



AUTOMATION

PART 1

Version 12.0

TABLE OF CONTENTS

PART 1

PREFACE	7
UNICODE	10
SURVEY	13
QUICK SURVEY OF THE MANUAL	15
GETTING STARTED	19
WINDOWS	21
KEYBOARD SHORT-CUTS	27
THE ON-LINE MANUAL	32
THE WORK AREA	35
SCREEN / IMAGE FUNCTIONS	38
BASIC DRAWING FACILITIES	45
ON DRAWING OBJECTS	47
LINES	65
ARCS/CIRCLES	97
TEXTS	99
SYMBOLS	133
FUNCTIONS ON COMPONENT LEVEL	220
AREAS	232
SUB DRAWINGS	244
RENAME SYMBOLS	278
INSERTING IMAGES	286
SURVEY AND REFERENCE DESIGNATIONS	289
THE EXPLORER WINDOW: THE PROJECTS TAB	291
THE OBJECT LISTER	296
FIND AND REPLACE	306
DESIGN CHECK	310
PRINTING	314
WINDOWS EXPLORER AS SYMBOL/PROJECT MENU	321
REFERENCE DESIGNATIONS	322
REFERENCES	353
AUTOMATICALLY UPDATED REFERENCES	355

CONTACT MIRRORS AND REFERENCE CROSSES	362
CURRENT PATH NUMBERS / REFERENCE SYSTEM.....	367
WIRE NUMBERS	375
AUTOMATIC WIRE NUMBERING	377
MANUAL PLACING OF WIRE NUMBERS.....	386
WIRE NUMBER FILES.....	389
SINGLE LINE DIAGRAMS	395
SINGLE LINE DIAGRAMS.....	397
AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS	399

PART 2

PLC FUNCTIONS.....	433
SURVEY OF PLC FUNCTIONS	435
WORKING WITH PLC'S	436
READING PLC I/O LISTS	449
PLC I/O ADDRESSING VIA EXCEL	452
CONNECTION SYMBOL FOR SIGNAL BUSES.....	454
INSTALLATION DRAWING.....	457
CREATING A GROUND PLAN.....	459
LAYERS, ELEVATION AND SCALING	465
TIPS FOR INSTALLATION DRAWING	471
THE PANEL BUILDER.....	473
WORKING WITH PROJECTS	481
ON PROJECTS	483
PAGES AND CHAPTERS.....	485
PROJECT TEMPLATES AND STANDARD PROJECTS	504
COPYING FULL PROJECTS INTO OTHER PROJECTS	506
SEVERAL PROJECTS SIMULTANEOUSLY	509
INSERTING LOGOS IN PROJECTS	517
LISTS.....	519
INSERT DRAWING HEADERS/FORMULAS	521
LIST SETTINGS	532
COMPONENT GROUPING	546
INSERTING UNIT DRAWINGS	558
DATABASES.....	559
USING THE DATABASE	561
CREATING COMPONENTS IN THE DATABASE.....	579
DATABASE SETTINGS.....	598
CHOOSING A DATABASE.....	613
PROGRAMMABLE BARCODE SCANNERS	615
MECHANICAL LOAD FROM THE DATABASE.....	616
FROM MECHANICAL TO ELECTRICAL DRAWING.....	625
EXPORT / IMPORT.....	627
EXPORTING LIST FILES.....	629
DWG AND DXF FILES	640
INTEGRATION WITH MICROSOFT EXCEL	646
INSERTING FILES FROM OTHER PROGRAMS / OLE	647
PLACING SYMBOLS VIA LISTS.....	654

OPENING FILES FROM THE TOOLS MENU.....	659
PC SCHEMATIC VIEWER.....	660
CREATE SYMBOLS, LISTS AND PROJECTS	663
PLANNING AHEAD.....	665
PAGE DATA AND PROJECT DATA DATAFIELDS	669
CREATING SYMBOLS.....	676
CREATING DATA SYMBOLS.....	711
CREATING SIGNAL SYMBOLS.....	717
CREATING DRAWING HEADERS	721
CREATING LISTS	732
CREATING UNIT DRAWINGS.....	748
CREATING TEMPLATES.....	752
UPDATING, MODULES AND LICENSES	761
MODULES	763
UPDATE AND AUTOMATIC INFORMATION.....	765
ADAPTING THE SYSTEM TO YOUR NEEDS.....	767
SHORT-CUT KEYS	769
COLORS.....	773
TANKING LICENSES.....	775
PREVENTING REPETITIVE STRAIN INJURIES.....	782
ACCESS CODES, REGISTRATION ETC.	785
PASSWORDS ON PROJECTS.....	787
LOCKING PROJECT FILES	788
REVISION CONTROL	790
REGISTRATION OF PC SCHEMATIC FILE TYPES	795
INDEXING.....	798
STARTING FUNCTIONS, BATCH PRINTING AND DDE.....	802
COMPRESSED SAVING / ZIPPING PROJECTS	806
APPENDICES	807
SETTINGS	809
DATAFIELDS	834
THE DATABASE FIELD PCSTYPE.....	843
THE DATABASE FIELD PINDATA.....	845
THE DATABASE FIELD PINDATA: COUNTING	846
DICTIONARY.....	850

PREFACE

Automation is a CAD-program for documenting PLC, wiring and power control circuit diagrams, and can furthermore be used for any kind of electrical installations.

The program is developed by the Danish software house PC|SCHEMATIC A/S.

In the same family of CAD programs you will find:

- PC|SCHEMATIC **PowerDistribution** for registration and survey of LV installations, for e.g. operations and maintenance.
- PC|SCHEMATIC **Tele** for registration and survey of tele installations, for e.g. documentation, maintenance and operations.
- PC|SCHEMATIC **Loop GI** for automatic generation of loop diagrams based on information from a database or for instance from Excel

PC|SCHEMATIC A/S also develop software for interfaces, as well as user-defined modules and applications, for the PC|SCHEMATIC programs.

Furthermore you can use **Automation** as a general CAD drawing tool. The program uses PC|SCHEMATIC's own file formats, but supports the CAD file formats DWG and DXF.



Automation RUNS UNDER WINDOWS VISTA

Automation IS "VISTA READY", AND FUNCTIONS CORRECTLY UNDER WINDOWS VISTA. QUITE A FEW CHANGES HAVE BEEN MADE TO ACCOMPLISH THIS. PLEASE NOTE THAT THE PROGRAM WILL BEHAVE SLIGHTLY DIFFERENT IN SOME SITUATIONS WHEN USING WINDOWS VISTA.

Special in version 12: Unicode

In **Automation** version 12 it is important to know that projects are saved in unicode. This has the consequence, that projects created in previous versions, now must be converted to version 12. Read more about this in "Unicode" on page 10.

Designing your own functions in Automation

You can now create your own functions in **Automation** by using the build-in interface for COM/OLE automation. The documentation for how to do this can be downloaded at www.pcschematic.com. You can also access the special forum for COM/OLE automation for the **Automation** program at the PC|SCHEMATIC userforum (direct address for the COM/OLE section at the forum: <http://forums.pcschematic.com/forums/59.aspx>).

Symbols and databases

All symbols in the program are created by PC|SCHEMATIC A/S, with license from Danish Standard / IEC. Where no overall standard exists, the symbols have been created in co-operation with either trade organisations, component vendors or **Automation** users.

Automation includes a database program, for which many vendors have created special databases. These can be loaded directly into the program. The extra advantage of these PC|SCHEMATIC databases, are that they, beyond the common database information — such as EAN number, description and prices —, contain electrical as well as mechanical symbols for the components. You also have the option to create your own component databases.

When buying the program, you will get access to component databases from several component vendors. These databases are created in Access format.

Furthermore, **Automation** can use databases, which are supported by ODBC, MDAC (Microsoft Data Access Components — standard on Windows 2000 and later) and BDE (or the so-called native drivers). In praxis this means that the program supports practically all database formats for Windows — for instance Oracle and Microsoft SQL Server.

Reducing Repetitive Strain Injuries

Along with the program comes the specially designed **Mouse Chasing System** for learning the program short-cut keys while working.

Support and Update

Support and updates are available, but the terms can differ from country to country. Please contact your local PC|SCHEMATIC agent for detailed information.

PC|SCHEMATIC A/S participates in a continuous dialog with the users of the program. The input from this dialog is the basis for the future development of the program.

OTHER MANUALS

For **Automation** the following additional manuals exist:

- The **Quick start** manual, which gives you a quick introduction to **Automation**.
- The **Information & Installation** manual, which gives you practical information as well as information on how to install the program. You also get information on how to install the license server software for multiuser network licenses.
- The **Project Generator** manual, which describes automatic generation of projects in **Automation**.
- The **Tools** manual, which describes how you can document symbols in projects or libraries automatically, how to set up parameters for DWG/DXF import/export, how to use the **PC|SCHEMATIC Translator** program, for translating project texts, as well as how you automatically create graphical **terminal plans**, graphical **cable plans** and graphical **connection plans**.
- The **Database** manual, which describes how to use the **PC|SCHEMATIC Database** program for creating and maintaining your database for **Automation**.

VIDEOS AND ARTICLES AT WWW.PCSHEMATIC.COM

At www.pcshchematic.com you can watch videos showing many of the available options in **Automation**.

At www.pcshcmeatic.com you can also a read survey article about the program, as well as articles about the basic drawing functions, PLC functions. lists and about working with projects.

PCISCHEMATIC FORUM AND FAQ

At www.pcschematic.com you also have access to the PC|SCHEMATIC forum, where you can exchange experiences with other PC|SCHEMATIC users.

UNICODE

Starting from version 12, **Automation** is now able to handle multiple regional character sets simultaneously. This is because the program now handles unicode.

WHAT IS UNICODE?

Up until now language settings of computers and programs have been more or less permanent. In Denmark, Germany, UK and other countries, our computers used a character set – WESTERN EUROPE/LATIN 1. This character set contains the letters and characters in western European alphabets, including Æ, Ø, Å, Ä, Ö and Ü. Other language areas have their own character sets, as for instance China, Japan and Russia.

Previously, it was only possible to work with one regional character set at a time. You could “cheat” the program to work with a different character set (like Russian) by changing some program settings — in **Automation** in **Settings => System**. This way it was possible — on for instance a English Windows installation — to work with a project, which was originally created using another character set.

Unicode, on the contrary, contains (almost) all existing characters — more than 100.000, each having its own meaning. "UNI CODE" = "one code".

When you have a program that supports Unicode, this program can show letters/characters from any alphabet in the same file. In **Automation** this means that when you have a project file, that will be used in Russia, this file can contain a Danish/English text and Russian characters.

This is because each character has its own unique code. However, the used font must contain all the needed characters. And this is not the case with all fonts.

The PC|SCHEMATIC font contains characters for Eastern and Western Europe and Russia.

SYSTEM SETTINGS FOR CONVERSION INTO UNICODE

The basic setting for conversion to Unicode-format, is made under **Settings => System**.

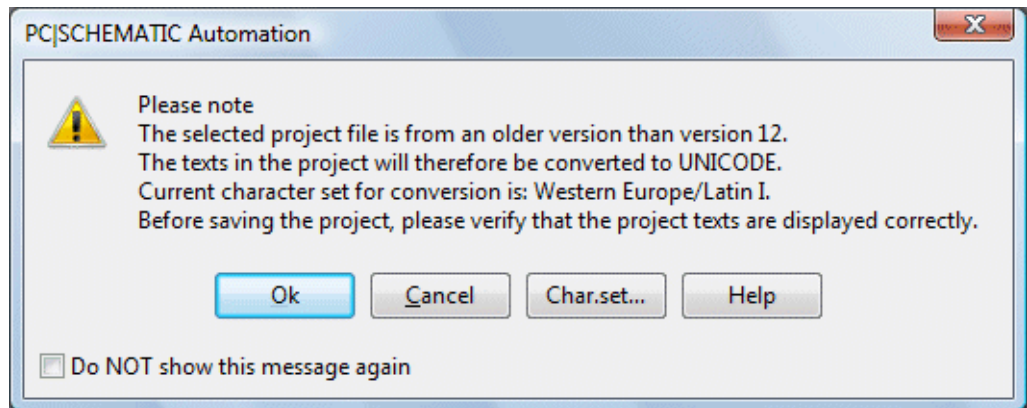
What happens during the conversion, is that all old files must be translated to the new format — that is converted from one character set to another. The program will guess the character set, as you can see in “Opening an existing project in version 12” on page 11.

THE PC|SCHEMATIC FONT AND UNICODE

The PC|SCHEMATIC font is not a Windows font and therefore it has limited Windows functionality. However, it conforms to the standards requirements concerning texts in technical documentation, and the font will be the standard font in most projects. It will also be converted to Unicode-format in all existing projects, that you continue to work with. New projects are saved in this format automatically.

OPENING AN EXISTING PROJECT IN VERSION 12

When you open a project created before version 12, the following dialog box appears:



You can choose not to display the dialog box again by selecting **Do NOT show this message again** at the bottom of the dialog box.

If you click the **Char. set** button, you will enter a dialog box, where you can see the recommended character set for the actual project. If this is not the correct character set, click **Select character set** where you can see all supported character sets.



IMPORTANT TO CHOOSE THE CORRECT CHARACTER SET

IT IS IMPORTANT, THAT YOU CHOOSE THE RIGHT CHARACTER SET FOR OLD FILES, AS YOU MIGHT RISK SAVING THE PROJECT IN A FORMAT THAT MAKES SHOWING OR RECOVERY OF THE ORIGINAL CHARACTERS IMPOSSIBLE.

THE PROGRAM CREATES BACKUPS OF OLD FILES IN THE OLD FORMAT, IF YOU CHOOSE TO. THOSE BACKUPS WILL BE SAVED AS FILENAME(ANSI).~PR/~SY. THIS MEANS, THAT IF YOU HAVE CHOSEN THE WRONG CODEPAGE FOR CONVERSION, YOU CAN GO BACK TO THE ORIGINAL FORMAT AND TRY AGAIN.

— IF YOU HAVEN'T CREATED PROJECTS INCLUDING "FOREIGN" ALPHABETS/ CHARACTERS, YOU WILL NOT EXPERIENCE ANY PROBLEMS, AND IT WILL BE SAFE FOR YOU TO TICK OFF THE DIALOG WHEN YOU SEE IT THE FIRST TIME.

The following character sets are supported:

- Central Europe
- Cyrillic
- Western Europe/Latin 1
- Greek
- Turkish
- Hebrew
- Arabic
- Baltic
- Vietnam
- Thai

- Japanese Shift-JIS
- Simplified Chinese GBK
- Korean
- Traditional Chinese Big5

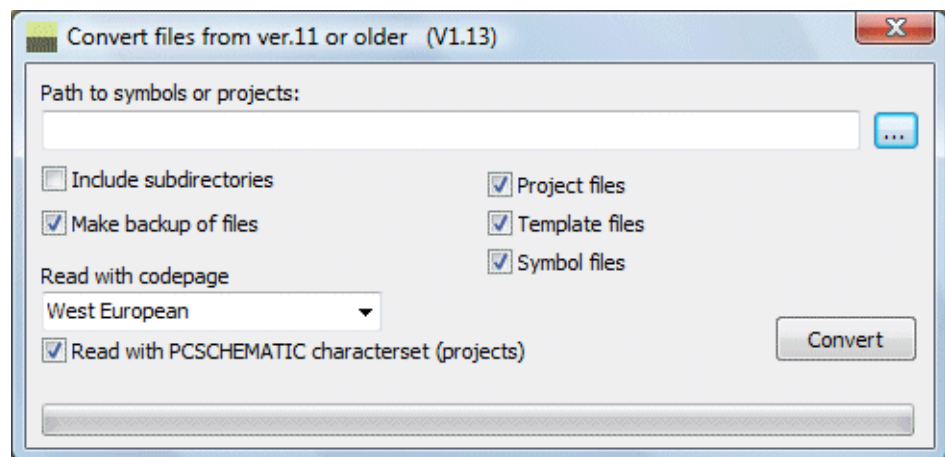
TEXTS IN SYMBOLS ARE CONVERTED AUTOMATICALLY

Conversion of old symbols function as as described above, but you will not enter the dialog when you load a new symbol. However, it is important that your settings are correct, and we recommend that you keep the setting for automatic backups in old version.

MODULE FOR CONVERTING PROJECTS TO UNICODE

You can also install a module to convert projects from versions prior to version 12:

- 1) Choose **File => Modules**
- 2) In the dialog box **Modules**: Click on **Convert files from ver. 11 or older** and click **Install**
- 3) In the dialog box **Modules**: Click **Close**
- 4) Close all active **Automation** projects
- 5) Choose **Tools => Convert files from ver. 11 or older**:



- 6) Select a folder and click **Convert** to convert all projects in the folder.



Survey

In this section you get a first impression of how to get started with the program.

To get the most of the manual, you should read this short section before going any further.

QUICK SURVEY OF THE MANUAL

To make the program as easy accessible as possible, the user interface for **Automation** have been developed as close to the Windows version of the Microsoft® Office-package as possible.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES THE CONTENTS OF THIS MANUAL.

If you are not familiar with Windows, you may read the brief introduction in the chapter “Windows” on page 21.

The manual itself is divided in sections. Below you can get a survey of the contents of these sections.

THE CONTENTS OF PART 1

SECTION	DESCRIPTION	SUBJECTS
"GETTING STARTED" (P. 19)	SCREEN FUNCTIONS IN THE PROGRAM ETC.	INTRODUCTION TO WINDOWS THE ONLINE MANUAL THE PROGRAM WORK AREA BASIC IMAGE FUNCTIONS
"BASIC DRAWING FACILITIES" (P. 45)	DETAILED DESCRIPTION OF THE BASIC DRAWING FACILITIES.	DRAWING DIAGRAMS / WORKING WITH LINES, SYMBOLS, TEXTS AND AREAS TRANSFERRING PROPERTIES BETWEEN OBJECTS ALIGNING AND TRIMMING FUNCTIONS AUTOMATIC LINE DRAWING / ROUTER MOUNTING CORRECT DRAWING TEXT LINKS FUNCTIONS ON COMPONENT LEVEL SUB DRAWINGS / DRAG'N DRAW RENAME SYMBOLS
"SURVEY AND REFERENCE DESIGNATIONS" (P. 289)	DESCRIPTION OF HOW TO FIND PAGES AND OBJECTS IN PROJECTS, PERFORM DESIGN CHECK AND PRINT. DESCRIPTION OF HOW TO USE REFERENCE DESIGNATIONS.	USING THE BUILD-IN EXPLORER USING THE OBJECT LISTER FIND AND REPLACE DESIGNCHECK PRINTING USING THE WINDOWS EXPLORER AS PROJECT/SYMBOL MENU REFERENCE DESIGNATIONS

SECTION	DESCRIPTION	SUBJECTS
"REFEREN- CES" (P. 353)	DESCRIPTION OF HOW TO USE REFERENCES IN THE PROGRAM.	AUTOMATICALLY UPDATED REFERENCES CONTACT MIRRORS REFERENCE CROSSES CURRENT PATH NUMBERS / REFERENCE SYSTEM
"WIRE NUM- BERS" (P. 375)	DESCRIPTION OF HOW TO USE WIRE NUMBERS.	AUTOMATIC WIRE NUMBERING MANUAL PLACING OF WIRE NUMBERS WIREDRAWER FILES

THE CONTENTS OF PART 2

SECTION	DESCRIPTION	SUBJECTS
"PLC FUNC- TIONS" (P. 433)	DESCRIPTION OF PLC FUNCTIONS.	SURVEY OF PLC FUNCTIONS WORKING WITH PLC'S SIGNAL BUSES READING PLC I/O FILES PLC I/O ADDRESSING VIA MICROSOFT EXCEL
"INSTALLA- TION DRA- WING" (P. 457)	DETAILED DESCRIPTION OF THE MORE DRAWING FACILITIES.	WORKING IN DIFFERENT LAYERS AND ELEVATIONS SCALING SINGLE LINE DIAGRAMS INSERTING IMAGES
"WORKING WITH PRO- JECTS" (P. 481)	THINGS YOU NEED TO KNOW WHEN YOU ARE CREATING PROJECTS.	ON PROJECTS WORKING WITH PAGES, PAGE TYPES AND PAGE FUNCTIONS PROJECT CHAPTERS PAGE AND PROJECT TEMPLATES TEMPLATE PROJECTS COPYING BETWEEN PROJECTS WORKING WITH MORE THAN ONE PROJECT OPEN STARTING ANOTHER PROJECT BY CLICKING
"LISTS" (P. 519)	HOW TO INSERT DRAWING HEADERS/LISTS.	INSERTING DRAWING HEADER/LIST LIST SETTINGS COMPONENT GROUPING UNIT DRAWINGS
"DATABASES" (P. 559)	HOW TO USE THE DATABASE IN YOUR PROJECTS.	USING AND CHOOSING A DATABASE GRAPHICAL CREATION OF COMPONENTS IN THE DATABASE PROGRAMMABLE BARCODE SCANNERS DATABASE SETTINGS REFERENCES FROM MECHANICAL TO ELECTRICAL PAGES MECHANICAL SYMBOLS FROM THE DATABASE
"EXPORT / IMPORT" (P. 627)	DESCRIPTION OF HOW TO IMPORT/EXPORT FILES.	LIST FILES DWG AND DXF FILES INTEGRATION WITH MICROSOFT EXCEL INSERTING OBJECTS FROM OTHER PROGRAMS / OLE OBJECTS PLACING SYMBOLS VIA LISTS PC SCHEMATIC VIEWER

SECTION	DESCRIPTION	SUBJECTS
"CREATE SYMBOLS, LISTS AND PROJECTS" (P. 663)	HOW TO ADAPT SYMBOLS, LISTS AND DATAFIELDS TO MEET YOUR SPECIFIC NEEDS, AND HOW TO CREATE PROJECTS THAT CAN BE USED AS TEMPLATES FOR OTHER PROJECTS.	CREATING STANDARD PROJECTS AND TEMPLATES SPECIFYING WHICH DATAFIELDS CAN BE USED IN PROJECTS DESIGNING SYMBOLS DESIGNING SIGNAL SYMBOLS DESIGNING DRAWING HEADERS/LISTS DESIGNING UNIT DRAWINGS
"UPDATING, MODULES AND LICENSES" . (P. 761)	INFORMATION ABOUT NETWORK SECURITY CHECKS, HOW TO INSTALL AND UPDATE ADDITIONAL PROGRAM MODULES, AND HOW TO ADAPT THE SYSTEM TO YOUR NEEDS.	PASSWORDS ON PROJECTS LOCKING PROJECT FILES INSTALLING MODULES REVISION MANAGEMENT ADAPTING THE SYSTEM SHORT-CUT KEYS SYSTEM COLORS REGISTERING PC SCHEMATIC FILE TYPES INDEXING DDE AND STARTING FUNCTIONS ZIPPING / COMPRESSED SAVING THE MOUSE CHASING SYSTEM
"APPENDICES" (P. 807)	FOR REFERENCE, AND FOR ADJUSTING THE PROGRAM SETTINGS.	PROGRAM SETTINGS DICTIONARY DATAFIELDS SPECIAL CHARACTERS IN THE PROGRAM INDEX

HOW TO GET STARTED

Start by installing the program, as described in "Installation" on page 27, or get the info from the **Information & Installation** manual.

Then proceed directly to the **Quick start** manual. Here you get a quick introduction to the program, and get ready to carry on with the detailed description of the program in "Basic Drawing facilities" on page 45.

Later on, you may return to the remaining part of "Getting started" on page 19, and get a better idea of the construction of the program.

You should also read "On projects" on page 483, which gives you a summary of the possibilities when you are working with projects in **Automation**.

— Use the chapter headings as well as the index, to receive information on the desired subjects as quickly as possible. The first chapter in some of the sections also contains a general overview of the section.

You can also use the program on-line manual, as described in "The On-line manual" on page 32.

Overall planning

In the section "Create Symbols, Lists and Projects" on page 663, you can get information on how to prepare your projects as easy as possible. In this section you design your own symbols and lists, and specify which datafields can be used in the project.

Appendix

In the last section of the manual, you can read about the various program settings, and how to adjust these to your needs. Here you will also find a dictionary, explaining commonly used terms in **Automation**, in Windows or in the manual itself. Finally, you can see the special characters, that can be applied in your project diagrams.

Finding your way around the projects

To find your way around the projects, it is recommended to read “Getting started” on page 19, as well as the chapter “Find and Replace” on page 306.

PC|SCHEMATIC website

For direct access to PC|SCHEMATIC’s homepage, choose **Help => Http://www.pcschematic.com**. Here you can for instance get information about the latest updates in the program.

Here you can also sign up for an e-mail service, which informs you when there are news on the **Automation** support page.

TYPOGRAPHY IN THE MANUAL

The following typographical elements are applied in the manual:

TYPOGRAPHY	USE
<i>ITALICS</i>	TEXTS WRITTEN IN <i>ITALICS</i> ARE TEXTS YOU HAVE TO TYPE IN YOURSELF.
BOLD	BOLD TEXTS — NOT INCLUDING HEADINGS — ARE USED FOR PROGRAM FUNCTIONS, BUTTONS AND MENUS.
BOLD AND ITALICS	ARE APPLIED FOR FIELD NAMES OR OPTIONS YOU CAN CLICK ON IN DIALOG BOXES.
KEYS <> []	THE CONTROL KEY IS WRITTEN AS <CTRL>, THE ESCAPE KEY AS <ESC>, THE ENTER KEY AS <ENTER>, THE DELETE KEY AS , WHILE THE SPACEBAR IS WRITTEN AS [SPACEBAR]. SINGLE LETTERS IN BRACKETS SUCH AS FOR INSTANCE [Z] SPECIFIES THAT THE "Z" KEY IS A SHORT-CUT KEY.
[CTRL+G]	IF IT SAYS [CTRL+G] THIS MEANS THAT YOU MUST PRESS DOWN THE "CONTROL" KEY AND THEN PRESS DOWN THE "G" KEY TO ACTIVATE THE SHORT-CUT.
MENUS =>	FILE => SAVE MEANS THAT YOU MUST CLICK ON THE MENU FILE (IN THE MENU BAR ON THE UPPER PART OF THE SCREEN) AND THEN CHOOSE THE MENU OPTION SAVE.



Furthermore, when you must click on one of the program buttons, this button will be displayed in the left margin of the text — facing where the button is mentioned. Buttons mentioned shortly after another, will be displayed following the order in which they are mentioned in the text.



Getting started

This section describes:

- Working with Windows programs..... (p. 21)
- Keyboard short-cuts (p. 27)
- The On-line Manual (p. 32)
- The Work area..... (p. 35)
- Applying basic Image functions..... (p. 38)

Reading this section will save time using the program.

On www.pcschematic.com you can see product videos, which can give you a survey of various functions in the program.

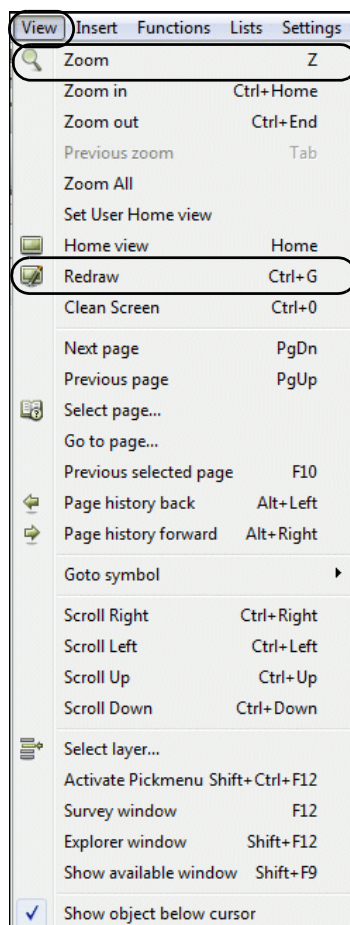
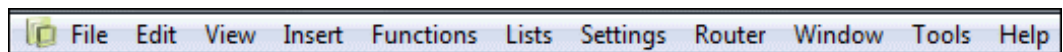
WINDOWS

This chapter describes how to use the basic Windows functions. Please skip this chapter if you are an experienced Windows user.

THIS CHAPTER DESCRIBES	
— MENUS	(P. 21)
— DIALOG BOXES	(P. 22)
— SHORT-CUT MENUS AND RIGHT-CLICKING	(P. 23)
— SWITCHING BETWEEN OPEN WINDOWS.....	(P. 23)
— USING THE MOUSE	(P. 23)
— DOCKING WINDOWS AND TOOLBARS.....	(P. 24)

MENUS

When you have opened a project in **Automation**, the uppermost line of the screen, will look like this:



This line is called the **Menu bar**, and clicking on any of the words here will make various menus appear. Here you specify which commands you want to execute in the program.

Choosing menus and commands in **Automation** follows the common directions for Windows. Below you will see the **View** menu, as an illustration of how to choose a command using the menus. Click on **View** in the menu bar, and the following menu will appear:

— As you can see, the letter **V** is underlined in the word **View**. This means that you can also choose this command by pressing [Alt+v].

Click on the option you want to choose — for instance **Screen**. You can also choose by using the arrow keys on the keyboard, and then press <Enter> when you have found the desired function.

Most of the commands have an underlined letter. Once the menu have been chosen, you can choose the command by pressing down the underlined letter. In the menu to the left, **Z**oom can for instance be chosen by pressing down [z].

If the command has got a short-cut key — which means the command can be chosen without activating the menu at all — the short-cut combination is displayed to the right of the command. You can for instance choose **Redraw** by pressing down [Ctrl+g].

If an option is *dimmed* — written in grey — you are not allowed to execute this command for the moment.

Three dots succeeding an option — for instance **Select page...** — signifies that you by clicking there will get more options — for instance in a new menu or a dialog box.

DIALOG BOXES

Below you will see a part of a dialog box (**Settings => Pointer/Screen**), together with an explanation of the various elements of the dialog box:

The screenshot shows a dialog box with several tabs: Project data, Page data, Pointer/Screen (selected), Directories, Page setup, Tools, Database, System, and Text/Symbol defaults. The 'Pointer/Screen' tab is active, showing three radio buttons for 'Grid display' (Off, Dotted, Squared), a 'Unit of measurement' dropdown menu set to 'mm', a 'Show message for 300 ms' slider, and four checked checkboxes: 'Hints', 'Show popup information', 'Show popup help', and 'Page tabs'. Arrows point from text boxes below to these elements.

Tabs →

Selected tab →

Grid display:
 Off
 Dotted
 Squared

Unit of measurement
 mm

Show message for 300 ms

Hints
 Show popup information
 Show popup help
 Page tabs

Round buttons within an area signifies *mutually exclusive* options. In this case it is therefore only possible to specify that the *grid* (please refer to the dictionary) is turned off, dotted or squared.

Click on the down-arrow, and a drop down menu will display which options you have got. In this case you will see a list containing the possible units of measurement.

Squares signifies functions you can enable or disable. When you have chosen a function, a check mark is placed within the square. In this case both **Hints** and **Page tabs** are selected.

The functions are *not* mutually exclusive, and can therefore be chosen at the same time.

SELECTING ELEMENTS IN LISTS

When elements are listed in a dialog box, these elements can be selected as described in “Selecting objects in the Object Lister” on page 298.

— See for instance “Copying, Cutting and Pasting more than one page” on page 494.

SORTING DATA IN COLUMNS

When a dialog box contains multiple columns, the contents of the dialog box can typically be sorted alphabetically based on the contents of the individual columns, as described in “The sorting order of the data in the Object Lister” on page 298.

OK and Cancel

In most of the dialog boxes there will be an **OK** and a **Cancel** button as well. The **OK** button causes you to leave the dialog box, and the settings you have specified will be effectuated. Clicking on the **Cancel** button, causes you to leave the dialog box *without* activating the changes you specified.

Please note that the buttons **OK** and **Cancel** is not displayed in the figure above. However, they are located in the lower part of the dialog box.

In stead of clicking **Cancel** you can press <Esc>.

Using the keyboard in dialog boxes

— See “Keyboard short-cuts” on page 27.

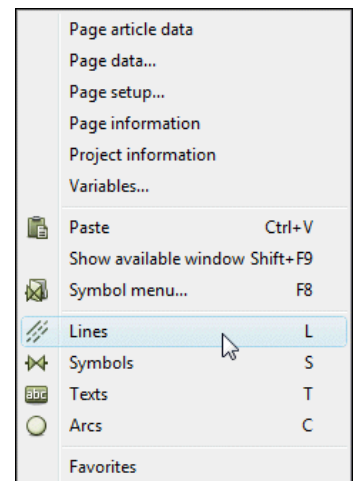
SHORT-CUT MENUS AND RIGHT-CLICKING

When you right-click on something in a project, you will practically always get a so-called short-cut menu. This menu gives you some options, which are relevant for what you clicked on.

It will therefore frequently be most efficient to right-click on an object and see which options you get.

The displayed menu appears by right-clicking on a page — the "paper" — in a project.

On new keyboards there also exists a short-cut menu key. Pressing this key has the same function as right-clicking on an object.



SWITCHING BETWEEN WINDOWS

When you want to change between several active projects (windows), press down <Ctrl> while pressing <Tab>. You then switch to another active project. You can also choose **Window** in the menu bar on the upper part of the screen, and then click on the open project you want to work on. If you have several project windows open simultaneously, you can also just click on the project you wish to work on. — Please refer to “Several Projects Simultaneously” on page 509.

When you press down the <Alt> key, and then press <Tab>, you can change to other active programs.

THE MOUSE

When you are working with Windows, you are using the mouse a great deal of the time. The operations you can perform with the mouse are displayed below:

OPERATION	DESCRIPTION
CLICK	PRESS DOWN AND RELEASE THE LEFT MOUSE BUTTON ONCE.
DOUBLE-CLICK	PRESS DOWN THE LEFT MOUSE BUTTON TWICE. IT IS IMPORTANT THAT YOU DO NOT MOVE THE MOUSE WHILE CLICKING.

OPERATION	DESCRIPTION
RIGHT-CLICK	PRESS DOWN AND RELEASE THE RIGHT MOUSE BUTTON ONCE.
DRAG AND DROP	<p>TO SELECT AN OBJECT POINT AT THE OBJECT WITH THE MOUSE CURSOR, AND PRESS THE LEFT MOUSE BUTTON. KEEP THE MOUSE BUTTON PRESSED DOWN WHILE MOVING THE MOUSE POINTER ACROSS THE SCREEN. YOU SAY THAT YOU <i>DRAG</i> THE MOUSE ACROSS THE SCREEN. WHEN YOU RELEASE THE MOUSE BUTTON, THE OBJECT IS PLACED WHERE YOU POINT WITH THE MOUSE CURSOR.</p> <p>YOU CAN MOVE BOTH OBJECTS, PROJECT PAGES AND ELEMENTS IN LISTS BY <i>DRAG AND DROP</i>.</p> <p>— SEE “MOVING A PLACED SYMBOL” ON PAGE 169, “DRAGGING A REFERENCE DESIGNATION TO A NEW POSITION IN THE DIALOG BOX REFERENCE DESIGNATIONS” ON PAGE 329 AND “DRAG PAGES BETWEEN PROJECTS” ON PAGE 510.</p>

The small arrow you control on the screen by moving the mouse, is called the mouse pointer. In Windows you can for instance specify the appearance of the mouse pointer, how rapidly it moves across the screen, and how fast you have to click when you double-click. Please refer to your Windows manual for further details.

DOCKING WINDOWS AND TOOLBARS

In many Windows programs you can move the toolbars and dock windows to the sides of the screen. In the following you can see how to do this in **Automation**.

Moving the Explorer window

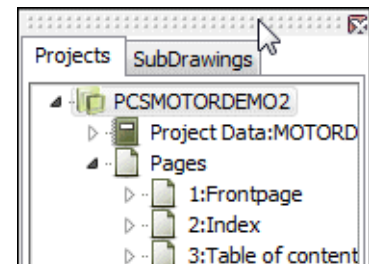
When you start **Automation** for the first time, the **Explorer** window is *docked* to the left toolbar. This means that it is placed at a fixed position just next to the left toolbar.

If you want to place the **Explorer** window at another location, you point at the two horizontal lines at the top of the **Explorer** window.

Then press down the mouse button, *drag* the window to another location on the screen, and release the mouse button.

If you just place it on the middle of the screen, it will cover for the details on the project page. When it is docked it is not in the way for viewing the page.

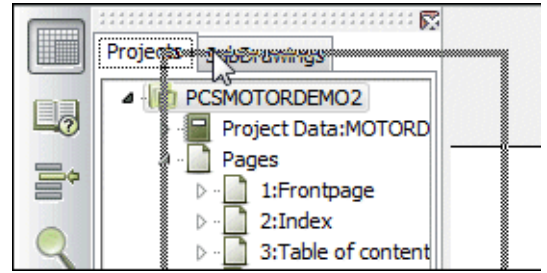
The **Explorer** window can be either a free window, or be docked to the left or to the right side of the screen.



Docking a window

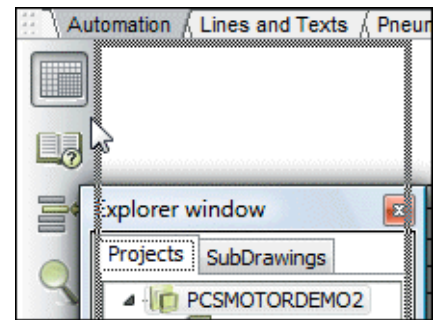
When you wish to dock the window, click on its title bar, keep the mouse button pressed down, and move the window.

The frame of the window is now displayed as a grey box.



When the mouse pointer — and *not* the edge of the grey box — is pointing on the edge you wish to dock the window to, release the mouse button.

Just before you release the button, the window will snap to its docked position.



Docking the Survey window

The **Survey** window can be docked in either the upper or the lower part of the **Explorer** window. Please note, that it is docked *inside* the **Explorer** window.

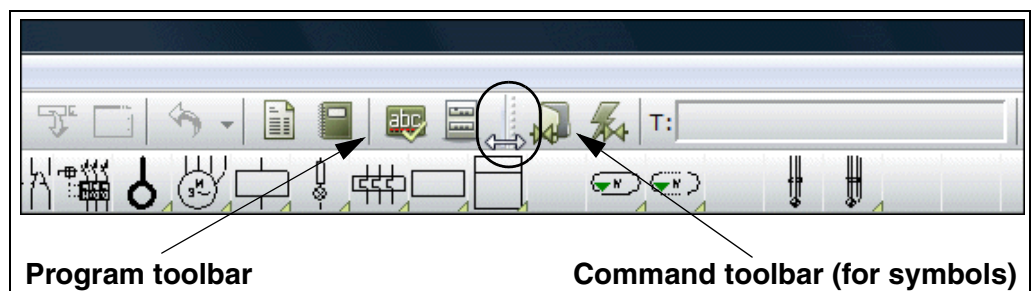
DOCKING THE PICKMENU

The pickmenu can also either be a free window, be docked below the upper toolbars, or be docked in either of the sides in the program window.

You can only dock one window along any edge of the program window. This means that if you have docked the **Explorer** window to the left-hand side, you will not be allowed to dock the pickmenu to the left-hand side as well.

Docking the Command Toolbar

The command toolbar can be docked, so that it is placed after the program toolbar using *drag and drop*.

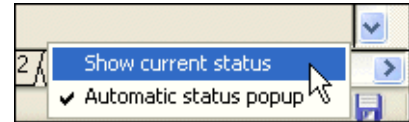
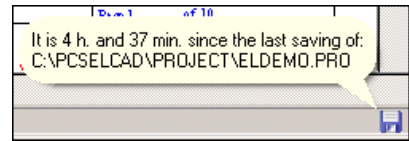


The command toolbar can not be moved/docked.

Status button / Info function for last save

In the right-hand side of the Status bar at the bottom of the screen, you will find a **Status button / Info function** for when the project was last saved.

When you click on it, it will display the period of time since last save. The function is activated automatically after 30 minutes, and is subsequently activated every five minutes, if the project is not saved. When you right-click on the button, a menu is displayed from which you can disable the automatic popup information.



KEYBOARD SHORT-CUTS

To reduce the risk for repetitive strain injuries from using the mouse too much, it is possible to work with **Automation** using the keyboard. This chapter describes these options.

Furthermore, general Windows principles for working without the mouse can be applied.

THIS CHAPTER DESCRIBES	
— CHOOSING MENU POINTS USING THE KEYBOARD	(P. 27)
— CHOOSING BUTTONS/FUNCTIONS USING THE KEYBOARD.....	(P. 28)
— CHOOSING IN DIALOG BOXES WITH THE KEYBOARD	(P. 28)
— SELECTION IN PICKMENUS USING THE KEYBOARD	(P. 29)
— PLACING OBJECTS USING THE KEYBOARD	(P. 29)
— CHOOSING OBJECTS USING THE KEYBOARD	(P. 30)
— CHOOSING MORE THAN ONE OBJECT WITH THE KEYBOARD.....	(P. 30)
— CHOOSING SYMBOLS WITH THE KEYBOARD	(P. 30)
— INCREASE/DECREASE SHORT-CUT USING THE KEYBOARD	(P. 31)
— SHORT-CUTS FOR ACTIVATING THE PENCIL.....	(P. 31)



CONCURRENT LEARNING OF SHORT-CUT KEYS

IN “PREVENTING REPETITIVE STRAIN INJURIES” ON PAGE 782, YOU CAN SEE HOW YOU CAN LEARN THE SHORT-CUT KEYS FOR THE COMMANDS YOU TYPICALLY CLICK ON WITH THE MOUSE — WHILE WORKING WITH THE PROGRAM.

CHOOSING MENU POINTS USING THE KEYBOARD

Choosing menu points using the keyboard follows the common Windows standards, as described in “Menus” on page 21.

— Please also refer to “Short-cut Keys” on page 769, where you will find a survey of the default short-cut keys, and you can read about how to define short-cut keys for all menu points in the toolbar.

CHOOSING BUTTONS/FUNCTIONS USING THE KEYBOARD

You can create short-cut keys for all buttons in the program. This is done by creating a short-cut key for the menu point that the button represents.



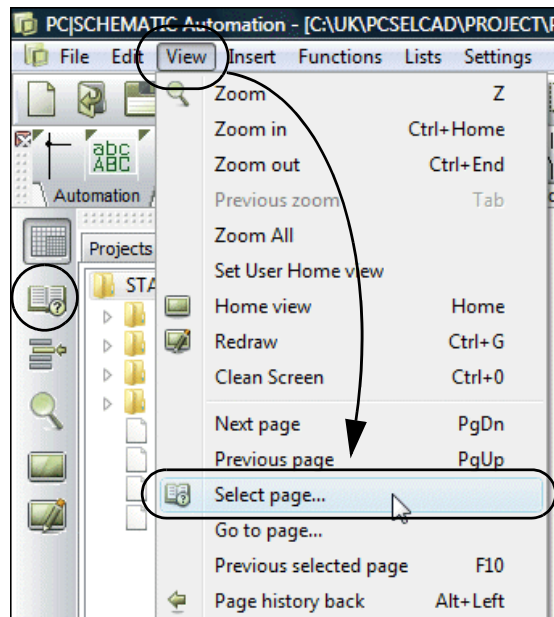
If for instance you wish to create a short-cut key for the **Page menu** button, you must create a short-cut key for the menu point **View => Select page**.



Please note that the **Page menu** button is displayed in front of the menu point, so that you can find the function in the menu easily.

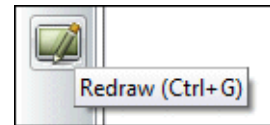
There are menu points for all buttons in the program.

— See “Short-cut Keys” on page 769 to see how to create keyboard short-cuts for all buttons.



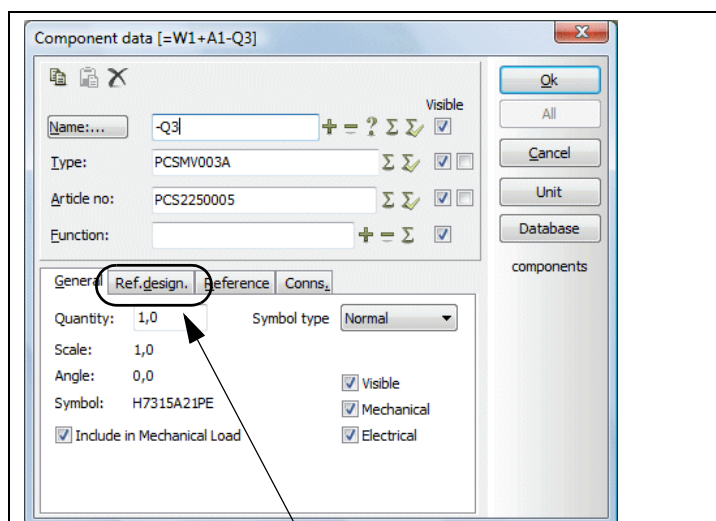
Short-cuts displayed as hints

When you let the mouse cursor rest over a button, the short-cut key is displayed as a *hint* in brackets after the button name.



CHOOSING IN DIALOG BOXES USING THE KEYBOARD

When you have opened a dialog box, you can choose the different fields, buttons or tabs by pressing down the <Alt> key, and press on the key for the underscored character for the option in the menu.



Press down <Alt> and press <d> to select the tab Ref.design.

If for instance you wish to choose the tab **Ref.design.** in the dialog box **Component data** above, press down the <Alt> tab, and press the <d> button — because **d** is the underscored character on the tab.

If you wish to select **OK**, press down the <Alt> key, and press the <O> key.

Using the tab key

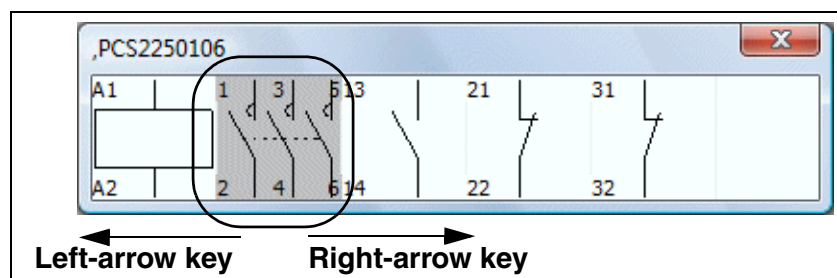
In each dialog box there is pre-defined order, in which you jump between the different options in the dialog box.

When you press the <Tab> key, you jump to the next option in the pre-defined order.

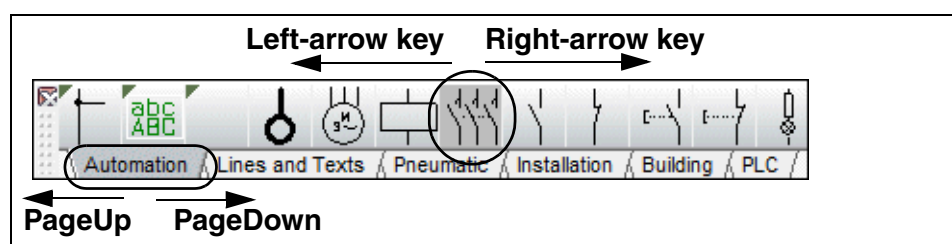
When you press down <Shift> and press the <Tab> key, you jump back to the previous option.

SELECTION IN PICKMENUS USING THE KEYBOARD

When you have selected a pickmenu for a component, the field with the first symbol in the pickmenu is highlighted. This indicates that you will get this symbol in the cross hairs, if you press <Enter>. You can use the right and the left arrow key to select another symbol in the pickmenu:



To use the keyboard to select symbols in the stationary pickmenu at the top of the screen, you must select **View => Activate Pickmenu** (Short-cut: <Shift+Ctrl+F12>).



You can then select the symbols by pressing the key-arrows, and change between the different pickmenus by pressing the <PageUp> and <PageDown> keys. When you press <Enter> you get the selected symbol in the cross hairs.

If a symbol has different states, you can select the symbol states by pressing the arrow-down and arrow-up keys.

PLACING OBJECTS USING THE KEYBOARD

When you have an object in the cross hairs, place it like this:

- 1) Use the arrow-keys, which moves the objects with snap (see “Snap” on page 42) in the direction of the chosen arrow.
- 2) Press <Enter> to place the object. — If you hereby enter a dialog box, please refer to “Choosing in dialog boxes using the keyboard” on page 28.
- 3) Press <Esc> to remove the object from the cross hairs.

SELECTING OBJECTS USING THE KEYBOARD

To move the cross hairs to select an object, do this:

- 1) Press down the <Alt> key, and use the arrow-keys: The cross hairs now jump to the next object on the page, in the direction specified by the arrow key.
- 2) Press <Enter>, and the object is selected.
- 3) Press <Enter> once more, and you enter the component data dialog box / the text dialog box.

You can now edit data for the selected object.

SELECTING MORE THAN ONE OBJECT USING THE KEYBOARD

To select more than one object using the keyboard, do this:

- 1) When the cross hairs is above an object: Press down <Ctrl> and press the <Enter> key.
- 2) Move the cross hairs to the next object, press down <Ctrl> and press <Enter> — etc. until all desired objects are selected.
- 3) To enter for instance the dialog box **Component data**: Place the cross hairs over one of the selected objects, and press <Enter> (without pressing down <Ctrl>).

You can now edit data for the selected objects.

SELECTING SYMBOLS USING THE KEYBOARD

Type in the symbol file name



When working with symbols, and you press down the [k] key, you enter the dialog box **Place symbol**, where you are asked to type in a symbol file name. — Read more about this in “Typing in the symbol file name directly” on page 148.

Type in the symbol article number



When working with symbols, and you press down the [v] key, you enter the dialog box **Place symbol**, where you are asked to type in a symbol article number. — Read more about this in “Typing in the Article Number directly” on page 149.

Type in the symbol type



When working with symbols, and you press down the [v] key, you enter the dialog box **Place symbol**, where you are asked to type in a symbol type. — Read more about this in “Typing in the Symbol Type directly” on page 150.

INCREMENT/DECREMENT USING THE KEYBOARD

See how to increment/decrement names for symbols, connection points and texts in “Short-cut keys for counting on texts” on page 117.

SHORT-CUTS FOR ACTIVATING THE PENCIL



When you are pressing a short-cut key to change from working with one type of drawing objects to another, you can activate the **Pencil** simultaneously, if you press the short-cut key twice.

Please note, that the short-cut key <Insert> always can be applied for the **Pencil**.

Lines

If for instance you press [l] to work with lines, you can press <Insert> to activate the **Pencil**.

You can also press [l][l] — which is pressing [l] twice. If you then want to deactivate the **Pencil**, you can do this by pressing [l] once more.

Circles

Correspondingly, the short-cut [c][c] will make you change to **Circles** with the **Pencil** active.

Texts

If you press [t][t], you activate **Texts**, and the cursor will be placed in the text field in the texts toolbar.

Symbols

If you press [s][s], you get the last selected symbol into the cross hairs. If you have not chosen a symbol since you opened the project, you enter the **Symbol menu**.

If you have a symbol in the cross hairs and press [s], the symbol disappears from the cross hairs. When you press [s] once more, the symbol returns to the cross hairs.

Zoom

When you want to make a new Zoom, you can apply the short-cut key [z]. If you on the other hand press [z][z], the full project page is displayed.

THE ON-LINE MANUAL

When using this on-line manual, you have the benefit of reading the manual directly on the screen while working with the program. The contents is identical to the A4 printable manual, only have the pages been adapted for reading on the screen.

THIS CHAPTER DESCRIBES	
— OPENING THE ON-LINE MANUAL	(P. 32)
— THE TABLE OF CONTENTS.....	(P. 33)
— THE BUTTONS IN THE ON-LINE MANUAL.....	(P. 33)
— ZOOMING AND ADAPTING TO YOUR SCREEN	(P. 33)
— DIRECT ACCESS TO THE INDEX	(P. 34)
— RETURNING TO Automation	(P. 34)
— PRINTING FROM THE ON-LINE MANUAL	(P. 34)

OPENING THE ON-LINE MANUAL

To open the On-line manual choose **Help** and select the manual.

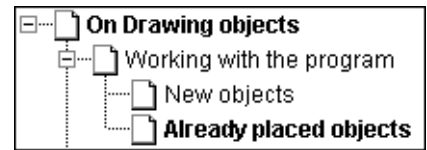
The screenshot shows the Adobe Acrobat 3D interface displaying the 'TEXTS' chapter. Annotations include:

- Click to view a chapter:** Points to the 'Texts' button in the top navigation bar.
- Zoom:** Points to the zoom controls in the top right corner.
- Fit the manual to your screen:** Points to the 'Fit' button in the top right corner.
- About the electronic manual:** Points to the help icon (question mark) in the top right corner.
- Return:** Points to the left arrow button in the right-hand navigation pane.
- Up and down:** Points to the up and down arrow buttons in the right-hand navigation pane.
- Click directly to Index:** Points to the 'Index' button in the right-hand navigation pane.
- Page no.:** Points to the page number '198' at the bottom right of the page.
- Click on the + to see the sub-chapters:** Points to the plus sign icon in the left-hand navigation pane.
- Click to view the sub-chapters:** Points to the 'THIS CHAPTER DESCRIBES' table of contents within the main text area.
- Click to jump to other chapters:** Points to the letter-based index (A-Z) in the right-hand navigation pane.

THE TABLE OF CONTENTS

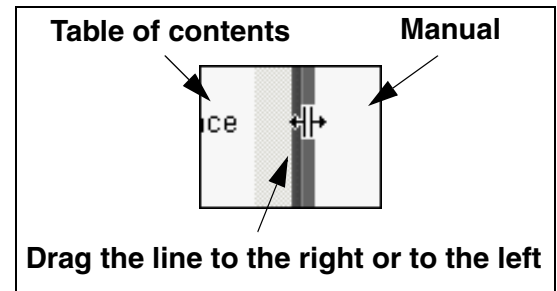
In the left-hand side of the manual, you can click directly on the section you wish to read.

A + in front of a section means that the chapter is divided into subsections.



When you click directly on a title of a section, the chosen section is displayed.

You can adjust the size of the table of contents on the screen. This is done by pointing on the line between the table of contents and the manual text, and then drag the line to either the left or the right.



You can turn the displaying of the table of contents on and off by pressing [F5].

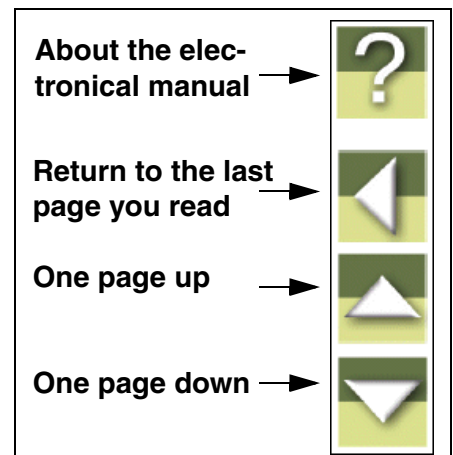
THE BUTTONS IN THE UPPER-RIGHT CORNER

In the upper-right corner of the on-line manual, you will find these four buttons.

When you click the ? button, you enter this chapter about using the on-line manual.

If you click the **Return** button, you jump back to the last page that you have viewed in the on-line manual.

When you click the up and down buttons, you leaf one page up or down respectively. You can also use the [PageUp] and [PageDown] buttons on the keyboard.



ZOOMING AND ADAPTING TO YOUR SCREEN



When you open the on-line manual initially, it is opened to a size fitting your screen. If you need to zoom in on a detail in the manual, you click the **Zoom** button in the upper-left corner of your screen.



To re-size the on-line manual to fit your screen, you click the **Fit in Window** button on the upper part of the screen, or press [Ctrl+0].

DIRECT ACCESS TO THE INDEX

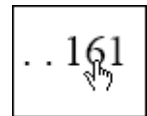
On the right-hand side of the on-line manual, you have direct access to the index.

The screenshot shows a vertical index of letters from M to XZ. The letter 'S' is highlighted with a hand cursor. A callout box titled "Selecting an area in all layers" explains that this option allows selecting everything in all layers for an instance of 'Area' and marking an area with a dashed box. It notes that if 'Symbols' were selected in the area, they would also be selected. Below this, a table of contents is visible with the entry "area in all layers" circled.

area in all layers	780
different types of objects	444
elements in lists	61
elevated objects	782

To find an explanation about fx. the **selection** function, you click **S** index letter in the right-hand side of the manual. This then looks up under **S** in the index. Now click on the dotted line after the word **Selection**, to bring up to the desired information.

Please note, that it is important that the cursor on screen has the appearance of a hand with a pointing index-finger, when you click the dotted line — or the page number. If this is not the case, you zoom in on the text, and therefore can no longer see the full page on screen.



Click the **Fit in Window** button or press [Ctrl+0] to restore full page view.

RETURNING TO Automation

In order to return to **Automation**, you just click on its program Windows button at the bottom of your screen — or press down the <Alt> button while pressing the <Tab> button.



The reason why you have to change back and forth in this way, is because that the manual is a document in the independent program **Adobe Acrobat reader**.

PRINTING FROM THE ON-LINE MANUAL



If you wish to read the manual on print, it is recommended to see the printed manual. To print out directly from the on-line manual, just click the **Print** button on the **Acrobat** program, and specify what to print out.

Please note, that the on-line manual buttons will also be printed.

THE WORK AREA

After starting **Automation**, you must choose whether to start a new project, or to continue working on an existing project.

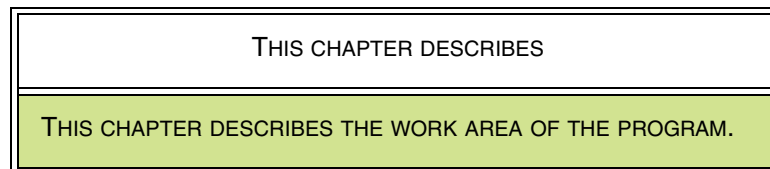


Because this chapter exclusively deals with the work area on the screen, click the **New document** (New project) button. (If you choose **File => New**, you can choose between different page and project templates, as described in “Pages and Chapters” on page 485).

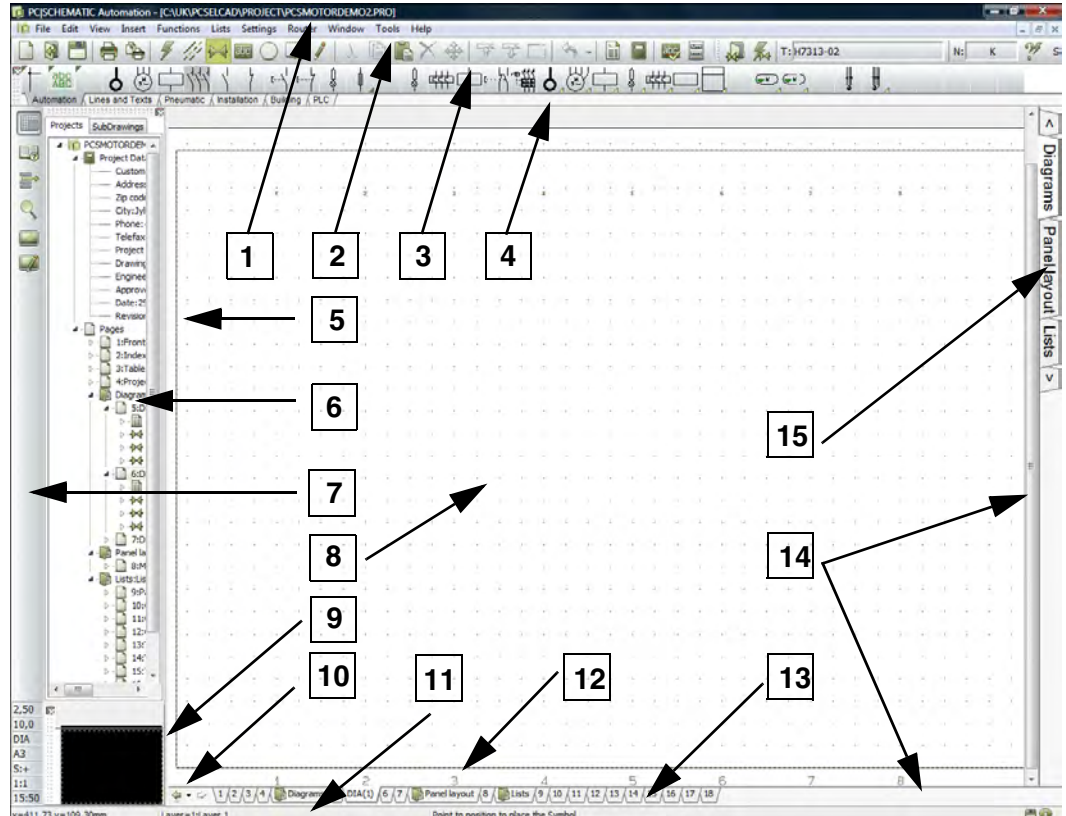
The program automatically starts up showing the dialog box **Project data**, with the tab **Settings** activated.

At the top of the dialog box **Settings** you will find some tabs, each containing a menu defining various program, page and project settings.

Click **Cancel**, **OK** or press the <Esc> key to leave the dialog box. — You can read more about this dialog box in “Settings” on page 809.



When you have opened a project, your screen should look like this:



— Please note that you can change the position of the windows and the toolbars, as described in “Docking Windows and Toolbars” on page 24.

1. The menu bar

In the menu bar you find the menus, that contains all the functions in the program. Choosing a subject in the menu bar, is either done using the mouse or by pressing down the <Alt> key while typing in the first underlined character in the menu. **Files** will for instance be selected by pressing [Alt+f]. — See “Menus” on page 21.

2. The program toolbar

The program toolbar contains program buttons. Here you can choose between the various program functions. The most common file and printing facilities are placed here, as well as the more general drawing and editing tools.

3. The command toolbar

The command toolbar has a varying appearance depending on which object type is selected in the program tool bar. It contains function and editing tools for the various drawing facilities: Lines, Symbols, Texts, Arcs and Areas. The different appearances of the toolbar for each of these is described in “Basic Drawing facilities” on page 45.

4. The pickmenu

Here you place the symbols, line types, text types and arc types you use the most, so that you can fetch them right away, and place directly in the diagram pages. When for instance you click on a symbol in the pickmenu, **Automation** change to working with symbols automatically, and you get the selected symbol in the cross hairs.

By clicking on the tabs at the bottom of the pickmenu, you select which of the created to display. (The *pickmenu* was previously named the *pickmenu* because it contained symbols only).

The pickmenu is described in “Fetching symbols from the Pickmenu” on page 136.

The pickmenu can be docked as described in “Docking the Pickmenu” on page 25.

5. The help frame

The help frame displays the standard margin on the paper. It can be deactivated easily — see “Pointer/Screen” on page 815. It is also possible to activate a printer help frame, displaying the margins of your printer.

6. The Explorer window

The **Explorer window** contains the **Projects** tab and the **SubDrawings** tab:

The Projects tab in the Explorer window

On the **Projects** tab you can get information about all open projects. You get a survey of the project pages, and a survey of all components on these pages. When for instance you click on an page symbol for a project page, the page is displayed on the screen. — See “The Explorer window: The Projects tab” on page 291.

The SubDrawings tab in the Explorer window

The **SubDrawings** tab is applied for handling e.g. placing, editing and displaying sub drawings. For instance you can *drag* sub drawings into the the active project page, or create new sub drawings based on copied areas from the active project.

— Read more about this in “Sub drawings” on page 244.

7. The left toolbar

The left toolbar contains various page and zoom functions, and — at the bottom — information about the page settings and status. These functions are elaborated in “Screen / Image Functions” on page 38.

8. The work area

The work area on the screen corresponds to the size of paper you have chosen to work on.

The size of the paper is specified in **Settings => Page setup**, as described in “Page setup” on page 822, or by inserting a drawing header, as described in “Insert drawing headers/formulas” on page 521.

9. The survey window

The survey window gives you a survey of the entire page. The part of the page that you are currently viewing on screen at the present (the *zoom*), is represented by the black box. — See “The Survey window” on page 41.

10. Page history buttons



By clicking on the **Page history** buttons, you can jump to the previously active pages in the project. — See “The Page history buttons” on page 486.

11. The status line

Among others you can see here coordinates, layer titles, guiding texts as well as various messages. When you let the mouse pointer rest over for instance a button on the screen, an explanatory text will appear here.

12. Current path numbers

Current path numbers are displayed in two different places: At the location you specify in your project, as well as on the lower part of the screen. When you zoom in on a part of a diagram, the current path numbers will still be displayed on the lower part of the screen. Thus you will always know where you are on the diagram.

— See “Current path numbers / Reference System” on page 367.

13. Page tabs

Click on the page tabs to jump back and forth between the pages. — See “Direct access to menus and Tabs” on page 43 and “Choosing pages outside the Page menu” on page 485.

14. Scrollbars

Having zoomed in on an area, you can make the area move by *dragging* the scrollbars. — See “The Scrollbars” on page 39.

15. Chapter tabs

When you click on a chapter tab, you jump to the first page in the selected chapter. — See “Chapters” on page 498.

SCREEN / IMAGE FUNCTIONS

This chapter has to do with the screen-related functions in the program. All buttons having this type of function, are located on the left hand side toolbar.

THIS CHAPTER DESCRIBES	
— THE FUNCTIONS ZOOM, SCROLL AND REDRAW	(P. 38)
— HOME VIEW AND REDRAW	(P. 40)
— SET USER HOME VIEW	(P. 40)
— SHOW EXPANDED DRAWING AREA	(P. 41)
— THE EXPLORER WINDOW	(P. 42)
— THE SURVEY WINDOW	(P. 41)
— SNAP	(P. 42)
— GRID	(P. 43)
— THE CROSS HAIRS	(P. 43)
— DIRECT ACCESS TO MENUS AND TABS	(P. 43)
— GENERAL PROTECTION ERROR	(P. 44)



ALL SHORT-CUT KEYS CAN BE CHANGED

PLEASE NOTE, THAT YOU CAN USE MANY SHORT-CUT KEYS IN THE PROGRAM. YOU CAN SEE THE PREDEFINED SHORT-CUT KEYS IN “SHORT-CUT KEYS” ON PAGE 769. THERE IT IS ALSO DESCRIBED HOW TO DEFINE YOUR OWN SHORT-CUT KEYS.

ZOOM, SCROLL, REDRAW

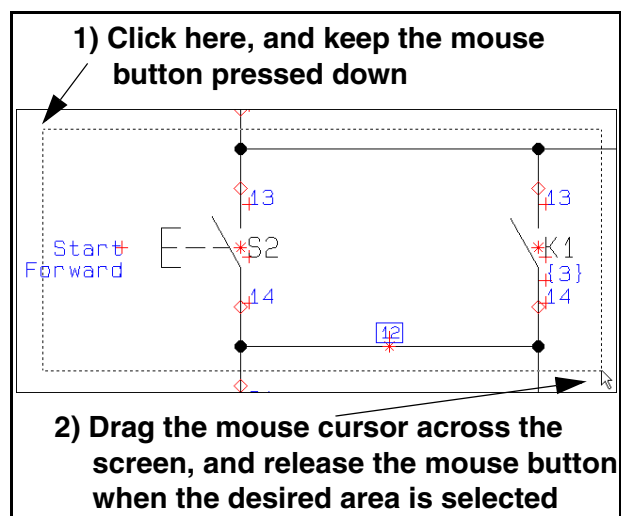
In **Automation** you can specify which part of the page you wish to see on the screen.

Zoom



When you want magnify a part of the page, you can click on the **Zoom** button, and then select an area on the screen using the mouse. Do this:

- 1) Click and keep the mouse button pressed down.
- 2) Drag the mouse over the area you wish to enlarge, and let go of the mouse button.



When you release the mouse button, the chosen area is now enlarged.

View => Zoom or the short-cut key [z] has a similar effect.

On a three-button mouse, the button in the middle can also be used for zoom.

The Zoom All function

When you choose **View => Zoom All**, a zoom is made so that all objects in the work area is displayed — in the closest zoom possible.

If there only are a few objects placed next to each other, the program will zoom in on these objects.

If there, on the other hand, are placed objects outside the boundaries of the page, the program will zoom out, so that it becomes possible to select these objects.

Zoom in/out

To zoom in, you can choose **View => Zoom in** (short-cut [Ctrl+Home]), and to zoom out you can choose **View => Zoom out** (short-cut [Ctrl+End]).

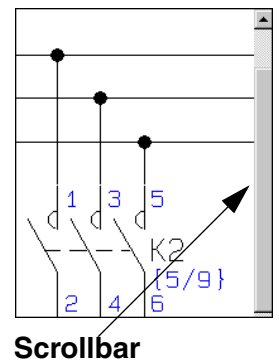
— Please also see “Zoom and Scroll for mouse with a wheel” on page 39.

The Scrollbars

When you press down the <Ctrl> key, you can use the arrow keys to move the zoom — e.g. the short-cut key [Ctrl+Right-Arrow].

Having zoomed in on an area, you can also apply the scrollbars in the right side and the bottom of the screen. When you click on a scroll bar and *drag* it to another location, the displayed zoom is moved correspondingly.

The use of scrollbars can be activated/deactivated as described in “Pointer/Screen” on page 815.



Zoom and Scroll for mouse with a wheel

If you have a mouse with a wheel, the button in the middle — the wheel — can also be used for zooming. This gives the following options:

PRESS	RESULTS IN
<WHEEL FORWARDS>	THE ZOOM IS MOVED UPWARDS ON THE PAGE
<WHEEL BACKWARDS>	THE ZOOM IS MOVED DOWNWARDS ON THE PAGE
<SHIFT+WHEEL FORWARDS>	THE ZOOM IS MOVED TO THE LEFT
<SHIFT+WHEEL BACKWARDS>	THE ZOOM IS MOVED TO THE RIGHT
<CTRL+WHEEL FORWARDS>	ZOOMS IN WHERE THE CROSS HAIRS IS LOCATED
<CTRL+WHEEL BACKWARDS>	ZOOMS OUT WHERE THE CROSS HAIRS IS LOCATED

Keep page zoom

By choosing **Settings => Pointer/Screen**, you can decide whether to keep the zoom when changing back and forth between pages in the project. You specify the zoom settings by deselecting **Keep zoom** and **Keep page zoom**. Read more about this in “Pointer/Screen” on page 815.

View previous zoom

When you select **View => Previous zoom**, you jump back to the previously applied zoom on the active page. Because the program memorizes the two last applied zooms, you can jump back and forth between these by using this function.



As a short-cut for this function, you can use the <Tab> key, and you can also activate the function by pressing the <Ctrl> key while clicking the **Zoom** button.

All pages memorizes the two most recently applied zooms, until the project is closed. Please note that this enables you to use the <Tab> button to jump directly from full screen to the most recently applied zoom.

HOME VIEW AND REDRAW



When you click the **Zoom to page** button, the entire page will be shown on the screen area. Choosing **View => Home view** gives the same result.

The short-cut keys are [Home] or [z][z].



To refresh the image on screen, click on the **Redraw** button. Alternatively select **View => Redraw**; the short-cut key is [Ctrl+g].

This updates the image on the screen, as well as the survey window.

SET USER HOME VIEW

When you only want **Automation** to display a specified section of the page when you click on the **Home view** button, do the following:



- 1) Choose **View => Set User Home view**.
- 2) The mouse cursor now changes appearance to a two-way arrow: *Click* to specify one corner of the new home view, and *click* once again to specify the opposite corner of the home view. (This is done similar to making a *zoom* — see “Zoom” on page 38).



The next time you click on the **Home view** button — or press [Home] — the specified area is displayed on the screen. This is the consequence for all project pages having the same page size, as the page you set the home view for — for instance **A4 horizontal**.

If you wish to create other Home views for other page formats, this is also possible.

Short-cut to creating User Home view



- 1) Click on the **Zoom** button.
- 2) The mouse cursor changes appearance to a two-way arrow: *Click* to specify one corner of the new Home View, and *press <Ctrl> down* and *click* once again to specify the opposite corner of the Home view. (See “Zoom” on page 38).

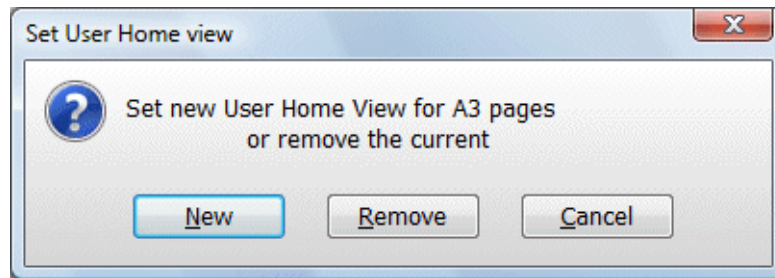
Display the full page, and not just the User Home view

When you wish to see the full page, and not just the defined Home view, press [zz].

Remove Home View settings



When you want the **Home view** button to display the full page again, you choose **View => Set User Home view** again. You then get the following message:



Click **Remove** to make the Home view button display the full page again, or click **New** to create a new Home view.

EXPANDED DRAWING AREA

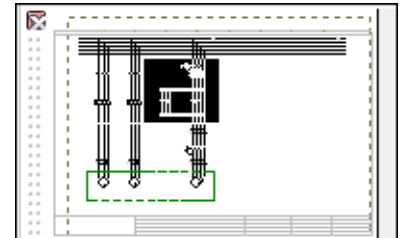
When you select **View => Clean screen**, the active project page will fill all of the program window. This makes the program toolbars and the **Explorer** window invisible.

You return to normal viewing by selecting **View => Clean screen** again.

The default short-cut key for the function is <Ctrl+0>.

THE SURVEY WINDOW

The survey window is an independent window which can be *docked* inside the **Explorer** window, or be placed anywhere on the screen. — For further details about the location of the **Survey** window, please refer to “Docking Windows and Toolbars” on page 24.



Moving the displayed area

A black box within the window shows which section of the drawing (which *zoom*) is presently being displayed on the full screen. Click on the black box and drag it until it covers the part of the drawing you want to see on screen. When the mouse pointer is pointing on the black box, you will be noticed that you are now able to move the box.

Zooming in the Survey window

You can zoom in and out using the survey window. By placing the mouse pointer on the boundary of the black box, it will change into a two-way arrow. You can now resize the zoom by dragging the boundary.

Objects displayed in the Survey window

Objects (see “On Drawing objects” on page 47) located on the page, will also be displayed in the survey window. This gives you the option to keep the survey of the page while working in a zoom. You can select a new zoom using the placed objects as guides. *Texts* will not be displayed in the **Survey** window.



The survey window is updated when you click **Redraw** or display a new project page.

Placing the Survey window

You can move the survey window by clicking on the windows menu bar, keeping the mouse button pressed down, and then drag the window to a new preferred location. The window can be enlarged or reduced by dragging its corners.

When you close down the program, the size and position of the window is memorised.

The **Survey** window can be activated/deactivated by choosing **View => Survey window**. Short-cut key [F12].

Docking the Survey window in the Explorer window

See how to dock the **Survey** window inside the **Explorer** window in “Docking the Survey window” on page 25.

THE EXPLORER WINDOW

In the left side of the screen you find the **Explorer** window. From here you can for instance choose what to display on the screen, change page data and project data, change symbol and cable component data, find symbols and close projects.

— Read more about this in “The Explorer window: The Projects tab” on page 291.

SNAP

When you place an object on a drawing, it is important that you can position it exactly where you want it. To support this, you can specify that objects only may be located in fixed spots, located at intervals of for instance 2.50mm. — If you then want to place a symbol, you will only be allowed to move it in steps of 2.50mm on the screen, which enables you to place the symbols accurately.

If the distance between the points where you can place the symbols is 2.50mm, you say that the *snap* is 2.50mm.



By clicking the **Snap** button in the left toolbar, you can toggle between normal snap (for instance 2.50mm) and fine snap (for instance 0.50mm).

If you are working in fine snap, the lower part of the left toolbar will display a red background under the size of the snap.

Choose **Settings => Page settings** to change the snap settings — or click on “2.5” in the lower part of the left toolbar.

If you have an object on the cross hairs that you want to place, you can place it using fine snap by pressing down <Shift>. After you place the object, the program automatically change back to normal snap.

Please note that 2.50mm is standard normal snap size for electro-technical drawings.

GRID

The dots you see placed on the entire page in a drawing, is called the *grid* of the drawing. You can change the distance between the dots by choosing **Settings => Page settings**.

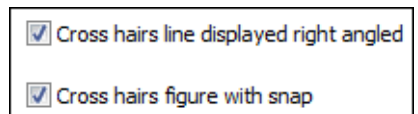
Choosing **Settings => Pointer/Screen**, you can disable the function completely, or choose to use squares instead of dots. See “Pointer/Screen” on page 815.

The size of the grid — in millimetres — only relates to what you see on screen, and not to the real size of the drawing. Thus changing the scale of the page from 1:1 to 1:50, will not make the grid change on the screen.

THE CROSS HAIRS

While you are working on a project drawing, the location of the cursor is indicated by a crossing vertical and horizontal line. This is called the *cross hairs*.

In **Settings => Pointer/Screen** you can see the cross hairs is set to be displayed right angled.



When you draw **Straight lines** (see “Straight lines” on page 66), a line will be displayed starting from where you clicked last, pointing towards the cross hairs.

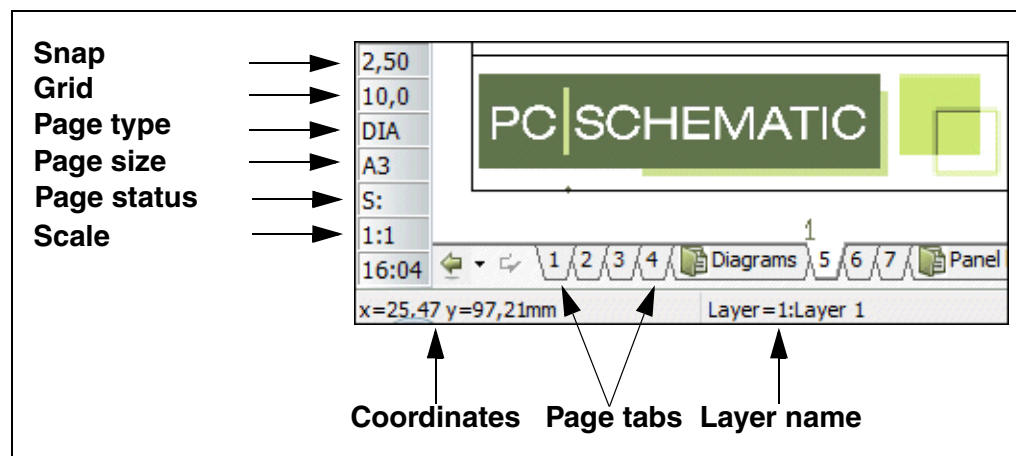
When you select **Cross hairs displayed right angled**, you will see the line that would be drawn if you clicked in this very moment. This line will not always be shown ending up in the cross hairs — due to the snap applied.

If the function is not selected, the line will always be shown from the place you clicked last and directly to the cross hairs. But this line is not necessarily the exact line that would be drawn if you clicked.

When **Cross hairs figure with snap** is selected, the figure (for instance a symbol) you have in the cross hairs will be shown in the nearest snap — exactly where it will be located when you click.

DIRECT ACCESS TO MENUS AND TABS

In the lower left-hand corner of the screen, you will see various information about fx. the snap settings and the title of the present layer. Clicking on these fields brings you directly into the menus, where these settings can be altered.



When you let the mouse cursor rest an instance over one of the fields, an explanatory text is displayed — as for **Snap** on the figure above.



Page status

In the field **Page status** the following status options exists:

- S:** The page has *not* been changed since last save.
- S:+** The page *has* been changed since last save.
- S:R+** The page has been changed since last revision start/stop, and has been changed since last save.

Page tabs

By clicking on the page tabs, you jump back and forth between the project pages. You can also use the short cut keys [PageUp] and [PageDown] to jump between the project pages.

Chapter tabs

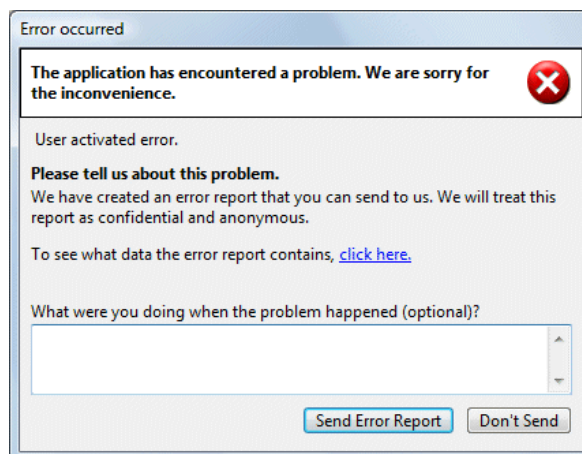
When you click on a chapter tab in the right-hand side of the screen, the first page in the displayed chapter is displayed.

GENERAL PROTECTION ERROR

If a General protection error occur, the line at the bottom of the screen starts flashing on a red background, giving a warning that the project should be saved under a new name, and that the program should be restarted.



At the same time this window is displayed:



Here you can tell us what happened just before the error occurred, which we would very much like to know, so taht we can correct it as soon as possible.

When you want to save the project, you automatically enter the dialog box **Save as**, to prevent that you save what might be a project containing errors, so that this project replace, the last saved version of the project. However, if you do let this project replace the last saved version of the project, **Automation** automatically creates a backup file (extension **.~pr**), so that the original version of the project also can be opened succeedingly.



Basic Drawing facilities

This section describes the basic drawing facilities in PCISCHMATIC Automation. Following topics are described:

- Working with drawing objects (p. 47)
- Transferring data between objects..... (p. 55)
- Align, Space and Trim functions (p. 59)
- Lines (p. 65)
- Arcs/Circles..... (p. 97)
- Texts..... (p. 99)
- Symbols..... (p. 133)
- Functions on Component level(p. 220)
- Areas.....(p. 232)
- Sub drawings.....(p. 244)
- Rename symbols.....(p. 278)
- Insert image(p. 286)



Read survey article
Please read the survey article about drawing
functions at www.pcschematic.com.

ON DRAWING OBJECTS

Any object you place on a drawing has got to be one of four different types of drawing objects: It is either a symbol, a text, a line or a circle.

Also there is the **Area**-command, which makes it possible to work independently of object types.

THIS CHAPTER DESCRIBES	
— THE BASIC WORKING MODES OF THE PROGRAM	(P. 48)
— SELECTING OBJECTS	(P. 49)
— AUTO COMMAND CHANGE	(P. 51)
— HOW TO WORK WITH SELECTED OBJECTS.....	(P. 51)
— SHOW OBJECT BELOW CURSOR	(P. 53)
— THE UNDO FUNCTION.....	(P. 54)
— TRANSFERRING PROPERTIES BETWEEN OBJECTS	(P. 56)
— ALIGN, TRIM AND SPACE	(P. 59)
— INVISIBLE OBJECTS	(P. 62)

In the Menu bar and the Program toolbar you have different options at hand, depending on which type of objects you have chosen to work with.

The buttons for command change

The buttons, you have to click on to select the various objects, look like this:

Symbols [s]

Texts [t]

Lines [l]

Circles [c]

Areas [a]



WHEN YOU CLICK ONE OF THESE BUTTONS YOUR OPTIONS CHANGE:

- THE PROGRAM TOOLBAR CHANGES
- THE OPTIONS IN THE PROGRAM MENUS CHANGES
- YOU ARE ONLY ALLOWED TO WORK WITH OBJECTS OF THE CHOSEN TYPE

— Please see “Auto command change” on page 51 also.

WORKING WITH THE PROGRAM

In **Automation** there are two different modes to work in.



Which mode you work in is determined by the **Draw** button (the **Pencil**).

You either:

- a) Draw/place *new objects* (activated/highlighted **Pencil**), or you work on
- b) *Already placed objects* (not activated Pencil)

New objects

When you draw/place new objects, you do so like this:

- 1) Select the type of objects you wish to work with
- 2) Activate the **Pencil**
- 3) Draw / place the objects



Thus if you want to draw a line, you first click on the **Lines** button (1), then click on the **Pencil** (2), and draw the line (3).

Please note that in several occasions the program activates the Pencil by itself. For instance this happens when you have typed in a text in the text field, or when you have picked a symbol in order to place it.

Already placed objects

If you, on the other hand, have to work with objects that have already been placed, the procedure is like this:

- 1) Select the type of objects you wish to work with — see “Auto command change” on page 51 also
- 2) Deactivate the pencil (Press <Esc>)
- 3) Select the object(s) you want to deal with
- 4) State what you want to do with the objects



If for instance you want to copy a symbol, you first click the **Symbols** button (1) and deactivate the pencil pressing <Esc> (2). Then select the symbol by clicking on it (3), and specify that you wish to copy the symbol by clicking **Copy** (4).

The copied symbol can now be placed.

Displaying relative coordinates

When moving or copying objects, the object coordinates are displayed in the status line at the bottom of the screen.

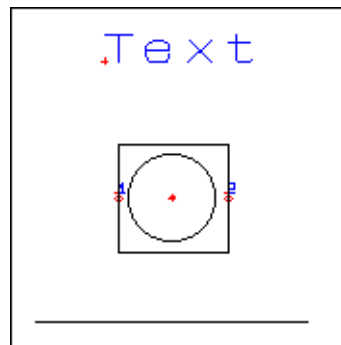
SELECTING OBJECTS

When you wish to do something with an existing object, you first have to select the object. This takes place in the following way:

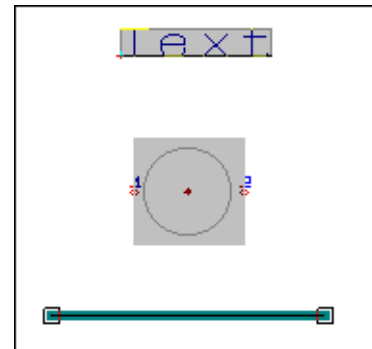
Click on the button which corresponds to the type of the object — for instance the Texts or the Symbols button. Make sure that the *Pencil* button is not selected; click on it or use <Esc>. Click on the object you wish to select.

The selection will be indicated as a colored field surrounding each chosen object individually.

Below you can see how deselected as well as selected objects appear on screen:



Deselected



Selected

When working with symbols, please note that a selection can include the entire symbol, or just one of its connection points.

If you select a text, the text toolbar (see “Dictionary” on page 850) will tell you which type of text you have chosen.

Remarks about selecting lines

A line is constructed of segments, depending on how many breaks it contains. First time you click on a line, all segments in the line will be selected. If you click at the line once more, only the segment you click on gets selected.

Electrical lines will be selected up to the next electrical point.

Selection by right-clicking

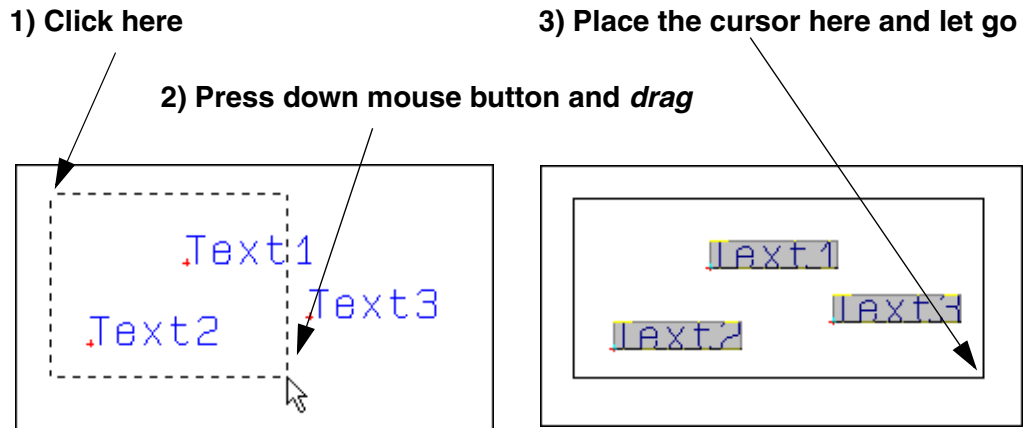
By right-clicking on an object (of the chosen object type), you will select the object and at the same time a short-cut menu will appear. In this menu you can e.g. choose **Move**, **Copy** or **Delete**. — Please refer to “Short-cut menus and Right-clicking” on page 23 for further details.

Selecting more objects of the same kind in one window



You can select more than one object of the same type at the same time. If for instance you want to select all texts in an area, click on the **Texts** button and select the desired area with the mouse. Do this by clicking in one corner of the area, and then, still pressing down the mouse button, drag the mouse to the opposite corner of the area. Thus a rectangle will be drawn with dashed lines, having one corner where you first clicked and the opposite corner where the mouse pointer is located

now. When all the desired objects are inside the rectangle, you let go of the mouse button. The texts in the area have now been selected.



If you wish to deselect one or more objects inside the window, press down the <Ctrl> key and click on the object(s). If you also wish to select objects outside the area, also press down the <Ctrl> key, and click on these objects.

Selecting more objects of the same kind using the mouse



Select the kind of objects you want to work with; for instance **Symbols**.

Press the <Ctrl> key while clicking on the desired objects. If you have selected a symbol you do not want after all, then click on the symbol again — still pressing the <Ctrl> key.

Selecting different types of objects in an area



Click on the **Area** button. Mark up a window as described in “Selecting more objects of the same kind in one window” on page 49. Hereby all object types inside the window are selected. If you wish to deselect one of the selected objects, press the <Ctrl> key and click on the object. If you also wish to select objects outside the area, also press down the <Ctrl> key, and click on these objects. — Read more about areas in “Areas” on page 232.

Selecting on different Layers and in different elevations

The ways of selecting objects described above, all assumes that you are working in the same *layer* as the objects you wish to select. — Read more about selecting and layers in “Selecting objects on different layers” on page 468, and read more about layers in general in “Layers, Elevation and Scaling” on page 465.

Selecting everything on a page



You can select everything on a page (including different layers) by selecting **Edit => Select all => All on page**. Short-cut [Ctrl+a]. If the **Area** button is selected all objects on the page will be selected. If **Symbols** are selected, all symbols on the page will be selected — and so on.

AUTO COMMAND CHANGE

If you want **Automation** to change command for you automatically, choose **Functions => Auto command change**.

When this function is activated, **Automation** automatically changes between **Symbols, Lines, Texts** and **Circles** when you want to select, move or delete objects. The program automatically changes to the correct command when the object is selected.

The function can be deactivated by choosing **Functions => Auto command change** again.



The function can also be activated/deactivated by clicking the **Auto command change** button, placed to the left of the **Lines** button.

— Please see “Show object below cursor” on page 53 also.

WORKING WITH SELECTED OBJECTS

When you want to copy, move, delete or rotate objects, you have to:

- 1) Choose the type of objects you are interested in
- 2) Select the objects you want to deal with
- 3) Copy / move / delete / rotate the objects

Step one and two are dealt with in “Selecting objects” on page 49, and step three will be elaborated in the sections to follow.

To see how to place / draw new objects, you have to read the following sections: “Lines” on page 65, “Arcs/Circles” on page 97, “Texts” on page 99 and “Symbols” on page 133.

Where no further remarks are added, the following functions covers working with **Symbols, Texts, Lines, Circles and Areas**.

Moving selected objects

When you want to move selected objects, you have three options at hand:



- a) Click the **Move** button. You thereby get the selected objects in the cross hairs, and just have to click where you want them placed.
- b) Use *click and draw*: Click on the selected object(s), keep the mouse button pressed down, and *drag* them across the screen. Let go of the mouse button when you can see the objects where you want them located. Read more about *drag and drop* in “The mouse” on page 23.
- c) Right-click inside the window. A menu emerge, and you choose **Move**. You get the selected object(s) in the cross hairs, and can move them across the screen. Click when the objects are where you want them located.

Remarks about moving Symbols

When you have to move a symbol that is connected to lines, you can do this in two different ways:

- a) The symbol remain connected to the lines: Click on the symbol, and *drag* it to the place you want it located.
- b) The lines are parked and the symbol can be moved freely: Press down the <Ctrl> key while clicking on the symbol, and *drag* it to where you want it placed.



You can also move the symbol freely by clicking on the symbol, and then press down <Ctrl> while clicking **Move**.

Furthermore, if you place a two-poled symbol on top of a conducting line, the symbol will automatically be connected to the line.

Copying selected objects

When you want to copy one or more selected objects, you have got two options:



- a) Click the **Copy** button. You now get a copy of the selected object(s) in the cross hairs, and can place the objects as many times as you want by clicking.
- b) Right-click on one of the selected objects. A menu emerge, and you choose **Copy**. You now likewise get a copy of the objects in the cross hairs, and can place the objects as many times as you want by clicking.

Deleting selected objects

If you want to delete one or more selected objects, you have also got two options:



- a) Click the **Delete** button or press . Hereby the objects are removed form the screen.
- b) Right-click on one of the selected objects. Choose **Delete** in the emerging menu. The objects are removed from the screen.



Please remember that you can undo your actions clicking the **Undo** button.

If parts of the deleted objects remain on screen, click the **Redraw** button to get the page refreshed.

Remarks when deleting a symbol

If you delete a symbol having conducting lines connected to it, the lines will be parked.

Rotating a selected object

This function works for **Texts**, **Symbols** and **Arcs**, but *not* for lines. The function also works for **Areas**, and lines within the selected area will also be rotated.

All objects are rotated counter-clockwise.

There are four ways of rotating objects:

- a) Press the [spacebar]. The object will be rotated 90 counter-clockwise.
- b) Right-click on the object and choose **Rotate**.



c) Click on the **Rotate** button in the toolbar.

d) Click in the field **V:**, type in the rotation angle and press <Enter>. The angle can have an accuracy of 1/10 degree. You can also click the down-arrow in the field, select an angle, and press <Enter>.

Remarks about rotating symbols



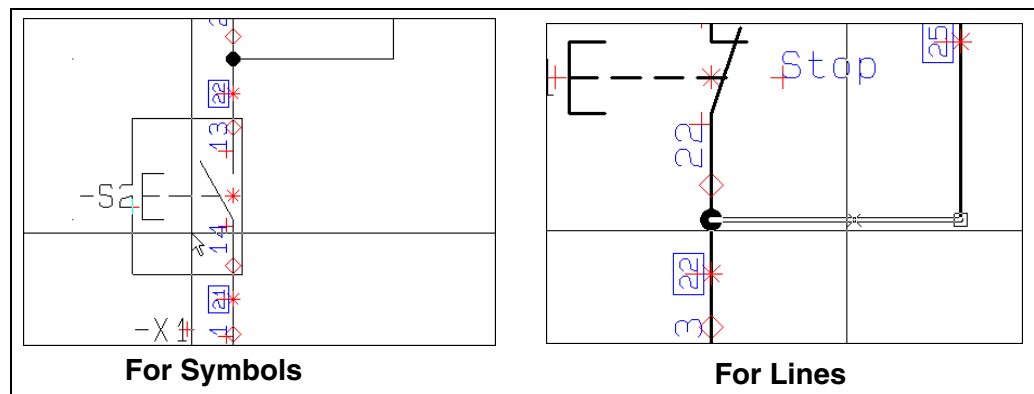
A symbol connected to conducting lines in both ends, will rotate 180 degrees, and the connection to the lines will be exchanged. Press down <Ctrl> when you click **Rotate** if you want the symbol to rotate only 90 degrees, and the lines will keep their original connections to the connection points.

Rotating areas

You can also rotate an entire area in one operation. To this, you must first have the area in the cross hairs in connection with either moving or copying, as described in “Rotating areas” on page 241.

SHOW OBJECT BELOW CURSOR

The function **View => Show object below cursor** temporarily frames the objects below the cursor, as the cursor is moved across the project page.



For lines it is also specified where you would connect to the line, if you clicked on it at the given moment.

— Please see “Auto command change” on page 51 also.

UNDO

An important function to know when working with objects, is the **Undo** function.



When you click the **Undo** button once (short-cut key [z]), you undo the last action the program has performed.

If you let the mouse cursor rest above the **Undo** button, the operation that you can undo, is displayed as a hint.

Undoing multiple actions/levels

When you click the down-button next to the **Undo** button, a list of the most recently performed actions are displayed. Here you can specify how many actions you wish to undo.

When for instance you click **Connection data -X3:2** in the figure to the right, all of the latest actions are undone, including when the connection data for **-X3:2** was changed.

See how many actions you undo

To begin with, the list only displays the five latest actions. When you place the cursor above the last point in the list, five more actions are displayed. This continues until you see all of the actions that can be undone.

The last point in the list also displays how many actions you have selected in the list at any moment.

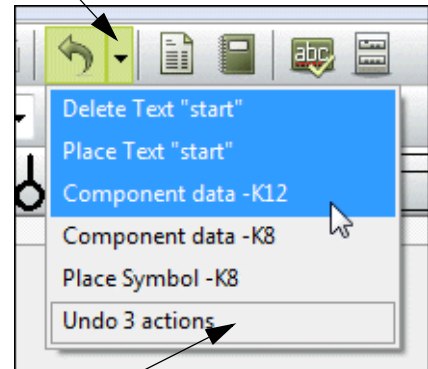
Specifying the number of undo levels

Under **Settings => System** you can specify how many actions/levels it shall be possible to undo at your system. — See “Undo levels” on page 829.

Some functions can not be undone

Please note that there are functions which you can not undo, such as using the automatic wire numbering or placing a drawing header on a page. In the first case you can save the project before assigning the wire numbers, and in the last case you have to take action yourself to remove the drawing header.

Click here, and the Undo list is displayed



Place the cursor here, and the next points in the list are displayed

TRANSFERRING DATA / PROPERTIES BETWEEN OBJECTS

You can transfer two types of information between objects of the same type in **Automation**:

- *Component Data*, which contains information about the *physical* components attached to the objects, and
- *Object properties*, which only have to do with the *appearance* of the objects

THIS CHAPTER DESCRIBES	
— HOW TO TRANSFER COMPONENT DATA BETWEEN OBJECTS	(P. 55)
— HOW TO TRANSFER OBJECT PROPERTIES BETWEEN OBJECTS.....	(P. 56)
— TRANSFERRING OBJECT PROPERTIES TO MORE THAN ONE OBJECT	(P. 56)

Transferring Component Data for Symbols or Lines

It is only possible to transfer *component data* between symbols or lines.

Transfer component data for Symbols



When you transfer component data from one symbol to another, start by clicking the **Symbols** button, and deactivate the Pencil (Press <Esc>).

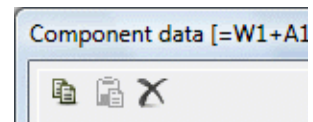
COPYING THE COMPONENT DATA



Right-click on the symbol you want to transfer component data *from*, and choose **Component data**. — Or click on the symbol and then click on the **Object data** button.



In the upper left hand corner of the **Component data** dialog box you click the **Copy component data** button. Leave the menu by pressing <Esc>.



TRANSFERRING COMPONENT DATA TO A SYMBOL



Right-click on the symbol you want to transfer the component data *to*, and click the **Paste component data** button.

You have now transferred component data from one symbol to another.

Correct the symbol name if you want to, and click **OK**.

TRANSFERRING COMPONENT DATA TO MORE THAN ONE SYMBOL

If you want to transfer the copied component data to more than one symbol, you can for instance drag a window around the relevant symbols, as described in “Selecting more objects of the same kind in one window” on page 49.

Then right-click inside the selected area, and choose **Component data**. Insert the copied component data by choosing **Edit => Paste**.

Please note, that if for instance **Type** and **Article data** are dimmed when you enter the dialog box, this tells you that the selected symbols do not have identical contents in these fields.

Transfer article data for Lines



If you want to transfer line article data from one line to another, the procedure is the same as described in “Transfer component data for Symbols” on page 55. The only difference is that you have to activate the **Lines** button instead of the Symbols button, and then choose **Line article data**.

Transferring Object properties between Objects

To all objects there are attached information about the attributes of the object, the so-called *object properties*. These attributes are for instance called *line properties* or *text properties*.

Below you can see which type of information you can transfer between the various types of objects:

OBJECT PROPERTIES	INFORMATION THAT CAN BE TRANSFERRED
LINE PROPERTIES	THE TYPE, WIDTH, LINE DISTANCE AND COLOR, AS WELL AS WHETHER THE LINE IS A CONDUCTING LINE AND WHETHER IT IS A JUMPER LINK — SEE E.G. “LINES” ON PAGE 65. — BUT <i>NOT</i> THE COURSE OF THE LINE, OR IF IT IS DRAWN AS FOR INSTANCE A SLANTED OR CURVED LINE. DURING THE CREATION OF A SYMBOL, <i>FOLLOW CONNECTED</i> IS ALSO TRANSFERRED. SEE “CREATING SYMBOLS” ON PAGE 676.
TEXT PROPERTIES	ALL INFORMATION IN THE DIALOG BOX TEXT PROPERTIES (P. 118). THE TEXT ITSELF, AND THE ANGLE IT MIGHT BE ROTATED WITH, IS <i>NOT</i> TRANSFERRED.
CIRCLE/ARC PROPERTIES	ALL INFORMATION IN THE CIRCLES TOOLBAR (P. 97).
SYMBOL PROPERTIES	ONLY THE SCALE OF THE SYMBOL.

Changing object properties on existing object

Select an object. In the toolbar (or in the dialog box **Text properties** for texts) you make the desired changes. Click on the **Transfer properties** button.



The changes have now been transferred to the chosen object.

Transfer object properties from one object to another

Select the object that carries the desired object properties, and click on the **Copy properties** button — or press down <Shift> while clicking on the object.



Select the object you want to transfer the properties to, and click on **Transfer properties**.

TRANSFER OBJECT PROPERTIES TO MORE THAN ONE OBJECT

It is possible to transfer object properties to several objects in a single operation. For instance you can give a number of texts the same height and/or color.

In this example you can see how this is done by placing the two texts *Text1* and *Text2* — each carrying different text properties — and then give both texts the same height without changing the other attributes of the texts.

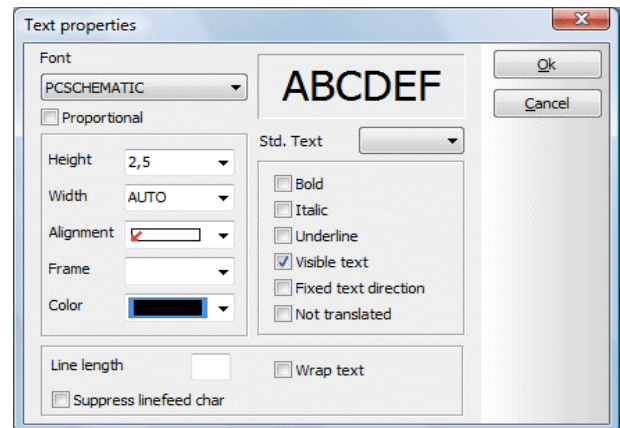


Place Text1

Click on **Texts**, click in the text field in the texts toolbar, type in *Text1*, and click on the **Text properties** button.

Set **Height** to 2.5, **Width** to *AUTO*, **Alignment** to *left-down* and **Color** to *Black*.

Click on **OK** and place the text somewhere on the page.

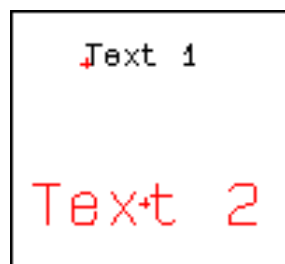
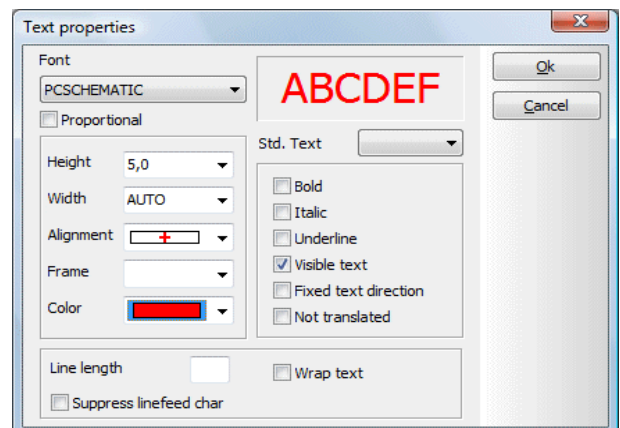


Place Text2



Then click in the text field in the texts toolbar once more, but type in *Text2* this time. Click on **Text properties**, and set **Height** to 5.0, **Width** to *AUTO*, **Alignment** to *centre-centre* and **Color** to *Red*.

Click **OK**, and place the text on the page.



Transfer text properties

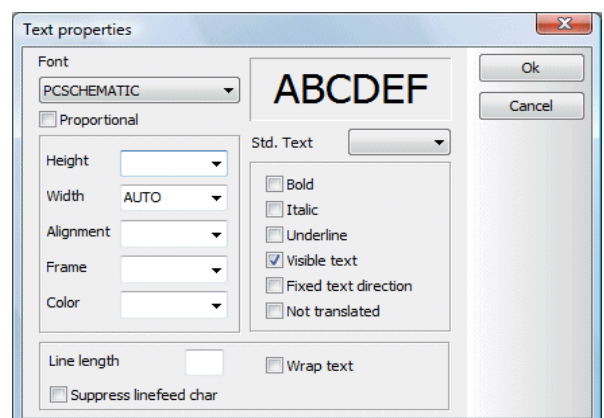


Select both of the texts, click on the **Copy properties** button and then on **Text properties**.

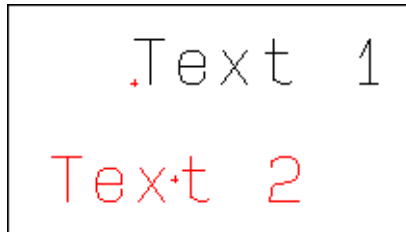
In the dialog box you now see which text properties the selected texts have in common — in this case both has the **Width AUTO**.

Now set **Height** to 8.0 and click **OK**.

As you can see, the texts now have the same height, but their other attributes have not been changed.



Text1 is still black and aligned *left-down*, while Text2 still is aligned *centre-centre* and red.



You can use the same procedure for lines and circles.



ALIGNING AND SPACING FUNCTIONS

In **Automation** you also have some functions for trimming and spacing objects.

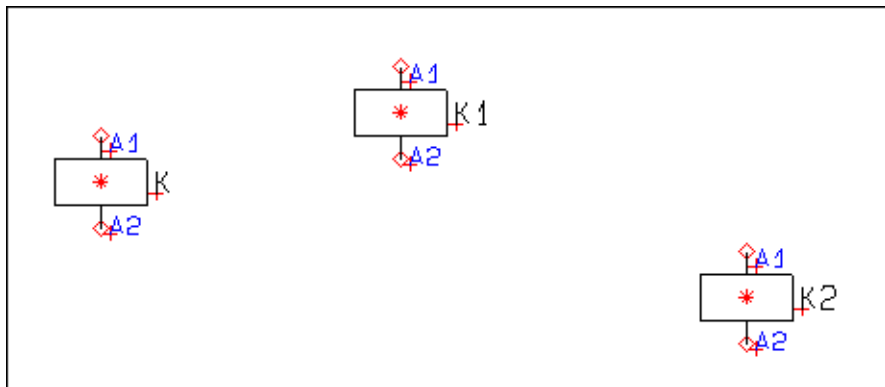
THIS CHAPTER DESCRIBES	
— THE ALIGN FUNCTION.....	(P. 59)
— THE SPACE FUNCTION.....	(P. 60)

The Align function

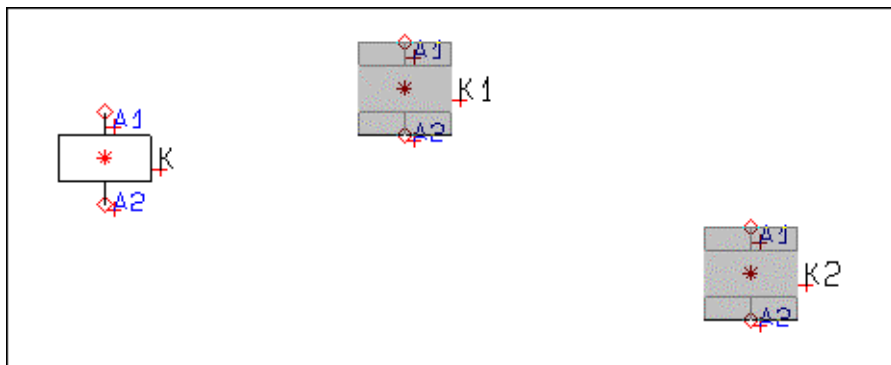
As you will see, it is possible to align objects in **Automation**.

In the following example symbols will be used, but you can do exactly the same using texts or circles.

These three symbols are to be aligned, at the same height as the symbol to the left:



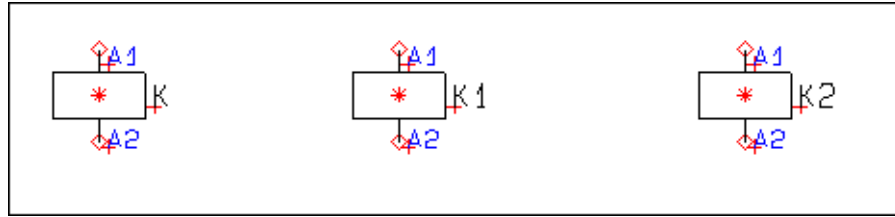
Make sure that the **Symbols** button is activated, then click on the right most symbol. Now press down <Ctrl> while clicking on the symbol in the middle. (Alternatively, you could drag a window around the two symbols). These two symbols have now been selected:



Select **Edit => Align**, or right-click inside the window and choose **Align**.

A rubber band starting in the symbol in the middle and ending in the cross hairs is now displayed. (See the guiding text in the bottom right-hand corner of the screen).

Click on the symbol to the left, and the two selected symbols are aligned with the symbol.



If the symbols had been closer to be arranged on a vertical line, they would have been placed on a vertical line instead.

Please note that the determining symbol may also be one of the selected symbols.

The Space function

There exists a similar function, placing symbols with fixed intervals. Unlike **Align** you relocate all the symbols when you use the **Space** command.

In the following example symbols are used once again, but texts or circles could be used as well.



Activate the **Symbols** button, and place some symbols as shown on the figure.

Then mark up a window surrounding the symbols, and choose **Edit => Space** — or right-click and choose **Space**.

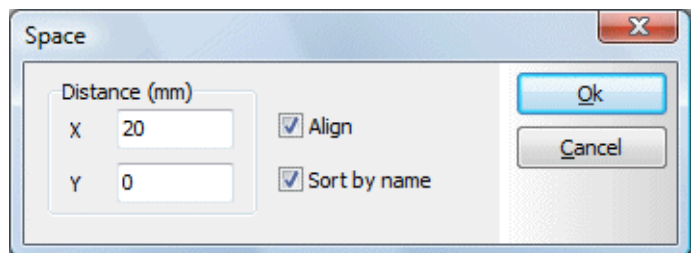
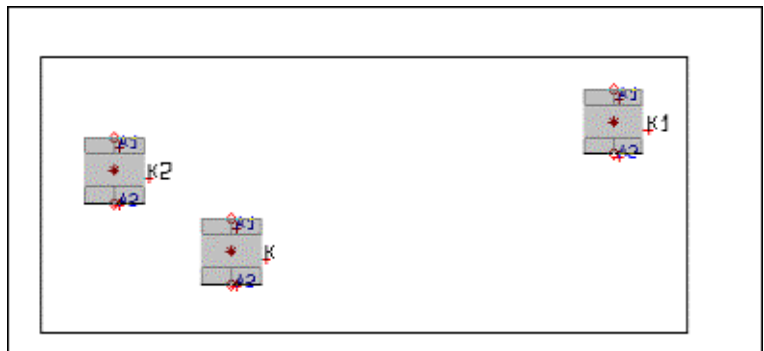
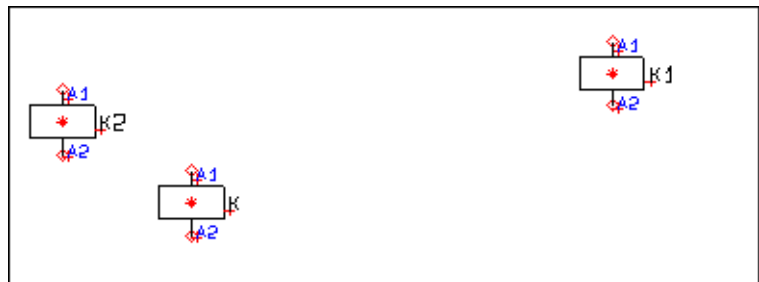
Please note the guiding text in the bottom right-hand corner of the screen.

A rubber band starting in the symbol in the middle and ending in the cross hairs is now displayed.

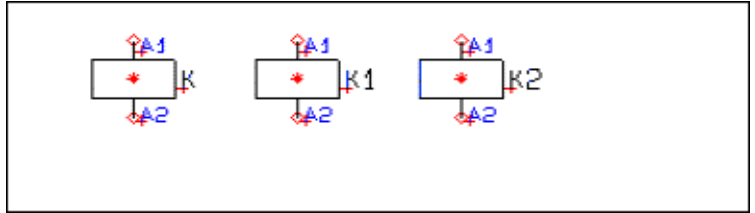
Click where you want the first of the symbols to be located.

A dialog box appear on screen, and you fill it out as shown.

Click **OK**.



The symbols are now placed aligned, with an interval of 20mm between them, sorted by name.



If you set the interval to be zero for **X** and **Y**, you will be asked to point

out where the first and last symbols have to be located.

When there is not enough space to place the symbols, you will be asked to point out a new starting point, from where the remaining symbols can be placed.

Sort by name

When you select **Sort by name** — only possible for symbols — the symbols are placed sorted by their names. If you have marked up the window around the symbols from left to right, the symbols will correspondingly be placed from left to right.

The first sorting criterion is the symbol name, then the Component Group number. If these are still identical, they are sorted according to the first connection name. — Read more about component grouping in “Component Grouping” on page 546.

MAKING OBJECTS INVISIBLE

All drawing objects — **Symbols, Lines, Texts** and **Circles/Arcs** — can be made invisible. This means that the objects will not be displayed on print, but will be displayed as dimmed, when the project pages are displayed on the screen.

THIS CHAPTER DESCRIBES	
— MAKING SYMBOLS INVISIBLE.....	(P. 62)
— MAKING LINES INVISIBLE	(P. 63)
— MAKING TEXTS INVISIBLE	(P. 63)
— MAKING CIRCLES/ARCS INVISIBLE	(P. 63)
— MAKING AREAS INVISIBLE	(P. 63)
— SUB DRAWINGS AND INVISIBLE OBJECTS	(P. 64)

Making symbols invisible

You can make symbols invisible like this:



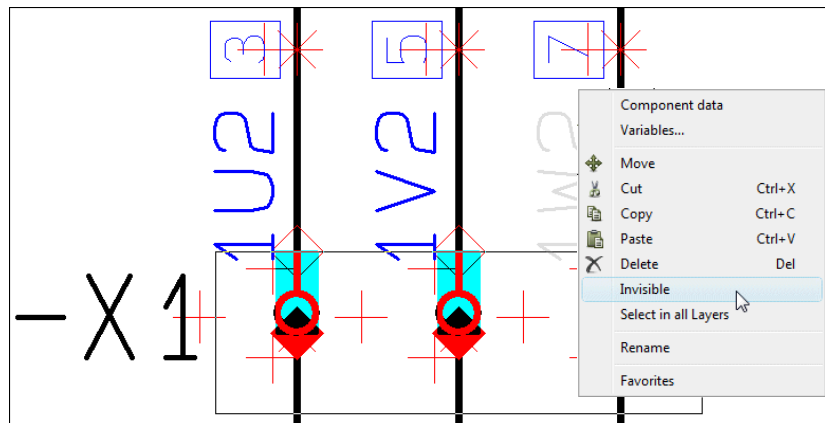
- a) Via the **Component data** dialog box: Click on the **Symbols** button, double-click on the symbol, which opens the dialog box **Component data**, where you remove the check mark in front of **Visible**, and click **OK**.

Or:



- b) By right-clicking: Click on the **Symbols** button, right-click on the symbol, and choose **Invisible**.

If you right-click after having selected an area in which both visible and invisible symbols exists, you will both be able to choose **Visible** and **Invisible**. You can then choose whether all symbols in the area shall be visible or invisible:



For symbols with reference crosses, the reference crosses also becomes invisible.

— Please read “Visible” on page 156 also.

Making lines invisible



Choose **Lines**, select an area containing the lines you wish to make invisible.

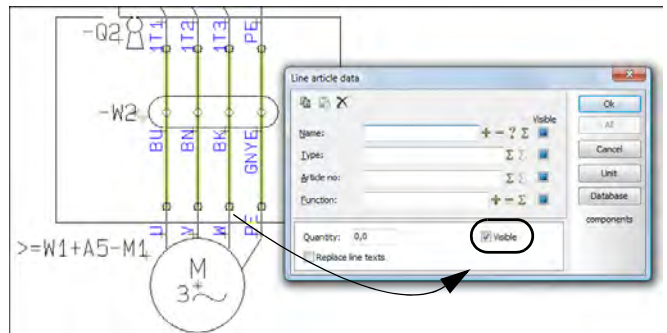
Then do the following:

- a) Via the dialog box **Line article data**: Right-click and choose **Component data**.

In the dialog box **Line article data**: Remove the **Visible** check mark, and click **OK**.

Or:

- b) Directly from the right-click menu: Right-click in the area, and click on **Invisible**.



Right-click and choose Component data

The lines are made visible again by right-clicking and then choosing **Visible**, or by entering the **Line article data** dialog box, and setting a check mark in front of **Visible**.

Making texts invisible

You can make texts invisible like this:



- a) Via the **Text properties** dialog box: Click the **Texts** button, click on the text, click on the **Text properties** button, remove the check mark in front of **Visible text**, and click **OK**. — See “Visible text” on page 121.

Or:



- b) By right-clicking: Click the **Texts** button, right-click on the text and choose **Invisible**.

Invisible texts are not displayed on screen, and can be selected by pressing the <Alt> button while clicking where the text would have been, if it was visible.



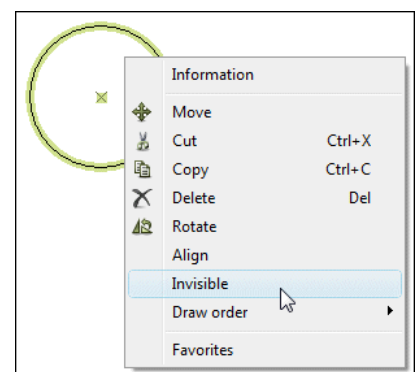
To make the text visible again, you must then click on the **Text properties** button, select **Visible text**, and click **OK**.

Making circles/arcs invisible



Choose **Circles**, right-click on the circle reference point, and choose **Invisible**.

To make the text visible again, right-click on the circle reference point again, and choose **Visible**.



Making areas invisible



- 1) Click on the **Area** button and select the area — see “Areas” on page 232.

2) Right-click in the area and choose **Invisible**.

If you right-click after having selected an area in which both visible and invisible objects exists, you will both be able to choose **Visible** and **Invisible**. Hereby you choose whether to make all objects in the area either visible or invisible. — See the image below “Making symbols invisible” on page 62.

Sub drawings and Invisible objects

When inserting sub drawings, the functions described above makes it easy to subsequently make parts of the sub drawings invisible.

— See “Sub drawings” on page 244.

LINES



When you work with lines, you always start by clicking the **Lines** button — or by using the short-cut key [I]. — See “Auto command change” on page 51 also. If you want to draw lines, you then click the **Pencil**, or right-click on the drawing area and choose **Draw Line**.

The general short-cut key for the Pencil is the [Ins] key, but you can also use the [I] key when you are working with Lines. Pressing the [Ins] or the [I] key activates/deactivates the Pencil.

THIS CHAPTER DESCRIBES	
— THE TWO TYPES OF LINES IN Automation	(P. 66)
— THE FUNCTIONS IN THE LINES TOOLBAR	(P. 66)
— DISCONNECTING DRAWN LINES	(P. 70)
— LINES IN THE PICKMENU	(P. 71)
— CONDUCTING (ELECTRICAL) LINES	(P. 71)
— ARTICLE DATA ON LINES	(P. 72)
— AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS	(P. 74)
— JUMPER LINKS	(P. 79)
— AUTOMATIC LINE DRAWING / ROUTER	(P. 81)
— EXTENDED LINES	(P. 82)
— FOLLOW CONNECTED LINES	(P. 82)
— THE TRIM LINE FUNCTION	(P. 84)
— DRAWING BY MEASURES	(P. 85)
— MEASUREMENT OBJECTS	(P. 87)
— MOUNTING CORRECT DRAWING	(P. 92)

— See how to move, copy and delete lines in “Working with selected objects” on page 51, and see how to transfer data between lines in “Transferring Data / Properties between Objects” on page 55.



AUTOMATIC LINE DRAWING

YOU CAN ALSO USE THE AUTOMATIC LINE DRAWING FUNCTION (ROUTER) IN **Automation**. THIS FUNCTION CAN BE ACTIVATED WHEN YOU ARE WORKING WITH LINES AND SYMBOLS.

— SEE “AUTOMATIC LINE DRAWING / ROUTER” ON PAGE 81.



AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS

Automation CAN INSERT ELECTRICAL POTENTIALS AT THE UPPER AND LOWER PART OF DIAGRAM PAGES AUTOMATICALLY. — SEE “AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS” ON PAGE 74.

THE TWO TYPES OF LINES IN Automation

You can draw two different types of lines in **Automation**:

- a) Conducting (electrical) lines
- b) Non conducting / free lines



If **Conducting lines** is active, the line can only be used as an electric connection. If not active, the line will be a general purpose (non conducting) line that can be used anywhere within the project.

When you click the Lines button, Conducting lines will be activated automatically.

Conducting lines are described in further detail in “Conducting (electrical) lines” on page 71.

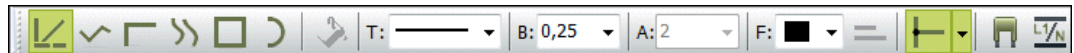


JUMPER LINKS

IT IS ALSO POSSIBLE TO DRAW JUMPER LINKS, AS DESCRIBED IN “JUMPER LINKS” ON PAGE 79.

THE LINES TOOLBAR

The toolbar for lines looks like this:



In the following you can see the functions of the buttons in the toolbar. — Please note that you can also place lines in the pickmenu, as described in “Lines in the Pickmenu” on page 71.

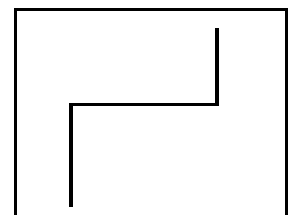
Straight lines

When you are drawing **Straight lines**, the lines are automatically drawn either right-angled or diagonally, then see “Pointer/Screen” on page 815).



Deactivate the Conducting lines button (click on the left-most button if it is highlighted), and activate the **Straight lines** button. Click where you want your line to start.

Then click once for each time you want your line to change direction, and double click to stop drawing the line — or click once and press <Esc> to stop drawing.



If the Pencil is deactivated (press <Esc>), you can change the course of the line by clicking and dragging in its terminal points, or where the line breaks.

When you try to move a line by dragging the line from a connection point, an extra line point is inserted on the line.

If you want to insert an extra line point, you can right-click on the line, and choose **Insert Line Point**.

Slanted lines

When you draw **Slanted lines**, you determine the angles of the lines yourself.

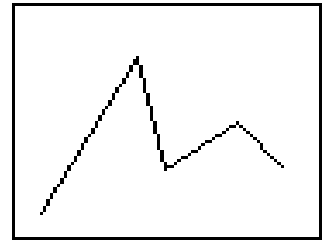


Click the **Slanted lines** button, and draw a line like the one you see here.

The line will become as you draw it, depending on the snap you are working in. — Read more about snap in “Snap” on page 42.

If the Pencil is deactivated (press <Esc>), you can change the course of the line by clicking and dragging in its terminal points, or where it changes its direction.

Breaks on the line is indicated as for straight lines.

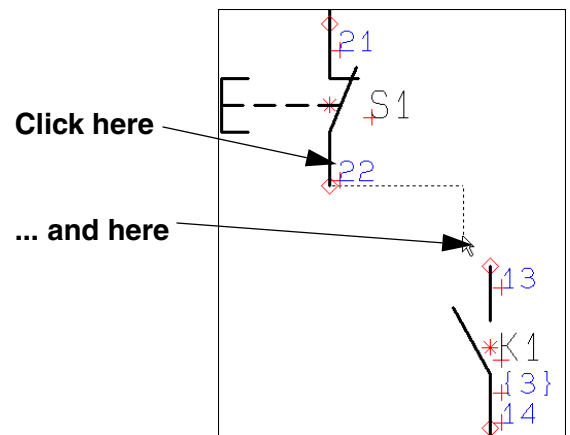


Angled lines



Drawing **Angled lines** you only have to point out where the line starts and ends. The program then creates an angled line automatically. To let the line bend in the opposite direction, press the [spacebar].

These are applied automatically in connection with mounting correct drawing. Please refer to “Mounting correct drawing” on page 92 for further details.



Curved lines

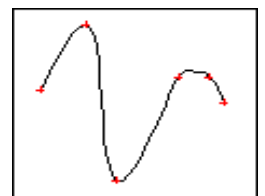


Click the **Curved lines** button.

Using this command enables you to draw curves. Click where you want the turnings of the curve. Note the marks (+) that appear on the curve when you finish drawing it.

If the Pencil is deactivated (press <Esc>), you can change the curve by clicking and dragging in its marks (+).

Curved lines can only be drawn using solid lines.



Round lines

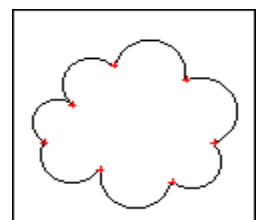


Click the **Round lines** button.

This command enables you to create coherent round lines. These can be used for corrections or for “comics bobbles.”

Round lines are half circles, drawn counter-clockwise. Round lines can only be drawn using solid lines.

As for curved lines, you can change the round lines by clicking and dragging in the marks (+).



Rectangles



Click the **Rectangle** button.

Thus you draw a rectangle: Click where you want one of the corners to be located, drag the mouse, and when the rectangle is as desired, click again. You have now drawn a rectangle.

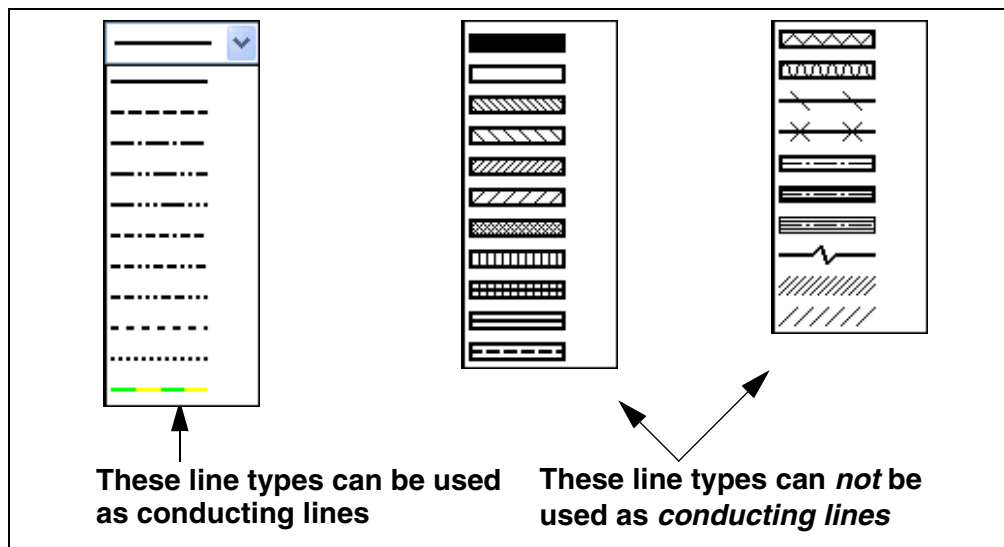
Filled areas



If you activate the **Filled areas** button *before* you start drawing, you can fill out colored areas in rectangles, circles and ovals as you draw lines. If **Filled areas** can not be selected, the button will be dimmed.

Line type: T

Click in the field specifying line types and select the type you wish to use for drawing.



— Please note that the lower of the line types to the left is yellow/green, for PE connections.

Line width: B

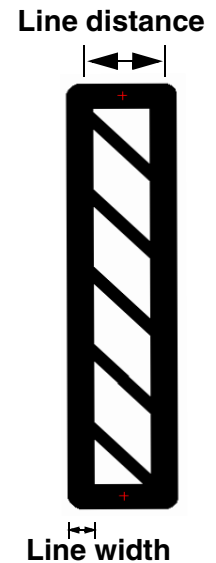
Here you determine the width of the pen used to draw the line. If the line type is hatched or similar, the line width is the width of the two borders of the line. (See the figure).

Line distance: A

For line types that are drawn with two full drawn lines (as a hatched line), you must specify the distance between these two lines. This is called the *line distance*, and it is measured from the centre of the one line to the centre of the other (See the figure).

Line color: F

When determining the color of the line, you can choose fourteen different colors. The color **NP** (Non Printable) will be shown on the screen, but will not be printed.

**Extend line**

See how to use the **Extend line** button in “Extended lines” on page 82.

Conducting lines

See how to use the **Conducting lines** button in “The two types of lines in Automation” on page 66.

Jumper links

See how to work with **Jumper links** in “Jumper links” on page 79.

Follow connected

When you are working in the Edit symbol mode, the button **Follow connected** also appears in the toolbar for Lines. Read more about this in “Follow connected lines” on page 82.

Insert potential

See how to use the **Insert potential button** for automatic insertion of electrical potentials in “Automatic insertion of electrical potentials” on page 74.

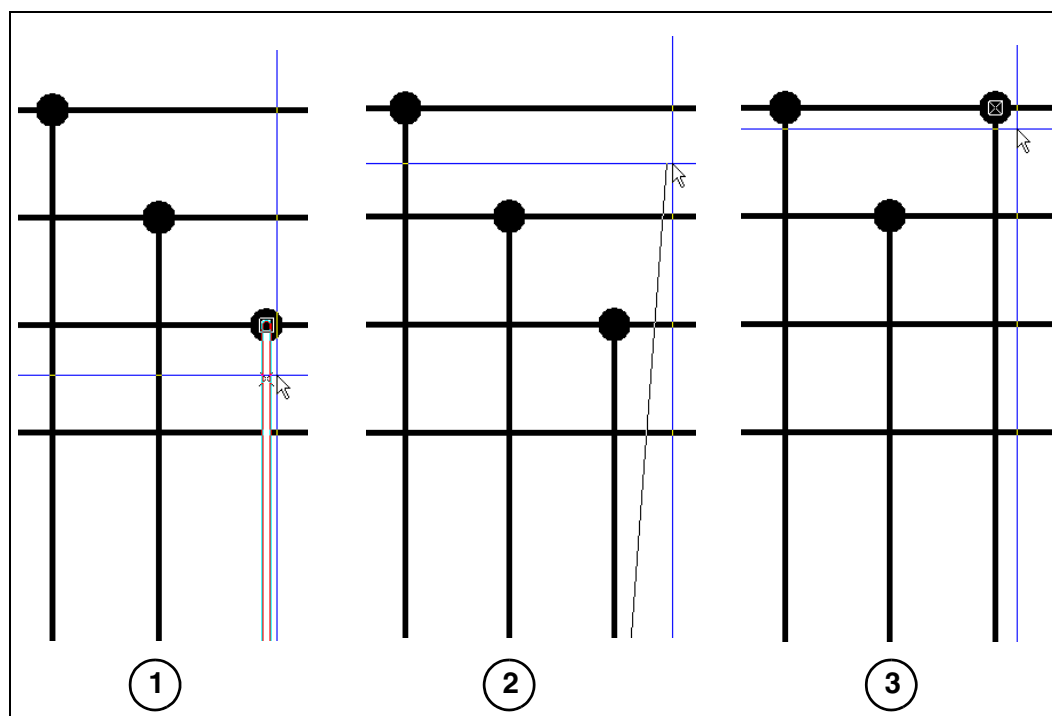
DISCONNECTING DRAWN LINES

You can disconnect a drawn line, and subsequently connect the end point of the line elsewhere in the diagram page. This can be done:

- For lines connected to symbols
- For lines connected to other lines

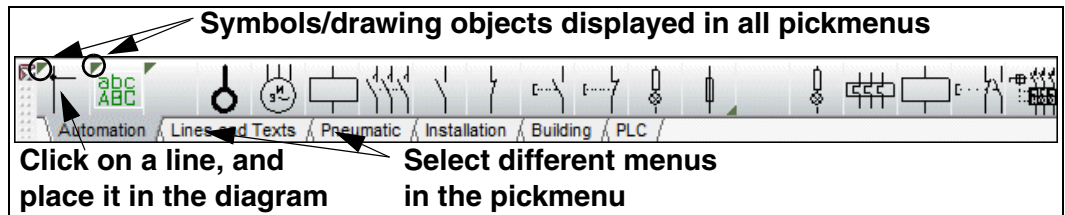
To move the end point for a drawn line, do this:

- 1) Press the <Ctrl> key while clicking on the line close to the end point. (*Do not click directly on the end point*, because this makes it unclear which of the lines meeting in the point, that you wish move).
- 2) Drag the end point to its new position.
- 3) Release the mouse button. The end point of the line is now moved.



LINES IN THE PICKMENU

To get quick access to drawing lines with predefined properties, you can place lines with the desired properties in the pickmenu at the upper part of the screen.



When you click on one of the *lines* in the pickmenu, the program automatically changes to working with **Lines** and the **Pencil** button is activated. You can then draw lines with the selected properties. — Read more about line properties in “Transferring Data / Properties between Objects” on page 55.

To insert a line type in the pickmenu, right-click on an empty cell in the pickmenu, and select **Place Line**. The currently applied line type is hereby inserted in the selected cell in the pickmenu.

If the pickmenu is not displayed on the screen, you can activate the pickmenu by selecting **Settings => Pointer/Screen** and click on **Pickmenu**.

The pickmenu can also contain symbols, texts and arcs. — Read more about the pickmenu in “Fetching symbols from the Pickmenu” on page 136 and “Adapting the Pickmenu” on page 173.

CONDUCTING (ELECTRICAL) LINES

A conducting line is an electrical line. It can start and end in electrical points only. An electrical point can be another conducting line, a connection point on a symbol or a signal.



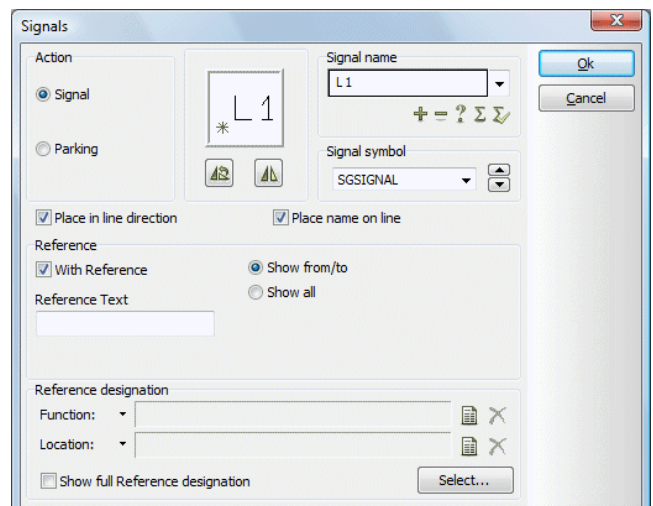
To specify that a line has to be conducting, you must activate the **Conducting lines** button.

If you try to start or end a conducting line without specifying a electrical connection, you will be asked about a signal name or whether the line should be parked.

A *signal symbol* with a *signal name* signifies an electrical connection.

This means that all electrical points with identical *signal names*, are the same electrical potential.

— Read more about signal symbols in “Signal symbols” on page 196, and about reference designations on signal symbols in “Variant names on signal symbols” on page 202.



Parking

If you want to wait to decide what the conducting line is to be connected to, you choose **Parking**. Thus the line gets no electrical connection. However, this is just a temporary solution, and there should never be parked lines in a completed project.

Warning when connecting non conducting lines to symbols

If you connect a non conducting line with a symbol, you will be warned about this. You will be allowed to do so if you insist, but the program draws no connection dot.

Show conducting

At any time you can choose **Functions => Show conducting** to see which lines are conducting. Conducting lines turn green, and non conducting lines turn red. To disable this function choose **Functions => Show conducting** once more.

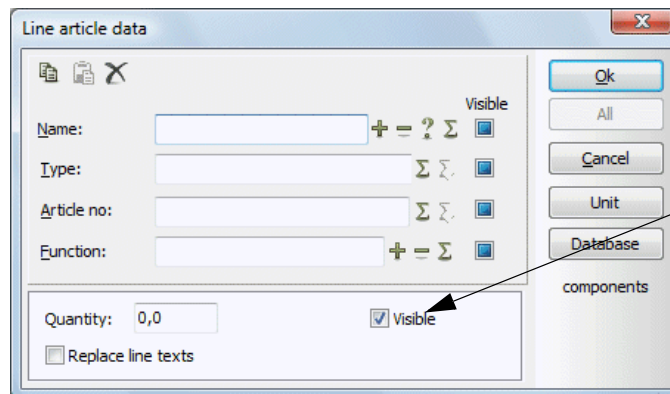
Transferring data for conducting lines

Please note that non conducting lines can become electrical by using the transfer data function.

— Read more about this in “Transferring Data / Properties between Objects” on page 55.

ARTICLE DATA ON LINES

When you wish to place article data on a line, deselect the Pencil and double-click on the line — or right-click on the line, and select **Line article data**. You then enter the dialog box **Line article data**:



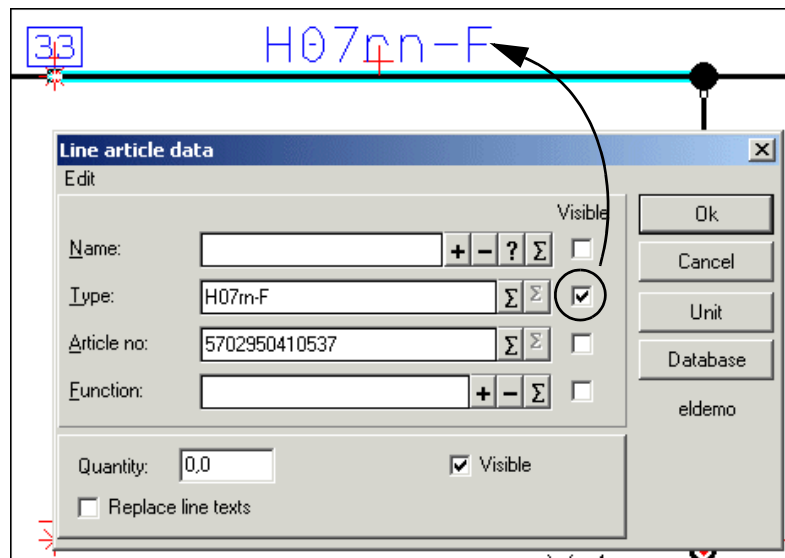
Specify if the line is *visible*

Here you can for instance click on **Database**, to fetch article data from the database — see “**Using the database**” on page 561, in particular “**Using the database for Lines**” on page 561.

— Please note that you can also attach article data to cable symbols, as described in “Working with cable symbols” on page 210.

Visible Line article data

When you select **Visible** for one of the article data fields, the contents of this field is displayed on the diagram, as illustrated for the **Type** field in the figure below.



When article data for a line is set to be visible, these are placed above the middle of the longest segment on the line. If this segment later disappears — e.g. by deletion or because a symbol is placed —, the text is made invisible.

You can move these article data texts, as you can move any other text in the program.

If you select **Replace line texts**, the texts are placed where the program would place these.

Pop-up information with Line article data

If **Settings => Pointer/Screen => Show popup information** is selected, article data for lines are displayed when the cursor is placed over lines — when the Pencil is not activated.

Datafields for Line article data

In all electrical lists — this meaning terminal, cable, connection and PLC lists — you can insert datafields with **Line-name**, **Line-type**, **Line-article no.** and **Line-function**. — See “Datafields” on page 834.

AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS

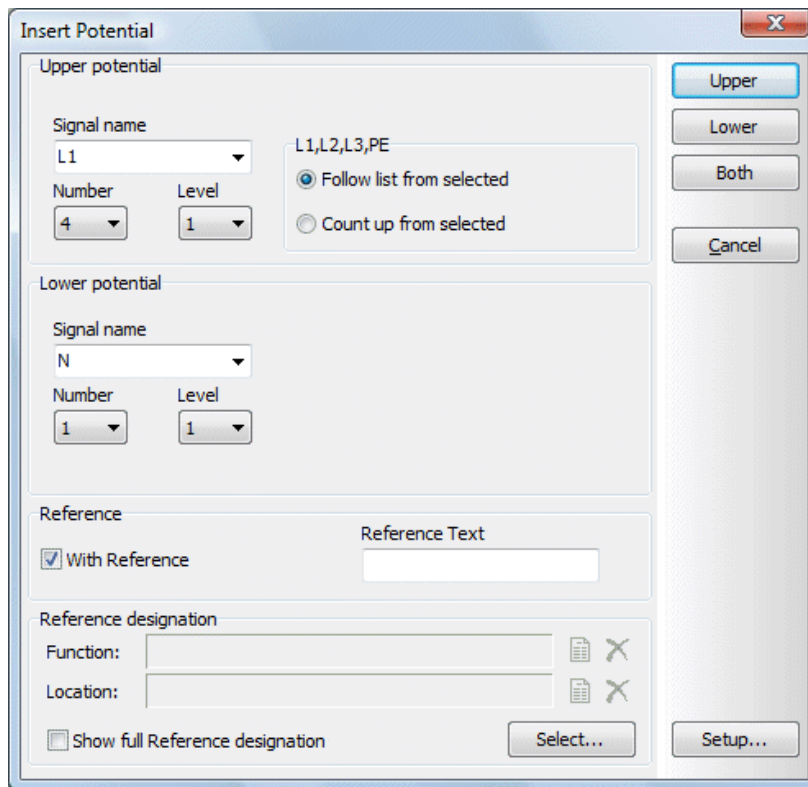
THIS CHAPTER DESCRIBES	
— INSERTING ELECTRICAL POTENTIALS AUTOMATICALLY	(P. 74)
— PLACING ELECTRICAL POTENTIALS ON THE UPPER PART OF THE PAGE.....	(P. 75)
— PLACING ELECTRICAL POTENTIALS ON THE LOWER PART OF THE PAGE	(P. 75)
— POSITION FOR THE ELECTRICAL POTENTIALS.....	(P. 76)
— REFERENCE TEXTS FOR AUTOMATICALLY PLACED ELECTRICAL POTENTIALS..	(P. 76)
— REFERENCE DESIGNATIONS FOR AUT. PLACED ELECTRICAL POTENTIALS	(P. 76)
— SETTINGS FOR AUTOMATICALLY PLACED ELECTRICAL POTENTIALS	(P. 76)
— INSERTING SIGNAL NAMES IN THE SIGNAL NAME LIST	(P. 78)

Inserting electrical potentials automatically



When you wish to let **Automation** insert electrical potentials automatically, click the **Insert Potential** button, or choose **Insert => Insert Potential**.

You then enter the dialog box **Insert potential**, where you specify which electrical potentials to insert:



From this dialog box you can place one or more electrical potentials, either on the upper or the lower part of the page — or both at the same time:

Inserting electrical potentials on the upper part of the page

In the **Number** field in the dialog box, you specify the number of potentials you wish to place at the upper part of the page.

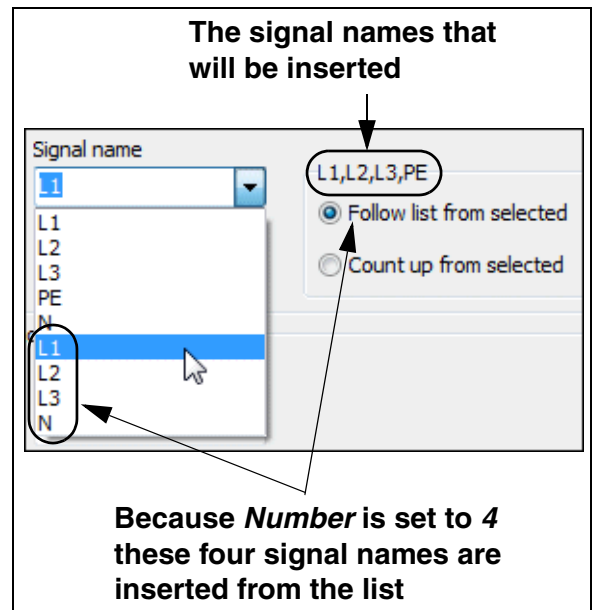
Subsequently, you can specify which signal names to insert.

You have two options:

- a) All signal names are fetched from the **Signal name** list: Select a signal name below **Signal name**, and click on **Follow list from selected**. The names for the selected **Number** of potentials, are selected from the **Signal name** list, starting from the selected signal name. If

Number is set to 4, and you have clicked on **L1** — as in the figure above — the potential names **L1**, **L2**, **L3** and **N** are inserted.

- b) The signal names are incremented starting with the selected **Signal name**: Choose a signal name under **Signal name**, and click on **Count up from selected**. The name on the *first* signal name is then selected from the **Signal name** list, and the subsequent names are incremented by one, as they are placed. If **Number** is set to 4, and you have clicked on **L1**, the electrical potentials **L1**, **L2**, **L3** and **L4** are inserted.



To only place electrical potentials on the upper part of the page, click the **Upper** button in the right-hand side of the dialog box.

— Please also refer to “Specify Level for the electrical potentials” on page 76, “Reference text for automatically placed electrical potentials” on page 76, “Reference designations for automatically placed electrical potentials” on page 76, “Settings for automatically placed electrical potentials” on page 76 and “Signal names on the Signal name list - for automatically inserted electrical potentials” on page 78.

Inserting electrical potentials on the lower part of the page

In the **Number** field below **Lower potential** in the dialog box, you specify the number of potentials you wish to place on the lower part of the page.

You can then specify which signal names you wish to apply, following the same principles as described in "Inserting electrical potentials on the upper part of the page" på side 75.

To only insert potentials on the lower part of the page, click the **Lower** button in the right-hand side of the dialog box.

If you wish to insert potentials on the upper and the lower part of the page in the same operation, click the **Both** button.

— See also “Specify Level for the electrical potentials” on page 76, “Reference text for automatically placed electrical potentials” on page 76, “Reference designations for automatically placed electrical potentials” on page 76, “Settings for automatically placed electrical potentials” on page 76 and “Signal names on the Signal name list - for automatically inserted electrical potentials” on page 78.

Specify Level for the electrical potentials

If it says **1** in the **Level** field, the first electrical potential is placed where the function is set up to place the first electrical potential.

If it e.g. says **3** in the **Level** field, the potentials are placed starting from where the function would have placed the third potential.

— See “Settings for automatically placed electrical potentials” on page 76.

Reference text for automatically placed electrical potentials

To use references for the inserted electrical potentials, select **With reference**, and type in a reference pretext in the **Reference text** field — if desired.

Reference designations for automatically placed electrical potentials

If you wish to attach reference designations to the automatically inserted electrical potentials, click on **Select** below **Reference designations**, and choose a reference designation as described in “Choosing reference designations when placing symbols” on page 337.

If you want to display the full reference designation for the potentials, select **Show full Reference designation**, as described in “Show full Reference designation” on page 338.



THE SELECTED LINE TYPE IS USED

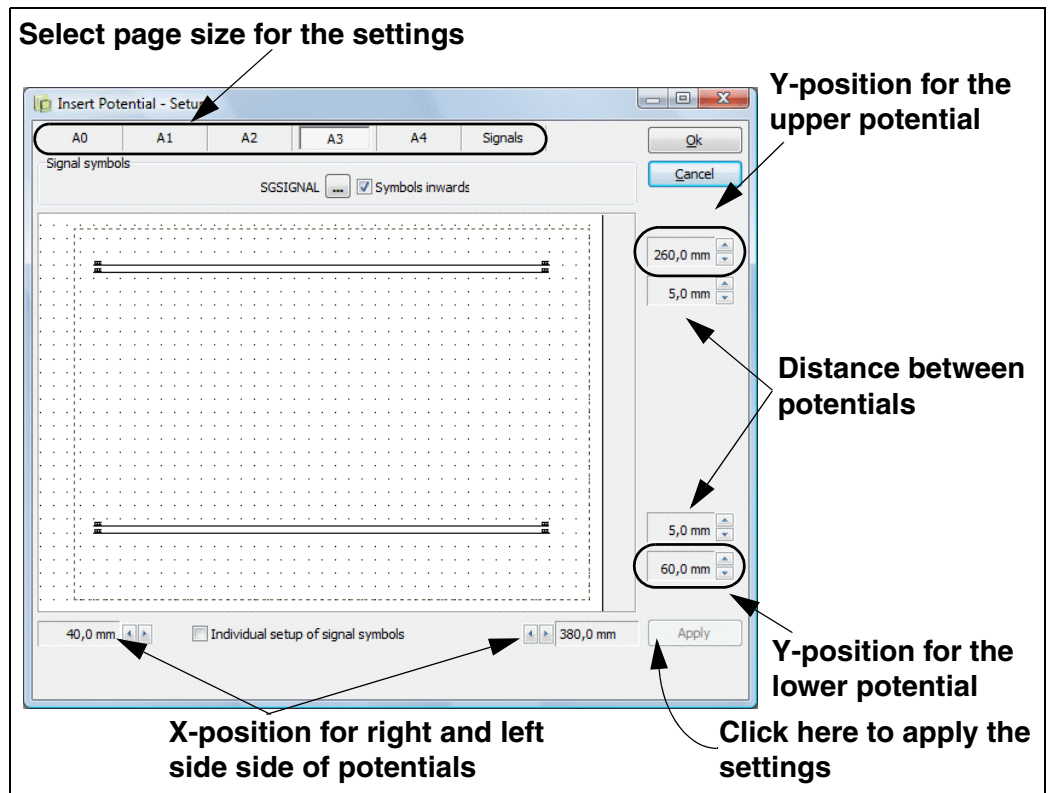
WHEN INSERTING ELECTRICAL POTENTIALS AUTOMATICALLY, THE LINE TYPE YOU HAVE SELECTED BEFORE CHOOSING **INSERT => INSERT POTENTIAL** IS USED FOR DRAWING THE POTENTIALS.

— SEE “LINE TYPE: T” ON PAGE 68.

Settings for automatically placed electrical potentials

The placing positions for the automatically placed electrical potentials can be set up individually for each individual page size.

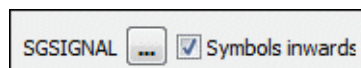
This can be done by choosing **Insert => Insert Potential**, and then click on **Setup**. You then enter the dialog box **Insert potential - Setup**:



Here you choose — at the top — which page size you wish to edit the settings for, make the changes, click on **Apply**, and then click **OK**.

Symbols inward

If you want to place the text for the signal symbols above the electrical potentials, click on **Symbols inward**.



Choosing signal symbols

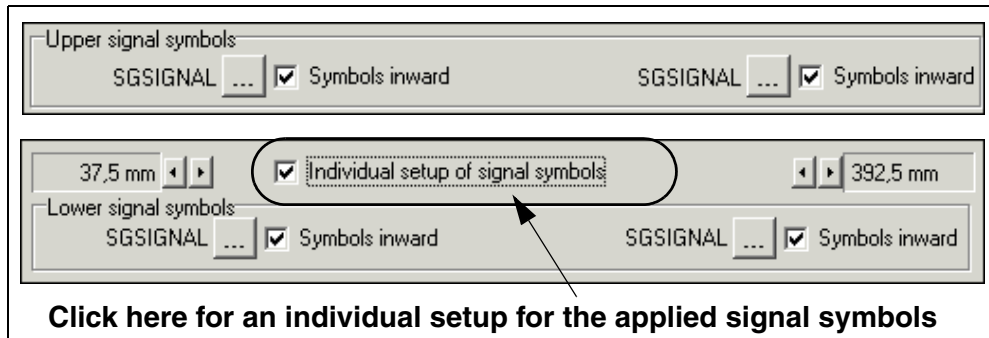


To select which signal symbol to apply, click the **SGSIGNAL** button, and you enter the **Symbol menu**, where you select the symbol.

Individual setup of signal symbols

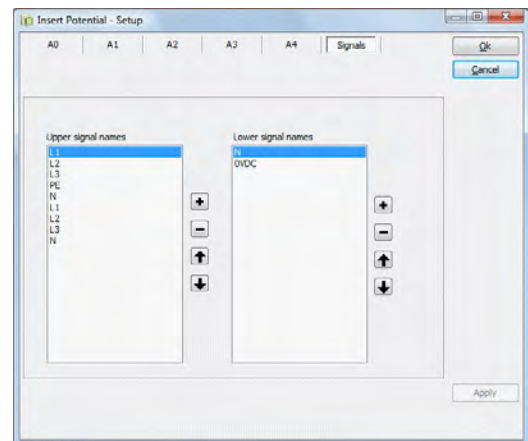
If you wish to use an individual setup for the applied signal symbols, click on **Individual setup of signal symbols**.

The dialog box then changes appearance, and you get the option to make an individual **Symbol inward** setup, and to choose signal symbol for all four possible positions for the signal symbols:



Signal names on the Signal name list - for automatically inserted electrical potentials

To specify which potential names that appears in the **Signal name** list — see “Inserting electrical potentials on the upper part of the page” on page 75 —, you must choose **Insert => Insert potential**, click on **Setup**, and here click on the tab **Signals**.



Here the columns **Upper signal names** and **Lower signal names** appears, for specifying the signal names in the **Signal name list** for the upper and lower electrical potentials respectively.

Edit the lists as described below, and then click **Apply**.



ADDING A SIGNAL NAME

Click on the **Plus** button, and type in a new signal name.



REMOVING A SIGNAL NAME

Click on the signal name in the list, and click the **Minus** button.



MOVING A SIGNAL NAME IN THE LIST

Click on the signal name in the list, and click the **Up / Down** arrows, until the name has the desired position in the list.

JUMPER LINKS

THIS CHAPTER DESCRIBES	
— ON DRAWING JUMPER LINKS	(P. 79)
— DRAWING JUMPER LINKS	(P. 79)
— JUMPER LINKS IN LISTS AND TERMINAL PLANS	(P. 80)

On drawing Jumper links

In **Automation** it is possible to work with jumper links. Succeedingly, these jumper links can be included in terminal lists and terminal plans automatically. Jumper links are *not* assigned wire numbers during automatic wire numbering.

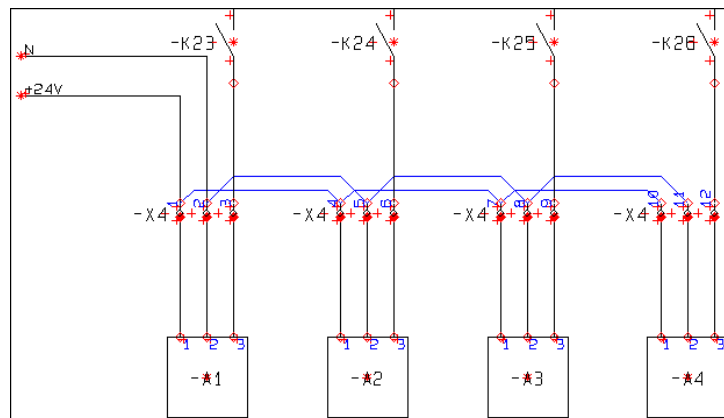
Drawing jumper links

To draw jumper links, do the following:



- 1) Click on the **Lines** button, and click on the **Pencil**.
- 2) Click on the **Jumper links** button, and draw the jumper links.

Jumper links can be drawn from all types of connection points. Jumper links can not start on the middle of another line, or on the middle of another jumper link.



When you start and end drawing a jumper link, a short message is displayed, informing you that you are drawing a jumper link.

Jumper lines in the pickmenu Lines and Texts

You can also place jumper links by clicking on the tab for the pickmenu **Lines and Texts**, and click on one of the jumper line symbols in the pickmenu.



The dot in the middle of the jumper line symbols displays the colour of the jumper lines.



THE LINE SETTINGS ARE MEMORIZED

THE PROGRAM AUTOMATICALLY USE THE LINE SETTINGS, WHICH YOU USED THE LAST TIME YOU WERE DRAWING JUMPER LINKS.

WHEN YOU FINISH DRAWING JUMPER LINKS, THE PROGRAM AUTOMATICALLY CHANGE BACK TO THE LINE SETTINGS, WHICH YOU USED BEFORE DRAWING JUMPER LINKS.

— READ MORE ABOUT LINE SETTINGS IN “THE LINES TOOLBAR” ON PAGE 66.

Changing jumper links to electrical lines - or the reverse

You can change a jumper link to an electrical line — or change an electrical line to a jumper link — by transferring line properties, as described in “Transferring Object properties between Objects” on page 56.

Show jumper links

When you choose **Functions => View Jumper Links**, all jumper links are displayed as green lines on the screen, while everything else are displayed in red. To disable the function, choose **Functions => View Jumper Links** once again.

Jumper links in Lists and Terminal plans

Under **Functions => Datafield**, you will now find the datafield **Terminals list => Jumper link**.

This datafield is displayed graphically in terminals lists. The result of placing this datafield can be seen in the list to the right.

When the datafield is placed on the left-hand side of the terminals, you must make it right-adjusted, so that it expands to the left.

When the datafield is placed on the right-hand side of the terminals, you must correspondingly make it left-adjusted, so that it expands to the right.

Jumper links in terminal plans

Jumper links can also be included in terminal plans. Here a special terminal symbol for displaying jumper links have been designed.

Read more about this in the **PC|SCHEMATIC Tools** manual.

External	Terminal	Intern
Name		Name
-A1:1	-X4:1	-K1:2
-A1:2	-X4:2	
-A1:3	-X4:3	-K23:2
-A2:1	-X4:4	
-A2:2	-X4:5	
-A2:3	-X4:6	-K24:2
-A3:1	-X4:7	
-A3:2	-X4:8	
-A3:3	-X4:9	-K25:2
-A4:1	-X4:10	
-A4:2	-X4:11	
-A4:3	-X4:12	-K26:2

Place right-adjusted data fields here

Place left-adjusted data fields here

AUTOMATIC LINE DRAWING / ROUTER

When you place symbols in a project, you can make **Automation** draw the symbol connections automatically.

When drawing lines, you can also let the program do a part of the job for you.



SHORT-CUT KEYS FOR OPTIMAL USE OF THE ROUTER

TO MAKE THE ROUTER FUNCTION THE BEST, YOU MUST CREATE SHORT-CUTS KEYS FOR THE ROUTER FUNCTIONS YOU USE THE MOST.

SEE "USER DEFINED SHORT-CUT KEYS" ON PAGE 771.

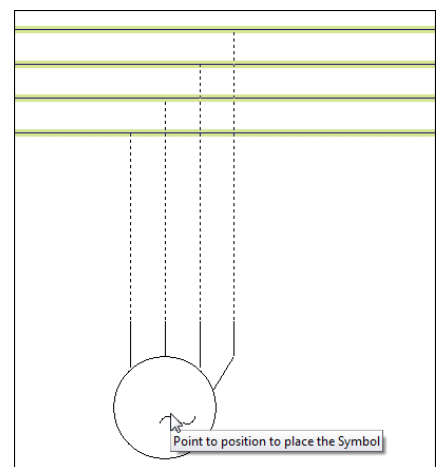
Automatic line drawing when placing symbols



To use the router when placing **Symbols**, you first choose **Router => Active**, to set a check mark in front of **Active**.

To deactivate the router, choose **Router => Active** again, so that the check mark disappears.

You will find the following options in the **Router** menu:



ROUTER =>	FUNCTION
ACTIVE [½]	STARTS AND STOPS THE ROUTER
ONLY NOT CONNECTED SYMBOLS BY MOVE	THE ROUTER IS ACTIVATED WHEN A SYMBOL IS MOVED, IF THE SYMBOL IS NOT CONNECTED.
ONLY IF ALL IN DIRECTION POSSIBLE	THERE WILL ONLY BE ROUTED WHEN ALL CONNECTIONS IN THE CHOSEN DIRECTION CAN BE ROUTED.
ONLY TO NEAREST DIRECTION [2]	THERE WILL ONLY BE ROUTED FROM THE SIDE OF THE SYMBOL, WHICH IS CLOSEST TO A LINE OR OTHER CONNECTION POINTS.
HORIZONTAL	THERE WILL ONLY BE ROUTED HORIZONTALLY.
LEFT	THERE WILL ONLY BE ROUTED TO THE LEFT.
RIGHT	THERE WILL ONLY BE ROUTED TO THE RIGHT.
VERTICAL	THERE WILL ONLY BE ROUTED VERTICALLY.
UP	THERE WILL ONLY BE ROUTED UPWARDS.
DOWN	THERE WILL ONLY BE ROUTED DOWNWARDS.
REVERSE ORDER [3]	CHANGES THE ROUTING ORDER, WHEN ROUTING TO MORE THAN ONE LINE.
SKIP OVER LINE [<]	THE ROUTER SKIPS A LINE AND ROUTES TO THE NEXT LINE.
ROUTE [1]	IF AN ALREADY PLACED SYMBOL IS SELECTED, THE SYMBOL CAN BE ROUTED.

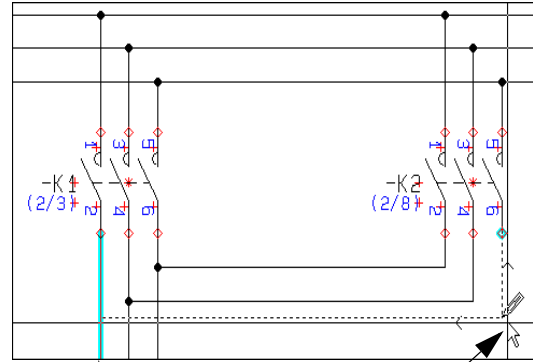
The router for lines



To apply the router for drawing **Lines** choose **Router => Active**, to set a check mark in front of **Active**.

To disable the router choose **Router => Active** again, and the check mark disappears.

You have the following options in the **Router** menu for lines:



- 1) Place the cursor here
- 2) Press the [**<**] button until the desired line is selected
- 3) Click to draw the connection

ROUTER =>	FUNCTION
ACTIVE [½]	STARTS AND STOPS THE ROUTER.
ANGLED [CTRL-SPACE-BAR]	CHANGES BETWEEN STRAIGHT AND ANGLED LINES
HORIZONTAL LINES	ROUTES ONLY HORIZONTAL LINES.
VERTICAL LINES	ROUTES ONLY VERTICAL LINES.
UP/LEFT	ROUTES UP/LEFT WHEN WORKING WITH ANGLED LINES.
DOWN/LEFT	ROUTES DOWN/LEFT WHEN WORKING WITH ANGLED LINES.
DOWN/RIGHT	ROUTES DOWN/RIGHT WHEN WORKING WITH ANGLED LINES.
UP/RIGHT	ROUTES UP/RIGHT WHEN WORKING WITH ANGLED LINES.
SKIP OVER LINE [<]	THE ROUTER SKIPS A LINE AND ROUTES TO THE NEXT LINE.

Pressing the spacebar changes the direction of the router.

EXTENDED LINES

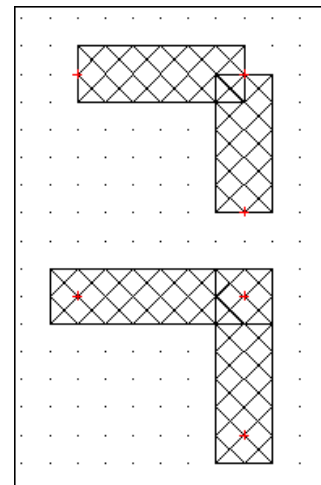


On all types of lines, you can determine that they shall be extended. Select the **Extend line** button on the toolbar. The line will now start half a line width before its starting point, and end half a line width after its ending point.

The lower example on the figure is drawn using **Extend line**.

When you draw continuously without stopping when the line breaks, extend lines will be chosen automatically.

The upper example is drawn as two independent lines, without selecting Extend line.



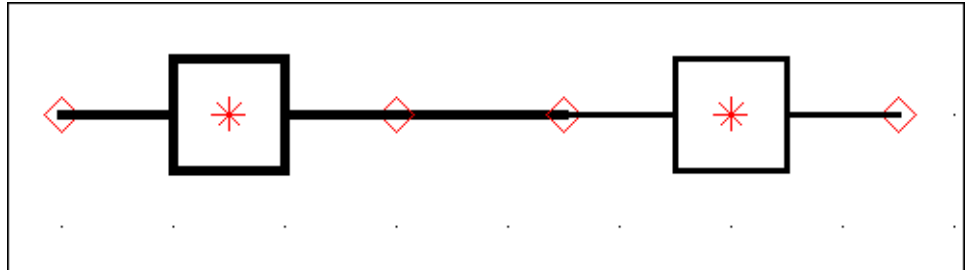
FOLLOW CONNECTED LINES

This button exists in Edit symbol only. If the symbol is designed using Follow connected lines, the lines in the symbol will automatically get the same width and color as the conducting lines that are connected to the symbol in the project.



In the symbol to the left all lines are drawn using **Follow connected**. Therefore all lines in the symbol gets the same width and color as the conducting line, that can be seen between the connection points of the two symbols. The other symbol is designed without Follow conducting.

A symbol designed using Follow connected, adapt to the last line that was connected to the symbol.



Follow connected can be deactivated as a general setting

Follow connected on symbols can be activated/deactivated by choosing **Settings => Text/Symbol defaults**, and then activate/deactivate **Activate Follow connected on symbols**.

— See “Text/Symbol defaults” on page 831, especially “Activate Follow connected on symbols” on page 832.

THE TRIM LINE FUNCTION

The **Trim line** command can be used for two purposes:

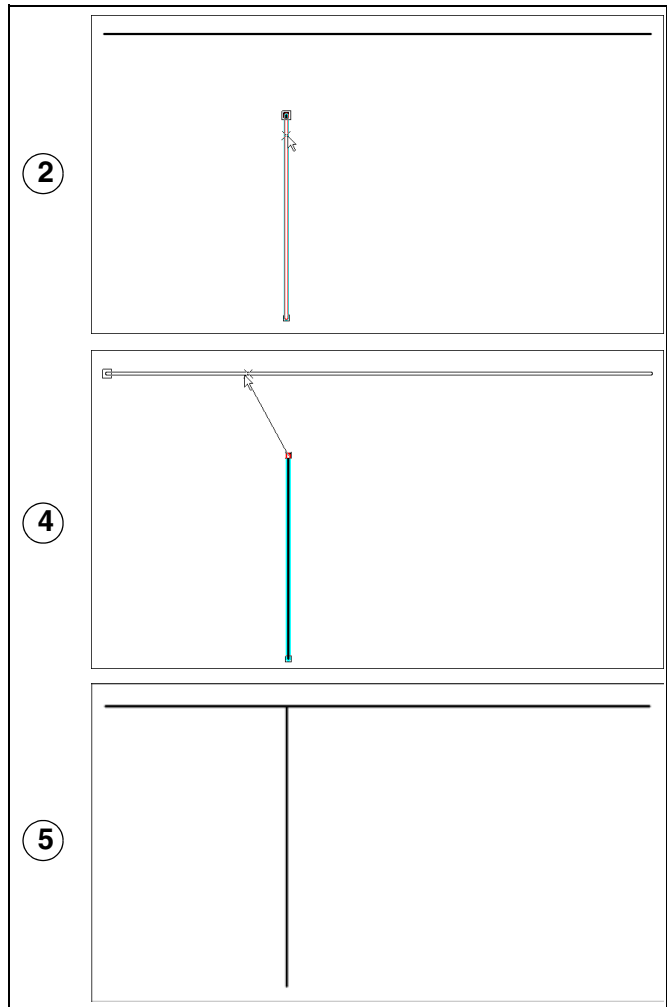
- 1) To extend a line, so that it meets another line.
- 2) To truncate a line, so that it meets another line.

Extend a line using Trim line

Do this to extend the vertical line (in the figure), so that it meets the horizontal line:

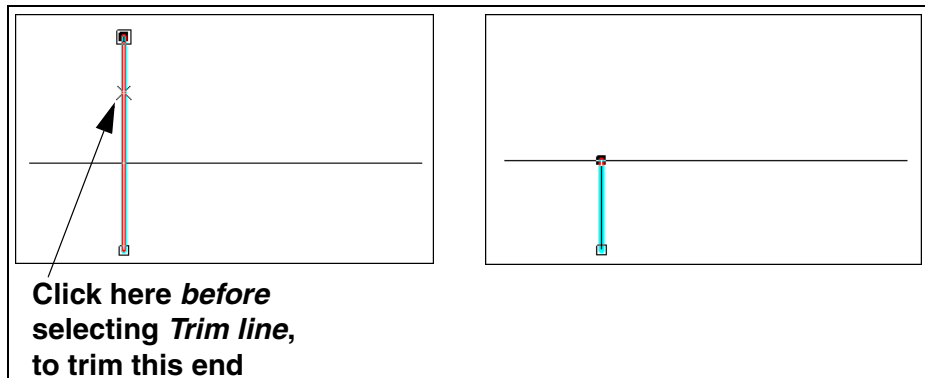


- 1) Click the **Lines** button, and press <Esc> to disable the Pencil.
- 2) Select the vertical line by clicking on it.
- 3) Choose **Edit => Trim line**.
- 4) A thin line starting from the selected symbol and ending in the cross hairs is now displayed: Click on the horizontal line.
- 5) The vertical line is extended, so that it meets the horizontal.



Trimming the selected end

Before applying the **Trim line** command, please click closest to that end of the line, which you wish to trim:



DRAWING BY MEASURES

It is also possible to type in the exact measures of what you want to draw.

In the status line at the bottom of the screen, you can see the X-Y co-ordinates for the cursors location on the screen.

The co-ordinates to the left state the absolute location, while the co-ordinates to the right states the co-ordinates relative to where you last clicked (also called the *relative* co-ordinates).

x=82,50 y=117,50mm x=37,50 y=20,00mm

Click on the X-Y co-ordinate field in the status line, or use the short-cut [Ctrl+i]. You now see a dialog box for typing in co-ordinates.

You can also enter this dialog box choosing **Functions => Coordinates**.

When you want to draw by measures, you have three options at hand:

Absolute coordinates

The measures are typed in referring to the origo of the page, located in the lower-left corner of the page.

Relative coordinates

The measures are typed in referring to the point last chosen on the page.

Polar coordinates

Here you draw by determining the length of the line, and the angle between the line and a horizontal line.

As an example of how to use absolute and relative co-ordinates, you can now draw a rectangle of 100 x 50 mm in different ways.



Click the **Lines** button, select line type, deactivate Conducting lines, and activate the **Pencil**.

Using Absolute coordinates

Click on the X-Y co-ordinate field — or press [Ctrl+i] — and choose **Absolute**.

Type in the X and Y co-ordinates, to state where the line shall start in relation to the origo of the page.

Start in the point 50, 100.

In the X field you type 50, in the Y field 100. You do not have to state it is in millimetres, due to the system settings. Click **OK**. (This setting can be changed in **Settings => Screen/Pointer**. Read more about this in “Pointer/Screen” on page 815).

Click on the X-Y co-ordinate field: Set X to 150 and Y to 100. Click **OK**.

Click on the X-Y co-ordinate field: Set X to 150 and Y to 150. Click **OK**.

Click on the X-Y co-ordinate field: Set X to 100 and Y to 150. Click **OK**.

Click on the X-Y co-ordinate field: Set X to *100* and Y to *100*. Click **OK**.

Press <Esc>. The rectangle is complete.

Using Relative coordinates

Start the line by clicking somewhere on the screen.

Click on the X-Y co-ordinate field — or press [Ctrl+i] — and choose **Relative**.

Press [Ctrl+i]: Set X to *100* and Y to *0*. Click **OK**.

Press [Ctrl+i]: Set X to *0* and Y to *50*. Click **OK**.

Press [Ctrl+i]: Set X to *-100* and Y to *0*. Click **OK**.

Press [Ctrl+i]: Set X to *0* and Y to *-50*. Click **OK**.

Press <Esc>. The rectangle is complete.

Using the Rectangle command with relative coordinates



Click on the **Lines**, the **Rectangle** and the **Pencil** buttons.

Start the rectangle by clicking where you want one of the corners of your rectangle.

Click on the X-Y co-ordinate field — or use short-cut [Ctrl+i] — and choose **Relative**.

To specify the opposite corner of the rectangle, you set X to *100* and Y to *50*.

Click **OK**.

The rectangle is now complete.



MEASUREMENT OBJECTS

THIS CHAPTER DESCRIBES	
— INSERTING MEASUREMENT OBJECTS	(P. 87)
— FREE MEASUREMENT	(P. 89)
— ROTATING MEASUREMENT OBJECTS	(P. 89)
— EDITING THE MEASUREMENT START AND END POINTS	(P. 90)
— EDITING MEASUREMENT DATA	(P. 91)
— MEASUREMENT OBJECTS AND LAYERS	(P. 91)
— DELETING MEASUREMENT OBJECTS	(P. 91)



MEASUREMENT ONLY ON GROUND PLAN / MECHANICAL PAGES

YOU CAN ONLY INSERT MEASUREMENT OBJECTS ON PAGES WITH THE PAGE TYPE **GROUND PLAN / MECHANICAL**.

— SEE “PAGE TYPE” ON PAGE 501.

Inserting measurement objects

To place measurement objects, which can place and calculate measurements between two points, choose **Functions => Measurement**.

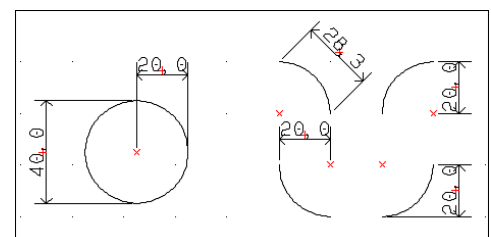
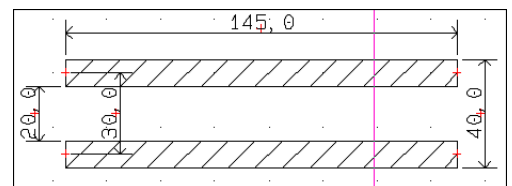
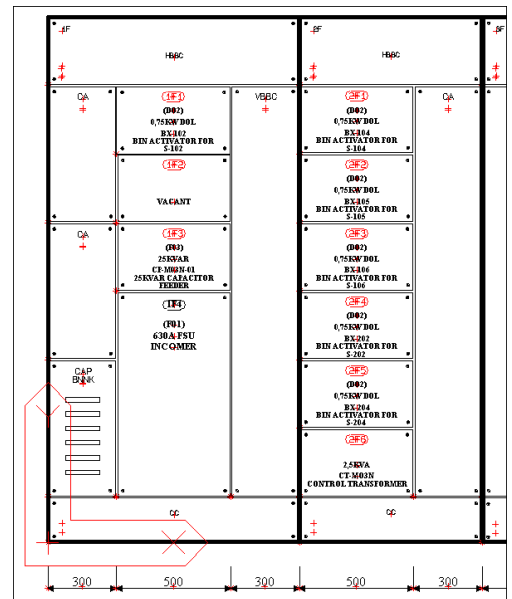
The starting point for the measurement object

You now get a pencil in the cross hairs, which tells you, that you can now point out the starting point of the measurement object.

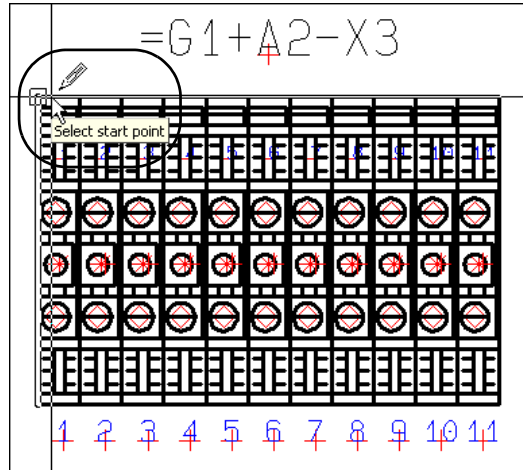
If you want to start the measurement from a symbol — e.g. from the symbol reference point or from a corner point on the symbol —, the program automatically examines the symbol definition, and finds the exact coordinates from this point.

If filled lines are applied, you can both measure from the center of the lines, and from the edges of the lines.

Circles can furthermore be measured at intervals of 90 degrees — see the figure to the right.



As displayed in the figure below, the program shows exactly which point you are selecting:



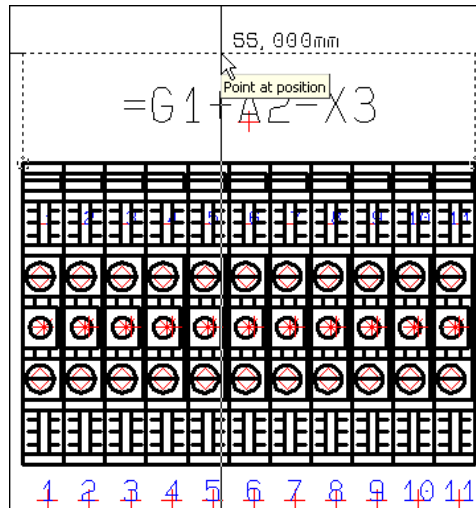
Click to point out the start point for the measurement object.

— When you wish to place a measurement object, which apparently does not relate to the placed objects on the drawing, please refer to “Free measurement” on page 89.

Pointing out the end point for the measurement object and place the measurement

Now click where you wish to place the end point for the measurement object.

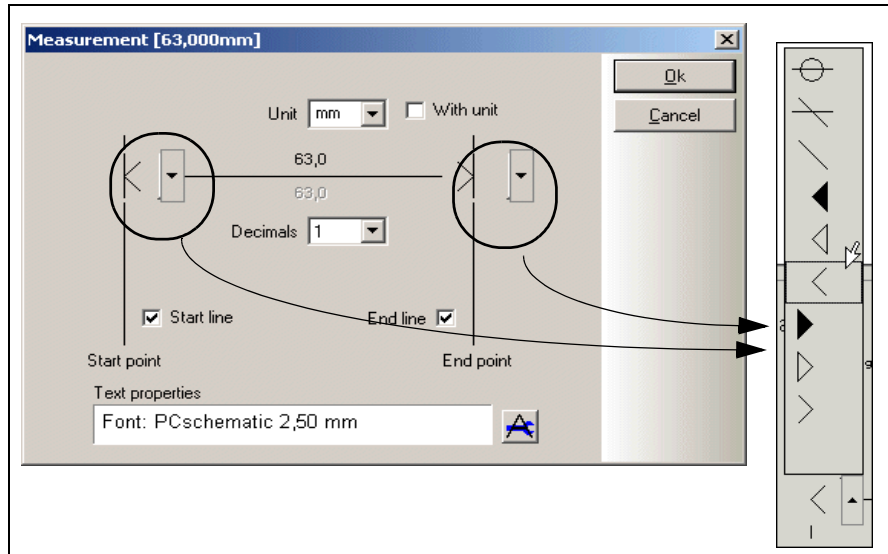
You now get the text containing the measurement itself in the cross hairs. — When start and end points does not have the same vertical or horizontal coordinates, you can rotate the measurement, as described in “Rotating the measurement object with the [Spacebar]” on page 89.



Click to place the measurement object.

Specifying the Measurement data

You now enter the dialog box **Measurement**:

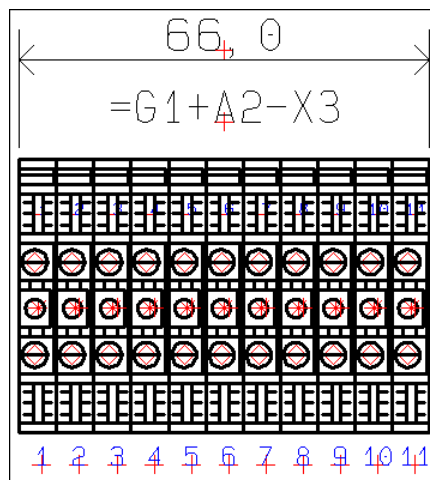


Here you can specify the appearance of the measurement object, whether to attach a unit specification to the measurement, and how many decimals to apply.



At the bottom of the dialog box, you can choose which font you wish to use for the measurement texts, by clicking the **Text properties** button.

Click **OK**, and the measurement is placed on the drawing:



Free measurement

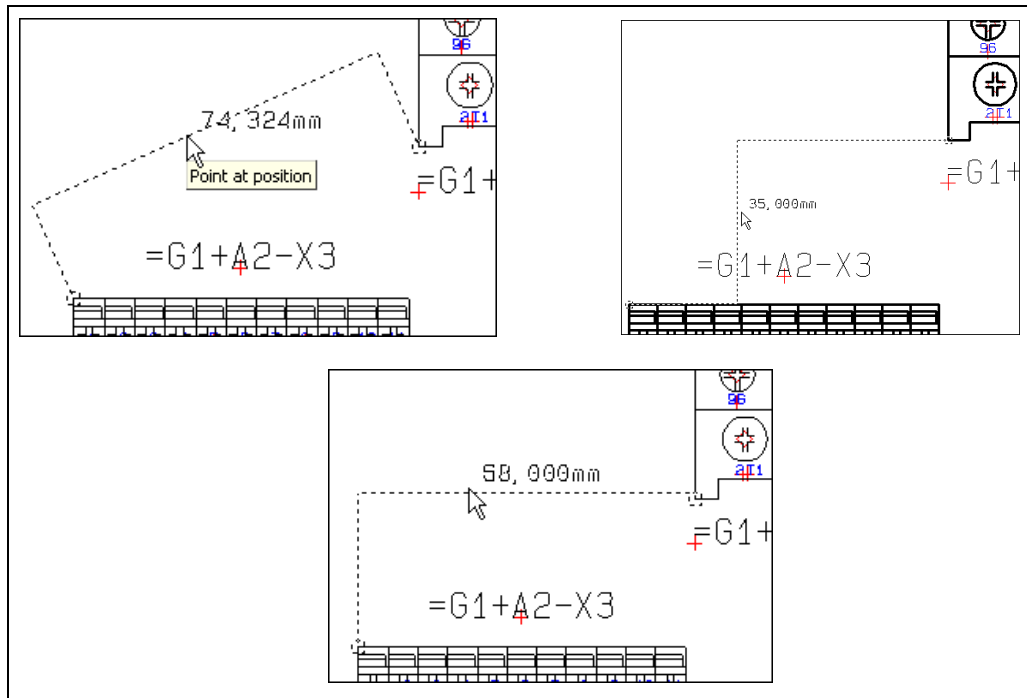
When you choose **Functions => Measurement**, the program will suggest start and end points for the measurement object, based on the objects in the diagram.

If you press the <Ctrl> button, you can place the start and end points for the measurement object freely. This is called free measurement in **Automation**.

Rotating the measurement object with the [Spacebar]

When the start and end points of a measurement object do not have the same coordinates vertically or horizontally, you can — when you have pointed out the

end point for the measurement object — use the [Spacebar] to rotate the measurement object:

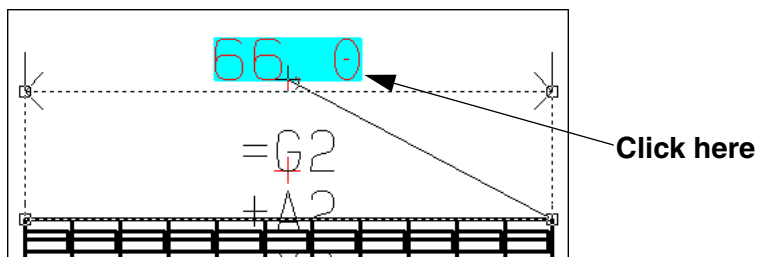


Editing the measurement start and end points

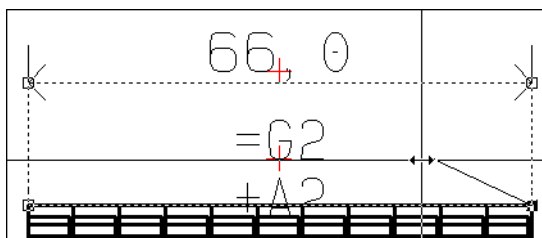
To edit the coordinates of start and/or end point of a measurement object, do the following:



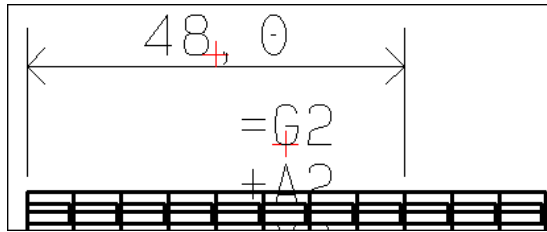
- 1) Click on the **Texts** button, and select the measurement object by clicking on the measurement text:



- 2) Now click on the measurement start/end point, and *drag* it to the desired position:



- 3) Release the mouse button, and the position of the start/end point(s) have now been changed:

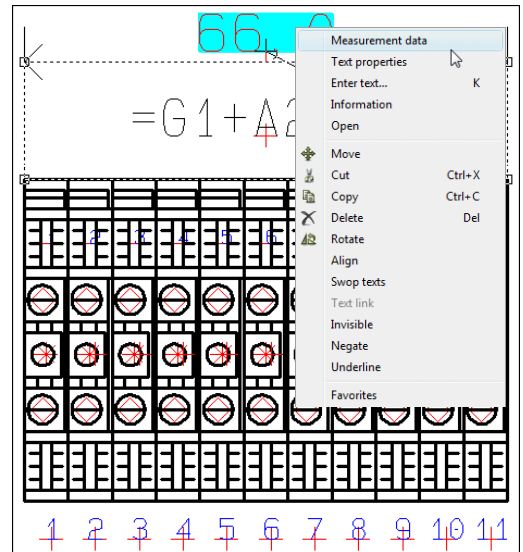


Editing Measurement data

To change the appearance of a measurement object, do this:



- 1) Click on the **Texts** button, right-click on the measurement text, and choose **Measurement data**.
- 2) You now enter the **Measurement data** dialog box, where you can edit the appearance of the measurement object — see “Specifying the Measurement data” on page 89.



Measurement objects and Layers

When you press the <Alt> key, the measurement function is able to see objects placed in other layers on the page. You can therefore choose to place the measurement objects in other layers on the page. When printing the page, it is then possible to control whether to print the measurement objects or not.

— See “Layers, Elevation and Scaling” on page 465, in particular “Printing pages with layers” on page 468.

Deleting a measurement object



To delete a measurement object, click the **Texts** button, select the measurement text, and click the **Delete** button.

MOUNTING CORRECT DRAWING

Automation is able to support mounting correct drawing automatically. — In the IEC 61082 standard this is also called *combined circuit and connection diagram*.

THIS CHAPTER DESCRIBES	
— ON MOUNTING CORRECT DRAWING	(P. 92)
— ACTIVATING MOUNTING CORRECT DRAWING	(P. 93)
— CHOOSING MOUNTING CORRECT DRAWING IN EXISTING PROJECT	(P. 94)
— SIGNAL NAMES IN MOUNTING CORRECT DRAWING	(P. 95)
— SUB DRAWINGS AND MOUNTING CORRECT DRAWING	(P. 95)
— CHANGING TO MOUNTING CORRECT DRAWING	(P. 95)
— CHANGING TO NOT MOUNTING CORRECT DRAWING	(P. 96)
— CONVERTING PROJECTS FROM VERSION 8.0 AND PREVIOUS	(P. 96)



MOUNTING CORRECT DRAWING RECOMMENDED

IF YOU ARE NOT SURE WHETHER TO DRAW MOUNTING CORRECT OR NOT IN YOUR DOCUMENTATION, IT IS RECOMMENDED TO DO SO IN **Automation**. THIS IS BECAUSE YOU CAN EASILY CONVERT MOUNTING CORRECT DRAWN PROJECTS TO NORMAL DRAWING MODE WITH CONNECTION DOTS, AS DESCRIBED IN “CHANGE TO NOT MOUNTING CORRECT DRAWING” ON PAGE 96.

On mounting correct drawing

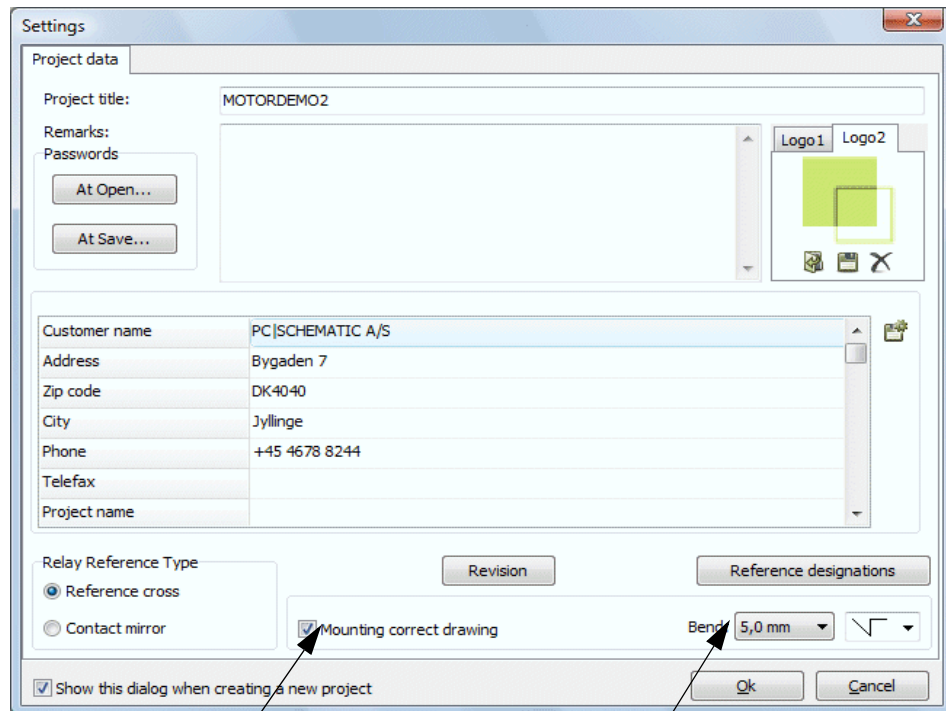
In **Automation** you can create mounting correct drawings on equal terms with how you normally draw diagrams with connection dots:

- Where a connection dot automatically occur in normal diagram drawings, a *branch figure* is inserted in mounting correct drawing mode.
- You can start and end lines on other lines — with automatical insertion of branch figures.
- You can also use automatical line drawing / the router for mounting correct drawn lines, starting/ending on other lines. — See “Automatic line drawing / Router” on page 81.
- Placing sub drawings by drag’n draw or by selecting **Insert => Insert sub drawing** also handles mounting correct drawing. — See “Sub drawings and mounting correct drawing” on page 95.
- Branch figures are moved automatically along with the lines, which they are placed on. You can therefore move e.g. an electrical potential **L1** — which is placed on the upper part of a page — up or down, just by moving one single line, branch figure or signal symbol on the electrical potential.

In mounting correct drawing mode, there are no limitations on what you are allowed to draw. Errors in mounting correct drawing, are exclusively found via the **Design Check** function. — See “Design check for Mounting correct drawing” on page 95.

Activating Mounting correct drawing

When you want to work in mounting correct drawing mode, you choose **Settings** => **Project data**, and select **Mounting correct drawing**.



Click here

Bend size

If you are already working in a project, you will be told if the project is not drawn mounting correct, when you enable **Mounting correct drawing**. Please read more about this in “Choosing Mounting correct drawing in an existing project” on page 94.

Bend

On the **Project data** tab, you also specify the size of the bend of the lines on mounting correct drawings.

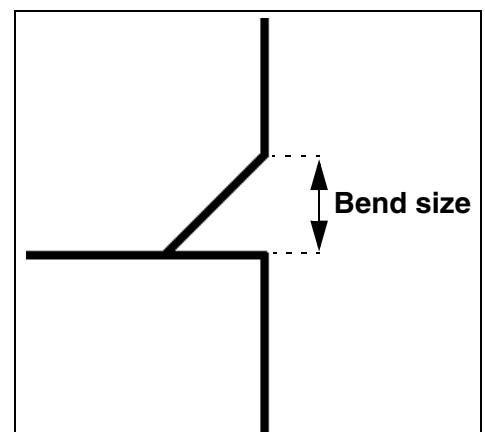
In the **Bend** field you can change the size of the bend by clicking the down-arrow, and selecting a bend size. In the figure to the right, you can see how the size of the bend is specified.

It is recommended *not* to use the bend size *5.0mm*, because this does not give a clear result when e.g. three electrical potentials are placed with a distance of 5,0 mm.

It is recommended to use a bend size of *2,0* or *2,5 mm*.

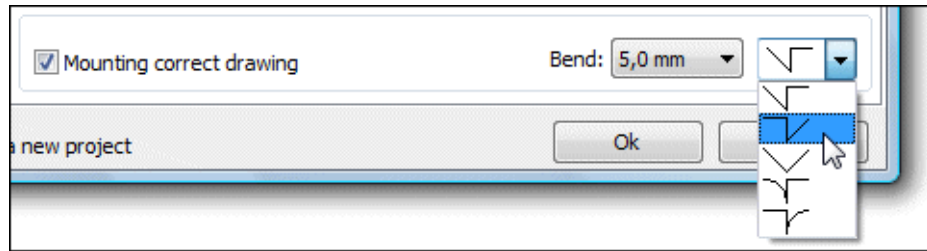
CHANGING THE APPLIED BEND SIZE IN A PROJECT

If you change the bend size used in a project, you will be asked whether to change the bend size in the existing branch figures in the project.



Branch figures

After the bend size, you can choose which branch figure that the program shall suggest initially, when drawing mounting correct:

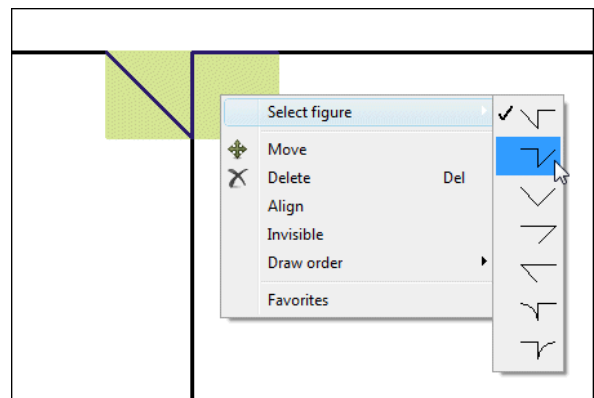


In mounting correct drawing mode, the program will automatically place the latest applied branch figure. When pressing the <Spacebar> repeatedly before drawing further, you can change to one of the other branch figures.

Changing branch figures on lines

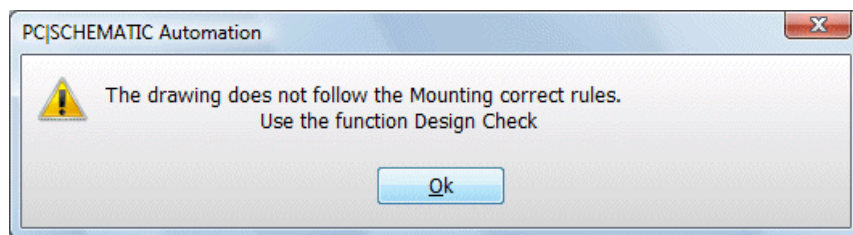
When you wish to use another branch figure on a line, then deactivate the Pencil, click on the branch figure and press the <Spacebar> until the desired figure appears.

Having deselected the Pencil, you can also right-click on the branch figure, select **Select figure**, and select the new branch figure — as illustrated to the right.



Choosing Mounting correct drawing in an existing project

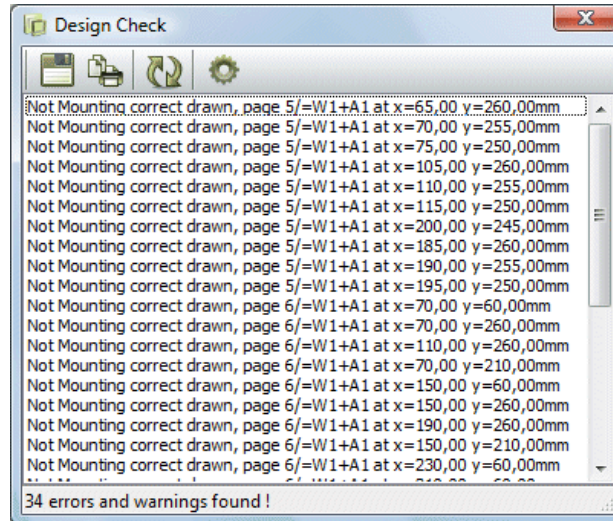
When you are working in a project, and select **Mounting correct drawing**, you will be informed if the project is not drawn mounting correct at the time being.



Design check for Mounting correct drawing

To locate the parts that are not mounting correct, you can then choose **Functions => Design check**, and enable **Mounting correct**.

When you click **OK** you get a list of the parts of the project, which are not mounting correct.



Starting from this list, you can then make sure that your project are mounting correct drawn.

— Read more about design check in “Design Check” on page 310.

Signal names in Mounting correct drawings

— See “Variant names on signal symbols” on page 202.

Sub drawings and mounting correct drawing

It is an advantage to draw all of your sub drawings mounting correct. Hereby you can use them for both mounting correct drawing, and for not mounting correct drawing.

When you *drag* one of these sub drawings into a project, which is not in mounting correct drawing mode, the branch figures are automatically converted to connection dots.

By drawing the sub drawings mounting correct, you therefore only need to maintain one single set of sub drawings, which can be applied for both purposes.

Change from Non mounting correct to Mounting correct drawing

Projects, which are drawn non mounting correct, can get branch figures inserted in stead of the connection dots like this:

- 1) Select **Mounting correct** in **Settings => Project data**.
- 2) Select **Functions => Special Functions => Convert Dots to Branches**.

— Please check that **Automation** have inserted appropriate branch figures where it is not obvious which figure to insert.

Change to Not mounting correct drawing

Projects, which are drawn mounting correct, can have inserted connection dots instead of branches like this:

- 1) De-select **Mounting correct** in **Settings => Project data**.
- 2) Select **Functions => Special Functions => Convert Branches to Dots**.

Converting mounting correct projects from version 8.0 and previous

Starting from **Automation** 9.0 an improved workflow for mounting correct drawing have been introduced. To convert mounting correct drawings from version 8.0 and previously, select **Functions => Special Functions => Convert to Branches**.

The connection dots in the project will then be converted to branch figures anywhere this is possible, and double-lines will be removed. — Please check that **Automation** have inserted appropriate branch figures where it is not obvious which figure to insert.

Wire numbers when converting mounting correct drawings from previous versions

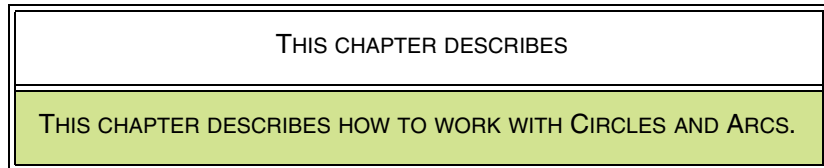
When converting mounting correct drawn projects from version 8.0 and previous, wire numbers will be moved, so that they uniquely identify which lines they are attached to.

ARCS/CIRCLES



When you want to draw circles, you click the **Circles** button and activate the **Pencil**. Circles have the short-cut key [c]. — See “Auto command change” on page 51 also.

The general short-cut key for the Pencil is the [Ins] key, but you can also use the [c] key when you are working with Circles. Pressing the [Ins] or the [c] key activates/deactivates the Pencil.



— See how to move, copy, rotate and delete circles and arcs in the section “Working with selected objects” on page 51, and see how to transfer data between circles/arcs in “Transferring Data / Properties between Objects” on page 55.

Please note, that circles and arcs can not be rotated.

The Circles toolbar

The toolbar for circles looks like this:



R is for the radius of the circle/arc.

V1 is for the arc start angle, **V2** is for the arc end angle. The drawing is made counter-clockwise.

To draw a full circle **V1** has to be 0 and **V2** has to be 360.

To draw a quarter circle **V1** can be 180 while **V2** is 270.

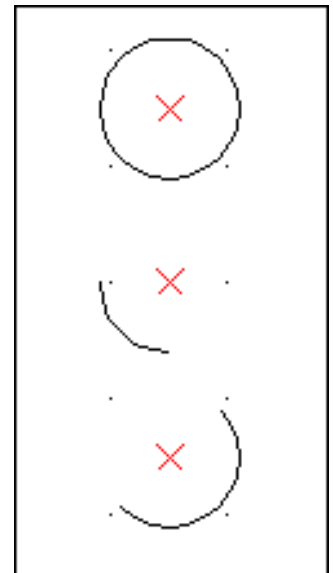
To draw a half circle **V1** can be 45 while **V2** is 225.



Whether the circle is to be filled out, is determined by clicking the **Filled circles** button on the toolbar.

Line width is determined in the field **B:**, while the color of the line and filling is determined in the field **F:**.

Selected arcs can be rotated by pressing the [Spacebar].



Circles / Arcs in the Pickmenu

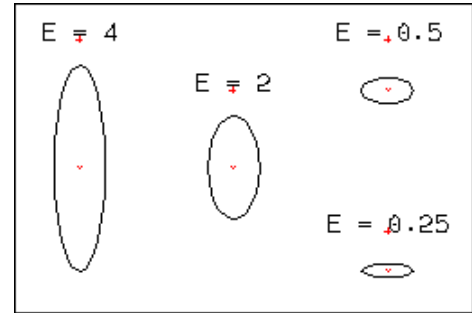
To get fast access to drawing arcs with predefined properties, you can place arcs with the desired properties in the pickmenu at the upper part of the screen. This is done as described for lines in “Lines in the Pickmenu” on page 71.

Ellipses

Using the circle command also enables you to draw ellipses. The last field **E**: in the toolbar of the circle command is an ellipse factor.

If the factor is set to 1 you will get an ordinary circle. Apart from this, see the impact of the different factors on the figure below.

When you fill out both **V1** and **V2** you can also draw arcs.



Converting circles and ellipses to lines

When you want to change a circle to lines, you do this by selecting the circle and then choosing **Edit => Convert to lines**.



When you do this, the circle is converted to lines.

To select these lines, you now have to click the **Lines** button.

Invisible arcs / circles

— See “Making circles/arcs invisible” on page 63.

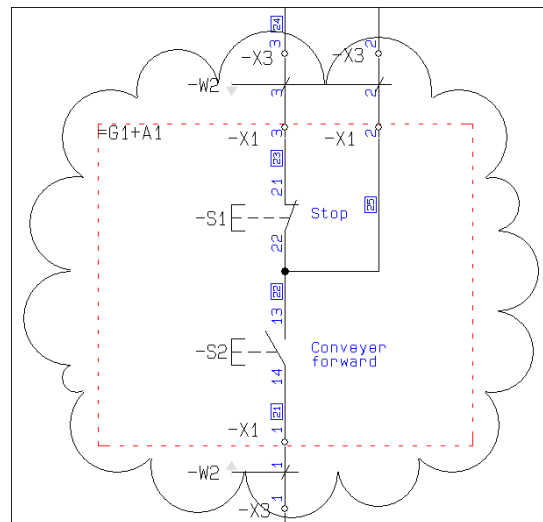
INSERT CLOUD

To insert a cloud to mark corrections easily, do this:

- 1) Choose **Insert => Insert Cloud**.
- 2) A pencil is now displayed in the cross hairs: Click to point out one corner for the cloud.
- 3) Click to point out the opposite corner.

The cloud is now drawn.

The applied line color and type is memorized for the next time you draw a cloud.



TEXTS



When you want to work with texts, you always start by clicking the **Texts** button — or press the short-cut key [t]. If you want to place new texts you click the **Pencil** button. — See “Auto command change” on page 51 also.

The general short-cut key for the Pencil is the [Ins] key, but you can also use the [t] key when you are working with Texts. Pressing the [Ins] or the [t] key activates/ deactivates the Pencil.

THIS CHAPTER DESCRIBES	
— THE TEXT TYPES IN Automation	(P. 99)
— PROJECT AND PAGE DATA	(P. 111)
— GENERAL TEXT FUNCTIONS	(P. 112)
— LOCKING TEXT TYPES	(P. 115)
— COUNTING ON TEXTS	(P. 116)
— THE APPEARANCE OF TEXTS / TEXT PROPERTIES	(P. 118)
— WHICH TEXTS ARE DISPLAYED IN PROJECTS.....	(P. 123)
— THE READING DIRECTION FOR TEXTS	(P. 126)
— TEXT LINKS.....	(P. 128)

— See how to move, copy, delete and rotate texts in “Working with selected objects” on page 51, and see how to transfer data between texts in “Transferring Data / Properties between Objects” on page 55.

TEXT TYPES IN Automation

There are various types of texts in **Automation**, as you will see below.

TEXT TYPE	DESCRIPTION	PAGE
FREE TEXTS	TEXTS, THAT CAN BE USED ANYWHERE IN THE PROJECT.	99
SYMBOL TEXTS	TEXTS, THAT ARE ATTACHED TO EACH SYMBOL, CONTAINING INFORMATION ON THE COMPONENT THAT IS COVERED BY THE SYMBOL.	104
CONNECTION POINT TEXTS	TEXTS, THAT ARE ATTACHED TO EACH OF A SYMBOLS CONNECTION POINTS.	106
DATAFIELDS	TEXT FIELDS THAT ARE FILLED OUT AUTOMATICALLY.	107

When the word *text* is mentioned in this manual, we will be referring to *free texts* — unless specified otherwise.

GENERAL PURPOSE / FREE TEXTS

General purpose/free texts are texts that can be applied anywhere in diagrams, in symbol definitions (see “Creating Symbols” on page 676) or in drawing headers/ formulas.

The free texts can not be transferred to project lists — as opposed to the other types of texts.

If a free text is a part of a symbol definition — see “Creating Symbols” on page 676 —, it can not be changed in a diagram later on.

Type in and place a free text



Click on the **Texts** and the **Pencil** button — or click inside the text field on the texts toolbar.

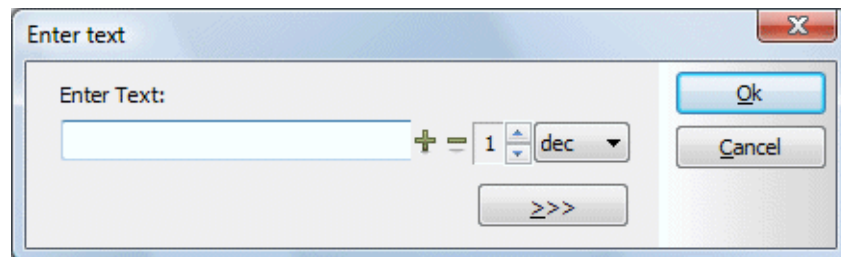


Type in the text as displayed above and press <Enter>. Please note that the toolbar informs you that you are dealing with a free text.

The text is now in the cross hairs. Click where you wish to place it.

You have now placed a free text on a project page.

When the Pencil is not activated, you can press the short-cut key [k], and you enter the dialog box **Place text** where you also can type in a text and then place it.



From here it is now possible to use the automatic counting functions, which can be found in the Texts toolbar. — Read more about these in “Counting on texts” on page 116.

Editing texts in projects

When you want to edit a text in a project, do the following:

- 1) Disable the Pencil (press <Esc>)
- 2) Click on the text and press the short-cut key [k]
- 3) Correct the text and press <Enter> — or click **OK**

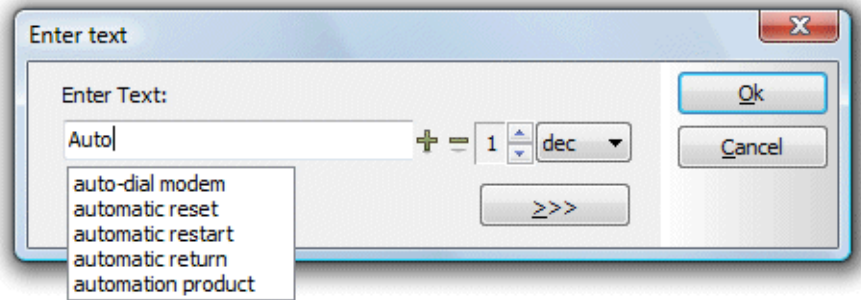
If you press [F7] to enter the **Object Lister**, you can see all the project texts in one place. From there you can also edit the texts.

— See “Checking and editing project texts via the Object Lister” on page 304.

Autocompletion of texts

Automation memorizes all of the texts which have been used in the program.

While you type in texts, the program therefore gives you some suggestions to which text you are typing in.



You can then choose between the suggestions in the list, or continue typing in the text.

If no list appears with texts having the same initial letters, you can press [Ctrl+Spacebar] and get a list of matching texts, that do not have the same initial letters.

Creating a list of texts

As mentioned, the program memorizes all the texts you type in.

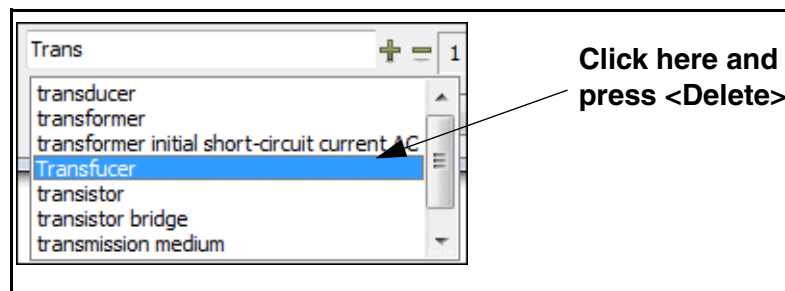
If you also want to add texts from existing projects, do like this:

- 1) Open the project containing the texts
- 2) Choose **Functions => Special functions => Transfer Project Texts to Auto Completion function**

This procedure is repeated for all the projects you want to transfer texts from.

Deleting texts from the autotext function

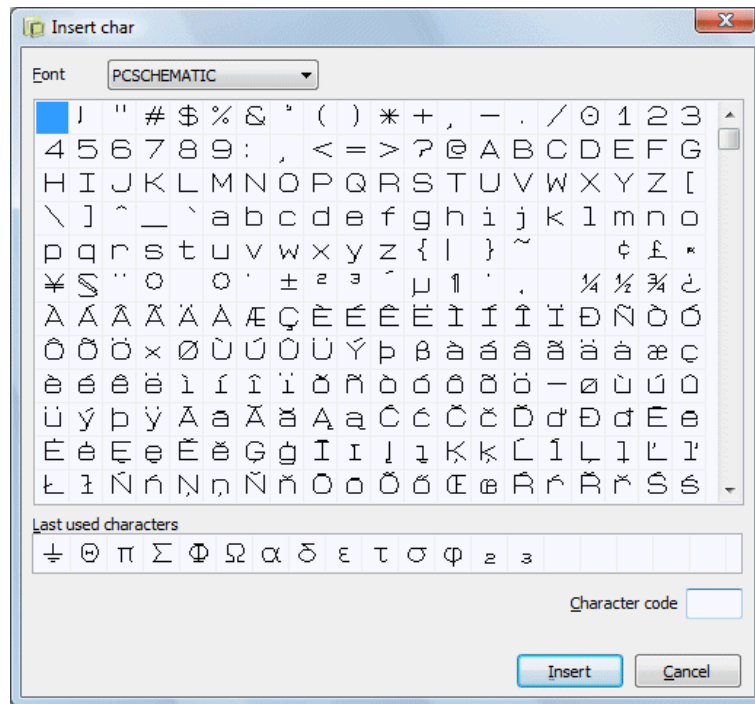
To delete texts from the autotext function, click on the undesired text in the autotext scroll menu, and press <Delete>.



Special characters

In the project pages you may also need special characters, which are not available on your keyboard. This goes for the signs for earth, frame and alternating current and characters from other european languages.

To insert a special character, right-click in the text field in the text toolbar, and choose **Insert char**:



In the dialog box **Insert char** you then select the desired character, and click **Insert**.

— Please refer to “Texts in more than one line” on page 112, “Negate (invert) text” on page 112 and “Underline text” on page 113.



SPECIAL CHARACTERS AND WINDOWS

THE FONTS YOU GET WITH OLDER VERSIONS OF WINDOWS, DOES NOT CONTAIN SPECIAL CHARACTERS. WHEN A SPECIAL CHARACTER DOES NOT EXIST IN A SELECTED FONT, A SQUARE IS DISPLAYED IN STEAD OF THE SPECIAL CHARACTER IN THE DIALOG BOX ABOVE.

THE FONTS YOU GET WITH WINDOWS VISTA AND SUCCEEDING VERSIONS CONTAINS SPECIAL CHARACTERS.

THE **PC|SCHEMATIC** FONT CONTAINS THE NECESSARY SPECIAL CHARACTERS.

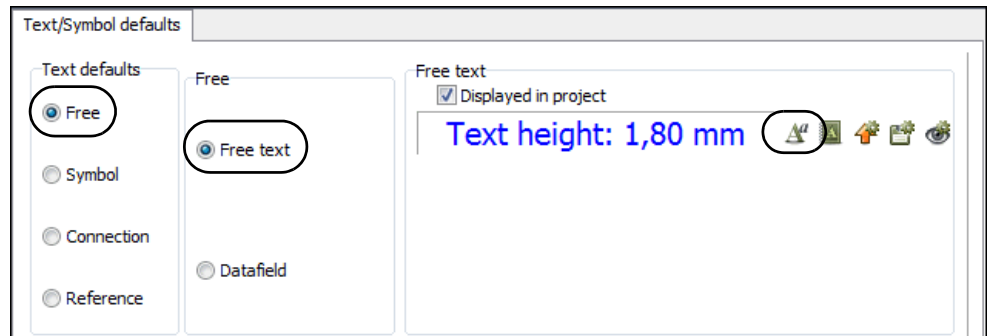
Default text properties for free texts

To edit the default text properties (text size etc.) which **Automation** suggests for free texts, do this:

- 1) Choose **Settings => Text/Symbol defaults**.



- 2) On the tab **Text/Symbol defaults**: Click on **Free**, click on **Free text** and click the **Text properties** button:



- 3) In the dialog box **Text properties**: Specify the desired text properties, as described in “Specifying text properties / Appearance of text” on page 118, and click **OK**.



- 4) In the dialog box **Text/Symbol defaults**: Click the **Save as default** button, if you also want to apply these settings in other projects.

- 5) Click **OK**.

TEXTS CONNECTED TO SYMBOLS

Symbols have attached symbol texts, and if the symbol has connection points, these have attached connection texts as well.

The fields for these texts can not be removed. However they neither have to be filled out nor visible. — Read more about symbol texts in “Creating Symbols” on page 676.

The symbol texts

To each symbol, some symbol texts are attached, which can contain the symbol/component name as well as other information about the component, which the symbol represents.

These symbol texts exist:

TEXT TYPE	DESCRIPTION
S NAME	THE NAME OF THE SYMBOL/COMPONENT — I.E. -K1, -Q34.
S TYPE	DESCRIBES THE TYPE OF THE COMPONENT — CAN BE FILLED OUT VIA THE DATABASE.
S ARTICLE NO	A TEXT TELLING EXACTLY WHICH COMPONENT A SYMBOL DESCRIBES. THIS BEING EITHER A EAN 13 NUMBER OR A STOCK NUMBER. CAN BE FILLED OUT VIA THE DATABASE.
S FUNCTION	THIS TEXT IS USED TO DESCRIBE THE FUNCTION OF THE SYMBOL. CAN BE FILLED OUT VIA THE DATABASE.

The information that you type into the symbol texts, can be used when for instance you want to generate parts lists automatically.

Filling out the symbol texts

When you place a symbol you automatically enter the **Component data** dialog box, where you can enter various information about the symbol, or just click **OK** and return later to fill in the missing data.

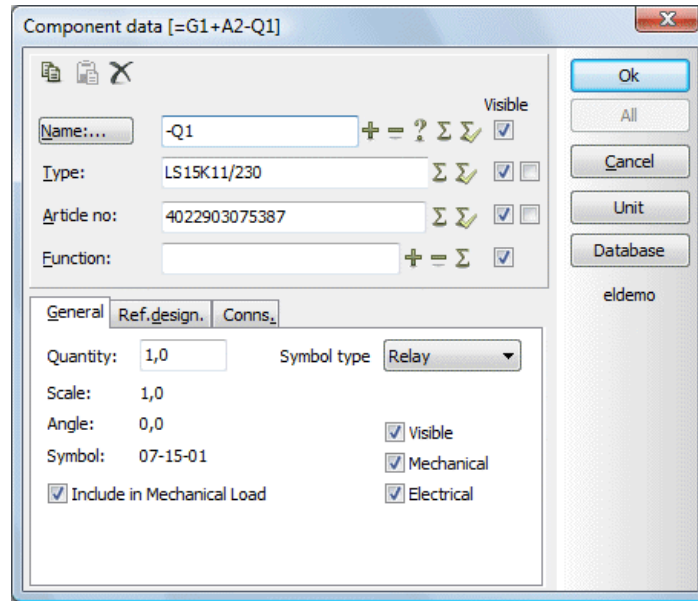
If you do not enter the **Component data** dialog box automatically, you can enable this function by entering **Settings => Pointer/Screen** and select **Ask for Names**. — Read more about this in “Pointer/Screen” on page 815.

When you return later to type in the symbol texts, you have to start by clicking the **Symbols** button, or use the short-cut key [s]. Then right-click on the symbol whose symbol texts you want to change, and choose **Component data** in the appearing menu.



Instead of right-clicking, you can select the symbol by clicking and then click the **Data** button in the program toolbar.

You now enter the dialog box **Component data**, that for instance could be filled out like this:

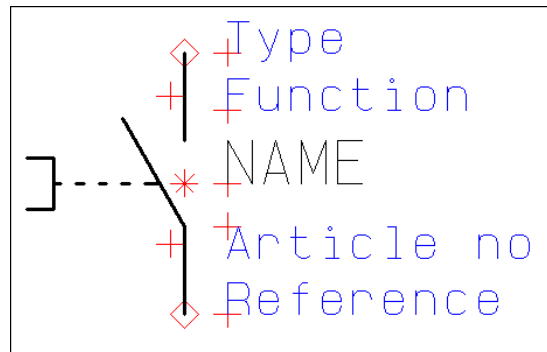


Here you can fill out the symbol texts, and decide which of the symbol texts that should be displayed together with the symbol in the project. It is also possible to type in the connection point names. This is described in “Placing and Naming symbols” on page 151.

Click **OK** when the data are as you want them to be.

Editing the symbol texts directly on a Diagram page

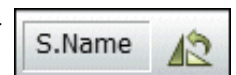
A symbol with visible symbol texts could look like this:



- Please note that references are only displayed when the symbol has references.
- See “Creating Symbols” on page 676, especially “Position for references to other symbols” on page 678.



You can change the contents of a visible symbol text directly on a diagram page, by first clicking on **Texts**, then disable the Pencil (press <Esc>), and click on the text you wish to change. In the texts toolbar you will see which type of symbol text you have selected.



To indicate to which symbol the symbol text is connected, a line is being displayed going from the reference point of the symbol to the symbol text. Click in the text field of the toolbar — or press [k] —, edit the text, and press <Enter>. The symbol text has now been changed.

Changing between the symbol texts by pressing [F5]/[F6]

Once you have selected a symbol text, you can change back and forth between the symbol texts connected to the same symbol by pressing [F5] and [F6]. When a symbol has references, pressing [F5]/[F6] will also jump to the reference text.

Please refer to “Locking text types” on page 115 for an easier way to change symbol texts.

Moving symbol texts on a Diagram page



To move a symbol text you also have to activate the **Texts** button. Then you can move the text as you can move any other object in **Automation**, for instance using *drag and drop*.

If you press down <Shift> while moving the text, you can place the text using fine *snap* — read more about snap in “Snap” on page 42.

CONNECTION POINT TEXTS

These texts are connected to the connection points of the symbol — where you connect conducting lines to the symbol — and contain information about these.

The following types of connection point texts exist:

CONNECTION POINT TEXT	CONTENTS
NAME (C.NAME)	THE CONNECTION NUMBER OF THE CONNECTION POINT.
FUNCTION (C.FUNC)	THE FUNCTION OF THE CONNECTION POINT.
LABEL (C.LABEL)	THE SHORT DESCRIPTION (FOR INSTANCE FOR PLC DRAWINGS).
DESCRIPTION (C.DESCR)	THE LONG DESCRIPTION (FOR INSTANCE FOR PLC DRAWINGS).

Connection point texts are filled out and moved like symbol texts. — Please refer to “Texts connected to symbols” on page 104 and “Symbol Connection points” on page 182.

When you have selected a connection point text, you can change back and forth between the texts connected to the connection point by pressing [F5] and [F6].

DATAFIELDS

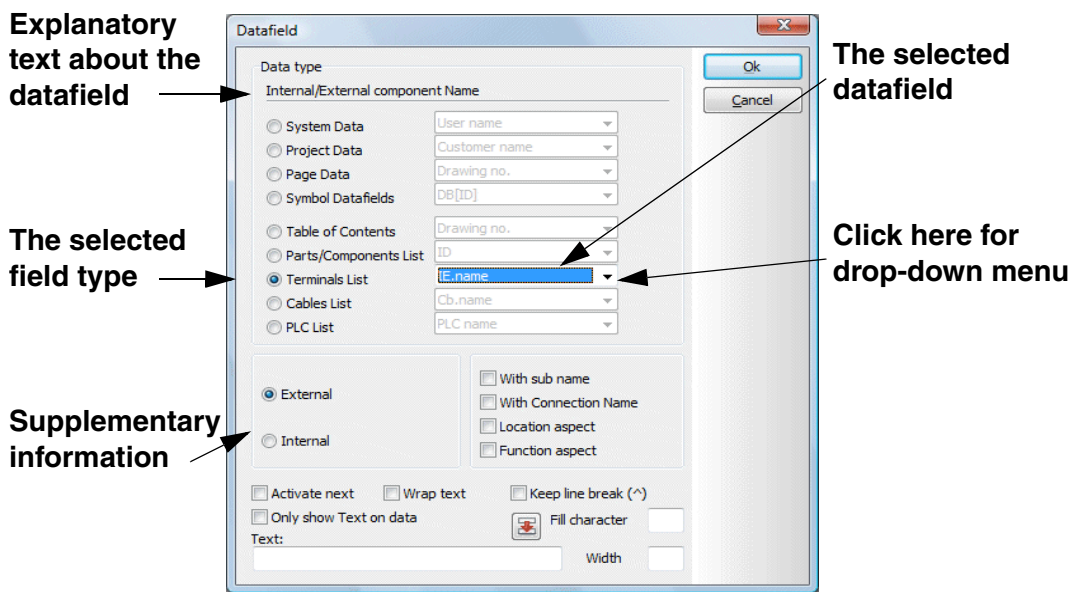
Another type of texts are called datafields. These can be filled out automatically by the program. Like free texts, they can be applied anywhere in drawings, in symbols and in drawing headers/formulas.

Choosing **Settings** you can specify the contents of some datafields once and for all. What you type in here will be inserted into the project any time you place one of these datafields. If you change the contents of one of the datafields in **Settings**, the project will be updated automatically. Read more about how to do this in “Project and Page data” on page 111.

— Please refer to “Page data and Project data Datafields” on page 669 for further details.

To insert a datafield choose **Insert => Insert datafield**.

You now enter the **Datafield** dialog box:



The various groups of datafields have the following contents:

DATAFIELD	CONTENTS
SYSTEM DATA	DATA ABOUT YOUR SYSTEM. FOR INSTANCE <i>USER NAME</i> WHICH IS THE NAME OF THE PERSON THAT IS TYPED IN AS USER IN SETTINGS => SYSTEM . <i>DATE</i> AND <i>TIME</i> ARE THE ACTUAL DATE AND TIME, MINUTES AND SECONDS.
PROJECT DATA	DATA CONCERNING THE ACTUAL PROJECT. YOU CAN DEFINE THESE IN SETTINGS => PROJECT DATA .
PAGE DATA	DATA CONCERNING THE PAGE YOU ARE WORKING ON IN THE PROJECT. THERE ARE SOME PRE-DEFINED PAGE DATA DATAFIELDS, SUCH AS THE PAGE NUMBER. YOU CAN DEFINE ADDITIONAL PAGE DATA DATAFIELDS IN SETTINGS => PAGE DATA .

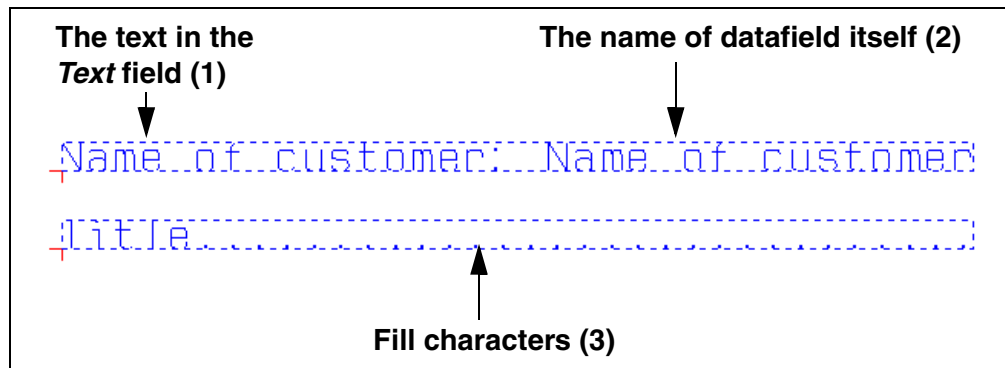
DATAFIELD	CONTENTS
SYMBOL DATAFIELDS	DATAFIELDS YOU CAN DEFINE BY YOURSELF. — READ MORE ABOUT THESE IN “CREATING DATA SYMBOLS” ON PAGE 711. YOU CAN ALSO PLACE SYMBOL DATAFIELDS FOR EACH INDIVIDUAL FIELD IN THE DATABASE. IN THE LIST OF SYMBOLDATA DATAFIELDS, THESE FIELDS ARE ASSIGNED THE NAMES DB[xxxxx] , WHERE xxxxx IS THE NAME ON THE DATAFIELD IN THE DATABASE.
TABLE OF CONTENTS	DATAFIELDS FOR TABLES OF CONTENTS.
PARTS/COMPONENTS LISTS	DATAFIELDS FOR PARTS- AND COMPONENTS LISTS.
TERMINALS LIST / CABLE LIST	DATAFIELDS FOR TERMINALS- AND CABLES LISTS. IN THE SUPPLEMENTARY INFORMATION BOX YOU CAN FOR INSTANCE DETERMINE WHETHER A DATAFIELD IS FOR THE EXTERNAL OR THE INTERNAL PART OF THE LIST.
PLC LIST	DATAFIELDS FOR PLC LISTS.

— See the survey of datafields in “Datafields” on page 834.

The other fields in the menu have following functions:

FIELD	FUNCTION
ACTIVATE NEXT	IS USED FOR LISTS WHEN A CERTAIN GROUP OF INFORMATIONS MUST BE REPEATED SEVERAL TIMES ON A PAGE. EACH TIME THE PROGRAM MEETS ACTIVATE NEXT , THE NEXT SET OF INFORMATION ON THE LIST CAN BE FILLED INTO THE LIST. SEE “DICTIONARY” ON PAGE 850.
WRAP TEXT	IF THE TEXT CONTAINS MORE CHARACTERS THAN STATED IN THE WIDTH FIELD, THE REMAINING PART OF THE TEXT WILL NOT BE DISPLAYED. IF YOU ACTIVATE WRAP TEXT , THE TEXT WILL CONTINUE ON AN EXTRA LINE BELOW.
KEEP LINEBREAK (^)	WHEN YOU SELECT KEEP LINEBREAK (^) , THE DATAFIELD WILL ALSO HAVE THE SPECIFIED LINEBREAK IN THE LISTS CREATED FROM THE PROJECT.
ONLY SHOW TEXT ON DATA	THIS OPTION ONLY APPEAR WHEN SOMETHING HAS BEEN TYPED INTO THE TEXT FIELD. IF YOU ACTIVATE THIS FUNCTION, THERE WILL NOT BE PRINTED ANYTHING ON THE SCREEN IN CASES WHERE THE TEXT FIELD IS FILLED OUT, BUT THERE ARE NO ACTUAL DATA IN THE DATAFIELD ITSELF. ESPECIALLY USED IN LISTS.
THE RED ARROW	TRANSFERS THE SELECTED DATAFIELD TEXT TO THE TEXT FIELD.
FILL CHARACTER	A CHARACTER THAT WILL BE FILLED IN FROM THE END OF THE TEXT AND IN THE REMAINING WIDTH OF THE DATAFIELD. FOR INSTANCE A LINE OF DOTS IN A TABLE OF CONTENTS BETWEEN THE PAGE TITLE AND THE PAGE NUMBER. (3) ON THE FOLLOWING FIGURE.
TEXT	HERE YOU CAN TYPE IN A TEXT TO PRECEDE THE CONTENTS OF THE DATAFIELD. (1) ON THE FOLLOWING FIGURE.
WIDTH	THE MAXIMUM WIDTH OF THE DATAFIELD (NUMBER OF CHARACTERS).

Two inserted datafields could for instance look like this:



Please note that this is how datafields look like when you are in Edit symbols — for instance because you want to create a list (Read more about creating lists in “Creating Lists” on page 732). If you just insert a datafield on a page, you will not see the *name* of the datafield itself (2), but the *contents* of the datafield.

Use **Insert => Insert Datafield** to select a datafield you want to insert. Click **OK** when the settings are as you want them.

You now get the chosen datafield into the cross hairs, and click where you want to place it.

When you want to edit a placed datafield, you right-click on the datafield and choose **Datafield**. This brings you into the **Datafield** dialog box.

Inserting the contents of the datafield in the text

By applying the % character, you can insert the contents of the datafield in the middle of the text in the **Text** field.

The following options exist:

INSERT	DESCRIPTION
%S	INSERTS THE VALUE/CONTENTS OF THE DATAFIELD UNCHANGED. IF YOU TYPE IN "PRICE: %S \$" — AND THE CONTENTS OF THE DATAFIELD IS 25 — THIS RESULTS IN THE TEXT "PRICE: 25 \$" — WHEN YOU HAVE TYPED IN THE CONTENTS OF THE DATAFIELD WITH NO DECIMALS, THE CONTENTS IS DISPLAYED WITHOUT DECIMALS. IF YOU HAVE TYPED IN THE CONTENTS WITH DECIMALS, THE CONTENTS IS DISPLAYED WITH DECIMALS.
%F	INSERTS THE VALUE OF THE DATAFIELD AS FLOATING POINT VALUE, WHEN POSSIBLE.

INSERT	DESCRIPTION
%.1F	<p>INSERTS THE CONTENTS OF THE DATAFIELD AS FLOATING POINT WITH ONE DECIMAL. YOU CAN CHOOSE FROM 0 TO 9 DECIMALS.</p> <p>IF YOU WRITE "PRICE: %.2F \$" — AND THE CONTENTS OF THE DATAFIELD IS 25 — THIS RESULTS IN THE TEXT "PRICE: 25,00 \$"</p> <p>IF YOU WRITE "%.3F KG." — AND THE CONTENTS OF THE DATAFIELD IS 17,65 — THIS RESULTS IN THE TEXT "17,650 KG."</p>
%-	<p>NO DATAFIELD VALUE IS INSERTED, AND THE TEXT IS ENDED RIGHT BEFORE THE % CHARACTER. IF FOR INSTANCE YOU WRITE "TOTALPRICE:%" THE TEXT "TOTALPRICE:" IS INSERTED IF THE CONTROLLING DATAFIELD HAS A CONTENT.</p> <p>THIS CAN FOR INSTANCE BE USED AT THE BOTTOM LINE OF A LIST, WHERE THE DATAFIELD PRICE 1 TOTAL DOES NOT HAVE A CONTENT UNTIL ALL PAGES IN THE LIST HAVE BEEN ADDED. TO MAKE THIS WORK, YOU MUST PLACE THE DATAFIELD PRICE 1 TOTAL WITH THE TEXT "TOTAL PRICE:%-" IN THE LEFT SIDE OF THE BOTTOM LINE, AND PLACE THE DATAFIELD PRICE TOTAL IN THE RIGHT SIDE OF THE BOTTOM LINE WITH THE TEXT "%S \$"</p> <p>SELECT ONLY SHOW TEXT ON DATA FOR BOTH DATAFIELDS.</p>

When you want to use the % character in a text, just type the % character twice. If you write "%.2f %%" this can result in the text "5,00 %".

PROJECT AND PAGE DATA

When you open a new project, the program will show you the tab **Project data** from the **Settings** dialog box. In this dialog box you also find the tab for **Page data**.

Here you can fill in various data for your project.

To fill out a field, you just click inside the field, and type in the information. You can jump between the fields by pressing the <Tab> key.

If you do not wish to enter these data now, press <Esc> or click **Cancel**.



If you want to fill out or change data later, choose **Settings => Project data / Page data**. Or you can click the **Project data** or the **Page data** button. You can also right-click on a project page, and select **Page data**.

The screenshot shows the 'Settings' dialog box with the 'Project data' tab selected. The 'Project title' field contains 'MOTORDEMO2'. The 'Remarks' and 'Passwords' fields are empty, with 'At Open...' and 'At Save...' buttons below them. A 'Logo' section shows two logos, 'Logo1' and 'Logo2', with a preview area. Below this is a table of customer information:

Customer name	PC SCHEMATIC A/S
Address	Bygaden 7
Zip code	DK4040
City	Jyllinge
Phone	+45 4678 8244
Telefax	
Project name	demo

At the bottom, there are radio buttons for 'Relay Reference Type' (selected: Reference cross, unselected: Contact mirror), a 'Revision' button, and 'Reference designations' buttons. There is also a checkbox for 'Mounting correct drawing' (checked) and a 'Bend' dropdown set to '5,0 mm'. At the very bottom, there is a checkbox 'Show this dialog when creating a new project' (checked) and 'Ok' and 'Cancel' buttons.

If you want to change the name of a datafield in the **Project** or **Page data** menu (the grey fields in the left hand side of the menu), just right-click on the field name you want to change. Then a menu appears, from which you can change the name of the field, delete the field, or add new datafields. However, you will not be allowed to change the *names* of project- or page datafields, that are being used in the present project. You can change the *contents* of the fields at any time.

— Read more about this in “Page data and Project data Datafields” on page 669.



Remember to click **Redraw** to see the effects of the changes in your project.

See how you design drawing headers in “Creating Drawing Headers” on page 721, and how to design your own lists in “Creating Lists” on page 732.

Activating the Auto text function for Page title

To activate the auto text function when typing in a **Page title**, press [Ctrl+Spacebar]. — See “Autocompletion of texts” on page 100.

GENERAL TEXT FUNCTIONS

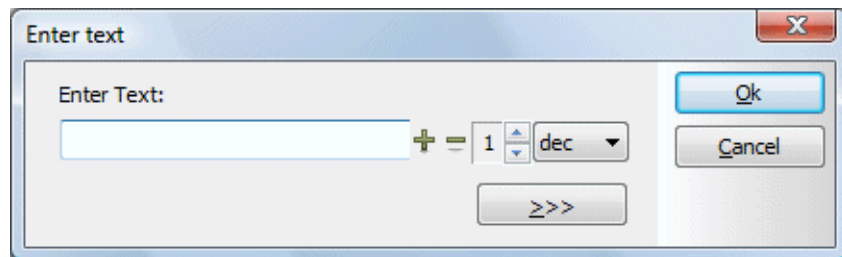
When you are working with texts, there are some text functions you can apply for all text types.

Changing a text

If you want to change a text on a drawing, you first select the text by *disabling the Pencil* (press <Esc>), and then click on the text. The text is now in the text field on the toolbar. Click inside the field, correct the text, and press <Enter>.

If you double-click on the text in the text field, the text will be selected, and you can type in another text instead.

When a text is selected, you can also press the short-cut key [k], and you will enter a dialog box where you also can type in the text.



When you select a text, the type of the text is displayed in the texts toolbar.

Changing a text in the cross hairs

When you have a text in the cross hairs, click in the text field (or press [k]), type in a new text, press <Enter> and place the text. This can also be done while copying.

Texts in more than one line

Insert the character ^ where you want a line break. — On your keyboard it is probably located to the left of the <Enter> key. The character will not appear before you type in the following character.

— See also “Line length” on page 120 and “Wrap text” on page 120.

Negate (invert) text

Type in the text in the text field, select it and press [Ctrl+PageUp]. The negate bar will only be placed above the selected part of the text.

Alternatively you can select the text in the toolbar, right-click and choose **Negate**.

If you deselect the Pencil (press <Esc>) and right-click on a text, you can also choose **Negate**.

You can remove the negation by selecting the text, and then choose **Negate** again.



NEGATED TEXTS AND CHANGING FONTS

IF YOU CHANGE FONT, YOU MUST BE AWARE THAT THIS CAN HAVE AN EFFECT ON NEGATED TEXTS. — SEE “SPECIAL CHARACTERS WHEN CHANGING THE PC|SCHEMATIC FONT TO A WINDOWS FONT” ON PAGE 119.

Underline text

Select the text as mentioned above and press [Ctrl+PageDown] — or select the text, right-click and choose **Underline**.

If you deselect the Pencil (press <Esc>) and right-click on a text, you can also choose **Under line**.

You can remove the underline by selecting the text, and then choose **Underline** again.



UNDERLINED TEXTS AND CHANGING FONTS

IF YOU CHANGE FONT, YOU MUST BE AWARE THAT THIS CAN HAVE AN EFFECT ON UNDERLINED TEXTS. — SEE “SPECIAL CHARACTERS WHEN CHANGING THE PC|SCHEMATIC FONT TO A WINDOWS FONT” ON PAGE 119.

Cut, copy and insert a text in the text field

A text in the text field can be transferred to the Windows clipboard, either by copying or by cutting from the text.

If you want to transfer text from the text field to the clipboard, you select the text you want to transfer, right-click and choose **Copy** in the menu that appear. A copy of the text is now placed on the clipboard.



If you want to transfer a text from the clipboard to a text field, then activate the **Texts** button. Right-click in the text field and choose **Insert**. Press <Enter> and place the text on the drawing.

The text stays on the clipboard until another text is transferred to the clipboard.

Swapping texts

To swap texts and datafields, you can apply the function **Edit => Swop texts**:

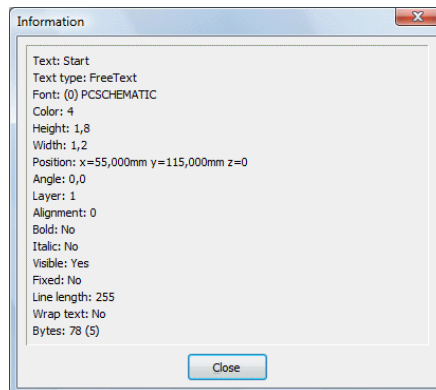
- 1) Select a text by clicking on it.
- 2) Choose **Edit => Swop texts**.
- 3) Click on the other text, that you want to swop the text with.

You can also right-click on a text, and choose **Swop texts**.

Information on texts



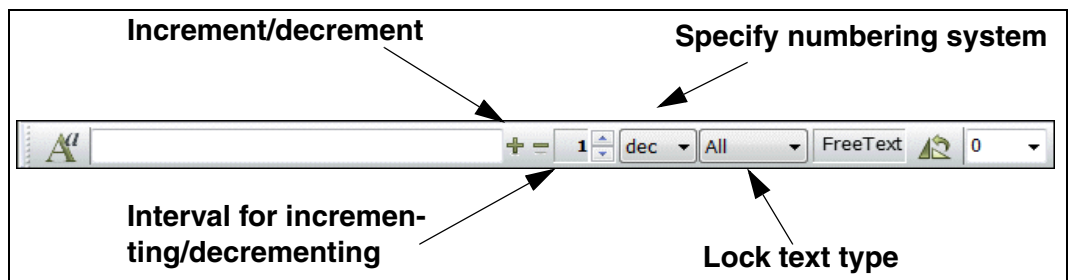
When you have chosen **Texts** and right-click on a text and choose **Information**, you enter the dialog box **Information**, which contains information about the text.



Here you find information about the text height, width, position, the used font, line length of the text, wrap text information etc.

LOCKING TEXT TYPES

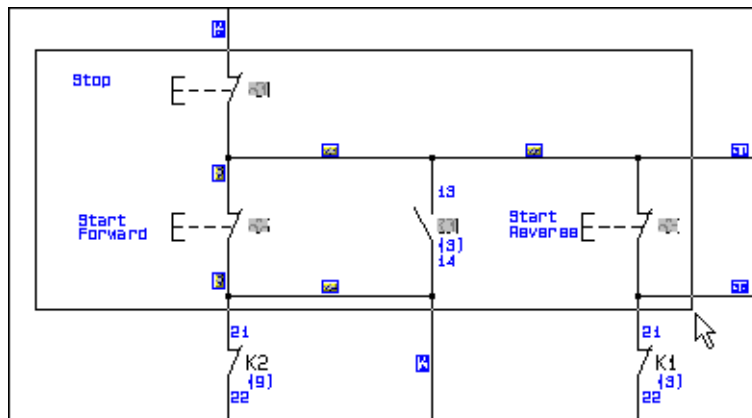
On the toolbar for texts it is possible to lock which type of texts you want to work with, as well as to carry out continuous counting on texts of the type chosen.



Locking a text type

On the toolbar for texts, you can specify, that you only want to work with texts of a specific text type.

If for instance you choose the type **S.Name**, only texts of the type **Symbol name** can be selected in the project. If you then drag a window around some symbols in a project, only their symbol names are selected.



When you have locked the type to **Symbol name**, you can then assign a name to a symbol by typing in the name in the text field in the texts toolbar, press <Enter>, and click on the symbol.

Doing it this way, you do *not* have to click on the reference point for the text **Symbol name** of the symbol.

If you want to place a connection point text, you lock the relevant text type, and click on the connection point itself.

When the text type is set to **All**, the selection of texts function on all text types.



RESET THE TEXT TYPE TO **All** AFTER USING THIS FUNCTION

TO BE ABLE TO SELECT ALL TYPES OF TEXTS IN THE PROJECT SUBSEQUENTLY, YOU SHOULD SET THE TEXT TYPE TO **All** AFTER USING THIS FUNCTION.

COUNTING ON TEXTS

When you want to place texts with increasing or decreasing numbers, you can specify according to which numbering system the texts shall be incremented/decremented. Here you can choose between binary, octal, decimal and hexadecimal. You can also choose to increment/decrement upper- or lower case letters (**ALFA** or **alfa** respectively).



The initial value is typed into the text field in the texts toolbar.

If the texts are to be incremented following a specified interval, you type in the interval in the **Interval** field, as displayed on page 115.



When you click on +, the text in the text field is incremented with the value specified in the **Interval** field. This increment follows the number system chosen. If the interval is set to **1**, and the number system to **dec**, the text **K1** will be incremented to **K2**.

If you click on - instead, the text will be decremented following similar guidelines.



When you press down <Ctrl> and click on +, the text will be incremented automatically each time you click on a new text.



Therefore, when you have chosen the text type **S.Name**, have typed in **K1** in the **Text** field and **2** in the **Interval** field, and then press down <Ctrl> and click on +, the first symbol you click on will get the symbol name **K1**, the following **K3**, then **K5** and so on.



If you press down <Ctrl> and click on -, the texts will be decremented automatically following the same directions.

Apply counting on texts in areas

To increment/decrement texts in areas, you do the following:



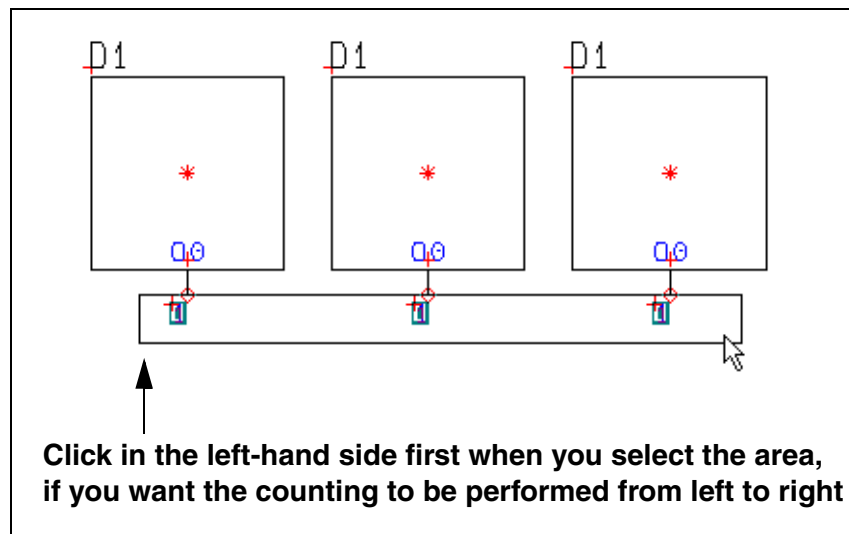
- 1) Click on **Texts**
- 2) Specify text type, interval and numbering system, as described above
- 3) Select the area containing the texts you want to count up/down (see below)
- 4) Type in the initial value in the text field in the toolbar



- 5) Press down <Ctrl> and click on +
- 6) Press <Enter>

The selected texts are now counted up according to the directions you have specified.

Please note, that the program registers where you clicked the first, when you selected the area. The counting of the texts starts from there.



If the counting should be carried out starting from the left-hand side, you therefore start by clicking in the left-hand side, when you select the area.

Texts can be counted up on several symbols at the same time — as illustrated on the figure above.

— Please refer to “Changing symbol names in an Area” on page 180 to see alternative ways to apply automatic counting on symbol names, and to “Choosing connection points in Areas” on page 184 to see how to do it for connection point names.

Short-cut keys for counting on texts

When you have selected a symbol, a text or a connection point on a symbol — or have a text in the cross hairs —, it is possible to increment or decrement the text/symbol name/connection name.

To increment, choose **Functions => Count up**, or press the [+] key on the numerical keyboard.

To decrement, choose **Functions => Count down**, or press the [-] key on the numerical keyboard.

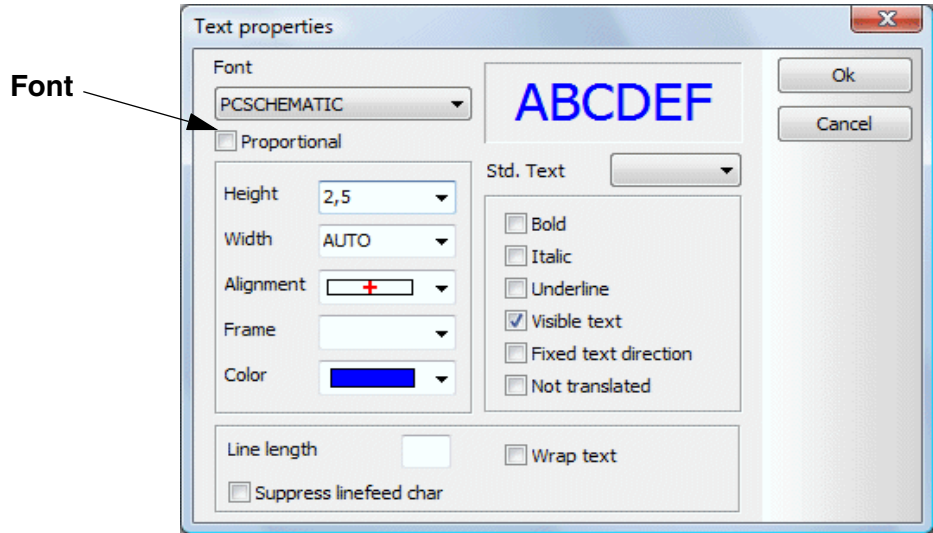
Increment/decrement is done according to the interval specified in the Texts toolbar.

— Both short-cut keys can be changed as described in “User defined short-cut keys” on page 771.

SPECIFYING TEXT PROPERTIES / APPEARANCE OF TEXT



Exit any dialog box that might be activated, click on the **Texts** button, and press the **Text properties** button.



When changing the various text settings, you click the down-arrow at the left hand side of the dialog box, and click on your choice in the drop down box that emerge.

Editing the properties of placed texts



When you want to edit the properties of a placed text, you first click the **Texts** button. Then click on the text, and click the **Text properties** button. — Or: Right-click on the text, and select **Text properties**.

Then make the changes described in this section.



SELECTING TEXT PROPERTIES VIA THE PICKMENU

YOU CAN SELECT DIFFERENT SETS OF TEXT PROPERTIES VIA THE PICKMENU, AS DESCRIBED IN "TEXTS IN THE PICKMENU" ON PAGE 122.

If for instance you want to type in the text height directly, you can also click in the **Height** field itself, and type in the desired height.

The **Text properties** dialog box contains the following fields:

Fonts

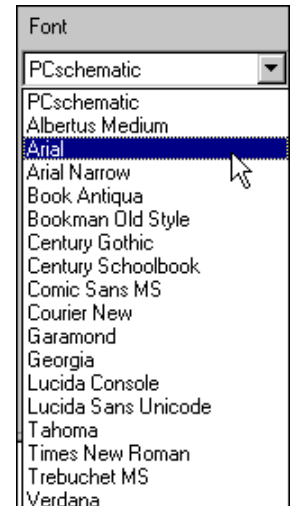
Here you choose which font you want to use.

On the list you can choose between the **PC|SCHEMATIC** font and the other fonts you have on the PC.

Please note that the Windows fonts should only be applied when strictly necessary. This is because they are not to be used on diagrams according to the EN standards, and because they take up extra space in the PC memory.

It can be an advantage to use these fonts in drawing headers, in tables of contents and in lists.

The **PC|SCHEMATIC** font follows the EN standards, and should therefore be used on diagrams.



Special characters when changing the PC|SCHEMATIC font to a Windows font

If you change texts written with the **PC|SCHEMATIC** font to be written with a Windows font, use of special characters such as negated characters, underlined characters will give an incorrect result. Earth characters and other special characters might also give an incorrect result, depending on which font you use.

— See “Exporting PDF files” on page 319 also.



SPECIAL CHARACTERS AND WINDOWS

THE FONTS YOU GET WITH OLDER VERSIONS OF WINDOWS, DOES NOT CONTAIN SPECIAL CHARACTERS. WHEN A SPECIAL CHARACTER DOES NOT EXIST IN A SELECTED FONT, A SQUARE IS DISPLAYED IN STEAD OF THE SPECIAL CHARACTER IN THE DIALOG BOX ABOVE.

THE FONTS YOU GET WITH WINDOWS VISTA AND SUCCEEDING VERSIONS CONTAINS SPECIAL CHARACTERS.

THE **PC|SCHEMATIC** FONT CONTAINS THE NECESSARY SPECIAL CHARACTERS.

Height

States the height of the text.

Width

States the width of the text. The width specifies the distance between characters and their appearance.

Selecting AUTO sets the width to 2/3 of the height of the text.

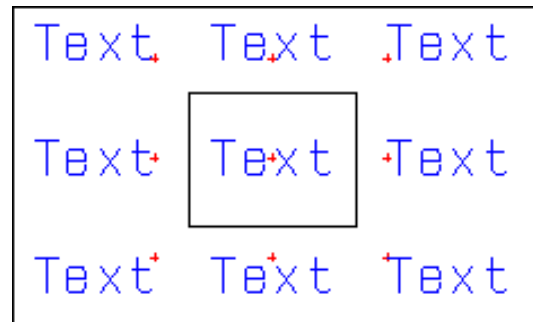
Alignment

There are nine different alignment options, such as *centre-centre* as shown above. When the text is placed in a point, it is actually the reference point of the text that is located in this point — as you can see on the figure.

In the field **Alignment** you determine where the reference point will be located in relation to the text.

If the text is rotated, it will be rotated around the reference point.

These reference points applies for both the **Automation** font and for Windows fonts — see “Fonts” on page 119.



Shortcut for changing text alignment

When you have selected a text, you can change the text alignment by pressing the <Shift> key while pressing the <Spacebar>.

When you have a text in the cross hairs, you can also change the text alignment by pressing the <Shift> key while pressing the <Spacebar>.

Frame

If you want a frame around the text, you pick one of the options in this field.

Color

States the color of text and frame. If you choose the color **NP** (Non Printable), you will see the text on the screen, but not on print.

— See “Visible text” on page 121 and “Making objects invisible” on page 62.

Line length

Here you can specify the maximal number of characters that can be included in the text line. If the text is longer than this, the extra characters are truncated.

Wrap text

Set a check mark here if the text shall continue on a new line, if the line length is exceeded.

Suppress lineshift char

When you select **Suppress lineshift char**, the manually entered lineshift characters are suppressed. This function is activated individually, via the **Text properties** dialog box, for each individual text you wish to use it on.

The function can e.g. be used in connection with PLC reference symbols, when you do not wish to display the I/O connection description text with the same lineshifts, which is adapted to the design of the PLC symbol. — See “Working with PLC’s” on page 436.

The function can also be used in connection with text links, where you wish to suppress the lineshifts for some of the texts in the text link. — See “Text links” on page 128.

Check marks

Bold text, **Italic text**, **Visible text** and **Fixed text direction** are enabled and disabled by clicking inside the check boxes. A check mark is displayed when they are enabled.

Bold text

The text will be bold.

Italic text

The text will be in italics.

Underlined text

The text will be underlined.

Visible text

If you remove the check mark in the **Visible text** check box, the text will not be shown in the project. Thus if you remove the check mark for a text containing a symbol name, the name of the symbol will not be shown in the project.



If you then click **Symbols**, right-click on the symbol and choose **Component data**, you can see that it now is stated here as well, that the symbol name is not visible. You can turn symbol texts on and off in the **Component data** dialog box *and* in the dialog box **Text properties**.

Fixed text direction

If this option is chosen, the text will always be directed as it is when you place it on the page. Therefore, if you change the general Reading direction of the page, the text will not change direction. Read more about reading direction in “Specify reading direction” on page 126.

When you have finished specifying the text properties, you click **OK**.

Not translated

In the field **Not translated** you can specify not to translate this text, when translating project texts via the translation program **PC|SCHEMATIC Translator**.

Select text properties from Text/Symbol defaults

In the **Text properties** dialog box it is also possible to select the settings in the dialog box from the settings for a specified text type in **Settings => Text/Symbol defaults**.

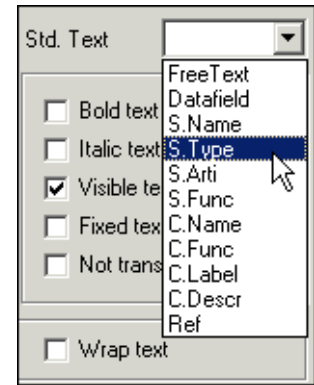
When you click on the down-arrow in the **Std. Text** field in the upper part of the dialog box, you get a list of the various text types, you can set the standard size for under **Settings => Text/Symbol defaults**.

Please note, that you can also select the default settings for free texts.

If for instance you click on **S.Type**, the settings in the dialog box **Text properties** will then become the same as the current default settings for symbol type texts.

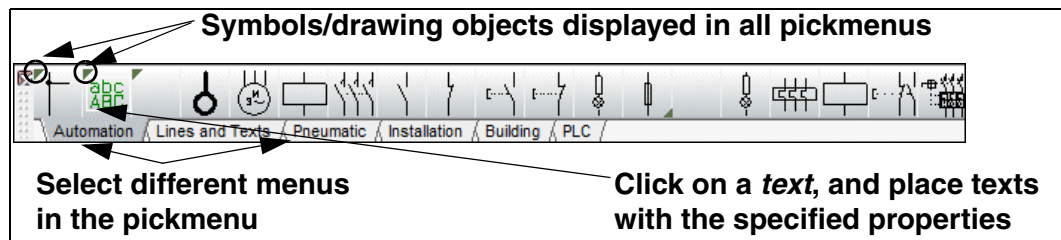
This makes it easier to place free texts having the same appearance as for instance symbol texts.

— Please refer to the “Text/Symbol defaults” on page 831 for further details on the default settings for texts.



Texts in the Pickmenu

To get fast access to drawing texts with predefined properties — see “Specifying text properties / Appearance of text” on page 118 —, you can place texts with the desired properties in the pickmenu at the upper part of the screen.



When you click on one of the *texts* in the pickmenu, the program automatically changes to working with **Texts**, and the cursor is placed in the text-field in the texts toolbar.

When you subsequently type in at text and press <Enter>, you get the text in the cross hairs. When you place the text in the project, the text gets the selected properties.

To insert a text type in the pickmenu, right-click on an empty cell in the pickmenu, and select **Place Text**. The currently applied text properties are hereby inserted in the cell in the pickmenu.

If the pickmenu is not displayed on the screen, you can activate the pickmenu by selecting **Settings => Pointer/Screen** and click on **Pickmenu**.

The pickmenu can also contain symbols, lines and arcs. — Read more about the pickmenu in “Fetching symbols from the Pickmenu” on page 136 and “Adapting the Pickmenu” on page 173.

TEXTS DISPLAYED IN THE PROJECT (GENERAL SETTING)

Entering **Settings => Text/Symbol defaults** you can — as a general project setting — specify which types of texts that can be displayed in the project.

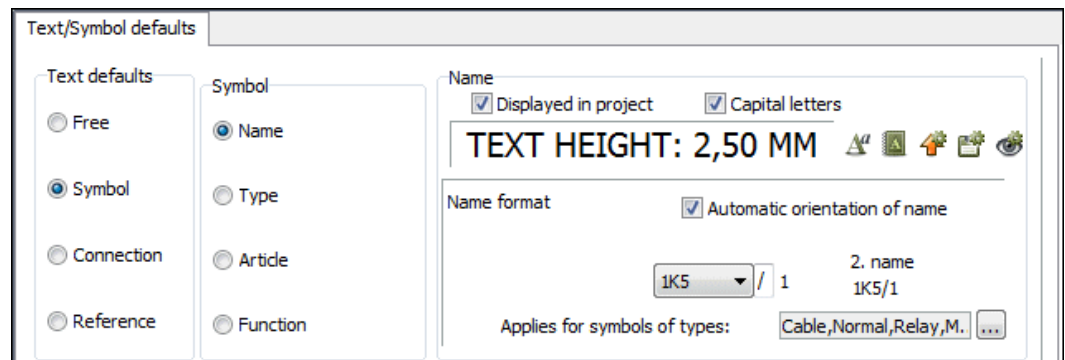
If you here decide that **Symbol article number** is *not to be shown* in the project, you will not be able to display the number for any symbol in the project — not even though you select the **Visible** field for **Name** in the **Component data** dialog box for the symbols in the project.

If you, on the other hand, in **Symbol defaults** have determined that for instance texts of the type **Symbol article number** can be shown in the project, you can for each individual symbol state that for this symbol, the number is not to be shown.

— Read more about this in “Texts connected to symbols” on page 104.

In the left hand side of the **Text defaults** section you determine which type of texts you want to change the settings for. On the right hand side you determine whether the various texts can be displayed in the project.

The contents of the right hand side of the text defaults vary depending on the text type chosen.



If for instance you want the option to display texts of the type **Symbol article number** in the project, you first click on **Symbol**, then on **Article**, and finally you select the **Displayed in project** box.

Click against **Capital letters** if you want the text in capital letters only. Hereby can only type in capitals, when you type in the text. Already placed texts are not changed.

Name format for symbol names

For symbol names you can determine whether the placing of the symbol on page and current path shall be included in the symbol name. Read more about this in “Naming symbols due to page and current path number” on page 370.



Transferring text settings to project texts

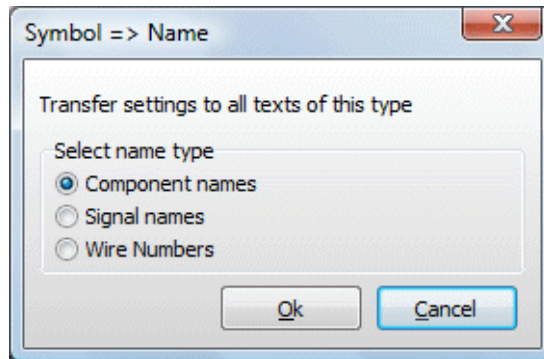
If you wish to transfer the text settings for the selected text type to the project, click the **Transfer text settings to project** button.

All project texts of the selected type hereby gets the settings (text height etc.) specified on the tab **Text/Symbol std.**



Transferring text settings for symbol names

When you click the **Transfer text settings to project** button for symbol names, you must subsequently specify whether to change the settings for **Component names**, **Signal names** or **Wire numbers**:



Select name type, and click **OK**.

Save as default



If you want to save the current settings as future standard, you click the **Save as default** button. Click **OK**. Please note that you only change the default settings for the type of text you are displaying in the dialog box at the present. If you click on the button with the settings shown above, you will only change the settings for symbol names.



If you want to change the default settings for more text types, you have to click the **Save as default** button each time you have changed the settings for one single text type.



If you want to fetch the current default settings for the text type chosen, you click the **Load defaults** button.



If you just want to see what the default settings are, you click the **Show defaults** button, which only displays the default settings as long as you press down the mouse button.

When you have saved settings as standard, new projects will get these settings. Previously created projects will not be changed.



When you return to the drawing, it is a good idea to click the **Redraw** button, to update the drawing with the new settings.

Changing text properties

On this tab you can also change the default appearance / which text properties the program shall suggest for the various text types.



When you have clicked your way to a text type, you determine its appearance by clicking the **Text properties** button. You now enter directly into the **Text properties**

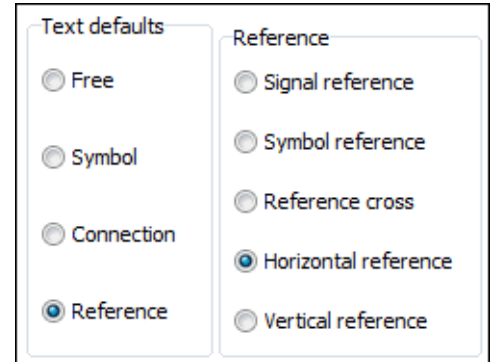
dialog box, as described in “Specifying text properties / Appearance of text” on page 118.

Other text standards

When you click on **Reference**, you can determine the appearance of the current path numbers by clicking on **Horizontal reference** and **Vertical reference** respectively.

If you click on **Reference cross** you can determine the appearance of the texts in the reference crosses that are placed in the project *subsequently*.

Clicking on **Symbol reference**, likewise determine the appearance of the reference texts between symbols that are placed in the project *succeedingly*. **Signal reference** can only be changed in the Edit symbol mode.



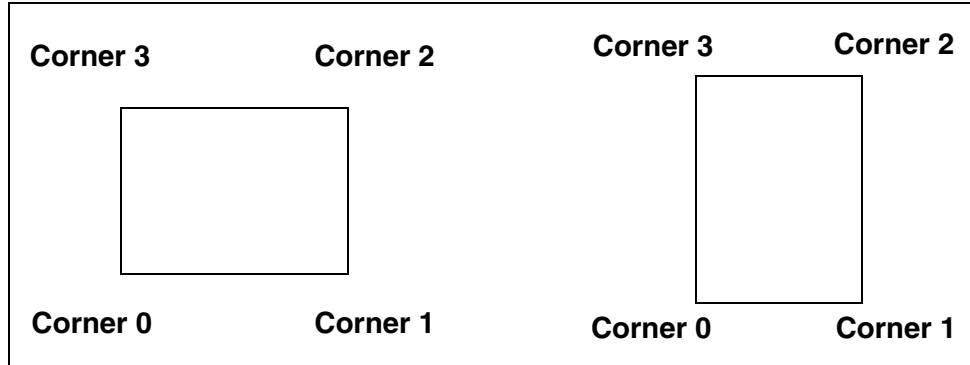
Text standards for Symbol and Connection

Please note that text defaults for **Symbol** and **Connection** only cover the text properties that will be suggested when you are designing symbols. Therefore they can only be changed in the Design/Edit symbol mode.

Already designed symbols will *not* be effected by the changes.

SPECIFY READING DIRECTION

The corners of a piece of paper is always numbered in such a way, that the bottom left hand corner has got number zero, the bottom right hand corner has number one, the upper right hand corner number two, and the upper left hand corner has number three.



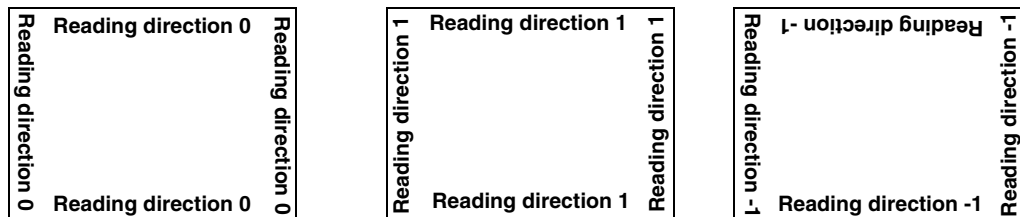
Whether the paper is in portrait or in landscape does not change this.

The texts on a drawing will normally only be read seen from two directions.

The texts on a landscape drawing will normally be read seen from the bottom right hand corner (corner one). However, if you are drawing on a landscape page, which you later print out and rotate to place it in a folder, you must choose reading direction 0 (texts are read from corner zero). The texts then appears correctly when placed in the folder.

The reading direction can be changed in **Settings => Page setup => Reading direction**. You can choose reading direction 0, 1, 2, 3 or -1. The number state the corner from which the texts can be read. This means that the rotation of texts, or of symbols with texts, will change automatically according to the chosen setting. If you wish to change the reading direction on a drawing, you just change the setting, and the texts will change accordingly.

If you set the reading direction to **-1** the texts will keep the orientation they have when you place them in the project. Thus they will not be oriented according to a specific reading direction.



If you have got a drawing in landscape, which after printing have to be inserted into a portfolio, you have to set the reading direction to **0**.

Please note that you have got the option to place individual texts with fixed reading direction. These will not be altered according to the general reading direction in **Page setup**. This facility can for instance be applied in company logos/drawings. See how to place texts with fixed reading direction in “Specifying text properties / Appearance of text” on page 118.

— See “Position of names when placing symbols” on page 127.

Position of names when placing symbols

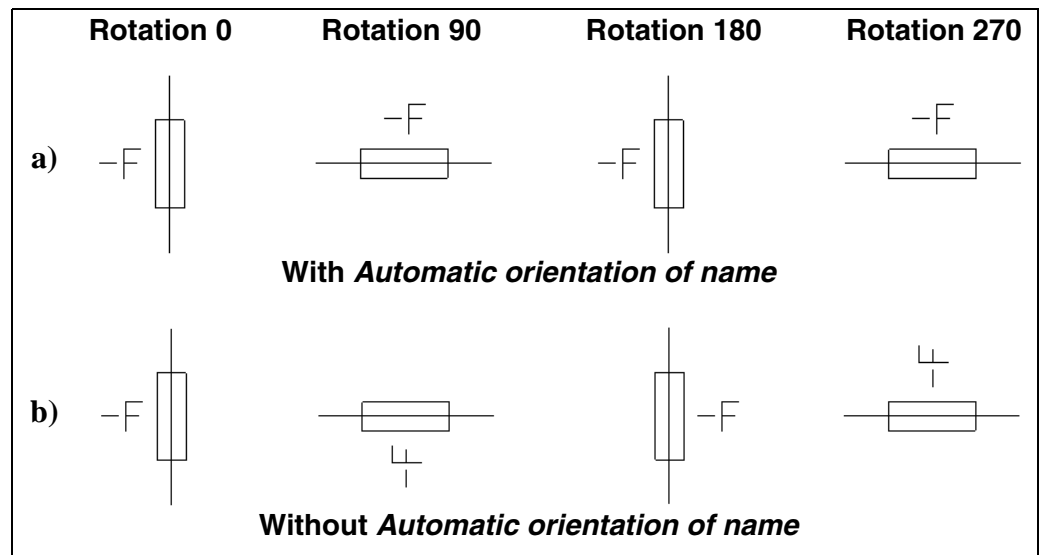


THIS ONLY APPLIES FOR SYMBOLS OF THE TYPE NORMAL

PLEASE NOTE, THAT THIS FUNCTION ONLY APPLIES FOR SYMBOLS OF THE TYPE **NORMAL**.

When symbols are rotated, symbol and connection names can be positioned in two different ways:

- a) Names are not rotated, and they are positioned above or to the left of the symbol, or
- b) Names are rotated with the symbol



How names are positioned in the project, is specified like this:

- 1) Choose **Settings => Text/symbol defaults**
- 2) Below **Text standards**: Select **Symbol** or **Connection**
- 3) Select or deselect **Automatic orientation of name**

The setting only applies for symbols subsequently placed in the project. Already placed symbols are not affected.

By default, **Automatic orientation of name** is selected for both symbol and connection names.



ROTATION: THE RELATION BETWEEN TEXT AND SYMBOL IS FIXED

WHEN YOU ROTATE A SYMBOL, THE SYMBOL TEXTS WILL BE ROTATED WITH THE SYMBOL.

TEXT LINKS

THIS CHAPTER DESCRIBES	
— WHAT IS A TEXT LINK?.....	(P. 128)
— INDICATION OF LINKED TEXTS.....	(P. 128)
— CREATING LINKED TEXTS FROM THE TEXTS COMMAND.....	(P. 128)
— CREATING LINKED TEXTS FROM THE SYMBOLS COMMAND	(P. 129)
— THE DIALOG BOX TEXT LINK	(P. 131)
— EDITING A TEXT LINK	(P. 131)
— DELETING A TEXT LINK.....	(P. 132)
— TEXT LINKS AND LOADING PLC I/O LISTS.....	(P. 132)
— MAINTAINING TEXT LINKS	(P. 132)

What is a text link?

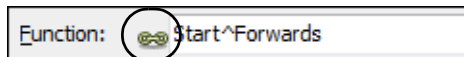
To ensure that all project texts referring to the same contents are always identical, **Automation** contains a function named *text links*. When you have linked two — or more — texts, and edit one of these texts, the other linked texts are changed automatically.

Any function, from which you can edit texts in the program, updates linked texts automatically when editing — see “Where to edit texts in link lists” on page 132.

This is also the case when importing PLC I/O data — see “Text links and importing PLC I/O lists” on page 132.

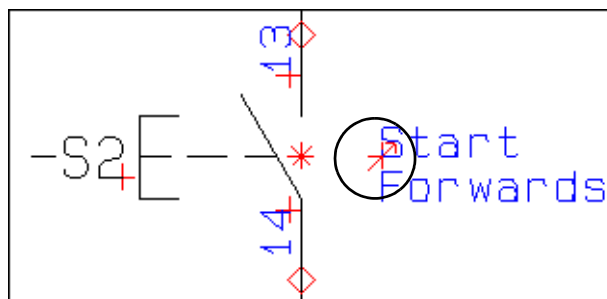
Indication of linked texts

When a text is linked, this is indicated in dialog boxes by a **Flash** in front of the text — for instance in the **Component data** dialog box:



This is also the case in the **Object Lister** — see “The Object Lister” on page 296.

In diagrams linked texts are indicated by an arrow drawn through the reference point of the text:

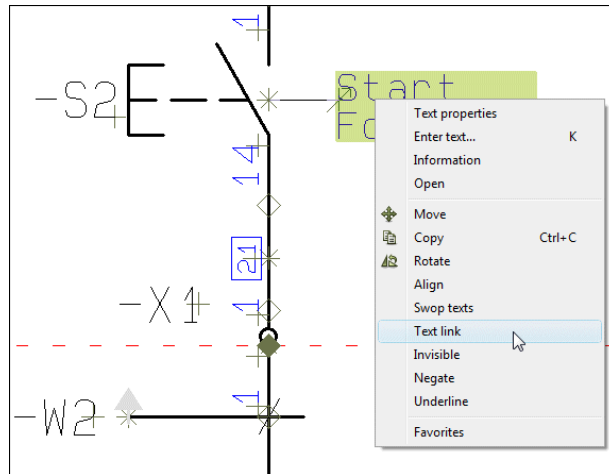


Creating a text link from the Texts command

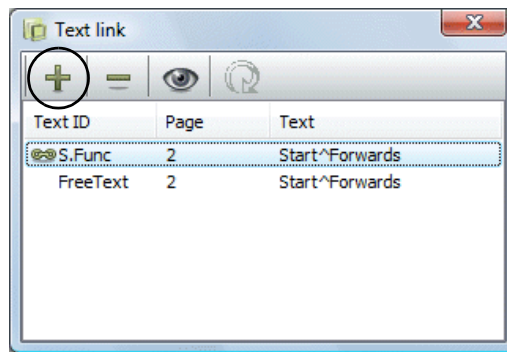
There are different ways of creating text links in **Automation**. When working with texts, do the following:



- 1) Select a text in the project — for instance click at the **Texts** button, then right-click on the text reference point, and choose **Text link**:



- 2) The dialog box **Text link** is displayed: Click on a new text in the project, and click on the **Add text to link** button to link the new text to the text in the dialog box:



- 3) Continue adding texts to the text links until all relevant texts are linked.



- 4) Close the dialog box **Text link** by clicking on the **Close** button in the upper right-hand corner of the dialog box.

— See “Creating text links from Component data and Connection data” on page 129 also.

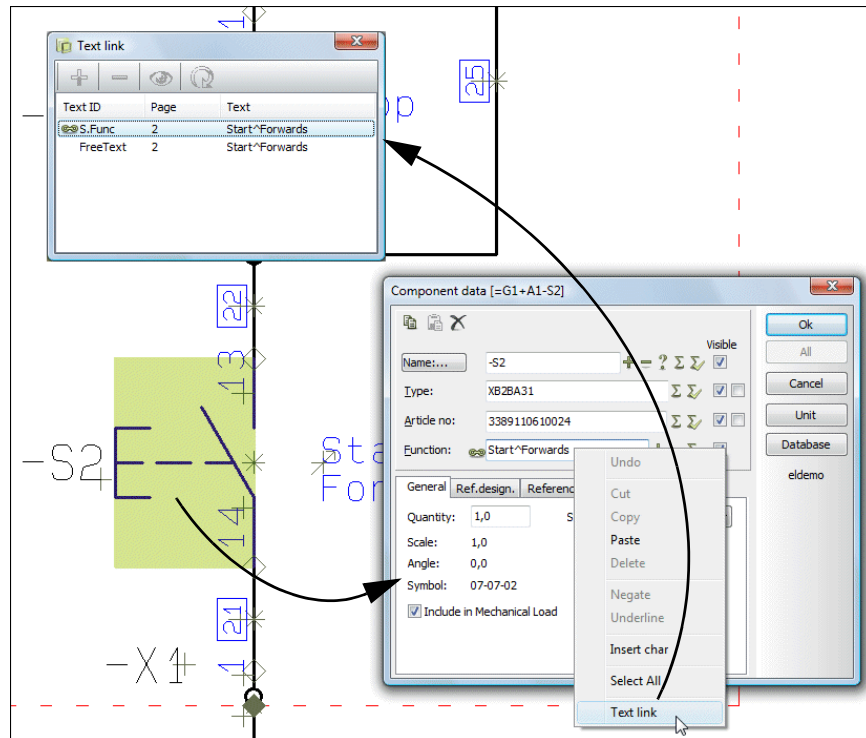
Creating text links from Component data and Connection data

When working with symbols, you can link symbol texts like this:



- 1) Click on **Symbols** — to start working with symbols.
- 2) Right-click on a symbol and select **Component data**.

- 3) In the dialog box **Component data**: Right-click in one of the text fields (see “Symbol names and Connection names can not be linked” on page 130), and select **Text link**.



- 4) You now enter the dialog box **Text link**, from which you can link texts. — If the dialog box **Text link** is already open, the selected text is added to the active link list in the dialog box **Text link**).



- 5) **Close** the dialog box **Component data** for the selected symbol, and add new texts to the link via the **Component data** dialog box for other symbols in the project, or shift to the **Texts** command and link more texts as described in “Creating a text link from the Texts command” on page 128.

- 6) Continue linking texts until all relevant texts are linked.



- 7) Close the dialog box **Text links** by clicking the **Close** button in the upper right-hand corner of the dialog box.

You can do exactly the same for connection points in the dialog box **Connection data**.



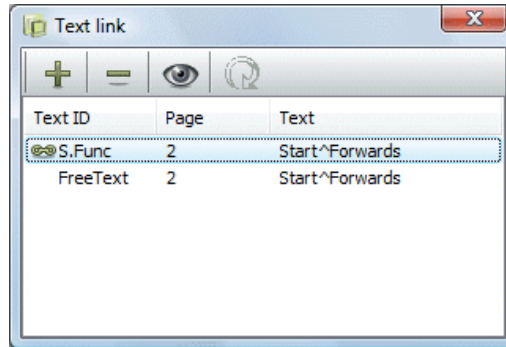
SYMBOL NAMES AND CONNECTION NAMES CAN NOT BE LINKED

IN THE DIALOG BOX **COMPONENT DATA** YOU CAN LINK ALL TEXTS — EXCEPT THE SYMBOL NAME.

IN THE DIALOG BOX **CONNECTION DATA** YOU CAN LINK ALL TEXTS — EXCEPT THE CONNECTION NAME.

The dialog box Text link

In the dialog box **Text link** a list of all texts which are linked together — a *link list* — is displayed:



Here you have the following options:

Linking texts to the texts in the link list



When working with **Texts**, and the dialog box **Text link** is open, you can add new texts to the link list by clicking on a text in the project, and then click on the **Add text to link** button in the dialog box **Text link**.

Removing texts from the selected link



When working with **Texts**, and the dialog box **Text link** is open, you can remove a text from the link list by clicking on the text in the dialog box, and then click on the **Remove text from link** button in the dialog box **Text link**.

Show text location

To jump to a text in the project from the dialog box **Text link**, you have the two following options:



- a) Click on the text in the dialog box **Text link**, and click on the **Show text location** button, or
- b) Double-click on the text in the dialog box **Text link**

The program now displays the page in the project where the text is placed, and selects the text. The dialog box **Text link** remains active on the screen.

Load link list of selected text



When the dialog box **Text link** is open, and you select a new text in **Automation**, and click on the **Load link list of selected text** button, the link list of the selected text is loaded in the dialog box.

Editing text links

When you wish to add texts to a link list after closing the dialog box **Text link** — or you wish to delete texts from the link list — do this:



- 1) Click on the **Texts** button — to work with texts.

2) Right-click on one of the texts in the link list, and open the dialog box by selecting **Text link**.



3) In the dialog box **Text link**: Add texts to the link list as described in “Creating a text link from the Texts command” on page 128, or remove a text from the link list by clicking on the **Remove text from link** button.



4) When you have finished editing: Close the dialog box by clicking on the **Close** button in the upper-right corner of the dialog box.

The text used in the text link list can also be edited from the **Object Lister**, just like you edit non-linked texts. — See “The Object Lister” on page 296.

Deleting a text link list



1) Click on the **Texts** button — to work with texts.

2) Right-click on one of texts in the link list, and open the dialog box **Text link** by selecting **Text link**.



3) Click on the **Remove text from link** button in the dialog box until only one text remains in the dialog box.



4) The links are now deleted: **Close** the dialog box.

Text links and importing PLC I/O lists

When you import a PLC I/O list in **Automation** — see “Reading PLC I/O lists” on page 449 — and a text hereby is changed (a PLC input/output description), which is linked to other texts in the project, these texts are automatically changed to the imported text.

Where to edit texts in link lists

The texts in the link lists can be maintained the following places:



- From the **Symbols** command: In the dialog boxes **Component data** and **Connection data** — see “Creating text links from Component data and Connection data” on page 129



- From the **Texts** command — see “Editing text links” on page 131

- From the **Object Lister** — see “The Object Lister” on page 296

- During import — see for instance “Text links and importing PLC I/O lists” on page 132

When you edit a linked text, all of the other linked texts are changed accordingly — no matter which way you edit the text.



SYMBOLS

In **Automation** you apply *symbols* when you want to represent an electrical component on a drawing. If for instance you want to place a lamp, you find the symbol for a lamp, and place it on the drawing.



When you want to work with Symbols, you always start by clicking the **Symbols** button — or use the short-cut key [s]. — See “Auto command change” on page 51 also.

The general short-cut key for the Pencil is the [Ins] key, but you can also use the [s] key when you are working with Symbols. Pressing the [Ins] or the [s] key activates/deactivates the Pencil.

THIS CHAPTER DESCRIBES	
— FETCHING SYMBOLS WITH AND WITHOUT USING THE DATABASE	(P. 135)
— PLACING AND NAMING SYMBOLS	(P. 151)
— SYMBOL SUB NAMES	(P. 160)
— SYMBOLS WITH STATES	(P. 162)
— UPDATING SYMBOLS FROM THE DATABASE	(P. 166)
— MOVING AND DELETING PLACED SYMBOLS	(P. 168)
— THE SYMBOL TOOLBAR	(P. 171)
— ADAPTING THE PICKMENU	(P. 173)
— CHANGING SYMBOL NAMES IN AREAS	(P. 180)
— SYMBOL CONNECTION POINTS	(P. 182)
— CHOOSING CONNECTION POINTS IN AREAS	(P. 184)
— THE PINSWOP FUNCTION	(P. 185)
— REPLACING SYMBOLS	(P. 187)
— AUTOMATIC CREATION OF SYMBOLS	(P. 190)
— THE SHOW NET FUNCTION AND THE NETLISTER	(P. 195)
— USING SIGNAL SYMBOLS	(P. 196)
— USING TERMINAL SYMBOLS	(P. 207)
— USING MULTILAYER TERMINAL SYMBOLS	(P. 208)
— USING CABLE SYMBOLS	(P. 210)
— USING ACCESSORY SYMBOLS	(P. 217)
— SYMBOL FOLDERS IN Automation	(P. 217)



FUNCTIONS ON COMPONENT LEVEL

MORE SYMBOL FUNCTIONS ARE DESCRIBED IN “FUNCTIONS ON COMPONENT LEVEL” ON PAGE 220. THE RELATIONSHIP BETWEEN COMPONENTS AND SYMBOLS ARE DESCRIBED IN “THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS” ON PAGE 221.

— Please refer to “Creating Data symbols” on page 711 to see how to use data symbols.

See how to move, copy, delete and rotate symbols in the section “Working with selected objects” on page 51, and see how to transfer object properties between symbols in “Transferring Component Data for Symbols or Lines” on page 55.



PLC FUNCTIONS

READ MORE ABOUT THE SPECIAL OPTIONS FOR WORKING WITH PLC SYMBOLS IN “PLC FUNCTIONS” ON PAGE 433.

Designing your own symbols

If you can not find the symbol you need among the symbols that come with **Automation**, then read “Creating Symbols” on page 676, to see how you design your own symbols.

SEE VIDEO ON **FETCHING SYMBOLS**

OUR WEBSITE

When working with symbols, you always start by clicking the **Symbols** button or by pressing the short-cut key [s].



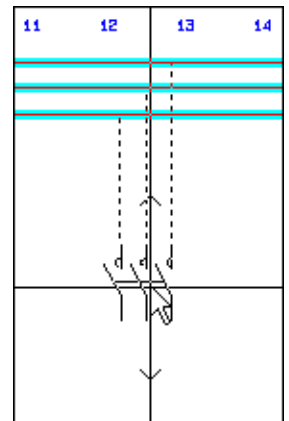
Beyond copying an already placed symbol, there are several ways of fetching a symbol in **Automation**:

WAY TO FETCH SYMBOLS	PAGE
<i>FROM THE PICKMENU</i>	136
<i>FROM THE SYMBOL MENU</i>	140
<i>FROM THE DATABASE</i>	147
<i>TYPING IN A SYMBOL FILENAME</i>	148
<i>TYPING IN AN ARTICLE NUMBER</i>	149
<i>TYPING IN A SYMBOL TYPE</i>	150
<i>DIRECT CREATION OF SYMBOLS</i>	190
<i>FROM WINDOWS EXPLORER</i>	321
<i>USING A BARCODE SCANNER</i>	615

Some of the functions are handy when you work without using the database, others when you know the exact article number of the component you want, others again when you know which type of component you want, but do not know its exact article number.

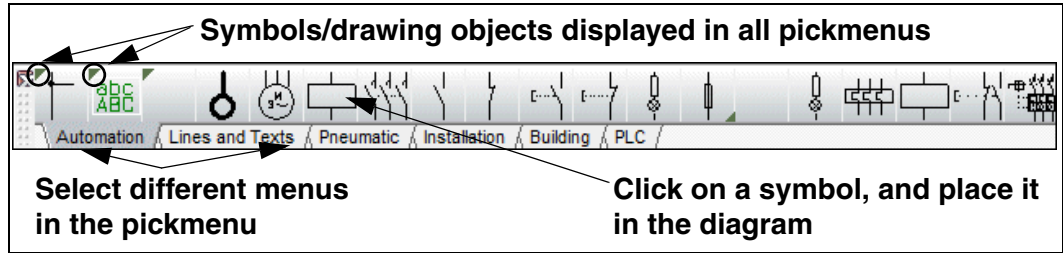

AUTOMATIC LINE DRAWING WHEN PLACING SYMBOLS

WHEN YOU PLACE A SYMBOL YOU CAN USE THE **ROUTER** FUNCTION, WHICH DRAWS THE LINES CONNECTING THE SYMBOL TO THE DIAGRAM AUTOMATICALLY. — READ MORE ABOUT THIS IN “AUTOMATIC LINE DRAWING / ROUTER” ON PAGE 81.



1. Fetching symbols from the Pickmenu

When working in **Automation**, you have the **Pickmenu** available on the upper part of the screen. Here you place the symbols — and other drawing objects — you use the most.



When you click on a *symbol* in the pickmenu, you get the symbol in the cross hairs, and can place it in the project. The program changes automatically to work with **Symbols**, and the **Pencil** is activated.

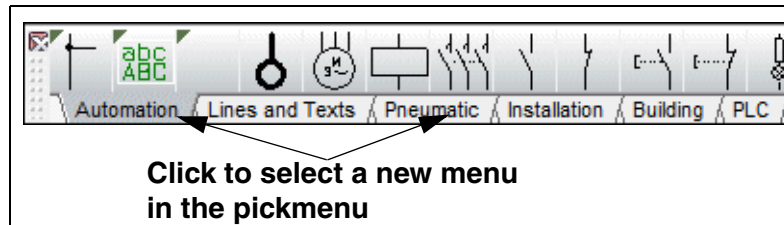
If the pickmenu is not displayed, choose **Settings => Pointer/Screen** and click on **Pickmenu**.

OTHER POSSIBILITIES WHEN USING THE PICKMENU	
— CHANGING BETWEEN MENUS IN THE PICKMENU	(P. 136)
— SYMBOLS / DRAWING OBJECTS DISPLAYED IN ALL PICKMENUS	(P. 137)
— SYMBOLS WITH COMPONENT DATA IN THE PICKMENU	(P. 137)
— ENTERING THE DATABASE USING THE PICKMENU	(P. 138)
— ALTERNATIVE ELECTRICAL SYMBOLS	(P. 138)
— SYMBOLS WITH STATES IN THE PICKMENU	(P. 139)
— POSSIBILITIES AND LIMITATIONS WHEN USING THE PICKMENU	(P. 139)

The pickmenu can also contain line types, text types and arcs. — Read more about the pickmenu in “Lines in the Pickmenu” on page 71, “Texts in the Pickmenu” on page 122 and “Circles / Arcs in the Pickmenu” on page 97.

Selecting the different Pickmenus

The pickmenu contains several different menus, which you select by clicking on the tabs at the bottom of the pickmenu.

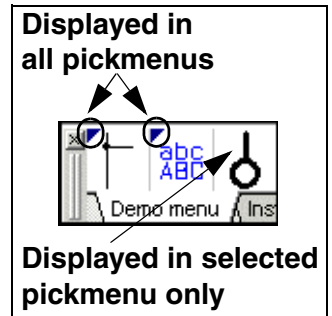


— See how to create new menus in the pickmenu in “Creating new menus in the Pickmenu” on page 175.

Symbols/ Drawing objects displayed in all Pickmenus

Symbols / drawing objects with a small triangle in the upper-left corner of the pickmenu cell, are displayed in all menus in the pickmenu.

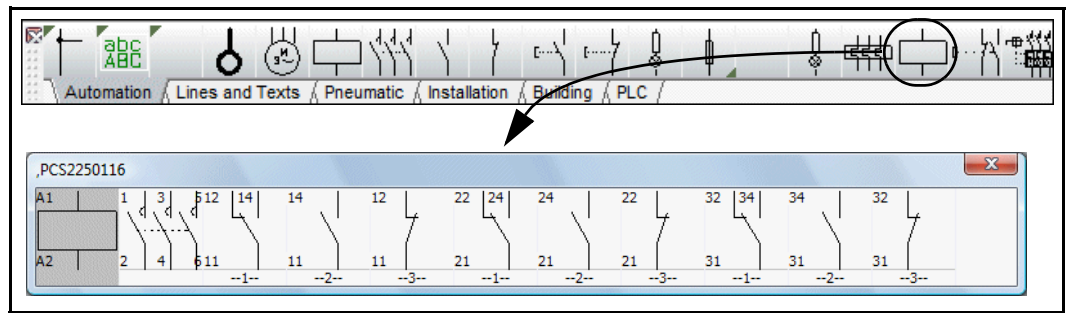
— See how to give a symbol / drawing object this status in “Inserting a Symbol / Drawing object in all Pickmenus” on page 173.



Symbols with component data in the pickmenu

The symbols in the pickmenu can be attached to components in the database. This way you automatically attach component data to the symbols when you place them in diagrams.

When you click on a symbol/component that contains more than one function in the database, you get a pickmenu containing the electrical symbols for the component.



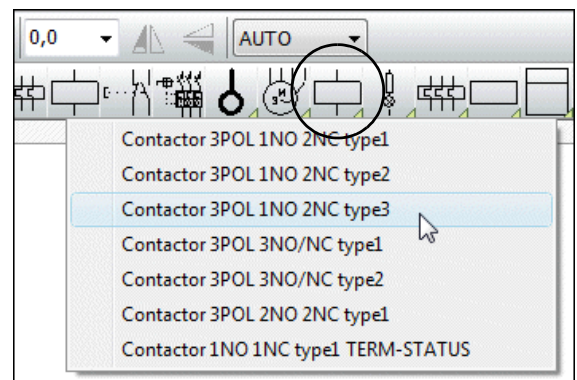
Click on one of the symbols and place it. Press [Ctrl+F9] to get this pickmenu for the component on screen again. — See “The function Show available” on page 224 or “The function Show available window” on page 226.

When you let the cursor rest for a moment over one of the symbols in the pickmenu, the component data is displayed below the symbol (as a *hint*) and at the bottom line of the screen.

SYMBOLS WITH MULTIPLE SETS OF DATA IN THE PICKMENU

You can also assign multiple sets of component data to a symbol (a cell) in the pickmenu. This way the same cell can contain the same component with e.g. different capacities.

When you let the mouse rest over a cell containing multiple sets of component data, a submenu is displayed for the different sets. — See “Automatic popup of submenu in the pickmenu” on page 174.



You can see that a symbol cell contains component data, by the light-blue rectangle in the lower-right corner of the cell.

— See how to create multiple sets of component data for a symbol in a cell in “Adding component data to symbols in the pickmenu” on page 175.

Entering the database using the pickmenu

If you press down <Ctrl> while clicking on a symbol in the pickmenu, you enter the database. Here you can see the components that have got the picked symbol as the first symbol in the **Symbol** field of the database. — See “Database settings” on page 598 and/or the **PC|SCHEMATIC Database** manual.

It is also possible to right-click on the symbol in the pickmenu, and choose **Data-base**.

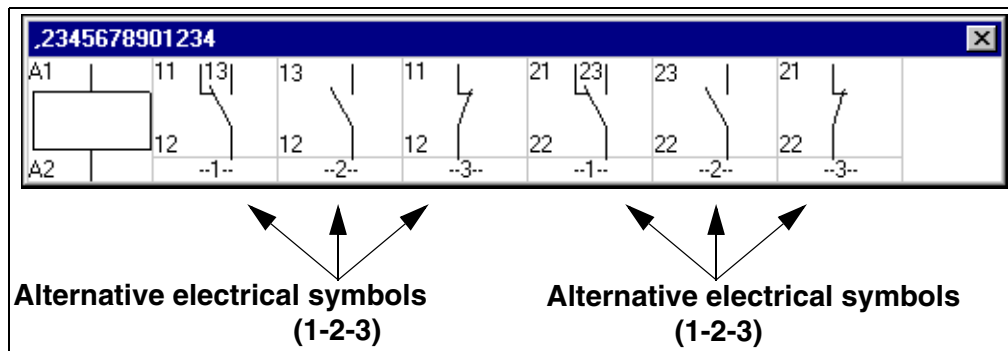
Select the desired component in the database and click **OK**.

Alternative electrical symbols

When you fetch a component via the database — for instance by using the pick-menu — it is some times possible to choose between alternative electrical symbols for the functions of the component.

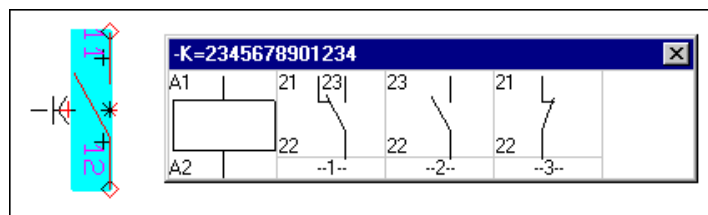
If for instance a component has a relay coil and two switch functions, the switch functions can be used as either a switch, a make contact or a break contact.

When you get the pickmenu for such a component, it could look like this:



When you choose one of these alternative symbols, the other alternatives for the function disappears.

If for instance you choose the symbol for the make contact for the first switch function, the two alternative symbols for switch and break contact disappears from the pickmenu for the component.



— Please also see “Symbols with states” on page 139.

Other options for using alternative symbols

There exists many other ways to use alternative symbols than just the example above.

Power control circuit symbols can for instance exist in a normal version and a version for one-line diagrams. You can also choose whether to use PLC modules or single I/O symbols.

— See how to specify the alternative symbols in the database in “Specifying alternative electrical symbols via the database” on page 584.

Symbols with states

Symbols with states can also be placed in the pickmenu. When a symbol in the pickmenu has states, a dark blue triangle is displayed in the lower-right corner of the cell in the pickmenu.

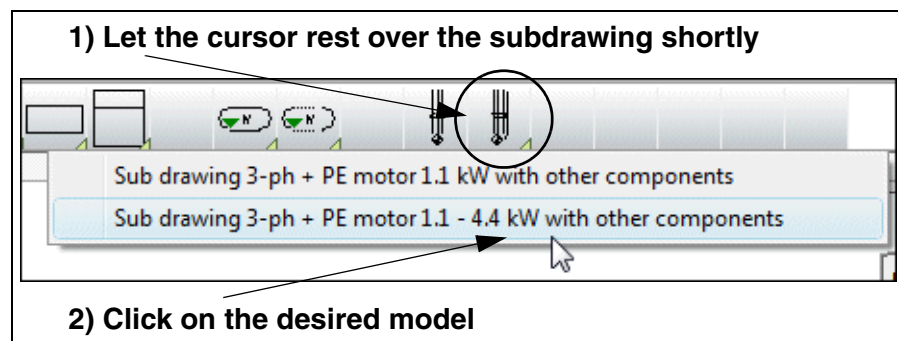
You can specify whether to display a submenu automatically, as described in “Automatic popup of submenus in the pickmenu” on page 174. When you let the mouse rest over the state for a symbol, the title for the state is displayed, if a title for the state exists.

— See “Symbols with states” on page 162, “Symbols with states for single line diagrams” on page 398 and “Creating symbols with states” on page 692.

Sub drawings in the Pickmenu

When you click on a subdrawing in the Pickmenu, you get the sub drawing in the crosshairs, and can place it in the project.

If the sub drawing contains data for different models, you select the model like this:



When you choose to place the sub drawing on a mechanical page, the mechanical symbols for the sub drawing are placed on the page.

— See how to place sub drawings in the Pickmenu in “Placing sub drawings in the Pickmenu” on page 177.



THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS

IN “MANUAL PLACING OF SUB DRAWINGS” ON PAGE 247 YOU CAN READ MORE ABOUT THE NORMALLY RECOMMENDED PROCEDURE FOR PLACING SUB DRAWINGS USING THE **SUBDRAWINGS** TAB IN THE **EXPLORER** WINDOW. READ MORE ABOUT SUB DRAWINGS IN THE CHAPTER “SUB DRAWINGS” ON PAGE 244.

Possibilities and limitations when using the pickmenu

You have the option of creating 999 different pickmenus. By clicking the up- and down arrows in the left side of the pickmenu, you can jump from one pickmenu to another. — See how to place symbols in the pickmenu in “Adapting the Pickmenu” on page 173.

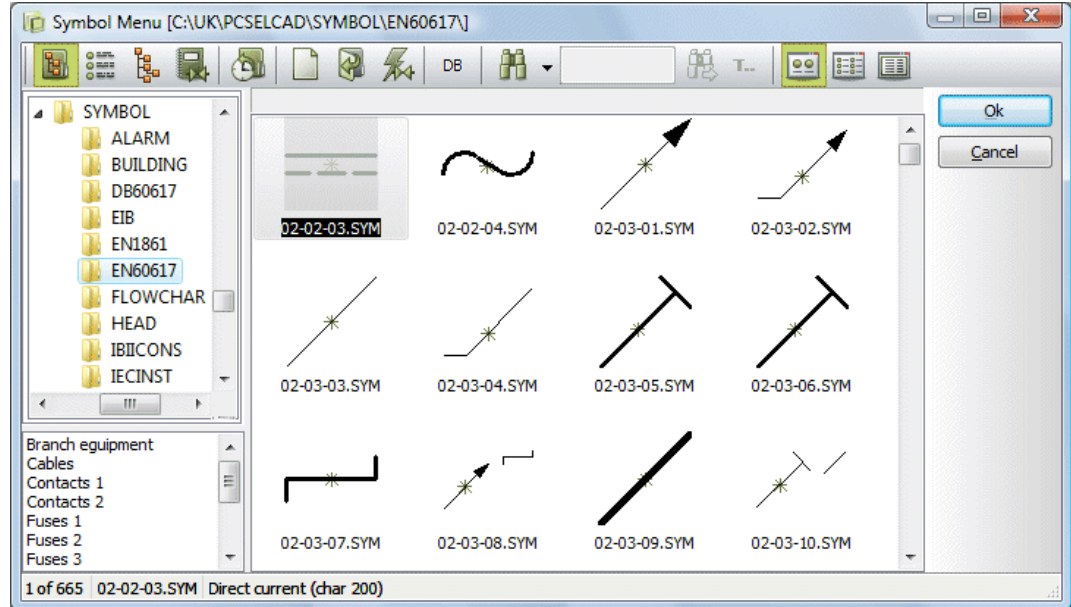
The pickmenu can be moved around on the screen as an ordinary window in Windows, and can also be docked. When it is docked it will always be displayed on the screen. You can read more about this in “Docking the Pickmenu” on page 25.



2. Fetching symbols using the Symbol menu

Click the **Symbols** button and the **Symbol menu** button — or press the short-cut key [F8], which can be applied from all commands.

In the **Symbol menu** you then pick the symbol you need:



Placing the found symbol

Click on the desired symbol and click **OK** — or double-click on the desired symbol —, and you return to your diagram with the symbol in the cross hairs.

Place the symbol in the project, and press <Esc> to remove the symbol from the cross hairs.

POSSIBILITIES IN THE SYMBOL MENU	
— SYMBOL LIBRARIES / PROJECT SYMBOLS IN THE SYMBOL MENU	(P. 141)
— REPRESENTATION OF SYMBOLS IN THE SYMBOL MENU	(P. 141)
— SEARCHING IN THE SYMBOL MENU	(P. 142)
— BOOKMARKS IN THE SYMBOL MENU.....	(P. 143)
— ENTERING THE DATABASE VIA THE SYMBOL MENU	(P. 146)
— CREATING NEW SYMBOLS FROM THE SYMBOL MENU	(P. 146)



AUTOMATIC UPDATE OF THE SYMBOL MENU

IF SYMBOLS, OR THE CONTENTS OF THE SYMBOL DIRECTORY DISPLAYED IN THE **SYMBOL MENU** CURRENTLY, ARE CHANGED FROM OUTSIDE THE PROGRAM — E.G. VIA WINDOWS EXPLORER OR OTHERS ON THE NET — THE **SYMBOL MENU** IS UPDATED AUTOMATICALLY.

Displaying symbol libraries / project symbols in the Symbol menu

When you wish to locate symbols in the **Symbol menu**, you can choose which symbols to display, in the following way:

Display Library via Folders



Click the **Select Libraries from Folders** button. You then get an explorer structure in the left-hand side of the **Symbol menu**, where you can find the folder in which the desired symbol library is placed.

In the folder **PcseIcad** on the **C:**-drive, you find the folder **Symbol**. In this folder you find a number of sub folders containing various symbol libraries.

When you have chosen a folder here, such as **EN60617**, you see these symbols displayed in the window in the **Symbol menu**. When there are more symbols than can be displayed in the window, there will be a scroll bar in the window.

The contents of the symbol libraries is described in “Symbol folders” on page 217.

If you need to know more about changing folders and opening files, please read “Windows” on page 21.

Selecting Libraries via Alias



Click on the **Select Library from Alias** button. Hereby a list of the created *aliases* in the left-hand side of the **Symbol menu**. Each of these aliases points on a folder, which contains a symbol library.

When installing **Automation**, aliases for the installed symbol libraries are created automatically.



YOU CAN USE ALIASES, WHICH POINTS ON SYMBOL LIBRARIES PLACED ALL OVER THE WORLD.

— Read more about *alias* in “Show directory alias” on page 820.

Showing the Project Symbols



Click on the **Show Project Symbols** button. Hereby all symbols in the project are displayed.

Show the most recently used symbols



Click on the **History** button. Hereby the last 50 symbols fetched via the **Symbol menu** are displayed. This also includes symbols fetched via the **Symbol menu** when working in other projects.

Representation of symbols in the Symbol menu

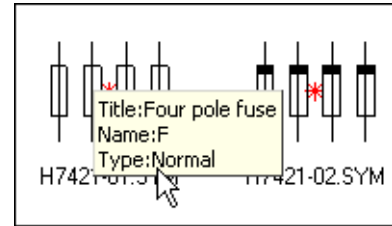
In the upper right-hand corner of the **Symbol menu**, you will find three buttons to specify how the symbols are displayed in the **Symbol menu**:

Symbols displayed as Large icons



When you click on the **Large icons** button, the symbols are displayed in the **Symbol menu**.

When placing the mouse cursor over a symbol, the **Title**, **Name** and **Type** of the symbol is displayed as a *hint*.

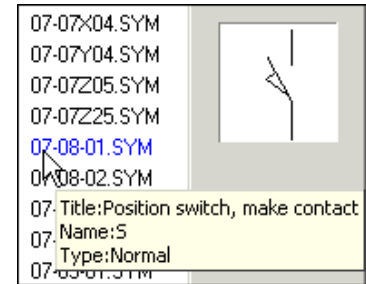


Symbol names displayed in a List



When you click on the **List** button, a list of the symbol file names are displayed.

When placing the mouse cursor over a symbol name, the symbol is displayed in the right-hand side of the menu, and the **Title**, **Name** and **Type** of the symbol is displayed as a *hint*.



Symbol names displayed with Details



When you click on the **Detail** button, a list of the symbol file names is displayed, containing detailed information on the selected symbols.

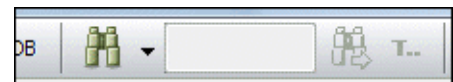
H7415L21.SYM	Four pole circuit breaker w. over-curre...	Q	With refer...
H7415M21.SYM	Four pole 3P+N circ.breaker w.aut.ov...	Q	With refer...
H7415N21.SYM	Three-pole +N automatic operating fuse	Q	Master ref...
H7418D01.SYM	Four pole HFI/HPFI disconnect	F	Normal
H7418E01.SYM	Four pole HPFI relay	F	Master ref...

When placing the mouse cursor over a line in the list, the symbol is displayed in the right-hand side of the menu.

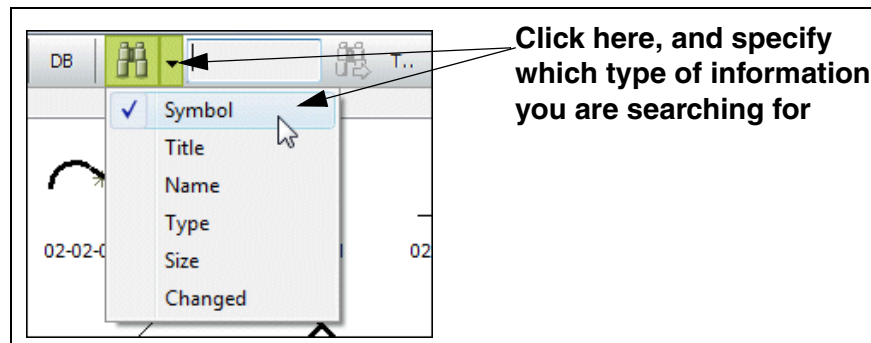
Searching in the Symbol menu



When you need to search after a symbol in the selected library, click the **Search** button — or press [Ctrl+f].



A search field is then displayed. Here you can type in the text you are searching for. Click on the down-arrow to the right of the Search button to specify which type of information you are searching for:



Then type in the symbol text you are searching for in the search field. As you type in the text, the program will select the first symbol in the symbol library, having a symbol text beginning with the text you have typed in so far. To jump to the next symbol matching the text, click the **Search again** button — or press [F3].

It makes no difference for the program whether the text has been written as upper case or lower case letters.

Begins with/Contains

If you take no special action, the program starts searching for texts beginning with the text you type in. When you wish to search for a text that just exists somewhere in the selected text type, click on the **Begins with/Contains** button, so that it changes appearance from **T...** to **..T...**



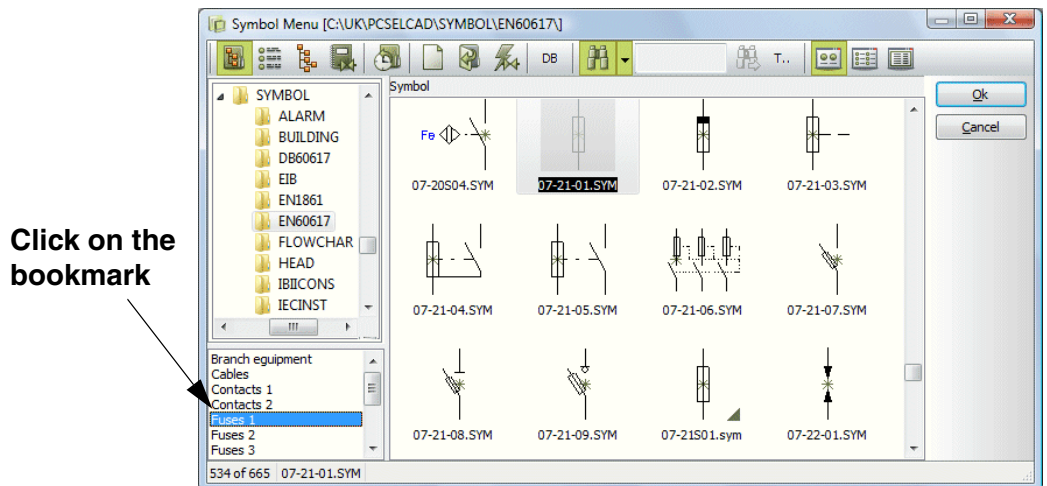
When this happens, the program searches for the text fragments anywhere in the selected text types.



To return to searching for texts beginning with the typed in text, click on the **Begins with/Contains** button again, so that it changes appearance from **..T..** to **T...**

Bookmarks in the Symbol menu

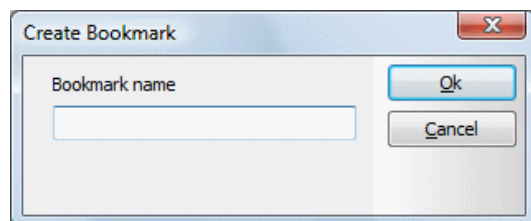
When the **Symbol menu** displays aliases or folders, it is possible to insert bookmarks. When you click on a bookmark, you jump down to the selected bookmark in the folder.



Creating a bookmark in the Symbol menu

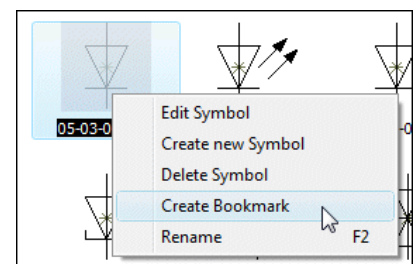
To create a bookmark, right-click on a symbol in the **Symbol menu**, and choose **Create bookmark**.

You then enter the dialog box **Create Bookmark**:



Type in the name of the bookmark, and click **OK**.

The bookmark is now displayed in the lower left-hand corner of the dialog box, when you select the alias or library in question in the **Symbol menu**.



Editing or Deleting created bookmarks

To edit or delete a created bookmark, right-click on the bookmark and choose **Edit bookmark** or **Delete bookmark**.

Bookmarks and copying symbol libraries

Information about bookmarks are saved in the relevant symbol libraries. This has the following consequences:

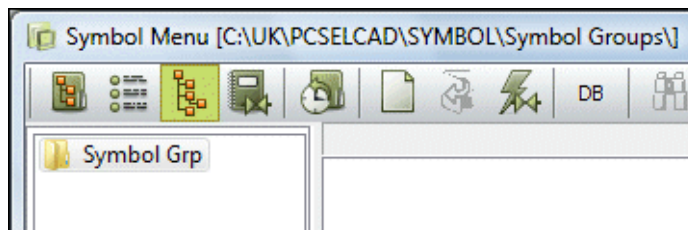
- 1) When there is access to the library from other PCs on the network, the bookmarks can also be seen from these PCs.
- 2) When you copy the library, the applied bookmarks are included automatically.

Hierarchical displaying of symbols in the Symbol menu

In **Automation**, the symbols are placed in different folders, without any hierarchical structure (see “Bookmarks in the Symbol menu” on page 143 also).



When you click the **Symbol groups** button in the **Symbol menu**, you get the option of creating your own hierarchical structure, which can make it easier for you to find the symbols. You then enter the folder **Symbol Grp**, which is placed under **Pcscad\Symbol**:

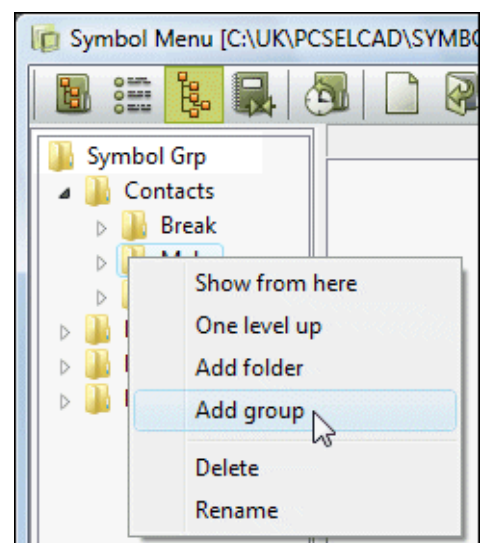


Here you can create your own grouping of the symbols, which you already have access to.

This is done by first creating a structure of folders with relevant names. In these folders, you create the so-called *groups*, and it is in these groups that you can place the symbols. Please note, that it is *not* possible to place the symbols in a *folder*, they can only be placed in a *group*.

Example on creating a hierarchical structure

- 1) Create the first folder: Right-click on the folder **Symbol Grp**, and select **Add folder**. Type in the folder name **Contacts**, and click **OK**.
- 2) Create sub folders: Right-click on the folder **Contacts**, and create the sub folder **Make**, just like you created the first folder. Under the **Contacts** folder, you also create the sub folders **Break** and **Switch**.



- 3) Create a group: Right-click on the folder **Switch** and select **Add group**. Type in the group name **1P**, and click **OK**. In this group **1P**, you can now place symbols.



SYMBOLS ARE NOT PLACED DIRECTLY IN THE GROUPS

THE SYMBOLS ARE NOT PLACED DIRECTLY IN THE CREATED GROUPS. INSTEAD, A *LINK* (OR A *REFERENCE*) IS CREATED TO THE FOLDER WHERE THE SYMBOL IS PLACED IN THE SYMBOL MENU. WHEN A SYMBOL IS CHANGED VIA THE EDIT SYMBOL FUNCTION, THE SYMBOLS ARE THEREFORE AUTOMATICALLY CHANGED IN ALL THE GROUPS, WHICH REFERS TO THE THE EDITED SYMBOL.

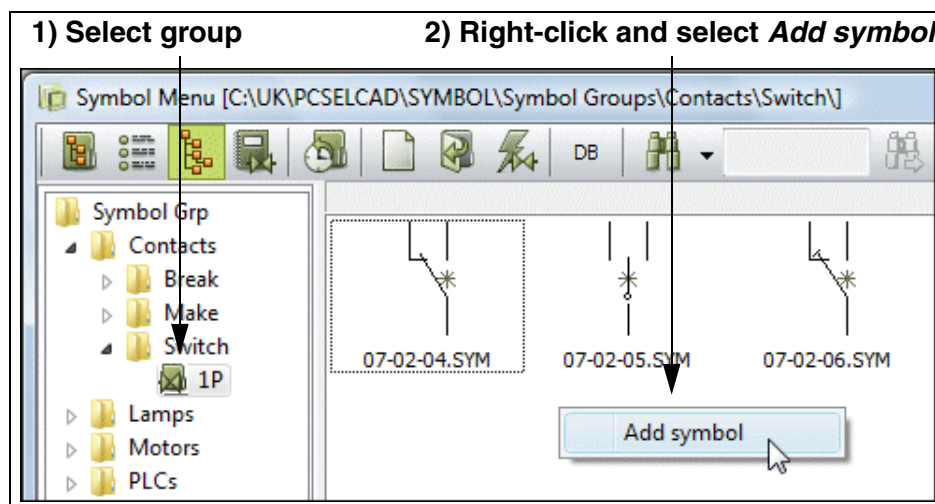
THE SAME SYMBOL CAN APPEAR IN DIFFERENT GROUPS.

You have now created a hierarchical structure, containing a group, in which you can place symbols.

Placing symbols in a hierarchical structure

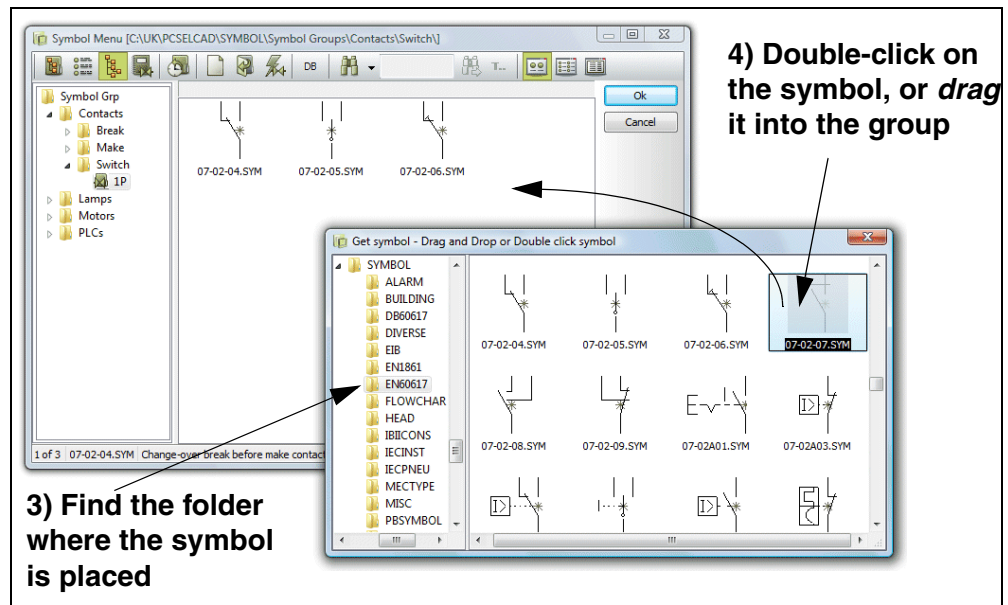
To place symbols in a group in the hierarchical structure, do this:

- 1) Select the group in the left side of the **Symbol menu**.
- 2) Right-click in the right side, and select **Add symbol**:



- 3) The dialog box **Get symbol** is now displayed: Select the folder where the symbol is placed, by clicking on the folder in the left side of the window.

- 4) Now find the symbol in the right side of the window, and double-click on the symbol to add it to the group. You can also *drag* the symbol from the **Get symbol** window, and place it in the group:



Remove symbol from group

If you wish to remove a symbol from a group, right-click on the symbol, and select **Remove symbol**.

Entering the database from the Symbol menu

To find symbols in the database which has attached symbols of a specific type, do the following:



- 1) Enter the **Symbol menu**.
- 2) Find the desired symbol, and click on it.



- 3) Click on the **Database** button, and you enter the **Database menu**.

In the database you now see all the components which has the selected symbol as the first symbol in the **Symbol** field in the database. — See “Database settings” on page 598 and/or the **PC|SCHEMATIC Database** manual.

— Read more about fetching symbols via the database in “Fetching symbols from the database” on page 147.

Creating symbols or Editing existing from the Symbol menu



To create a new symbol, click on the **Create new symbol** button.

When you wish to edit an existing symbol, click on it in the **Symbol menu**, and click on the **Edit symbol** button.

— Read more about this in “Creating Symbols” on page 676.



3. Fetching symbols from the database

If you already know which components you are going to use, you can fetch the symbol directly from the database.

Entering the database



Click the **Symbols** button, press down <Ctrl> and click the **Symbol menu** button. You then enter the database directly, if it is on (See “Using the database” on page 561 for how to turn a database on). You can also press the short-cut key [d].

Select component group →

Choose e.g. Fabricate →

Components →

Searching and navigation →

EANNUMBER	TYPE	DESCRIPTION	SOURCE
1122094094	LS 37.22	Contactor, 190W, LS 37.22, 230V AC	AEG
4022903095584	8775	Overload relay, 20-32A	AEG
3389110231632	LCID4011M7	Contactor, LCID4011M7, 220V-40A	GRUPE SCHNEIDER
3389110232936	LR2D33S3	Overload relay, 23-32A	GRUPE SCHNEIDER
402903075387	L519K11	Contactor, 190W, L519K11, 230V AC	AEG
1122001339	L519K11	Contactor, 190W, L519K11, 24V AC	AEG
5702423013623	CB-HO	Auxiliary contact, 1NO	DANFOSS
5702423015932	CB-NC	Auxiliary contact, 1NC	DANFOSS
5702423006175	ETB	Time delay unit, off-delay 0,5-20s, 110-240V	DANFOSS
5702423009237	CI 32	Contactor, 3P 32 A, 220/230 V AC	DANFOSS
5702423010189	TI 80	Overload relay, 22-32 A	DANFOSS
3389110230253	LA1DN01	Auxiliary contactor, 1NC	GRUPE SCHNEIDER
3389110230284	LA1DN10	Auxiliary contactor, 1NO	GRUPE SCHNEIDER
3389110230321	LA1DN22	Auxiliary contactor, 2NO+2NC	GRUPE SCHNEIDER

OPTIONS IN THE DATABASE MENU	
— SELECTING A SYMBOL IN THE DATABASE	(P. 147)
— PLACING SELECTED SYMBOLS FROM THE DATABASE	(P. 148)
— DISPLAY THE PICKMENU FOR THE COMPONENT AGAIN	(P. 148)

Selecting a symbol in the database

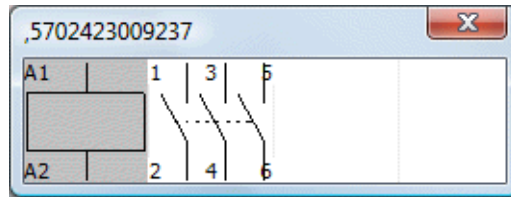
Click on a folder, for instance **Automatic switches / connection material**, and choose e.g. a **Fabricate** in the upper right-hand corner of the dialog box, and select a component.

Select for instance the component with EAN number **4022903075387** — as displayed above. Click **OK**.

— Read more about searching in the **Database menu** in “Using the database” on page 561.

Placing the selected symbols from the database

A special pickmenu, containing all the electrical symbols for the component, now appear on the screen.



If the component consists of just one symbol, you automatically get this symbol in the cross hairs.

Otherwise, click on the symbol you want to use, and place it where you want it. When you click on a symbol, the pickmenu disappear automatically.

Displaying the pickmenu for the component again

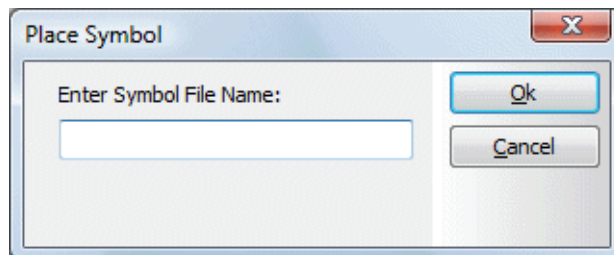
To make the pickmenu reappear, choose **Functions => Show available again** or press the short-cut key [Ctrl+F9]. Please refer to “The function Show available” on page 224 for further details.

— See also “The function Show available window” on page 226.

4. Typing in the symbol file name directly



When you have activated **Symbols** and press the short-cut key [k], you enter this dialog box:



Type in the file name of the symbol you want, and press <Enter>. The program now search the contents of the folder you used last time you used the **Symbol menu**. If the program does not find the symbol there, the program searches the symbol libraries with *alias* — as described in “Show directory alias” on page 820.

If the program can not find a matching symbol, you will enter the **Symbol menu**. If there exists a symbol in the **Symbol menu** whose name starts with the text you typed in, it will be selected.

— In “Automatic creation of symbols” on page 190 you can see how to create symbols directly from this dialog box.

5. Typing in the Article Number directly



When you have activated **Symbols** and choose **Functions => Database => Find Article** — or press the short-cut key [v] — you enter this dialog box:

The dialog box titled "Place Symbol" has a close button (X) in the top right corner. It contains a text input field labeled "Enter Symbol Article:" with the number "40" entered. To the right of the input field are two buttons: "Ok" and "Cancel".

Here you type in as much you can remember of the article number, and click **OK**.

You then enter the database, which displays all components having an article number matching what you typed in.

If you typed in *40* — as in the figure above — the database will look like this:

The screenshot shows the "Database" window with a tree view on the left and a table of search results. The table has columns for EANNUMBER, TYPE, DESCRIPTION, and SOURCE. The first row is selected, showing a push button with plot light from SIEMENS.

EANNUMBER	TYPE	DESCRIPTION	SOURCE
4001869040813	STE4702	Push button with plot light, 1NC	SIEMENS
4008190100032	WDU 6	Screw clamp terminal, blue	WEXO
4008190163440	WDU 6	Screw clamp terminal	WEXO
4008190297749	WDU 2,5	Screw clamp terminal, black	WEXO
4008190297756	WDU 2,5	Screw clamp terminal, white	WEXO
4008190455149	WDU 2,5	Screw clamp terminal, grey	WEXO
4008190455248	WDU 4	Screw clamp terminal, brown	WEXO
4008190455293	WDU 4	Screw clamp terminal, grey	WEXO
4011209317055	3RA1210-0BA15-QAP0	LOAD FEEDER FUSELESS REVERSING DUTY	
4015080297437	RL-GN/FR	Signal lamp 2,4W/130V, 220-240V	KLÖCKNER MOELLER
4022903075387	LS15K11	Contactora, 15KW, LS15K11, 230V AC	AEG
4022903085584	B77S	Overload relay, 20-32A	AEG

At the bottom of the window, there are search filters: "Show approved components only" (unchecked), "Hide obsolete components" (checked), "Count Records:" (set to 1), and search criteria: "EANNUMBER begins with" (40), "TYPE begins with" (empty), and "and only in Blank" (unchecked). There are also "Ok" and "Cancel" buttons.

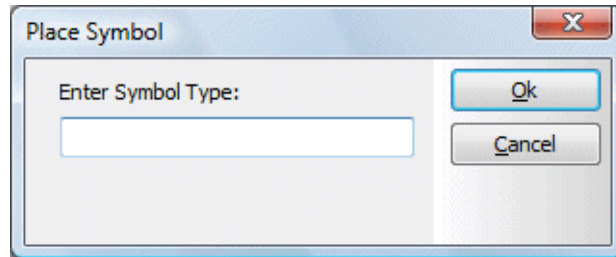
Click on the desired component, and click **OK**. You now get the electrical symbol for the component in the cross hairs — or a pickmenu containing all electrical symbols for the component.

If you type in a complete article number, you get the symbol directly in the cross hairs without entering the database menu.



6. Typing in the Symbol Type directly

When you have activated **Symbols** and choose **Functions => Database => Find Type** — or press the short-cut key [b] — you enter this dialog box:



Here you type in as much you can remember of the symbol type, and click **OK**.

You then enter the database, which displays all components having a symbol type matching what you typed in.

If you typed in *S* you get a list of all components in the database of a type starting with **S** — similar to explained above. Click on the desired component, and click **OK**. You now get the electrical symbol for the component in the cross hairs, or a pickmenu containing all electrical symbols for the component.

If you type in a complete type specification, you get the symbol directly in the cross hairs without entering the database menu.



UPPER AND LOWER CASE CHARACTERS IN THE DATABASE

WHEN YOU TYPE IN THE SYMBOL TYPE, THE PROGRAM DISTINGUISHES BETWEEN UPPERCASE AND LOWERCASE LETTERS WHEN YOU ARE APPLYING A DATABASE FORMAT, WHICH IS ABLE TO MAKE THIS DISTINCTION.

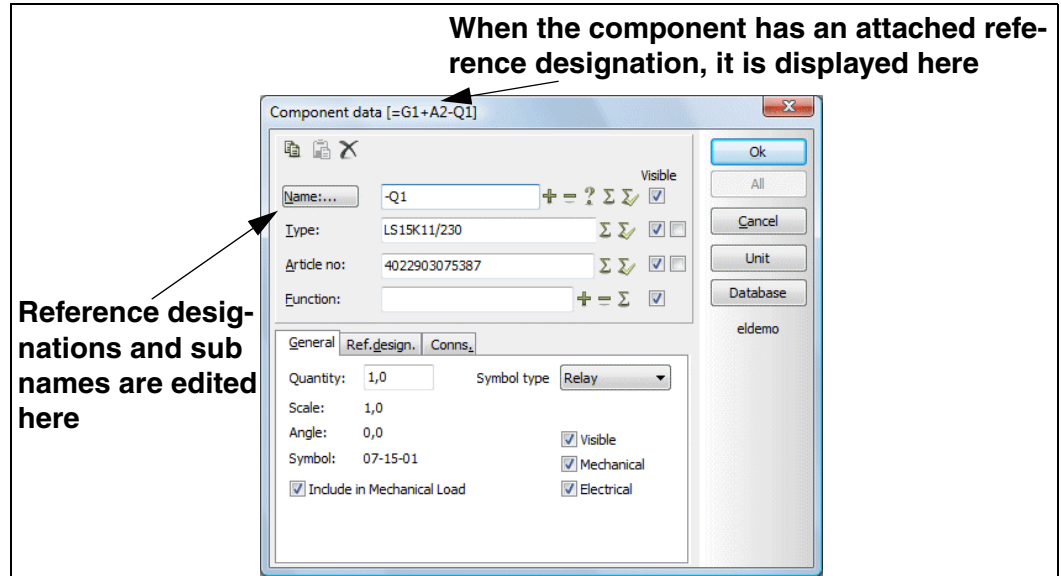
IF FOR INSTANCE YOU ARE USING AN ACCESS DATABASE, THERE WILL BE NO DISTINCTION BETWEEN UPPER AND LOWER CASE CHARACTERS.

THE DBASE FORMAT, ON THE OTHER HAND, MAKES THIS DISTINCTION.

PLACING AND NAMING SYMBOLS

When you have got a symbol in the cross hairs — using one of the ways mentioned in “Fetching symbols” on page 135 — you can place it by clicking where you wish to place it on the drawing.

When you place the symbol, you will automatically be asked about the **Component data** in the following dialog box:



In the dialog box **Component data** above you type in the component name, and other relevant information. — Please refer to “Texts connected to symbols” on page 104 for details about the contents of these text fields.

If you have selected the symbol via the database, all the fields are filled out automatically — except for the symbol/component name.

Click **OK** when you are satisfied with the settings, or click **Cancel** to cancel the changes in the contents of the dialog box.

Press <Esc> to remove the symbol from the cross hairs.



CHANGES CAN BE TRANSFERRED TO ALL SYMBOLS FOR THE COMPONENT
WHEN TEXT PARAMETERS ARE CHANGED IN THE DIALOG BOX COMPONENT DATA, THE MESSAGE *ALL=CHANGE ALL SYMBOLS FOR THE COMPONENT* IS DISPLAYED IN THE UPPER-LEFT CORNER OF THE DIALOG BOX.



IF YOU THEN CLICK THE **ALL** BUTTON IN THE DIALOG BOX, THE CHANGES IN THE FIELDS **NAME, TYPE, ARTICLE NO, FUNCTION** AND THE REFERENCE DESIGNATIONS ARE TRANSFERRED TO **ALL** THE OTHER SYMBOLS FOR THE COMPONENT. AT THE SAME TIME ALL SYMBOLS FOR THE COMPONENT ARE UPDATED FROM THE DATABASE, AS DESCRIBED IN “UPDATE FROM DATABASE WHEN CHANGING SYMBOL ARTICLE NO” ON PAGE 223.

IF YOU CLICK **OK**, DATA IS ONLY CHANGED FOR THE SELECTED SYMBOL.

Automatic naming of symbols



When you activate the **Auto name** button, the symbols you place are assigned the next available symbol name for that type of symbol without entering the **Component data** dialog box.

The button is located in the Symbol toolbar on the upper part of the screen. — See “The Symbol toolbar” on page 171.



AUTO NAMING OF SYMBOLS ON AUTOMATIC PROJECT GENERATION

WHEN GENERATING PROJECTS AUTOMATICALLY, THE SYMBOLS IN THE CREATED PROJECT CAN BE NAMED CONSECUTIVELY AUTOMATICALLY.

— READ MORE ABOUT THIS IN “GENERATING AND UPDATING PROJECTS VIA EXCEL” ON PAGE 614, “MERGING PROJECTS VIA EXCEL” ON PAGE 625 AND “AUTO NAME SYMBOLS DURING GENERATION” ON PAGE 630.






How not to enter the dialog box Component data automatically

If you do not want to enter this dialog box each time you place a symbol, you can disable the function by *deselecting Ask for names* in **Settings => Pointer/Screen**. — See “Pointer/Screen” on page 815.


OPTIONS IN THE DIALOG BOX COMPONENT DATA	
— SPECIFYING THE COMPONENT NAME	(P. 153)
— DISPLAYING/EDITING THE PARTS OF A COMPONENT NAME	(P. 154)
— SPECIFYING TYPE AND ARTICLE NO.....	(P. 154)
— SPECIFYING FUNCTION	(P. 155)
— VISIBLE/INVISIBLE TEXTS	(P. 155)
— THE TAB GENERAL	(P. 155)
— THE TAB REF.DESIGN.	(P. 156)
— THE TAB REFERENCE	(P. 157)
— CHANGING CONNECTION DATA.....	(P. 157)
— THE TAB SYMBOL DATA	(P. 159)
— COPYING COMPONENT DATA	(P. 159)
— COLLECTING INFORMATION FROM THE DATABASE.....	(P. 159)
— ATTACHING UNIT DRAWINGS TO SYMBOLS.....	(P. 159)

Specifying the Component name

In the **Name** field in the dialog box **Component data** you can either type in a name, or use one of the following options:


BUTTON	FUNCTION
	Increase the number in the symbol name by one.
	Decrease the number in the symbol name by one.
	Gives you the next available symbol name for the relevant symbol type.
	Gives you a list, where you can choose between the already used symbol names.
	Gives you a selected list of symbol names for all relevant symbols in the actual situation.

Automatic counting in the Component data dialog box

 If you press down <Ctrl> when you click on + in the dialog box **Component data**, the names of the following symbols of the same type that you place, will be increased by one as you place them in the project.

When this function is activated, you will see a + in front of the **Name** field in the dialog box.

 If you press down <Ctrl> when you click on -, they will automatically be decreased by one. A - will be displayed in front of **Name** when this function is activated.

 If you press <Ctrl> while clicking on ?, subsequently placed symbols will automatically be assigned the next available symbol name. When this function is activated, a ? will be displayed in front of the **Name** field.

Other options for naming symbols

To see how to rename more symbols at the same time, please refer to “Changing symbol names in an Area” on page 180.

Please note that you can also change the symbol texts directly on the diagram page, as described in “Editing the symbol texts directly on a Diagram page” on page 105.

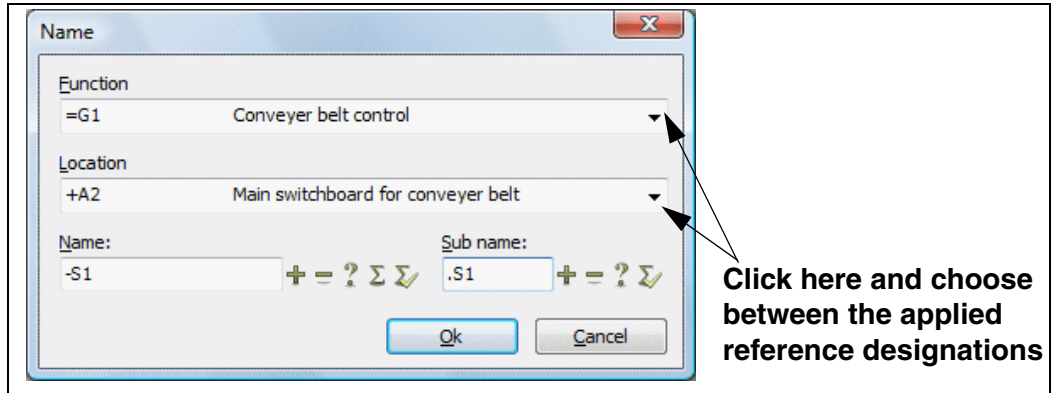
More advanced ways to change texts directly on diagrams are described in “Locking text types” and “Counting on texts” on page 116.

Current path names in symbol names

To include current path names in symbol names automatically, please refer to “Naming symbols due to page and current path number” on page 370.

Button for Displaying/Editing the parts of a Component name

The text for the **Name** field in the dialog box **Component data** is also a button, which you can click on. When clicking this **Name** button, you enter the following dialog box:



In this dialog box you can see and edit the different parts of the symbol name: **Name (-)**, **Function (=)**, **Location (+)** and **Sub name**.

— Please also see “The tab Ref.design.” on page 156 and “Sub names on symbols” on page 160.

Options for the Type and Article no fields

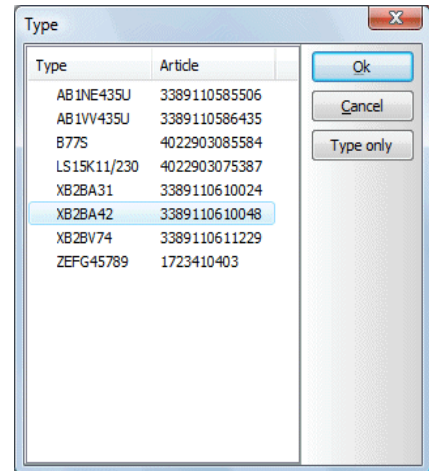


For the fields **Type** and **Article no** in the dialog box **Component data**, you can either type in data, or fetch these via the **Used** and **Used related** buttons.

Here the **Used** button displays all used types and article numbers in the project, while the **Used related** button displays all used types and article numbers in the project for symbols of the same type.

In the appearing dialog box you can either choose type or article number separately, or both at the same time.

— Article number and type data can also be transferred automatically when choosing components via the database, as described in “Fetching symbols from the database” on page 147.



Update from database when changing article number

When changing the **Article no** for a symbol, all symbols for the component are updated, if you subsequently select **Change all symbols for the component** and click **OK** in the **Component data** dialog box. If the new article number exists in the database, the symbols for the component are also updated with component data from the database.

The options for how this is done can be specified in **Settings => Database => Update Components when changing Article No. => Settings**, as described in “Update from Database when changing symbol Article no” on page 223.

— See also “Updating symbols from the database” on page 166.



Options for the Function field

For the **Function** field in the dialog box **Component data**, you can either type in data, or apply the buttons **Used**, **Count up** and **Count down**.

The **Used** button displays the currently applied texts in the project, while the **Count up** / **Count down** buttons counts up/down on the current function text.

Visible texts

By setting a check mark in the **Visible** fields, you determine which information will be displayed in the project. Read more about the texts in this dialog box in “Texts connected to symbols” on page 104.

Please note that the texts will only be displayed in the project if you have not disabled this in **Settings => Text/Symbol defaults** — see “Texts displayed in the project (general setting)” on page 123.

The tab General

On the tab **General** in the dialog box **Component data** you can see information about the symbol, such as the symbol file name, the symbol type, scale, visibility and rotation angle.

Quantity

The field **Quantity** is for instance used for single line diagrams (see “Single line diagrams” on page 397), and specifies how many occurrences of the component the symbol represents.

For instance this can be used if you need five support symbols and only want to place one symbol on the drawing. In the project lists, five support symbols will then be included.

Symbol type

By clicking the down-arrow in the field **Symbol type** in the dialog box **Component data** you can change the symbol type. — Read more about symbol types in “Symbol types” on page 686.

Symbol data in Electrical and Mechanical lists

You can also specify whether the symbol shall be represented in electrical and/or mechanical lists.

When you select **Electrical**, the symbol will be represented on terminals-, cables- and PLC lists and in connections list files, if the symbol contains the type of information you choose to include in the various lists.

If you select **Mechanical**, the symbol will be represented in components- and parts lists.

When you place a symbol, both **Electrical** and **Mechanical** are selected automatically, unless specified otherwise in the symbol definition. — See “Symbol settings: Advanced” on page 688.

Visible

When **Visible** is *not* selected, the symbol is not displayed on print — See “Making objects invisible” on page 62.

Include in Mechanical Load

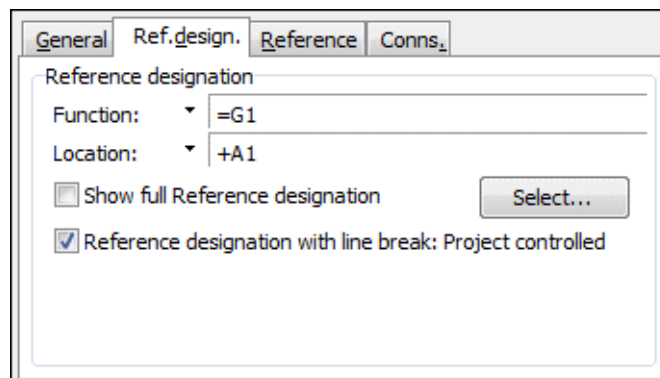
If **Include in Mechanical Load** is selected for the symbol, a mechanical symbol will be placed for the symbol during mechanical load, if a mechanical symbol exists for the component in the database.

Please note that this setting is selected by default, and that the setting covers all symbols for the component.

— Read more about mechanical load in “Mechanical Load from the Database” on page 616.

The tab Ref.design.

When you use reference designations in the active project, the tab **Ref.design.** will also appear in the dialog box **Component data**.

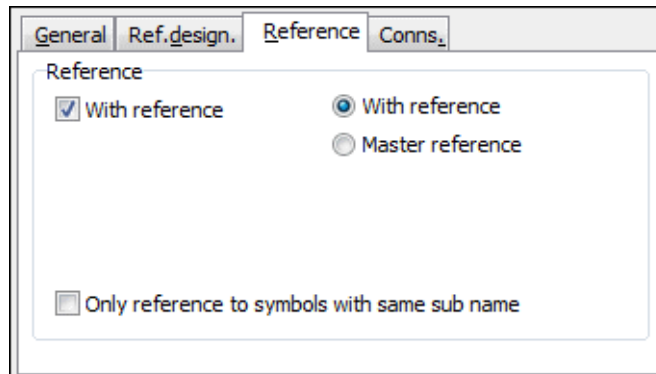


From here you choose the reference designations for the symbol, as described in “Reference designations for symbols” on page 336. — Another way of selecting reference designations for the symbol, is described in “Button for Displaying/Editing the parts of a Component name” on page 154.

— See also “Show full Reference designation” on page 338 and “Individual control of reference designations with line breaks from Component data” on page 338.

The tab Reference

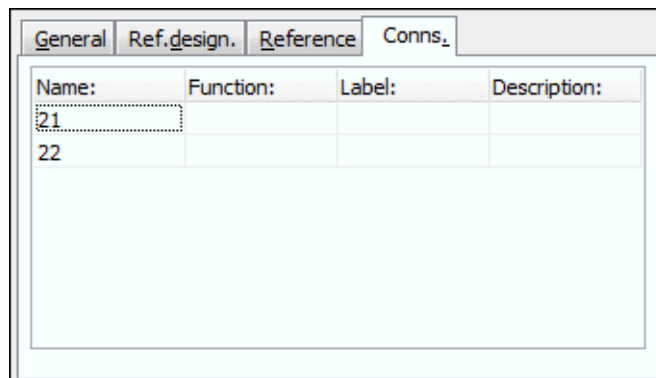
In **Automation** *all* symbols can have attached references. For symbols which does not have attached references by default, you will therefore find an extra **Reference** tab in the dialog box **Component data**.



— Read more about the use of this tab in “References on symbols” on page 357.

Changing Connection data

If you click on the tab **Conns.**, and click in one of the text fields, you can change the connection texts for the symbols connection points.



If the connection names are changed in this dialog box, they are set to be visible automatically.

This can also be done from the dialog box **Connection data**, as described in “Symbol Connection points” on page 182.

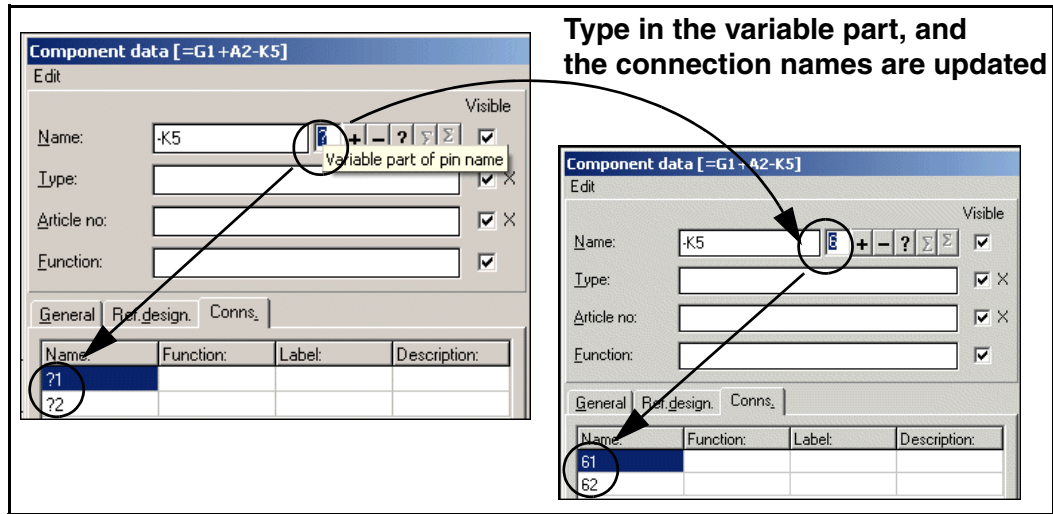
Changing connection Name from the Component data box

From the dialog box **Component data** you can edit the symbol connection names in two ways:

CONNECTION NAMES WITH VARIABLE PART IN THE SYMBOL DEFINITION

If there exists a variable part in the connection name either in the **PINDATA** information from the database or in the symbol definition — see “Connection data with

a variable part in the name” on page 681 —, there will exist an extra text field in the dialog box **Component data**.

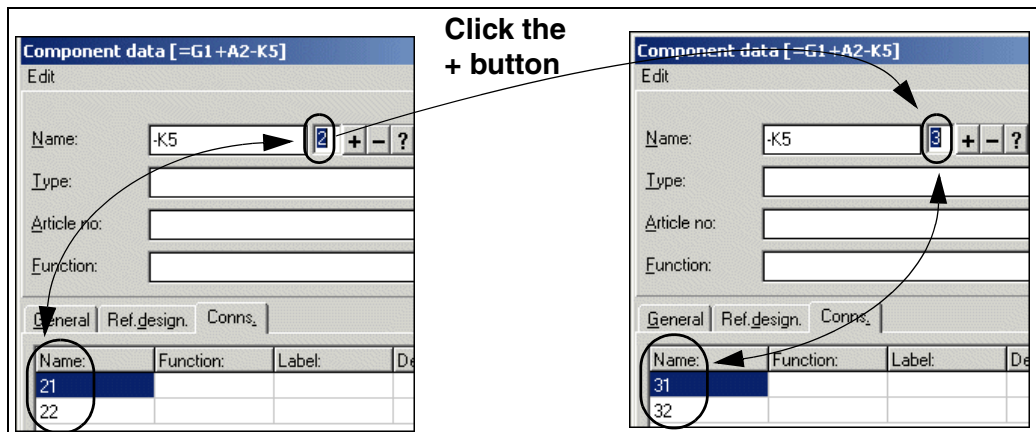


Here you can type in the variable part of the connection name. If there is a question mark, this represents one single character. If there is an asterisk, you can type in all of the connection name. This can for instance be applied for terminal symbols. — See “Wildcards/jokers” on page 540 for further information about the wildcards ? and *.

Please note in the figure above that the connection names ?1 and ?2 are both changed to 61 and 62 respectively, when 6 is written in the text field for the variable part of the connection name.



When you have selected the text field **Variable part of pin name**, the buttons **Count up**, **Count down**, and **Next free** for automatic counting has effect on this text field:



These button functions as described in “Specifying the Component name” on page 153.

— Please refer to “Symbols with variable connection names” on page 589 and “Connection data with a variable part in the name” on page 681.

EDITING CONNECTION NAMES DIRECTLY FROM THE NAME FIELD IN COMPONENT DATA
 You can also change the connection names directly from the **Name** field in the **Component data** dialog box. If you type in the **Name** -K1:13,14, the connection points are named **13** and **14** respectively. The connection names are changed

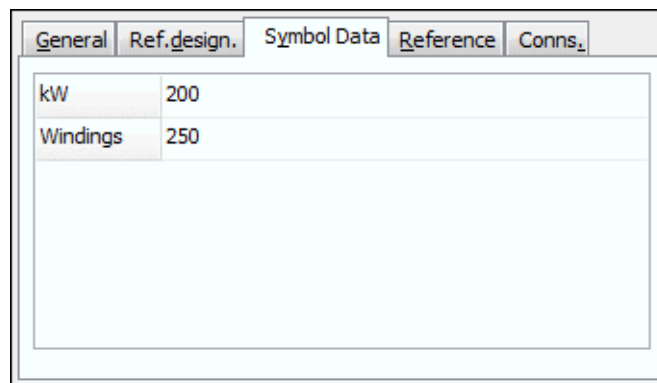
according to the numbering order of the connection points in the definition of the symbol. — For further details about this, please refer to “Place connection points” on page 679.

If the connection names are changed in the **Component data** dialog box, they are automatically set to be **Visible**.

The tab Symbol Data

When a symbol is created containing extra datafields — see “Extra symbol Datafields in symbols” on page 691 — the tab **Symbol Data** — will also appear.

Click on the tab **Symbol Data** and type in the relevant values.



When you place a symbol containing datafields of the type **Project data** or **Page data**, which does not exist in the project, you will be asked whether to include these datafields in the project.

Copying Component data

If you wish to copy component data from one symbol to another, please refer to “Transfer component data for Symbols” on page 55.

Collecting information from the database

Information about the symbol can be collected directly from the database, which you enter by clicking **Database**. Here you find the appropriate component, and click **OK**.

When you have selected the symbols for the component from the database, all text fields, except the **Name** field, are filled out with information from the database.

— Read more about this in “Using the database” on page 561.

Attaching Unit drawings to Symbols

If you connect a unit drawing to the symbol, you do this by clicking on **Unit**. This is elaborated in “Attach a symbol on a diagram to the unit drawing” on page 558.

SUB NAMES ON SYMBOLS

In **Automation** sub names on symbols can be handled intelligently.

What is sub names on symbols?

Sub names on symbols are applied where a component contains functions, which must be identified separately.

This could for instance be the case for a reversing contactor with two coils, where it must be possible to identify which contacts that are controlled by the individual coils. The names of the two coils could here be **-Q1.Q1** and **-Q1.Q2**.

Another example could be a push-button with a build-in lamp, where the symbols could be named **-S1.S** and **-S1.P** respectively.



FUNCTIONALLY UNDERSTOOD AS DIFFERENT COMPONENTS

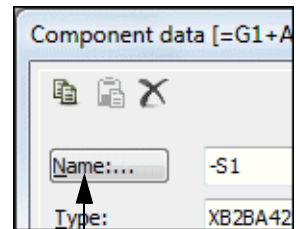
SYMBOLS FOR THE SAME COMPONENT, WHICH HAS DIFFERENT SUB NAMES, ARE FUNCTIONALLY UNDERSTOOD AS DIFFERENT COMPONENTS IN THE PROJECT, WHILE THEY IN PARTS LISTS ARE UNDERSTOOD AS PARTS OF THE SAME COMPONENT.

IF A COMPONENT THEREFORE CONTAINS TWO COILS WITH THE NAMES **-Q1.Q1** AND **-Q1.Q2**, THESE ARE EACH ASSIGNED THEIR OWN REFERENCE CROSS IN THE PROJECT, BUT ARE REPRESENTED AS ONE COMPONENT IN THE PARTS LIST.

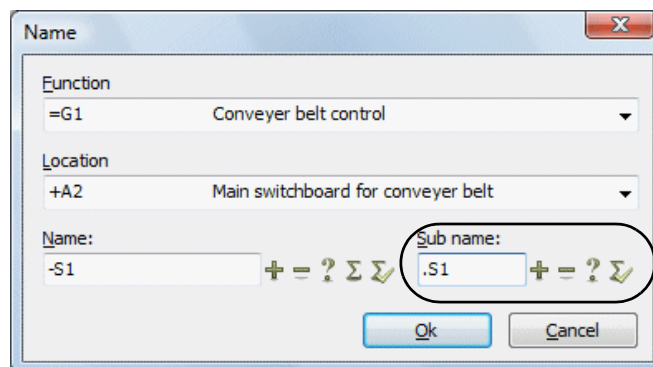
Specifying sub names on symbols

To specify a sub name for a symbol, do this:

- 1) Enter the dialog box **Component data** for the symbol — e.g. by double-clicking on the symbol.
- 2) In the dialog box **Component data**: Click on **Name**:



2) Click here



- 3) In the dialog box **Name**: Click in the field **Sub name**, and type in a sub name — or use the buttons **Count up**, **Count down**, **Next free** or **Used related** to specify the sub name.

4) Click **OK**, and click **OK** in the dialog box **Component data**.



SUB NAMES CAN ONLY BE SPECIFIED IN THE DIALOG BOX NAME

SUB NAMES CAN *ONLY* BE SPECIFIED IN THE DIALOG BOX **NAME**.

THEREFORE, IF YOU WRITE *-Q1.1* IN THE FIELD **NAME** IN THE DIALOG BOX **COMPONENT DATA**, THIS IS UNDERSTOOD AS THE COMPONENT NAME, AND *NOT* AS A COMPONENT NAME FOLLOWED BY A SUB NAME.

BECAUSE OF THIS, THE SUB NAME IS NOT DISPLAYED IN THE **NAME** FIELD IN THE DIALOG BOX **COMPONENT DATA**.

Sub names can be specified in the database

Sub names can also be specified via the database, as described in “Specifying symbol sub names from the database” on page 595.

Sub names included in lists

Sub names on components can also be included in all electrical lists. For this purpose, all datafields containing a component name, have included the option **With sub name** in the dialog box **DataField**. — See “Datafields” on page 107.

SYMBOLS WITH STATES

Symbols can be placed with different *states* on the project pages, so that the symbols change appearance when their state number is changed.

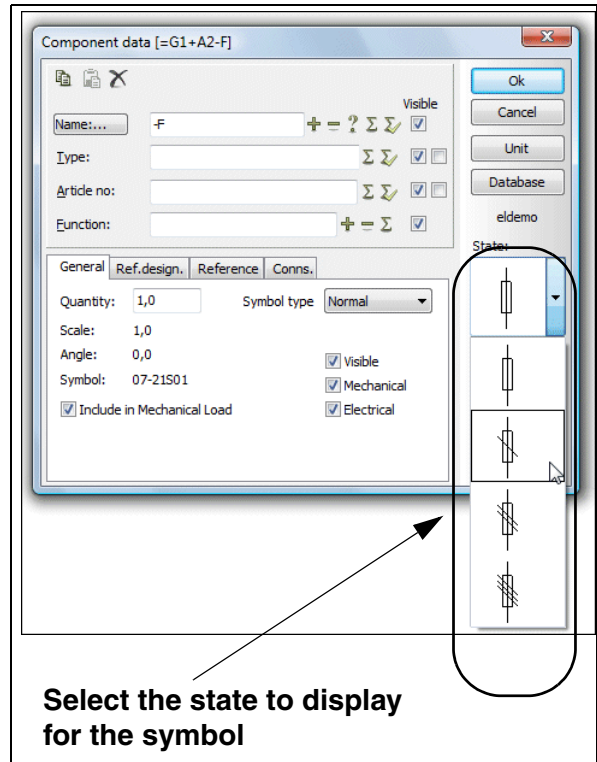
This is only possible for symbols, which have been designed with different states. — See how to create symbols with states in “Creating symbols with states” on page 692.

Select a state for the symbol

When you click to place a symbol with a special state in the project, you must select the state in the **Component data** dialog box — see the figure to the right.

When you subsequently need to change the symbol state, enter the **Component data** dialog for the symbol, select a new state, and click **OK**.

— Please also see “Symbols with states for single line diagrams” on page 398 and “Placing symbols with selected states in the pickmenu” on page 176.



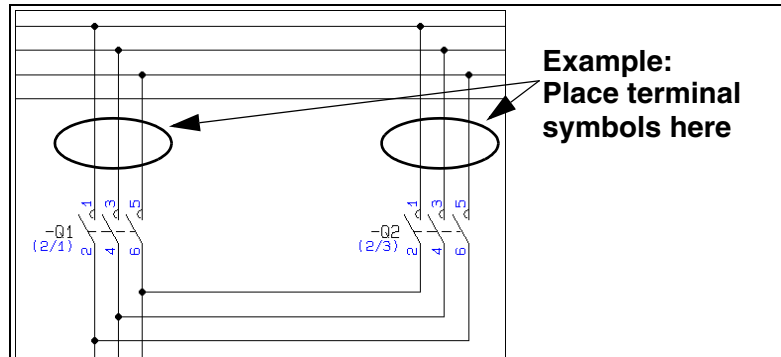
SHIFT STATE WITH <SHIFT + SPACEBAR>

WHEN YOU HAVE SELECTED A SYMBOL CONTAINING STATES, YOU CAN SHIFT BETWEEN THE STATES BY PRESSING <SHIFT> WHILE PRESSING THE <SPACEBAR>. THIS IS ALSO THE CASE WHEN YOU HAVE THE SYMBOL IN THE CROSS HAIRS.

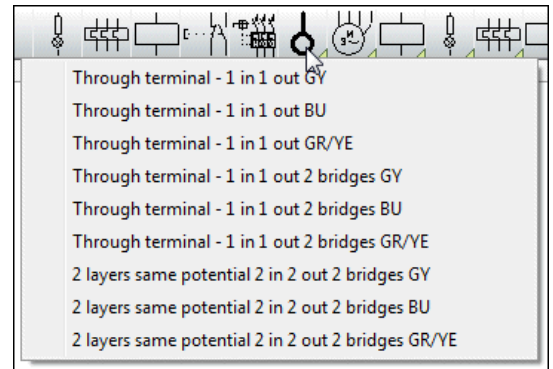
MULTIPLACING SYMBOLS

In **Automation** it is possible to place multiple symbols of the same kind in one single operation — for instance when placing terminals.

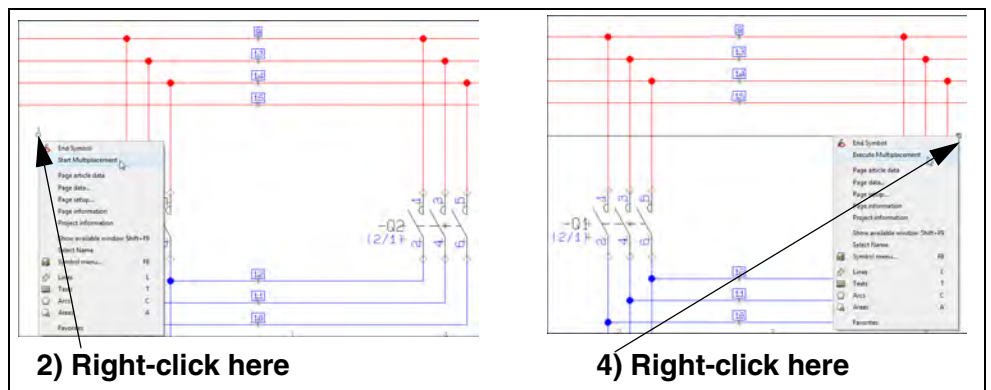
Do this:



- 1) Select a symbol, e.g. a terminal from the pickmenu.
- 2) Right-click on the page, and select **Start Multiplacement**. (See the figure above).
- 3) A "line" now emerges from where you right-clicked, ending in the cross hairs. Move the cursor so that the line crosses all of the lines that you wish to place the symbols on. — See also “Skipping line segments when multiplacing symbols” on page 164.

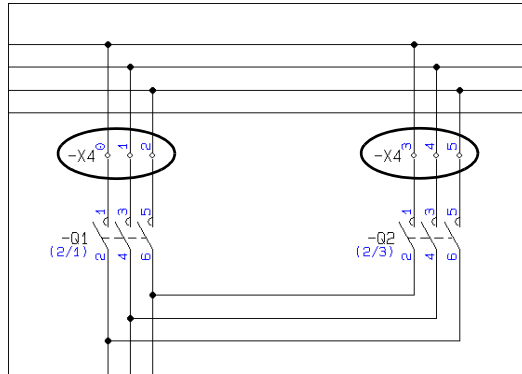


- 4) Right-click on the page, and select **Execute Multiplacement**.



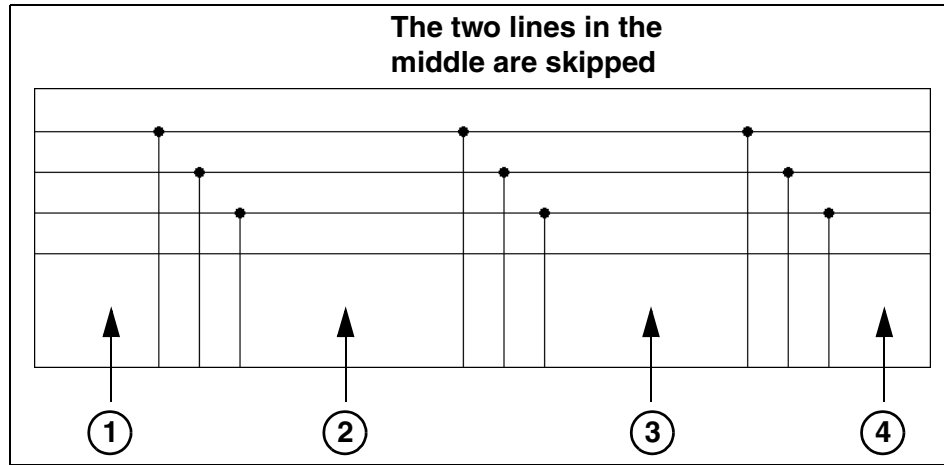
- 5) In the dialog box **Component data**: Specify symbol names and click **OK**. — See also “Placing and Naming symbols” on page 151 for how to name connection names for terminals directly from the **Symbol** menu.

6) The terminals are now placed with assigned connection names:

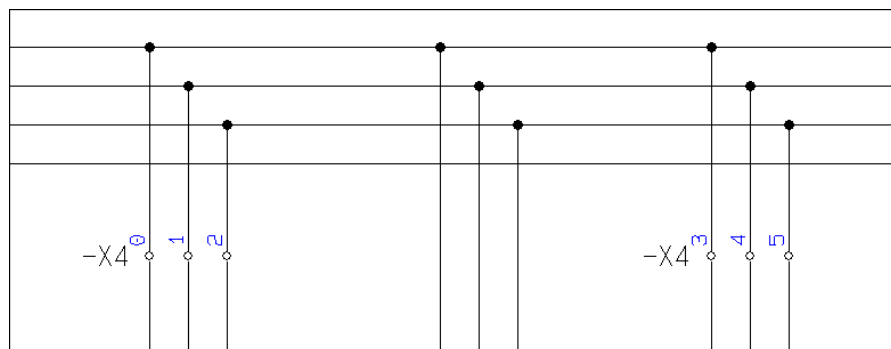


Skipping line segments when multiplying symbols

To skip one or more line segments when multiplying symbols, do this:



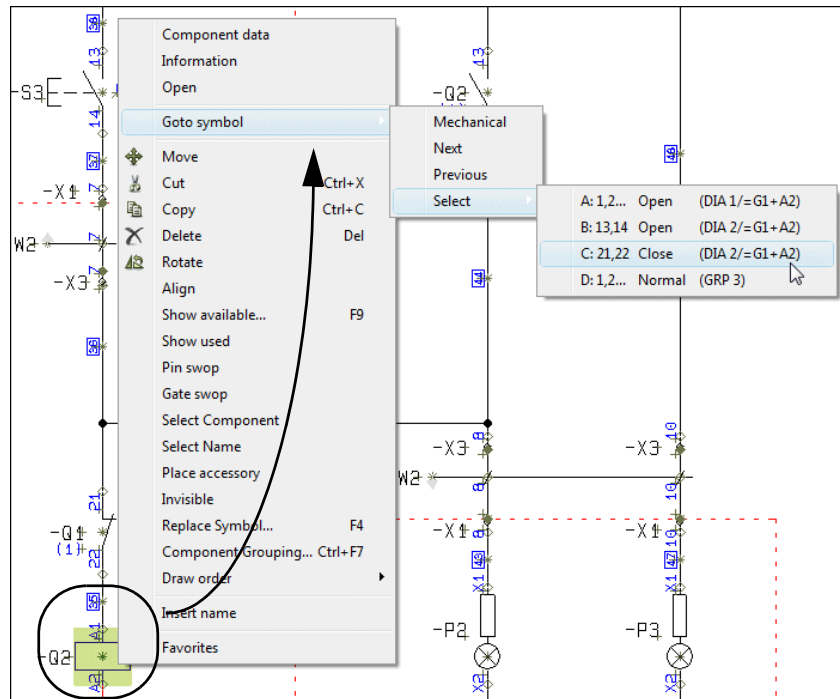
- 1) Select a symbol, e.g. a terminal, from the pickmenu. Right-click, and select **Start Multiplacement** — see the figure above.
- 2) Click *before* the lines you wish to skip.
- 3) Press <Ctrl> and click *after* the lines you wish to skip.
- 4) Right-click *after* the lines you wish to place symbols on, and select **Execute multiplacement**.



— See also “Multiplying symbols” on page 163.

NAVIGATION BETWEEN SYMBOLS FOR THE SAME COMPONENT

When you have selected a symbol and choose **View => Goto symbol** — or if you right-click on a symbol and choose **Goto symbol** — you can select which of the other symbols for the component to jump to in the project:



In the **Goto symbol** function, you have the following options:

- **Mechanical:** Hereby you jump to the mechanical drawing, when such a drawing exist. The mechanical symbol for the component is selected, and the cross hairs points out the symbol.
- **Diagram:** The first of the diagram symbols for the component is selected, and the cross hairs points out the symbol.
- **Next:** The next diagram symbol for the component is selected, and the cross hairs points out the symbol.
- **Previous:** The previous diagram symbol for the component is selected, and the cross hairs points out the symbol.
- **Select:** Here a menu containing all symbols for the component is displayed. For each symbol you see the names of the first two connection points, as well as the symbol and page type, the page number and reference designation (if any). When you select a symbol in the menu, the symbol is selected in the project, and the cross hairs points out the symbol.



JUMPING BETWEEN SYMBOLS BY CLICKING ON REFERENCES ON THE PAGE

PLEASE ALSO SEE “AUTOMATICALLY UPDATED REFERENCES” ON PAGE 355, IN PARTICULAR “AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK” ON PAGE 355. HERE YOU SEE HOW TO JUMP BACK AND FORTH BETWEEN THE SYMBOLS FOR THE COMPONENT, BY DOUBLE-CLICKING ON REFERENCES PLACED NEXT TO THE SYMBOLS ON THE PROJECT PAGES.

UPDATING SYMBOLS FROM THE DATABASE

To make sure that e.g. the type texts and/or the connection names for the symbols for a component — or for all symbols on a page or in the project — matches the given information for the specified EAN numbers in the database, do the following:

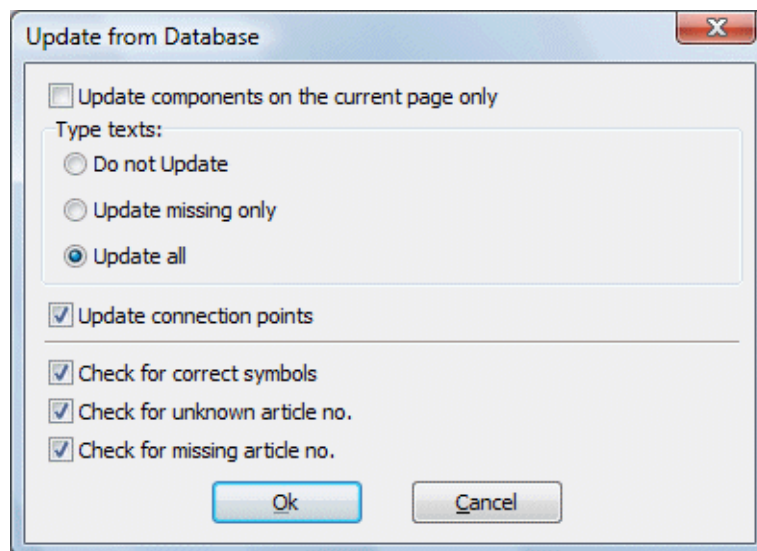


UPDATING SYMBOLS WHEN CHANGING ARTICLE NUMBERS

WHEN CHANGING THE ARTICLE NUMBER FOR A SYMBOL, ALL SYMBOLS FOR THE COMPONENT CAN ALSO BE UPDATED FROM THE DATABASE.

— SEE “UPDATE FROM DATABASE WHEN CHANGING SYMBOL ARTICLE NO” ON PAGE 223 AND “UPDATE FROM DATABASE WHEN CHANGING ARTICLE NUMBER” ON PAGE 154.

- 1) Select the symbol(s) you wish to update from the database. If no symbols are selected, all symbols in the project — or on the selected page — can be updated.
- 2) Choose **Functions => Database => Update from Database**.
- 3) You now enter the dialog box **Update from Database**:



Here you have the following options:

OPTION	DESCRIPTION
UPDATE ONLY COMPONENTS ON THE CURRENT PAGE	THIS OPTION IS ONLY DISPLAYED WHEN YOU HAVE NOT SELECTED ANY SYMBOLS ON THE PROJECT PAGE. WHEN YOU SELECT THIS OPTION, ONLY THE SYMBOLS ON THE PAGE ARE UPDATED (AS WELL AS ALL RELATED COMPONENTS IN THE PROJECT). WHEN YOU DO <i>NOT</i> SELECT THIS OPTION, ALL COMPONENTS IN THE PROJECT ARE UPDATED.
TYPE TEXTS	YOU CAN HERE CHOOSE EITHER Do Not Update , Update Missing Only OR Update All .
FUNCTION TEXTS	IF YOU IN THE DATABASE SETTINGS HAVE SPECIFIED THAT THE PROGRAM SHALL TRANSFER CONTENTS TO THE FUNCTION FIELD FOR SYMBOLS, YOU WILL ALSO GET THE OPTIONS Do Not Update , Update Missing Only AND Update All FOR FUNCTION TEXTS. (THIS IS SPECIFIED IN SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA IN THE FUNCTION FIELD — SEE “TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES” ON PAGE 600).
UPDATE CONNECTION POINTS	UPDATES ALL CONNECTION POINTS, SO THAT THEY ARE ASSIGNED THE CONNECTION POINT NAMES DESCRIBED IN THE DATABASE.
CHECK FOR CORRECT SYMBOLS	ISSUES A WARNING FOR EACH PLACE ANOTHER SYMBOL HAS BEEN USED, THAN THE ONE SPECIFIED IN THE DATABASE FOR THE ARTICLE NUMBER, AND FOR EACH PLACE THERE E.G. HAS BEEN USED A BREAK CONTACT TOO MUCH, COMPARED TO WHICH FUNCTION THE COMPONENT CONTAINS ACCORDING TO THE DATABASE. FOR SECURITY REASONS, YOU SUBSEQUENTLY HAVE TO MAKE THE NECESSARY CHANGES MANUALLY.
CHECK FOR UNKNOWN ARTICLE NO.	ISSUES A WARNING FOR EACH PLACE ARTICLE NUMBERS HAVE BEEN USED, WHICH DOES NOT EXIST IN THE DATABASE. FOR SECURITY REASONS, YOU SUBSEQUENTLY HAVE TO MAKE THE NECESSARY CHANGES MANUALLY.
CHECK FOR MISSING ARTICLE NO.	ISSUES A WARNING FOR EACH PLACE NO ARTICLE NUMBERS HAVE BEEN SPECIFIED. FOR SECURITY REASONS, YOU SUBSEQUENTLY HAVE TO MAKE THE NECESSARY CHANGES MANUALLY.

Select the desired settings, and click **OK**.

- 4) The type texts for the symbols and/or connection names are now updated, and you get a status for the update. Here you can click on each individual warning in the window, and the program points out the symbol in the project. The window remains on the screen until you click the **Close** button in the upper-right corner.

MOVING AND DELETING PLACED SYMBOLS

In the following it is described how to move and delete symbols in **Automation**. You can also *cut* and *paste* symbols, as described in “Cutting and Pasting areas” on page 239.

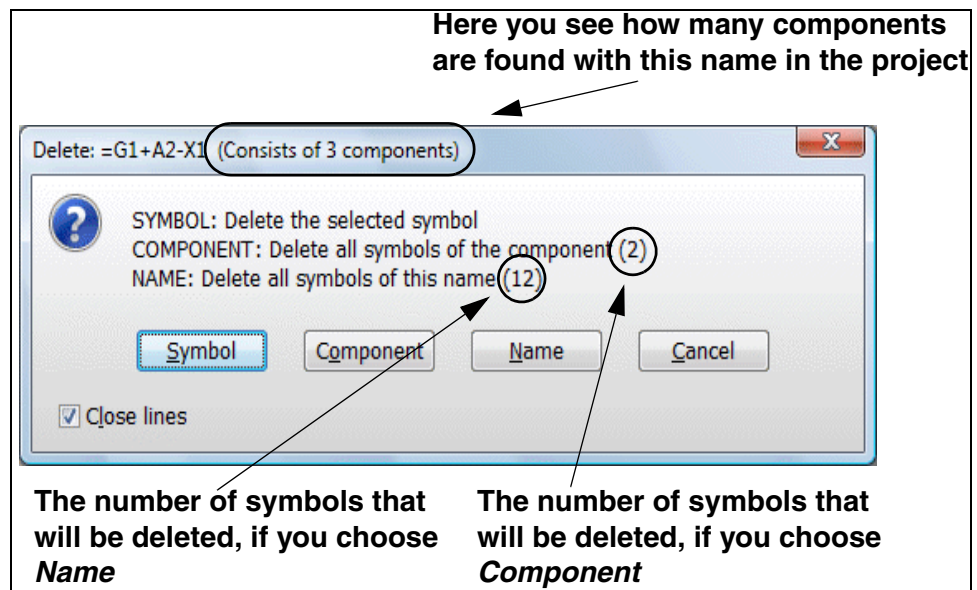
Deleting a placed symbol

You can delete a placed symbol in two different ways:

a) Deleting a symbol and Redrawing the lines



- 1) Click the **Symbols** button, and press <Esc> to deselect the Pencil.
- 2) Click the **Delete** button, and the dialog box **Delete** is displayed:



- 3) You now have the following options:
 - Click on **Symbol**: Only the selected symbol is deleted.
 - Click on **Component**: All symbols for the selected component is deleted (in the figure above, you can see that the physical terminal, from which the symbol is selected, consists of two symbols).
 - Click on **Name**: All symbols with this name are deleted (the terminal row in the figure above consists of 2 physical components, represented by 12 symbols in the project).
 - Click **Cancel**: No action is taken.
 - If you select **Close lines**, the lines are closed for all deleted symbols — whenever this is possible. If you do not select **Close lines**, the lines are parked.

The symbol is now deleted, and the line which the symbol was placed on, is now closed — if this is possible.

b) Deleting a symbol and Parking the lines

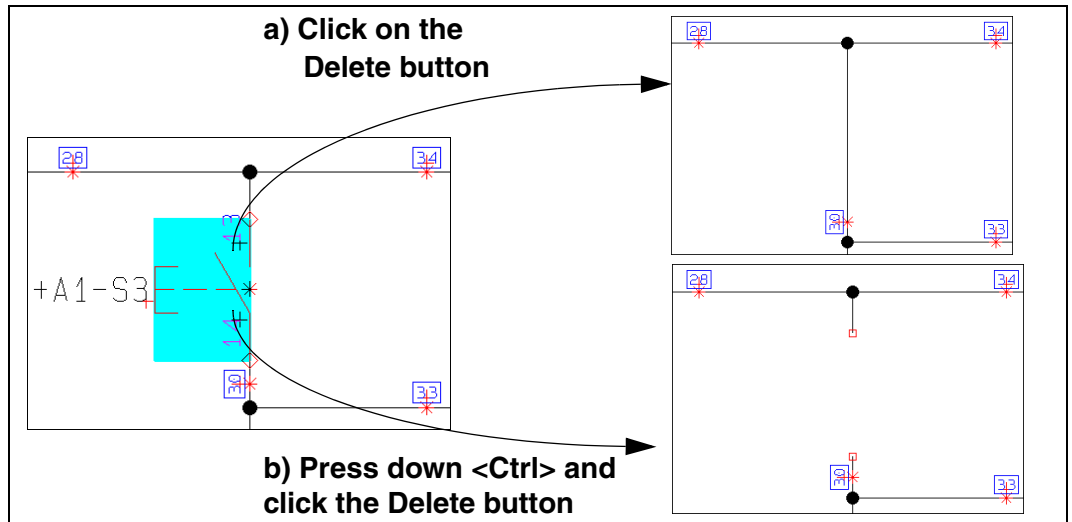


- 1) Click on the **Symbols** button —to start working with symbols.



- 2) Press down <Ctrl> and click the **Delete** button: You are now asked whether to delete all symbols with the current symbol name: Answer **Yes** or **No**.

The symbol is now deleted, and the lines which the symbol was connected to, are parked.



Deleting more than one symbol

You can also delete more than one symbol at the same time, by selecting the symbols at the same time — as described in “Changing symbol names in an Area” on page 180 — and then delete them as described above.

Moving a placed symbol

You can move a placed symbol in two different ways:

a) Moving a symbol with Connected lines



- 1) Click on the **Symbols** button — to start working with symbols.
- 2) Click on the symbol, press down the mouse button, and *drag* the symbol to a new location.

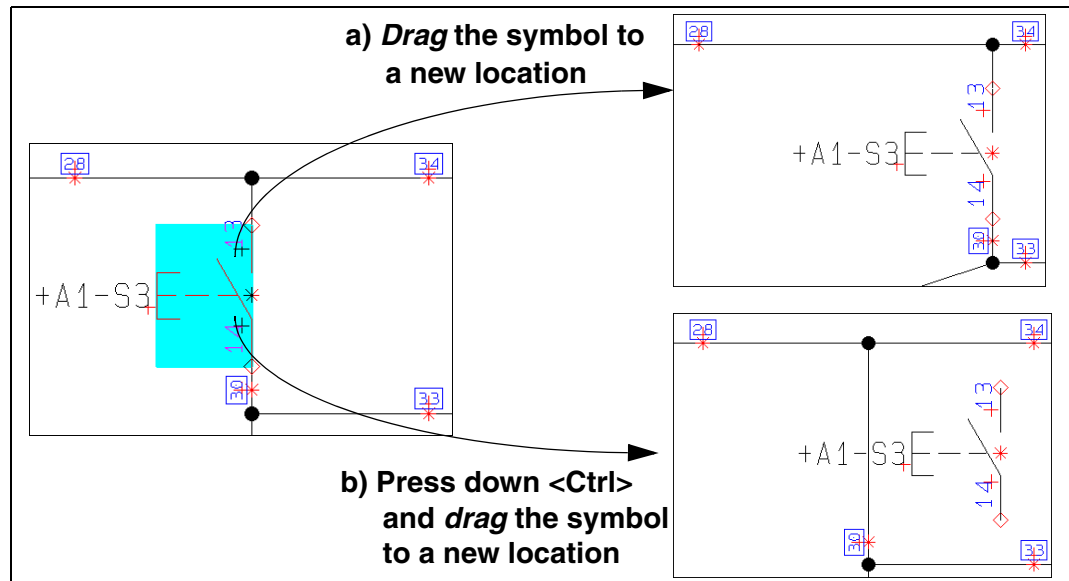
The symbol is now moved, and is still connected to the same lines as before.

b) Moving a symbol and Parking the connected lines



- 1) Click on the **Symbols** button — to start working with symbols.
- 2) Press down the <Ctrl> key, and *drag* the symbol to a new location.

The symbol is now moved, and the lines which it was previously connected to, now becomes connected.



Moving more symbols at the same time

You can also move more symbols at the same time by selecting the relevant symbols — as described in “Changing symbol names in an Area” on page 180 — and then move them as described above.

Moving symbols with overlapping connection points

When two symbols are placed so that their connection points are connected directly, and one of the symbols are moved, a connecting line is automatically drawn between the two previously connected connection points.

THE SYMBOL TOOLBAR

When you have selected a symbol, you get information about the symbol in the symbol toolbar:



T: The file name of the symbol you last had in the cross hairs

N: The name of the currently selected symbol

S: The scale of the symbol

Automatic naming of symbols

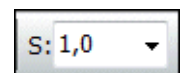


When the **Auto Name** button is activated, symbols are automatically assigned the next available symbol name when placed in the project.

The automatic naming function can also be started/stopped by choosing **Functions => Auto Name**.

Scaling symbols

If you want to scale one or more symbols up or down, you can use Symbol scale. You can scale a symbol by picking a scale factor from the drop down list and press <Enter>, or type in a value and press <Enter>. This can be done either when you have got the symbol in the cross hairs, or by selecting the symbol(s), typing in a value and then click the



Transfer properties button. Symbol scale is rarely used in electrical diagrams (not standard).

Read more about scaling symbols in “Symbol settings: Advanced” on page 688.

Mirroring symbols



Select a symbol, click the **Mirror symbol vertically** button, and the symbol will be mirrored vertically. Or click the **Mirror symbol horizontally** button, and the symbol will be mirrored horizontally.

Rotating symbols



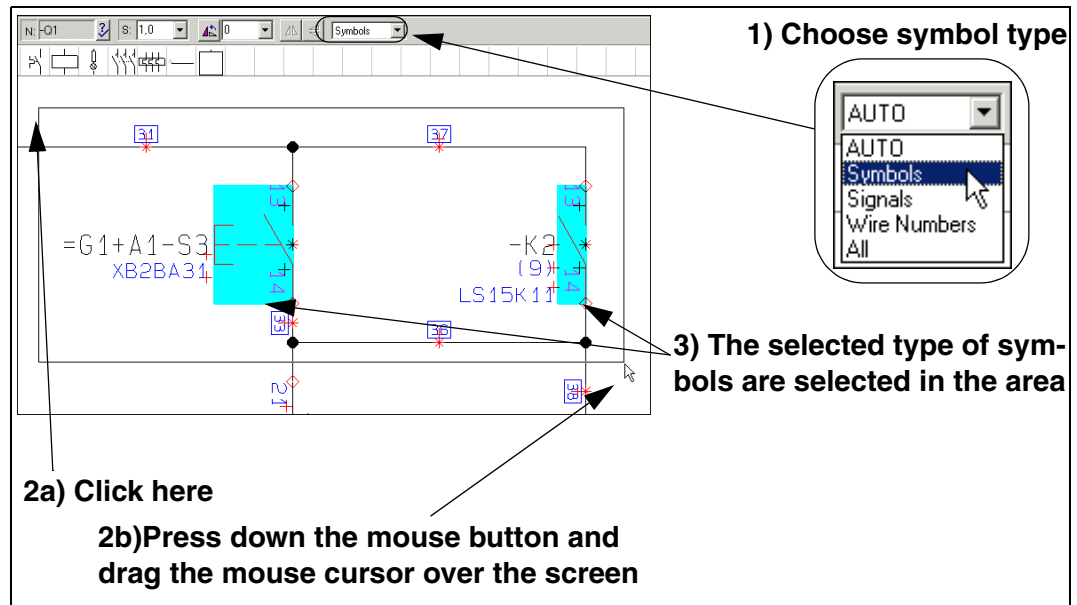
By clicking the Rotate symbol button, you can rotate the selected symbol — see “Rotating a selected object” on page 52.

Choose symbol type in an area on the screen

When you are working with symbols, and mark up an area on the screen using the mouse, you can specify which of the symbols in the area to select. Do the following:

- 1) Choose which type of symbols to select: **AUTO** (= the most common symbols in the area), **Symbols** (all symbols, which are *not* wire numbers or signals), **Signals**, **Wire numbers** or **All** (all symbols of any type in the area).
- 2) Select the area using the mouse, and release the mouse button.

The selected symbols in the area are now selected.



Selecting symbols using Select all or by selecting in areas

When you choose **Edit => Select All => Everything on page** (short-cut key [Ctrl+a]), or when you select symbols in an area, all symbols in the area becomes selected, no matter the type.

ADAPTING THE PICKMENU

In the pickmenu you can place the symbols you use the most for easy access. Below the following aspects are elaborated,

— See how to fetch symbols — and other drawing objects — from the pickmenu in “Fetching symbols from the Pickmenu” on page 136.

The pickmenu can also contain line types, text types and arcs. — Read more about this in “Lines in the Pickmenu” on page 71, “Texts in the Pickmenu” on page 122 and “Circles / Arcs in the Pickmenu” on page 97.

THIS CHAPTER DESCRIBES	
— PLACING SYMBOLS IN THE PICKMENU	(P. 173)
— PLACING SYMBOLS IN THE PICKMENU WITH COMPONENT DATA	(P. 173)
— INSRETING A SYMBOL/DRAWING OBJECT IN ALL PICKMENUS	(P. 173)
— CREATING A NEW MENU IN THE PICKMENU	(P. 175)
— SPECIFYING COMPONENT DATA IN THE PICKMENU	(P. 175)
— PLACING SYMBOLS IN THE PICKMENU VIA THE DATABASE.....	(P. 176)
— PLACING SYMBOLS WITH SELECTED STATES IN THE PICKMENU	(P. 176)
— RIGHT-CLICKING IN THE PICKMENU	(P. 178)

Placing symbols in the pickmenu

To place a symbol directly in an empty cell in the pickmenu, you have to have a symbol in the cross hairs, and then click inside an empty cell in the pickmenu. You can fetch the symbol from either the **Symbol menu** or from a drawing.

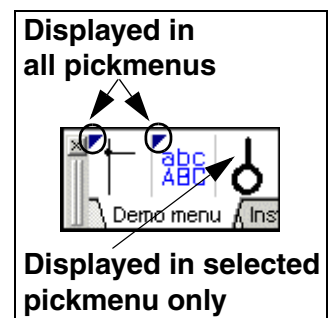
Or, you can right-click inside an empty cell in the pickmenu and choose **Place symbol**, which brings you into the **Symbol menu**, where you click your way to the symbol you want to place in the pickmenu, and click **OK**.

Include component data when inserting a symbol in the pickmenu

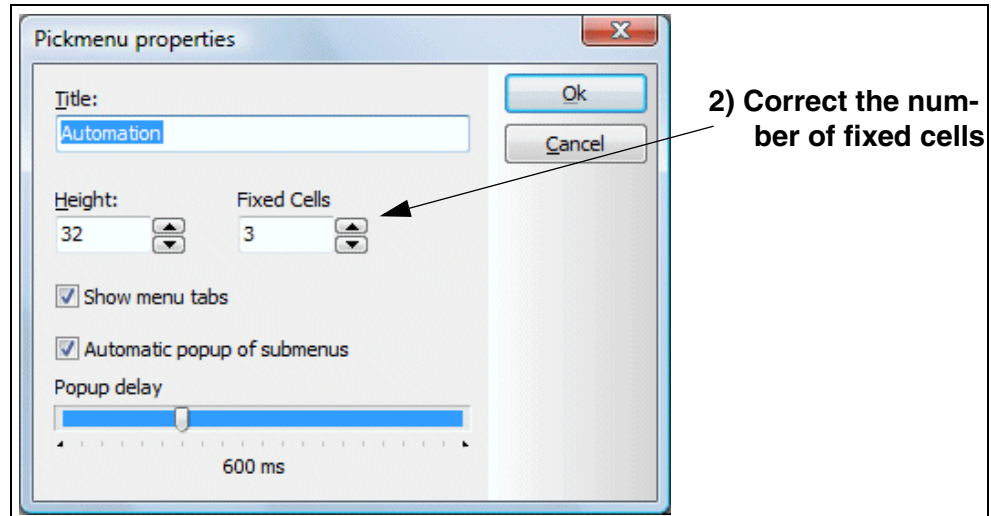
When you hold a symbol in the cross hairs, and press down <Ctrl> while placing the symbol in the pickmenu, the component data is also transferred to the symbol in the pickmenu.

Inserting a Symbol / Drawing object in all Pickmenus

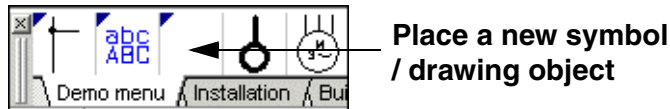
To insert one or more drawing object(s) to be displayed in all pickmenus, do the following:



- 1) Right-click on the pickmenu, select **Properties**, and you enter the dialog box **Pickmenu properties**:



- 2) Set the number of fixed cells, click **OK**, and the new cell(s) are displayed:



- 3) Place a new symbol / drawing object in the cell, as described in “Placing symbols in the pickmenu” on page 173 or “Include component data when inserting a symbol in the pickmenu” on page 173.



- 4) The symbol is now represented in all pickmenus.

Removing fixed cells from the pickmenus

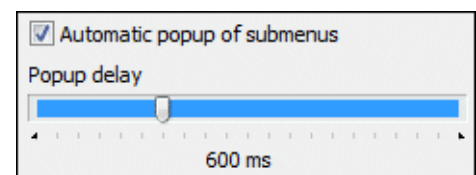
To remove a number of symbols from being displayed in all pickmenus, right-click on the pickmenu, select **Properties**, and reduce the number of **Fixed Cells** to desired number of cells. Please note that the right-most fixed cells are removed first.

Automatic popup of submenus in the pickmenu

You can specify whether to display a popup menu for symbols with states (see “Symbols with states” on page 162) or for symbols with multiple sets of article data attached (see “Symbols with multiple sets of data in the Pickmenu” on page 137), when you let the mouse rest over the symbol in the pickmenu.

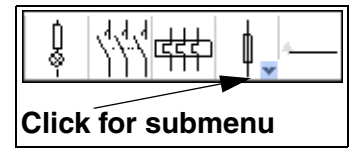
To do this, right-click in the pickmenu, and choose **Properties**. You then enter the dialog box **Pickmenu properties**, where you select **Automatic popup of submenus**.

Under **Popup delay** you can specify whether the popup menu appears instantly, when the mouse cursor rests over a symbol with states / sets of article data, or



whether to insert a delay before the popup menu appears. To insert a delay has the advantage that submenus are not displayed when the mouse cursor just passes the pickmenu, in order to make selections in other of the program menus or tool-bars.

When **Automatic popup of submenus** is not activated, small buttons are displayed in the pickmenu in the relevant cells. When you click this button, the submenu is displayed.



Creating new menus in the Pickmenu

To create a new menu in the pickmenu, do the following:

- 1) Right-click on a cell in the pickmenu (and *not* one of the tabs), and select **Create new Menu**.
- 2) The menu is now created: Right-click on a cell in the pickmenu (and *not* one of the tabs), and select **Properties**.
- 3) In the dialog box **Pickmenu properties**: Name the menu, and click **OK**.

Deleting a menu

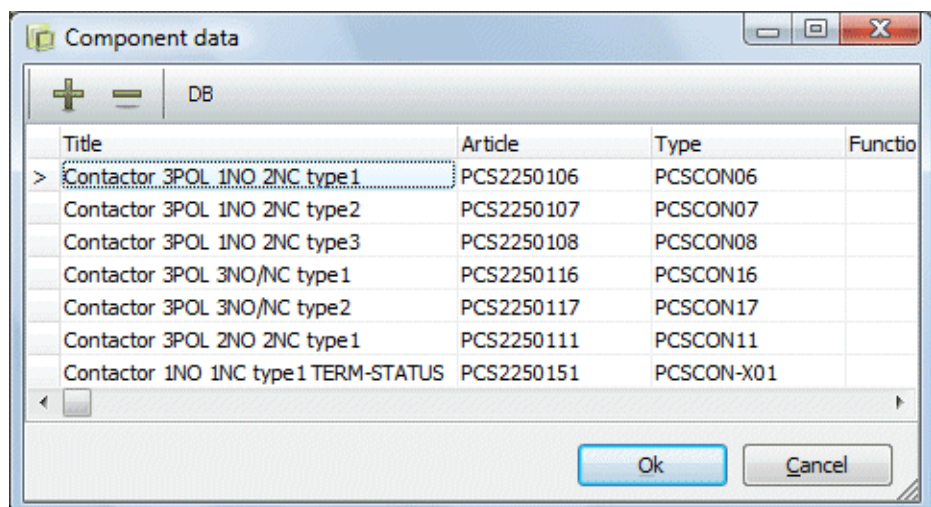
To delete a menu, do the following:

- 1) Select the menu by clicking on the tab for the menu at the bottom of the pickmenu.
- 2) Right-click on a cell in the pickmenu (and *not* on one of the tabs), select **Delete Menu**, and answer **OK** to delete the menu.

Adding component data to symbols in the pickmenu

You can attach multiple sets of component data to the symbols in the pickmenu.

To do this, right-click on a symbol in the pickmenu and choose **Component Article Data**. You then enter the **Component data** dialog box:



Here you have the following options:

Create a new line with component data



Click the **Plus** button. In the new empty line you can either type in component data by clicking in the relevant fields, or click the **DB** button to enter the **Database menu** to select a component. — You can read more about the **Database menu** in “About the Database menu” on page 565.



When you have filled out a component line in the dialog box, and then click the **DB** button, you are asked whether to overwrite the selected line, when returning from the database. You also get the option of choosing **Add**, to create a new line with component data.

Deleting a line with component data



Click on the line you wish to delete, and then click the **Minus** button.

Multiple symbols of the same kind in the pickmenu

Placing multiple symbols of the same kind in the pickmenu, is also an option. When you place a symbol which exists in the pickmenu already, you are informed about this.

When you let the mouse cursor rest over one of the symbols in the pickmenu, the symbol title is displayed both below the symbol (as a *hint*), and at the bottom of the screen.

Placing symbols in the pickmenu using the database



It is also an option to place a symbol in the pickmenu using the database. When you right-click on an empty cell in the pickmenu, and choose **Place symbol**, you enter the **Symbol menu**, where you click on the **Database** button in the dialog box.

You now enter the database, where you choose the desired component, and click **OK**. The symbol and its attached component data is hereby being transferred to the pickmenu.

If the component contains more than one symbol, the first of its symbols is placed in the pickmenu. — See “The function Show available” on page 224.

Placing symbols with selected states in the pickmenu

You can also place symbols, with a specified state selected, in the symbol toolbar. To transfer the selected state to the symbol toolbar, press down <Ctrl> when clicking the symbol in the symbol toolbar.

— Read more about symbols with states in “Symbols with states” on page 162 and “Creating symbols with states” on page 692.

Placing sub drawings in the Pickmenu

In order to place sub drawings in the Pickmenu, this is required:

- 1) Use a database for **Automation**, which contains a field for sub drawings.
- 2) A unique article number must be created for each sub drawing, that you wish to place in the Pickmenu.
- 3) The article number for each individual sub drawing in the database must contain a link to the sub drawing, that you wish to place in the Pickmenu. — See how to do this in “URL links” on page 610, especially “Example for sub-drawings” on page 611.

In order to place a sub drawing in the Pickmenu, do this:

- 1) Place a symbol in the Pickmenu.
- 2) Right-click on the symbol in the Pickmenu, and choose **Component Article Data**.
- 3) In the dialog box **Component data**: Click the **Database** button.
- 4) In the **Database menu**: Select the article number for the sub drawing and click **OK**.
- 5) In the dialog box **Component data**: Click **OK**.



The sub drawing can now be placed in projects via the Pickmenu, as described in “Sub drawings in the Pickmenu” on page 139.



THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS

YOU CAN READ MORE ABOUT SUB DRAWINGS IN THE CHAPTER “SUB DRAWINGS” ON PAGE 244. THERE YOU CAN ALSO READ ABOUT THE RECOMMENDED PROCEDURE FOR PLACING SUB DRAWINGS USING THE **SUBDRAWINGS** TAB IN THE **EXPLORER** WINDOW.

Right-clicking in the pickmenu

To the left in the pickmenu title bar the name of the menu is displayed. To the right there are two arrows, with which you can scroll back and forth between the various pickmenus.

If the active pickmenu contains more symbols than can be shown at a time, there will also be a scroll bar at the bottom of the pickmenu.

Insert new cell

Right-click on a filled out cell in the pickmenu, choose **Insert Cell**, and an empty cell is inserted where you right-clicked.

Delete cell

Right-click on a cell and choose **Delete Cell**. The cell is now deleted. This goes for empty cells as well cells containing symbols.

Create new menu

Right-click on the pickmenu, and choose **Create new Menu**.

Give a menu a title

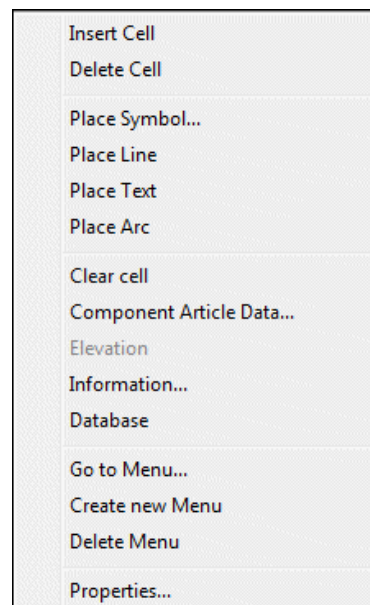
Right-click and choose **Properties**, and type in a title for the active pickmenu. Here you can also determine the size of the pickmenu on the screen. Click **OK**. The title can be seen after the number of the pickmenu.

Go to a specific pickmenu

Right-click and choose **Go to Menu**. You now get a survey of the menus, containing their numbers and titles. Click on the menu you wish to use.

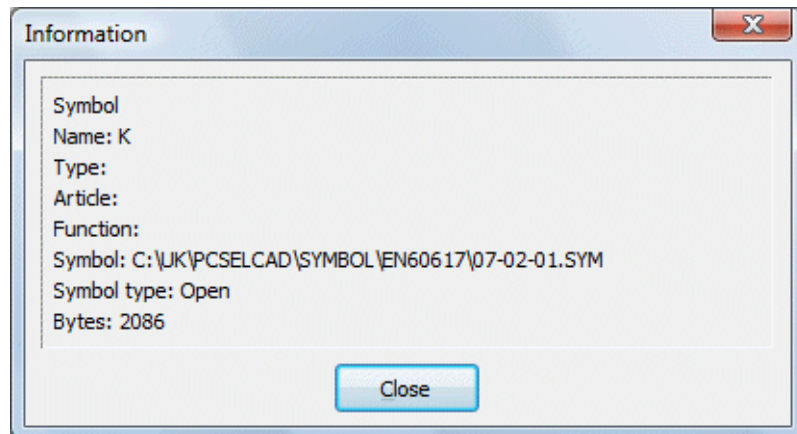
Delete a menu

Right-click and choose **Delete Menu**. The displayed pickmenu — and all of its symbols — will be deleted.



Information about the symbol

Right-click on the symbol in the pickmenu and choose **Symbol information**.

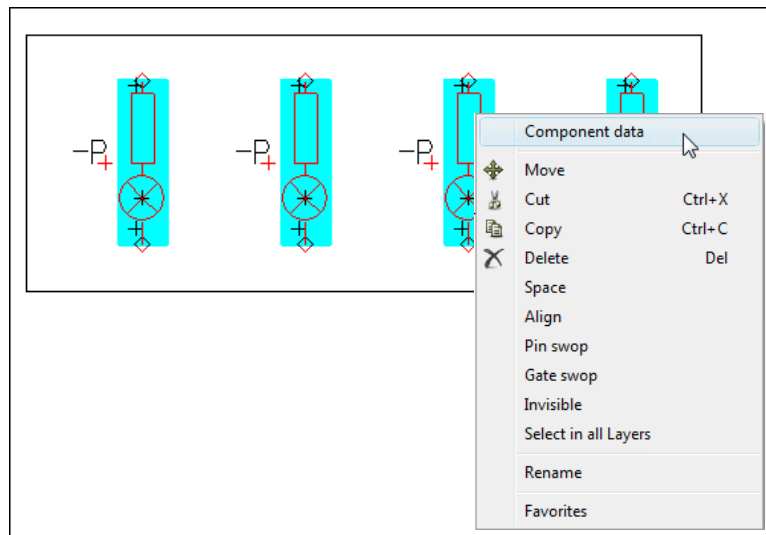


CHANGING SYMBOL NAMES IN AN AREA

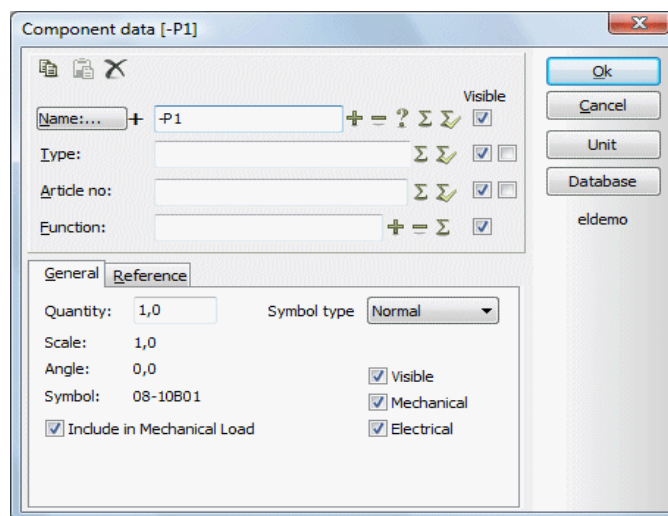
THIS CHAPTER DESCRIBES	
— CHANGING MORE THAN ONE SYMBOL NAME	(P. 180)
— THE NAMING ORDER OF SYMBOLS	(P. 181)
— OTHER OPTIONS.....	(P. 181)

Changing more than one symbol name

When you want to change more than one symbol name at the same time, you can select the symbols by dragging a window around them, right-click inside the selected area, and choose **Component data**.

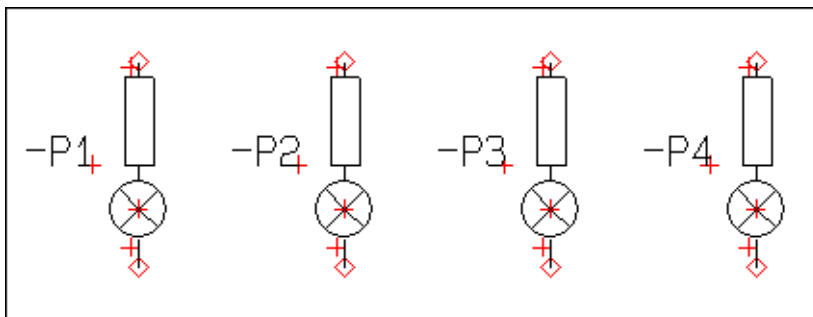


You then enter the **Component data** dialog box.



? In the **Component data** dialog box you apply the automatic counting facility by typing in the name of the first symbol, pressing <Ctrl> while clicking on ?, and then

click **OK**. — Please refer to “Automatic counting in the Component data dialog box” on page 153 for further details.



The symbol names are now changed. Click outside the selected area to deselect the symbols.

The naming order of the symbols

It is important *how* you select the symbols. If you click in the left-hand side of the screen, and then drag a window to the right, the numbering of the symbols will be carried out from left to right.

If you select the window from right to left, the symbol names are named from the right to the left. Thus, the counting direction follows the movement you make when you select the window.

If you select the symbols by <Ctrl>-clicking, the symbols are named in the order you selected them.

Other options

See how to do the same for connection points in “Choosing connection points in Areas” on page 184.

You can also change the symbol texts directly on the diagram page, as described in “Editing the symbol texts directly on a Diagram page” on page 105.

Other advanced ways to change texts directly on diagrams are described in “Locking text types” on page 115 and “Counting on texts” on page 116.



RENAME SYMBOLS - OR INSERT SYMBOL NAMES

IN “RENAME SYMBOLS” ON PAGE 278 YOU CAN SEE HOW TO RENAME SELECTED SYMBOLS — OR HOW TO RENAME ALL PROJECT SYMBOLS.

IF YOU NEED TO INSERT NEW/AVAILABLE SYMBOL NAMES OR CONNECTION NAMES, YOU CAN SEE HOW TO DO THIS IN “INSERT NEW SYMBOL NAMES OR CONNECTION NAMES” ON PAGE 284.

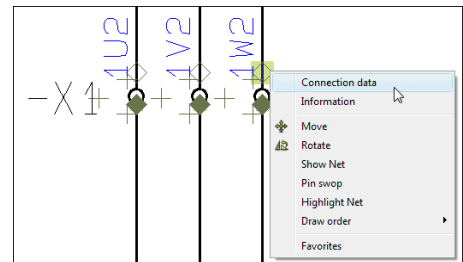
SYMBOL CONNECTION POINTS

THIS CHAPTER DESCRIBES	
— CONNECTION POINTS FOR SYMBOLS CAN BE SELECTED.....	(P. 182)
— SPECIFYING CONNECTION DATA	(P. 182)
— EDITING CONNECTION TEXTS FROM COMPONENT DATA DIALOG BOX	(P. 183)
— EDIT NEXT/PREVIOUS FOR CONNECTION POINTS	(P. 183)

Connection points for symbols can be selected

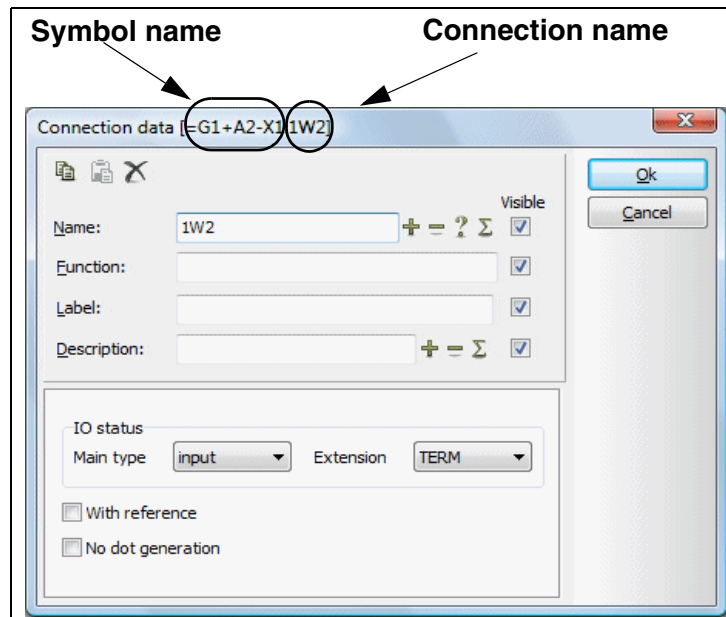
The connection points for the symbols can be selected as well as the symbols themselves.

On the figure to the right, a connection point of a terminal symbol is selected.



Specifying Connection data

If you right-click on a symbols connection point and then choose **Connection data**, you can change the connection data for the connection point — e.g. the connection name.



— If you accidentally select the whole symbol, and not just the connection point, just right-click on the connection point again.

In this dialog box you can type in connection data, and specify which connection point texts is visible on the diagram. The connection point texts are described in “Connection point texts” on page 106.

The **Visible** setting only applies for the selected connection point. The texts are only visible if they are allowed to be so by the over-all visibility settings for connection texts. Please refer to “Texts displayed in the project (general setting)” on page 123 for further details.

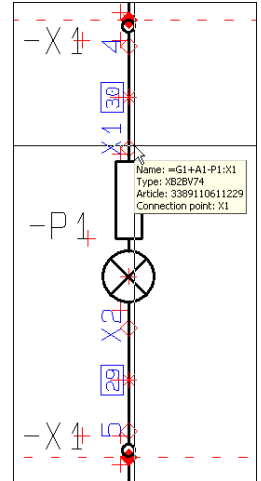
The function of the four buttons in the dialog box, is described in “Specifying the Component name” on page 153.

With reference

— See “With reference” on page 681.

No dot generation

— See “No dot generation” on page 681.

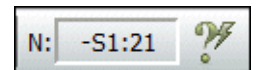


Connection data as Info Tips

When the cross hairs is placed above a connection point, the connection data for the connection point is displayed as an *Info Tip*.

Connection name displayed in the toolbar Symbols

When a connection point is selected, the connection name is displayed in the symbol name field in the toolbar Symbols. This also applies for invisible connection names.



Editing connection texts from the Component data box

When you click on the **Conns.** tab in the **Component data** dialog box, you can edit the connection texts by clicking in the relevant fields, and then type in the texts. — Please refer to “Changing Connection data” on page 157.

Edit Next/Previous for Connection points

When you have selected a connection point, you can jump back and forth between the connection points for the same symbol, by pressing [F5] and [F6] — or by choosing **Edit => Next** or **Edit => Previous**. You jump between the connection points according to their connection numbers.

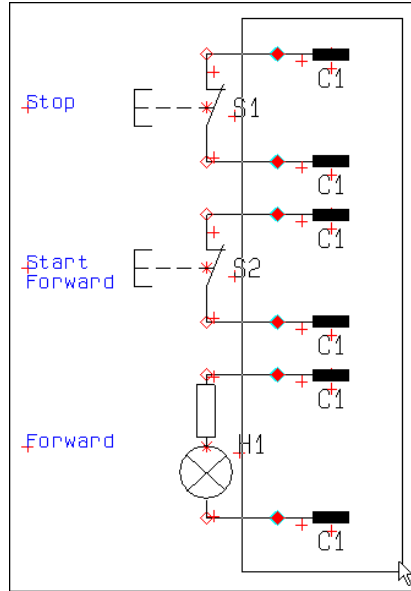
The function also applies for symbol texts.

CHOOSING CONNECTION POINTS IN AREAS

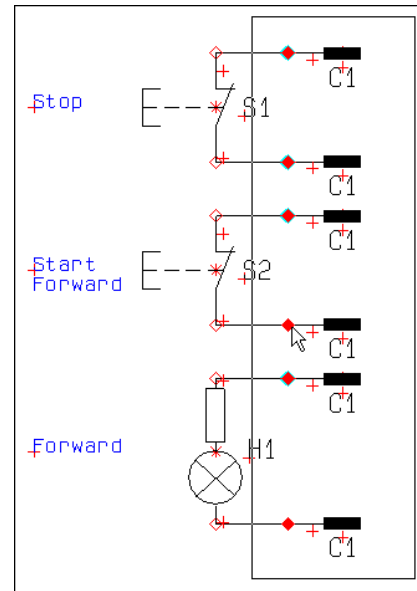


When you have activated the **Symbols** function, start dragging a window around an area and press <Ctrl> when releasing the mouse button, you select the connection points in the area. The *symbols* in the area are not selected.

If you press <Ctrl> and click on connection points outside the area, these will also be included. If you press <Ctrl> and click on a selected connection point, it becomes deselected.



Select the connection points by pressing <Ctrl> when you release the mouse button



Press <Ctrl> and click on the unwanted connection points in the area

When you have selected an area and deselected the unwanted connection points — as displayed above — you right-click inside the area and choose **Connection data**.



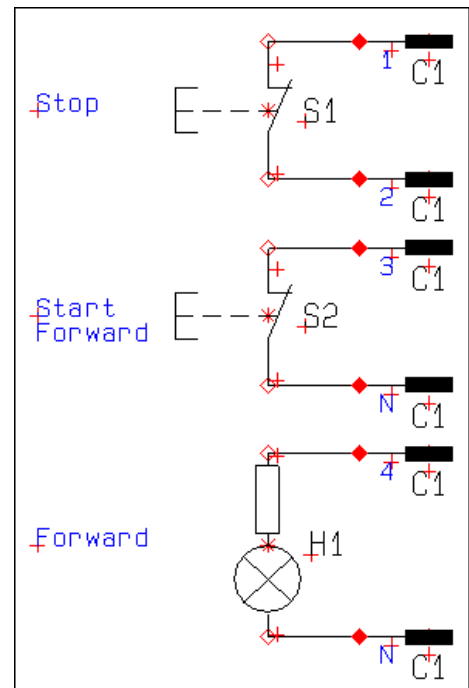
Then apply the automatic counting facility by typing in the first **Name (1)**, pressing <Ctrl> while clicking on +, and then click **OK**. The selected connection points are then renumbered automatically.

— Please refer to “Automatic counting in the Component data dialog box” on page 153 for further details.

Then rename the remaining connection points one by one.

Numbering order for <Ctrl> click

If you select the connection points one by one by pressing <Ctrl> while clicking on the connection points, they will be renumbered according to the order in which you selected them.



PINSWOP

Letting the connection points of symbols swop places is an option, under the name *pinswop*. This can be done within the same symbol, as well as between symbols.

THIS CHAPTER DESCRIBES	
— PINSWOP FOR SYMBOLS WITH TWO CONNECTION POINTS	(P. 185)
— PINSWOP IN AREAS FOR SYMBOLS WITH TWO CONNECTION POINTS	(P. 185)
— FREE PINSWOP.....	(P. 186)
— MOVING CONNECTION POINTS.....	(P. 186)

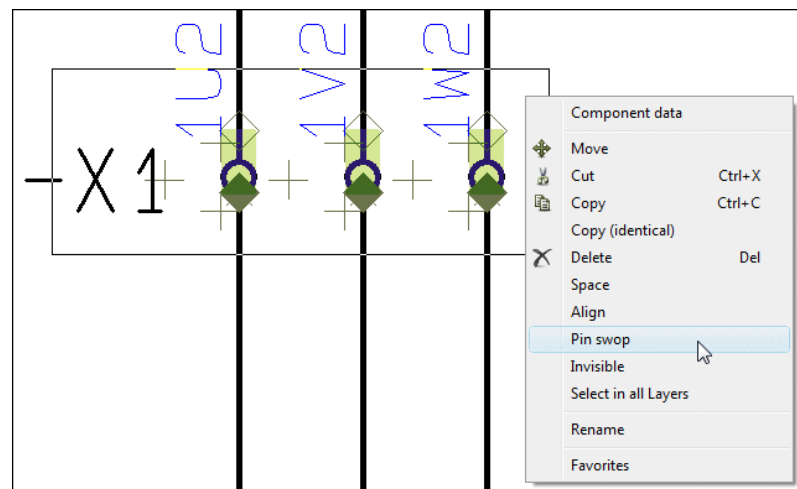
Pinswop for symbols with two connection points

To perform a pinswop on a symbol with two connection points, you first click the **Symbols** button, and right-click on the symbol. Then choose **Pinswop**, and the two connection points of the symbol swop places.



Pinswop in Areas for symbols with two connection points

To perform a pinswop on all symbols with two connection points within an area, you also click on the **Symbols** button first, select an area, and right-click within the area. Then choose **Pinswop**, and the connection points of all symbols with two connection points within the area swop places.



In this way you can for instance easily change a terminal row, from having the output side turning upwards to downwards. — Please refer to “Creating terminal symbols” on page 697 for further details about the input- and output side of terminals.

The connection points in the area for the symbols not having two connection points are left unchanged.

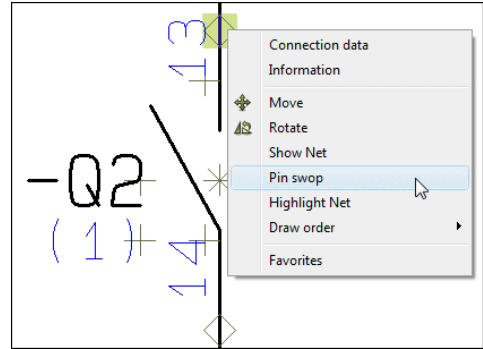
The function can also be activated by choosing **Edit => Pinswop**.

Free Pinswop

Swapping connection points between symbols, and within symbols having more than two connection points, is also an option.

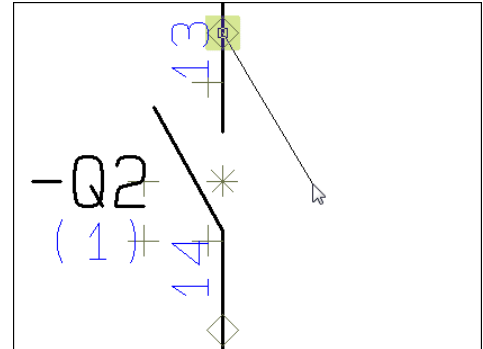


To do this, you also have to activate the **Symbols** button. Then right-click on one of the two *connection points*, you want to swop places.



Now choose **Pinswop**, and you get a rubber band in the cross hairs. Then click on the connection point you want to swop with, and the connection points swop places.

Please note, that this swopping can be performed within the same symbol, or from one symbol to another.



Moving connection points

To move a connection point, do this:



- 1) Activate the **Symbols** button, right-click on the connection point, and choose **Move**.
- 2) You now get the connection point in the cross hairs: Click where you wish to place the connection point.

When you move connection points on symbols, connected lines are moved also.



DO NOT MOVE CONNECTION POINTS ON STANDARD SYMBOLS

THE POSSIBILITY TO MOVE CONNECTION POINTS IS DESIGNED FOR SPECIAL PURPOSES, SUCH AS BLOCK SYMBOLS.

CONNECTION POINTS CAN *NOT* BE MOVED OUTSIDE THE FRAME OF THE SYMBOL (DISPLAYED WHEN SELECTING SYMBOLS).



REPLACING SYMBOLS

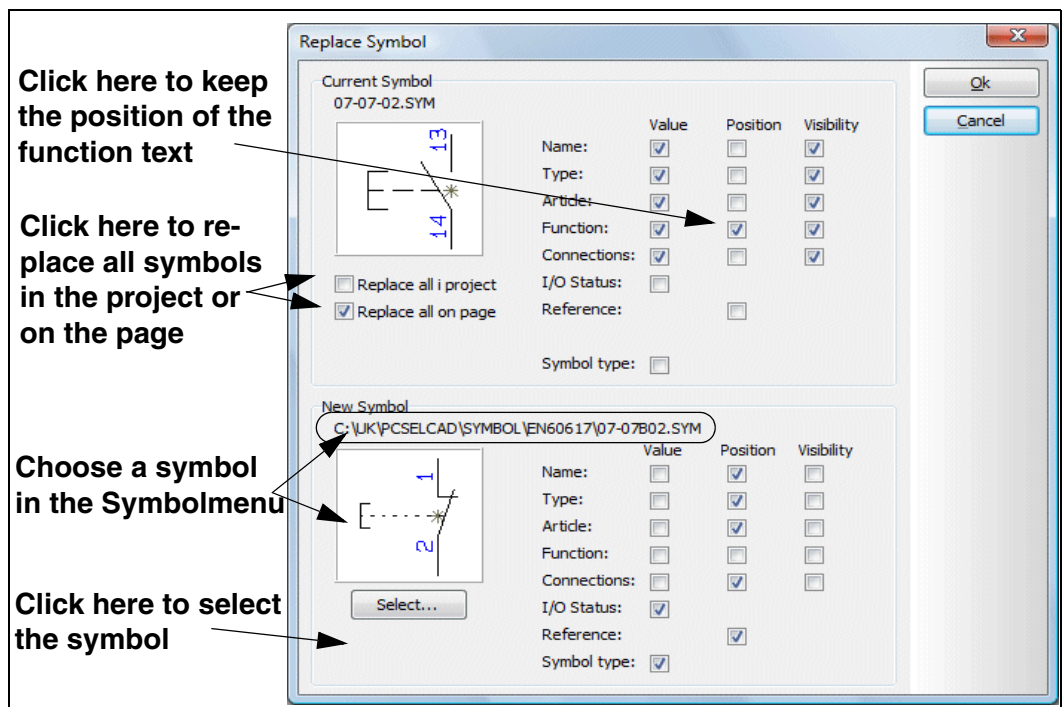
THIS CHAPTER DESCRIBES	
— REPLACING SYMBOLS AUTOMATICALLY	(P. 187)
— CONNECTION POINTS WHEN REPLACING SYMBOLS	(P. 189)
— TEXT PROPERTIES WHEN REPLACING SYMBOLS	(P. 189)
— REPLACING OLDER VERSIONS OF SYMBOLS	(P. 189)

Replacing symbols automatically

Automatic replacing of all occurrences of a symbol in a project is also an option.

If you decide to use e.g. an open contact instead of a close contact, you can replace the symbols in the project.

To do so select one of the open contacts, and choose **Functions => Replace symbol** (short-cut key [F4]). You then enter the following dialog box:



When you enter the dialog box, it is presumed that you wish to keep the contents/ value of the symbol texts from the *old* symbol, while the positions of the texts follows the text-positions for the *new* symbol.

The symbol text Value when replacing

When you set a check mark for a **Value** for some of the texts for the present symbol, these texts are not replaced when you change symbol.

The I/O status and Symboltype for the symbol

By selecting/deselecting **I/O status** you specify whether to keep the I/O status for the old symbol, or to use the I/O status for the new symbol.

By selecting/deselecting **Symboltype**, you specify whether to keep the symboltype for the old symbol, or to use the symboltype for the new symbol.

The symbol text Position when replacing

You can for instance enable **Position** for the function text of the present symbol. This ensures you, that the *location* and the *orientation* of the function text remains unchanged, even though you change symbol.

The symbol text Visibility when replacing symbols

Likewise you can specify whether the selected **Visibility** shall remain unchanged for the new symbol, or whether the visibility shall be specified by the symbol definition for the new symbol. — By keeping the visibility for the original symbol when replacing terminal symbols, you can therefore avoid having to change the visibility for the names of the terminal symbols after the replacement.

Replace all symbols in the project or on the page

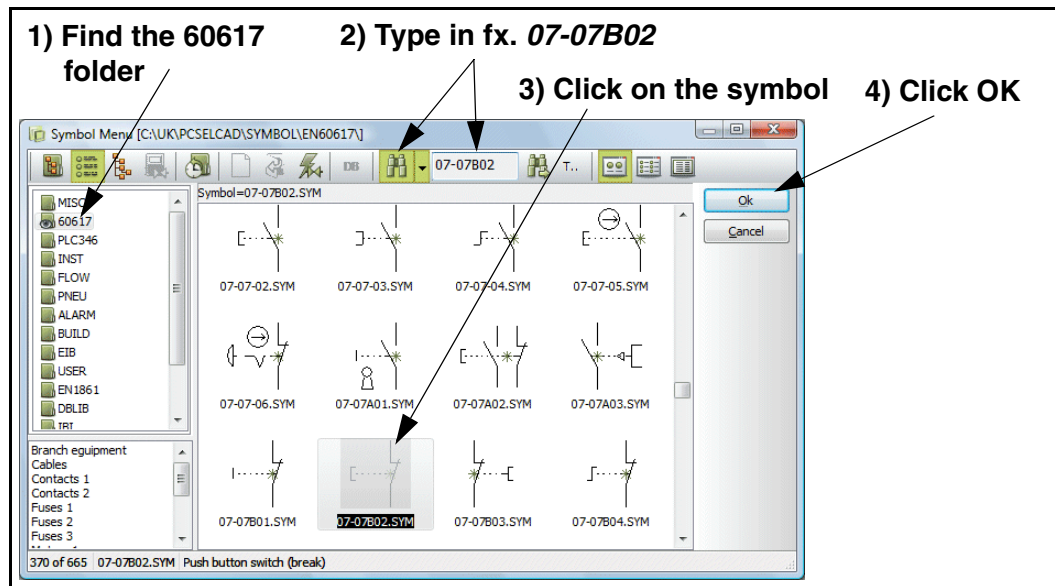
Select **Replace all in project** in order to replace all symbols, of the selected type, in the project. If you wish to replace all symbols of the selected type on the active page in the project, select **Replace all on page**.

If you do not select any of these options, only the selected symbol will be replaced.

Selecting a new symbol

Click on **Select** at the bottom of the dialog box, to choose which symbol you wish to use instead.

You then enter the **Symbol menu**, where for instance you choose the symbol **07-07B02**.



Click **OK**. — See “Fetching symbols using the Symbol menu” on page 140.

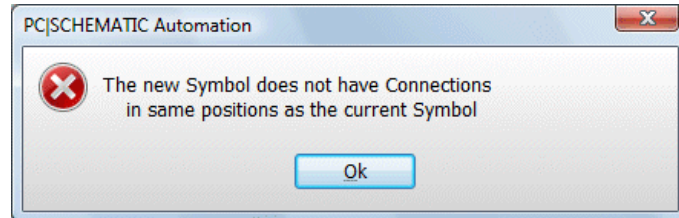
You now return to the dialog box **Replace Symbol**, where you click **OK**.

The symbol(s) have now been replaced in the project.

— Please note, that it is possible to undo the replace symbol function.

Connection points when replacing symbols

If the connection points of the old and the new symbol are not placed on identical positions, you get a warning:



Click **OK**. However, you can still choose to replace the symbol(s) by clicking **OK** in the **Replace Symbol** dialog box.



YOU ONLY GET THIS WARNING WHEN THE CONNECTION POINTS ARE CONNECTED ELECTRICALLY. WHEN YOU CLICK **OK** THE NOT MATCHING CONNECTION POINTS ARE PARKED.

If the new symbol has less connection points than the old one, you get an error message. If the new symbol has *more* connection points, this will be accepted when it has connection points on the same locations as the old one.

Transferring connection point texts when Replacing symbols

For symbols with just one connection point, the **Replace symbol** function also transfers connection point texts from the old symbol to the new symbol, when these connection points does not have the same position.

When the symbols have more than one connection point, and some of these connection points does not have the same positions, the connection point texts are transferred when the connection points are created with the same name in the symbol libraries.

Text properties when replacing symbols

The text properties for the symbol texts follows the definition of the new symbol.

Replacing older versions of symbols

Using the **Replace symbols** function you can replace all the old versions of the project symbols. You can see that a newer version exists of a symbol, if it has a question mark (?) in its symbol file name.

AUTOMATIC CREATION OF SYMBOLS

In **Automation** it is possible to create rectangular and circular symbols automatically, by entering their dimensions. This can both be done "on the fly" while working on a project, and by specifying the symbol dimensions via the database.

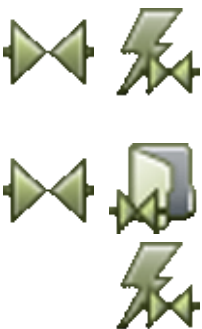
THIS CHAPTER DESCRIBES
— AUTOMATIC CREATION OF SYMBOLS — USING THE SYMBOL GENERATOR ... (P. 190)
— AUTOMATIC CREATION OF SYMBOLS — BY SYNTAX (P. 192)
— SYNTAX FOR DIRECT CREATION OF SYMBOLS (P. 193)
— AUTOMATIC CREATION OF ELECTRICAL SYMBOLS FROM THE DATABASE (P. 194)
— AUTOMATIC CREATION OF MECHANICAL SYMBOLS FROM THE DATABASE (P. 194)

When you want **Automation** to generate symbols for you automatically, while working on a project, you can do this in two ways:

- a) Using the **Symbol generator**: Read “**Automatic generation of symbols — using the Symbol generator**” on page 190. (If you enter the **Symbol generator** this way, you can furthermore save the symbol, and continue working on the symbol, as described in “Creating Symbols” on page 676).
- b) By manually typing in the syntax for the appearance of the symbol: Read “AUTOMATIC CREATION OF ELECTRICAL AND MECHANICAL SYMBOLS - BY SYNTAX” ON PAGE 192.

Automatic generation of symbols — using the Symbol generator

You can enter the **Symbol generator** in four different ways:



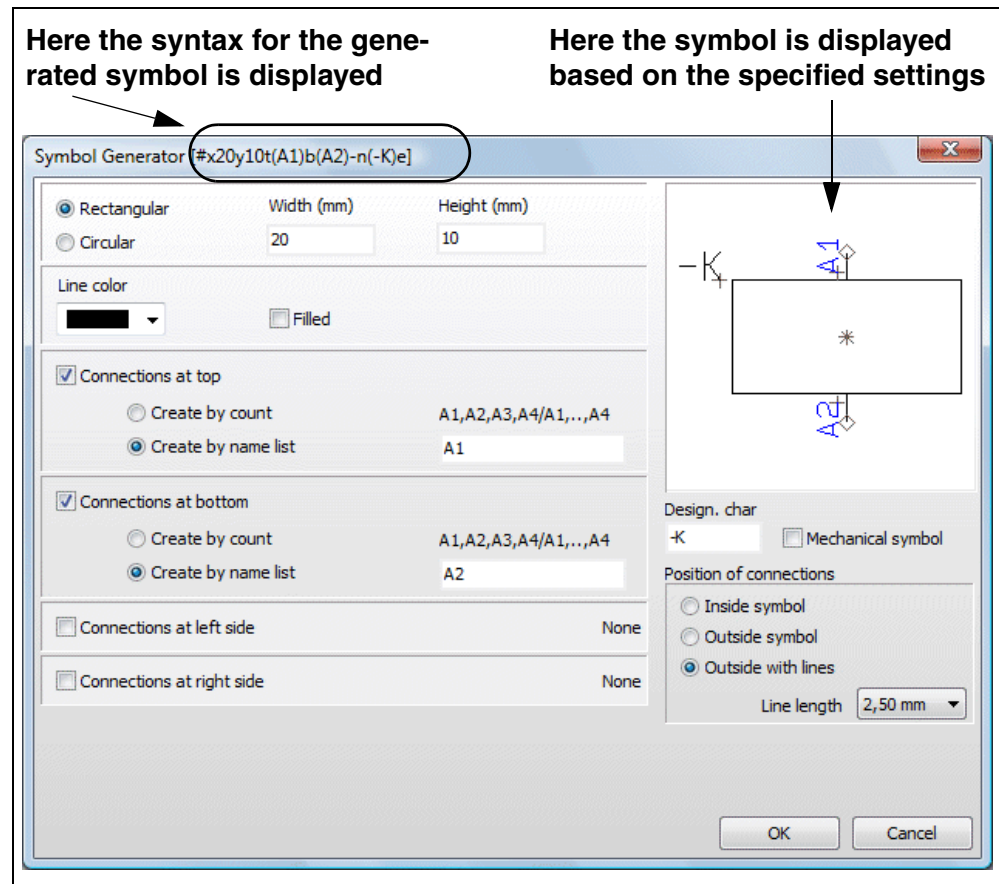
- a) Select **Symbols** and click the **Symbol generator** button.
- b) Select **Functions => Symbol generator**.
- c) Select **Symbols** and click the **Symbol menu** button. In the **Symbol menu**: Click the **Symbol generator** button.
- d) When editing symbols: Click the **Symbol generator** button.

You now enter the **Symbol generator**.

Options in the Symbol generator

In the **Symbol generator** you specify the design of the symbol. In the window in the top-right corner of the dialog box, you can see the symbol, based on the pre-

sent settings. The syntax for the generated symbol is displayed in the title bar for the dialog box:



SELECT RECTANGULAR OR CIRCULAR

At the top of the dialog box, you select whether to create a **Rectangular** or a **Circular** symbol, and specify **Width/Height** or **Radius**.

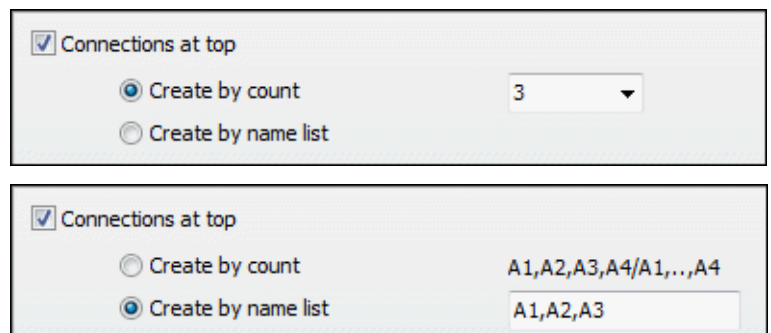
LINE COLOR AND FILL COLOR

Under **Line color**, you select which color to use for the lines in the symbol. If you select **Filled**, you can also select a fill color.

CONNECTIONS: NUMBER, NAMING AND POSITION

You can place connections to the symbol on the top, at the bottom, and at the right and the left side of the symbol.

All of these places, you can either specify a number of connections (which names the connections 1,2,3,...), or specify the names by typing in a name list. — Please remember to see the selected connections in the upper-right corner of the dialog box.



SPECIFYING A DESIGN CHARACTER

In the right side of the dialog box, you can specify a designation character for the symbol. Please note that it is not necessary to place a dash (-) before the symbol name, because the program can handle this for you automatically for all symbols in the project. — See “Preceding “-” in symbol names” on page 339.

ELECTRICAL OR MECHANICAL SYMBOL

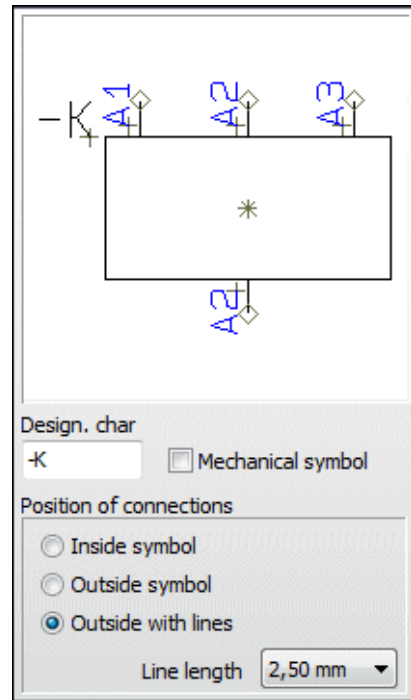
By default the symbol is created as an electrical symbol, but if you select **Mechanical symbol**, it is created as a mechanical symbol.

Beyond the different properties connected to these two different types of symbols, the texts are placed and oriented differently for the two symbol types.

POSITION OF CONNECTIONS

Under **Position of connections** you can choose between:

- **Inside symbol:** The connections are placed on the symbol, and the connection texts are placed inside the symbol.
- **Outside symbol:** The connections are placed on the symbol, and the connection texts are placed outside the symbol.
- **Outside with lines:** The connections are placed outside the symbol, with connecting lines to the connections. You can specify the length of the connecting lines in the field **Line length**.

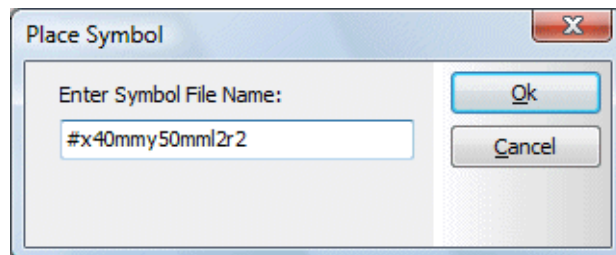


Automatic creation of electrical and mechanical symbols - By syntax

When you need to create a simple symbol quickly, do the following:

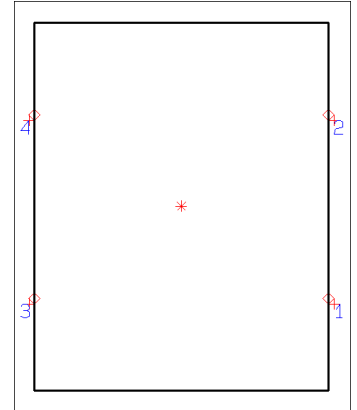


- 1) Click the **Symbols** button, and press the [k] button. The dialog box **Place Symbol** is displayed:



- 2) Type in the dimensions of the symbol, and the number of connection points — see “Syntax for automatic creation of symbols” on page 193.
- 3) Place the symbol in the project, and you enter the dialog box **Component data**.
- 4) Specify the component data — for instance just the symbol name — and click **OK**.

You have now created a symbol, which you have placed directly in the project.



Symbol type for automatically created symbols

Symbols, which are created automatically, are assigned the symbol type **Normal**.

SYNTAX FOR AUTOMATIC CREATION OF SYMBOLS



ACCORDING TO IEC / EN STANDARDS

WHEN YOU GENERATE SYMBOLS AUTOMATICALLY ON ELECTRICAL PAGES, THE IEC / EN STANDARDS FOR PLACING TEXTS ARE APPLIED AUTOMATICALLY.

Two types of symbols can be created directly: Circle- and Box symbols.

Circle symbols

When you type **#r40mm**, the program creates a circle with radius 40 mm as a symbol.

Box symbols

To create a box symbol with the dimensions 20x30 mm, type **#x20mmy30mm**.

Specifying connection points

To assign two connection points to the left and to the right, add **l2r2** after the physical dimensions of the symbol. The letter **l** stands for "left", the letter **r** stands for "right".

To assign two connection points on each side of the symbol above, you must therefore write **#x40mmy50mml2r2**.

When you write **l2**, the counting is done top-down. If you write **l-2**, the counting is done bottom-up.

Accordingly, you can write **t** for "top" and **b** for "bottom". If you also wish to specify the connection point names, you can write these in brackets in stead of the number of connection points.

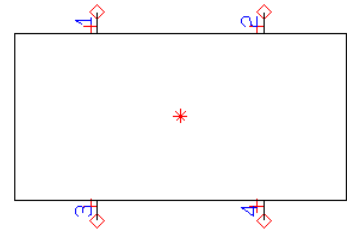
Thus you can write **#x40mmy50mmt(13,21)b(14,22)**. Hereby you get a symbol with the dimensions 40x50, with the connection points **13** and **21** on the top, and the connection points **14** and **22** at the bottom.

Connecting line for connection points

To place a 2,5mm connecting line to the connection points in the symbol, add a "-" to the end of the format string.

E.g.: **#x40mmy20mmt2b2-**

All connection points are hereby assigned a 2,5mm connecting line.

*Placing the connection point names*

If you take no special action, the connection point names are placed outside the symbol. When you want these to be placed inside the symbol, write *i* at the end of the text. If you write **#x10mmy20mml2r2i**, you get a symbol with two connection points to the left and two to the right, where the connection point names are placed inside the symbol.

For symbols placed on mechanical pages, the names are typically placed inside the symbol, while they are typically placed on the outside for symbols on electrical pages.

Automatic creation of Electrical symbols via the Database

To create a symbol directly from the database, type in the symbol dimensions in the **PCSTYPE** field for the component.

Read more about the syntax in "Syntax for automatic creation of symbols" on page 193.

Automatic creation of Mechanical symbols via the Database

To any component in the database, you can attach a mechanical symbol. The name of the file you have saved this symbol in, you can write in the **MECTYPE** field.

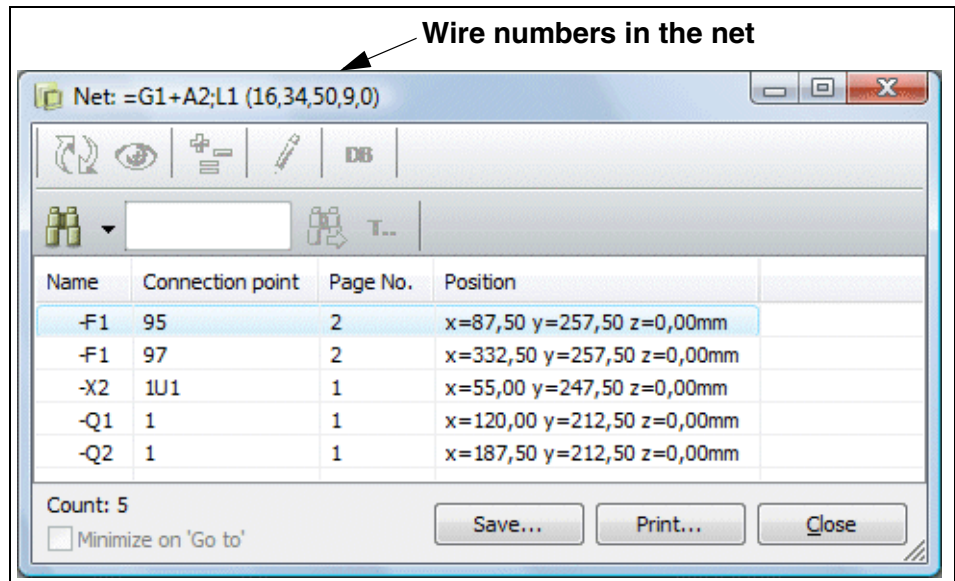
Creating a Mechanical symbol automatically

When you wish **Automation** to automatically create a symbol with the dimensions 30x50 mm, you must write **#x30mmy50mm** in the **MECTYPE** field. If you instead write **#r40mm**, the program creates a circle with radius 40 mm as a mechanical symbol. Read more about this in "Syntax for automatic creation of symbols" on page 193.

— Read more about mechanical symbols in "Mechanical symbols" on page 593.

SHOW NET

If you right-click on a connection point of a symbol — or on a signal symbol — and choose **Show net**, you can see which symbols are connected to the same potential.



It is also specified which connection points on the symbols are connected to the potential, and the position of the symbols are also displayed.

After the name of the net, the wire numbers in the net are displayed in brackets, in the title bar of the dialog box.

If you double-click on one of the lines, you return to the diagram, and the cross hairs point out the location of the connection point of the chosen symbol.

Generate connections lists

To generate a connections list file, choose **Lists => Connections List File**.

— Read more about generating list files in “Exporting List Files” on page 629.

— See also “Connections list for wire numbering” on page 391.

SIGNAL SYMBOLS

THIS CHAPTER DESCRIBES	
— WHAT IS A SIGNAL SYMBOL.....	(P. 196)
— THE APPEARANCE OF SIGNAL SYMBOLS.....	(P. 197)
— DESIGNING SIGNAL SYMBOLS.....	(P. 197)
— PLACING SIGNAL SYMBOLS WHEN DRAWING LINES.....	(P. 197)
— SIGNAL SYMBOLS WITH REFERENCE DESIGNATIONS.....	(P. 200)
— PLACING SIGNAL SYMBOLS ON PARKED LINES.....	(P. 201)
— PLACING SIGNAL SYMBOLS DIRECTLY.....	(P. 201)
— VARIANT NAMES ON SIGNAL SYMBOLS.....	(P. 202)
— INSERT REFERENCE DESIGNATIONS ON SIGNALS AUTOMATICALLY.....	(P. 203)
— UNREFERRED SYGNAL SYMBOLS.....	(P. 203)
— IMPLIED SIGNALS FOR TERMINALS AND CABLES.....	(P. 204)

What is a signal symbol

A special kind of symbols are called signal symbols. They are used when you start or end a conducting line on a non electrical point in a diagram, or simply wish to name an electrical potential for easy identification.

A signal symbol can be placed in all electrical points, and the signal symbol specifies an electrical connection.

All signal symbols with the same name are connected electrically

To identify an electrical connection from a point, you place a signal symbol, and assign a name to this symbol.

The program views all signal symbols with the same signal *name* as the same electrical connection (potential). This means that all signal symbols for instance using the name **L1** are connected electrically.

Please note that it is only the *name* of the potential, that identifies the electrical connection. This means that you can use the signal *symbol* itself for different potentials, if only you remember to assign different names to the signal symbols.

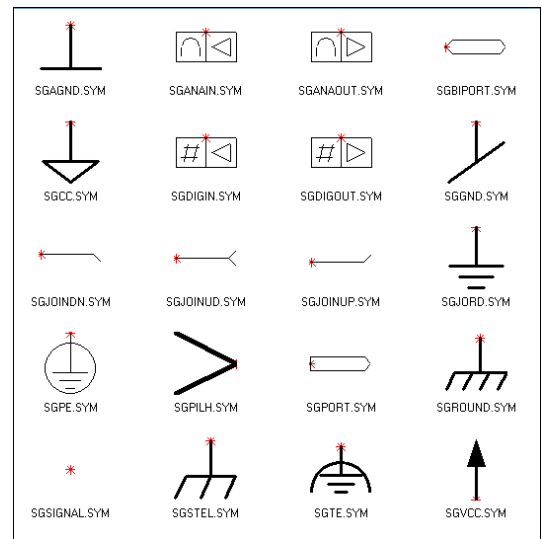
— See “Automatic insertion of electrical potentials” on page 74.

The appearance of signal symbols

Signal symbols can for instance look as displayed to the right:

Various types of signal symbols are located on the **symbol** folder, in the sub folder **MISC**.

To distinguish the signal symbols from other types symbols, they all have names starting with **SG**.



Designing signal symbols

See how to design your own signal symbols in “Creating Signal Symbols” on page 717.

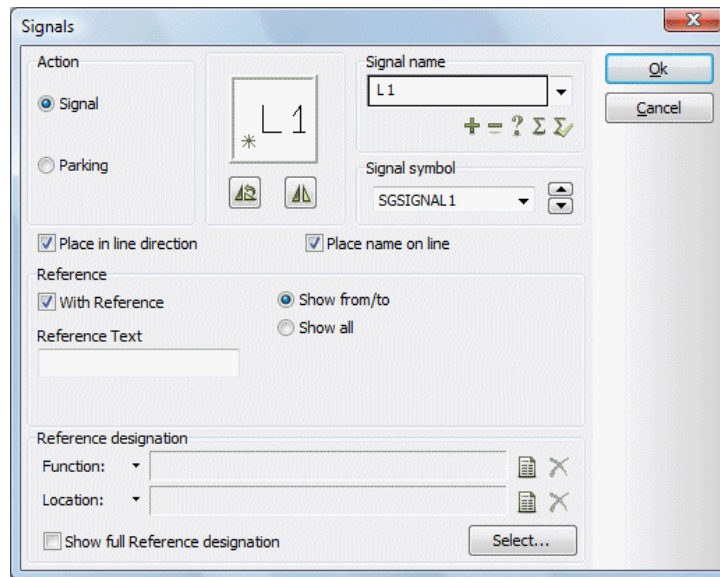
Placing signal symbols when drawing lines



Activate the **Pencil**, click on **Lines**, and make sure the **Conducting lines** button is activated (Highlighted).

Then click on the diagram where you want the signal symbol to be located.

You now enter the dialog box **Signals**:



Parking

If you wish to wait to connect the conducting line, you can *park* the line. To do this click on **Parking**, and then click **OK**. Hereby you get a conducting line with no electrical connection.

However, this is a temporary solution only, which gives you the option to return later on, and then assign a name to the signal.






— See how to locate parkings in a project in “Design Check” on page 310.

Placing the signal symbol

If you, on the other hand, wish to place a signal symbol, click on **Signal**.

SPECIFY SIGNAL NAME

In the field **Signal name** you can either type in a name, or use one of the following options:

BUTTON	FUNCTION
	INCREASE THE NUMBER IN THE SIGNAL NAME BY ONE.
	DECREASE THE NUMBER IN THE SIGNAL NAME BY ONE.
	GIVES YOU THE NEXT AVAILABLE SIGNAL NAME OF THE SYMBOL TYPE CHOSEN.
	GIVES YOU A LIST, WHERE YOU CAN CHOOSE BETWEEN THE SIGNAL NAMES ALREADY USED IN THE PROJECT.
	GIVES YOU A LIST OF USED VARIANT NAMES FOR THE SELECTED SIGNAL SYMBOL — SEE “VARIANT NAMES ON SIGNAL SYMBOLS” ON PAGE 202.

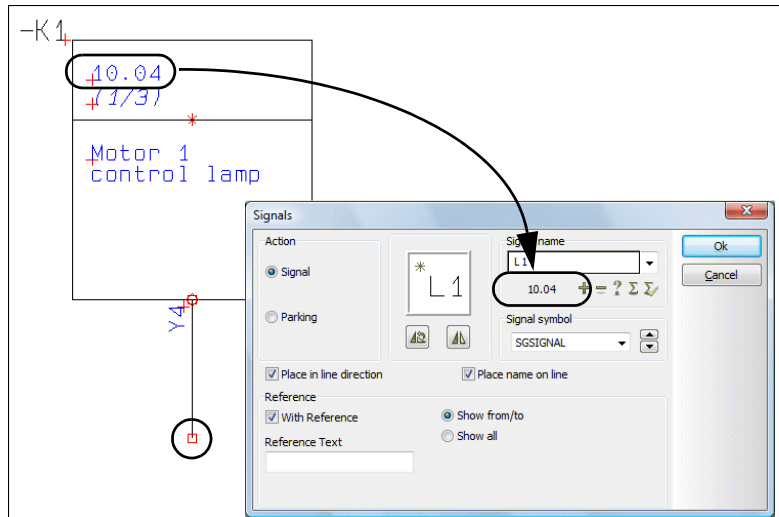
You can also click on the down-arrow in the field **Signal name**. This gives you the option to choose between predefined signal names.

— See the section “Frequently used signal symbol Names” on page 720 to see how you can add names to this list by yourself.



SIGNAL NAMES ASSIGNED FROM FUNCTION TEXT / IO ADDRESS

WHEN YOU START A LINE FROM A CONNECTION POINT, AND LET IT END WITH A SIGNAL, AN EXTRA BUTTON IS DISPLAYED IN THE **SIGNALS** DIALOG BOX. WHEN YOU CLICK THIS BUTTON, THE SIGNAL NAME IS FETCHED FROM THE *FUNCTION TEXT* FROM THE CONNECTION POINT, FROM WHICH THE LINE STARTED:



WHEN THE LINE STARTS FROM A PLC CONNECTION POINT — AS IN THE FIGURE ABOVE —, YOU CAN THEREFORE EASILY ASSIGN THE ADDRESS FOR THE PLC CONNECTION TO THE SIGNAL NAME. THE SHORT-CUT KEY FOR THIS FUNCTION IS <ALT+S>.

SELECT SIGNAL SYMBOL

When you click the up- and down-arrow in the **Signal symbol** field in the right hand side of the dialog box, you can scroll between the signal symbols that are known to the system. The chosen symbol will then be shown in the dialog box.

— See how to place signals in this menu in the section “Frequently used signal symbols” on page 719.

ROTATE OR MIRROR THE SIGNAL SYMBOL



Below the symbol itself, you can click the buttons **Mirror vertically** or **Rotate** to mirror or rotate the symbol.

SIGNAL SYMBOLS FOLLOWING THE LINE DIRECTION

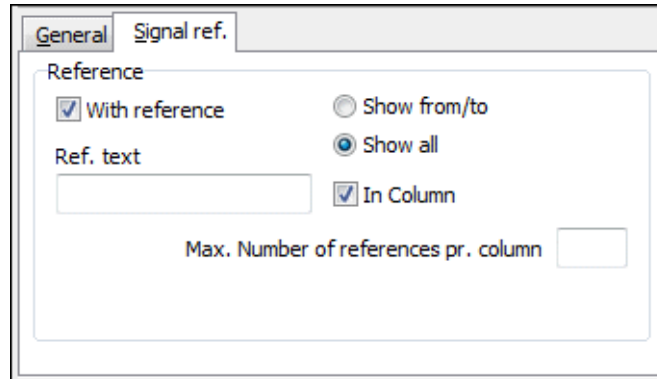
When you select **Place in line direction**, the signal symbol will follow the direction of the line. When you also select **Place name on line**, the name of the signal symbol will be displayed above the line. The program memorizes the selected settings for the different signal symbols in the project — for up to 25 different signal symbols.

PAGE REFERENCES ON SIGNAL SYMBOLS

In **With reference** you can assign the symbol reference texts, that refers to the other places the same signal symbol is being used.

Selecting **Show from/to** sets references to the nearest previous and to the next following page on which the same signal name occur.

Clicking on **Show all** will show references to all places the same signal name occurs. The program can handle references of an unlimited length.



When you select **Show all** and **In Column**, you can specify the maximum number of references in each column in the field **Max. Number of references pr. column**.

If you click in the field **Reference text**, you can type in the reference text, that will be displayed in front of the reference — such as *Page_*.

REFERENCE DESIGNATIONS ON SIGNAL SYMBOLS

When you wish to include reference designations in the signal name, you can do two things:

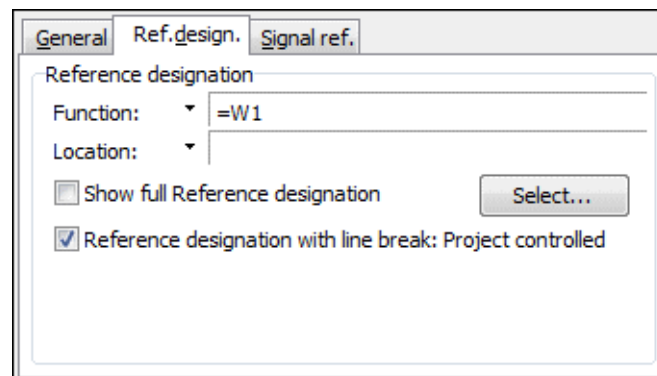
- a) Click on **Select**, and you get the option of selecting references via the dialog box **Reference designations**, as described in “Reference designations for symbols” on page 336, or:



- b) Click on the button **Insert current page Function/Location aspect** for the fields **Function** and/or **Location**. This assigns the function/location aspect to the signal name.



If you click the **Delete** button for the fields **Function / Location**, the displayed function/location aspect is deleted.



If you click on **Show full Reference designation**, the full reference designation is displayed for the symbol. If not, only parts of the reference designation different from the page reference designation, are displayed.

CLICK OK WHEN THE SETTINGS ARE OK

When you are contempt with the settings, click **OK**, otherwise click **Cancel**.

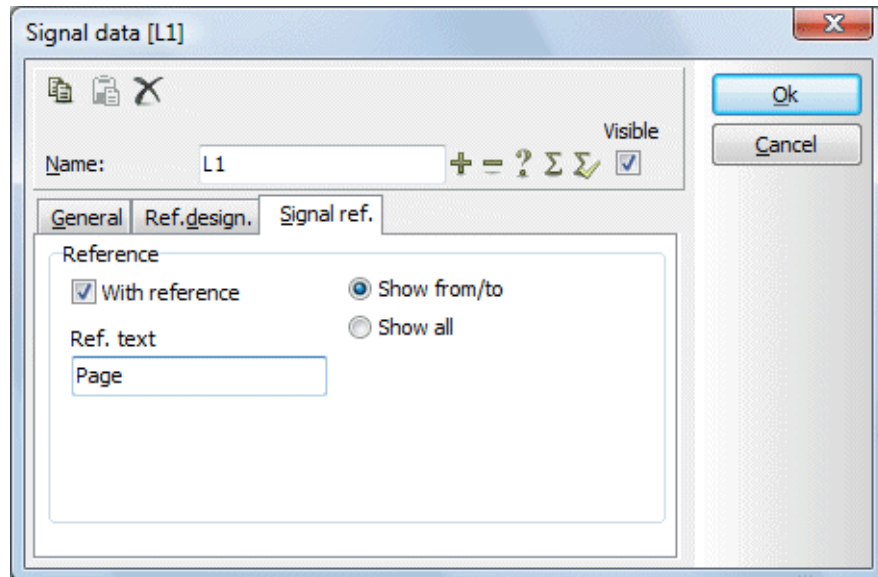


Placing signal symbols on parked lines

To get the signal symbol in the cross hairs, click the **Symbols** button, and fetch a signal symbol — just like you would fetch any other symbol. Read more about this in “Fetching symbols” on page 135.

When you have the signal symbol in the cross hairs, you can place it on a parked line. — See “Parking” on page 197.

When you click on the parking point on the drawing, you enter this dialog box:



Because you have already selected a signal symbol, you are not asked about this here. Fill out the dialog box following the guidelines given above.

Placing signal symbols directly

You can place a symbol in the cross hairs — for instance via the **Symbol menu** — and place it by clicking on the page. This makes you enter the **Signal data** menu, which you fill out and click **OK**.

The program now automatically activate the Lines command, and you get a line in the cross hairs, starting from the signal symbol.

If you delete the line later on, the signal symbol remains on the page.

Signal symbols can also be placed directly on the connection point of a symbol, or on a conducting wire.

Variant names on signal symbols

Automation handles variant names on signal symbols:

What are variant names on signal symbols used for?

Variant names on signal symbols are applied for specifying branches of the same signal.

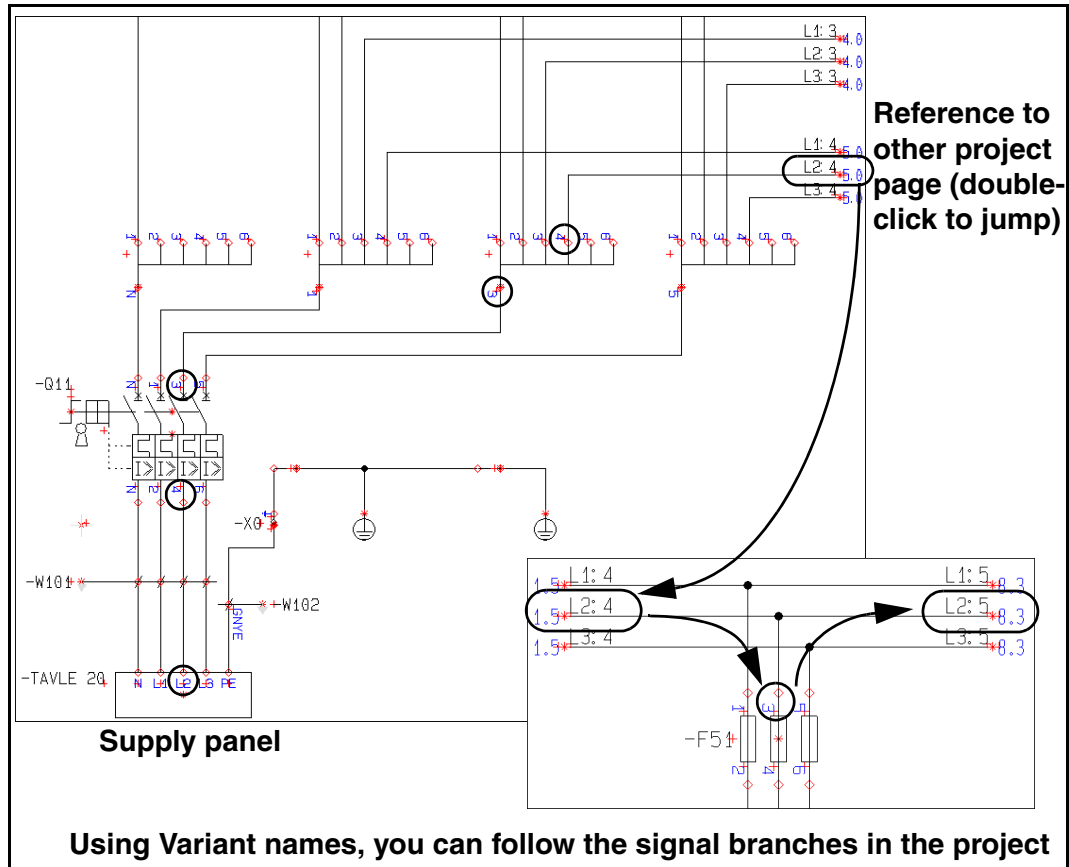
This being for instance the source point from which three signals with the name **L1** branches off. These three branches can for instance be named **L1:1**, **L1:2** and **L1:3**.

The program sees these as the same signal.

Furthermore, the signal flow follows the variant names. You can change variant name on a page, where for instance the name in the left-hand side is **L2:4** and in the right-hand side is **L2:5**. — See the figure below.

This makes it possible to keep track of the signal flow.

This supports the correct automatic placing of wire numbers for signals, and can make the diagrams mounting correct, with respect to signals.



Comments on variant names

Comments can be attached to variant names, as for instance **L1:1 (230 VAC)**.

Show signal

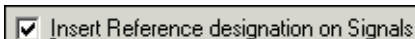
When you select **Functions => Show signal**, you enter a dialog box from where you select a signal name according to the same principle as described in “Find all symbols with identical names” on page 227.

When you have selected a signal name and click **OK**, the selected signal is displayed in green on all project diagram pages, while everything else is displayed in red.

To deselect the function, choose **Functions => Show signal** again.

Insert Reference designations on Signals automatically

When you wish to let **Automation** insert reference designations on signal symbols automatically, choose **Settings => Project data => Reference designations**, and select ***Insert Reference designations on Signals***.



When you place signal symbols on *pages* with assigned reference designations, the signal symbol will then get these reference designations.

When you place signal symbols in *areas* with assigned reference designations, the signal symbol will then also get these reference designations.

Insert Reference designations on Signals is by default *not* selected when installing the program. You can always place reference designations manually.



SIGNAL NAMES WITH REF. DESIGNATIONS ACCORDING TO EN61346

A SEMICOLON (;) IS AUTOMATICALLY PLACED BETWEEN SIGNAL NAMES AND REFERENCE DESIGNATIONS, AS PRESCRIBED IN EN61346.

Unreferenced signal symbols

If you wish to set all unreferenced signal symbol in the project invisible, just choose **Functions => Special Functions => Set unreferenced signal symbols invisible**.

— See also “Unreferenced cable symbols” on page 216.

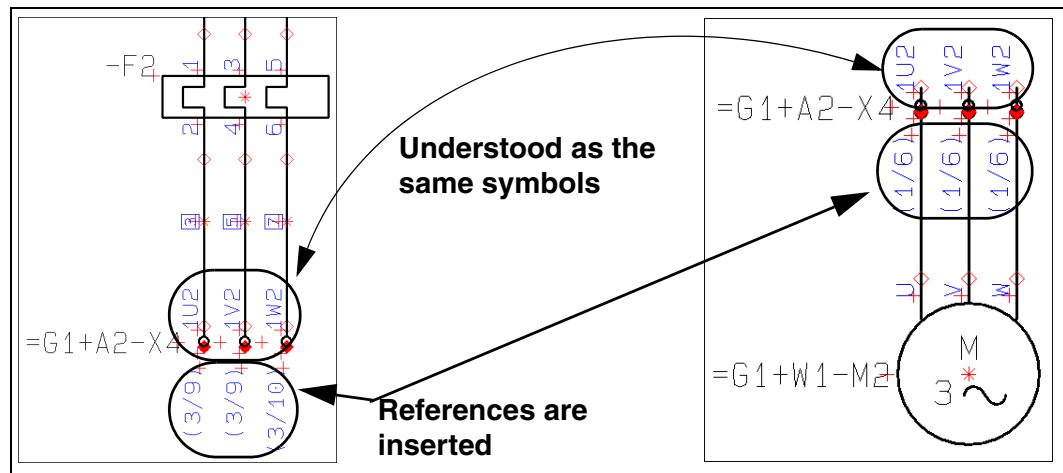
Implicit references for terminal and cable connection points

In some cases references/connections between connection points will be implicit:

Implicit signals for terminals only connected on the internal or external side

When two terminal symbols with the same names and the same connection names, both connection points have activated **With reference**, and both are only connected in one end, they are understood as two representations of the same symbol.

References will automatically be displayed for the connection points:



The terminal symbols in the figure above are understood as representations of the same symbols, because:

- 1) They have same connection names (**1U2**, **1V2** and **1W2**) for the same terminals ($=G1+A2-X4$)
- 2) **With reference** is activated for the *connection points* on the terminal symbols, and

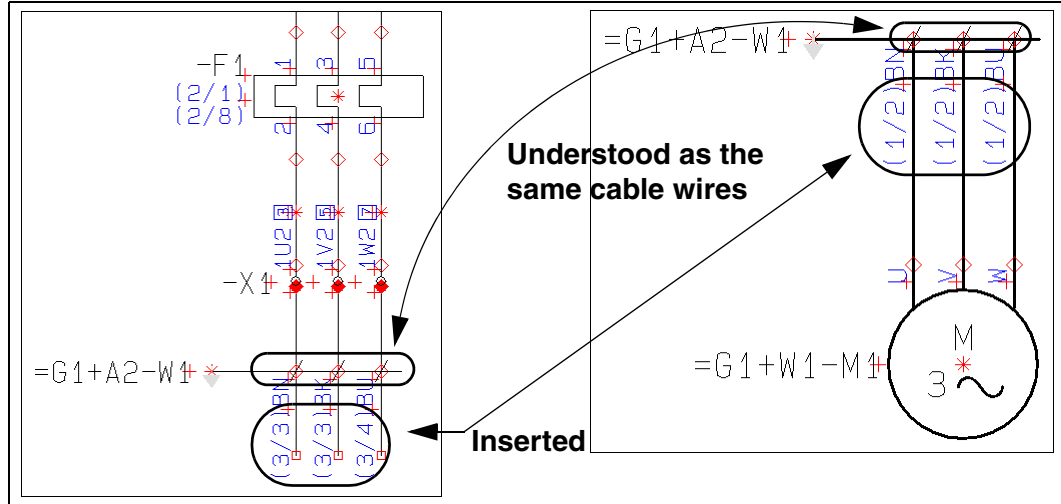
3) Both sets of terminals are only connected in one end

**To select multiple *connection points*:
Remember to press <Ctrl> before releasing the mouse button**

Set *With reference* on the *connection points* for *both* rows of terminals

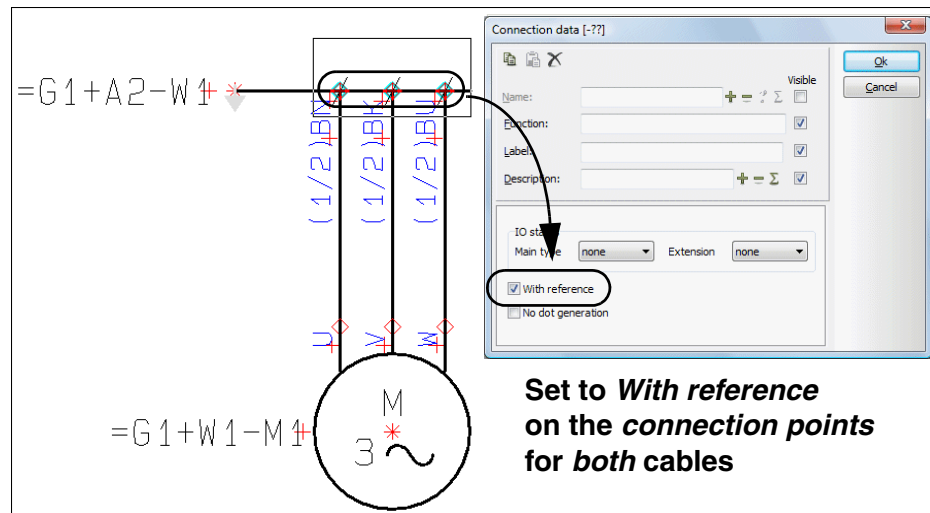
Implicit signals on cable wires

When two cable symbols with the same names and the same cable wire names, and the cable wire connection points for both cables have activated **With reference**, and the cable wire connection points for both cables are only connected in one direction, the cable wires for the symbols are understood as two representations of the same cable wires for the same symbol.



The cable wires in the figure above are understood as the same symbols, because:

- 1) They have the same cable wire names (**BN**, **BK** and **BU**) for the same cable (**=G1+A2-W1**)
- 2) **With reference** is activated for the cable wire *connection points* for both of the cable symbols, and
- 3) Both sets of cable wires are only connected in one direction



TERMINAL SYMBOLS

When you place terminal symbols, you indicate which connection points are *input*, and which are *output* for the terminal. This is important when you fill out the terminals lists of the project. — Read more about terminals lists in “Creating a Terminals list” on page 739.

— See “Multi layer terminals” on page 208.

— See “Multiplacing symbols” on page 163.

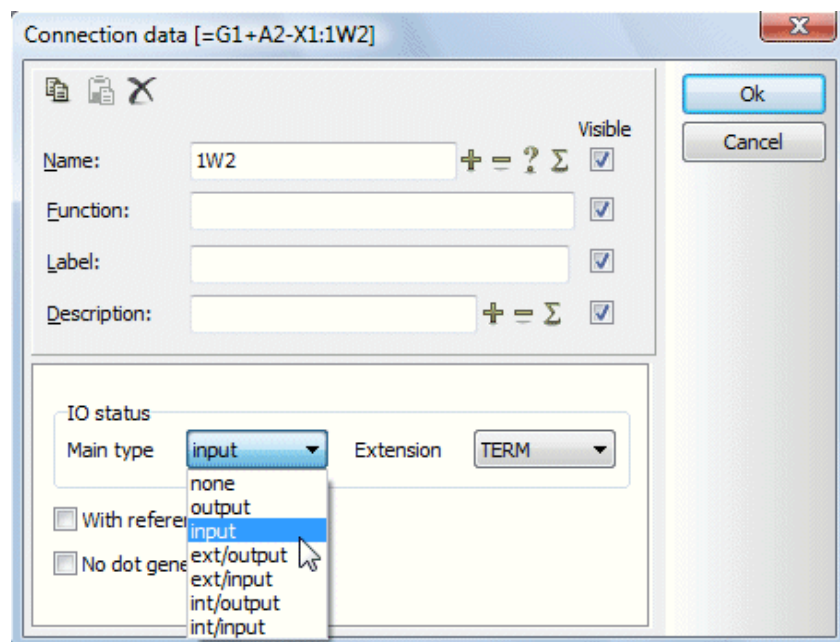
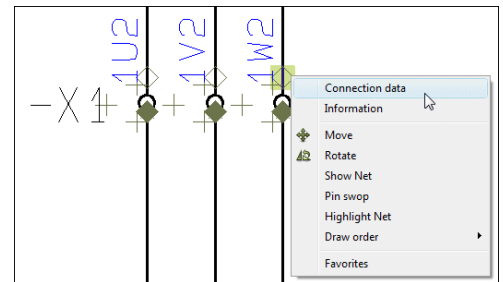
Input and Output on terminal symbols

On the terminal symbol itself, the *output* connection point is solid red.

If you want to change whether a connection point is *input* or *output*, you click the **Symbols** button, right-click on the connection point, and choose **Connection data**.



Clicking on the down-arrow in the **Main type** field in the **Connection data** dialog box, enables you to choose whether the connection point is of the type *input* or *output* — or *ext/output*, *ext/input*, *int/output* or *int/input* for terminals with additional status **PLC**, which you can read more about in “Creating terminal symbols with an additional status” on page 699, “Terminal symbols with additional status PLC” on page 700 in particular.



For terminals **Extension** is set to **TERM** — or to **TERM/PLC**, see “Creating terminal symbols with an additional status” on page 699, “Terminal symbols with additional status PLC” on page 700 in particular.

If you accidentally select the entire symbol instead of the connection point, then right-click on the connection point once more.

Changing connection names for terminal rows

See “Renumbering pinnumbers for a terminal row using Select name” on page 229.

Designing terminal symbols

See how to design terminal symbols in “Creating terminal symbols” on page 697.

MULTI LAYER TERMINALS

THIS CHAPTER DESCRIBES	
— ON MULTI LAYER TERMINALS	(P. 208)
— MULTI LAYER TERMINALS ON DIAGRAM PAGES.....	(P. 208)
— MULTI LAYER TERMINALS ON MECHANICAL PAGES	(P. 209)

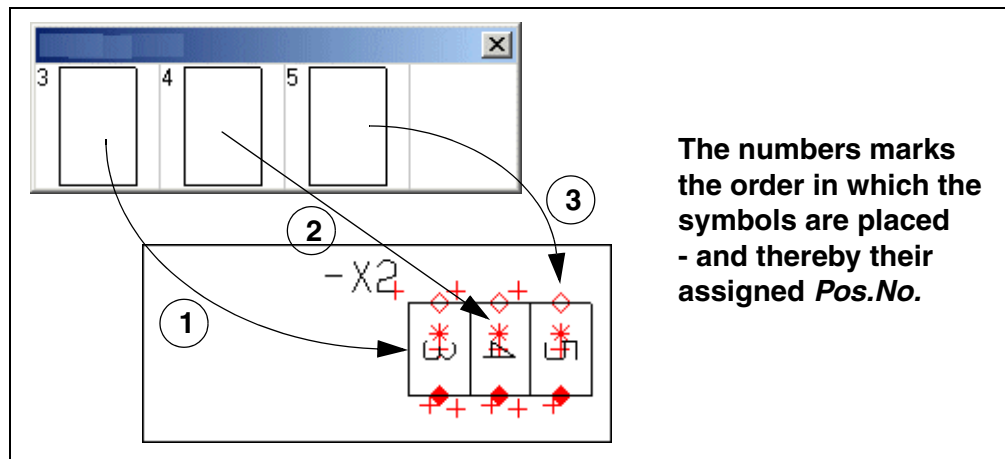
On multi layer terminals

In order to handle multi layer terminals, an extra column have been added in the component grouping named **Pos.no.**, that specifies which layer on the terminal the terminal symbol refers to.

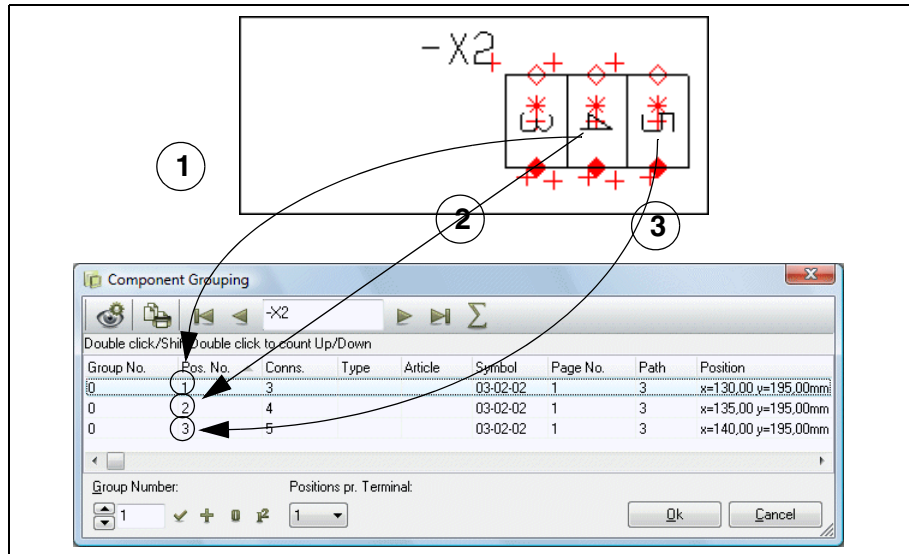
— See “Component Grouping” on page 546.

Multi layer terminals on diagram pages

When a multi layer terminal with more than one terminal symbol is placed via the database, the first placed symbol from the database pickmenu is assigned **Pos.No.** 1. The next placed is assigned number 2 etc.

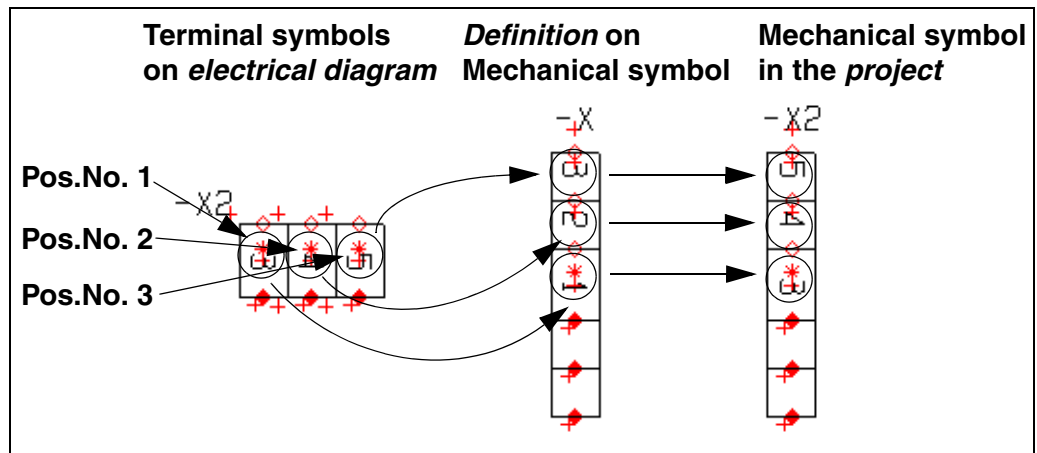


The position numbers can be specified/changed in **Functions => Component Grouping**.



Multi layer terminals on Mechanical pages

The connection point having the connection number **1** on the mechanical symbol, is assigned the connection number from the connection point having **Pos.No. 1** on the symbol on the diagram.



WORKING WITH CABLE SYMBOLS

This section deals with how to apply cable symbols on your diagrams.

— See “Creating cable symbols” on page 701 and “Creating busbar symbols” on page 703.

The database in **Automation** can contain information about cable wire names. If you are using the database, the names of the cable wires can therefore be supplied automatically.

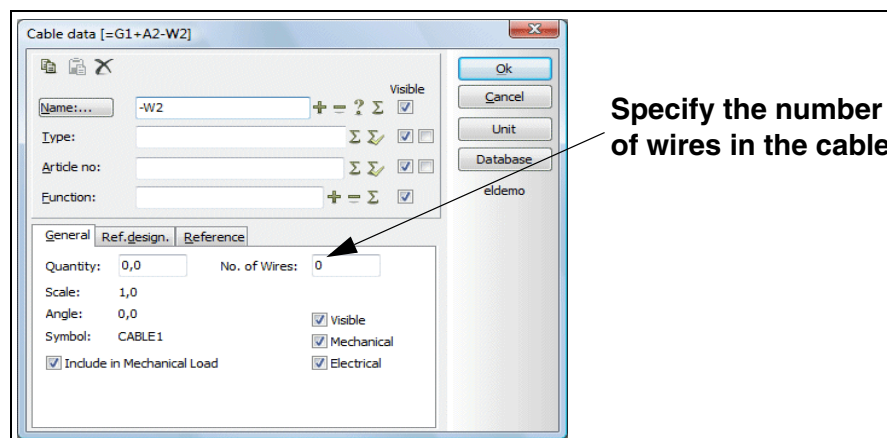
When you work with cables you enter different dialog boxes when selecting the cable wires, depending on whether the system has cable wire names for the cable or not.

You can find cable symbols in the pickmenu, and by opening the folder **MISC** from the **Symbol menu**. In the database, you will find cables in the folder **Flexible cables**.

THIS CHAPTER DESCRIBES	
— PLACING CABLE SYMBOLS WITHOUT WIRE NAMES FROM THE DATABASE	(P. 210)
— PLACING CABLE SYMBOLS WITH WIRE NAMES FROM THE DATABASE	(P. 213)
— PLACING SYMBOLS FOR THE SAME CABLE IN ONE OPERATION	(P. 214)
— MOVING CONNECTION POINTS FOR CABLE WIRES	(P. 214)
— ADDING CABLE WIRES TO PLACED CABLES	(P. 215)
— REMOVING CABLE WIRES FROM PLACED CABLES	(P. 215)
— ADDING A SCREEN ON A PLACED CABLE	(P. 216)
— REVERSE DIRECTION ON PLACED CABLES	(P. 216)
— UNREFERENCED CABLE SYMBOLS	(P. 216)
— SETTINGS FOR DEFAULT CABLE DIRECTION	(P. 217)

Placing cable symbols without Cable wire names from the Database

When you have a cable symbol in the cross hairs, you place it next to some of the wires you want to include into the cable. You then enter the **Cable data** dialog box:



Number of cable wires in the cable

In the field **No. of Wires**, you can specify how many wires that exists in this cable. When this amount of wires have been placed, it will not be possible to place any more wires for this cable. — If the cable is fetched via the database, the field **No. of Wires** is filled out automatically.



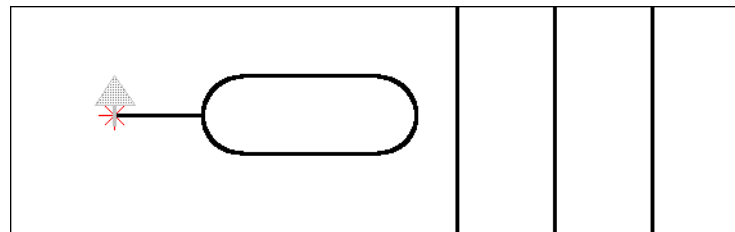
NUMBER OF WIRES IN COMPONENTS LISTS

WHEN CREATING COMPONENTS LISTS, IT IS POSSIBLE TO INSERT INFORMATION ABOUT THE NUMBER OF WIRES AND USED WIRES FOR THE CABLES IN THE LIST VIA THE DATAFIELD **CABLE WIRES**.

Filling out the dialog box Cable data

Apart from this, you can fill out the dialog box **Cable data** as described in “Placing and Naming symbols” on page 151. Click **OK**.

The illustration below is a part of a larger drawing, where the three vertical lines are wires.



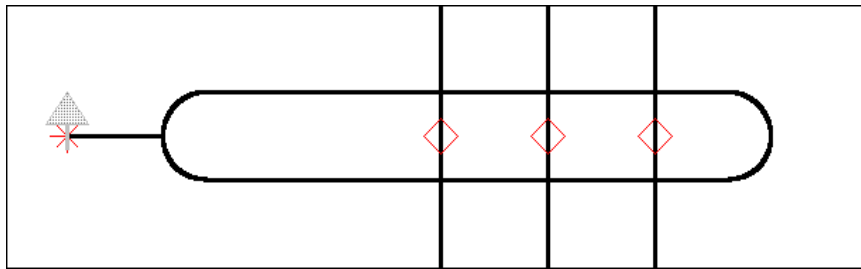
When you place the cable, a rubber band now starts in the reference point of the symbol, and ends up in the cross hairs. This indicates that you can now point out which wires to include in the cable.

You have two options: Pointing out the wires individually, or selecting more than one wire at the time:

Pointing out individual cable wires

Click on the wires you want to include one at a time. Each time you click on a wire, you are asked to type in **Connection data**.

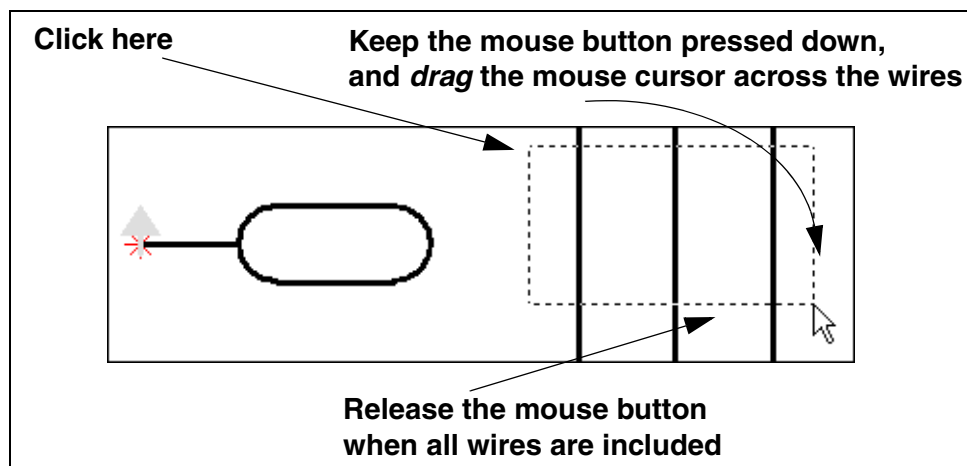
When you click **OK** the symbol expands to include this wire as well.



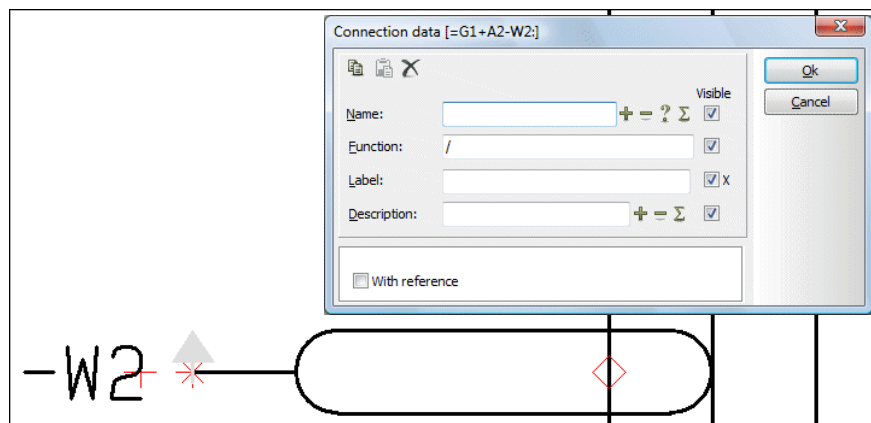
When you do not wish to include more wires into the cable you press <Esc>, and the "rubber band" disappears. If the number of cable wires is specified in the dialog box **Cable data**, the "rubber band" disappears when the specified number of cable wires have been placed in the project.


Pointing out more than one cable wire in a single operation

When you have placed the cable symbol — and a thin line (the "rubber band") starts in the reference point of the symbol, and ends up in the cross hairs — you can mark up a window over the wires, which you wish to include in the cable:



When you release the mouse button, you enter the dialog box **Connection data**, from where you can name cable wires — for instance by using automatic counting:



 If for instance you write *1* in the **Name** field, press down <Ctrl> while clicking on **+**, and then click **OK**, the three wires in the example will get the names **1**, **2** and **3** respectively. — See e.g. “Automatic naming of symbols” on page 171.

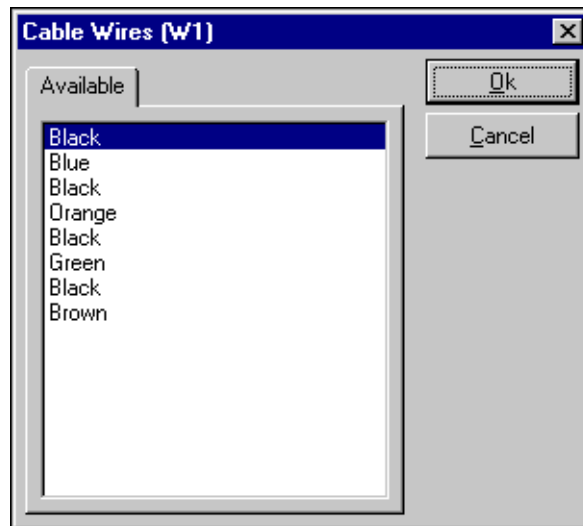
When you are using a cable with specified wire names from the database, the selected wires are automatically assigned wire names according to the order in which the wires appear in the database.

Place cables with Cable wire names from the database

When you use the database, you can let the database supply the cable wire names automatically.

The procedure for selecting the cable symbol, and pointing out the cable wires is the same as described in “Placing cable symbols without Cable wire names from the Database” on page 210.

Only, when you select a cable wire, you enter this dialog box instead:



Here you choose which of the cable conductors you wish to apply, and click **OK**.

Each time you enter this dialog box, you can see which cable wire names are **Available** and which are **Used**.

If you have changed a cable wire name to a name unknown to the database, you also see the tab **Unknown**.

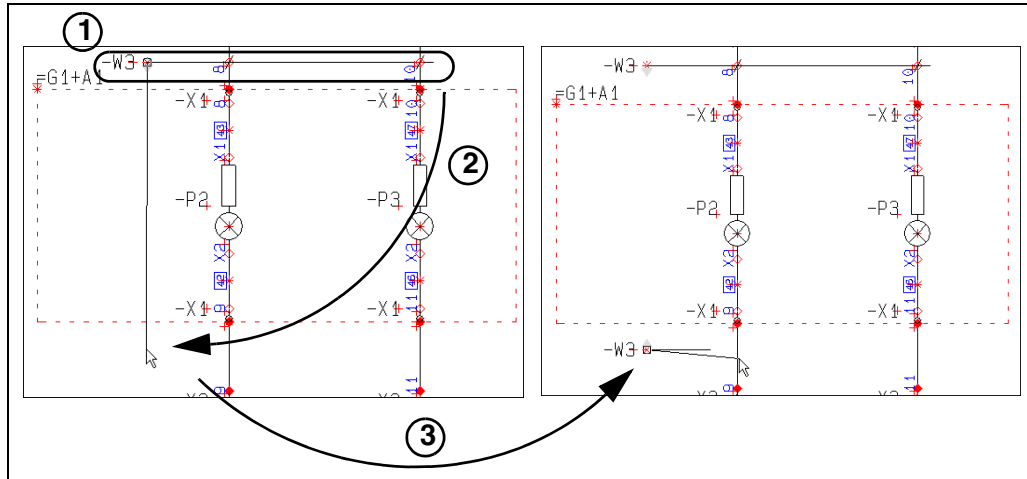
— Please note that it is also possible to choose cable wires from the tab **Used**. This makes it possible to display the cable wire more than one place in the project.

Placing symbols for the same cable in a single operation

You can place more than one symbol for the same cable at different locations in the diagrams, in a single operation. When these symbols have the same cable name, the program knows they represent the same cable.

Select the cable symbol only once

When you place a cable symbol and include some cable wires in the cable, and then click exactly over, or exactly under, the cable symbol, a new cable symbol with the same cable name is placed in the diagram:



To do as in the figure above, do the following:

- 1) Place the cable symbol **-W3** and include the cable wires in the cable
- 2) Do *not* press <Esc> to stop including cable wires, but click vertically under the **-W3** cable symbol
- 3) A new cable symbol with the name **-W3** is placed in the diagram

FREE PLACING OF THE SUBSEQUENT CABLE SYMBOL

When you need to place the other cable symbol for the same cable anywhere in the diagram, do the following:

- 1) Place the cable symbol **-W3** and include the cable wires in the cable
- 2) Do *not* press <Esc> to stop including cable wires, but press <Ctrl>, and click where you wish to place the next cable symbol
- 3) A new cable symbol with the name **-W3** is now placed where you clicked

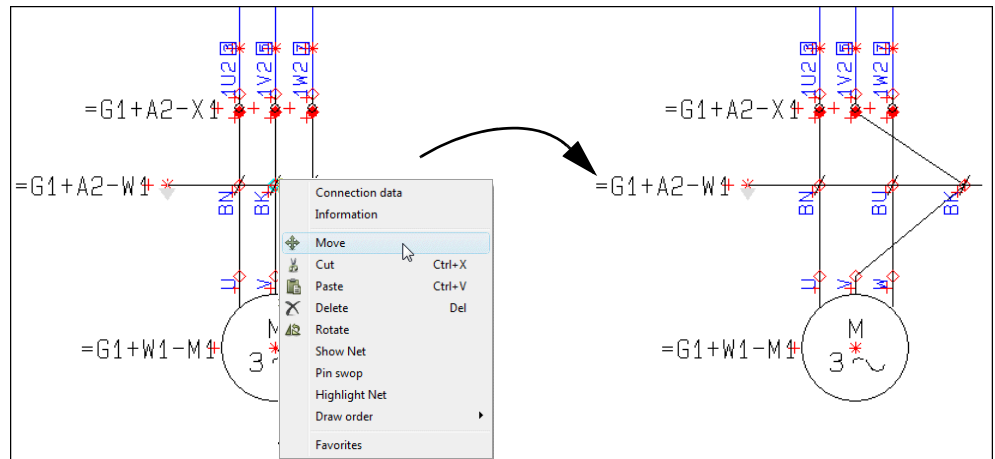
Moving connection points for cable wires

It is possible to move the connection points for a cable in relation to the cable symbol. The connection points can only be moved along the axis of the symbol, and can not be moved to the opposite side of the reference point of the symbol. To move a connection point, do the following:



- 1) Make sure that the **Symbols** button is activated, right-click on the connection point, and select **Move**

- 2) You now get the connection point in the cross hairs: Click where you wish to place the connection point.



Moving connection points for cable wires in areas

If you select the connection points in an area — by dragging a window over them and pressing <Ctrl> when releasing the mouse button — you can also move the selected connection points for the cable in the area.

Adding wires to cables



If you want to add another wire to the cable after having deselected the cable symbol, first make sure the **Symbols** button is activated, then right-click on the cable symbol, and choose **Add cable core**. A rubber band now radiates from the reference point and ends in the cross hairs.

Now click on the wire(s) that are to be included in the cable, and each time you click on a wire, the symbol expands to include this wire as well.

If you are using a cable with color codes from the database, you will be asked to select color codes for each individual wire, as described in “Place cables with Cable wire names from the database” on page 213.

You can also select more than one wire in one operation, as described in “Pointing out more than one cable wire in a single operation” on page 212.

Press <Esc> when you have finished.

Removing a wire from a cable



If you later on want to remove a core from the cable, you make sure the **Symbols** button is activated, right-click on the connection point for the wire you want to exclude, and choose **Delete**.

Adding a screen to a placed cable symbol



THE CABLE SYMBOL MUST HAVE A SCREEN CONNECTION POINT

TO USE THE FUNCTION **ADD CABLE SCREEN**, THE APPLIED CABLE SYMBOL MUST HAVE A CONNECTION POINT FOR A CABLE SCREEN IN ITS SYMBOL DEFINITION.

— SEE “CONNECTION POINT FOR CABLE SCREEN” ON PAGE 701.

To add a cable screen to a placed cable symbol, do this:



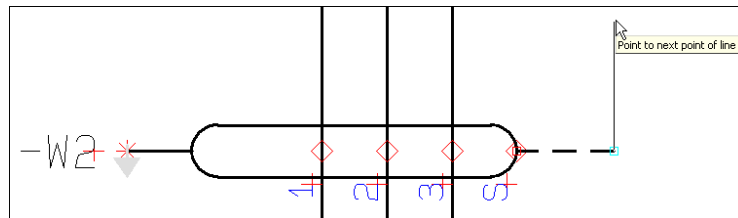
1) Click the **Symbol** button — to work with symbols.

2) Right-click on the cable symbol, and choose **Add Cable screen**.

3) In the dialog box **Connection data**: Type in a name for the cable screen, and click **OK**.



4) A connection point for the cable screen is now inserted on the cable symbol, the program changes to the **Lines** command automatically, and initiates the drawing of a line starting from the connection point for the cable screen:



5) Change line type if necessary, and connect the line to the desired symbol.



LINE SETTINGS MEMORIZED FOR CABLE SCREENS

THE PROGRAM AUTOMATICALLY USE THE LINE SETTINGS, THAT YOU USED THE LAST TIME YOU CONNECTED A CABLE SCREEN.

AFTER CONNECTING THE CABLE SCREEN, THE PROGRAM AUTOMATICALLY SWITCH BACK TO THE LINE SETTINGS, THAT YOU USED BEFORE CONNECTING THE CABLE SCREEN.

— READ MORE ABOUT LINE SETTINGS IN “THE LINES TOOLBAR” ON PAGE 66.

Reversing the cable direction

To reverse the direction of the cable, right-click on the reference point of the cable, and choose **Reverse direction**.

The direction is of importance when the cable lists are filled out.

Unreferenced cable symbols

When you create a project by using sub drawings or sub projects, where one end of a cable is displayed on one project page, and the other end is displayed on

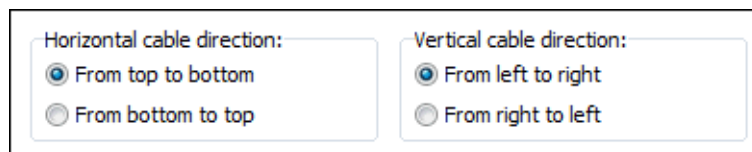
another project page, it may occur that one end of a cable is not used in a project. If **With reference** is activated for the cable connection points, these will be unreferenced.

These cables can be set to be invisible by choosing **Functions => Special Functions => Set unreferenced cable symbols invisible**.

— See also “Unreferenced signal symbols” on page 203.

Default Cable direction

In **Settings => Text/Symbol default** you can specify the cable direction as an over-all setting, vertically as well as horizontally.



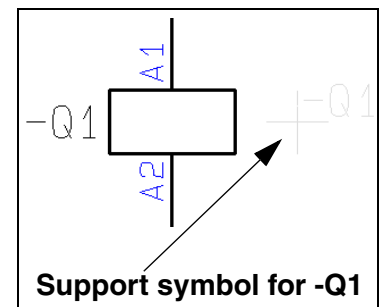
USING ACCESSORY SYMBOLS

To specify that a symbol needs an accessory, do this:



1) Make sure the **Symbols** button is activated, right-click on the symbol, and choose **Place accessory** (or choose **Functions => Place Accessory**).

2) You now get the accessory symbol in the cross hairs: Click where you wish to place the accessory symbol — e.g. next to the symbol.



3) In the dialog box **Component data**: The accessory symbol is automatically assigned the name of the selected symbol (here **-Q1**). Type in the article number for the accessory, or click on **Database**, to select the accessory component in the database.

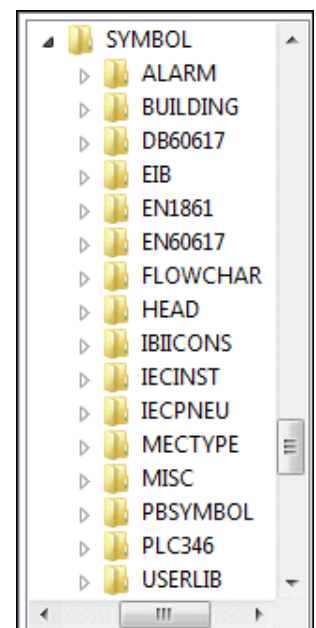
Data from accessory symbols are included in the lists, but the accessory symbol itself is not included when printing the diagram pages.

SYMBOL FOLDERS

The symbols in **Automation** are located in various folders, depending on their use.

When you wish to fetch a symbol, it is a good idea to know in which folder the symbol is located.

Typically, the symbols will be located in folders under the folder **c:\Pcscad\symbols**. Here is a survey of the contents of these folders:



FOLDER	CONTENTS
ALARM	SYMBOLS FOR ALARM SYSTEMS. NOT STANDARD.
BUILDING	SYMBOLS FOR GROUND PLAN DRAWINGS: DOORS, WINDOWS, WHITE GOODS ETC.
DB60617	SYMBOLS CREATED BECAUSE OF THE DATABASES FOR THE PROGRAM. THE SYMBOLS FOLLOWS THE STANDARD EN 60617.
DEMOMECH	EXCLUSIVELY MECHANICAL SYMBOLS FOR SWITCHBOARD DRAWINGS. AT PRESENT THE FOLDER ONLY CONTAINS EXAMPLES USED IN "AUTOMATION PROJECT TUTORIAL" IN THE <i>PCschematic Tutorial</i> MANUAL.
EIB	SYMBOLS FOR INTELLIGENT BUILDING INSTALLATION (EIB).
EN1861	SYMBOLS ACCORDING TO EN 1861, REFRIGERATING SYSTEMS AND HEAT PUMPS - SYSTEM FLOW DIAGRAMS AND PIPING AND INSTRUMENT DIAGRAMS - LAYOUT AND SYMBOLS.
EN60617	<p>THE FOLDER CONTAINS SYMBOLS ACCORDING TO THE EUROPEAN STANDARD IEC/EN 60617, GRAPHICAL SYMBOLS FOR DIAGRAMS. THE SYMBOLS FOLLOWS EN 81714-2, CONSTRUCTION OF GRAPHICAL SYMBOLS. THE SYMBOL NAMES FOLLOWS THE EN 61346 STANDARD FOR LETTER CODES.</p> <p>PC SCHEMATIC HAS A LICENSE FOR USING THE IEC STANDARDS. THE SYMBOLS ARE SAVED UNDER THEIR SERIAL NUMBER, FOR INSTANCE 04-02-14.</p> <p>IF A SYMBOL IN THE IEC STANDARD IS ONLY SHOWN AS A MAKE-FUNCTION (N/O, NORMALLY OPENED SYMBOL), AN EVENTUAL CORRESPONDING BREAK-FUNCTION (N/C, NORMALLY CLOSED SYMBOL) WILL THEN HAVE A B INSTEAD OF THE LAST HYPHEN - AND AN S IF THE OPPOSITE IS THE CASE.</p> <p>POWER CONTROL CIRCUIT SYMBOLS, THAT ARE NOT COVERED BY THE IEC STANDARDS, HAVE BEEN ASSIGNED A NAME STARTING WITH H FOLLOWED BY AN IEC SERIAL NUMBER, WHERE 0 IS EXCLUDED. THE FIRST HYPHEN HAVE BEEN REPLACED BY THE NUMBER OF CONTACTS. FOR INSTANCE: H7303-04. AT THE SAME TIME A NUMBER OF SYMBOLS NOT SHOWN IN IEC HAVE BEEN CREATED, FOLLOWING THE STANDARD.</p> <p>THE SYMBOLS ARE DESIGNED WITHOUT A MINUS IN FRONT OF THE SYMBOL NAMES. THIS IS BECAUSE THE PROGRAM AUTOMATICALLY INSERTS A MINUS IN FRONT OF THE NAMES OF THE SYMBOLS YOU PLACE IN THE PROJECTS, IF YOU CHOOSE SETTINGS => POINTER/SCREEN AND SELECT INSERT - IN FRONT OF SYMBOL NAMES — SEE "POINTER/SCREEN" ON PAGE 815.</p> <p>IF YOU CHOOSE FUNCTIONS => SPECIAL FUNCTIONS => PREFIX ALL SYMBOL NAMES WITH -, A MINUS IS INSERTED IN FRONT OF ALL SYMBOL NAMES IN THE SYMBOLS IN THE PROJECT.</p>
FLOWCHAR	SYMBOLS FOR (COMPUTER) FLOWCHARTS.
HEAD	DRAWING HEADERS, FORMULAS FOR PARTS LISTS AND TABLES OF CONTENTS AND UNIT TABLES FOR UNIT DRAWINGS.
IBIICONS	IBI LIBRARY, DESIGNED BY COWI A/S.
IECINST	SYMBOLS FOR ELECTRICAL INSTALLATIONS, FROM EN 60617, GRAPHICAL SYMBOLS FOR DIAGRAMS - PART 11: ARCHITECTURAL AND TOPOGRAPHICAL INSTALLATION PLANS AND DIAGRAMS.
IECPNEU	SYMBOLS FOR PNEUMATICS AND HYDRAULICS, FROM EN 60617, FLUID POWER SYSTEMS AND COMPONENTS - GRAPHIC SYMBOLS AND CIRCUIT DIAGRAMS ACC. TO ISO 1219 AND BASIC SYMBOLS ACC. TO EN 60617.
MISC	SPECIAL SYMBOLS, WHICH ARE NOT OFFICIAL STANDARD, BUT ARE INCLUDED IN Automation : SIGNAL SYMBOLS, CABLE SYMBOLS, LOGOS, PLC REFERENCE SYMBOLS ETC.

FOLDER	CONTENTS
<i>PLC346</i>	PLC SYMBOLS WITH LETTER CODES ACCORDING TO EN 61346. THE PLC SYMBOLS ARE CREATED IN COOPERATION WITH A BOARD OF PLC VENDORS. NOT STANDARD.
<i>USER</i>	FOLDER FOR YOUR OWN SYMBOLS.

FUNCTIONS ON COMPONENT LEVEL

When you are working with symbols in **Automation**, the individual symbol will frequently just be one symbol among other symbols for the functions in a component. Therefore, there are both functions for working on symbol level and component level.

THIS CHAPTER DESCRIBES	
— THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS	(P. 221)
— FETCHING SYMBOLS FOR A COMPONENT VIA THE DATABASE	(P. 222)
— UPDATING FROM THE DATABASE WHEN CHANGING ARTICLE NO.	(P. 223)
— THE FUNCTION SHOW AVAILABLE	(P. 224)
— THE FUNCTION SHOW AVAILABLE WINDOW	(P. 226)
— FINDING SYMBOLS BASED ON THE SYMBOL NAME	(P. 227)
— SELECT COMPONENT	(P. 231)
— PLACE AVAILABLE SYMBOLS FOR A COMPONENT	(P. 231)

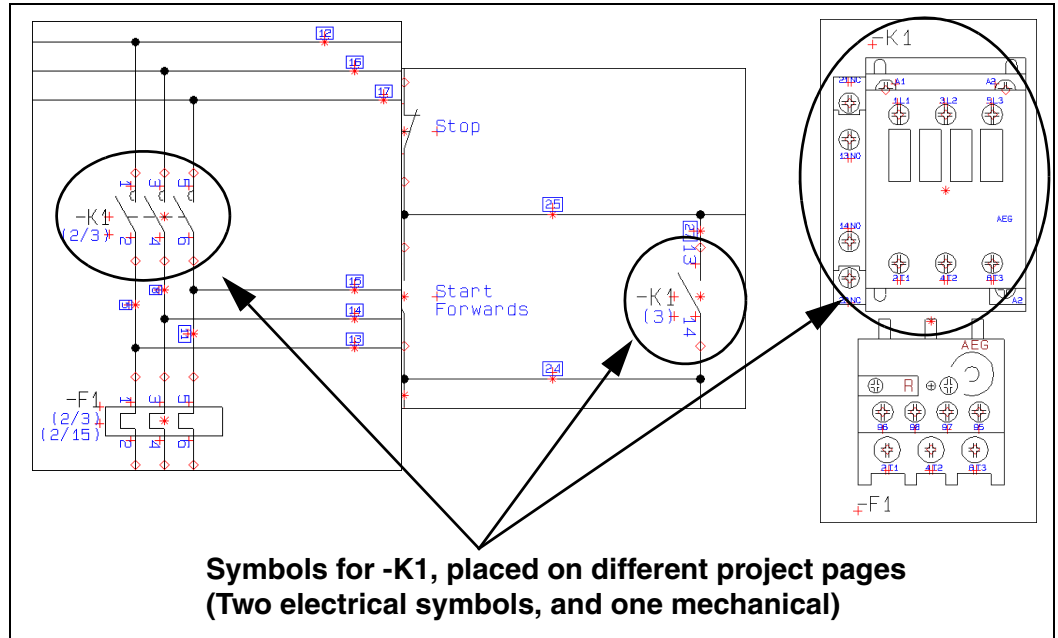
- For other ways to get a survey of the symbols in the project, please refer to “The Object Lister” on page 296.
- See also “Fetching symbols” on page 135 and “Placing and Naming symbols” on page 151.

THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS

Components containing more than one function, have an electrical symbol for each function in the component.

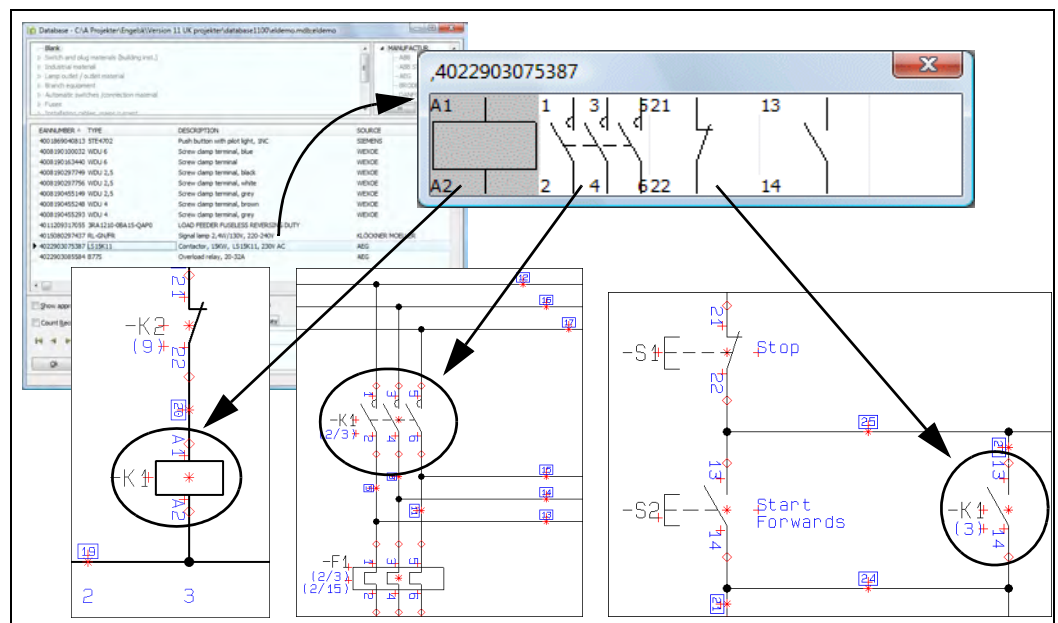
These electrical symbols can be placed freely in the diagram pages in the project.

To specify that two symbols belongs to the same component, they must have the same symbol name.



When you type in the same symbol name for two electrical symbols in a project, this specifies that they are part of the same component — please also see “Function assemblies” on page 222.

When you choose a component via the database, you automatically get its electrical symbols on the screen, and can place these in the diagrams. — See “Fetching symbols for a Component from the Database” on page 222.



— See “Update from Database when changing symbol Article no” on page 223 also.

Function assemblies

When you are working with advanced contactors, the symbols for **-K1** can be an assembly of several components.

The program knows that two — or more — symbols belong to the same component when they have the same name *and* article number.

This also applies for terminal rows and other function assemblies.

— See “Sub names on symbols” on page 160 also.

Symbols, lists and databases

When you have attached article numbers to the symbols on the diagrams, the program can update lists automatically by fetching the necessary information from the database.

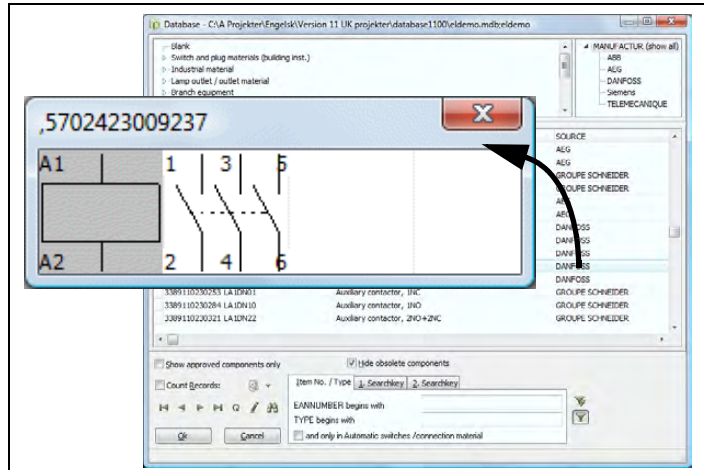
— Read more about this in “Using the database” on page 561.

Practically all components in the component vendor databases for **Automation** also have an attached mechanical symbol, which can be used for mechanical layouts. — Read more about this in “Mechanical Load from the Database” on page 616.

FETCHING SYMBOLS FOR A COMPONENT FROM THE DATABASE

When you place the electrical symbols for a component using the database, you get a pick-menu for the electrical symbols available for the component. These symbols can then be placed in the project diagrams.

This is described in further detail in “Fetching symbols from the database” on page 147.



Placing symbols using the database, enables you to generate project lists automatically. You will find the relation between components, databases and automatic updating of lists described in further detail in “Using the database” on page 561.

UPDATE FROM DATABASE WHEN CHANGING SYMBOL ARTICLE NO

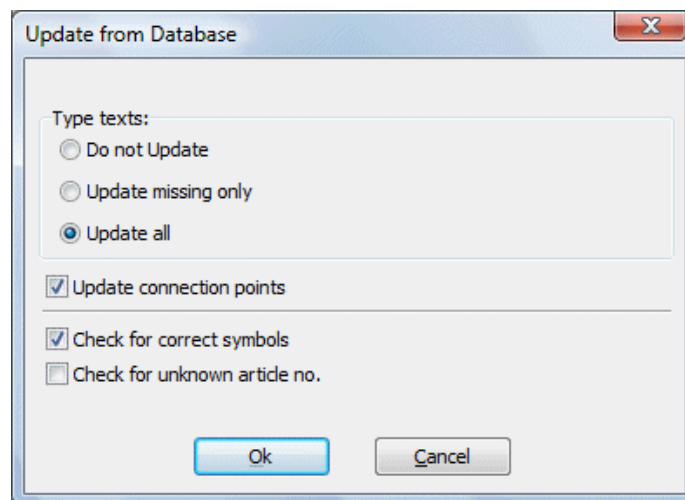
When the **Article no** is changed for one of the symbols for the component, it is possible to have the component updated from the database, as described in “Update from database when changing article number” on page 154.

Therefore, if you have changed **Article no** for one symbol for the component -K1, and subsequently select **Change Data for all symbols for the component** and click **OK** in the dialog box **Component data**, all symbols for the component -K1 are updated.

Settings for updating individual components via the Database

The settings for how to update via the database, are specified the following way:

- 1) Choose **Settings => Database**.
- 2) Select **Update components when changing Article No.**, and click on **Settings**.
- 3) You now enter the dialog box **Update from Database**:



- 4) Here you specify how to update the symbols for individual components when changing article number — see a description of the options in “Updating symbols from the database” on page 166.



UPDATING COMPONENT, PAGE OR PROJECT

IT IS ALSO POSSIBLE TO UPDATE COMPONENTS WITHOUT CHANGING ARTICLE NUMBER, AND TO UPDATE ALL COMPONENTS ON THE ACTIVE PAGE, OR IN THE ENTIRE PROJECT.

— SEE HOW TO DO THIS IN “UPDATING SYMBOLS FROM THE DATABASE” ON PAGE 166.

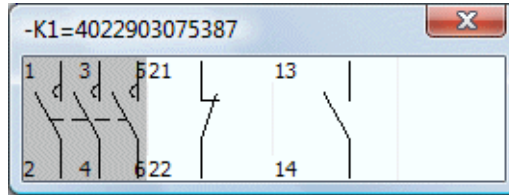
THE FUNCTION SHOW AVAILABLE

When you place a symbol via a pickmenu from the database, the pickmenu disappears.

To get the pickmenu for the component back on screen, right-click on one of the electrical symbols for the component — e.g. the one you just placed —, and choose **Show available**.

— See also “The function Show available window” on page 226.

You now see the electrical symbols for the component, that has still not been used in the project.



Also, you can click on a symbol and choose **Functions => Show available**, or press the short-cut key [F9].

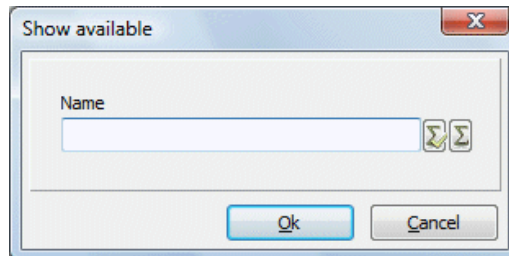
Show available again

When you choose **Functions => Show available again**, or press the short-cut key [Ctrl+F9], the pickmenu for the component is displayed again. This makes it easier to work with for instance PLC's and multi terminals.

Lists of Used names in the dialog box Show available



When you have selected **Symbols**, and choose **Functions => Show available**, you enter the following dialog box:



When you click on the **Used names** button you get a list of all the components that are used in the project.



When you click on the **Used names with available symbols** button, a list of all project components with available symbols is displayed.

Click on the component you want to use electrical symbols from, and you enter a pickmenu for the component. This way you can fetch symbols from components, that are placed on other pages in the project.



IF A SYMBOL CONTAINS **FUNCTION** OR **TYPE** INFORMATION, THIS INFORMATION IS TRANSFERRED TO THE OTHER SYMBOLS FOR THE COMPONENT WHEN YOU USE THE **SHOW AVAILABLE** FUNCTION.

IF THE DATABASE CONTAINS INFORMATION ABOUT **FUNCTION** OR **TYPE**, THIS INFORMATION IS ADDED TO THE PLACED SYMBOL IN STEAD.



BUTTON FOR SELECTED LIST FOR ACTUAL FUNCTION

WHEN A BUTTON WITH THE SAME APPEARANCE AS THE **USED NAMES WITH AVAILABLE SYMBOLS** BUTTON APPEARS IN A DIALOG BOX, IT CAN GIVE YOU A SELECTED LIST, BASED ON THE NEEDS IN THE ACTUAL SITUATION.

WHEN FOR INSTANCE YOU ARE ABOUT TO PLACE A BREAK CONTACT, THE PROGRAM DISPLAYS THE POSSIBLE NAMES FOR A BREAK CONTACT — SUCH AS FOR INSTANCE THE NAMES FOR THE APPLIED CONTACTORS AND PUSH-BUTTONS. WHEN FOR INSTANCE YOU ARE PLACING A TERMINAL, ONLY NAMES FOR THE APPLIED TERMINALS ARE DISPLAYED.

Wildcards/jokers in Show available

You can also use wildcards/jokers in the dialog box **Show available** — as described in “Wildcards/jokers” on page 540.

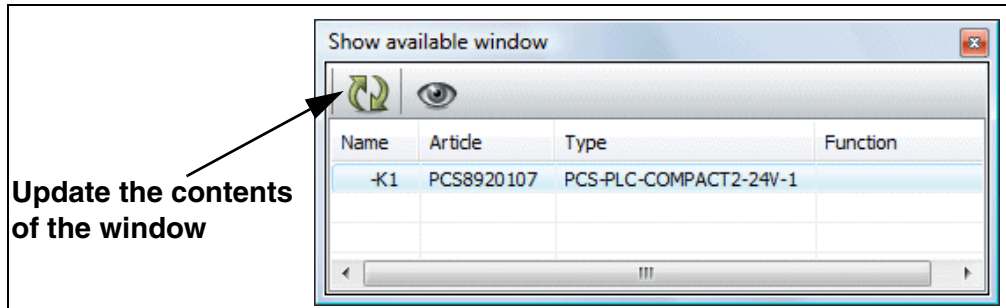
Show available for Mechanical symbols

When only a mechanical symbol is placed for a component, **Show available** can also be used for placing the electrical symbols for the component. Read more about this in “From Mechanical to Electrical drawing” on page 625.

SEE VIDEO ON OUR WEBSITE THE FUNCTION SHOW AVAILABLE WINDOW



The function **View => Show available window** displays all components in the project, that has available functions/symbols.

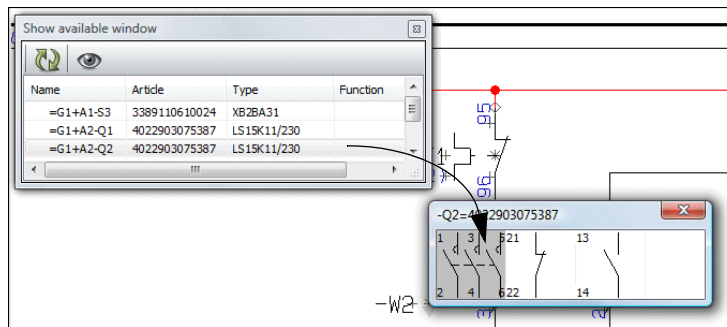


Update the contents of the window

You can also right-click on the page, and choose **Show available window**.

Pickmenus for components - via the Show available window

When you double-click on one of the components in the **Show available window**, you get a pickmenu for the component, containing symbols for all available functions in the component:



The **Show available window** remains on screen when you double-click on a component in the list, and you therefore continuously have easy access to all components with available functions/symbols in the project.

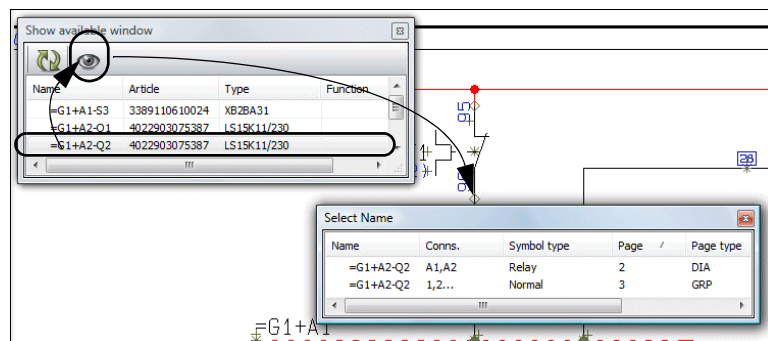


The window is closed by clicking the **Close** button in the dialog box.

Select name window for components - via the Show available window



If you click on a component in the window, and then click the **Show where the component is placed** button, the **Select name** window is displayed for the component.



— See “Find all symbols with identical names” on page 227.

FINDING SYMBOLS BASED ON THE COMPONENT NAME



THE SELECT NAME FUNCTION CAN BE USED FOR:

- FINDING SYMBOLS IN THE PROJECT QUICKLY
- EDITING COMPONENT DATA FOR MORE THAN ONE SYMBOL AT THE TIME

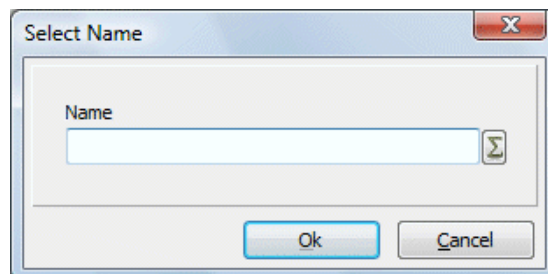
THIS CHAPTER DESCRIBES	
— FIND ALL SYMBOLS WITH IDENTICAL NAMES	(P. 227)
— SEARCHING FOR MORE COMPONENTS USING WILDCARDS	(P. 228)
— JUMP TO THE NEXT SYMBOL IN THE SELECT NAME LIST AUTOMATICALLY	(P. 228)
— CHANGING SYMBOL REF. DESIGNATIONS USING SELECT NAME.....	(P. 229)
— RENUMBER CONNECTION NAMES FOR TERMINAL ROW USING SELECT NAME(P.	229)

— See “Select Component” on page 231 also.

Find all symbols with identical names

When you want to get a survey of where the electrical symbols for the component are placed in the project, you choose **Functions => Select name**.

You then enter a dialog box where you specify the name of the component:



When you wish just to find the symbols for a specific component, type in the component name, and press <Enter>.



When you need a list of component names in the project, click the **Used names** button, select a name from the list of component names, and click **OK**.

When there is more than one symbol with the same name, the **Select name** window is displayed:

Name	Conns.	Symbol type	Page /	Page type	Position
=G1+A2-Q1	1,2...	Open	1	DIA	x=125,00 y=205,00mm
=G1+A2-Q1	A1,A2	Relay	2	DIA	x=87,50 y=100,00mm
=G1+A2-Q1	13,14	Open	2	DIA	x=157,50 y=202,50mm
=G1+A2-Q1	21,22	Close	2	DIA	x=227,50 y=122,50mm
=G1+A2-Q1	1,2...	Normal	3	GRP	x=251,00 y=374,00mm

Here you can see all the places the symbol name occurs in the project, as well as different information about the symbols; their connection names, symbol types, which project page they are placed on, the page type of this page, and their position on the page.

You can remove symbols from the window by selecting them and press <Delete>. (Please note that this does not remove the symbols from the project, but *only* from the **Select name** window). Read more about selecting in “Selecting objects in the Object Lister” on page 298.

The contents of the window can be sorted as described in “The sorting order of the data in the Object Lister” on page 298.

Searching after more components using Wildcards

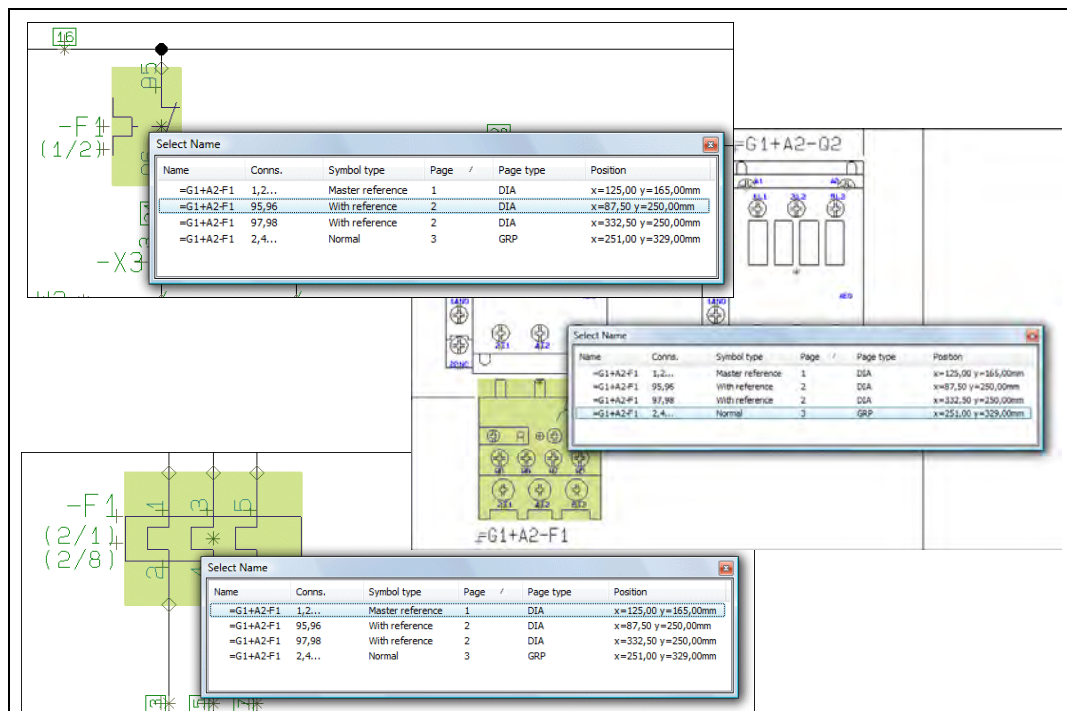
When you wish to find all components of a specific type, or do you not know all of the component name, you can use the so-called wildcards — as described in “Wildcards/jokers” on page 540.

When for instance you write * and click **OK** all project symbols are selected, as if you had pressed <Ctrl> and selected the symbols one at the time.

If you have followed the naming standard about - in symbol names, you can also write *-*, whereby all project symbols are selected — *except* wire number symbols and signals (when these do not contain a "-" in their symbol names).

Jump to a symbol from the Select name list

When you click on the different electrical symbols in the **Select name** box, you automatically jump to where the selected symbols are placed in the project — having the cross hairs pointing out the symbol.



You can also activate the function by right-clicking on a symbol, and then choose **Select Name**.

Changing the symbol reference designations using **Select name**

When for instance you wish to change reference designations, you can do the following:



- 1) Activate the **Symbols** button, right-click on the page and choose **Select Name**.
- 2) In the dialog box **Select Name**: Write for instance ***+A1*** (see “Searching after more components using Wildcards” on page 228) and click **OK**.

Name	Conns.	Symbol type	Page	Page type	Position
=G1+A1-X1	1	Terminal	2	DIA	x=87,50 y=160,00mm
=G1+A1-S2	13,14	Normal	2	DIA	x=87,50 y=177,50mm
=G1+A1-S1	21,22	Normal	2	DIA	x=87,50 y=202,50mm
=G1+A1-X1	3	Terminal	2	DIA	x=87,50 y=220,00mm
=G1+A1-X1	2	Terminal	2	DIA	x=105,00 y=220,00mm
=G1+A1-X1	5	Terminal	2	DIA	x=157,50 y=82,50mm
=G1+A1-P1	X1,X2	Normal	2	DIA	x=157,50 y=100,00mm
=G1+A1-X1	4	Terminal	2	DIA	x=157,50 y=125,00mm
=G1+A1-X1	7	Terminal	2	DIA	x=227,50 y=182,50mm



- 3) In the window **Select Name**: Remove the symbols from the list, which you do *not* wish to change the reference designations, and click on the **Component data** button in the toolbar for symbols.
- 4) In the dialog box **Component data**: Click on the tab **Ref. design.** and click on **Select**.
- 5) In the dialog box **Reference designations**: Select the new reference designation — for instance **+A2** —, and click **OK**.
- 6) In the dialog box **Component data**: Click **OK**.



- 7) In the window **Select name** the new reference designations are now displayed. Click the **Close** button in the upper right-hand corner to close the window.

Renumbering pinnumbers for a terminal row using **Select name**

Do you wish to renumber pinnumbers for a terminal row, so that the pinnumbers increase from the start to the end of the project, do the following:



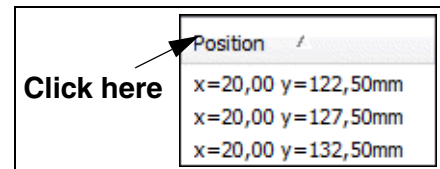
- 1) Click on the **Symbols** button, right-click on the page and choose **Select Name**.



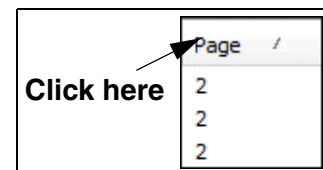
- 2) In the dialog box **Select Name**: Click on the **Used** button, select the terminal row (for instance =G1+A2-X3) and click **OK** — hereby all symbols for =G1+A2-X3 are selected in the project:

Name	Conns.	Symbol type	Page	Page type	Position
=G1+A2-X3	1	Terminal	2	DIA	x=87,50 y=147,50mm
=G1+A2-X3	3	Terminal	2	DIA	x=87,50 y=232,50mm
=G1+A2-X3	2	Terminal	2	DIA	x=105,00 y=232,50mm
=G1+A2-X3	5	Terminal	2	DIA	x=157,50 y=67,50mm
=G1+A2-X3	4	Terminal	2	DIA	x=157,50 y=137,50mm
=G1+A2-X3	7	Terminal	2	DIA	x=227,50 y=167,50mm
=G1+A2-X3	6	Terminal	2	DIA	x=227,50 y=235,00mm
=G1+A2-X3	9	Terminal	2	DIA	x=297,50 y=67,50mm
=G1+A2-X3	8	Terminal	2	DIA	x=297,50 y=137,50mm
=G1+A2-X3	11	Terminal	2	DIA	x=332,50 y=67,50mm
=G1+A2-X3	10	Terminal	2	DIA	x=332,50 y=137,50mm
=G1+A2-X3	1,1	Normal	3	GRP	x=196,00 y=199,00mm
=G1+A2-X3	2,2	Normal	3	GRP	x=202,00 y=199,00mm
=G1+A2-X3	3,3	Normal	3	GRP	x=208,00 y=199,00mm
=G1+A2-X3	4,4	Normal	3	GRP	x=214,00 y=199,00mm

- 3) In the **Select name** window: Click on the column header **Position** so that the displayed arrow in the field points upwards — hereby the symbols are sorted according to the order they are placed on the pages.



- 4) Click on the column header **Page** so that the displayed arrow in the field points upwards — hereby the symbols are sorted in page order, and according to the order they are placed on the project pages.



- 5) Click on the **Component data** button in the toolbar for symbols.



- 6) In the dialog box **Component data**: Assign names to the pinnumbers — write for instance -X3:1, press down <Ctrl> and click on **+**, and click **OK**. The pinnumbers are hereby incremented according to the order in which they are placed in the list (and hereby in the order in which they are placed in the project).

Name	Conns.	Symbol type	Page	Page type	Position
=G1+A2-X3	1	Terminal	2	DIA	x=87,50 y=147,50mm
=G1+A2-X3	2	Terminal	2	DIA	x=87,50 y=232,50mm
=G1+A2-X3	3	Terminal	2	DIA	x=105,00 y=232,50mm
=G1+A2-X3	4	Terminal	2	DIA	x=157,50 y=67,50mm
=G1+A2-X3	5	Terminal	2	DIA	x=157,50 y=137,50mm
=G1+A2-X3	6	Terminal	2	DIA	x=227,50 y=167,50mm
=G1+A2-X3	7	Terminal	2	DIA	x=227,50 y=235,00mm
=G1+A2-X3	8	Terminal	2	DIA	x=297,50 y=67,50mm
=G1+A2-X3	9	Terminal	2	DIA	x=297,50 y=137,50mm
=G1+A2-X3	10	Terminal	2	DIA	x=332,50 y=67,50mm
=G1+A2-X3	11	Terminal	2	DIA	x=332,50 y=137,50mm

Please note that existing holes in the numbering disappears at the same time.

Changing component data for the selected symbols

Changing component data from the **Select name** dialog box is done like this:

1) Select the symbols in the **Select name** dialog box.



2) Click on the **Object data** button — on the top of the screen —, to change article data for *all* of the selected symbols at the same time.

3) In the **Component data** dialog box: Click for instance on **Database** and select a new component in the database.

SELECT COMPONENT

If you right-click on a symbol and choose **Select Component**, you select to all symbols having the same name *and* the same article number as the selected symbol.

This is an option because a component can consist of several parts, each having their own article number.

— See “Finding symbols based on the Component Name” on page 227 also.

PLACE AVAILABLE SYMBOLS

If you want to place all symbols which have not been placed for a component, you choose **Functions => Place available Symbols**. You then enter this dialog box, where you click on **Load selected** and type in the symbol name.

Click **OK**, and the cross hairs changes appearance to a pencil. Click where you want the remaining symbols to be placed.

The symbols are now placed on top of each other.

In order to separate the symbols, drag a window around them, right-click within the selected area, and use the **Space** function — as described in “The Space function” on page 60.

If you choose **Load all** in the dialog box, all unused symbols for all components in the project will be placed.

If you choose **Load selected** and for instance write **-K***, all symbols starting with **-K** are placed. — Read more about this in “Wildcards/jokers” on page 540.

AREAS

In **Automation** you use the **Area** command when you want to select all symbols, lines, texts and circles from an area in a project.



To work with areas, start by clicking on the **Area** button, or by pressing the short-cut key [a].

THIS CHAPTER DESCRIBES	
— SELECTING OBJECTS IN AREAS	(P. 232)
— INTELLIGENT COPYING OF AREAS	(P. 233)
— ASSIGNING REFERENCE DESIGNATIONS WHEN COPYING AREAS	(P. 235)
— MOVING OR DELETING SELECTED AREAS	(P. 237)
— CUTTING AND PASTING AREAS	(P. 239)
— MOVING OR COPYING AREAS TO OTHER PROJECT PAGES	(P. 238)
— MAKING AREAS INVISIBLE	(P. 237)
— ROTATING AREAS	(P. 241)
— SCALING AREAS	(P. 241)
— COPYING AREAS TO SYMBOL DEFINITIONS	(P. 243)



INSERTING SUB DRAWINGS

PLEASE NOTE THAT YOU CAN ALSO INSERT SUB DRAWINGS — “SUB DRAWINGS” ON PAGE 244.

SELECTING ALL TYPES OF OBJECTS IN AN AREA



Click on the **Area** button. Mark up a window. All object types within the window will now be selected. If you want to exclude any objects, press the <Ctrl> key and click on the objects.

If you also want to select objects outside the area, press down <Ctrl> and click on the objects.

— Read more about selecting objects in “Selecting objects” on page 49.

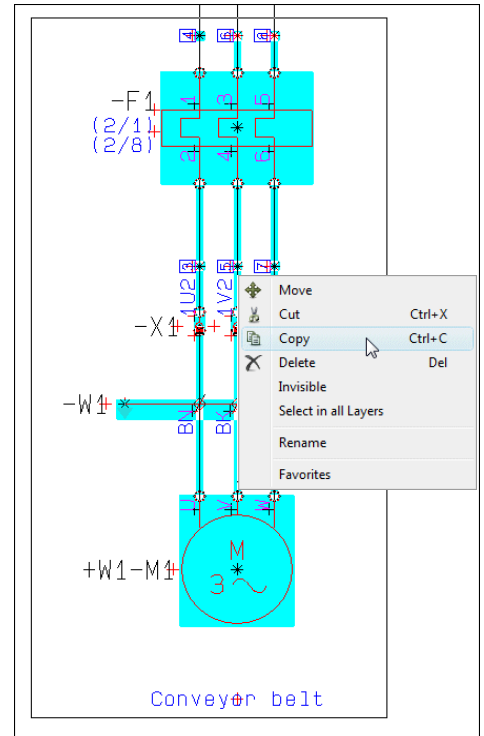
SEE VIDEO ON OUR WEBSITE **INTELLIGENT COPYING OF AREAS**



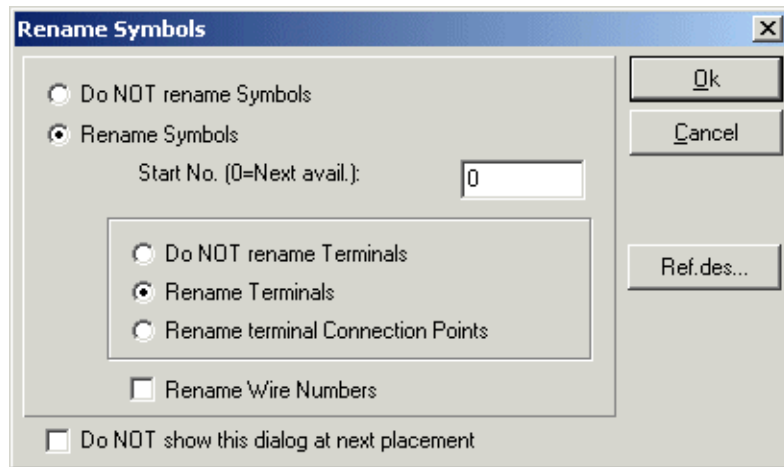
When you want to copy an area in **Automation** you click the **Area** button, and use the mouse to select the area you want to copy — see “Selecting all types of objects in an area” on page 232.

You can then either:

- a) Click on the **Copy** button on the upper part of the screen, and then click where you wish to place the copy
- b) Right-click inside the selected area and choose **Copy**, and click where you wish to place the copy
- c) Click inside the area, press down <Ctrl> and *drag* the copied area to where you wish to place the copy



When you place the copied area, you enter a dialog box, where you specify the directions for renaming the names in the copied area:



The options in this dialog box depends on which types of symbols exists in the copied area.

FULFILLING STANDARDS WHEN COPYING AREAS
<p>ESPECIALLY IN CONNECTION WITH NAMING OF TERMINALS, DETAILED RULES APPLIES FOR NAMING/NUMBERING. THUS MOTOR TERMINALS, SUPPLY TERMINALS AND INPUT- AND OUTPUT TERMINALS ARE NUMBERED ACCORDING TO DIFFERENT PRINCIPLES.</p> <p>WHEN YOU COPY AN AREA THE PROGRAM FIRST EXAMINES WHETHER THE AREA IS DRAWN ACCORDING TO THE RULES IN <i>EN60204-1, 14.2.1 CONDUCTOR AND CABLE RUNS</i>. IF THIS IS THE CASE, THE PROGRAM THEN ANALYSE WHICH TERMINAL TYPES IT CONTAINS, AND RENAMES THESE ACCORDING TO THE DIFFERENT PRINCIPLES.</p> <p>BECAUSE NO HOLES ARE ALLOWED IN THE NUMBERING, THE PROGRAM ALSO KEEPS TRACK OF WHICH NAMES/NUMBERS ARE USED FOR THE DIFFERENT TYPES OF TERMINALS.</p>

Renaming symbols when copying areas

If you want to renumber the symbols in the area, click on **Rename symbols**. If you want to renumber all symbols in the area, so that they start from a specific number, you specify this number in the field **Start No. (0=Next avail.)**.

When you only want to assign the next available symbol name to the symbols, just type 0 in this field. If for instance you write 20, the symbols are renumbered starting from 20.

When you do not want to renumber the symbols, click on **Do NOT rename Symbols**.

Please note that you can copy the area as many times as you want. Press <Esc> for removing the copied area from the cross hairs.

Page/Current path numbers in symbols in copied areas

If you use page/current path numbers in the names of the copied area, the names will adjust to the new page/current path numbers automatically.

Renaming terminals when copying areas

With respect to renumbering terminals, you can here choose between the following options:

OPTION	CONSEQUENCE
DO NOT RENAME TERMINALS	NEITHER THE TERMINALS NOR THEIR CONNECTION POINTS ARE RENUMBERED.
RENAME TERMINALS	ONLY THE TERMINAL NAMES ARE RENUMBERED, NOT THEIR CONNECTION POINTS.
RENAME TERMINAL CONNECTION POINTS	THE NAMES OF THE TERMINAL CONNECTION POINTS ARE RENUMBERED, THE TERMINAL NAMES ARE NOT CHANGED. — SEE “FULFILLING STANDARDS WHEN COPYING AREAS” ON PAGE 234.

Renaming wire numbers when copying areas

If the copied area contains wire numbers, you can click on **Rename Wire Numbers**. This assigns the next available wire numbers to the copied area. The wire numbers are assigned according to the principle you have used to apply for wire numbering — See “Automatic wire numbering” on page 377.

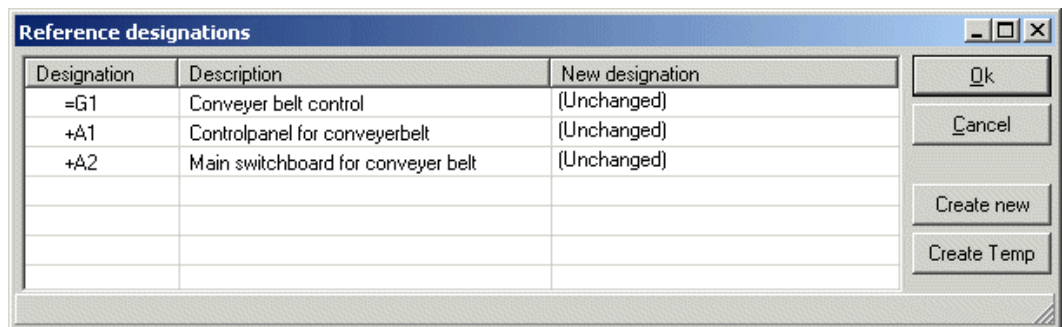
Copying an area to another page

If you want to move the copied area to another project page, see how to do this in “Moving or Copying areas to other pages” on page 238.

ASSIGNING REFERENCE DESIGNATIONS WHEN COPYING AREAS

If reference designations are used in the area you copy, the button **Ref.des.** is displayed in the dialog box **Rename symbols**. This gives you the option to assign other — or new — reference designations to the copied area.

When you click the **Ref.des.** button, you enter this dialog box:



In the columns **Designation** and **Description** the current reference designations are displayed along with their descriptions.

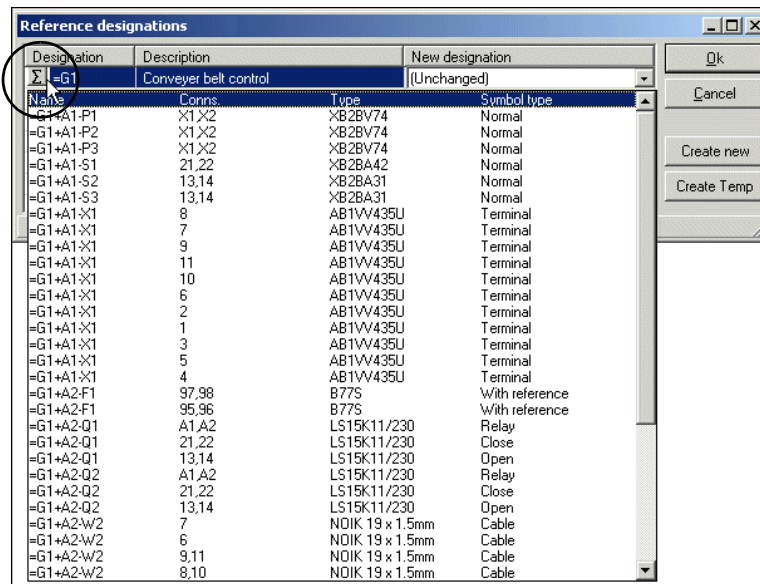


REF. DESIGNATIONS FOR SUB DRAWINGS, PAGES AND FULL PROJECTS

THIS CHAPTER DESCRIBES HOW TO EDIT REFERENCE DESIGNATIONS WHEN COPYING AREAS. YOU HAVE THE EXACT SAME OPTIONS WHEN COPYING FULL PAGES AND WHEN INSERTING STANDARD PROJECTS.

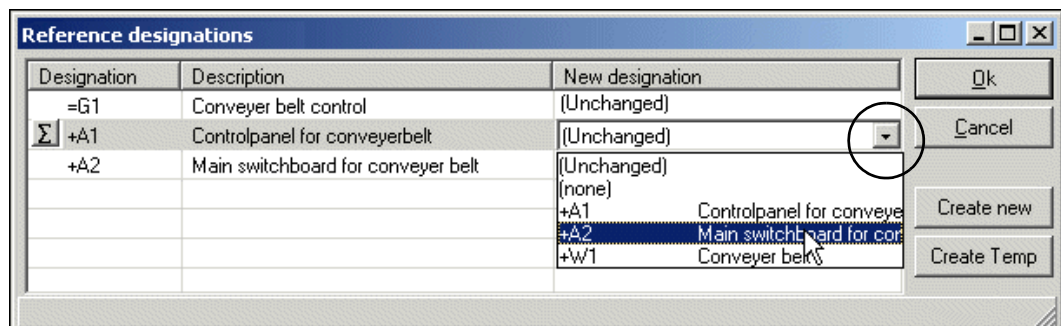
See symbols using the listed reference designations

Σ When you click on the line for one of the reference designations in the dialog box, a **Show symbols** button is displayed. When you click on this button, all symbols using this reference designation in the copied area are displayed.



Editing a reference designation

If you wish to edit a reference designation, first click on the line with the reference designation. Then click on the down-arrow in the column **New designation**, and you get the option of selecting a new reference designation:



By default (**Unchanged**) is selected, but you get the option of choosing by all applied reference designations in the project. You can also choose (**none**), to delete this reference designation from the symbols in the copied area.

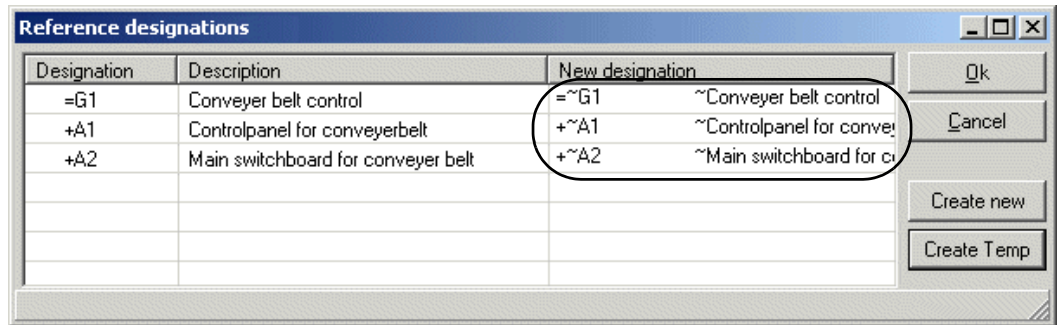
CREATING A NEW REFERENCE DESIGNATION

When you need to create a new reference designation, click the **Create new** button to the right in the dialog box, and you enter the dialog box **Reference designations**, from which you can create new reference designations. — See how to do this in “**Defining reference designations for the project**” on page 325, in particular “**Specifying function/location reference designations**” on page 326.

ASSIGNING TEMPORARY REFERENCE DESIGNATIONS

When you can not decide how to name reference designations in the copied area, you can create temporary reference designations, which you can use until you have found the correct names. You do this by clicking the **Create Temp** button in

the right-hand side in the dialog box. Hereby all reference designations in the dialog box are assigned a temporary reference designation:



These temporary reference designations are assigned a ~ sign after the reference designation prefix. This temporary reference designation is created from the initial reference designation, so that **=G1** becomes **=~G1**. If this temporary value is already used in the project, the temporary reference designation with the next available name using the same reference letter is applied. If **=~G1** is used, **=~G2** is then used in the project — if not used already. The description remains unchanged, with a preceding ~ character.

The temporary reference designations are deleted automatically, when they are no longer used in the project.

MOVING OR DELETING SELECTED AREAS

Moving and deleting of areas is done as described in “Working with selected objects” on page 51.

If you want to move an area to another page in the project, please see “Moving or Copying areas to other pages” on page 238.

MAKING AREAS INVISIBLE

— See “Making areas invisible” on page 63.

MOVING OR COPYING AREAS TO OTHER PAGES

When you have an area in the cross hairs — when moving or when copying — you can move the area to another page in the project.

This is done by clicking the page tab for the relevant page, which brings you to the new page, where you still carry the area in the cross hairs.

Click to place the area.

When you have a copied area in the cross hairs, you can change project page by pressing the [PageUp] and [PageDown] buttons, or by using the **Explorer** window.

— See “Copying an area from one project to another” on page 511 also.

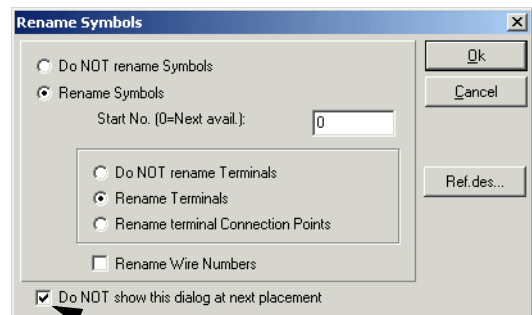
The dialog box Rename Symbols can be deactivated after first placement

When you have copied an area and insert it for the first time, you enter the dialog box **Rename Symbols**.

Here you specify how to rename the copied symbols.

When you select **Do NOT show this dialog at next placement**, these settings are memorized the next time the area is placed — and dialog box is therefore not displayed.

When placing a new copied area, the dialog box appears again.



Click here NOT to show the dialog box again

Renaming settings are memorized for copying Areas

The settings in the dialog box **Rename Symbols** are memorized by the program.

Component grouping when copying terminals

When copying terminals — individually or in areas and pages — **Automation** keeps the individual component grouping, so that the copied symbols are separated from the originals.

— See “Component Grouping” on page 546.

Copying an area to another project

When you have the area in the cross hairs, you can copy it to another project by:

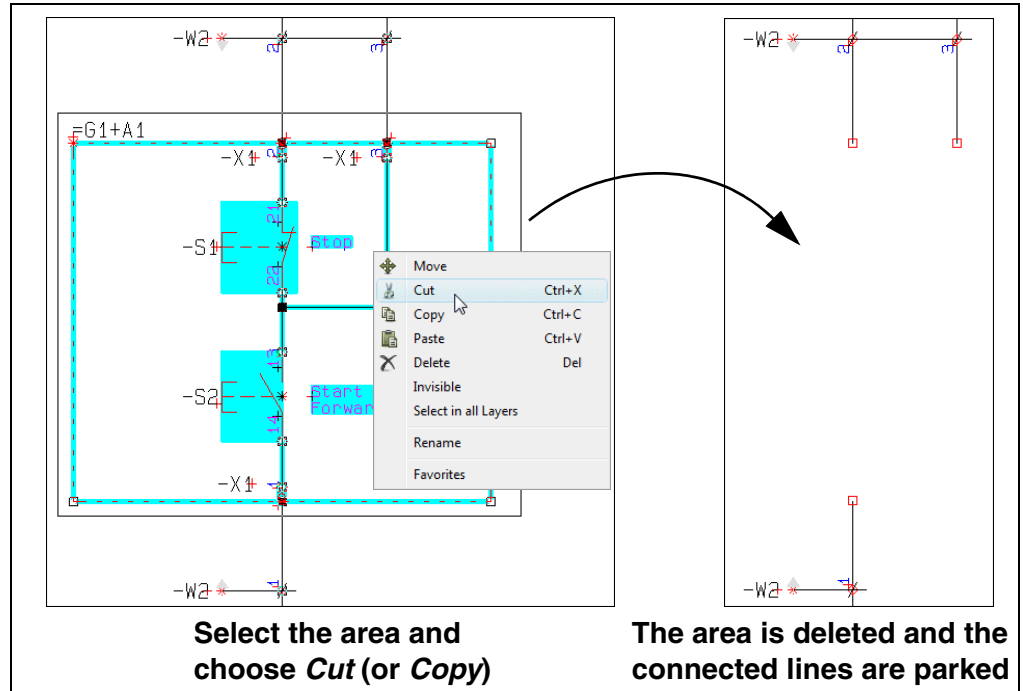
- a) Clicking on the other project in the **Explorer** window, and press [Ctrl+v].
- b) Following the description in “Copying an area from one project to another” on page 511.

CUTTING AND PASTING AREAS

You can cut and paste an area in the following way:

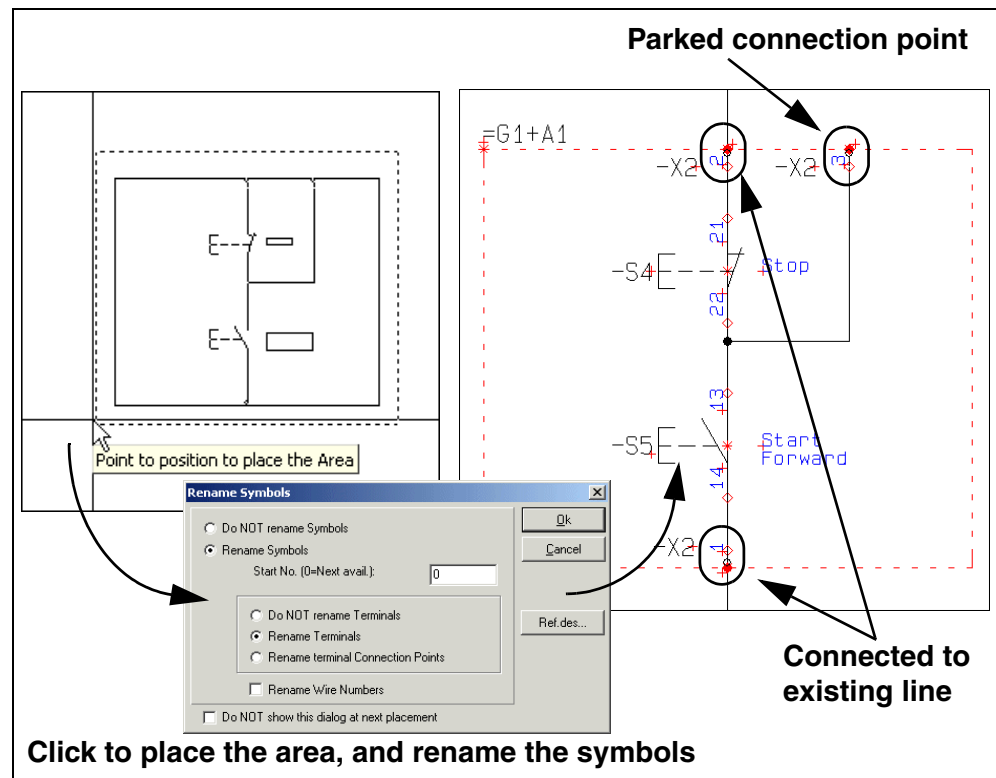


- 1) Click on the **Area** button, and select the area by dragging a window over it with the mouse.
- 2) Click on the **Cut** button, or right-click inside the area and choose **Cut**.



- 3) The area is deleted: Shift to another page — if necessary — and click the **Paste** button. You now get the area in the cross hairs.
- 4) Click where you wish to place the area, and you are asked how/whether to rename the symbols in the area, as described in “Intelligent copying of areas” on page 233. Select the appropriate options, and click **OK**. — See the figure below.

5) The area is now placed at its new position, with updated symbol names.



Lines are cut along the edges of areas when copying

When cutting a connected area, **Automation** cuts the lines connected to the area, along the edges of the selected area. Where the area is cut out, the connecting lines are parked.

When you place the area, the lines in the area are connected to other lines, if you place them on these. Otherwise, the lines are parked.

Only *lines* are cut. Symbols, texts and circles can not be cut in two this way.



USING THE CUT FUNCTION FOR OTHER COMMANDS

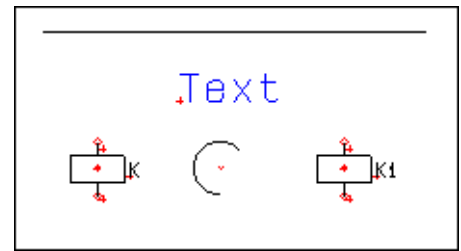
THE *CUT* FUNCTION CAN ALSO BE USED WHEN WORKING WITH SYMBOLS, TEXTS, LINES AND CIRCLES.

ROTATING AREAS

You have also got the option to rotate an entire area. All kinds of objects in the area — symbols, texts, lines and circles/arcs — will be rotated.



First click the **Area** button, and then select the area you want to rotate by dragging a window around it.

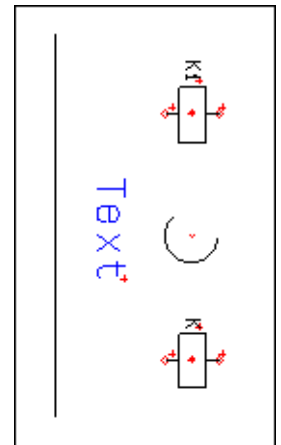


Click inside the area, keep the mouse button pressed down and move the mouse. Press [spacebar]. Hereby the area is rotated around the point the mouse is pointing at.

Every time you press [spacebar] the area is rotated 90 degrees counter-clockwise.



When you have selected the area, you can also click the **Move** button, click in the angle field in the command toolbar, and type in the angle in which the area is to be rotated.



SCALING AREAS

You can use scaling when you make drawings as well as when you create/edit symbols (see “Creating Symbols” on page 676).

To scale an area do the following:

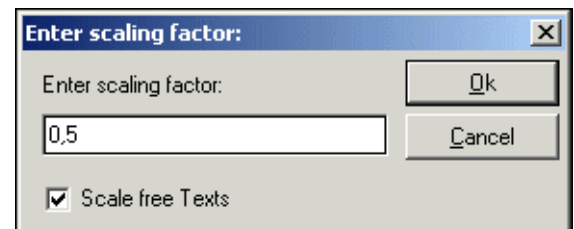


1) Click on the **Area** button, and mark up the area you want to scale

2) Choose **Edit => Scale**

3) Type in a scaling factor. — If for instance you want the area in half size, you type in 0,5. In double size, you type in 2

4) Click **OK**

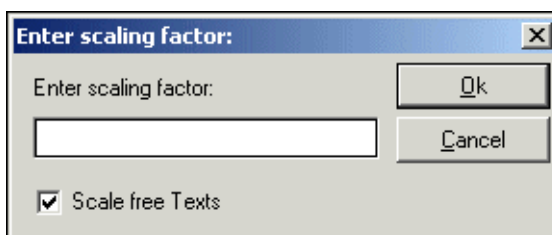


The area have now been scaled. If you are editing a symbol, this can require adaptation of texts, and moving of the reference point, because they are now placed in another distance to the scaled objects.

— See for instance how to scale a company logo, to make it fit a drawing header in “Placing a Company Logo on a drawing header” on page 731.

Scaling texts in areas

When you have selected an area and choose **Edit => Scale**, it is possible to specify whether to scale the free texts in the area. This is done by choosing **Scale free texts** in the dialog box:



Scaling everything on a page



If you want to scale everything on the entire page, you activate the **Area** button, choose **Edit => Select all => All on page**, and choose **Edit => Scale** as above. Read more about selections in areas in “Selecting objects” on page 49.

Specifying the scale of the page

You can also specify the scale of the entire page. — Read more about this in “Setting the scale of the page” on page 469.

SEE VIDEO ON **COPYING AREAS INTO EDIT SYMBOL MODE**



To copy an area from a project drawing into a symbol, do the following:



- 1) Click the **Area** button, and select the area you wish to copy.
- 2) Right-click, and choose **Copy**.
- 3) Click on the **Symbols** button, and then on the **Symbol menu** button.
- 4) In the **Symbol menu**: Click on the **Create new Symbol** button.
- 5) In the **Edit symbol** mode: Choose **Edit => Paste**.
- 6) You now get the area in the cross hairs: Click to place the area in the symbol definition.
- 7) Press <Esc> to remove the area from the cross hairs.

The copied area is now inserted in the Symbol editor.



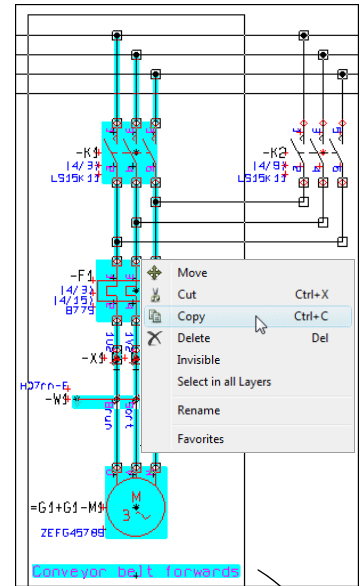
ALL SYMBOLS IN THE INSERTED AREA ARE AUTOMATICALLY CONVERTED TO LINES WHEN THE AREA IS INSERTED IN THE SYMBOL DEFINITION.

You can now place the connection points for the new symbol, edit the symbol etc. as described in "Creating Symbols" on page 676.

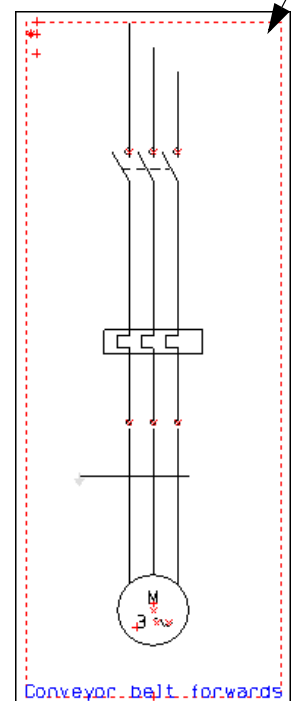
COPYING AREAS BETWEEN PROJECTS



THE COPIED AREA CAN ALSO BE INSERTED DIRECTLY INTO ANOTHER **Automation** PROJECT, BY JUMPING TO THE OTHER PROJECT, AND CHOOSING **EDIT => PASTE**.



Copying an area from a diagram to a symbol definition





SUB DRAWINGS

In **Automation** it is possible to save parts of diagrams as *sub drawings*, which you can then *drag* directly into project pages in other projects. This is in **Automation** called *Drag'n Draw*.

— Hereby you can also drag project pages or full projects into the active project.

All of the symbols in the sub drawings can have individual article data attached, and you can — as a general setting — specify the settings for how to rename the symbols, when the sub drawings are placed in projects.

Furthermore you can create different sets of *model values* for each individual sub drawing, so that you can choose between placing e.g. a 1 kW or a 15 kW motor-starter circuit when placing the sub drawings.

You can also apply the sub drawings in connection with automatic project generation, as described in the **Project generator** manual.

THIS CHAPTER DESCRIBES	
— THE SUBDRAWINGS TAB IN THE EXPLORER WINDOW	(P. 245)
— ON INSERTING SUB DRAWINGS	(P. 246)
— MANUAL PLACING OF SUB DRAWINGS	(P. 247)
— PLACING SUB DRAWINGS BY DRAG'N DRAW	(P. 252)
— DRAG'N DRAW SETUP	(P. 253)
— CREATING SUB DRAWINGS FOR MANUAL PLACING	(P. 255)
— EDITING SUB DRAWINGS	(P. 256)
— CREATING SUB DRAWINGS BY COPYING FROM PROJECTS	(P. 256)
— ON PAGES AND SUB DRAWINGS FOR DRAG'N DRAW	(P. 259)
— CREATING PROJECT PAGES FOR AUTOMATIC PLACING OF SUB DRAWINGS ..	(P. 260)
— PREPARING SUB DRAWINGS FOR DRAG'N DRAW / AUTOMATIC PLACING	(P. 265)
— THE FUNCTION OF THE INSERTION POINTS	(P. 267)
— INSERTING VARIABLES IN SUB DRAWINGS, PAGES AND PROJECTS	(P. 269)
— CREATING MODEL DATA FOR SUB DRAWINGS	(P. 272)
— INVISIBLE AREAS AND SUB DRAWINGS	(P. 275)
— SUB DRAWINGS WITH INTELLIGENT LINE CONNECTION	(P. 276)



SUB DRAWINGS IN NETWORKS

WHEN YOU ARE WORKING IN A NETWORK, IT COULD BE A GOOD IDEA TO PRODUCE A SET OF STANDARD SUB DRAWINGS, AND PLACE THESE IN A FOLDER, WHICH EVERYBODY HAS ACCESS TO IN THE NETWORK.

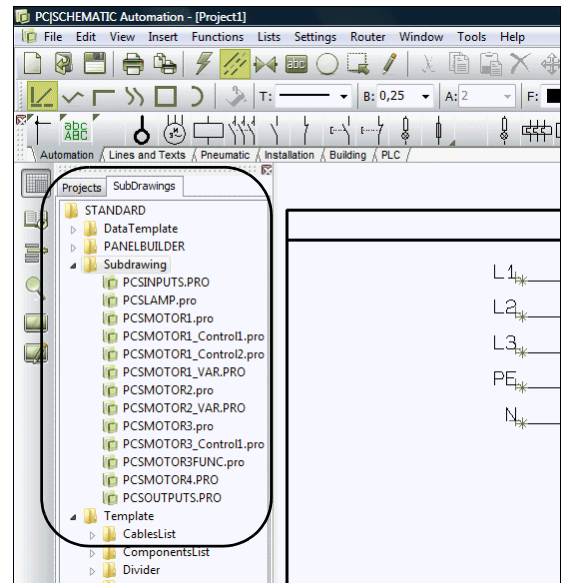
THE EXPLORER WINDOW: THE SUBDRAWINGS TAB

In the **Explorer window** in the left-hand side of the screen, you find the tab **SubDrawings**.

From here you can fetch the sub drawings and projects/project pages, which you choose to *drag* into the active project.

If the **Explorer** window is not displayed on the screen, please read “General settings for the Explorer window” on page 291.

— Read more about the **Explorer window** in “The Explorer window: The Projects tab” on page 291.



Specifying what to display on the SubDrawings tab

To select — and lock — which folder to display the contents of on the **SubDrawings** tab, you can right-click on the folder in the tab.

You can then choose between:

Show from here: The **SubDrawings** tab displays the contents of the selected folder.

One level up: The **SubDrawings** tab displays the contents of the folder one level above the selected folder.

Show all: The **SubDrawings** tab displays everything you have access to from your PC.

Other options in the right-click menu

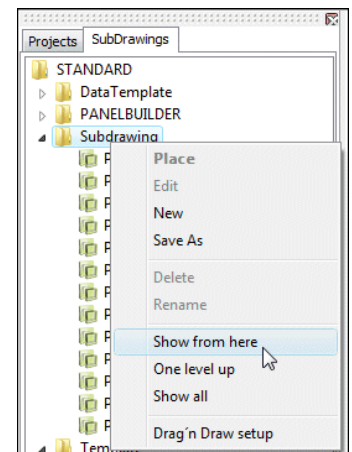
Beyond this, you can choose between:

Place: Hereby you get the selected sub drawing in the cross hairs, and can place it in the active project, as described in “Manual placing of sub drawings” on page 247.

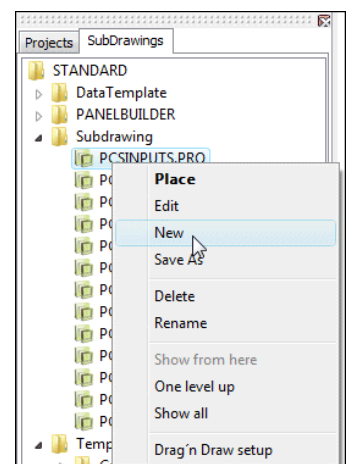
Insert: This places the sub drawing on the diagram page. Is only possible on pages containing insertion points.

Edit: Hereby the selected sub drawing is opened in the **Edit sub drawing** mode, and you can edit the sub drawing, as described in “Editing sub drawings” on page 256.

New: Hereby the program enters the **Edit sub drawing** mode, from which you can create a new sub drawing, as described in “Creating sub drawings by Copying from projects” on page 256.



(Click on Folder)



(Click on Sub drawing)

Delete: Hereby the selected sub drawing is deleted. It is not possible to delete folders from the **SubDrawings** tab.

Rename: Hereby you can rename the selected sub drawing.

Drag'n Draw setup: — See “Drag'n Draw setup” on page 253.

ON INSERTING/PLACING SUB DRAWINGS

When inserting sub drawings in **Automation**, this can be done in two different ways:

- 1) *Manually:* When you *drag* the sub drawing *from* the **SubDrawings** tab into the project page, you get the sub drawing in the cross hairs, and place it manually on the project page. — See “Manual placing of sub drawings” on page 247.
- 2) *By Drag'n Draw / Automatic placing of sub drawings:* When you *drag* the sub drawing *from* the **SubDrawings** tab, and *release* it as soon as it is over the project page, the sub drawing is connected automatically e.g. to the drawn potentials on the pages. — See “Placing sub drawings with drag'n draw” on page 252.

For both methods applies: Control of Article data, Texts and Model data

When applying both methods, the symbols in the sub drawings can contain article data — e.g. from the database. Furthermore, all texts can be controlled via variables, and you can select between different sets of model data for the sub drawings, if variables and/or sets of model data, that have been created for sub drawings.

For manual placing of sub drawings

All sub drawings can be placed manually.

You can start by creating sub drawings for manual placing, and later prepare them for applying drag'n draw.

For Drag'n Draw

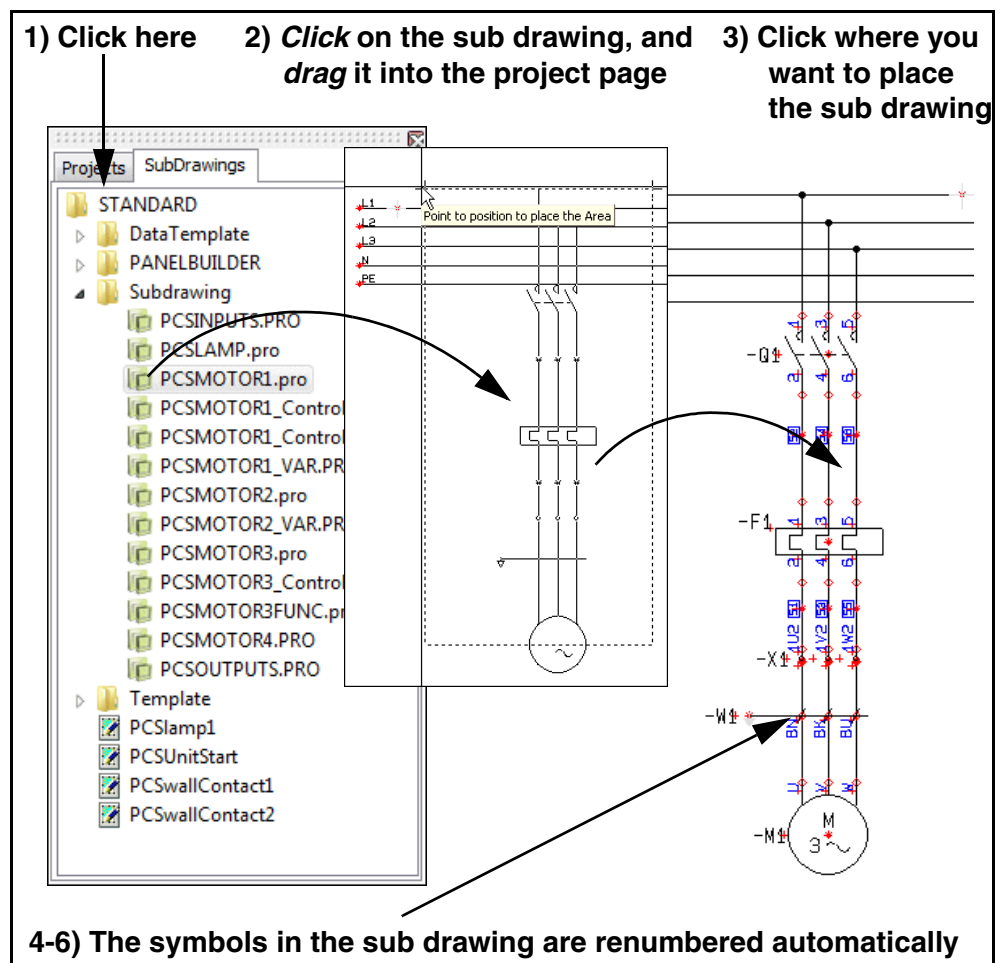
To get the full benefit of applying drag'n draw, you need to make some extra preparations, as described in “On Pages and Sub drawings for Drag'n Draw” on page 259.

Having made these preparations, you achieve a faster workflow — and you have made practically speaking all of the necessary preparations for applying sub drawings for automatic project generation, as described in the **Project generator** manual.

MANUAL PLACING OF SUB DRAWINGS

To insert a sub drawing *manually* — see “On inserting/placing sub drawings” on page 246 — on the active project page, do the following:

- 1) Click on the **SubDrawings** tab in the **Explorer** window.
If the **Explorer** window is not visible, press <Ctrl+F12> or choose **Settings => Pointer/Screen** and select **Explorer window**).
- 2) Click on the name of the desired sub drawing, and *drag* it into the project page.
- 3) You now get the sub drawing in the cross hairs, and click where you wish to place the sub drawing in the project.



If the sub drawing contains Model data

- 4) If different *models* have been created for the selected sub drawing, you enter the dialog box **Rename symbols**, in which you select which model you want to apply. — Read more about this in “Inserting sub drawings: With model data” on page 248.

The sub drawing is now inserted: Symbols are renamed and Article numbers added

- 5) The sub drawing is now placed in the project page, and:
— The symbols have been renumbered automatically (see “Drag’n Draw

setup” on page 253)

— The symbols in the placed sub drawing contains the same article numbers, texts and variables as the sub drawing in the **SubDrawings** tab.



THE SUB DRAWING IS NOW INSERTED

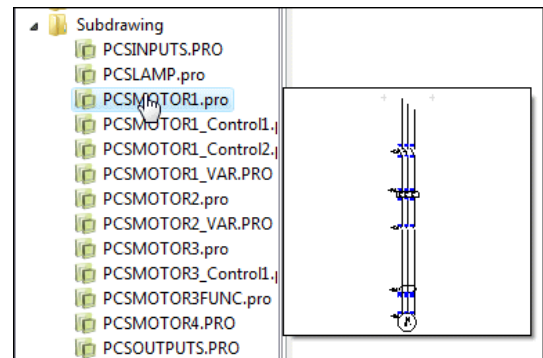
THE SUB DRAWING IS NOW INSERTED IN THE PROJECT, AND YOU CAN CONTINUE WORKING WITH THE SYMBOLS, LINES ETC. IN THE SUB DRAWING, JUST AS IF YOU HAD DRAWN IT FROM SCRATCH.

— Please also see “Creating Project pages for aut. placing of sub drawings” on page 260 and see how to apply sub drawings for automatic project generation in the **Project generator** manual.

Sub drawings displayed in window

When you *click* on a sub drawing in the **SubDrawings** tab, the sub drawing is displayed in a small window, so that you can see which sub drawing you are selecting.

To *select* the sub drawing, *double click* on the name of the sub drawing — or *click and drag* in the same operation. Hereby you get the sub drawing in the cross hairs, and you can place it in the project.

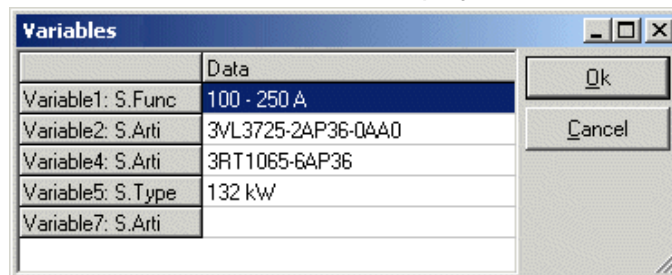
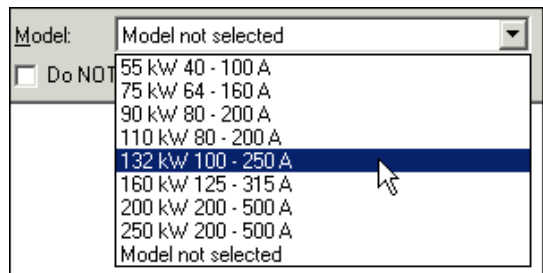


Inserting sub drawings: With model data

When you insert a sub drawing — see “Manual placing of sub drawings” on page 247 —, where the sub drawing have different sets of model data attached, you can select which model to apply at the bottom of the dialog box

Rename symbols:

When you have selected which model data to assign to the inserted sub drawing, you can click the **Variables** button in the upper-right corner of the dialog box, and the values of variables for the model are displayed:



Here you can edit data — if necessary — by double clicking on a field and then editing the text. When you have placed the sub drawing in the project, you can edit the texts in via this dialog box, as described in “Editing variables for inserted sub drawings” on page 249.

Click **OK**, and click **OK** in the dialog box **Rename symbols**.

Variable markers (*/@..*) in sub drawings are removed when sub drawings are placed in projects.

— See also “Drag’n Draw setup” on page 253, “Drag’n Draw settings: For model data” on page 253 in particular.

— See also how to create models for a sub drawing in “Creating Model data for sub drawings” on page 272.



THE PURPOSE OF VARIABLES

VARIABLES MAKES IT POSSIBLE TO EDIT SOME OF THE DATA, THAT CAN BE INCLUDED IN A SUB DRAWING.

— READ MORE ABOUT THIS IN “INSERTING VARIABLES IN SUB DRAWINGS, PAGES AND PROJECTS” ON PAGE 269, SPECIALLY “INSERTING VARIABLE MARKERS ON A SUB DRAWING (OR ON A DRAWING TEMPLATE)” ON PAGE 269.

Editing variables for inserted sub drawings

To edit the text fields, which had variables on an inserted sub drawing, do this:

- 1) Select the project page on which the sub drawing is inserted.
- 2) Choose **Edit => Variables**.
- 3) You now enter the dialog box **Variables**.

	Section2	Section3	Section4	Se
Variable1	=G1+A2-Q1	=G1+A2-Q2	=G1+A2-Q3	=E
Variable2	4022903075387	4022903075387	4022903075387	40
Variable3	-F1	-F2	-F3	-F
Variable4	4022903085584	4022903085584	4022903085584	40
Variable5	-X1	-X2	-X3	-X
Variable6	-W1	-W2	-W3	-W
Variable7	5702950410537	5702950410537	5702950410537	57
Variable8	-M1	-M2	-M3	-M

Click on the variable field, and the cross hairs points out its position on the page

Here you see a survey of all variables on the page. If more than one sub drawing is inserted on the page, these are all displayed in the dialog box. When in doubt about which datafield is listed in the dialog box, click on it in the dialog box, and the cross hairs points it out on the diagram page.

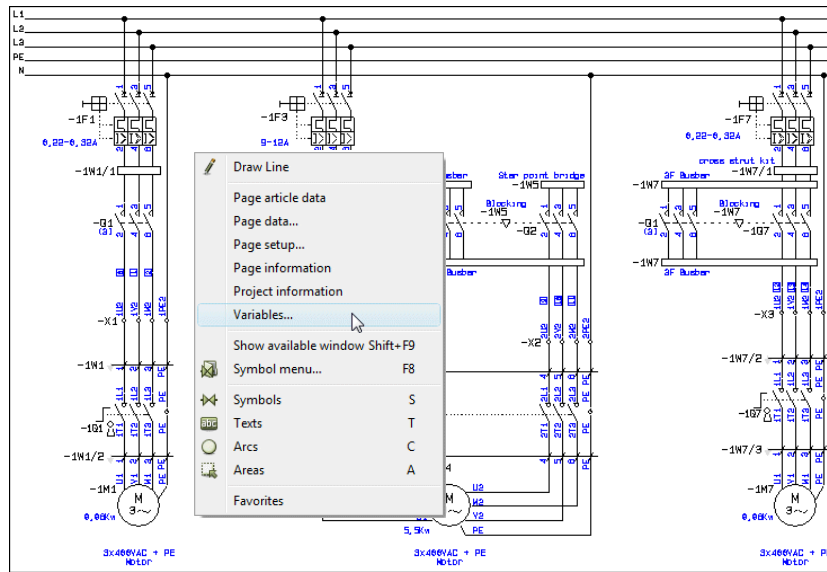
- 4) Double-click on one of the fields in the dialog box, and edit the text. Continue like this until all relevant texts are edited.
- 5) Click **OK**.

You can also edit the texts in the sub drawing exactly as you would edit any other text in **Automation**.

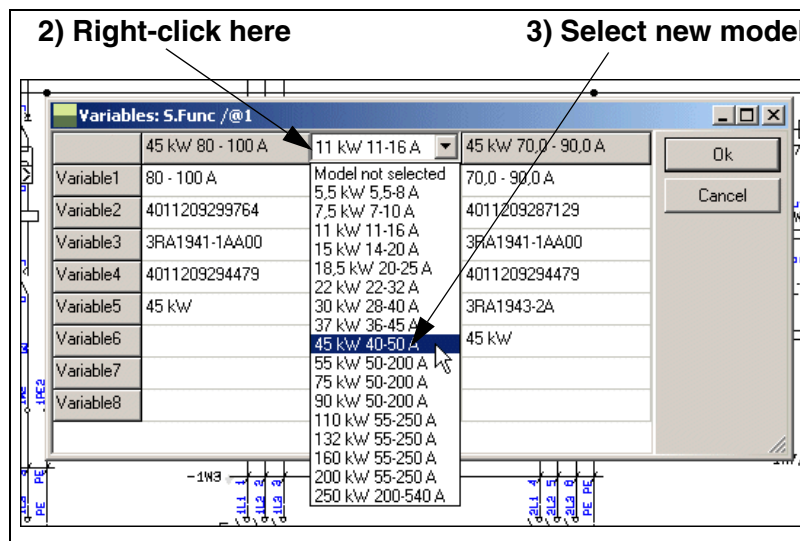
Sub drawings: Changing the selected model

When you have placed a sub drawing containing model data, you can subsequently change the selected model:

- 1) Right-click on the page, and select **Variables**:



- 2) In the dialog box **Variables**: Right-click on the column header for the section (sub drawing), for which you wish to change model.
- 3) You now see all of the models for the sub drawing. Click on the desired model:



The model data for the sub drawing are now changed.

It is possible to change model data for any sub drawing in the project. You can also do this for sub drawings, that has been placed in the project by automatic project generation.

Scope for changing models

All other symbols in the project, that are a part of the applied symbols in the sub drawing, will be updated according to the new model automatically.

This meaning, that if the sub drawing e.g. contains a symbol with the name **-K3**, all project symbols for the component **-K3** will be updated with the new data.

When the newly selected model implicates that some symbols change state, this will also be the case for the implicated symbols outside the sub drawing.

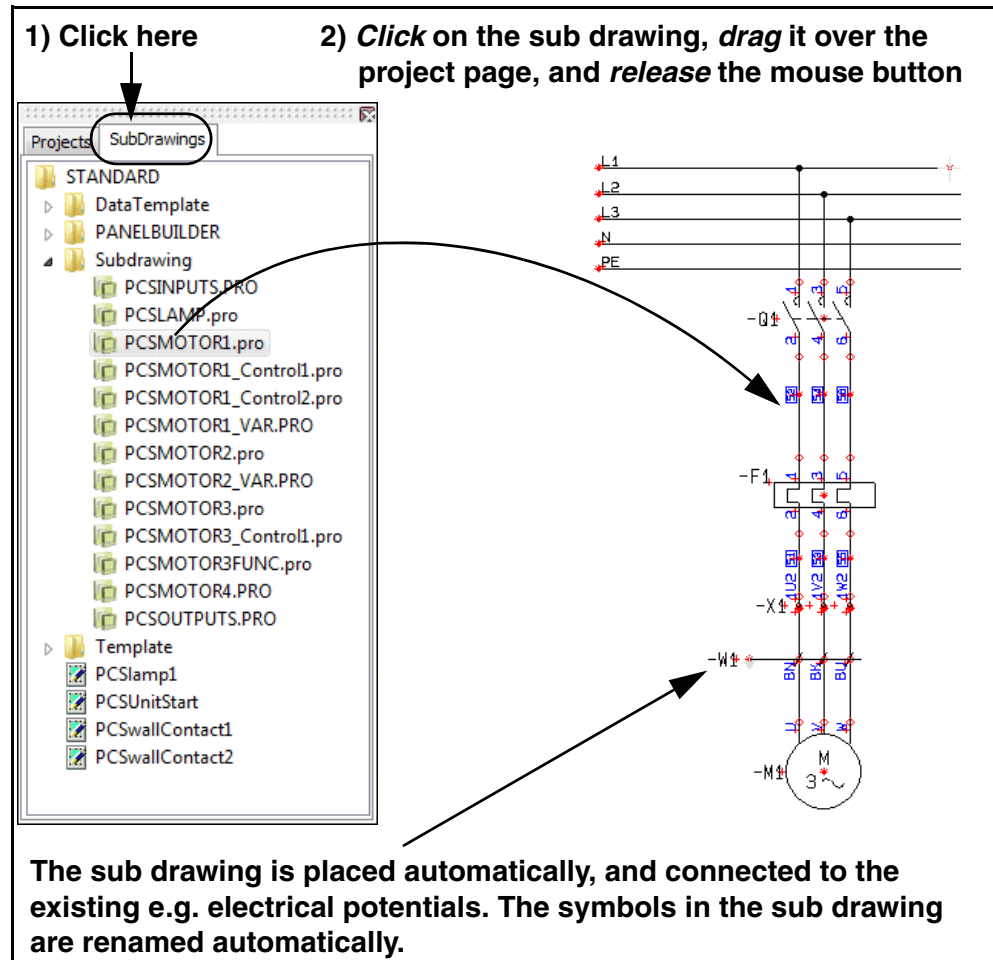


PLACING SUB DRAWINGS WITH DRAG’N DRAW

To place sub drawings by applying the drag’n draw technique, both the sub drawing and the pages they are placed in, must be prepared for drag’n draw / automatic placing. — See “On Pages and Sub drawings for Drag’n Draw” on page 259.

When the project pages and the sub drawings are prepared, you can place the sub drawings like this:

- 1) Click on the **SubDrawings** tab in the **Explorer** window.
(If the **Explorer** window is not visible, then choose **Settings => Pointer/Screen** and select **Explorer window**).
- 2) Click on the name of the desired sub drawing, *drag* it into the project page, and *release* it as soon as it is over the project page — or just press the <Ctrl> key and *click* on the sub drawing. Hereby the sub drawing is placed automatically on the project page, and connects automatically to e.g. the electrical potentials on the page:

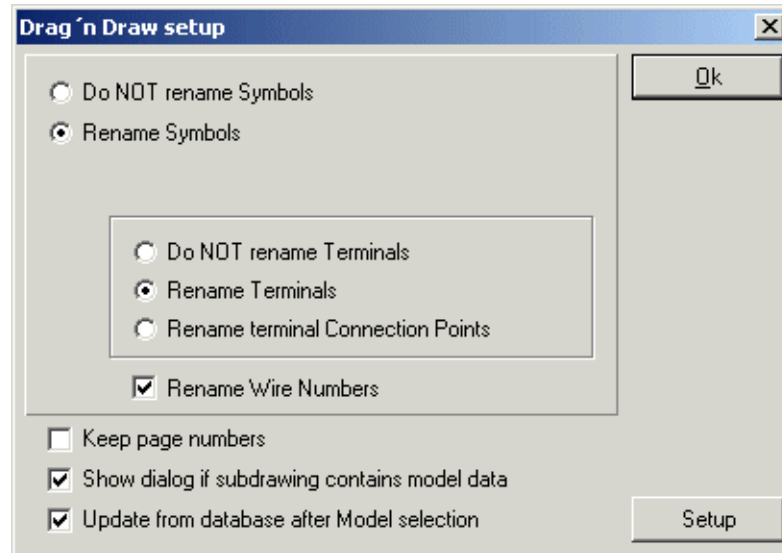


If variables are placed in the sub drawing, or if you can select different sets of model data for the sub drawing, you now get the option of specifying which data to insert in the sub drawing. You can read more about this by continuing the explanation under “If the sub drawing contains Model data” on page 247.

DRAG'N DRAW SETUP

When sub drawings are placed automatically in project pages, **Automation** can rename symbol names in these sub drawings automatically.

To specify how to do this, right-click somewhere inside the **SubDrawings** tab, and select **Drag'n Draw setup**:



Here you specify whether to rename symbols in the sub drawings you insert, and how to rename these symbols. — Read more about this in “Intelligent copying of areas” on page 233.

Beyond this, you have the following options:

Keep page numbers

Because you can also *drag* full pages / projects into the active project, you can select **Keep page numbers** to specify that you wish to keep the existing page numbers, so that these are not changed when you drag new pages into the project. This includes both the page numbers in the active project, and the page numbers in the project / the project pages, which you drag into the project.

Drag'n Draw settings: For model data

When you select **Show dialog if sub drawing contains model data**, you always enter the dialog box **Rename symbols** — see “**Manual placing of sub drawings**” on page 247, “Inserting sub drawings: With model data” on page 248 in particular — when placing a sub drawing containing model data. From this dialog box you can select which model you wish to place.

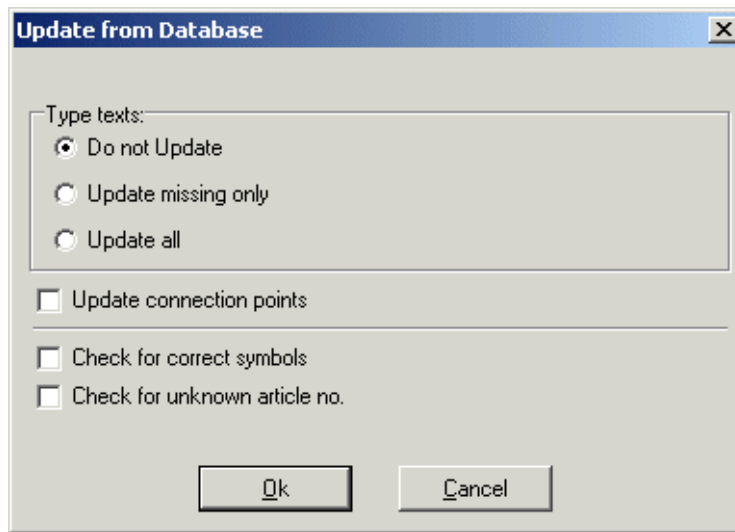
If the option is *not* selected, the sub drawing is placed without model data, or with the data from the original sub drawing, if the sub drawing contains data values.

Update from database after Model selection

When you select **Update from database after Model selection**, the sub drawing is updated from the database, when you select the model.

If this option is *not* selected, the symbols in the sub drawing are assigned exactly the same data values as the sub drawing.

When you select **Update from database after Model selection**, a **Setup** button appears in the dialog box: Click on it, and you enter the dialog box **Update from database**:



Here you specify how to perform the update from the database. These options are described in "Updating symbols from the database" on page 166.

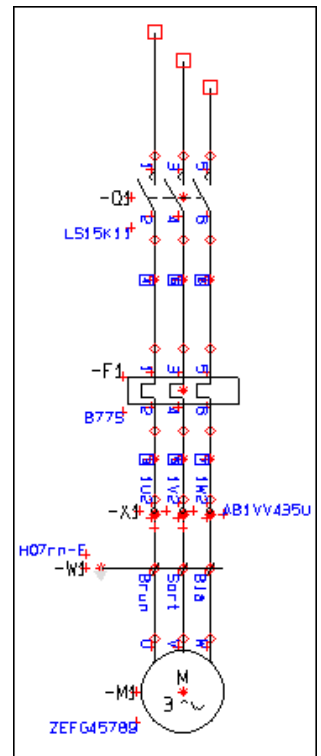
CREATING SUB DRAWINGS FOR MANUAL PLACING

A sub drawing can be any project, containing one single diagram page. — See also “Creating sub drawings by Copying from projects” on page 256.

For instance you can create a sub drawing like this:

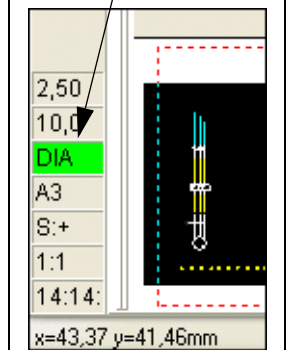
Enter the Edit Sub drawing mode

- 1) In the **SubDrawings** tab in the **Explorer** window: Right-click in the folder that you wish to place the sub drawing in, and choose **New**:



- 2) **Automation** now changes to the **Edit sub drawing** mode, which you can see because the **Page type** field at the bottom of the left toolbar blinks green. You can now create a new sub drawing.
- 3) Draw a sub drawing — for instance as displayed in the figure above — and attach article data from the database to the symbols, if necessary. It has no importance where on the page you draw the sub drawing. — You can also insert a copied area from an existing project, as described in “Creating sub drawings by Copying from projects” on page 256.
- 4) Continue as described in “Inserting e.g. variable markers and/or model data” on page 257.

Blinks green in the Edit sub drawing mode



LAYERS ARE FLATTENED TO ONE LAYER

WHEN YOU PLACE A SUB DRAWING IN A PROJECT, THE LAYERS IN THE SUB DRAWING ARE FLATTENED, SO THAT ALL INFORMATION IN THE SUB DRAWING IS PLACED IN ONE LAYER ONLY. THE LAYER STRUCTURE IN THE SUB DRAWING IS THEREFORE LOST, WHEN THE SUB DRAWING IS PLACED IN THE PROJECT.



DO NOT INCLUDE LOGOS IN SUB DRAWINGS.

EDITING SUB DRAWINGS

To edit a sub drawing, right-click on the sub drawing in the **SubDrawings** tab in the **Explorer** window, and choose **Edit**. Hereby you can edit the sub drawing as described in “Creating sub drawings for manual placing” on page 255.



CREATING SUB DRAWINGS BY COPYING FROM PROJECTS

If you wish to create a sub drawing based on existing material from a project, you can do like this:

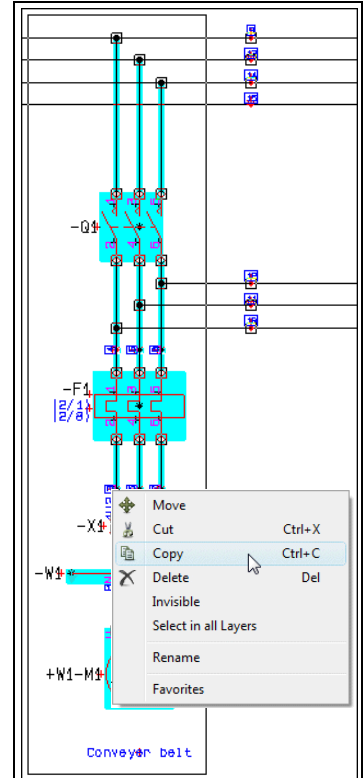
Copying the desired area



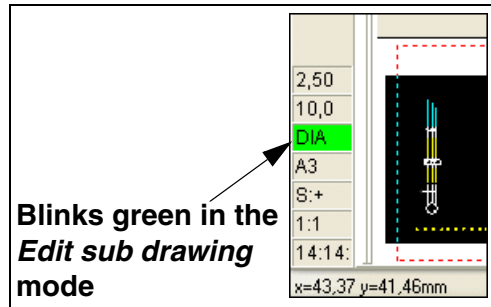
- 1) Click the **Area** button.
- 2) Select the area you wish to copy, right-click in the area, and choose **Copy** — see the figure to the right.

Entering the Edit sub drawing mode

- 3) On the **SubDrawings** tab in the **Explorer window**: Right-click on the folder you wish to place the sub drawing in, and select **New**:
- 4) **Automation** now change to the **Edit sub drawing** mode, which you can see because the **Page type** field at the bottom of the left toolbar blinks green. You can now create a new sub drawing.



Copying an area from a project ...



Blinks green in the Edit sub drawing mode



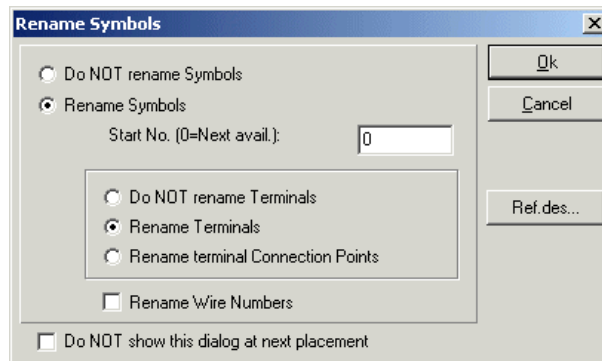
SAVE COPIED AREAS USING SAVE AS

INSTEAD OF CHOOSING **NEW**, YOU CAN CHOOSE **SAVE AS**. YOU ENTER THE DIALOG BOX **SAVE AS**, WHERE YOU TYPE IN THE NAME OF THE SUB DRAWING AND CLICK **SAVE**. THE SUB DRAWING IS THEN CREATED, AND IS DISPLAYED IN THE SELECTED FOLDER IN THE **SUB-DRAWINGS** TAB.

TO EDIT THE SUB DRAWING, YOU MUST THEN RIGHT-CLICK ON IT AND CHOOSE **EDIT**.

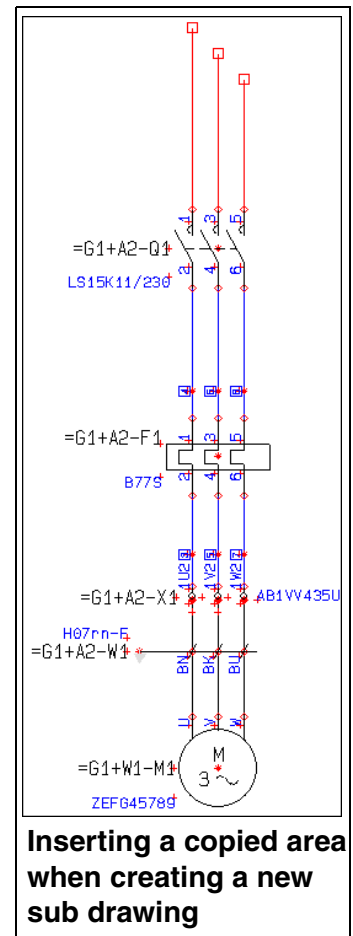
Inserting the copied area

- 5) Right-click on the page and choose **Paste**.
- 6) You are now asked whether to rename the symbols in the area — and get the option of changing reference designations (if any) in the area:



— See which options you have in “Intelligent copying of areas” on page 233. Select the desired settings and click **OK**.

- 7) The copied area is now displayed including all component data and reference designations for the symbols in the area. — See the figure to the right. Press <Esc> to remove the copied area from the cross hairs.

*Adapting the sub drawing*

- 8) Now adapt the copied area, so that it becomes exactly the sub drawing you want.

Inserting e.g. variable markers and/or model data

- 9) If you want to, you can now place *variable markers* in the sub drawing, as described in “*Inserting variables in Sub drawings, Pages and Projects*” on page 269, and you can create different sets of *model data* for the sub drawing, as described in “*Creating Model data for sub drawings*” on page 272.
- 10) If you wish to use the sub drawing for *drag’n draw* or in automatic project generation, you must also place *insertion points* in the sub drawing, as described in “*The function of the insertion points*” on page 267. (It might be easier to place the insertion points in the project *before* you copy the area — see “*Points to consider in Project generation*” in the **Project generator** manual).

Saving the sub drawing

- 11) When you have adapted the sub drawing, click on the **Save** button, and you enter the **Save as** dialog box. Here the same folder is selected as the one you clicked on in the **SubDrawings** tab, when you chose **New**. Assign a name to the sub drawing, and click **Save**.

12) Choose **File => Close** to leave the edit sub drawing mode.

You have now created a new sub drawing from a copied area in a project, and have inserted the sub drawing in a folder, which is accessible from the **SubDrawings** tab in the **Explorer window**.



SUB DRAWINGS FOR DRAG’N DRAW / AUTOMATIC PLACING

IF YOU WISH TO USE THE SUB DRAWING FOR DRAG’N DRAW / AUTOMATIC PLACING, YOU MUST ALSO READ “ON PAGES AND SUB DRAWINGS FOR DRAG’N DRAW” ON PAGE 259 AND “PREPARING SUB DRAWINGS FOR DRAG’N DRAW / AUT. PLACING” ON PAGE 265.

ON PAGES AND SUB DRAWINGS FOR DRAG’N DRAW

To apply the automatic functions in **Automation** for *drag’n draw* / automatic placing of sub drawings, you need to create:

- 1) The *project pages* where you place the sub drawings. Here you must specify the area in which you can place sub drawings on the page.
- 2) The *sub drawings* which **Automation** shall be able to place automatically on the project pages. Here you must specify how much space the sub drawing needs when it is placed in a project.

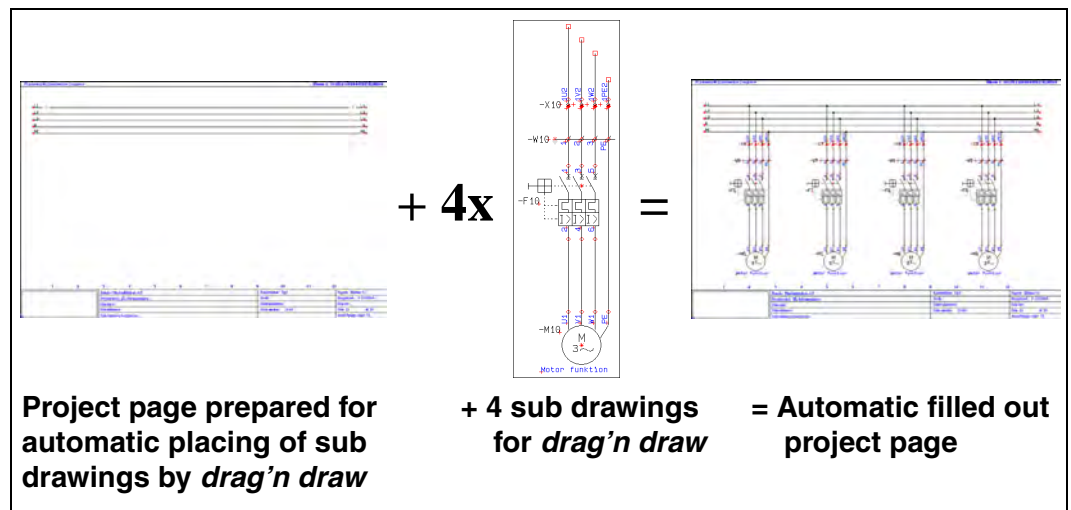
The project pages are created just like you create sub drawings — see “Preparing Sub drawings for Drag’n Draw / Aut. placing” on page 265. Here it is also described how to place the so-called *insertion points*, used for specifying the size of sub drawings and for specifying in which area sub drawings can be placed on project pages.

On project pages you will — as opposed to sub drawings — also place a drawing header.

Variable markers and Models

In project pages you can, just as for sub drawings, insert *variable markers* for the texts in the page / sub drawing, that you wish to specify the contents of, when you *drag* the page or sub drawing into the project.

For sub drawings you can furthermore specify different sets of *model data*, which you can choose between when placing the individual sub drawing.



Example

An example on creating a drawing template can be found in “Creating Project pages for aut. placing of sub drawings” on page 260.

CREATING PROJECT PAGES FOR AUT. PLACING OF SUB DRAWINGS

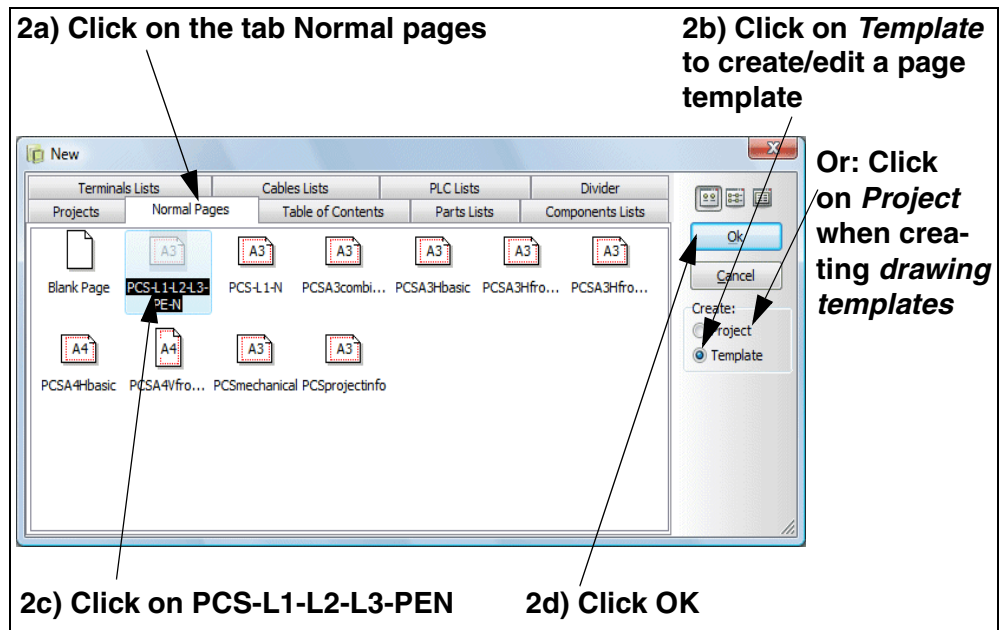
When using *drag'n draw* you can make **Automation** place sub drawings on project pages automatically.

To get full benefit of this powerful tool, you must first create a *project page*, on which you specify within which area the sub drawings can be placed automatically. As you will see, you will only have to create this page once.

Page Templates prepared for automatic placing of sub drawings

To see the design of a page template prepared for automatic placing of sub drawings, do this:

1) Choose **File => New**, and you enter the dialog box **New**:



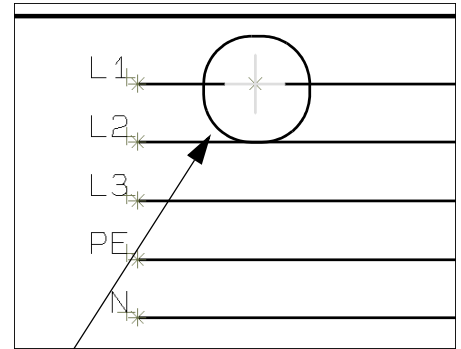
2) Click on the tab **Normal pages**, click on **Template** in the right-hand side, click on the page **PCS-L1-L2-L3-PEN**, and click **OK**.



AUTOMATIC PROJECT GENERATION: CREATING DRAWING TEMPLATES

WHEN YOU WISH TO CREATE A *DRAWING TEMPLATE* FOR AUTOMATIC GENERATION OF PROJECTS — SEE "CREATING DRAWING TEMPLATES FOR AUTOMATIC PROJECT GENERATION" IN THE **Project generator** MANUAL — YOU MUST CHOOSE **PROJECT**, AND SAVE THE CREATED PROJECT IN THE FOLDER WHERE YOU KEEP DRAWING TEMPLATES. — SEE "SETTINGS FOR THE PROJECT GENERATOR" IN THE **Project generator** MANUAL.

- 3) **Zoom** in on the upper left corner of the page. Here an *insertion point* symbol is placed, as you can see in the figure to the right.
- 4) In the other end of the electrical potentials, there is placed another insertion point symbol. These two symbols specifies the area in which subdrawings can be placed automatically. It is important that these two symbols are aligned.
- 5) Choose **File => Close** to exit the page template.



Insertion point symbol



THE INSERTION POINTS SPECIFIES WHERE THE SUB DRAWINGS ARE INSERTED

THE TWO INSERTION POINT SYMBOLS SPECIFIES THE AREA IN WHICH **Automation** CAN NOW PLACE THE SUB DRAWINGS AUTOMATICALLY.

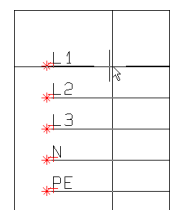
— READ MORE ABOUT INSERTION POINT SYMBOLS IN “THE FUNCTION OF THE INSERTION POINTS” ON PAGE 267.

— SEE “POINTS TO CONSIDER IN PROJECT GENERATION” IN THE **Project generator** MANUAL ALSO.

Placing insertion points on project pages for automatic placing of sub drawings

If you are creating your own page template, that you wish to prepare for automatic placing of subdrawings, you can place the insertion point symbols like this:

- 1) Start by opening an existing page template, as described in “Page Templates prepared for automatic placing of sub drawings” on page 260. You might choose to click on **BlankPage**, to start on an entirely new page.
- 2) Insert a drawing header, electrical potentials and whatever you wish to place in the page template.
- 3) Click on the **Symbols** button and then on the **Symbol menu** button.
- 4) In the **Symbol menu**: Choose the Alias or the folder **MISC**, select the symbol **INSPPOINT.SYM**, and click **OK**. — Read more about the **Symbol menu** in “Fetching symbols using the Symbol menu” on page 140.
- 5) Place the insertion point symbol — for instance at the *right-hand* side of **L1**.
- 6) Insert another insertion point symbol — for instance at the *left-hand* side of **L1** — and press <Esc> to remove the symbol from the cross hairs. It is important that the two insertion point symbols are aligned.
- 7) Click on the **Save** button.



- 8) In the dialog box **Save**: Write for instance the name *Example.std*, select the folder **Pcscad\Standard\Template\Normal**, and click **Save**.
(If you have created a *drawing template* for automatic generation of projects, you must remember to save it in the folder where you save drawing templates. — See "Settings for the Project Generator" in the **Project generator** manual).
- 9) Choose **File => Close** to close the page.
You have now created a template page, prepared for automatic placing of sub drawings by *drag'n draw*.



Placing sub drawings automatically on a project page

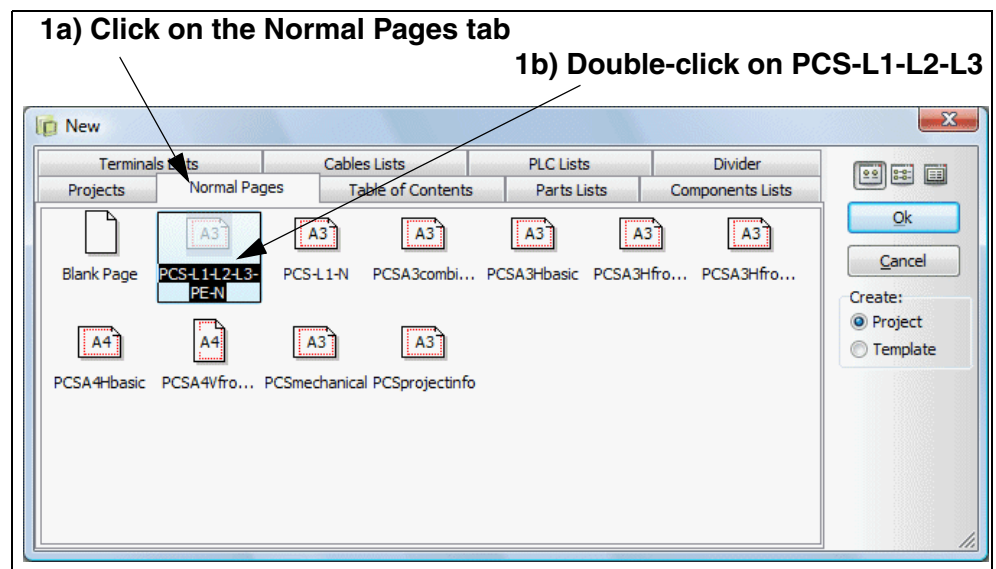


THE FOLLOWING IS BASED ON THAT YOU USE A *TEMPLATE PAGE* — AS DESCRIBED ABOVE —, PREPARED FOR AUTOMATIC PLACING OF SUB DRAWINGS BY DRAG'N DRAW.

IF YOU HAVE CREATED A *DRAWING TEMPLATE* FOR AUTOMATIC GENERATION OF PROJECTS, YOU MUST INSTEAD DRAG IN THIS PAGE FROM THE FOLDER YOU HAVE SPECIFIED AS **DEFAULT FOLDER FOR DRAWING TEMPLATES AND SUB DRAWINGS** FOR THE CREATED TEMPLATE — SEE "SETTINGS FOR THE PROJECT GENERATOR" IN THE **Project generator** MANUAL.

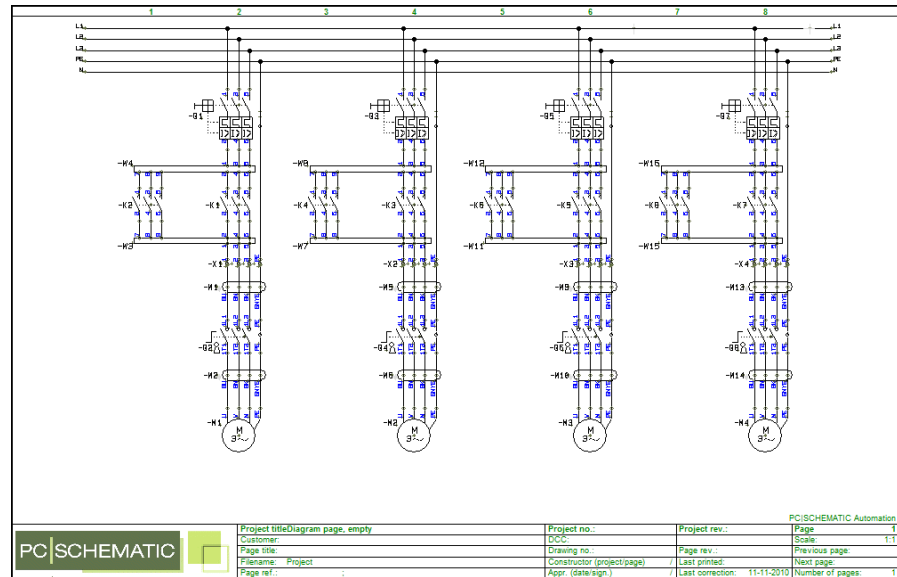
This is how you place a subdrawing automatically on a project page using drag'n draw:

- 1) Insert a new page that is prepared for automatic insertion of subdrawings: Choose **File => New**. In the dialog box **New**, click on the tab **Normal pages**, and double-click on **PCS-L1-L2-L3-PEN**:



- 2) When you have clicked **OK** in the **Settings** dialog box, you can see the two insertion points on **L1**, as described in "Page Templates prepared for automatic placing of sub drawings" on page 260.
- 3) In the **Explorer window**: Open the folder **Pcselcad\Standard\Subdrawing**, click on the sub drawing **PCSMOTOR3.PRO** (which have already been prepared for drag'n draw — see "Preparing Sub drawings for Drag'n Draw / Aut. placing" on page 265), drag the sub drawing into the project page, and release it as soon as you have dragged it out of the **Explorer window**.
- 4) You are now asked whether and how to rename the symbols in the subdrawing, and also get the option of choosing a subdrawing model, at the bottom of the dialog box. Click **OK**.
- 5) **Automation** have now placed the sub drawing on the page automatically, and have connected the sub drawing to the electrical potentials on the page.

- 6) Now *drag* the sub drawing **PCSMOTOR3.PRO** into the project page three more times. The sub drawing have now been placed four times on the page by using *drag'n draw*:



— If you could only get room to place the sub drawing three times on the page, this is because the insertion points on the page are placed too close to make room for four sub drawings on the page. For the future use of the page, you might therefore consider to make room for exactly four sub drawings on the page. — See e.g. "Points to consider in Project generation" in the **Project generator** manual, "Creating sub drawings as Modules" in particular.



AUTOMATIC RENAMING AND ARTICLE DATA

ALL SYMBOLS, TERMINALS AND CABLES HAVE HEREBY BEEN RENAMED AUTOMATICALLY. FURTHERMORE ALL SYMBOLS CONTAIN DATA FROM THE DATABASE, BECAUSE THE PLACED SUB DRAWING CONTAINED ARTICLE NUMBERS FOR THE SYMBOLS.

— SEE "DRAG'N DRAW SETUP" ON PAGE 253 TO SEE HOW TO SPECIFY THE SETUP FOR RENAMING ETC. WHEN APPLYING DRAG'N DRAW.

Subsequent update of text fields with variables

Though the variable markers are removed when placing the sub drawings, you still have the option of editing text fields from the sub drawing, which had variables.

— See "Editing variables for inserted sub drawings" on page 249.

PREPARING SUB DRAWINGS FOR DRAG'N DRAW / AUT. PLACING

When a sub drawing has been prepared for automatic placing, **Automation** can automatically place the sub drawing correct, when you *drag* it into a page which has been prepared for this. — See “Placing sub drawings with drag'n draw” on page 252.

— See how to prepare project pages for automatic placing of sub drawings by *drag'n draw* in “Creating Project pages for aut. placing of sub drawings” on page 260.



SUB DRAWINGS FOR AUTOMATIC GENERATION OF PROJECTS

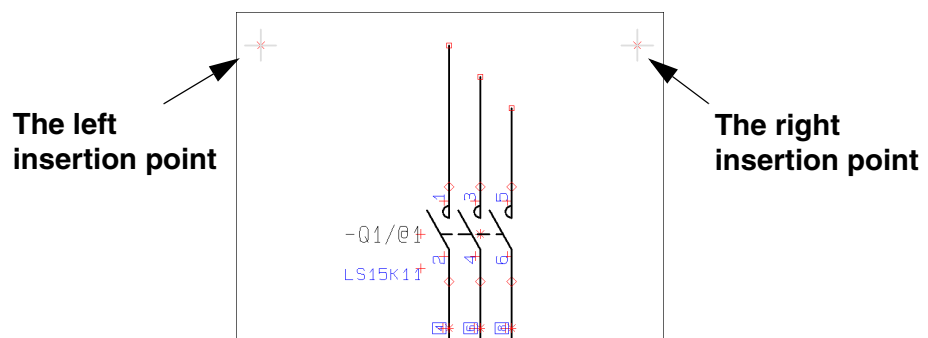
WHEN YOU WISH TO APPLY A SUB DRAWING FOR AUTOMATIC PROJECT GENERATION, IT *MUST* BE PREPARED FOR AUTOMATIC PLACING.

— READ MORE ABOUT THIS IN "GENERATE AND UPDATE PROJECTS AUTOMATICALLY" IN THE **Project generator** MANUAL. PLEASE PAY ATTENTION TO THE GENERAL CONSIDERATIONS ON CREATING SUB DRAWINGS IN "POINTS TO CONSIDER IN PROJECT GENERATION".

- 1) Open an existing sub drawing, or draw a new one, as described in “Creating sub drawings for manual placing” on page 255, or copy an area from an existing project, and create a sub drawing from this area, as described in “Creating sub drawings by Copying from projects” on page 256.



- 2) Click on the **Symbols** button, and then on the **Symbol menu** button.
- 3) In the **Symbol menu**: Choose the Alias or the folder **MISC**, choose the symbol **INSPOINT.SYM**, and click **OK**. — See e.g. the figure under “Placing insertion points on project pages for automatic placing of sub drawings” on page 261 or read more about the **Symbol menu** in “Fetching symbols using the Symbol menu” on page 140.
- 4) Place the two insertion points, so that they mark how much space the sub drawing needs horizontally during the automatic placing — see “The function of the insertion points” on page 267 (and see "Points to consider in Project generation" in the **Project generator** manual).



- 5) If you also wish to control the *texts* in the sub drawing (e.g. for automatic project generation — see the **Automation**): Insert some so-called *variable markers*, which makes it possible to control the texts of the sub drawing when placing the sub drawing (or to control the texts from a *project definition file* during automatic project generation). Read how to do this in “Inserting variables in Sub drawings, Pages and Projects” on page 269.

- 6) *If the sub drawing shall contain sets of **model data** — which you subsequently will be able to choose between when placing the sub drawing by **drag'n draw** —, then also create these, as described in “Creating Model data for sub drawings” on page 272.*



- 7) **Save** the sub drawing in a folder where you place sub drawings. — If the sub drawing shall be used for automatic generation of projects, you can for instance save it in the folder which is set to **Default folder for drawing templates and sub drawings**, as described in "Settings for the Project Generator" in the **Project generator** manual.

You have now created a sub drawing, which can be used for drag'n draw / automatic placing, as described in “Placing sub drawings with drag'n draw” on page 252. There you can also see a complete workflow for placing sub drawings on project pages using *drag'n draw*.

THE FUNCTION OF THE INSERTION POINTS

When you only wish to apply a *sub drawing* for manual placing, you do not need to place insertion point symbols on the sub drawing. However, if you need to be able to insert the part drawing automatically — by *drag'n draw* or by automatic project generation —, it *must* have two insertion points. — See “Insertion points and automatic placing of sub drawings” on page 267.

The left insertion point on the part drawing is the point you get in the cross hairs, when you place the sub drawing manually. You could therefore choose to insert just one single insertion point with just this purpose.

The insertion point symbols of the sub drawings are removed when you place the sub drawing on a project page.

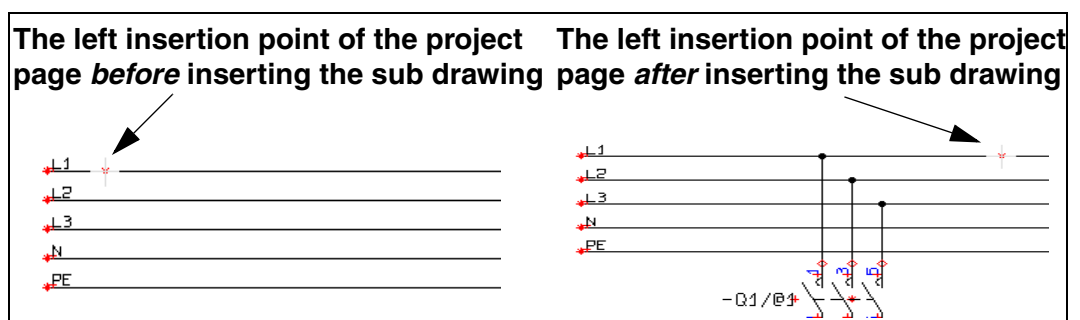
The created insertion point symbols are not displayed when printing project pages. This is because they are created using the color NP.

Insertion points and automatic placing of sub drawings

When using automatic placing of sub drawings, there exists two sets of insertion points:

- 1) The insertion points of the *sub drawing*: They tell the program how much space the sub drawing needs when it is placed automatically.
- 2) The insertion points of the *project page*: They tell the program where sub drawings can be placed on the page. (The project page can be fetched directly from a template page, or it can be a drawing template — see “On Pages and Sub drawings for Drag'n Draw” on page 259).

During the automatic placing the sub drawing is placed so that its left insertion point is placed on the left insertion point of the *project page*. The left insertion point of the project page is then moved, so that it now is placed on the position of the right insertion point of the sub drawing. The insertion points of the sub drawing are deleted from the page.



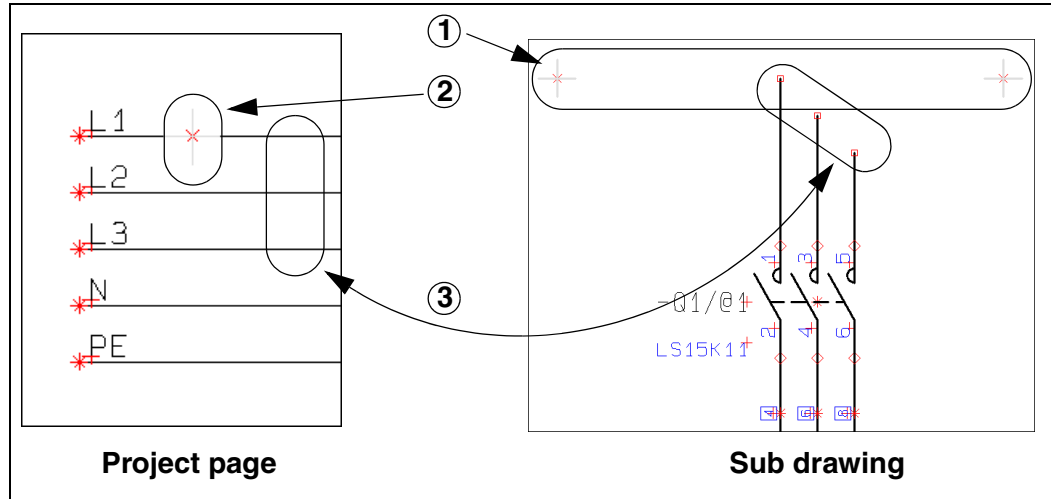
A sub drawing can not be placed automatically if there is not enough room for this sub drawing between the insertion points on the project page.

Insertion points and automatic project generation

During automatic project generation — see "Generate and Update projects automatically" in the **Project generator** manual — the situation mentioned above would result in inserting an extra project page, on which the sub drawing would be placed.

Placing the insertion point symbols carefully

When applying insertion point symbols, you must be careful to let the insertion points for the sub drawings match the electrical lines on the drawing templates, on which the sub drawings are going to be placed.



It is important to make sure of the following (observe the figure above):

- 1) Place the insertion point symbols on the *sub drawing* so that they are at the same height as the uppermost parked line.
- 2) Also place the insertion point symbols on the *project page* on the uppermost of the phases to which the sub drawing is going to be connected.
- 3) Adjust the vertical distance between the parked lines on the *sub drawing* to the distance between the phases on the *project page*.
- 4) Make sure to place the insertion points on the sub drawing so that there are room enough for the texts, which can be inserted via automatic project generation.

— Please also read "Points to consider in Project generation" in the **Project generator** manual. Here you will find a survey of additional aspects to consider, when creating sub drawings.



SUB DRAWINGS CAN ALSO BE PLACED VERTICALLY

ABOVE YOU FIND A DESCRIPTION OF PLACING SUB DRAWINGS FROM THE LEFT TO THE RIGHT.

PLEASE NOTE THAT SUB DRAWINGS CAN ALSO BE PLACED VERTICALLY FROM THE TOP TO THE BOTTOM.

INSERTING VARIABLES IN SUB DRAWINGS, PAGES AND PROJECTS

When a sub drawing contains variables, you can for instance control symbol article numbers, symbol datafields e.g. containing motor data, or free texts placed under electrical circuits. — See e.g. “Manual placing of sub drawings” on page 247.

To do this you must:

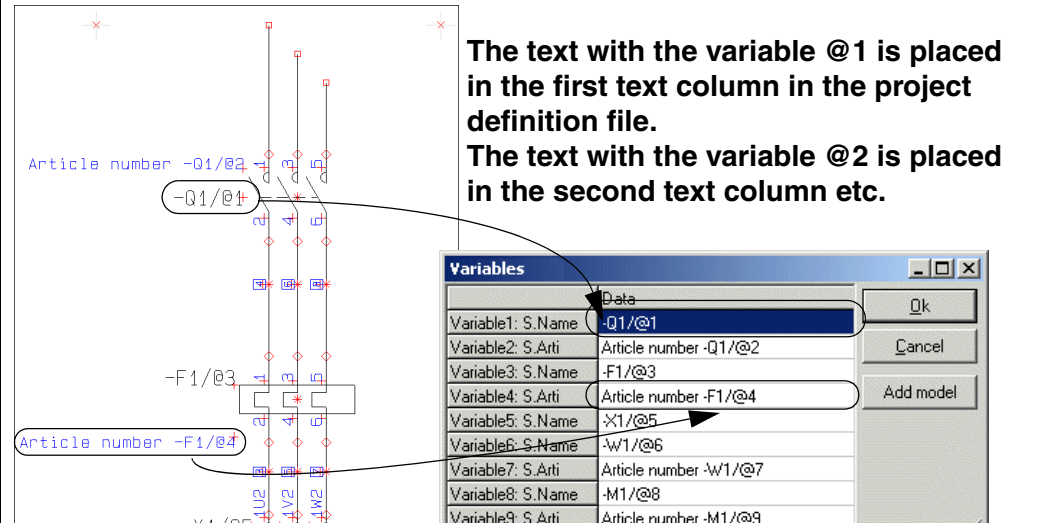
- 1) Insert *variables* in the sub drawing, as described in “Inserting variable markers on a sub drawing (or on a drawing template)” on page 269.
- 2) Create a *variable list*, as described in “Creating a variable list for a sub drawing / drawing template” on page 271.

When applying sub drawings for automatic project generation, you can control *all* texts and names in the project. — See the **Project generator** manual.

Inserting variable markers on a sub drawing (or on a drawing template)

When you wish to create variables, you do this by inserting so-called *variable markers* in the project. All data can be controlled via these variables.

To insert a variable marker, you write */@1* behind the first text in the sub drawing / project, that you wish to turn into a variable. Then you write */@2* behind the next, then */@3*, and so on, until all variables in the sub drawing have been specified.



The diagram shows an electrical circuit with several components labeled with variable markers. The labels are: "Article number -Q1/@1", "-Q1/@1", "-F1/@3", "Article number -F1/@4", "-X1/@5", "L1", "L2", "L3", "L4", "L5", "L6", "L7", "L8", "L9", "L10", "L11", "L12", "L13", "L14", "L15", "L16", "L17", "L18", "L19", "L20", "L21", "L22", "L23", "L24", "L25", "L26", "L27", "L28", "L29", "L30", "L31", "L32", "L33", "L34", "L35", "L36", "L37", "L38", "L39", "L40", "L41", "L42", "L43", "L44", "L45", "L46", "L47", "L48", "L49", "L50".

The **Variables** dialog box shows the following list:

Variable	Data
Variable1: S.Name	-Q1/@1
Variable2: S.Arte	Article number -Q1/@2
Variable3: S.Name	-F1/@3
Variable4: S.Arte	Article number -F1/@4
Variable5: S.Name	-X1/@5
Variable6: S.Name	-W1/@6
Variable7: S.Arte	Article number -W1/@7
Variable8: S.Name	-M1/@8
Variable9: S.Arte	Article number -M1/@9

The text with the variable @1 is placed in the first text column in the project definition file.
The text with the variable @2 is placed in the second text column etc.



CREATE A VARIABLE LIST

WHEN YOU HAVE PLACED THE RELEVANT VARIABLES ON THE PAGE, CHOOSE **TOOLS => CREATE VARIABLE LIST IN (SUB) DRAWING** TO CREATE A VARIABLE LIST FOR THE SUB DRAWING (OR DRAWING TEMPLATE).

When you right-click on the page and choose **Variables**, you can subsequently see the created variables and edit them, if you wish to.

— Please also see “Editing variables for inserted sub drawings” on page 249 and “Manual placing of sub drawings” on page 247.

Variables specifying parts of a project text

A variable can also be inserted so that it is only *part* of an otherwise constant project text. This is specified by placing the variable between the characters "<" and ">".

When for instance you write **-M</@1>**, and the variable **@1** is assigned the value **7**, this results in the text **-M7**.

This can for instance be useful if you are using a system where several symbols in a sub drawing must always have the same number. If for instance the symbols in the sub drawing shall be named **-F1**, **-Q1**, **-W1** and **-M1** one place, and **-F2**, **-Q2**, **-W2** and **-M2** in the next, this can be controlled using a single variable: **-F</@1>**, **-Q</@1>**, **-W</@1>** and **-M</@1>**.

DEFAULT VALUE FOR INSERTED VARIABLES

When an inserted variable has a text between the "<"-character and a "/"-character, this is regarded as the default value for the variable.

This can for instance be applied when creating project definition files, so that you can make it clear to yourself what to insert at this exact position in the file. If for instance you write **Control of <MOTOR/@1>**, the variable **@1** will get the value **MOTOR** in the project definition file.

Sub drawings with model data

If you wish to create different *sets* of variable values for a sub drawing, so that you can choose between different *models* when placing the sub drawing, please read "Creating Model data for sub drawings" on page 272.



VARIABLES ON ORDINARY PROJECT PAGES

YOU CAN ALSO INSERT VARIABLES ON ORDINARY PROJECT PAGES.

Variables for Page title and Page data

On drawing templates — and in ordinary project pages — you can also insert variables on all **Page data** datafields, and in the **Page title**.

When you have inserted these variables in the pages, and have executed **Tools => Create variable list**, you can choose **Edit =>**

Page data	Page setup	Page remarks
Page title:		
Control and Signal Circuit diagram/@1		
<input checked="" type="checkbox"/> Include this page in Table of Contents.		
Page data		
Drawing no.		
Engineer		/@2
Approved by:		
Approved date:		

Variables. Here the created variables for the page will always be placed in the first section:

	Section1	Section2	Section3
Variable1	Control and Signal Circuit diagram/@1	-X9	-X10
Variable2	/@2	-W9	-W10
Variable3		4022903075387	3389110586411
Variable4		-F9	-F10
Variable5		4022903085584	4022903085584
Variable6		-M9	-M10
Variable7		1723410405	1723410406

Variables for Page article data - including the page reference

It is also possible to create variable for **Page article data** for pages (**Name**, **Type**, **Article no** and **Function**) on drawing templates for automatic project generation via Excel.

When you wish to be able to control the page reference designation, place a variable marker in the **Name** field in the **Page article data** dialog box for the page.

You enter the dialog box **Page article data** by right-clicking on the page, and then choose **Page article data**.

Variables on Symbol datafields

If you are using symbols with specially created symbol datafields — see “Extra symbol Datafields in symbols” on page 691 —, you can also insert variables on these.

Creating a variable list for a sub drawing / drawing template

When you have placed the relevant variables on the page, choose **Tools => Create variable list in (Sub) drawing**. You have now created a variable list.

Click **OK**.

CREATING MODEL DATA FOR SUB DRAWINGS

You can use the same sub drawing for representing different *models* for e.g. a motorstarter circuit. Hereby you use one single drawing, but attach different article data / texts to the symbols in the sub drawing.

The same sub drawing can therefore be used for e.g. a 55 kW and a 250 kW motorstarter circuit.



VIDEO ABOUT SUB DRAWINGS WITH MODEL DATA

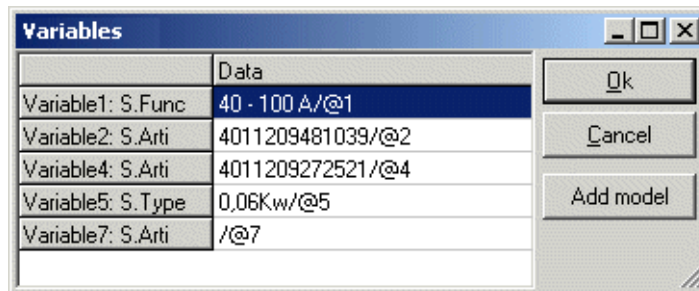
ON WWW.PCSCHMATIC.COM UNDER **VIDEOS**, YOU CAN SEE A VIDEO ABOUT HOW TO WORK WITH SUB DRAWINGS WITH MODEL DATA.

To create models for a sub drawing, do this:

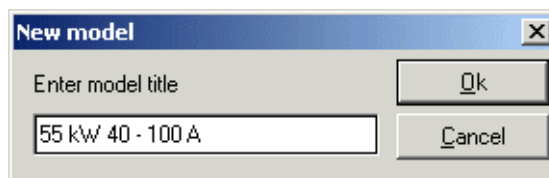
- 1) Draw the sub drawing, as described in “Creating sub drawings for manual placing” on page 255 or “Creating sub drawings by Copying from projects” on page 256. — If the sub drawing shall be used for *drag’n draw*, please also see “Preparing Sub drawings for Drag’n Draw / Aut. placing” on page 265.
- 2) Insert variable markers on the texts in the sub drawing, that are different from model to model. — See “Inserting variables in Sub drawings, Pages and Projects” on page 269.

Creating models for a sub drawing

- 3) While you are working on the sub drawing (in the **Edit sub drawing** mode): Right-click on the page, and choose **Variables**:

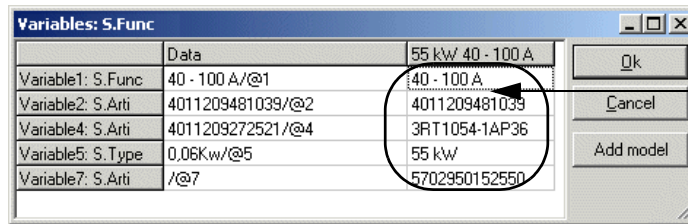


- 4) In the dialog box **Variables**: Click on **Add model**:



- 5) In the dialog box **New model**: Type in the name on the new model you wish to create, and click **OK**.
- 6) In the dialog box **Variables**: Double-click in the fields for the different variables in the model, and type in the data for the model — or right-click on the

field and select **Database**, and you enter the **Database menu**, from which you select the desired component:

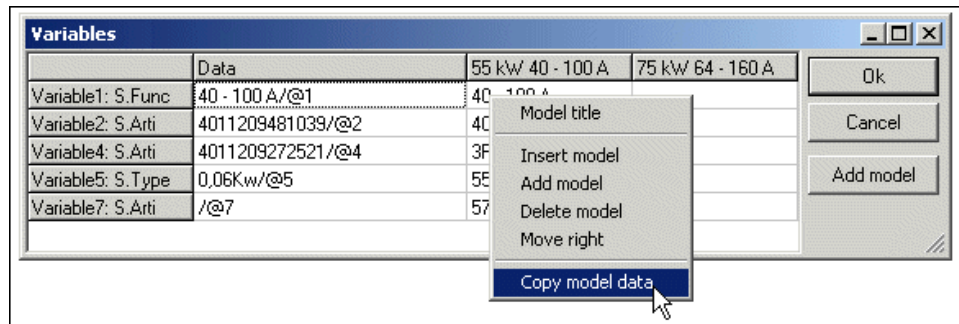


Double-click in the fields, and type in the data

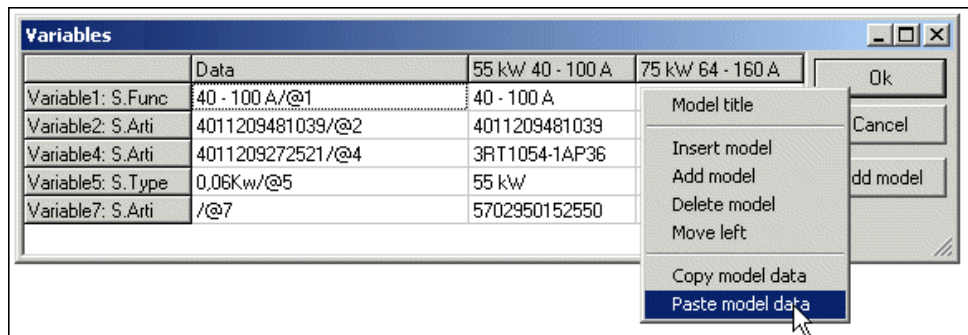
Creating a new model by copying data

7) Click on **Add model** and type in a title for the new model.

8) Right-click on the first created model, and choose **Copy model data**:

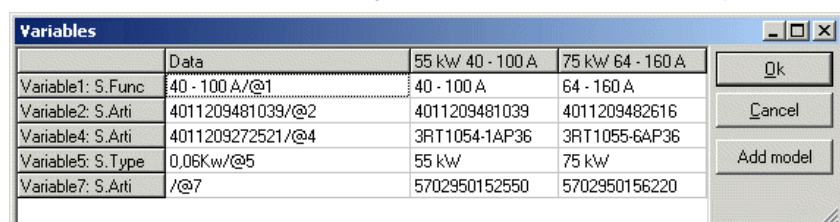


9) Right-click on the new model, and choose **Paste model data**:



10) Answer **OK** to overwrite the model data for the new model (do *not* select to insert the title), and the data are inserted as data for the new model.

11) For the text fields you wish to edit: Double-click on the field, and edit the contents. — Or right-click on the field and select **Database**, and you enter the **Database menu**, from which you select the desired component.



12) Click **OK**.

You have now created model values for a sub drawing. The created sets of model values, are the model values you can choose from, when the sub drawing is placed in a project. — See “Manual placing of sub drawings” on page 247, “Inserting sub drawings: With model data” on page 248 in particular.



LINE ARTICLE DATA CAN ALSO CONTAIN VARIABLES

LINE ARTICLE DATA IN SUB DRAWINGS CAN ALSO CONTAIN VARIABLES. THIS APPLIES FOR BOTH DRAG’N DRAW AND IN CONNECTION WITH AUTOMATIC PROJECT GENERATION.

Creating a model variable

For use in automatic project generation, you can create a so-called *model variable*, so that you via the *project definition file* can specify which of the models in the sub drawings to insert.

Read more about this in "Create model variable" in the **Project generator** manual.

Model data: Options when right-clicking

When you right-click in the dialog box **Variables** when creating model data, you get the following options:

Database: Gives you the option of choosing components from the database. Appears when right-clicking on an **S.Article** or a **S.Type** variable.

Model title: You can edit the title for the selected model.

Model variable: See “Creating a model variable” on page 274.

Insert model: Inserts a new model where you have clicked.

Add model: Adds a new model.

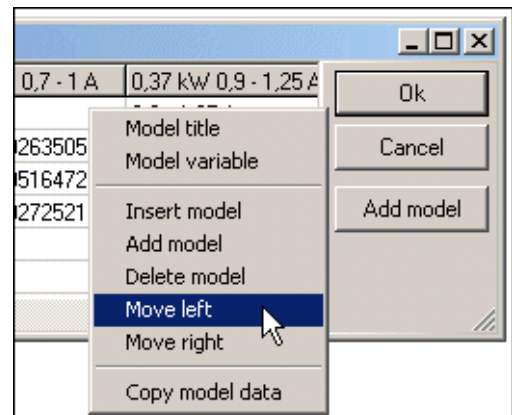
Delete model: Deletes the selected model.

Move left: Moves the selected model one position to the left. — See also “Drag and drop Models in the dialog box” on page 274.

Move right: Moves the selected model one position to the right. — See also “Drag and drop Models in the dialog box” on page 274.

Copy model data: Copies the values for the selected model — see “Creating a new model by copying data” on page 273.

Insert model data: Inserts the copied model values — see “Creating a new model by copying data” on page 273.



Drag and drop Models in the dialog box

To move a model to another position in the dialog box, you can *click* on the title of the model, and *drag* it to another position in the dialog box.

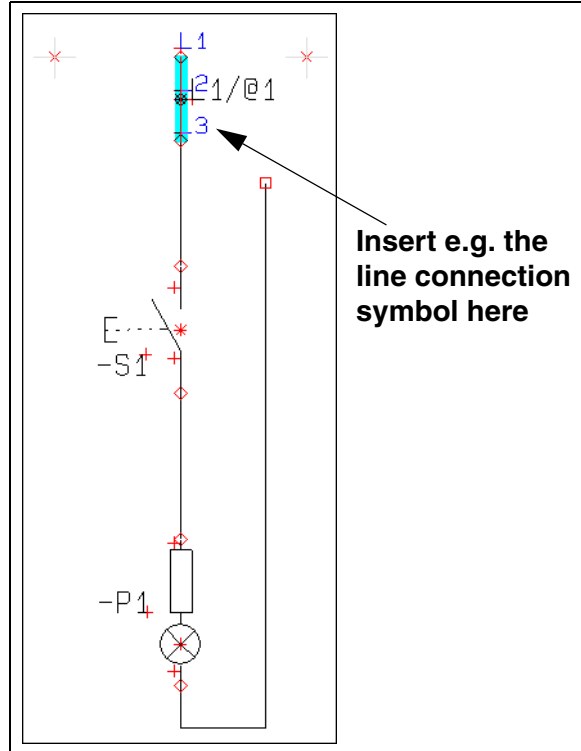
INVISIBLE AREAS AND SUB DRAWINGS

When inserting sub drawings, you can subsequently make parts of these sub drawings invisible, as described in “Making objects invisible” on page 62.

SUB DRAWINGS WITH INTELLIGENT LINE CONNECTION

To create sub drawings with intelligent line connection, you must use a line connection symbol, as described in “Creating Line Connection symbols” on page 709.

This line connection symbol can for instance be placed as displayed in the figure below.



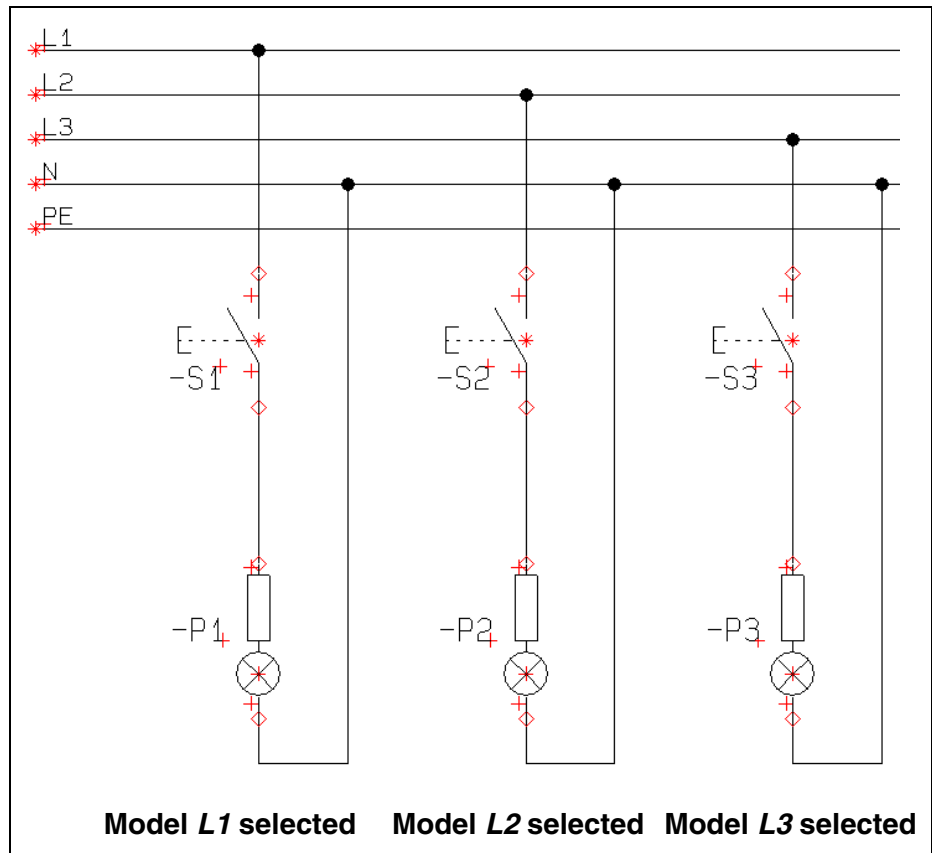
Controlled via variable marker in the symbol name

To control the connection, you must insert a variable marker in the symbol name. When the variable in the name gets the same value as one of the connection points — e.g. **3** —, the wire, which is connected to the symbol, will connect to where connection point **3** is placed. It can therefore be controlled if for instance a wire in a sub drawing connects to **L1**, **L2** or **L3**.

This aspect can therefore be different depending on which *model* of the sub drawing you choose. Below the models **L1**, **L2** and **L3** have been created:

Variables	Data	L1	L2	L3
Variable1: S.Name	L1/@1	L1	L2	L3

When you place the sub drawing in a project, you can therefore get the following result by placing the three different models:



When the sub drawing is placed on the project page, the line connection symbol is converted to an ordinary line, and is therefore no longer a symbol.

RENAME SYMBOLS

When you select **Tools => Rename symbols**, you enter the dialog box **Rename selected symbols**, from which you can rename the used symbols in the project.

THIS CHAPTER DESCRIBES	
—	GENERAL INFO ON THE DIALOG BOX RENAME SELECTED SYMBOLS(P. 278)
—	SELECTING SYMBOLS IN THE DIALOG BOX(P. 280)
—	DIRECTION FOR RENUMBERING SYMBOLS(P. 281)
—	SPECIFY START VALUE(P. 282)
—	SHOW NEW SYMBOL NAMES(P. 282)
—	NAMING WHEN RENUMBERING TERMINALS(P. 282)
—	INSERTING SYMBOL NAMES OR CONNECTION NAMES(P. 284)
—	SORTING ON COLUMN CAPTIONS(P. 285)

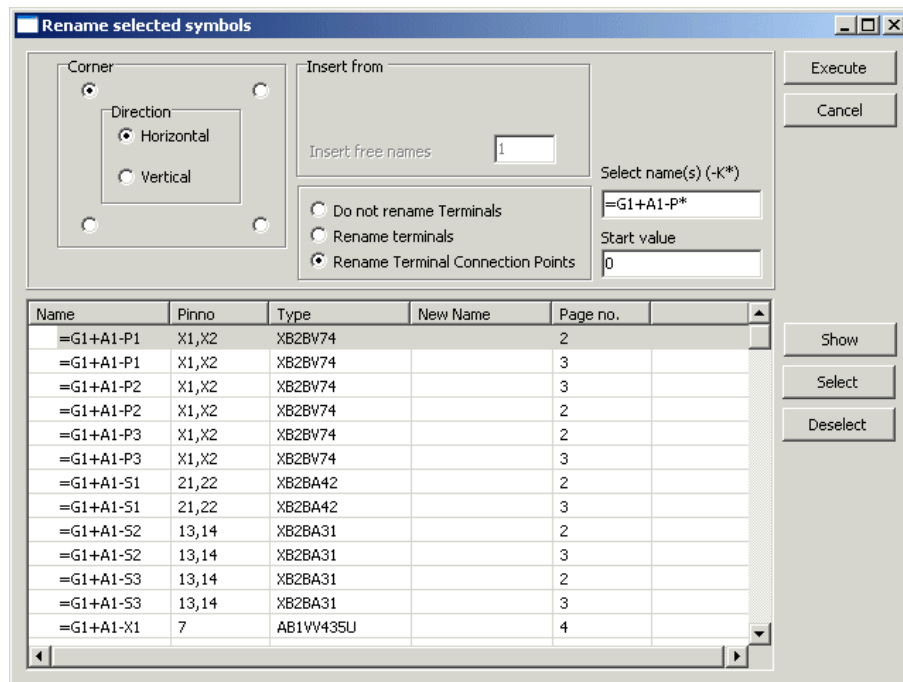


INSERT SYMBOL / CONNECTION NAMES

IT IS ALSO POSSIBLE TO RIGHT-CLICK ON A SYMBOL IN THE PROJECT, AND SELECT **INSERT NAME**. — READ MORE ABOUT THIS IN “INSERT NEW SYMBOL NAMES OR CONNECTION NAMES” ON PAGE 284.

GENERAL INFO ON THE DIALOG BOX RENAME SELECTED SYMBOLS

When you select **Tools => Rename symbols**, you enter the dialog box **Rename selected symbols**:



When you wish to rename symbols in the project, do this:

- 1) Select which symbols to rename — see “Selecting symbols in the Rename symbols dialog box” on page 280.
- 2) Specify a direction for renaming the symbols — see “Direction for renaming symbols” on page 281.
- 3) Specify how to rename terminals (if any) — see “Naming when renaming terminals” on page 282.
- 4) Specify a start number for the renamed symbols, see which names this will result in, and execute the renaming — see “Specify Start value for Renaming symbols” on page 282.

The function **Rename selected symbols** can also be handy when you wish to keep control of the symbol names when merging projects.



WORKS ONLY WHEN USING FREE NAME FORMAT FOR SYMBOLS

THIS FUNCTION ONLY WORKS IF YOU USE FREE NAMING FORMAT FOR THE SYMBOLS IN THE PROJECT — THIS MEANS THAT YOU DO *NOT* USE CURRENT PATH NUMBERING.

IF YOU DO NOT USE FREE NAMING FORMAT, THE FUNCTION CAN ONLY BE USED FOR RENAMING TERMINALS.

Which symbol names are renamed

Only symbol names ending with a number, are renamed/renumbered. If a symbol name contains a number *before* a letter, this number is not changed.

Undo renumbering



Please note that it is possible to undo the renaming by using the **Undo** function *after* closing the dialog box **Rename selected symbols**.

SELECTING SYMBOLS IN THE RENAME SYMBOLS DIALOG BOX

When you click on **Execute**, all symbols listed in the dialog box **Rename selected symbols** are renamed.

You can select which symbols to rename in three different ways:

1. Selecting/deselecting in the list

When you select symbols in the list, you can use the <Shift> and/or the <Ctrl> key, as you normally do in Windows, and e.g. in the **Object Lister**. — See “Selecting objects in the Object Lister” on page 298.

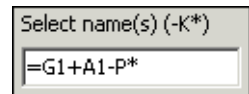
Name	Pinno	Type	New Name	New Pinno	Page no.
=G1+A1-P2	X1,X2	XB2BV74			3
=G1+A1-P2	X1,X2	XB2BV74			2
=G1+A1-P3	X1,X2	XB2BV74			2
=G1+A1-P3	X1,X2	XB2BV74			3
=G1+A1-S1	21,22	XB2BA42			2
=G1+A1-S1	21,22	XB2BA42			3
=G1+A1-S2	13,14	XB2BA31			2
=G1+A1-S2	13,14	XB2BA31			3
=G1+A1-S3	13,14	XB2BA31			2
=G1+A1-S3	13,14	XB2BA31			3
=G1+A1-X1	7	AB1VV435U			4
=G1+A1-X1	9	AB1VV435U			4
=G1+A1-X1	8	AB1VV435U			4

The selected symbols can then either be selected or deselected:

- If you click on **Select** in the right-hand side of the dialog box: The selected symbols are now the only symbols in the list.
- If you click on **Deselect** in the right-hand side of the dialog box: The selected symbols are deleted from the list. (You can also press the <Delete> button to delete the selected symbols from the list).

2. Selecting by specifying symbol names

In the field **Select name(s)** you can type in which symbols you wish to select, among the names already displayed in the list.



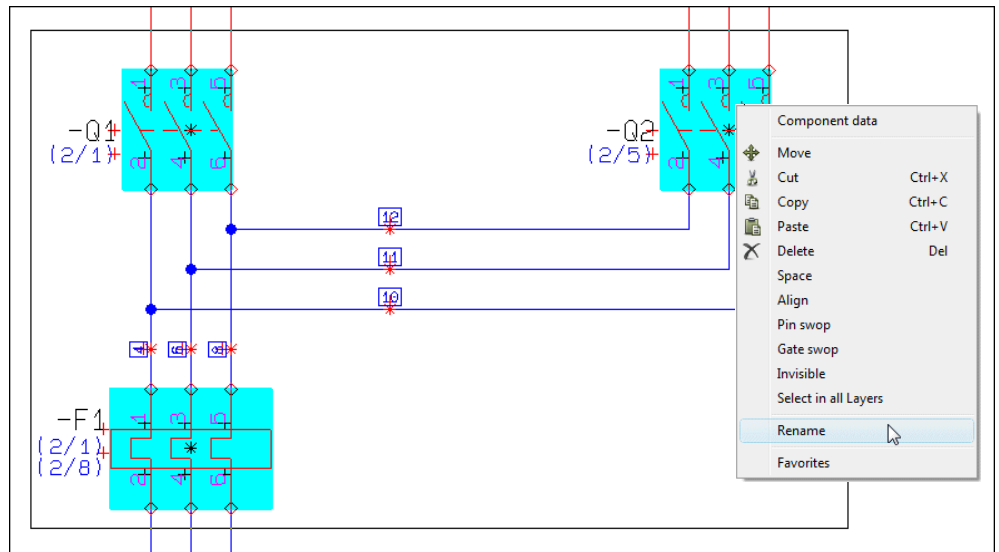
Then click on **Select** in the right-hand side of the dialog box. Now only symbols fulfilling the entered criterion are displayed.

3. Selecting project symbols by right-clicking

You can also select multiple symbols in the project — *before* entering the dialog box **Rename selected symbols**:

- 3a) Select the symbols by dragging a window over them with the mouse — if necessary please see “Selecting objects” on page 49, in particular “Selecting more objects of the same kind in one window” on page 49.

3b) Right-click and select **Rename** (see the figure below) — or select **Tools => Rename symbols**.



3c) You now enter the dialog box **Rename selected symbols**, where only the selected symbols are displayed:

Name	Pinno	Type	New Name	New Pinno	Page no.
=G1+A2-F1	1,2..(6)	B775			1
=G1+A2-Q1	1,2..(6)	LS15K11/230			1
=G1+A2-Q2	1,2..(6)	LS15K11/230			1



THE OTHER SYMBOLS FOR THE COMPONENT ARE ALSO RENAMED

PLEASE NOTE, THAT ONLY THE SELECTED SYMBOLS ARE DISPLAYED IN THE DIALOG BOX. WHEN A COMPONENT HAS MORE SYMBOLS IN THE PROJECT, WITH THE SAME NAME AS THE SELECTED SYMBOL, ALL SYMBOLS FOR THE COMPONENT ARE RENAMED IN THE PROJECT.

DIRECTION FOR RENAMING SYMBOLS

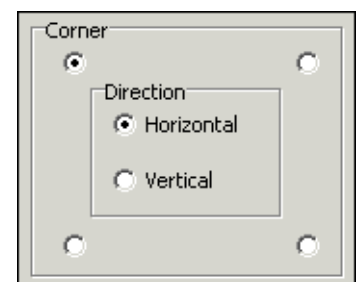
When you rename symbols, it is only their position in the project, that determines which names the symbols are assigned.

The direction for the renaming always starts from the first project page, and ends at the last project page.

How the renaming is performed on the individual project page, can be specified in the upper-left corner of the dialog box **Rename selected symbols**.

Here you select one of the four corners, to specify from which corner to start the renaming.

Furthermore, you specify the numbering direction **Vertical** or **Horizontal**.

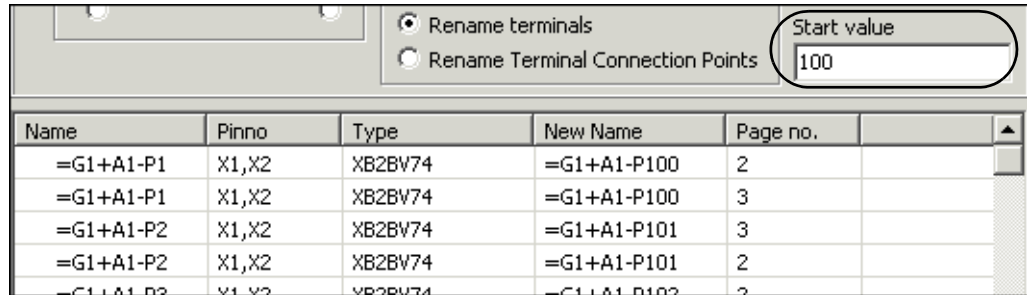


What is important for which number a component is assigned, is therefore the first time a symbol for the component appears in the project.

— The function and location aspects in reference designations (if any) remains unchanged under the renaming.

SPECIFY START VALUE FOR RENAMING SYMBOLS

When you wish to specify a start value for the renamed symbols, type in the number in the **Start value** field.



Hereby the new symbol names starts with e.g. the number 100.

SHOW NEW SYMBOL NAMES BEFORE RENAMING SYMBOLS

When you have selected which symbols to rename, have specified a direction for the renaming, and have specified a start value (optional), you are ready to execute the renaming.

Before you do this, you can click the **Show** button in the right-hand side of the dialog box. The new names are now displayed in the column **New Name** — see the figure above.

When you are satisfied with the new names, click the **Execute** button, and the listed symbols are renamed in the project.



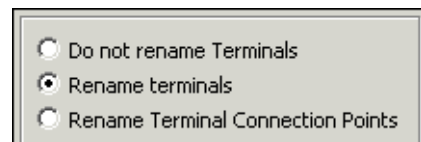
UNDO THE RENAMING

PLEASE NOTE THAT IT IS POSSIBLE TO UNDO THE RENAMING BY USING THE **UNDO** FUNCTION, AFTER CLOSING THE DIALOG BOX **RENAME SELECTED SYMBOLS**.

NAMING WHEN RENAMING TERMINALS

In the dialog box **Rename selected symbols** you can specify how to rename the terminals in the list.

You have the following options:



OPTIONS	ACTION
DO NOT RENAME TERMINALS	NO TERMINALS AND NO TERMINAL CONNECTION POINTS ARE RENAMED.

OPTIONS	ACTION
RENAME TERMINALS	ONLY THE TERMINALS ARE RENAMED. TERMINAL CONNECTION POINTS ARE <i>NOT</i> RENAMED.
RENAME TERMINAL CONNECTION POINTS	THE TERMINAL CONNECTION POINTS ARE RENAMED. TERMINAL NAMES ARE NOT RENAMED. — SEE “FULFILLING STANDARDS WHEN COPYING AREAS” ON PAGE 234.

— These options are precisely the same as the options for intelligent copying of areas. See more about this in “Intelligent copying of areas” on page 233.



RULES FOR RENAMING TERMINALS

TERMINAL CONNECTION NAMES ARE ONLY RENAMED WHEN THESE ARE NUMBERS. TERMINAL CONNECTION NAMES ARE ONLY RENAMED WHEN BOTH (/ALL) CONNECTION NAMES FOR A TERMINAL ARE IDENTICAL.

Example on renaming terminals

When you select e.g. **Rename Terminal Connection Points**, and click on **Show**, you can see the new connection names for the terminals in the column **New Pinno**:

Name	Pinno	Type	New Name	New Pinno	Page no.
=G1+A1-X1	7	AB1VW435U	=G1+A1-X1	6	4
=G1+A1-X1	9	AB1VW435U	=G1+A1-X1	8	4
=G1+A1-X1	8	AB1VW435U	=G1+A1-X1	9	4
=G1+A1-X1	7	AB1VW435U	=G1+A1-X1	6	3
=G1+A1-X1	8	AB1VW435U	=G1+A1-X1	9	3
=G1+A1-X1	6	AB1VW435U	=G1+A1-X1	7	3
=G1+A1-X1	6	AB1VW435U	=G1+A1-X1	7	4

— See also “Insert new Symbol names or Connection names” on page 284.

INSERT NEW SYMBOL NAMES OR CONNECTION NAMES

It is also possible to use the **Rename selected symbols** dialog box to insert new symbol names or connection names in the project:

1. Insert new symbol names

To insert e.g. three new symbol names starting from **-P2** in the project, do this:

- 1a) Select **-P2** in the project.
- 1b) Select **Functions => Rename symbols** — or right-click on **-P2** and select **Insert name**.
- 1c) In the dialog box **Insert name**: Click in the **Insert new names** field, and type in how many names to insert:

Insert from =G1+A1-P2
 From =G1+A1-P2 .. =G1+A1-P3
 To =G1+A1-P5 .. =G1+A1-P6
 Insert free names

- 1d) Click on **Execute**.

The symbol **-P2** is hereby renamed to **-P5**, and the subsequent **-P** symbols are renamed correspondingly. (**-P3** are renamed to **-P6** — etc.).

The symbol names **-P2**, **-P3** and **-P4** are now available in the project.

2. Insert new connection names on terminals

To insert e.g. five new connection names in terminal **-X1**, starting from connection **4**, do this:

- 2a) Select the terminal which has the connection **4** in the project.
- 2b) Select **Functions => Rename symbols**. (Or right-click directly on the terminal symbol and select **Insert name**).
- 2c) In the dialog box **Insert name**: Click in the **Insert free names** field, and type in how many connection names to insert:

Insert from =G1+A2-X1
 From =G1+A2-X1:4 .. =G1+A2-X1:6
 To =G1+A2-X1:9 .. =G1+A2-X1:11
 Insert free names

Terminal Name
 Terminal connection name

- 2d) Click on **Execute**.

The new connection names are now inserted for the terminal, according to the same principle as described in “Insert new symbol names” on page 284.

SORTING ACC. TO COLUMN NAMES IN THE DIALOG BOX

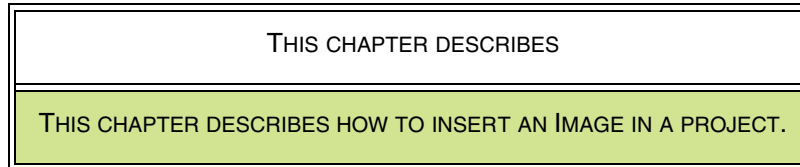
When you open the dialog box **Rename selected symbols**, the information in the dialog box are sorted by symbol name, but if you wish to e.g. sort according to type, you just click on the column caption **Type**.

When you wish to sort in reverse order, just click on the column caption again. The sorting is alphabetical/numerical.

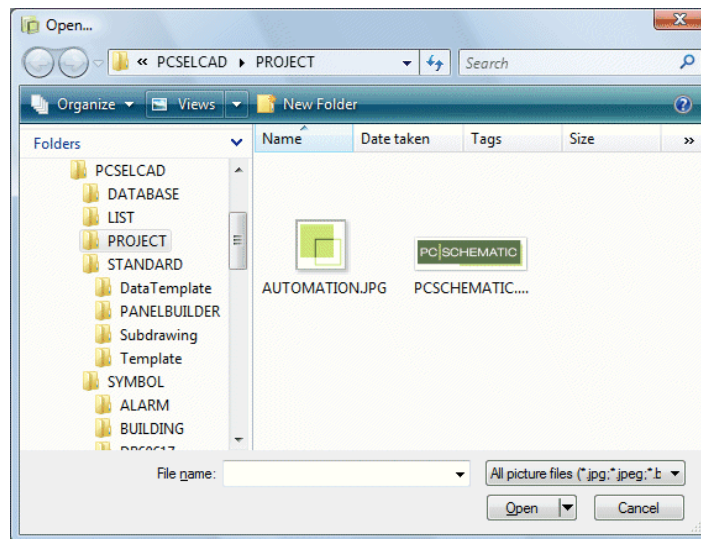
All columns can be sorted in both directions.

INSERTING IMAGES

If you want to insert an image on a page in your project, you choose **Insert => Insert Picture**. You can choose this function independent from which type of drawing objects you are working with.



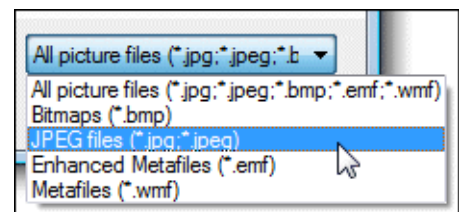
You now enter the dialog box **Open**, where you determine which file to insert into the project, and click **Open**.



Here you can choose between inserting the image with or without a link. — See more about this on the figure above.

The file types you can open from here are either **.bmp** files, **.jpg** files, metafiles or enhanced metafiles.

If you need to insert other file types/image formats, it is possible that you can insert the image as an OLE object, which is described in “Inserting files from other programs / OLE” on page 647.



When you have opened the image, it is placed in the cross hairs, and you click where you want to place it on the page.

Move, delete, copy or rotate inserted image



If you want to select an image, you click the **Area** button, and then click on the reference point of the image — or, alternatively, drag a window around the reference point.

If you right-click on the selected image, this menu appear:



Here you determine to either move, copy, delete or rotate the selected image. Images can not be rotated.

Scaling an image

When you want to change the size of an image, do like this:

- 1) Double-click on the image
- 2) Point on one of the corners of the image, so that the mouse pointer changes to a two-way arrow



- 3) Press the mouse button, *drag* the image to the desired size, and release the mouse button
- 4) Click somewhere outside the image, to deselect the image

You can also scale an image by choosing **Edit => Scale**, as described in “Scaling areas” on page 241.



Survey and Reference Designations

This section describes functions, which gives you a survey of projects:

- The Explorer (p. 291)
- The Object Lister(p. 296)
- Find and Replace(p. 306)
- Design check..... (p. 310)
- Printing..... (p. 314)
- The Explorer as Project/Symbol menu (p. 321)
- Reference designations(p. 322)

THE EXPLORER WINDOW: THE PROJECTS TAB

The **Explorer** window contains two different tabs: The **Projects** tab and the **Sub-Drawings** tab. This chapter describes the **Projects** tab, as well as the general options in the **Explorer** window. The **SubDrawings** tab is described in “Sub drawings” on page 244.

In the **Projects** tab in the **Explorer** window, you can display all types of information about the active projects, and can navigate back and forth between the project pages in all of the open projects. Furthermore, you can edit Project data, Page Data and component data directly from the **Explorer** window.

THIS CHAPTER DESCRIBES	
— GENERAL SETTINGS FOR THE EXPLORER WINDOW	(P. 291)
— DISPLAYING AND HIDING INFORMATION IN THE PROJECTS TAB	(P. 292)
— EDITING PROJECT, PAGE AND ARTICLE DATA FROM THE PROJECTS TAB ...	(P. 292)
— SELECTING SYMBOLS FROM THE PROJECTS TAB	(P. 294)
— JUMPING BETWEEN PROJECT PAGES IN DIFFERENT PROJECTS	(P. 295)
— UPDATING THE PROJECTS TAB	(P. 295)
— CLOSING PROJECTS FROM THE PROJECTS TAB	(P. 295)

GENERAL SETTINGS FOR THE EXPLORER WINDOW

You can specify where to place the **Explorer** window, as explained in “Docking Windows and Toolbars” on page 24.

If you do not want to use the **Explorer** window, you can turn it off by choosing **Settings => Pointer/Screen**, and deselect **Explorer window**.

View Explorer window

View => Explorer window is a menu point which shows/hides the **Explorer** window, if **Explorer window** has been selected in **Settings => Pointer/Screen**. The function has the short-cut key <Shift+F12>, so that you can show/hide the **Explorer** window quickly.

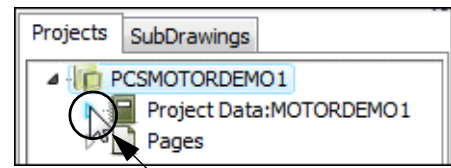
The Explorer window in Edit symbol mode

When you are designing symbols — being in the Edit symbol mode — you most likely do not need the **Explorer** window.

Therefore you can choose **Settings => Pointer/Screen => Explorer window** and subsequently deselect **Show in Edit symbol**, which specifies whether to show the **Explorer** window in the Edit symbol mode.

DISPLAY AND HIDE INFORMATION IN THE EXPLORER WINDOW

In the **Projects** tab in the **Explorer** window, you can display all the information attached to a specific object, by clicking the small triangle icon next to the object. — Please see the figure to the right.

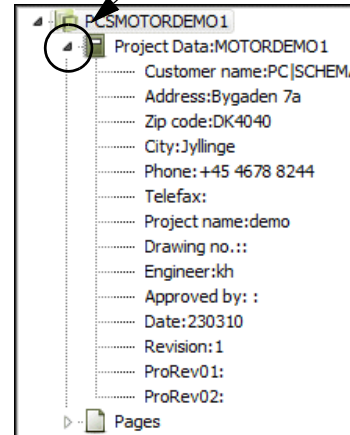


Click here to open

Click here to close

To stop displaying the extra information, click the triangle icon once again.

If for instance you click on the triangle icon in the **Project Data** line above, you see the project data for the project.



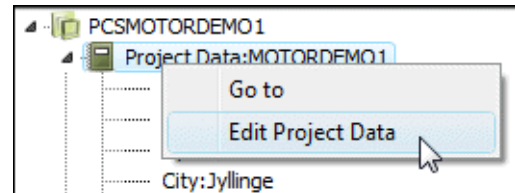
As you can see, the symbol in the **Project Data** line has now changed to a different triangle icon. When you click on this triangle, the project data datafields are hidden, and the **Explorer** window looks like the first figure again.

CHANGING DATA FROM THE PROJECTS TAB

From the **Projects** tab in the **Explorer** window you can change project data, page data and component data.

Editing Project data from Projects tab

To edit Project data from the **Projects** tab, right-click on the line **Project data**, and choose **Edit Project data**.



You then enter the dialog box **Project data**, from where you can edit the project data for the project. This dialog box is described in “Project and Page data” on page 111.

Make the changes and click **OK**.

Editing the contents of a single Project data datafield

When you want to edit the contents of a single Project data datafield — for instance **Customer name** — you right-click on the datafield in the **Explorer** window, and choose **Edit Customer name**. You then enter a dialog box where you make the changes and click **OK**.

Editing Page data and Project data

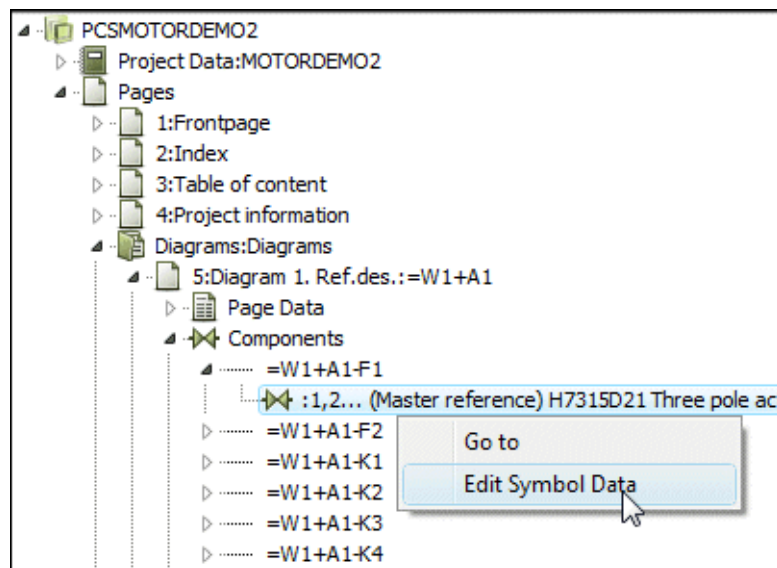
Page data datafields are changed directly from the **Projects** tab in the **Explorer** window, as described in “Editing the contents of a single Project data datafield” on page 292.

When you edit a Page data datafield, you also get the option of changing the contents of all Page data datafields of the chosen type in the entire project by choosing **Change on All Pages**.

The contents of Page data datafields can also be edited from the **Object Lister** (short-cut key [F7]), as described in “Editing Page data in the Object Lister” on page 304.

Editing Component data from the Projects tab in the Explorer window

When you want to edit the Component data for at symbol, this can be done by right-clicking on the symbol in the **Projects** tab in the **Explorer** window, and choosing **Edit Symbol Data**.



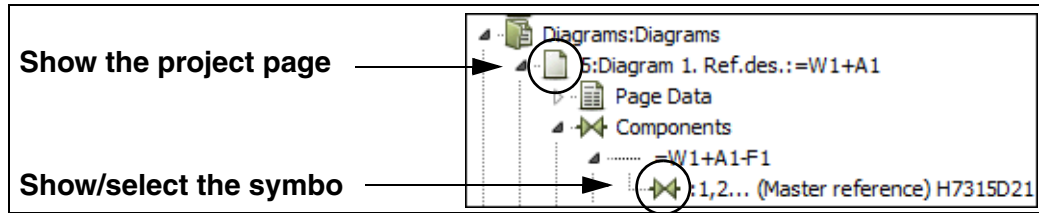
This brings you into the dialog box **Component data**, where you edit the component data and click **OK**.

You can also make changes in article data for cables and signals from the **Projects** tab in the **Explorer** window.

Component data and cable article data can also be changed directly from the **Object Lister** (short-cut key [F7]), as described in “Check and edit component data and cable data” on page 303.

SELECTING A SYMBOL ON A PAGE

Next to each symbol in the **Projects** tab in the **Explorer** window, you'll find a small symbol icon. When you click the icon, the symbol is selected in the project. When you click the icon for a **page**, the selected page is displayed:



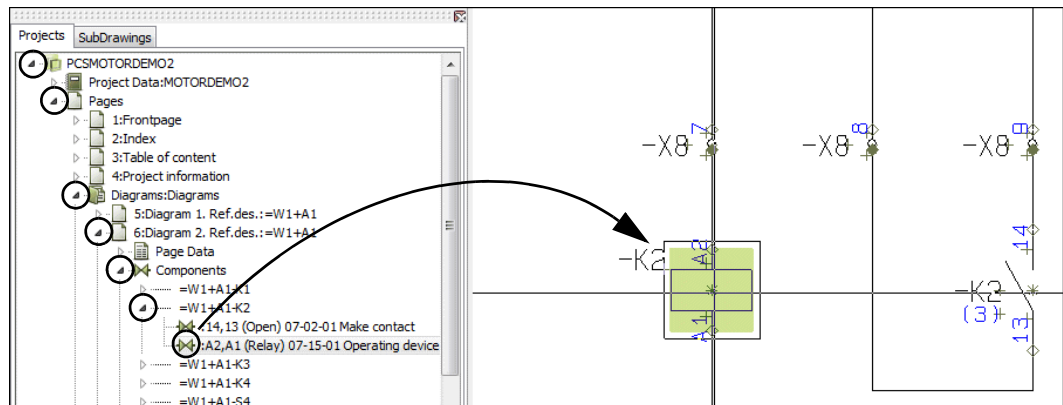
In the project **PCSMOTORDEMO2** you can for instance do this in the **Projects** tab: Click the triangle icon next to **PCSMOTORDEMO2** and click the triangle icon next to **Pages**. You now get a survey of all the pages in the project.

Now click the triangle icon next to the chapter **Diagrams**, and click the triangle icon next to the page **6:Diagram 2**. You can now see that you can get more information about **Page Data**, **Components**, **Cables** and **Signals** on the page.

Open **Components**, and then open **=W1+A1-K1**.

You can now see all the symbols on the page for the selected component — in this case two.

When you click on the symbol icon next to the relay symbol, the symbol is displayed/selected in the project, as displayed below:



When you then click on the symbol icon next to the make contact, the cross-hairs then points out the make contact for you.

The line you clicked on can contain information about the connection names for the symbol (**A2, A1**), the type of the symbol (**Relay**), the symbol file name (**07-15-01**) and the title of the symbol. Due to the size of the **Explorer** window, you will probably not be able to see all that information. If you let the mouse pointer rest a short while over the line, the entire line will be displayed.

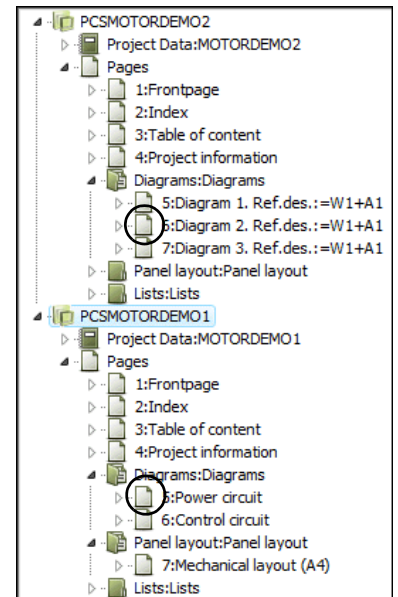
Please note, that you can do the same for **Cables** and **Signals**.

JUMP BETWEEN PAGES IN MORE THAN ONE PROJECT

When you have more than one project open at the time, you can find all the open projects in the **Projects** tab in the **Explorer** window.

When you open **Pages** for the projects, you can jump back and forth between the pages in the different projects, just by clicking on the page icon for the relevant project page.

By opening **Project Data** for the projects, you can for instance compare project data for the projects.

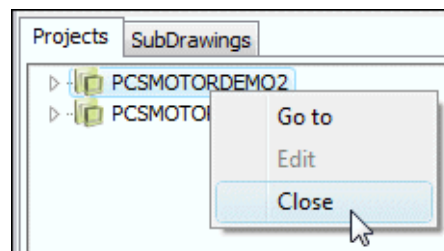


UPDATING THE EXPLORER WINDOW

When you delete a project page, which is displayed in the **Explorer** window, this page is also deleted from the **Explorer** window.

CLOSING PROJECTS FROM THE PROJECTS TAB

If you want to close a project from the **Projects** tab in the **Explorer** window, right-click on the project file name and choose **Close**.



You can also right-click on a project in the **Projects** tab in the **Explorer** window, and choose **Go to**. This activates the last page you worked on in the project.



THE OBJECT LISTER

To navigate quickly in your projects, you can use the **Object Lister**.

The **Object Lister** makes it possible for you to locate any given component in a project and/or information about the applied symbols/components.

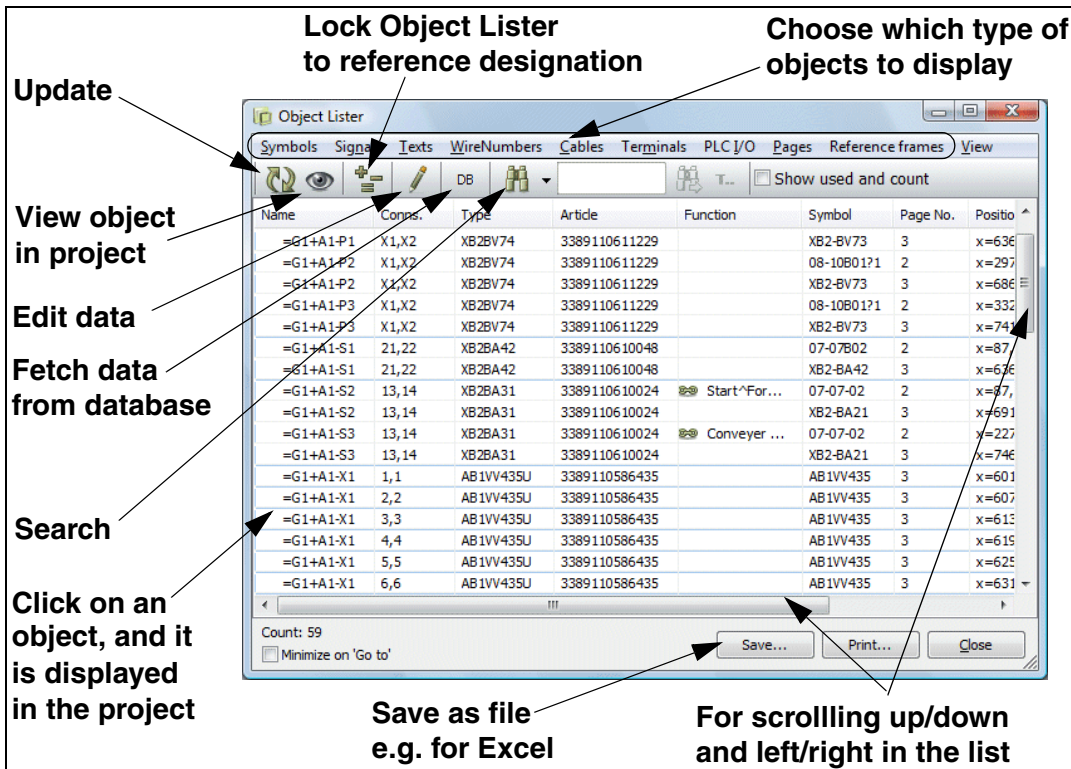
From the **Object Lister** you can also edit component data, PLC I/O data, page data and free texts for all of the project. Furthermore, you can print and create files containing selected information to for instance Excel.

— See also “The Explorer window: The Projects tab” on page 291 and “Find and Replace” on page 306.

THIS CHAPTER DESCRIBES	
— ON THE DIALOG BOX OBJECT LISTER	(P. 297)
— SELECTING OBJECTS VIA THE OBJECT LISTER	(P. 298)
— LOCKING THE OBJECT LISTER TO A REFERENCE DESIGNATION	(P. 298)
— THE SORTING ORDER OF DATA IN THE OBJECT LISTER.....	(P. 298)
— SELECTING OBJECTS	(P. 298)
— COUNTING ON SELECTED OBJECTS	(P. 299)
— DISPLAYING INFORMATION	(P. 300)
— PRINTING	(P. 300)
— SAVE TO FILE	(P. 301)
— CHOOSING NEW COMPONENTS IN THE DATABASE.....	(P. 301)
— SEARCHING.....	(P. 302)
— SHOWING THE NUMBER OF FOUND OBJECTS	(P. 302)
— EDITING SYMBOL AND CABLE DATA VIA THE OBJECT LISTER	(P. 303)
— EDITING PLC I/O DATA VIA THE OBJECT LISTER.....	(P. 303)
— EDITING PROJECT TEXTS VIA THE OBJECT LISTER.....	(P. 304)
— EDITING PAGE DATA VIA THE OBJECT LISTER	(P. 304)
— OBJECT LISTER FOR ONE COMPONENT	(P. 304)
— TEXT LINKS IN THE OBJECT LISTER.....	(P. 305)

ON THE OBJECT LISTER DIALOG BOX

To enter the Object lister you choose **Functions => Object Lister**, or press the short-cut key [F7]:



By clicking in the upper-left corner you determine whether to find **Symbols, Signals, Texts, Wire numbers, Cables, PLC I/O, Pages** or **Reference frames**. Please note, that the dialog box is not filled out until you have chosen one of these options.

If you, as above, have chosen **Symbols**, all project symbols will appear in alphabetical order — independently of which page you are working on at present.



THE OBJECT LISTER REMAINS ON SCREEN

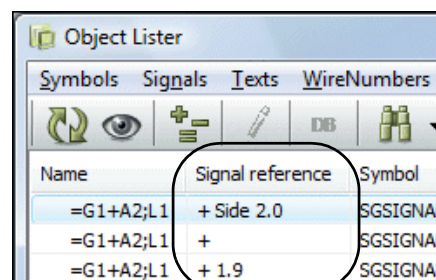
THE **OBJECT LISTER** REMAINS ON SCREEN WHILE YOU ARE WORKING WITH THE PROJECT. WHEN THE **OBJECT LISTER** DISPLAYS DATA, THE PROGRAM UPDATES THESE DATA AUTOMATICALLY, WHEN YOU MAKE CHANGES IN THE PROJECT.

— THE **OBJECT LISTER** CAN BE SET TO BECOME TRANSPARENT, WHEN YOU ARE WORKING ON THE DRAWING AREA, WHILE YOU LET THE **OBJECT LISTER** REMAIN ON THE SCREEN — SEE “**POINTER/SCREEN**” ON PAGE 815.

Signals in the Object Lister

When you select **Signals** in the **Object Lister**, the signal references are also displayed:

+ before the reference, means that the signal is **With reference**, and displays **From/To**.



* before the reference, means that **With reference** is not selected, and displays **All**.

- before the reference, means that **With reference** is not selected.

SELECTING OBJECTS DIRECTLY FROM THE OBJECT LISTER

If for instance you have chosen to work with **Symbols**, and wish to find the symbol **=G1+A2-Q2** in the project, scroll down to it and double-click on **=G1+A2-Q2**. This automatically places you on the page in the project where the symbol is located, and the symbol will be selected.

This also applies for other types of objects in the **Object Lister**.

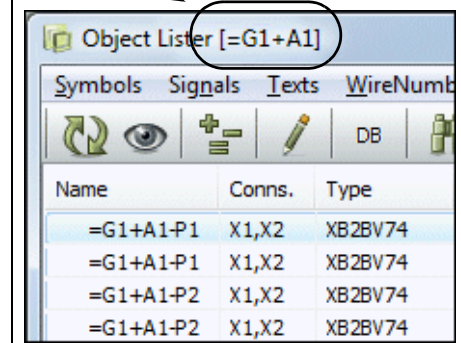
LOCKING THE OBJECT LISTER TO A REFERENCE DESIGNATION



When you click on the **Lock list to reference designation** button, you enter the dialog box **Reference designation**, from which you can select which reference designation to display in the **Object Lister**.

Subsequently, only objects with the selected reference designation are displayed in the **Object Lister**. In the title bar of the **Object Lister**, you can see the selected reference designation.

Only objects with this reference designation is displayed



Show all objects with any reference designation again



To make the **Object Lister** displays objects with any project designation again, click on the **Lock list to reference designation** button, and select **Top node = none** for both the function and the location aspect in the dialog box **Reference designations**.

THE SORTING ORDER OF THE DATA IN THE OBJECT LISTER

Initially the objects will be sorted by symbol name (page name and text contents respectively), but if you want them sorted by for instance article, click on the column heading **Article**.

When you wish to see the objects in reverse order, click the column heading again. The sorting order is alphabetical/numerical. You can sort all columns in both directions.

SELECTING OBJECTS IN THE OBJECT LISTER

Selecting objects in the Object Lister follows the common Windows guidelines:

Selecting a single object in the Object Lister

To select one single object in the **Object Lister**, just click on it.

Choosing a block of objects in the Object Lister

To select a block of objects listed in succession in the **Object Lister**, click on the first object in the block, and press down <Shift> while clicking on the last object in the block.

Selecting and deselecting single objects in the Object Lister

To select more than one object individually, press down <Ctrl> while clicking on the objects. When you wish to deselect a selected object, without deselecting the other selected objects, you must also press down <Ctrl> while clicking on the object.

Selected objects and Printing in the Object Lister

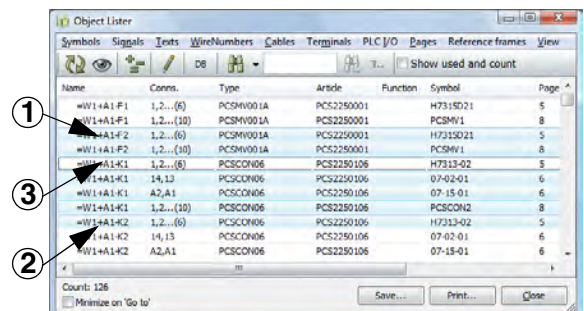
When you have selected more than one object when clicking on **Print** in the **Object Lister**, only the selected objects are printed. When only one object is selected, all of the list is printed. — See “Printing from the Object Lister” on page 300.

This also applies when you click on **Save** to save a file. — See “Save as File from the Object Lister” on page 301.

Example on choosing objects in the Object Lister

To select the selected objects in the **Object Lister** below, do the following:

- 1) Click on the first object in the list.
- 2) Press down <Shift>, and click on the last object in the list.
- 3) Press down <Ctrl>, and click on the object you wish to deselect.



The chosen objects can now for instance be printed, saved to a file, or the article data for the selected objects can be edited.

COUNTING ON SELECTED OBJECTS IN THE OBJECT LISTER

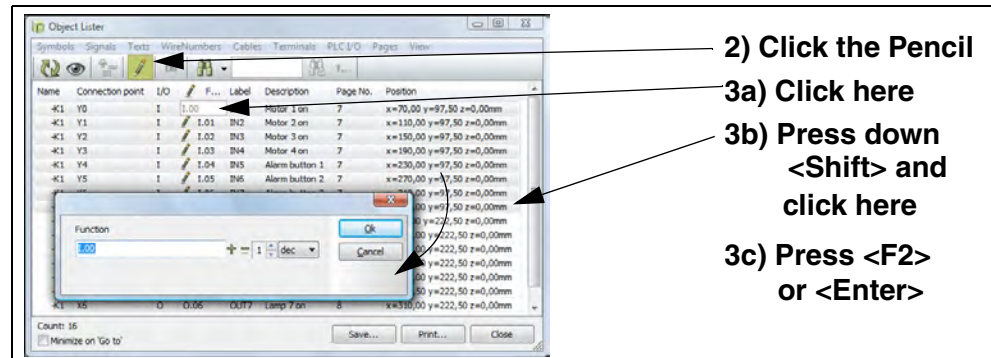
When you have selected more than one object in the **Object Lister**, it is possible to apply the automatic counting functions in the program for various of the object texts.

When for instance you wish to change the function texts (the PLC addresses) for the output on a PLC, do the following:

- 1) In the **Object Lister**: Click on **PLC I/O** — to work with PLCs.
- 2) Click on the **Pencil** — to edit texts in the **Object Lister**.



- 3) Select the relevant symbols — see “Selecting objects in the Object Lister” on page 298 —, and press <Enter> or <F2>:



- 4) You now get the option to change the selected texts automatically, as described in “Changing a text” on page 112. Click **OK**.

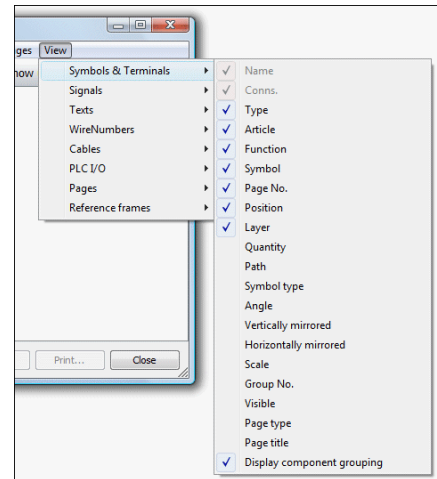
DISPLAYING INFORMATION IN THE OBJECT LISTER

Under the menu point **View**, you can specify which information to display in the **Object Lister** for the different types of objects.

If for instance you choose **View => Symbols**, you can see which information that is displayed for symbols in the **Object Lister**.

Viewing the different types of information can be selected/deselected by placing/removing a check mark in front of each type of information.

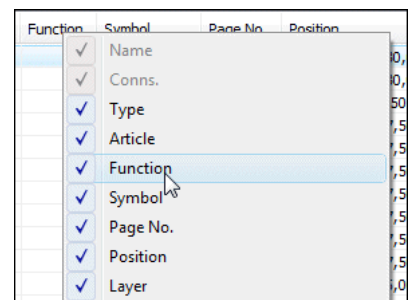
When a check mark is placed in front of a type of information which is dimmed, this specifies that this information can *not* be deselected in the **Object Lister**.



Removing a column from the Object Lister by right-clicking

When you right-click on a column heading in the **Object Lister**, a right-click menu appears, from which you can also specify not to display a column in the **Object Lister**.

In the example to the right there is right-clicked on the column heading **Function**, and a right-click menu appeared. When **Function** is deselected, this column is no longer displayed in the **Object Lister**.



PRINTING FROM THE OBJECT LISTER

When you have selected more than one object, only the selected objects are printed, when you click on **Print**. When only one object is selected, all of the list is printed. — Read more about selecting objects in “Selecting objects in the Object Lister” on page 298.

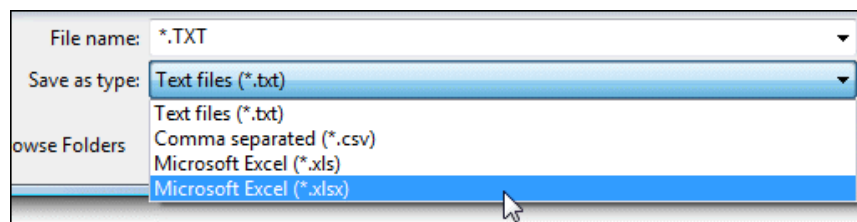
When you click on **Print**, you enter the dialog box **Print**, in which you choose printer and print the selected.

SAVE AS FILE FROM THE OBJECT LISTER

When you have selected more than one object, only the selected objects are saved, when you click on **Save**. When only one object is selected, all of the list is saved. — Read more about selecting objects in “Selecting objects in the Object Lister” on page 298.

When you click on **Save** you enter the dialog box **Save**. Here you can save the selected objects in three different ways:

- 1) **Text files (*.txt)**: The file is saved in column format.
- 2) **Comma separated (*.csv)**: The file is saved in comma separated format.
- 3) **Microsoft Excel (*.xls)**: The file is saved in Microsoft Excel format.



Select file type, type in a file name, and click **OK**.



EXCEL MUST BE INSTALLED

IT IS ONLY POSSIBLE TO SAVE IN EXCEL FORMAT WHEN EXCEL IS INSTALLED ON THE PC.

THIS IS BECAUSE **Automation** LETS EXCEL HANDLE THE SAVING. THIS WAY YOU ARE NOT DEPENDANT OF WHETHER **Automation** CAN HANDLE THE VERSION OF EXCEL INSTALLED ON YOUR PC.

CHOOSING NEW COMPONENTS IN DATABASE VIA OBJECT LISTER

This is only possible when working with **Symbols** or **Cables** in the **Object Lister**:



When you click the **DB** button, you enter the database, located at the article number of the chosen component. You can then change the component in the project by choosing another article number in the database.

Only the selected symbol(s) in the **Object Lister** have their data changed. — See “Selecting objects in the Object Lister” on page 298.

SEARCHING IN THE OBJECT LISTER



When you click on the **Find** button, you can search among the selected objects in the **Object Lister**.



Click on the **Down arrow**, and choose which type of text you wish to search on.

When for instance you choose **Type** this is marked in the column **Type**.

Conns.	Type	Article
1,2...(6)	PCSMV001a	PCS2250001
1,2...(10)	PCSMV001a	PCS2250001

Shows that you are searching on **Type**

Then type in the search text. While you are typing in the text, the **Object Lister** automatically jumps down to the first object in the list, that matches what you have typed in for the moment.



Click on the **Search again** button to find the next object in the list, that matches what you have typed in. When no objects matches what you have typed in, the Search again button is dimmed.

Automation can either search for texts that *begins with* what you have typed in, or just *contains* the text.



This depends on whether the **Begins with** button is activated, or the **Contains** button is activated. The button is placed to the right of the Find again button, and changes appearance between **Begins with** and **Contains** when you click on it.

SHOW USED AND COUNT

When you select **Show used and count**, each symbol name will only be displayed once in the list.

You can also see how many times each symbol is used in the project.

Name	Count
=W1+A1-F1	2
=W1+A1-F2	2
=W1+A1-K1	4
=W1+A1-K2	4
=W1+A1-K3	4

When you double click on a symbol name in the list, you get the same detailed information about the symbol as you can see in "Object lister for selected symbol" on page 304.

To return to the list of all project symbols, select **Show used and count** again.

CHECK AND EDIT COMPONENT DATA AND CABLE DATA



When you click on **Symbols** and then click the **Pencil**, you can correct the article data for the symbols directly from the **Object Lister**.

Name	Conns.	Type	Article
=W1+A1-F1	1,2...(6)	PCSMV001A	PCS2250001
=W1+A1-F1	1,2...(10)	PCSMV001A	PCS2250001
=W1+A1-F2	1,2...(6)	PCSMV001A	PCS2250001

If for instance you want to change the symbol type, click in the **Type** field for the relevant symbol, and make the change.

When you click on **Cables** you can make similar changes for **Cable data**.

See how to edit article data for more than one cable at the time in “Editing PLC I/O data via the Object Lister” on page 303.

EDITING PLC I/O DATA VIA THE OBJECT LISTER

To edit PLC I/O data in **Automation**, click on **PLC I/O** in the **Object Lister**.

You can then choose either **Used I/O** — which gives you all used PLC I/O symbols in the project — or **From reference symbols** — which gives you all mentioned connection points in the PLC reference symbols in the project.

Name	Connection point	I/O	Function	Label	Description	Page No.	Position
-K1	Y0	I	I.00	IN1	Motor 1 on	7	x=70,00 y=97,50 z=0,00m
-K1	Y1	I	I.01	IN2	Motor 2 on	7	x=110,00 y=97,50 z=0,00
-K1	Y2	I	I.02	IN3	Motor 3 on	7	x=150,00 y=97,50 z=0,00
-K1	Y3	I	I.03	IN4	Motor 4 on	7	x=190,00 y=97,50 z=0,00
-K1	Y4	I	I.04	IN5	Alarm button 1	7	x=230,00 y=97,50 z=0,00
-K1	Y5	I	I.05	IN6	Alarm button 2	7	x=270,00 y=97,50 z=0,00
-K1	Y6	I	I.06	IN7	Alarm button 3	7	x=310,00 y=97,50 z=0,00
-K1	Y7	I	I.07	IN8	Alarm button 4	7	x=350,00 y=97,50 z=0,00
-K1	Y8	O	O.00	OUT1	Lamp 1 on	8	x=70,00 y=222,50 z=0,00



When you click on the **Pencil**, you can start editing PLC I/O data directly from the **Object Lister**.

Searching for PLC I/O data is done as described in “Searching in the Object Lister” on page 302.

See how to edit PLC I/O data for more than one PLC symbol at a time in “Counting on selected objects in the Object Lister” on page 299.

PLC I/O data can be saved to a file — for instance in Excel format — as described in “Save as File from the Object Lister” on page 301.

CHECKING AND EDITING PROJECT TEXTS VIA THE OBJECT LISTER

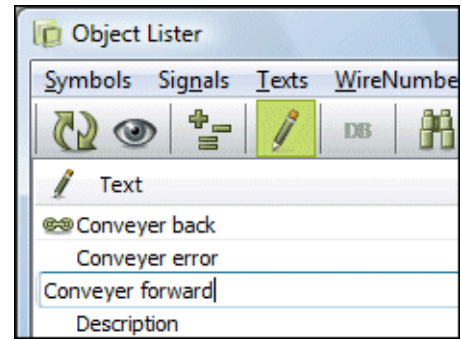
It is a good idea to check the project texts by clicking on **Texts**. If you need to make corrections, click the **Pencil**, and click in the text field you wish to edit.



The changes is transferred to the diagrams automatically.

When you want to jump to the text in the project, first deactivate the Pencil, then double-click on the text in the **Object Lister**.

— See how to edit more than one text at a time in “Counting on selected objects in the Object Lister” on page 299.

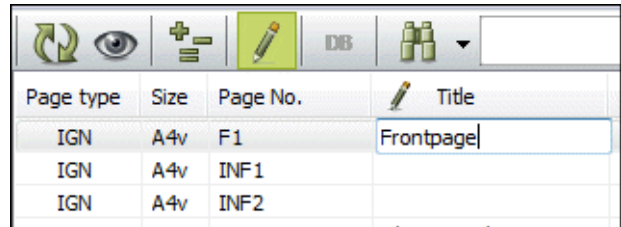


EDITING PAGE DATA IN THE OBJECT LISTER

When you click on **Pages**, you get a survey of all the project pages.



When you then click the **Pencil**, you get the option of editing the page titles.

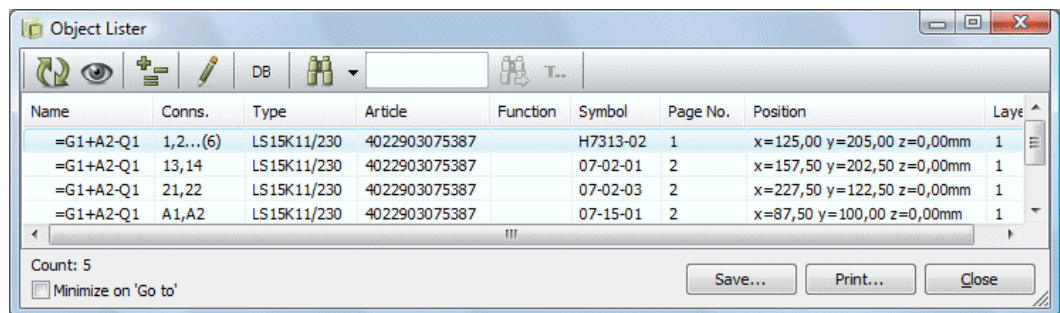
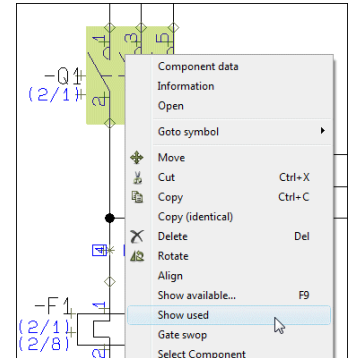


OBJECT LISTER FOR SELECTED SYMBOL

If you select a symbol when you are working in the project — for instance **-F1** — and press [F7], you will get a survey of all the places **-F1** is used in the project.

Alternatively you can right-click on a symbol, and choose **Show used**.

You then get the following survey of the symbol:



These functions are also available for signals and wire numbers.

Because the **Object Lister** remains on the screen, you can subsequently select another symbol in the project, and click the **Update** button in the **Object Lister**.



This updates the **Object Lister** with information from the component, you have selected now.

When you want the list on print, click on **Print**. Click **Save** to save it as a file.



Click on the **Pencil** to correct the component data, or click the **DB** button to find the symbol in the database.

TEXT LINKS IN THE OBJECT LISTER

Information about linked text is also displayed in the **Object Lister**.

— See “Text links” on page 128.

FIND AND REPLACE

To find objects in projects quickly, you either use the *Object Lister* — see “The Object Lister” on page 296 — or the *Find* function, which is described in this chapter.

The Object Lister gives you the possibility to locate components in the project and/or locate article data for the applied symbols/components. Also, you can change texts using the Object Lister.

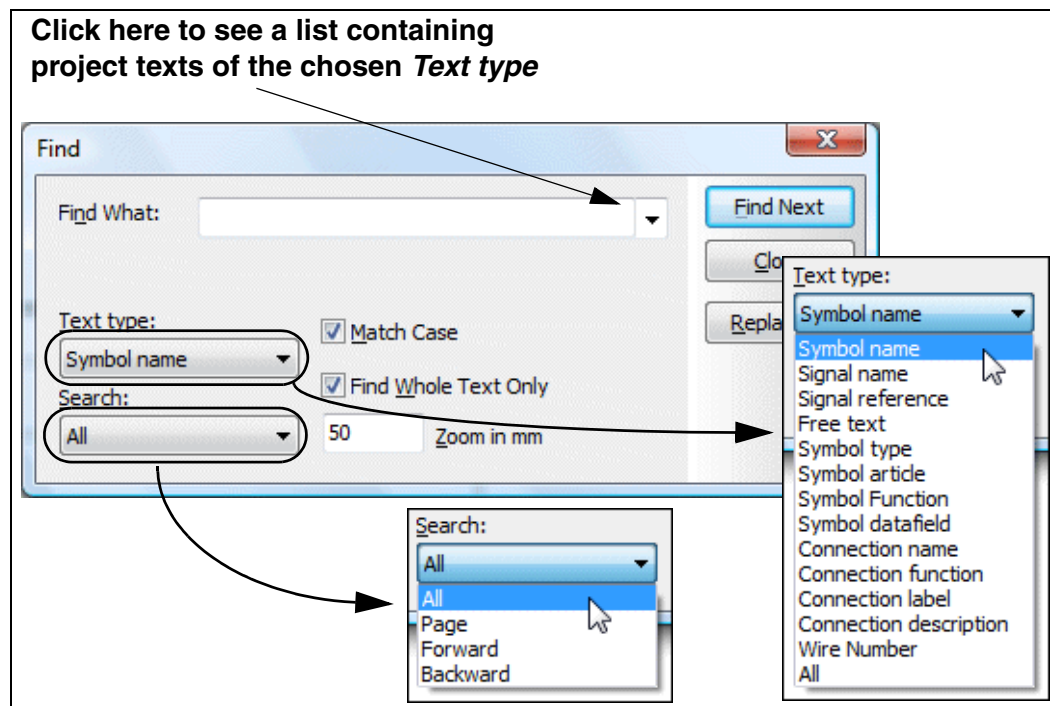
The Find function gives you the option to find symbol texts, and the replace function gives you the option to replace them as well.

THIS CHAPTER DESCRIBES	
— FIND SYMBOL TEXTS	(P. 306)
— REPLACE SYMBOL TEXTS.....	(P. 308)
— THE DIALOG BOX FIND AND REPLACE REMAINS ON SCREEN.....	(P. 308)
— NEGATED AND UNDERLINED TEXTS IN FIND AND REPLACE	(P. 309)

— Other ways to find *symbols* quickly are described in “Functions on Component Level” on page 220.

FINDING SYMBOL TEXTS

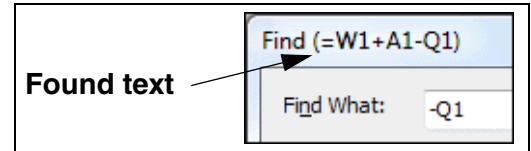
If you want to find symbol texts, you choose **Edit => Find** (short-cut key [Ctrl+f]) and enter this dialog box:



Here you type in the type of text you want to find, and state which type of symbol text you want to find, and in which direction you want the search carried out. The search direction determines whether you search the full project, the page you are working on, or in the direction forwards or backwards. — Please note that if you search for **Symbol datafields**, you search for the *values* in the symbol datafields. Correspondingly, you can only *replace* the values of the symbol datafields, and *not* their names.

When you set **Text type** to **All, Automation** search in all texts types in the project.

When a text is found, the text is displayed in the dialog box.



Text recognition

If you select **Match Case**, you will only find the texts using upper case and lower case letters exactly the way you type them in. If you write *X1*, the text *x1* will not be found. However, this text will be found if you have *not* selected **Match case**.

If you select **Find Whole Text Only**, you will only find the symbol texts that contains exactly the letters/characters you typed in. If you have typed in *Component*, it will *not* find the text *Component A38*. But this text will be found if you have *not* selected **Find Whole Text Only**.

Please note, that if you have *not* selected **Find Whole Text Only**, the program will find all texts having the text you typed in as a part of the text. If you typed in *tact*, it would find the text *contact*.

Searching with Reference designations

When searching for symbols with reference designations, it is possible only to specify either the location or the function part of the reference designation.

If you type in the search text *=G1-Q1* and select **Match Whole text only**, you will therefore also find the symbols for the component *=G1+A2-Q1*.

Zoom during Find and Replace

When a found text is selected on a diagram, it is sometimes an advantage to zoom in on the symbol. In the **Zoom in mm** field you specify how close you want to zoom in on the found text.

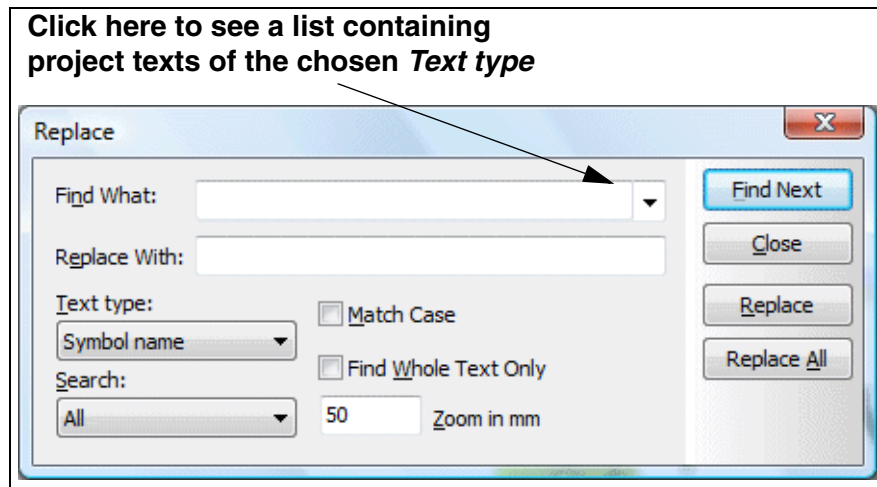
Find next symbol

When you want to find the text in the project, you click on **Find Next**, and the next text matching the search criteria will be found.

If you exit the **Find** dialog box, you will find the next text matching the search criteria by choosing **Edit => Find next**, or by pressing the [F3] key.

REPLACING SYMBOL TEXTS

If you want to find and replace symbol texts, you choose **Edit => Replace** and enter this dialog box:



You specify what you want to *find* as explained above, and in the **Replace with** field you type in the text you want to replace.

When you have clicked on **Find Next**, and a symbol text has been found, you click on **Replace** to replace the text. After replacing the text, you automatically jump to the next text matching the search criterion.

If you do not want to change the text you have found, just click on **Find next**, and you jump to the next text matching the search criterion.

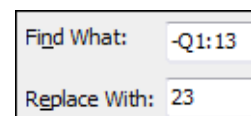
Clicking on **Replace All** will replace all matching texts in the specified area.

Click on **Close** when you have finished.

— Please note that if you search for **Symbol datafields**, you search for the *values* in the symbol datafields. Correspondingly, you can only *replace* the values of the symbol datafields, and *not* their names.

Find and Replace for connection names

Connection names can also be replaced from the dialog box **Replace**. If you wish to replace the connection name **13** for the symbol **-Q1**, write **-Q1:13** in the **Find What** field, and only write the new connection name in the **Replace with** field — e.g. **23** — because the program knows that it is the connection name, that is going to be replaced.



FIND AND REPLACE REMAINS ON SCREEN

When you have chosen **Find** and **Replace**, the dialog box remains on screen while you perform operations, so that you can continue searching easily.

— The **Find and Replace** dialog box can be set to be transparent, when working on the drawing area while the **Find and Replace** dialog box remains on screen — see “Pointer/Screen” on page 815.

NEGATED AND UNDERLINED TEXTS IN FIND AND REPLACE

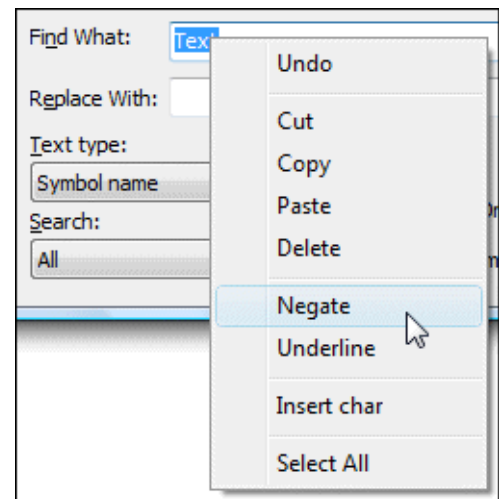
In the dialog boxes for **Edit => Find** and **Edit => Replace** you can search for negated and underlined texts.

To search for a negated text, do the following:

- 1) Type in the text, and select the part of the text that is negated.
- 2) Right-click on the selected, and choose **Negate**.

Searching for underlined texts takes place by choosing **Underline**.

To neutralise the negation of a text, select the text, right-click and choose **Negate** again. To neutralise the underlining of an underlined text, choose **Underline**.



DESIGN CHECK

The designcheck function can give you some help checking your design.

THIS CHAPTER DESCRIBES	
— PERFORMING A DESIGN CHECK.....	(P. 310)
— THE DESIGN CHECK DIALOG BOX.....	(P. 311)
— THE DESIGN CHECK ERROR MESSAGES.....	(P. 312)
— DISABLING USED MORE THAN ONCE IN THE SYMBOL DEFINITION.....	(P. 313)

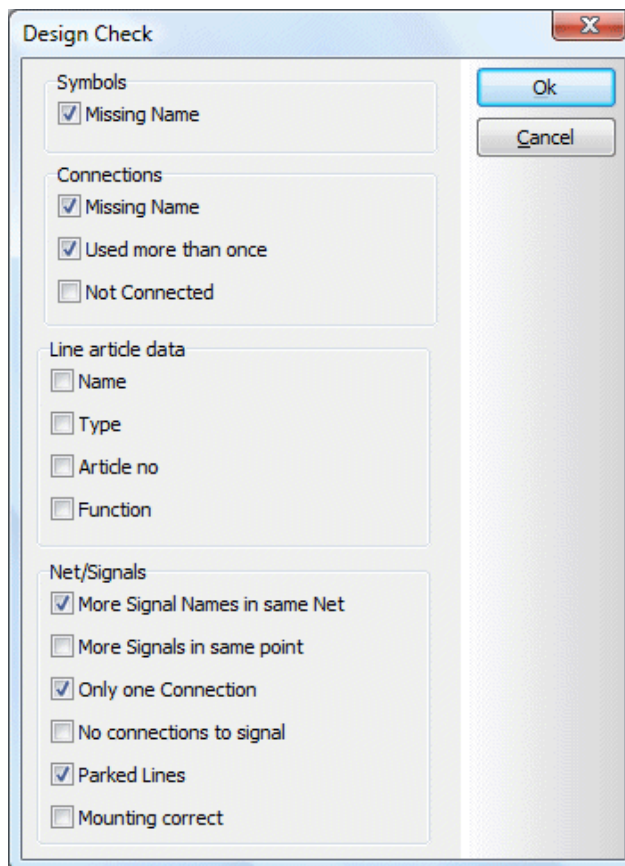


USE THE DESIGN CHECK FUNCTION FREQUENTLY

IT IS A GOOD IDEA TO APPLY THE DESIGN CHECK FUNCTION FROM TIME TO TIME WHILE YOU ARE DRAWING. THIS WAY YOU DO NOT HAVE TO BOTHER WITH TOO MANY ERROR MESSAGES AT THE SAME TIME.

PERFORMING A DESIGN CHECK

To activate the function, choose **Functions => Design Check**, which gives you the following options:



Select which of the options you want your project checked for. To maintain a survey, it might be a good idea to check the project for one design check option at the time.

— Please note that the check for **Mounting correct** only appears if you have chosen to work in mounting mode. — See “Mounting correct drawing” on page 92.



ACTIONS PERFORMED BEFORE DESIGN CHECK CAN NOT BE UNDONE

WHEN EXECUTING A DESIGN CHECK, THE PROJECT IS AUTOMATICALLY OPTIMISED.

THEREFORE, YOU DO NOT HAVE THE OPTION TO UNDO THE ACTIONS MADE BEFORE EXECUTING THE DESIGN CHECK.

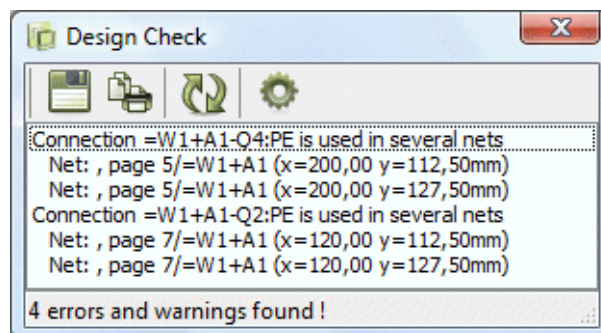
WHEN YOU ACTIVATE THE DESIGN CHECK FUNCTION, LINES WHICH HAVE BEEN EDITED ARE CORRECTED TO BE CONTINUOUS LINES.

Click on **OK**.

If the design check finds errors of the selected types, you will get a list of errors and warnings — as described in “The Design Check dialog box” on page 311. Otherwise, you will be told that no errors have been found.

THE DESIGN CHECK DIALOG BOX

When you choose **Functions => Design Check**, and click **OK** to execute a design check, you enter the dialog box **Design Check**:



This dialog box remains on screen while you are working in the project. — The dialog box can be set to be transparent, as described in “Pointer/Screen” on page 815.

Here you have the following options:

Double-click on errors in the list

When you double-click on an error message in the list, the program jumps to the place in the project, which have provoked the message, and the relevant object is pointed out by the cross hairs.

Update the Design Check list



When you have made changes in the project, you can click the **Update list** button. This automatically updates the list according to the performed changes.

When you have corrected the errors on the list, there may still be errors in the project, due to subsequent obsolete errors.

Therefore it is a good idea to perform another design check, once you have corrected the errors mentioned in the original list.

Accordingly, it may happen that correcting one error will make more than one error message disappear.

Design check settings



When you click the **Design Check settings** button, you enter the **Design Check** menu, from where you specify which types of design check you wish to perform. The design check list is automatically updated according to these directions.

Save and Print

Furthermore, you have the following options:



Click on the **Save** button to save the list in a file.

Click the **Print** button to print the list.

Close the Design Check list



To close the design check list, click the **Close** button in the upper right-hand corner of the dialog box.

ERROR MESSAGES

When you perform a design check, you may see the following error messages, accompanied by information about the page on which the error occurs, and the coordinates for exactly where on the page it occurs:

ERROR MESSAGE	SIGNIFIES
<i>MISSING SYMBOL NAME</i>	THE SYMBOL ON THE POSITION STATED HAVE NOT BEEN ASSIGNED A SYMBOL NAME.
<i>MISSING CONNECTION NAME</i>	YOU HAVE NOT ASSIGNED A NAME FOR THE CONNECTION POINT OF THE SYMBOL ON THE STATED POSITION.
<i>CONNECTION K1:14 IS USED IN SEVERAL NETS L1 AND L2, ARE USED IN THE SAME NET</i>	THE CONNECTION POINT WITH THE CONNECTION POINT NAME 14 IS USED IN MORE THAN ONE ELECTRICAL POTENTIAL.
<i>ONLY ONE CONNECTION IN NET L7</i>	YOU HAVE DRAWN A CONDUCTING LINE WITH JUST A SIGNAL SYMBOL — AND NOTHING ELSE. A CONNECTION TO A SYMBOL, OR TO ANOTHER SIGNAL SYMBOL, COULD BE MISSING.
<i>PARKED LINE</i>	YOU HAVE PARKED A LINE ON THE STATED POSITION.
<i>MORE SIGNALS IN SAME POINT</i>	YOU HAVE PLACED MORE THAN ONE SIGNAL SYMBOL AT THE SAME POSITION.
<i>X1:1U1 IS NOT CONNECTED</i>	THE CONNECTION POINT IS NOT CONNECTED.

If you are having difficulties handling errors in your project, please contact your local support service.

DESELECT USED MORE THAN ONCE IN THE SYMBOL DEFINITION

It is possible to *deselect* **Functions => Design Check => Used more than once** for individual symbols via the symbol definition for a symbol.

This is because it for some symbols — specially for pneumatics symbols — is allowed to have more connection points with the same name.

— Read more about this in “No double check” on page 682.

PRINTING

In **Automation** you can choose between just printing out the page you are presently working on, without changing any page settings, or specify which pages you wish to print, and how they shall be printed etc.

THIS CHAPTER DESCRIBES	
— OPENING THE PRINT DIALOG BOX	(P. 314)
— THE RIGHT-HAND SIDE OF THE DIALOG BOX	(P. 315)
— SELECT PAGES	(P. 315)
— SETUP	(P. 317)
— CHOOSE PRINTER	(P. 318)
— BATCH PRINTING	(P. 318)
— IF DIAGRAMS ARE MINIMIZED WHEN PRINTING.....	(P. 318)
— EXPORTING PDF FILES	(P. 319)
— EXPORTING DWG/DXF FILES.....	(P. 320)

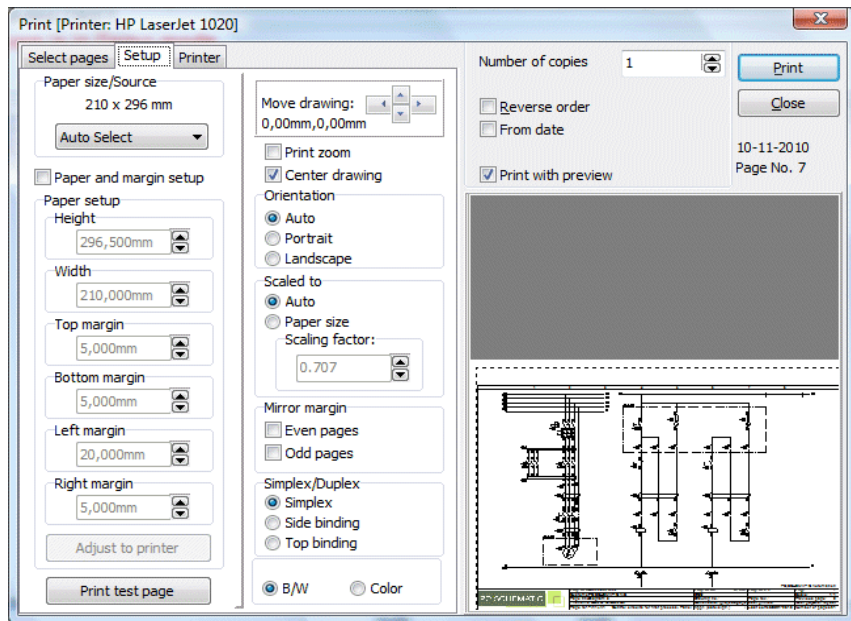


If you only want to print out the page you are working on, and do not wish to make changes in the printer settings, click the **Print this page** button.

OPENING THE PRINT DIALOG BOX



The printing dialog box shown below is activated by choosing **File => Print** or by clicking on the **Print pages** button on the program toolbar. The short-cut key is [Ctrl+p].



THE RIGHT-HAND SIDE OF THE PRINTER DIALOG BOX

The right-hand side of the dialog box is identical for the tabs **Select pages**, **Setup** and **Printer**. Here you get an overview of what printing a page will result in.

You can also determine the number of copies, whether you want the pages printed in reverse order, or if you only want to print pages changed later than a specified date.

Projects are printed out as one document. This means the printing can not be interrupted by other jobs on a network printer.

When printing more than one copy of the project, an information is sent to the printer, specifying the number of copies. — This, however, requires that the printer driver is able to handle this information. If the printer driver can handle the information "10 copies", this is obviously a lot faster than if the program should send the project to the printer ten times.

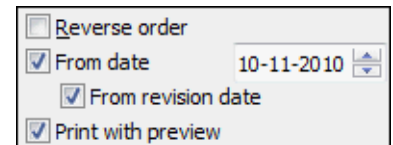
When more than one page and more than one copy is chosen, you can also choose **Collate**, if the chosen printer driver can handle the collate information.

If you wish to display the pages in the dialog box **Print** as they are printed, click on **Print with preview**.

The print settings are saved when you click on **Close**, and are in effect the next time you print pages.

Printing for Revision

If you are working on a revision — see "Revision control" on page 790 — and select **From date**, you get the option of choosing **From revision date**. This makes it possible only to print pages which has been changed since the start of the revision.



— See "Printing during revision control" on page 791 also.

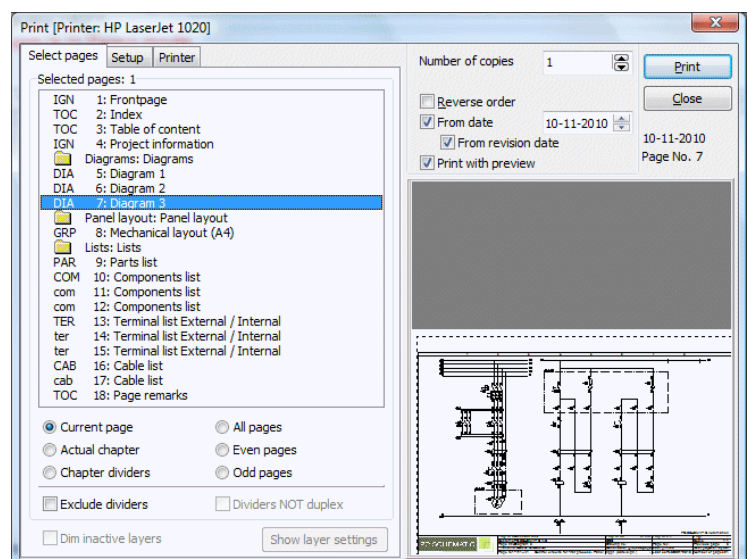
SELECT PAGES

Choosing the **Select pages** tab, you specify which pages to print.

If you want to print page 2 to 4 in the project, you click on page 2, and press down <Shift> while clicking on page 4.

If you, on the other hand, want to select page 1 and 3, you click on page 1, and press down <Ctrl> while clicking on page 3.

If you choose **All pages**, **Even pages**, **Odd pages** or **Current page**, the selected pages are singled out in the list of project pages.



Pages marked with a preceding ! character, have been changed since the project was opened.

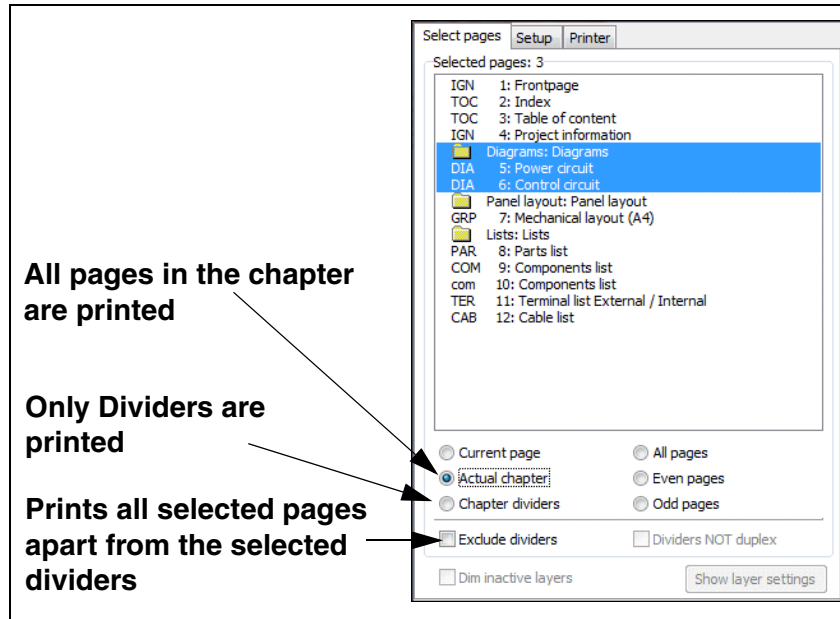
Pages marked with a preceding + character, have been changed since the project was last saved.

Printing Chapters and Dividers



When you click the **Print pages** button and click on the tab **Select pages**, you have the option to handle chapters and dividers easily:

This for instance makes it possible to print dividers on a different type of paper.



When you are printing on a duplex printer, you can also specify not to print dividers on duplex, by clicking on **Dividers NOT duplex**. This checkbox is dimmed if the selected printer can not handle duplex.

Printing layers

If you click on **Show layer settings**, you can specify which layers on the drawing to print. — See “Printing pages with layers” on page 468.

If you select **Dim not active layers**, will those layers, which are dimmed when displayed on screen, also be dimmed when printing. This is only an option when there exists more than one layer on the page.

— Read more about this in “The Layer menu” on page 466.

SETUP

Selecting **Setup** enables you to define margins and orientation. But to do this, you first have to select **Paper and margin setup**.

When you click **Adjust to printer**, the program gives you the printable area of your printer through the figures in the **margin** fields.

When you have done this, you will see the area, specified by the margin settings, displayed by a red dash line. At the same time you will see the area the drawing fills out, indicated by a fully drawn rectangle.

Also there are a number of functions, which make it easier to alter the printout to meet your requirements. The drawing can be scaled up and down, and can be moved on the paper.

If you want a margin with room for holes in the paper, you must set the left margin to 20mm.

If you click on **Auto** below **Orientation**, **Automation** automatically orientates the page according to the paper.

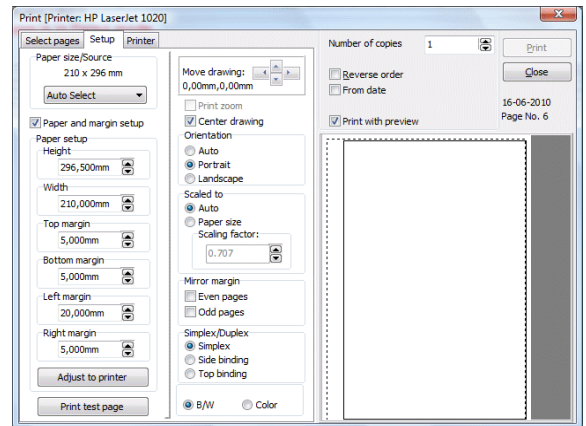
If you deselect **Centre drawing** and set **Scale to** and **Orientation** to **Auto**, your drawing will be adapted to the chosen printer — with respect to the location and scaling.

If you choose **Even pages** under **Mirror margin**, the margin will be mirrored for the even pages in the project — In this context the first page in the project is page number 1. If you choose **Odd pages** the margin of the odd pages will be mirrored.

When you choose **Duplex** the project will be printed in duplex. — Margins are only mirrored when you choose **Even pages** or **Odd pages**.

Choose **Color** when you want to print the project in color.

On black and white printers, selecting **Color** gives you a printout using a grey tone scale. Please note that your printer might convert some colors to the color white. The color white will not be seen on the print.

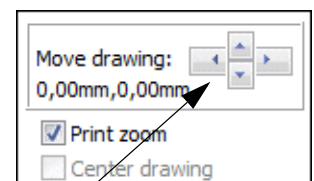


Print zoom

To print a zoom, do the following:



- 1) Click on the **Zoom** button, and make a zoom — see “Zoom” on page 38.
- 2) Click the **Print pages** button, and click on the tab **Settings**.
- 3) Choose **Print zoom**. If you need to move the selected zoom, click on the arrows next to **Move drawing**. When you press down <Shift> while clicking the arrows, the zoom is moved with three times bigger steps.



Click here to move the chosen zoom

4) Click on **Print**, and the chosen zoom is printed.

Print test page

If you are having difficulties printing the drawing on your printer, click **Print test page**. This will make the printer print out a test page, displaying the printable area of your printer, drawn with a thin line along the edges. The right, left, top and bottom margins will also be printed in figures. These are given using one thousandth of a millimetre, making **5865um** equal to **5.865mm**. These settings are specified in your printer driver, and therefore are not changeable from the **Automation** program. If the test print does not include all four lines, you have not chosen the correct printer driver for your printer.

A biased left or right margin, suggests that the printer is not feeding the paper correctly. If the printable area is biased towards the left- or the right hand side, placing the paper differently in the tray might help.

If you are having further difficulties, please contact your local supplier.

CHOOSE PRINTER

The last option is the **Printer** tab, which opens up the general Windows **Print setup** dialog box. The settings in this dialog box have lower priority than the ones you define in the rest of the dialog box **Print**, and can be overruled.

When all settings have been set, click on **Print** in the right hand side of the dialog box.

The last used printer and settings for each printer is memorized by the program.

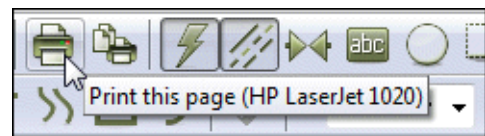
Alternative PCISCHEMATIC font

— See “Applying the Arial font for printing to PDF files” on page 320.

Displaying the printer name as a hint



When you let the mouse cursor rest over the **Print pages** button, the name of the selected printer is displayed as a *hint*.



BATCH PRINTING

Read more about this in “Batch printing” on page 804.

IF DIAGRAMS ARE MINIMIZED WHEN PRINTING

When printing, **Automation** squeezes all objects in the work area for each diagram page. This may result in that the drawing might be placed in e.g. the lower-left corner of the printed page. This is because objects are placed outside the frame of the normal diagram page.

Read more about this in “The Zoom All function during DWG/DXF import” on page 640 and “The Zoom All function” on page 39.

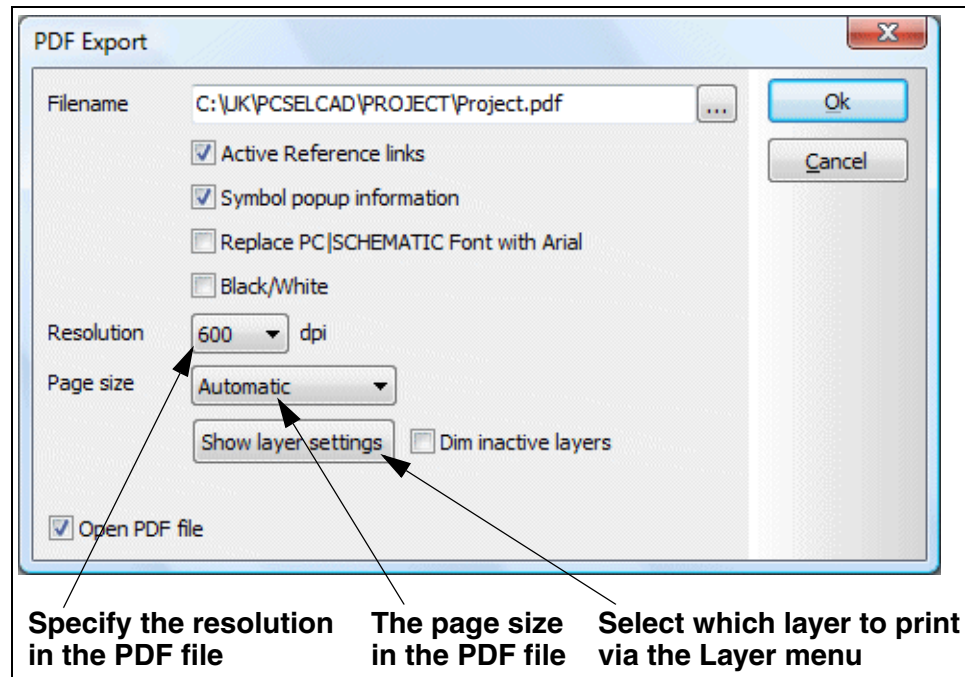
EXPORTING PDF FILES

In **Automation** you can export the active project as a PDF file.

You can start the export in two ways:

- Choose **File => Export => As PDF**, or
- Choose **File => Save as** and choose **PDF fiels (*.pdf)** below **File type** at the bottom of the dialog box.

This brings you into the dialog box **PDF export**:



In the dialog box you have the following options:

Active reference links

When you select **Active reference links**, links are inserted on the following objects in the PDF file:

- Active links between symbols in project pages
- Active links from components in lists to components in the project
- Active links from chapter tables of contents to the mentioned project pages

As you can see, the generated PDF files contains almost all links and references from the original project files.

Symbol popup information

When you select **Symbol project information**, the generated PDF file will contain popup information for the symbols in the project pages.

Fonts when creating PDF files

When printing a project as a PDF file, all **PC|SCHEMATIC** fonts will be turned into graphics. Due to the search facilities in the **Acrobat** program, it is possible to search on texts using *Windows* fonts in the PDF file.

Please note, that *it is not correct to use Windows fonts on diagram pages according to the standards.*

APPLYING THE ARIAL FONT FOR PRINTING TO PDF FILES

When a project is printed to a PDF file, it is possible to substitute the **PC|SCHEMATIC** font with the **Arial** font, which is the Windows font most similar to a CAD font. This function is specially designed for creating PDF files.

This makes it possible to apply the **PC|SCHEMATIC** font in the project, *and* to get texts which you can search on in the PDF file. These texts will also become smoother in the PDF files than texts using the **PC|SCHEMATIC** font.

To do this, you must select the **Printer** tab, and select **Alternative PC|SCHEMATIC font: Arial**.

SPECIAL CHARACTERS IN PDF FILES

If you change texts written with the **PC|SCHEMATIC** font to a Windows font, negation and underlining will give an incorrect result. The earth character and other special characters may also give incorrect results, depending on which font you use. — See “Special characters when changing the PC|SCHEMATIC font to a Windows font” on page 119.

**SPECIAL CHARACTERS AND WINDOWS**

THE FONTS YOU GET WITH OLDER VERSIONS OF WINDOWS, DOES NOT CONTAIN SPECIAL CHARACTERS. WHEN A SPECIAL CHARACTER DOES NOT EXIST IN A SELECTED FONT, A SQUARE IS DISPLAYED IN STEAD OF THE SPECIAL CHARACTER.

THE FONTS YOU GET WITH WINDOWS VISTA AND SUCCEEDING VERSIONS CONTAINS SPECIAL CHARACTERS.

THE **PC|SCHEMATIC** FONT CONTAINS THE NECESSARY SPECIAL CHARACTERS.

Command files for converting projects to PDF format

It is possible to create command files, which automatically saves **Automation** projects as PDF files. Read more about this in “Command files for saving projects as PDF, DWG and DXF files” on page 803.

EXPORTING DWG AND DXF FILES

In **Automation** you can both export and import DWG and DXF files. Read more about this in “DWG and DXF files” on page 640.

WINDOWS EXPLORER AS SYMBOL/PROJECT MENU

THIS CHAPTER DESCRIBES

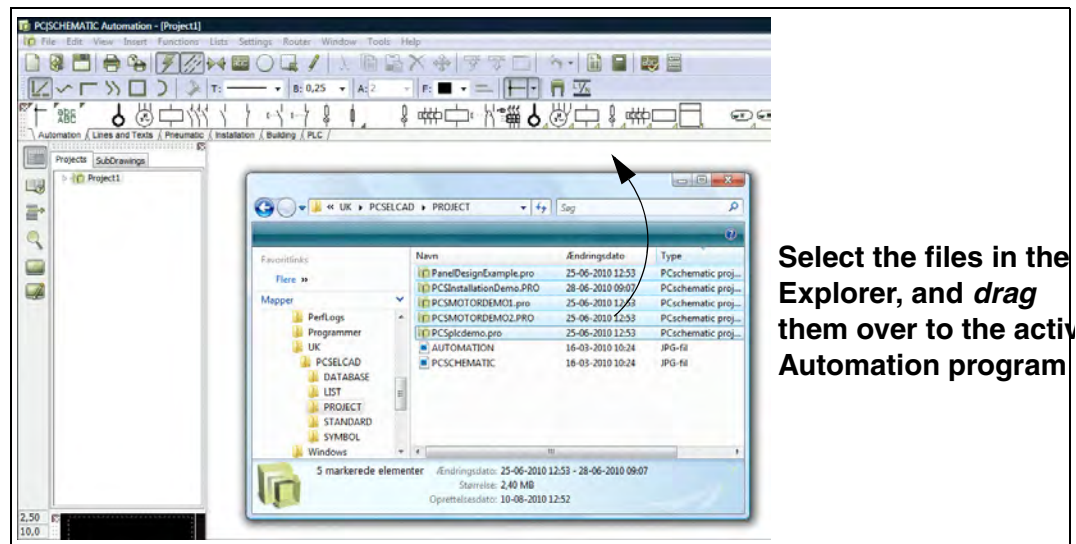
- USING THE WINDOWS EXPLORER AS A PROJECT MENU(P. 321)
- USING THE WINDOWS EXPLORER AS A SYMBOL MENU.....(P. 321)

USING THE WINDOWS EXPLORER AS A PROJECT MENU

In **Automation** it is possible to use **Windows Explorer** as a kind of *Project menu*.

When you have opened Windows Explorer, you can therefore *drag* project files from the Explorer to the **Automation** program, whereby the projects are opened.

When you select several projects at a time, all of the selected projects will be opened in **Automation**:



Select the files in the Explorer, and *drag* them over to the active Automation program

USING THE WINDOWS EXPLORER AS A SYMBOL MENU

In **Automation** it is possible to apply **Windows explorer** as a kind of *Symbol menu*.

As described for projects, you can apply the Windows Explorer to drag *symbols* into an open project. Hereby you get the symbol directly in the cross hairs in **Automation**. You can therefore apply the Windows Explorer as an alternative symbol menu, which you can keep open while working with **Automation**.

You can only fetch one symbol at the time from the Windows Explorer.

If you open two copies of Windows Explorer, you can apply the one to open projects from, and the other to fetch symbols from.

REFERENCE DESIGNATIONS

In **Automation** you can apply reference designations in the naming of your symbols. This makes it for instance possible to follow the IEC/EN 61346 standard (IEC750 previously) for naming of symbols.

The reference designations in **Automation** can be defined freely, and is therefore not tied up to a specific standard. This makes it both possible to follow an internal company standard, and to follow changing international standards. If you need to change standard for using reference designations, this is also an option.

THIS CHAPTER DESCRIBES	
— ON REFERENCE DESIGNATIONS	(P. 322)
— DEFINING REFERENCE DESIGNATIONS	(P. 325)
— REFERENCE DESIGNATIONS FOR SYMBOLS	(P. 336)
— REFERENCE DESIGNATIONS FOR PAGES	(P. 340)
— REFERENCE DESIGNATIONS IN AREAS ON PAGES	(P. 342)
— REFERENCE DESIGNATIONS FOR CHAPTERS	(P. 346)
— REFERENCE DESIGNATIONS FOR MORE THAN ONE PAGE AT A TIME.....	(P. 346)
— REFERENCE DESIGNATIONS VIA CHAPTER TABS.....	(P. 349)
— DATAFIELDS FOR SPECIFYING REFERENCE DESIGNATIONS IN PROJECTS.....	(P. 351)
— LISTS ACC. TO FUNCTION OR LOCATION ASPECT	(P. 352)



LEARN MORE ABOUT REFERENCE DESIGNATIONS

IN THE BOOK "ELECTRO-TECHNICAL DOCUMENTATION ACCORDING TO STANDARDS", YOU CAN READ MORE ABOUT APPLYING REFERENCE DESIGNATIONS IN ELECTRO-TECHNICAL DOCUMENTATION. THE BOOK IS PUBLISHED BY PC|SCHEMATIC A/S, AND IS NOW IN ITS 2. EDITION.

ON REFERENCE DESIGNATIONS

— Please skip this explanation if you are already familiar with reference designations.

Reference designations are used for identifying components, functions or locations where e.g. a component is placed. The reference designation for a component must be placed next to the symbol in the diagram, and on / next to the physical component, where it is placed/mounted.

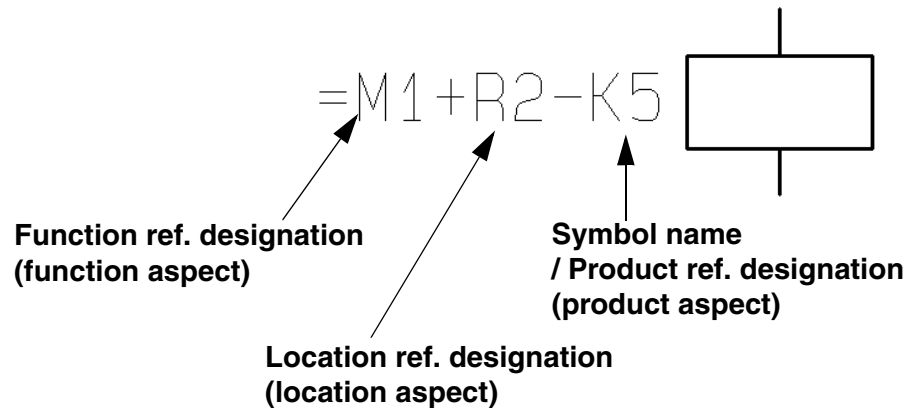
A reference designation must consist of a prefix, followed by a capital letter, followed by an integer. For instance **-M01**.

A reference designation must be attached to one of three different aspects, depending on how you need to observe what the reference designation describes:

- When you observe a function (function aspect), the prefix = is applied
- When you observe a location (location aspect), the prefix + is applied

- When you observe a product (product aspect), the prefix - is applied

For a symbol you can see the following three reference designations:



The reference designation in the block $=M1+R2-K5$ is interpreted like this: The product/component $-K5$ exists in the location $+R2$ and is attached to the function $=M1$.



PRODUCT REFERENCE DESIGNATION / SYMBOL NAME

IN STEAD OF THE *PRODUCT REFERENCE DESIGNATION* FOR A SYMBOL, THE TERM *SYMBOL NAME* IS USED IN THE REST OF THIS MANUAL.

In the example above the function designation $=M1$ specifies that the symbol/component is a part of the function $=M1$. This could for instance be "Pumping1". The location designation $+R2$ — for where the component is placed — could for instance be "Hall2".

The product designation $-K5$ is used to identify the component among the components being a part of Pumping1 ($=M1$), which also exist in Hall2 ($+R2$).

By using the reference designation system with the three different aspects, you can for instance describe a complete industrial plant. The reference designation system gives you the possibility to make one list containing all products, another containing all functions, and a third containing all locations required to identify everything in the industrial plant.

It is possible to specify that all symbols on a project page e.g. belongs to Pumping1 ($=M1$) which is located in Hall2 ($+R2$). If you had specified the reference designation $=M1+R2$ in the title block of the drawing header in the example above, you would only have need to display the reference designation $-K5$ next to the symbol.

If there exists a symbol/component which is not attached to the reference designations for the page, **Automation** automatically displays those of the reference designations for the symbol, which differs from the reference designations for the page.

Before creating the reference designations, you should have a survey of which functions, as well as which switchboards/racks/drawers or other location aspects, you will need in your project.

Character for specifying full reference designation (exclude concatenation)

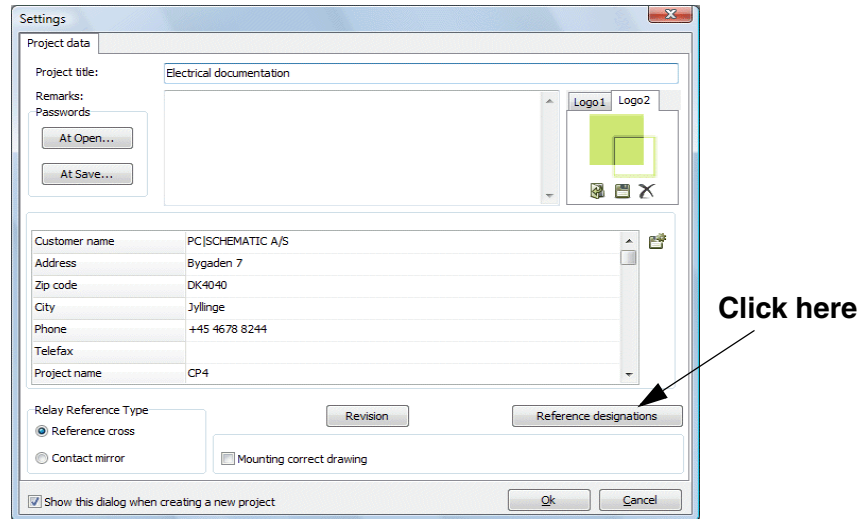
If the character > is displayed in front of a reference designation — e.g. >**K1** — this means that **-K1** is the *full* reference designation for the component. **-K1** therefore uniquely identifies the component.

Please note, that this character can therefore also appear when you have forgotten to specify a reference designation for the component.

The > character is called "exclude concatenation", which you can read more about in "Use full exclude of concatenation" on page 331.

DEFINING REFERENCE DESIGNATIONS FOR THE PROJECT

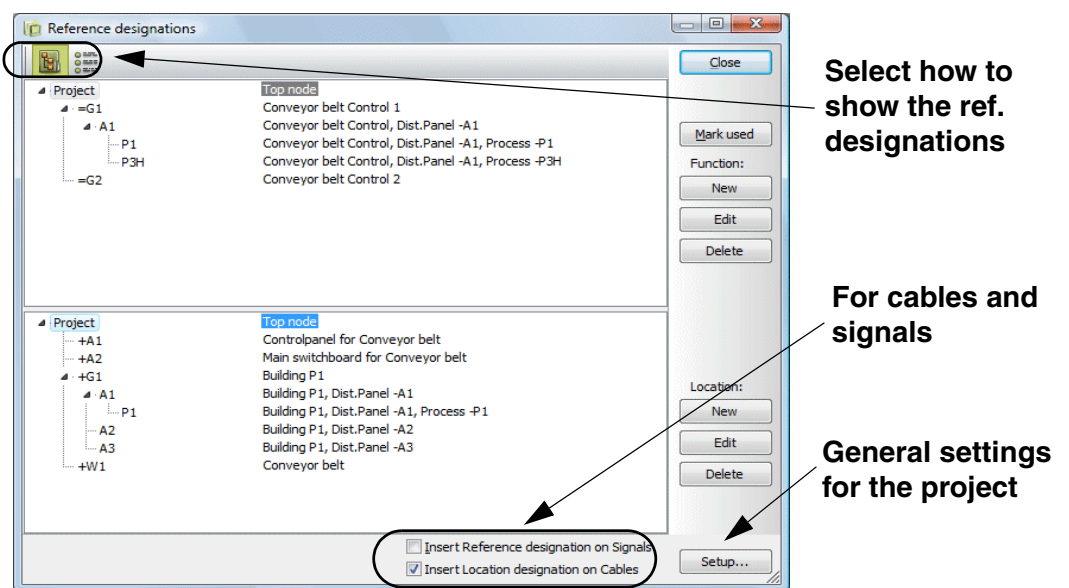
To create reference designations in your project, you enter **Settings => Project data**, which is the only place in **Automation** where you can see anything about reference designations, as long as no reference designations have been defined in the project.



REFERENCE DESIGNATIONS MUST BE DEFINED FIRST

IT WILL NOT BE POSSIBLE TO REFER TO REFERENCE DESIGNATIONS ANYWHERE IN **Automation** UNTIL YOU HAVE CREATED REFERENCE DESIGNATIONS IN THIS DIALOG BOX. HAVING DONE THIS, HOWEVER, YOU CAN SELECT REFERENCE DESIGNATIONS IN **SETTINGS => PAGE DATA** AS WELL AS IN THE DIALOG BOX **COMPONENT DATA**.

Here you click on **Reference designations**, and enter the dialog box **Reference designations**:



In the dialog box **Reference designations** you can create and edit the **Function** and **Location** aspects for the reference designations.

The first time you enter the dialog box, no **Function** or **Location** aspects will be specified.

OPTIONS IN THE DIALOG BOX REFERENCE DESIGNATIONS	
— DISPLAYING REFERENCE DESIGNATIONS IN THE DIALOG BOX.....	(P. 326)
— SPECIFYING FUNCTION/LOCATION ASPECTS	(P. 326)
— SEE USED FUNCTION/LOCATION ASPECTS	(P. 328)
— SORTING FUNCTION/LOCATION ASPECTS BY COLUMN	(P. 329)
— EDITING FUNCTION/LOCATION ASPECTS	(P. 329)
— DELETING FUNCTION/LOCATION ASPECTS	(P. 330)

Displaying reference designations in the dialog box Reference designations

In the upper-left corner of the dialog box **Reference designations** you can specify how to display the reference designations in the dialog box:

Display reference designations as hierarchy / Tree



When you click the **Show as tree** button, the reference designations are displayed as a hierarchical structure — see below.

The reference designations are hereby displayed as a folder structure, just like a folder structure you would find in the Windows Explorer, with similar possibilities for copying, deleting structure parts etc.

Display reference designations as a list



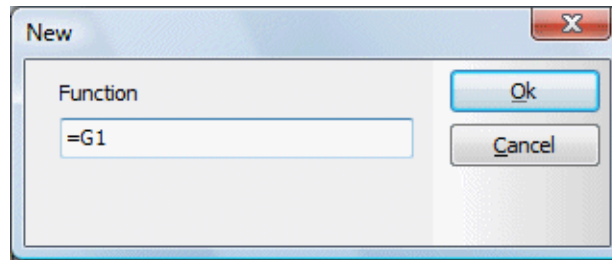
When you click the **Show as list** button, the reference designations are displayed as a list, with all of the reference designations displayed at the same level — see below.

<p>The screenshot shows a tree view under a 'Project' node. The tree structure is as follows:</p> <ul style="list-style-type: none"> Project <ul style="list-style-type: none"> =G1 <ul style="list-style-type: none"> A1 <ul style="list-style-type: none"> P1 P3H =G2 	<table border="1"> <thead> <tr> <th>Function</th> <th>Description</th> </tr> </thead> <tbody> <tr><td>=G1</td><td>Conveyor belt Control 1</td></tr> <tr><td>=G1A1</td><td>Conveyor belt Control, Dist.Panel -A1</td></tr> <tr><td>=G1A1P1</td><td>Conveyor belt Control, Dist.Panel -A1, Process -P1</td></tr> <tr><td>=G1A1P3H</td><td>Conveyor belt Control, Dist.Panel -A1, Process -P3H</td></tr> <tr><td>=G2</td><td>Conveyor belt Control 2</td></tr> </tbody> </table> <table border="1"> <thead> <tr> <th>Location</th> <th>Description</th> </tr> </thead> <tbody> <tr><td>+G1</td><td>Building P1</td></tr> <tr><td>+G1A1</td><td>Building P1, Dist.Panel -A1</td></tr> <tr><td>+G1A1P1</td><td>Building P1, Dist.Panel -A1, Process -P1</td></tr> <tr><td>+G1A2</td><td>Building P1, Dist.Panel -A2</td></tr> <tr><td>+G1A3</td><td>Building P1, Dist.Panel -A3</td></tr> <tr><td>+A1</td><td>Controlpanel for Conveyor belt</td></tr> <tr><td>+A2</td><td>Main switchboard for Conveyor belt</td></tr> <tr><td>+W1</td><td>Conveyor belt</td></tr> </tbody> </table>	Function	Description	=G1	Conveyor belt Control 1	=G1A1	Conveyor belt Control, Dist.Panel -A1	=G1A1P1	Conveyor belt Control, Dist.Panel -A1, Process -P1	=G1A1P3H	Conveyor belt Control, Dist.Panel -A1, Process -P3H	=G2	Conveyor belt Control 2	Location	Description	+G1	Building P1	+G1A1	Building P1, Dist.Panel -A1	+G1A1P1	Building P1, Dist.Panel -A1, Process -P1	+G1A2	Building P1, Dist.Panel -A2	+G1A3	Building P1, Dist.Panel -A3	+A1	Controlpanel for Conveyor belt	+A2	Main switchboard for Conveyor belt	+W1	Conveyor belt
Function	Description																														
=G1	Conveyor belt Control 1																														
=G1A1	Conveyor belt Control, Dist.Panel -A1																														
=G1A1P1	Conveyor belt Control, Dist.Panel -A1, Process -P1																														
=G1A1P3H	Conveyor belt Control, Dist.Panel -A1, Process -P3H																														
=G2	Conveyor belt Control 2																														
Location	Description																														
+G1	Building P1																														
+G1A1	Building P1, Dist.Panel -A1																														
+G1A1P1	Building P1, Dist.Panel -A1, Process -P1																														
+G1A2	Building P1, Dist.Panel -A2																														
+G1A3	Building P1, Dist.Panel -A3																														
+A1	Controlpanel for Conveyor belt																														
+A2	Main switchboard for Conveyor belt																														
+W1	Conveyor belt																														
Hierarchial display of reference designations	List display of reference designations																														

Specifying function/location reference designations

When you want to add a function aspect, click on the **New** button next to the **Function** window — or press the [Insert] key, if you have already selected a function aspect in the dialog box.

You then enter the dialog box **New**:



The character = will be suggested as the first sign in the reference designation, but you can also use a character of your own choice.

If the dialog box displays reference designations in list form, and a function reference designation is selected in the dialog box, the dialog box **New** will be filled out with this reference designation — as for instance =**G1** in the dialog box above. This is a help for when you type in many reference designations starting with the same text.

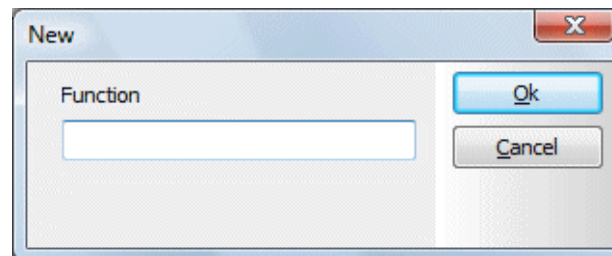
Type in the name of the new function aspect and click **OK**.



IEC / EN STANDARD

THE CHARACTERS = AND + ARE IEC / EN STANDARD FOR REFERENCE DESIGNATIONS.

Then type in the description for the reference designation.



Click **OK**.

If you are displaying the reference designations as a list — see “Displaying reference designations in the dialog box Reference designations” on page 326 — the function aspect is placed below the selected function aspect in the dialog box.

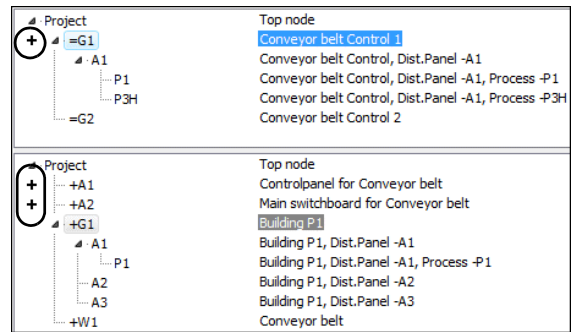
Click on **New** again — or press the [Insert] key — to add more function reference designations.

Location reference designations are specified correspondingly.

See used reference designations

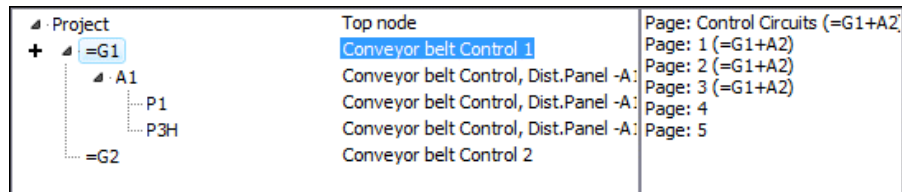
When you wish to see which reference designations are currently used in the project, click on **Marked used**. This inserts a + in front of the used reference designations.

This + remains visible until the dialog box is closed. The next time the dialog box is opened, the + characters are removed.



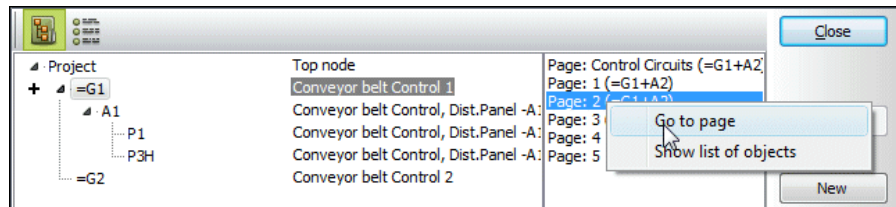
See on which page the reference designations are used

If you wish to see on which page a reference designation is used, you can right-click on the reference designation and choose **Show pages** — or click on the reference designation and press the <spacebar> —, and the pages are displayed in the right-hand side of the dialog box:

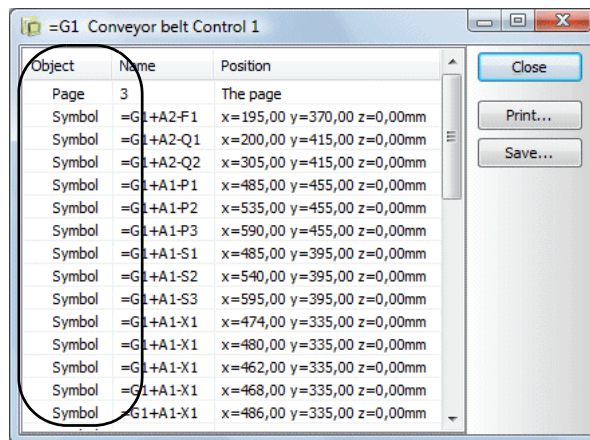


If you see the reference designations in list form, you can also double-click on the reference designation, to see on which pages the reference designation is used.

If you then right-click on a page, and choose **Go to page**, the selected page is displayed in the project:



If you right-click on a page, and choose **Show list of objects**, a list is displayed, containing all the objects on the page with the selected reference designation:



In the left-hand side of the list, you can see the type of the objects. The list can be printed, or saved as a file.

Sorting reference designations by columns

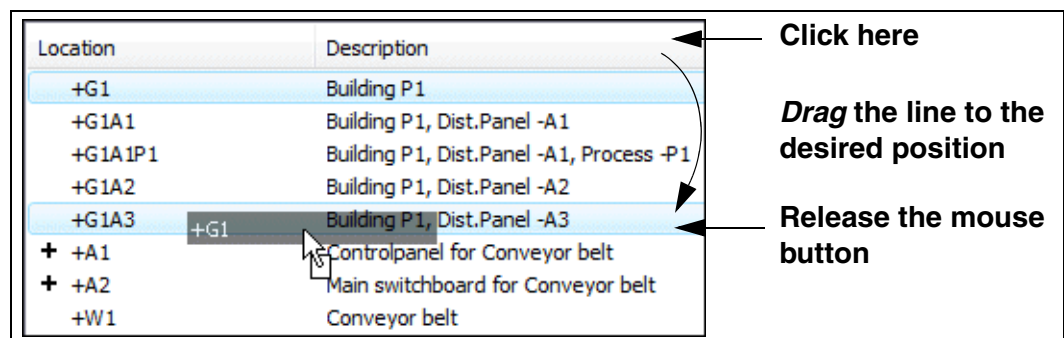
Please note: *This only applies for displaying reference designations as a list* — see “Display reference designations as a list” on page 326.

In the dialog box **Reference designations** you can sort the reference designations alphabetically based on the columns **Function aspect**, **Location aspect** and **Description** for function and location aspects.

This is done as described in “The sorting order of the data in the Object Lister” on page 298.

Dragging a reference designation to a new position in the dialog box Reference designations

When you wish to move a reference designation to another position in the list of reference designations, do this by *clicking* on the reference designation, and then *drag* it to the desired position:



The function/location aspect is placed just *below* the function/location aspect, that you point on when you release the mouse button. If you *drag* a function/location aspect beyond the top or the bottom of the list, the list starts scrolling. The last used function/location aspect will be memorized for the next time the dialog box is opened.

— See “The mouse” on page 23 for further details about drag and drop.

Editing reference designations

If you wish to change a function/location aspect, right-click on it and choose **Edit** — or press the [F2] key.

If you change a reference designation, and there exists other reference designations starting with this reference designation, you will be asked whether these should be changed also. If you change the reference designation =**G1** to =**G2**, you can for instance have the reference designation =**G1A2** changed to =**G2A2** automatically.

Changing reference designations for more than one page

To change reference designations for more than one page at a time, see “Edit reference designations for more than one page” on page 346.

Changing reference designations for Chapters

Changing reference designations for a full chapter, see “Editing reference designations for chapters” on page 346.

Deleting reference designations

To delete a reference designation, click on it and click **Delete** — or press the [Delete] key. You can also right-click, and select **Delete unused**, to delete the unused reference designations.

Inserting reference designations on signals

When you wish to assign signal references to signals, you must select **Insert Reference designations on Signals** — please refer to “Insert Reference designations on Signals automatically” on page 203.

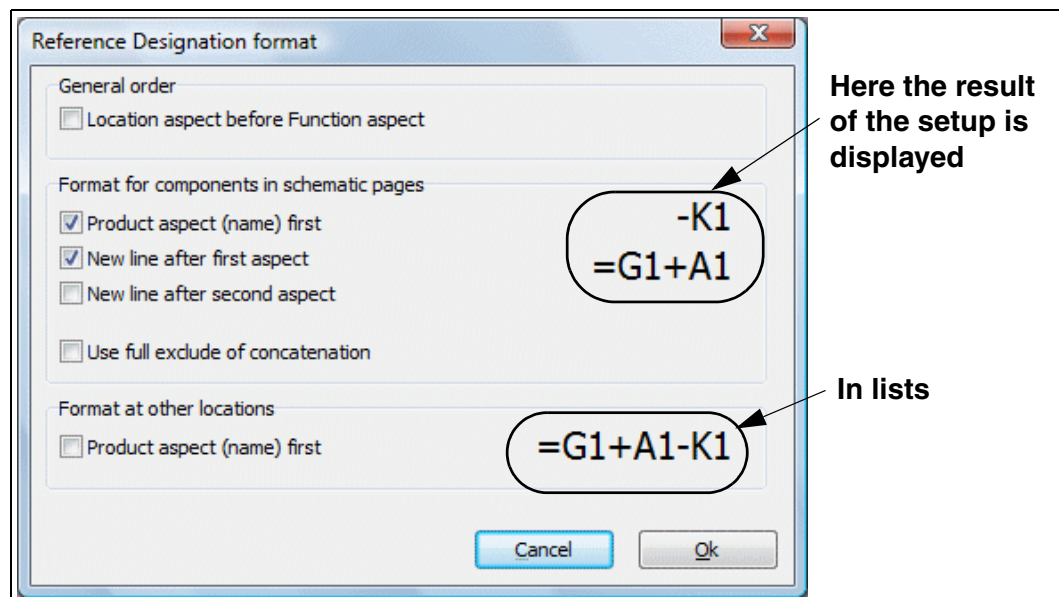
Inserting the location aspect for cables

Because no location aspect is typically assigned to cables (a cable runs from one location to another), you can choose **Settings => Project data => Reference designations**, and select/deselect the check mark next to **Insert Location designations on Cables**.

Hereby you specify whether to automatically insert a location aspect for a cable, when the cable is placed on a page with reference designations.

FORMAT FOR REFERENCE DESIGNATIONS

In the dialog box **Reference designations**: Click on **Setup** in the lower-left corner, to enter the dialog box **Reference Designation format**. Here you specify how to display reference designations in the project:



The consequences of the selected settings are displayed in the right-hand side of the dialog box. You have the following options:

General order: Location aspect before Function aspect

By default, the function aspect is displayed before the location aspect. When you select **Location aspect before Function aspect**, the location aspect is displayed before the function aspect. This applies to all project pages.

In schematic pages: Product aspect (name) first

When you select **Product aspect (name) first**, the product aspect (the name) is displayed first in the reference designation. If not selected, the product aspect (name) is placed last in the reference designation. This applies all project pages except lists.

In schematic pages: Reference designations with line breaks

By selecting **New line after first aspect** and/or **New line after second aspect**, you can split the reference designation in multiple lines.

— See “Individual control of reference designations with line breaks from Component data” on page 338 and “The tab Ref.design.” on page 156.

Use full exclude of concatenation

— See “Use full exclude of concatenation” on page 331.

Format at other locations: Product aspect (name) first

When you select **Product aspect (name) first** at the bottom of the dialog box, the product aspect (name) will be displayed first in lists. If not selected, the product aspect (name) is displayed last in lists.

USE FULL EXCLUDE OF CONCATENATION

In the dialog box **Reference Designation format**, you can select/deselect **Use full exclude of concatenation**.

What is concatenation?

Concatenation means "joining two character strings end to end", and is used in this context to describe how you understand the reference designations displayed on the project pages.

In order to make the reference designations take up less space on the project pages, it is implied that the reference designation for the page (or reference frame) is also a part of the symbol name — even though the page reference designation does not appear in front of the symbol name on the page. If the page has the reference designation =G1+A1, and the symbol name -P1 is displayed on the page, you will interpret this as the symbol has the reference designation =G1+A1-P1. The page reference designation and the symbol name has hereby been concatenated.



CHARACTER FOR "EXCLUDE CONCATENATION"

WHEN THE CHARACTER > IS THE FIRST CHARACTER IN THE REFERENCE DESIGNATION, THIS MEANS THAT THE REFERENCE DESIGNATION IS THE **FULL** REFERENCE DESIGNATION FOR THE SYMBOL. THE DISPLAYED REFERENCE DESIGNATION SHALL THEREFORE **NOT** BE CONCATENATED WITH THE PAGE REFERENCE DESIGNATION.

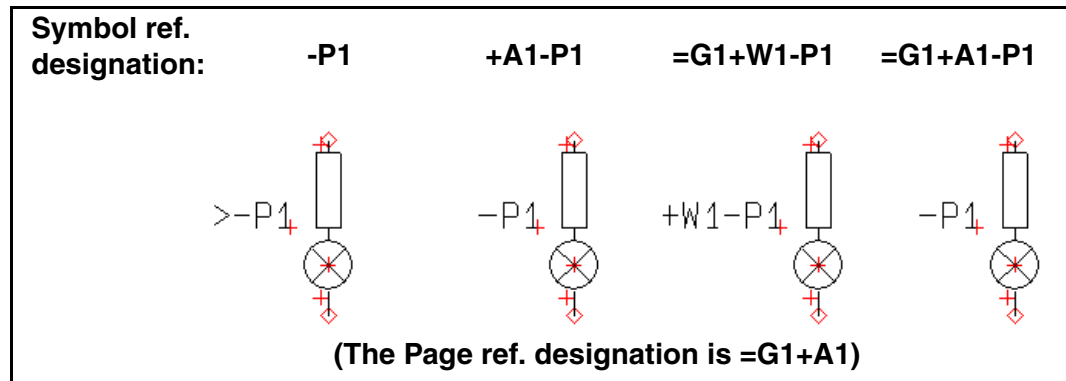
IF IT THEREFORE SAYS >+A1-P1 FOR A SYMBOL, THEN +A1-P1 IS THE **FULL** REFERENCE DESIGNATION FOR THE SYMBOL.

The way that **Automation 9.0** — and previous versions — have handled displaying the reference designations on the schematic pages, corresponds to *not* activating **Use full exclude of concatenation**.

If you do activate **Use full exclude of concatenation**, there will not exist any possibilities for misunderstandings in how to interpret reference designations on project pages, which is elaborated in the examples below:

Example without Use full exclude of concatenation

Example on four symbols, where the page reference is **=G1+A1**, and where **Use full exclude of concatenation** is *not* activated:

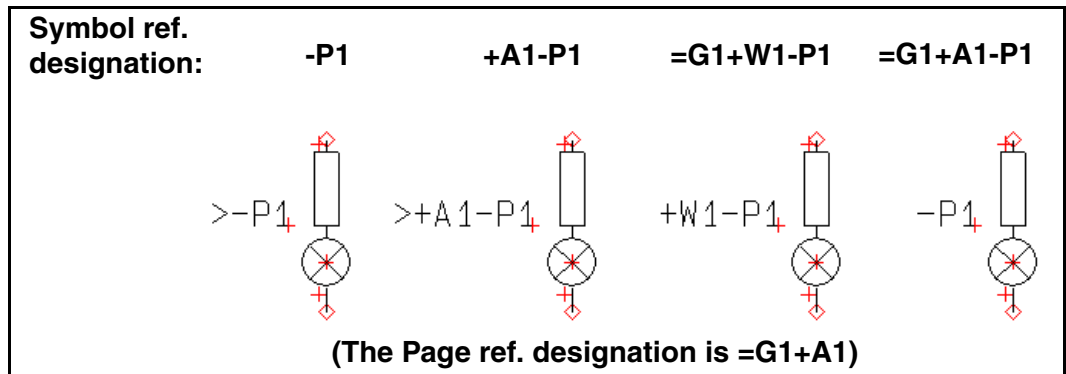


Explanation:

- 1) The name **>-P1** is displayed: This means that the full reference designation is displayed, which is correct.
- 2) The name **-P1** is displayed: Because the displayed reference designation does not start with the > character, this is interpreted as that the page reference designation must be concatenated with the displayed symbol name. The symbol reference designation will therefore be interpreted as **=G1+A1-P1**, which is *not correct*, because **=G1** is not part of the symbol reference designation.
- 3) The name **+W1-P1** is displayed: Because it does not start with the > character, we must concatenate with the page reference designation. Because the location aspect is included in the name, we will only concatenate with the function aspect. Therefore the reference designation will be interpreted as **=G1+W1-P1**, which is correct.
- 4) The name **-P1** is displayed: Because it does not start with the > character, we must concatenate with the page reference designation. You will therefore understand the reference designation as **=G1+A1-P1**, which is correct.

Example with Use full exclude of concatenation

Example on four symbols, where the page reference is **=G1+A1**, and where **Use full exclude of concatenation** is activated. (This means that everywhere possible, **Automation** will use full exclusion of concatenation):



- 1) The name **>-P1** is displayed: This means that the full reference designation is displayed, which is correct.
- 2) The name **>+A1-P1** is displayed: This again means that the full reference designation is displayed, which is correct.
- 3) The name **+W1-P1** is displayed: Because it does not start with the > character, we must concatenate with the page reference designation. Because the location aspect is included in the name, we will only concatenate with the function aspect. Therefore the reference designation will be interpreted as **=G1+W1-P1**, which is correct.
- 4) The name **-P1** is displayed: Because it does not start with the > character, we must concatenate with the page reference designation. You will therefore understand the reference designation as **=G1+A1-P1**, which is correct.

Close the dialog box Reference Designations

Click on **Close**, and then on **OK** under **Project data**, to return to your project.

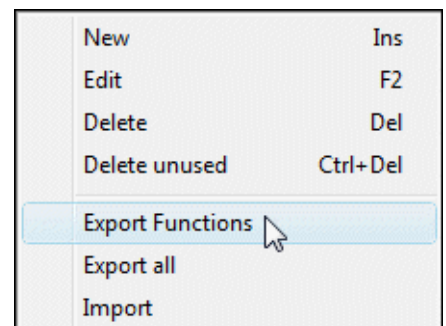
IMPORT AND EXPORT OF REFERENCE DESIGNATIONS

Reference designations can be exported/imported to/from Excel (*.xls files) and to/from commaseparated files (*.csv files).

If you right-click in the **Function** or **Location** window in the dialog box **Reference designations**, you get the option of exporting or importing reference designations, by choosing **Export** or **Import** respectively.

If you have right-clicked in the **Functions** window, you can also choose **Export Functions**, as you can choose **Export Locations** if you right-clicked in the **Locations** window.

If no reference designations have been defined in the project yet, an **Import** button will also appear in the right-hand side of the dialog box.



When you choose to export reference designations, you enter a dialog box, where you specify where to save the reference designations file, and specify whether to save the file as an Excel file or as a CSV file.

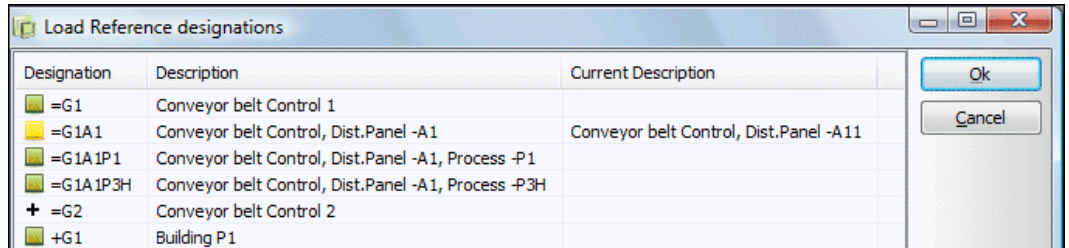
Format for Reference designations files

The applied format for reference designations files, is displayed in the figure below:

	A	B	
1	=G1	Conveyor belt Control 1	Format in Excel files
2	=G1A1	Conveyor belt Control, Dist.Panel -A1	
3	=G1A1P1	Conveyor belt Control, Dist.Panel -A1, Process -P1	
4	=G1A1P3H	Conveyor belt Control, Dist.Panel -A1, Process -P3H	
5	=G2	Conveyor belt Control 2	
6	+G1	Building P1	
7	+G1A1	Building P1, Dist.Panel -A1	
8	+G1A1P1	Building P1, Dist.Panel -A1, Process -P1	
9	+G1A2	Building P1, Dist.Panel -A2	
10	+G1A3	Building P1, Dist.Panel -A3	
11	+A1	Controlpanel for Conveyor belt	
12	+A2	Main switchboard for Conveyor belt	
13	+W1	Conveyor belt	
<pre>=G1;"Conveyor belt Control 1" =G1A1;"Conveyor belt Control, Dist.Panel -A1" =G1A1P1;"Conveyor belt Control, Dist.Panel -A1, Process -P1" =G1A1P3H;"Conveyor belt Control, Dist.Panel -A1, Process -P3H" =G2;"Conveyor belt Control 2" +G1;"Building P1" +G1A1;"Building P1, Dist.Panel -A1" +G1A1P1;"Building P1, Dist.Panel -A1, Process -P1" +G1A2;"Building P1, Dist.Panel -A2" +G1A3;"Building P1, Dist.Panel -A3" +A1;"Controlpanel for Conveyor belt" +A2;"Main switchboard for Conveyor belt" +W1;"Conveyor belt"</pre>			Format in CSV files

Importing reference designations

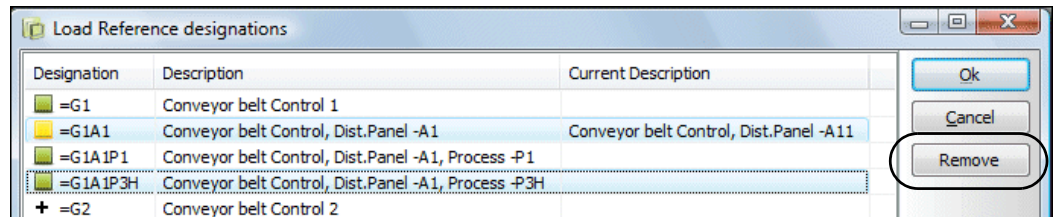
When you import reference designations, you enter the following dialog box:



To the left in the dialog box, you can see the status for the different reference designations:

+	NEW REFERENCE DESIGNATION, WHICH IS ADDED TO THE PROJECT.
GREEN SQUARE	REFERENCE DESIGNATION, WHICH ALREADY EXISTS IN THE PROJECT.
YELLOW SQUARE	THE REFERENCE DESIGNATION ALREADY EXISTS IN THE PROJECT, BUT HAS ANOTHER DESCRIPTION. AFTER IMPORTING, THE IMPORTED DESCRIPTION FOR THE REFERENCE DESIGNATION WILL BE USED IN THE PROJECT.

If you do not wish to import some of the reference designations, select the unwanted reference designations in the dialog box, and click the **Remove** button in the right-hand side of the dialog box:



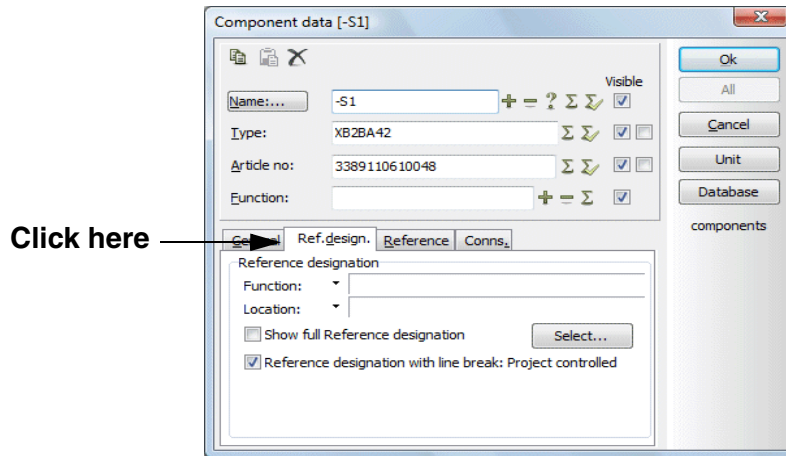
Then import the remaining reference designations by clicking **OK**.

REFERENCE DESIGNATIONS FOR SYMBOLS

THIS CHAPTER DESCRIBES	
— REFERENCE DESIGNATIONS WHEN PLACING SYMBOLS	(P. 336)
— EDITING REF. DES. FOR ALREADY PLACED SYMBOLS.....	(P. 339)
— EDITING REF. DES. USING THE SELECT NAME FUNCTION.....	(P. 339)
— PRECEEDING "-" IN SYMBOL NAMES.....	(P. 339)
— INSERTING REFERENCE DESIGNATIONS FOR SIGNAL SYMBOLS	(P. 339)

Selecting Reference designations when placing symbols

When you have activated the reference designations in the **Project data** dialog box, you can apply reference designations when you place symbols in the project.

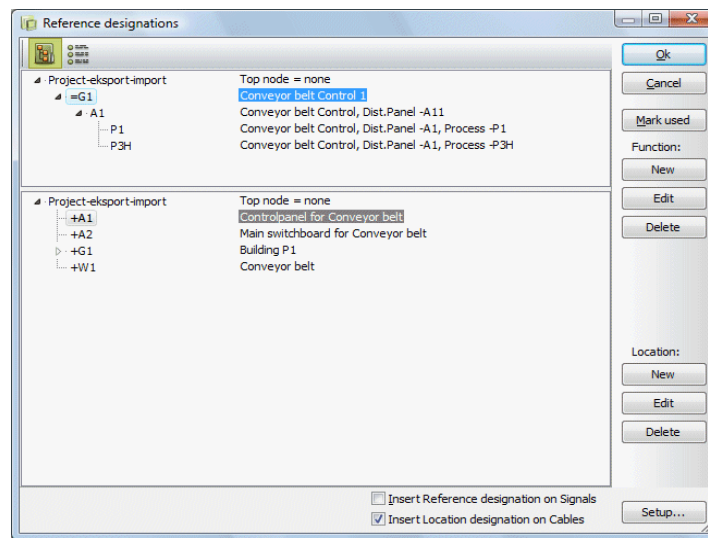


IF THE TAB REF.DESIGN DOES NOT APPEAR

IF THE REFERENCE DESIGNATION PART OF THE DIALOG BOX DOES NOT EXIST, THIS IS BECAUSE NO REFERENCE DESIGNATIONS HAVE BEEN DEFINED IN THE **PROJECT DATA** DIALOG BOX. — SEE HOW TO CREATE THESE IN “DEFINING REFERENCE DESIGNATIONS FOR THE PROJECT” ON PAGE 325.

Choosing reference designations when placing symbols

When you click on on the tab **Ref. design.** and then **Select**, you enter this dialog box:



ONLY CREATED ASPECT TYPES DISPLAYED

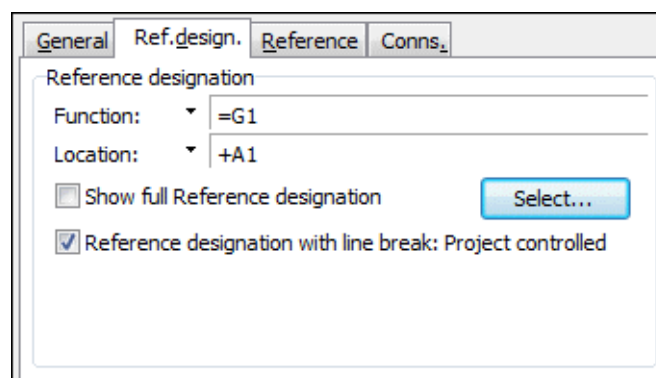
WHEN PLACING SYMBOLS, THE DIALOG BOX **REFERENCE DESIGNATIONS** ONLY DISPLAYS THE ASPECT TYPES, WHICH HAVE BEEN CREATED IN THE PROJECT.

IF FOR INSTANCE YOU HAVE NOT CREATED A LOCATION REFERENCE DESIGNATION '+' THIS PART OF THE DIALOG BOX IS NOT DISPLAYED, AND THE FUNCTION REFERENCE DESIGNATION WINDOW FILLS ALL OF THE DIALOG BOX.

Here you select the desired reference designation, and click **OK**. If one of the aspects shall not be a part of the symbol name, you click on **Top node = none** under the aspect.

If you wish to create a new reference designation, click on **New**, and create the reference designation — see “Specifying function/location reference designations” on page 326.

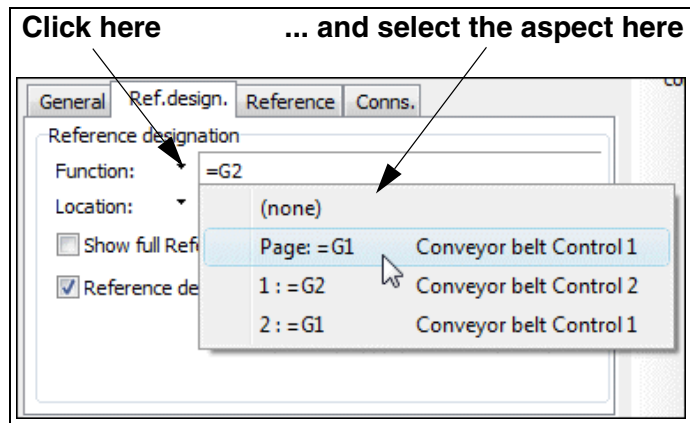
When you return to the **Component data** dialog box, you can see the reference designation on the tab **Ref. design.** in the lower part of the dialog box.



Displaying the page reference designation and the most recently applied reference designations

When you click on the down-arrow next to the **Function** field, the function reference designation for the page and the most recently applied function reference designations are displayed. You can then select a function reference designation in the list.

The function reference designation for the page and the most recently applied function reference designation are displayed in the top of the list.



You can do just the same for location reference designations.

SHORT-CUT KEYS

These functions has the short-cut keys <Ctrl+f> for function reference designations and <Ctrl+l> for location reference designations. The **Ref.design.** tab is automatically displayed when the short-cut keys are pressed.

Show full Reference designation

Because the page can be assigned a reference designation, and because the symbol can be placed in an area on the page containing reference designations, only the part of the reference designations that are different from the page or the area is displayed.

If you wish to display the full reference designation, select **Show full Reference designation**.

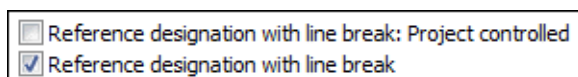
— See “Character for specifying full reference designation (exclude concatenation)” on page 324 also.

Individual control of reference designations with line breaks from Component data

Under **Settings => Project data => Reference designations => Setup** you can specify that the reference designations are displayed in more that one line, as a general setting for the project.

In the dialog box **Component data** on the tab **Ref.design.**, you can — for the selected symbol — *deselect* this setting by removing the check mark in the check box **Reference designation with line break: Project controlled**.

When the check mark is removed, you get the following option:



You can then select/deselect **Reference designation with line break** for the selected symbol, independent of the general project settings.

Type in the symbol name / product reference

Now type in the symbol name and click **OK**.

Editing reference designations for placed symbols

When you want to assign reference designations to a group of placed symbols, do the following:



- 1) Click the **Symbols** button and select the symbols
- 2) Right-click on the selected symbols, and choose **Component data**
- 3) In the dialog box **Component data**: Click on the tab **Ref.design.** and click on **Select**
- 4) In the dialog box **Reference Designations**: Choose reference designation and click **OK**
- 5) In the dialog box **Component data**: Click **OK**
- 6) Click somewhere on the page — to deselect the symbols

You have now assigned reference designations to the selected symbols.

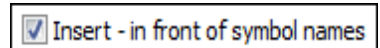
Editing reference designations using the Select name function

— See “Changing the symbol reference designations using Select name” on page 229.

Preceding “-” in symbol names

The symbols in the **EN60617** folder does not have a preceding “-” in their symbol names.

Therefore enter **Settings => Pointer/Screen** and select **Insert - in front of symbol names**. All symbols you place in the following, will have a preceding “-” in their symbol names.



Adding “-” in already placed symbols

If you have an old project without “-” in the symbol names, this prefix can be added automatically to all symbols in the project, by choosing **Functions => Special Functions => Prefix all symbol names with -**.

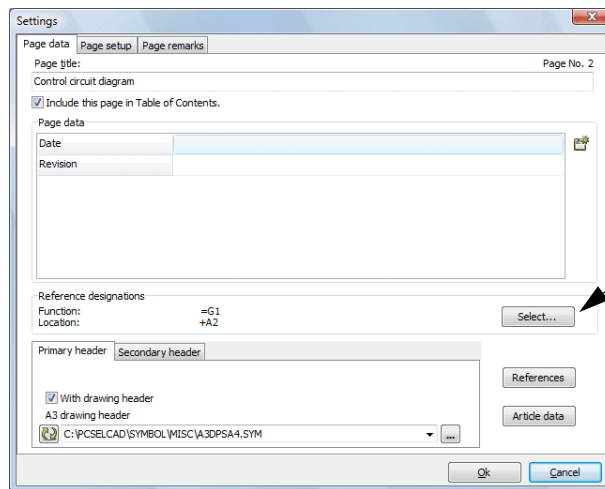
Inserting reference designations on signal symbols and symbol references

— See “Reference designations on symbol and signal references” on page 360.

REFERENCE DESIGNATIONS FOR A FULL PAGE

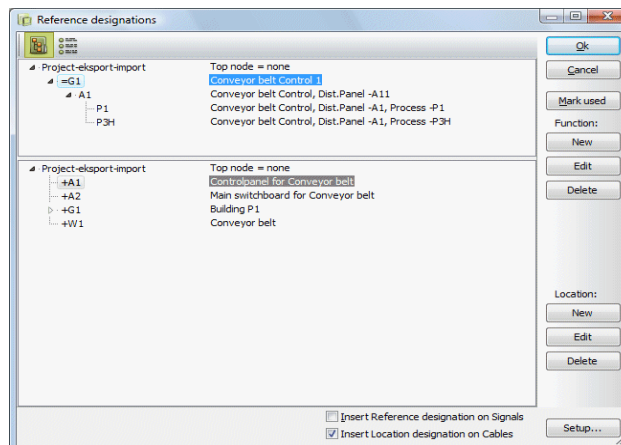
It is also possible to specify reference designations for a full page.

To do this you enter **Settings => Page data**.



Click here

Here you click on **Select** under **Reference designations**, and enter the dialog box **Reference designations**:



Then click on the reference designations you wish to select for the page and click **OK**.

You now return to the dialog box **Page data**.

The page reference is not displayed on the page

Please note, that it is not displayed on the project page, that reference designations have been attached to it. — See “Inserting a drawing header displaying reference designations” on page 341.

Click **OK**.



ALREADY PLACED SYMBOLS ARE NOT ASSIGNED NEW REF. DESIGNATIONS

SYMBOLS *ALREADY PLACED* ON THE PAGE, WILL NOT AUTOMATICALLY GET THE NEW REFERENCE DESIGNATION FOR THE PAGE.

SYMBOLS PLACED *AFTER ASSIGNING THE REFERENCE DESIGNATION* WILL AUTOMATICALLY GET THE NEW REFERENCE DESIGNATIONS FOR THE PAGE.

Symbol names on pages with assigned reference designations

Every symbol you place on a page with reference designations, are assigned the reference designation of the page — unless you specify otherwise for the individual symbols in their respective **Component data** dialog boxes.

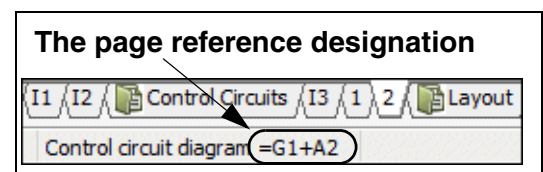
Because the symbols are conceived as belonging to the reference designation of the page, this reference designation is not displayed as a part of the symbol name on the page. Reference designations will only be displayed when they are different from the reference designation of the page.

Using the Show available function on pages with reference designations

When you via the **Show available** function place symbols for a component, which have already been assigned a reference designation, this symbol keeps the assigned reference designation.

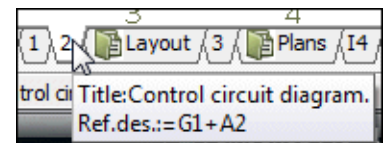
The page reference designation displayed in the Status line

In the status line at the bottom of the screen you can see the reference designation of the page along with the page title. Click **OK**.



Page reference designation displayed as popup information

When you let the mouse cursor rest over a page tab, the page reference designation is displayed in the popup information.



Inserting a drawing header displaying reference designations

If you do not take action yourself, the reference designation of the page is not displayed on the project page itself. Therefore it is a good idea to place a drawing header on the page, in which the reference designation of the page is displayed.

To do this, you must place the datafield **Reference designation** of the type **Page data** in the drawing header. — See “Datafield for reference designations in projects” on page 351.

Please refer to “Creating Drawing Headers” on page 721 for further details on creating drawing headers.

Inserting a datafield with reference designations

You can also place the datafield directly on the page.

To do this choose **Insert => Insert datafield**. Then click on **Page Data**, choose the datafield **Reference designation**, and click **OK**.

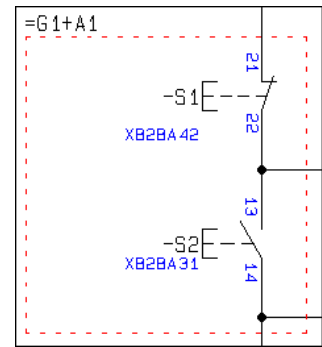
The datafields **Next used page** and **Previous used page** can also display the reference designation of the pages they refer to.

REFERENCE DESIGNATIONS FOR AREAS ON PAGES

It is also possible to specify that areas on pages shall have an attached reference designation.

To do this choose **Insert => Insert Reference Frame**.

You then get a pencil in the cross hairs, and draw a frame for the area with the reference designation.



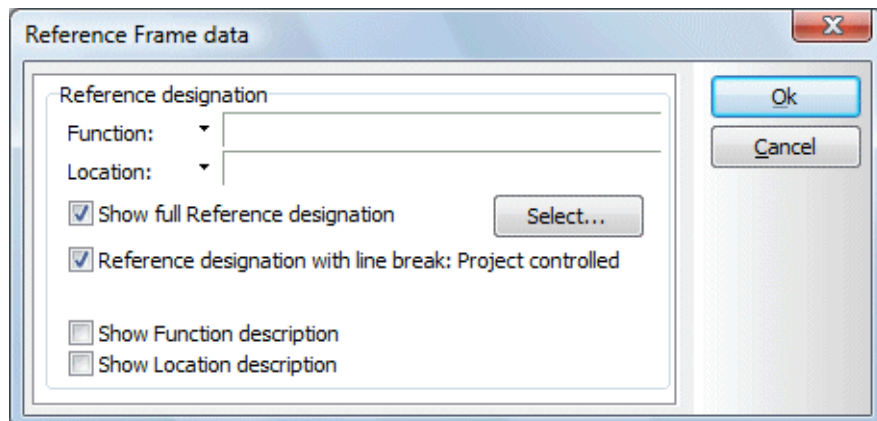
THE PROGRAM MEMORIZES THE LINES SETTINGS

THE PROGRAM AUTOMATICALLY USES THE LAST APPLIED LINES SETTINGS FOR **INSERT REFERENCE FRAME**.

AFTER DRAWING THE REFERENCE FRAME, THE PROGRAM AUTOMATICALLY CHANGES TO THE LINES SETTINGS, WHICH YOU WERE USING BEFORE INSERTING THE REFERENCE FRAME.

— READ MORE ABOUT LINES SETTINGS IN “THE LINES TOOLBAR” ON PAGE 66.

When you have drawn the frame, you enter this dialog box:



Here you select the relevant reference designation as described in “Selecting Reference designations when placing symbols” on page 336, and click **OK**.

You now return to the above dialog box, where the selected reference designation is displayed.

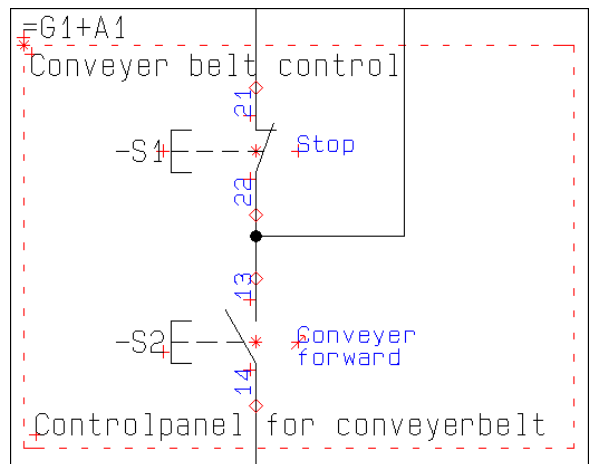
Because the page also can have assigned a reference designation, the only the part of the reference designation which is displayed is the part which is different from the reference designation of the page.

Show full Reference designation

When you want the full reference designation to be displayed, select **Show full Reference designation**.

Show Function description and Show Location description

If you select **Show Function description** and/or **Show Location description**, the function description and/or the location description for the reference frame will be displayed. Here the description for the function aspect will be placed starting in the upper-left corner of the frame, while the location will be placed starting in the lower-left corner of the frame.



Both texts can be moved freely, and their text data can be edited, as for other texts in the program. —

See “Moving symbol texts on a Diagram page” on page 106 and “Specifying text properties / Appearance of text” on page 118.

If the frame is moved, the texts are moved relative to the frame, which is also the case for the reference designation text.

Individual control of reference designations with line break

— See “Individual control of reference designations with line breaks from Component data” on page 338.

Click OK

When the dialog box is filled out, click **OK**.



DRAW THE REFERENCE FRAME BEFORE PLACING THE SYMBOLS

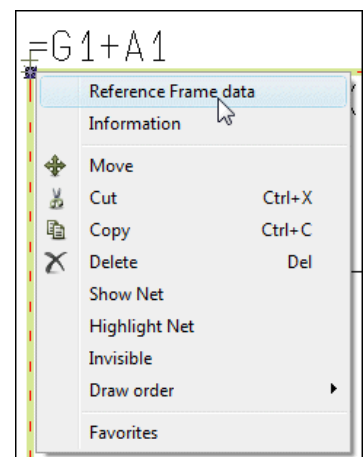
WHEN YOU CREATE REFERENCE DESIGNATIONS FOR AREAS YOU *FIRST* DRAW THE FRAME, AND *THEN* PLACE THE SYMBOLS IN THE FRAME. THIS WAY THE SYMBOLS ARE ASSIGNED THE REFERENCE DESIGNATIONS FOR THE AREA.

Editing reference designations for areas

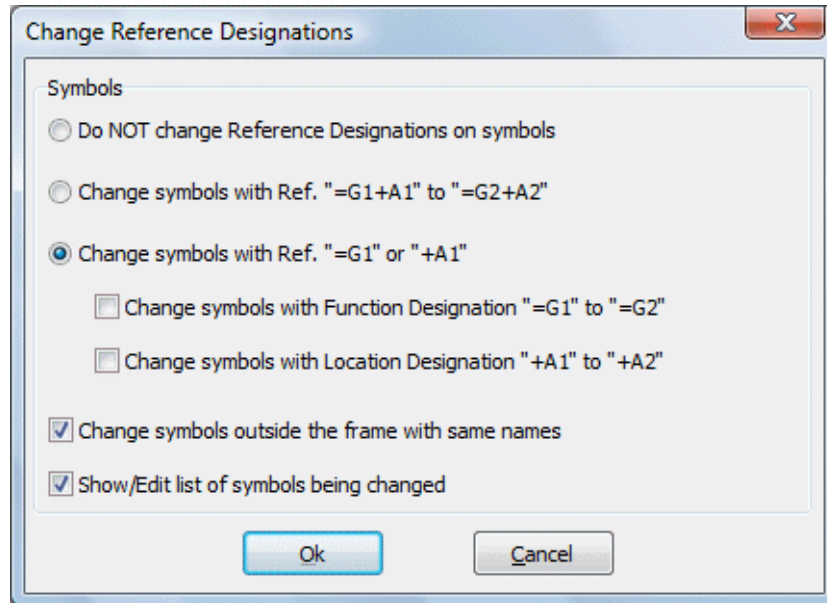


If you want to change the reference designation for the area, click the **Lines** button, right-click on the frame, and choose **Reference Frame data**.

You then enter the dialog box **Reference Frame data**, as described in “Reference designations for Areas on pages” on page 342. Here you type in the new reference designations for the area / reference frame.



When you change the reference designation on a reference frame, you are asked whether/how to change the reference designations for symbols placed *inside* the reference frame *with* the previous reference designation for the frame.



Specifying which symbols inside the frame to assign new reference designations to

In this dialog box you specify how/whether to update the symbols:

- **Do NOT change Reference Designations on symbols:** The reference designations remains unchanged for the symbols inside the reference frame
- **Change symbols with Ref. "=G1+A1" to "=G2+A2":** Symbols with the same full reference designation as the reference frame, are updated
- **Change symbols with Ref. "=G1" or "+A1":** You get the option to further specify whether the function and/or location aspect to change for symbols with matching function/location aspect inside the frame

Change symbols outside the frame with same names

If you choose **Change symbols outside the frame with same names**, all symbols outside the frame are changed, if they have the same symbol name *and* reference designation, as any of the symbols inside the frame.

If the reference frame is placed on a page, which you have just copied into the project, you will typically *not* select this option — so that you do not change names on other project pages by accident.

When there are symbols inside the reference frame, which are part of a component, for which there also are placed symbols outside the reference frame, it can be important that **Change symbols outside the frame with same names** is selected. This ensures you that all symbols for the same component still have the same reference designations. — If this is not the case, the program will no longer understand these symbols as being parts of the same component.

Show/Edit list of symbols being placed

If you choose **Show/Edit list of symbols being placed**, you subsequently enter a dialog box where you can see which symbols **Automation** will change reference

designations for. Here you can also specify not to assign new reference designations to some of the symbols in the list, by removing the check mark for these symbols in the list. — See also “Show/Edit list of symbols being changed” on page 350.

Click OK

Make the desired changes in the dialog box **Change Reference designations**, and click **OK**.

See all reference frames in the project

To see all areas with reference designations in the project, you can enter the **Object Lister** and choose **Reference Frames**.

— See “The Object Lister” on page 296.

EDITING REFERENCE DESIGNATIONS FOR CHAPTERS



To assign the same reference designation to a full chapter, enter the **Page menu** — as described in “Edit reference designations for more than one page” on page 346 —, select all pages in the project, and change the reference designations for the pages.



YOU MUST ASSIGN REF.DES. MANUALLY TO NEW PAGES IN CHAPTERS

WHEN YOU INSERT A NEW PAGE IN A CHAPTER, THE PAGE IS *NOT* ASSIGNED REFERENCE DESIGNATIONS AUTOMATICALLY.

YOU MUST DO THIS MANUALLY, AS DESCRIBED IN “REFERENCE DESIGNATIONS FOR A FULL PAGE” ON PAGE 340.

— Please refer to “Change reference designations via chapter Divider pages” on page 349 also.

EDIT REFERENCE DESIGNATIONS FOR MORE THAN ONE PAGE

To edit reference designations for one or more pages in one operation, do this:

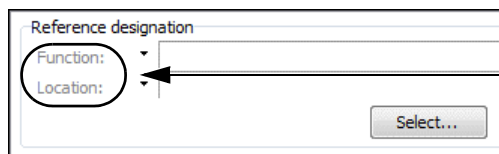


- 1) Click the **Pagemenu** button, or select **View => Select page**.
- 2) In the **Page menu**: Select the pages for which you wish to change reference designations — eg. by pressing down the <Ctrl> key, and clicking on the pages one by one.
- 3) Click on **Article data**.



IF THE PAGES DO NOT HAVE IDENTICAL REFERENCE DESIGNATIONS

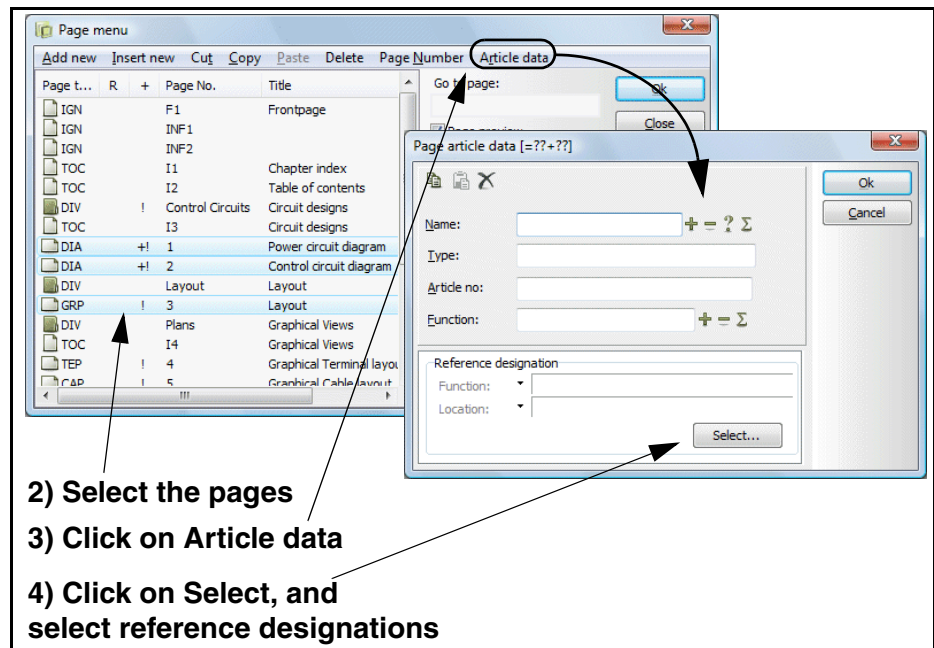
IN THE DIALOG BOX **PAGE ARTICLE DATA**: IF THE TEXTS **FUNCTION** AND/OR **LOCATION** ARE DIMMED, THIS MEANS THAT THE SELECTED PAGES DO NOT HAVE IDENTICAL REFERENCE DESIGNATIONS.



NOTE: If *dimmed*, only the ref.des. of the pages can be changed

IF THIS IS THE CASE, IT IS ONLY POSSIBLE TO EDIT THE REFERENCE DESIGNATIONS FOR THE *PAGES*. THE REFERENCE DESIGNATIONS FOR THE *SYMBOLS* WILL *NOT* BE CHANGED.

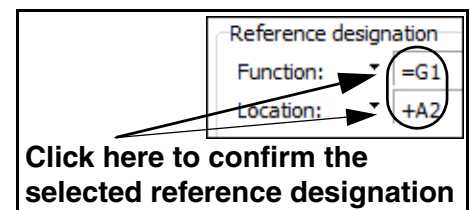
- 4) In the dialog box **Page article data**: Click on **Select** under **Reference designations**:



- 5) In the dialog box **Reference designations**: Select the new reference designations, and click **OK**. What to do next depends on whether the selected pages had identical reference designations:

If the pages did NOT have identical reference designations

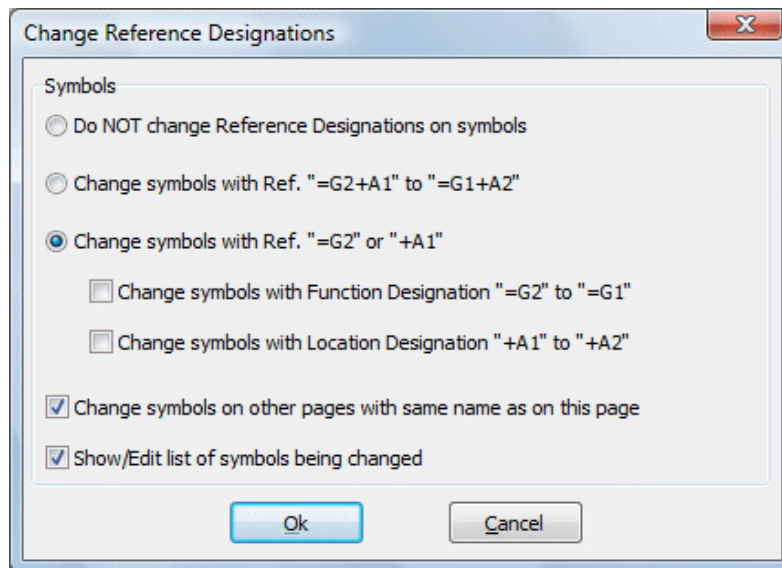
- 6a) If the selected pages did *not* have identical reference designations, do the following in the dialog box **Page article data**: Click on one — or both — of the reference designations, to confirm that you wish to select this reference designation. This causes **Function** and/or **Location** to be written in black, and the selected pages are assigned the reference designation — which is displayed in the title bar of the dialog box **Page article data**. (Please note, that *only the pages*, and *not the symbols* on the pages, will have their reference designations changed — as described in the box “If the pages do not have identical reference designations” on page 346). Klik **OK**.



If the pages did have identical reference designations

- 6b) If the selected pages had *identical* reference designations, you enter the dialog box **Change Reference Designations**. Here you get the option of

changing reference designations for the implicated symbols:



— Read about the options in this dialog box in “Options in the dialog box Change Reference Designations” on page 349. Choose how to change reference designations for the involved symbols, and click **OK**. If you have chosen **Show/Edit list of symbols being changed**, please also see “Show/Edit list of symbols being changed” on page 350.

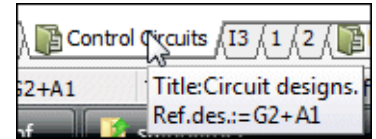
Click OK in the Page menu

7) You now return to the **Page menu**: Click **OK**.

CHANGE REFERENCE DESIGNATIONS VIA CHAPTER DIVIDER PAGES

To change reference designations via a chapter tab, do this:

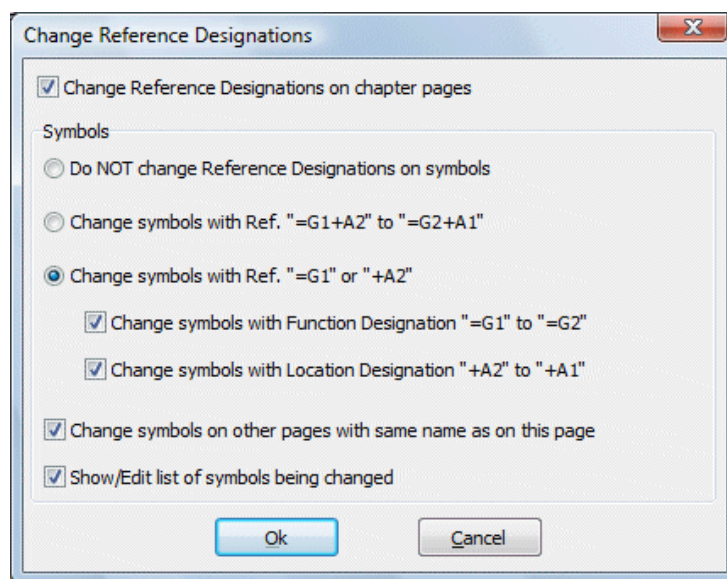
- 1) Click on the page tab for the divider page for the chapter (The chapter front page).
- 2) Select **Settings => Page data**, and click on **Select** below **Reference designations**.



- 3) In the dialog box **Reference designations**: Choose reference designations, and click **OK**.

Options in the dialog box Change Reference Designations

- 4) You now enter the dialog box **Reference designations**:



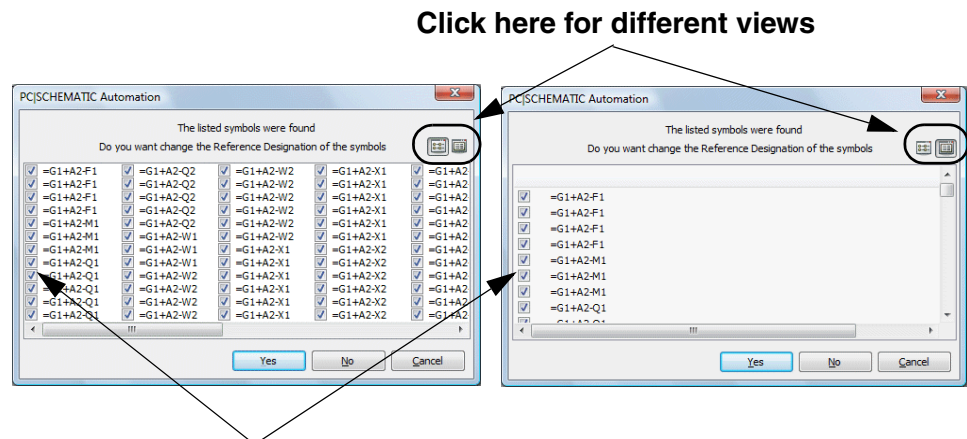
Here you are asked whether to change reference designations on the pages in the chapter having the same reference designations as the divider page, and which directions to apply for changing the symbols. You have the following options:

OPTION	EFFECT
CHANGE REFERENCE DESIGNATIONS ON CHAPTER PAGES	SELECT THIS OPTION IN ORDER TO CHANGE REFERENCE DESIGNATIONS FOR THE CHAPTER PAGES HAVING THE SAME REFERENCE DESIGNATIONS AS THE DIVIDER PAGE. YOU WILL NOT BE ABLE TO SELECT THE OPTIONS BELOW, BEFORE YOU HAVE SELECTED THIS OPTION.
DO NOT CHANGE REFERENCE DESIGNATIONS ON SYMBOLS	CLICK HERE IN ORDER NOT TO CHANGE REFERENCE DESIGNATIONS ON ANY SYMBOLS.
CHANGE SYMBOLS WITH REF. "X" TO "Y"	CLICK HERE TO CHANGE BOTH THE FUNCTION AND THE LOCATION REFERENCE DESIGNATION.
CHANGE SYMBOLS WITH REF. "X" OR "Y"	CLICK HERE TO CHANGE EITHER THE FUNCTION OR THE LOCATION REFERENCE DESIGNATION.

OPTION	EFFECT
CHANGE SYMBOLS WITH FUNCTION REFERENCE DESIGNATION "X" TO "Y"	FIRST CHOOSE CHANGE SYMBOLS WITH REF. "X" OR "Y": THEN CLICK HERE ONLY TO CHANGE FUNCTION REFERENCE DESIGNATION.
CHANGE SYMBOLS WITH LOCATION REFERENCE DESIGNATION "X" TO "Y"	FIRST CHOOSE CHANGE SYMBOLS WITH REF. "X" OR "Y": THEN CLICK HERE ONLY TO CHANGE LOCATION REFERENCE DESIGNATION.
CHANGE SYMBOLS ON OTHER PAGES WITH SAME NAME AS ON THIS PAGE	CLICK HERE, SO THAT YOU ALSO CHANGE REFERENCE DESIGNATIONS FOR SYMBOLS OUTSIDE THE CHAPTER, WHICH ARE A PART OF THOSE OF THE COMPONENTS IN THE CHAPTER, WHICH WILL HAVE THEIR REFERENCE DESIGNATIONS CHANGED.
SHOW/EDIT LIST OF SYMBOLS BEING CHANGED	CLICK HERE TO GET A LIST OF WHICH SYMBOLS THAT ARE GOING TO BE CHANGED. IN THIS LIST YOU CAN DESELECT SYMBOLS, FOR WHICH YOU DO NOT WISH TO CHANGE REFERENCE DESIGNATIONS.

Show/Edit list of symbols being changed

5) If you have chosen **Show/Edit list of symbols being changed**, you now enter the following dialog box:



Click here to select/deselect symbols

Remove the check mark in front of the symbols, that shall not have their reference designations changed, and click **OK**.

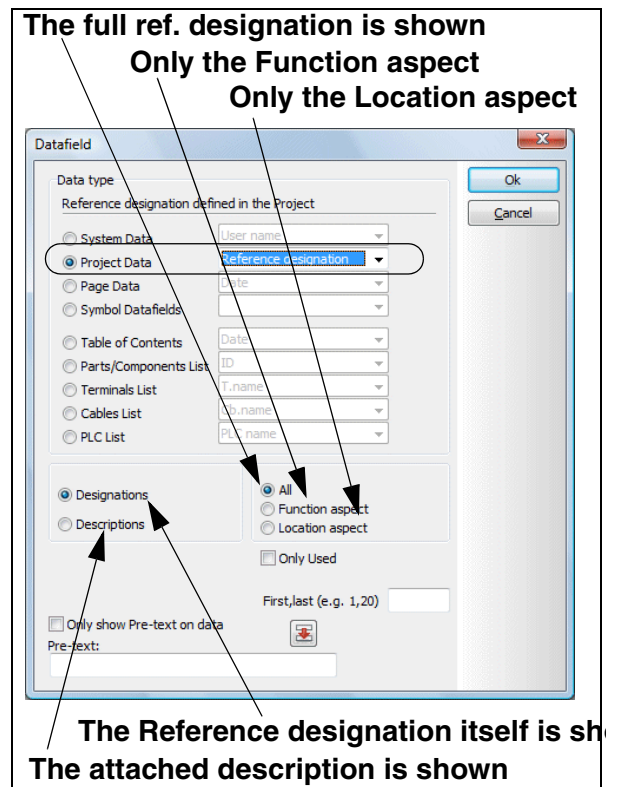
6) In the dialog box **Settings**: Click **OK**

You have now changed reference designations for the desired pages and symbols.

DATAFIELD FOR REFERENCE DESIGNATIONS IN PROJECTS

When you wish to place a datafield containing reference designations in a list or on a diagram page, do the following:

- 1) Choose **Insert => Insert Datafield**.
- 2) In the dialog box **Datafield**: Click on **Project Data**, and choose the datafield **Reference designation**.
- 3) Choose the relevant part of the reference designation — as displayed in the figure to the right.
- 4) Click **OK**.



REFERENCE DESIGNATIONS IN ALPHABETICAL ORDER

WHEN REFERENCE DESIGNATIONS ARE BEING PRINTED VIA THIS DATAFIELD, THEY ALWAYS APPEAR IN ALPHABETICAL ORDER.

Specify which reference designations to display

When you have selected the datafield **Reference designation** in the **Project data** group, you can choose only to display a limited part of the reference designations.

First,last (e.g. 1,20)

This is done in the field **First,last**, where you type in the number of the first and the last reference designation you wish to display.

When for instance you type in *1,20* the first 20 reference designations are displayed, if you type in *21,40* the subsequent 20 reference designations are displayed. This way it is possible to list the reference designations over several pages or and/or columns, when a project contains many reference designations.

LISTS FOR EACH FUNCTION OR LOCATION

When you apply reference designations, it is possible to create separate lists for each Function or Location in the project.

This is done by clicking on the list in the project you want divided following these criteria. Here you choose **Lists => List settings** and enter the dialog box **List setup**, where you click on the **Criteria** tab.

If you want a list of every symbol having the **Function** aspect **=P1** — which for instance could be a pump — you fill out the **Criteria** tab like this:

Field	Operator	Text	A<>a
Name	=	=P1*	<input type="checkbox"/>

Or fill out the **Criteria** tab like this to get a list of the assembly **+A1**:

Field	Operator	Text	A<>a
Name	=	=*A1*	<input type="checkbox"/>

— Please refer to “Specifying Criteria for the components to include” on page 539 for further details on criteria on lists.



References

This section describes the use of references in PCISCHMATIC Automation:

- Automatically updated references(p. 355)
- Contact mirrors and Reference crosses(p. 362)
- Reference system settings(p. 367)



AUTOMATICALLY UPDATED REFERENCES

When you place more than one symbol for the same component in a project, the program can automatically create references between these symbols on the project pages.

Next to each symbol you can therefore see references to the other symbols for the same component. The appearance of these references you can determine yourself.

THIS CHAPTER DESCRIBES	
— INSERTING A REFERENCE SYSTEM.....	(P. 355)
— AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK	(P. 355)
— REFERENCES ON SYMBOLS.....	(P. 357)
— APPEARANCE OF REFERENCES	(P. 360)
— REFERENCE DESIGNATIONS ON SYMBOL AND SIGNAL REFERENCES.....	(P. 360)
— REFERENCES FOR PLCs	(P. 361)

— Please see to “Contact mirrors and Reference crosses” on page 362.

— Please also see “Navigation between symbols for the same component” on page 165.

INSERTING A REFERENCE SYSTEM

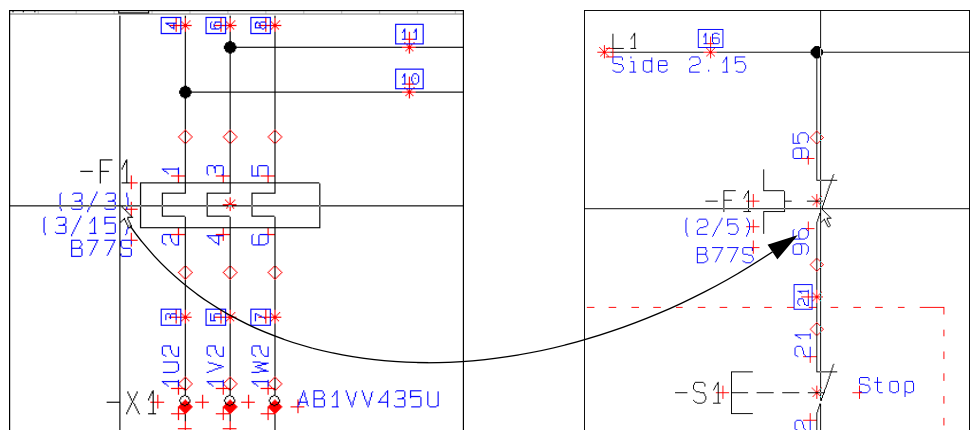
In order to let the program create appropriate references, you must first create/ adapt the reference system for the diagram pages, so that it meets your demands. Read more about this in “Setting up the current paths” on page 367.



AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK

To jump to a symbol which is referred to, do the following:

- 1) Double-click on the reference.

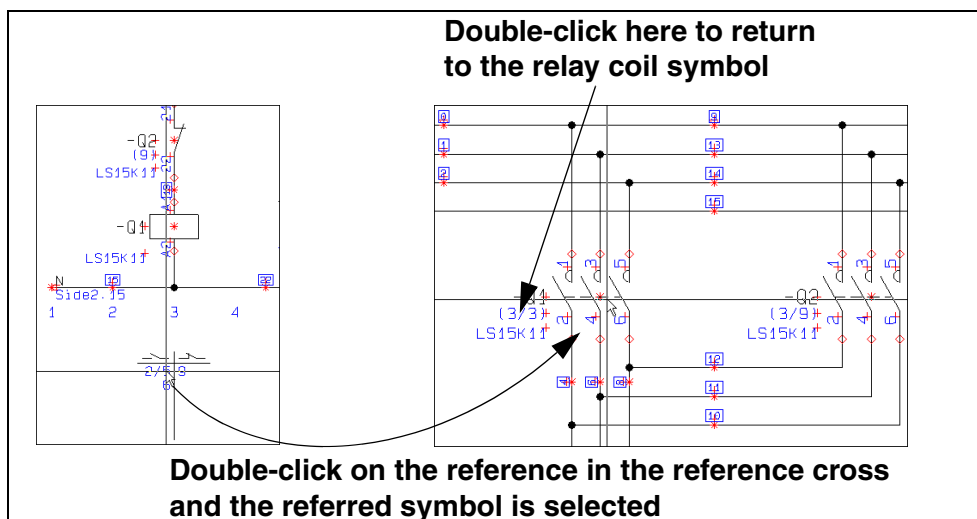


Double-click on the reference, and the referred symbol is selected

- 2) The program now automatically jumps to the project page, where the referred to symbol is placed. The referred symbol is selected by the cross hairs.

Automatic jump to reference from reference cross

Correspondingly you can double-click on the reference in a reference cross or in a contact mirror:



When you double-click on the reference for a symbol in the reference cross/contact mirror, you return to the symbol with the symbol having the symbol type **Relay**.



SYMBOL REFERENCES ARE ACTIVE IN ALL COMMANDS

WHEN YOU DOUBLE-CLICK ON A REFERENCE FOR A SYMBOL, **Automation** JUMPS TO THE REFERRED SYMBOL — NO MATTER IF YOU WORK WITH LINES, TEXTS, CIRCLES, SYMBOLS OR AREAS.

REFERENCES ON SYMBOLS

In **Automation** you can apply references on *all* types of symbols.

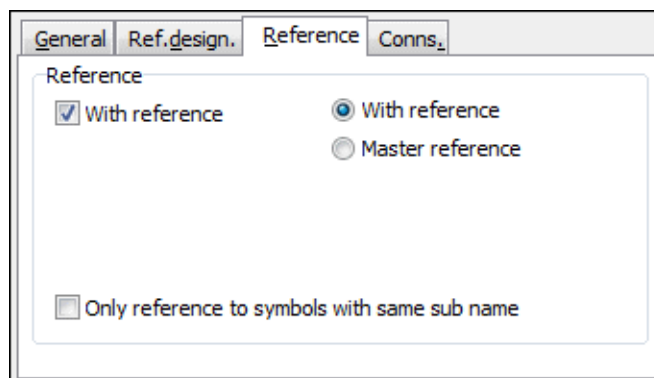
How the reference appears for a symbol, primarily depends on the **symbol type** of the symbol. However, some symbols are created without symbol type, and therefore has no references.

Inserting references on symbols without reference

To insert references on symbols without references — or to change the selected reference for a symbol — do the following:



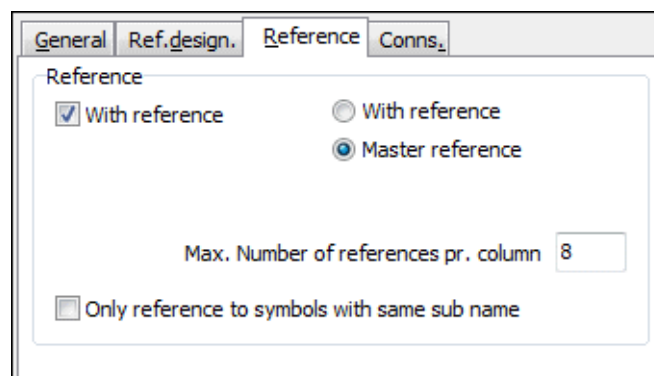
- 1) Click on the **Symbols** button, right-click on the symbol, and choose **Component data** — or double-click on the symbol.
- 2) In the dialog box **Component data**: Click on the **Reference** tab, choose **With reference** and then either **Master reference** or **With reference**, and click **OK**.



Choosing the reference type Master reference

If you select the reference type **Master reference**, a list is created next to the symbol, containing references to the other symbols having the same symbol name — if these are set to **With reference**.

It is possible to set more than one symbol with the same symbol name to **Master reference**. However, from the symbols set to **With reference**, there are only a reference to the first symbol with **Master reference** in the documentation.



References can be placed in columns. How many references to place in each column, is specified in the field **Max. Number of references pr. column**.

When you choose **Master reference** for PLC symbols, a PLC I/O reference cross is placed next to the symbol.

Choosing the reference type *With reference*

When you choose the reference type **With reference**, there will be placed a reference next to the symbol, which refers to the symbol(s) that has the same symbol name and are set to **Master reference**.

When there are no symbols with the same symbol name which are set to **Master reference**, a reference is placed to the preceding and the succeeding symbol in the project. Hereby a chain of references between symbols with the same symbol name is created. This makes it easy to run through the project and find all symbols having the same symbol name. All you need to do is to double-click on the references, and you jump to the next — or previous — symbol in the chain.

Sub names and references

If you only want references between symbols with the same subnames within a component, do this:

- 1) Double-click on one of the symbols for the component.
- 2) In the dialog box **Component data**: Click on the **Reference** tab at the bottom of the dialog box.
- 3) In the **Reference** tab: Select **Only reference to symbols with same sub name**.

If **Only reference to symbols with same sub name** is *not* selected, references will appear for all symbols with the same name, no matter what their sub names are.

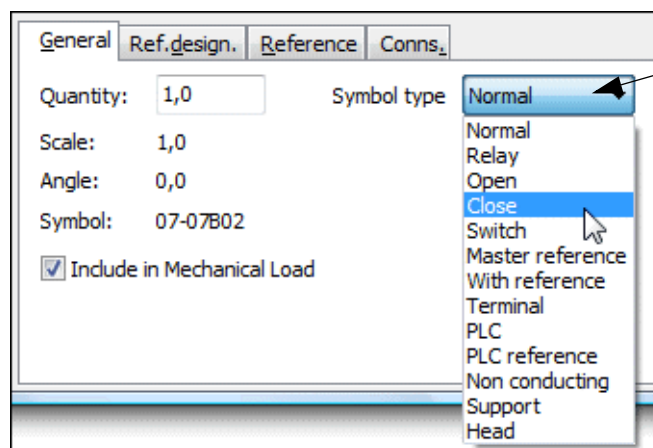
Symbol type and References

The symbol type is specified in the symbol definition — see “Creating Symbols” on page 676; see "Symbol types" in particular.

However, you can change the symbol type for a symbol like this:



- 1) Click on the **Symbols** button — to work with symbols.
- 2) Right-click on the symbol and choose **Component data**.
- 3) Click on the **General** tab, and click on the down-arrow in the **Symbol type** field:



Click here to choose symbol type

4) You can now choose between the following symbol types:

SYMBOL TYPE	STANDARD REFERENCES FOR THE SYMBOL TYPE
NORMAL	THE SYMBOL IS NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
RELAY	A REFERENCE CROSS/CONTACT MIRROR IS CREATED FOR THE SYMBOL, WHEN PLACED ON A DIAGRAM.
OPEN	THE SYMBOL WILL BE CONCEIVED AS AN OPEN CONTACT, AND THERE WILL BE PLACED A REFERENCE TO THE SYMBOL IN THE REFERENCE CROSS/CONTACT MIRROR BELOW THE RELAY COIL SYMBOL FOR THE COMPONENT. THE SYMBOL WILL ALSO GET A REFERENCE TO THE RELAY COIL SYMBOL, WHEN THE SYMBOL IS PLACED IN THE PROJECT.
CLOSE	THE SYMBOL WILL BE CONCEIVED AS A CLOSE CONTACT, AND THERE WILL BE PLACED A REFERENCE TO THE SYMBOL IN THE REFERENCE CROSS/CONTACT MIRROR BELOW THE RELAY COIL SYMBOL FOR THE COMPONENT. THE SYMBOL WILL ALSO GET A REFERENCE TO THE RELAY COIL SYMBOL, WHEN THE SYMBOL IS PLACED IN THE PROJECT.
SWITCH	THE SYMBOL WILL BE CONCEIVED AS A SWITCH, AND THERE WILL BE PLACED A REFERENCE TO THE SYMBOL IN THE REFERENCE CROSS/CONTACT MIRROR BELOW THE RELAY COIL SYMBOL FOR THE COMPONENT. IN REFERENCE CROSSES, THERE WILL BE A REFERENCE TO THE SYMBOL IN BOTH THE OPEN AND THE CLOSE CONTACT SIDE. THE SYMBOL WILL ALSO GET A REFERENCE TO THE RELAY COIL SYMBOL, WHEN THE SYMBOL IS PLACED IN THE PROJECT.
MASTER REFERENCE	THE SYMBOL WILL HAVE REFERENCES TO ALL OTHER SYMBOLS OF THE SAME SYMBOL NAME. FOR INSTANCE HAND OPERATED CONTACTS
WITH REFERENCE	REFERS EITHER TO A SYMBOL WITH MASTER REFERENCE , OR TO SYMBOLS IN THE CLOSEST PROCEEDING OR SUCCEEDING CURRENT PATH, WHICH ARE PART OF THE SAME COMPONENT.
TERMINAL	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
PLC	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
NON CONDUCTING	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
SUPPORT	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.

5) Choose symbol type and click **OK**.



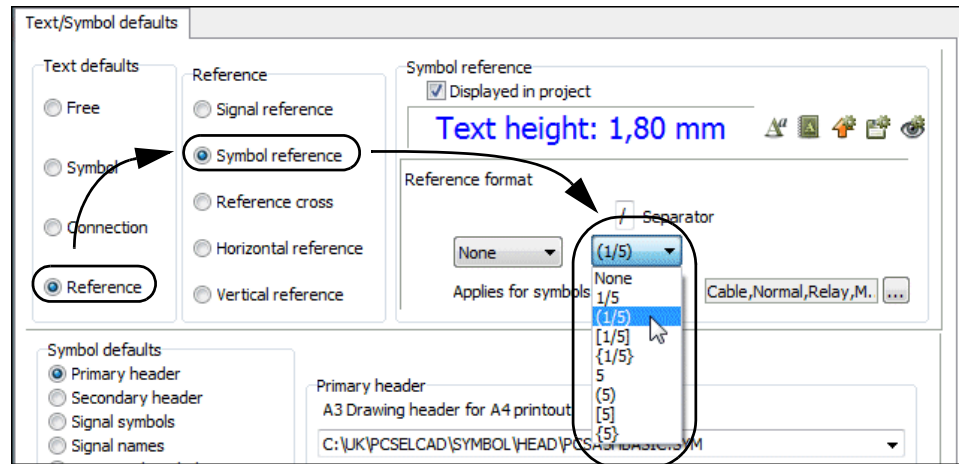
ALL SYMBOL TYPES CAN BE CREATED WITH ACTIVE WITH REFERENCE

ALL SYMBOL TYPES CAN BE CREATED WITH **WITH REFERENCE** ACTIVE VIA THEIR SYMBOL DEFINITION. — READ MORE ABOUT THIS IN “**CREATING SYMBOLS**” ON PAGE 676, “SYMBOL SETTINGS: REFERENCE” ON PAGE 689 IN PARTICULAR.

APPEARANCE OF REFERENCES

To specify the appearance of the reference, do the following:

- 1) Choose **Settings => Text/Symbol defaults**.
- 2) Click on **References**.
- 3) Then click on **Symbol reference** to specify the appearance of the symbol references.
- 4) You now get the following options:



- 5) Now choose **Reference format** and **Separator** and brackets if necessary.



- 6) When you wish also to change the appearance of the text, click the **Text properties** button, as described in “Specifying text properties / Appearance of text” on page 118.
- 7) Click **OK**.
- 8) Choose **Functions => Update references** — and the changes are displayed in the project.

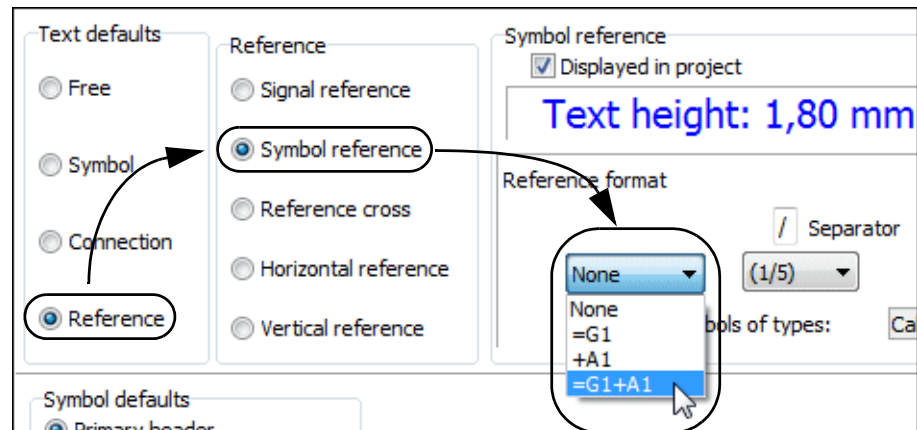
When you wish to change the appearance for references for signal symbols, just choose **Signal reference** in step 3 above.

REFERENCE DESIGNATIONS ON SYMBOL AND SIGNAL REFERENCES

To let references for symbols contain reference designations for the page that are referred to, do the following:

- 1) Choose **Settings => Text/Symbol defaults**.
- 2) Click on **References**.
- 3) Then click on **Symbol reference** to specify the appearance of the references for symbols.

- 4) Now choose a reference format with reference designations, as displayed in the figure below:



- 5) Click **OK**.

Signal symbol references with reference designations

You can, correspondingly, activate reference designations for signal symbols by choosing **Signal reference** in stead of **Symbol reference** in the procedure above.

REFERENCES FOR PLCs

— See "Jumping between PLC ref. symbols and PLC I/O symbols" på side 444.



CONTACT MIRRORS AND REFERENCE CROSSES

When displaying references for symbols with the type **Relay** — see “References on symbols” on page 357 — you can either use *reference crosses* or *contact mirrors*.

This chapter describes how to use reference crosses and contact mirrors in the program.

THIS CHAPTER DESCRIBES	
— ON CONTACT MIRRORS	(P. 363)
— ON REFERENCE CROSSES	(P. 364)
— CHOOSING BETWEEN CONTACT MIRRORS AND REFERENCE CROSSES	(P. 365)
— CONTACT MIRRORS ARE CREATED AUTOMATICALLY BY THE PROGRAM	(P. 365)

— Please read “Setting up the current paths” on page 367 to see how to set the reference system on the project pages.

— Please also read “Automatically updated references” on page 355 for general information on references in **Automation**.



CONTACT MIRRORS GIVES QUICK ACCESS TO AVAILABLE SYMBOLS

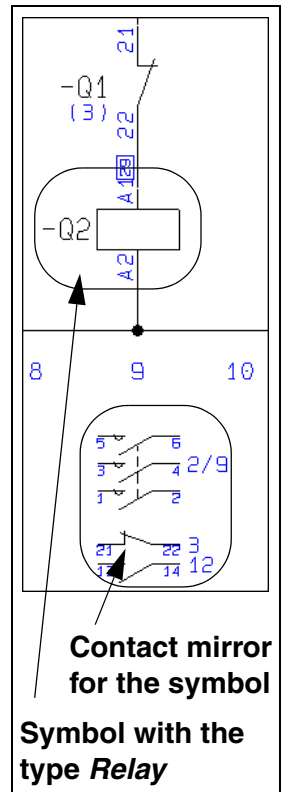
CONTACT MIRRORS ALSO GIVES YOU THE OPTION TO GET THE AVAILABLE SYMBOLS FOR A COMPONENT IN THE CROSS HAIRS RIGHT AWAY.

ON CONTACT MIRRORS

If you have set the program to use contact mirrors — see “Choosing between contact mirrors and reference crosses” on page 365 — and place a symbol with an article number from the attached database, the other symbols for the component are displayed at the bottom of the page.

Thus contact mirrors are a graphical representation of the other symbols for the component. The contact mirror is placed under the symbol for the component, which has the symbol type **Relay**.

When for instance you place a relay coil symbol via the database, the other symbols for the component are displayed below the relay coil symbol at the bottom of the page — as displayed on the figure to the right. In the contact mirror for the relay coil symbol you see a three-poled open contact, a close contact and an open contact.



Contact mirrors and changes in the article number

When you change the article number for a symbol, the contact mirror will be changed, so that it matches the information about the article number in the database.

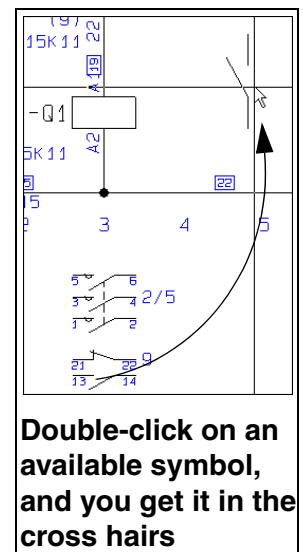
Direct jump to the other symbols for the component

For the symbols, which are already placed in the project, a reference to their location is displayed. When you double-click on one of these symbols, you jump directly to the symbol in the project. — See “Automatic jump between references on double-click” on page 355.

Get available symbols from the contact mirror directly into the cross hairs

When you double-click on one of the symbols in the contact mirror, which is not placed in the project, you get the symbol in the cross hairs, and can then place it in the project.

When a symbol is deleted from the project, the reference to the symbol is removed from the contact mirror, and you can once again see that the symbol is available in the contact mirror.



ON REFERENCE CROSSES

When you are using reference crosses, a reference cross is displayed under all symbols of the symbol type **Relay**.

Under the open contact in the reference cross, references to all the open contacts for the component are displayed. Also, references to all the close contacts are displayed below the close contact in the reference cross.

When you double-click on one of these references, you jump directly to the symbol in the project, which the reference refers to. — See “Automatic jump between references on double-click” on page 355.

Available symbols are not displayed in reference crosses

Available symbols for the component are not displayed in the reference cross.

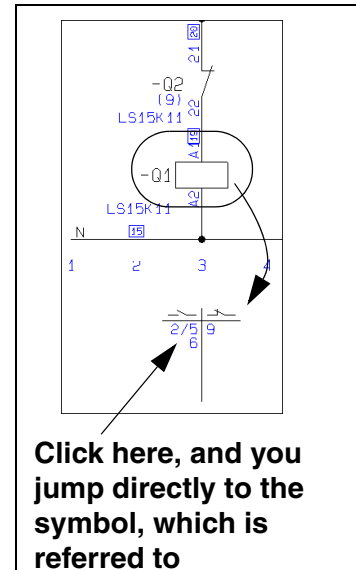
When you are using reference crosses, you therefore do not have the option to get available symbols directly in the cross hairs, as when you use contact mirrors — see “Get available symbols from the contact mirror directly into the cross hairs” on page 363.

Reference crosses for symbols with sub names

For components containing symbols with sub names, independent reference crosses are created for each of the relay coil symbols for the component, which has independent sub names.

Hereby components with several functions can be separated functionally on the project pages, and still appear as one component in the project lists.

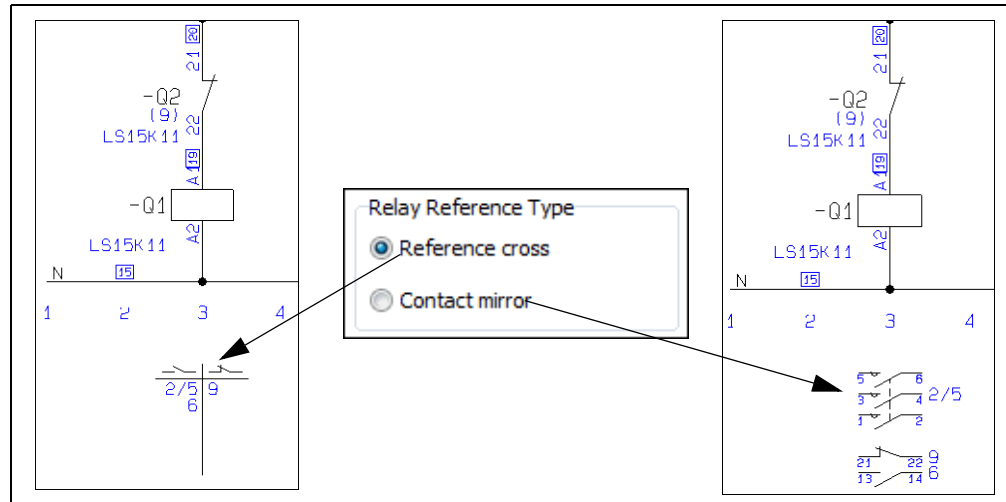
— See “Sub names on symbols” på side 160, especially “Functionally understood as different components” på side 160.



CHOOSING BETWEEN CONTACT MIRRORS AND REFERENCE CROSSES



To choose whether to use contact mirrors or reference crosses in **Automation**, you must click the **Project data** button, and click on either **Reference cross** or **Contact mirror** in the lower-left corner of the dialog box, and click **OK**.



You can, at any time, choose whether you wish to use contact mirrors or reference crosses.

If **Contact mirror** is activated without an attached database, the contact mirrors only contains the symbols, which are used in the project.



SWITCHING BETWEEN CONTACT MIRRORS AND REF. CROSSES

YOU CAN CHOOSE TO USE CONTACT MIRRORS WHILE WORKING WITH THE PROJECT, AND THEN SWITCH TO REFERENCE CROSSES WHEN DELIVERING THE PROJECT. — OR VICE VERSA.

HOWEVER, THE DATABASE MUST BE ATTACHED WHEN CHOOSING TO WORK WITH CONTACT MIRRORS, BECAUSE THE PROGRAM THEN RUNS THROUGH THE PROJECT, IN ORDER TO CREATE CONTACT MIRRORS. IF THE DATABASE IS NOT ATTACHED AT THIS TIME, AVAILABLE CONTACTS ARE NOT DISPLAYED.

CONTACT MIRRORS ARE CREATED AUTOMATICALLY

In **Automation** the program creates the contact mirrors for you automatically, so that they contains all the relevant symbols. When for instance you place a relay coil symbol from a component from the database, the program automatically creates a contact mirror containing the other symbols for the component under the relay coil symbol in the project.

If you later place other symbols with the same symbol name in the project, where these symbols does not have an article number — or has an unknown article number — the symbols are added to the bottom of the contact mirror. When these symbols are deleted in the project, they are removed from the contact mirror.

Creating contact mirrors without using the database

When you are working without an attached database, and place a symbol with the symbol type **Relay**, no contact mirror is created for the symbol.

The contact mirror is not created until you place another symbol with the same symbol name in the project. After this, symbols will be added to the contact mirror as you place more symbols with the same symbol name. The symbols will therefore appear in the order in which they are placed in the project.

When a symbol is placed using an attached database, the contact mirror for the symbol is memorized, also when the database is no longer attached. This applies as long as the article number for the symbol remains unchanged.

Contact mirrors for symbols without article numbers

What is mentioned above, only applies when you are working with an attached database, and place a symbol without an article number — or a symbol with an unknown article number.

When you assign an article number to a symbol, the contact mirror for the symbol is created/re-organized according to the order of the symbols for the component in the database (In the PCSTYPE field in the database — see “Database settings” on page 598).

CURRENT PATH NUMBERS / REFERENCE SYSTEM

The reference system / current path numbers is a coordinate system you can use on your diagrams. It will be used when you refer to the position of the individual components, for instance in a reference cross. The current path numbers / reference systems can be located along the horizontal or vertical axis, or along both axis simultaneously.

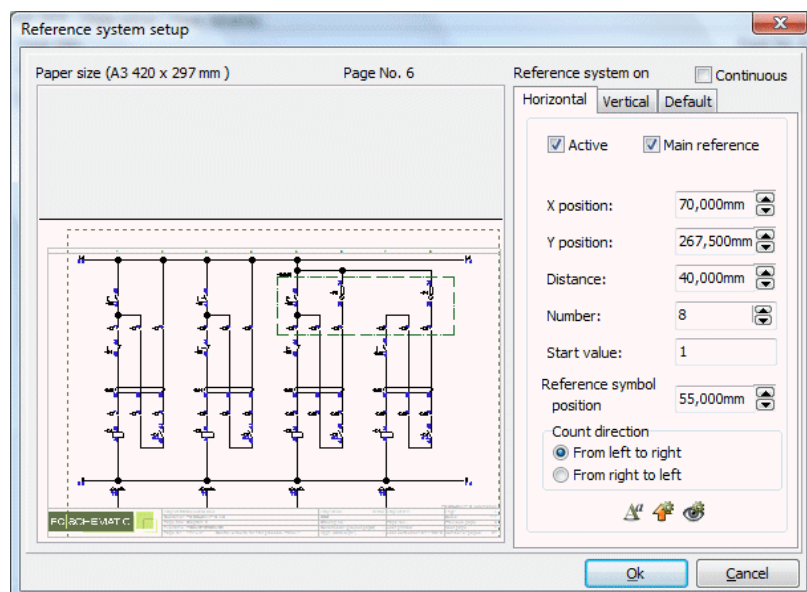
You can determine to let the number of the current path, and/or page a symbol is located on, be part of the name of the symbol. This is called *current path naming*.

THIS CHAPTER DESCRIBES	
— SETTING UP THE CURRENT PATHS.....	(P. 367)
— THE HORIZONTAL REFERENCE SYSTEM	(P. 368)
— THE VERTICAL REFERENCE SYSTEM	(P. 368)
— COPYING THE REFERENCE SETTINGS.....	(P. 369)
— CHANGING THE REFERENCE CROSSES.....	(P. 369)
— NAMING SYMBOLS ACCORDING TO CURRENT PATH NUMBERS	(P. 370)
— SYMBOL NAMES WHEN MOVING OR COPYING	(P. 372)
— CURRENT PATH NUMBERS AND AUTOMATIC PROJECT GENERATION.....	(P. 373)
— NAMING DUE TO CHANGES IN THE PAGE MENU	(P. 373)

SETTING UP THE CURRENT PATHS

To determine where and how to locate the current path numbers, you choose **Settings => Page data => References**.

You then enter the **Reference system setup** dialog box:



HORIZONTAL REFERENCE SYSTEM

In the field **Active** on the tab **Horizontal** you determine whether you want to use references along the x-axis on your diagram or not.

In the **Main reference** check box you determine whether these current path numbers are to be used when referring between symbols.

If **Main reference** is selected a reference cross will be placed on the diagram for each individual symbol having a relay status (See “Symbol types” on page 686). These will be filled out with references to the contacts of the same component. If **Main reference** is deselected, no reference crosses will be placed on the diagram.

In the fields **X position (mm)** and **Y position (mm)**, you determine where to place the first current path; in the **Distance (mm)** field you determine the distance between the current paths; and in **Number** you choose how many current path numbers you want on the chosen page.

In the **Start value** field, you determine the start value of the first current path on the diagram, and in **Counting direction** you choose whether to count from left to right, or from right to left.

If you want the current paths on the different pages in the project to be numbered in succession, select the **Continuous** check box. If the page you are working on has no preceding diagram, the field will be dimmed and can not be selected.

If you have selected **Continuous** on a page and then change the initial value, you automatically change the initial value on the *first* of the coherent pages.

In the **Reference symbol position** field you determine the position of the reference crosses.



By clicking on the **Text properties** button, you determine the size of the characters/figures in the numbering of the current paths.

Default settings



If you want to see the present default settings for the tab **Horizontal**, you click the **Show defaults** button. The default settings are displayed as long as the button is pressed down.



To load the defaults for the tab **Horizontal**, you click the **Load defaults** button.

VERTICAL REFERENCE SYSTEM

By clicking on the tab **Vertical** you will be able to determine corresponding settings for the current path numbers along the vertical axis. If you wish, you can use letters instead of numbers.

COPYING THE REFERENCE SETTINGS



If you want these settings for the current path numbers to be used on the diagrams you create from now on, you click on the tab **Default**, and then click on the **Save as default** button. Hereby you save *all* settings made on the tabs **Horizontal** and **Vertical**.

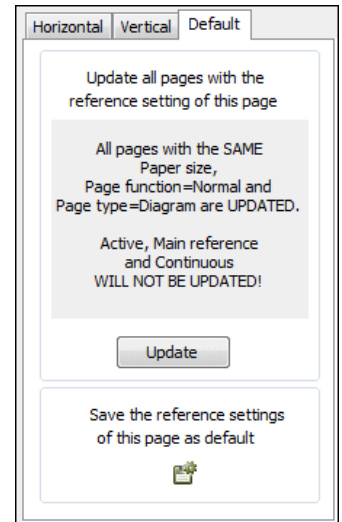
Please note, that clicking on the Save as standard button will not effect any pages in the project.

If you want to update the other pages in the project with the chosen settings, you click on **Update**.

Hereby you update all pages having the **Page function Normal**, the **Page type Diagram**, and also the same **Page size** as the page you are working on at present.

Check marks in the check boxes **Active**, **Main reference** and **Continuous** are not updated/changed.

— Read more about **Page type** and **Page function** in “Pages and Chapters” on page 485.



CHANGING THE REFERENCE CROSSES

The reference crosses are defined as symbols, so in order to change these, you have to change their symbol definitions. — See “Creating Symbols” on page 676.

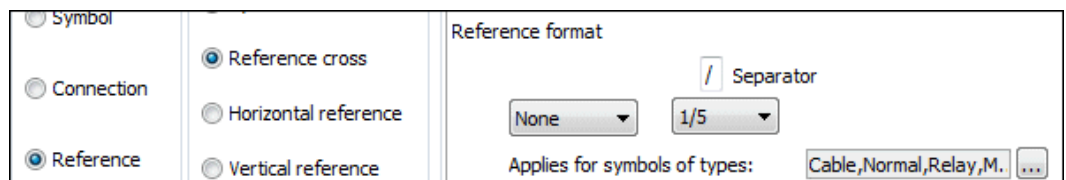
You choose which reference cross to use in the project by first choosing **Settings => Text/Symbol defaults**, then click on **Others**, on **Reference cross**, and on **Add**. You then enter the **Symbol menu**, where you select the symbol for the reference cross you wish to use.

The reference crosses are defined as symbols, so when you wish to change these, you must edit their symbol definitions. — See “Creating Symbols” on page 676.



If you enter **Settings => Text/Symbol defaults**, click on **Reference** and then on **Reference cross**, you can edit the appearance of the *text* in the reference crosses by clicking on the **Text properties** button. Please note, that the appearance will only be changed for the reference crosses that are placed in the project *after* you made the changes.

You can also choose which separator to use in the reference crosses:

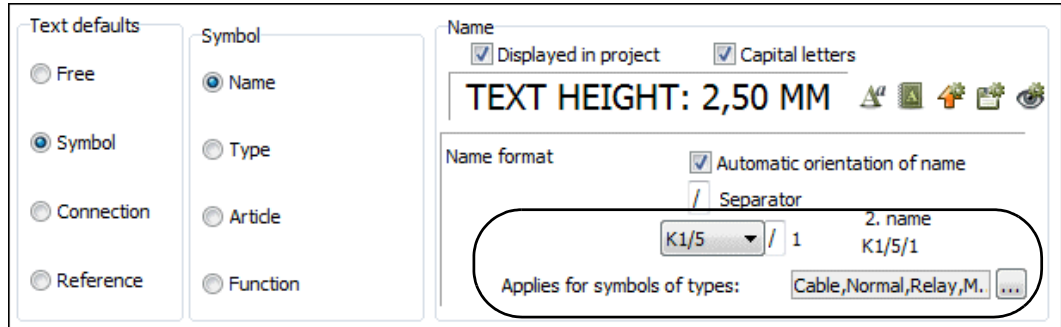


Then choose **Functions => Update references** and the entire project is updated with the new references.

NAMING SYMBOLS DUE TO PAGE AND CURRENT PATH NUMBER

If you want the page and the number of the current path, in which the symbols are located, to be part of the symbol names, you can let **Automation** handle this automatically.

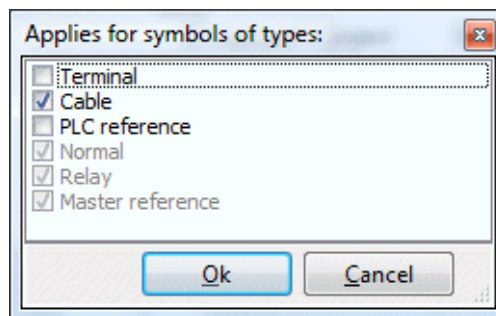
To arrange this, you choose **Settings => Text/Symbol standards**, and click on **Symbols**, and then on **Name**.



In the field **Name format** you can choose between different formats for the symbol names.



When you click the **Browse** button, you can see which symbol types the settings applies for:



Please note that the selected name formats always applies for symbols of the type **Normal**, **Relay** and **Main reference**.

Name formats

You can here choose between the following name formats:

THE NAME FORMAT FREE

If you choose **Free**, the symbol names become exactly as you type them.

THE NAME FORMAT K1/5

If you choose **K1/5**, the location of the symbol will automatically be included as the last part of the symbol name.

After the letter (**K**) follows the page number (**1**) a separator (**/**) and the number of the current path (**5**) in which the symbol is placed.

If you have chosen this type, you can specify which separator should be used between the page number and the current path number. You do this in the **Separator** field.

THE NAME FORMAT 1K5

If you choose **1K5**, the *page number* will be located before, and the *current path number* after the code letter.

THE NAME FORMAT K5

If you choose **K5**, only the current path number is included in the symbol name. This only makes sense if you are using **Continuous** current path numbers.

THE NAME FORMAT K15

If you choose **K15**, the last digit specifies the current path number. Other digits after the code letter specifies the page number. If for instance it says **-P146** on the diagram page, this refers to page 14, current path 6. This format can only be used if the current path numbers does not exceed one character — e.g. 0 to 9.

THE NAME FORMAT K105

If you choose **K105**, the last two digits specifies the current path number. Other characters after the code letter specifies the page number. If for instance it says **-P1406** this refers to page 14, current path 6. This format can only be used if the current path numbers does not exceed two characters — e.g. 0 to 99.



NAMING FORMATS WITH CURRENT PATH NUMBERS AND THE IEC/EN 61346 STANDARD

THE STANDARD IEC/EN 61346 DOES NOT MENTION CURRENT PATH NUMBERING.

IF YOU USE THE FORMAT **K1/5** OR **1K5**, THIS WILL BE IN CONTRADICTION TO THE NAMING RULES IN THE STANDARD.

BUT IF YOU CHOOSE THE FORMAT **K5**, **K15** OR **K105**, AND EXPLAINS THE SIGNIFICANCE OF THE APPLIED FORMAT IN THE DOCUMENTATION, YOU CAN FOLLOW THE STANDARDS IN THIS CONTEXT.

Changing symbol names to current path names

When you in this dialog box specifies that current path numbers shall be included in the symbol names, this only has an effect for the symbols you place *subsequently*.

If you want to include current path numbers in the *placed* symbols, choose **Functions => Special Functions => Create CurrentPath Numbers on all Symbols**.

You are then asked whether to rename all symbols. Click **OK**.

The option **Create CurrentPath Numbers on all Symbols** is only displayed when you have specified that current path numbers shall be included in the symbol names in the project.

Generation names

If you place more symbols of the same type in the same current path, the symbols will still be assigned unique names.

If for instance you place a lamp in current path sixteen on page three, it gets the name **H3.16**. If you then place more lamps in the same current path, they'll be assigned the names **H3.16.1** and **H3.16.2** — and so on. In this context the addi-

ons are called *generation names*, and are added to be able to distinguish between the symbols.

The generation names are assigned according to the order in which you place the symbols, and has nothing to do with their position in the current path.

In the **Second separator** field, you determine which sign will be used to separate the generation name from the current path number.

Save as default



If you want the chosen settings to be in effect for any new project you create, you click on the **Save as default** button.



If you want to load the present default settings for symbol names, you click the **Load defaults** button.

If you just want to see the defaults, you click the **Show defaults** button, which only displays the defaults as long as you keep the mouse button pressed down.

Symbol names in previously created projects will not be affected.

— Read more about default settings and about this dialog box in “Texts displayed in the project (general setting)” on page 123.

Changing between name format types

If you change from using the name format **Free** to a name format with included current path numbers, this will not have any effect on symbols already placed in your project. The change only concern the symbols you place after having made the changes.

If you want to change symbol names for symbols, that have been named using the name format **Free**, in order to make them follow page and current path numbers, you must do as described in “Make a symbol regain its adaptability” on page 373.

If you change from one current path name format to another, the changes will affect all symbol names, which contains the correct page and current path number for the symbol.

During moving or copying, symbols having these name formats, furthermore have the capability to have their names corrected, corresponding to page and current path numbers of their new location.

SYMBOL NAMES WHEN MOVING OR COPYING

If you *copy* a symbol with a current path name, the copy of the symbol automatically receives a name corresponding to which page/current path you place it on.

If you, on the other hand, *move* a symbol, you will be asked whether or not to rename the symbol.

If you answer **Yes**, the symbol is assigned a name corresponding to the new location of the symbol.

If your answer is **No**, the symbol keeps its old name, but *at the same time it loses the ability automatically to be assigned a current path name during moving/copying.*

Copying or moving more objects at the same time, follows the same directions as mentioned above.

Make a symbol regain its adaptability

If you have a symbol that has lost its adaptability, you can manually change its name to what it should be called, according to its present location.

This makes the symbol name regain its adaptability.

NAMING DUE TO CHANGES IN THE PAGE MENU

When you make changes on project pages in the **Page menu**, this may have consequences for pages containing symbols with current path names. (You can read more about this in "The Page menu and current path numbers" on page 489).

If this is the case, you will get a warning. If you answer **Yes** to rename the symbols, their names are changed according to the new page and current path numbers. If you copy/move the symbols later on, they will still be able to be assigned new current path names.

If your answer is **No**, the symbol names lose their adaptability. Thus, a symbol having the name **K3.5** will remain the name **K3.5** on its new location. But the program will no longer recognize this as a current path name. In this case the symbol will from then on be regarded as a symbol having the name format type **Free**.

— See how to let the symbol regain its adaptability in "Make a symbol regain its adaptability" above.

CURRENT PATH NUMBERS AND AUTOMATIC PROJECT GENERATION

When applying the automatic project generation facilities in **Automation**, all symbols in the generated project can automatically be assigned current path numbers.

— Read more about this in the **Project Generator** manual.



Wire numbers

This section describes functions for automatic and manual wire numbering:

- Automatic wire numbering..... (p. 377)
- Manual wire numbering(p. 386)
- Wire number files(p. 389)

AUTOMATIC WIRE NUMBERING

Automation can assign wire numbers to your drawings automatically. You can also assign wire numbers manually, and combine these two ways of working.

THIS CHAPTER DESCRIBES	
— STARTING THE AUTOMATIC WIRE NUMBERING	(P. 377)
— LOCKING THE AUTOMATIC WIRE NUMBERING	(P. 378)
— FUNCTIONS	(P. 379)
— POTENTIAL NUMBERING	(P. 379)
— WIRE NUMBERING.....	(P. 380)
— FORMAT	(P. 383)
— NUMBER SYMBOL	(P. 383)
— EXCLUDE SIGNALS	(P. 385)

— Please refer to “Manual placing of wire numbers” on page 386 to see how to place wire numbers manually.

STARTING THE AUTOMATIC WIRE NUMBERING

If you want to insert wire numbers automatically on your diagrams, you choose **Functions => Wire numbers**. This makes you enter the dialog box **Wire numbers**, which contains various tabs. Here you determine how the automatic wire numbering is to be carried out.

If you need to place wire numbers manually, this is described in “Manual placing of wire numbers” on page 386.



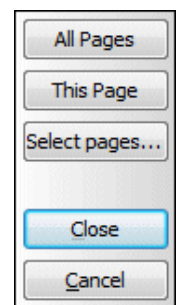
No UNDO FOR WIRE NUMBERING

BEFORE ASSIGNING WIRE NUMBERS AUTOMATICALLY, IT SHOULD STRESSED THAT *IT IS NOT POSSIBLE TO UNDO THIS FUNCTION*. THEREFORE, IT IS A GOOD IDEA TO SAVE THE PROJECT BEFORE ACTIVATING THE AUTOMATIC WIRE NUMBERING.

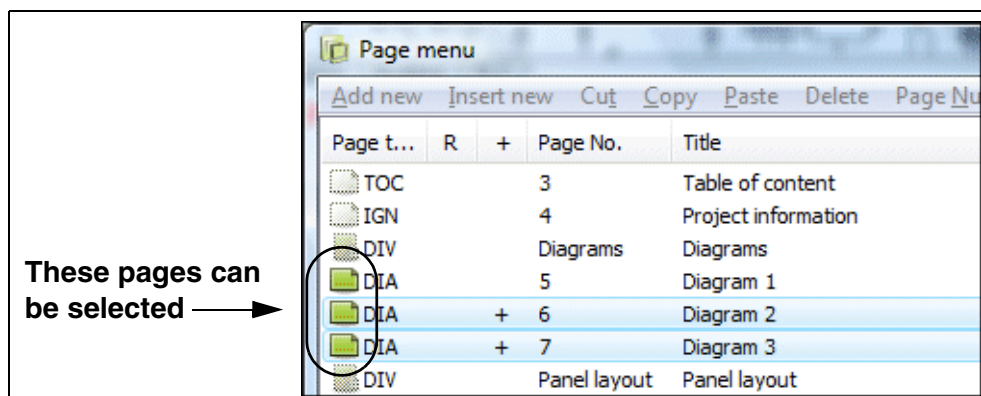
When the settings are appropriate, you click either **All Pages**, **This page** or **Select pages** to carry out the automatic wire numbering, following the directions you have set up.

When you click on **All Pages**, the wire numbering is made for all pages in the project.

When you click on **This Page**, the wire numbering is only made for the current page.



When you click on **Select pages**, you enter a special version of the **Page menu**:



The pages where it is possible to perform wire numbering, are indicated by a green page icon. Select the pages where you wish to assign wire numbers, and click **OK**.

If you wish to keep the settings in the dialog box **Wire Numbers**, but not carry out the wire numbering right now, click on **Close**. If you do not wish to keep the new settings, click **Cancel**.

Wire numbering settings in the project file

The directions for the wire numbering you set up here, are remembered in the project file. Thus you can perform automatic wire numbering following different guidelines in different projects, if you want.

When you start up a new project, the wire number settings used in the last active project will be suggested.

Wire numbers in the Object lister

To locate wire numbers in the project, you can use the Object lister, as described in "The Object Lister" on page 296.

When you use the **Show net** function, you get access to the same function, as described in "Show net" on page 195.

LOCKING THE AUTOMATIC WIRE NUMBERING

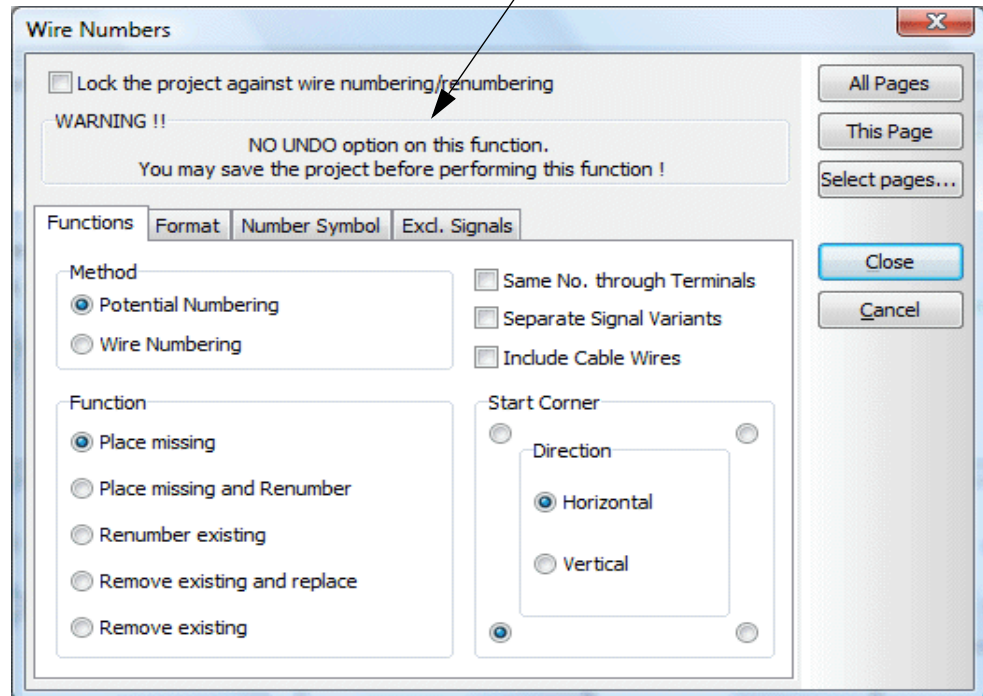
At the top of the dialog box **Wire numbers** you can lock the project against wire numbering/renumbering by selecting **Lock the project against wire numbering/renumbering**.

This makes it possible to avoid renumbering a project by mistake.

FUNCTIONS

On the wire number tab **Functions**, you determine the principles for the automatic wire numbering.

Please observe the warning



Basically, there are two different principles for performing the wire numbering, namely potential numbering and wire numbering.

Potential numbering

To select potential numbering, you click on **Potential numbering** in the upper-left corner of the dialog box.

Hereby each electrical potential is assigned its own wire number. If the same electrical potential is separated by signal symbols (see “Signal symbols” on page 196) — for instance because it is used on more pages — the same wire number will be applied each place this potential exists.

The wire number is placed in the middle of the longest line-segment in the potential, but may be moved succeedingly if you wish.

If you want the same wire number on each side of a terminal, you click on **Same number through terminals**.

Due to the fact that cables are components of their own right, cable wires will not be numbered unless you check off **Include cable wires**.

If you wish to interpret signal variants as unique potentials, select **Separate Signal Variants**.

Wire numbering

If you want to assign a unique number to each wire in the entire project, you check off **Wire Numbering** in the upper-left corner of the dialog box.

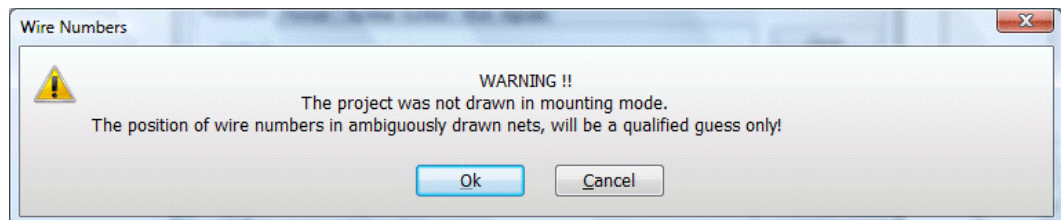


THE WIRE NUMBERING IS A QUALIFIED GUESS

IF YOU CANNOT TELL FROM THE DRAWING HOW THE WIRE NUMBERS SHOULD BE PLACED, YOU CANNOT ASSUME THAT THE PROGRAM CAN GUESS WHERE THE WIRE NUMBERS SHOULD BE LOCATED!

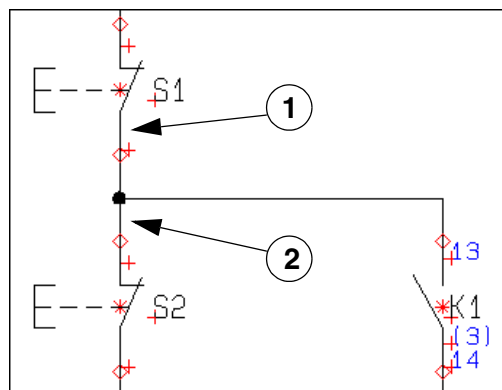
THE PROGRAM WILL PLACE THE CORRECT AMOUNT OF WIRE NUMBERS WITHIN THE INDIVIDUAL POTENTIALS, AND YOU WILL THEREFORE RECEIVE THE CORRECT AMOUNT OF WIRE NUMBERS IN YOUR WIRE NUMBERS FILE. — BUT THERE IS NO GUARANTEE THAT THE NUMBERS WILL BE LOCATED ON THE CORRECT WIRES, IF YOU DO NOT DRAW IN MOUNTING MODE.

Therefore, if you want to perform automatic wire numbering, you receive this warning:

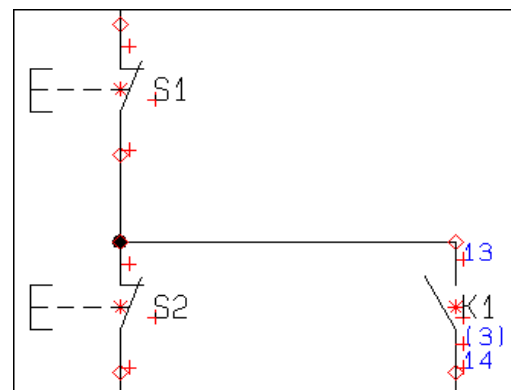


Unambiguous drawing

To make it perfectly clear how the wires should be installed, you can make the drawing unambiguous:



Ambiguous



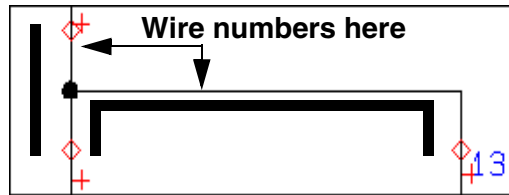
Unambiguous

On the ambiguously drawn figure above, it is not clear whether the horizontal wire connects **K1** to **S1**, or from **K1** to **S2**.

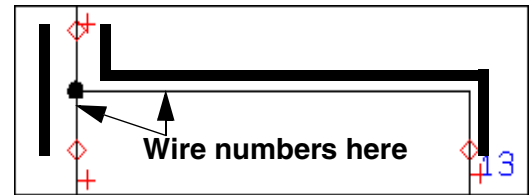
On the other hand, the unambiguously drawn figure clearly states that the wire connects **K1** to **S2**.

It only makes sense to place a wire number on a drawn line, which only represents one wire. However, on the ambiguous drawing above, it is not clear whether the

line from **S1** to the connection dot (**1**), or the line from the connection dot to **S2** represents both wires. On the ambiguous drawing, the symbols could be connected in two different ways:



Option 1



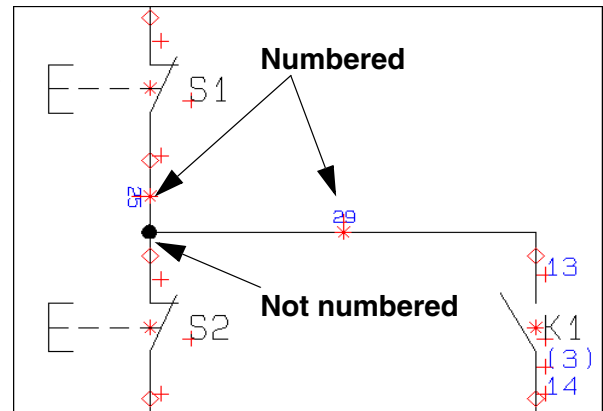
Option 2

In this case, the program must make a qualified guess.

The program guess

Automation will make a guess which polyline represents more than one wire. These are not assigned a wire number.

— In this context a polyline is the line drawn *from* the connection point of a symbol *to* the connection dot in which the line meets another line. On the figure, one of the poly-line is the line starting in the connection point **13**, and ending in the connection dot.

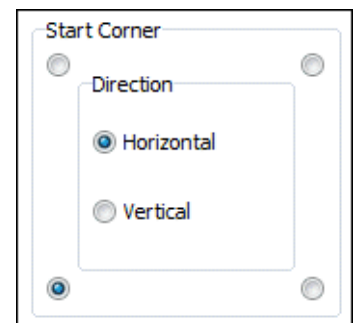


In an electrical potential with for instance three connected symbols, two lines will be included. The program will then make a guess, that the shortest polyline represents both wires. Please bear this in mind when you draw diagrams.

Numbering order

To specify which corner of the page to start the wire numbering from, you select one of the four corners.

Also you specify whether the counting direction shall be **Horizontal** or **Vertical**.



Function

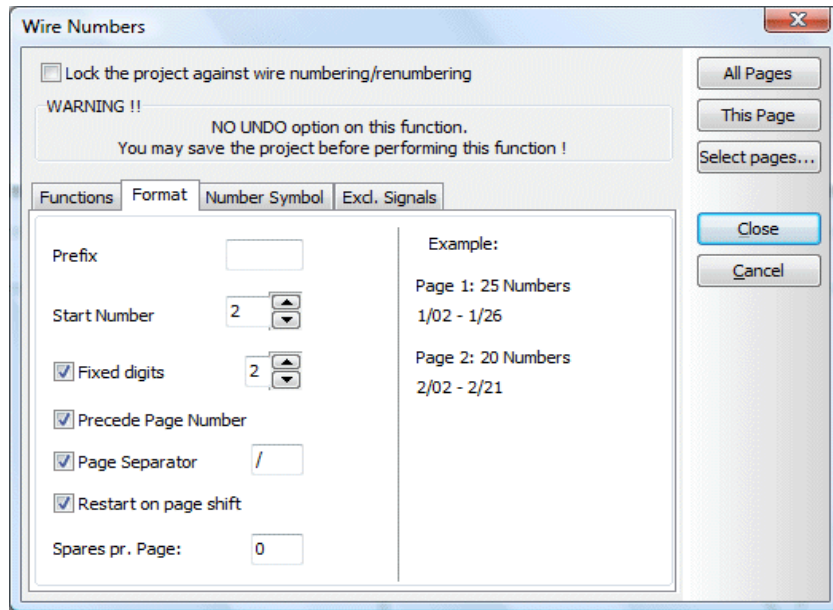
In the lower-left corner of the tab, you determine how to assign wire numbers, and where to place the wire numbers:

FUNCTION	ACTION
PLACE MISSING	DOES <i>NOT</i> ALTER WIRE NUMBERS THAT HAVE ALREADY BEEN PLACED, BUT ASSIGNS WIRE NUMBERS TO WIRES WITHOUT NUMBERS.
PLACE MISSING AND RENUMBER	PLACES NEW WIRE NUMBERS ON WIRES WITHOUT WIRE NUMBERS, AND ALLOWS ALREADY PLACED WIRE NUMBERS TO KEEP THEIR <i>POSITION</i> ON THE DIAGRAMS, BUT ALL WIRE NUMBERS ARE RENUMBERED.

FUNCTION	ACTION
<i>RENUMBER EXISTING</i>	THIS FUNCTION IS USED WHEN YOU HAVE PLACED ALL THE WIRE NUMBERS, AND DO NOT WISH TO PLACE MORE NUMBERS. THEREFORE IT ONLY RENUMBERS ALREADY PLACED WIRE NUMBERS - WITHOUT EXAMINING WHETHER OTHER WIRES LACKS WIRE NUMBERS.
<i>REMOVE EXISTING AND REPLACE</i>	ALL EXISTING WIRE NUMBERS ARE REMOVED COMPLETELY, AND NEW WIRE NUMBERS ARE PLACED. THE EARLIER POSITION OF THE WIRE NUMBERS ARE NOT REMEMBERED.
<i>REMOVE EXISTING</i>	REMOVES ALL WIRE NUMBERS FROM THE PROJECT.

Please notice that the further down you go on the list, the larger is the extent of the actions. — Please remember that **Undo** is no option here!

FORMAT FOR AUTOMATIC WIRE NUMBERS



Here you determine the numbering principles for the wire numbers.

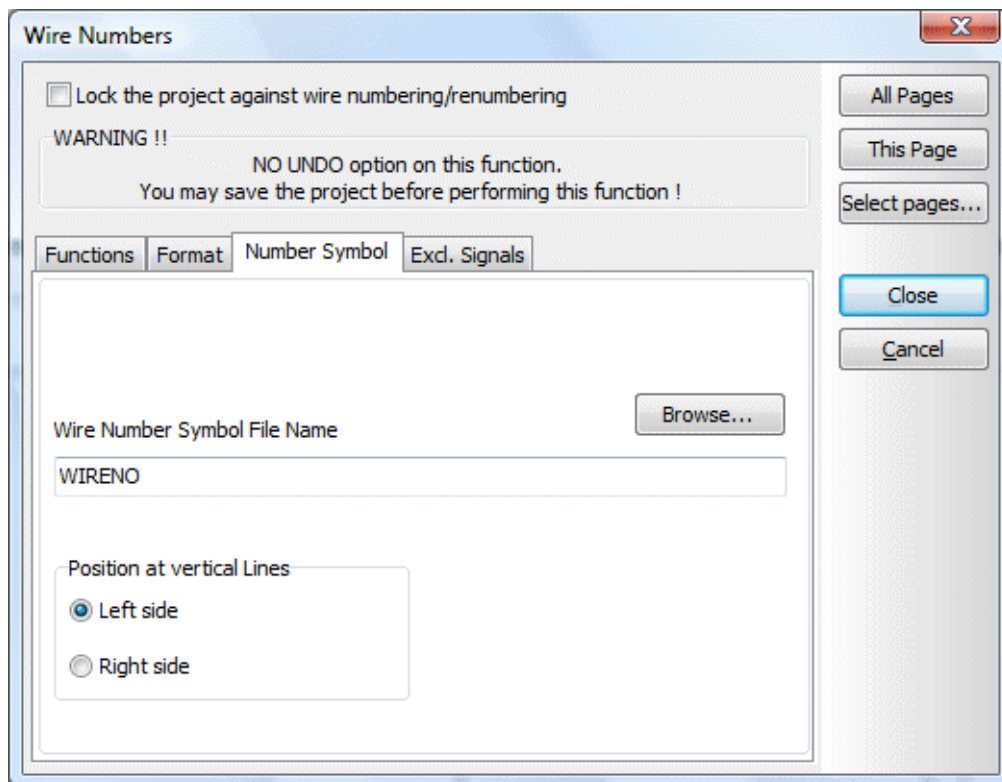
FORMAT	ACTION
PREFIX	HERE YOU CAN PLACE A PREFIX , WHICH IS PLACED IN THE BEGINNING OF THE WIRE NUMBERS.
START NUMBER	HERE YOU CAN SPECIFY A START NUMBER FOR THE WIRE NUMBERS.
FIXED DIGITS	HERE YOU STATE WHETHER THE WIRE NUMBERS SHALL CONTAIN A FIXED NUMBER OF DIGITS — FROM TWO TO NINE.
PRECEDE PAGE NUMBER	PLACES THE PAGE NUMBER IN THE FIRST PART OF THE WIRE NUMBER. — CAN ONLY BE CHOSEN IF FIXED DIGITS HAVE BEEN CHECKED OFF. INSERTION OF A NEW PAGE BETWEEN PAGES, WHICH USES THIS TYPE OF WIRE NUMBERS, WILL NOT CHANGE WIRE NUMBERS AUTOMATICALLY.
PAGE SEPARATOR	INSERTS A USER DEFINED SEPARATOR BETWEEN THE PAGE NUMBER AND THE WIRE NUMBER.
RESTART ON PAGE SHIFT	MAKES THE WIRE NUMBERS START ALL OVER ON EACH PAGE. — CAN ONLY BE SELECTED IF PRECEDE PAGE NUMBER IS SELECTED.
SPARES PR. PAGE	RESERVES AN EXTRA POOL OF AVAILABLE WIRE NUMBERS FOR EACH DIAGRAM PAGE. THUS YOU MAKE ROOM FOR WIRE NUMBERS THAT ARE APPLIED LATER ON, WITHOUT THIS CAUSING ANY CHANGES TO THE OTHER WIRE NUMBERS IN THE PROJECT. — DOES NOT MAKE MUCH SENSE IF YOU HAVE CHECKED OFF RESTART ON PAGE SHIFT .

Please note, that in the right-hand side of the tab, you can see an example on how the wire numbers would appear in a (fictional) project.

NUMBER SYMBOL FOR AUTOMATIC WIRE NUMBERING

When you assign a wire number to a wire, this is done by placing a wire number symbol, having the wire number as its symbol name.

When you click on the tab **Number symbol**, you state which symbol will be used as the wire number symbol.



If you want to change which symbol is being used, you click on **Browse**, whereby you enter the **Symbol menu**. Here you click on the desired wire number symbol, and click **OK**.

You can only apply symbols of the type **Wire number**.

Position of wire number symbol

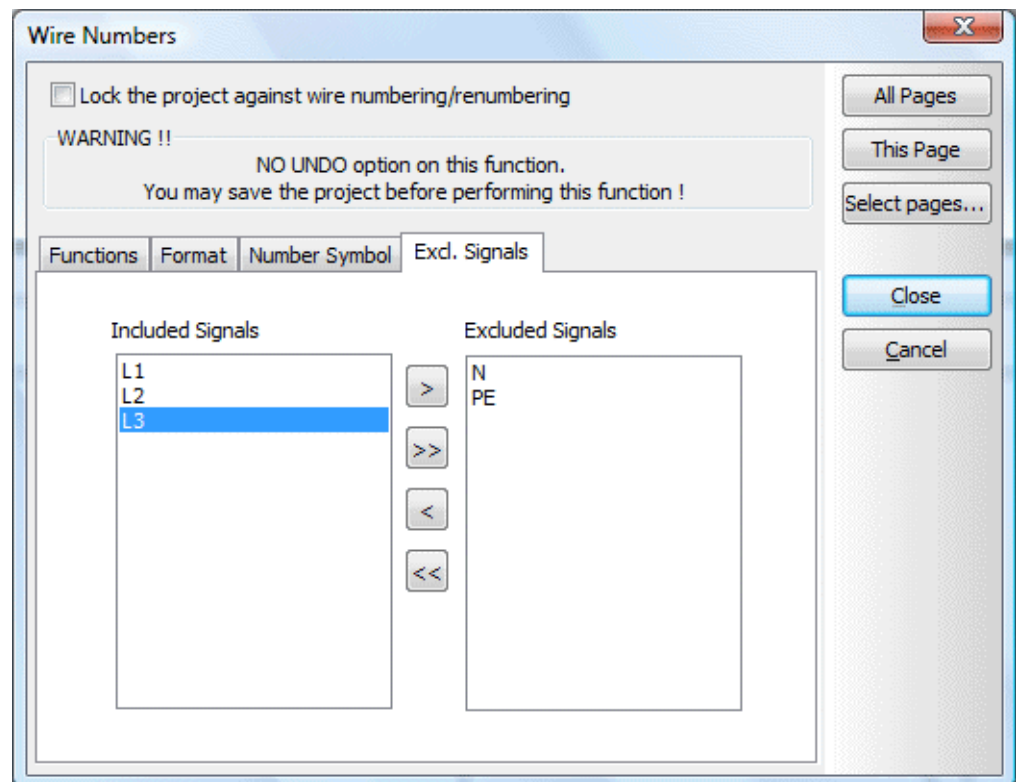
In the lower-left corner, you specify whether the wire number symbols are placed on the **Left side** or on the **Right side** of vertical lines.

Design your own wire number symbol

If you want to design your own wire number symbols, you can see how to design symbols in “Creating Symbols” on page 676.

EXCLUDE SIGNALS FROM AUTOMATIC WIRE NUMBERING

Because some users do not wish to number certain signals, it is possible to determine which signals should be excluded from the automatic wire numbering.



Select the signals you want to transfer and click on the **Transfer** button, or click on the **Transfer all** button to exclude all signals from the automatic wire numbering.



Correspondingly you can select excluded signals in the right hand side of the tab, and revert them by clicking on the **Revert** button, or click on the **Revert all** button to get all excluded signals back on the included list of signals to be numbered.

MANUAL PLACING OF WIRE NUMBERS

If you want to make sure that certain wires in the project are assigned specific wire numbers, you can apply the wire numbers manually.

THIS CHAPTER DESCRIBES	
— FETCHING A WIRE NUMBER SYMBOL.....	(P. 386)
— PLACING A WIRE NUMBER SYMBOL	(P. 387)
— AUTOMATICALLY INCREMENTING OF WIRE NUMBER NAMES	(P. 388)

It is a good strategy to place the wire numbers *you* want to place first, and then apply the automatic wire numbering function, as described in “Automatic wire numbering” on page 377.

FETCHING A WIRE NUMBER SYMBOL

The wire numbers are placed by a wire number symbol, which you place on the line that shall have the wire number.



To fetch a wire number symbol you click on the **Symbols** and the **Symbol menu** button, to enter the **Symbol menu**. Or press [F8] to enter the menu directly.

Here you choose a wire number symbol — for instance the symbol **WIRENO** in the **MISC** folder — and click **OK**.

Please note, that this wire number must be of the symbol type **Wire number**.

Place the wire number symbol in the pickmenu

Also, you can place the wire number symbol in the pickmenu, as described in “Placing symbols in the pickmenu” on page 173.

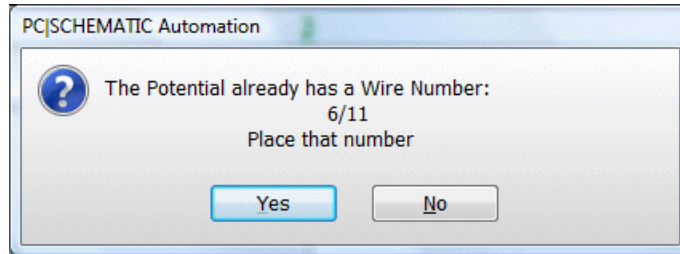
This speeds up the manual placing of the symbol.

Please refer to “Fetching symbols” on page 135 for further details about fetching symbols.

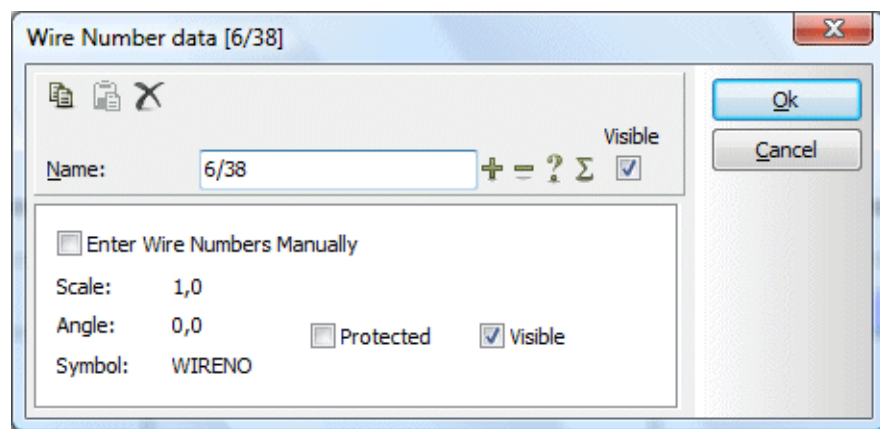
PLACING A WIRE NUMBER SYMBOL

You now have the wire number symbol in the cross hairs, and click on the wire you want to place the symbol on.

If the wire already has a wire number, you will get this message:



If you want to place another wire number, click **No**, and you enter the dialog box **Wire number data**.



If **Enter Wire Numbers Manually** is *not* selected, the program automatically suggest the next available wire number — according to the wire numbering principle specified in **Functions => Wire Numbers**.

If **Enter Wire Numbers Manually** is selected, you can specify the wire number names independent of these settings. This way the program supports, that you can apply a specific numbering principle for the naming of your manually placed wire numbers.

If you select **Protected**, the wire number will not be renamed or moved automatically when you change the wire numbers automatically in **Functions => Wire numbers**.

AUTOMATIC INCREMENTING OF MANUALLY PLACED WIRE NUMBERS



Having selected **Enter Wire Numbers Manually**, you can for instance type in *L1/1*, press down <Ctrl>, and click on **Plus**. Hereby you activate the automatic incrementing of the manually placed wire number names. Click **OK**.

When you place the wire number in a potential already having a wire number, you can select **Change all in Potential to L1/1**. This way you rename all wire numbers in the potential. This function is particularly useful when you have copied diagrams from other projects.

Now click on another wire, and notice that the name of the new wire number now have been incremented to *L1/2*. Click **OK**.

OTHER FUNCTIONS IN THE DIALOG BOX

By selecting **Visible** in the dialog box, you specify that the wire number should be visible on the diagrams.

The buttons **+**, **-**, **?** and Σ function as described in “Placing and Naming symbols” on page 151.

WIRE NUMBER FILES

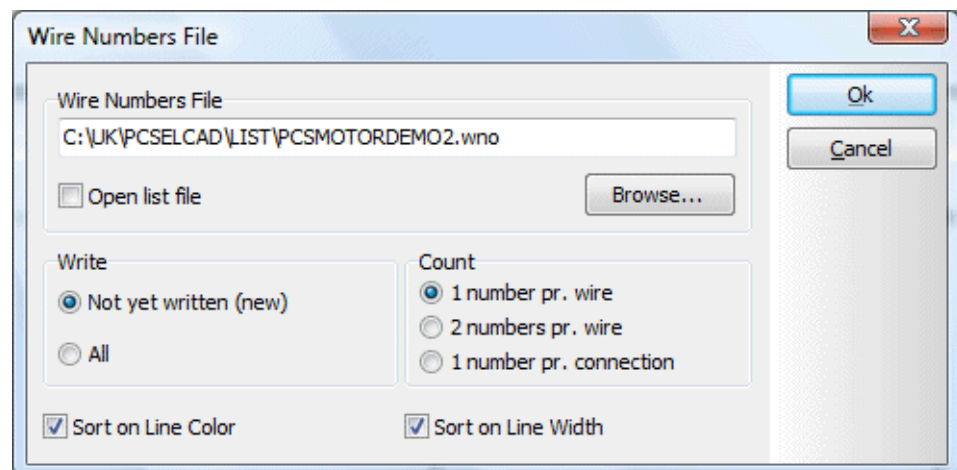
In **Automation** you can create wire number files for a wire numbering tool. This chapter describes how to create wire number files.

THIS CHAPTER DESCRIBES	
— CREATING A WIRE NUMBER FILE.....	(P. 389)
— CREATING A CONNECTIONS LIST FILE FOR WIRE NUMBERING.....	(P. 391)

CREATING A WIRE NUMBER FILE

When you want have your wire numbers saved in a file, you choose **Lists => Wire number file**.

You then enter this dialog box:



Here you have the following options:

New or All wire numbers

The program keeps track of which wire numbers have been saved in a file. Therefore clicking on **Not yet written (new)** only gives you a file with wire numbers, which have not been written to a wire number file earlier.

If you want all wire numbers in the project included in the file, you click on **All**.

How many numbers pr. wire

If you only want a single wire number pr. wire, click on **1 number pr. wire**, if you want two wire numbers for each wire, you click on **2 numbers pr. wire**.

WIRE NUMBERS PR. CONNECTION

If you choose **1 number pr. connection**, one wire number pr. connection is created. This function only has relevance for potential numbering, where two wires under the same terminal has the same number, and therefore can use the same wire number label.

Wire numbers to file according to Line color and Line width

It is furthermore possible to create wire number files, which is sorted according to line color and/or line width for the line on which the wire number is placed.

This makes it possible succeedingly to handle the wire numbers differently in the wire numbering tool, in which the file is loaded.

To sort the wire numbers, select **Sort on Line Color** and/or **Sort on Line Width**.

Specify name and file format for the wire number file

In the field **Wire Numbers File** you type in the name of the wire number file, as well as the path that leads to the file. The extension of the file specifies the file format — see the options in the figure below.

By clicking on **Browse**, you enter the dialog box **Save as**. Here you can find the folder and/or the file name you want to use for the file, and specify the file format for the wirenumber file.

OPENING THE WIRE NUMBER FILE

If you select **Open list file** in the dialog box **Wire Numbers File**, the wire number file is subsequently opened in a program according to the file format. If for instance you selected **Microsoft Excel (*.xls)**, the file is opened in Excel, if you have this program installed on your PC.

Creating the wire number file

When you click **OK**, a wire number file is created, which can be loaded in a wire numbering tool, from which you can print the wire numbers to place on the actual wires.

CONNECTIONS LIST FOR WIRE NUMBERING

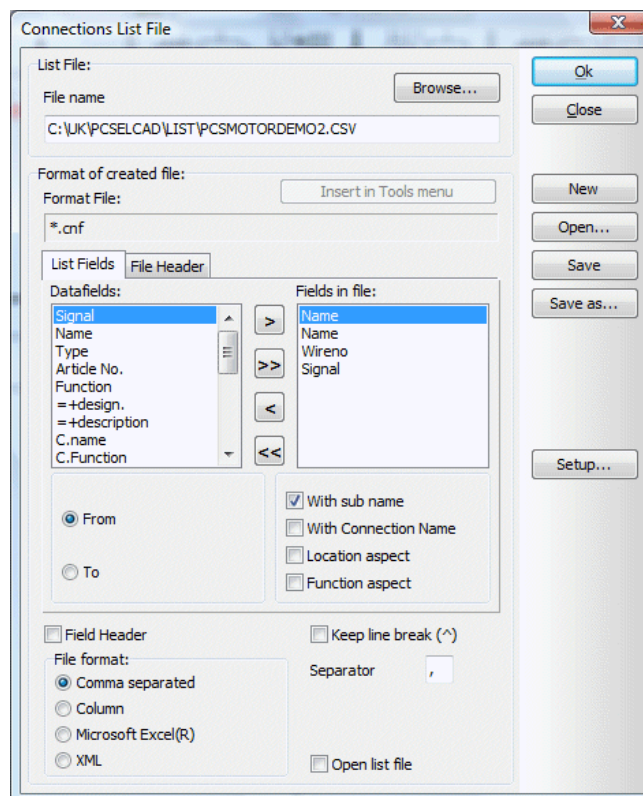
In **Automation** you can create a connections list file, which contains component names with connections for all electrically connected components. Hereby a complete connections list is displayed, where name and connection is displayed for each individual connection.

This list can successfully be sent to a wire numbering tool, from which you can have all the necessary wire numbers printed.

It is also possible to create connections lists with wire numbers, so that the list can be used for mounting.

Creating a connections list for wire numbering

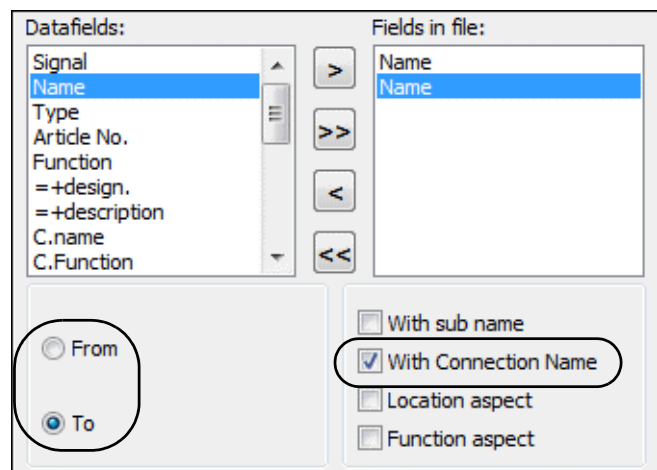
Choose **Lists => Connectionslist for file**, and you enter this dialog box:



If you create a connections list file, which only contains component names with connections for all of the electrically connected components, just choose the datafield **Name** twice.

When you then select **With Connection Name** for both of these datafields, and set the one to **From** and the other to **To**.

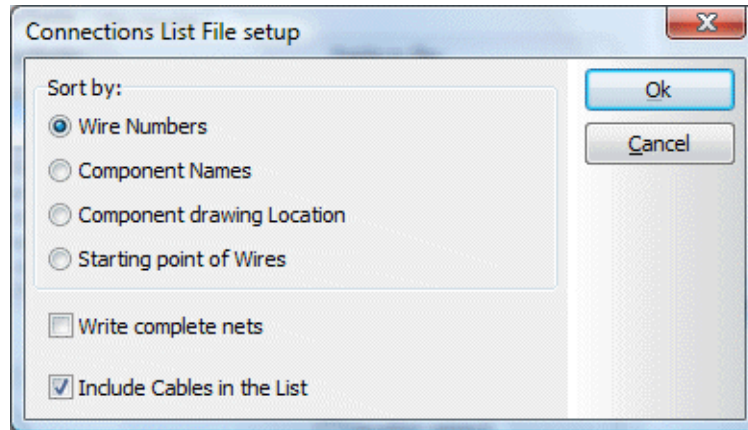
— Read more about the options in this dialog box in



“Specifying the contents of the file” on page 634, and see “The buttons in the right-hand side of the dialog box” on page 633 also.

Settings for Connections lists for file

If you click on **Setup**, you enter the following dialog box:



Here you have the following options for specifying how to sort the list:

WIRE NUMBERS

The file is sorted by wire numbers, so that the lowest wire numbers are placed first.

COMPONENT NAMES

The file is sorted alphabetically by component names. Thus for instance all wires attached to the component **-K1** comes first, then wires attached to **-K2** etc.

COMPONENT DRAWING LOCATION

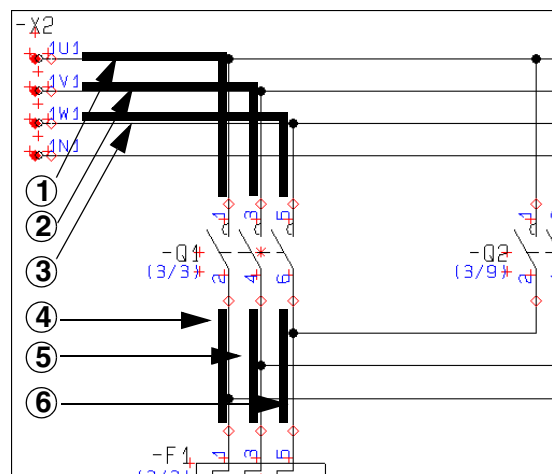
The file is sorted according to the pages in the project. First comes everything from page **1**, then everything from page **2** etc.

STARTING POINT OF WIRES

The file is sorted by the starting points, as displayed in the figure to the right.



IF YOU NEED TO CREATE CONNECTIONS LIST FILES ACCORDING TO OTHER DIRECTIONS, PLEASE CONTACT PC|SCHEMATIC A/S.



WRITE COMPLETE NETS

If you select **Write complete nets**, the connections in the same nets are listed in succession.

INCLUDE CABLES IN LISTS

Select **Include cables in lists**, in order to include cables in the list.

CLICK OK

Select the desired setting, and click **OK**.

— Read more about lists to files in “Exporting List Files” on page 629.



Single line diagrams

This section describes functions for single line diagrams:

- Single line diagrams.....(p. 397)
- Generating single line diagrams automatically(p. 399)



SINGLE LINE DIAGRAMS

Automation supports single line diagrams — except for PLCs.

This means that you can draw lines that represents several connections. Correspondingly, you can place symbols, representing several symbols of the same kind.

THIS CHAPTER DESCRIBES	
— SIGNAL NAMES IN SINGLE LINE DIAGRAMS	(P. 397)
— CONNECTION NAMES IN SINGLE LINE DIAGRAMS	(P. 398)
— SYMBOL NAMES IN SINGLE LINE DIAGRAMS.....	(P. 398)
— SYMBOLS FOR SINGLE LINE DIAGRAMS IN PICKMENU	(P. 398)
— SYMBOLS WITH STATES FOR SINGLE LINE DIAGRAMS	(P. 398)

There are three places where you can specify that a line or a symbol represents more lines or components. This is in the *signal names*, the *connection names* and the *symbol names*.

Single line diagrams and lists

These lines and symbols are automatically unpacked in the lists of the project. When it comes to lists, it therefore makes no difference whether you have created single line diagrams or ordinary diagrams.

SIGNAL NAMES IN SINGLE LINE DIAGRAMS

As an example, you can draw the signals **L1**, **L2** and **L3** with just single line.

The only thing you need to do, is to assign the signal name *L1,L2,L3* to the signal, as displayed on figure 2.

On the symbol **K1**, the connection name of the contact to the left is set to *1*, which specifies it is connected to **L1**.

The connection name of the contact in the middle gets the name *,3*. This specifies that it is connected to **L2**, because **L2** is specified as the second signal in the signal name.

Correspondingly, the name *,,5* tells you that the contact to the right is connected to the signal **L3**.

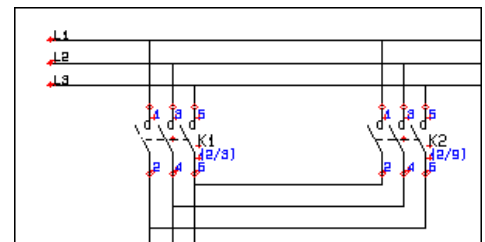


Figure 1

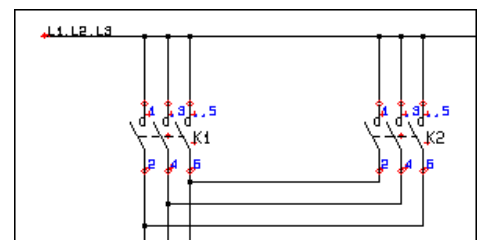


Figure 2

CONNECTION NAMES IN SINGLE LINE DIAGRAM

On figure 3, the symbol for **K1** has been replaced by a symbol, which only displays one contact.

However, the three names in the connection points of **K1**, specify that the symbol represents a component having three contacts.

Therefore the connection names still specify the same connections as on the other figures.

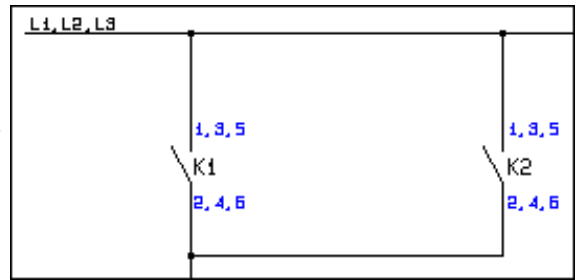
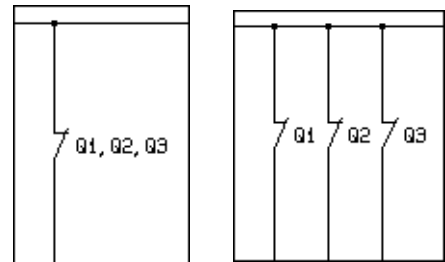


Figure 3

SYMBOL NAMES IN SINGLE LINE DIAGRAMS

By assigning the names **Q1, Q2, Q3** to a symbol, you can in a similar way let a symbol represent *three* — or more — symbols of the same type.

These two figures represent two different ways of drawing the same connection.



SYMBOLS FOR SINGLE LINE DIAGRAMS IN PICKMENU

See how to place symbols for single line diagrams in the pickmenu for a component from the database in “Specifying alternative electrical symbols via the database” on page 584.

SYMBOLS WITH STATES FOR SINGLE LINE DIAGRAMS

Symbols can be created with different *states*, so that the symbols changes appearance when their state number is changed.

Hereby the same symbol can be applied for single line diagrams, where e.g. lines across the symbol indicates the number of phases used.

- See how to apply symbols with states in “Symbols with states” on page 162.
- See how to create symbols with states in “Creating symbols with states” on page 692.

AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS

Automation can automatically generate single line diagrams based on multi-line diagrams and panel drawings.

THIS CHAPTER DESCRIBES	
— INSTALLING THE MODULE SINGLELINE DIAGRAM.....	(P. 399)
— PREPARING FOR AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS	(P. 400)
— STARTING THE MODULE SINGLELINE DIAGRAM	(P. 400)
— SETTINGS FOR THE MODULE SINGLELINE DIAGRAM	(P. 402)

— Please also see “Single line diagrams” on page 397.

INSTALLING THE MODULE SINGLELINE DIAGRAMS

In order to generate single line diagrams automatically in **Automation**, you must first install the module **Singleline Diagram**:

- 1) Choose **File => Modules**.
- 2) Click on **Singleline Diagram** and click **Install**.
- 3) In the dialog box **Modules** it is now displayed that the module is installed:
Click **Close**.

The module is activated by choosing **Tools => Singleline Diagram**.

PREPARATIONS FOR AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS

In order to be able to generate single line diagrams automatically, you must first do this:

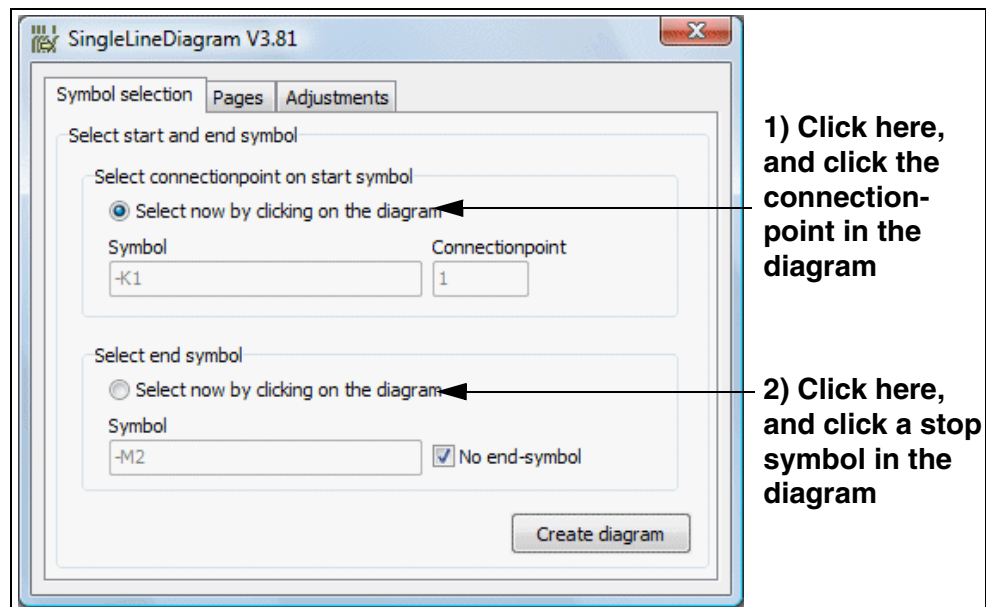
- 1) Create single line symbols for the applied components in the diagram.
- 2) In the applied database: Add a field to the database, which makes it possible to specify which single line symbol to use for each individual component — for instance the field *SINGLELINESYMBOL*.
- 3) In the applied database: Specify which single line symbol, that belongs to which of the applied components.
- 4) Adjust the settings in **Automation**, so that the program knows from which database fields to fetch the single line symbols: Choose **Settings => Database => Database settings**, click on the tab **Component data**, and specify in the *Singleline symbol* field which datafield in the database that **Automation** fetches single line symbols from.

— See an example on how to do it in “Single line diagram example” on page 403.

GENERATING A SINGLE LINE DIAGRAM

To generate a single line diagram using the **Singleline Diagram** module, do this:

- 1) Choose **Tools => Singleline Diagram**:



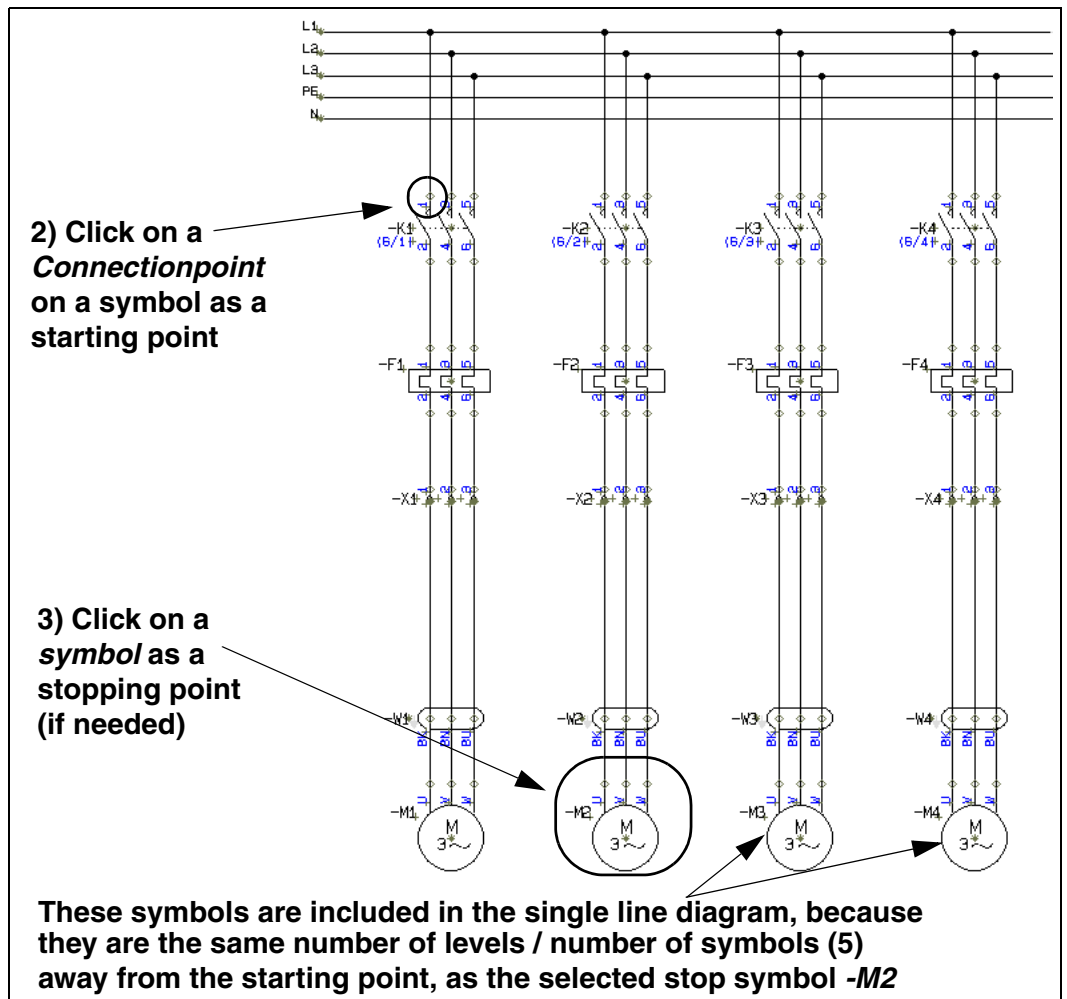
- 2) Click on the symbol *connectionpoint* in the diagram, from which you want the single line diagram to start, as displayed in the figure "Point out start and end point for the single line diagram" below.
 - If you are creating the diagram based on a file from the **Panel Builder**, the start symbol is found automatically.
- 3) Select a stop symbol, if necessary. — If the selected stop symbol is for instance 5 symbols away from the startingpoint, the generated single line

diagram will contain all symbols connected with the connection point, which are up to 5 symbols/levels away from this. See the figure "Point out start and end point for the single line diagram" below. However, it is seldom necessary to specify a stop symbol.

4) Click on **Generate diagram**.

The single line diagram is now generated based on the settings specified on the tabs **Pages** and **Adjustments**.

Point out start and end point for the single line diagram

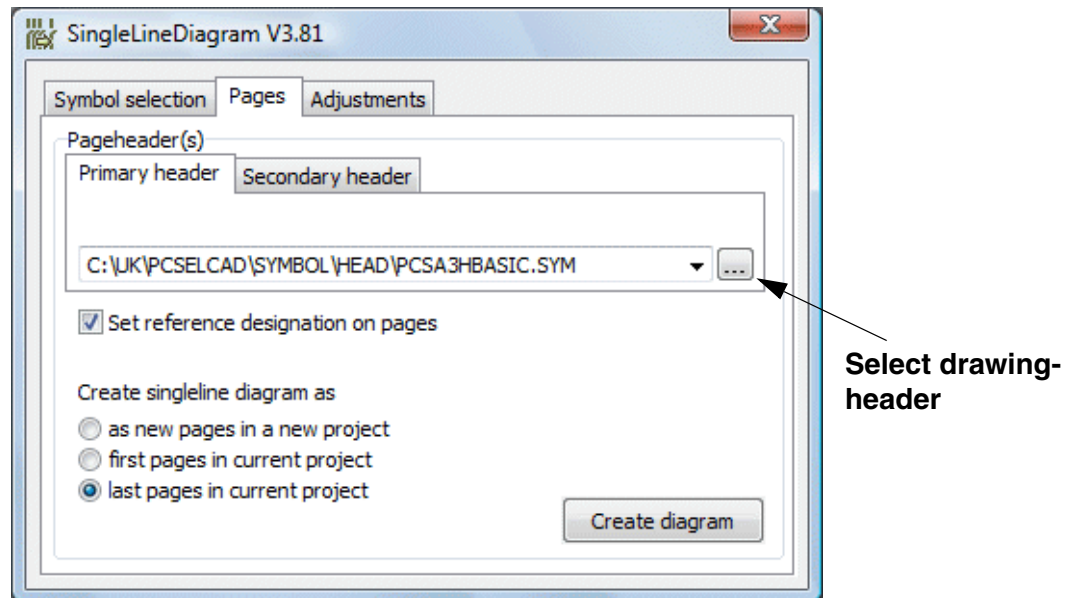


SETTINGS FOR THE MODULE SINGLELINE DIAGRAM

In the dialog box **Singleline Diagram** there are different settings for how to generate single line diagrams:

The Pages tab

On the **Pages** tab, you specify how to create the generated pages.

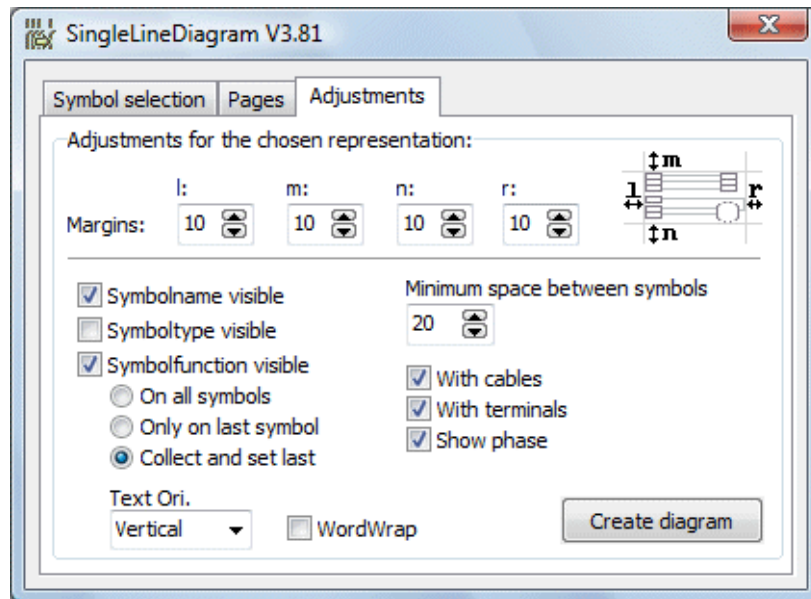


You can choose:

- Choose which drawing header(s) to use on the pages.
- Whether to insert reference designations on the pages.
- Whether to place the single line diagrams at the beginning or at the end of the project, or whether to create the single line diagrams as a new project. If you choose to let the single line diagrams be a part of the original project, the single line diagram drawings will be created as a new chapter along with a chapter tab page.

The Adjustments tab

On the **Adjustments** tab you specify what to include in the single line diagrams, and how to place the objects on the page.



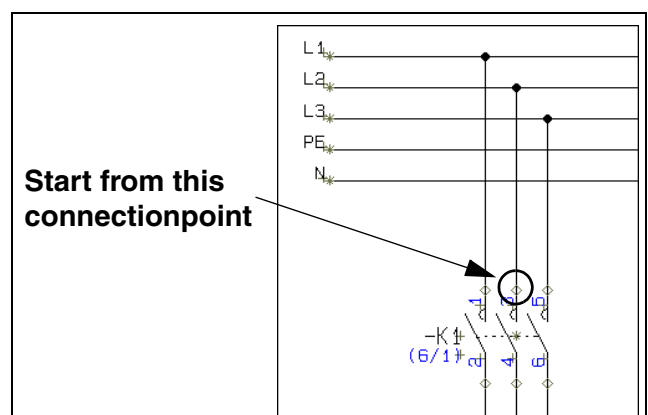
Here you specify:

- The margins
- The minimum space between the placed symbols on the pages
- Which symbol texts to display
- The orientation of function texts for symbols
- Whether to include cables and/or terminals in the diagrams
- Whether to display phases in the diagram

SINGLE LINE DIAGRAM EXAMPLE

If you wish to try the **Single-line Diagram** module, you can open the project **PCSMOTORDEMO1**, start the **Singleline Diagram** module, and generate the single line diagram, as described in "Generating a single line diagram" on page 400.

Select the connectionpoint displayed in the figure to the right.



Please note that you must use the included **Pcsdemo** database, in order to be able to create the single line diagram in this example.

Singleline Diagram and the Panel Builder

If you wish to try the **Singleline Diagram** module along with the **Panel Builder**, do this:

- 1) Select a test database, which contains the necessary single line symbols for the components. See how to do this in “Shift database for the Panel builder” on page 478.
- 2) Open the **Project** folder, and open the project **PanelDesignExample.pro**.
- 3) Start the **Singleline Diagram** module and generate the single line diagram as described in “Generating a single line diagram” on page 400.



INDEX



INDEX FOR PART 1 AND PART 2

- A**
- Absolute measures 85
 - Access 613
 - Accessory symbols 217
 - Activate next 108, 736, 850
 - Active 466
 - Active, check box 368
 - Actual page 468
 - Adaptability
 - symbol names 373
 - Adapting the system to your needs 767
 - Add a wire to a cable 215
 - Add new 488, 507
 - Additional 603
 - Adjust to printer 317
 - Adobe Acrobat Reader 34
 - ALARM, symbol folder 218
 - Alias
 - correct 821
 - Align 59, 619
 - Align grid to 1. Reference 815
 - Alignment 120
 - All pages 315
 - All visible 466
 - Already placed objects 48
 - Alternative electrical symbols 584
 - Alternative symbols
 - and PINDATA 586
 - for PLC 585
 - Ambiguous drawing 380
 - ANSI
 - paper size 502
 - Apply addresses 441
 - Approved components
 - in Database menu 605
 - Arcs/circles 47, 97
 - toolbar 97
 - Area button 47
 - Areas
 - change symbol names in 180
 - change symbol names in, naming order 181
 - choosing connection points in 184
 - copy
 - between projects 243
 - into Edit symbol 243
 - reference designations intelligently 235
 - copy between projects 511
 - copy intelligently 233
 - counting on texts in areas 116
 - filled 68
 - invisible 63
 - rotate 241
 - scaling texts in 242
 - Article data 814
 - on lines 72
 - transfer for symbols or lines 55
 - Article number
 - type in directly 149
 - Articles from unit drawings 534
 - Ask for Names 104, 816
 - Assign article number to unit parts list 749
 - Asterisk 540
 - Auto hide scrollbars 817
 - Autocompletion of texts 100
 - Automatic
 - blocking of mouse clicks 784
 - insertion of electrical potentials 74
 - line drawing 81
 - recovery information 828
 - wire numbering 377
 - Automatic information 765
 - Automatic switches, database folder 147
- B**
- Background color 773
 - Baltic characters 768, 829
 - Barcode scanners 615
 - Base addressing
 - for PLC's 437
 - Batch printing 804
 - on net installation 804
 - on stand-alone installation 804
 - BDE 613
 - Bend 93
 - Black background 817
 - Bold text 121
 - Bookmarks
 - in Symbol menu 143
 - Bottom to top 455
 - BUILDING, symbol folder 218
 - Busbar symbols 703
 - Busbar, symbol type 687
 - Busses 454
- C**
- Cable list, page function 491
 - Cable symbols 210
 - add wire to 215
 - color codes 213
 - create 701
 - implicit references on cable wires 206
 - invisible 217
 - place in one operation 214
 - remove wire from a cable 215
 - reverse direction 216
 - same cable in more places 216
 - Cables
 - move

connection points	214	lists in columns	536
Cables lists		COM/OLE automation	7
create	740	Combined circuit and connection diagram	92
data fields	108	Command bar	36
list setup	543	Command toolbar	850
Cancel button	23	Company and User name	828
Capital letters	123	Company logo	
Cascade	509	on drawing header	731
CD		remove	745
component vendors	8	Component	
Change		insert in database	580
between projects	513	Component databases	
between windows	23	how does the database work	562
column of data fields	746	Component Grouping	
created drawing header	745	and copying	551
data field	746	menu	547
free texts	745	Component vendors	
name format types	372	CD with databases	8
symbol in folder	710	Components	
symbol names in Area	180	approved and obsolete in Database menu	605
symbol names in Area, naming order	181	changing symbol data for Component	230
symbol type	710	create in database	579
text	107	from existing project	580
text in the cross hair	112	fetching component symbols from database	222
Chapters	485	find in database	566
chapter tabs collapsed	485	functions on component level	220
Character set	768	grouping	546
choose	829	having several electrical symbols	583
Chinese version		insert new names	284
PCschematic Viewer	660	navigation between symbols	165
Choose language	829	place available symbols	231
Circles	47, 97	rename selected	278
align	59	direction for renaming	281
convert to lines	98	start number	282
full	97	which names are renamed	279
half	97	rrename	
invisible	63	when copying areas	233
quarter	97	sorting order	548
space	60	survey of symbols with same Symbol Name	227
toolbar	97	Components list, page function	491
Circles button	97	Components lists	108
Click	23, 850	create	739
Clipboard		files	629
copy to	516	Conducting (electrical) lines	71
Close, symbol status	584	Conducting lines button	71
Close, symbol type	686	Connection data	182
Codepage	644	Connection is used in several nets	312
Color	120, 773	Connection names	588, 851
background	773	in one line diagrams	398
dotted grid	773	in single line diagrams	398
help objects	774	insert new	284
lines	69	Connection point button	694
object selection	774	Connection point texts	106
Squared grid	773	Connection points	182, 851
squared grid-10	773	choosing in areas	184
Color codes for cables	213	edit next/previous	183
Columns		place	679
change column of data fields	746	Connections lists	195

wire numbers	392	in symbol names	370
Contact mirrors		setup	367
about	363	Curved lines	67
and changing article numbers	363	Curved lines button	67
choose	365	Czech version	
created automatically by the program	365	PCschematic Viewer	660
creating without using database	365	D	
get available symbols in the cross hairs via	363	Damaged files	828
jump to reference from	356	Data	
jump to the other symbols for the component	363	transfer between objects	55
Continuous	368	Data field dialog box	107
Coordinate field	460	Data fields	107, 851
Copy		and PLC lists	741
and component grouping	551	change	746
area form one project to another	511	change column of	746
areas between projects	243	copy rows of	736
areas intelligently	233	create symbol data fields	712
reference designations	235	insert	725
drawing headers	514	missing	524
project into another	506	page/project data	669
rows of data fields	736	reference designations	351
selected objects	52	types	725
symbol names when copying	372	Data symbols	
to other programs	516	and lists	715
Copy button	52	create	711
Copy properties button	56	place in project	713
Correct		with predefined values	715
directories or alias	821	Data, symbol type	687, 713
Unit drawing/parts list	751	DATABASE	824
Counting		Database	826
on texts	116	1. and 2. Searchkey	568
on texts in areas	116	1.Searchkey	567
Counting direction	368	CD from component vendors	8
Counting direction for naming	181	choose a DB alias in PC SCHEMATIC	614
Create		choose database from PC SCHEMATIC Database	614
cable list	740	se	614
components list	739	choose database menu	608
diagram page w. list	742	choose from other DB program	614
drawing headers	721	choose Menu using other DB programs	609
lists	732	choosing a database	613
PLC lists	740	create component	579
signal symbol	717	from existing project	580
symbol	676	Database menu	
Table of Contents	739	approved and obsolete components	605
terminals lists	739	Product choice or Fabricate choice	604
unit drawing	748	editing records	572
Criteria, tab	539	enter using the pickmenu	138
Cross hairs	43, 851	fields	
object selection	774	file names in	612
Cross hairs / Small cross hairs	815	links	612
Cross hairs figure with snap	816	find components	566
Cross hairs line displayed right angled	816	for lines	561
Ctrl key	23	how does the database work	562
Current page	315	insert component in database	580
Current path naming	367	interface	598
Current path numbers		mechanical load	616
default settings	368	menu structures in other DB programs	614
		place cable using color codes from database	

213	
place symbol in pickmenu using database	176
programmable barcode scanners	615
searching in the database	565
settings	598
transfer information from	563
translate fields	598
update symbols from	166, 223, 827
use	561
Database setup	826
Datatemplates	
create variablelist	271
dBase	613
dBASE files	614
DDE functions	804
Default vaule	
for variable	848
Default, tab	369
Delete	
page with a list on	747
selected objects	52
symbol	168
symbol from folder	710
templates	758
Delete button	52
DEMOMEK, symbol folder	218
Design	
cable list	740
components list	739
data symbol	712
diagram page w. list	742
lists	732
PLC lists	740
signal symbol	717
symbols	676
Table of Contents	739
Terminals lists	739
unit drawings	748
Design Check	
dialog box	311
double-click on error in list	311
save and print	312
update design check list	311
Designcheck	310
Diagram	
print	314
Diagram, page type	501
Diagrams	
loop diagrams	7
Dialog boxes	22
selecting elements in lists	22
Dialogue	8
Dictionary	850
Different types of objects, select	232
Dimmed	22, 851
Direct access	
to menus	43
Directories	819, 851
most used	820
Directory alias	
show	820
Displayed in project	123
Distance (mm)	368
Distance, lines	69
Divider, page function	492
Dock	851
a window	25
Explorer window	24
Symbol pickmenu	25
Toolbars	25
Document object =>	
Edit	651
Open	651, 652
Properties	652
Dotted grid color	773
Double-click	23, 851
Drag	
pages between projects	510
Drag and drop	24, 851
Draw	
house walls	461
inner walls	462
Draw button	65
Drawing	
by measures	85
measurement objects	87
print	314
standard projects	504
unambiguous	380
with elevation	469
Drawing headers	814
add to list of drawing headers	729
and copying	514
and layers	467
change already created	745
company logo on	731
create	721
insert	521
predefined	730
remove form list of drawing headers	729
replace	526
Drawing objects	47, 852
Drawing templates	
insert variables	269
insertion points	267
Drive	852
Drop down menu	22
Dutch version	
PC SCHEMATIC Viewer	660
DWG files	640
export	643
export project as	643
export symbol as	645
import	640
import as symbol	643
open	641

open new project by importing	642	EN1861	218
DXF files	640	EN60204-1,14.2.1, standard	234
export	643	EN60617, symbol folder	218
export project as	643	EN61346	322
export symbol as	645	signal names according to	203
import	640	Enter Wire Numbers Manually	387
import as symbol	643	Error	
open	641	general protection error	44
open new project by importing	642	Error messages	312
E		Even pages	315
Edit		Excel	
templates	758	integration with, survey	646
Edit =>		place symbols via lists	654
Align	59	PLC I/O addressing via	452
Convert to lines	98	Explorer window	291, 817
Copy to Clipboard	516	displaying and hiding information	292
Data tags	270	move	24
Find	306	Selecting a symbol on a page	294
Find next	307	Export	
Insert special	650	DWG and DXF files	643
Join signal	454, 455	PLC I/O data to Excel	452
Links	653	project as DWG or DXF file	643
Next	183	symbol as DWG or DXF file	645
Paste	243	Extend line button	82
Pinswop	185	Extended lines	82
Previous	183	Extensions	
Replace	308	of list files	638
Scale	287	F	
Select all =>		FABRICATE (Show all)	566
All on layer	468	Fabricate choice	
All on page	50	in Database menu	604
Space	60, 619	Factor, ellipses	98
Variables	249	Favorites	767
Editing symbols for a building	464	Field headings	635
EIB, symbol folder	218	File =>	
Electrical installations		Close	511
in different elevations	463	Close and return to PC schematic project	651
Electrical lines	71	Modules	764
Electrical potentials	196	New	35, 505, 753
automatic insertion of	74	New =>	
on lower part of page	75	PC schematic project file	796
on upper part of page	75	Open Unit	751
settings for	76	Print	314
Electrical symbols		Print =>	
components having more	583	Select pages =>	468
create automatically	192	Save unit as	750
box symbols	193	File =>.Send to	484
circle symbols	193	File format	635
connection points	193	File type registration, dialog box	795
from database	194	Files	852
place based on placed mechanical symbols	625	commasep. files	
Elements of a project	483	place symbols via	654
Elevation	469, 852	DWG/DXF	640
lines changign elevation	469	file names	
select elevated objects	469	in database fields	612
Elevation window	852	from Object Lister	301
Ellipses	98	lists as files	629
Empty lines between Terminal Rows	543	pindata files	589

registering PCschematic filtypes	795
wire number	389
Fill character	108
Filled areas	68
Find Next	307
Find Whole Text Only	307
Fixed digits	383
Fixed text direction	121
Flat buttons	817
FLOWCHAR, symbol folder	218
Folders	852
change symbol in	710
delete symbol from	710
most used	820
symbol folders	217
Follow connected button	83, 718
Follow connected lines	82
Font	119
Foreign languages	768
Format file	630
Formulas	
insert	521
projects	504
Frame	120
Free texts	99, 852
change	745
Ftp	
links in database fields	612
Full circle	97
Function aspects	322, 325
Functions =>	
Auto command change	51
Auto Name	171
Centre drawing	503
Component Grouping	209, 547
Coordinates	85
Count down	117
Count up	117
Database =>	
Find Article	149
Find Type	150
Insert in Database	580
Update from Database	166
Design Check	310, 311
Design Check =>	
Used more than once	
deselect	313
Generate isometry	503
Measurement	87
Mechanical load	617
Object Lister	297
Place Accessory	217
Place available Symbols	231
Replace symbol	187
Select name	227
Show available	224
Show available again	148, 224
Show conducting	72
Show signal	202

Special Functions =>	
Convert Branches to Dots	96
Convert Dots to Branches	95
Convert to Branches	96
Create CurrentPath No on all Symb	371
Exchange Article with Alt. Article on all Symbols	600
Prefix all symbol names with -	339, 816
Set all articles as approved in the database	607
Set unreferenced cable symbols invisible	217
Set unreferenced signal symbols invisible	203
Transfer data from PLCref. to PLCsymb.	447
Transfer data from PLCsymb. to PLCref.	447
Transfer Proj. Texts to Auto Compl. func.	101
Update references	360, 369
View Jumper Links	80
Wire numbers	377

G

General protection error	44
General purpose texts	99
General, tab	534
Generation names	371
German version	
PCschematic Viewer	660
Grid	43, 852
Grid display	815
Ground plan	459
place symbols on	463
with elevation	469
Ground plan, page type	501
Group Number	550
Grouping	
components	546
Guess	
wire numbers	380

H

Half circle	97
Height	119
Help	32
Help =>	
Http	
//www.pcschematic.com	18
Help frame	36, 818
Help objects color	774
Hints	817, 852
Home view	
set User Home view	40
HOMEDIR	824
Horizontal reference	125
Horizontal, tab	368
Hot-line	8
House drawing	459
Http	
links in database fields	612
Hungarian version	
PCschematic Viewer	660
Hydraulics	
place symbols via lists	654

Hydraulics, symbols	218	automatic information and update	765
I		Interval in minutes	828
I/O lists		Invert text	112
read	449	Invisible	466
I/O symbols		areas	63
create PLC I/O symbols	704	circles	63
IEC750 standard	322	lines	63
IECINST, symbol folder	218	objects	62
IECPNEU, symbol folder	218	Sub drawings	64
Ignored, page function	491	symbols	62
Image		texts	63
insert	59	ISO	
scale	287	paper size	502
Image functions	38	Isometric, page type	501
Import		Isometrical drawing	503
DWG and DXF files	640	Italic text	121
as page in project	641	J	
PLC I/O data from Excel	453	JIS	
Include Additional from Database	535	paper size	502
Include cable wires	379	Join symbols	
Include Lines with article data	535	define	720
Index		Jokers	540
on-line manual	34	Jumper links	79
Indexed variables	849	in lists and terminal plans	80
Indexing	798	K	
Information		Keep page zoom	39, 817
automatic information	765	Keep zoom	39, 816
Insert		Komponenter	
data fields	725	rename selected	
image	59	selecting symbols	280
list	529	L	
logo	517	Language	829
page for mechanical drawing	616	Layers	466
unit drawing	558	and drawing headers	467
Insert - in front of symbol names	339, 816	change using the mouse	467
Insert =>		print page with	468
Insert data field	107, 834	Left tool bar	37
Insert object	648, 649	Left toolbar	852
Insert Picture	286	License key	829
Insert Potential	74	Licenses	
Insert Reference Frame	342	SuperPro monitor	781
Insert sub drawing	92	tanking licenses	775
Insert new	488	transfer licenses by tanking	776
symbols or connection names	284	transfer licenses via license files	777
Insertion points	267	Lines	47, 65
Install modules	764	article data	72
Installation		Automatic line drawing	82
pickmenu	463	automatic wire numbering	377
Installations		by measures	85
LV installations	7	changing elevation	469
instdemo.pro, demo project	463	color	69
Intelligent		curved	67
copying of areas	233	distance	69
reference designations	235	electrical	71
Interface		extended	82
database	598	follow connected lines	82
Internet			

- in symbols 679
- invisible 63
- jumper links 79
- rectangles 68
- round 67
- router 82
- select elevated 469
- slanted 67
- straight 66
- toolbar 66
- transfer article data 55
- trim 84
- type 68
- types for ground plans 461
- width 69
- Links
 - edit 653
 - in database fields 612
- List =>
 - Read PLC I/O List 449
- List files
 - buttons in the dialog box 633
 - create 630
 - criteria 634
 - example 637
 - extensions 638
 - field headings 635
 - file format 635
 - matching the system you export to 634
 - reference designations in 636
 - the contents 634
 - the file name 634
 - the format file 630
- List of drawing headers 729
- List of used objects 306
- List setup
 - terminals- cables- and PLC lists 543
- List symbols 742
- Lists
 - add to the list of drawing headers 738
 - as files 629
 - columns 536
 - company logo on 731
 - connections lists 195
 - wire numbers 392
 - create 732
 - criteria 539
 - data symbols on 715
 - delete page with list on 747
 - diagram page w. list 742
 - for each Higher level or Location 352
 - insert 529
 - jumper links in 80
 - list symbols 742
 - missing data fields 524
 - order in 546, 550
 - place symbols via lists 654
 - predefined 744

- references to location of mechanical symbols .
 - 622
- remove form page 747
- settings 532
- symbols
 - sub names in lists 161
- unit parts lists 748
- update 531
- using reference designations 352
- XML format 632
- Lists =>
 - Connections List File 195
 - Connectionslist for file 391
 - List Settings 545
 - List settings 533, 534, 543
 - Parts List File 630, 637
 - PLC List File 452
 - Read Components List =>
 - Create from diagram page 657
 - Create from mechanical page 657
 - Load from file 654
 - Update all lists 531
 - Update lists 533
 - Update parts lists 531
 - Update Table of Contents 491
 - Update Unit Drawing 750
 - Wire number file 389
- Load defaults button 368
- Load previous project 829
- Load selected 617
- Load standard button 124
- Location aspects 322, 325
- Lock
 - after system break down 788
 - delete lock information manually 788
 - project files 788
 - project in use 788
 - text types 115
- Logo
 - insert 517
- loop diagrams 7
- LV installations 7
- M**
 - Mailto
 - links in database fields 612
 - Main reference, check box 368
 - Manual 32
 - Survey of 15
 - Manual placing of wire numbers 386
 - Mapping
 - database fields 598
 - Margin 317
 - Master reference, symbol type 687
 - Match Case 307
 - Maximize 853
 - MCS 782
 - MDAC 613
 - Measurement 87

free measurement	89	block for clicking on a function	783
reference points and measurement	89	disable	784
rotate	89	Mouse pointer	853
Measurement objects		Move	
layers	91	Explorer window	24
Measures		origo	459
absolute	85	reference points	724
polar	85	selected objects	51
relative	85	Survey window	25
Measures, drawing by	85	symbol	169
Mechanical drawing		Symbol names when move	372
insert page	616	Symbol pickmenu	25
Mechanical load		Toolbars	25
from database	616	Move button	51
references to mechanical symbols in lists	622	MSSQL	613
Mechanical symbols	194, 593, 603	Multi layer terminals	208
create automatically	192, 194	on diagram pages	208
box symbols	193	on mechanical pages	209
circle symbols	193	Multi symbols	694
connection points	193	fetch	695
from database	194	Multi-level terminals	590
start by drawing mechanically	625	Multisignal, symbol type	687
MECTYPE	583	N	
Menu bar	21, 36, 853	Name format	370, 831
Menus	21, 853	Name format types	
choose database menu	608	change	372
direct access	43	Native drivers	8
Microsoft Excel		Navigation	
integration with, survey	646	between symbols for component	165
Microsoft SQL Server	8	Negate	112
Minimize	853	Negate text	112
Mirror		NET	824
symbols	171	Net	
Mirror symbol horizontally button	171	tanking licenses	775
Mirror symbol vertically button	171	Net router	816
MISC, symbol folder	218	Networks	
Missing connection name	312	sub drawings in	244
Missing data fields	524	New document button	35
Missing symbol name	312	New versions	8
Modules	763	No dot generation	698
install and uninstall	764	No visible	466
start a module	764	Non conducting symbol	853
update and re-install	764	Non conducting, symbol type	687
which can you install	763	Normal, page function	491
More signals in same point	312	Normal, symbol status	584
Mounting correct drawing	92	Normal, symbol type	686
activate	93	Norms	
bend	93	page norms	502
choosing in an existing project	94	Not yet written (New)	389
convert from previous verisons	96	NP	853
convert to automatically	95	Number	368
Mounting mode	380	Numbering order	381
Mouse		O	
change layer	467	Object Lister	306
Favorites	767	check project texts	304
mouse pointer	24	counting on selected objects	299
Mouse Chasing System	8, 782		
automatic blocking of mouse clicks	784		

- editing PLC I/O data 303
- print 306
- printing from 300
- save as file 301
- selecting objects in 298
- show used and count 302
- typing in PLC I/O data 443
- Object properties 56
 - change on existing object 56
 - transfer to more objects 56
- Object selection
 - Cross hairs 774
- Object selection color 774
- Objects 47, 853
 - align 59
 - already placed 48
 - invisible 62
 - new/draw/place 48
 - select different types of 232
 - select elevated 469
 - space 60
 - survey 306
 - transfer data between 55
- Obsolete components
 - in Database menu 605
- ODBC interface 8, 613
- Odd pages 315
- OK button 23
- OLE automation 7
- OLE objects
 - edit links 653
 - properties 652
- On all pages 526
- On this page 526
- One line diagrams
 - connection names 398
- On-line manual 32
 - adapt to screen 33
 - buttons 33
 - index 34
 - index-finger 34
 - open 32
 - printing from 34
 - return to PCschematic 34
 - Table of Contents 33
 - zoom 33
- Only one connection in net 312
- Only show text on data 108
- Open
 - DWG and DXF files 641
- Open dialog box 507
- Open files from Tools menu 659
- Open, symbol status 584
- Open, symbol type 686
- Operator 539
- Oracle 8, 613
- Order
 - in lists 546, 550

- Origo
 - move 459
- Ortho 45 816
- Other programs
 - copy to 516
- Output 314
- Overall planning 665
- P**
- Page
 - page setup 500
 - transfer to other pages 500
- Page data 813
 - data fields 107, 669
 - add 672
 - in standard project 667
 - type in 111
- Page Data, data field type 725
- Page format 502
- Page function 822
- Page history buttons 486
- Page menu 487
 - copying pages 494
 - naming 494
 - symbol names when changes 373
- Page settings 35, 459
- Page setup 822
- Page shift between Terminal Rows 543
- Page tabs 817, 853
- Page type 822
- Page types 501
- Page zoom
 - keep 39
- Pages 485
 - copy 494
 - create new 489
 - create page templates 753
 - drag between projects 510
 - insert drawing header 522
 - Object Lister 297
 - page history 486
 - Page menu 487
 - reference designations 340
 - displaying reference designation on page 341
 - remove list form 747
- Pages and layers 535
- Panel builder
 - creating own components 477
 - datafields 476
 - drawing headers for 477
 - try the Panel builder 478
- Panel builder
 - project template 478
- Paper and margin setup 317
- Paper size 502, 822
- Paradox files 614
- Parked line 312
- Parking 72, 853
- Parts list, page function 491

Parts lists	108	symbols	219
files	629	synchronize PLC data	447
order in	551	text links	132
prices	604	typing in I/O data via Object Lister	443
Parts/Components list, data field type	725	update PLC projects from previous versions	447
Password		workflow	
projects	787	for PLC's not in the database	445
Path	853	with PLC's from database	438
PBCODE, datafield for Panel builder	477	working with PLC's	436
PBCURRENT, datafield for Panel builder	476	PLC functions	
PBMODULES, datafield for Panel builder	476	Apply addresses	441
PBPHASES, datafield for Panel builder	476	PLC list, data field type	725
PBSYMBOL, datafield for Panel builder	477	PLC list, page function	491
PC SCHEMATIC Loop	7	PLC lists	
PC SCHEMATIC PowerDistribution	7	and data fields	741
PCschematic		create	740
registering filetypes	795	data fields	108
PCSTYPE	583	list setup	543
Pencil button	65	Pneumatics	
Pickmenu	36, 136	pickmenu	463
attach data to symbol in pickmenu	175	Pneumatics symbols	218
enter database from pickmenu	138	connection points with same name	313
for installations and pneumatics	463	place symbols via lists	654
place symbol in pickmenu using database	176	Pointer	815
place symbol with data	173	Pointer/Screen	815
place wire number symbol in	386	Polar	85
Pin names	602	Polish characters	768, 829
PINDATA	583	Polish version	
Pindata	588	PC SCHEMATIC Viewer	660
Pindata files	589	Position of name when placing symbols	127
Place		Posno1	750
available Symbols	231	Potential numbering	379
symbols	151	Potentials	196
Place missing	381	automatic insertion of	74
Place missing and renumber	381	on lower part of page	75
Planning	665	on upper part of page	75
PLC	436	settings for	76
addressing directly from database	437	PowerDistribution	7
base addressing	437	Precede page number	383
choose format file for I/O file	449	Predefined drawing headers	730
contents of I/O file	450	Predefined lists	744
create PLC I/O symbols	704	Prefix	383
create PLC ref. symbols	707	Preparations	665
Data on I/O symbol connection points	442	Previous versions	
displaying changes in project acc. to I/O file	451	update PLC projects from	447
I/O addressing via Excel	452	Price 1	601
I/O data		Primary	535
edit	303	Print	314
jump PLC ref. symb. / PLC I/O symb.	444	batch printing	804
PLC reference symbols	436	on net installation	804
more than one	437	on stand-alone installation	804
read I/O lists	449	chapters and dividers	316
re-addressing	440	from Object Lister	300
specify I/O file filename	450	object lists	306
survey over functions	435	page with layers	468
symbol folder	219	zoom	317
symbol status	584	Print pages button	314
symbol type	687	Print test page	318
		Print this page button	314

Printer help frame	818
Printer, tab	318
Product choice	
in Database menu	604
Program settings	35
Program toolbar	36, 854
Programmable barcode scanners	615
Programs	
copy to other	516
Project data	811
data fields	107, 669
add	672
in standard project	667
type in	111
Project Data, data field type	725
Project definition file	
controlling project texts from	269
Project menu	
Windows Explorer as	321
Project oriented program	483
Project settings	35
Project templates	
create	666
Projects	
about	483
compressed saving	806
controlling texts from project definition file	269
copy areas between	511
copy project into project	506
copying areas between	243
create project templates	755
from existing project	757
creating project templates	666
drag pages between projects	510
elements of	483
export as DWG or DXF file	643
insert logo	517
insert sub drawing	247
jump between several projects	295
lock file	788
lock project in use	788
open from symbol	514
password	787
planning	665
same project in two versions	515
searching for texts via Windows	799
send by e-mail	484
simultaneous projects	509
survey of	515
zip	806
Properties =>	
Project data	796
Protected	387
Q	
Qualified guess	380
Quarter circle	97
Question mark	540

R

Re-addressing	
PLC's	440
Reading direction	126
Recovery information	828
Re-creating files	828
Rectangle button	68, 461
Rectangles	68
with relative coordinates	86
Red arrow	108
Redraw button	40
Reducing repetitive strain injuries	782
Reference cross	368
Reference crosses	125
about	364
choose	365
jump to reference from	356
Reference designations	322
data field for specifying	351
defining	325
displaying on page	341
for a page	340
in list files	636
insert a data field	341
insert auto. on signal symbols	203
prefix all symbol names with -	339
symbol names on pages with assigned	341
symbols	336
Reference letter	601
Reference point button	718
Reference points	818, 854
move	679, 724
selection	737
Reference symbol position	368
Reference symbols	
PLC reference symbols	436
Reference system	
default settings	368
setup	367
Reference system setup, dialog box	367
Reference text	200
Reference, symbol type	687
References	814
appearance	360
automatic update of ref.	355
implicit references on cable wires	206
implicit references on terminals	204
jump to ref. from ref. cross/contact mirror	356
jump to reference	355
on symbols	357
symbol type and reference	358
Refresh	40
Register command	805
Registration	
of PC schematic ELautomation	805
of PC schematic filetypes	795
Re-install modules	764
Relative	85

Relay, symbol status	584	Save with backup	819
Relay, symbol type	686	Scale	
Remove		image	287
company logo	745	scale factor	823
list from page	747	symbols	171, 688
Remove blocking	784	texts in areas	242
Remove existing	382	Screen functions	38
Remove existing and replace	382	Scroll	854
Rename	467	Search	
selected symbols	278, 279	in database	565
symbols from Symbol menu	690	Searchkeys	
Renumber existing	382	in database	567
Repetitive strain injuries		Secondary	535
block for clicking on a function	783	Select	
Repetitive strain injuries, reducing	8, 782	area in all layers	468
Replace		different types of objects	232
drawing header	526	elements in lists	22
Replace All	308	elevated objects	469
Replace symbol texts	308	everything on a layer	468
Replace symbols	187	everything on a page	50
connection points	189	more objects of same kind	49
text properties	189	object on different layer	468
Reports		objects	49
loop reports	7	on different layers	468
Reset group numbers button	548	right-click	49
Restart on page shift	383	several objects using mouse	50
Restarting the program	44	symbol by right-click	171
Reverse direction	216	symbol from database	147
Right-click	24, 854	symbol from pickmenu	136
select	49	symbol from the symbol menu	140
short-cuts	770	symbol using k-key	148
symbols	171	Select all layers	468
Rotate		Select Component	231
areas	241	Select database file	826
selected object	52	Select pages, tab	315
texts	120	Select set of characters	829
Rotate button	53	Selected objects	
Round buttons	22	copy	52
Round lines	67	delete	52
Round lines button	67	move	51
Router	81	rotate	52
Router =>		working with	51
Active	81	Selection	
Rows		reference points	737
copy rows of data fields	736	Semi ISOmetric, page type	501
Russian characters	768, 829	Semicolon	588
Russian version		Send project by e-mail	484
PCschematic Viewer	660	Servers	829
S		Settings	35, 809
S Article no	104	data base	598
S Function	104	lists	532
S Name	104	Settings =>	
S Type	104	Cursor/Screen =>	
Same number through terminals	379	Colors	773
Save		Database =>	826
symbol	685	Database settings =>	
Save as default button	124	Component data	567
		Database menu	614

Database setup =>	598	Show message in (ms)	816
Component data	567	Show net	195
Component search	604	Show popup help	817
Select database file	613, 614	Show popup information	817
Directories	515, 819	Show scrollbars	817
Directories =>		Show used	304
Database	589	Show used and count	302
Page data =>	111, 813	Signal buses	454
Reference designations	325, 340	Signal names	
References	367, 622	in single line diagrams	397
With drawing header	522	Signal symbols	196, 854
Page setup	500, 822	create	717
grid	43	frequently used	719
page function	492	frequently used names of	720
page type	501	insert reference designations automatically	203
snap	42	invisible	203
Page setup =>		place	197
Reading direction	126	place directly	201
Pointer/Screen	39, 104, 815	place on parked line	201
Ask for names	152	variant names	202
grid	43	Signals	
Pointer/Screen =>		Object Lister	297
Shortcuts	771	Simultaneous	
Project data	811	projects	509
Project data =>	111	Single line diagrams	397
Password	787	connection names	398
Reference designations	325	signal names	397
Settings	809	symbol names	398
System	828	symbols with states	398
System =>		Single line symbols	
Compressed saving	806	connection names for	588
File type registration	795	Slanted lines	67
License Tank	776	Slanted lines button	67
SuperPro monitor	781	Snap	42, 854
Text/Symbol defaults	831	Snap button	42
Drawing headers	729	Sort by name	61, 619
References	360	Sorting order	
Signal name	720	of components	548
Signal symbols	719	Space	60
symbol data fields	712	Spares pr. Page	383
Text defaults	370	Special characters	102, 112
Tools	824	Specify character set	768
Settings => Page setup		Specify symbol status	593
Paper size	502	SQL Server	8
Setup, tab	317	Squared grid color	773
Several projects		Squared grid-10 color	773
jump between projects	295	Squares	22
Short-cut keys	769	Standard characters	768
learning while working	784	Standard projects	504
user defined	771	convert to templates	757
Show		copy into project	506
symbol texts	685	Standard/Project file	507
Show all	200	Standard/Project file, page function	492
Show available	224	Standards	
Show available window	226	EN1861	218
Show defaults button	124, 368	EN60204-1,14.2.1	234
Show directory alias	820	EN60617	218
Show from/to	200	EN61346	218, 322
Show layer settings	316		
Show list of objects	328		

EN81714-2	218	select library via folders	141
IEC/EN60617	218	show project symbols	141
IEC750	322	Windows Explorer as	321
ISO1219	218	Symbol pickmenu	36, 818
page standards	502	dock/move	25
Start number	383	Symbol reference	125
Start value	368	Symbol settings button	718
Starting functions	802	Symbol texts	
load project with page open	802	replace	308
load project with symbol selected	802	Symbol toolbar	
loading specific pages from project	802	select symbol type in area on screen	171
States		Symbols	47, 133
symbols with states	162	accessory symbols	217
Status bar	855	align	59
Straight lines	66	alternative electrical symbols	584
Straight lines button	66	alternative symbols	
Straighten lines	816	PINDATA	586
Strategy	665	alternative symbols for PLC	585
Sub drawings	244	attach unit drawing to	558
automatic placing	252	automatic line drawing/router	81
create	255	busbar symbols	703
by copying	256	cable symbols	210
create variablelist	271	invisible	217
insert on project page	247	cables with color codes	213
insert variables	269	change in folder	710
insertion points	267	change names in Area	180
invisible areas	64	change names in Area, naming order	181
networks	244	change type	710
Sub names		changing symbol data for Component	230
on symbols	160	component having several electrical	583
SuperPro monitor	781	component level functions	220
Support	8	connection names for single line symbols	588
Support, symbol type	687	connection points	182
Survey		create	676
of projects	515	default text settings	695
of the manual	15	create automatically	192
of used objects	306	box symbols	193
Survey window	37, 817, 855	circle symbols	193
move/dock	25	connection points	193
Switch, symbol status	584	from database	194
Switch, symbol type	686	create data symbol	711
Symbol article data =>		create symbol data fields	712
Unit	558	current path numbers	370
Symbol article data dialog box	104	data symbols	
Symbol Data Field, data field type	725	with predefined values	715
Symbol Data Fields		data symbols and lists	715
data fields	108	Database	
Symbol defaults	831	update symbols from	827
Symbol definition	588	delete	168
Symbol folders	217	delete from folder	710
Symbol manipulator	695	Edit symbol mode	
Symbol menu		default text settings	695
bookmarks	143	editing for a building	464
create symbol / edit existing	146	enter database using pickmenu	138
enter the database from	146	export as DWG or DXF file	645
representation of symbols	141	fetching symbols for component from database	222
search in	142	insert new symbol names	284
select library via alias	141	invisible	62

join symbols, define	720	show texts	685
lines in	679	signal busses	454
list symbols	742	signal symbols	196
load another symbol into	696	invisible	203
mechanical	194, 593	space	60
create automatically	194	specify symbol status	593
mirror	171	states	162
move	169	in single line diagrams	398
move reference points	679	sub names	160
multi symbols	694	functionally understood as	160
name format	370	included in lists	161
names on pages with reference designations	341	specified from database	161
names regain adaptability	373	specify	160
names when changes in Page menu	373	what is	160
names when move/copy	372	survey of symbols with same Symbol Name	227
navigation between	165	symbol manipulator	695
Object Lister	297	symbol names	
open project from	514	in single line diagrams	398
pickmenu		position of name when placing symbols	127
attach data to symbol in pickmenu	175	symbol type and reference	358
enter database using pickmenu	138	terminals	
pickmenus for many components	226	multi layer	208
place symbol in pickmenu using database	176	on diagram pages	208
place symbol with data	173	on mechanical pages	209
symbols with states	139	texts connected to	104
place and name	151	toolbar	171
place available symbols	231	transfer article data	55
place on ground plan	463	type in Article Number directly	149
place via lists	654	type in Symbol Type directly	150
PLC		types	686
create PLC I/O symbols	704	update	
create PLC ref. symbols	707	from database	166
PLC symbols, working with	436	when changing article number	166
position of name when placing symbols	127	update from database	223
reference designations	336	wire number	383
references on	357	with extra connection points	695
rename		Symbols button	135
when copying areas	233	Symbols text	
rename from Symbol menu	690	S Article no	104
rename selected	278	S Function	104
direction for renaming	281	S name	104
selecting symbols	280	S type	104
start number	282	SYSDIR	824
terminals	282	System	828
which names are renamed	279	System break down	
replace symbols	187	lock after	788
save	685	System breakdown	828
scale	171, 688	System Data	
select		data fields	107
from database	147	System Data, data field type	725
from pickmenu	136	T	
from symbol menu	140	Tab key	23
right-click	171	Table of Contents	
using k-key	148	create	739
select elevated	469	data fields	108
select symbol type in area on screen	171	On-line manual	33
select using Explorer window	294	Table of Contents, data field type	725
show available	224	Table of Contents, page function	491
show net	195		

Templates	752	change free texts	745
create from standard projects	757	change in cross hair	112
delete	758	check project texts	304
edit	758	connected to symbol	104
insertion points	267	controlling from project definition file	269
on networks	752	counting on texts	116
page templates		counting on texts in areas	116
create	753	data fields	107
project templates		reference designations	351
create	755	default text settings	
from existing project	757	in Edit symbol mode	695
template projects	504	displayed in the project (overall setting)	123
Terminal plans		fixed text direction	121
jumper links in	80	frame	120
Terminal symbols	207	free texts	
create	697	default properties	103
implicit references on terminals	204	height	119
Terminal, symbol status	584	in more than one line	112
Terminal, symbol type	687	invert/negate	112
Terminals		invisible	63
multi layer	208	italics	121
on diagram pages	208	linking texts	128
on mechanical pages	209	lock text types	115
multi-level terminals	590	Object Lister	297
rename	282	rotate	120
Terminals list, data field type	108, 725	scaling texts in areas	242
Terminals list, page function	491	searching for texts via Windows	799
Terminals lists		space	60
create	739	special characters	112
list setup	543	specify text properties	118
order in	551	text links	128
Test page, print	318	create from Symbols command	129
Text	108	create from Texts command	128
Text defaults	831	delete	132
Text fields		edit	131
place	724	in PLC I/O lists	132
Text link		indication of linked texts	128
dialog box	131	what is a text link	128
Text links	128	toolbar	100
create from Symbols command	129	type in	100
create from Texts command	128	underline	113
delete	132	underlined	121
edit	131	visible text	121
in PLC I/O lists	132	width	119
indication of linked texts	128	Texts button	99
what is a text link	128	Tile horizontal	509
Text properties		Tile vertical	509
specify	118	Time-out	829
Text properties button	118	Toolbars	
Text/Symbol defaults	831	arcs/circles	97
Texts	47, 99	dock/move	25
align	59	lines	66
alignment	120	symbols	171
appearance	118	texts	100
auto completion	100	Tools =>	824
auto text		Create variable list	269, 270, 271
delete texts	101	Database	598
bold	121	Mouse Chasing System	782
change	107	Rename symbols	278

Show textfile 659
 Singleline diagram 399
 Tools menu
 open files from 659
 Top to bottom 455
 Transfer Group Number button 550
 Transfer information
 from database 563
 Translate
 database fields 598
 Trim line 84
 Type in texts 100
 Types
 line types 68
 symbol types 686

U
 Unambiguous drawing 380
 Underline 113
 Underline text 113
 Underlined text 121
 Undo 54
 Undo button 54
 Uninstall modules 764
 Unit Drawing, page function 491
 Unit drawings
 attach to symbol 558
 correct 751
 create 748
 create using drawing of entire component 750
 insert 558
 open 515
 save 750
 Unit of measurement 816
 Unit parts lists 748
 assign article number to 749
 correct 751
 without unit drawings 750
 Unregister command 805
 Update 8
 automatic information 765
 modules 764
 Update lists 531
 Update, button 369
 User defined short-cut keys 771
 User Home view
 set 40
 User name 828
 USERLIB, symbol folder 219

V
 Variables 848
 create variablelist 271
 indexed variables 849
 insert 269
 insertion with default value 848
 normal variable 848
 possible control via var. in symbol names . 849

that inserts a value in a text 848
 with default value 848
 Variant names
 on signal symbols 202
 Vertical reference 125
 Vertical, tab 368
 View =>
 Activate Pickmenu 29
 Clean screen 41
 Go to page 485
 Goto symbol 165
 Home view 40
 Page history back 486
 Page history forward 486
 Previous zoom 40
 Redraw 40
 Select page 28, 346, 487
 Set User Home view 40
 Show available window 226
 Show object below cursor 53
 Show texts 685
 Survey window 42
 Zoom 39
 Zoom All 39
 DWG/DXF 640
 Zoom in 39
 Zoom out 39
 Visible text 121

W
 Warnings 312
 Width 108, 119
 Width, lines 69
 Wildcards 540
 Window
 several windows open 513
 Window =>
 Arrange Icons 509
 Close All 509
 Tile 510
 Tile horizontal 511
 Windows 21
 change 23
 indexing service 798
 Windows Explorer
 as Project menu 321
 as Symbol menu 321
 Wire
 add to cable 215
 remove from cable 215
 Wire number, symbol type 687
 Wire numbers
 auto. incr. on manually placed wire no's . 388
 automatic 377
 connections lists 392
 Excl. signals, tab 385
 fetch symbol 386
 files 389

Format	383
Functions	379
manual placing	386
Number symbol, tab	383
numbering order	381
place symbol in pickmenu	386
place symbol manually	387
With reference	199
With reference, symbol status	584
With reference, symbol type	687
Word	516
Work area	35, 37, 855
Wrap text	108
X	
X position (mm)	368
x,y position	622
XML	
lists in XML format	632
Y	
Y position (mm)	368
Z	
Zip	
advantages by zipping	806
on zipping	806
project files	806
Zoom	855
keep page zoom	39
previous zoom	40
print	317
Zoom button	38
Zoom to page button	40

