrovalves		
🖶 🚽 Bu	uilding		
🖶 🚽 Bu	uttons, switches		
- T	Emergency		
	Push buttons		
÷	Switches		
C	apacitors		
🗄 🚽 Ci	rcuit-breakers		
	Select	Can	cel

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.

∎ Туре

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.

PLC components have their own unique set of types.

Note

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** sin on the relay -K1.



Δ1

-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 **6** on the Manufacturer part and circuits tab click Search reactions Manufacturers part selection.

7 Filter parts.

On the Filters tab use the following criteria.

🔑 Manufacturer part selec	tion									_		Х
🔏 Classification 🌱 Filt	ters		d.							English		~
🔀 Remo	ove filters	Ę	e Sear	n	Au	tomatic refresh						
□ ♥ Classification		~	Number	Reference		Description (En	Manufacturer	Class	3	Туре		G ^
	C Any class		1 1	15030		1 N/O 20A 230V AC	Schneider Electric	Contactor re	lays,	Base	3	
In the class	Same class		2 1	15031		1 N/O 20A 230V A	Schneider Electric	Contactor re	lays,	Base	3	
	C Same base class		3 1	15185			Schneider Electric	Contactor re	lays,	Base	3	
Class:	Contactor relays, rel		4 1	15186			Schneider Electric	Contactor re	lays,	Base	3	
🗉 🌱 General			5 1	15231		MINe timer (delay	Schneider Electric	Contactor re	ays,	Base	3	
Eibrary:	<all -<="" libr="" project="" th="" the=""><td></td><td>6 1</td><td>15232</td><td></td><td></td><td>Schneider Electric</td><td>Contactor re</td><td>ays,</td><td>Base</td><td>3</td><td></td></all>		6 1	15232			Schneider Electric	Contactor re	ays,	Base	3	
Туре:	Base •	1		15233			Schneider Electric	Contactor re	ays,	Base	3	
Mark root:	<all></all>		8	15315			Schneider Electric	Contactor re	ays,	Base	5	
- Manufacturer part			9 I	15310			Schneider Electric	Contactor re	dave.	Baco	5	~
Manufacturer:	Schneider Electric 🔻	<										>
A Reference:		15										
Series:						ΨT						
Article number:			Manufacture	r		Reference		Descri	State	Description	Terminal	number
Description (English):										NO Contact	13, 14	
Commercial reference (E										Relay coil	A1, A2	
Supplier										NO power	,	
Creation date										NO power	1	
+ Modification date										NO power	1	
- Circuit												
A Number of circuits:	<all></all>											
Number of terminals	<all></all>		_									
		× (>	<			>
Number of manufacturer parts	s found: 25118								Select		Close	

Click **Search** 2 to update the listed parts.

Тір

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**.

🔑 Manufacturer part selection									×
🔏 Classification 🌱 Filters		\rightarrow					English		\sim
Remove filters		Cearch Search	Au	tomatic refresh					
	A N	umber Referen	ce 🔺	Description (En	Manufacturer	Class	Ту	pe	Circui
Classification C Any class In the class C Same class C Same ba C Same b	s ss class relays, rel oject libr • * * * * * * * * * * * * *	GC6340M5 GC6340M5 GC6340M5 GC6340M5 GC6340M5 GC6340M5 GC6340M5 GC6340M5	5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5	Modular cont. 4N	Schneider Electric Schneider Electric Schneider Electric Schneider Electric Schneider Electric Schneider Electric Schneider Electric Schneider Electric	Contactor rela Contactor rela Contactor rela Contactor rela Contactor rela Contactor rela Contactor rela Contactor rela Contactor rela	ys, Base ys, Base ys, Base ys, Base ys, Base ys, Base ys, Base ys, Base ys, Base ys, Base	5 5 5 5 5 5 5 5 5 5 5 5	
Manufacturer: Schneider Reference: GC6340MS Series:	Electric •	÷ + E		*					
Article number: Description (English): Commercial reference (E		Manufacturer Schneider Electric	GC6340N	Reference M527		Descri	State Descript NO powe NO powe	ion Termina r 1, 2 r 3, 4	l number
Supplier Creation date Modification date Circuit Sumber of circuits: <all></all>	•						NO powe NO powe Relay coi	r 5, 6 r 7, 8 A 1, A 2 act 13, 14	
Number of terminals <all></all>	× v <					>	Select	Close	>

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 $+ \blacksquare$ 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.

Search Add manufacturer Ad	delectrical assembly Manufacturer par	Properties t	↑ ↓ Move Mov up dow	circuit						
Manufacturer parts Manufacturer Schneider Electric	Reference		tion Schoolder	Description	(English)					
Circu site	Properties Circuits Add m	Circuits,	terminals	Add	💰 Inse	rt	× Dele	te		
State Description Terr NO power con 1, 2 NO power con 1, 2 NO power con 3, 6 NO power con 5, 6 NO power con 7, 8 Relay coil A1, A NO contact 13, 1 NO NO NO	Number 0 1 2 3 4 4 <	Relay coil NO power cont NO power cont NO power cont NO power cont	Type tact tact tact tact		▼ A1,A2 ▼ 1,2 ▼ 3,4 ▼ 5,6 ▼ 7,8	nal marks	Group 	Sym	ibol >	
	Terminals	Add	•	nsert	X Dele	te				
	Circuit 0 0	Index 0 1	Mark A1 A2	Orientation ···· Undefined	<pre> Termination </pre> <pre> <pre></pre></pre>	Max. wire n 99 99	Mnemonic 	Maximum w 0	Minimum wi 0 0	1 1
	<								>	

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.

Manufact	urer part properties	Schneider Electric : GC634	DM527					>
💡 Propertie	es 🚜 Circuits, terr	ninals						
ircuits								
→ Add	multiple	🖧 Add	🕰 Insert	🗙 Delete				
Number		Туре	Terminal marks	Group		Symbol		
	Relay coil		▼ A1,A2					Т
	NO power contact		 ▼ 1,2 ▼ 3,4 		4	Add	l	٦.
	NO power contact		▼ 5,6		4	Insert		
	NO power contact		▼ 7,8		×	Delete	Del	
					۲	Assign syml	ool 📐	
					×	Remove syn	nbol	
£					-		>	

Right-click **Symbol** column and click **Assign symbol** (26) 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



💕 Properti	es 🔥 Circuits, terminals				
Circuits					
Je Standard Add	multiple	idd 🖧 Insert	🗙 Delete		
Number	Туре	Terminal marks	Group	Symbol	
0	Relay coil	▼ A1,A2			
1	NO power contact	▼ 1,2	(1)	TR-EL035	
2	NO power contact	▼ 3,4	(1)	TR-EL035	
3	NO power contact	▼ 5,6	(1)	TR-EL035	
	NO	-17.0			

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.

Circuits								
State	Description	Terminal number	Associated symbol	Reference	e Group			
	Relay coil	A1, A2	05-2	LC1D1210B	37			
	NO power contact	1/L1, 2/T1		LC1D1210B	37			
	NO power contact	3/L2, 4/T2		LC1D1210B				
	NO power contact	5/L3, 6/T3		LC1D1210B				
	NO Contact	13, 14	05-4	Circuit as	sociation		x	
	NC power contact	21, 22						-
	NG Contact	1	05-3		Circuit types	are diffe	rent.	
	NO Contact			I 🙂 г		to contin	0.162	
				۲ I	you want	to contin	lide.	
				c	ircuit types: "N	C Contact"	and "NC power contact".	
					A. 17			
	- Circuits				🔻 Yes			
	State Des	cription Termin	al number Associat					
	Belay co	sil Δ1 Δ2	05-2		🔿 No			
		ercontact 1/112	VT1					
	NO pow	er contact 3/12/4	/T2		C1D121087			_21
	NO pow	er contact 5/1.3 6	/T3	1	C1D1210B7		⊥l 1 ⊥l 2-k	a L
	NO Cont	act 13.14	05-4	1	C1D1210B7			5 T
	NC powe	er contact 21.22	05-5	L.	C1D1210B7		05-	-2
	NO Cont	act .						
							Ī	
	[

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.

Ci	ircuits –					
	State	Description	Terminal number	Associated	Reference	Group
		Relay coil	A1, A2		GC6340M527	
		NO power contact	1, 2		GC6340M527	
		NO power contact	3, 4		GC6340M527	
	4	NO power contact	5, 6		GC6340M527	
		NO power contact	7, 8	05-2	GC6340M527	
		NO Contact	13, 14	05-4		

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

C	Circuits					
	State	Description	Terminal number	Associated symbol	Reference	Group
		Relay coil	A1, A2	05-2	GC6340M527	
		NO power contact	1, 2		GC6340M527	
		NO power contact	3, 4		GC6340M527	
		NO power contact	5, 6		GC6340M527	
		NO power contact	7, 8		GC6340M527	
		NO Contact	13, 14	05-4		

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.

🖋 Sele	ct circuit from co	omponent manufa	acturer part: K	I		×
Compon	ent circuits				Symbol from manufacturer part	
State	Description	Terminal number	Reference	Default symbol	#TAG	#P TAKP TAKP TAG 2
	Relay coil	A1, A2	GC6340M527		#RFF MAN	
	NO power co	1, 2	GC6340M527	TR-EL035	#DEE DEE	
	NO power co	3, 4	GC6340M527	TR-EL035	#REF_REF=	-7 - 7 - 7 - 4
	NO power co	5, 6	GC6340M527	TR-EL035	#ID_1	
	NO power co	7, 8	GC6340M527		#REF_ART_NUM	#P_T #49_5#49_4 AG_3#

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

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



Electrical Assemblies

8 8 8

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 electrical assembly will be applied.

Blectrical a	issembly	properties											
6 Propertie	s 📳 l	User data	A Manufac	turer parts									
PI	Add		💥 Dele	te									
Manufacturer	Refere	nce	Description (E	English)	d	assification	Туре						
lussmann Iussmann Iussmann	1DP89 SC-60 SC-60	3-Pole Industrial Fuse Block Time Delay / Slow Blow Class Time Delay / Slow Blow Class		: Block w Class G Fu w Class G Fu	Fuses, Disconnectors Base Fuse Fuses, Disconnectors Acces Fuse Fuses, Disconnectors Acces		Base Accessory Accessory						
ssmann	SC-60	Number	Reference		Description	(English)	Manufacturer	Clas	s	Туре	Cire	cuit	
			✓ 1 S	SC Class 3P I	Fuse 3 Po	ole Fuse aci	tessories E	Bussmann	Black boxes	5	Electrical assembly	0	>
		1	+		+	1							
		Manu	facturer	Refere	ence	De	scription (English)		State	Description	Terminal number	r Associa	
		⊡	issmann	SC Class 3	P Fuse	3 Pole Fuse ac	ccessories		4	Fuse, Disconnector	*,*		
		- 4	Bussmann	1DP89		3-Pole Indust	rial Fuse Block		1	Fuse, Disconnector	*,*		
		- 4	Bussmann	SC-60		Time Delay / S	Slow Blow Class	G Fuse	4	Fuse, Disconnector	*,*		
		- 4	Bussmann	SC-60		Time Delay / S	Slow Blow Class	G Fuse					
		- 14	Bussmann	SC-60		Time Delay / S	Slow Blow Class	G Fuse					

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 LessonO5 Case Study. For more information, see <i>Unarchive parts</i> . on page 57.
Procedure	Employ the parts manager to create an electrical assembly and apply it to a project component.
Where to Find It	CommandManager: Library > Manufacturer parts >> >
	 Add manufacturers part > Add Electrical Assembly Shortcut Menu: Right-click a component or symbol and click Symbol or Component Properties Manufacturer part and circuits > Add Electrical Assembly
	15 Locate symbol. With drawing 04-Power open Zoom into fuse -F2.
	 16 Create electrical assembly. Right-click on -F2 and click Symbol properties, then click Manufacturer parts and circuits tab and finally click Add electrical ecoembly.

Add electrical assembly 🦄.

17 Assign accessory parts.

Click Add \nearrow to define the parts that make up the electrical assembly.

Search and Add the following parts and click Select.

🖋 Manufacturer part selecti	on								— 🗆	\times
🔏 Classification 🌱 Filte	rs							Englis	h	~
Remo	ove filters	<u>ا</u>	Ce Search	Automat	ic refresh					
Classification	1	Num	ber Reference	e 🔺 Descript	ion (English)	Manufacturer	Clas	s	Type	
In the class	 O Any class ○ Same class ○ Same base class 	1 1 1 1 2 1 3	1DP89 KTU-700 SC-60	3-Pole Indus FUSE - CLAS Time Delay /	trial Fuse Block S L Slow Blow Cl	Bussmann Bussmann Bussmann	Fuses, Disc Fuses, Disc Fuses, Disc	onnect onnect onnect	Base Base Accessory	
Class:	Fuses, Disconnectors\Fu									
General										
🗧 Library:	<all></all>									
Туре:	<all></all>									
Mark root:	<all></all>									
😑 🌱 Manufacturer part		<								>
🗡 Manufacturer:	Bussmann 🔹	1			-					
Series:		1	Manufacturer	Reference	1	Description (English)		State	Description	Termi
Article number:		- r Z	Bussmann	1DP89	3-Pole Indu	strial Fuse Block		-		_
Description (English):			Bussmann	SC-60	Time Delay	/ Slow Blow Class	G Fuse			
Commercial reference (En			Bussmann	SC-60	Time Delay	/ Slow Blow Class	G Fuse			
Supplier			Bussmann	SC-60	Time Delay	/ Slow Blow Class	G Fuse			
Creation date										
Modification date										
Circuit										
🖧 Number of circuits:	<all></all>									
Number of terminals:	<all></all>							<		>
				1				-		
Number of manufacturer parts	found: 3						Se	lect	Close	

18 Electrical Assembly data.

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

2	Electrical assembly properties		_		×
e	Properties 🎽 Manufacturer parts				
	General				^
	🖋 Reference:	SC Class 3P Fuse parts			
	🖋 Manufacturer:	Bussmann			-
	Class:	Fuses, Disconnectors\Fuses			
	Туре:	Electrical assembly			
	Article number:				
	External ID:				
	Series:				
	Data sheet:			0	pen
Đ	Supplier				
Đ	Illustration				
Đ	Size				
Đ	Use				
Đ	Control				
Đ	Manufacturer data				
	Manufacturer part				
Ξ	Description				_
	secription (English):	3 Pole Fuse Accessories			
	📰 Commercial reference (English):				
Θ	User data				
	User data 1:				
	User data 2:				
	Translatable data				~
S	ıpplier				~
					~
	<< Previous Next >>	[OK	Can	cel

Click OK.

19 Add Electrical Assembly to catalog.

Once the assembly is created a prompt will be displayed allowing for it to be applied to the current component only, or be added to the catalog for use in all projects. Click **Yes** to add the part to the catalog.



20 Close the project.

With the assembly applied to the component right-click project name in the Documents side panel and click **Close** \Box .

Component properties									_		×
💕 Mark and data 🥓 Man	ufacturer part and	d circuits									
۶ 🖌	I'FY	× 6	x	1		+	×			\uparrow	1
Search Add manufacturer part	Add electrical assembly	Delete Prop	erties Move up	Move down	Edit terminals	Add virtua circuits	Delete virtual circuit	Switch symb circuit	ol Dissociate symbol circuit	Up	Down
	Manufactur	er part						Circuit			
Manufacturer parts Manufacturer	Refe	rence			Description (English)					
Bussmann	SC Class 3P F	use parts	3 Pole Fuse A	ccessories							
	1DP89		3-Pole Indust	rial Fuse B	lock						
- 🔽 Bussmann	SC-60		Time Delay /	Slow Blow	Class G Fue	se					
🔽 Bussmann	SC-60		Time Delay /	Slow Blow	Class G Fu	se					
	SC-60		Time Delay /	Slow Blow	Class G Fue	se					
Circuits											
State Description 1	Terminal n As	sociated	Reference	Grou	Þ						
Fuse, Disconn*,	• 04-	6 SC	-60								
Fuse, Disconn*,	• 04-	6 SC	-60								
Fuse, Disconn*,	• 04-	6 SC	-60								
						_					
							🔺 Customize		ОК	Car	ncel

Exercise 3: Manufacturers Parts

Procedure

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 57
- *Unarchive wizard*. on page 58
- *Find a component*. on page 40
- Finding Manufacturer Parts on page 59
- *Filter parts*. on page 62
- *Refined filter.* on page 63
- *Symbol circuit association*. on page 68

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 $% \mathcal{A} = \mathcal{A}$

manufacturers parts j.

Browse to the LessonO5\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** \blacksquare command to locate component K1.

M Search component: St	art Exercise 05							×
Mark Mark	к1		the Description		🗇 Master			
- Location]		🐀 Function		 Destination			
🖧 Class]		🖋 Manufacturer part		Component type	<all></all>		•
			Fil	ters				
						*	* (\$
Start Exercise 05 Start Exercise 05 Start Exercise 05 Start Exercise 05 Start Exercise 04 Start Exercise 044 Start Exercise 044	; 5 - NO Contact 10 - NO Contact 3 - Relay coil 5 - NC Contact 5 - NC Contact 5 - 3 poles NO power cont	act			 		. Z	046
							Close	

9 Component properties.

Right-click the component and open its **Properties 6**.

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 first listed 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.

Circuits				
State	Description	Terminal number	Associated symbol Gro	quo
	Relay coil	A1, A2	04-3	
	NO power contact	1, 2	03-5	
	NO power contact	3, 4	03-5	
	NO power contact	5, 6	03-5	
	NO Contact	53, 54	04-6	
	NC Contact	61, 62	05-5	
4	NO power contact	X1, X2		
	NO Contact		04-10	

11 Force circuit association.

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

r	Circuits				
	State	Description	Terminal number	Associated symbol	Group
		Relay coil	A1, A2	04-3	
		NO power contact	1, 2	03-5	
		NO power contact	3, 4	03-5	
		NO power contact	5, 6	03-5	
		NO Contact	53, 54	04-6	
		NC Contact	61, 62	05-5	
		NO power contact	X1, X2	04-10	

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 \square 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_LessonO4.proj from the folder LessonO4\Case Study.
Procedure	Utilize different tools to insert and create associations between

± 🕞 +L1

034486

+L1+L1 +L1+L2

=F1-F1 =F1-K1 =F1-K2

=F1-Q1

=F1-Q3 =F1-T1

=F1-X1 =F1-X2

C7K12015M7 =F1-N1

🗸 🤹 036200

036200 <u>ه</u> ک 036200

🔽 🤹 036200 🔽 🥨 034486 034486

<u>ن</u>

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 2.

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.



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

4

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 LessonO4\ 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** \gg .

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 5**.

Question

Why is everything transparent?

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



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.								
om fy								
^								
*								

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

		neeren innee		
Pool				
		_	-	0000
	 antin and	a inclusion par		

Click **Cancel** to end the command.

8 Insert a component.

Expand component =F1-Q1 and right-click on part OO6557 then click **Insert** \gg .

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 LessonO4\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.					
Тір	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 1**.

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** \square 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 82
- Associate component part. on page 80

To start the exercise unarchive and open the file

Using an Archive File

Procedure

Start_ExerciseO4.proj from the folder LessonO4\Exercises. Insert a component and associate a component to an existing

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_LessonO5.proj from the folder LessonO5\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].

2 Route wires.

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

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



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

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

Tip

3 Results without Paths.

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



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 FeatureManager design tree scroll to the bottom of the list. Right-click EWS [~ 24V _ Control]20 and click **Delete** × and then **Yes** to remove the sketched wires.

5 Sketches.

Ensure that **View sketches** \square 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 93).



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.

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.



Тір

10 Exit Hide Sketch.

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

Selecting **View sketches** ^[C] 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 <i>SOLIDWORKS Route</i> on page 96 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 96.

?

 $\overline{}$

 $\overline{}$

 $\overline{}$

~

 $\overline{}$

 $\overline{}$

0

ĉ

ĉ

Draw Graph Delete Graph

*→

150

Use first axis connection 0.50mm

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 T 					



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] EWS [N L1 L2 L3 Phase 2] EWS [N L1 L2 L3 _ Phase 3] EWS [N L1 L2 L3 _ Neutral wire] EWS [N L1 L2 L3 _ Protection] EWS [~ 24V _ Control] 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 95) will map to a route sub-assembly that includes a cable part.



13 Delete sketches.

Click the FeatureManager, scroll to the bottom of the list and multi select the following 3D sketches.

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

Right-click on them and click **Delete** and **Yes to all** when promoted to remove the sketched lines.

14 SOLIDWORKS route.

Click Route wires \mathbf{X} and the options SOLIDWORKS Route, Use Splines and All components. Click OK.



Note

Where to Find It

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.

CommandManager: SOLIDWORKS Electrical 3D >
 Segregation ()

15 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**.



16 Route Wires.

Click **Route wires t** 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.



17 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 6** .

Manufacturer part properties Legrand : 036200		— 🗆 X
😚 Properties 💰 Circuits, terminals		
Series:		,
Data sheet:		Open
Supplier		
Supplier name:		*
Stock number:		
Information		
🖧 Number of circuits:	0	
Number of terminals:	0	
Illustration		
🔁 Line diagram:		
Scheme:		
🤏 3D part:		
O 2D footprint:		
Connection label:		
I Printed circuit board file:		
Size		
Width:	25 mm	
Height:	25 mm	
Depth:	2000 mm	
Adjustable:	V	
Length:	500 mm	
Duct filling ratio (%):	11	
🗉 Use		

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

Click OK.

18 Report lengths.

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

The wire lengths are listed by wire style.

📰 Rep	oort manager: Start Lesson 05											-		×
Add	Delete Properties Schedule U	Ç Jpdate	Ac	dd ter Filte	Edit filter	Generate drawings	Exce	I Txt rt export	XML XML export	Parameter				
Order	Description			→		Origin		Destin	ation	Wire numb	er Section	Lengt	h (mm)	^
R 1	PLC Inputs / Outputs list	< <u>N</u>	22		=F1+L	1+L1-H2:X1		=F1+L1+L2	2-K2:	10	2.1 (mm²)	1763.88		
2	List of the cables	<n< th=""><th>31</th><th></th><th>=F1+L</th><th>1+L1-H2:X2</th><th></th><th>=F1+L1+L1</th><th>1-H1</th><th>N-8</th><th>2.1 (mm²)</th><th>318.46</th><th></th><th></th></n<>	31		=F1+L	1+L1-H2:X2		=F1+L1+L1	1-H1	N-8	2.1 (mm²)	318.46		
✓ ✓ Bill Of Materials grouped by manuf		. <n< th=""><th>13</th><th></th><th>=F1+L</th><th>1+L1-S1:1</th><th></th><th>=F1+L1+L2</th><th>2-K1:</th><th>1</th><th>2.1 (mm²)</th><th>1458.84</th><th></th><th></th></n<>	13		=F1+L	1+L1-S1:1		=F1+L1+L2	2-K1:	1	2.1 (mm ²)	1458.84		
=4	List of wires by line style	<n< th=""><th>10</th><th></th><th>=F1+L</th><th>1+L1-S1:1</th><th></th><th>=F1+L1+L2</th><th>2-X1</th><th>1</th><th>2.1 (mm²)</th><th>910.82</th><th></th><th></th></n<>	10		=F1+L	1+L1-S1:1		=F1+L1+L2	2-X1	1	2.1 (mm ²)	910.82		
25	Drawings list	<n< th=""><th>5</th><th></th><th>=F1+L</th><th>1+L1-S1:2</th><th></th><th>F1+L1+L1</th><th>1-52:3</th><th>2</th><th>2.1 (mm²)</th><th>295.45</th><th></th><th></th></n<>	5		=F1+L	1+L1-S1:2		F1+L1+L1	1-52:3	2	2.1 (mm ²)	295.45		
		li	11		=F1+L	1+L1-S2:3		=F1+L1+L2	2-K1:	2	2.1 (mm ²)	1438.07		
			12		=F1+L	1+L1-S2:4		=F1+L1+L2	2-X1	3	2.1 (mm²)	950.32		
		li	6		=F1+L	1+L1-S3:14		=F1+L1+L1	1-S4:	1	2.1 (mm ²)	254.96		
<			<i>,</i> [^]		-							1000.00		<u> </u>
			`											-
														Ŷ
													Close	

Note

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

19 Close the project.

On the **Electrical Project Document** panel right-click the project name and click **Close project** \square 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 91
- *Route Wires* on page 93

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** \square selecting to **Save** when prompted.