



AUTOMATION

PART 1

15/3-2013

TABLE OF CONTENTS

PART 1

PREFACE	7
UNICODE	10
SURVEY	13
QUICK SURVEY OF THE MANUAL	15
GETTING STARTED	21
WINDOWS	23
KEYBOARD SHORT-CUTS	29
THE ON-LINE MANUAL	35
THE WORK AREA	38
SCREEN / IMAGE FUNCTIONS	42
BASIC DRAWING FACILITIES	51
ON DRAWING OBJECTS	53
LINES	72
ARCS/CIRCLES	100
TEXTS	102
SYMBOLS	137
FUNCTIONS ON COMPONENT LEVEL	228
AREAS	244
SUB DRAWINGS	256
RENAME SYMBOLS	289
INSERTING IMAGES	297
SURVEY AND REFERENCE DESIGNATIONS	299
THE EXPLORER WINDOW: THE PROJECTS TAB	301
THE OBJECT LISTER	306
FIND AND REPLACE	316
DESIGN CHECK	320
PRINTING	326
WINDOWS EXPLORER AS SYMBOL/PROJECT MENU	334
REFERENCE DESIGNATIONS	335
REFERENCES	365
AUTOMATICALLY UPDATED REFERENCES	367

CONTACT MIRRORS AND REFERENCE CROSSES	374
CURRENT PATH NUMBERS / REFERENCE SYSTEM.....	379
WIRE NUMBERS	387
AUTOMATIC WIRE NUMBERING	389
MANUAL PLACING OF WIRE NUMBERS.....	398
WIRE NUMBER FILES.....	401
SINGLE LINE DIAGRAMS	405
SINGLE LINE DIAGRAMS	407
AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS	409
PLC FUNCTIONS	415
SURVEY OF PLC FUNCTIONS	417
WORKING WITH PLCS.....	418
READING PLC I/O LISTS	432
PLC I/O ADDRESSING VIA EXCEL	435
CONNECTION SYMBOL FOR SIGNAL BUSES	437

PART 2

ELECTRICAL INSTALLATION DRAWING.....	469
CREATING A GROUND PLAN.....	471
LAYERS, ELEVATION AND SCALING.....	477
TIPS FOR INSTALLATION DRAWING.....	484
WORKING WITH PROJECTS.....	487
ON PROJECTS.....	489
PAGES AND CHAPTERS.....	491
PROJECT TEMPLATES AND STANDARD PROJECTS.....	511
COPYING FULL PROJECTS INTO OTHER PROJECTS.....	513
SEVERAL PROJECTS SIMULTANEOUSLY.....	516
OVERVIEW PROJECTS.....	524
YOU HAVE NOW ESTABLISHED A CONNECTION BETWEEN THE SYM- BOL FOR THE ELECTRICAL PANEL AND THE PROJECT THAT IT IS DOCUMENTED IN.....	525
INSERTING LOGOS IN PROJECTS.....	527
LISTS.....	529
INSERT DRAWING HEADERS/FORMULAS.....	531
LIST SETTINGS.....	542
COMPONENT GROUPING.....	556
INSERTING UNIT DRAWINGS.....	568
DATABASES.....	569
USING THE DATABASE.....	571
CREATING COMPONENTS IN THE DATABASE.....	589
DATABASE SETTINGS.....	608
CHOOSING A DATABASE.....	623
PROGRAMMABLE BARCODE SCANNERS.....	624
MECHANICAL SYMBOLS.....	625
MECHANICAL SYMBOLS - OVERVIEW.....	627
PLACING MECHANICAL SYMBOLS VIA THE DATABASE.....	628
FROM MECHANICAL TO ELECTRICAL DRAWING.....	639
MEASUREMENT OBJECTS.....	641
EXPORT / IMPORT.....	647
EXPORTING LIST FILES.....	649
DWG AND DXF FILES.....	660
INTEGRATION WITH MICROSOFT EXCEL.....	666
INSERTING FILES FROM OTHER PROGRAMS / OLE.....	667
PLACING SYMBOLS VIA LISTS.....	674
OPENING FILES FROM THE TOOLS MENU.....	679

PC SCHEMATIC VIEWER.....	680
CREATE SYMBOLS, LISTS AND PROJECTS	683
PLANNING AHEAD.....	685
PAGE DATA AND PROJECT DATA DATAFIELDS	689
CREATING SYMBOLS.....	696
CREATING DATA SYMBOLS.....	732
CREATING SIGNAL SYMBOLS.....	738
CREATING DRAWING HEADERS	742
CREATING LISTS	752
CREATING UNIT DRAWINGS	767
CREATING TEMPLATES.....	771
UPDATING, MODULES AND LICENSES	779
MODULES	781
UPDATE AND AUTOMATIC INFORMATION.....	784
ADAPTING THE SYSTEM TO YOUR NEEDS.....	786
SHORT-CUT KEYS	788
COLORS.....	792
TANKING LICENSES.....	794
PREVENTING REPETITIVE STRAIN INJURIES.....	800
ACCESS CODES, REGISTRATION E.T.C.....	803
PASSWORDS ON PROJECTS.....	805
LOCKING PROJECT FILES	806
REVISION CONTROL	808
REGISTRATION OF PC SCHEMATIC FILE TYPES	813
INDEXING.....	816
STARTING FUNCTIONS, BATCH PRINTING AND DDE.....	821
COMPRESSED SAVING / ZIPPING PROJECTS	825
APPENDICES	827
SETTINGS	829
DATAFIELDS	855
THE DATABASE FIELD PCSTYPE.....	866
THE DATABASE FIELD PINDATA.....	868
THE DATABASE FIELD PINDATA: COUNTING	869
DICTIONARY.....	873

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 telephone and data installations, for e.g. documentation, maintenance and operations.
- PC|SCHEMATIC **Loop** for automatic generation of loop diagrams based on information from a database or for instance from Excel
- PC|SCHEMATIC **Cablemanager** for cable management and cable routing.

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.

For version 12 and later versions: Unicode

In **Automation** version 12 unicode was introduced. This has the consequence, that projects created in version 11 and previous versions, now must be converted to unicode in order to be opened correctly. 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. You also have the option of executing these functions from **Tools => Scripts**. To do this, you must first install **Scripts** under **Files => Modules**.

The documentation for how to do this can be downloaded at www.pcschematic.com under **Download => Modules, drivers etc.** 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 cooperation 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). In praxis this means that the program supports practically any database format 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**.
- Various tutorial manuals also exist — such as the **Automation project - Tutorial**, the **Electrical installation - Tutorial**, the **PLC project - Tutorial** and the **Panelbuilder - Tutorial**.

All of these manuals can be downloaded at www.pcschematic.com/manuals.

VIDEOS AND ARTICLES AT WWW.PCSCHEMATIC.COM

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

At www.pcschematic.com/articles 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

Since version 12, **Automation** has been able to handle multiple regional character sets simultaneously. This is because the program 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 a project created before version 12” on page 11.

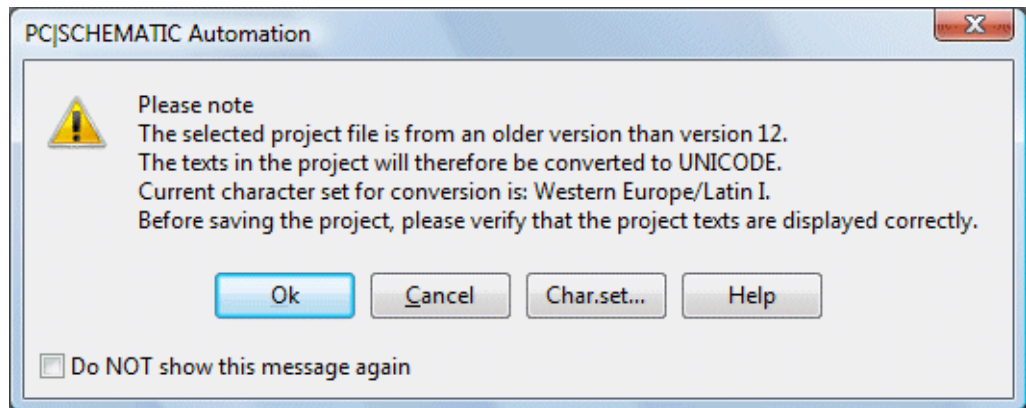
THE PC|SCHEMATIC FONT AND UNICODE

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.

The PC|SCHEMATIC font 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 A PROJECT CREATED BEFORE 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.

When you are pleased with the selected character set, and do not wish to enter the dialog box again the next time you open a project created before version 12, select ***Do NOT show this message again***.



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 FILE-NAME(ANSI).~PR/~SY. THIS MEANS, THAT IF YOU HAVE CHOSEN THE WRONG CODE PAGE 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

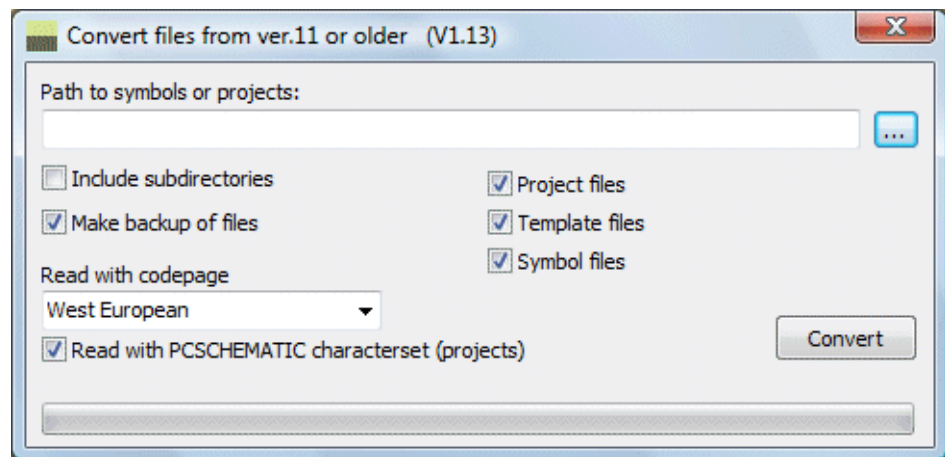
TEXTS IN SYMBOLS ARE CONVERTED AUTOMATICALLY

Conversion of old symbols function as 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 quite close to the Windows version of the Microsoft® Office-package.

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 23.

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. 21	SCREEN FUNCTIONS IN THE PROGRAM E.T.C.	INTRODUCTION TO WINDOWS THE ONLINE MANUAL THE PROGRAM WORK AREA BASIC IMAGE FUNCTIONS
"BASIC DRAWING FACILITIES" P. 51	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. 299	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 DESIGN CHECK PRINTING USING THE WINDOWS EXPLORER AS PROJECT/SYMBOL MENU REFERENCE DESIGNATIONS
"REFERENCES" P. 365	DESCRIPTION OF HOW TO USE REFERENCES IN THE PROGRAM.	AUTOMATICALLY UPDATED REFERENCES CONTACT MIRRORS REFERENCE CROSSES CURRENT PATH NUMBERS / REFERENCE SYSTEM

SECTION	DESCRIPTION	SUBJECTS
"WIRE NUMBERS" P. 387	DESCRIPTION OF HOW TO USE WIRE NUMBERS.	AUTOMATIC WIRE NUMBERING MANUAL PLACING OF WIRE NUMBERS WIRENUMBER FILES
"SINGLE LINE DIAGRAMS" P. 405	DESCRIPTION OF HOW TO WORK WITH SINGLE LINE DIAGRAMS.	ABOUT SINGLE LINE DIAGRAMS AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS
"PLC FUNCTIONS" P. 415	DESCRIPTION OF PLC FUNCTIONS.	SURVEY OF PLC FUNCTIONS WORKING WITH PLCs SIGNAL BUSES READING PLC I/O FILES PLC I/O ADDRESSING VIA MICROSOFT EXCEL

THE CONTENTS OF PART 2

SECTION	DESCRIPTION	SUBJECTS
"ELECTRICAL INSTALLATION DRAWING" P. 469	DESCRIPTION OF DRAWING FUNCTIONS WITH PARTICULAR INTEREST FOR ELECTRICAL INSTALLATION DRAWING.	GROUND PLAN DRAWING WORKING IN DIFFERENT LAYERS AND ELEVATIONS SCALING TIPS FOR ELECTRICAL INSTALLATION DRAWING
"WORKING WITH PROJECTS" P. 487	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. 529	HOW TO INSERT DRAWING HEADERS/LISTS.	INSERTING DRAWING HEADER/LIST LIST SETTINGS COMPONENT GROUPING UNIT DRAWINGS
"DATABASES" P. 569	HOW TO USE THE DATABASE IN YOUR PROJECTS.	USING AND CHOOSING A DATABASE GRAPHICAL CREATION OF COMPONENTS IN THE DATABASE DATABASE SETTINGS PROGRAMMABLE BARCODE SCANNERS
"MECHANICAL SYMBOLS" P. 625	GET AN OVERVIEW OVER HOW TO WORK WITH MECHANICAL SYMBOLS	HOW TO WORK WITH MECHANICAL SYMBOLS PLACE MECHANICAL SYMBOLS, BASED ON THE PLACED ELECTRICAL SYMBOLS, VIA THE DATABASE PLACE ELECTRICAL SYMBOLS, BASED ON THE PLACED MECHANICAL SYMBOLS, VIA THE DATABASE

SECTION	DESCRIPTION	SUBJECTS
"EXPORT / IMPORT" P. 647	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
"CREATE SYMBOLS, LISTS AND PROJECTS" P. 683	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. 779	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. 827	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 the independent **Information & Installation** manual.

If you are familiar with Windows, you can then read the independent mini manual "Motor control - Tutorial" — or for instance "Electrical installation - Tutorial". Here you get a quick introduction on how to create a project, and get ready to carry on with the detailed description of the program in "Basic Drawing facilities" on page 51. You can also read the "Quick start" manual, which gives you a quick introduction to the program.

If you do not have these manuals, you can download them at www.pcschematic.com/manuals.

Later on, you may return to the remaining part of "Getting started" on page 21, and get a better idea of the construction of the program.

You should also read "On projects" on page 489, 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 35.

Overall planning

In the section “Create Symbols, Lists and Projects” on page 683, 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 21, as well as the chapter “Find and Replace” on page 316.

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.

You can also download updates for your version of **Automation** at the website.

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.
<i>BOLD AND ITALICS</i>	ARE APPLIED FOR FIELD NAMES OR OPTIONS YOU CAN CLICK ON IN DIALOG BOXES.

TYPOGRAPHY	USE
KEYS <> <CTRL+G>	ALL SHORT-CUT KEYS ON THE KEYBOARD ARE ENCLOSED IN BRACKETS — SUCH AS FOR THE DELETE KEY. 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. SINGLE LETTERS IN BRACKETS SUCH AS FOR INSTANCE <Z> SPECIFIES THAT THE "Z" KEY IS A SHORT-CUT KEY.
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. 23
- Keyboard short-cuts p. 29
- The On-line Manual p. 35
- The Work area..... p. 38
- Applying basic Image functions..... p. 42

Reading this section will save time using the program.

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



Installation of the program

See how to install the program in our separate installation manual, which can be downloaded at www.pcschematic.com/manuals.

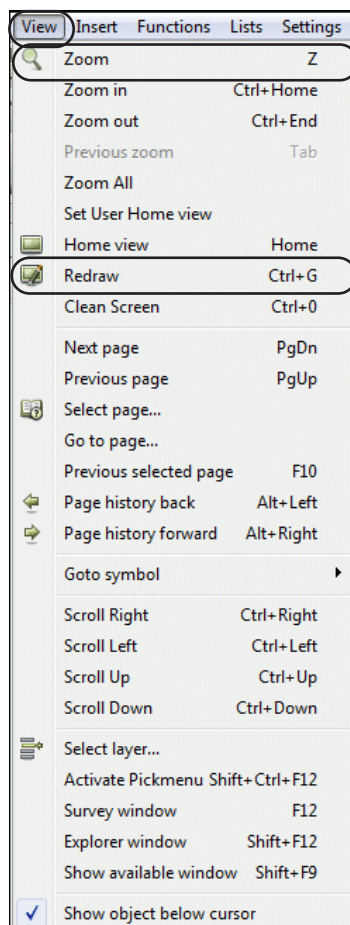
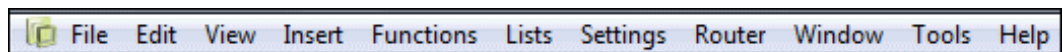
WINDOWS

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. 23
— DIALOG BOXES	P. 24
— SHORT-CUT MENUS AND RIGHT-CLICKING.....	P. 25
— SWITCHING BETWEEN OPEN WINDOWS.....	P. 25
— USING THE MOUSE.....	P. 25
— DOCKING WINDOWS AND TOOLBARS.....	P. 26

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 titled 'Pointer/Screen' 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 and contains the following controls:

- Grid display:** Three radio buttons: 'Off' (selected), 'Dotted', and 'Squared'.
- Unit of measurement:** A dropdown menu showing 'mm'.
- Show message for 300 ms:** A slider control.
- Checkboxes:** 'Hints', 'Show popup information', 'Show popup help', and 'Page tabs' (all checked).

Annotations with arrows point to these elements:

- Tabs:** Points to the tab bar at the top.
- Selected tab:** Points to the 'Pointer/Screen' tab.
- Grid display:** Points to the radio buttons.
- Unit of measurement:** Points to the dropdown arrow.
- Show message for 300 ms:** Points to the slider.
- Page tabs:** Points to the 'Page tabs' checkbox.

Explanatory text below the dialog box:

- Grid display:** 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.
- Unit of measurement:** 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.
- Page tabs:** 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 309.

— See for instance “Copying, Cutting and Pasting more than one page” on page 500.

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 309.

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 29.

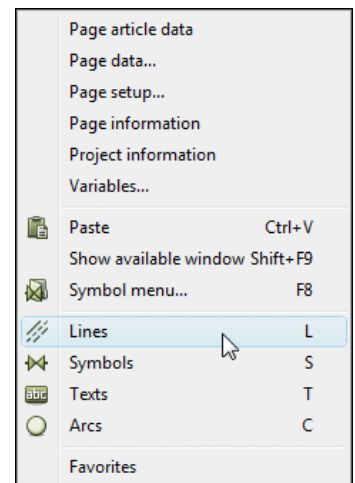
SHORT-CUT MENUS AND RIGHT-CLICKING

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 516.

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 174, “DRAGGING A REFERENCE DESIGNATION TO A NEW POSITION IN THE DIALOG BOX REFERENCE DESIGNATIONS” ON PAGE 342 AND “DRAG PAGES BETWEEN PROJECTS” ON PAGE 517.</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

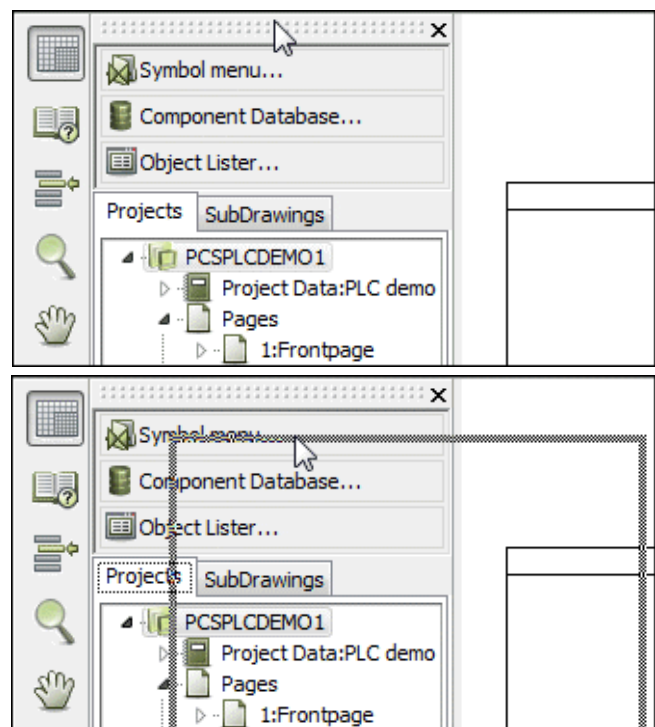
In many Windows programs you can move the toolbars and dock windows to the sides of the screen. In the following you can see how to do this in **Automation**.

When you work on a PC with two monitors, you can get a larger work area by moving for instance the **Show available window** to the secondary monitor.

Moving the Explorer window or the Show available 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. Correspondingly, the **Show available window** can be docked along the right-hand side of the screen.

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. (When you wish to move the **Show available window** you point at the title bar).



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. The **Show available window** can either be an independent window, or can be docked along the right-hand side of the work area.

Docking a window

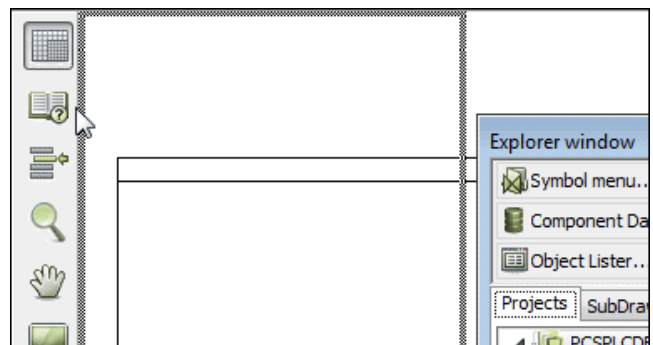
When you wish to dock the window, click on its title bar, keep the mouse button pressed down, and move the window.

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 at 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.

In order to dock the **Show available window** along the right-hand side of the screen, you must point at the right-hand side of the screen.



Docking the Survey window or the Show available 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. Correspondingly, the **Show available** pickmenu can be docked at the bottom of the **Show available window**.

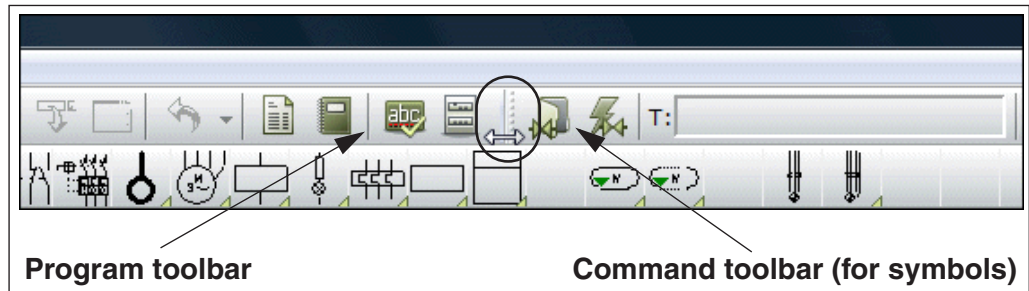
DOCKING THE PICKMENU

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.

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. 29
— CHOOSING BUTTONS/FUNCTIONS USING THE KEYBOARD.....	P. 30
— NAVIGATION IN THE EXPLORER WINDOW.....	P. 30
— CHOOSING IN DIALOG BOXES WITH THE KEYBOARD	P. 31
— SELECTION IN PICKMENUS USING THE KEYBOARD	P. 31
— PLACING OBJECTS USING THE KEYBOARD	P. 32
— CHOOSING OBJECTS USING THE KEYBOARD	P. 32
— CHOOSING MORE THAN ONE OBJECT WITH THE KEYBOARD	P. 32
— CHOOSING SYMBOLS WITH THE KEYBOARD	P. 33
— INCREASE/DECREASE SHORT-CUT USING THE KEYBOARD	P. 33
— SHORT-CUTS FOR ACTIVATING THE PENCIL.....	P. 33



CONCURRENT LEARNING OF SHORT-CUT KEYS

IN “PREVENTING REPETITIVE STRAIN INJURIES” ON PAGE 800, 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 23.

— Please also refer to “Short-cut Keys” on page 788, 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.



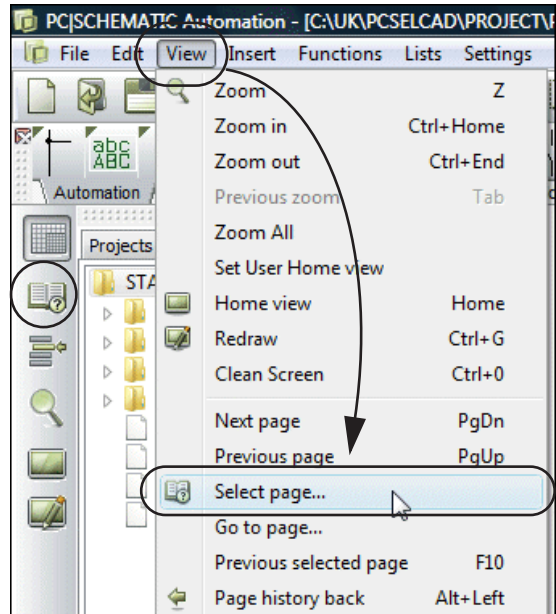
If for instance you wish to create a short-cut key for the **Page menu** button, you must create a short-cut key for the menu point **View => Select page**.



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.

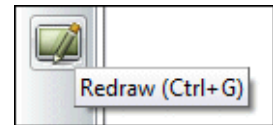
There are menu points for all buttons in the program.

— See “Short-cut Keys” on page 788 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.



NAVIGATION IN THE EXPLORER WINDOW USING THE KEYBOARD

In the **Explorer window** you use the arrow-keys to navigate like this:

<Up-arrow>: Move the cursor upwards in the **Explorer window**

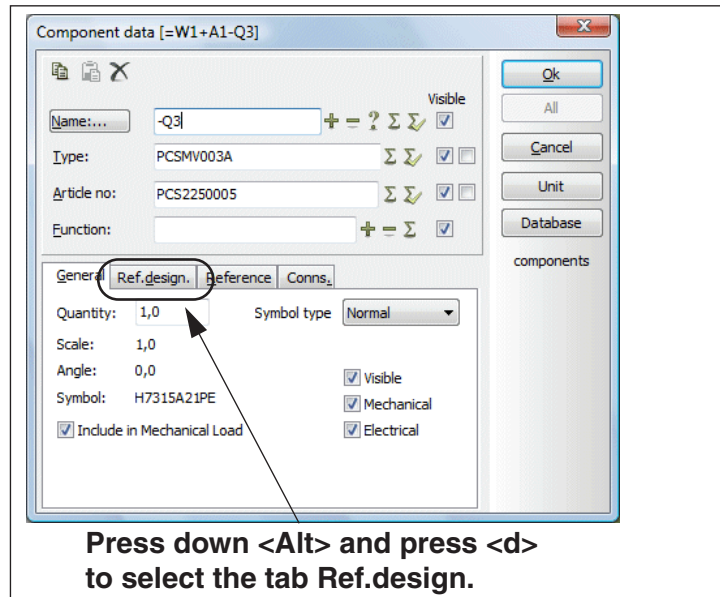
<Down-arrow>: Move the cursor downwards in the **Explorer window**

<Right-arrow>: Open folder

<Left-arrow>: Close folder

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.



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

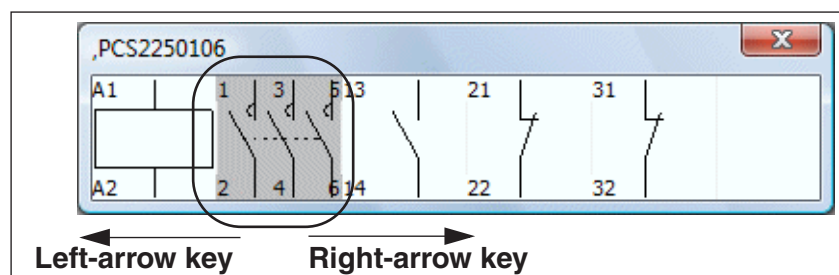
In each dialog box there is predefined order, in which you jump between the different options in the dialog box.

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

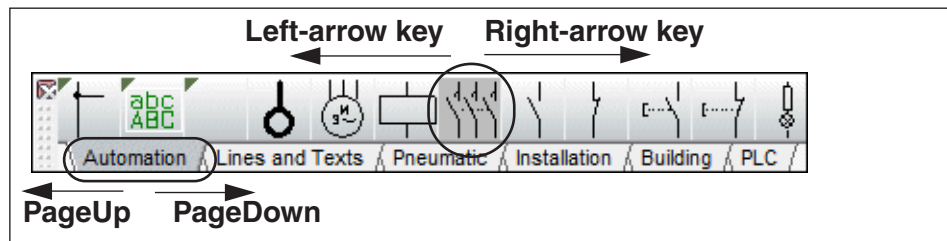
When you press down <Shift> and press the <Tab> key, you jump back to the previous option.

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 47) 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 31.
- 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 153.

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 154.

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 155.

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 121.

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. 35
— THE TABLE OF CONTENTS	P. 36
— THE BUTTONS IN THE ON-LINE MANUAL	P. 36
— ZOOMING AND ADAPTING TO YOUR SCREEN	P. 36
— DIRECT ACCESS TO THE INDEX	P. 37
— RETURNING TO Automation	P. 37
— PRINTING FROM THE ON-LINE MANUAL	P. 37

OPENING THE ON-LINE MANUAL

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

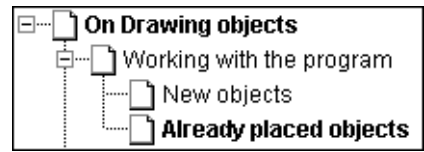
The screenshot shows the Adobe Acrobat 3D interface displaying the 'TEXTS' chapter. The interface includes a left sidebar with a tree view, a main content area, and a right sidebar with navigation controls. Labels with arrows point to various features:

- Click to view a chapter:** Points to the tree view in the left sidebar.
- Zoom:** Points to the zoom percentage (127%) in the top toolbar.
- Fit the manual to your screen:** Points to the 'Fit' button in the top toolbar.
- About the electronic manual:** Points to the question mark icon in the right sidebar.
- Return:** Points to the left arrow icon in the right sidebar.
- Up and down:** Points to the up and down arrow icons in the right sidebar.
- Click directly to Index:** Points to the 'I' icon in the right sidebar.
- 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 a plus sign icon in the left sidebar.
- Click to view the sub-chapters:** Points to a plus sign icon in the left sidebar.
- Click to jump to other chapters:** Points to a plus sign icon in the left sidebar.

THE TABLE OF CONTENTS

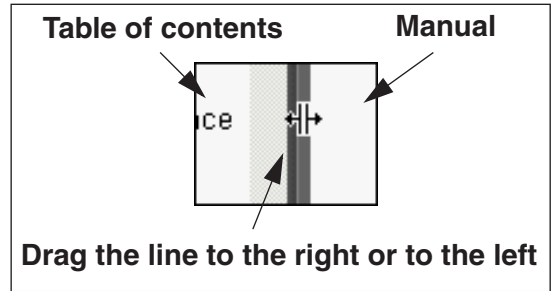
In the left-hand side of the manual, you can click directly on the section you wish to read.

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>.

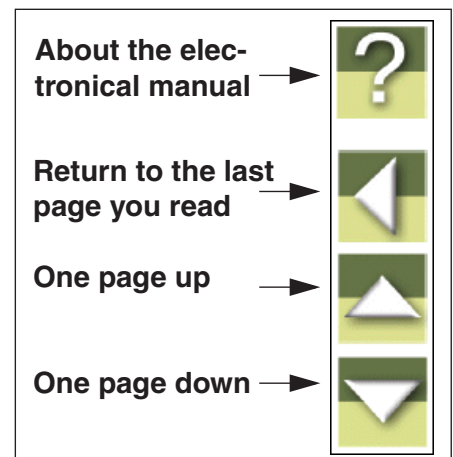
THE BUTTONS IN THE UPPER-RIGHT CORNER

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

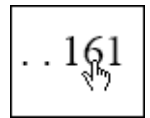
On the right-hand side of the on-line manual, you have direct access to the index.

The screenshot shows a vertical index of letters from M to X-Z. The letter 'S' is circled, and a callout box titled "Select" points to it. Another callout box titled "Selecting an area in all layers" points to a dashed selection box on the page. Below the index is a table of contents:

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 491).

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

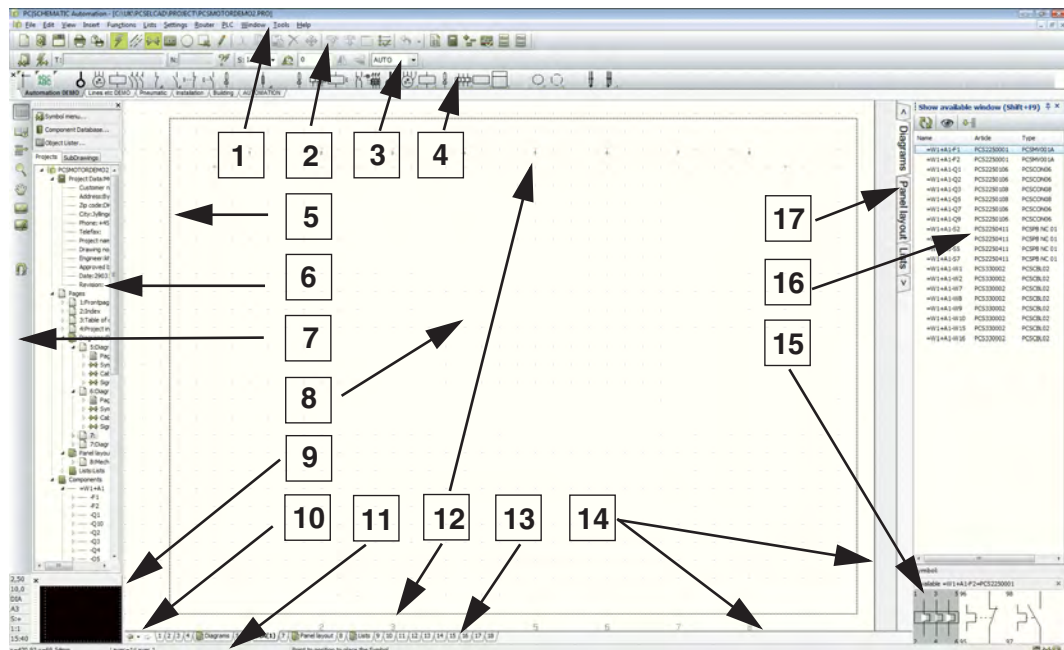
At the top of the dialog box **Settings** you will find some tabs, each containing a menu defining various program, page and project settings.

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 829.

THIS CHAPTER DESCRIBES

THIS CHAPTER DESCRIBES THE WORK AREA OF THE PROGRAM.

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 26.

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 23.

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 51.

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 140, and can be docked as described in “Docking the Pickmenu” on page 27.

5. The help frame

The help frame displays the standard margin on the paper. It can be deactivated easily — see “Pointer/Screen” on page 836. It is also possible to activate a printer help frame, displaying the margins of your printer.

6. The Explorer window

In the upper part of the **Explorer window** you have direct access to:

- The **Symbol menu**: Here you have access to all of the available symbols in the program. Read more about this in “Fetching symbols using the Symbol menu” on page 144.
- The **Component database**: Here you have access to all of the available components in the database. Read more about this in “Fetching symbols from the database” on page 152.
- The **Object lister**: Here you for instance get an overview of any types of texts attached to objects in the project — such as texts attached to different types of symbols. Read more about this in “The Object Lister” on page 306.

Furthermore, the **Explorer window** contains two tabs: 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 301.

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 256.

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 42.

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 843, or by inserting a drawing header, as described in “Insert drawing headers/formulas” on page 531.

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 46.

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 492.

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 379.

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 48 and “Choosing pages outside the Page menu” on page 491.

14. Scrollbars

If you zoomed in on an area, you can make the area move by *dragging* the scrollbars. The scrollbars are only visible when you have zoomed in on an area. — See “The Scrollbars” on page 43.

15. The Show available window

In the **Show available window** you see all components in the project with available symbols (unused functions). Read more about this in “The Show available window” on page 234.

16. Available symbols for the selected symbol in the Show available window

When you have selected a component in the **Show available window**, you can see the available symbols for the component here.

17. Chapter tabs

When you click on a chapter tab, you jump to the first page in the selected chapter. — See “Chapters” on page 504.

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 PAN	P. 42
— HOME VIEW AND REDRAW	P. 44
— SET USER HOME VIEW	P. 45
— SHOW EXPANDED DRAWING AREA	P. 46
— THE EXPLORER WINDOW	P. 47
— THE SURVEY WINDOW	P. 46
— SNAP	P. 47
— GRID	P. 47
— THE CROSS HAIRS	P. 47
— DIRECT ACCESS TO MENUS AND TABS	P. 48
— GENERAL PROTECTION ERROR	P. 49



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 788. THERE IT IS ALSO DESCRIBED HOW TO DEFINE YOUR OWN SHORT-CUT KEYS.

ZOOM, SCROLL AND PAN

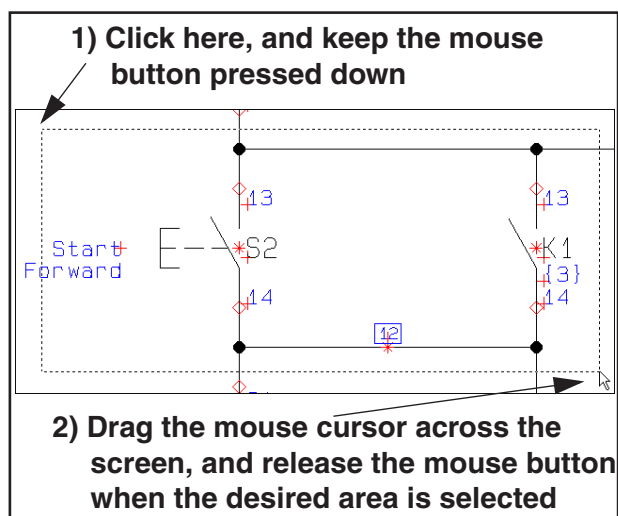
In **Automation** you can specify which part of the page you wish to see on the screen.

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.



1) Click here, and keep the mouse button pressed down

2) Drag the mouse cursor across the screen, and release the mouse button when the desired area is selected

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>).

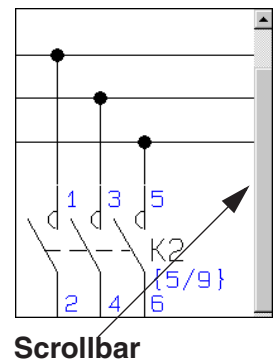
— Please also see “Zoom and Scroll for mouse with a wheel” on page 43.

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+RightArrow>.

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 836.



Zoom and Scroll for mouse with a wheel

If you have a mouse with a wheel between the left and right mouse buttons, the wheel can also be used for zooming and scrolling.

Exactly how it works, is specified under **Settings => Pointer/Screen**.

Mouse wheel set to Scroll

If **Settings => Pointer/Screen => Mouse wheel** is set to *Scroll*, the mouse wheel works like this:

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

Mouse wheel set to Zoom

If **Settings => Pointer/Screen => Mouse wheel** is set to *Zoom*, the mouse wheel works like this:

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

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 836.

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.

Pan



When you have zoomed in on an area, you can click the **Pan** button ("the hand") in the left toolbar. The cursor then changes appearance to a hand.

When you then press down the left mouse button, you can drag the page (pan), so that you see exactly the part of the page, that you wish to see.

As long as the **Pan** button is activated, you can not select any objects on the project pages. To continue working in the project, you therefore need to deactivate the **Pan** button by clicking it again, or by pressing <Esc>.

If you have a mouse with a middle mouse button, you can choose to use the middle mouse button for panning: Select **Settings => Pointer/Screen => Middle mouse button** and choose **Pan**.

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 42).



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 42).

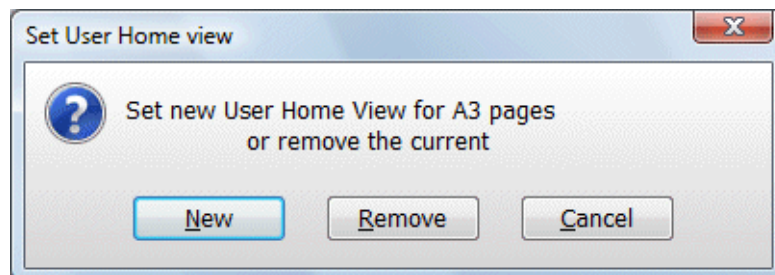
Display the full page, and not just the User Home view

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

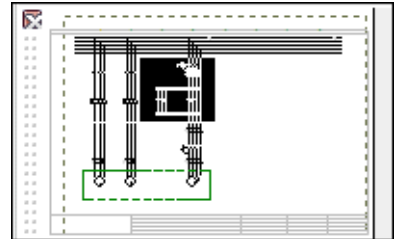
When you select **View => Clean screen**, the active project page will fill all of the program window. This makes the program toolbars and the **Explorer** window invisible.

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 26.



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 53) 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 or the Show available window” on page 27.

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 301.

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**.

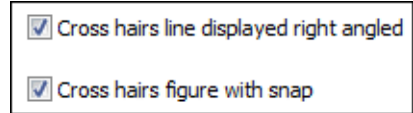
Choosing **Settings => Pointer/Screen**, you can disable the function completely, or choose to use squares instead of dots. See “Pointer/Screen” on page 836.

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 73), a line will be displayed starting from where you clicked last, pointing towards the cross hairs.

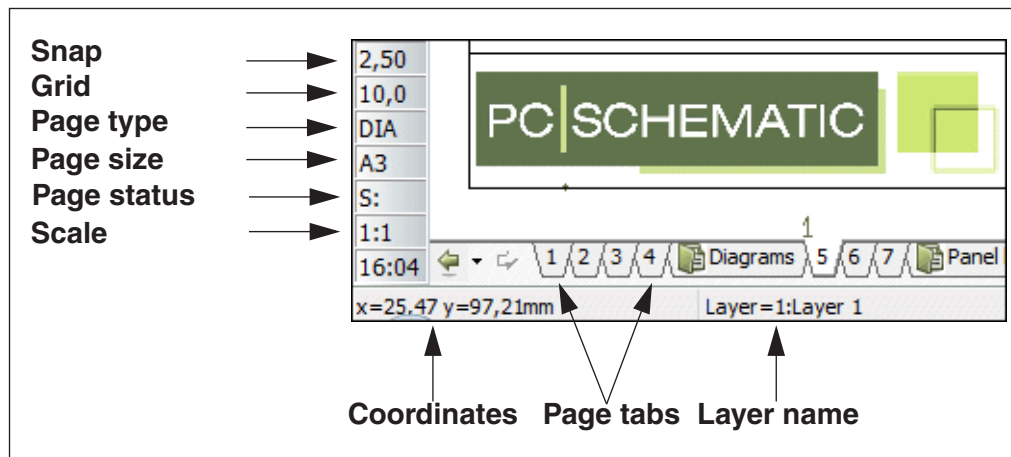
When you select **Cross hairs displayed right angled**, you will see the line that would be drawn if you clicked in this very moment. This line will not always be shown ending up in the cross hairs — due to the snap applied.

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.

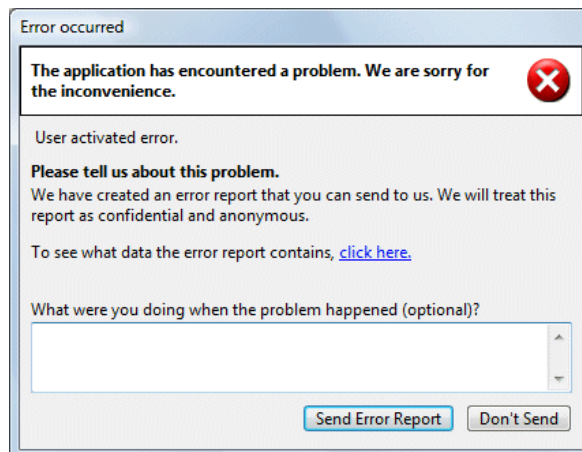


GENERAL PROTECTION ERROR

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. 53
- Transferring data between objects..... p. 62
- Align, Space and Trim functions p. 66
- Lines..... p. 72
- Arcs/Circles..... p. 100
- Texts..... p. 102
- Symbols..... p. 137
- Functions on Component level p. 228
- Areas..... p. 244
- Sub drawings..... p. 256
- Rename symbols..... p. 289
- Insert image p. 297



Read survey article

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

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. 54
— SELECTING OBJECTS	P. 55
— AUTO COMMAND CHANGE	P. 57
— HOW TO WORK WITH SELECTED OBJECTS.....	P. 58
— SHOW OBJECT BELOW CURSOR	P. 60
— THE UNDO FUNCTION	P. 60
— TRANSFERRING PROPERTIES BETWEEN OBJECTS	P. 63
— ALIGN, TRIM AND SPACE.....	P. 66
— INVISIBLE OBJECTS	P. 69

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>
(s. 137)

Texts <t>
(s. 102)

Lines <l>
(s. 72)

Circles <c>
(s. 100)

Areas <a>
(s. 244)



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 57 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 57 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.

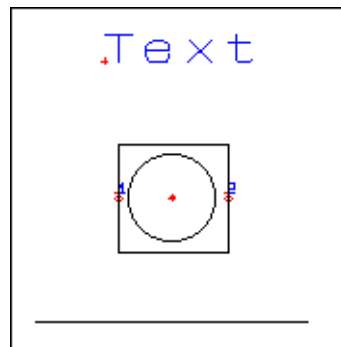
SELECTING OBJECTS

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

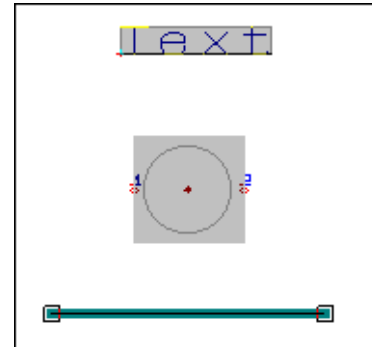
Click on the button which corresponds to the type of the object — for instance the Texts or the Symbols button. Make sure that the *Pencil* button is not selected; click on it or use <Esc>. Click on the object you wish to select.

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 873) 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 25 for further details.

Selecting more objects of the same kind in one window

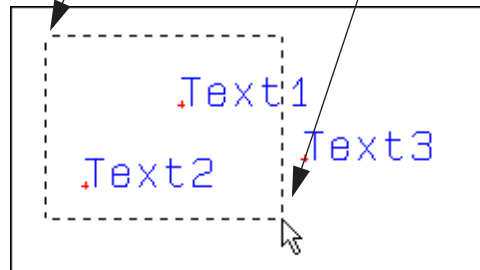


You can select more than one object of the same type at the same time. If for instance you want to select all texts in an area, click on the **Texts** button and select the desired area with the mouse. Do this by clicking in one corner of the area, and then, still pressing down the mouse button, drag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

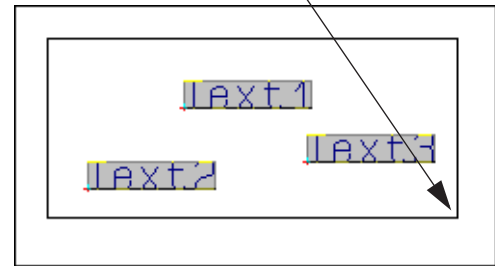
pointer is located now. When all the desired objects are inside the marquee rectangle, release the mouse button. The texts in the area have now been selected.

1) Click here

2) Press down the mouse button and *drag*



3) Place the cursor here and release the mouse button



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



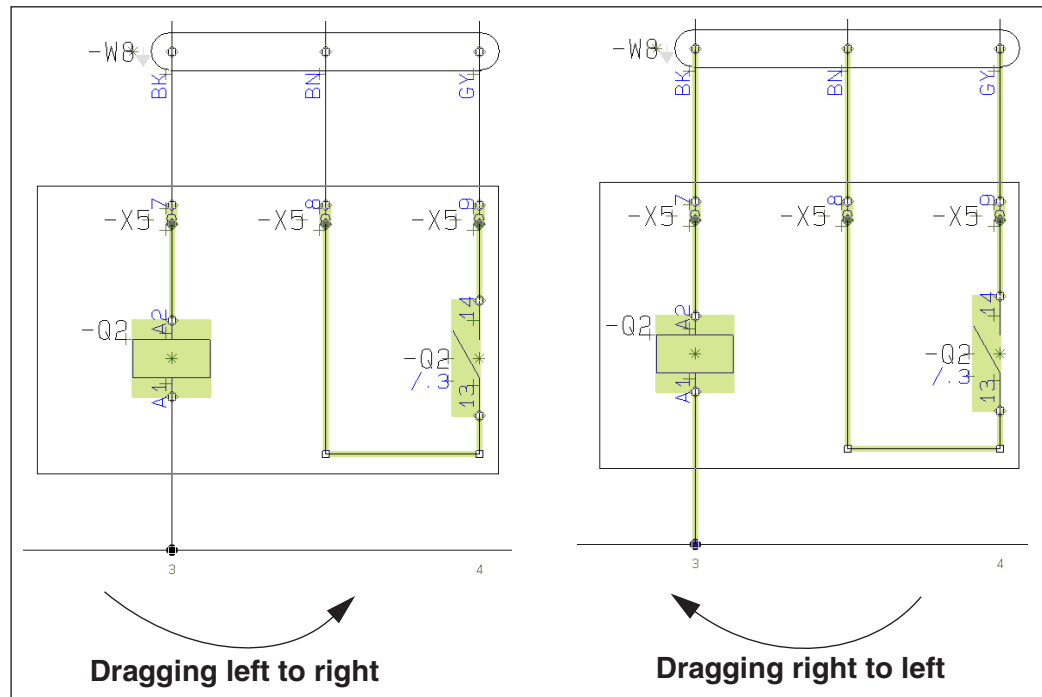
Click on the **Area** button. Drag a marquee rectangle around the objects as described in “Selecting more objects of the same kind in one window” on page 55. Hereby all object types inside the window are selected. If you wish to deselect one of the selected objects, press the <Ctrl> key and click on 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 244.

Dragging the marquee rectangle left to right — or right to left

How many objects you select, when you drag the marquee rectangle, depends on whether you drag the mouse *left to right* or *right to left*.

When you select objects by dragging left to right, you only select objects where all parts of the object is inside the marquee rectangle and objects having their reference points inside the marquee rectangle.

When you select objects by dragging right to left, all objects with just a fraction inside the marquee rectangle are selected.



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 481, and read more about layers in general in “Layers, Elevation and Scaling” on page 477.

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** is 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 60 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 55, 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 72, “Arcs/Circles” on page 100, “Texts” on page 102 and “Symbols” on page 137.

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 25.
- 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.

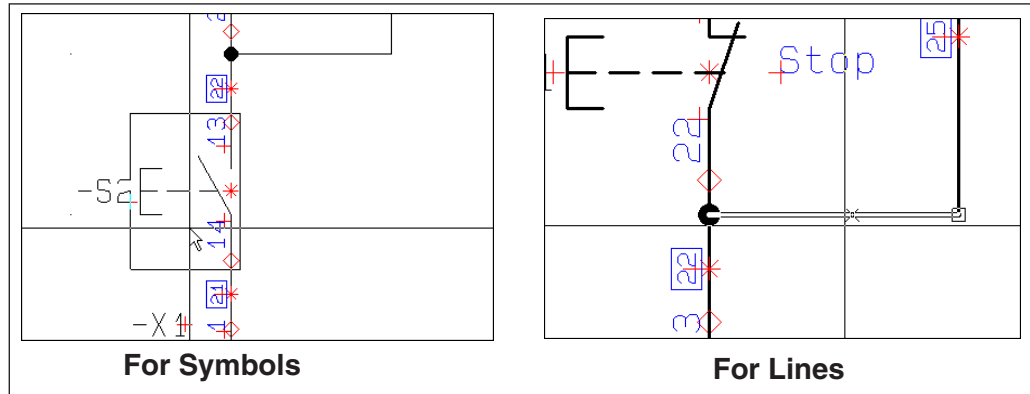


Rotating areas

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 253.

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 57 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.

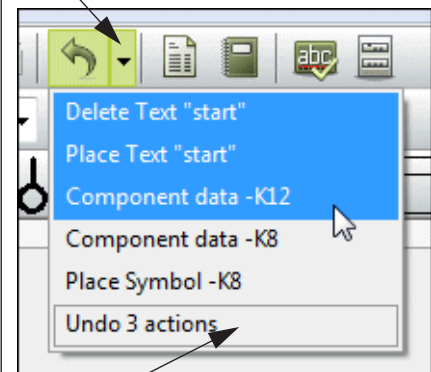
If you let the mouse cursor rest above the **Undo** button, the operation that you can undo, is displayed as a hint.

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.

Click here, and the Undo list is displayed



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

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 850.

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.

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. 62
— HOW TO TRANSFER OBJECT PROPERTIES BETWEEN OBJECTS.....	P. 63
— TRANSFERRING OBJECT PROPERTIES TO MORE THAN ONE OBJECT.....	P. 63

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>).

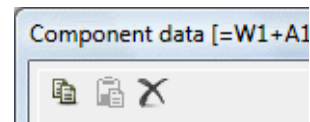
COPYING THE COMPONENT DATA



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 55.

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 62. 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 72. — 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 696.
TEXT PROPERTIES	ALL INFORMATION IN THE DIALOG BOX TEXT PROPERTIES P. 122. 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. 100.
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.

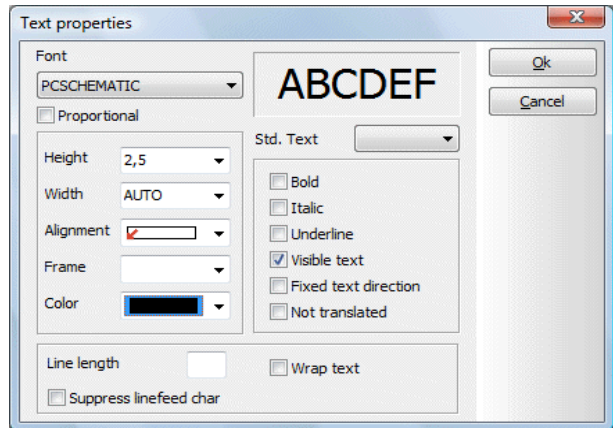
Place Text1



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.

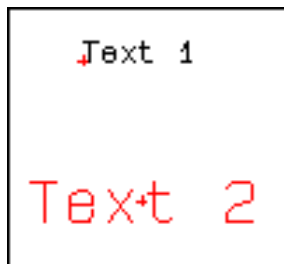
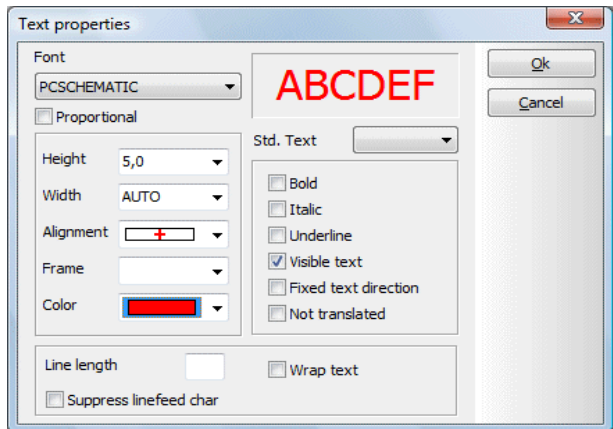


Place Text2



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

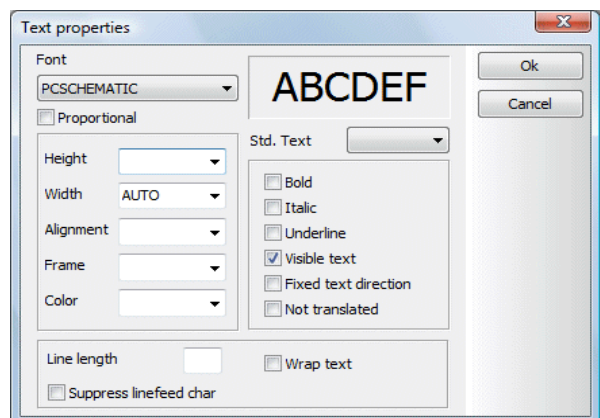


Select both of the texts, click on the **Copy properties** button and then on **Text properties**.

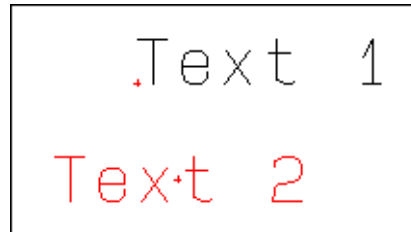
In the dialog box you now see which text properties the selected texts have in common — in this case both has the **Width** *AUTO*.

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.

SEE VIDEO ON **ALIGNING AND SPACING FUNCTIONS**



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

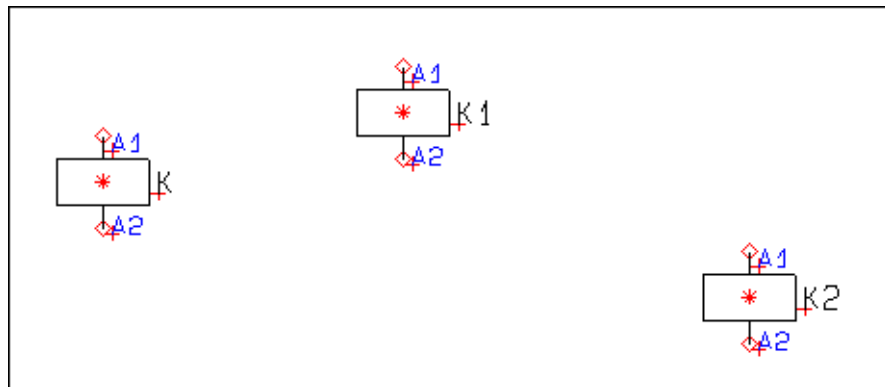
THIS CHAPTER DESCRIBES	
— THE ALIGN FUNCTION	P. 66
— THE SPACE FUNCTION	P. 67

The Align function

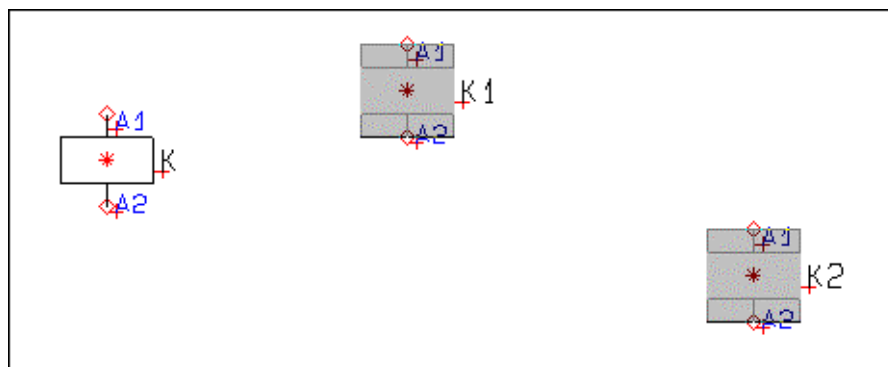
As you will see, it is possible to align objects in **Automation**.

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:



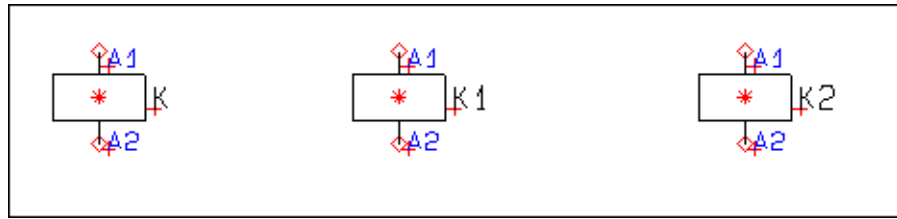
Make sure that the **Symbols** button is activated, then click on the right most symbol. Now press down <Ctrl> while clicking on the symbol in the middle. (Alternatively, you could drag a window around the two symbols). These two symbols have now been selected:



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**.

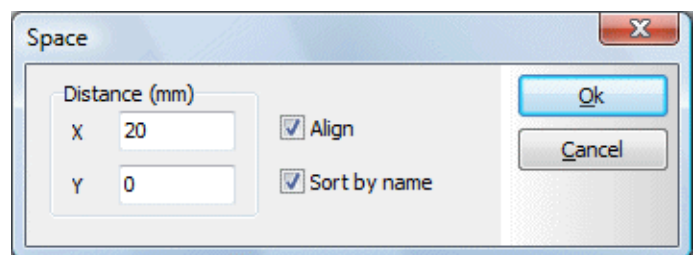
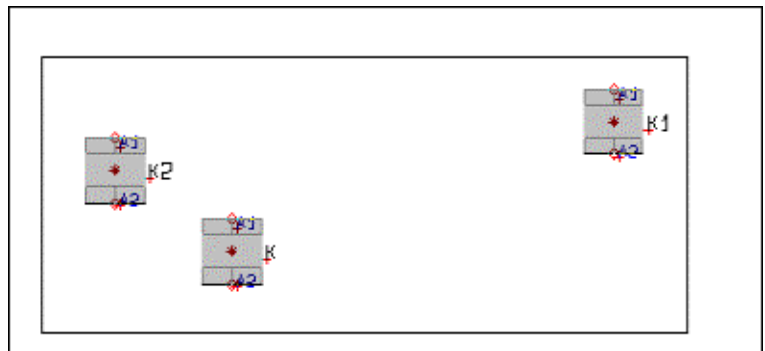
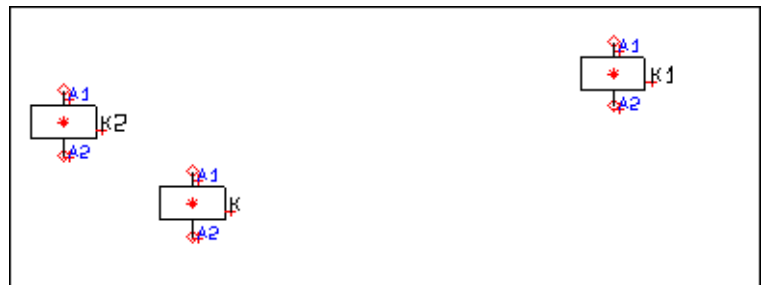
Please note the guiding text in the bottom right-hand corner of the screen.

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

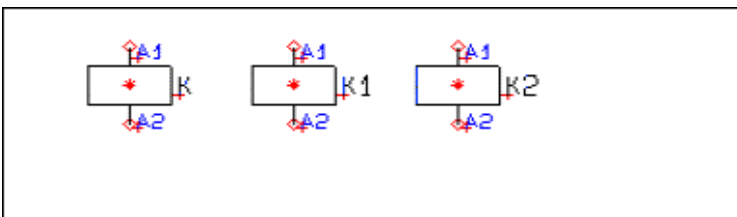
Click where you want the first of the symbols to be located.

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 556.

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. 69
— MAKING LINES INVISIBLE	P. 70
— MAKING TEXTS INVISIBLE	P. 70
— MAKING CIRCLES/ARCS INVISIBLE	P. 70
— MAKING AREAS INVISIBLE	P. 70
— SUB DRAWINGS AND INVISIBLE OBJECTS	P. 71

Making symbols invisible

You can make symbols invisible like this:

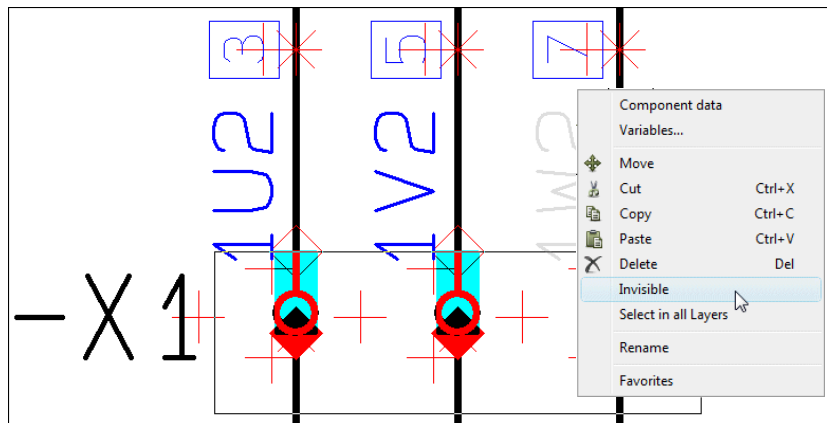


- a) Via the **Component data** dialog box: Click on the **Symbols** button, double-click on the symbol, which opens the dialog box **Component data**, where you remove the check mark in front of **Visible**, and click **OK**.
Or:



- 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 161 also.

Making lines invisible



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

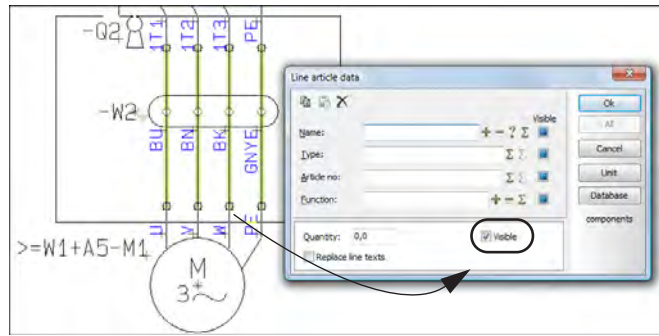
Then do the following:

- 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 125.

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.



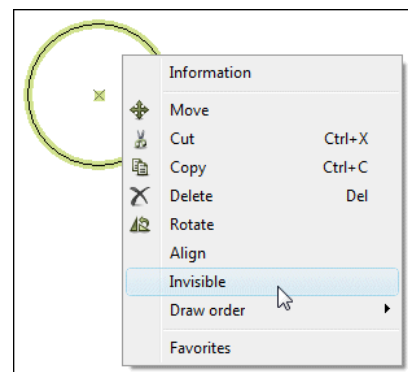
To make the text visible again, you must then click on the **Text properties** button, select **Visible text**, and click **OK**.

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 244.

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 69.

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 256.

LINES



When you work with lines, you always start by clicking the **Lines** button — or by using the short-cut key <l>. — See “Auto command change” on page 57 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 <l> key when you are working with Lines. Pressing the <Ins> or the <l> key activates/deactivates the Pencil.

THIS CHAPTER DESCRIBES	
— THE TWO TYPES OF LINES IN Automation	P. 73
— THE FUNCTIONS IN THE LINES TOOLBAR	P. 73
— DISCONNECTING DRAWN LINES	P. 77
— LINES IN THE PICKMENU	P. 78
— CONDUCTING (ELECTRICAL) LINES.....	P. 78
— ARTICLE DATA ON LINES	P. 79
— AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS	P. 81
— JUMPER LINKS.....	P. 86
— AUTOMATIC LINE DRAWING / ROUTER.....	P. 89
— EXTENDED LINES.....	P. 90
— FOLLOW CONNECTED LINES	P. 90
— THE TRIM LINE FUNCTION	P. 92
— DRAWING BY MEASURES.....	P. 93
— MOUNTING CORRECT DRAWING.....	P. 95

— See how to move, copy and delete lines in “Working with selected objects” on page 58, and see how to transfer data between lines in “Transferring Data / Properties between Objects” on page 62.



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 89.



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 81.

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 78.



JUMPER LINKS

IT IS ALSO POSSIBLE TO DRAW JUMPER LINKS, AS DESCRIBED IN “JUMPER LINKS” ON PAGE 86.

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 78.

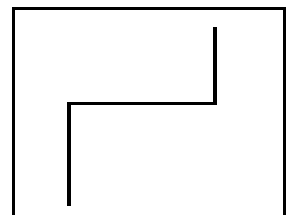
Straight lines

When you are drawing **Straight lines**, the lines are automatically drawn either right-angled or diagonally, then see “Pointer/Screen” on page 836).



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.

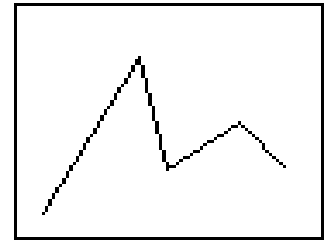


Click the **Slanted lines** button, and draw a line like the one you see here.

The line will become as you draw it, depending on the snap you are working in. — Read more about snap in “Snap” on page 47.

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.

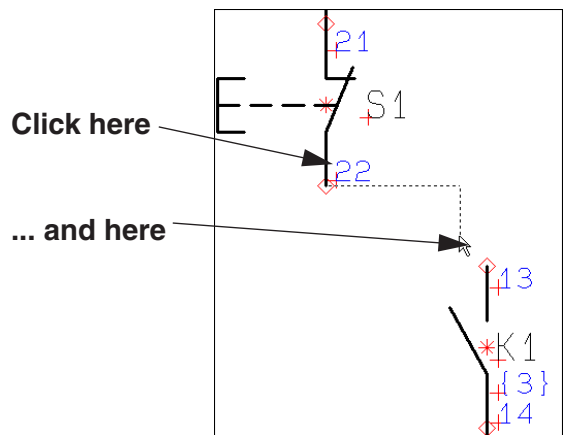


Angled lines



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 95 for further details.



Curved lines

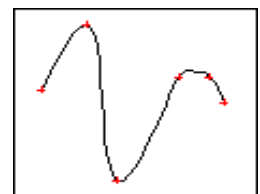


Click the **Curved lines** button.

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

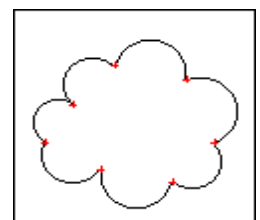


Click the **Round lines** button.

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.

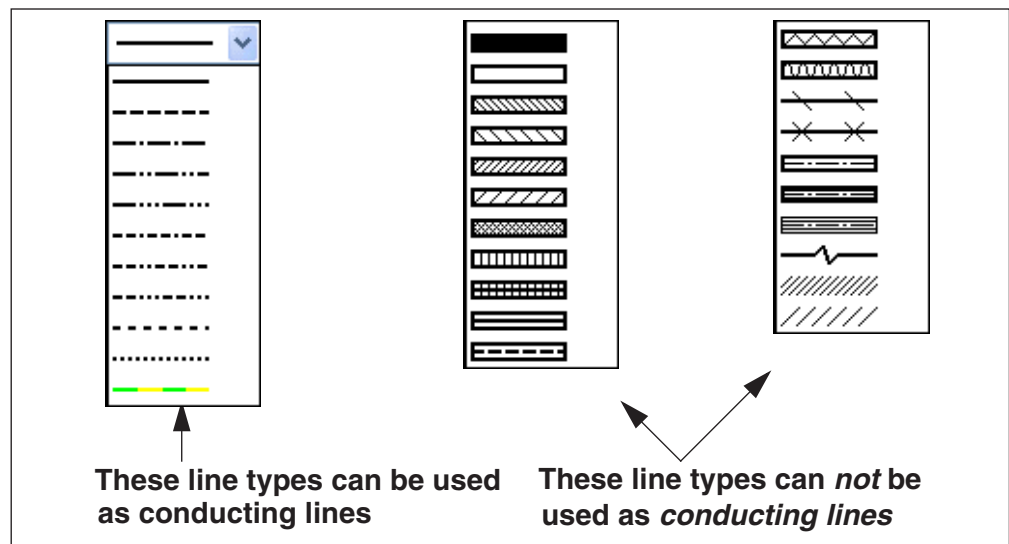
Filled areas



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.



SHORT CUT FOR CHANGE OF LINE TYPE

IN ORDER TO QUICKLY CHANGE LINE TYPE FOR A DRAWN LINE, JUST SELECT THE LINE AND PRESS <SPACEBAR>. HEREBY YOU SWITCH DOWN TO THE NEXT AVAILABLE LINE TYPE IN THE LIST. IF YOU PRESS <SHIFT> WHILE PRESSING <SPACEBAR>, YOU MOVE UPWARDS IN THE LIST OF LINE TYPES. YOU CAN ONLY SWITCH BETWEEN SIMILAR LINE TYPES — MEANING LINE TYPES THAT CAN BE USED FOR CONDUCTING AND NON-CONDUCTING LINES RESPECTIVELY.

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.

Line distance



Line width

Extend line



See how to use the **Extend line** button in “Extended lines” on page 90.

Conducting lines



See how to use the **Conducting lines** button in “The two types of lines in Automation” on page 73.

Jumper links



See how to work with **Jumper links** in “Jumper links” on page 86.

Mounting correct drawing



See how to apply **Mounting correct drawing** in “Mounting correct drawing” on page 95.

Without wire number



When you use the **Without wire number** button, you can deselect wire numbering for individual lines in the project. Read more about this in “Deselecting wire numbering for individual lines” on page 390.

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 90.

Insert potential



See how to use the **Insert potential button** for automatic insertion of electrical potentials in “Automatic insertion of electrical potentials” on page 81.

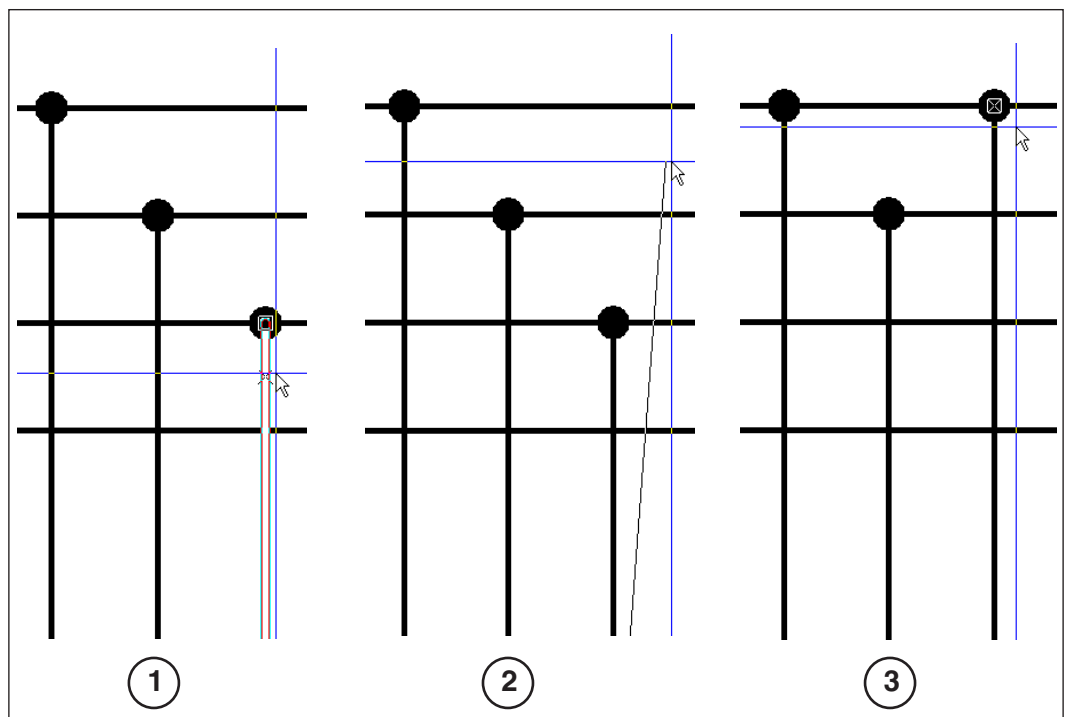
DISCONNECTING DRAWN LINES

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

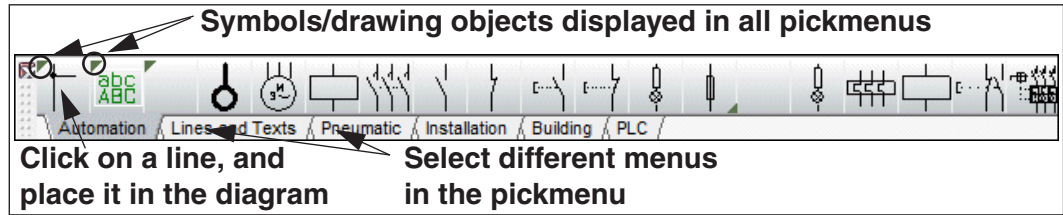
To move the end point for a drawn line, do this:

- 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 62.

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 140 and “Adapting the Pickmenu” on page 178.

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.



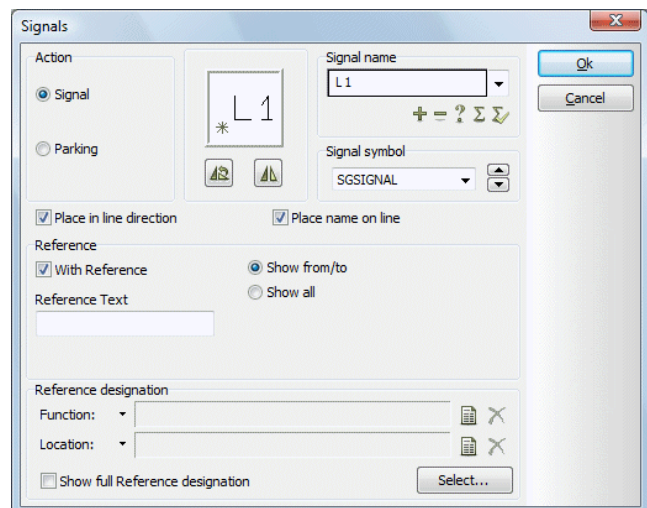
To specify that a line has to be conducting, you must activate the **Conducting lines** button.

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 203, and about reference designations on signal symbols in “Variant names on signal symbols” on page 209.



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.

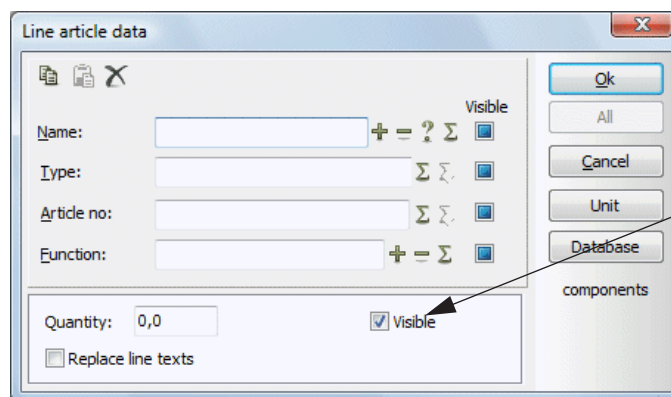
Transferring data for conducting lines

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 62.

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**:



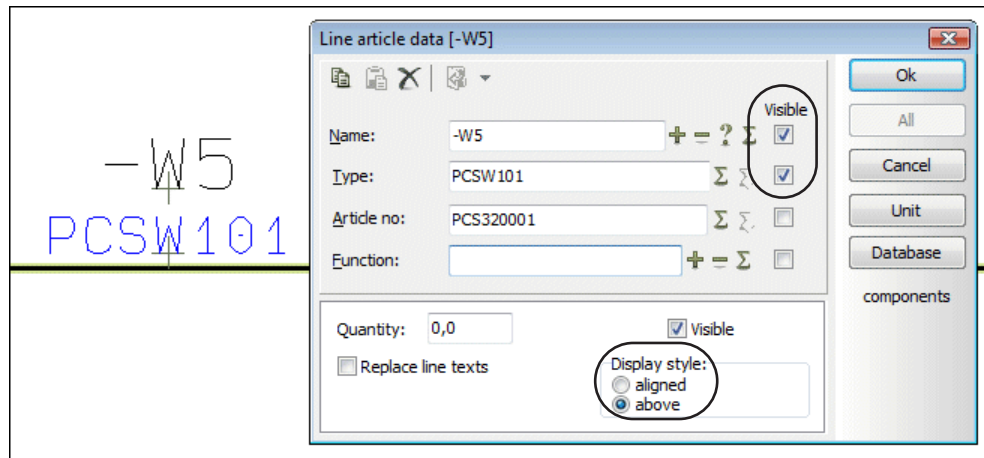
Specify if the line is *visible*

Here you can for instance click on **Database**, to fetch article data from the database — see “**Using the database**” on page 571, in particular “**Using the database for Lines**” on page 571.

— Please note that you can also attach article data to cable symbols, as described in “Working with cable symbols” on page 218.

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 **Name** and **Type** fields 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 specify which article data to display for lines, as a general setting for the project. To do this, select **Settings => Text/Symbol defaults** and then choose **Line** under **Text defaults**. Read more about this in “Texts displayed in the project (general setting)” on page 127.

You can also specify whether to place the texts **Aligned** or **Above** (as in the figure above).

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 855.

AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS

THIS CHAPTER DESCRIBES	
— INSERTING ELECTRICAL POTENTIALS AUTOMATICALLY	P. 81
— PLACING ELECTRICAL POTENTIALS ON THE UPPER PART OF THE PAGE.....	P. 82
— PLACING ELECTRICAL POTENTIALS ON THE LOWER PART OF THE PAGE	P. 82
— POSITION FOR THE ELECTRICAL POTENTIALS.....	P. 83
— REFERENCE TEXTS FOR AUTOMATICALLY PLACED ELECTRICAL POTENTIALS....	P. 83
— REFERENCE DESIGNATIONS FOR AUT. PLACED ELECTRICAL POTENTIALS	P. 83
— SETTINGS FOR AUTOMATICALLY PLACED ELECTRICAL POTENTIALS	P. 83
— INSERTING SIGNAL NAMES IN THE SIGNAL NAME LIST	P. 85

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.

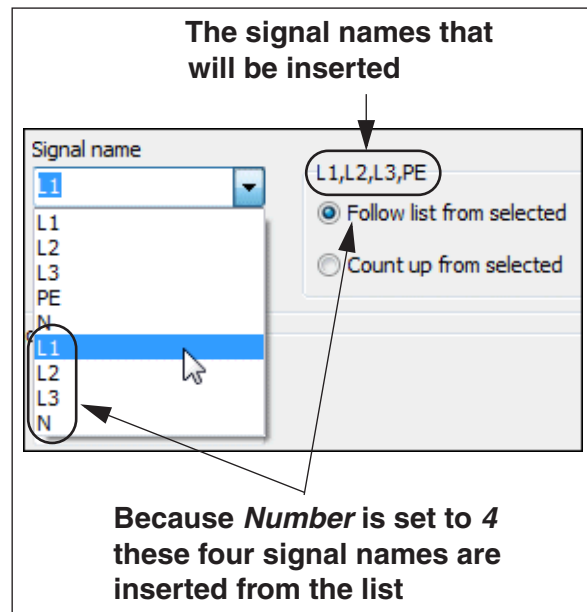
Subsequently, you can specify which signal names to insert.

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 83, “Reference text for automatically placed electrical potentials” on page 83, “Reference designations for automatically placed electrical potentials” on page 83, “Settings for automatically placed electrical potentials” on page 83 and “Signal names on the Signal name list - for automatically inserted electrical potentials” on page 85.

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 82.

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 83, “Reference text for automatically placed electrical potentials” on page 83, “Reference designations

for automatically placed electrical potentials” on page 83, “Settings for automatically placed electrical potentials” on page 83 and “Signal names on the Signal name list - for automatically inserted electrical potentials” on page 85.

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 83.

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 350.

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 351.



THE SELECTED LINE TYPE IS USED

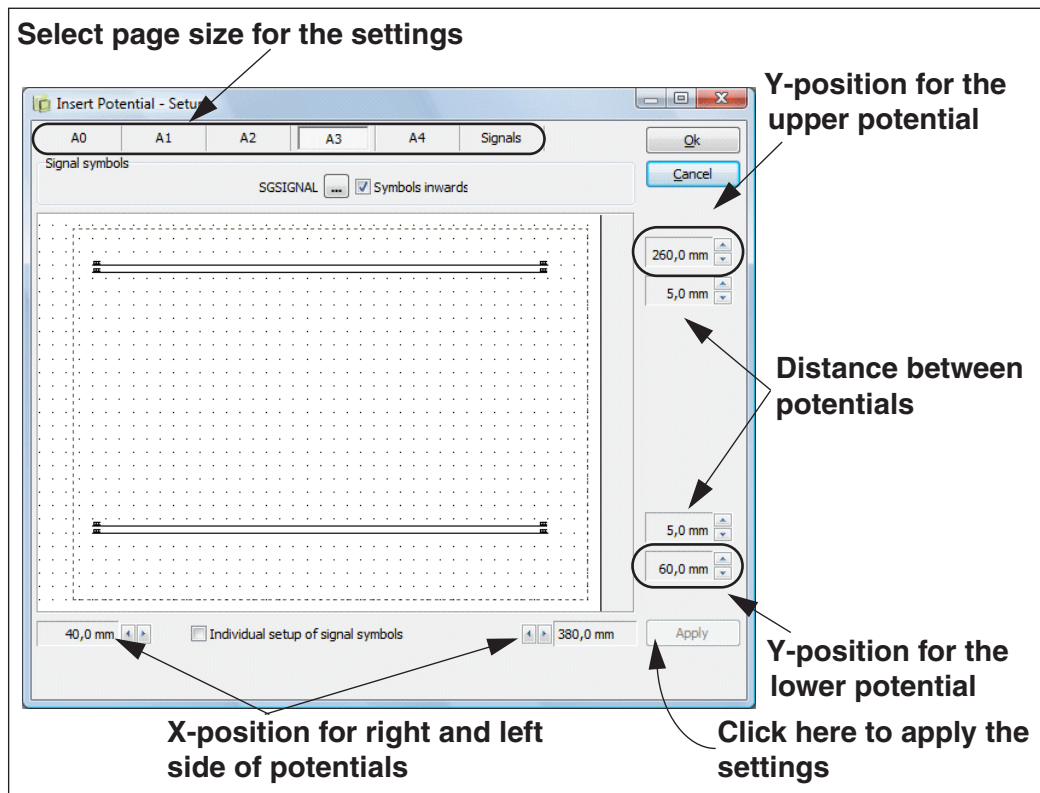
WHEN INSERTING ELECTRICAL POTENTIALS AUTOMATICALLY, THE LINE TYPE YOU HAVE SELECTED BEFORE CHOOSING **INSERT => INSERT POTENTIAL** IS USED FOR DRAWING THE POTENTIALS.

— SEE “LINE TYPE: T” ON PAGE 75.

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.

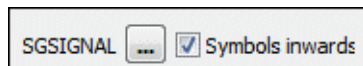
This can be done by choosing **Insert => Insert Potential**, and then click on **Setup**. You then enter the dialog box **Insert potential - Setup**:



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

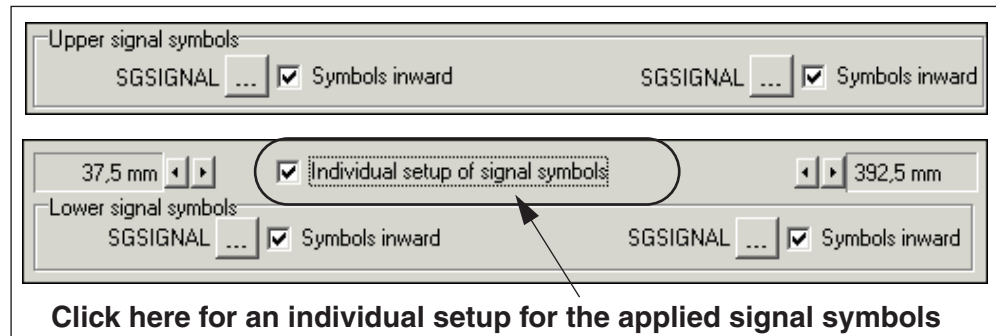


To select which signal symbol to apply, click the **SGSIGNAL** button, and you enter the **Symbol menu**, where you select the symbol.

Individual setup of signal symbols

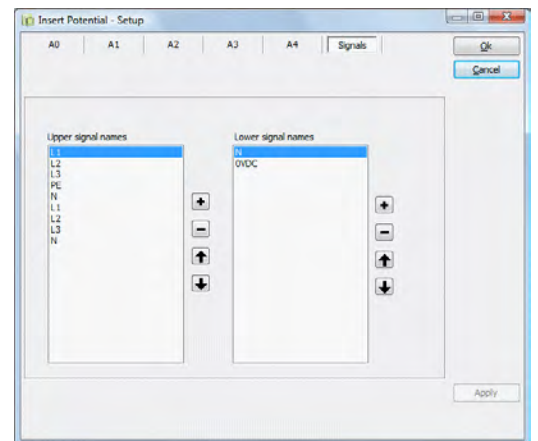
If you wish to use an individual setup for the applied signal symbols, click on **Individual setup of signal symbols**.

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 82 —, 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. 86
— DRAWING JUMPER LINKS	P. 86
— JUMPER LINKS IN LISTS AND TERMINAL PLANS	P. 87

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.

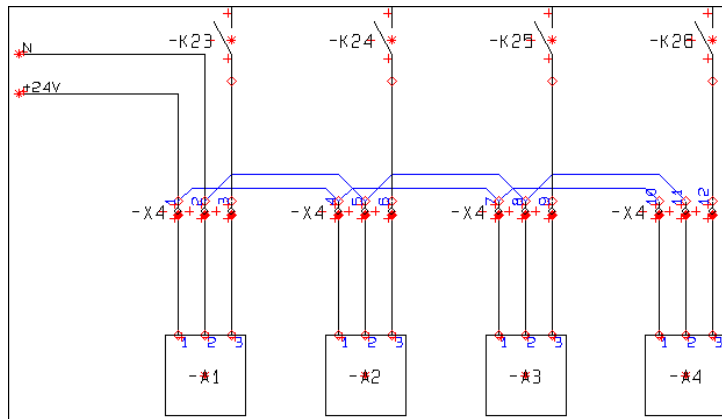
Drawing jumper links

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.

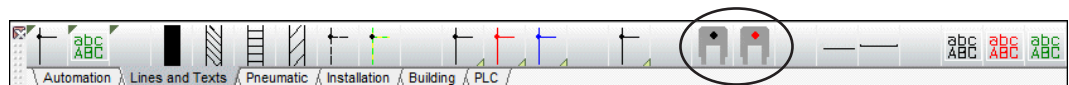
Jumper links can be drawn from all types of connection points. Jumper links can not start on the middle of another line, or on the middle of another jumper link.



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 73.

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 63.

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.

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

Article data for jumper links in parts and components lists

You can also get article data from jumper links in the parts and components lists.

— See “Article data on lines” on page 79 and “Which data to include” on page 544.

CORRECT AMOUNT OF JUMPER LINKS IN PARTS LISTS

In the **PINDATA** field of the database, you can specify how many *connections* a jumper link with a specific article number has. When a jumper link for instance has two connections (three terminals), write #2 in the **PINDATA** field. For a jumper link

for one connection (two terminals), write #1. If you do not specify any value, it is set to 1 by default.

For jumper links, you can use the field **UnitPrPack** to specify the number of possible connections. This makes it possible to specify the needed amount of jumper units in the parts list. (The field **UnitPrPack** is specified in the database under **Settings => Database => Database Setup => Component data** next to the field **Units/Pack**).

If the number of connections is not specified for the jumper link in the database, it will appear with the quantity 1 in the parts list. — Both when article data is specified for the jumper link and when it is not.

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.

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 790.

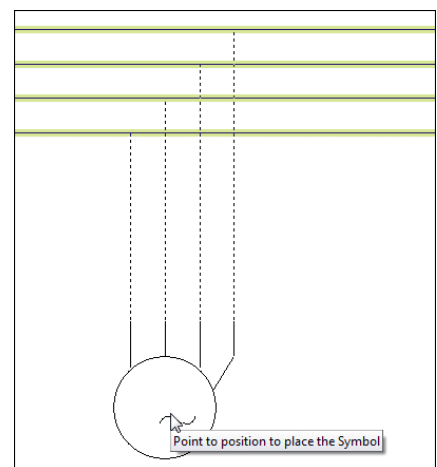
Automatic line drawing when placing symbols



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.

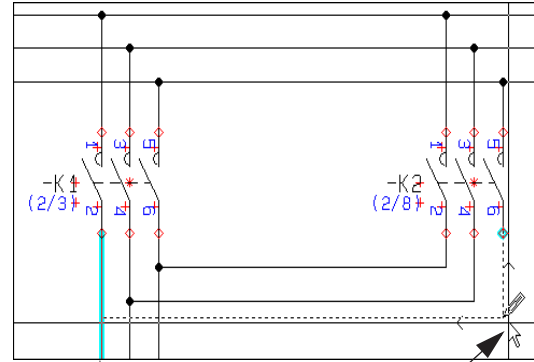


The router for lines

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 <1/2>	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

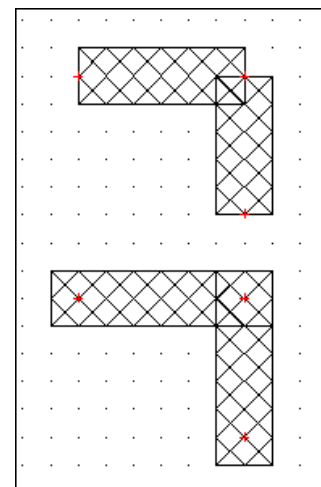


On all types of lines, you can determine that they shall be extended. Select the **Extend line** button on the toolbar. The line will now start half a line width before its starting point, and end half a line width after its ending point.

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.



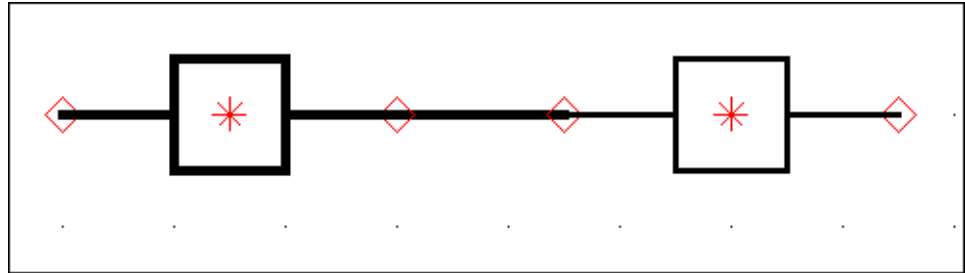
FOLLOW CONNECTED LINES

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 852, especially “Activate Follow connected on symbols” on page 853.

THE TRIM LINE FUNCTION

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

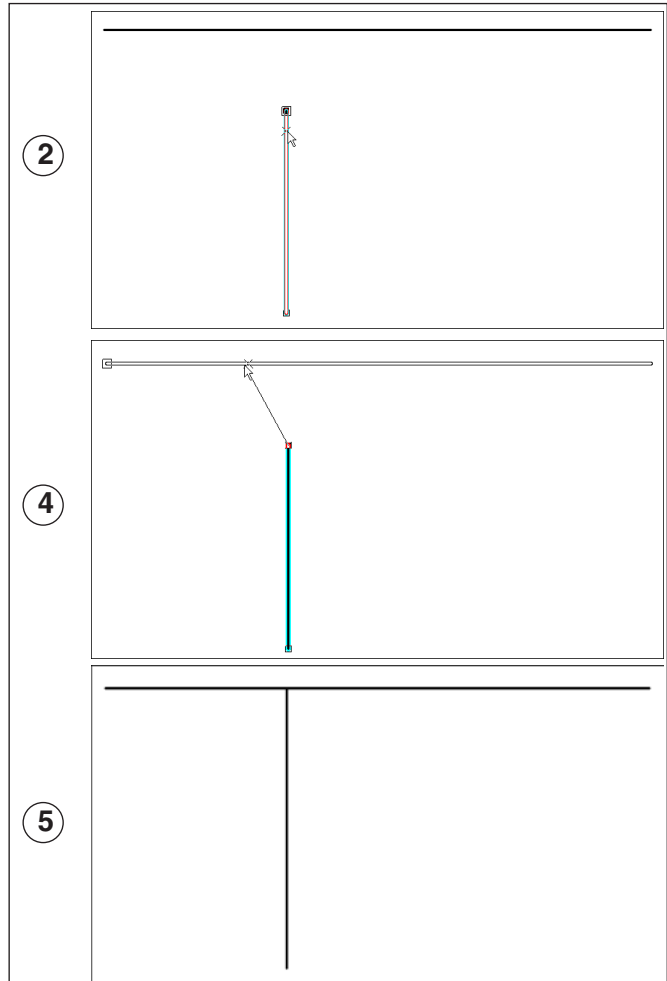
- 1) To extend a line, so that it meets another line.
- 2) To truncate a line, so that it meets another line.

Extend a line using Trim line

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

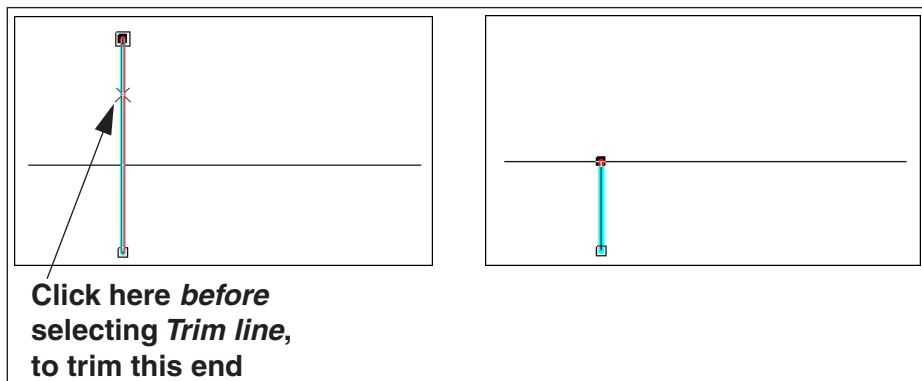


- 1) Click the **Lines** button, and press <Esc> to disable the Pencil.
- 2) Select the vertical line by clicking on it.
- 3) Choose **Edit => Trim line**.
- 4) A thin line starting from the selected symbol and ending in the cross hairs is now displayed: Click on the horizontal line.
- 5) The vertical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

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

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

Click on the X-Y coordinate field in the status line, or use the short-cut <Ctrl+i>. You now see a dialog box for typing in coordinates.

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 coordinates, 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 coordinate field — or press <Ctrl+i> — and choose **Absolute**.

Type in the X and Y coordinates, 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 836).

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

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

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

Click on the X-Y coordinate 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 coordinate 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 coordinate 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.

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. 95
— SETTINGS FOR MOUNTING CORRECT DRAWING.....	P. 96
— DESIGN CHECK FOR MOUNTING CORRECT DRAWING	P. 97
— SIGNAL NAMES IN MOUNTING CORRECT DRAWING	P. 98
— SUB DRAWINGS AND MOUNTING CORRECT DRAWING	P. 98
— CHANGING TO MOUNTING CORRECT DRAWING	P. 98
— CHANGING TO NOT MOUNTING CORRECT DRAWING	P. 98
— CONVERTING PROJECTS FROM VERSION 8.0 AND PREVIOUS	P. 99



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 98.

On mounting correct drawing

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

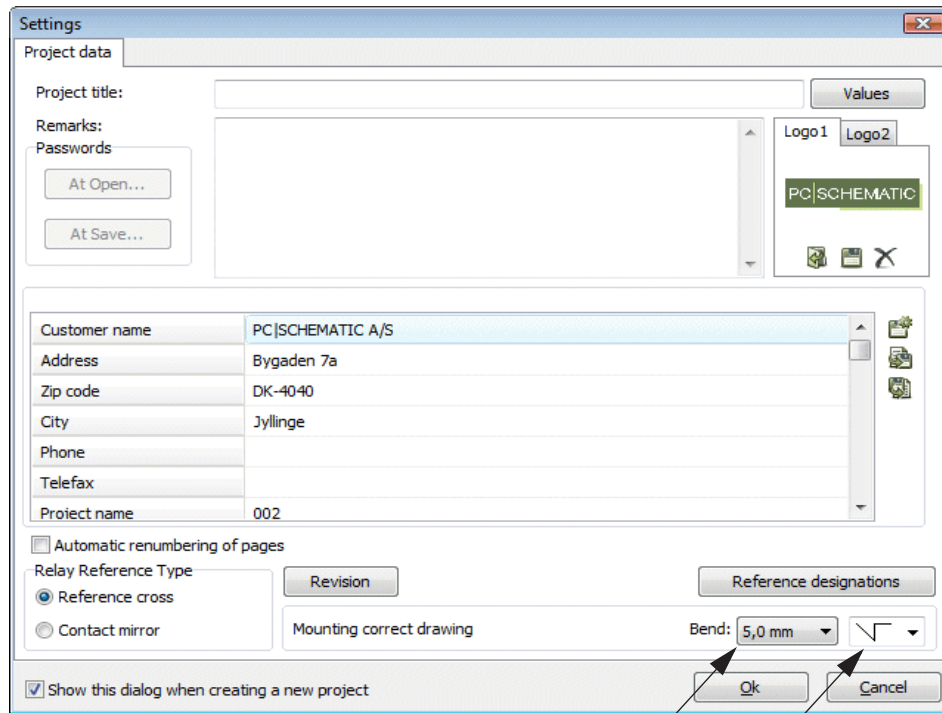
- In **Automation** you can mix mounting correct drawing and normal drawing mode.
- You switch between mounting correct drawing and normal drawing mode by clicking the **Mounting correct drawing** button.
- Where a connection dot automatically occur in normal diagram drawings, a *branch figure* is inserted in mounting correct drawing mode.
- You can start and end lines on other lines — with automatic insertion of branch figures.
- You can also use automatic line drawing / the router for mounting correct drawn lines, starting/ending on other lines. — See “Automatic line drawing / Router” on page 89.
- 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 98.
- 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 97.

Settings for Mounting correct drawing

When you want to specify how to work with mounting correct drawings, choose **Settings => Project data**:



Bend size **Branch figure**

Here you can specify the bend size and which branch figure to suggest as default.

Bend

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

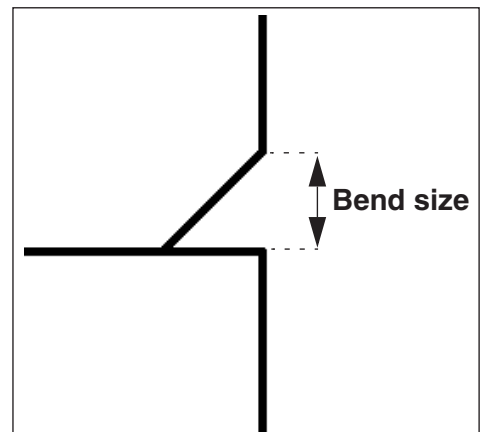
In the **Bend** field you can change the size of the bend by clicking the down-arrow, and selecting a bend size. In the figure to the right, you can see how the size of the bend is specified.

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*.

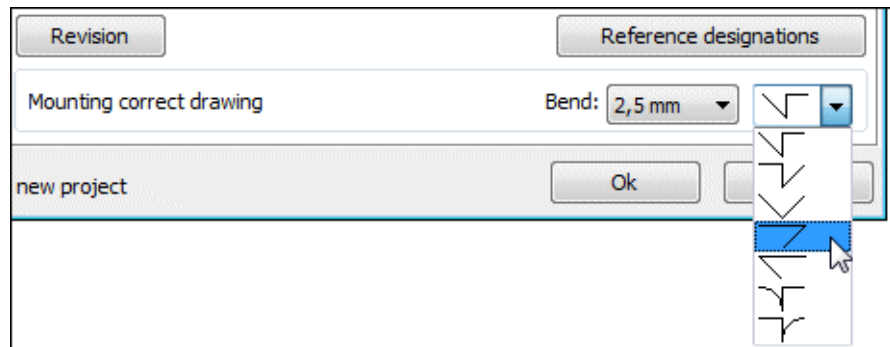
CHANGING THE APPLIED BEND SIZE IN A PROJECT

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:

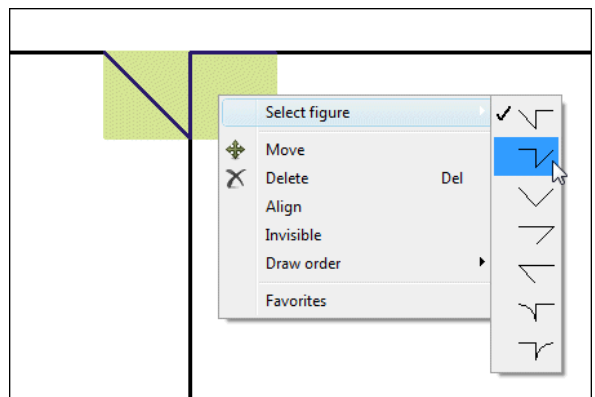


When drawing mounting correct, the program will automatically place the latest applied branch figure. When pressing the <Spacebar> repeatedly before drawing further, you can change to one of the other branch figures.

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.



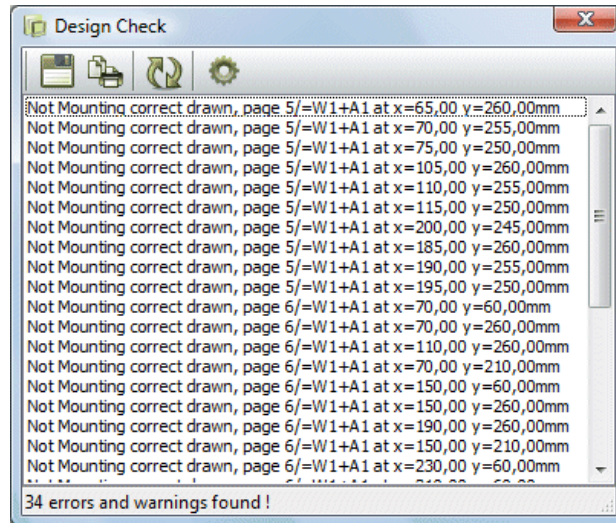
Design check for Mounting correct drawing

To locate what is not mounting correct in the project, do this:



- 1) Click the **Design check** button — or choose **Functions => Design check**.
- 2) In the dialog box **Design check**: Click the **Design check settings** button.
- 3) In the dialog box **Design check settings**: Select **Mounting correct** and click **OK**.

- 4) In the **Design check** dialog box you now get a list of all the parts of the project, that are not drawn mounting correct:



Based on this list, you can correct the project and make sure it is mounting correct. (Please remember that you can click each line in the list, to see where the project is not mounting correct).

- 5) — Read more about design check in “Design Check” on page 320.

Signal names in Mounting correct drawings

— See “Variant names on signal symbols” on page 209.

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 not drawn mounting correct, can get branch figures inserted in stead of the connection by selecting **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 in stead of branches by selecting **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 has been implemented. 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

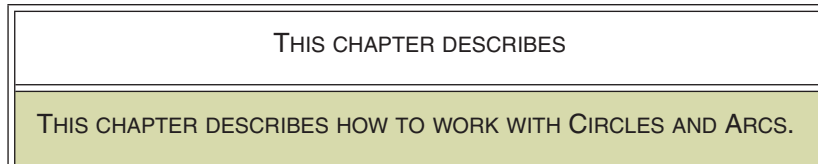
When converting mounting correct drawn projects from version 8.0 and previous, wire numbers will be moved, so that they uniquely identify which lines they are attached to.

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 57 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.

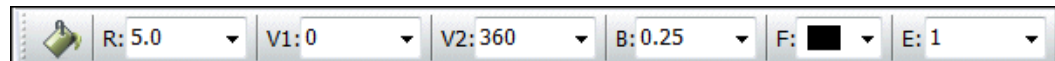


— See how to move, copy, rotate and delete circles and arcs in the section “Working with selected objects” on page 58, and see how to transfer data between circles/arcs in “Transferring Data / Properties between Objects” on page 62.

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.

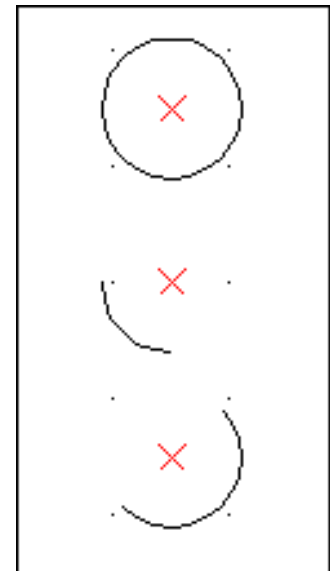
To draw a half circle **V1** can be 45 while **V2** is 225.



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

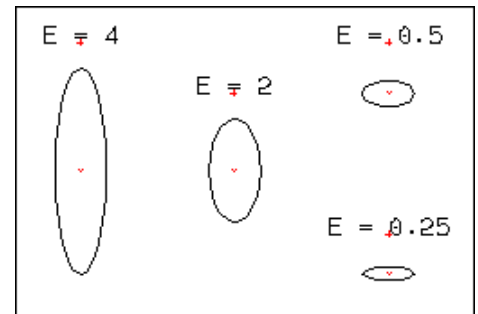
To get fast access to drawing arcs with predefined properties, you can place arcs with the desired properties in the pickmenu at the upper part of the screen. This is done as described for lines in “Lines in the Pickmenu” on page 78.

Ellipses

Using the circle command also enables you to draw ellipses. The last field **E**: in the tool-bar 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.

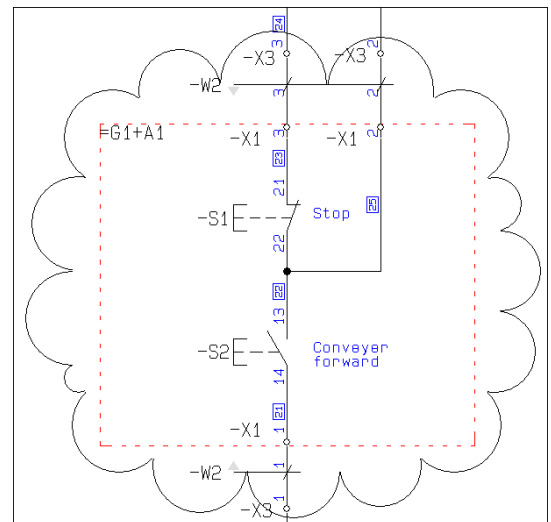
Invisible arcs / circles

— See “Making circles/arcs invisible” on page 70.

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.
- 4) You now enter the dialog box **Enter text**: Type in a text, which will be placed in the middle of the cloud, and click **OK** — or press <Esc> if you do not wish to place a text in the cloud.
- 5) Draw a new cloud, or press <Esc> when you do not wish to draw more clouds.



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 57 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. 102
— PROJECT AND PAGE DATA.....	P. 114
— GENERAL TEXT FUNCTIONS	P. 116
— LOCKING TEXT TYPES.....	P. 119
— COUNTING ON TEXTS	P. 120
— THE APPEARANCE OF TEXTS / TEXT PROPERTIES	P. 122
— WHICH TEXTS ARE DISPLAYED IN PROJECTS	P. 127
— THE READING DIRECTION FOR TEXTS	P. 130
— TEXT LINKS	P. 132

— See how to move, copy, delete and rotate texts in “Working with selected objects” on page 58, and see how to transfer data between texts in “Transferring Data / Properties between Objects” on page 62.

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.	102
SYMBOL TEXTS	TEXTS, THAT ARE ATTACHED TO EACH SYMBOL, CONTAINING INFORMATION ON THE COMPONENT THAT IS COVERED BY THE SYMBOL.	107
CONNECTION POINT TEXTS	TEXTS, THAT ARE ATTACHED TO EACH OF A SYMBOLS CONNECTION POINTS.	109
DATAFIELDS	TEXT FIELDS THAT ARE FILLED OUT AUTOMATICALLY.	110

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 696) or in drawing headers/formulas.

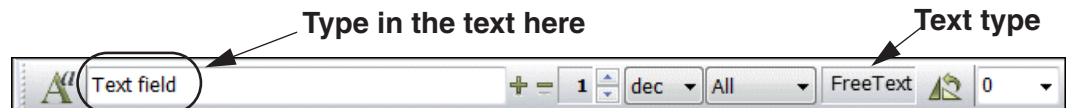
The free texts can not be transferred to project lists — as opposed to the other types of texts.

If a free text is a part of a symbol definition — see “Creating Symbols” on page 696 —, 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.

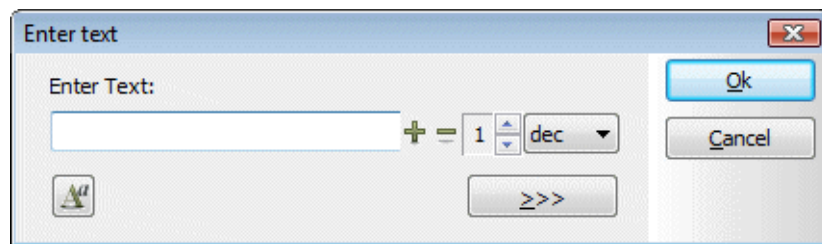


Type in the text as displayed above and press <Enter>. Please note that the toolbar informs you that you are dealing with a free text.

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 120.

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**

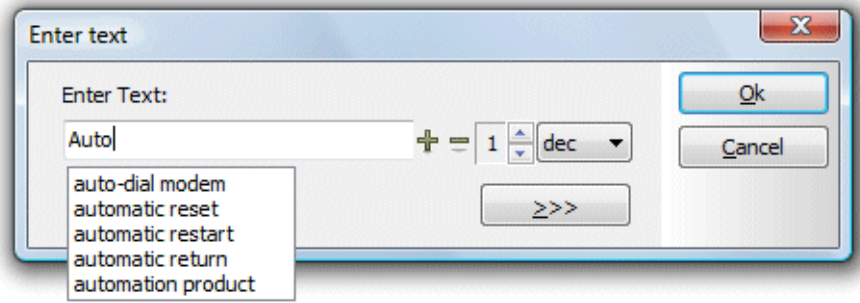
If you press <F7> to enter the **Object Lister**, you can see all the project texts in one place. From there you can also edit the texts.

— See “Checking and editing project texts via the Object Lister” on page 314.

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.

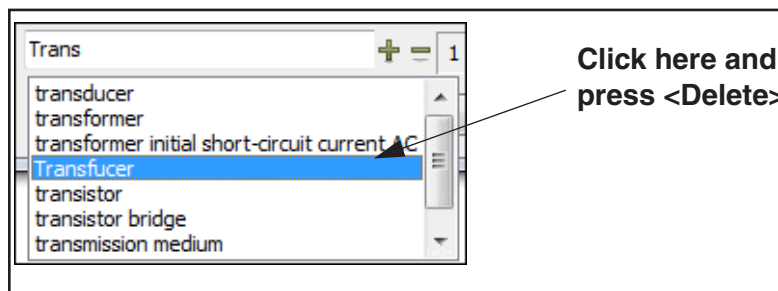
If you also want to add texts from existing projects, do like this:

- 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

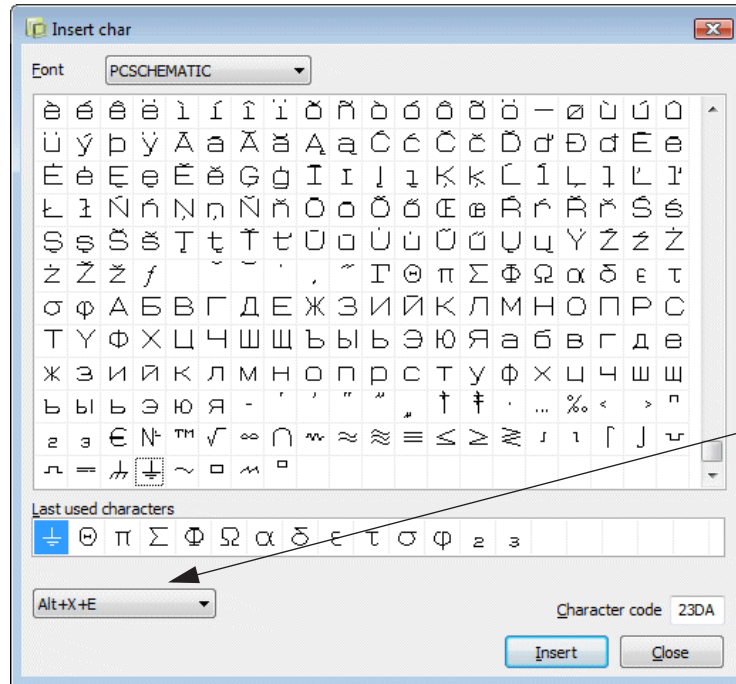
To delete texts from the autotext function, click on the undesired text in the autotext scroll menu, and press <Delete>.



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 etc.

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



Here you can specify a short-cut key for the selected character

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

Right-clicking in dialog boxes and inserting special characters

When you need to insert a special character in a text field in a dialog box, just right-click and choose **Insert char**. You then enter the dialog box **Insert char**, and can insert the character as described above.

— Please refer to “Texts in more than one line” on page 116, “Negate (invert) text” on page 116 and “Underline text” on page 117.



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.

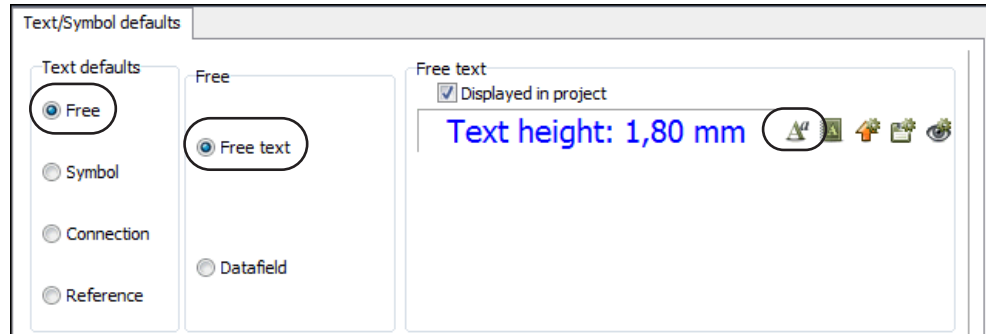
Default text properties for free texts

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 122, 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 696.

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 836.

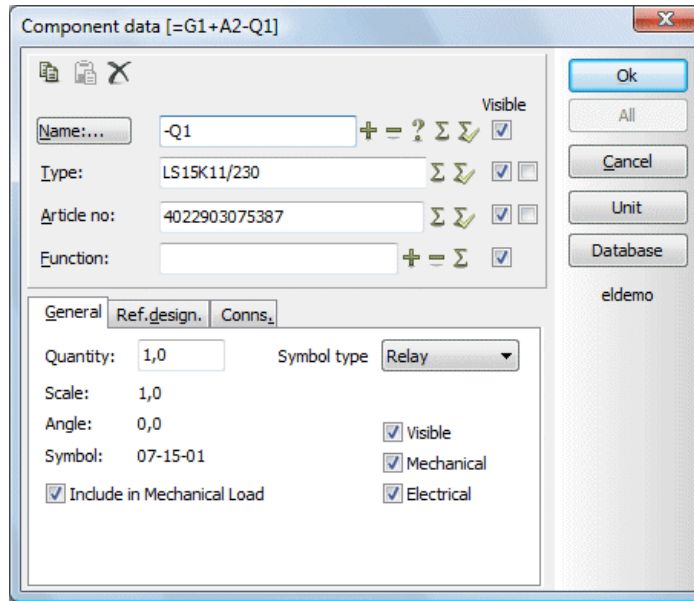


When you return later to type in the symbol texts, you have to start by clicking the **Symbols** button, or use the short-cut key <s>. Then right-click on the symbol whose symbol texts you want to change, and choose **Component data** in the appearing menu.



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:

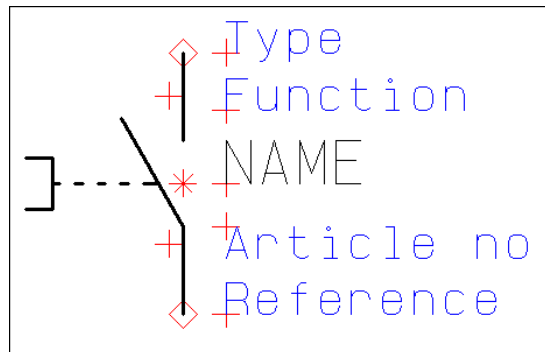


Here you can fill out the symbol texts, and decide which of the symbol texts that should be displayed together with the symbol in the project. It is also possible to type in the connection point names. This is described in “Placing and Naming symbols” on page 156.

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

Editing the symbol texts directly on a Diagram page

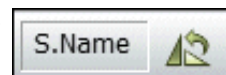
A symbol with visible symbol texts could look like this:



- Please note that references are only displayed when the symbol has references.
- See “Creating Symbols” on page 696, especially “Position for references to other symbols” on page 698.



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 119 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 47.

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 107 and “Symbol Connection points” on page 187.

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

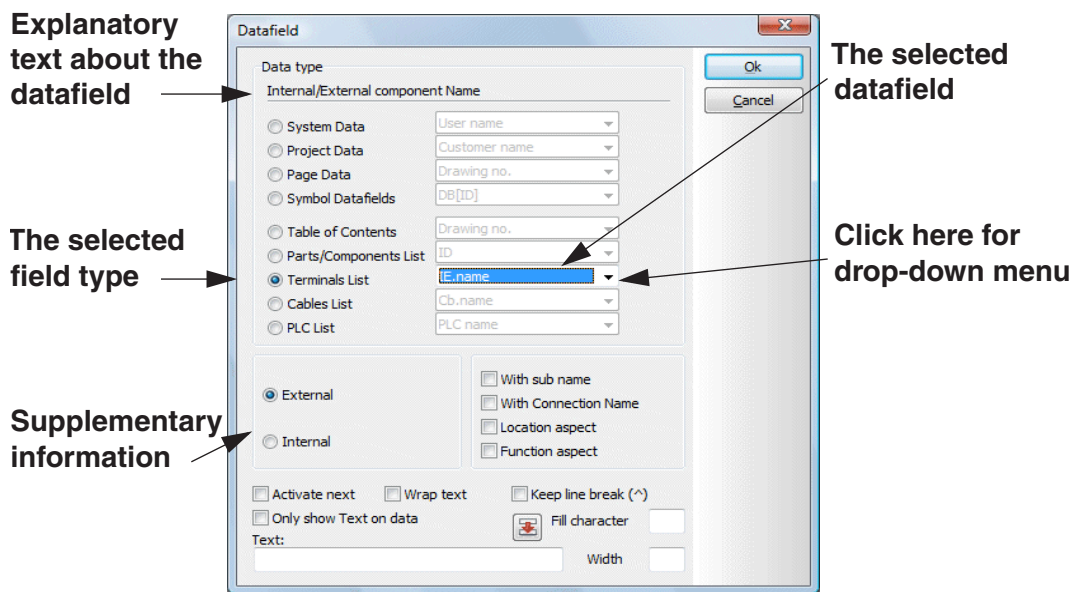
Another type of texts are called datafields. These can be filled out automatically by the program. Like free texts, they can be applied anywhere in drawings, in symbols and in drawing headers/formulas.

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 114.

— Please refer to “Page data and Project data Datafields” on page 689 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 PREDEFINED PAGE DATA DATAFIELDS, SUCH AS THE PAGE NUMBER. YOU CAN DEFINE ADDITIONAL PAGE DATA DATAFIELDS IN SETTINGS => PAGE DATA .

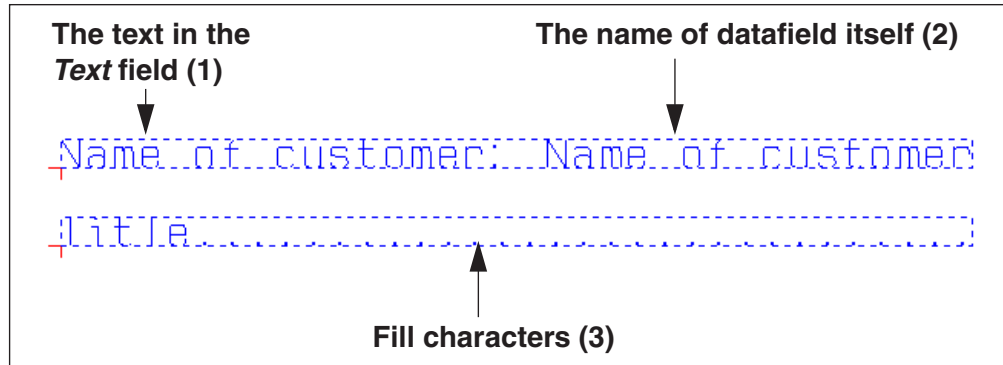
DATAFIELD	CONTENTS
SYMBOL DATAFIELDS	DATAFIELDS YOU CAN DEFINE BY YOURSELF. — READ MORE ABOUT THESE IN “CREATING DATA SYMBOLS” ON PAGE 732. YOU CAN ALSO PLACE SYMBOL DATAFIELDS FOR EACH INDIVIDUAL FIELD IN THE DATABASE. IN THE LIST OF SYMBOLDATA DATAFIELDS, THESE FIELDS ARE ASSIGNED THE NAMES DB<xxxxx> , 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 855.

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 873.
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 LINE BREAK 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 FILL CHARACTER	TRANSFERS THE SELECTED DATAFIELD TEXT TO THE TEXT FIELD. 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 752). 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 "TOTAL-PRICE:" 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**.

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 689.



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 742, and how to design your own lists in “Creating Lists” on page 752.

Data list for Project and Page title

To the right of the field **Project title** on the **Project data** tab, you can click the **Values** button. When you click the button, you can create/edit a list with possible project titles. When such a list is created, a down-arrow is displayed next to the **Project title** field, when you click inside the text field. If you click the down-arrow, you can subsequently choose between the listed project titles.

To the right of the **Page title** field on the **Page data** tab, there is a similar **Value** button, which gives you the same option for page titles.

This makes it easier to apply standardized project and page titles in projects.

Activating the Auto text function for Page title

To activate the auto text function when typing in a **Page title**, press <Ctrl+Space-bar>. — See “Autocompletion of texts” on page 103.

GENERAL TEXT FUNCTIONS

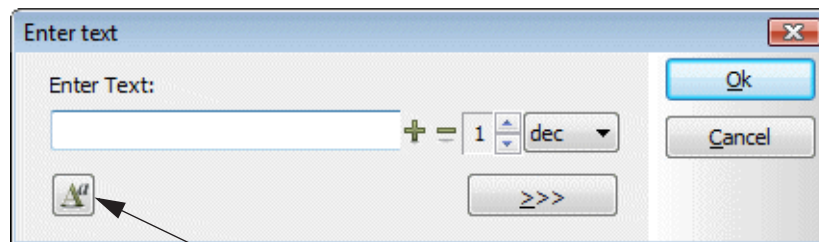
When you are working with texts, there are some text functions you can apply for all text types.

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.



Here you can change text properties

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 124 and “Wrap text” on page 124.

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 123.

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 123.

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.

Swopping texts

To swop texts and datafields, you can apply the function **Edit => Swop texts**:

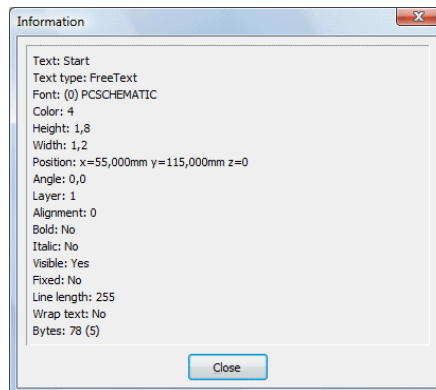
- 1) Select a text by clicking on it.
- 2) Choose **Edit => Swop texts**.
- 3) Click on the other text, that you want to swop the text with.

You can also right-click on a text, and choose **Swop texts**.

Information on texts



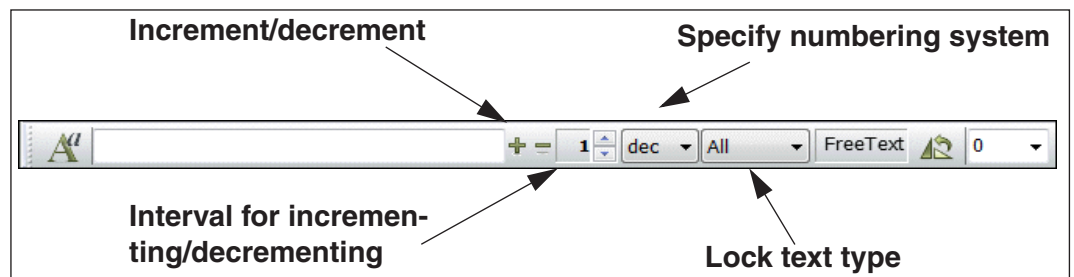
When you have chosen **Texts** and right-click on a text and choose **Information**, you enter the dialog box **Information**, which contains information about the text.



Here you find information about the text height, width, position, the used font, line length of the text, wrap text information e.t.c.

LOCKING TEXT TYPES

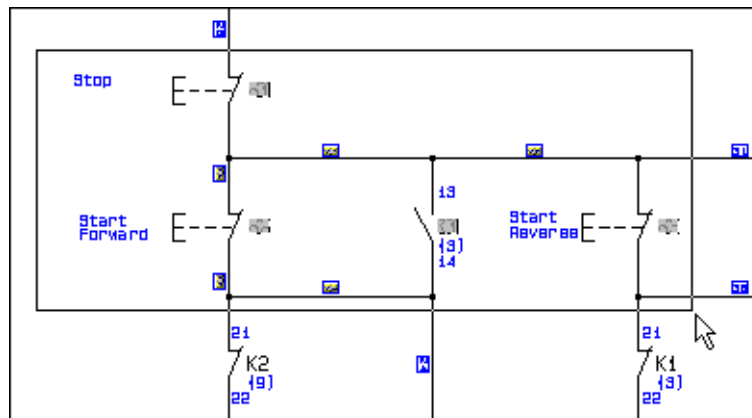
On the toolbar for texts it is possible to lock which type of texts you want to work with, as well as to carry out continuous counting on texts of the type chosen.



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 119.



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:



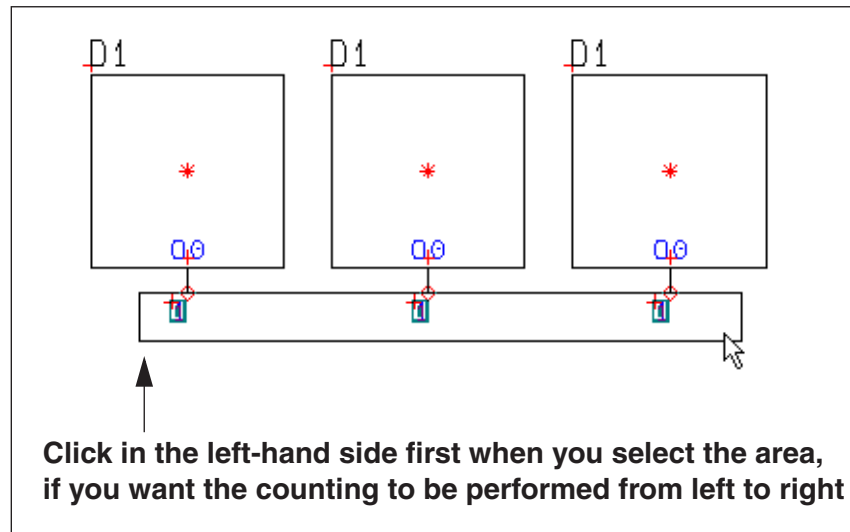
- 1) Click on **Texts**
- 2) Specify text type, interval and numbering system, as described above
- 3) Select the area containing the texts you want to count up/down (see below)
- 4) Type in the initial value in the text field in the toolbar



- 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 185 to see alternative ways to apply automatic counting on symbol names, and to “Choosing connection points in Areas” on page 189 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.

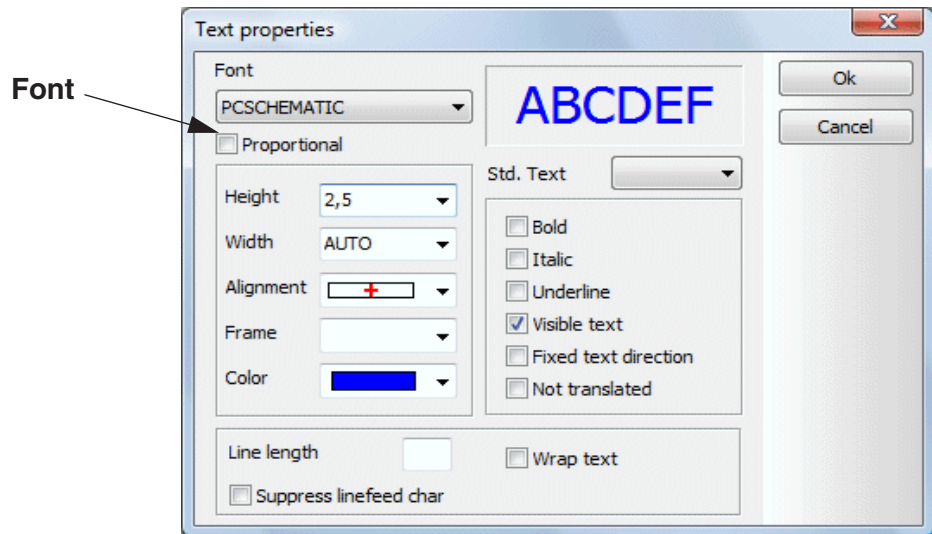
Increment/decrement is done according to the interval specified in the Texts toolbar.

— Both short-cut keys can be changed as described in “User defined short-cut keys” on page 790.

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 126.

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

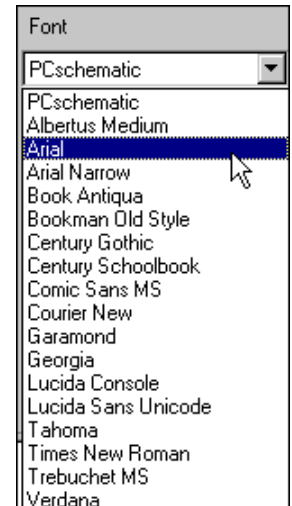
Here you choose which font you want to use.

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 331 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.

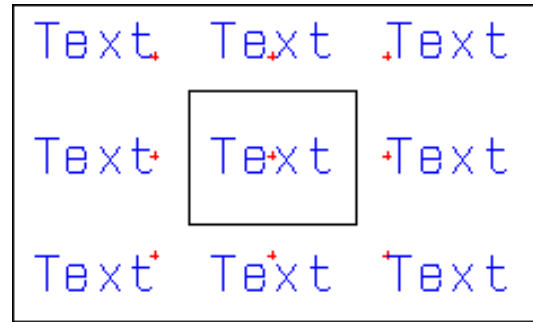
Alignment

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 123.



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 125 and “Making objects invisible” on page 69.

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 line shift 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 line shifts, which is adapted to the design of the PLC symbol. — See “Working with PLCs” on page 418.

The function can also be used in connection with text links, where you wish to suppress the line shifts for some of the texts in the text link. — See “Text links” on page 132.

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 130.

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**.

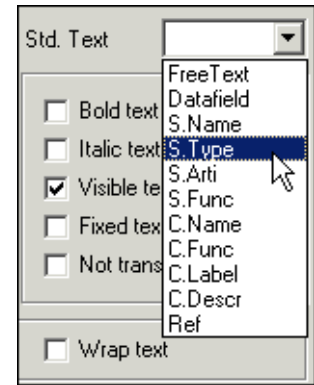
When you click on the down-arrow in the **Std. Text** field in the upper part of the dialog box, you get a list of the various text types, you can set the standard size for under **Settings => Text/Symbol defaults**.

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.

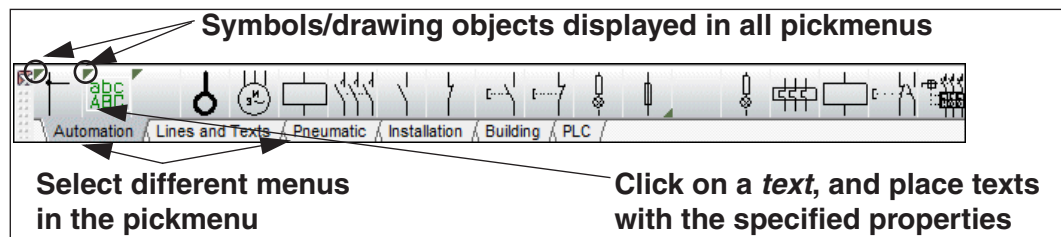
This makes it easier to place free texts having the same appearance as for instance symbol texts.

— Please refer to the “Text/Symbol defaults” on page 852 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 122 —, 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 140 and “Adapting the Pickmenu” on page 178.

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.

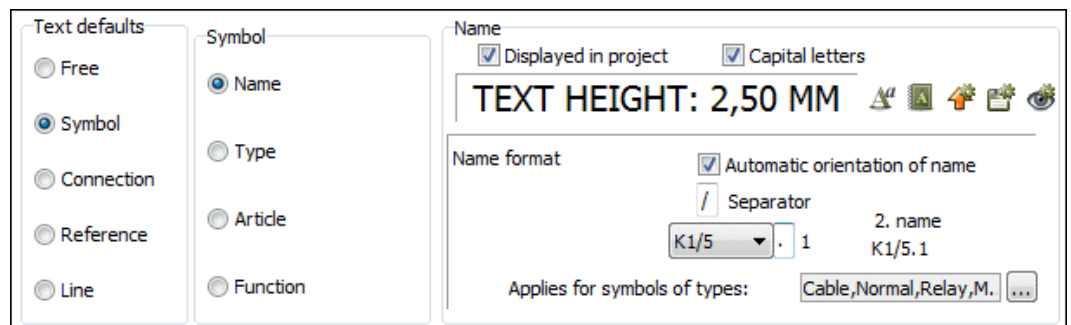
If you here decide that **Symbol article number** is *not to be shown* in the project, you will not be able to display the number for any symbol in the project — not even though you select the **Visible** field for **Name** in the **Component data** dialog box for the symbols in the project.

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 107.

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 382.

Transferring text settings to project texts



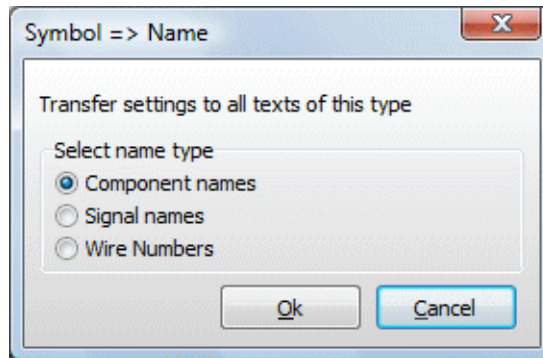
If you wish to transfer the text settings for the selected text type to the project, click the **Transfer text settings to project** button.

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**

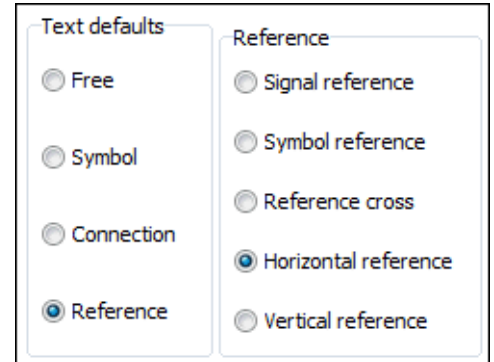
dialog box, as described in “Specifying text properties / Appearance of text” on page 122.

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.



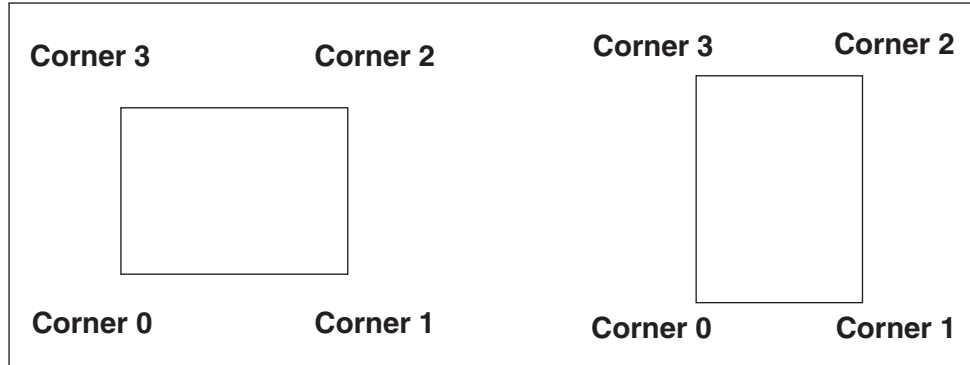
Text standards for Symbol and Connection

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.



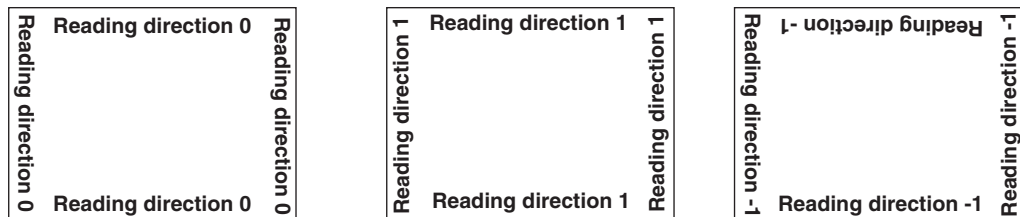
Whether the paper is in portrait or in landscape does not change this.

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 122.

— See “Position of names when placing symbols” on page 131.

Position of names when placing symbols

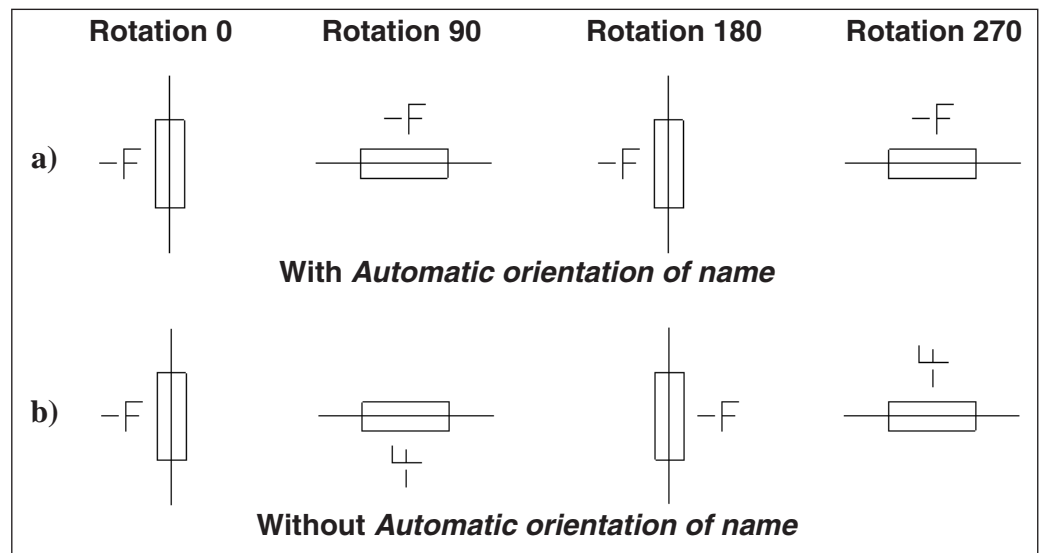


THIS ONLY APPLIES FOR SYMBOLS OF THE TYPE NORMAL

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:

- Names are not rotated, and they are positioned above or to the left of the symbol, or
- Names are rotated with the symbol



How names are positioned in the project, is specified like this:

- Choose **Settings => Text/symbol defaults**
- Below **Text standards**: Select **Symbol** or **Connection**
- 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. 132
— INDICATION OF LINKED TEXTS	P. 132
— CREATING LINKED TEXTS FROM THE TEXTS COMMAND	P. 132
— CREATING LINKED TEXTS FROM THE SYMBOLS COMMAND	P. 133
— THE DIALOG BOX TEXT LINK	P. 135
— EDITING A TEXT LINK	P. 135
— DELETING A TEXT LINK	P. 136
— TEXT LINKS AND LOADING PLC I/O LISTS	P. 136
— MAINTAINING TEXT LINKS	P. 136

What is a text link?

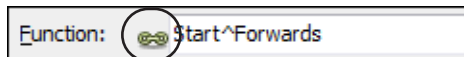
To ensure that all project texts referring to the same contents are always identical, **Automation** contains a function named *text links*. When you have linked two — or more — texts, and edit one of these texts, the other linked texts are changed automatically.

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 136.

This is also the case when importing PLC I/O data — see “Text links and importing PLC I/O lists” on page 136.

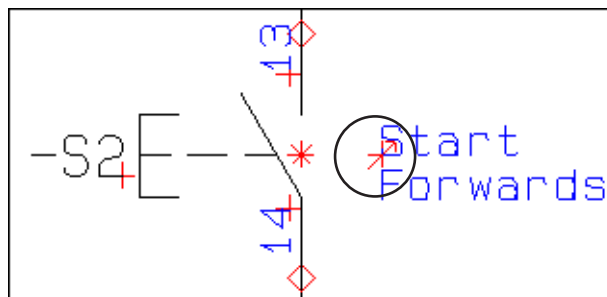
Indication of linked texts

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 306.

In diagrams linked texts are indicated by an arrow drawn through the reference point of the text:

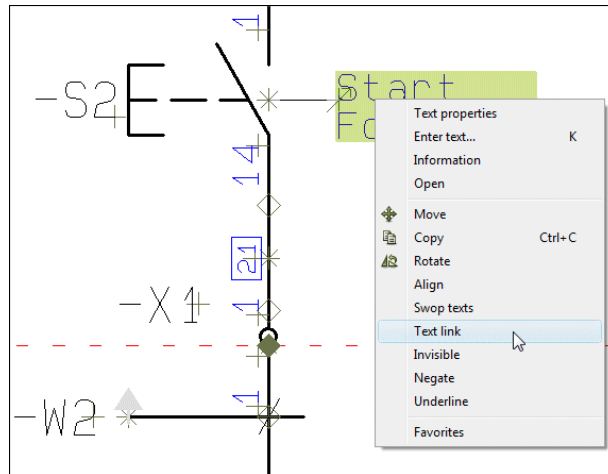


Creating a text link from the Texts command

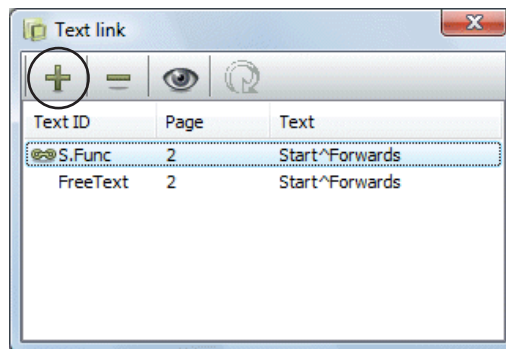
There are different ways of creating text links in **Automation**. When working with texts, do the following:



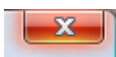
- 1) Select a text in the project — for instance click at the **Texts** button, then right-click on the text reference point, and choose **Text link**:



- 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 133 also.

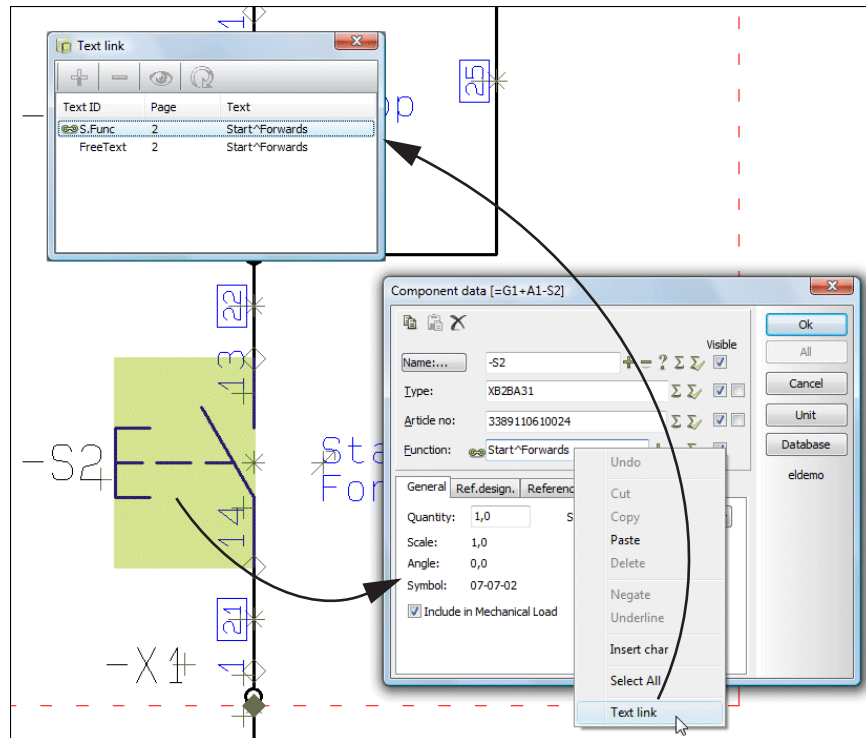
Creating text links from Component data and Connection data

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 134), and select **Text link**.

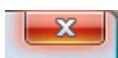


- 4) You now enter the dialog box **Text link**, from which you can link texts. — If the dialog box **Text link** is already open, the selected text is added to the active link list in the dialog box **Text link**).



- 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 132.

- 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**.



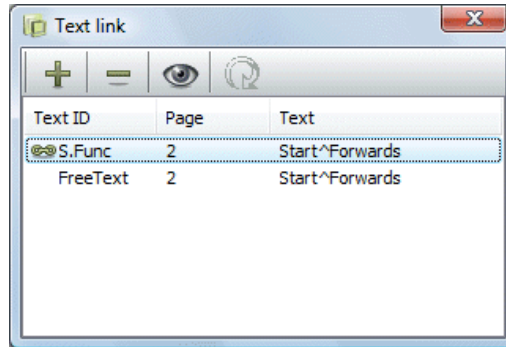
SYMBOL NAMES AND CONNECTION NAMES CAN NOT BE LINKED

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 132, 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 306.

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 432 — 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 133



- From the **Texts** command — see “Editing text links” on page 135

- From the **Object Lister** — see “The Object Lister” on page 306

- During import — see for instance “Text links and importing PLC I/O lists” on page 136

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. (This chapter is mostly about *electrical symbols*. If you wish to know more about *mechanical symbols*, please see “Mechanical symbols” on page 625).



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 57 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. 139
— PLACING AND NAMING SYMBOLS	P. 156
— SYMBOL SUB NAMES	P. 165
— SYMBOLS WITH STATES	P. 167
— UPDATING SYMBOLS FROM THE DATABASE	P. 171
— MOVING AND DELETING PLACED SYMBOLS	P. 173
— THE SYMBOL TOOLBAR	P. 176
— ADAPTING THE PICKMENU	P. 178
— CHANGING SYMBOL NAMES IN AREAS.....	P. 185
— SYMBOL CONNECTION POINTS	P. 187
— CHOOSING CONNECTION POINTS IN AREAS.....	P. 189
— THE PIN SWOP FUNCTION	P. 190
— REPLACING SYMBOLS	P. 193
— AUTOMATIC CREATION OF SYMBOLS.....	P. 197
— THE SHOW NET FUNCTION AND THE NETLISTER	P. 202
— USING SIGNAL SYMBOLS	P. 203
— USING TERMINAL SYMBOLS.....	P. 214
— USING MULTILAYER TERMINAL SYMBOLS.....	P. 215
— USING CABLE SYMBOLS.....	P. 218
— USING ACCESSORY SYMBOLS.....	P. 225
— SYMBOL FOLDERS IN Automation	P. 225



FUNCTIONS ON COMPONENT LEVEL

MORE SYMBOL FUNCTIONS ARE DESCRIBED IN “FUNCTIONS ON COMPONENT LEVEL” ON PAGE 228. THE RELATIONSHIP BETWEEN COMPONENTS AND SYMBOLS ARE DESCRIBED IN “THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS” ON PAGE 229.

— Please refer to “Creating Data symbols” on page 732 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 58, and see how to transfer object properties between symbols in “Transferring Component Data for Symbols or Lines” on page 62.



PLC FUNCTIONS

READ MORE ABOUT THE SPECIAL OPTIONS FOR WORKING WITH PLC SYMBOLS IN “PLC FUNCTIONS” ON PAGE 415.

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 696, 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>.



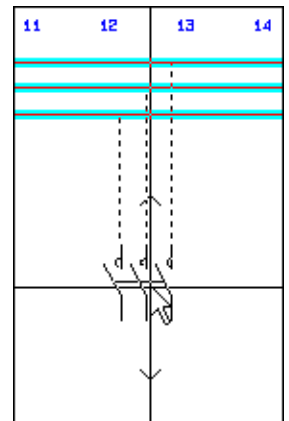
Beyond copying an already placed symbol, there are several ways of fetching a symbol in **Automation**:

WAY TO FETCH SYMBOLS	PAGE
<i>FROM THE PICKMENU</i>	140
<i>FROM THE SYMBOL MENU</i>	144
<i>FROM THE DATABASE</i>	152
<i>TYPING IN A SYMBOL FILENAME</i>	153
<i>TYPING IN AN ARTICLE NUMBER</i>	154
<i>TYPING IN A SYMBOL TYPE</i>	155
<i>DIRECT CREATION OF SYMBOLS</i>	197
<i>FROM WINDOWS EXPLORER</i>	334
<i>USING A BARCODE SCANNER</i>	624

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.

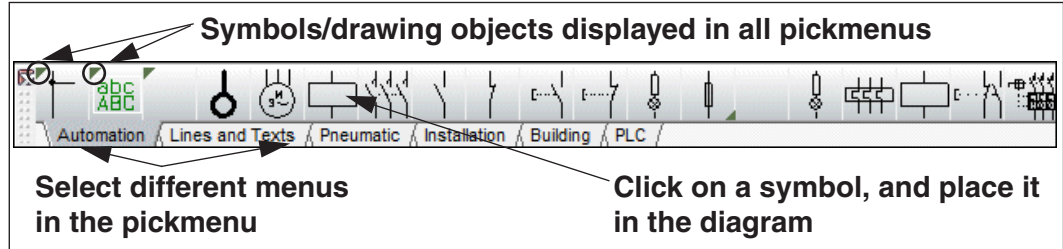
**AUTOMATIC LINE DRAWING WHEN PLACING SYMBOLS**

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 89.



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.

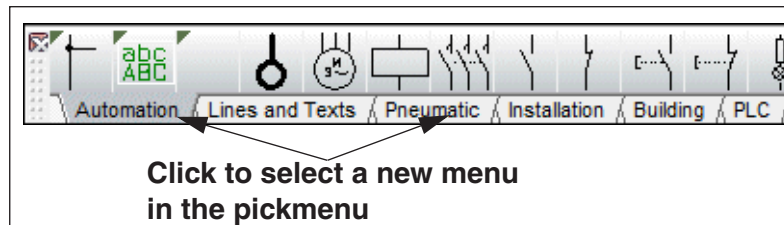
If the pickmenu is not displayed, choose **Settings => Pointer/Screen** and click on **Pickmenu**.

OTHER POSSIBILITIES WHEN USING THE PICKMENU	
— CHANGING BETWEEN MENUS IN THE PICKMENU	P. 140
— SYMBOLS / DRAWING OBJECTS DISPLAYED IN ALL PICKMENUS	P. 141
— SYMBOLS WITH COMPONENT DATA IN THE PICKMENU	P. 141
— ENTERING THE DATABASE USING THE PICKMENU	P. 142
— ALTERNATIVE ELECTRICAL SYMBOLS	P. 142
— SYMBOLS WITH STATES IN THE PICKMENU	P. 143
— POSSIBILITIES AND LIMITATIONS WHEN USING THE PICKMENU	P. 143

The pickmenu can also contain line types, text types and arcs. — Read more about the pickmenu in “Lines in the Pickmenu” on page 78, “Texts in the Pickmenu” on page 126 and “Circles / Arcs in the Pickmenu” on page 100.

Selecting the different Pickmenus

The pickmenu contains several different menus, which you select by clicking on the tabs at the bottom of the pickmenu.

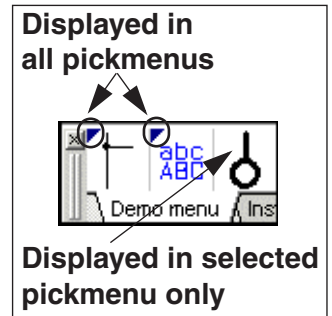


— See how to create new menus in the pickmenu in “Creating new menus in the Pickmenu” on page 180.

Symbols/ Drawing objects displayed in all Pick-menus

Symbols / drawing objects with a small triangle in the upper-left corner of the pickmenu cell, are displayed in all menus in the pickmenu.

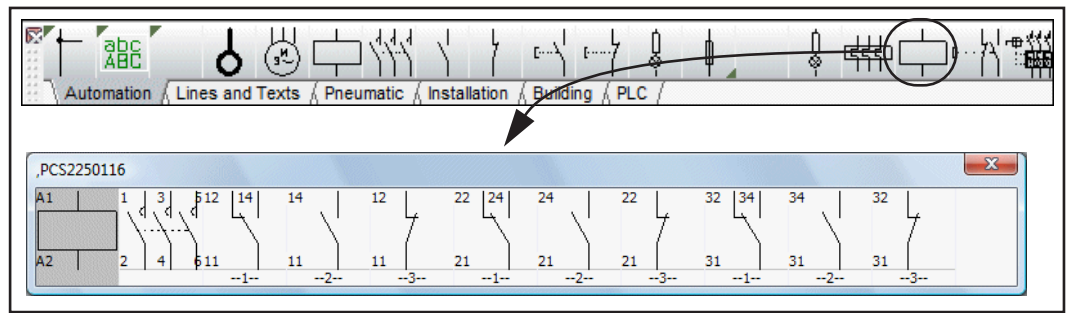
— See how to give a symbol / drawing object this status in “Inserting a Symbol / Drawing object in all Pick-menus” on page 178.



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.



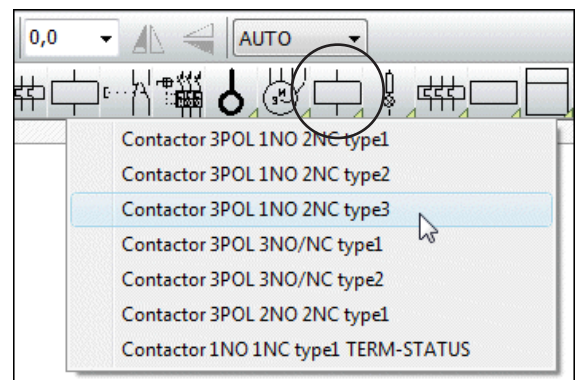
Click on one of the symbols and place it. Press <Ctrl+F9> to get this pickmenu for the component on screen again. — See “The function Show available” on page 232 or “The Show available window” on page 234.

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 sub-menus in the pickmenu” on page 179.



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 180.

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 608 and/or the **PC|SCHEMATIC Database** manual.

It is also possible to right-click on the symbol in the pickmenu, and choose **Data-base**.

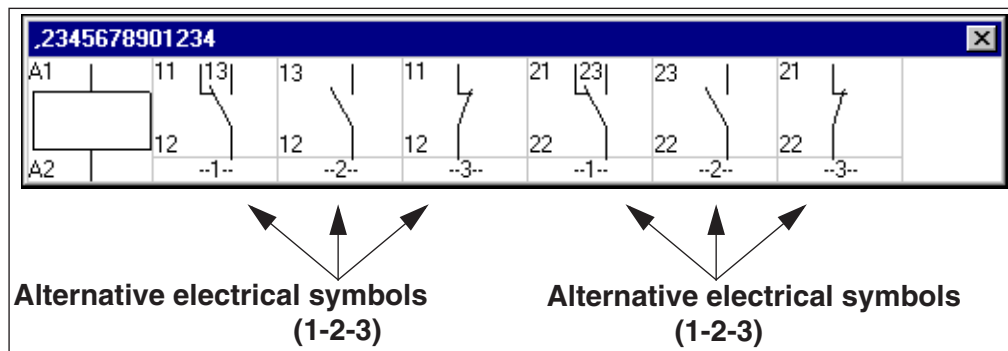
Select the desired component in the database and click **OK**.

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.

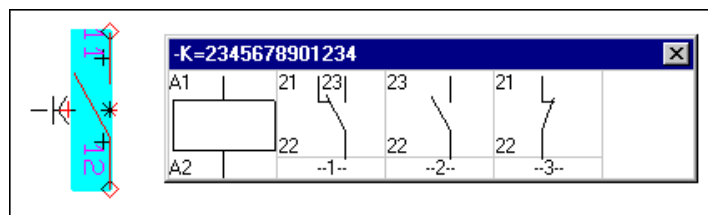
If for instance a component has a relay coil and two switch functions, the switch functions can be used as either a switch, a make contact or a break contact.

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 form the pickmenu for the component.



— Please also see “Symbols with states” on page 143.

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 594.

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.

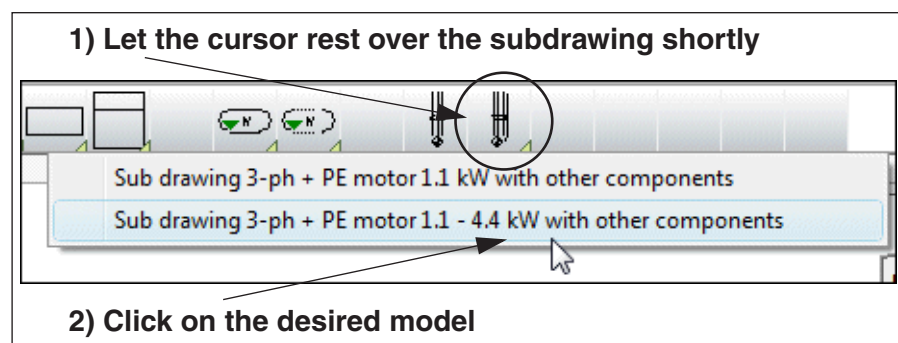
You can specify whether to display a submenu automatically, as described in “Automatic popup of submenus in the pickmenu” on page 179. When you let the mouse rest over the state for a symbol, the title for the state is displayed, if a title for the state exists.

— See “Symbols with states” on page 167, “Symbols with states for single line diagrams” on page 408 and “Creating symbols with states” on page 712.

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 182.

**THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS**

IN “MANUAL PLACING OF SUB DRAWINGS” ON PAGE 259 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 256.

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 178.

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 27.

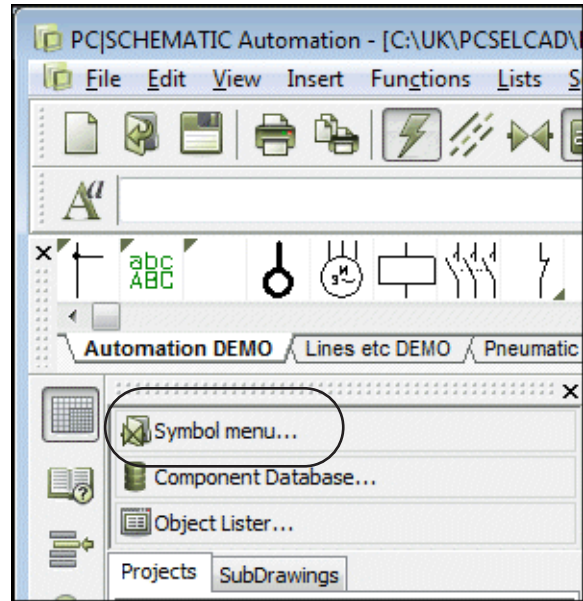
2. Fetching symbols using the Symbol menu



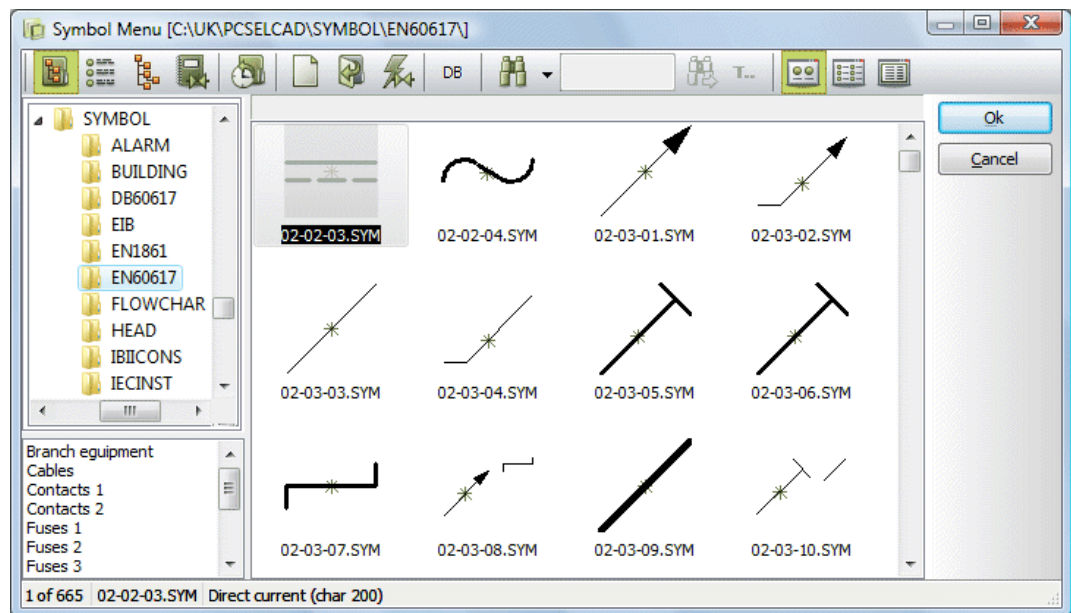
Click the **Symbols** button and the **Symbol menu** button — or press the short-cut key <F8>.

You can also click the **Symbol menu** button in the upper part of the **Explorer** window, as displayed to the right. This button is always available, no matter which command you are working in.

Alternatively, you can select **Functions => Symbol menu**.



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. 146
— REPRESENTATION OF SYMBOLS IN THE SYMBOL MENU.....	P. 146
— SEARCHING IN THE SYMBOL MENU	P. 147
— BOOKMARKS IN THE SYMBOL MENU	P. 148
— ENTERING THE DATABASE VIA THE SYMBOL MENU	P. 151
— CREATING NEW SYMBOLS FROM THE SYMBOL MENU	P. 151



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 225.

If you need to know more about changing folders and opening files, please read “Windows” on page 23.

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 841.

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

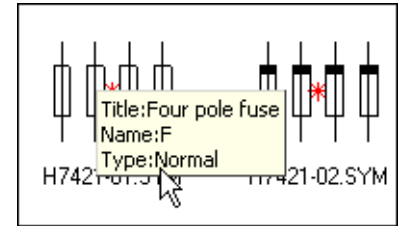
In the upper right-hand corner of the **Symbol menu**, you will find three buttons to specify how the symbols are displayed in the **Symbol menu**:

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*.

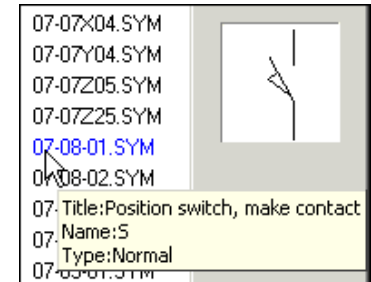


Symbol names displayed in a List



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.

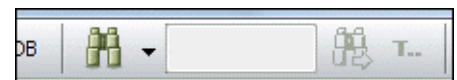
H7415L21.SYM	Four pole circuit breaker w. over-curre...	Q	With refer...
H7415M21.SYM	Four pole 3P+N circ.breaker w.aut.ov...	Q	With refer...
H7415N21.SYM	Three-pole +N automatic operating fuse	Q	Master ref...
H7418D01.SYM	Four pole HFI/HPFI disconnecter	F	Normal
H7418E01.SYM	Four pole HPFI relay	F	Master ref...

When placing the mouse cursor over a line in the list, the symbol is displayed in the right-hand side of the menu.

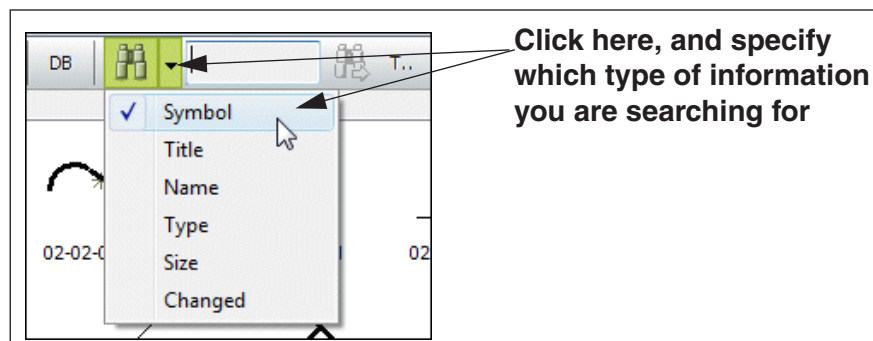
Searching in the Symbol menu



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...**



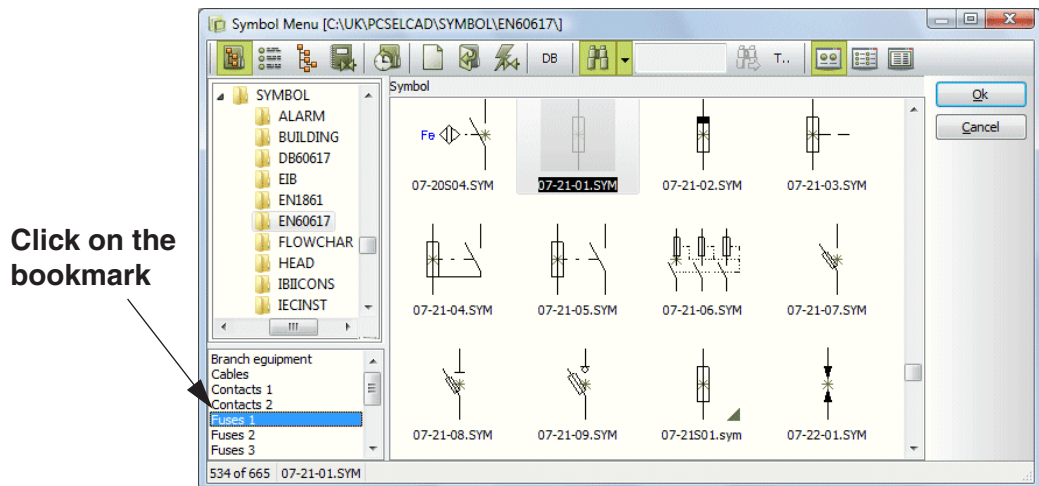
When this happens, the program searches for the text fragments anywhere in the selected text types.



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

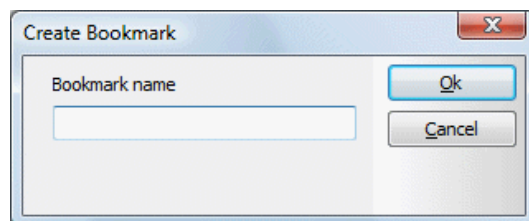
When the **Symbol menu** displays aliases or folders, it is possible to insert bookmarks. When you click on a bookmark, you jump down to the selected bookmark in the folder.



Creating a bookmark in the Symbol menu

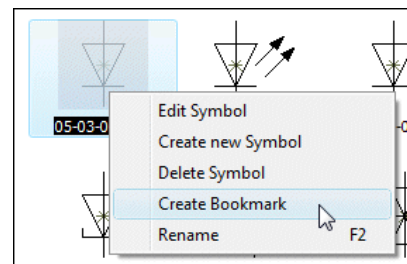
To create a bookmark, right-click on a symbol in the **Symbol menu**, and choose **Create bookmark**.

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:

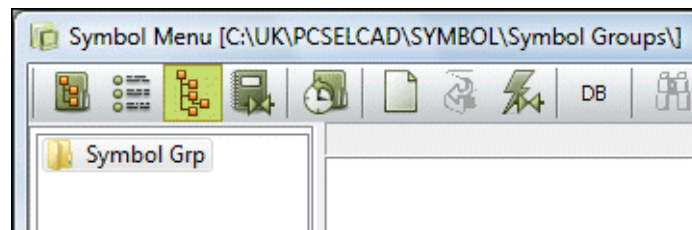
- 1) When there is access to the library from other PCs on the network, the bookmarks can also be seen from these PCs.
- 2) When you copy the library, the applied bookmarks are included automatically.

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 148 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\Symbols**:

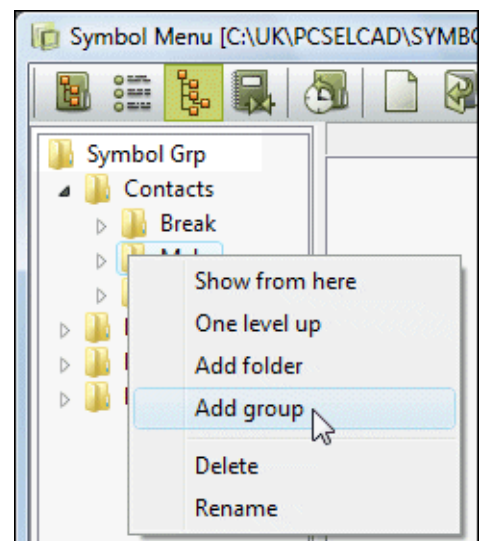


Here you can create your own grouping of the symbols, which you already have access to.

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.

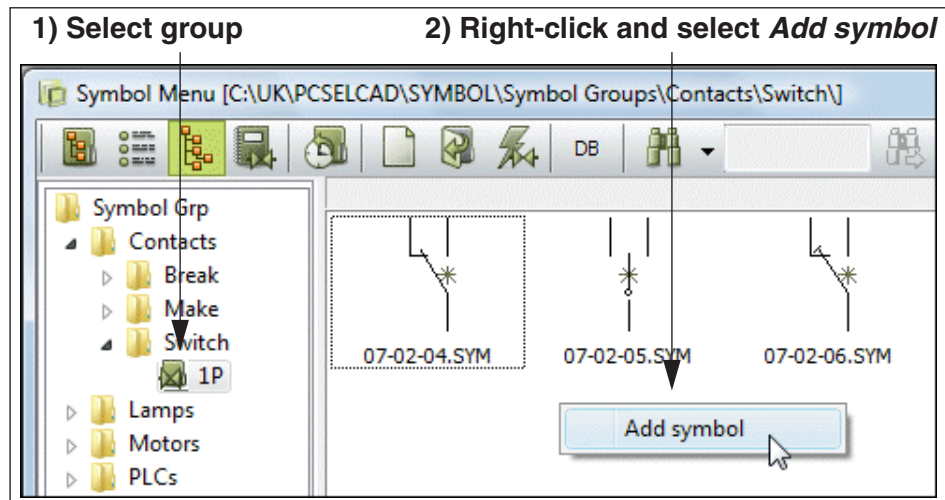
THE SAME SYMBOL CAN APPEAR IN DIFFERENT GROUPS.

You have now created a hierarchical structure, containing a group, in which you can place symbols.

Placing symbols in a hierarchical structure

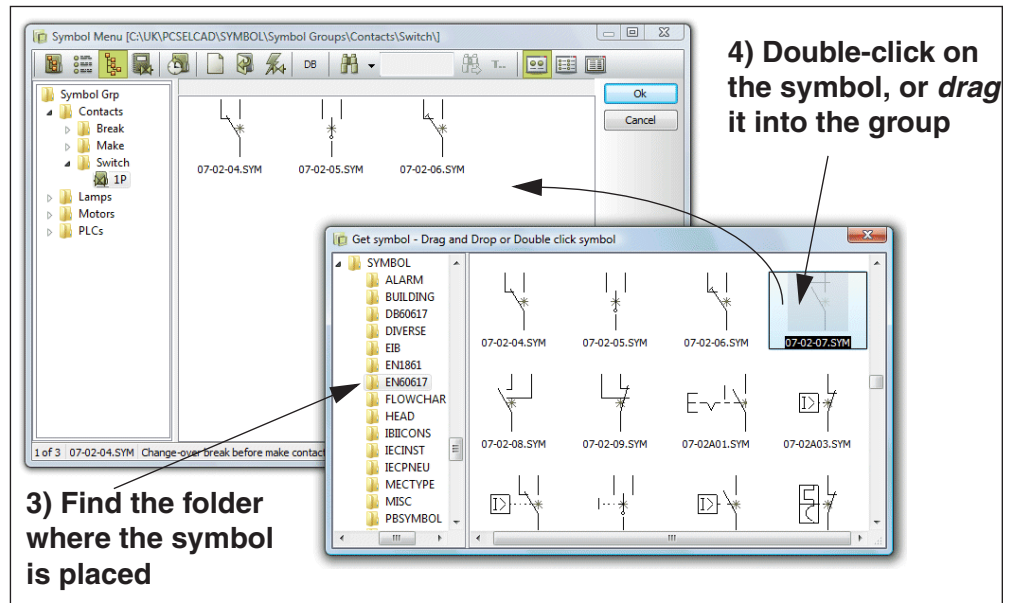
To place symbols in a group in the hierarchical structure, do this:

- 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 608 and/or the **PC|SCHEMATIC Database** manual.

— Read more about fetching symbols via the database in “Fetching symbols from the database” on page 152.

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 696.



3. Fetching symbols from the database

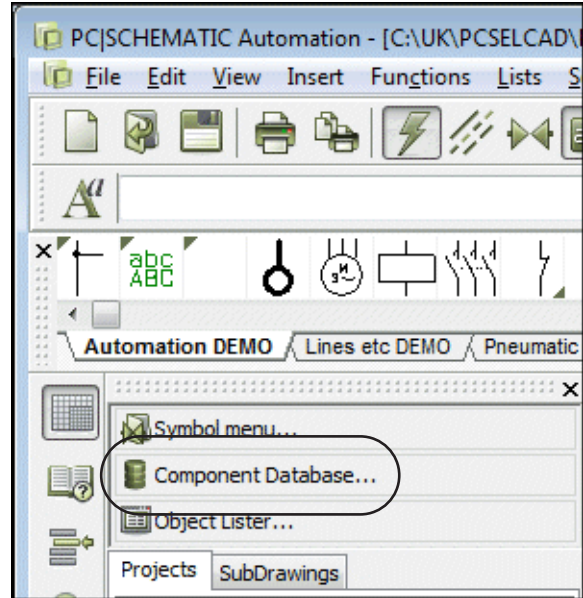
If you already know which components you are going to use, you can fetch the symbol directly from the database.

Entering the database



Click the **Symbols** button, press down <Ctrl> and click the **Symbol menu** button — or click the **Component Database** button in the upper part of the **Explorer** window, as displayed to the right. This button is always available, no matter which command you are working in.

You then enter the database directly, if it is on (See “Using the database” on page 571 for how to turn a database on). You can also press the short-cut key <d>.



EANNUMBER	TYPE	DESCRIPTION	SOURCE
1122094094	LS 37.22	Contactor, 19KW, LS 37.22, 230V AC	AEG
4022903095594	0775	Overload relay, 20-32A	AEG
3389110231632	LC ID-4011M7	Contactor, LC ID-4011M7, 220V-40A	GRUPE SCHNEIDER
3389110232936	LR2D3353	Overload relay, 23-32A	GRUPE SCHNEIDER
4022903076387	LS19K11	Contactor, 19KW, LS19K11, 230V AC	AEG
1122001330	LS19K11	Contactor, 19KW, LS19K11, 24V AC	AEG
5702423013623	CB4HC	Auxiliary contact, 3NO	DANFOSS
5702423015932	CB4HC	Auxiliary contact, 3NC	DANFOSS
5702423006175	ETD	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	LA IDN01	Auxiliary contactor, 1NC	GRUPE SCHNEIDER
3389110230284	LA IDN10	Auxiliary contactor, 1NO	GRUPE SCHNEIDER
2389110230321	LA IDN22	Auxiliary contactor, 2NO+2NC	GRUPE SCHNEIDER

OPTIONS IN THE DATABASE MENU	
— SELECTING A SYMBOL IN THE DATABASE	P. 153
— PLACING SELECTED SYMBOLS FROM THE DATABASE	P. 153
— DISPLAY THE PICKMENU FOR THE COMPONENT AGAIN	P. 153

Selecting a symbol in the database

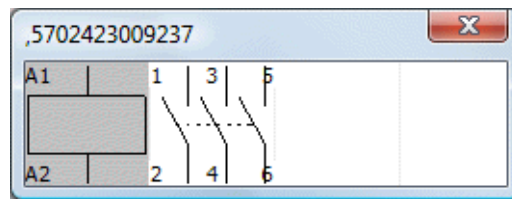
Click on a folder, for instance **Automatic switches / connection material**, and choose e.g. a **Fabricate** in the upper right-hand corner of the dialog box, and select a component.

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 571.

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 disappears automatically.

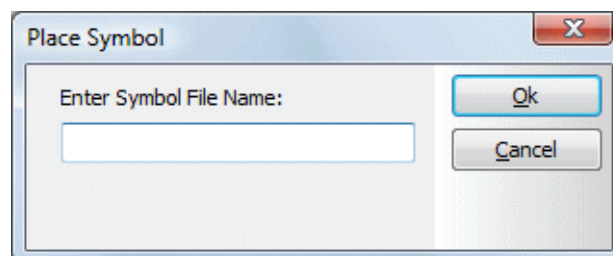
Displaying the pickmenu for the component again

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 232 for further details.

— See also “The Show available window” on page 234.

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 searches 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 841.

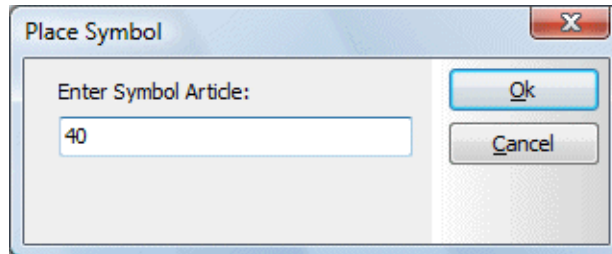
If the program can not find a matching symbol, you will enter the **Symbol menu**. If there exists a symbol in the **Symbol menu** whose name starts with the text you typed in, it will be selected.

— In “Automatic creation of symbols” on page 197 you can see how to create symbols directly from this dialog box.

5. Typing in the Article Number directly

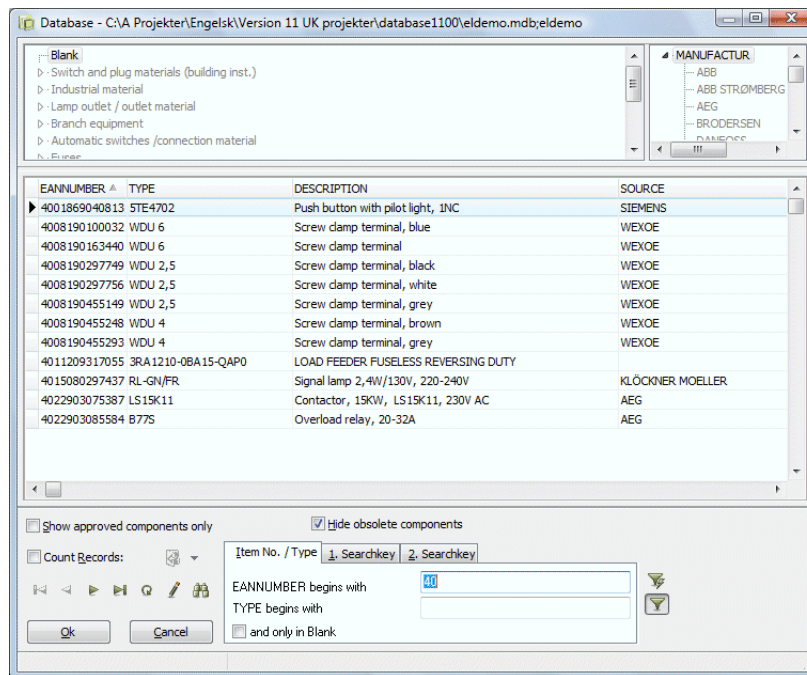


When you have activated **Symbols** and choose **Functions => Database => Find Article** — or press the short-cut key <v> — you enter this dialog box:



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:



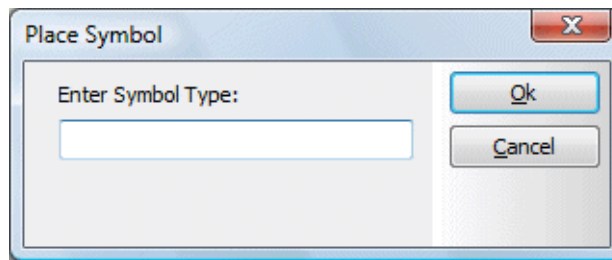
Click on the desired component, and click **OK**. You now get the electrical symbol for the component in the cross hairs — or a pickmenu containing all electrical symbols for the component.

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 — 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.

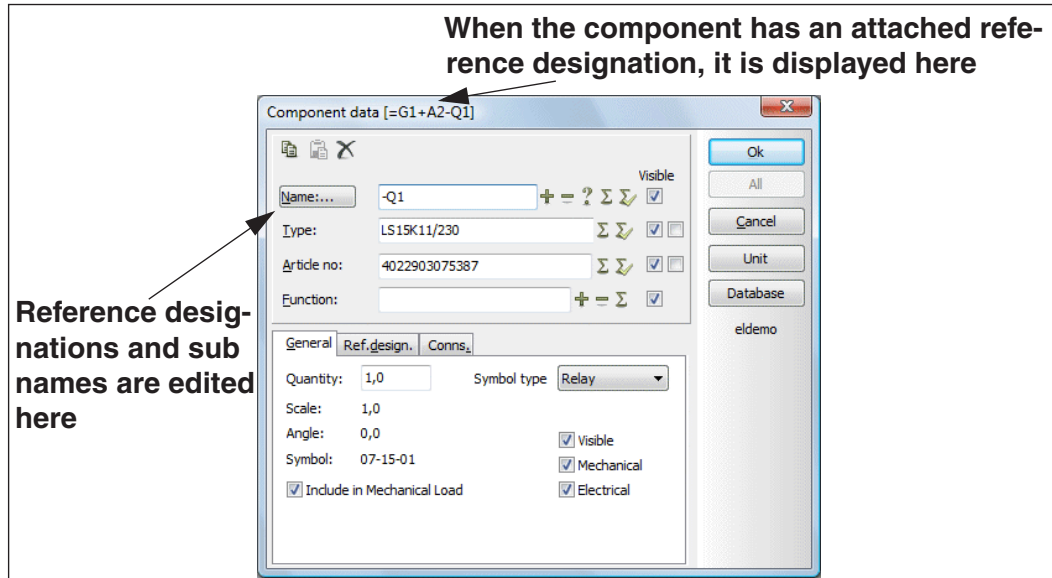
IF FOR INSTANCE YOU ARE USING AN ACCESS DATABASE, THERE WILL BE NO DISTINCTION BETWEEN UPPER AND LOWER CASE CHARACTERS.

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 139 — 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 107 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 231.

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 176.



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 836.

OPTIONS IN THE DIALOG BOX COMPONENT DATA	
— SPECIFYING THE COMPONENT NAME	P. 158
— DISPLAYING/EDITING THE PARTS OF A COMPONENT NAME	P. 159
— SPECIFYING TYPE AND ARTICLE NO.....	P. 159
— SPECIFYING FUNCTION.....	P. 160
— VISIBLE/INVISIBLE TEXTS	P. 160
— THE TAB GENERAL	P. 160
— THE TAB REF.DESIGN.	P. 161
— THE TAB REFERENCE	P. 162
— CHANGING CONNECTION DATA.....	P. 162
— THE TAB SYMBOL DATA	P. 164
— COPYING COMPONENT DATA	P. 164
— COLLECTING INFORMATION FROM THE DATABASE	P. 164
— ATTACHING UNIT DRAWINGS TO SYMBOLS.....	P. 164

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 185.

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 108.

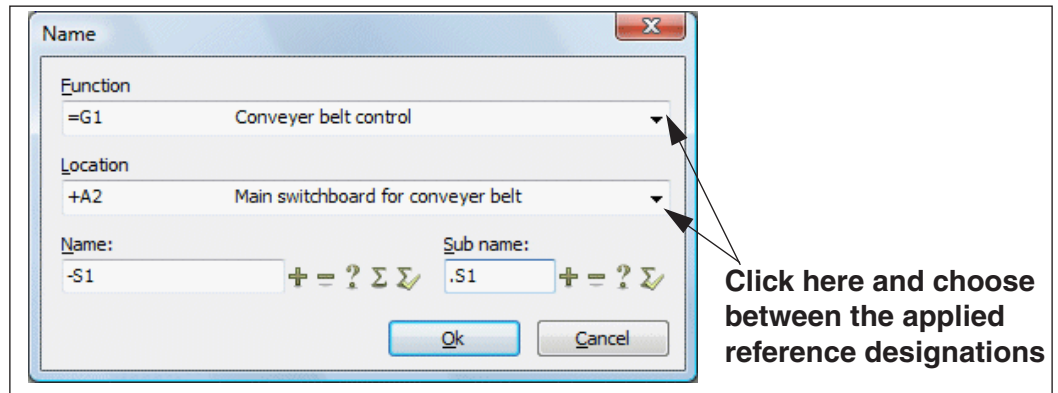
More advanced ways to change texts directly on diagrams are described in "Locking text types" and “Counting on texts” on page 120.

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 382.

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 161 and “Sub names on symbols” on page 165.

Options for the Type and Article no fields

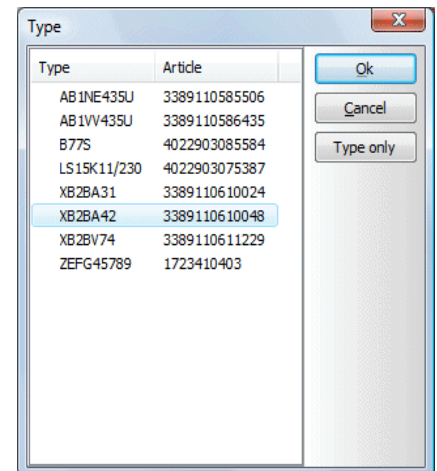


For the fields **Type** and **Article no** in the dialog box **Component data**, you can either type in data, or fetch these via the **Used** and **Used related** buttons.

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 152.



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 231.

— See also “Updating symbols from the database” on page 171.



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 107.

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 127.

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 407), 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 706.

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 708.

Visible

When **Visible** is *not* selected, the symbol is not displayed on print — See “Making objects invisible” on page 69.

Include in Mechanical Load

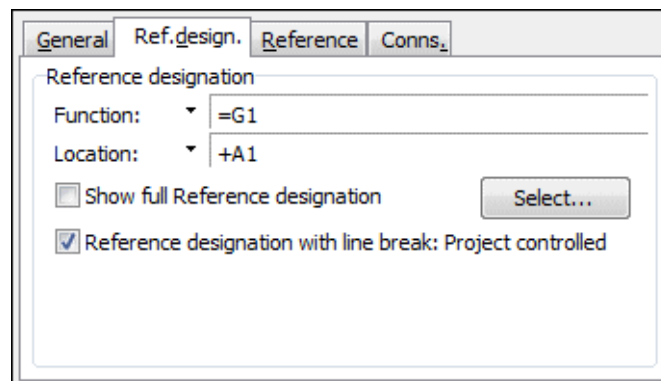
If **Include in Mechanical Load** is selected for the symbol, a mechanical symbol will be placed for the symbol during mechanical load, if a mechanical symbol exists for the component in the database.

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 “Placing mechanical symbols via the database” on page 628.

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**.

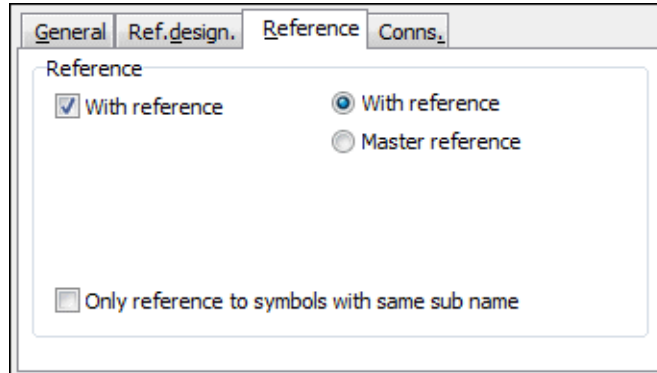


From here you choose the reference designations for the symbol, as described in “Reference designations for symbols” on page 349. — Another way of selecting reference designations for the symbol, is described in “Button for Displaying/Editing the parts of a Component name” on page 159.

— See also “Show full Reference designation” on page 351 and “Individual control of reference designations with line breaks from Component data” on page 351.

The tab Reference

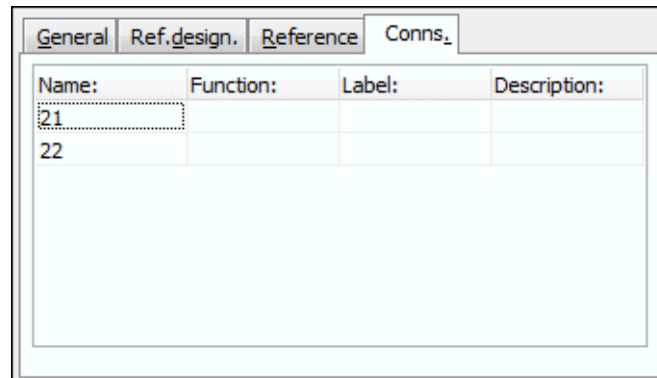
In **Automation** all symbols can have attached references. For symbols which does not have attached references by default, you will therefore find an extra **Reference** tab in the dialog box **Component data**.



— Read more about the use of this tab in “References on symbols” on page 369.

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 187.

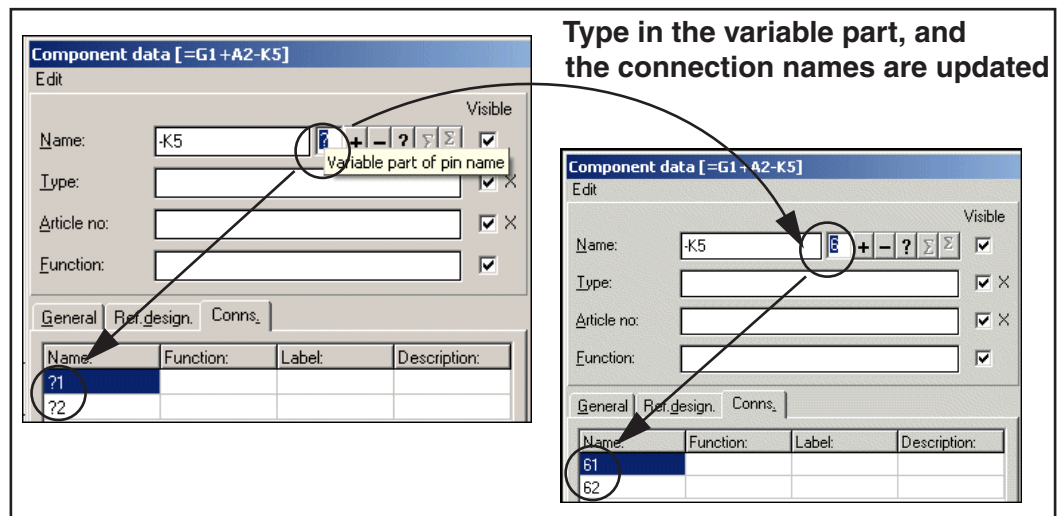
Changing connection Name from the Component data box

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 701 —, there will exist an extra text field in the

dialog box **Component data**.

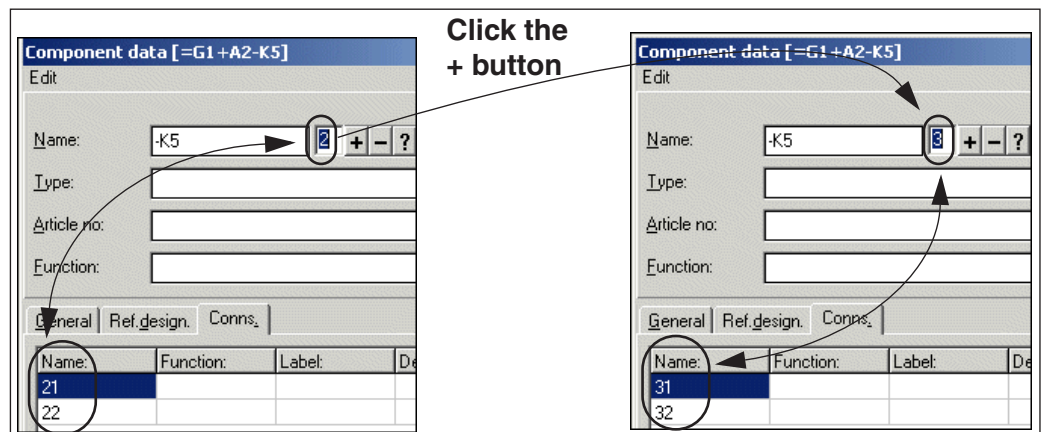


Here you can type in the variable part of the connection name. If there is a question mark, this represents one single character. If there is an asterisk, you can type in all of the connection name. This can for instance be applied for terminal symbols. — See “Wildcards/jokers” on page 550 for further 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 158.

— Please refer to “Symbols with variable connection names” on page 599 and “Connection data with a variable part in the name” on page 701.

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

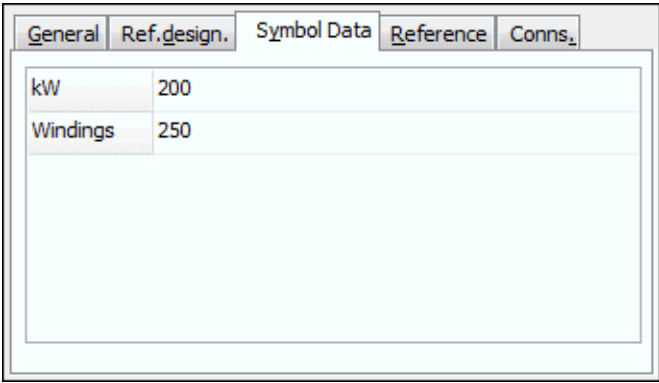
symbol. — For further details about this, please refer to “Place connection points” on page 699.

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 711 — the tab **Symbol Data** — will also appear.

Click on the tab **Symbol Data** and type in the relevant values.



kW	200
Windings	250

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 62.

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 571.

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 568.

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

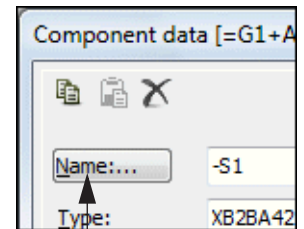
SYMBOLS FOR THE SAME COMPONENT, WHICH HAS DIFFERENT SUB NAMES, ARE FUNCTIONALLY UNDERSTOOD AS DIFFERENT COMPONENTS IN THE PROJECT, WHILE THEY IN PARTS LISTS ARE UNDERSTOOD AS PARTS OF THE SAME COMPONENT.

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.

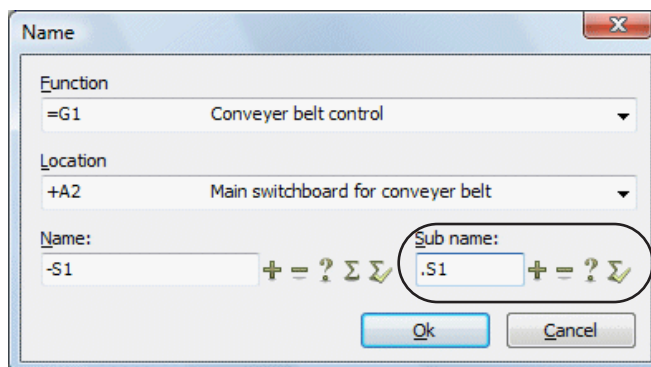
Specifying sub names on symbols

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 606.

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 110.

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.

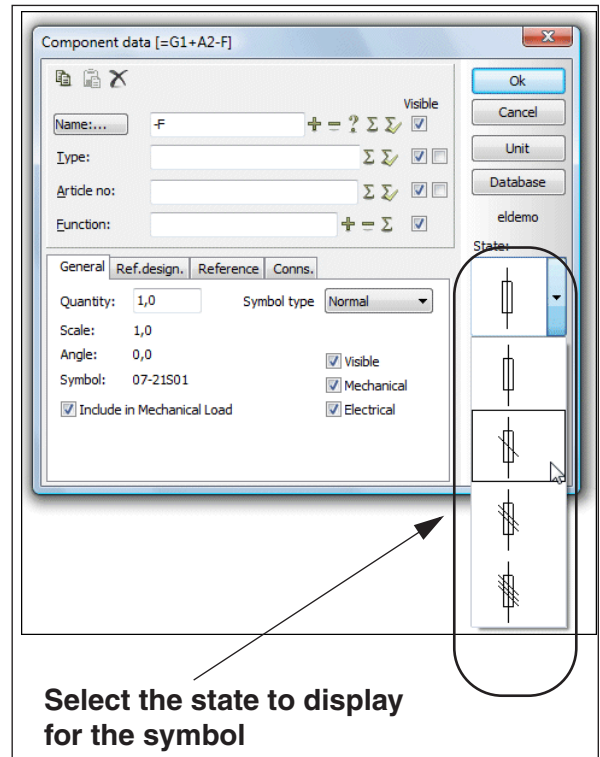
This is only possible for symbols, which have been designed with different states. — See how to create symbols with states in “Creating symbols with states” on page 712.

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 408 and “Placing symbols with selected states in the pickmenu” on page 181.



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.

Symbols with states and Show available

If it is defined in the database that a symbol is to be displayed with a specific state selected, and this state has been changed to another state for a placed symbol, this is interpreted as if the symbol has *not* been placed in the project. In **Show available** the symbol will therefore still appear, and it will be displayed with the state defined in the database.

If *no* state is specified for the symbol in the database, the symbol disappears from **Show available**, when it is placed in the project — no matter which state is selected for the symbol in the project.

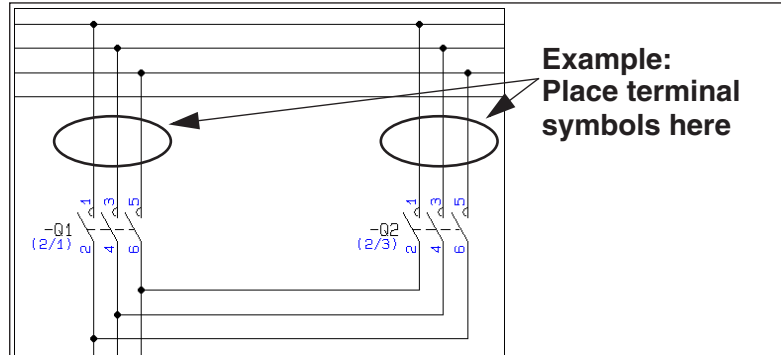
— Read more about **Show available** in “The Show available window” on page 234.

— See how to specify states for symbols in the database in “Specifying symbol state for electrical symbols via the database” on page 596.

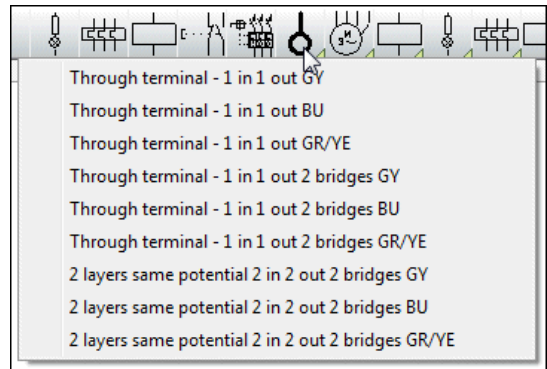
MULTIPLACING SYMBOLS

In **Automation** it is possible to place multiple symbols of the same kind in one single operation — for instance when placing terminals.

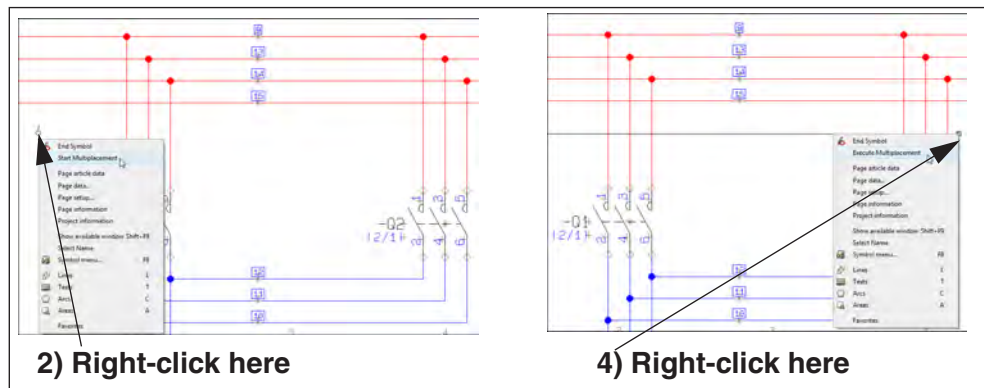
Do this:



- 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 169.

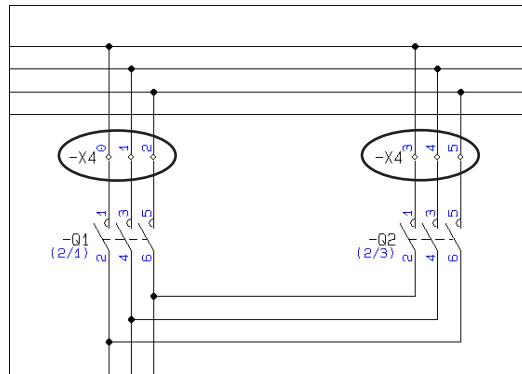


- 4) Right-click on the page, and select **Execute Multiplacement**.



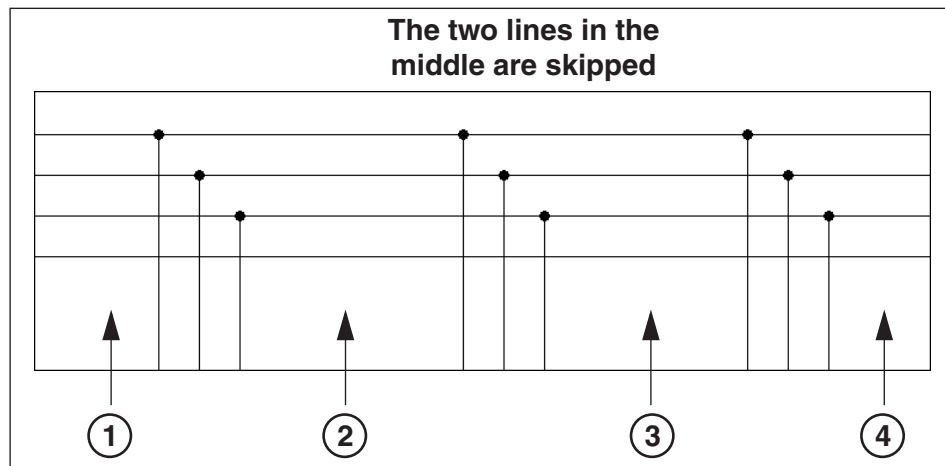
- 5) In the dialog box **Component data**: Specify symbol names and click **OK**. — See also “Placing and Naming symbols” on page 156 for how to name connection names for terminals directly from the **Symbol** menu.

6) The terminals are now placed with assigned connection names:

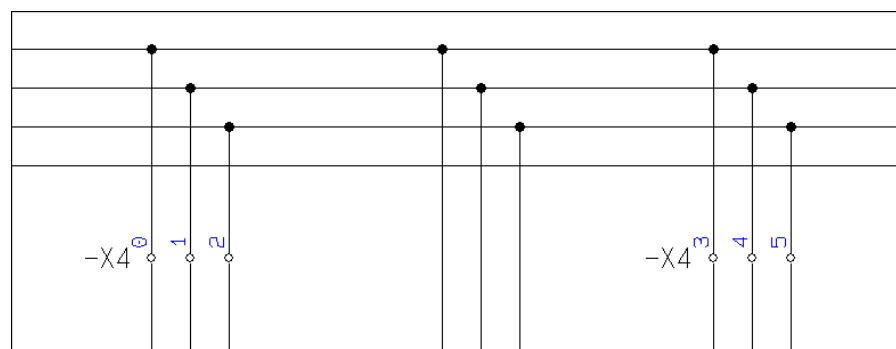


Skipping line segments when multiplacing symbols

To skip one or more line segments when multiplacing symbols, do this:



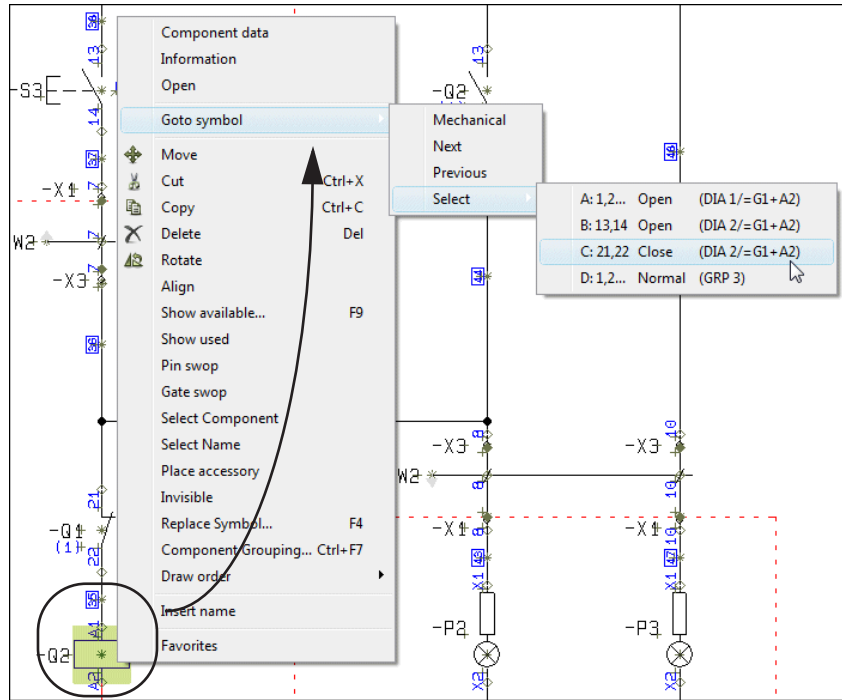
- 1) Select a symbol, e.g. a terminal, from the pickmenu. Right-click, and select **Start Multiplacement** — see the figure above.
- 2) Click *before* the lines you wish to skip.
- 3) Press <Ctrl> and click *after* the lines you wish to skip.
- 4) Right-click *after* the lines you wish to place symbols on, and select **Execute multiplacement**.



— See also “Multiplacing symbols” on page 168.

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 367, IN PARTICULAR “AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK” ON PAGE 367. 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:

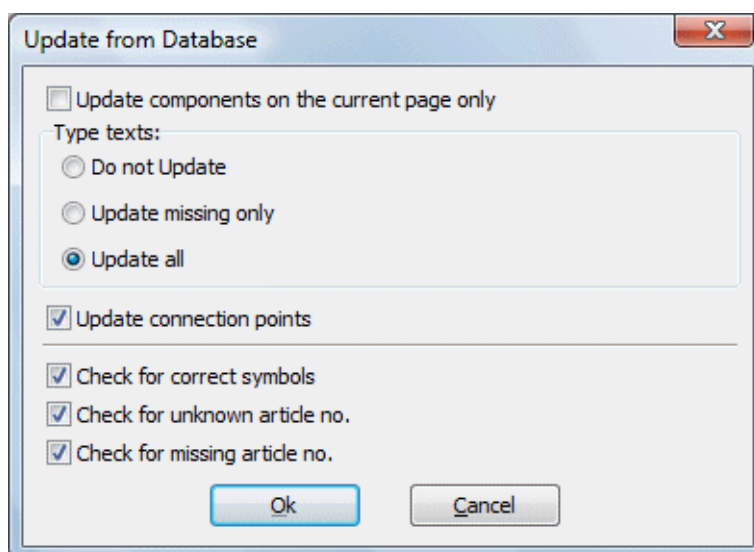


UPDATING SYMBOLS WHEN CHANGING ARTICLE NUMBERS

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 231 AND “UPDATE FROM DATABASE WHEN CHANGING ARTICLE NUMBER” ON PAGE 159.

- 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 610).
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 251.

Deleting a placed symbol

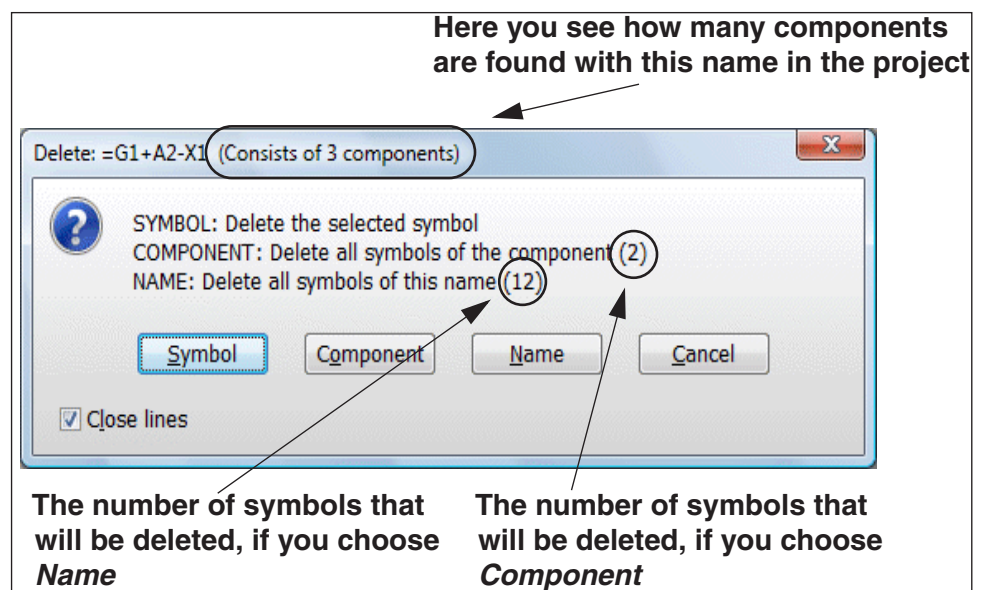
You can delete a placed symbol in two different ways:

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

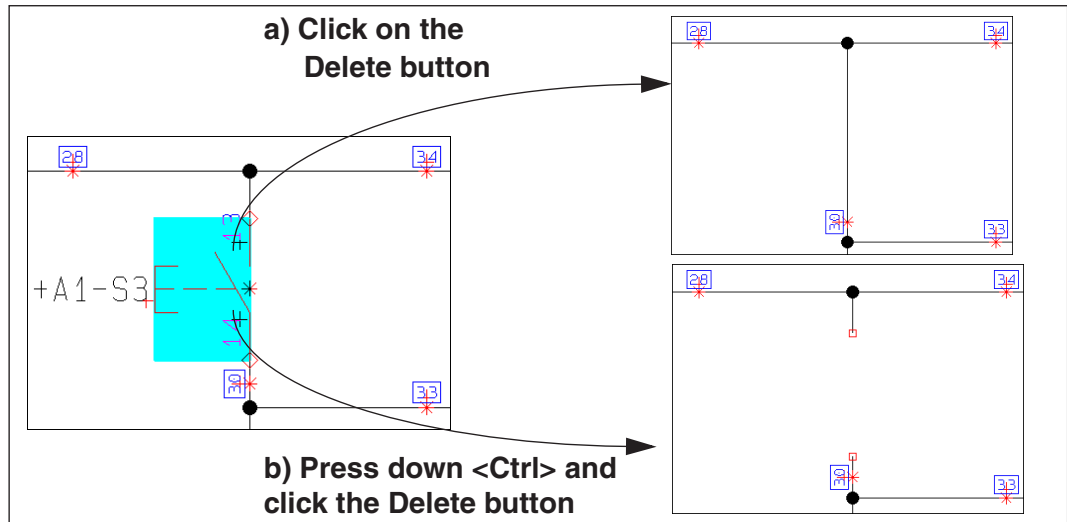


1) Click on the **Symbols** button —to start working with symbols.



- 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 185 — 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.

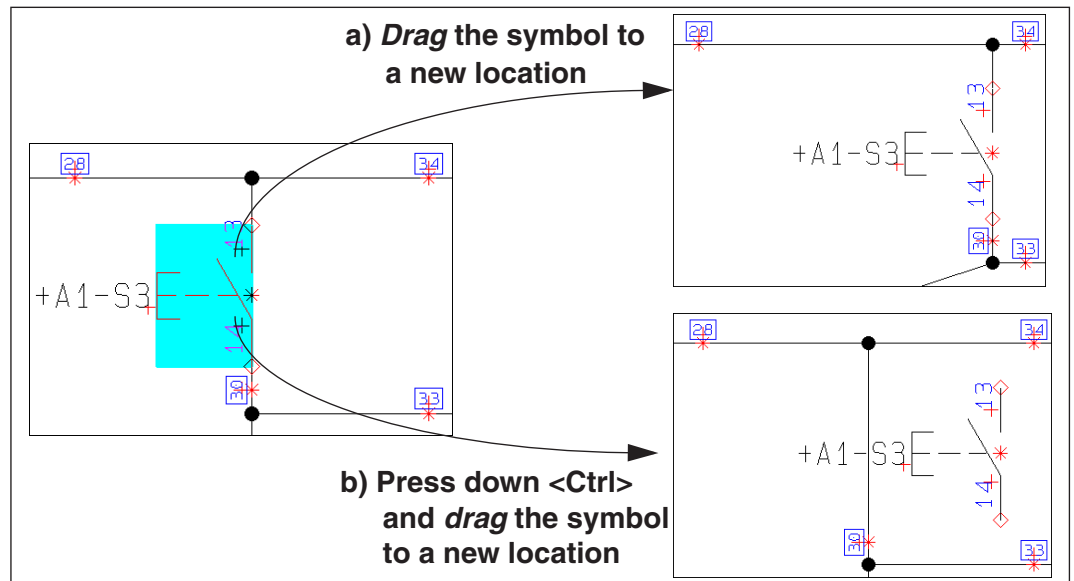
The symbol is now moved, and is still connected to the same lines as before.

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 185 — 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

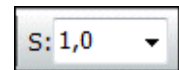


When the **Auto Name** button is activated, symbols are automatically assigned the next available symbol name when placed in the project.

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 708.

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 59.

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.

1) Choose symbol type

2a) Click here

2b) Press down the mouse button and drag the mouse cursor over the screen

3) The selected type of symbols are selected in the area

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 140.

The pickmenu can also contain line types, text types and arcs. — Read more about this in “Lines in the Pickmenu” on page 78, “Texts in the Pickmenu” on page 126 and “Circles / Arcs in the Pickmenu” on page 100.

THIS CHAPTER DESCRIBES	
— PLACING SYMBOLS IN THE PICKMENU	P. 178
— PLACING SYMBOLS IN THE PICKMENU WITH COMPONENT DATA	P. 178
— INSERTING A SYMBOL/DRAWING OBJECT IN ALL PICKMENUS	P. 178
— CREATING A NEW MENU IN THE PICKMENU	P. 180
— SPECIFYING COMPONENT DATA IN THE PICKMENU	P. 180
— PLACING SYMBOLS IN THE PICKMENU VIA THE DATABASE.....	P. 181
— PLACING SYMBOLS WITH SELECTED STATES IN THE PICKMENU	P. 181
— RIGHT-CLICKING IN THE PICKMENU	P. 183

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.

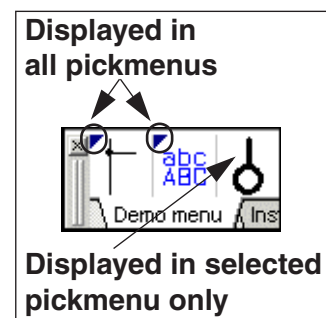
Or, you can right-click inside an empty cell in the pickmenu and choose **Place symbol**, which brings you into the **Symbol menu**, where you click your way to the symbol you want to place in the pickmenu, and click **OK**.

Include component data when inserting a symbol in the pickmenu

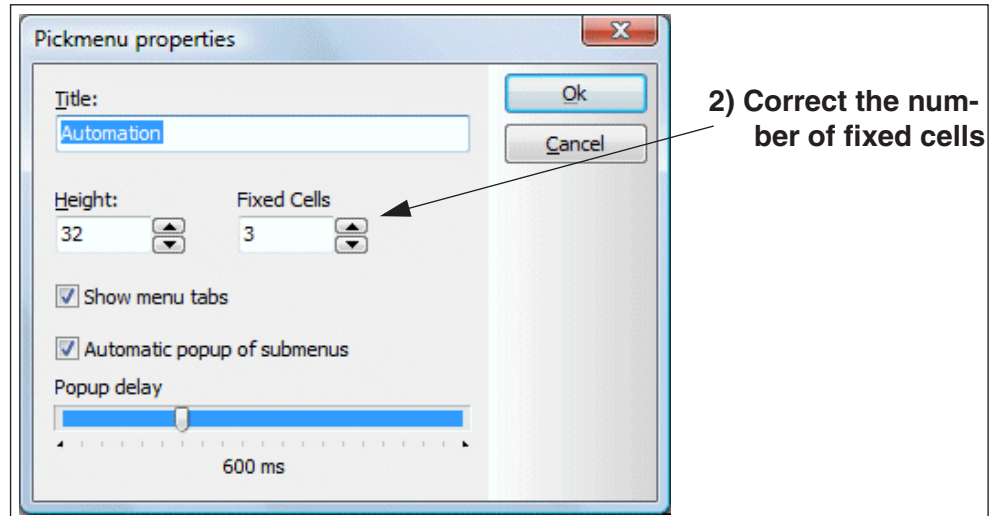
When you hold a symbol in the cross hairs, and press down <Ctrl> while placing the symbol in the pickmenu, the component data is also transferred to the symbol in the pickmenu.

Inserting a Symbol / Drawing object in all Pickmenus

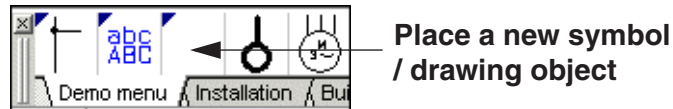
To insert one or more drawing object(s) to be displayed in all pickmenus, do the following:



- 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 178 or “Include component data when inserting a symbol in the pickmenu” on page 178.



- 4) The symbol is now represented in all pickmenus.

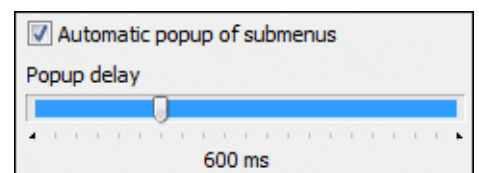
Removing fixed cells from the pickmenus

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 167) or for symbols with multiple sets of article data attached (see “Symbols with multiple sets of data in the Pickmenu” on page 141), when you let the mouse rest over the symbol in the pickmenu.

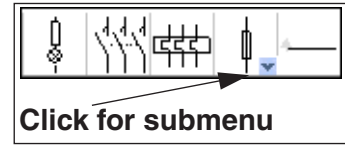
To do this, right-click in the pickmenu, and choose **Properties**. You then enter the dialog box **Pickmenu properties**, where you select **Automatic popup of submenus**.



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 toolbars.

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

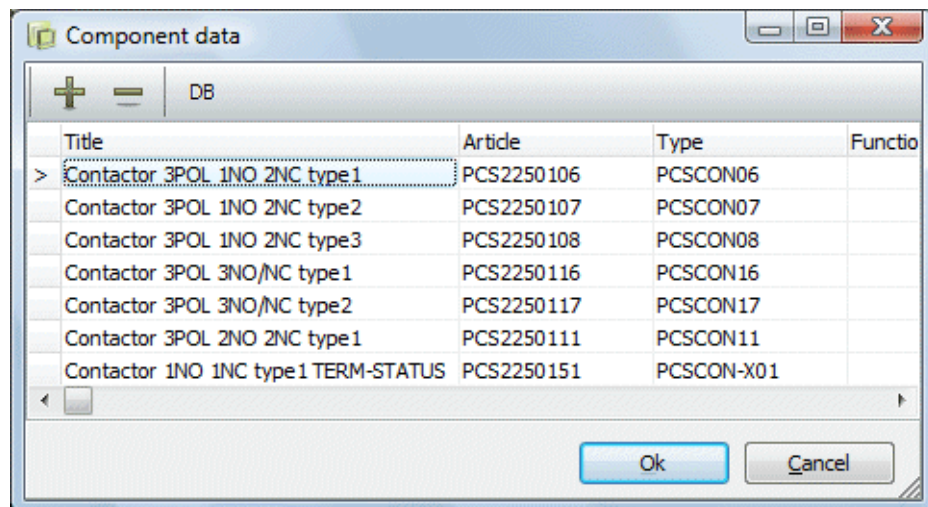
To delete a menu, do the following:

- 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 575.



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 232.

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 167 and “Creating symbols with states” on page 712.

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 620, especially “Example for sub-drawings” on page 621.

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 143.



THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS

YOU CAN READ MORE ABOUT SUB DRAWINGS IN THE CHAPTER “SUB DRAWINGS” ON PAGE 256. 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.

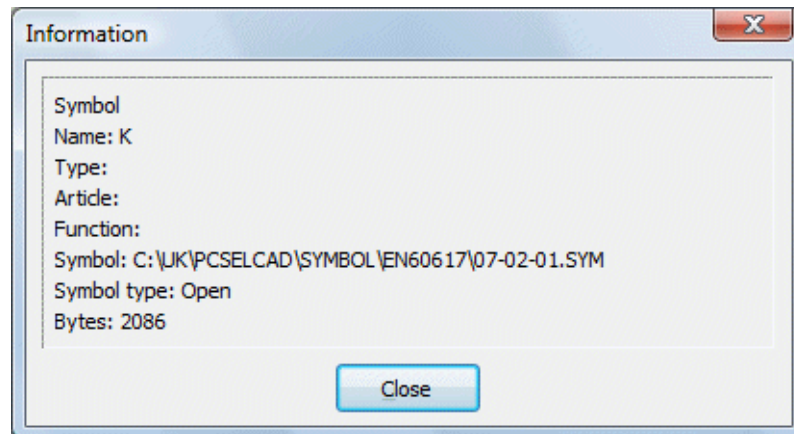
Delete a menu

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**.

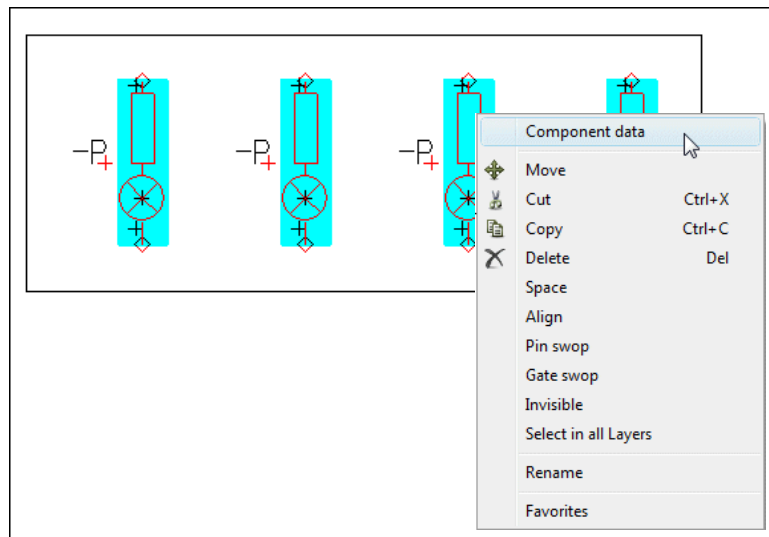


CHANGING SYMBOL NAMES IN AN AREA

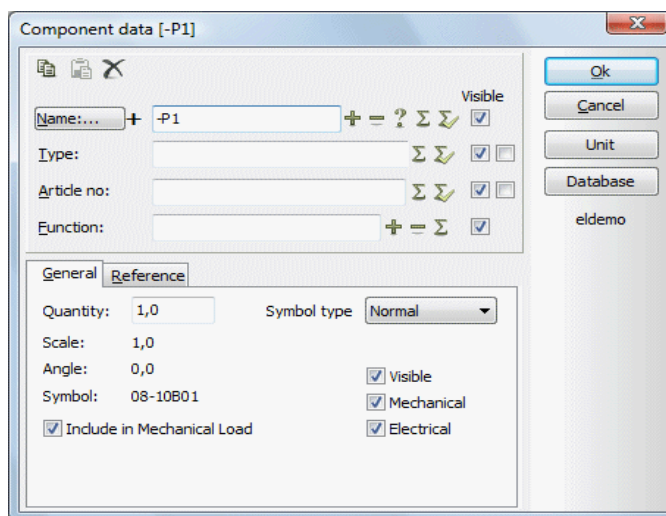
THIS CHAPTER DESCRIBES	
— CHANGING MORE THAN ONE SYMBOL NAME	P. 185
— THE NAMING ORDER OF SYMBOLS	P. 186
— OTHER OPTIONS	P. 186

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**.

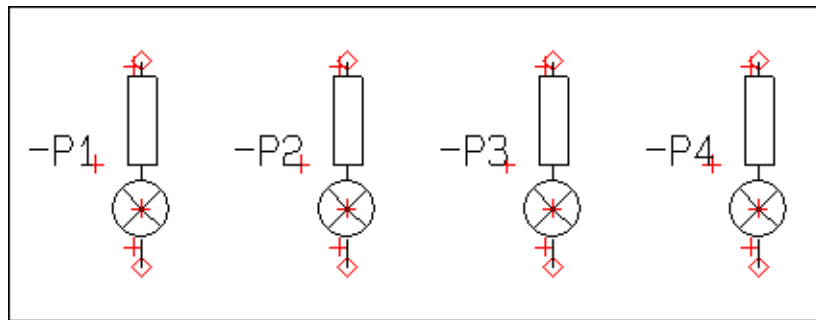


You then enter the **Component data** dialog box.



? 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 158 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 189.

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 108.

Other advanced ways to change texts directly on diagrams are described in “Locking text types” on page 119 and “Counting on texts” on page 120.



RENAME SYMBOLS - OR INSERT SYMBOL NAMES

IN “RENAME SYMBOLS” ON PAGE 289 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 295.

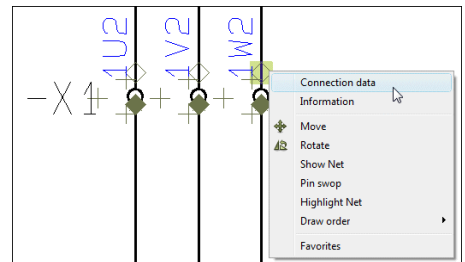
SYMBOL CONNECTION POINTS

THIS CHAPTER DESCRIBES	
— CONNECTION POINTS FOR SYMBOLS CAN BE SELECTED.....	P. 187
— SPECIFYING CONNECTION DATA	P. 187
— EDITING CONNECTION TEXTS FROM COMPONENT DATA DIALOG BOX	P. 188
— EDIT NEXT/PREVIOUS FOR CONNECTION POINTS	P. 188

Connection points for symbols can be selected

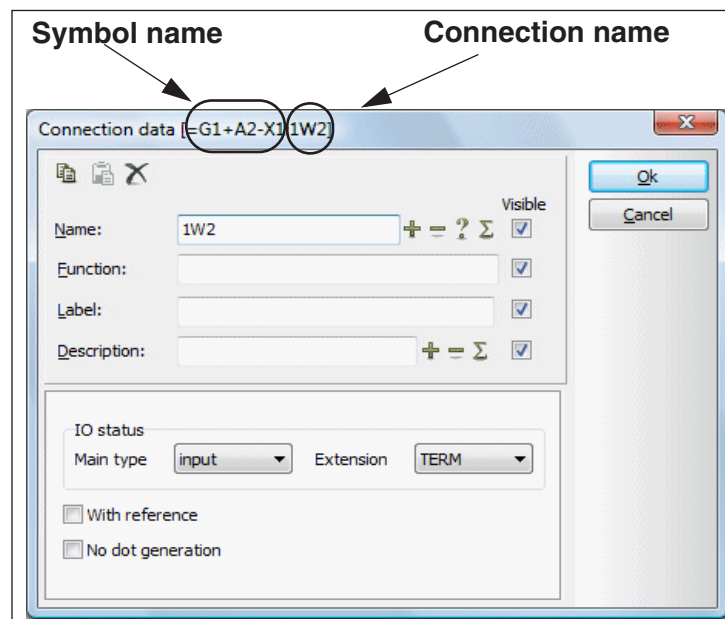
The connection points for the symbols can be selected as well as the symbols themselves.

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 109.

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 127 for further details.

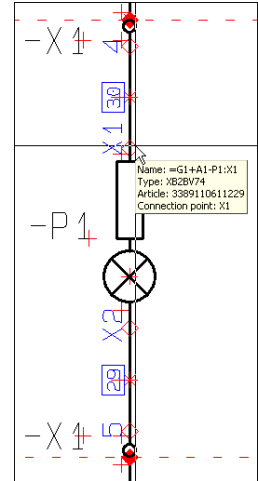
The function of the four buttons in the dialog box, is described in “Specifying the Component name” on page 158.

With reference

— See “With reference” on page 701.

No dot generation

— See “No dot generation” on page 701.

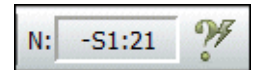


Connection data as Info Tips

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 162.

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.

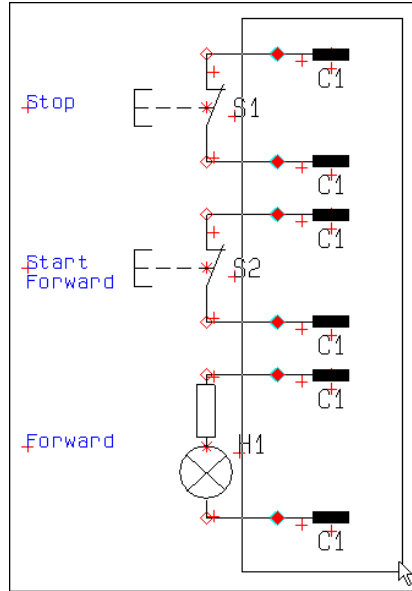
The function also applies for symbol texts.

CHOOSING CONNECTION POINTS IN AREAS

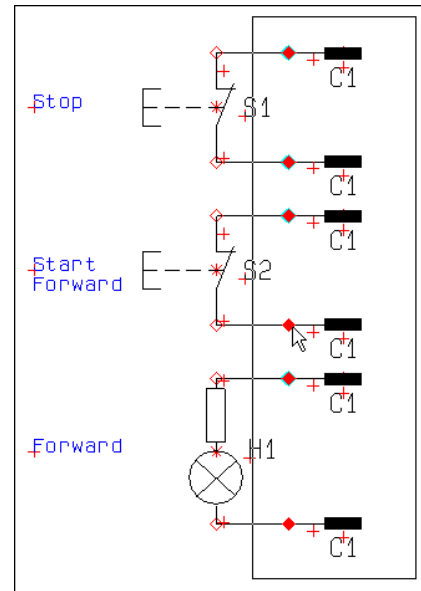


When you have activated the **Symbols** function, start dragging a window around an area and press <Ctrl> when releasing the mouse button, you select the connection points in the area. The *symbols* in the area are not selected.

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**.



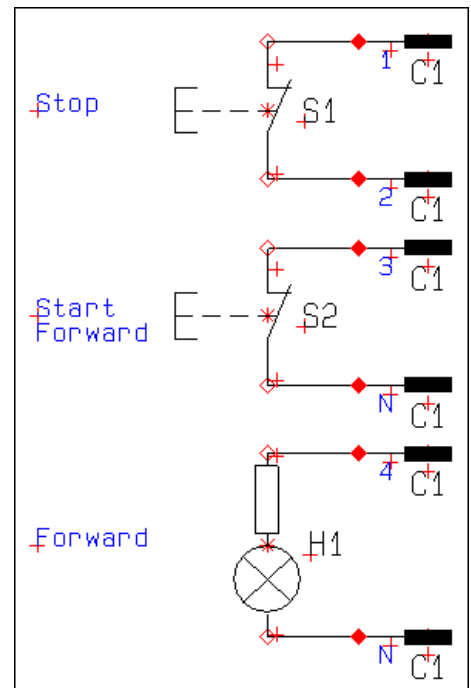
Then apply the automatic counting facility by typing in the first **Name (1)**, pressing <Ctrl> while clicking on +, and then click **OK**. The selected connection points are then renumbered automatically.

— Please refer to “Automatic counting in the Component data dialog box” on page 158 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.



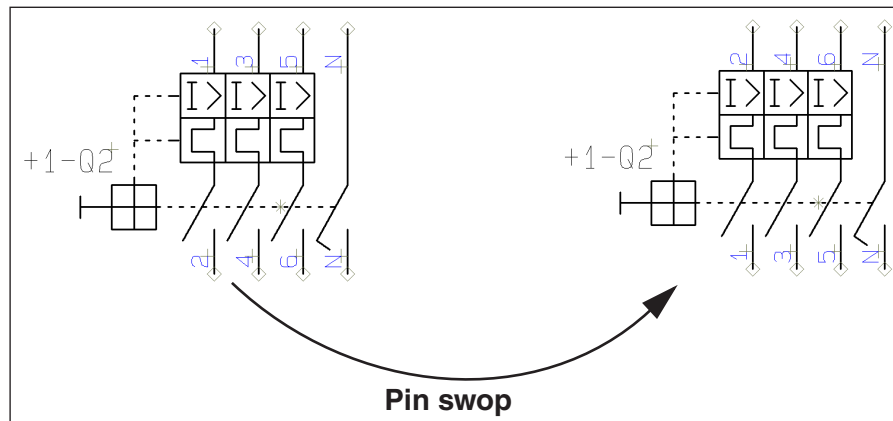
PIN SWOP

Letting the connection points of symbols swop places is an option, under the name *pin swop*. This can be done within the same symbol, as well as between symbols.

THIS CHAPTER DESCRIBES	
— PIN SWOP FOR SYMBOLS	P. 190
— PIN SWOP FOR SYMBOLS IN AREAS	P. 190
— FREE PIN SWOP	P. 191
— MOVING CONNECTION POINTS	P. 191

Pin swop for symbols

To perform a pin swop on a symbol with an even number of connection points, you first click the **Symbols** button, and right-click on the symbol. Then choose **Pin swop**, and the two connection points of the symbol swop places.



PIN SWOP - FOR MANUALLY CREATED SYMBOLS

WHEN YOU HAVE CREATED YOUR OWN SYMBOLS, PLEASE REMEMBER TO FOLLOW THESE SIMPLE GUIDELINES:

IN THE DEFINITION OF THE SYMBOL IN THE *SYMBOL LIBRARY* YOU MUST MAKE SURE THAT THE CONNECTION POINTS AT THE ONE SIDE OF THE SYMBOL ONLY CONTAIN EVEN NUMBERS, WHILE THE CONNECTION POINTS AT THE OTHER SIDE ONLY CONTAIN ODD NUMBERS.

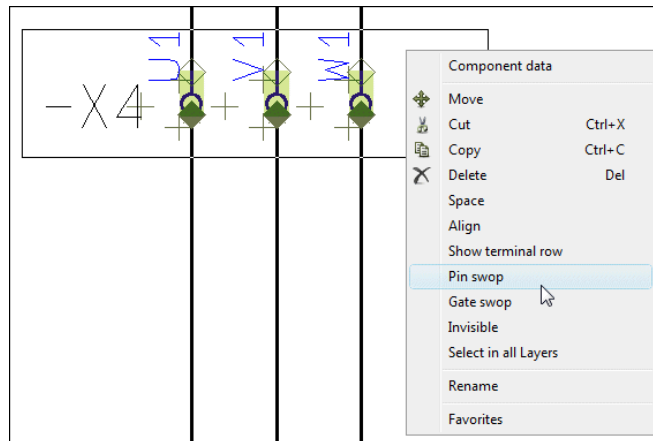
PIN SWOP WORKS FOR ALL OF THE SYMBOLS THAT ARE INCLUDED IN THE PROGRAM.

Pin swop for symbols in Areas

To perform a pin swop on all symbols with an even number of connection points within an area, you also click on the **Symbols** button first, select an area, and right-



click inside the area. Then choose **Pin swop**, and the connection points of the symbols inside the area swop places.



In this way you can for instance easily change a terminal row, from having the output side turning upwards to turning downwards. — Please refer to “Creating terminal symbols” on page 717 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.

Connection points for all symbols inside the area with an odd number of connection points, remain unchanged.

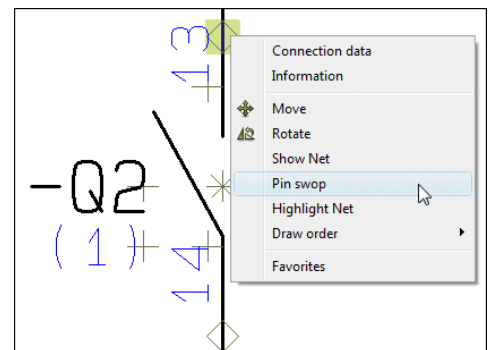
The function can also be activated by choosing **Edit => Pin swop**.

Free pin swop

Swapping connection points between symbols, and within symbols having more than two connection points, is also an option.

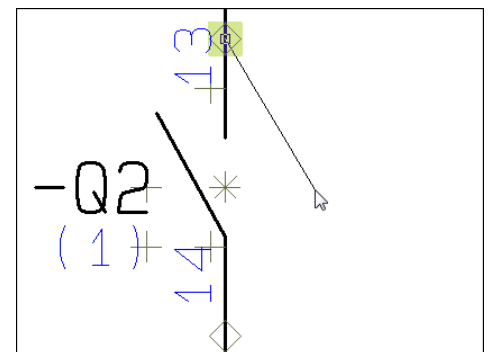


To do this, you also have to activate the **Symbols** button. Then right-click on one of the two *connection points*, you want to swop places.



Now choose **Pin swop**, 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).



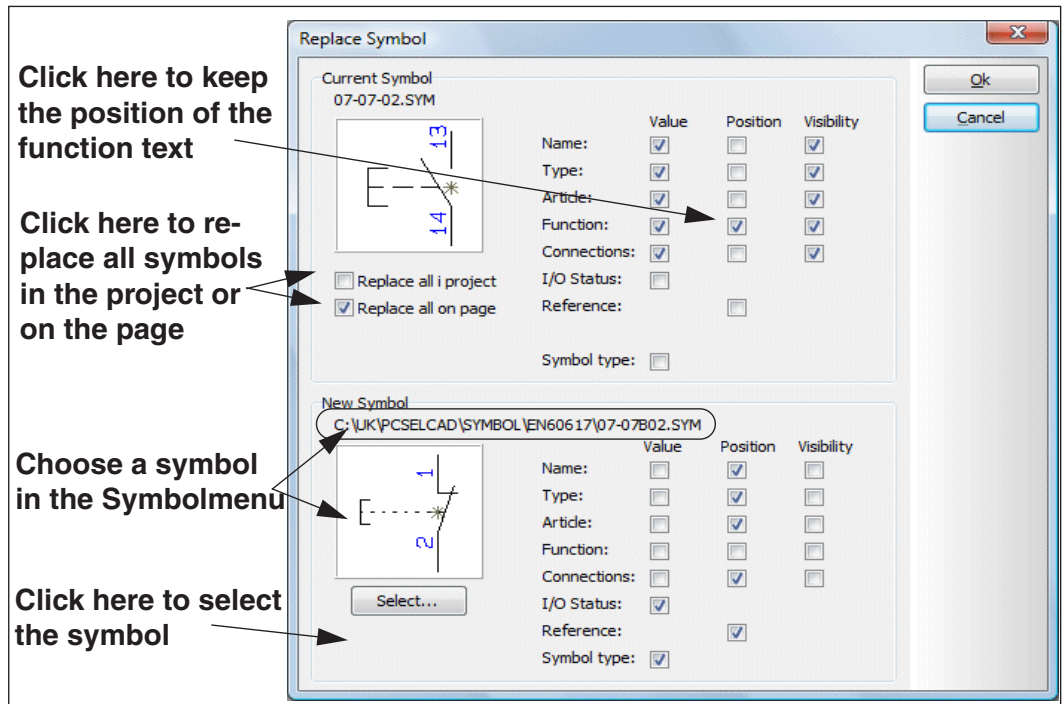
REPLACING SYMBOLS

THIS CHAPTER DESCRIBES	
— REPLACING SYMBOLS AUTOMATICALLY	P. 193
— CONNECTION POINTS WHEN REPLACING SYMBOLS	P. 195
— TEXT PROPERTIES WHEN REPLACING SYMBOLS	P. 195
— REPLACING OLDER VERSIONS OF SYMBOLS	P. 195
— EDIT A SYMBOL - AND REPLACE IT ON THE PROJECT PAGES	P. 195

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 Symbol type 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 symbol type for the old symbol, or to use the symbol type 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

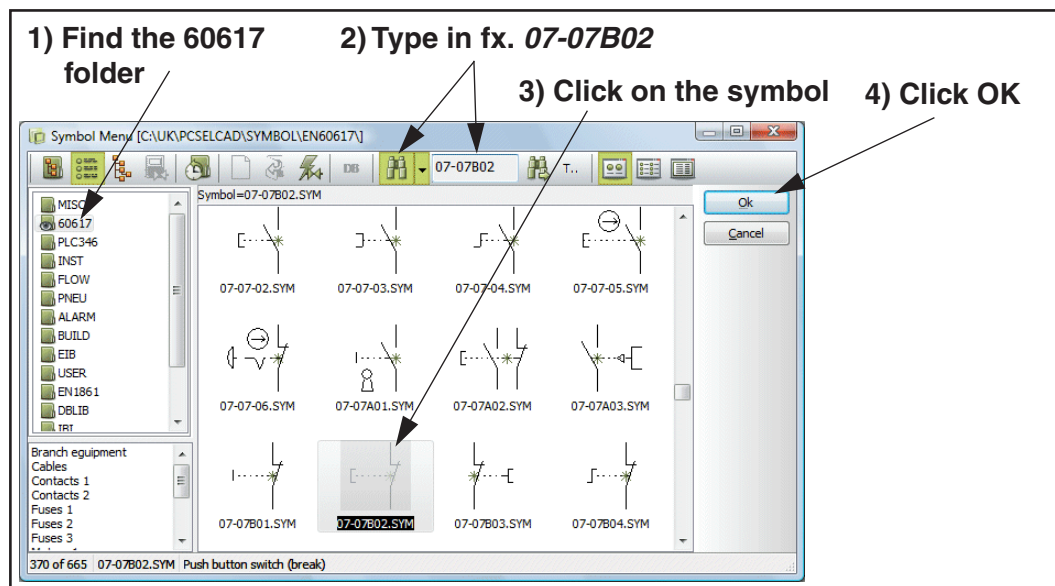
Select **Replace all in project** in order to replace all symbols, of the selected type, in the project. If you wish to replace all symbols of the selected type on the active page in the project, select **Replace all on page**.

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 144.

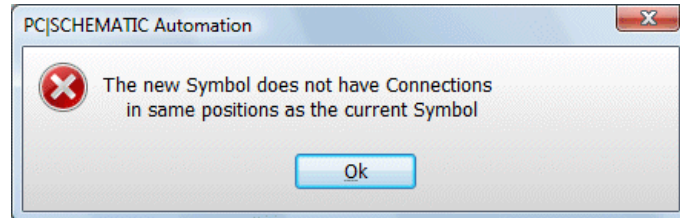
You now return to the dialog box **Replace Symbol**, where you click **OK**.

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.

Edit a symbol - and replace it on project pages

To edit a symbol used in the project, and subsequently replace all occurrences of the symbol in the project, do this:

- 1) Right-click on the symbol on a project page, and choose **Edit symbol**.
- 2) In the *Edit symbol* mode: You can now edit the symbol — as described in “Creating Symbols” on page 696.
- 3) When the symbol has been edited and saved: Select **Files => Replace symbol**.
- 4) In the **Replace symbol** dialog box: You can now replace the old version of the symbol with the new version — as described in “Replacing symbols” on page 193.

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. 197
— AUTOMATIC CREATION OF SYMBOLS — BY SYNTAX	P. 199
— SYNTAX FOR DIRECT CREATION OF SYMBOLS.....	P. 200
— AUTOMATIC CREATION OF ELECTRICAL SYMBOLS FROM THE DATABASE	P. 201
— AUTOMATIC CREATION OF MECHANICAL SYMBOLS FROM THE DATABASE	P. 201

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 197. (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 696).
- 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 199.

Automatic generation of symbols — using the Symbol generator

You can enter the **Symbol generator** in four different ways:



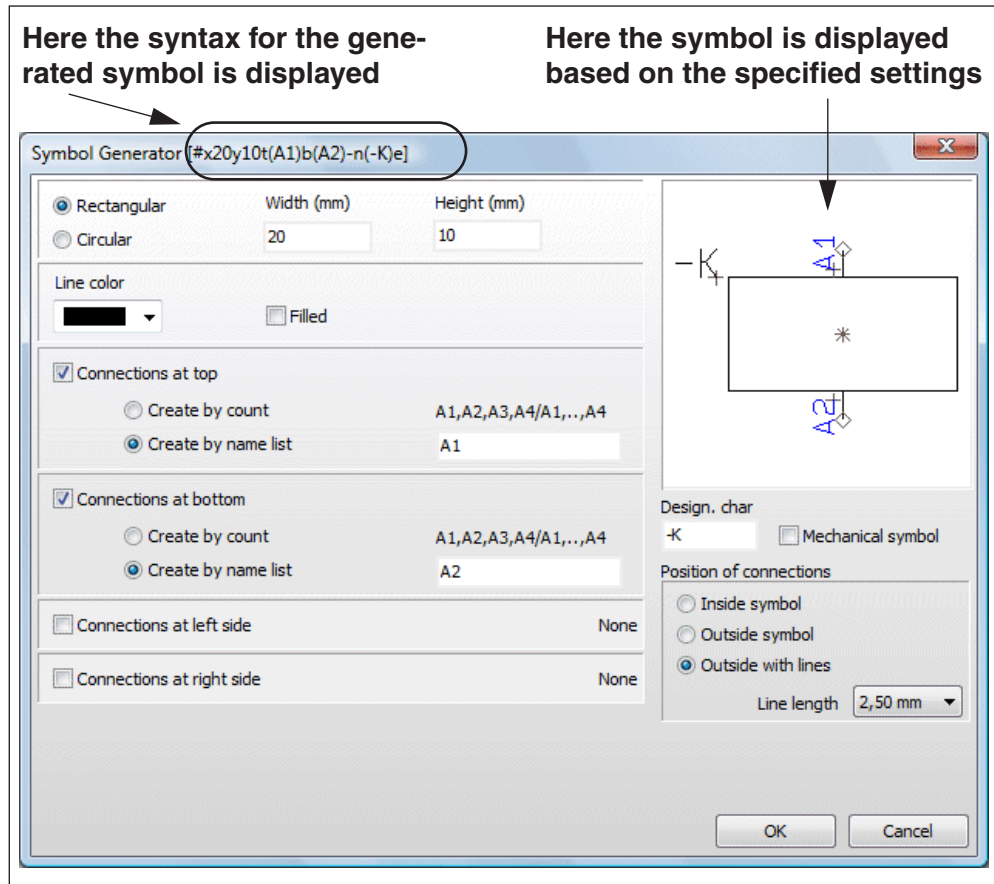
- a) Select **Symbols** and click the **Symbol generator** button.
- b) Select **Functions => Symbol generator**.
- c) Select **Symbols** and click the **Symbol menu** button. In the **Symbol menu**: Click the **Symbol generator** button.
- d) When editing symbols: Click the **Symbol generator** button.

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**.

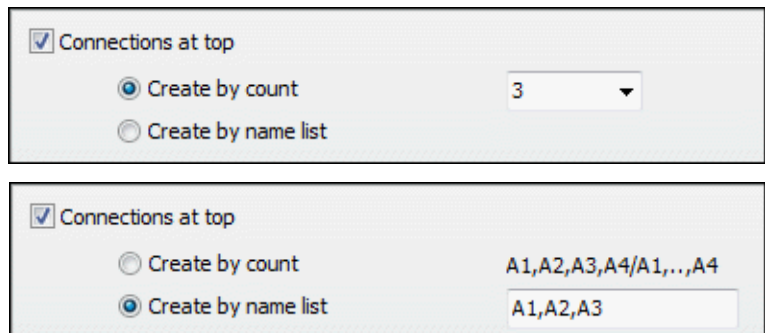
LINE COLOR AND FILL COLOR

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.



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 352.

ELECTRICAL OR MECHANICAL SYMBOL

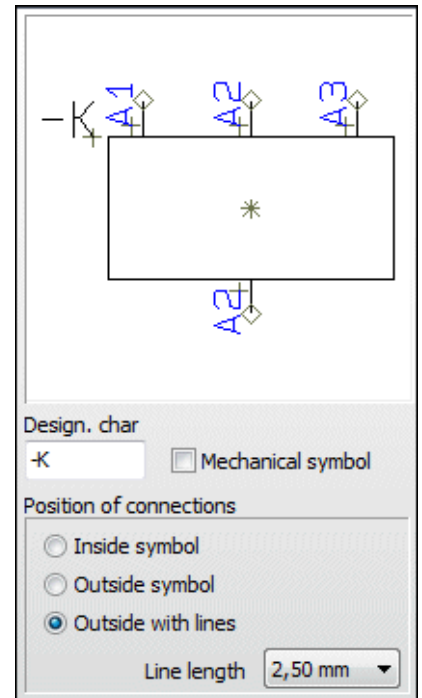
By default the symbol is created as an electrical symbol, but if you select **Mechanical symbol**, it is created as a mechanical symbol.

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**.

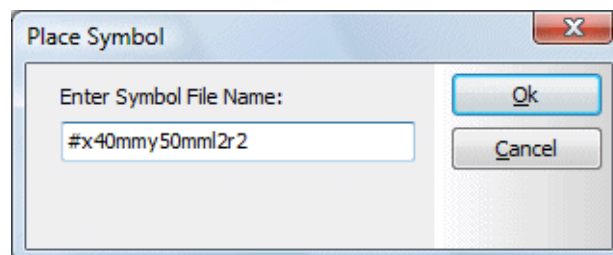


Automatic creation of electrical and mechanical symbols - By syntax

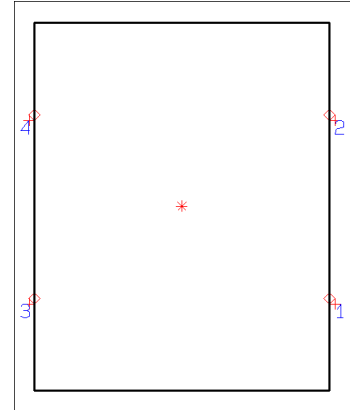
When you need to create a simple symbol quickly, do the following:



- 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 200.
- 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.

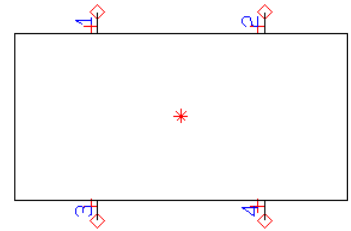
Thus you can write **#x40mmy50mmt(13,21)b(14,22)**. Hereby you get a symbol with the dimensions 40x50, with the connection points **13** and **21** on the top, and the connection points **14** and **22** at the bottom.

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 200.

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.

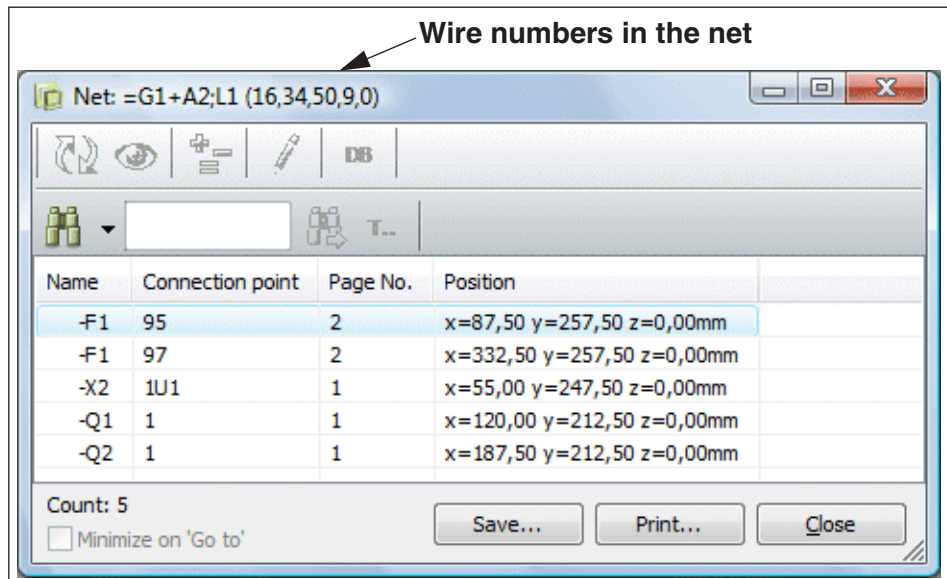
Creating a Mechanical symbol automatically

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 200.

— Read more about mechanical symbols in "Mechanical symbols" on page 603.

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 649.
- See also “Connections list for wire numbering” on page 402.

SIGNAL SYMBOLS

THIS CHAPTER DESCRIBES	
— WHAT IS A SIGNAL SYMBOL	P. 203
— THE APPEARANCE OF SIGNAL SYMBOLS	P. 204
— DESIGNING SIGNAL SYMBOLS	P. 204
— PLACING SIGNAL SYMBOLS WHEN DRAWING LINES.....	P. 204
— SIGNAL SYMBOLS WITH REFERENCE DESIGNATIONS	P. 207
— PLACING SIGNAL SYMBOLS ON PARKED LINES	P. 208
— PLACING SIGNAL SYMBOLS DIRECTLY	P. 208
— VARIANT NAMES ON SIGNAL SYMBOLS	P. 209
— INSERT REFERENCE DESIGNATIONS ON SIGNALS AUTOMATICALLY	P. 210
— UNREFERRED SIGNAL SYMBOLS	P. 210
— IMPLIED SIGNALS FOR TERMINALS AND CABLES	P. 211

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.

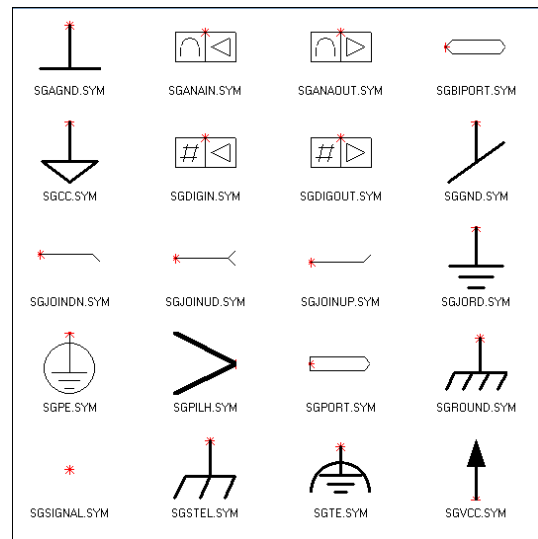
— See “Automatic insertion of electrical potentials” on page 81.

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 738.

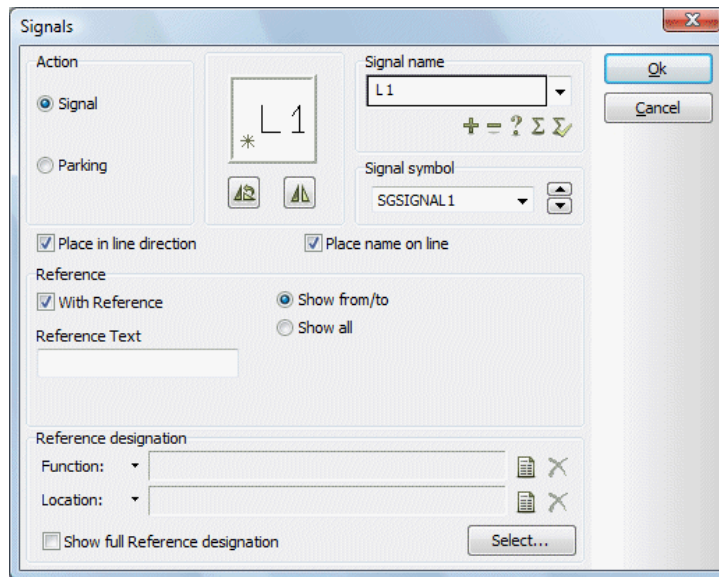
Placing signal symbols when drawing lines



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 320.

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
	<i>INCREASE THE NUMBER IN THE SIGNAL NAME BY ONE.</i>
	<i>DECREASE THE NUMBER IN THE SIGNAL NAME BY ONE.</i>
	<i>GIVES YOU THE NEXT AVAILABLE SIGNAL NAME OF THE SYMBOL TYPE CHOSEN.</i>
	<i>GIVES YOU A LIST, WHERE YOU CAN CHOOSE BETWEEN THE SIGNAL NAMES ALREADY USED IN THE PROJECT.</i>
	<i>GIVES YOU A LIST OF USED VARIANT NAMES FOR THE SELECTED SIGNAL SYMBOL — SEE “VARIANT NAMES ON SIGNAL SYMBOLS” ON PAGE 209.</i>

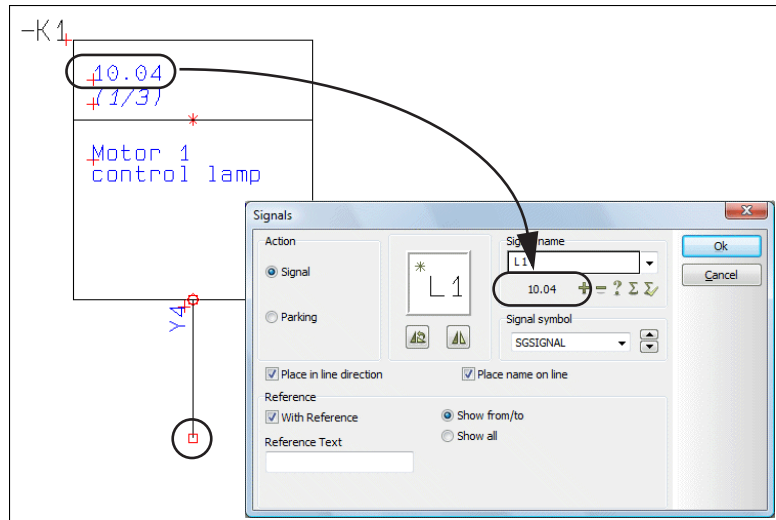
You can also click on the down-arrow in the field **Signal name**. This gives you the option to choose between predefined signal names.

— See the section “Frequently used signal symbol Names” on page 741 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 740.

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

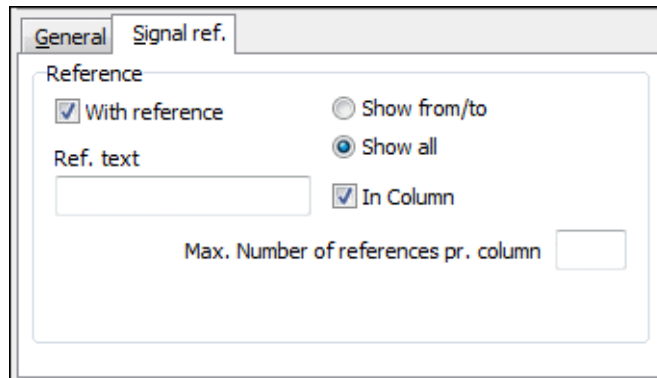
When you select **Place in line direction**, the signal symbol will follow the direction of the line. When you also select **Place name on line**, the name of the signal symbol will be displayed above the line. The program memorizes the selected settings for the different signal symbols in the project — 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:

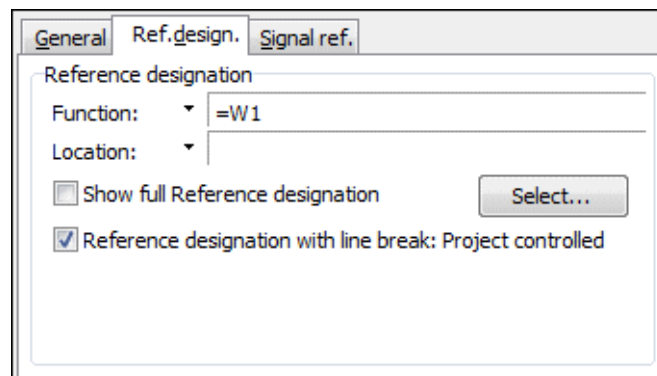
- a) Click on **Select**, and you get the option of selecting references via the dialog box **Reference designations**, as described in “Reference designations for symbols” on page 349, or:



- 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 content with the settings, click **OK**, otherwise click **Cancel**.

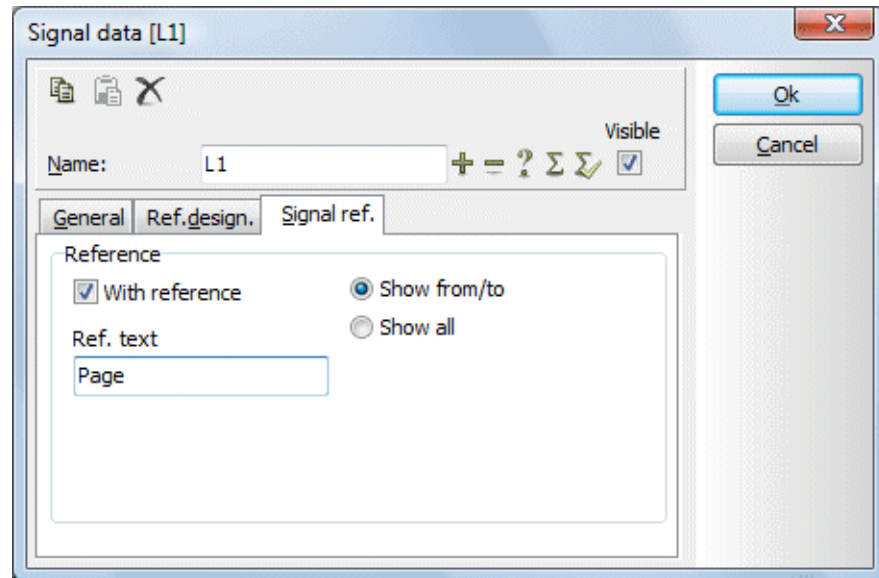
Placing signal symbols on parked lines



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 139.

When you have the signal symbol in the cross hairs, you can place it on a parked line. — See “Parking” on page 204.

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.

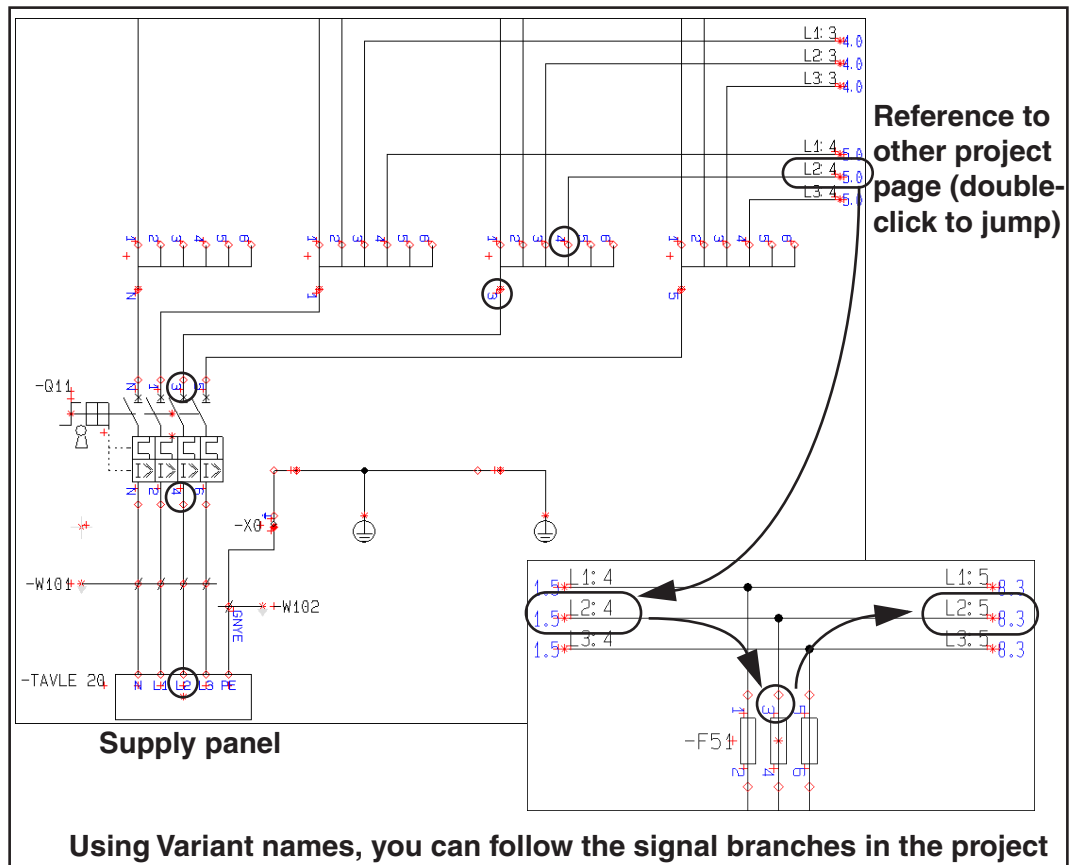
This being for instance the source point from which three signals with the name **L1** branches off. These three branches can for instance be named **L1:1**, **L1:2** and **L1:3**.

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

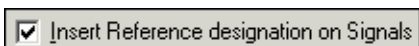
symbols with identical names” on page 238.

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 224.

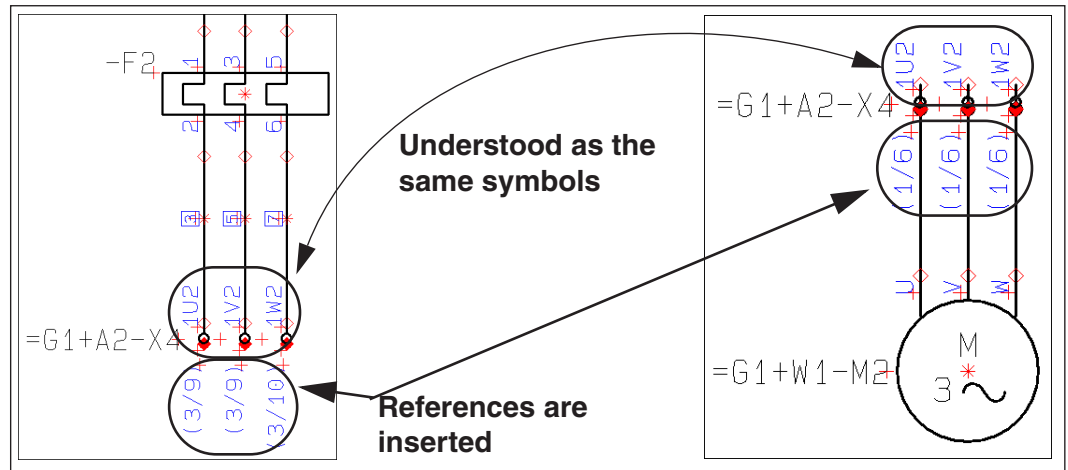
Implicit references for terminal and cable connection points

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

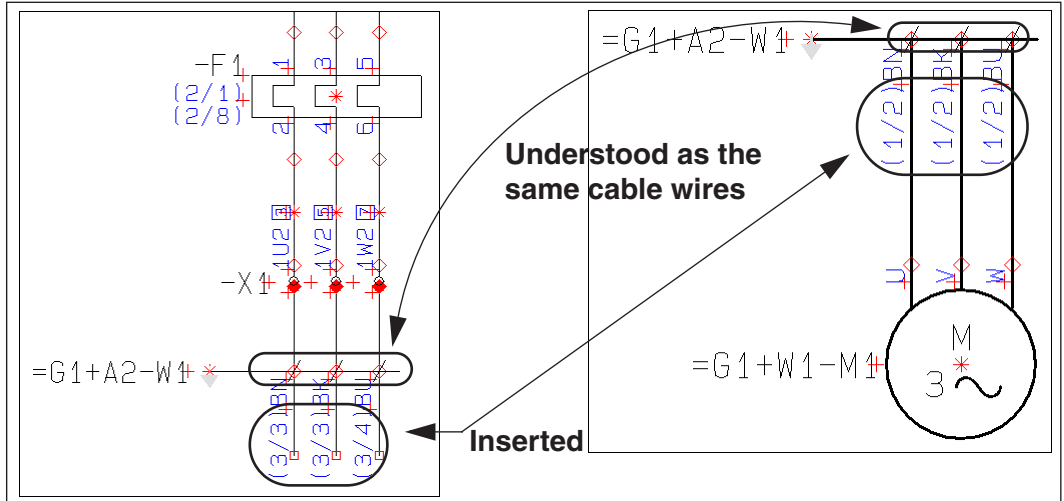
3) Both sets of terminals are only connected in one end

**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**

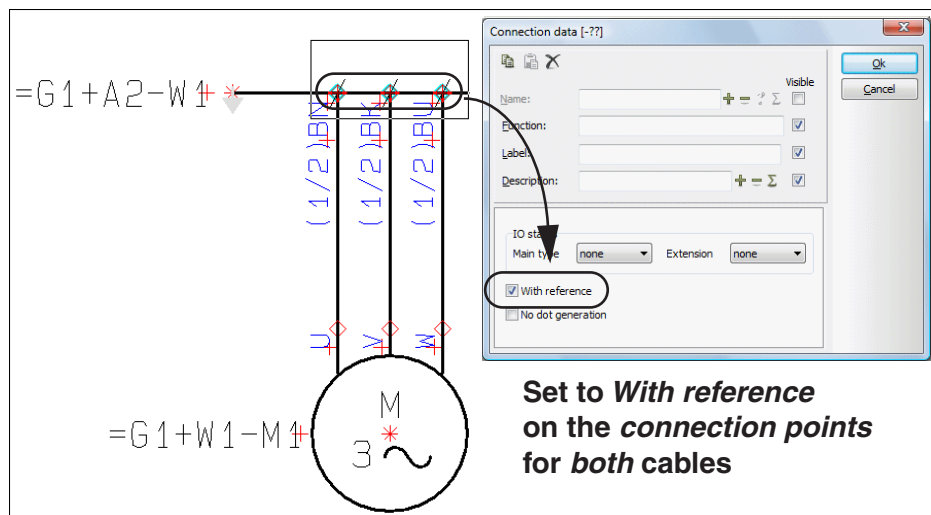
Implicit signals on cable wires

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



Set to With reference on the connection points for both cables

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 759.

— See “Multi layer terminals” on page 215.

— See “Multiplacing symbols” on page 168.

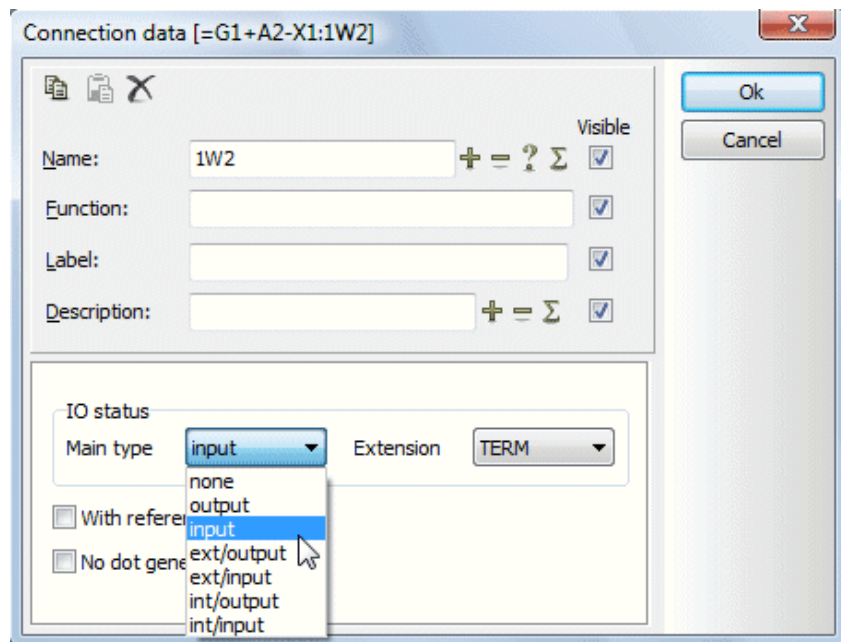
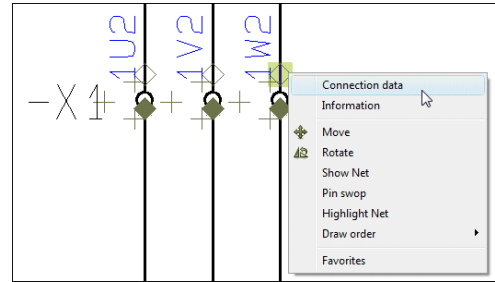
Input and Output on terminal symbols

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 additional symbol types” on page 720, “Terminal symbols with additional symbol type PLC” on page 721 in particular.

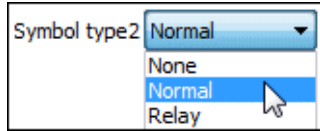


For terminals **Extension** is set to **TERM** — or to **TERM/PLC**, see “Creating terminal symbols with additional symbol types” on page 720, “Terminal symbols with additional symbol type PLC” on page 721 in particular.

If you accidentally select the entire symbol instead of the connection point, then right-click on the connection point once more.

Different terminal connection names on the input and output side

When you have placed a terminal symbol on a project page, and wish to use different connection names on the input and output side, do this:



1) Select the terminal: Choose **Symbols** and double-click on the terminal symbol.

2) In the dialog box **Component data**: Set **Symboltype2** to *Normal* and click **OK**.

3) You can now double-click on the connection points and name them individually via the **Connection data** dialog box.

Changing connection names for terminal rows

See “Renumbering pinnumbers for a terminal row using Select name” on page 241.

Designing terminal symbols

See how to design terminal symbols in “Creating terminal symbols” on page 717.

MULTI LAYER TERMINALS

THIS CHAPTER DESCRIBES	
— ON MULTI LAYER TERMINALS	P. 215
— MULTI LAYER TERMINALS ON DIAGRAM PAGES	P. 216
— MULTI LAYER TERMINALS ON MECHANICAL PAGES	P. 217

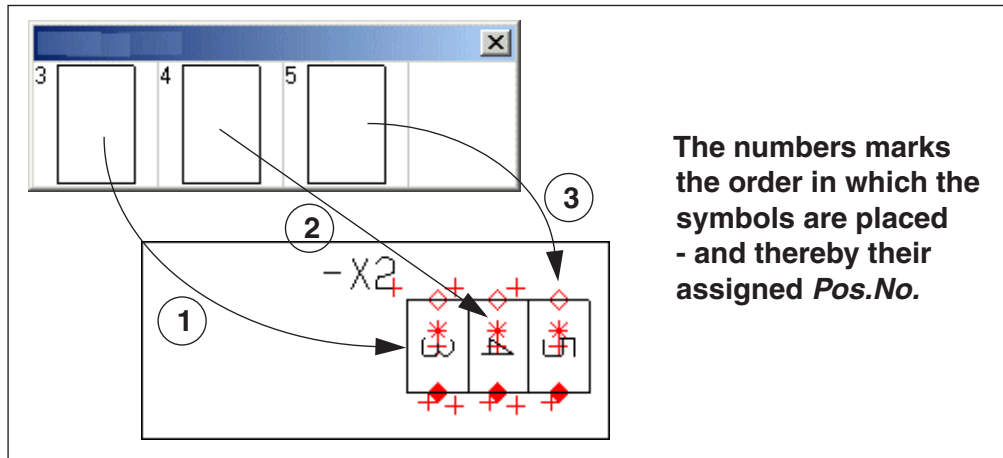
On multi layer terminals

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.

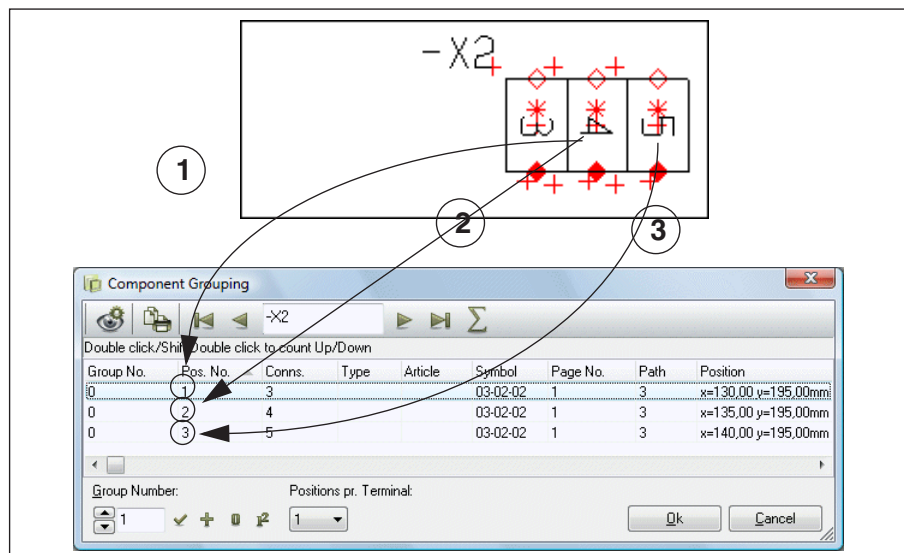
— See “Component Grouping” on page 556.

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.

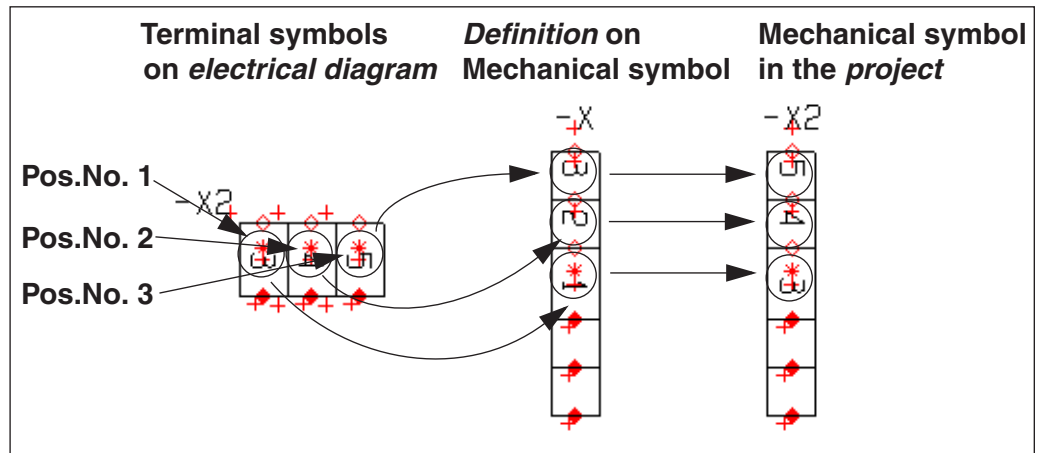


The position numbers can be specified/changed in **Functions => Component Grouping**.



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 722 and “Creating busbar symbols” on page 724.

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.

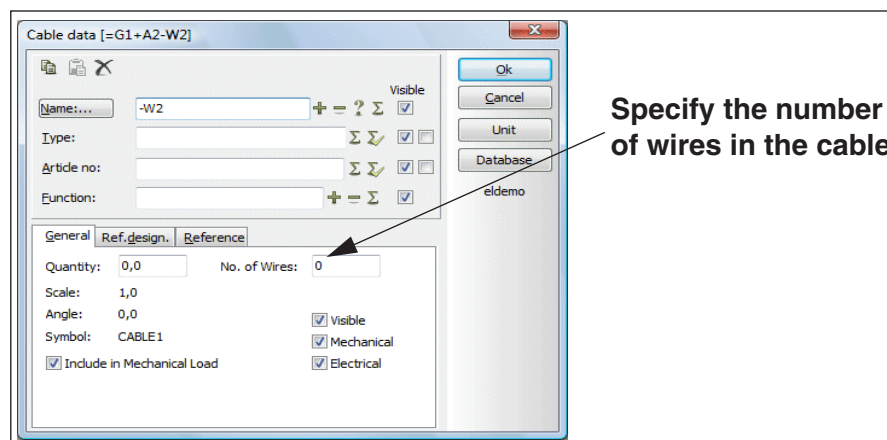
When you work with cables you enter different dialog boxes when selecting the cable wires, depending on whether the system has cable wire names for the cable or not.

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. 218
— PLACING CABLE SYMBOLS WITH WIRE NAMES FROM THE DATABASE	P. 221
— PLACING SYMBOLS FOR THE SAME CABLE IN ONE OPERATION	P. 222
— MOVING CONNECTION POINTS FOR CABLE WIRES	P. 222
— ADDING CABLE WIRES TO PLACED CABLES	P. 223
— REMOVING CABLE WIRES FROM PLACED CABLES	P. 223
— ADDING A SCREEN ON A PLACED CABLE	P. 224
— REVERSE DIRECTION ON PLACED CABLES	P. 224
— UNREFERENCED CABLE SYMBOLS.....	P. 224
— SETTINGS FOR DEFAULT CABLE DIRECTION	P. 225

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.



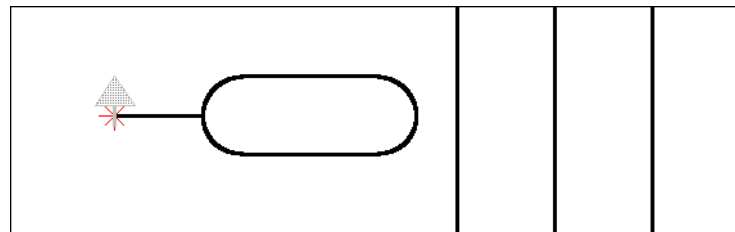
NUMBER OF WIRES IN COMPONENTS LISTS

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 156. Click **OK**.

The illustration below is a part of a larger drawing, where the three vertical lines are wires.



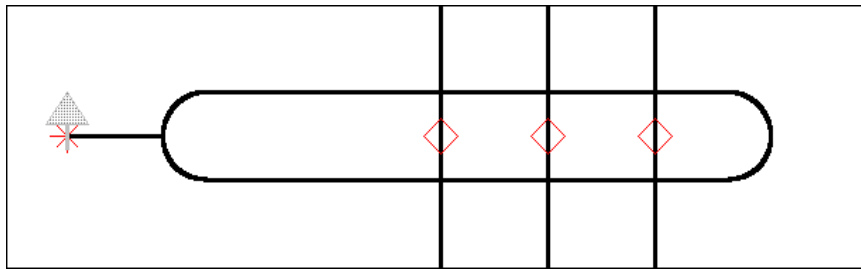
When you place the cable, a rubber band now starts in the reference point of the symbol, and ends up in the cross hairs. This indicates that you can now point out which wires to include in the cable.

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**.

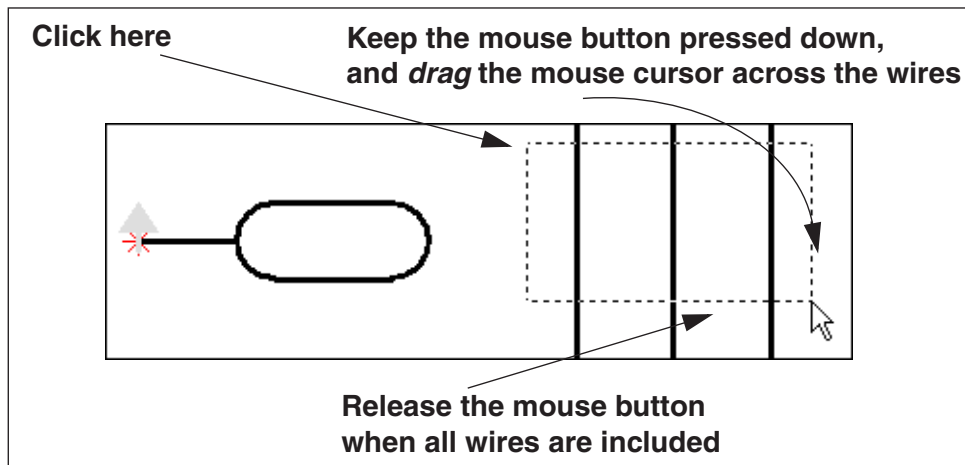
When you click **OK** the symbol expands to include this wire as well.



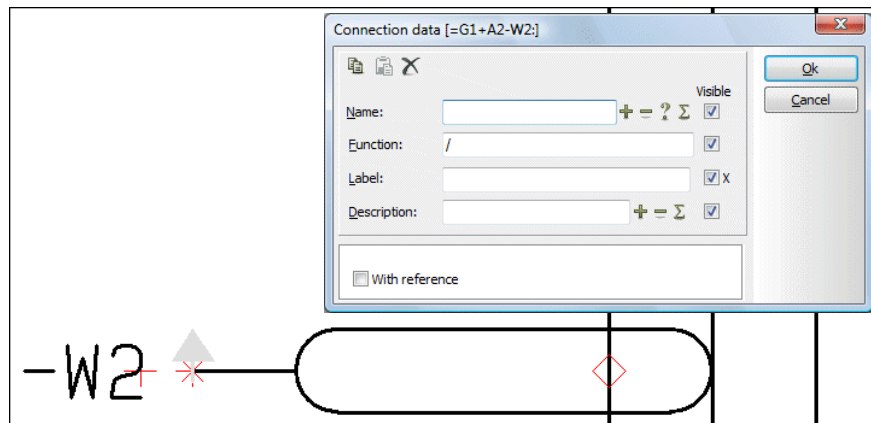
When you do not wish to include more wires into the cable you press <Esc>, and the "rubber band" disappears. If the number of cable wires is specified in the dialog box **Cable data**, the "rubber band" disappears when the specified number of cable wires have been placed in the project.

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 176.

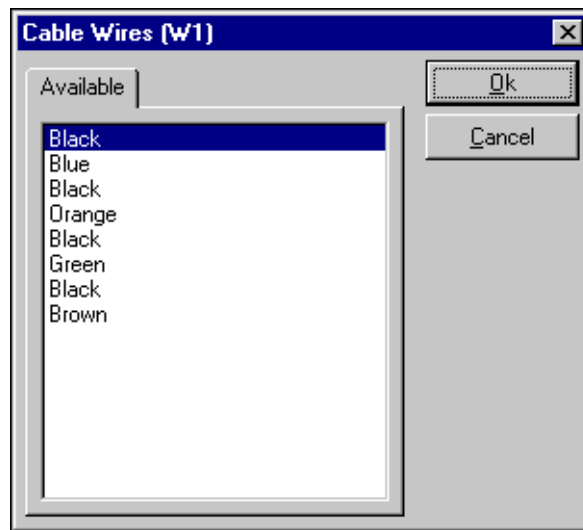
When you are using a cable with specified wire names from the database, the selected wires are automatically assigned wire names according to the order in which the wires appear in the database.

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 218.

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**.

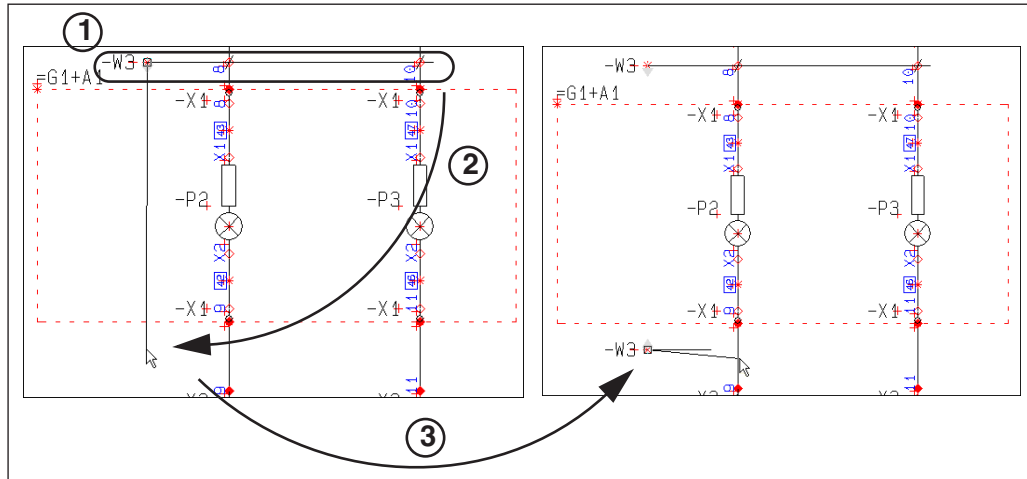
— Please note that it is also possible to choose cable wires from the tab **Used**. This makes it possible to display the cable wire more than one place in the project.

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

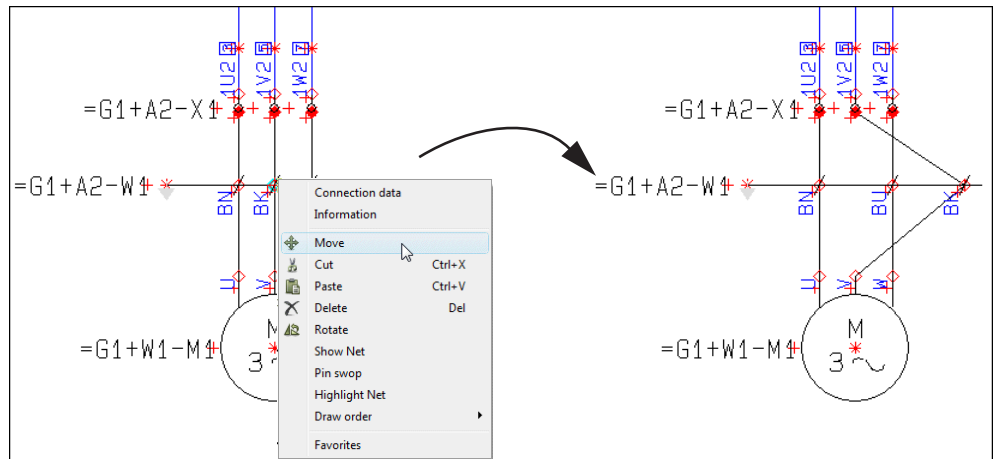
Moving connection points for cable wires

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 221.

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 220.

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 722.

To add a cable screen to a placed cable symbol, do this:



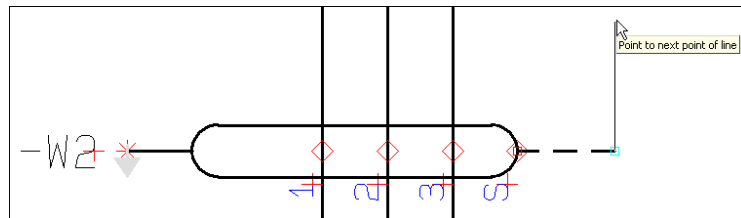
1) Click the **Symbol** button — to work with symbols.

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 73.

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 210.

Default Cable direction

In **Settings => Text/Symbol default** you can specify the cable direction as an over-all setting, vertically as well as horizontally.

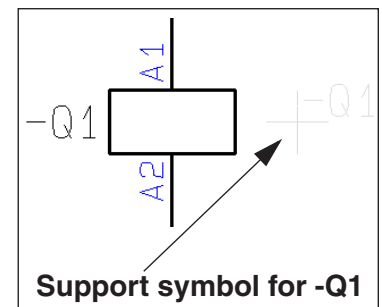
USING ACCESSORY SYMBOLS

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.

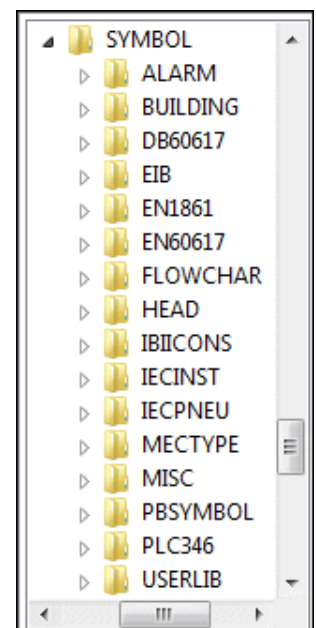
Data from accessory symbols are included in the lists, but the accessory symbol itself is not included when printing the diagram pages.

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 836.</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. 229
— FETCHING SYMBOLS FOR A COMPONENT VIA THE DATABASE	P. 230
— UPDATING FROM THE DATABASE WHEN CHANGING ARTICLE NO.	P. 231
— THE FUNCTION SHOW AVAILABLE.....	P. 232
— THE FUNCTION SHOW AVAILABLE WINDOW	P. 234
— FINDING SYMBOLS BASED ON THE SYMBOL NAME	P. 238
— SELECT COMPONENT.....	P. 242
— PLACE AVAILABLE SYMBOLS FOR A COMPONENT	P. 242

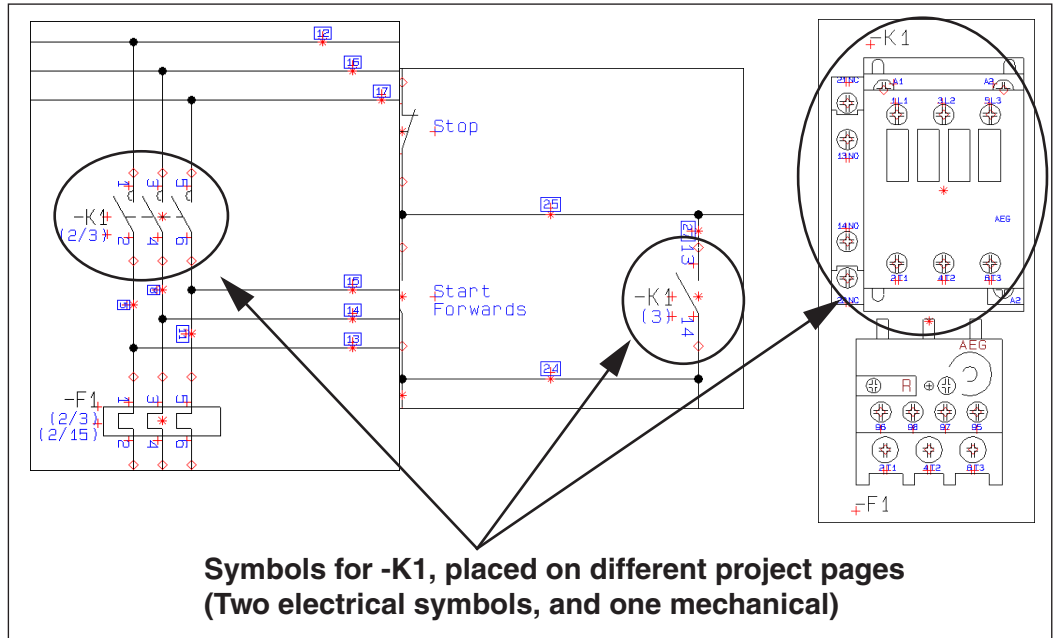
- For other ways to get a survey of the symbols in the project, please refer to “The Object Lister” on page 306.
- See also “Fetching symbols” on page 139 and “Placing and Naming symbols” on page 156.

THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS

Components containing more than one function, have an electrical symbol for each function in the component.

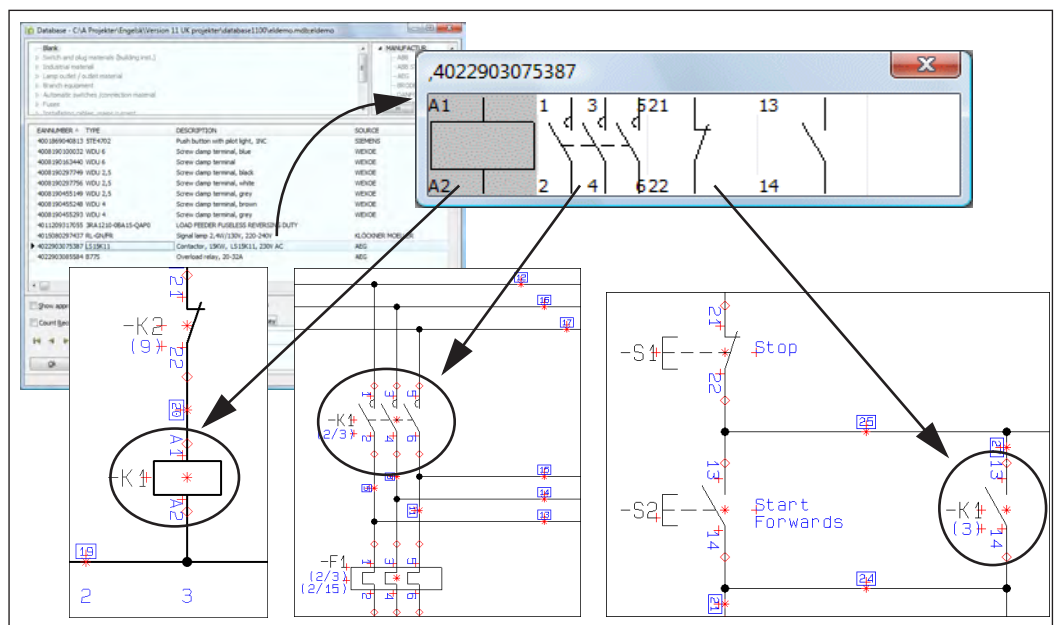
These electrical symbols can be placed freely in the diagram pages in the project.

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 230.

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 230.



— See “Update from Database when changing symbol Article no” on page 231 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 165 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 571.

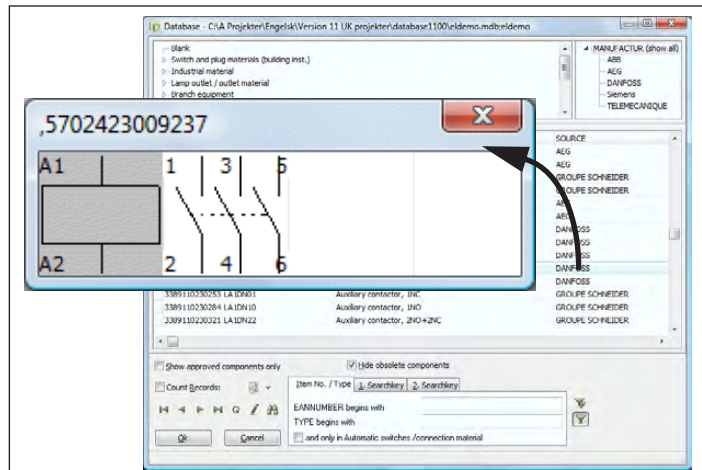
Practically all components in the component vendor databases for **Automation** also have an attached mechanical symbol, which can be used for mechanical layouts. — Read more about this in “Placing mechanical symbols via the database” on page 628.

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 152.

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 571.



UPDATE FROM DATABASE WHEN CHANGING SYMBOL ARTICLE NO

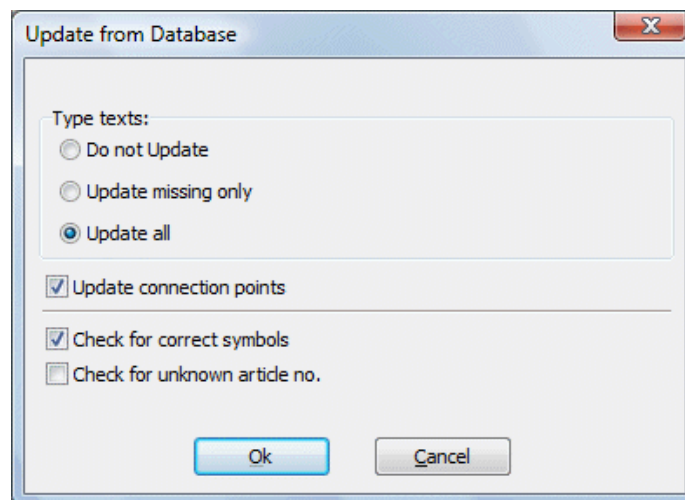
When the **Article no** is changed for one of the symbols for the component, it is possible to have the component updated from the database, as described in “Update from database when changing article number” on page 159.

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 171.



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 171.

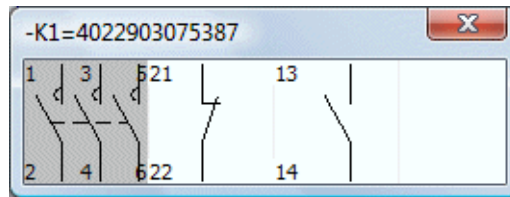
THE FUNCTION SHOW AVAILABLE

When you place a symbol via a pickmenu from the database, the pickmenu disappears just after you select a symbol from it. (However, this is not the case if the **Show available window** is displayed permanently on the screen. In that case **Show available** remains visible at the bottom-right corner of the screen — see “The Show available window” on page 234).

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 Show available window” on page 234.

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>.

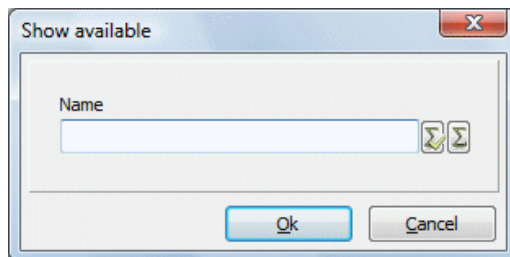
Show available again

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 PLCs 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 550.

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 639.



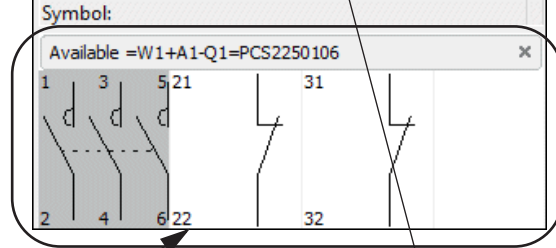
THE SHOW AVAILABLE WINDOW

SEE VIDEO ON OUR WEBSITE

The **Show available window** gives you an overview of all used components in the project, that still has available/unused symbols (component functions).

Show available window (Shift+F9)

Name	Article	Type	Func...
=W1+A1-F1	PCS2250001	PCSMV001A	
=W1+A1-F2	PCS2250001	PCSMV001A	
=W1+A1-Q1	PCS2250106	PCSCON06	
=W1+A1-Q2	PCS2250106	PCSCON06	
=W1+A1-Q3	PCS2250108	PCSCON08	
=W1+A1-Q5	PCS2250108	PCSCON08	
=W1+A1-Q7	PCS2250106	PCSCON06	
=W1+A1-Q9	PCS2250106	PCSCON06	
=W1+A1-S2	PCS2250411	PCSPB NC 01	
=W1+A1-S4	PCS2250411	PCSPB NC 01	
=W1+A1-S5	PCS2250411	PCSPB NC 01	
=W1+A1-S7	PCS2250411	PCSPB NC 01	
=W1+A1-W1	PCS330002	PCSCBL02	
=W1+A1-W2	PCS330002	PCSCBL02	
=W1+A1-W7	PCS330002	PCSCBL02	
=W1+A1-W8	PCS330002	PCSCBL02	
=W1+A1-W9	PCS330002	PCSCBL02	
=W1+A1-W10	PCS330002	PCSCBL02	



Selected component
Available symbols for the selected component

The window only displays symbols relevant to the page that you are presently working on:

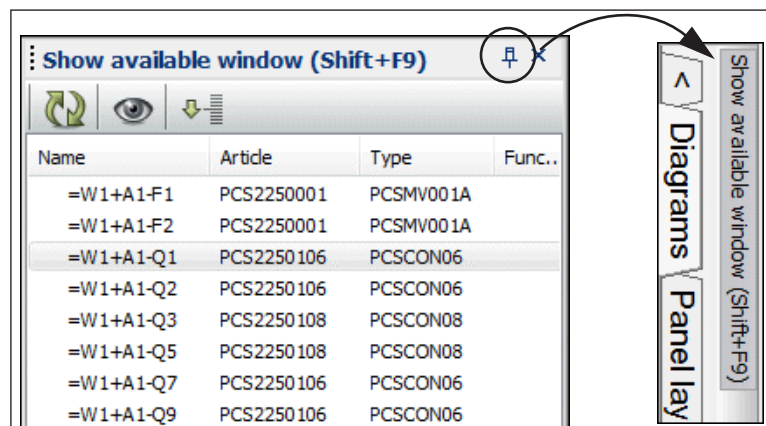
- When you are working on a *Diagram page*, all components with available/unused *electrical* symbols are displayed.
- When you are working on a *Mechanical page*, all components with available/unused *mechanical* symbols are displayed.
- If you have chosen to work with another type of symbols on the page, all components of these "other symbols" with available/unused symbols are displayed. See more about this option in "Other symbols" on page 612, "Specifying that you want to apply "other symbols" on a page" on page 613 in particular.

Positioning the Show available window

The **Show available window** is displayed automatically in the right-hand side of the screen, when you under **Settings => Pointer/Screen** activate **Auto-activate Show avail. window**.

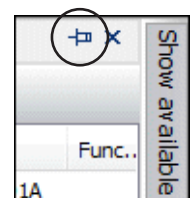
Dock, lock or unlock the Show available window

The window can be docked or moved as described in "Docking Windows and Tool-bars" on page 26. When the window is docked, you can also click the **Unlock panel** icon.



Hereby the **Show available window** is only displayed as a tab at the right-hand side of the screen. When you subsequently click the tab, the **Show available window** is displayed for some seconds, after which only the **Show available window** tab is displayed again.

In order to display the **Show available window** permanently on the screen, just click the **Unlock panel** icon in the **Show available window**.



Placing an available symbol for a component

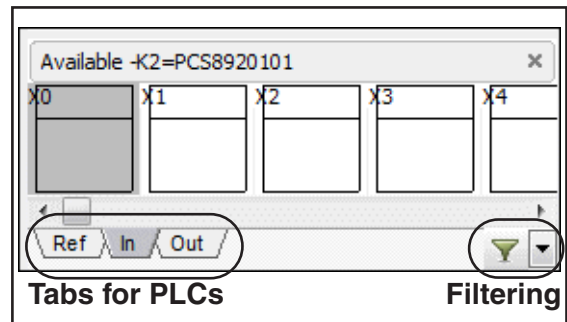
When you click on a component in the list in the upper part of the **Show available window**, you can see the available symbols for the component at the bottom of the window.

When you click on one of the available symbols for the component at the bottom of the window, you get the symbol in the cross hairs. You can now place the symbol in the project — with attached article data and component name.

Symbols for PLCs in Show available

For components with many attached symbols, there can be tabs at the bottom of **Show available**, so that you can access the symbols you need more easily.

For PLCs there can therefore be tabs for reference symbols, input and output.



FILTERING FOR DISPLAYING PLC SYMBOLS IN SHOW AVAILABLE



There is also a **Filter** button which is used for filtering which symbols to display in **Show available** for the PLC. The filter is activated/deactivated by clicking the **Filter** button. The settings for the filtering is specified by clicking the down-arrow to the right of the Filter button.

You can see that the filter is activated when the background colour for the Filter button is green.

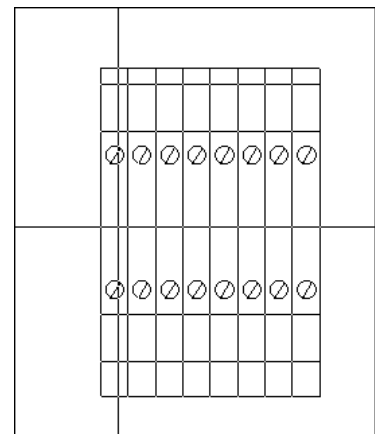
Selecting all available symbols for a component

You can also right-click on one of the symbols for the component in the symbol pickmenu at the bottom of the window, and choose **Select all**. This gives you all of the available symbols for the component in the cross hairs, and they are placed side by side, when you place them on a project page.

Select all - on mechanical pages

When you are working on a Mechanical page, the **Show available window** shows all components with available *mechanical* symbols in the project.

When for instance you use **Select all** to place terminal symbols on a mechanical page, you get all mechanical terminal symbols in the cross hairs, so that you can place them side by side.

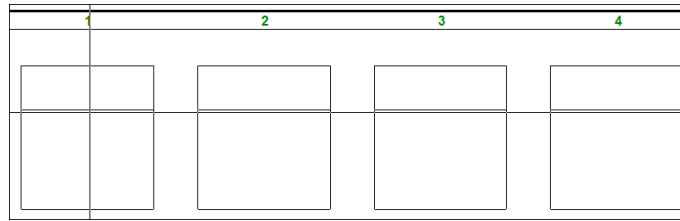


Select all - for PLCs

When for instance you have selected all output symbols for a PLC, and wish to place them side by side on a page with predefined current path numbers, **Automation** takes this into consideration:

- When between 4 and 16 current path numbers have been defined for the page, the PLC symbols will be placed so that the distance between the PLC symbols is the same as the distance between the current path numbers

- There will not be placed more PLC symbols on the page, than the number of defined current path numbers specified for the page

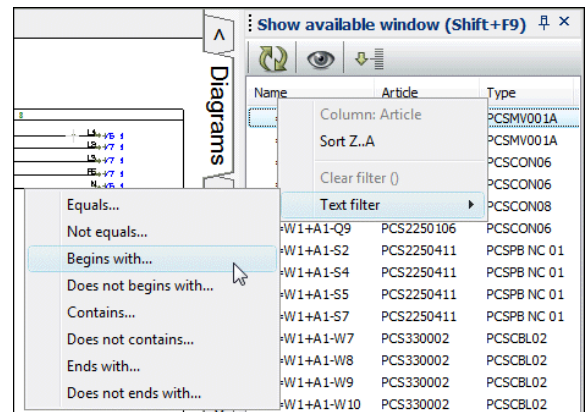


Displaying components in the Show available window

Because many components can be listed in the **Show available window**, there also exist some functions for sorting, filtering and hiding components in the list:

Sorting and filtering in the Show available window

If you right-click on the column names at the top of the **Show available window**, you get the option of specifying the sorting order in which the components appear in the **Show available window**.



Furthermore, under **Text filter** you can filter which components to show in the list.

Hide selected in the Show available window



When you select a number of components in the **Show available window**, you can then click the **Hide selected** button — or right-click and choose **Hide selected**. The components then no longer appears in the list. In stead a line named **Hidden components** is displayed at the bottom of the list.

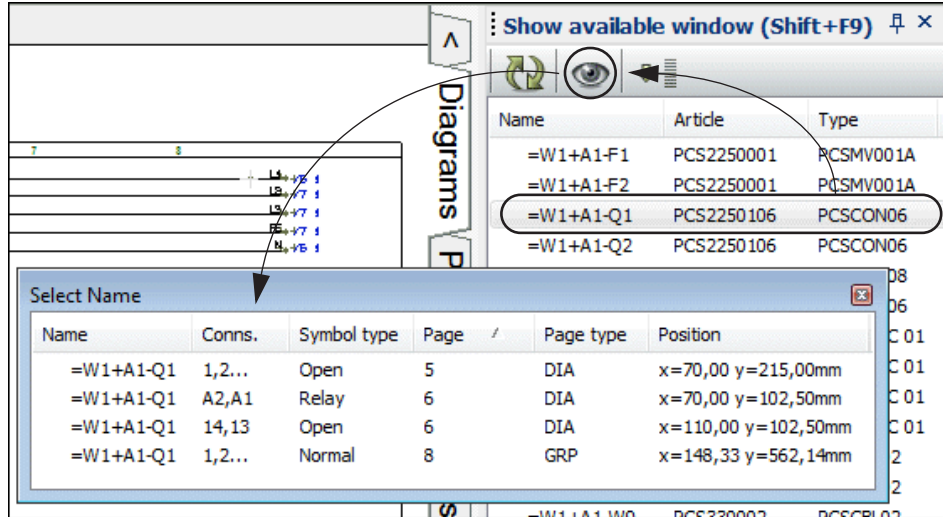


In order to see the hidden components, just double-click the **Hidden components** line. When you want the hidden components to appear in the list again, select them and click the **Unhide selected** button — or right-click the components and choose **Unhide selected**.

Select name window for a component - via the Show available window



When you click 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.



Here you can see where the other symbols for the component are placed in the project.

— Read more about this in “Find all symbols with identical names” on page 238.

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

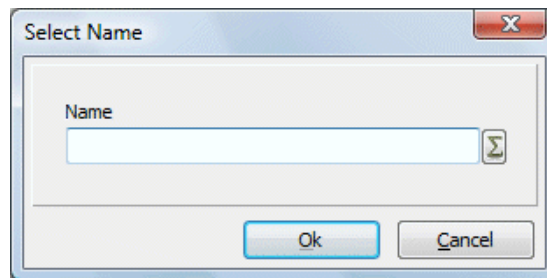
THIS CHAPTER DESCRIBES	
— FIND ALL SYMBOLS WITH IDENTICAL NAMES	P. 238
— SEARCHING FOR MORE COMPONENTS USING WILDCARDS	P. 239
— JUMP TO THE NEXT SYMBOL IN THE SELECT NAME LIST AUTOMATICALLY	P. 240
— CHANGING SYMBOL REF. DESIGNATIONS USING SELECT NAME.....	P. 240
— RENUMBER CONNECTION NAMES FOR TERMINAL ROW USING SELECT NAME .	P. 241

— See “Select Component” on page 242 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 309.

The contents of the window can be sorted as described in “The sorting order of the data in the Object Lister” on page 309.

Searching after more components using Wildcards

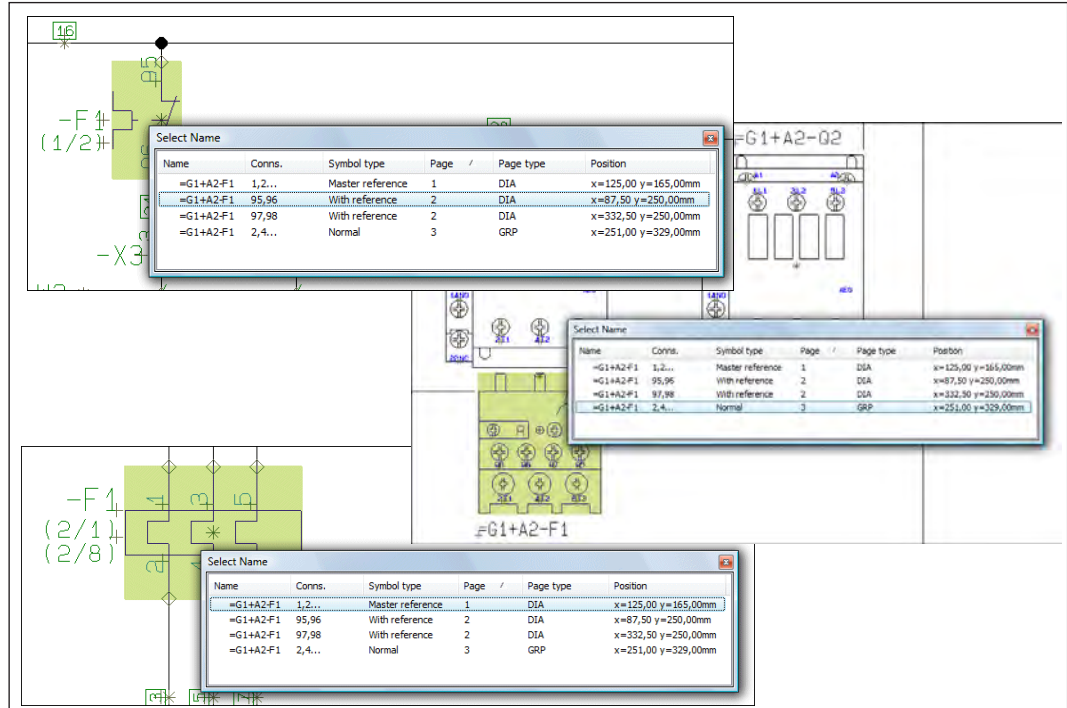
When you wish to find all components of a specific type, or do you not know all of the component name, you can use the so-called wildcards — as described in “Wildcards/jokers” on page 550.

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.



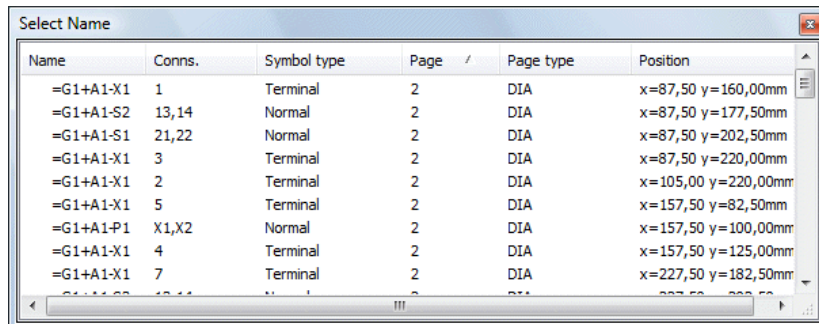
You can also activate the function by right-clicking on a symbol, and then choose **Select Name**.

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 239) and click **OK**.



- 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:



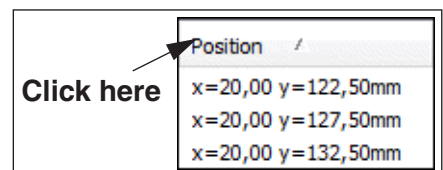
- 1) Click on the **Symbols** button, right-click on the page and choose **Select Name**.



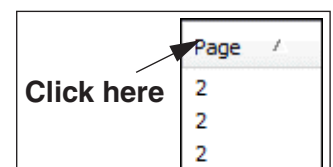
- 2) In the dialog box **Select Name**: Click on the **Used** button, select the terminal row (for instance =G1+A2-X3) and click **OK** — hereby all symbols for =G1+A2-X3 are selected in the project:

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 pin numbers — 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 238 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 67.

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 550.

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. 244
— REFERENCE POINTS WHEN SELECTED AREAS ARE MOVED/COPIED	P. 244
— INTELLIGENT COPYING OF AREAS	P. 245
— ASSIGNING REFERENCE DESIGNATIONS WHEN COPYING AREAS	P. 247
— MOVING OR DELETING SELECTED AREAS.....	P. 249
— CUTTING AND PASTING AREAS	P. 251
— MOVING OR COPYING AREAS TO OTHER PROJECT PAGES.....	P. 250
— MAKING AREAS INVISIBLE.....	P. 249
— ROTATING AREAS	P. 253
— SCALING AREAS	P. 253
— COPYING AREAS TO SYMBOL DEFINITIONS.....	P. 255



INSERTING SUB DRAWINGS

PLEASE NOTE THAT YOU CAN ALSO INSERT SUB DRAWINGS — “SUB DRAWINGS” ON PAGE 256.

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.

Please note that it makes a difference whether you select the objects *right to left* or *left to right*. Read more about this in “Dragging the marquee rectangle left to right — or right to left” on page 56.

— Read more about selecting objects in “Selecting objects” on page 55.

REFERENCE POINTS WHEN SELECTED AREAS ARE MOVED/COPIED

When you move a selected area, the lower-left corner of the selected area is applied as reference point. However, if you right-click inside the selected area and choose **Select area reference point with Move**, you can subsequently point out the reference point for the moved area yourself.

When copying you can also right-click and choose **Select area reference point with Copy**.

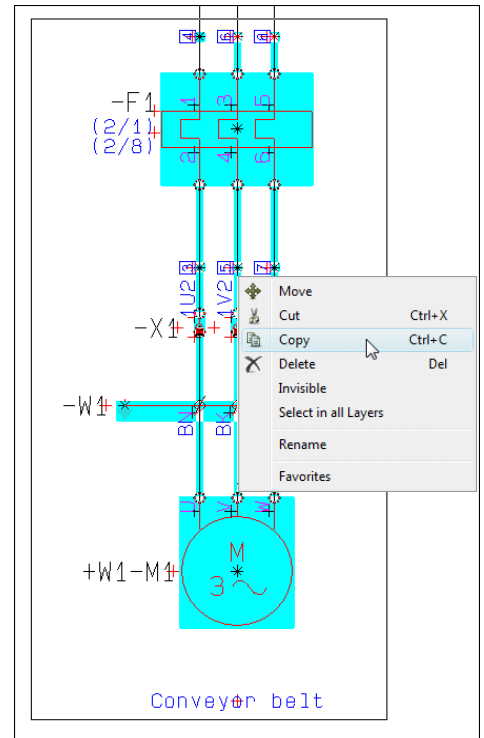


INTELLIGENT COPYING OF AREAS

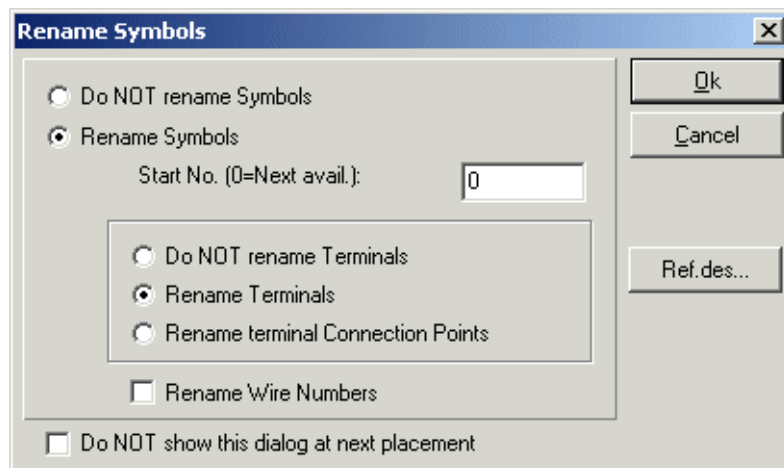
When you want to copy an area in **Automation** you click the **Area** button, and use the mouse to select the area you want to copy — see “Selecting all types of objects in an area” on page 244.

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 246.

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 389.

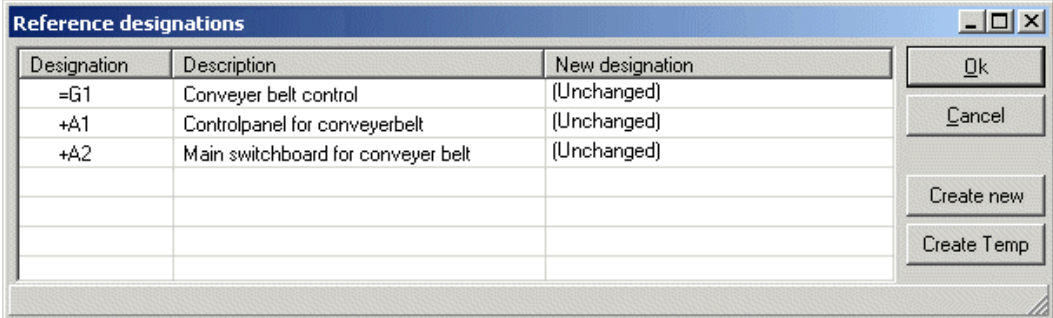
Copying an area to another page

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 250.

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:



Designation	Description	New designation
=G1	Conveyer belt control	(Unchanged)
+A1	Controlpanel for conveyerbelt	(Unchanged)
+A2	Main switchboard for conveyer belt	(Unchanged)

In the columns **Designation** and **Description** the current reference designations are displayed along with their descriptions.

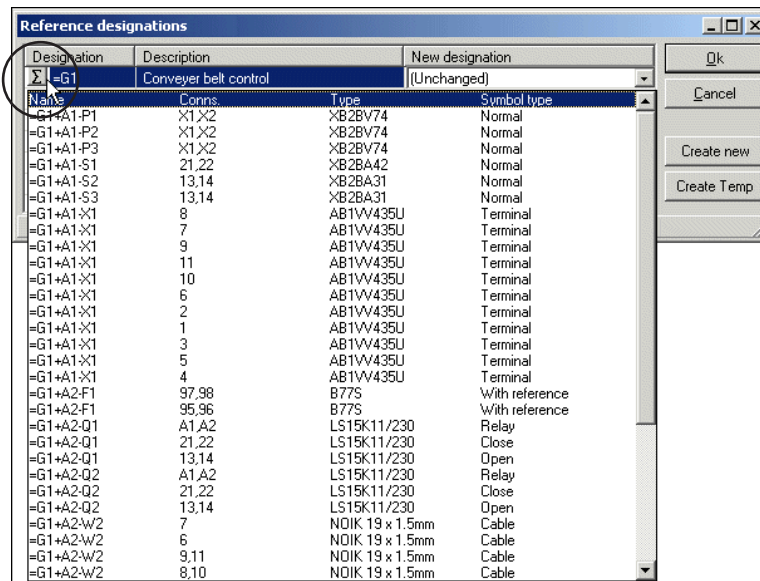


REF. DESIGNATIONS FOR SUB DRAWINGS, PAGES AND FULL PROJECTS

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.

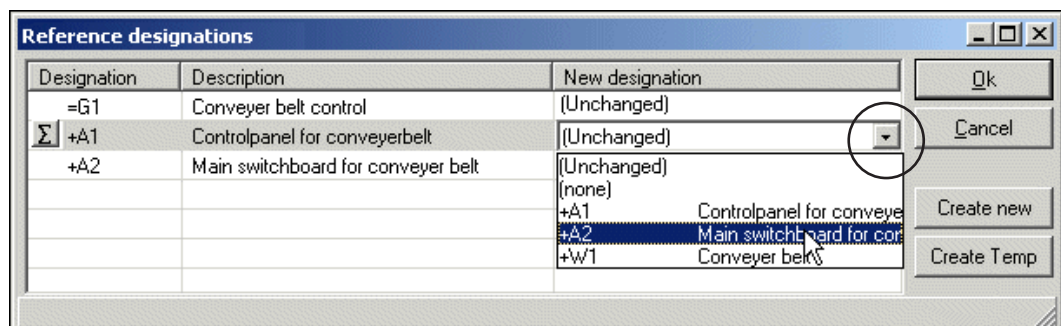
See symbols using the listed reference designations

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.

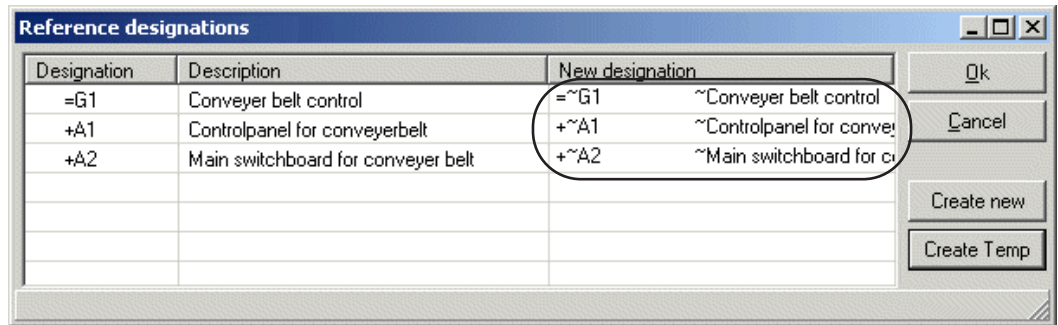
CREATING A NEW REFERENCE DESIGNATION

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 338, in particular “**Specifying function/location reference designations**” on page 340.

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 58.

If you want to move an area to another page in the project, please see “Moving or Copying areas to other pages” on page 250.

MAKING AREAS INVISIBLE

— See “Making areas invisible” on page 70.

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 518 also.

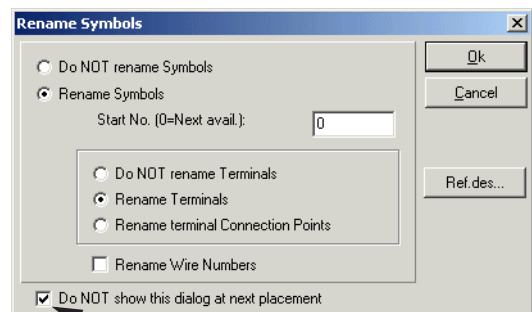
The dialog box Rename Symbols can be deactivated after first placement

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 556.

Copying an area to another project

When you have the area in the cross hairs, you can copy it to another project by:

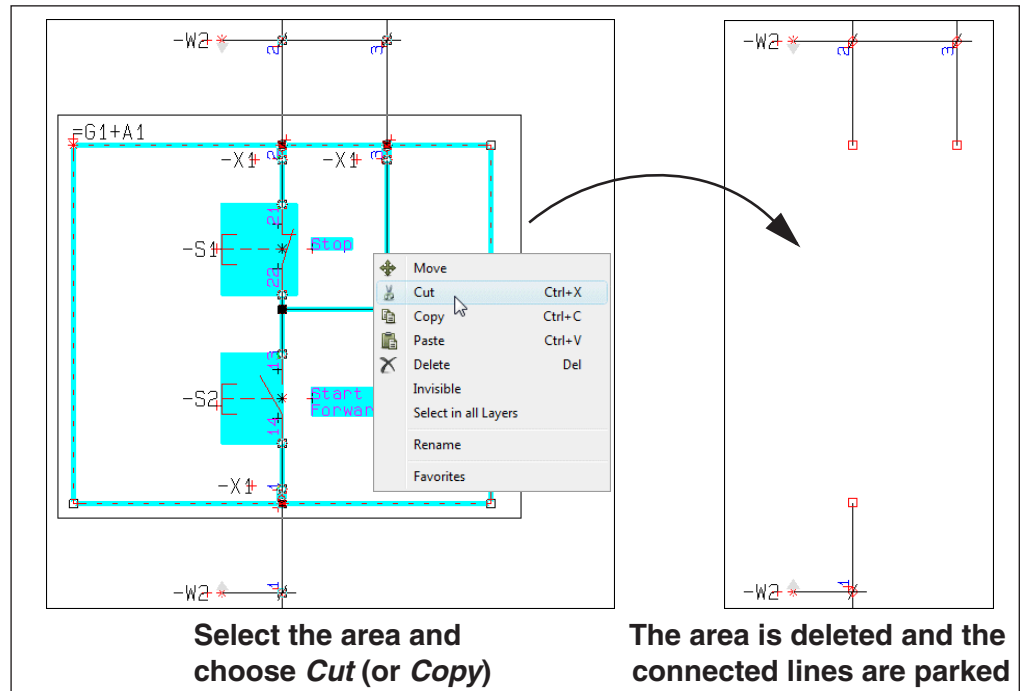
- a) Clicking on the other project in the **Explorer** window, and press <Ctrl+v>.
- b) Following the description in “Copying an area from one project to another” on page 518.

CUTTING AND PASTING AREAS

You can cut and paste an area in the following way:

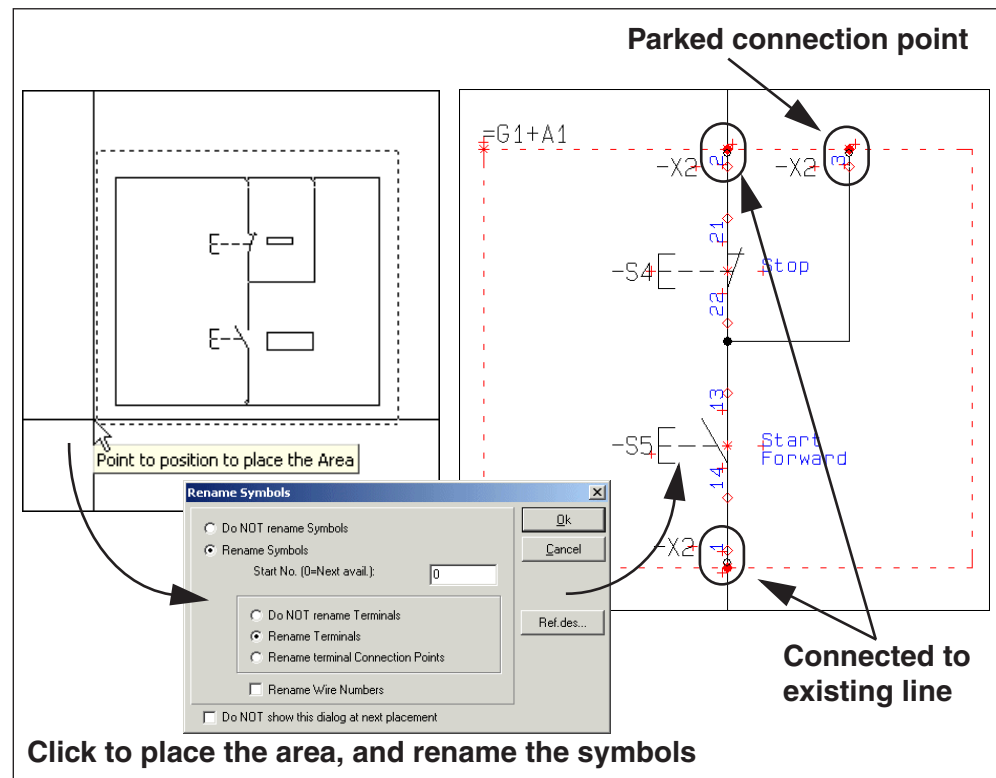


- 1) Click on the **Area** button, and select the area by dragging a window over it with the mouse.
- 2) Click on the **Cut** button, or right-click inside the area and choose **Cut**.



- 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 245. 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

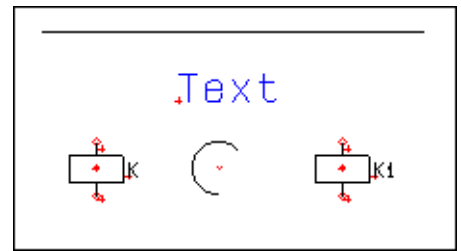
THE *CUT* FUNCTION CAN ALSO BE USED WHEN WORKING WITH SYMBOLS, TEXTS, LINES AND CIRCLES.

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.

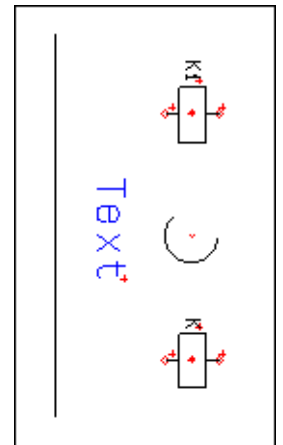


Click inside the area, keep the mouse button pressed down and move the mouse. Press <spacebar>. Hereby the area is rotated around the point the mouse is pointing at.

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 696).

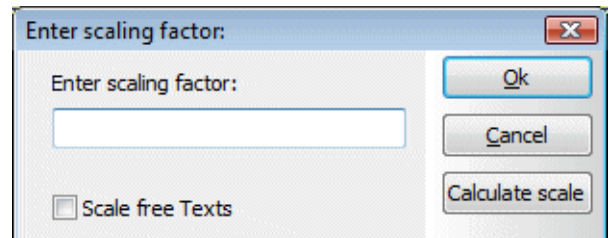
To scale an area do the following:



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.



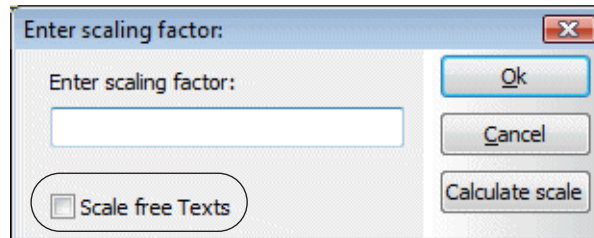
CALCULATE SCALE

IF YOU CLICK THE **CALCULATE SCALE** BUTTON IN THE DIALOG BOX **ENTER SCALING FACTOR**, YOU ARE ASKED TO POINT OUT TWO POINTS ON THE DRAWING, AND ARE SUBSEQUENTLY ASKED TO TYPE IN THE PHYSICAL DISTANCE BETWEEN THESE POINTS. THE PROGRAM THEN CALCULATES THE SCALING FACTOR AUTOMATICALLY. PLEASE REMEMBER TO ENTER A UNIT OF MEASUREMENT WHEN SPECIFYING THE PHYSICAL DISTANCE BETWEEN THE POINTS.

— 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 751.

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 55.

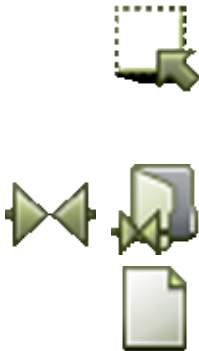
Specifying the scale of the page

You can also specify the scale of the entire page. — Read more about this in “Setting the scale of the page” on page 483.



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.



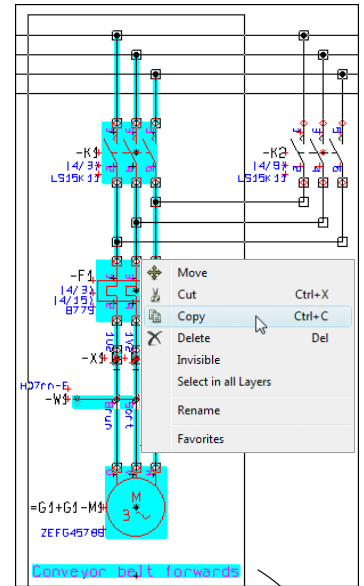
ALL SYMBOLS IN THE INSERTED AREA ARE AUTOMATICALLY CONVERTED TO LINES WHEN THE AREA IS INSERTED IN THE SYMBOL DEFINITION.

You can now place the connection points for the new symbol, edit the symbol etc. as described in "Creating Symbols" on page 696.

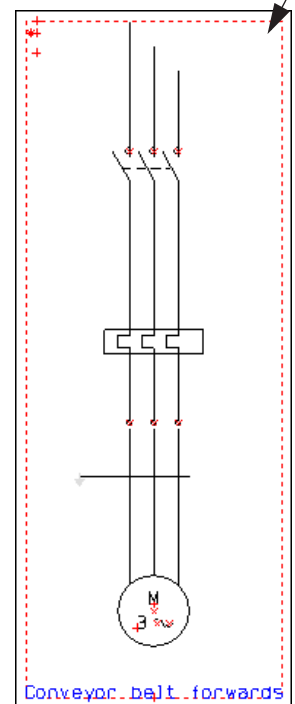
COPYING AREAS BETWEEN PROJECTS



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. 257
— ON INSERTING SUB DRAWINGS.....	P. 258
— MANUAL PLACING OF SUB DRAWINGS	P. 259
— PLACING SUB DRAWINGS BY DRAG'N DRAW.....	P. 264
— DRAG'N DRAW SETUP.....	P. 265
— CREATING SUB DRAWINGS FOR MANUAL PLACING	P. 267
— EDITING SUB DRAWINGS	P. 268
— CREATING SUB DRAWINGS BY COPYING FROM PROJECTS	P. 268
— ON PAGES AND SUB DRAWINGS FOR DRAG'N DRAW	P. 271
— CREATING PROJECT PAGES FOR AUTOMATIC PLACING OF SUB DRAWINGS....	P. 272
— PREPARING SUB DRAWINGS FOR DRAG'N DRAW / AUTOMATIC PLACING.....	P. 277
— THE FUNCTION OF THE INSERTION POINTS	P. 279
— INSERTING VARIABLES IN SUB DRAWINGS, PAGES AND PROJECTS	P. 281
— CREATING MODEL DATA FOR SUB DRAWINGS.....	P. 284
— INVISIBLE AREAS AND SUB DRAWINGS.....	P. 287
— SUB DRAWINGS WITH INTELLIGENT LINE CONNECTION	P. 287



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.

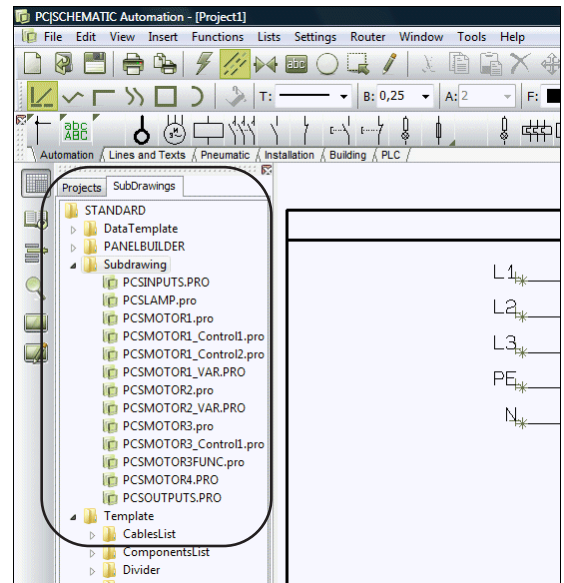
THE EXPLORER WINDOW: THE SUBDRAWINGS TAB

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 301.

— Read more about the **Explorer window** in “The Explorer window: The Projects tab” on page 301.



Specifying what to display on the SubDrawings tab

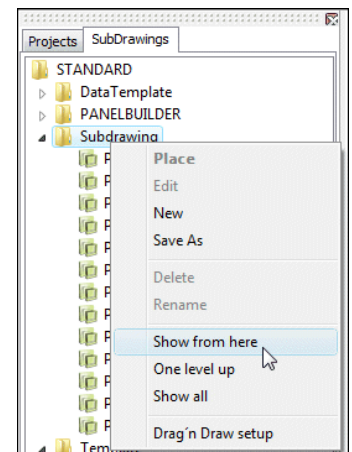
To select — and lock — which folder to display the contents of on the **SubDrawings** tab, you can right-click on the folder in the tab.

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.



(Click on Folder)

Other options in the right-click menu

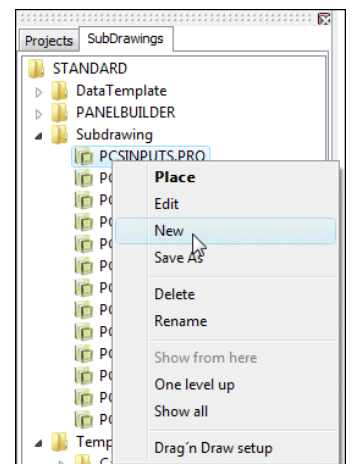
Beyond this, you can choose between:

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 259.

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 268.

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 268.



(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 265.

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 259.
- 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 264.

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

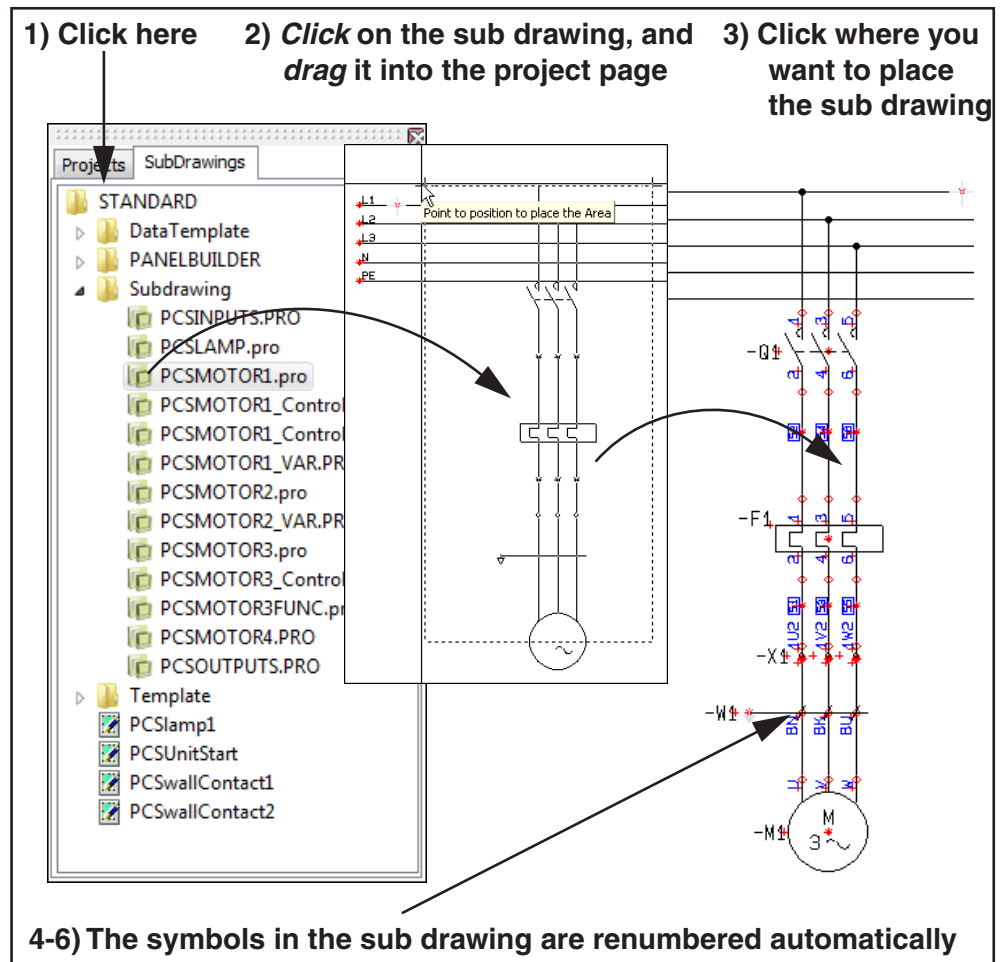
To get the full benefit of applying drag'n draw, you need to make some extra preparations, as described in “On Pages and Sub drawings for Drag'n Draw” on page 271.

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 258 — 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 260.

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 265)

— 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

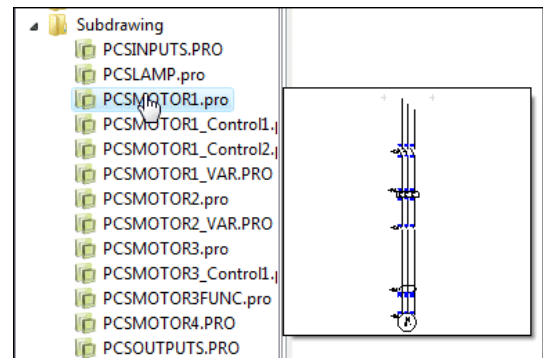
THE SUB DRAWING IS NOW INSERTED IN THE PROJECT, AND YOU CAN CONTINUE WORKING WITH THE SYMBOLS, LINES ETC. IN THE SUB DRAWING, JUST AS IF YOU HAD DRAWN IT FROM SCRATCH.

— Please also see “Creating Project pages for aut. placing of sub drawings” on page 272 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.

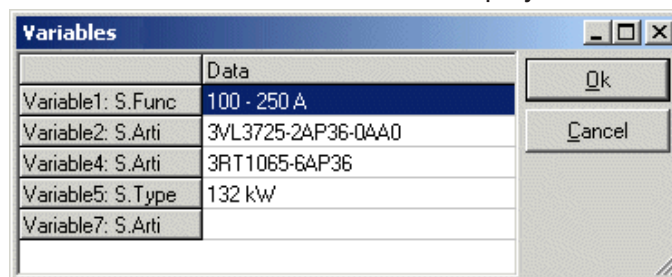
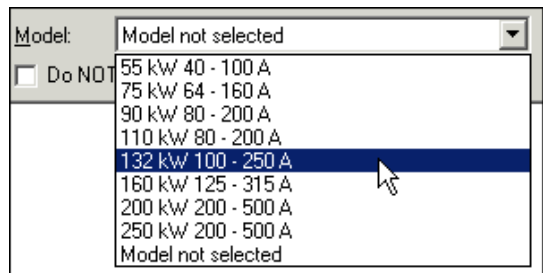


Inserting sub drawings: With model data

When you insert a sub drawing — see “Manual placing of sub drawings” on page 259 —, 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 261.

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 265, “Drag’n Draw settings: For model data” on page 265 in particular.

— See also how to create models for a sub drawing in “Creating Model data for sub drawings” on page 284.



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 281, SPECIALLY “INSERTING VARIABLE MARKERS ON A SUB DRAWING (OR ON A DRAWING TEMPLATE)” ON PAGE 281.

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.

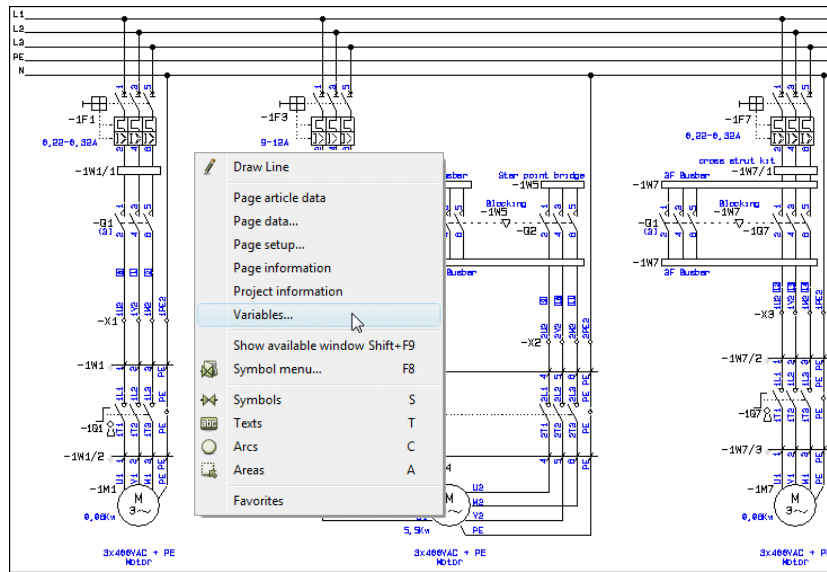
- 4) Double-click on one of the fields in the dialog box, and edit the text. Continue like this until all relevant texts are edited.
- 5) Click **OK**.

You can also edit the texts in the sub drawing exactly as you would edit any other text in **Automation**.

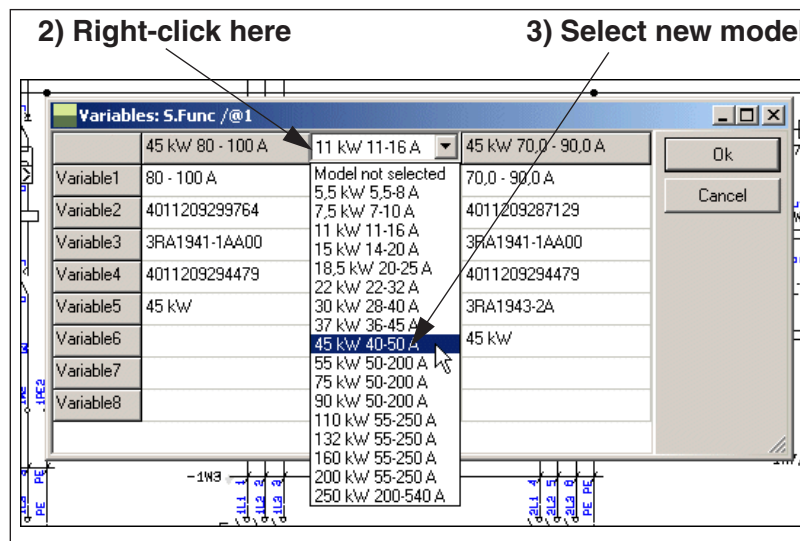
Sub drawings: Changing the selected model

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.

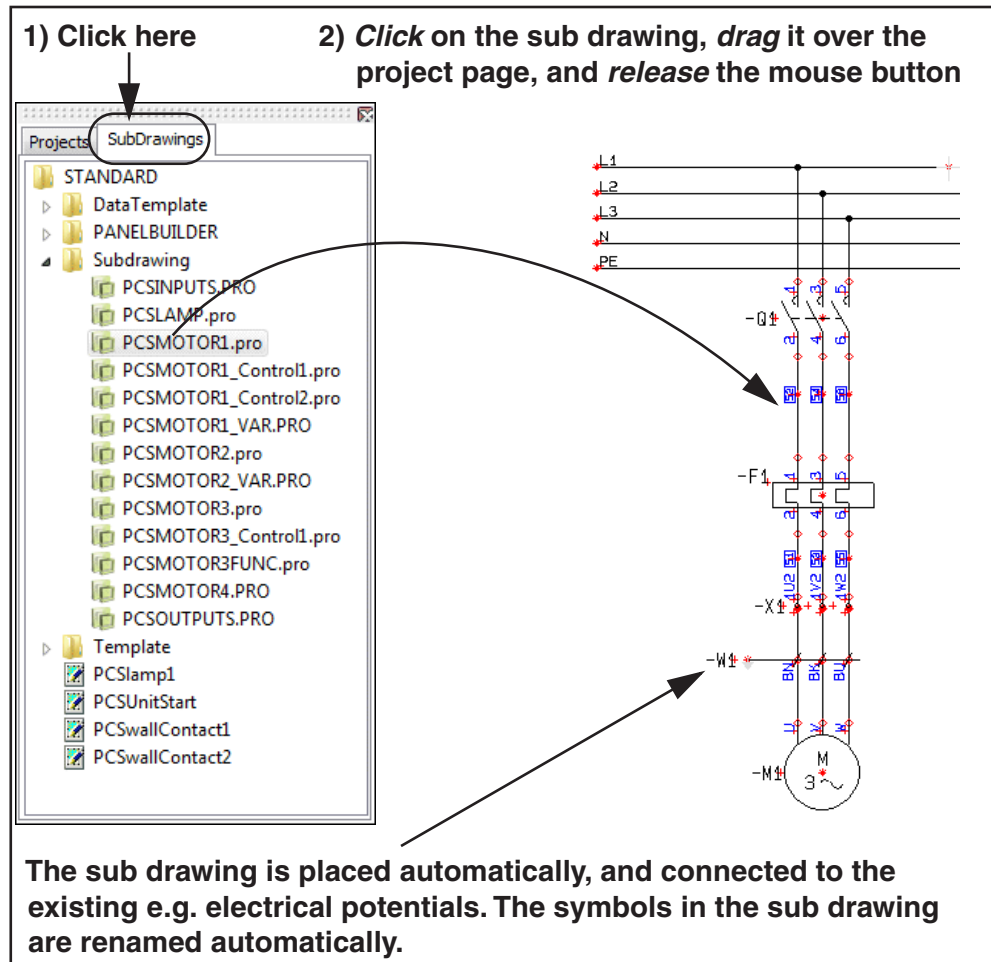


PLACING SUB DRAWINGS WITH DRAG’N DRAW

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 271.

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:

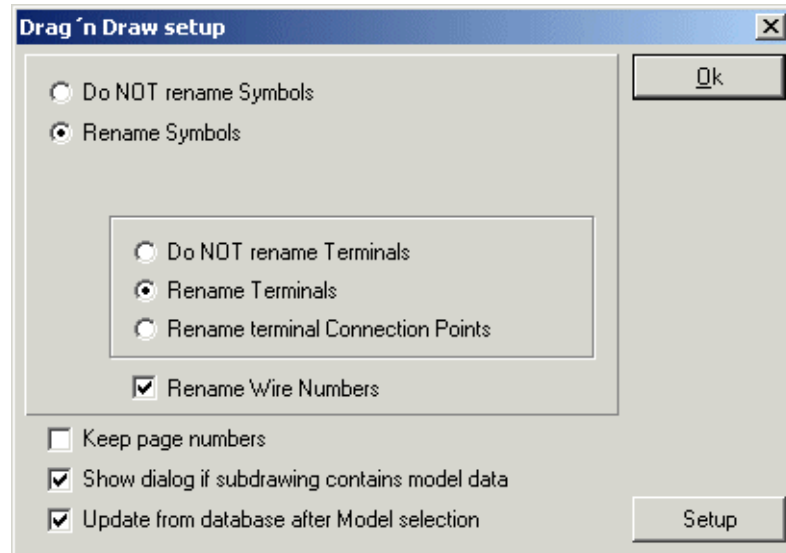


If variables are placed in the sub drawing, or if you can select different sets of model data for the sub drawing, you now get the option of specifying which data to insert in the sub drawing. You can read more about this by continuing the explanation under “If the sub drawing contains Model data” on page 259.

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 245.

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 259, “Inserting sub drawings: With model data” on page 260 in particular — when placing a sub drawing containing model data. From this dialog box you can select which model you wish to place.

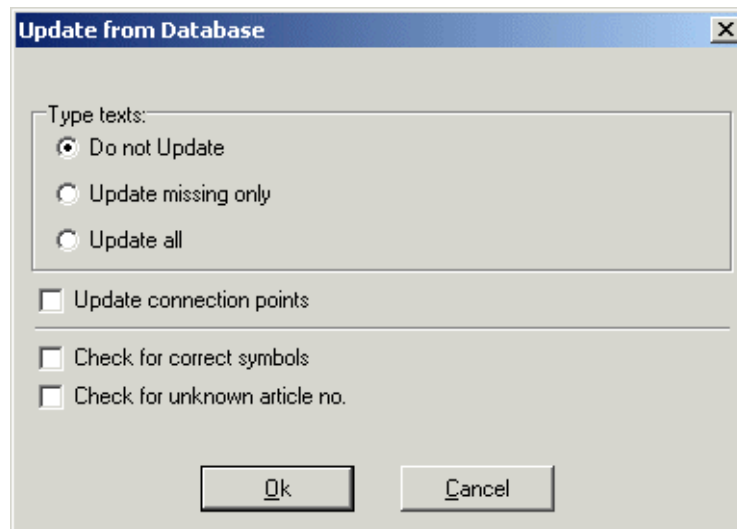
If the option is *not* selected, the sub drawing is placed without model data, or with the data from the original sub drawing, if the sub drawing contains data values.

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 171.

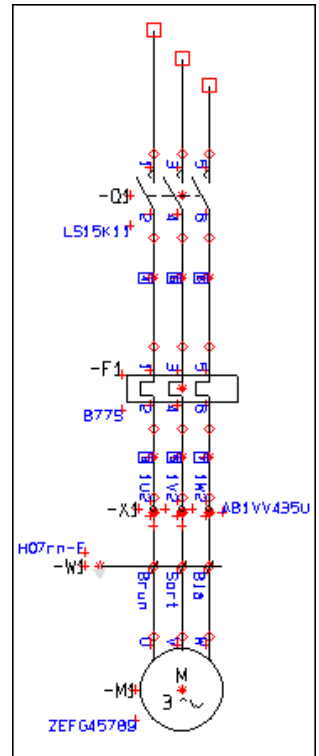
CREATING SUB DRAWINGS FOR MANUAL PLACING

A sub drawing can be any project, containing one single diagram page. — See also “Creating sub drawings by Copying from projects” on page 268.

For instance you can create a sub drawing like this:

Enter the Edit Sub drawing mode

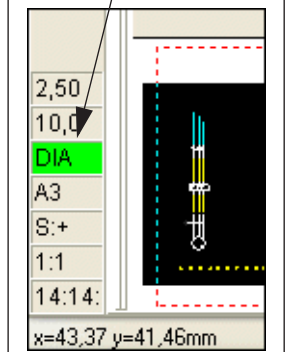
- 1) In the **SubDrawings** tab in the **Explorer** window: Right-click in the folder that you wish to place the sub drawing in, and choose **New**:



- 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.

Blinks green in the Edit sub drawing mode

- 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 268.



- 4) Continue as described in “Inserting e.g. variable markers and/or model data” on page 269.



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 267.

SEE VIDEO ON OUR WEBSITE **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:

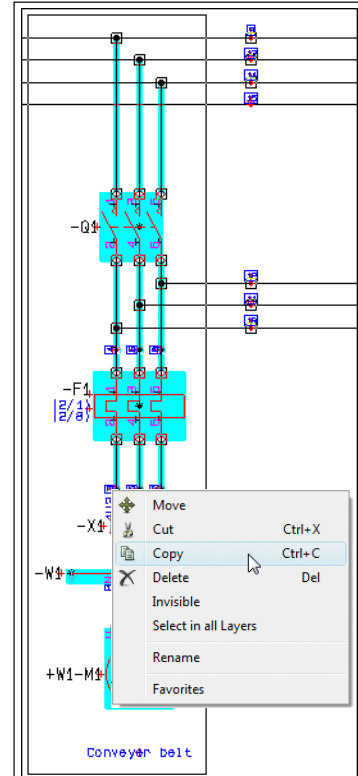
Copying the desired area



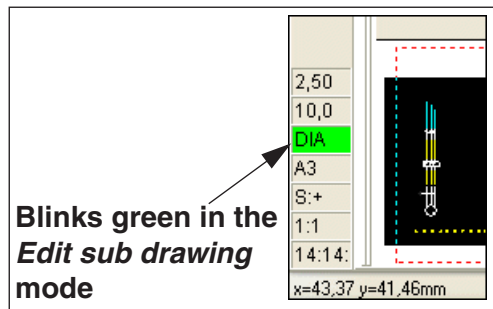
- 1) Click the **Area** button.
- 2) Select the area you wish to copy, right-click in the area, and choose **Copy** — see the figure to the right.

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



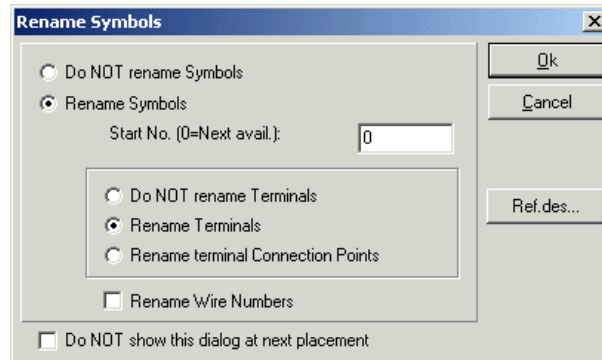
SAVE COPIED AREAS USING SAVE AS

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**.

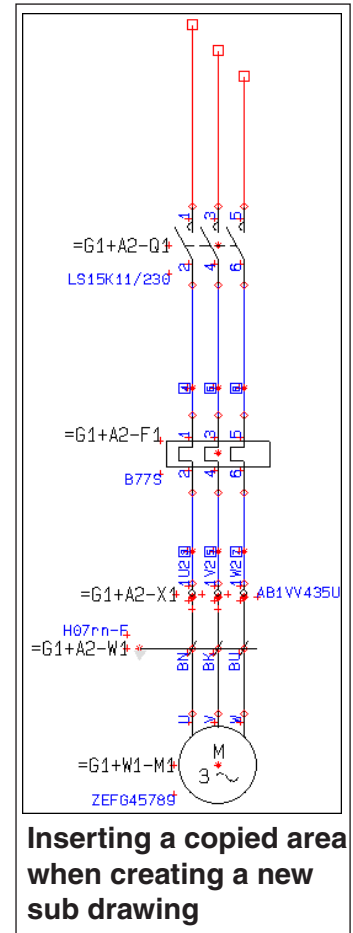
Inserting the copied area

- 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 245. 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 281, and you can create different sets of *model data* for the sub drawing, as described in “*Creating Model data for sub drawings*” on page 284.
- 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 279. (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 271 AND “PREPARING SUB DRAWINGS FOR DRAG’N DRAW / AUT. PLACING” ON PAGE 277.

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.

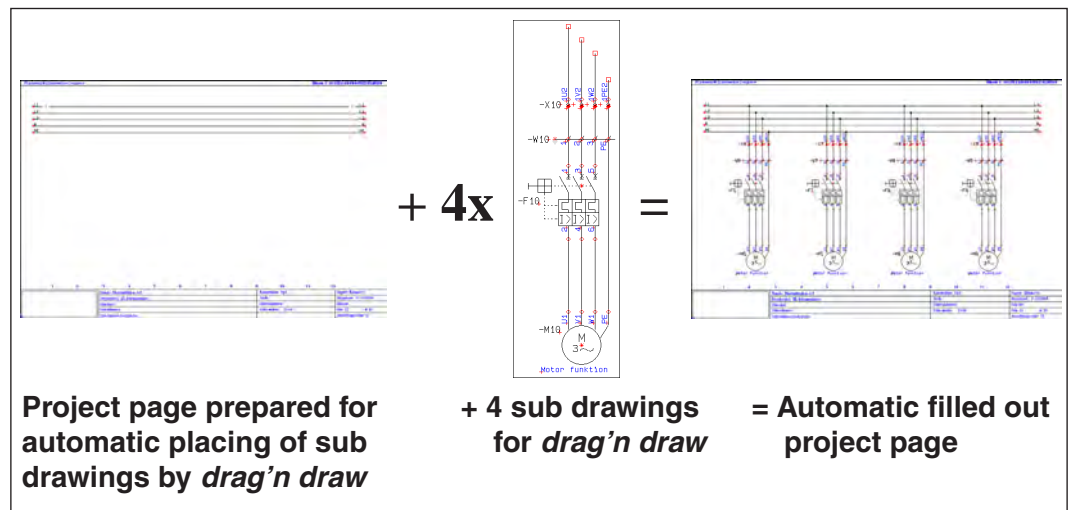
The project pages are created just like you create sub drawings — see “Preparing Sub drawings for Drag’n Draw / Aut. placing” on page 277. Here it is also described how to place the so-called *insertion points*, used for specifying the size of sub drawings and for specifying in which area sub drawings 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 272.

CREATING PROJECT PAGES FOR AUT. PLACING OF SUB DRAWINGS

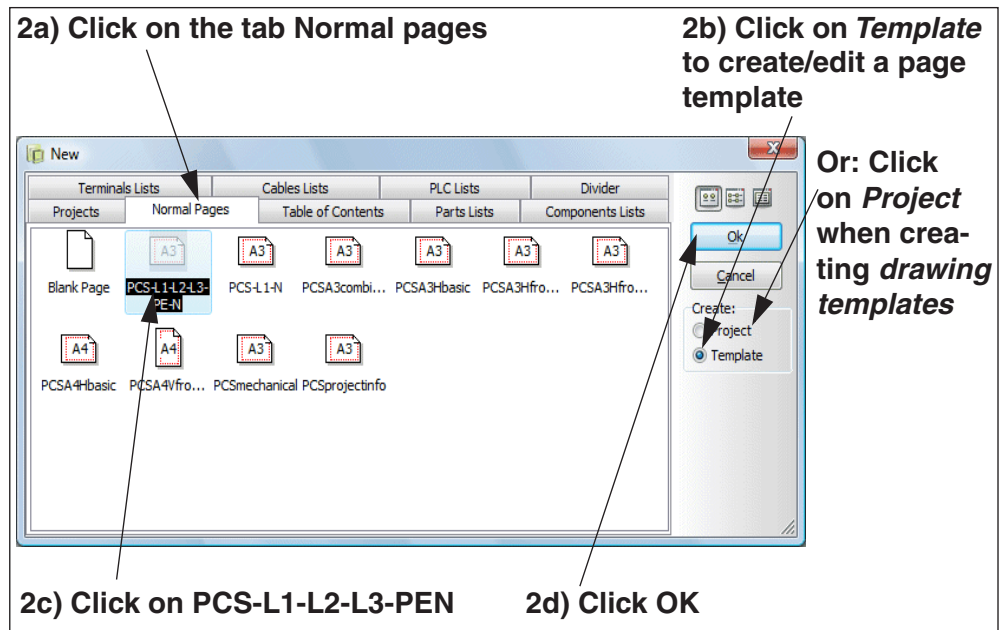
When using *drag'n draw* you can make **Automation** place sub drawings on project pages automatically.

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**:



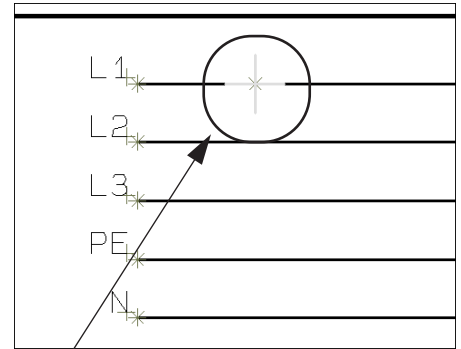
2) Click on the tab **Normal pages**, click on **Template** in the right-hand side, click on the page **PCS-L1-L2-L3-PEN**, and click **OK**.



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 sub drawings 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 279.

— SEE “POINTS TO CONSIDER IN PROJECT GENERATION” IN THE **Project generator** MANUAL ALSO.

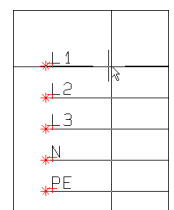
Placing insertion points on project pages for automatic placing of sub drawings

If you are creating your own page template, that you wish to prepare for automatic placing of sub drawings, 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 272. 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 144.
- 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 **Pcseicad\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

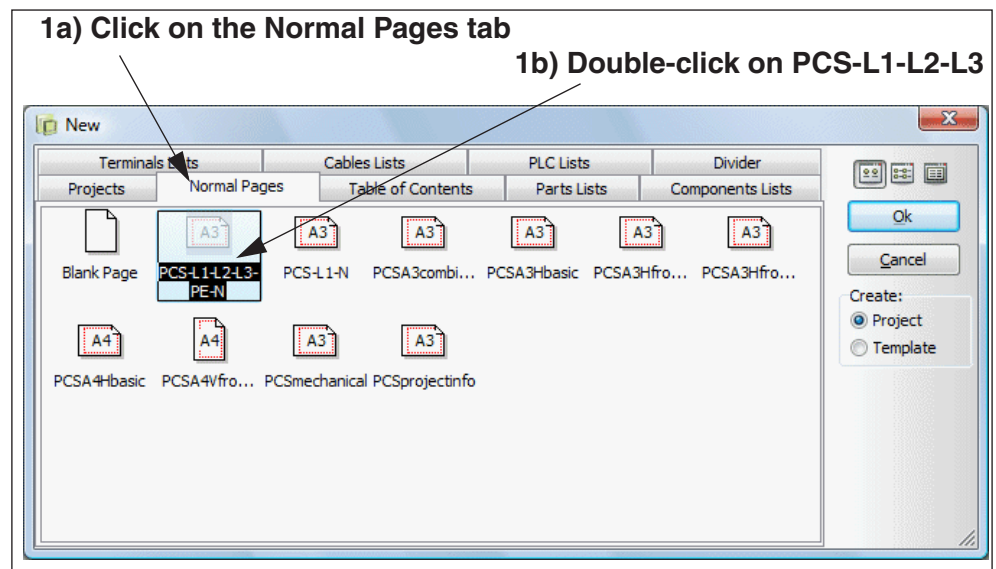


THE FOLLOWING IS BASED ON THAT YOU USE A *TEMPLATE PAGE* — AS DESCRIBED ABOVE —, PREPARED FOR AUTOMATIC PLACING OF SUB DRAWINGS BY DRAG’N DRAW.

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.

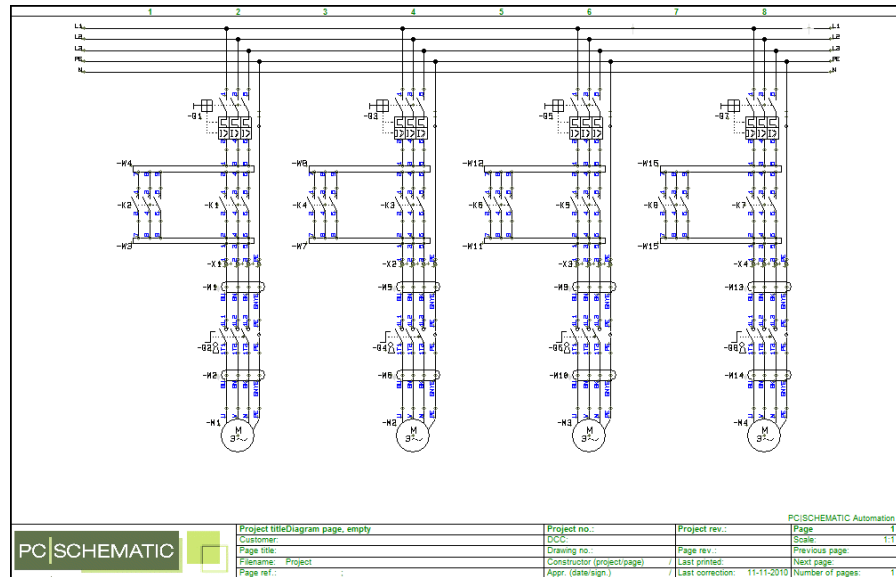
This is how you place a subdrawing automatically on a project page using drag’n draw:

- 1) Insert a new page that is prepared for automatic insertion of sub drawings: 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 272.
- 3) In the **Explorer window**: Open the folder **Pcscad\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 277), 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 sub drawing, and also get the option of choosing a sub drawing 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 265 TO SEE HOW TO SPECIFY THE SETUP FOR RENAMING E.T.C. 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 261.

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 264.

— 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 272.



SUB DRAWINGS FOR AUTOMATIC GENERATION OF PROJECTS

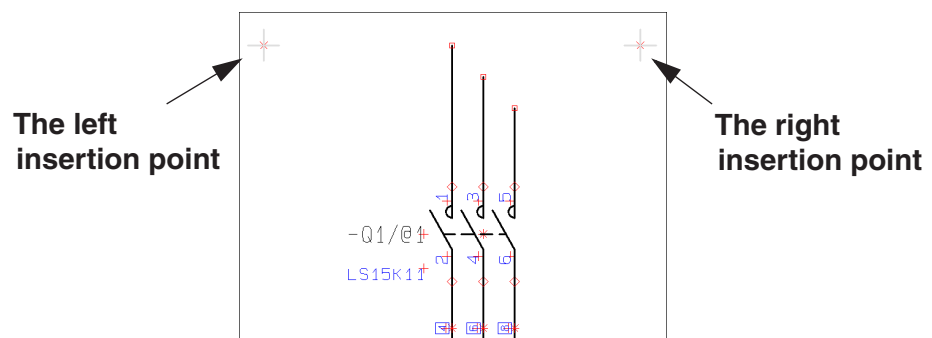
WHEN YOU WISH TO APPLY A SUB DRAWING FOR AUTOMATIC PROJECT GENERATION, IT *MUST* BE PREPARED FOR AUTOMATIC PLACING.

— 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 267, 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 268.



- 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 273 or read more about the **Symbol menu** in “Fetching symbols using the Symbol menu” on page 144.
- 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 279 (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 281.

- 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 284.*



- 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 264. 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 279.

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.

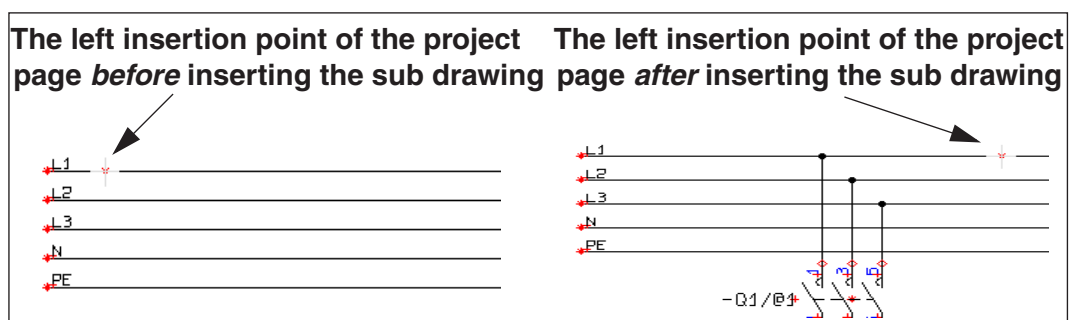
The created insertion point symbols are not displayed when printing project pages. This is because they are created using the color NP.

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 271).

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.



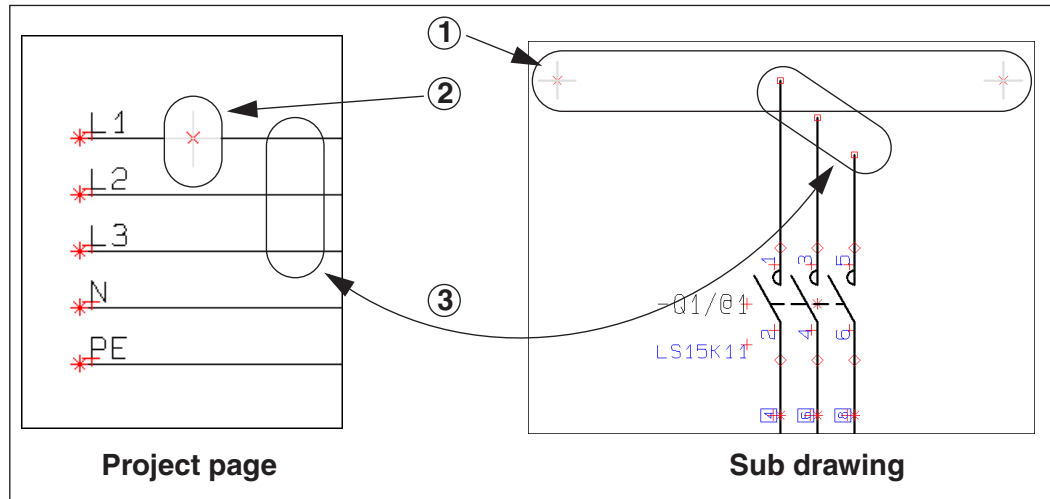
A sub drawing can not be placed automatically if there is not enough room for this sub drawing between the insertion points on the project page.

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 259.

To do this you must:

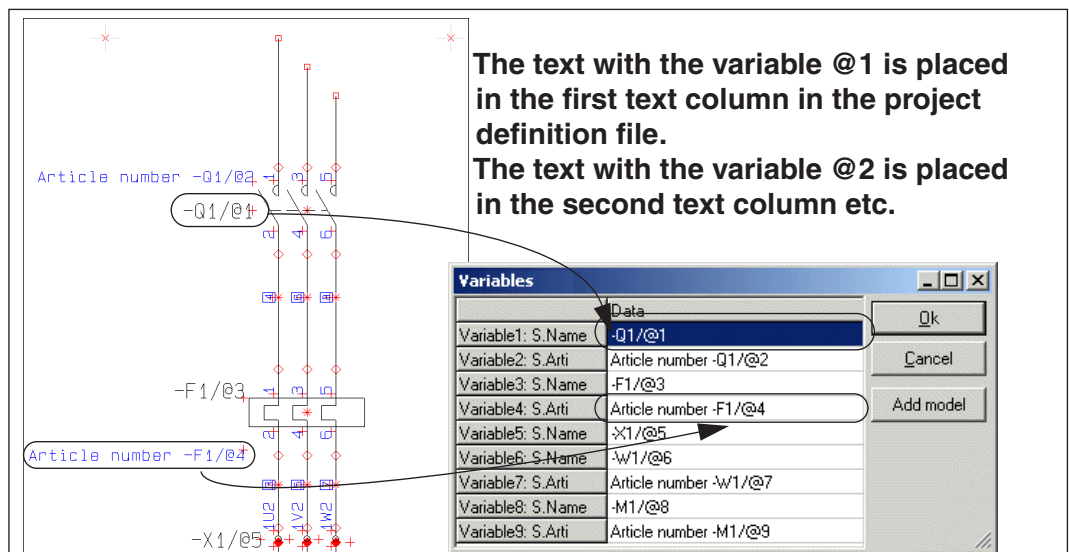
- 1) Insert *variables* in the sub drawing, as described in “Inserting variable markers on a sub drawing (or on a drawing template)” on page 281.
- 2) Create a *variable list*, as described in “Creating a variable list for a sub drawing / drawing template” on page 283.

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", and "-X1/@5".

The "Variables" dialog box is open, showing a list of variables with their names and values. The variables are:

Variable	Name	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 261 and “Manual placing of sub drawings” on page 259.

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 284.



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 => Cre-**

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:			

ate variable list, you can choose **Edit => 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 711 —, 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 motor starter 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 motor starter circuit.



VIDEO ABOUT SUB DRAWINGS WITH MODEL DATA

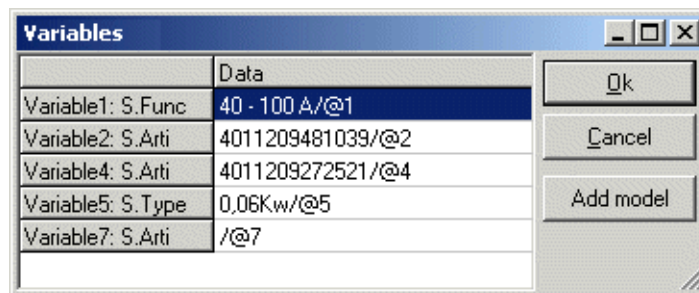
ON WWW.PCSCHMATIC.COM UNDER **VIDEOS**, YOU CAN SEE A VIDEO ABOUT HOW TO WORK WITH SUB DRAWINGS WITH MODEL DATA.

To create models for a sub drawing, do this:

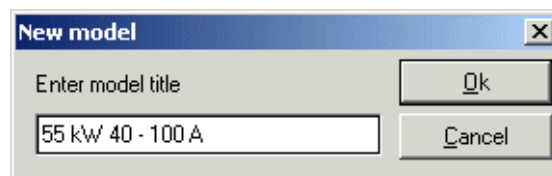
- 1) Draw the sub drawing, as described in “Creating sub drawings for manual placing” on page 267 or “Creating sub drawings by Copying from projects” on page 268. — If the sub drawing shall be used for *drag’n draw*, please also see “Preparing Sub drawings for Drag’n Draw / Aut. placing” on page 277.
- 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 281.

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**:

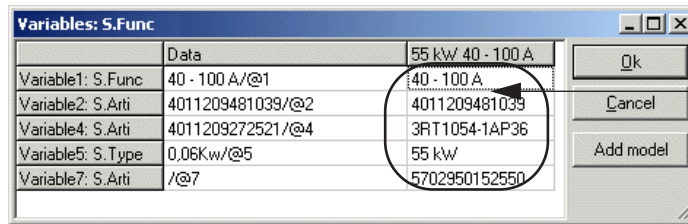


- 4) In the dialog box **Variables**: Click on **Add model**:



- 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

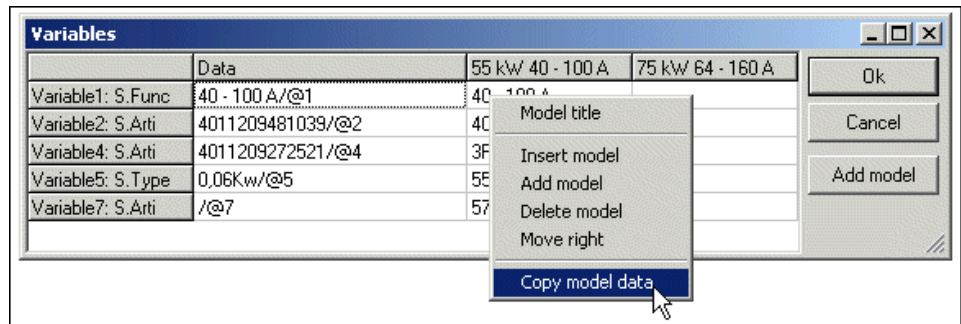
field and select **Database**, and you enter the **Database menu**, from which you select the desired component:



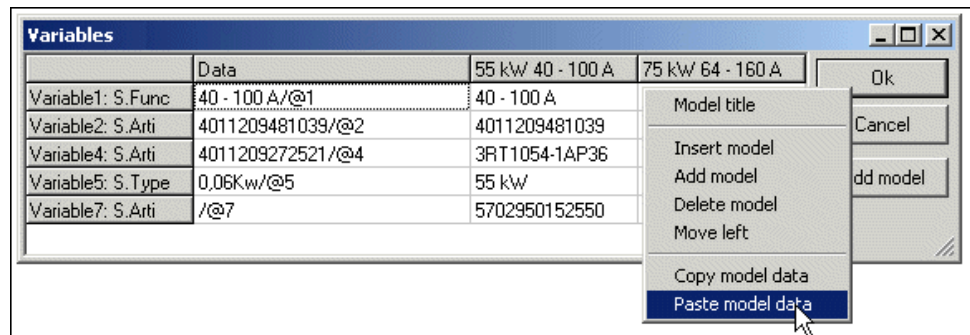
Double-click in the fields, and type in the data

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**:

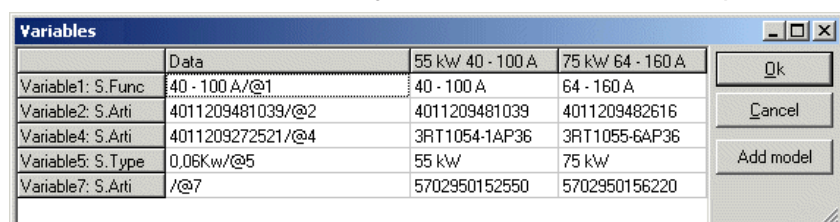


- 9) Right-click on the new model, and choose **Paste model data**:



- 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 259, “Inserting sub drawings: With model data” on page 260 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 data-base. 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 286.

Insert model: Inserts a new model where you have clicked.

Add model: Adds a new model.

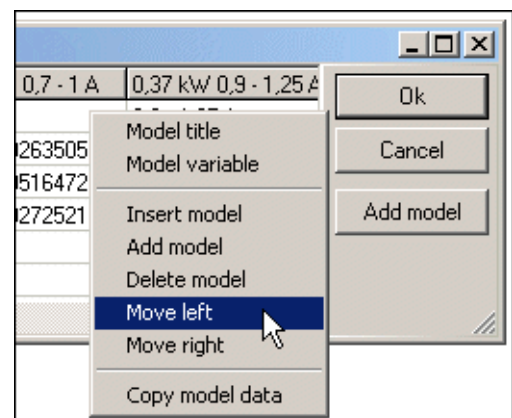
Delete model: Deletes the selected model.

Move left: Moves the selected model one position to the left. — See also “Drag and drop Models in the dialog box” on page 286.

Move right: Moves the selected model one position to the right. — See also “Drag and drop Models in the dialog box” on page 286.

Copy model data: Copies the values for the selected model — see “Creating a new model by copying data” on page 285.

Insert model data: Inserts the copied model values — see “Creating a new model by copying data” on page 285.



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.

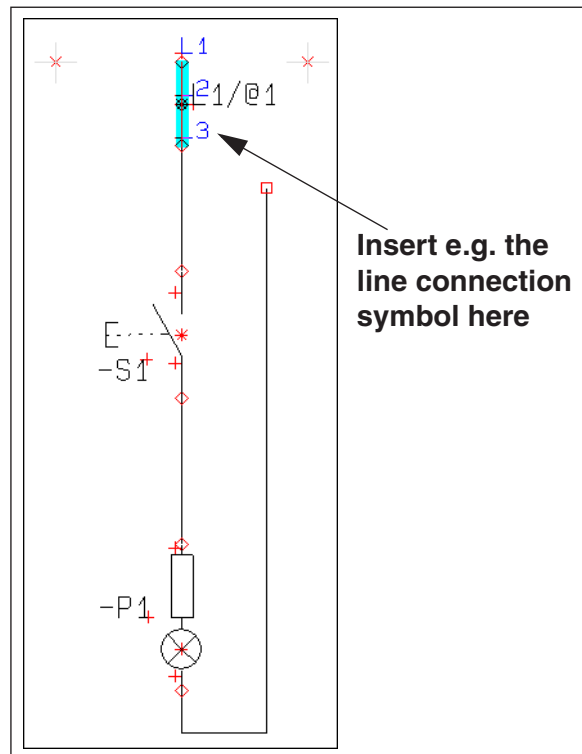
INVISIBLE AREAS AND SUB DRAWINGS

When inserting sub drawings, you can subsequently make parts of these sub drawings invisible, as described in “Making objects invisible” on page 69.

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 730.

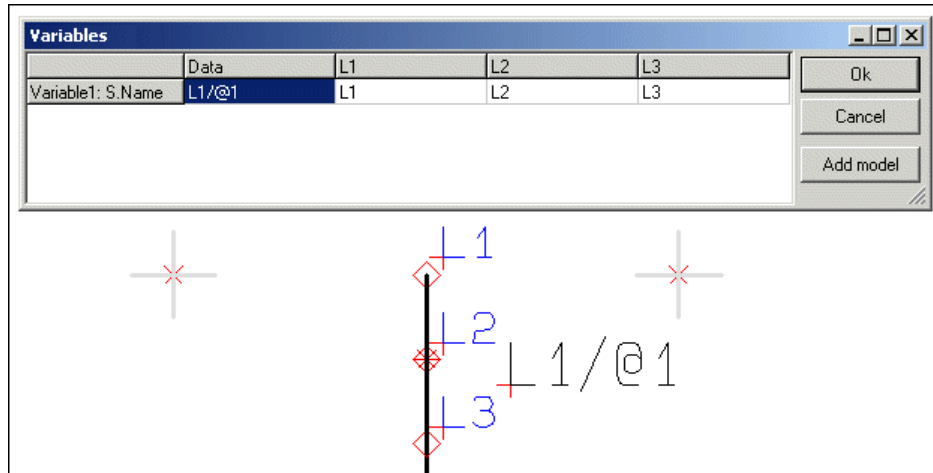
This line connection symbol can for instance be placed as displayed in the figure below.



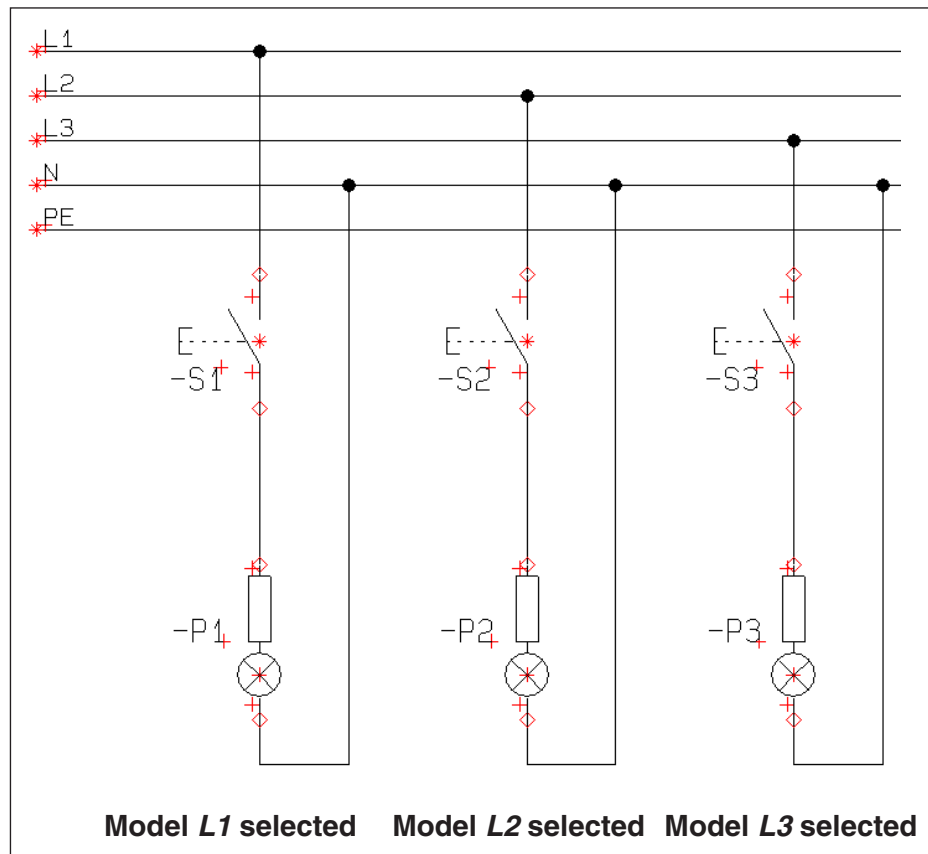
Controlled via variable marker in the symbol name

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:



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. 289
— SELECTING SYMBOLS IN THE DIALOG BOX	P. 291
— DIRECTION FOR RENUMBERING SYMBOLS	P. 292
— SPECIFY START VALUE	P. 293
— SHOW NEW SYMBOL NAMES	P. 293
— NAMING WHEN RENUMBERING TERMINALS	P. 293
— INSERTING SYMBOL NAMES OR CONNECTION NAMES	P. 295
— SORTING ON COLUMN CAPTIONS	P. 296

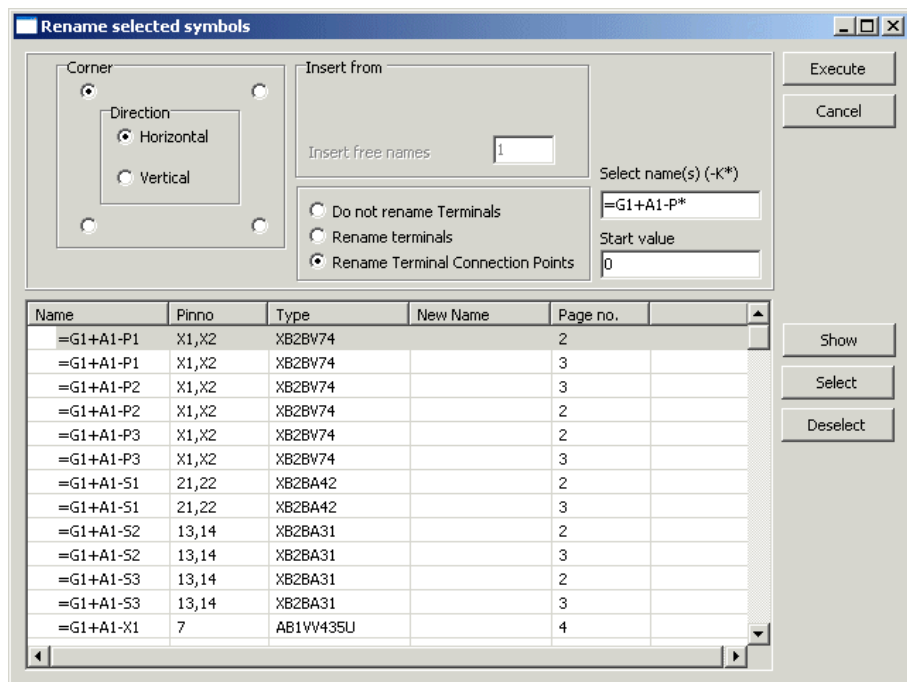


INSERT SYMBOL / CONNECTION NAMES

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 295.

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 291.
- 2) Specify a direction for renaming the symbols — see “Direction for renaming symbols” on page 292.
- 3) Specify how to rename terminals (if any) — see “Naming when renaming terminals” on page 293.
- 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 293.

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 309.

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.

Select name(s) (-K*)

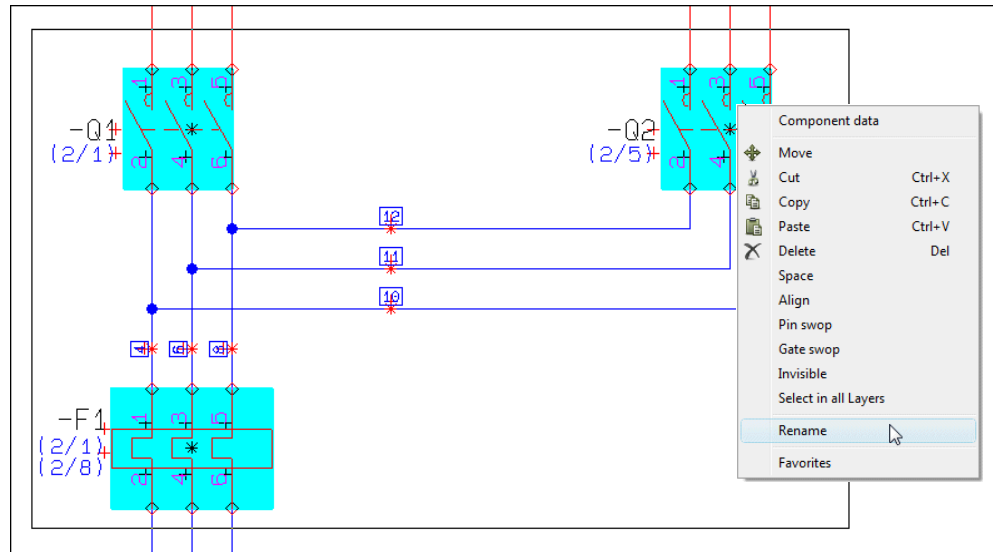
Then click on **Select** in the right-hand side of the dialog box. Now only symbols fulfilling the entered criterion are displayed.

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 55, in particular “Selecting more objects of the same kind in one window” on page 55.

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

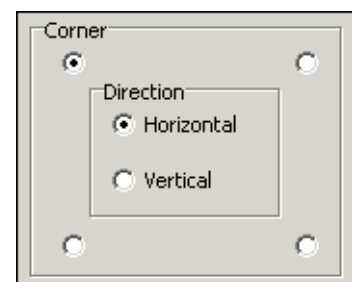
When you rename symbols, it is only their position in the project, that determines which names the symbols are assigned.

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 **Horisontal**.

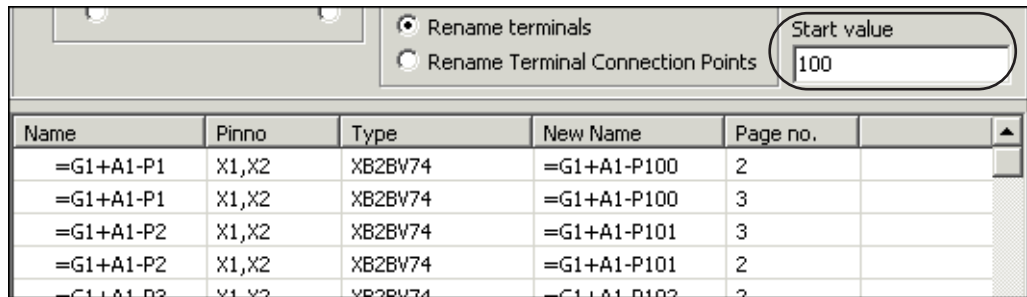


What is important for which number a component is assigned, is therefore the first time a symbol for the component appears in the project.

— 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.



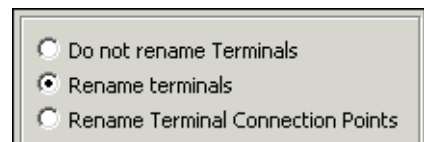
UNDO THE RENAMING

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 246.

— 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 245.



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 295.

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 295.

SORTING ACC. TO COLUMN NAMES IN THE DIALOG BOX

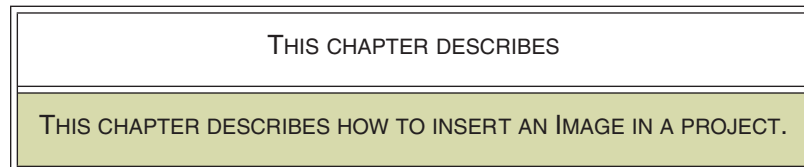
When you open the dialog box **Rename selected symbols**, the information in the dialog box are sorted by symbol name, but if you wish to e.g. sort according to type, you just click on the column caption **Type**.

When you wish to sort in reverse order, just click on the column caption again. The sorting is alphabetical/numerical.

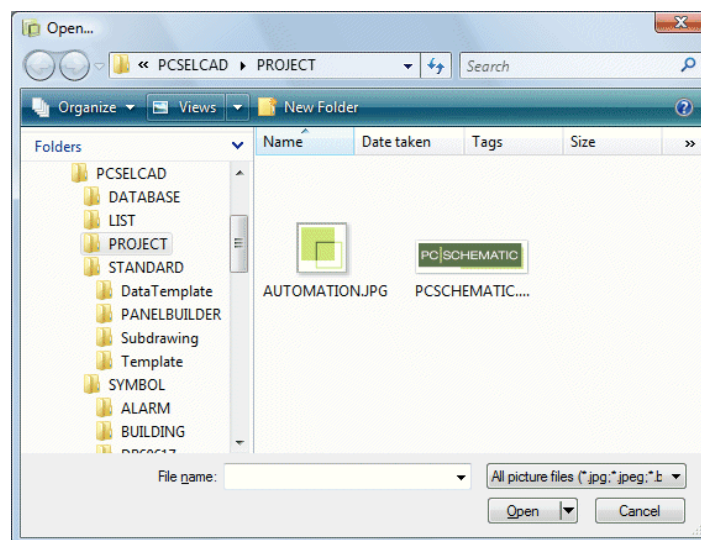
All columns can be sorted in both directions.

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.



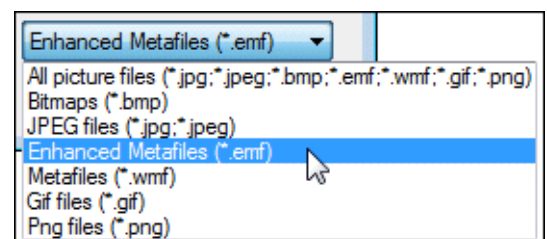
You now enter the dialog box **Open**, where you determine which file to insert into the project, and click **Open**.



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, enhanced metafiles, **.gif** files or **.png** files.

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 667.



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 253.



Survey and Reference Designations

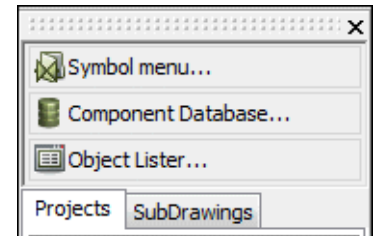
This section describes functions, which gives you a survey of projects:

- The Explorer p. 301
- The Object Lister p. 306
- Find and Replace p. 316
- Design check..... p. 320
- Printing..... p. 326
- The Explorer as Project/Symbol menu p. 334
- Reference designations..... p. 335

THE EXPLORER WINDOW: THE PROJECTS TAB

The **Explorer** window contains two different tabs: The **Projects** tab and the **SubDrawings** 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 256.

There are also three buttons at the top of the **Explorer window**, which are described elsewhere: The **Symbol menu** button (see “Fetching symbols using the Symbol menu” on page 144), the **Component database** button (see “Fetching symbols from the database” on page 152) and the **Object Lister** button (see “The Object Lister” on page 306).



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. 301
— DISPLAYING AND HIDING INFORMATION IN THE PROJECTS TAB	P. 302
— EDITING PROJECT, PAGE AND ARTICLE DATA FROM THE PROJECTS TAB	P. 302
— SELECTING SYMBOLS FROM THE PROJECTS TAB	P. 304
— JUMPING BETWEEN PROJECT PAGES IN DIFFERENT PROJECTS	P. 305
— UPDATING THE PROJECTS TAB	P. 305
— CLOSING PROJECTS FROM THE PROJECTS TAB	P. 305

GENERAL SETTINGS FOR THE EXPLORER WINDOW

You can specify where to place the **Explorer** window, as explained in “Docking Windows and Toolbars” on page 26.

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.

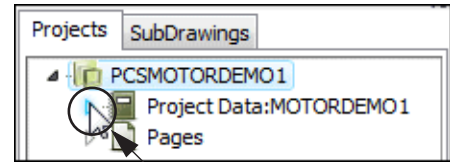
The Explorer window in Edit symbol mode

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.

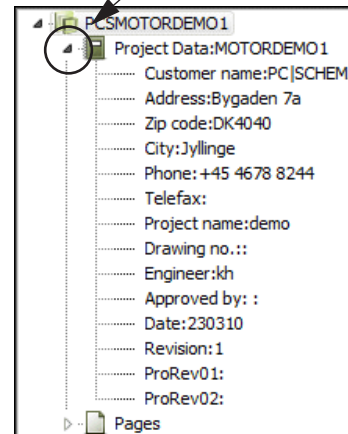


Click here to open

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.



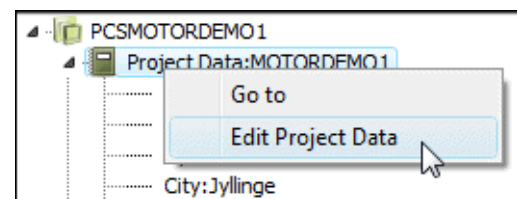
As you can see, the symbol in the **Project Data** line has now changed to a different triangle icon. When you click on this triangle, the project data datafields are hidden, and the **Explorer** window looks like the first figure again.

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 114.

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

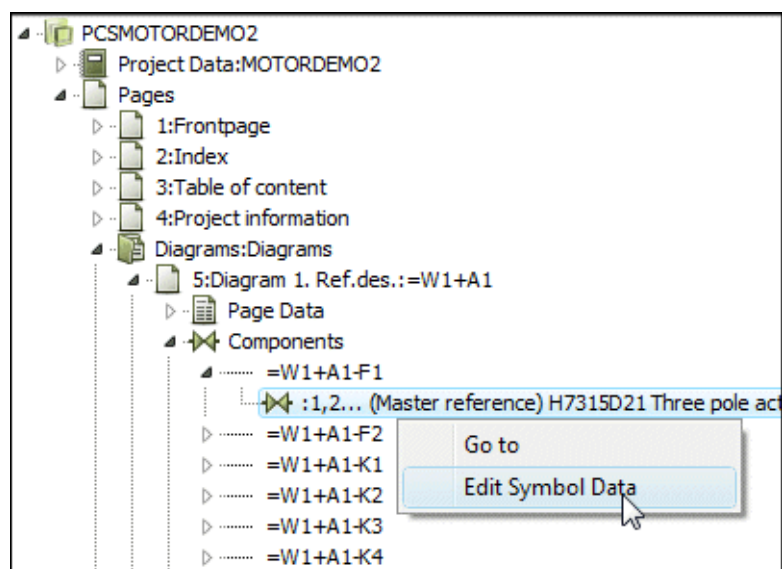
Page data datafields are changed directly from the **Projects** tab in the **Explorer** window, as described in “Editing the contents of a single Project data datafield” on page 302.

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 315.

Editing Component data from the Projects tab in the Explorer window

When you want to edit the Component data for a symbol, this can be done by right-clicking on the symbol in the **Projects** tab in the **Explorer** window, and choosing **Edit Symbol Data**.



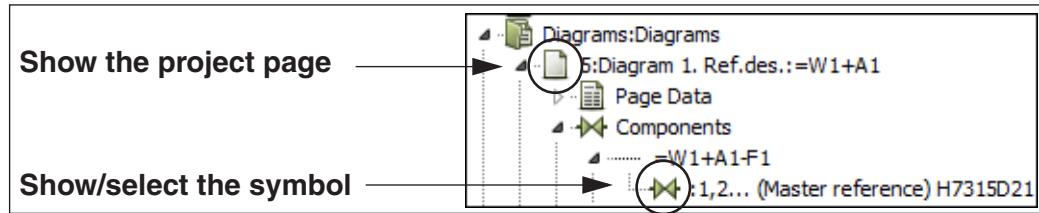
This brings you into the dialog box **Component data**, where you edit the component data and click **OK**.

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 313.

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:



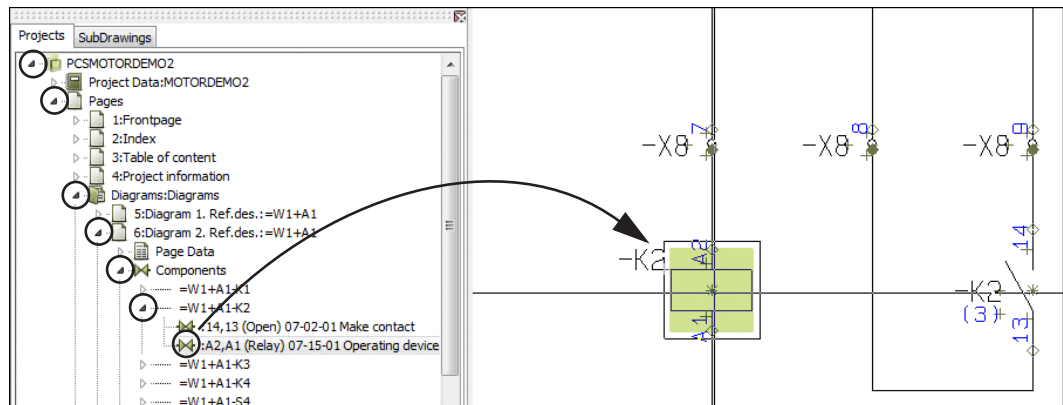
In the project **PCSMOTORDEMO2** you can for instance do this in the **Projects** tab: Click the triangle icon next to **PCSMOTORDEMO2** and click the triangle icon next to **Pages**. You now get a survey of all the pages in the project.

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.

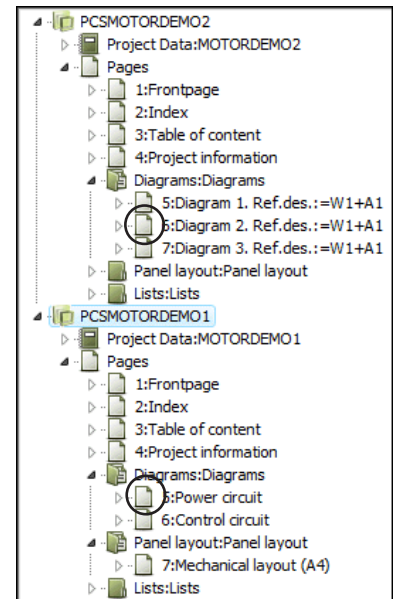
Please note, that you can do the same for **Cables** and **Signals**.

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.

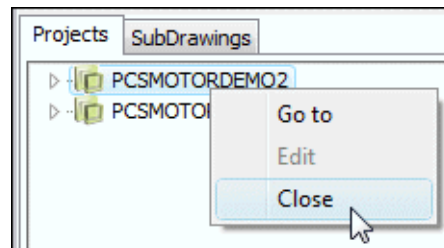


UPDATING THE EXPLORER WINDOW

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 301 and “Find and Replace” on page 316.

THIS CHAPTER DESCRIBES	
— ON THE DIALOG BOX OBJECT LISTER	P. 307
— SELECTING OBJECTS VIA THE OBJECT LISTER	P. 308
— LOCKING THE OBJECT LISTER TO A REFERENCE DESIGNATION	P. 308
— THE SORTING ORDER OF DATA IN THE OBJECT LISTER.....	P. 309
— SELECTING OBJECTS	P. 309
— COUNTING ON SELECTED OBJECTS.....	P. 310
— DISPLAYING INFORMATION	P. 311
— PRINTING	P. 311
— SAVE TO FILE	P. 311
— CHOOSING NEW COMPONENTS IN THE DATABASE.....	P. 312
— SEARCHING.....	P. 312
— SHOWING THE NUMBER OF FOUND OBJECTS	P. 313
— EDITING SYMBOL AND CABLE DATA VIA THE OBJECT LISTER	P. 313
— EDITING PLC I/O DATA VIA THE OBJECT LISTER.....	P. 313
— EDITING PROJECT TEXTS VIA THE OBJECT LISTER.....	P. 314
— EDITING PAGE DATA VIA THE OBJECT LISTER	P. 315
— OBJECT LISTER FOR ONE COMPONENT	P. 315
— TEXT LINKS IN THE OBJECT LISTER.....	P. 315

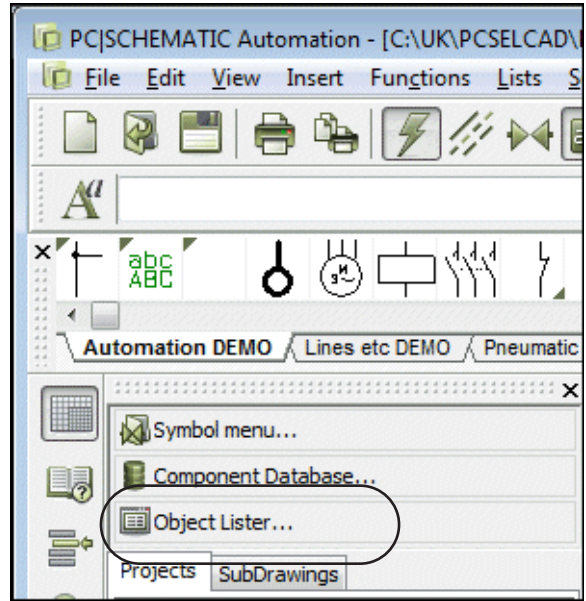
ON THE OBJECT LISTER DIALOG BOX

To enter the Object lister you choose **Functions => Object Lister**, or press the short-cut key <F7>.

You can also click the **Object Lister** button in the upper part of the **Explorer** window, when you wish to enter the **Object Lister**.

When you enter the **Object Lister**, you must first specify which type of objects that you want to see the attached texts/data for. *Then* the **Object Lister** is filled with data.

By choosing an object type at the top of the **Object Lister**, you determine whether to show **Symbols, Signals, Texts, Wire numbers, Cables, PLC I/O, Pages, Lines** (only lines with article data are displayed) or **Reference frames**.



Update

View object in project

Edit data

Fetch data from database

Search

Click on an object, and it is displayed in the project

Lock Object Lister to reference designation

Choose which type of objects to display

Save as file e.g. for Excel

For scrolling up/down and left/right in the list

Name	Conn.	Type	Article	Function	Symbol	Page No.	Position
=W1+A1-F1	1,2...(6)	PCSMV001A	PCS2250001		H7315D21	5	x=70,00 y=...
=W1+A1-F1	1,2...(10)	PCSMV001A	PCS2250001		PCSMV1	8	x=132,27 y=...
=W1+A1-F2	1,2...(6)	PCSMV001A	PCS2250001		H7315D21	5	x=110,00 y=...
=W1+A1-F2	1,2...(10)	PCSMV001A	PCS2250001		PCSMV1	8	x=186,27 y=...
=W1+A1-Q1	1,2...(6)	PCSCON06	PCS2250106		H7313-02	5	x=70,00 y=...
=W1+A1-Q1	14,13	PCSCON06	PCS2250106		07-02-01	6	x=110,00 y=...
=W1+A1-Q1	A2,A1	PCSCON06	PCS2250106		07-15-01	6	x=70,00 y=...
=W1+A1-Q1	1,2...(10)	PCSCON06	PCS2250106		PCSCON2	8	x=148,73 y=...
=W1+A1-Q2	1,2...(6)	PCSCON06	PCS2250106		H7313-02	5	x=110,00 y=...
=W1+A1-Q2	14,13	PCSCON06	PCS2250106		07-02-01	6	x=190,00 y=...
=W1+A1-Q2	A2,A1	PCSCON06	PCS2250106		07-15-01	6	x=150,00 y=...
=W1+A1-Q2	1,2...(10)	PCSCON06	PCS2250106		PCSCON2	8	x=191,83 y=...
=W1+A1-Q3	1,2...(6)	PCSCON08	PCS2250108		H7313-02	5	x=150,00 y=...
=W1+A1-Q3	13,14	PCSCON08	PCS2250108		07-02-01	6	x=350,00 y=...
=W1+A1-Q3	A1,A2	PCSCON08	PCS2250108		07-15-01	6	x=310,00 y=...
=W1+A1-Q3	1,2...(10)	PCSCON08	PCS2250108		PCSCON2	8	x=278,83 y=...

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 836.

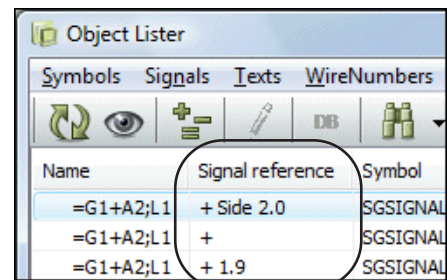
Signals in the Object Lister

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**.

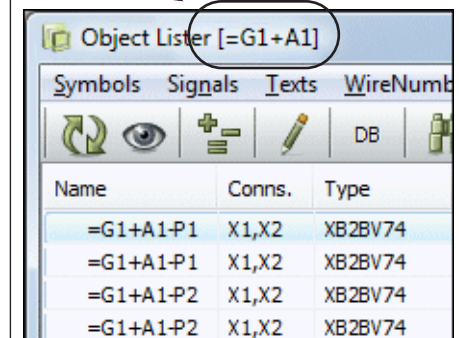
LOCKING THE OBJECT LISTER TO A REFERENCE DESIGNATION



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

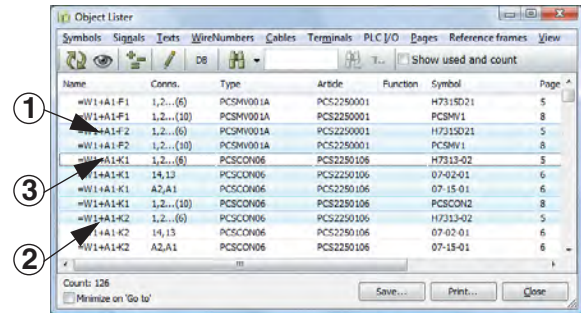
When you have selected more than one object when clicking on **Print** in the **Object Lister**, only the selected objects are printed. When only one object is selected, all of the list is printed. — See “Printing from the Object Lister” on page 311.

This also applies when you click on **Save** to save a file. — See “Save as File from the Object Lister” on page 311.

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.



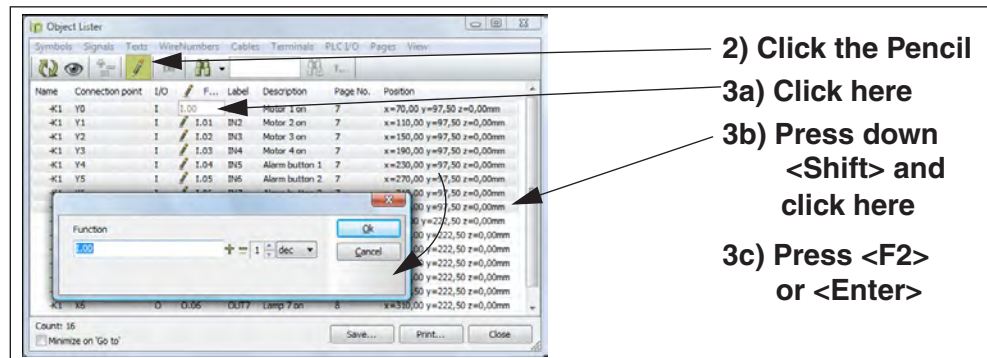
The chosen objects can now for instance be printed, saved to a file, or the article data for the selected objects can be edited.

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 309 —, 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 116. Click **OK**.

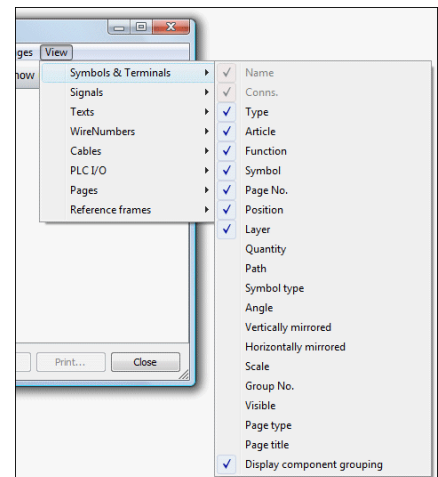
DISPLAYING INFORMATION IN THE OBJECT LISTER

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.

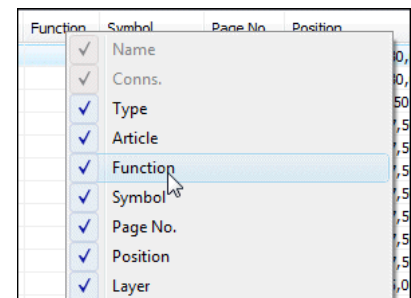
When a check mark is placed in front of a type of information which is dimmed, this specifies that this information can *not* be deselected in the **Object Lister**.



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 309.

When you click on **Print**, you enter the dialog box **Print**, in which you choose printer and print the selected.

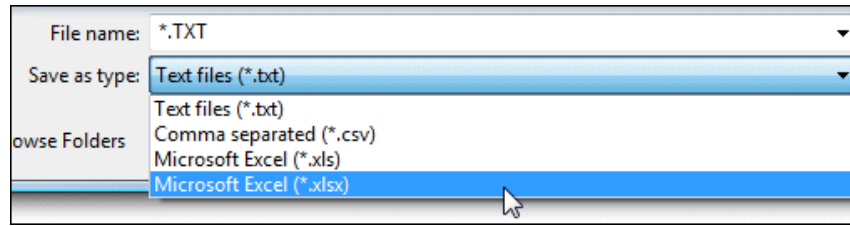
SAVE AS FILE FROM THE OBJECT LISTER

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 309.

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 309.

SEARCHING IN THE OBJECT LISTER



When you click on the **Find** button, you can search among the selected objects in the **Object Lister**.

Click here and choose what to search for

Type in the search text and press <Enter>



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.

Name	Count
=W1+A1-F1	2
=W1+A1-F2	2
=W1+A1-K1	4
=W1+A1-K2	4
=W1+A1-K3	4

You can also see how many times each symbol is used in the project.

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 315.

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)	PCSMV00 1A	PCS2250001
=W1+A1-F2	1,2...(6)	PCSMV00 1A	PCS2250001

If for instance you want to change the symbol type, click in the **Type** field for the relevant symbol, and make the change.

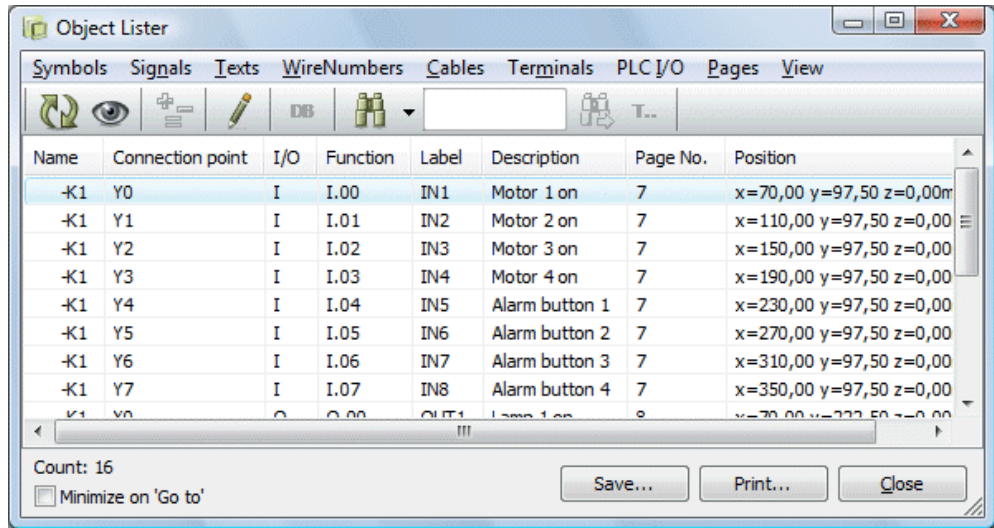
When you click on **Cables** you can make similar changes for **Cable data**.

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 313.

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.



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 312.

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 310.

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 311.

CHECKING AND EDITING PROJECT TEXTS VIA THE OBJECT LISTER

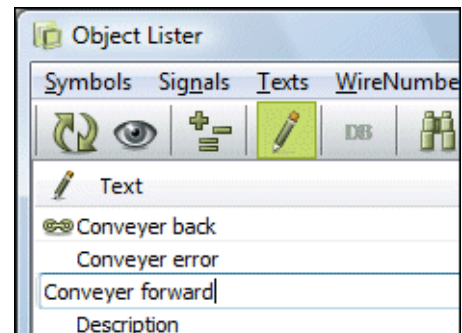


It is a good idea to check the project texts by clicking on **Texts**. If you need to make corrections, click the **Pencil**, and click in the text field you wish to edit.

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 310.



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.

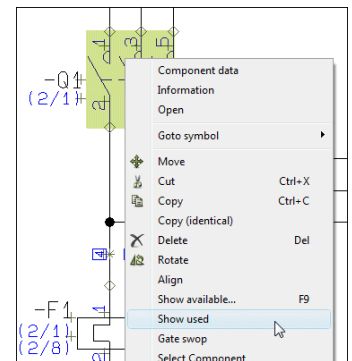
Page type	Size	Page No.	Title
IGN	A4v	F1	Frontpage
IGN	A4v	INF1	
IGN	A4v	INF2	

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:



Name	Conns.	Type	Article	Function	Symbol	Page No.	Position	Layer
=G1+A2-Q1	1,2...(6)	LS15K11/230	4022903075387		H7313-02	1	x=125,00 y=205,00 z=0,00mm	1
=G1+A2-Q1	13,14	LS15K11/230	4022903075387		07-02-01	2	x=157,50 y=202,50 z=0,00mm	1
=G1+A2-Q1	21,22	LS15K11/230	4022903075387		07-02-03	2	x=227,50 y=122,50 z=0,00mm	1
=G1+A2-Q1	A1,A2	LS15K11/230	4022903075387		07-15-01	2	x=87,50 y=100,00 z=0,00mm	1

Count: 5
 Minimize on 'Go to'

Buttons: Save... Print... Close

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 132.

FIND AND REPLACE

To find objects in projects quickly, you either use the *Object Lister* — see “The Object Lister” on page 306 — 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.

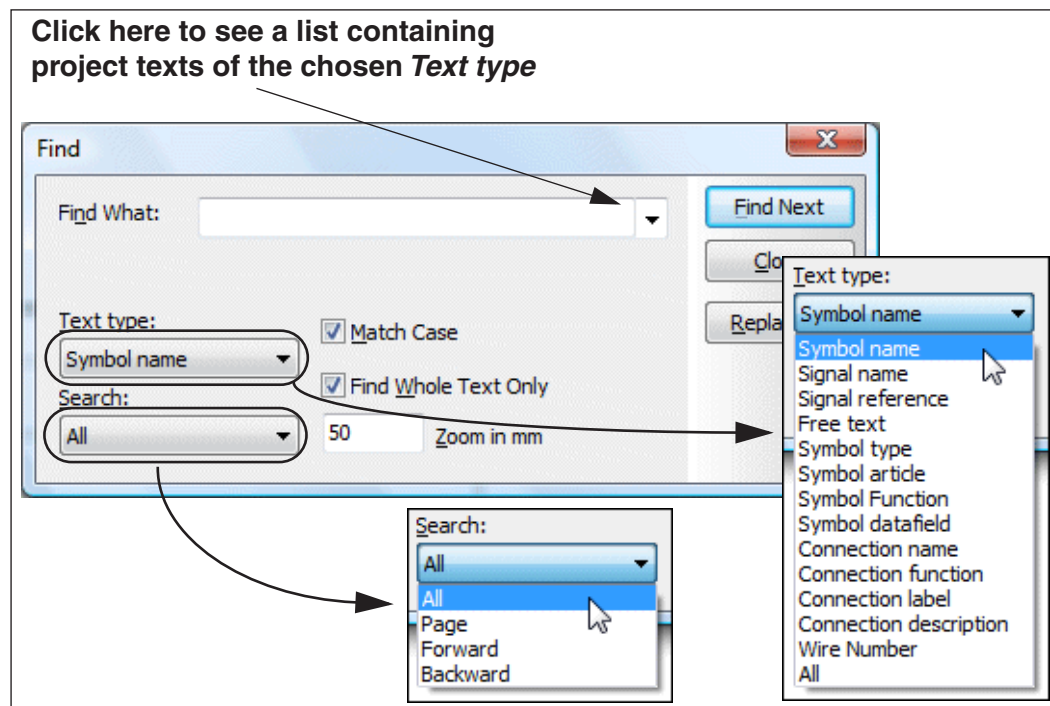
The Find function gives you the option to find symbol texts, and the replace function gives you the option to replace them as well.

THIS CHAPTER DESCRIBES	
— FIND SYMBOL TEXTS.....	P. 316
— REPLACE SYMBOL TEXTS.....	P. 318
— THE DIALOG BOX FIND AND REPLACE REMAINS ON SCREEN.....	P. 318
— NEGATED AND UNDERLINED TEXTS IN FIND AND REPLACE	P. 319

— Other ways to find *symbols* quickly are described in “Functions on Component Level” on page 228.

FINDING SYMBOL TEXTS

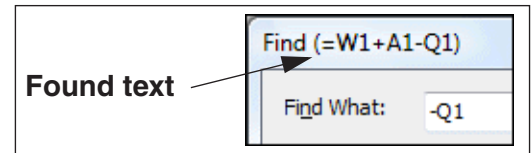
If you want to find symbol texts, you choose **Edit => Find** (short-cut key <Ctrl+f>) and enter this dialog box:



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.

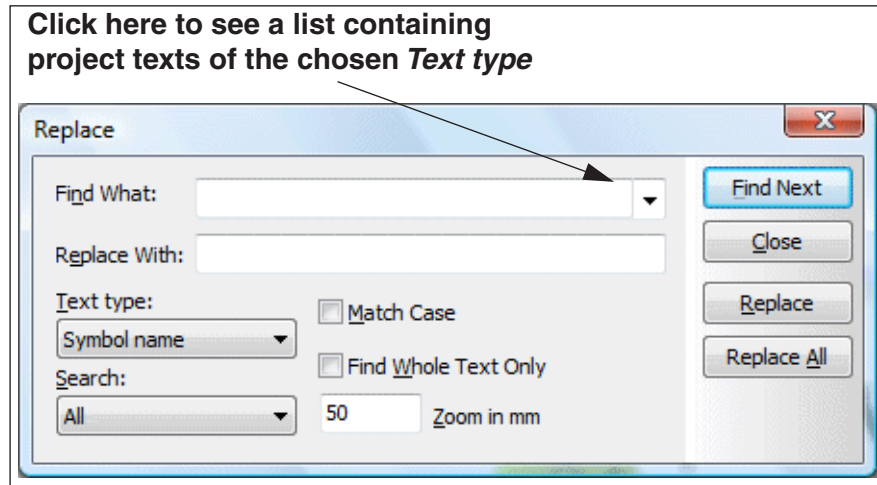
Find next symbol

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.

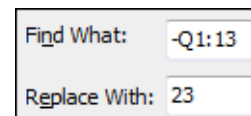
Clicking on **Replace All** will replace all matching texts in the specified area.

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 836.

NEGATED AND UNDERLINED TEXTS IN FIND AND REPLACE

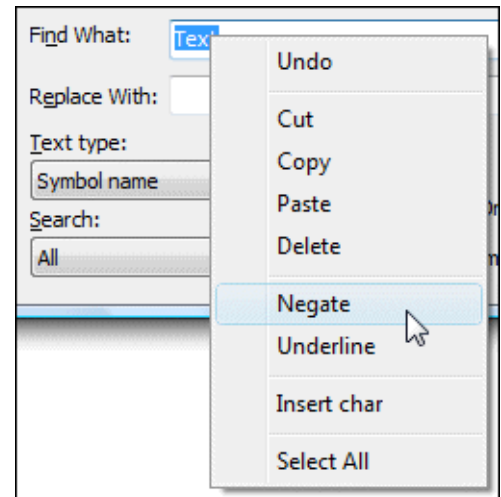
In the dialog boxes for **Edit => Find** and **Edit => Replace** you can search for negated and underlined texts.

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

When using the design check function in a project, you are notified about the most obvious drawing errors:

- If for instance you want your diagrams to be mounting correct, you can check the project for connection dots
- If you have chosen to create very detailed connection lists, you can check for consistent line data
- Or you can make a simple check whether the diagrams contain short-circuiting or unconnected and/or parked lines

Thus the design check function can support you, when you wish to check your design.

THIS CHAPTER DESCRIBES	
— PERFORMING A DESIGN CHECK	P. 320
— THE DESIGN CHECK DIALOG BOX	P. 323
— DISABLING USED MORE THAN ONCE IN THE SYMBOL DEFINITION	P. 324
— DESIGN CHECK SETTINGS	P. 324



USE THE DESIGN CHECK FUNCTION FREQUENTLY

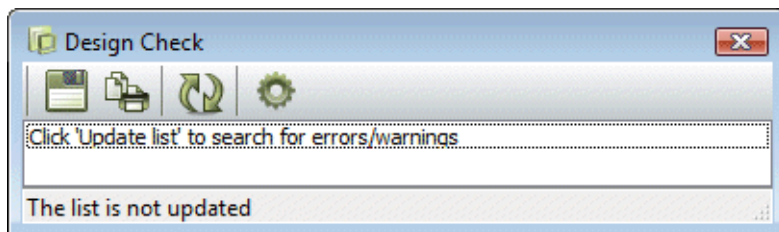
IT IS A GOOD IDEA TO APPLY THE DESIGN CHECK FUNCTION FROM TIME TO TIME WHILE YOU ARE DRAWING. THIS WAY YOU DO NOT HAVE TO BOTHER WITH TOO MANY ERROR MESSAGES AT THE SAME TIME.

YOU CAN ALSO CHOOSE TO HAVE SOME DESIGN CHECKS MADE ONLINE, IE WHILE DRAWING.

PERFORMING A DESIGN CHECK

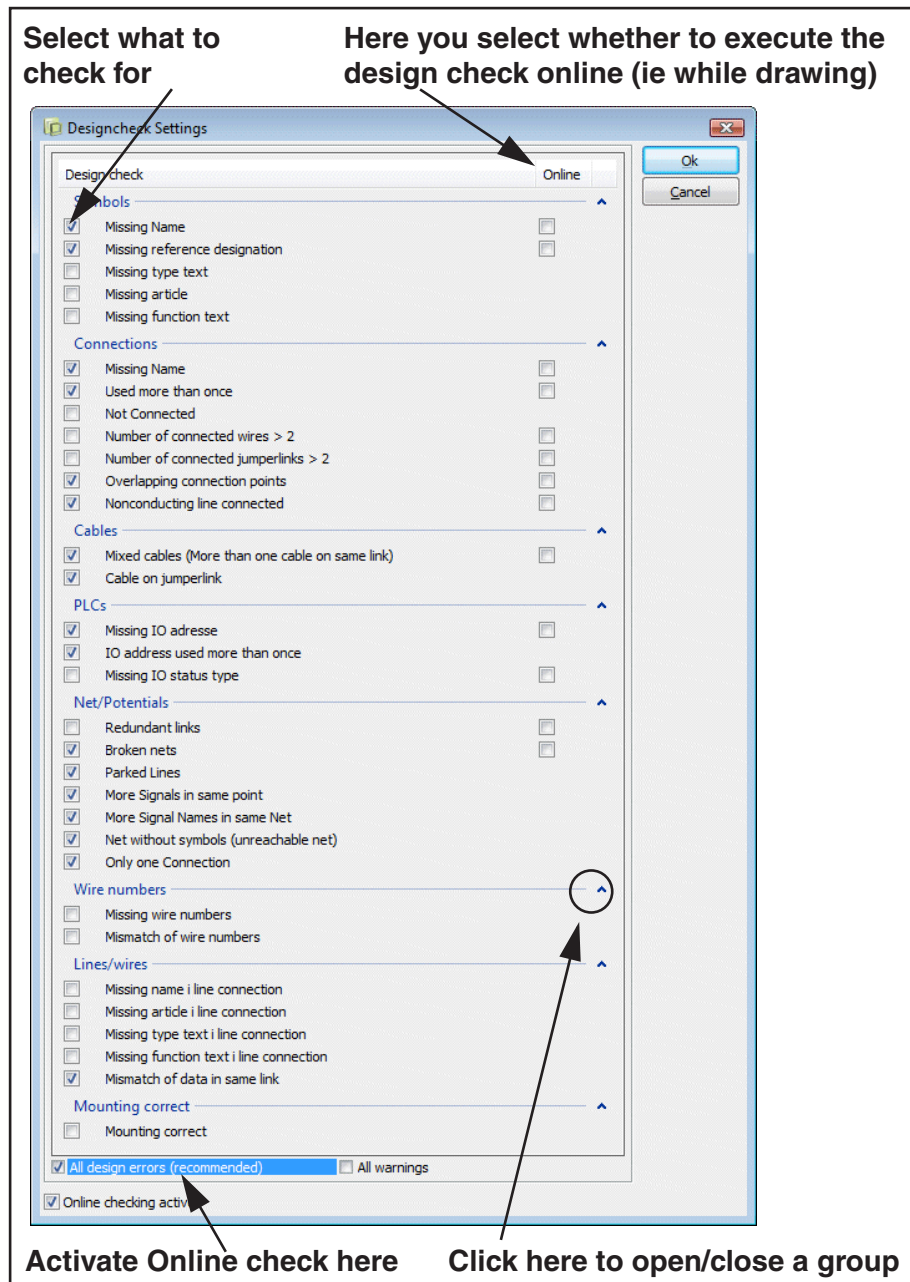


- 1) To activate the function, choose **Functions => Design Check**, or click the **Design Check** button.



- 2) In the **Design Check** dialog box: Click the **Design Check settings** button.

3) In the dialog box **Design Check settings** you now get the following options:



4) Select which errors to check the project for. To maintain the overview, it may be a good idea just to check the project for one type of errors at a time. If you select **All design errors (recommended)** at the bottom of the dialog box, all checks for design errors are selected in the dialog box.

If you select **All warnings** all checks are selected in the dialog box.



ACTIONS PERFORMED BEFORE DESIGN CHECK CAN NOT BE UNDONE

WHEN YOU EXECUTE A DESIGN CHECK, THE PROJECT IS ALSO OPTIMIZED. THEREFORE YOU DO *NOT* HAVE THE OPTION OF UNDOING ACTIONS PERFORMED *BEFORE* YOU EXECUTED THE DESIGN CHECK. DURING THE OPTIMIZATION PROCESS, LINES THAT ARE DRAWN IN SEVERAL STAGES, ARE FOR INSTANCE CORRECTED TO COHERENT LINES.

You can read more about the options in the dialog box in “Design check settings” on page 324.

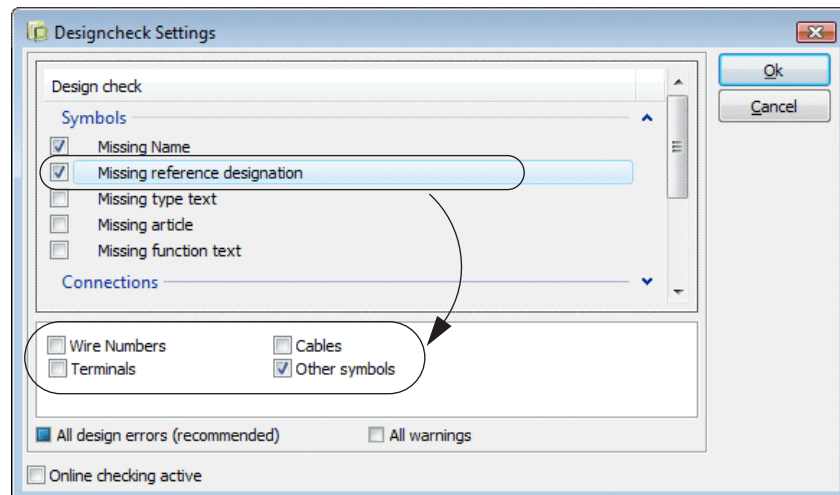
5) Click **OK**.

6) If the design check finds errors, these are now displayed in the dialog box **Design Check**, as described in “The Design Check dialog box” on page 323. If this is not the case, you are told that no errors are found.

Design check for Symbols

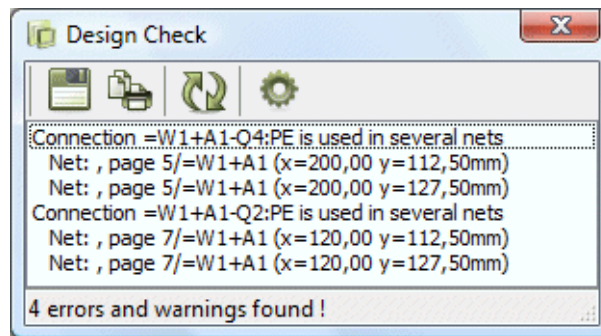
For each check under **Symbols** at the top of the dialog box, you can specify whether the check shall be performed for **Wire numbers**, **Cables**, **Terminals** and/or **Other symbols**. These options appear *at the bottom* of the dialog box, when you have selected one of the options under **Symbols** at the top of the dialog box.

In the figure below, the check for **Missing reference designations** is made for **Other symbols**.



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 836.

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.

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 702.

DESIGN CHECK SETTINGS

Below you get an overview of the options in the dialog box Design Check settings:

GROUP	CHECK FOR	MEANING
SYMBOLS	MISSING NAME, TYPE, ARTICLE NUMBER, FUNCTION	CHECK FOR SYMBOLS WITHOUT NAME/TYPE/ARTICLE NUMBER/FUNCTION IN COMPONENT DATA
CONNECTIONS	MISSING NAME	CONNECTIONS WITHOUT NAMES DO NOT MAKE SENSE IN ELECTRICAL LISTS, SUCH AS CONNECTION LISTS, TERMINAL LISTS, CABLE LISTS AND GRAPHICAL PLANS. BEWARE, THAT ALL PCS SYMBOLS HAVE NAMES, EVEN WHEN THEY ARE INVISIBLE.
	USED MORE THAN ONCE	IT IS AN ERROR IF A COMPONENT'S TERMINAL IS CONNECTED TO TWO OR MORE DIFFERENT NETS/POTENTIALS.
	NOT CONNECTED	UNCONNECTED TERMINALS ARE ALLOWED BY THE PROGRAM, ALSO WHEN GENERATING LISTS. HOWEVER, YOU CAN CHECK FOR UNCONNECTED TERMINALS. NOT CONNECTED CONNECTIONS ARE ALSO FOUND, WHEN A JUMPER/LINE PASSES TERMINALS <i>WITHOUT</i> ACTUALLY CONNECTING. GRAPHICALLY THEY <i>APPEAR TO BE</i> CONNECTED.
	NUMBER OF CONNECTED WIRES >2	MORE THAN TWO WIRES ARE CONNECTED TO THE CONNECTION POINT. THE NUMBER CAN BE CHANGED.
	NUMBER OF CONNECTED JUMPER LINKS >2	MORE THAN TWO JUMPER LINKS ARE CONNECTED TO THE CONNECTION POINT. THE NUMBER CAN BE CHANGED.
	OVERLAPPING CONNECTION POINTS	WHEN TWO CONNECTION POINTS ARE POSITIONED ON TOP OF EACH OTHER.
	NON-CONDUCTING LINE CONNECTED	YOU GET A WARNING WHEN YOU TRY TO CONNECT A NON-CONDUCTING LINE TO A TERMINAL, BUT YOU CAN IGNORE THE WARNING. THIS OPTION CAN SEARCH FOR NON-CONDUCTING CONNECTIONS IN THE PROJECT.

CABLES	MIXED CABLES	CHECK FOR MIXED CABLES MEANING MORE THAN ONE CABLE ON THE SAME LINK. (PREVIOUSLY THE CHECK WAS ONLY AVAILABLE IN CABLE LISTS).
	CABLE ON JUMPER LINK	JUMPER LINKS CAN NOT BE PART OF A CABLE.
PLCs	MISSING IO ADDRESS	MISSING IO ADDRESSES ON A PLC SYMBOL IS AN ERROR.
	IO ADDRESS USED MORE THAN ONCE	CHECK WHETHER THE SAME ADDRESS MORE THAN ONCE IN THE PROJECT.
	MISSING IO STATUS TYPE	IF YOU DECIDE TO USE THIS FUNCTION, YOU CAN CHECK FOR PLC SYMBOLS WITHOUT STATUS TYPE.
NET/ POTENTIALS	REDUNDANT LINES	CHECKS WHETHER A CONNECTION STILL EXISTS WHEN A CONDUCTOR IS REMOVED.
	BROKEN NETS	WHEN MOUNTING CORRECT DRAWING IS APPLIED INCORRECTLY, YOU CAN END UP WITH NETS WITHOUT CONNECTIONS. THIS FUNCTION CAN ALSO BE USED FOR CHECKING PROJECTS FROM PREVIOUS VERSIONS.
	PARKED LINES	CHECK FOR LINES WITHOUT CONNECTION TO SIGNALS OR OTHER SYMBOLS OR LINES.
	MORE SIGNALS IN SAME POINT	CHECK FOR MORE THAN ONE SIGNAL SYMBOL IN THE SAME POINT.
	MORE SIGNALS IN SAME NET	SHORT CIRCUIT – CHECK FOR MORE SIGNAL NAMES IN THE SAME NET.
	NET WITHOUT SYMBOLS (UNREACHABLE NET)	BOXES AND THE LIKE DRAWN WITH CONDUCTING LINES. THESE WILL BE CONSIDERED AS NETS BY THE PROGRAM, BUT THEY WILL BE UNREACHABLE.
	ONLY ONE CONNECTION	A NET WITH ONLY ONE CONNECTION CAN NOT HAVE ELECTRICAL FUNCTIONS.
WIRE NUMBERS	MISSING OR MISMATCH	LINKS WITHOUT WIRE NUMBERS OR DIFFERENT WIRE NUMBERS ON THE SAME LINK.
LINES/ WIRES	MISSING NAME, TYPE, ARTICLE NUMBER, FUNCTION	LINES WITHOUT NAME/TYPE/ARTICLE NUMBER/FUNCTION IN COMPONENT DATA.
	MISMATCH OF DATA IN SAME LINK	WHEN YOU USE DETAILED CONNECTION LISTS, THAT CONTAIN LINE DATA, THE PROGRAM CAN CHECK FOR DIFFERENT DATA (MISMATCH) ON TWO OR MORE SEGMENTS IN THE SAME LINK.
MOUNTING CORRECT	MOUNTING CORRECT	CHECKS WHETHER ALL CONNECTIONS ARE MADE WITH BENDS AND NOT DOTS.

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. 326
— THE RIGHT-HAND SIDE OF THE DIALOG BOX	P. 327
— SELECT PAGES	P. 327
— SETUP	P. 329
— CHOOSE PRINTER	P. 330
— BATCH PRINTING	P. 331
— IF DIAGRAMS ARE MINIMIZED WHEN PRINTING.....	P. 331
— EXPORTING PDF FILES	P. 331
— EXPORTING DWG/DXF FILES.....	P. 333

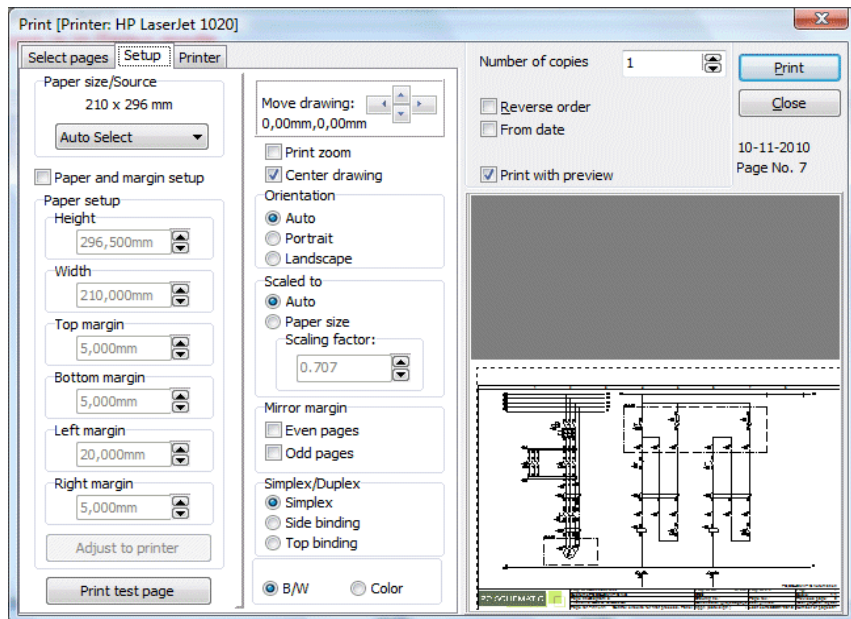


If you only want to print out the page you are working on, and do not wish to make changes in the printer settings, click the **Print this page** button.

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.

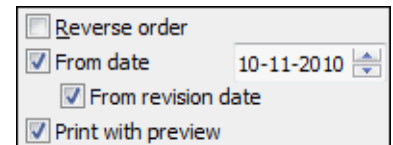
When more than one page and more than one copy is chosen, you can also choose **Collate**, if the chosen printer driver can handle the collate information.

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 808 — 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 809 also.

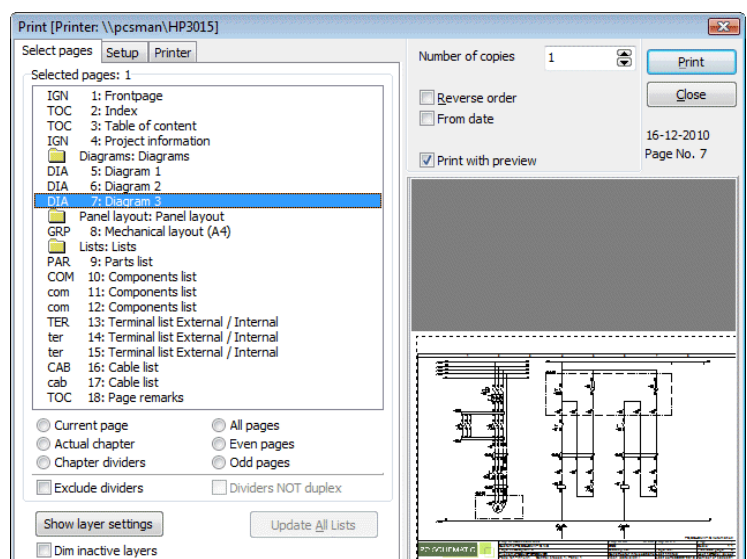
SELECT PAGES

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.

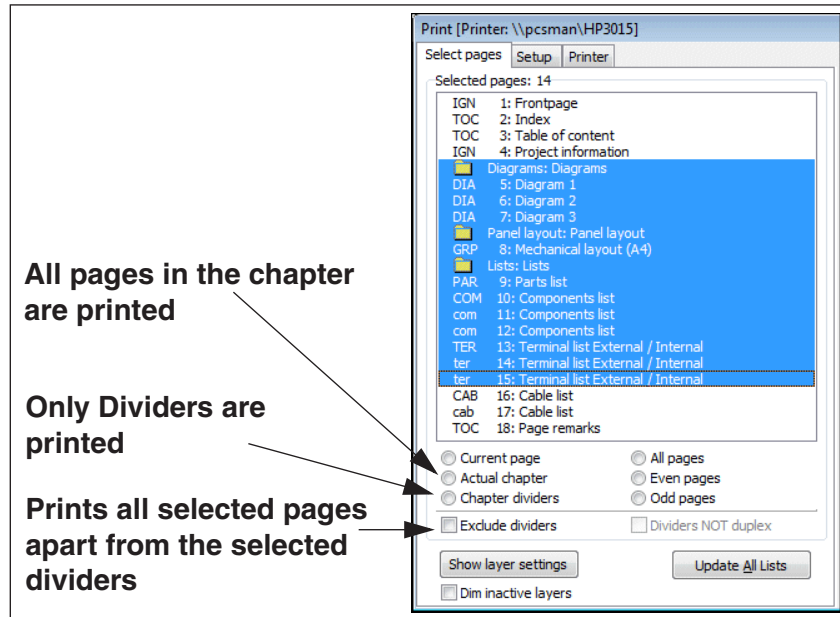
Pages marked with a preceding + character, have been changed since the project was last saved.

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 check box is dimmed if the selected printer can not handle duplex.

Printing layers

If you click on **Show layer settings**, you can specify which layers or layer groups on the drawing to print. — See “Printing pages with layers” on page 481.

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 478.

Printing lists

When one of the pages, that you wish to print, is a list page, the button **Update all lists** is displayed at the bottom of the **Select pages** tab.

When you click **Update all lists** before printing, the most recent changes in the project are included in the list pages.

However, if you do *not* click **Update all lists** before printing, the list pages are printed as they appear in the project. Therefore, if any changes has been made in

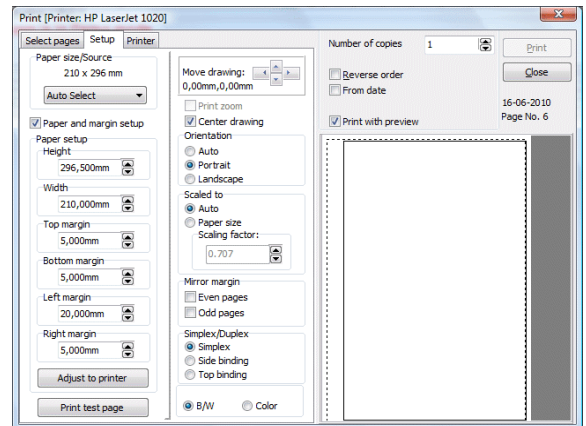
the project since the list pages were last updated, these changes are *not* included in the printed list pages.

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

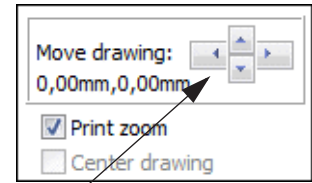
To print a zoom, do the following:



- 1) Click on the **Zoom** button, and make a zoom — see “Zoom” on page 42.



- 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.
- 4) Click on **Print**, and the chosen zoom is printed.



Click here to move the chosen zoom

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, where you choose which printer to use for printing. Here you also have the option of opening the general Windows dialog box for print settings. 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. However, if you have selected **Ask when changing printer name**, you will be asked whether to remember the chosen printer, when you select a new printer.

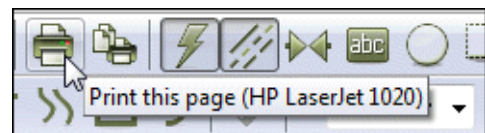
Alternative PCISCHEMATIC font

In the field **Alternative PCSCHEMATIC font** you can specify whether to replace the PCSCHEMATIC font when printing.

— See “Applying other fonts when printing/exporting PDF files” on page 332.

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 823.

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 660 and “The Zoom All function” on page 43.

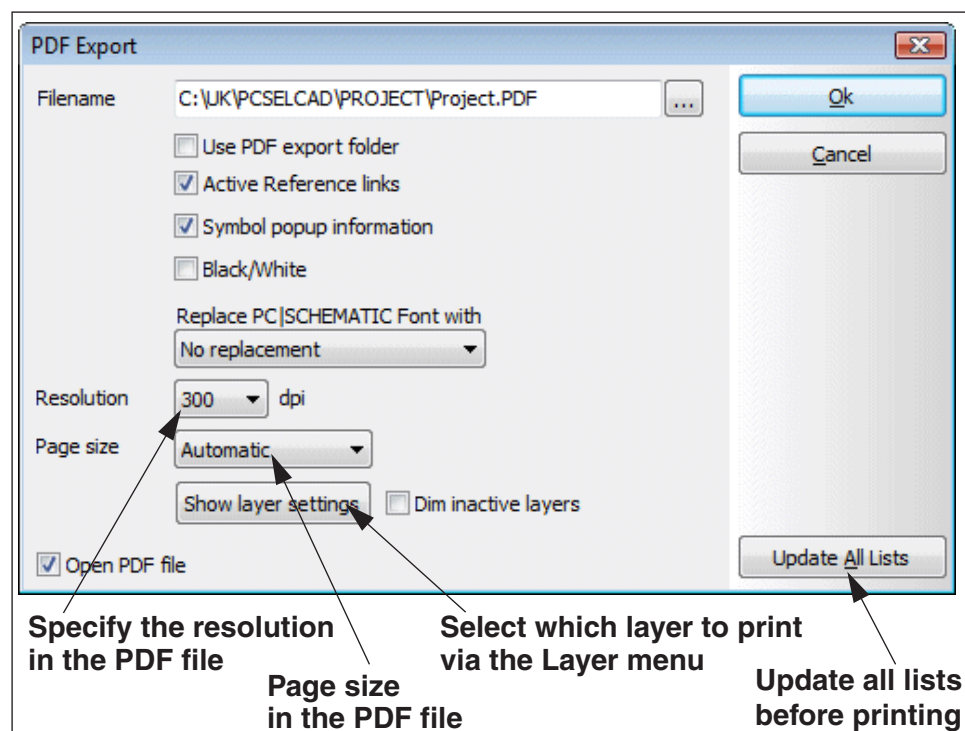
EXPORTING PDF FILES

In **Automation** you can export the active project as a PDF file.

You can start the export in two ways:

- Choose **File => Export to => 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:

Use PDF export folder

When you select **Use PDF export folder**, the PDF file is saved in the folder specified for *Directories* for PDF files. Read more about this in “Directories” on

page 840. If this option is not selected, the PDF file is saved in the same folder as the project.

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 exporting/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 OTHER FONTS WHEN PRINTING/EXPORTING PDF FILES

When printing/exporting PDF files, it is possible to substitute the **PC|SCHEMATIC** font with for instance the **Arial** font, which is the *Windows* font that is 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 *Arial* font in the **Alternative PC|SCHEMATIC font** field. You also have the option of choosing other fonts.

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 123.



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 822.

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 660.

WINDOWS EXPLORER AS SYMBOL/PROJECT MENU

THIS CHAPTER DESCRIBES

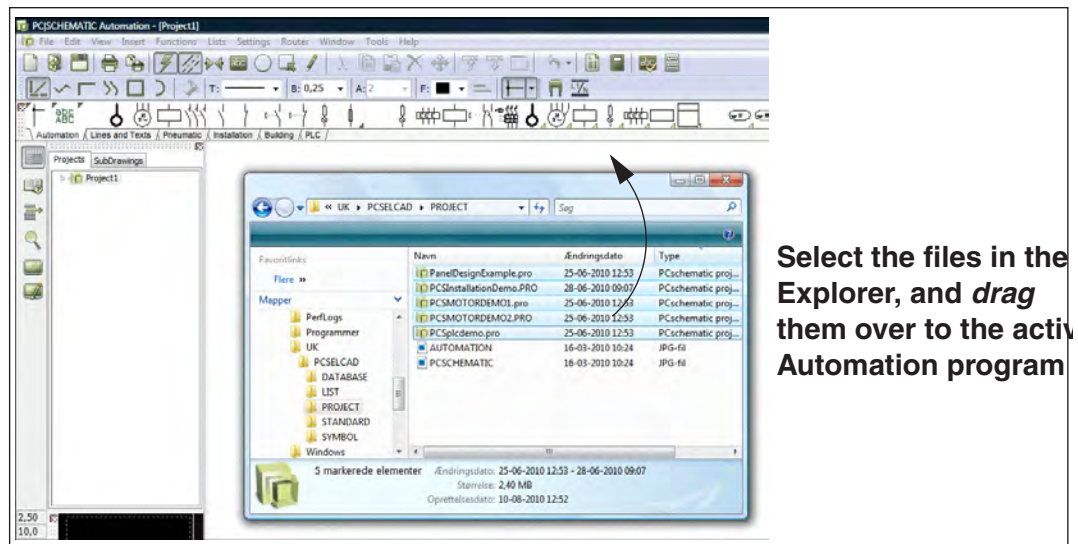
- USING THE WINDOWS EXPLORER AS A PROJECT MENU..... P. 334
- USING THE WINDOWS EXPLORER AS A SYMBOL MENU P. 334

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**:



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. 335
— DEFINING REFERENCE DESIGNATIONS	P. 338
— REFERENCE DESIGNATIONS FOR SYMBOLS	P. 349
— REFERENCE DESIGNATIONS FOR PAGES	P. 353
— REFERENCE FRAMES / REFERENCE DESIGNATIONS IN AREAS	P. 354
— REFERENCE DESIGNATIONS FOR CHAPTERS	P. 358
— REFERENCE DESIGNATIONS FOR MORE THAN ONE PAGE AT A TIME	P. 358
— REFERENCE DESIGNATIONS VIA CHAPTER TABS	P. 361
— DATAFIELDS FOR SPECIFYING REFERENCE DESIGNATIONS IN PROJECTS	P. 363
— LISTS ACCORDING TO FUNCTION OR LOCATION ASPECT	P. 364



LEARN MORE ABOUT REFERENCE DESIGNATIONS

IN THE BOOK "ELECTRICAL DOCUMENTATION ACCORDING TO STANDARDS", YOU CAN READ MORE ABOUT APPLYING REFERENCE DESIGNATIONS IN ELECTRICAL DOCUMENTATION. THE BOOK IS PUBLISHED BY PC|SCHEMATIC A/S, AND IS NOW IN ITS 3RD 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.

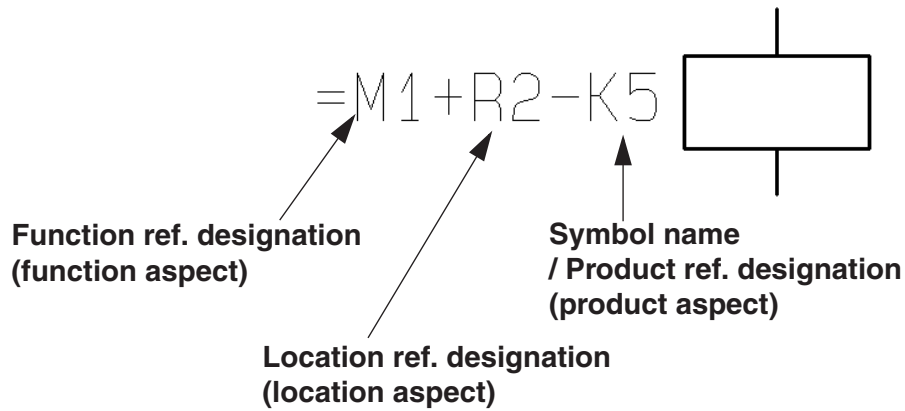
A reference designation must consist of a prefix, followed by a capital letter, followed by an integer. For instance **-M01**.

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.

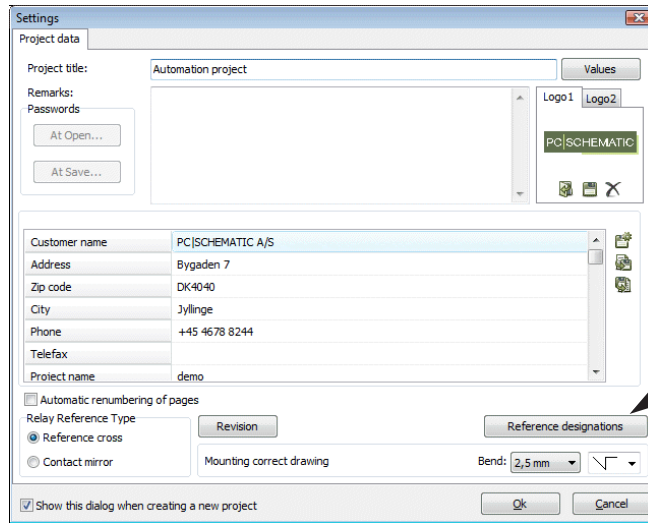
Please note, that this character can therefore also appear when you have forgotten to specify a reference designation for the component.

The > character is called "exclude concatenation", which you can read more about in "Use full exclude of concatenation" on page 345.

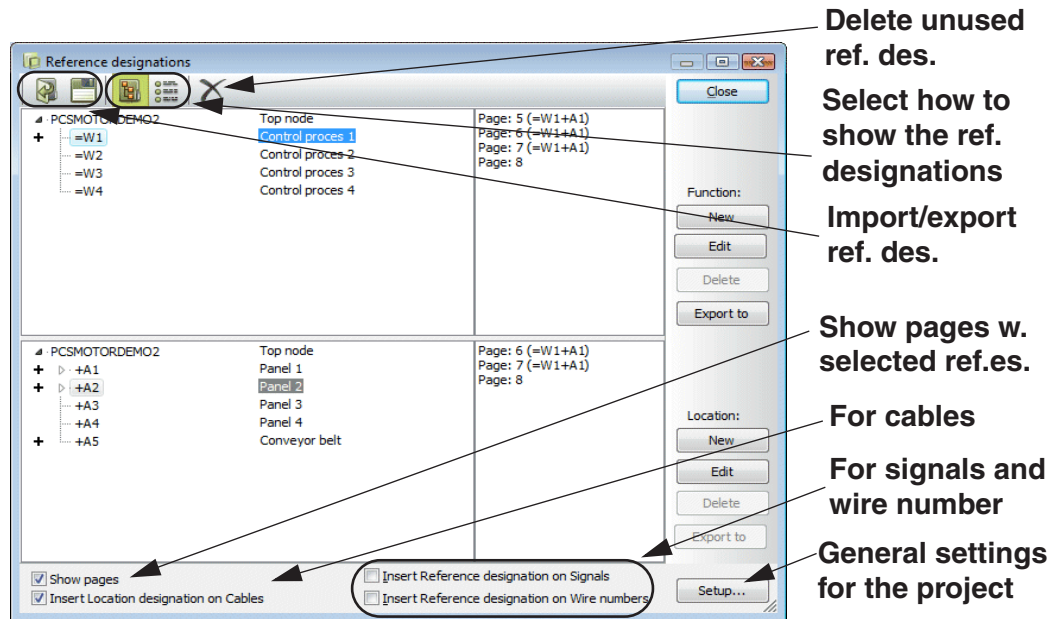


DEFINING REFERENCE DESIGNATIONS FOR THE PROJECT

To create reference designations in your project, you either click the **Reference designations** button, choose **Settings => Project data** and click **Reference designations**.



You then 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.



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 THE **REFERENCE DESIGNATIONS** DIALOG BOX. HAVING DONE THIS, HOWEVER, YOU CAN SELECT REFERENCE DESIGNATIONS IN **SETTINGS => PAGE DATA** AS WELL AS IN THE DIALOG BOX **COMPONENT DATA**.

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. 339
— SPECIFYING FUNCTION/LOCATION ASPECTS	P. 340
— SEE USED FUNCTION/LOCATION ASPECTS	P. 341
— SORTING FUNCTION/LOCATION ASPECTS BY COLUMN	P. 341
— EDITING FUNCTION/LOCATION ASPECTS	P. 342
— DELETING FUNCTION/LOCATION ASPECTS	P. 343

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



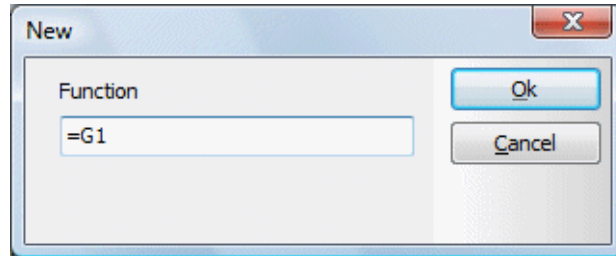
When you click the **Show as list** button, the reference designations are displayed as a list, with all of the reference designations displayed at the same level — see below.

<p>Hierarchical display of reference designations</p>	<p>List display of reference designations</p>
--------------------------------------------------------------	------------------------------------------------------

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.

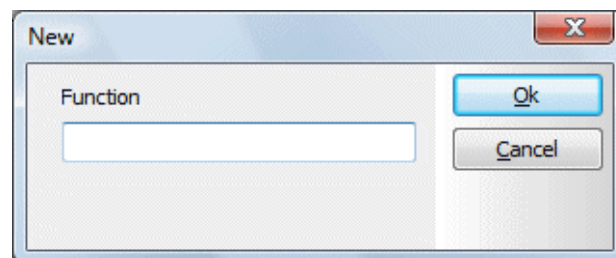
Type in the name of the new function aspect and click **OK**.



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 339 — 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

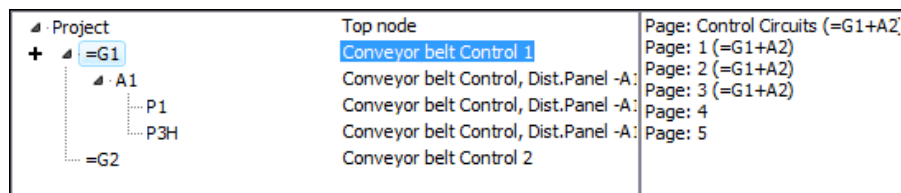
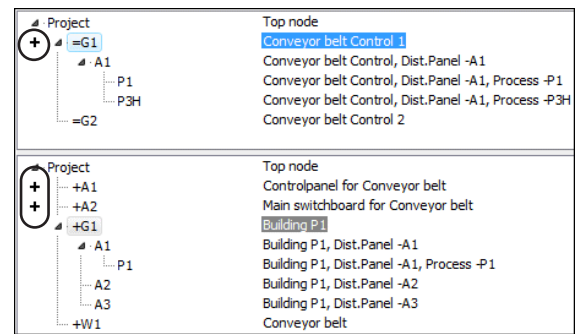
The reference designations currently applied in the project are marked with a + next to the reference designation.



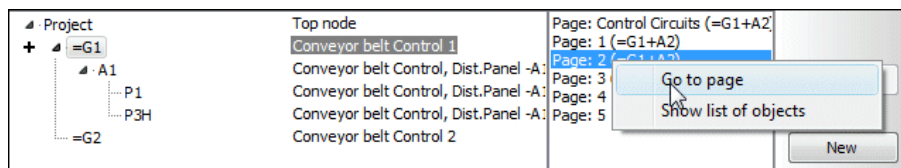
If you wish to delete the unused reference designations in the project, click the **Delete unused** button.

See on which page the reference designations are used

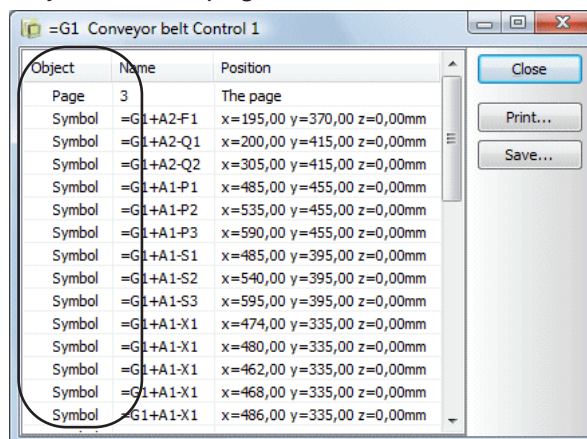
If you wish to see on which page a reference designation is used, select **Show pages** in the bottom-left corner of the dialog box, and the pages are displayed in the right-hand side of the dialog box:



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

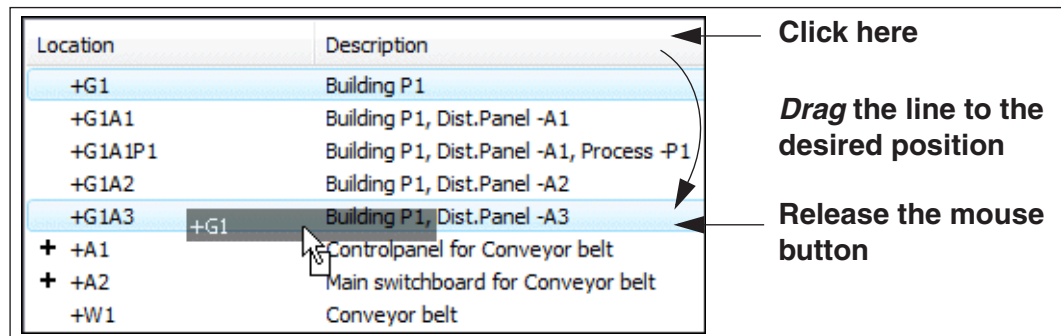
Please note: *This only applies for displaying reference designations as a list* — see “Display reference designations as a list” on page 339.

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 309.

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.

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 358.

Changing reference designations for Chapters

Changing reference designations for a full chapter, see “Editing reference designations for chapters” on page 358.

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 210.

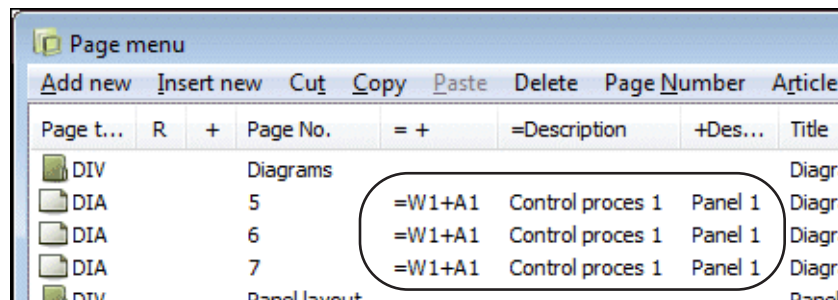
Inserting the location aspect for cables

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.

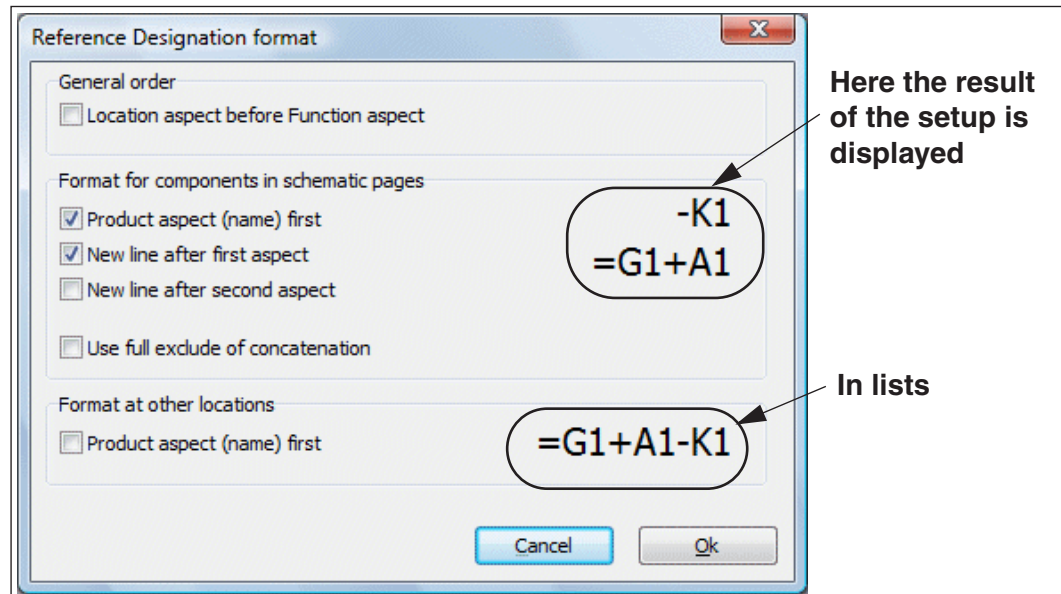
Reference designations in the Page menu

When you use reference designations in a project, the reference designations also appear in the **Page menu**.



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 351 and “The tab Ref.design.” on page 161.

Use full exclude of concatenation

— See “Use full exclude of concatenation” on page 345.

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.

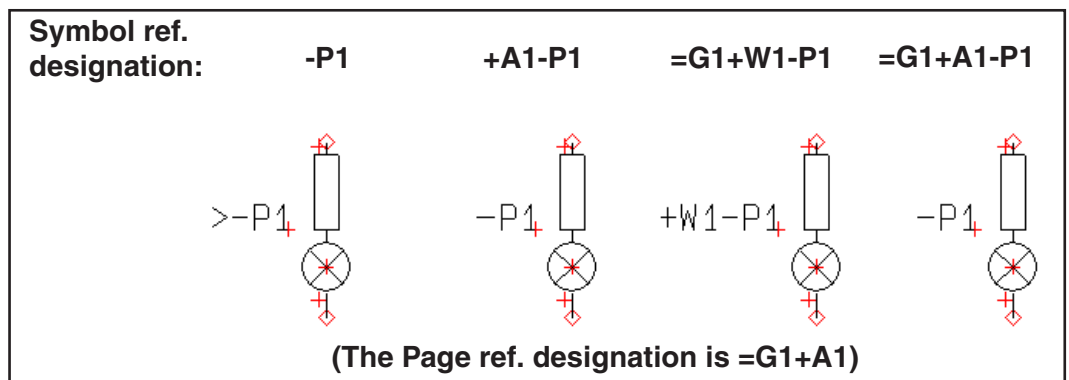
IF IT THEREFORE SAYS **>+A1-P1** FOR A SYMBOL, THEN **+A1-P1** IS THE **FULL** REFERENCE DESIGNATION FOR THE SYMBOL.

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:

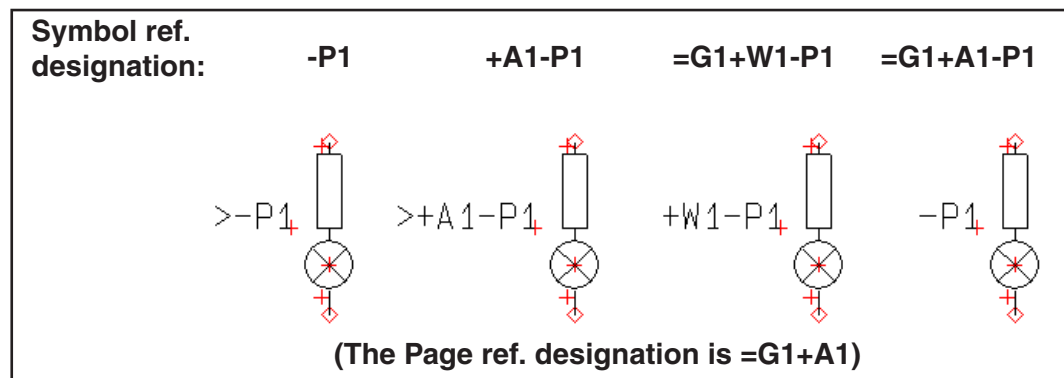


Explanation:

- 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 comma separated files (*.csv files).



If you wish to import reference designations, click the **Import** button, and you get the option of choosing which file you wish to import reference designations from. If you wish to export the project reference designations to a file, click the **Export all** button.

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.

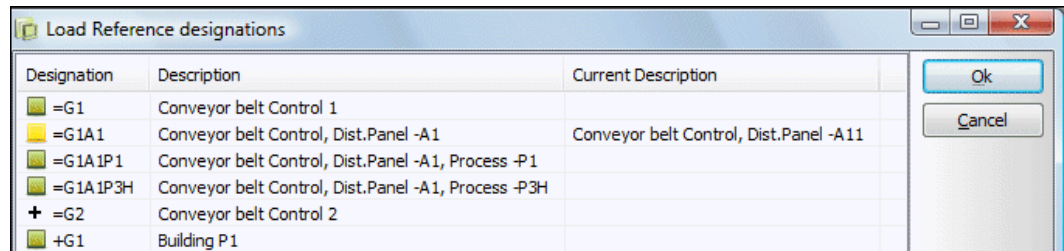
Format for Reference designations files

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

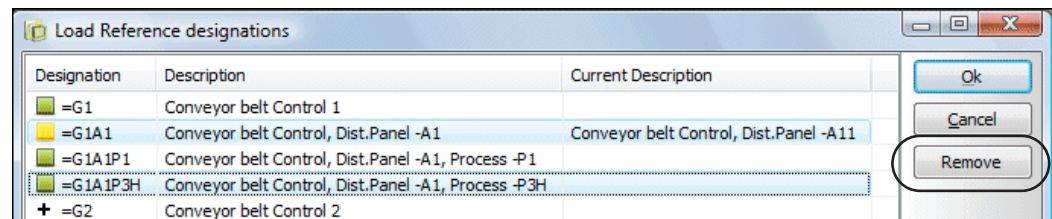
When you import reference designations, you enter the following dialog box:



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:



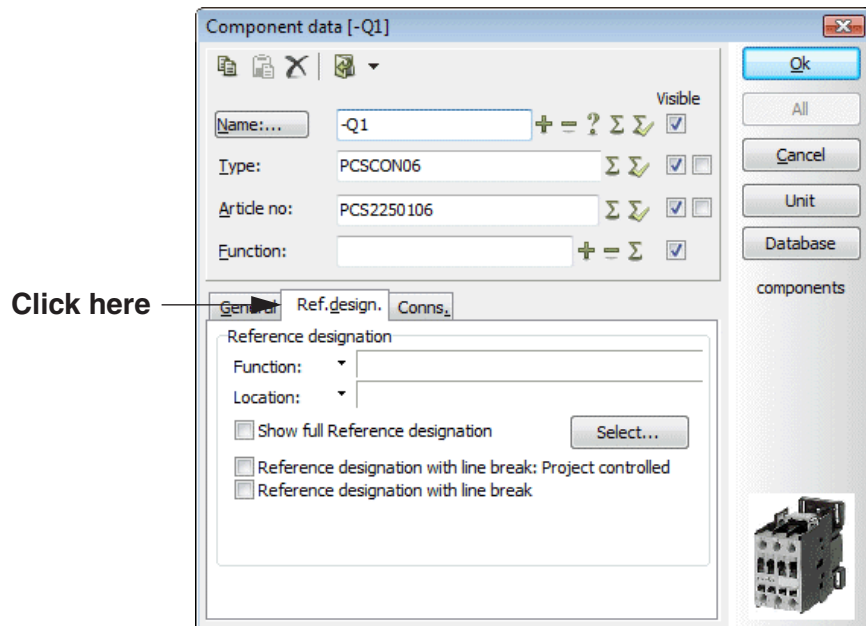
Then import the remaining reference designations by clicking **OK**.

REFERENCE DESIGNATIONS FOR SYMBOLS

THIS CHAPTER DESCRIBES	
— REFERENCE DESIGNATIONS WHEN PLACING SYMBOLS.....	P. 349
— EDITING REF. DES. FOR ALREADY PLACED SYMBOLS.....	P. 352
— EDITING REF. DES. USING THE SELECT NAME FUNCTION.....	P. 352
— PRECEEDING "-" IN SYMBOL NAMES.....	P. 352
— INSERTING REFERENCE DESIGNATIONS FOR SIGNAL SYMBOLS	P. 352

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.

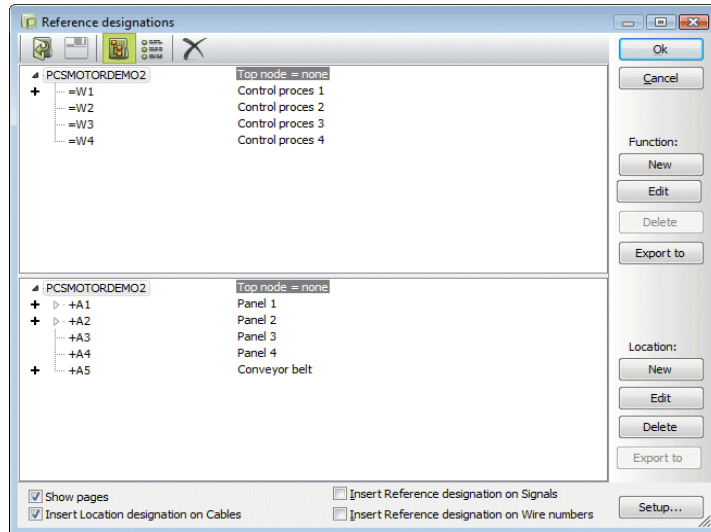


IF THE TAB REF.DESIGN DOES NOT APPEAR

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 338.

Choosing reference designations when placing symbols

When you click on the tab **Ref. design.** and then **Select**, you enter this dialog box:



ONLY CREATED ASPECT TYPES DISPLAYED

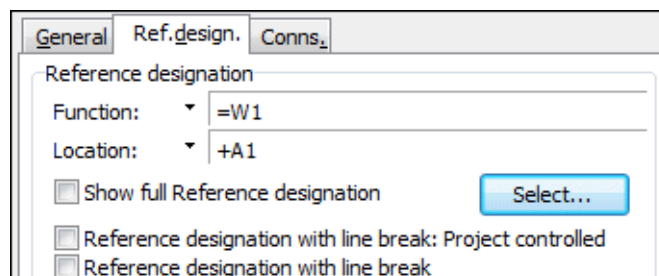
WHEN PLACING SYMBOLS, THE DIALOG BOX **REFERENCE DESIGNATIONS** ONLY DISPLAYS THE ASPECT TYPES, WHICH HAVE BEEN CREATED IN THE PROJECT.

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 340.

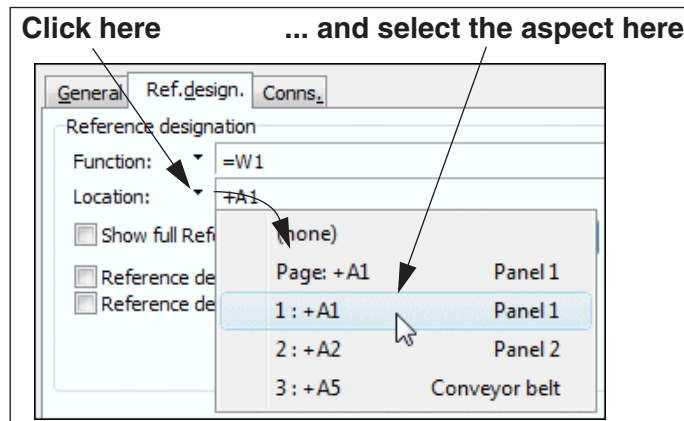
When you return to the **Component data** dialog box, you can see the reference designation on the tab **Ref. design.** in the lower part of the dialog box.



Displaying the page reference designation and the most recently applied reference designations

When you click on the down-arrow next to the **Location** field, the function reference designation for the page and the most recently applied location reference designations are displayed. You can then select a location 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**.

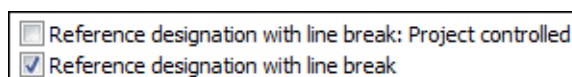
— See “Character for specifying full reference designation (exclude concatenation)” on page 337 also.

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.

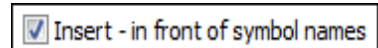
Editing reference designations using the Select name function

— See “Changing the symbol reference designations using Select name” on page 240.

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 372.

REFERENCE DESIGNATIONS FOR A FULL PAGE

It is also possible to specify reference designations for a full page:

- 1) Choose **Settings => Page data**.
- 2) In the dialog box **Page data**: Click on **Select** under **Reference designations**.
- 3) In the dialog box **Reference designations**: Click on the reference designations you wish to select for the page and click **OK**.
- 4) If symbols are already placed on the page, you are now asked whether to change their reference designations. — Read more about this in “Changing reference designations” on page 356.

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 354.

Click **OK**.

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.

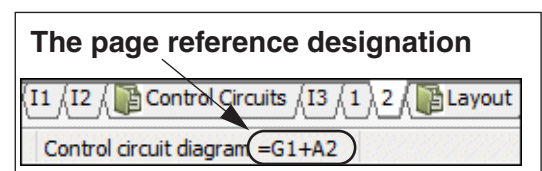
Because the symbols are conceived as belonging to the reference designation of the page, this reference designation is not displayed as a part of the symbol name on the page. Reference designations will only be displayed when they are different from the reference designation of the page.

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.

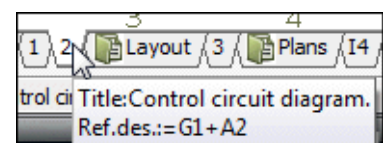
The page reference designation displayed in the Status line

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 363.

Please refer to “Creating Drawing Headers” on page 742 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**.

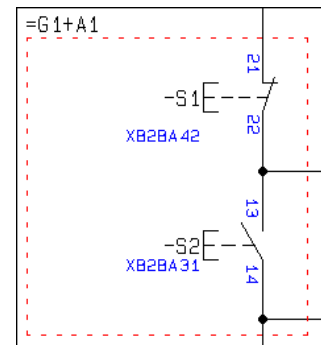
The datafields **Next used page** and **Previous used page** can also display the reference designation of the pages they refer to.

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.



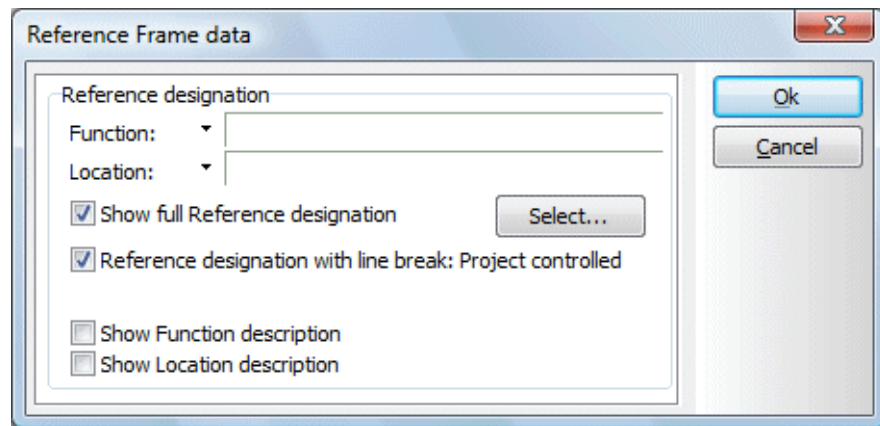
THE PROGRAM MEMORIZES THE LINES SETTINGS

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 73.

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 349, 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.

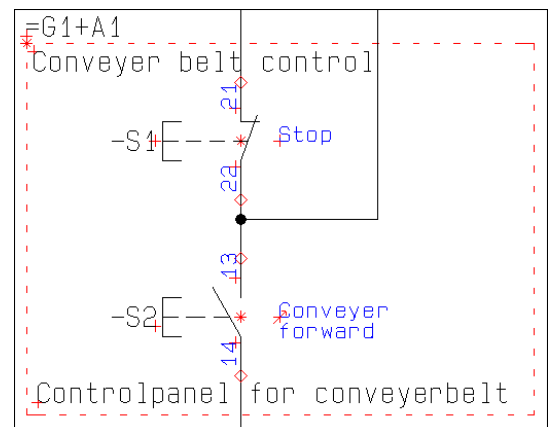
Both texts can be moved freely, and their text data can be edited, as for other texts in the program. — See

“Moving symbol texts on a Diagram page” on page 109 and “Specifying text properties / Appearance of text” on page 122.

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 351.



Click **OK**

When the dialog box is filled out, click **OK**.

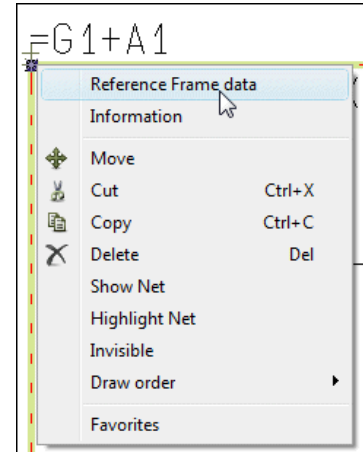
If symbols are already placed inside the area, that you specified for the reference frame, you are now asked whether to change the reference designations for these symbols. — Read more about this in “Changing reference designations” on page 356.

Editing reference designations for areas



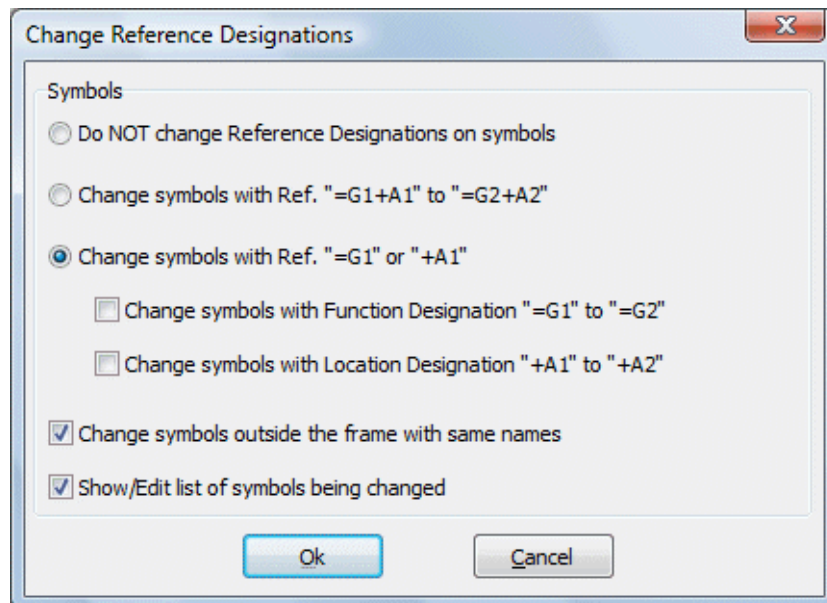
If you want to change the reference designation for the area, click the **Lines** button, right-click on the frame, and choose **Reference Frame data**.

You then enter the dialog box **Reference Frame data**, as described in “Reference designations for Areas on pages” on page 354. Here you type in the new reference designations for the area / reference frame.



Changing reference designations

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 362.

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 306.

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 358 —, 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 353.

— Please refer to “Change reference designations via chapter Divider pages” on page 361 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**.



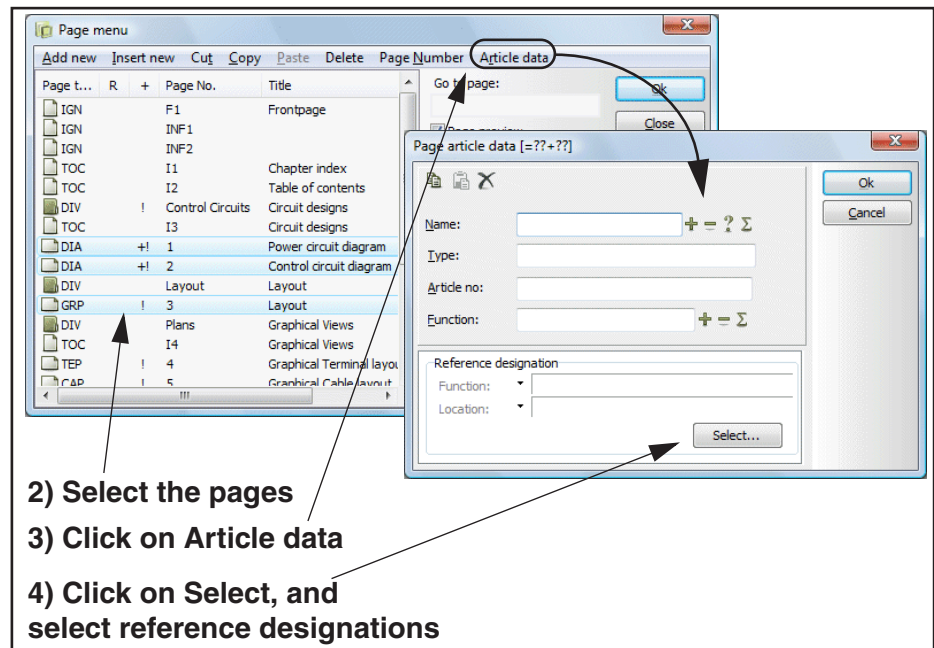
IF THE PAGES DO NOT HAVE IDENTICAL REFERENCE DESIGNATIONS

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.

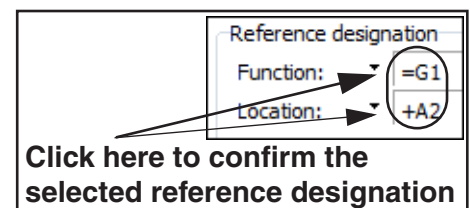
- 4) In the dialog box **Page article data**: Click on **Select** under **Reference designations**:



- 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

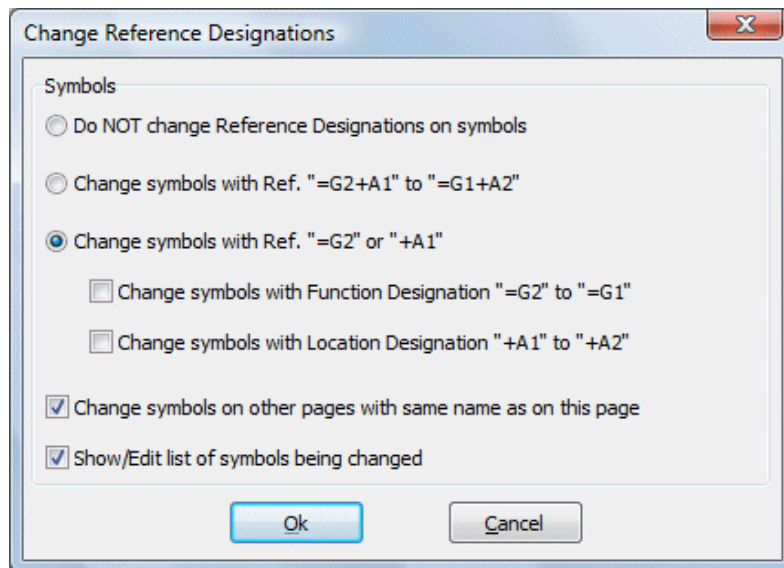
- 6a) If the selected pages did *not* have identical reference designations, do the following in the dialog box **Page article data**: Click on one — or both — of the reference designations, to confirm that you wish to select this reference designation. This causes **Function** and/or **Location** to 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 358). Click **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 361. 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 362.

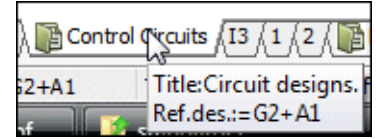
Click OK in the Page menu

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:

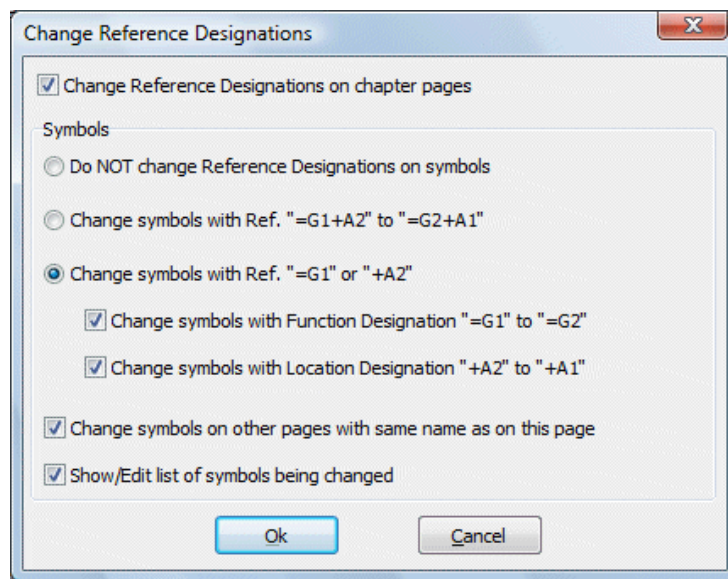
- 1) Click on the page tab for the divider page for the chapter (The chapter front page).
- 2) Select **Settings => Page data**, and click on **Select** below **Reference designations**.



- 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**:



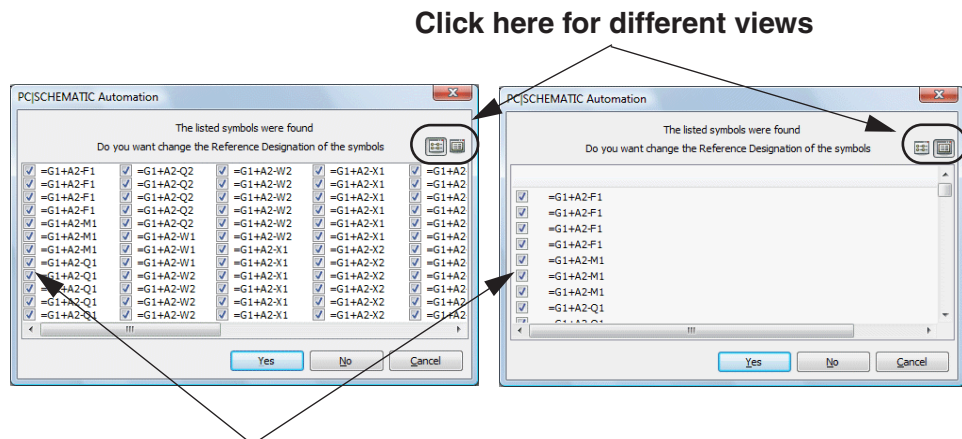
Here you are asked whether to change reference designations on the pages in the chapter having the same reference designations as the divider page, and which directions to apply for changing the symbols. You have the following options:

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 for different views

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**.

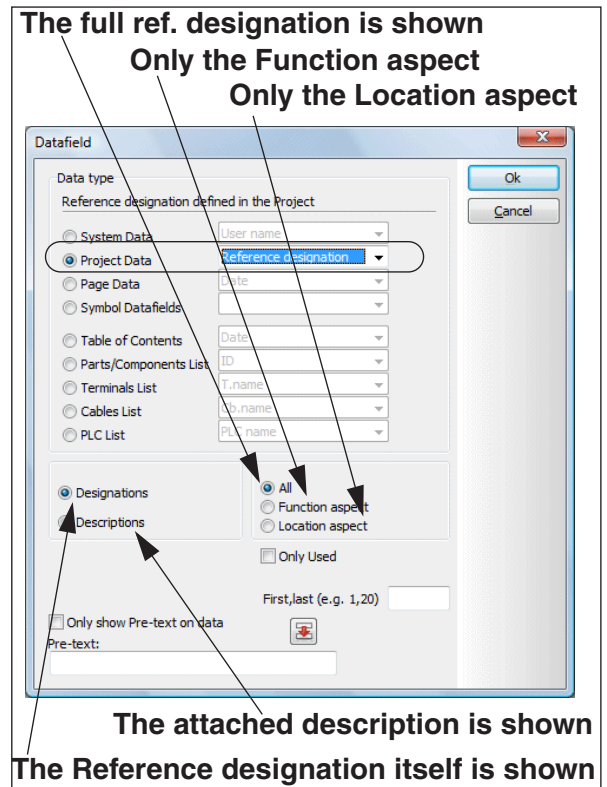
6) In the dialog box **Settings**: Click **OK**

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 549 for further details on criteria on lists.



References

This section describes the use of references in PCISCHMATIC Automation:

- Automatically updated references p. 367
- Contact mirrors and Reference crosses p. 374
- Reference system settings p. 379



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. 367
— AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK.....	P. 367
— REFERENCES ON SYMBOLS.....	P. 369
— APPEARANCE OF REFERENCES	P. 372
— REFERENCE DESIGNATIONS ON SYMBOL AND SIGNAL REFERENCES.....	P. 372
— REFERENCES FOR PLCs	P. 373

— Please see to “Contact mirrors and Reference crosses” on page 374.

— Please also see “Navigation between symbols for the same component” on page 170.

INSERTING A REFERENCE SYSTEM

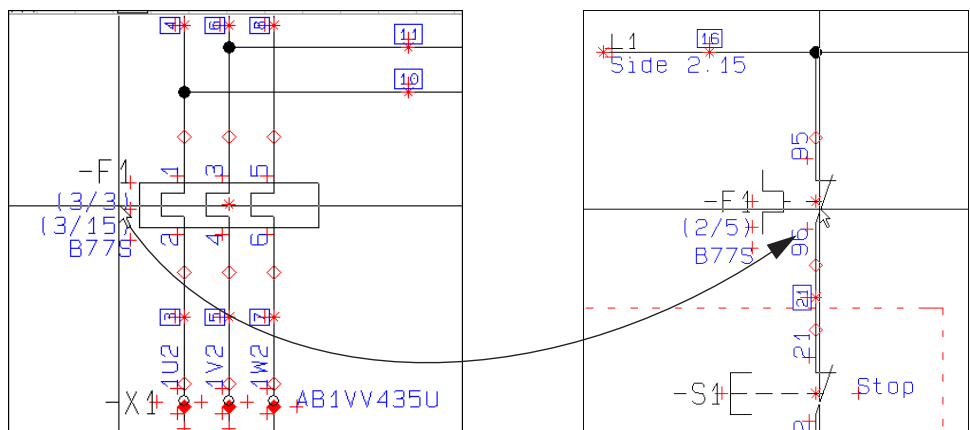
In order to let the program create appropriate references, you must first create/ adapt the reference system for the diagram pages, so that it meets your demands. Read more about this in “Setting up the current paths” on page 379.



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.

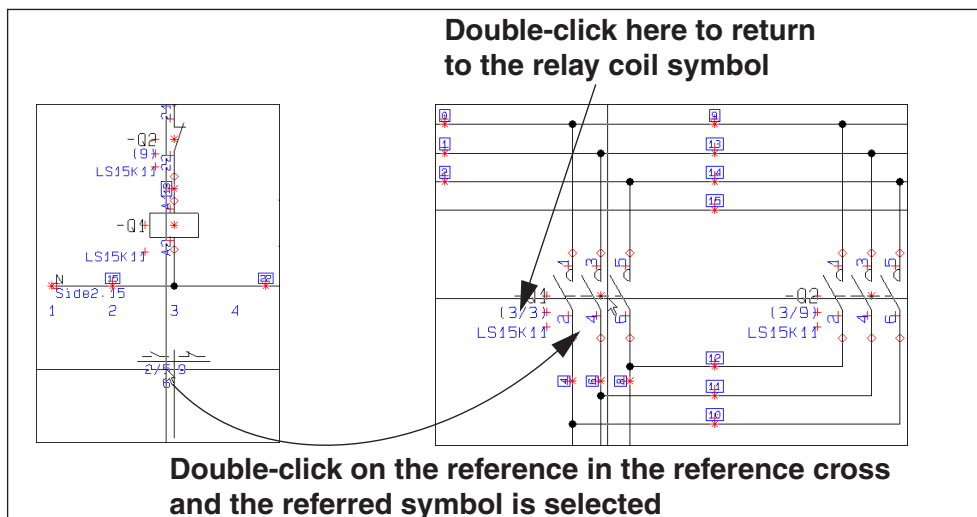


Double-click on the reference, and the referred symbol is selected

- 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.

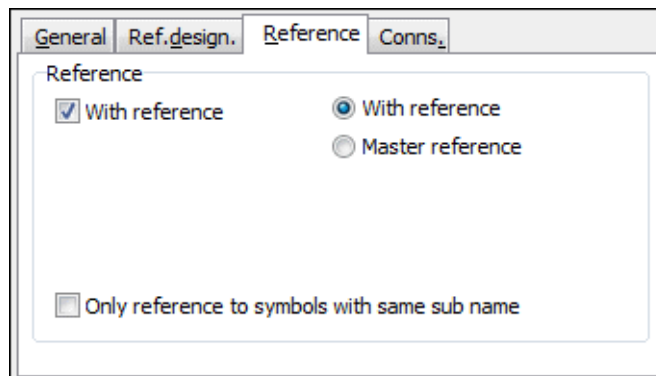
How the reference appears for a symbol, primarily depends on the **symbol type** of the symbol. However, some symbols are created without symbol type, and therefore has no references.

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:



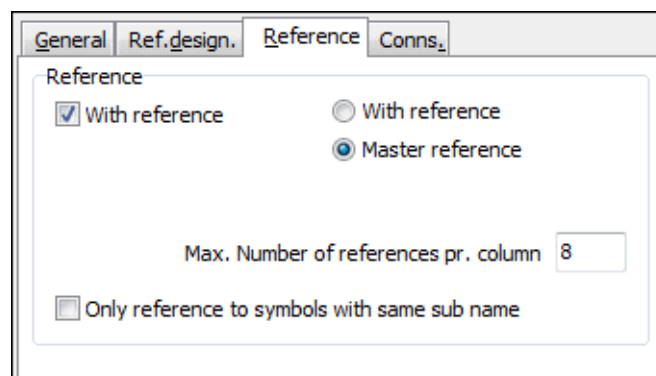
- 1) Click on the **Symbols** button, right-click on the symbol, and choose **Component data** — or double-click on the symbol.
- 2) In the dialog box **Component data**: Click on the **Reference** tab, choose **With reference** and then either **Master reference** or **With reference**, and click **OK**.



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 sub names 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.

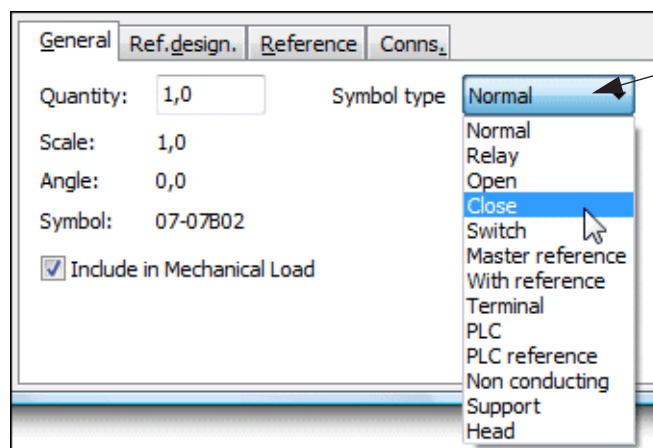
Symbol type and References

The symbol type is specified in the symbol definition — see “Creating Symbols” on page 696; 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**.



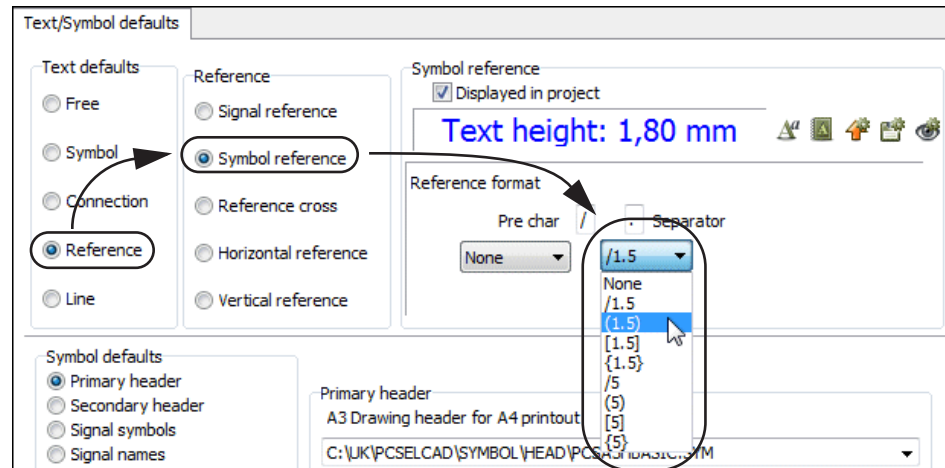
ALL SYMBOL TYPES CAN BE CREATED WITH ACTIVE WITH REFERENCE

ALL SYMBOL TYPES CAN BE CREATED WITH **WITH REFERENCE** ACTIVE VIA THEIR SYMBOL DEFINITION. — READ MORE ABOUT THIS IN “**CREATING SYMBOLS**” ON PAGE 696, “SYMBOL SETTINGS: REFERENCE” ON PAGE 709 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**, **Pre char** 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 122.

- 7) Click **OK**.

- 8) Choose **Functions => Update references** — and the changes are displayed in the project.

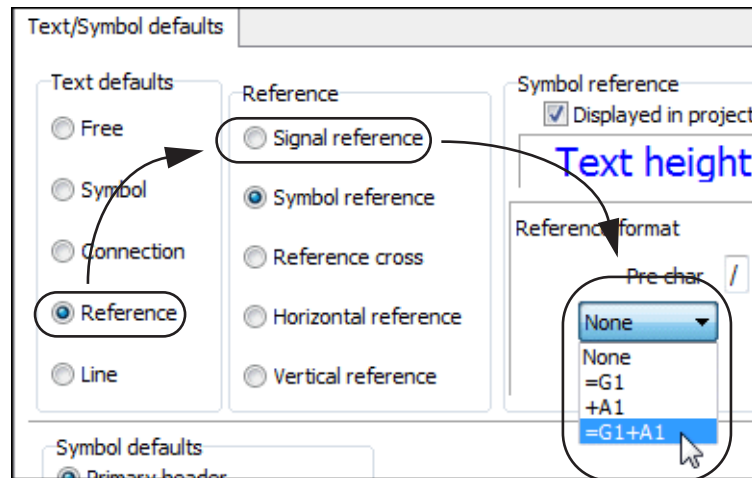
When you wish to change the appearance for references for signal symbols, just choose **Signal reference** in step 3 above.

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 427.



CONTACT MIRRORS AND REFERENCE CROSSES

When displaying references for symbols with the type **Relay** — see “References on symbols” on page 369 — 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. 375
— ON REFERENCE CROSSES	P. 376
— CHOOSING BETWEEN CONTACT MIRRORS AND REFERENCE CROSSES	P. 377
— CONTACT MIRRORS ARE CREATED AUTOMATICALLY BY THE PROGRAM	P. 377

- Please read “Setting up the current paths” on page 379 to see how to set the reference system on the project pages.
- Please also read “Automatically updated references” on page 367 for general information on references in **Automation**.



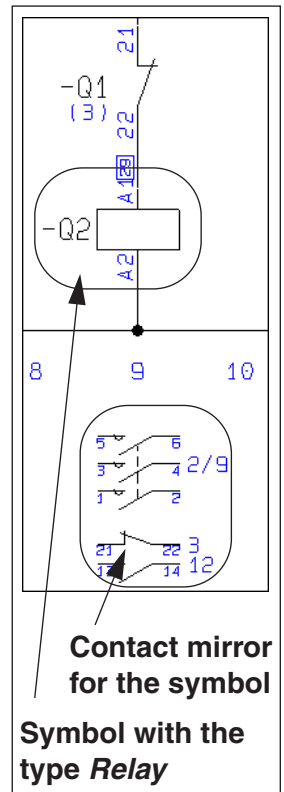
CONTACT MIRRORS GIVES QUICK ACCESS TO AVAILABLE SYMBOLS
 CONTACT MIRRORS ALSO GIVES YOU THE OPTION TO GET THE AVAILABLE SYMBOLS FOR A COMPONENT IN THE CROSS HAIRS RIGHT AWAY.

ON CONTACT MIRRORS

If you have set the program to use contact mirrors — see “Choosing between contact mirrors and reference crosses” on page 377 — 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 relay 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.

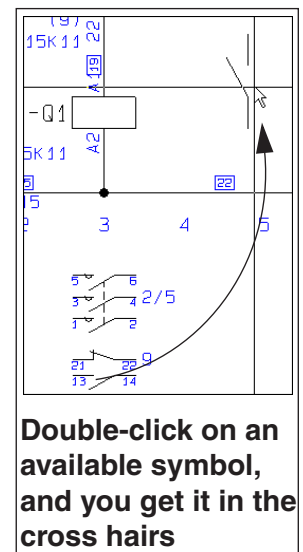
Direct jump to the other symbols for the component

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 367.

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 367.

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 375.

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.

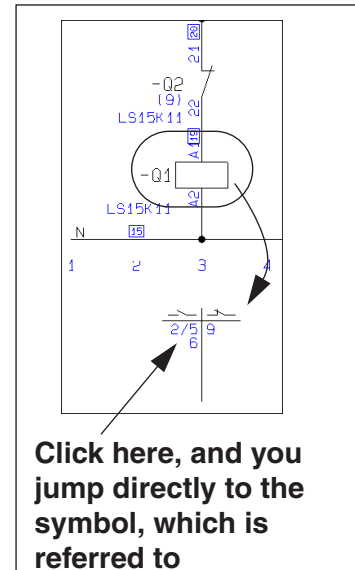
— See “Sub names on symbols” på side 165, especially “Functionally understood as different components” på side 165.

Move reference cross

To move a reference cross, do this:

- 1) Right-click on the relay coil and choose **Move reference symbol**
- 2) Click to point out the new position of the reference cross

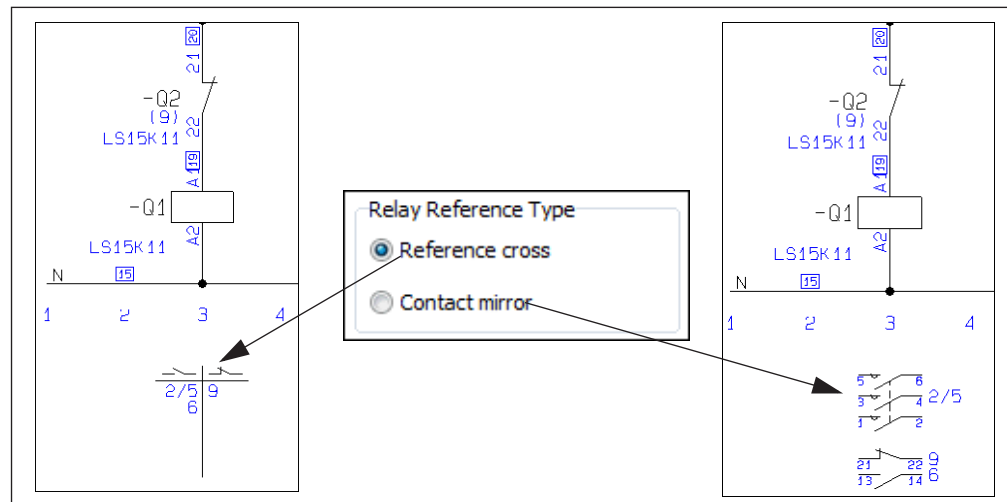
The reference cross is now moved to the new position.



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 608).

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.

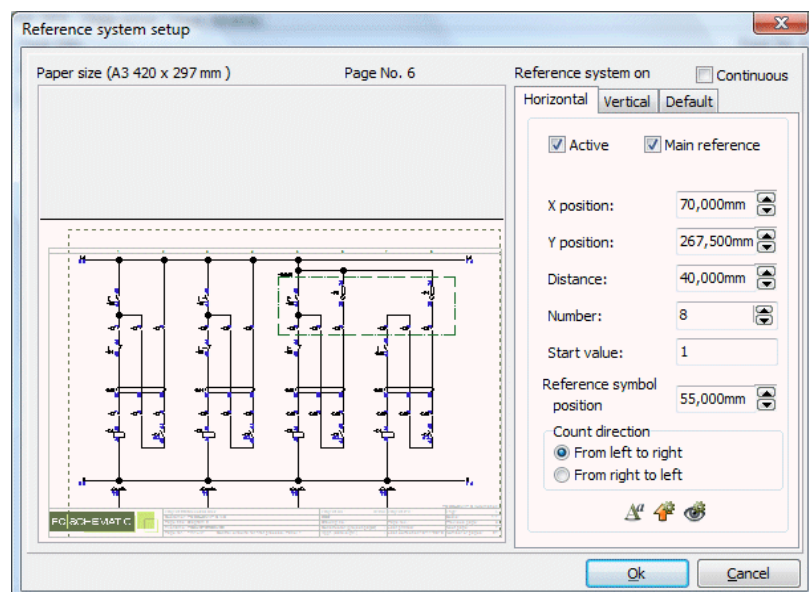
You can determine to let the number of the current path, and/or page a symbol is located on, be part of the name of the symbol. This is called *current path naming*.

THIS CHAPTER DESCRIBES	
— SETTING UP THE CURRENT PATHS.....	P. 379
— THE HORIZONTAL REFERENCE SYSTEM.....	P. 380
— THE VERTICAL REFERENCE SYSTEM	P. 380
— COPYING THE REFERENCE SETTINGS.....	P. 381
— CHANGING THE REFERENCE CROSSES.....	P. 381
— NAMING SYMBOLS ACCORDING TO CURRENT PATH NUMBERS	P. 382
— SYMBOL NAMES WHEN MOVING OR COPYING	P. 384
— CURRENT PATH NUMBERS AND AUTOMATIC PROJECT GENERATION.....	P. 385
— NAMING DUE TO CHANGES IN THE PAGE MENU	P. 385

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 706). 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**.

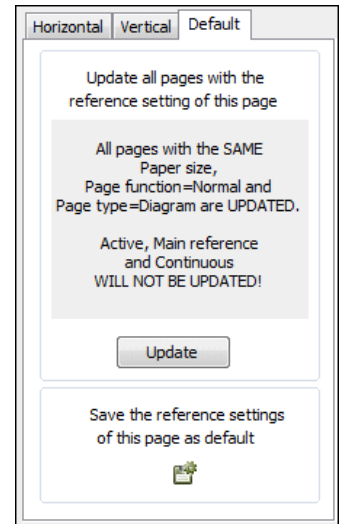
Please note, that clicking on the Save as standard button will not effect any pages in the project.

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 491.



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 696.

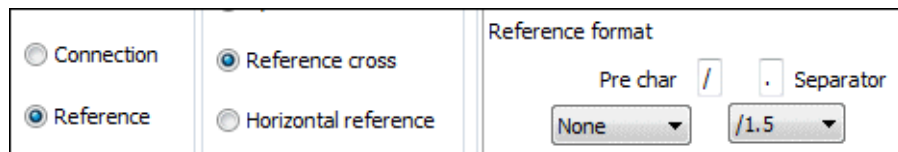
You choose which reference cross to use in the project by first choosing **Settings => Text/Symbol defaults**, then click on **Others**, on **Reference cross**, and on **Add**. You then enter the **Symbol menu**, where you select the symbol for the reference cross you wish to use.

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 696.



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:

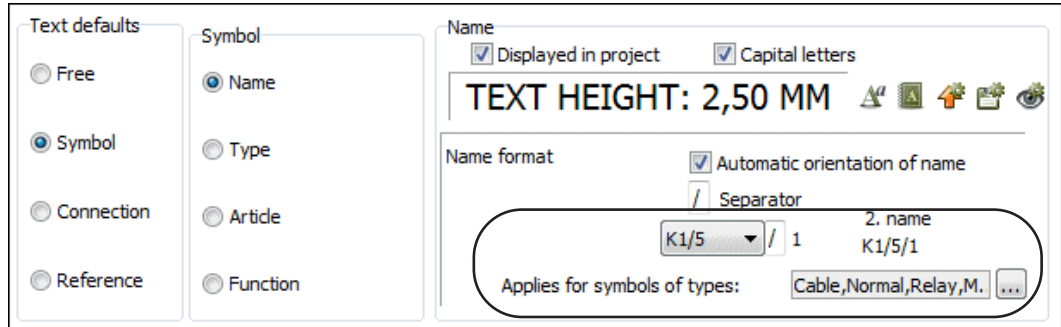


Then choose **Functions => Update references** and the entire project is updated with the new references.

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.

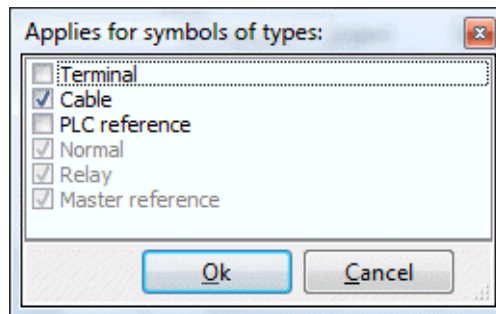
To arrange this, you choose **Settings => Text/Symbol standards**, and click on **Symbols**, and then on **Name**.



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. When you choose this format, you also get the option of choosing a different separator than **/**.

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. Under **2. name** you can see the result of the chosen name format.

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 *successfully*.

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-

tions 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 127.

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 385.

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 495).

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. 389
- Manual wire numbering p. 398
- Wire number files p. 401

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. 389
— DESELECTING WIRE NUMBERING FOR INDIVIDUAL LINES	P. 390
— LOCKING THE AUTOMATIC WIRE NUMBERING	P. 390
— FUNCTIONS	P. 391
— POTENTIAL NUMBERING.....	P. 391
— WIRE NUMBERING.....	P. 392
— FORMAT	P. 395
— NUMBER SYMBOL	P. 395
— EXCLUDE SIGNALS	P. 397

— Please refer to “Manual placing of wire numbers” on page 398 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 398.



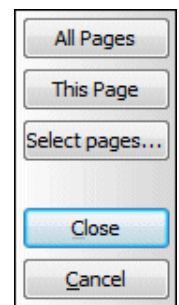
No UNDO FOR AUTOMATIC WIRE NUMBERING

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.

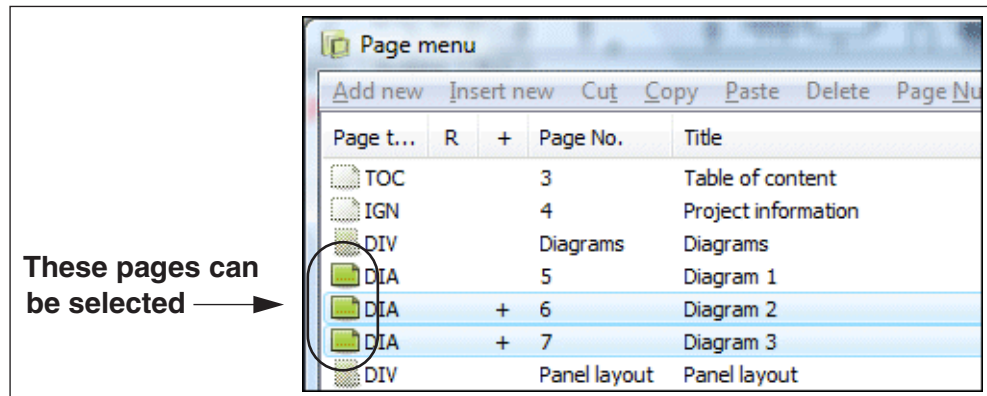
When the settings are appropriate, you click either **All Pages, This page** or **Select pages** to carry out the automatic wire numbering, following the directions you have set up.

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 306.

When you use the **Show net** function, you get access to the same function, as described in "Show net" on page 202.

DESELECTING WIRE NUMBERING FOR INDIVIDUAL LINES



- 1) Choose **Lines** and click the line.
- 2) Choose **Edit => Without wire number** — or click the **Without wire number** button.

You can also right-click the line and choose **Without wire number**.

— In order to make it possible to assign wire numbers to the line again, just right-click the line and choose **Without wire number**.

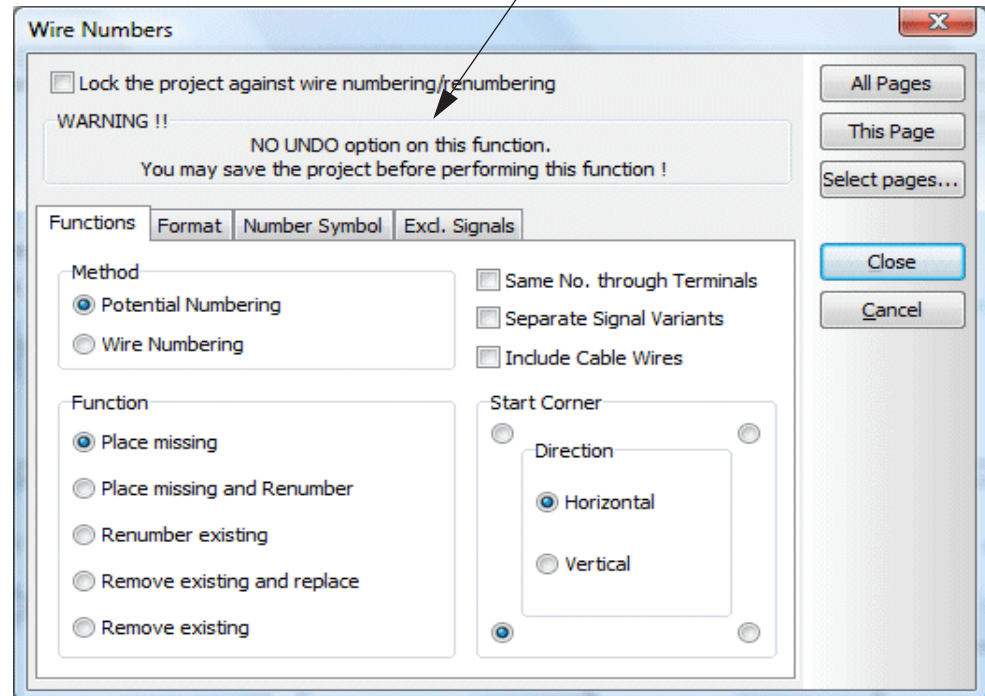
LOCKING THE AUTOMATIC WIRE NUMBERING

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 203) — 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.

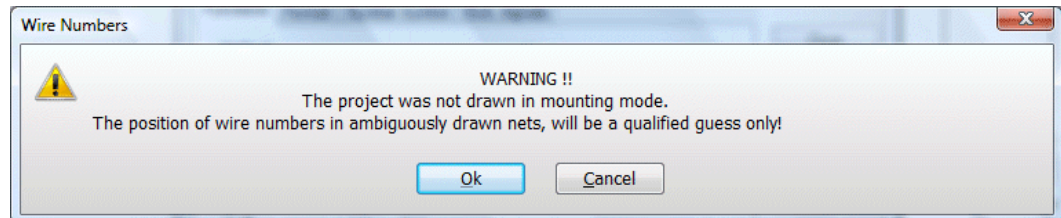


THE WIRE NUMBERING IS A QUALIFIED GUESS

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!

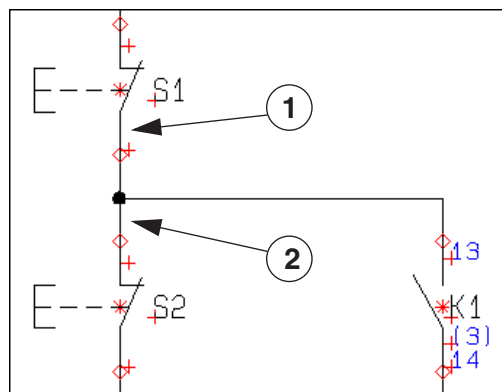
THE PROGRAM WILL PLACE THE CORRECT AMOUNT OF WIRE NUMBERS WITHIN THE INDIVIDUAL POTENTIALS, AND YOU WILL THEREFORE RECEIVE THE CORRECT AMOUNT OF WIRE NUMBERS IN YOUR WIRE NUMBERS FILE. — BUT THERE IS NO GUARANTEE THAT THE NUMBERS WILL BE LOCATED ON THE CORRECT WIRES, IF YOU DO NOT DRAW IN MOUNTING MODE.

Therefore, if you want to perform automatic wire numbering, you receive this warning:

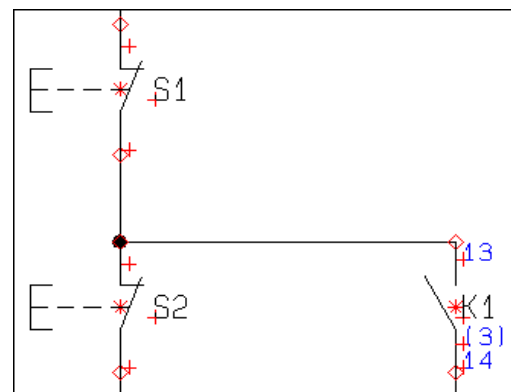


Unambiguous drawing

To make it perfectly clear how the wires should be installed, you can make the drawing unambiguous:



Ambiguous



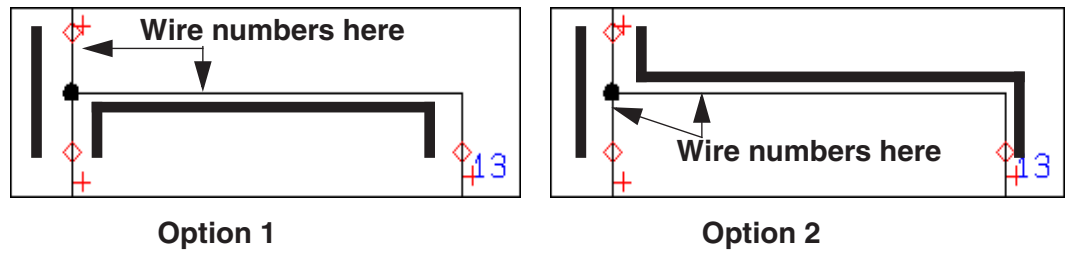
Unambiguous

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:

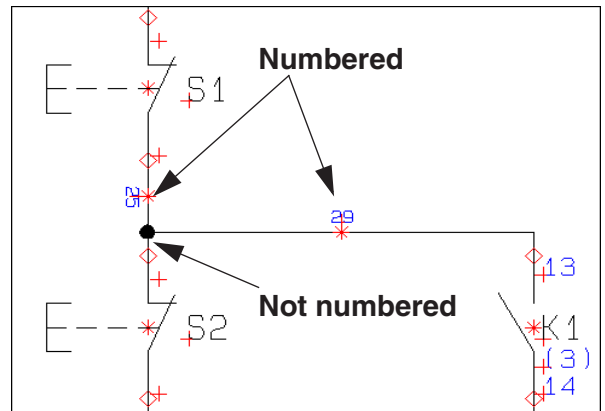


In this case, the program must make a qualified guess.

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 polylines is the line starting in the connection point **13**, and ending in the connection dot.

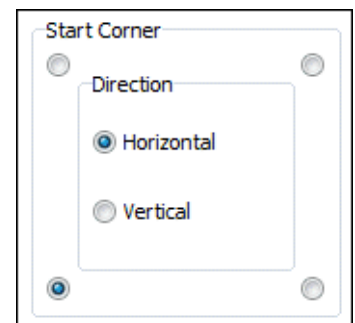


In an electrical potential with for instance three connected symbols, two lines will be included. The program will then make a guess, that the shortest polyline represents both wires. Please bear this in mind when you draw diagrams.

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

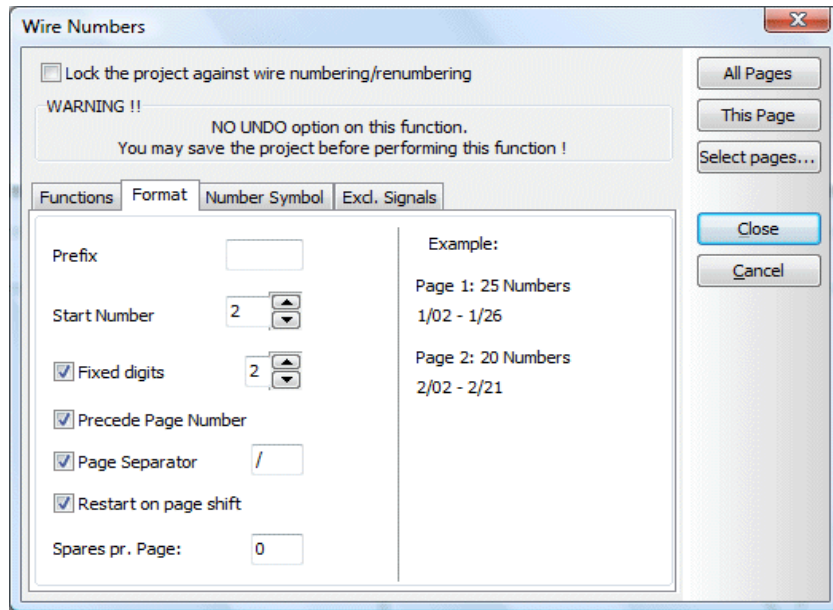
In the lower-left corner of the tab, you determine how to assign wire numbers, and where to place the wire numbers:

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.

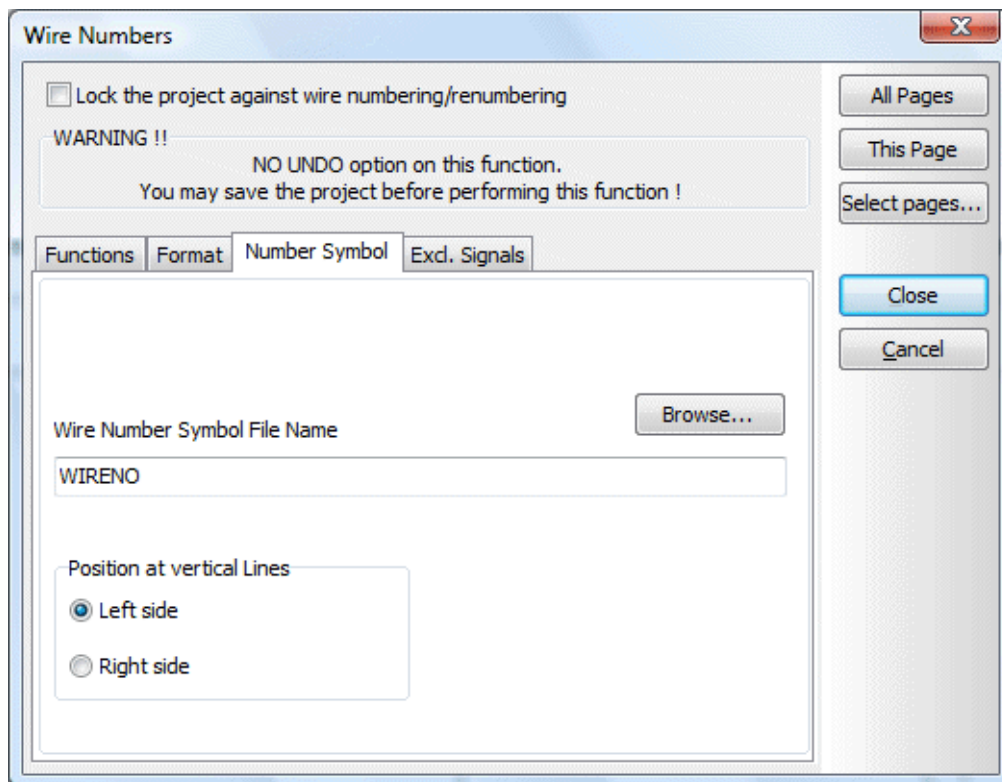
FORMAT	ACTION
PREFIX	HERE YOU CAN PLACE A PREFIX , WHICH IS PLACED IN THE BEGINNING OF THE WIRE NUMBERS.
START NUMBER	HERE YOU CAN SPECIFY A START NUMBER FOR THE WIRE NUMBERS.
FIXED DIGITS	HERE YOU STATE WHETHER THE WIRE NUMBERS SHALL CONTAIN A FIXED NUMBER OF DIGITS — FROM TWO TO NINE.
PRECEDE PAGE 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

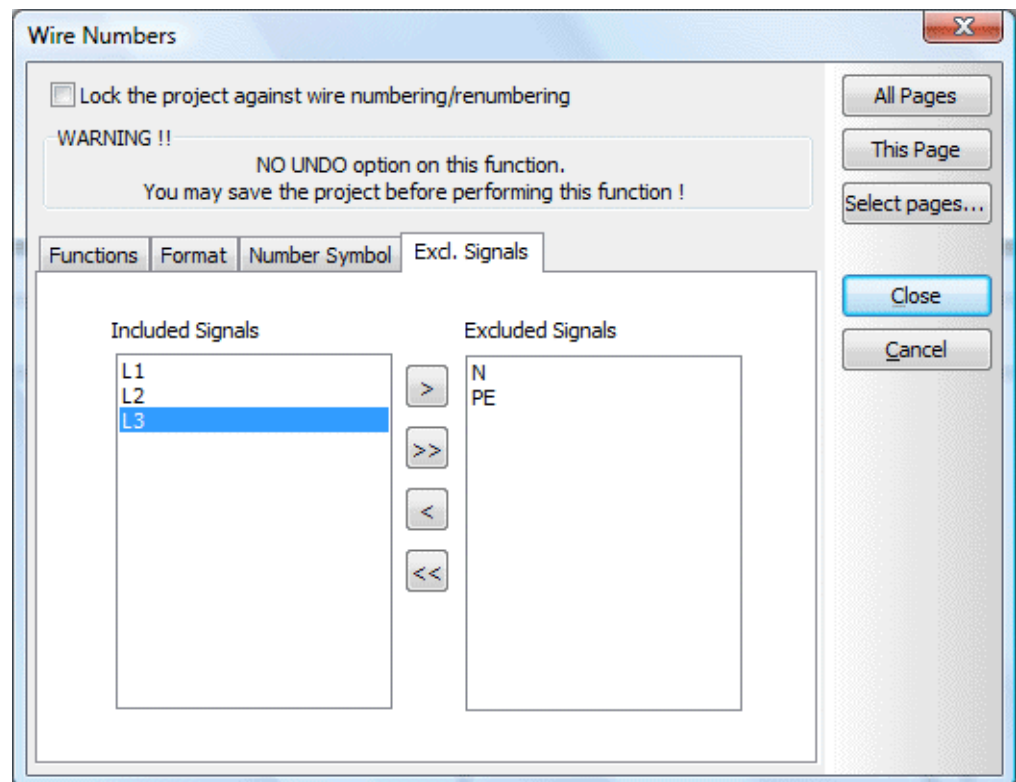
In the lower-left corner, you specify whether the wire number symbols are placed on the **Left side** or on the **Right side** of vertical lines.

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 696.

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. 398
— PLACING A WIRE NUMBER SYMBOL.....	P. 399
— AUTOMATICALLY INCREMENTING OF WIRE NUMBER NAMES	P. 399

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 389.

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 178.

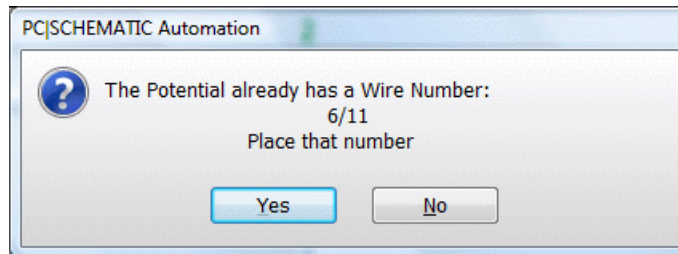
This speeds up the manual placing of the symbol.

Please refer to “Fetching symbols” on page 139 for further details about fetching symbols.

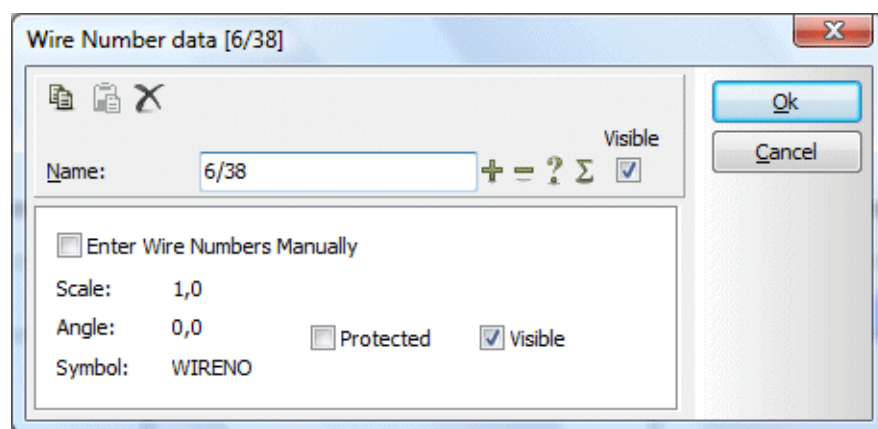
PLACING A WIRE NUMBER SYMBOL

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 156.

WIRE NUMBER FILES

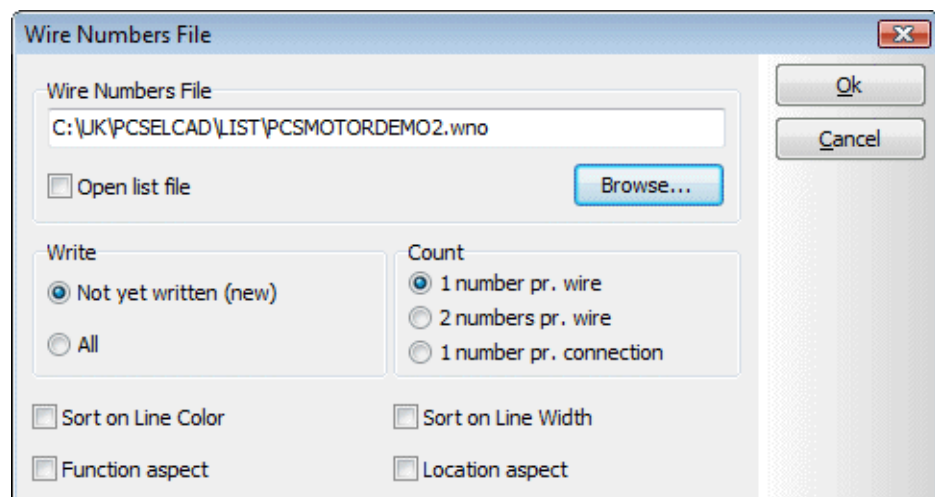
In **Automation** you can create wire number files for a wire numbering tool. This chapter describes how to create wire number files.

THIS CHAPTER DESCRIBES	
— CREATING A WIRE NUMBER FILE.....	P. 401
— CREATING A CONNECTIONS LIST FILE FOR WIRE NUMBERING.....	P. 402

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**.

Wire numbers to file according to Function aspect and Location aspect

If you use reference designations on wire numbers, you can also specify whether to include the *Function aspect* and/or the *Location aspect* in the file.

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 wire number 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

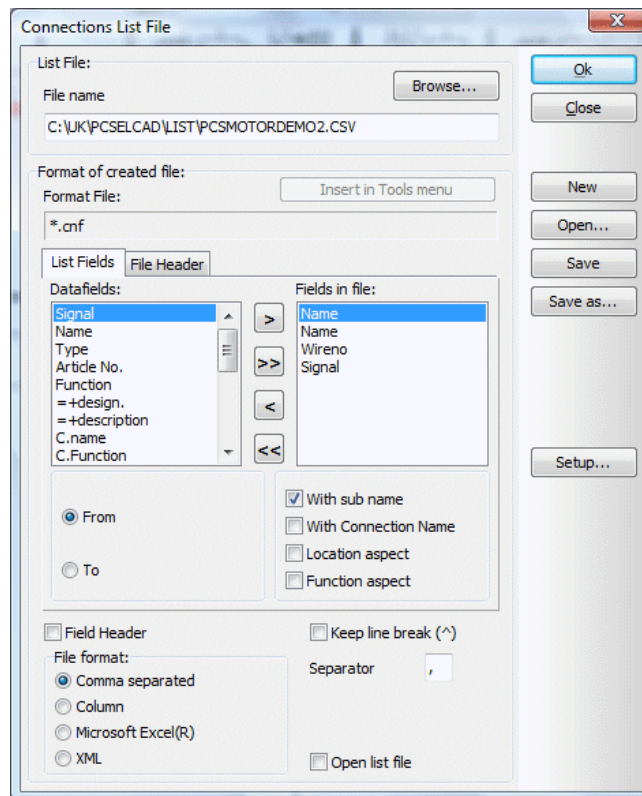
In **Automation** you can create a connections list file, which contains component names with connections for all electrically connected components. Hereby a complete connections list is displayed, where name and connection is displayed for each individual connection.

This list can succeedingly 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

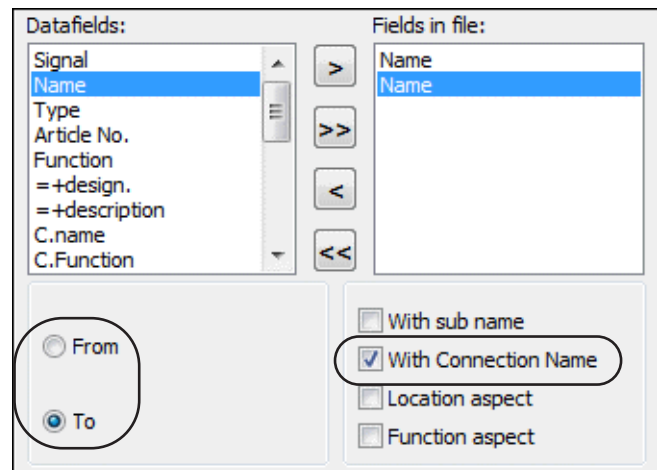
Choose **Lists => Connectionlist for file**, and you enter this dialog box:



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.

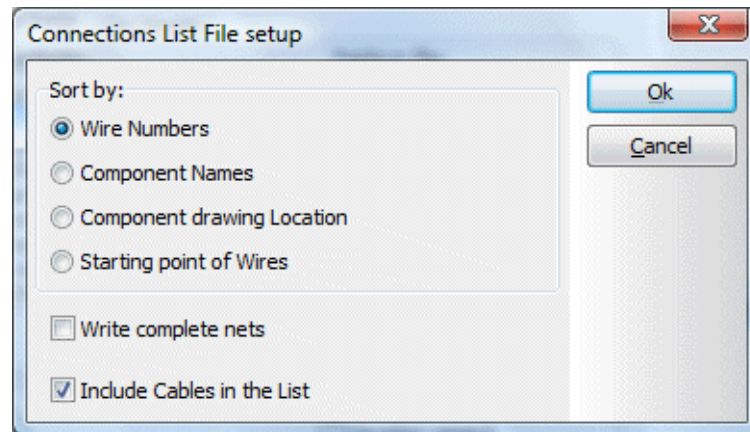
When you then select **With Connection Name** for both of these datafields, and set the one to **From** and the other to **To**.

— Read more about the options in this dialog box in “Specifying the contents of the file” on page 654, and see “The buttons in the right-hand side of the dialog box” on page 653 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

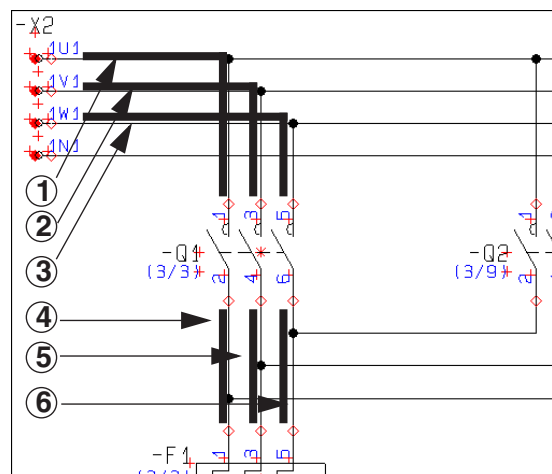
The file is sorted according to the pages in the project. First comes everything from page **1**, then everything from page **2** etc.

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 649.



Single line diagrams

This section describes functions for single line diagrams:

- Single line diagrams p. 407
- Generating single line diagrams automatically p. 409

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. 407
— CONNECTION NAMES IN SINGLE LINE DIAGRAMS	P. 408
— SYMBOL NAMES IN SINGLE LINE DIAGRAMS	P. 408
— SYMBOLS FOR SINGLE LINE DIAGRAMS IN PICKMENU	P. 408
— SYMBOLS WITH STATES FOR SINGLE LINE DIAGRAMS	P. 408

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**.

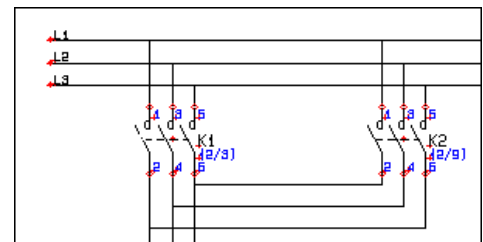


Figure 1

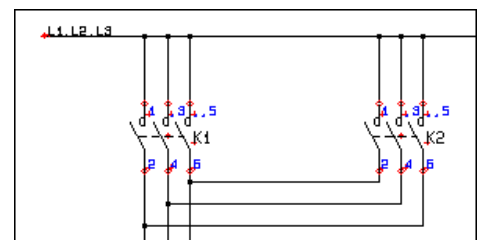


Figure 2

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.

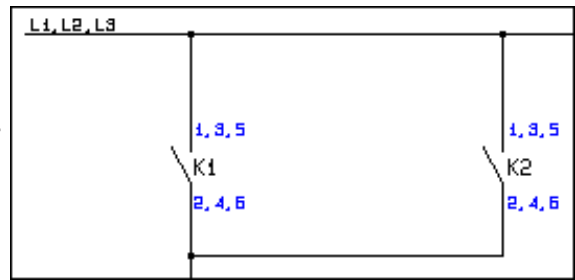
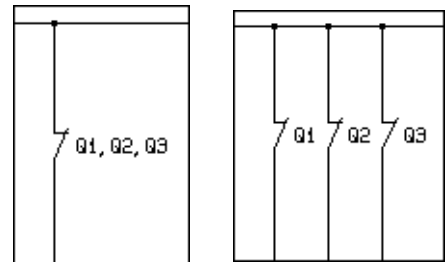


Figure 3

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 594.

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 167.
- See how to create symbols with states in “Creating symbols with states” on page 712.

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. 409
— PREPARING FOR AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS	P. 410
— STARTING THE MODULE SINGLELINE DIAGRAM.....	P. 410
— SETTINGS FOR THE MODULE SINGLELINE DIAGRAM	P. 412

— Please also see “Single line diagrams” on page 407.

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:

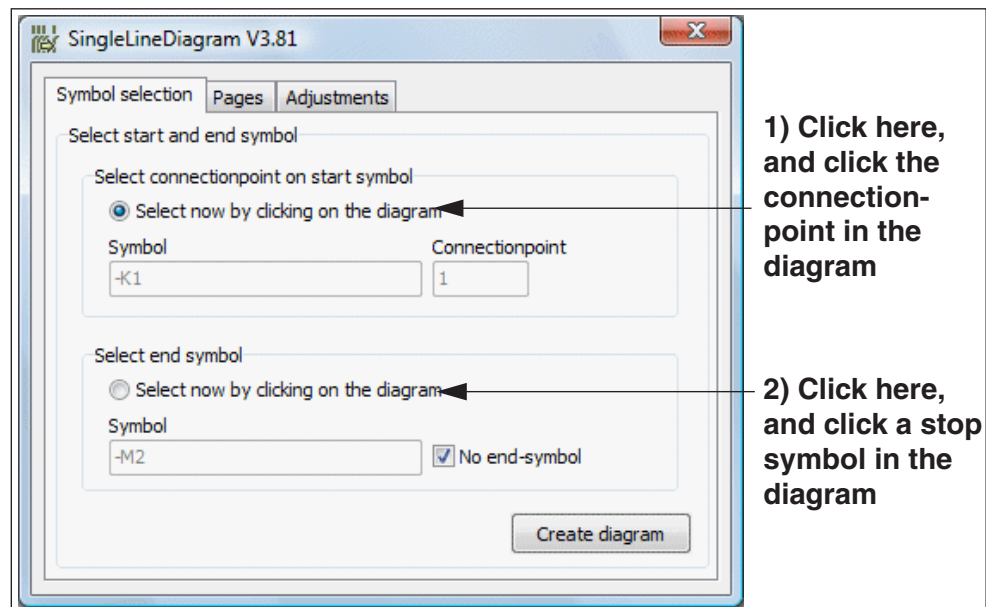
- 1) Create single line symbols for the applied components in the diagram.
- 2) In the applied database: Add a field to the database, which makes it possible to specify which single line symbol to use for each individual component — for instance the field *SINGLELINESYMBOL*.
- 3) In the applied database: 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 413.

GENERATING A SINGLE LINE DIAGRAM

To generate a single line diagram using the **Singleline Diagram** module, do this:

- 1) Choose **Tools => Singleline Diagram**:



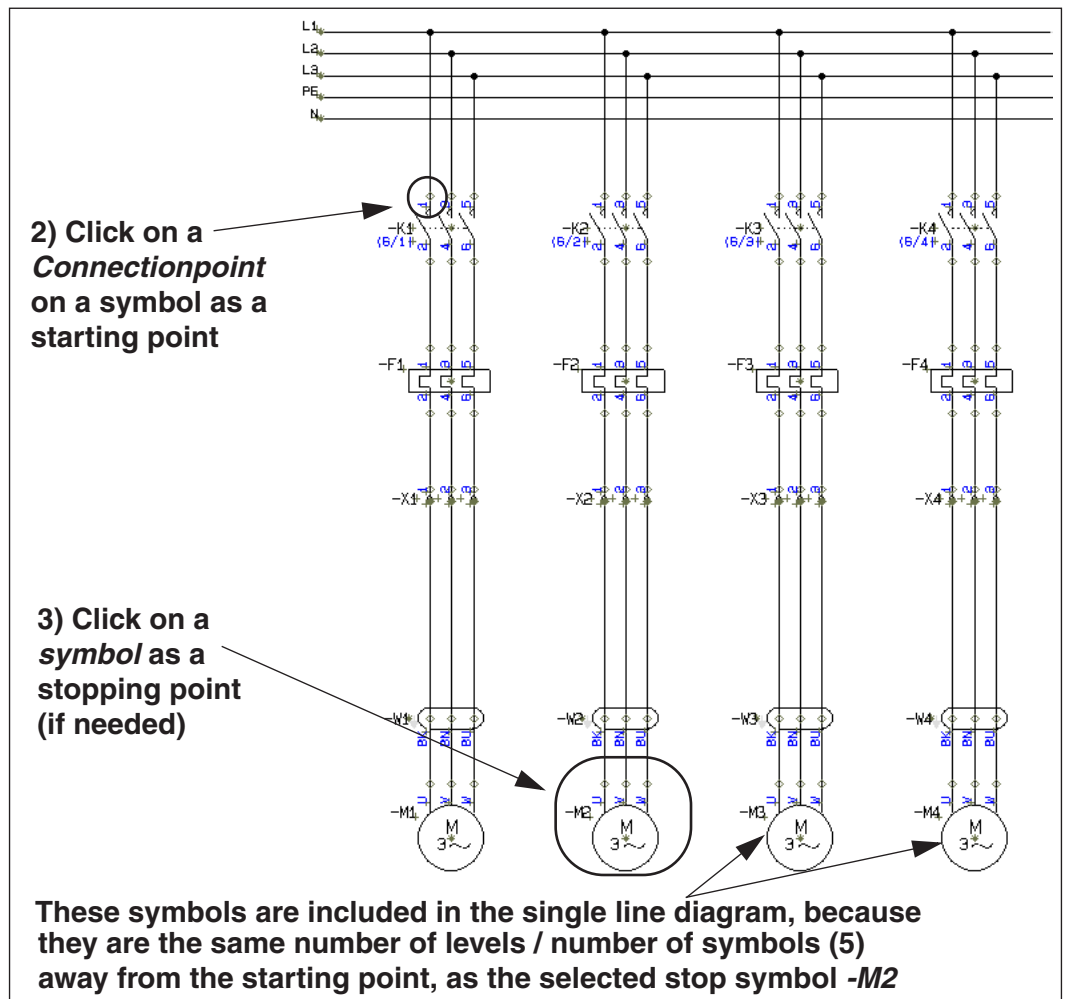
- 2) Click on the symbol *connectionpoint* in the diagram, from which you want the single line diagram to start, as displayed in the figure "Point out start and end point for the single line diagram" below.
 - If you are creating the diagram based on a file from the **Panel Builder**, the start symbol 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

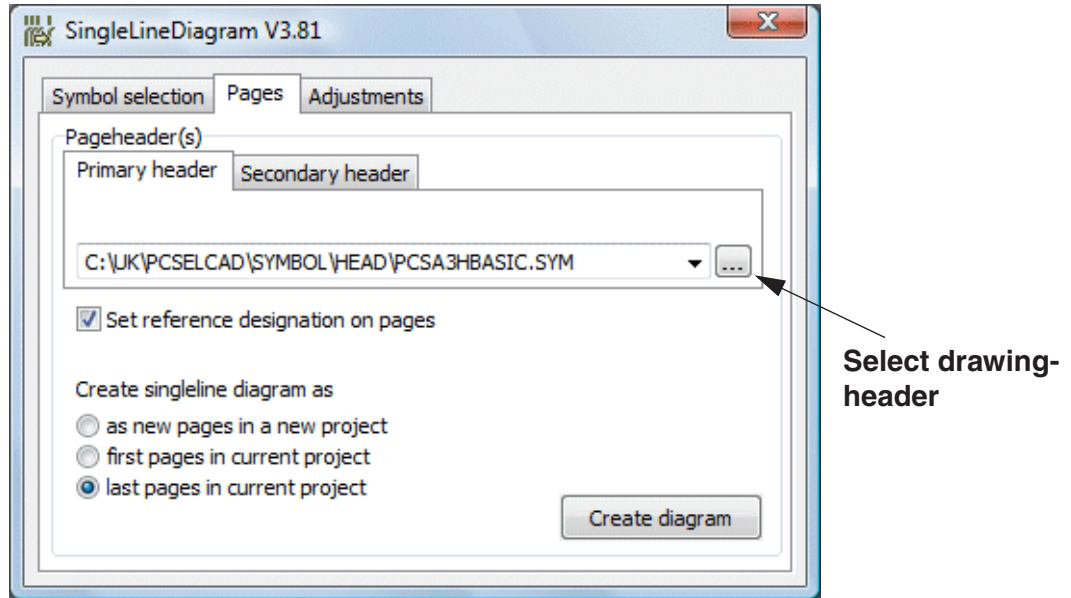


SETTINGS FOR THE MODULE SINGLELINE DIAGRAM

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.

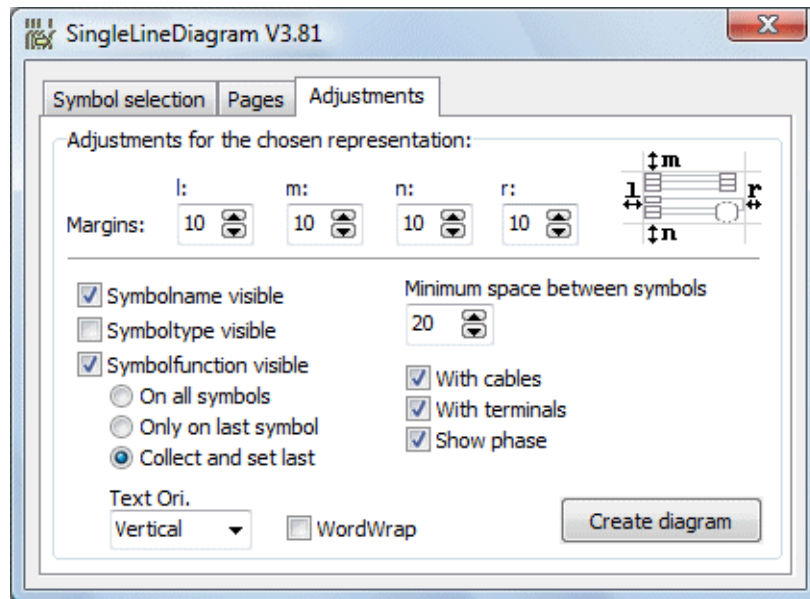


You can choose:

- 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.



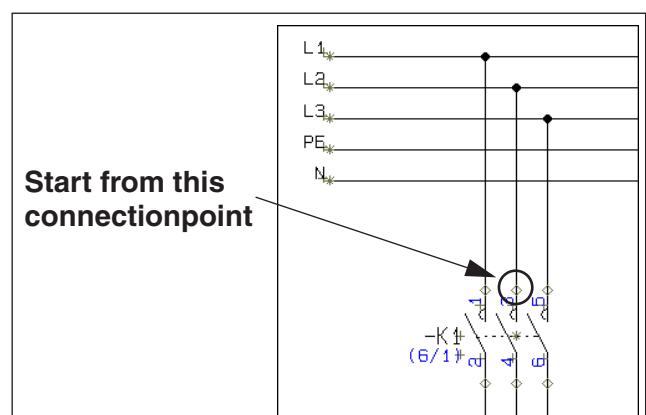
Here you specify:

- 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 **Singleline 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 410.

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 484.
- 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 410.



PLC functions

This section describes:

- Survey of PLC functions p. 417
- Working with PLCs p. 418
- Importing PLC I/O data p. 432
- Addressing PLC I/O data via Excel p. 435
- Connection symbol for signal buses p. 437



Read survey article

Read the survey article on working with PLCs at www.pcschematic.com/articles.

At www.pcschematic.com/manuals you can download the mini manual “PLC tutorial”, which guides you through creating a PLC project.



SURVEY OF PLC FUNCTIONS

THIS CHAPTER DESCRIBES

THIS CHAPTER GIVES A SURVEY OF WHICH OPTIONS YOU HAVE WHEN WORKING WITH PLCs IN **Automation**.

When working with PLCs in **Automation** you have the following options:

- 1) Working with PLCs: See “Working with PLCs” on page 418.
- 2) Working with PLCs and sub drawings: Read the mini manual "PLC tutorial".
- 3) Exporting/Importing PLC I/O data: See “PLC I/O addressing via Excel” on page 435.
- 4) Controlling PLC related texts for other symbols: See “Text links” on page 132, especially “Text links and importing PLC I/O lists” on page 136.
- 5) Editing PLC I/O data via the Object Lister: See “Typing in PLC I/O data via the Object Lister” on page 426 and “Editing PLC I/O data via the Object Lister” on page 313.
- 6) PLC I/O addressing via Excel: “PLC I/O addressing via Excel” on page 435.
- 7) Addressing PLC symbols via the database: See “Addressing PLC symbols via the database” on page 601.
- 8) Applying alternative PLC symbols: See “Alternative symbols for PLCs” on page 595.
- 9) Connection symbol for signal buses: See “Connection symbol for signal buses” on page 437.



READ THE MINI MANUAL "PLC TUTORIAL"

TO GET AN INTRODUCTION TO WORKING WITH PLCs, IT IS RECOMMENDED TO READ THE "PLC TUTORIAL". HERE YOU FIND A DESCRIPTION OF THE OPTIONS YOU HAVE IN THE PLC MENU.

THE TUTORIAL CAN BE DOWNLOADED AT www.pcschematic.com/manuals.



WORKING WITH PLCs

THIS CHAPTER DESCRIBES	
— ON PLC REFERENCE SYMBOLS.....	P. 418
— MORE THAN ONE PLC REFERENCE SYMBOL FOR THE SAME PLC COMPONENTP.	419
— ASSIGNING ADDRESSES TO PLCs VIA THE DATABASE	P. 419
— WORKFLOW FOR PLCs VIA THE DATABASE.....	P. 420
— RE-ADDRESSING PLCs.....	P. 423
— PLC DATA ON THE CONNECTION POINTS OF THE I/O SYMBOLS.....	P. 425
— TYPING IN PLC I/O DATA VIA THE OBJECT LISTER.....	P. 426
— JUMPING BETWEEN PLC REFERENCE SYMBOLS AND PLC I/O SYMBOLS	P. 427
— WORKFLOW FOR PLCs NOT IN THE DATABASE.....	P. 428
— SYNCHRONIZING PLC DATA WHEN COPYING BETWEEN PROJECTS	P. 430
— UPDATING PLC PROJECTS FROM PREVIOUS VERSIONS	P. 430

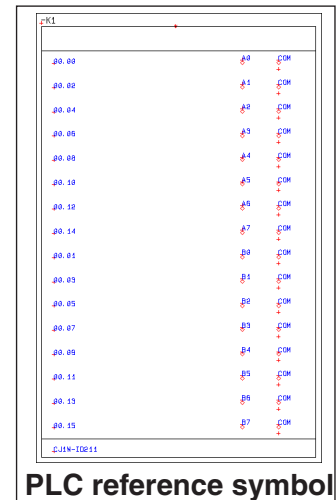
— Please also refer to “PLC I/O addressing via Excel” on page 435, in particular “Export of PLC I/O data to Excel” on page 435.

ON PLC REFERENCE SYMBOLS

When working with PLCs in **Automation**, you should — as your first action — place a PLC reference symbol, which carries the data for the PLC, and functions as a kind of control centre for the PLC.

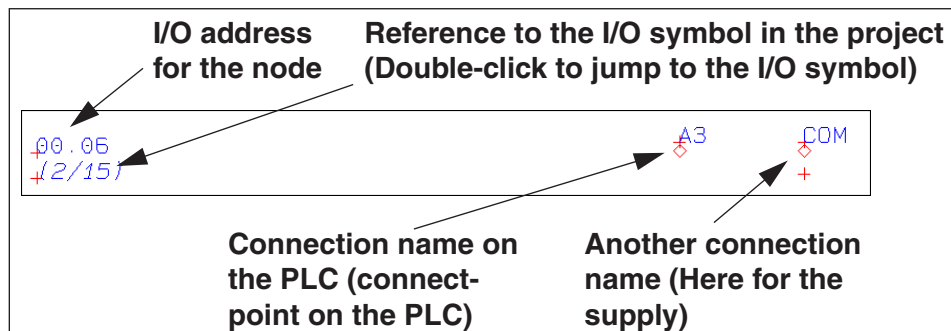
When you have placed the PLC reference symbol, you will then typically load the I/O list for the PLC, or type in I/O data manually in the program. Not until then will you start placing the PLC I/O symbols in **Automation**.

— Read more about the workflow in “Workflow for PLCs from the database” on page 420.



Each line in the PLC reference symbol represents a node

In each individual line in the PLC reference symbol, you can see information about one individual node (input/output) on the PLC:



Each of these nodes on the PLC can be represented as individual PLC I/O symbols in the project drawings. You can also apply PLC symbols representing e.g. eight nodes.

All re-addressing is performed via the PLC reference symbol

All general re-addressing of PLCs / PLC modules, takes place via the PLC reference symbols, as described in “Re-addressing PLCs” on page 423.

Creating PLC reference symbols

See how to create PLC reference symbols in “Creating Symbols” on page 696, in particular “Creating PLC reference symbols” on page 728.

MORE PLC REFERENCE SYMBOLS FOR THE SAME PLC COMPONENT

You can also control PLC I/O data for a PLC component by using more than one PLC reference symbol for the component.

— Please also refer to “Re-addressing when more PLC ref. symbols for a component” on page 423.

ASSIGNING PLC ADDRESSES DIRECTLY FROM THE DATABASE

All PLC components can be assigned addresses directly from the database. This saves time, and reduces the risk for errors when typing in data.

Read more about this in “Creating components in the database” on page 589, especially “Addressing PLC symbols via the database” on page 601.

As an example on this, you can refer to the PLC with EAN number **PCS8920107** in the included database.

Base addressing of PLCs recommended

It is recommended that all PLCs in the database has a base addressing, so that they are assigned the addresses they would have, if they were placed in position 0 (lowest address).

This also applies for PLCs — e.g. PLC modules — which does not have fixed addresses.

This is because the base addressing tells the program which addressing principle should be applied for this particular PLC during subsequent re-addressing in **Automation**.

PLCs in the included database

In the database that you get along with the **Automation** software, you find the PLCs under the **Electronic systems** folder, where they are placed in the **PLC control systems** sub folder.

.00	00	00
.01	01	01
.02	02	02
.03	03	03
.04	04	04
.05	05	05
.06	06	06
.07	07	07
.08	08	08
.09	09	09
.10	10	10
.11	11	11
.12	12	12
.13	13	13
.14	14	14
.15	15	15

PLC addressed directly from the database

WORKFLOW FOR PLCs FROM THE DATABASE

When you are using PLCs which have been posted in your database for **Automation** — see “Creating components in the database” on page 589 and “Addressing PLC symbols via the database” on page 601 —, the following workflow is recommended:



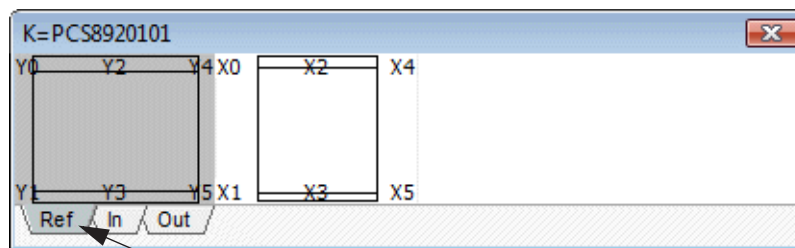
PLCs AND SUB DRAWINGS

WHEN YOU WISH TO WORK WITH PLCs AND USE SUB DRAWINGS, PLEASE READ THE MINI MANUAL "PLC TUTORIAL". HERE YOU FIND A DESCRIPTION OF THE ADVANCED POSSIBILITIES YOU HAVE FOR DOING THIS.

THE MANUAL CAN BE DOWNLOADED AT www.pcschematic.com/manuals.

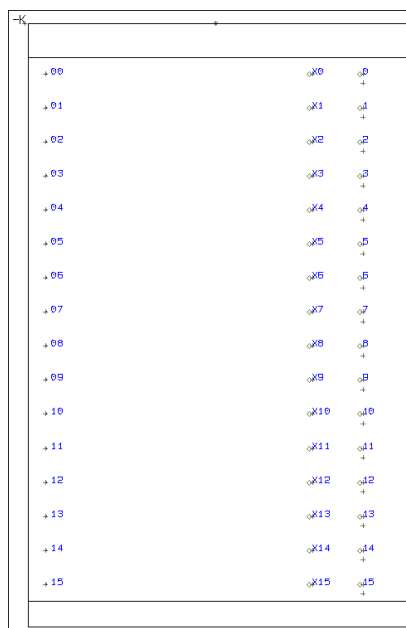
1. Select a PLC and place the PLC reference symbol

- 1a) Select the PLC in the database, or choose a PLC which has been placed in the program pickmenu with data from the database. — Select for instance the PLC with EAN number **PCS8920101** in the included database. You now get a pickmenu for the component:



The PLC reference symbols are placed on this tab

- 1b) Choose the first of the PLC reference symbols (in the figure above: the reference symbol for the outputs), place it on a project page, and assign a name to the component in the dialog box **Component data**. The reference symbol is now placed on the project page:



1c) From the PLC reference symbol, you can now control all data for the PLC in the project. In each individual line in the PLC reference symbol, you can see information about an individual node in the PLC, represented by I/O symbols for the PLC in the project — see “Each line in the PLC reference symbol represents a node” on page 418.

2. Loading the PLC I/O list

2a) Now add labels and descriptions to the PLC in one of the two following ways:

- Load an I/O list for the PLC, as described in “Reading PLC I/O lists” on page 432. Load e.g. the file **Pcsexcelio.xls** in the **Lists** folder (apply the format file **ExcellIO.plc** — the PLC must be named **-K1**), which matches the selected PLC with EAN number **PCS8920101** in the included database. Or:
- Type in labels and descriptions for the PLC via the **Object Lister**, as described in “Typing in PLC I/O data via the Object Lister” on page 426. (Please also read “Re-addressing PLCs” on page 423).

3. Placing PLC I/O symbols in projects

3a) You can now place the I/O symbols for the PLC in the project: If the **Show available window** is opened, the pickmenu for the PLC is displayed at the bottom of the window. (If not, right-click on the PLC reference symbol, choose **Show available**, and the pickmenu for the PLC appears). In the pickmenu for the PLC: Pick an I/O symbol, and place it in the project.

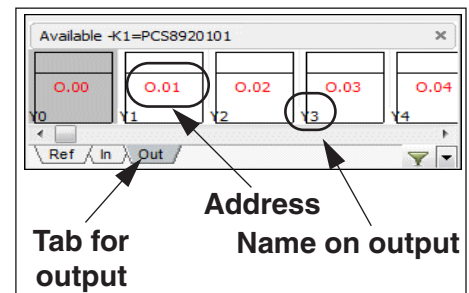


ADDRESSES ON I/O SYMBOLS ARE DISPLAYED IN THE PLC PICKMENU

IN THE PICKMENU FOR THE PLC, THE ADDRESS FOR EACH I/O SYMBOL APPEARS IN RED WRITING.

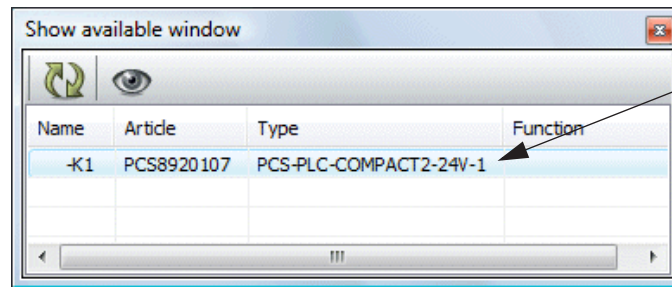
CONNECTION NAMES ARE DISPLAYED AS FOR OTHER SYMBOLS.

— SEE MORE ABOUT PLC SYMBOLS IN SHOW AVAILABLE IN “SYMBOLS FOR PLCs IN SHOW AVAILABLE” ON PAGE 236.



3b) Now place the remaining PLC I/O symbols. If the **Show available window** is not displayed, it can be a good idea to choose **View => Show available window**, which gives you direct access to choose the pickmenu for the PLC — and makes it easier to keep the survey, while working with several PLCs at

a time.



Double-click here and the pickmenu for the PLC is displayed

— See “The Show available window” on page 234.

See also:

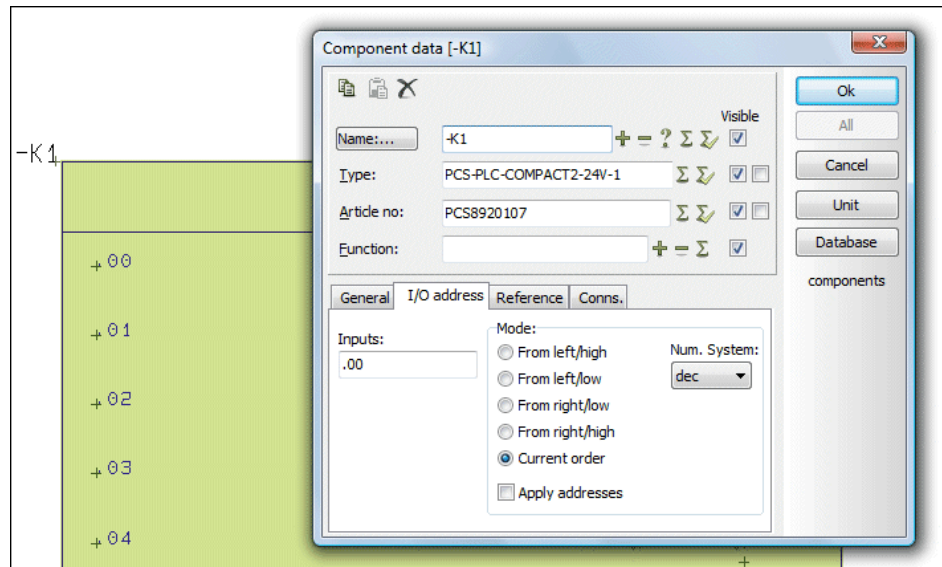
- “More PLC reference symbols for the same PLC component” on page 419.
- “Re-addressing PLCs” on page 423.
- “PLC data on the I/O symbol connection points” on page 425.
- “On PLC reference symbols” on page 418.
- You can also read the independent mini manual "PLC tutorial". Here you are introduced to the best ways of working with PLCs. You can also see how to use the advanced options for keeping track the addressing when working with PLCs and sub drawings. The manual can be downloaded at www.pcschematic.com/manuals.

RE-ADDRESSING PLCs

All re-addressing of PLC modules in **Automation** takes place via the PLC reference symbols.

To re-address, do the following:

- 1) Right-click on the PLC reference symbol, choose **Component data**, and click on the tab **I/O address**:



- 2) Type in a new start address for the PLC in the left-hand side, click on **From left/high** — or pick another corner to start numbering from —, choose **Number system**, and click **OK**. (See “Options on the tab I/O address in the Component data dialog box” on page 423).



RE-ADDRESSING WHEN MORE PLC REF. SYMBOLS FOR A COMPONENT

ONLY INPUTS/OUTPUTS ATTACHED TO THE PLC REFERENCE SYMBOL IS RE-ADDRESSED. WHEN FOR INSTANCE TWO PLC REFERENCE SYMBOLS ARE ATTACHED TO A COMPONENT, ONLY THE INPUTS/OUTPUTS REPRESENTED IN THE SELECTED PLC REFERENCE SYMBOL ARE RE-ADDRESSED.

WHEN YOU WISH TO RE-ADDRESS INPUTS/OUTPUTS FOR MORE PLC REFERENCE SYMBOLS FOR THE COMPONENT, CHOOSE ALL RELEVANT PLC REFERENCE SYMBOLS *BEFORE* ENTERING THE **COMPONENT DATA** DIALOG BOX AND RE-ADDRESS.

Options on the tab I/O address in the Component data dialog box

On the tab **I/O address** in the **Component data** dialog box, you have the following options:

Specify start address

In the left-hand side you type in the start address in the field **Input** and/or **Output**.

Mode

Under **Mode** you can specify from which corner of the PLC reference symbol the numbering shall begin. Here **From left/upper** will be selected by default, which will cover most situations.

Choosing a numbering system

In the field **Num. System** you can choose whether to number according to the binary, the octal, the decimal or the hexadecimal numbering system.



RENUMBERING PLC ADDRESSES WITH COMPOUND NUMBERING SYSTEMS

WHEN RENUMBERING OF PLC ADDRESSES, **Automation** ALSO HANDLES RENUMBERING FOR MIXED NUMBERING SYSTEMS — E.G. DECIMAL NUMBERING *BEFORE* THE PERIOD COMBINED WITH OCTAL NUMBERING *AFTER*. FOR INSTANCE CAN **I17.0-I17.7** BE RENUMBERED TO **I18.0-18.7**.

IN THE FIELD **NUM. SYSTEM** YOU MUST SPECIFY THE NUMBERING SYSTEM APPLIED *AFTER* THE PERIOD.

Applying the addresses

To apply the addresses, **Apply addresses** must be activated. **Apply addresses** is deactivated each time you enter the dialog box. This ensures you that you do not activate a re-numbering by clicking **OK** after making changes in the upper part of the dialog box. When you make changes on the tab **I/O address**, **Apply addresses** is activated automatically.

Changing addresses on PLC I/O symbols

When you change the address on a *PLC I/O symbol*, this has the effect that all data for this address in the PLC reference symbol, is transferred to the PLC I/O symbol.

It is not recommended to change the address for a PLC I/O symbol to an address which does not exist in the PLC reference symbol. The only thing you should do in this context, is to specify that the PLC I/O symbol now represents another input/output for the same PLC.

— Read more about this in “PLC data on the I/O symbol connection points” on page 425.

Exporting PLC I/O data

Read “Export of PLC I/O data to Excel” on page 435 for information about how to export PLC I/O data from **Automation**.

PLC DATA ON THE I/O SYMBOL CONNECTION POINTS

When working with PLC I/O symbols, data for the PLC inputs/outputs are attached to the *connection points* on the PLC I/O symbols. This makes it possible to either document PLC's using PLC I/O symbols representing e.g. eight inputs/outputs, or by PLC I/O symbols representing a single input/output in each symbol.

Double-click on the *connection point*, Or: Right-click on the symbol, and see PLC data for the node

and see PLC data for the node

choose PLC I/O data ...

... and edit data via the Object Lister

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=0,00mm

The I/O address determines the other texts in the dialog box

It is the address of the PLC I/O symbol connection point, that determines the contents in the other fields for the connection point. When you double-click on the connection point and enter the dialog box **Connection data**, this address is specified in the **I/O addr.** field.

When you change the I/O address to another I/O address for the PLC — e.g. to the address *00.06* in the figure above — and click **OK**, **Name**, **Label** and **Description** for the node / connection point is updated with the data specified for the I/O address in the reference symbol for the PLC.



RE-ADDRESSING PLCs ONLY POSSIBLE FROM PLC REFERENCE SYMBOLS

ALL GENERAL RE-ADDRESSING OF PLCs TAKES PLACE VIA THE PLC REFERENCE SYMBOLS. — SEE “RE-ADDRESSING PLCs” ON PAGE 423.

Changes in Label and Description are updated in the PLC ref. symbol

When you change **Label** or **Description** in the dialog box **Connection data**, this information is updated in the reference symbol for the PLC. — Please note that you also can edit the **Label** and **Description** texts via the **Object Lister**, as described in “Typing in PLC I/O data via the Object Lister” on page 426.

When you change **Label** and/or **Description** and at the same time change the I/O address in the dialog box, the changes in **Label/Description** will be overwritten by the new data, which are fetched — via the address — from the PLC reference symbol.

TYPING IN PLC I/O DATA VIA THE OBJECT LISTER



RE-ADDRESSING PLC'S

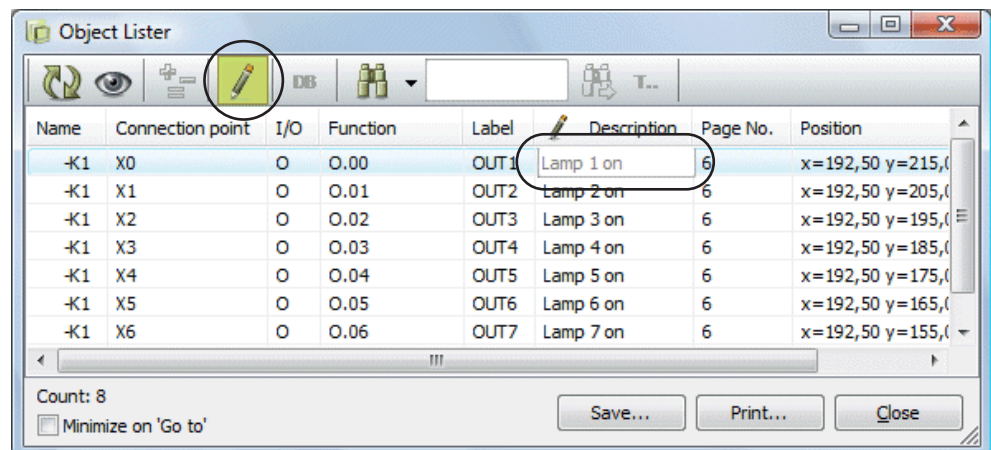
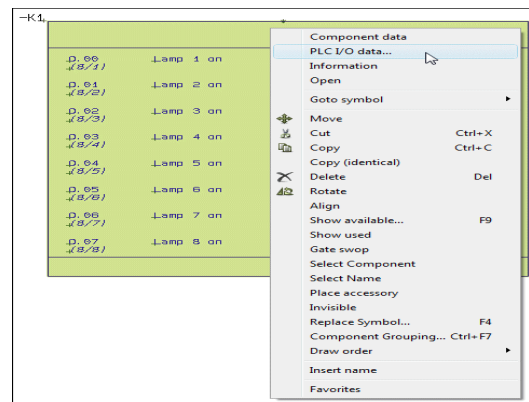
IF YOU WISH TO RE-ADDRESS A PLC, PLEASE READ "RE-ADDRESSING PLC'S" ON PAGE 423.

When you wish to type in / edit PLC I/O data manually in **Automation**, do the following:

- 1) Right-click on the reference symbol for the PLC, and click on **PLC I/O data**.
- 2) You now enter the **Object Lister**, where PLC I/O data are displayed for the selected PLC reference symbol.

If the data for the PLC is spread over more than one PLC reference symbol, only I/O data for the inputs/outputs represented by the selected PLC reference symbol is displayed.

- 3) In the **Object Lister**: Click on the **Pencil**, double-click on the relevant fields (or click and press <F2> / <Enter>), and type in the new data.



4) Click **OK**, and the change(s) are transferred to the PLC.



EDITS ONLY I/O DATA ATTACHED TO THE SELECTED PLC REF. SYMBOL

WHEN YOU RIGHT-CLICK AND CHOOSE **PLC I/O DATA**, YOU CAN ONLY EDIT THE I/O DATA ATTACHED TO THIS REFERENCE SYMBOL.

THIS MAKES IT EASIER TO MAINTAIN THE SURVEY WHEN WORKING WITH LARGE PLCs WITH MORE THAN ONE ATTACHED PLC REFERENCE SYMBOL.

Read more about the **Object Lister** in “The Object Lister” on page 306. Here you can read about the options for applying automatic counting functions, and the possibilities for saving data in various file formats.

— Please read “Reading PLC I/O lists” on page 432 also.

SEE VIDEO ON

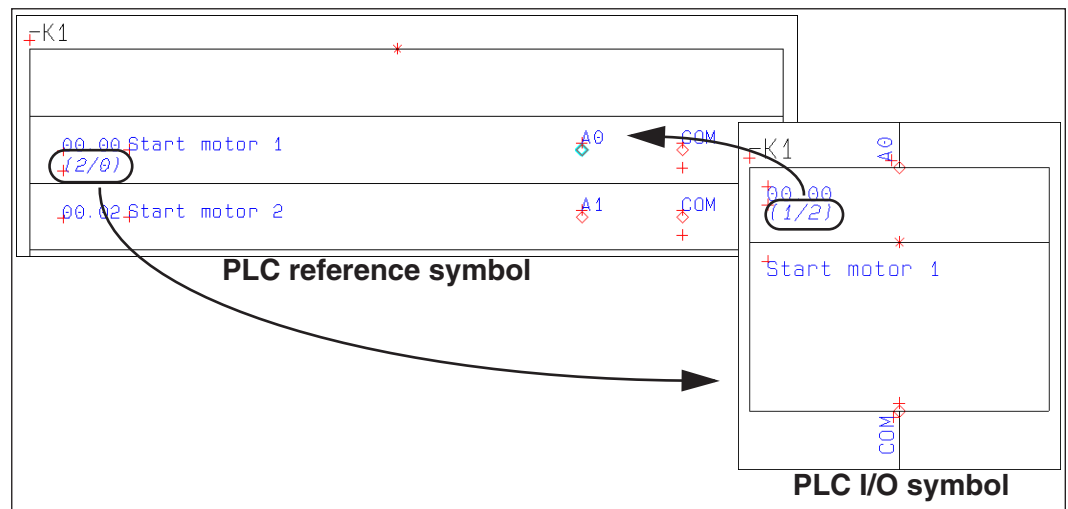


OUR WEBSITE

JUMPING BETWEEN PLC REF. SYMBOLS AND PLC I/O SYMBOLS

When you have placed an I/O symbol, you can — in the PLC reference symbol — double-click on the reference to the I/O symbol, and you jump to the I/O symbol.

On the PLC I/O symbol you can double-click on the reference to the PLC reference symbol, and you jump to the line for the selected I/O symbol in the PLC reference symbol:



— Read more about references in “Automatically updated references” on page 367.

WORKFLOW FOR PLCs NOT REPRESENTED IN THE DATABASE



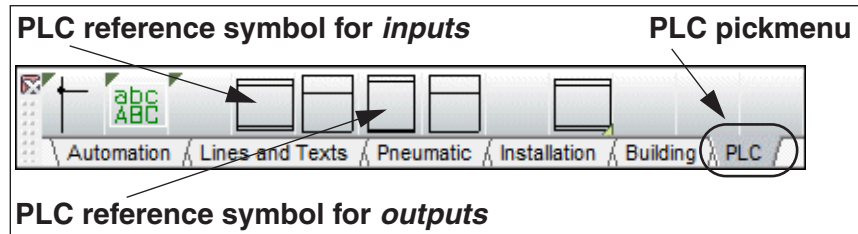
IT IS RECOMMENDED TO ADD THE APPLIED PLCs TO THE DATABASE

IT IS STRONGLY RECOMMENDED TO ADD THE APPLIED PLC'S TO THE DATABASE. THIS CAN SAVE YOU A LOT OF TIME IN THE FUTURE, AND REDUCE THE RISK FOR ERRORS. — PLEASE ALSO SEE “ASSIGNING PLC ADDRESSES DIRECTLY FROM THE DATABASE” ON PAGE 419.

When you choose to work with PLCs, which does not exist in your database for **Automation**, you should do the following:

1. Select PLC reference symbol(s) for the component

- 1a) Select an appropriate PLC reference symbol, and place it in the project. You can for instance choose one of the two PLC reference symbols placed to the right in the pickmenu:



— If you need another already created PLC reference symbol, these can be found via the **Symbol menu** in the folder **PLC346**, where the PLC reference symbol names starts with **PLCRef**.

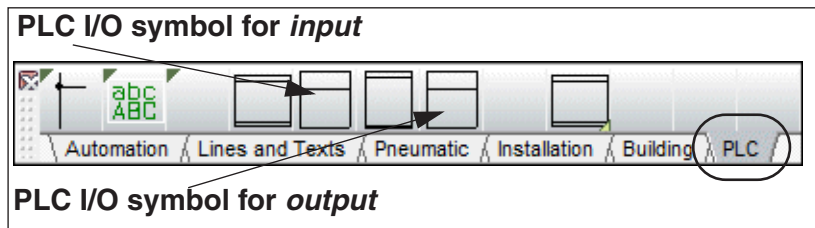
- 1b) Click to place the PLC reference symbol, and name it in the dialog box **Component data**. If you need more than one PLC reference symbol for the same PLC component, just click to place these also, and assign the same name to these symbols.
- 1c) Then address the PLC reference symbol(s) as described in “Re-addressing PLCs” on page 423, so that the addresses matches this PLC.

2. Type in data for the PLC

- 2a) Type in data for the PLC, as described in “Typing in PLC I/O data via the Object Lister” on page 426.

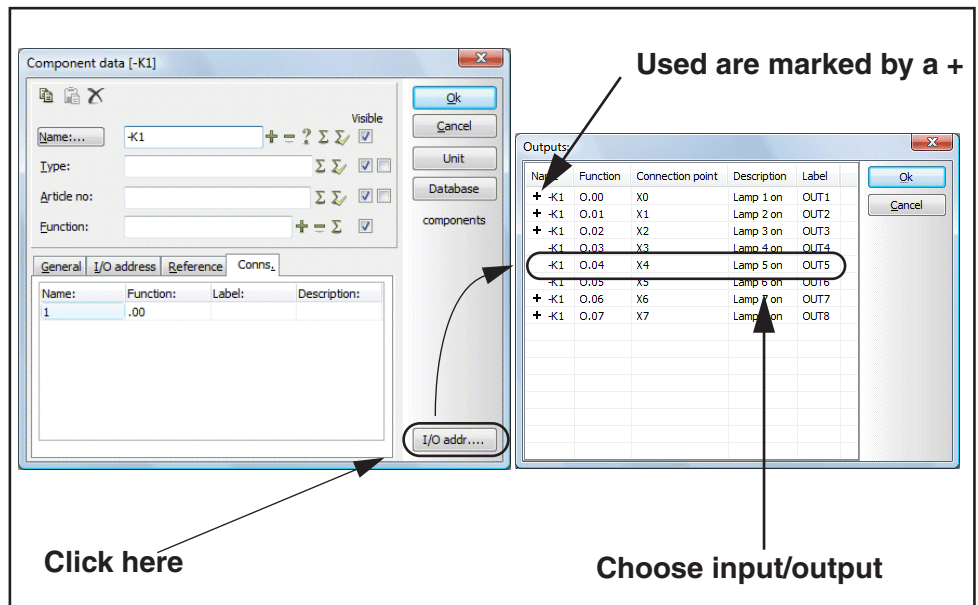
3. Place the PLC I/O symbols and choose PLC I/O data

3a) Place the PLC I/O symbols for the PLC by selecting them in the pickmenu at the upper part of the screen:



Select the symbol, and click to place it on the project page.
 — If you need another already created PLC I/O symbol, these can be found via the **Symbol menu** in the folder **PLC346**.

3b) When you enter the dialog box **Component data**, you type in a component name in the **Name** field and click on the **I/O addr** button in the lower-right corner of the dialog box:



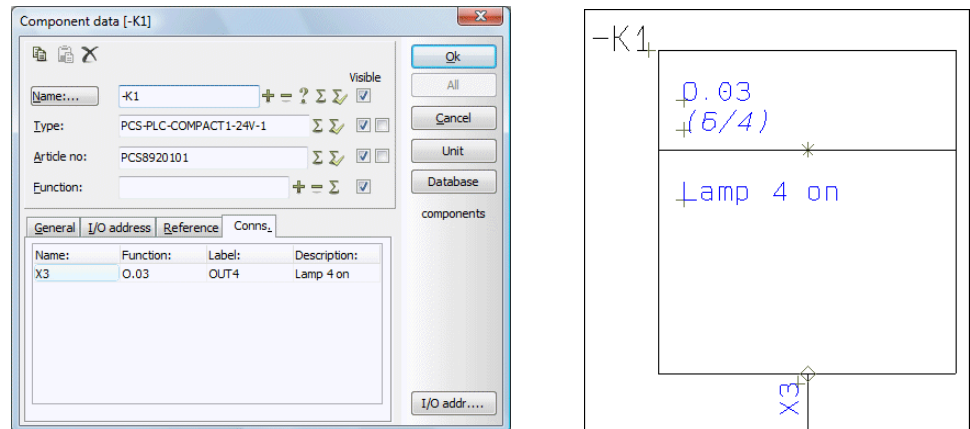
3c) You now enter the dialog box **Inputs** (or **Outputs**), where you click on the relevant input/output, and click **OK**.
 Please note that used inputs/outputs are marked by a preceding **+**.



CHOOSE INPUTS/OUTPUTS FOR ALL PLCs IN THE PROJECT

IF FOR INSTANCE YOU ONLY TYPE IN **-K** IN THE **NAME** FIELD, INPUTS/OUTPUTS FOR ALL PLCs IN THE PROJECT WITH NAMES BEGINNING WITH **-K** ARE DISPLAYED.

3d) When returning to the **Component data** dialog box, you will see that it now carries data for the component, and when you click **OK**, you will subsequently see that the PLC symbol now carries the selected I/O data:



3e) Continue like this, until you have placed the relevant symbols.

See also:

- “Workflow for PLCs from the database” on page 420.
- “Re-addressing PLCs” on page 423.
- “PLC data on the I/O symbol connection points” on page 425.

SYNCHRONIZING PLC DATA WHEN COPYING BETWEEN PROJECTS

If you merge project pages from different projects, so that the pages containing the PLC symbols come from one project, while the pages with the PLC reference symbols come from another, no synchronization are made between the PLC symbols and the PLC reference symbols.

This is because the program does not know whether to transfer the values from the PLC reference symbols to the PLC symbols — or the reverse.

You specify this yourself, by using one of the following functions:

- **Functions => Special Functions => Transfer data from PLC reference symbols to PLC symbols, or**
- **Functions => Special Functions => Transfer data from PLC symbols to PLC reference symbols.**

UPDATING PLC PROJECTS FROM PREVIOUS VERSIONS

When you wish to update a PLC project from an earlier version of **Automation** than version 7.0, to get the full benefit of applying PLC reference symbols, do the following:



- 1) Enter the **Symbol menu**, click on **Project symbols** and update the symbol definition on the applied PLC I/O symbols, so that they follow the guidelines described in “Creating PLC I/O symbols” on page 725.
- 2) In the project: Replace the applied PLC I/O symbols with the new versions of the symbols. Even though you save the updated PLC I/O symbols under the same names as before, the symbols in the project must still be replaced with the updated versions. — See “Replacing symbols” on page 193.

- 3) Create PLC reference symbols for all PLCs in the project. — See “Creating PLC reference symbols” on page 728.
(You might use the predefined PLC reference symbols in the **PLC346** folder).
- 4) Update the PLCs in the database, so that they now have PLC reference symbols and connection names for the PLC reference symbols. — See “Creating PLC components in the database” on page 606.
Because the PLCs most likely are already addressed in the existing project, it is not strictly necessary to address the PLC’s in the database, though it is recommended because of the subsequent use of the PLCs in other projects. — See “Assigning PLC addresses directly from the database” on page 419.
- 5) Right-click on one of the PLC I/O symbols in the project, and choose **Show available**.
- 6) In the pickmenu for the selected PLC: Click on the PLC reference symbol, and place it an appropriate place in the project. Please note that the PLC reference symbol is automatically filled out with data from the placed PLC I/O symbols.
- 7) Repeat the two steps above for all PLCs in the project.

The project is now updated.

READING PLC I/O LISTS

In **Automation** you can read PLC I/O lists from PLC programming tools, and have the changes transferred directly to your diagrams.

THIS CHAPTER DESCRIBES	
— CHOOSING THE FORMAT FILE	P. 432
— SPECIFYING THE FILE NAME	P. 433
— DISPLAYING THE CONTENTS OF THE FILE.....	P. 433
— IF MORE THAN ONE PLC IN THE PROJECT.....	P. 434
— DISPLAYING THE CHANGES IN THE PROJECT	P. 434

The reading procedure is build up step by step, so that you only make one decision in each dialog box. This is to avoid errors, because errors under the reading could be crucial.

CHOOSING FORMAT FILE

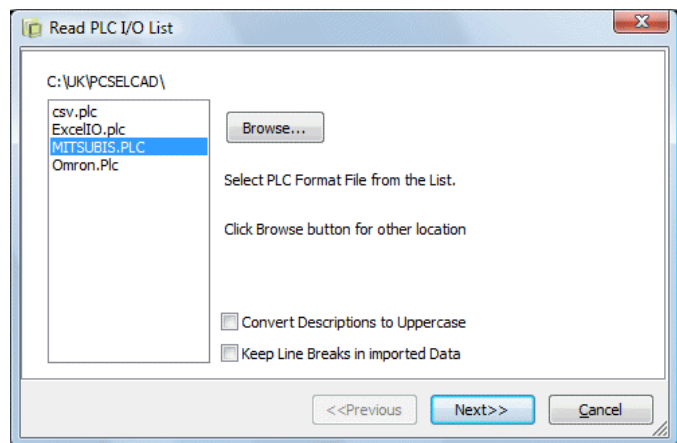
To start reading a PLC I/O list, choose **PLC => Read PLC I/O List**.

In the first dialog box, you choose which format file to use, when you read the PLC I/O list.

This format file contains information about how the contents of the PLC I/O file shall be interpreted.

Here you choose a format file from the displayed list, or click on **Browse** and choose another format file.

As you can see, you have the option of converting description texts on PLC connections to upper case characters during the import, by selecting **Convert Description to Uppercase**. You can also specify to keep line breaks in the imported data by selecting **Keep Line Breaks in imported Data**.



Click **Next**.

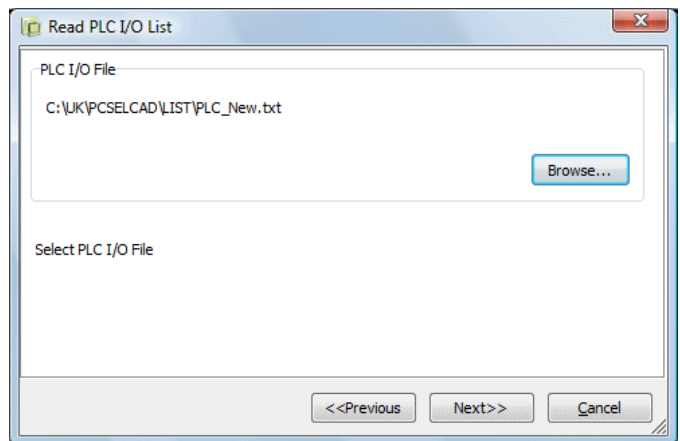
— For further information about format files, please refer to “The format file” on page 650.

SPECIFYING THE FILE NAME

You are then asked about the name of the PLC I/O file itself.

To keep the survey of the project file names, it is recommended to let the file have the same name as the project — but of course having another extension (such as **.txt** for text files, or **.xls** for Excel files).

Click on **Browse** if you need to locate another file, and then click **Next**.



THE CONTENTS OF THE FILE

You can then see the contents of the file you are going to read, in order to make sure, that you have chosen the correct file.

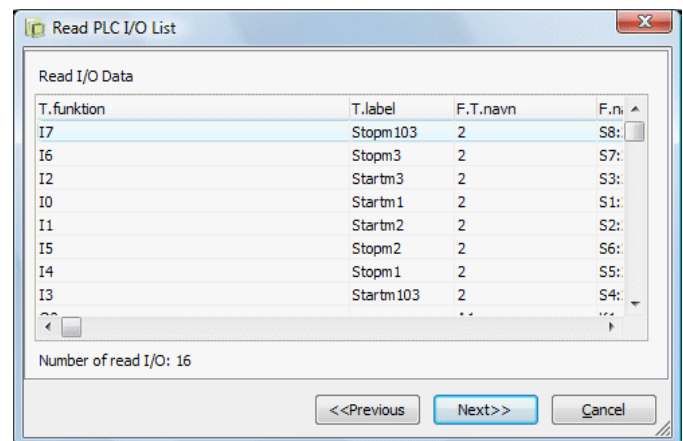
The contents of the file will be displayed in columns, for clarity — even though it might actually be a comma separated file. If the column width is incorrect, this can also be detected here.

Apply the elevator in the right-hand side of the dialog box, to see the entire contents of the file.

Please notice, that it is not possible to make changes in the file here.

Click **Next** if no errors are found in the file.

If you find an error, click **Cancel**, and the file will not be read.

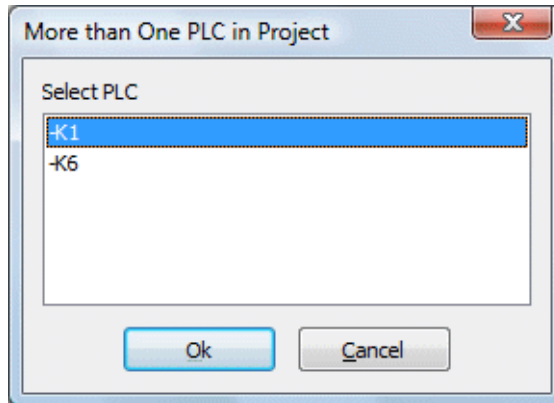


CONNECTION NAMES AS PART OF PLC NAMES

DURING LOAD OF PLC I/O LISTS, THE CONNECTION NAMES OF THE PLC I/O'S CAN BE A PART OF THE PLC NAME, SO THAT YOU DO NOT NEED A SEPARATE COLUMN FOR CONNECTION NAMES IN THE PLC I/O FILE. (E.G. **K1:1**).

IF MORE THAN ONE PLC EXISTS IN THE PROJECT

If no name has been specified for the PLC in the file, and there are more than one PLC in the project, you must now select which PLC the file contents matches:



Click on the PLC, and click **OK**.

DISPLAYING THE CHANGES IN RELATION TO THE PROJECT

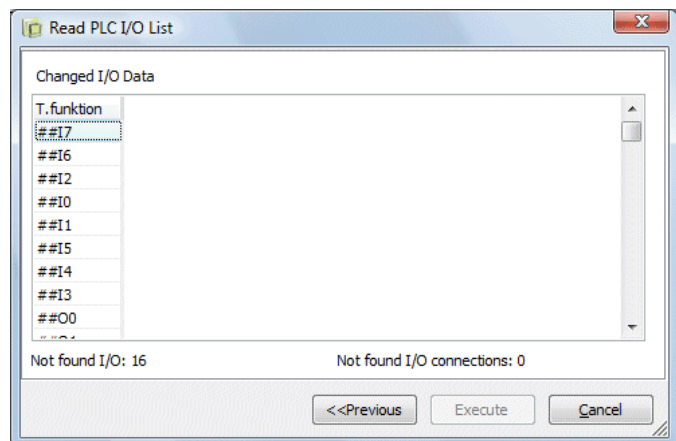
It is then displayed which changes the PLC programmer have made, compared to the information in the project.

When there are ## characters in a datafield, this means that the program has not been able to locate this datafield in the project.

If the sign ^ already is applied to signify line breaks, the program estimates which column width the use of this sign corresponds to. In the following, the program places ^ signs in the new texts, in order to make them follow a similar column width.

If you need to, click on **Print** to print out the changes for documentation purposes, or to pass it on to the PLC programmer.

If everything is in order, then click on **Execute**. It is not until you click here, that the program reads the PLC I/O list.



PLC I/O ADDRESSING VIA EXCEL

In **Automation** it is possible to export PLC I/O data to an Excel file, which can be opened in Excel automatically.

THIS CHAPTER DESCRIBES	
— EXPORTING PLC I/O DATA TO EXCEL	P. 435
— IMPORTING PLC I/O DATA FROM EXCEL.....	P. 436

For further details about the options in the individual dialog boxes, please refer to “Reading PLC I/O lists” on page 432.



EXCEL MUST BE INSTALLED

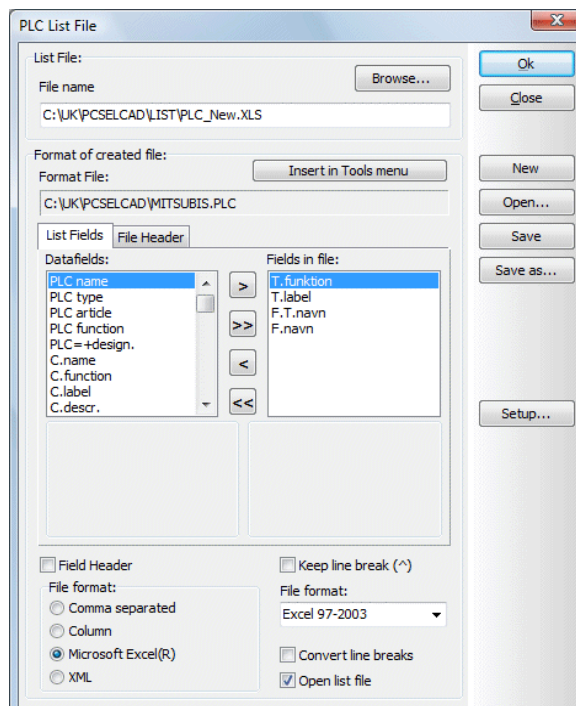
IT IS ONLY POSSIBLE TO SAVE AND OPEN FILES IN EXCEL FORMAT IF EXCEL IS INSTALLED ON THE PC.

READ MORE ABOUT THIS IN “EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT” ON PAGE 666.

EXPORT OF PLC I/O DATA TO EXCEL

In a project containing PLCs — such as **Plcdemo.pro** —, exporting PLC I/O data can be done like this:

- 1) Choose **PLC => PLC List File:**



- 2) Click on **Open** — to open a format file specifying which information to include in the generated file. (You can also specify this information in the dialog box manually).
- 3) In the dialog box **Open**: Select the format file **ExcellIO.plc** in the folder **Pcselcad** — or another suitable format file for the export — and click **OK**.
- 4) The settings are now as displayed in the dialog box above. Please note that the file format is set to **Microsoft Excel(R)**, and that **Run Microsoft Excel(R)** is selected.
- 5) Select **Field Header** in the dialog box, so that you get a header in the Excel sheet, that informs you about the contents of the columns.
- 6) Click **OK**.
- 7) When there is more than one PLC in the project, you now enter a window, where you select which PLC(s) to export PLC I/O data for, or whether to export PLC I/O data for.
Select a PLC, or select multiple PLC's — e.g. by pressing <Ctrl> and clicking on them — or select all by clicking on **All**.
- 8) Click **OK**.
- 9) Excel is now started, and the created PLC I/O file is displayed in Excel.

IMPORTING PLC I/O DATA FROM EXCEL

In **Automation** you can update I/O addresses having a PLC name and connection number in the PLC I/O list file.

See how to import a PLC I/O list in Excel format in “Reading PLC I/O lists” on page 432.

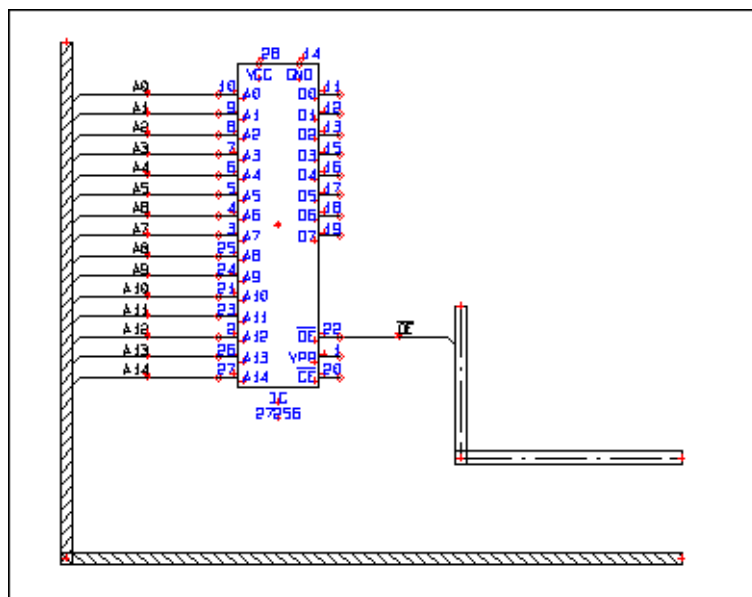
CONNECTION SYMBOL FOR SIGNAL BUSES

THIS CHAPTER DESCRIBES

THIS CHAPTER DESCRIBES HOW TO WORK WITH CONNECTION SYMBOLS FOR SIGNAL BUSES.

When you want to display a signal bus on the drawing, you do this by drawing a wide non conducting line.

— Please refer to “Creating Join symbols” on page 741 to see how to create join symbols.

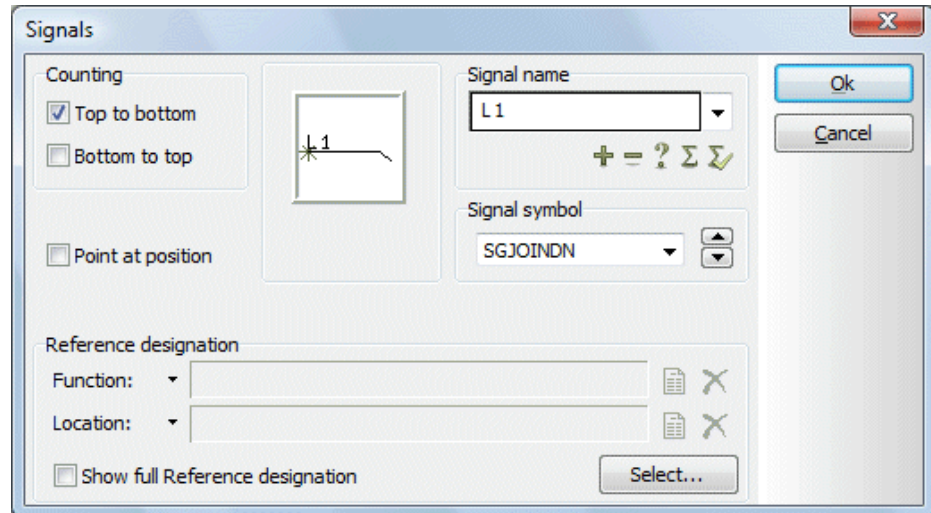


To specify a signal bus as displayed above, do this:



- 1) Find a symbol for an IC or a PLC and place it on the drawing.
- 2) Activate the **Pencil** and click the **Lines** button, select a wide line type — for instance like the one above — and set the line width (**A:**) to an appropriate size. On the drawing above it is set to 2.5 mm.
- 3) Draw a line like the left-most line on the drawing.
- 4) Then draw a line like the one on the right side of the symbol. The lines must not be conducting (which they also can not be when you are using a wide line type).
- 5) You have now drawn the two signal buses: Deactivate the Pencil (press <Esc>), select a thin line type, and click on the line to the left of the circuit. Thus the line is selected.
- 6) Then choose **Edit => Join signal**.

- 7) A rubber band now emerge starting from the selected line and ending up in the cross hairs. Mark up a window that includes all the connection points on the left side of the circuit.
- 8) The following dialog box appears:



- 9) Below **Signal name** you either type in a name, or specify the name using the buttons under the **Signal name** field, as described in “Specify Signal name” on page 205.
If you click on the up- or down-arrow in the right hand side of the dialog box, you can choose between the signal symbols, that are defined in the system. The symbol itself will also be shown in the window inside the dialog box. In the upper-left corner you state whether the number in the signal names shall be assigned from **Top to bottom** or from **Bottom to top**. Click **OK** when you are satisfied with the settings.
- 10) Press <Esc>, and the line is no longer selected.
- 11) If you just want to connect one connection point to the signal symbol, you can try to click on the thick line at the right-hand side of the circuit. Thus the line is selected.
- 12) Then choose **Edit => Join signal**.
- 13) A rubber band now appears starting from the line and ending up in the cross hairs. Now click on the connection point on the circuit you wish to connect to the bus.
- 14) A dialog box, like the one above, now appears on the screen, and you continue as described above.
- 15) When you have connected the first connection point to the signal bus, you will see that there is still a rubber band radiating from the signal bus to the cross hairs. If you want to connect more legs to the bus, you can now carry on connecting their connection points individually.
- 16) If you want to determine yourself where on the line the signal symbol shall be located, just select **Point at position** in the dialog box. When you click

OK after doing this, the next thing to do is to click between the signal bus and the connection point, where you want the signal symbol to be placed.

The program keeps full control over where the signals enters and leaves the signal bus via the signal names. When for instance the signal name *A0* is connected somewhere to the signal bus, the signal continue the next place *A0* is connected to the signal bus.



CHANGE SIGNAL SYMBOL BY PRESSING <SPACEBAR>

WHEN YOU HAVE SELECTED A SIGNAL SYMBOL, YOU CAN CHANGE BETWEEN THE PREDEFINED SIGNAL SYMBOLS BY PRESSING <SPACEBAR>.



INDEX



INDEX FOR PART 1 AND PART 2

A

Absolute measures 93
 Access 623
 Accessories
 for cables 604
 Accessory symbols 225
 Activate next 111, 756, 873
 Active 478
 Active, check box 380
 Actual page 481
 Adaptability
 symbol names 385
 Adapting the system to your needs 786
 Add a wire to a cable 223
 Add new 494, 514
 Additional 613
 Adjust to printer 329
 Adobe Acrobat Reader 37
 ALARM, symbol folder 226
 Alias
 correct 842
 Align 66, 631
 Align grid to 1. Reference 836
 Alignment 124
 All pages 327
 All visible 478
 Already placed objects 54
 Alternative electrical symbols 594
 Alternative symbols
 and PINDATA 596
 for PLC 595
 Ambiguous drawing 392
 ANSI
 paper size 509
 Apply addresses 424
 Approved components
 in Database menu 614
 Arcs/circles 53, 100
 toolbar 100
 Area button 53
 Areas
 change symbol names in 185
 change symbol names in, naming order 186
 choosing connection points in 189
 copy
 between projects 255
 into Edit symbol 255
 reference designations intelligently 247
 copy between projects 518
 copy intelligently 245
 counting on texts in areas 120
 filled 75
 invisible 70
 rotate 253

 scaling texts in 254
 Article data 835
 on lines 79
 transfer for symbols or lines 62
 Article number
 type in directly 154
 Articles from unit drawings 544
 Ask for Names 107, 837
 Assign article number to unit parts list 768
 Asterisk 550
 Auto hide scrollbars 838
 Autocompletion of texts 103
 Automatic
 blocking of mouse clicks 802
 insertion of electrical potentials 81
 line drawing 89
 recovery information 849
 wire numbering 389
 Automatic information 784
 Automatic renumbering of pages 832
 Automatic switches, database folder 153

B

Background color 792
 Baltic characters 787, 850
 Barcode scanners 624
 Base addressing
 for PLC's 419
 Batch printing 823
 on net installation 823
 on stand-alone installation 823
 BDE 623
 Bend 96
 Black background 839
 Bold text 125
 Bookmarks
 in Symbol menu 148
 Bottom to top 438
 BUILDING, symbol folder 226
 Busbar symbols 724
 Busbar, symbol type 707
 Busses 437

C

Cable list, page function 497
 Cable symbols 218
 accessories for cables 604
 add wire to 223
 color codes 221
 create 722
 implicit references on cable wires 213
 invisible 225
 place in one operation 222
 remove wire from a cable 223

reverse direction	224	Squared grid	792
same cable in more places	224	squared grid-10	792
Cables		Color codes for cables	221
move		Columns	
connection points	222	change column of data fields	765
Cables lists		lists in columns	546
create	760	COM/OLE automation	7
data fields	111	Combined circuit and connection diagram	95
list setup	553	Command bar	39
Calculate scale	253	Command toolbar	873
Cancel button	25	Company and User name	849
Capital letters	127	Company logo	
Cascade	516	on drawing header	751
CD		remove	764
component vendors	8	Component	
Change		insert in database	591
between projects	520	Component databases	
between windows	25	how does the database work	572
column of data fields	765	Component Grouping	
created drawing header	763	and copying	562
data field	764	menu	557
free texts	764	Component vendors	
name format types	384	CD with databases	8
symbol in folder	731	Components	
symbol names in Area	185	approved and obsolete in Database menu	614
symbol names in Area, naming order	186	changing symbol data for Component	242
symbol type	731	create in database	589
text	110	from existing project	590
text in the cross hair	116	fetching component symbols from database	230
Chapters	491	find in database	576
chapter tabs collapsed	491	functions on component level	228
Character set	787	grouping	556
choose	850	having several electrical symbols	593
Chinese version		insert new names	295
PC SCHEMATIC Viewer	680	navigation between symbols	170
Choose language	850	place available symbols	242
Circles	53, 100	rename selected	289
align	66	direction for renaming	292
convert to lines	101	start number	293
full	100	which names are renamed	290
half	100	rrename	
invisible	70	when copying areas	245
quater	100	sorting order	558
space	67	survey of symbols with same Symbol Name	238
toolbar	100	Components list, page function	497
Circles button	100	Components lists	111
Click	25, 873	create	759
Clipboard		files	649
copy to	523	Conducting (electrical) lines	78
Close, symbol status	594	Conducting lines button	78
Close, symbol type	706	Connection data	187
Codepage	664	Connection names	598, 873
Color	124, 792	in one line diagrams	408
background	792	in single line diagrams	408
dotted grid	792	insert new	295
help objects	793	Connection point button	714
lines	76	Connection point texts	109
object selection	793	Connection points	187, 873

choosing in areas	189	Cross hairs line displayed right angled	837
edit next/previous	188	Ctrl key	25
place	699	Current page	327
Connections lists	202	Current path naming	379
to file		Current path numbers	
with Line data	658	default settings	380
wire numbers	404	in symbol names	382
Contact mirrors		setup	379
about	375	Curved lines	74
and changing article numbers	375	Curved lines button	74
choose	377	Czech version	
created automatically by the program	377	PC SCHEMATIC Viewer	680
creating without using database	377	D	
get available symbols in the cross hairs via	375	Damaged files	849
jump to reference from	368	Data	
jump to the other symbols for the component	375	transfer between objects	62
Continuous	380	Data field dialog box	110
Coordinate field	472	Data fields	110, 874
Copy		and PLC lists	761
and component grouping	562	change	764
area form one project to another	518	change column of	765
areas between projects	255	copy rows of	756
areas intelligently	245	create symbol data fields	733
reference designations	247	insert	747
drawing headers	521	missing	534
project into another	513	page/project data	689
rows of data fields	756	reference designations	363
selected objects	58	types	746
symbol names when copying	384	Data symbols	
to other programs	523	and lists	736
Copy button	59	create	732
Copy properties button	63	place in project	734
Correct		with predefined values	736
directories or alias	842	Data, symbol type	707, 734
Unit drawing/parts list	770	DATABASE	845
Counting		Database	847
on texts	120	1. and 2. Searchkey	578
on texts in areas	120	1.Searchkey	577
Counting direction	380	CD from component vendors	8
Counting direction for naming	186	choose a DB alias in PC SCHEMATIC	623
Create		choose database menu	617
cable list	760	choose Menu using other DB programs	619
components list	759	choosing a database	623
diagram page w. list	762	create component	589
drawing headers	742	from existing project	590
lists	752	Database menu	
PLC lists	760	approved and obsolete components	614
signal symbol	738	Product choice or Fabricate choice	613
symbol	696	editing records	582
Table of Contents	759	enter using the pickmenu	142
terminals lists	759	fields	
unit drawing	767	file names in	622
Criteria, tab	549	links	622
Cross hairs	47, 874	find components	576
object selection	793	for lines	571
Cross hairs / Small cross hairs	836	how does the database work	572
Cross hairs figure with snap	837	insert component in database	591
		interface	608

mechanical load 628
 place cable using color codes from database .
 221
 place symbol in pickmenu using database 181
 programmable barcode scanners 624
 searching in the database 575
 settings 608
 transfer information from 573
 translate fields 608
 update symbols from 171, 231, 848
 use 571
 Database setup 847
 Datatemplates
 create variablelist 283
 DDE functions 823
 Default vaule
 for variable 871
 Default, tab 381
 Delete
 page with a list on 766
 selected objects 59
 symbol 173
 symbol from folder 731
 templates 777
 Delete button 59
 DEMOMEK, symbol folder 226
 deselect wirenumbering 390
 Design
 cable list 760
 components list 759
 data symbol 733
 diagram page w. list 762
 lists 752
 PLC lists 760
 signal symbol 738
 symbols 696
 Table of Contents 759
 Terminals lists 759
 unit drawings 767
 Design Check
 dialog box 323
 double-click on error in list 323
 save and print 323
 update design check list 323
 Designcheck 320
 Diagram
 print 326
 Diagram, page type 508
 Diagrams
 loop diagrams 7
 Dialog boxes 24
 selecting elements in lists 24
 Dialogue 8
 Dictionary 873
 Different types of objects, select 244
 Dimmed 24, 874
 Direct access
 to menus 48

Directories 840, 874
 most used 841
 Directory alias
 show 841
 Displayed in project 127
 Distance (mm) 380
 Distance, lines 76
 Divider, page function 498
 Dock 874
 a window 27
 Explorer window 26
 Symbol pickmenu 27
 Toolbars 28
 Document object =>
 Edit 671
 Open 671, 672
 Properties 672
 Dotted grid color 792
 Double-click 25, 874
 Drag
 pages between projects 517
 Drag and drop 26, 874
 Draw
 house walls 473
 inner walls 474
 Draw button 72
 Drawing
 by measures 93
 measurement objects 641
 print 326
 standard projects 511
 unambiguous 392
 with elevation 482
 Drawing headers 835
 add to list of drawing headers 750
 and copying 521
 and layers 480
 change already created 763
 company logo on 751
 create 742
 insert 531
 predefined 751
 remove form list of drawing headers 750
 replace 535
 Drawing objects 53, 875
 Drawing templates
 insert variables 281
 insertion points 279
 Drive 875
 Drop down menu 24
 Dutch version
 PC|SCHEMATIC Viewer 680
 DWG files 660
 calculate scale 253
 export 663
 export project as 663
 export symbol as 665
 import 660

import as symbol	663	lines changign elevation	482
open	661	select elevated objects	482
open new project by importing	662	Elevation window	875
DXF files	660	Ellipses	101
calculate scale	253	Empty lines between Terminal Rows	553
export	663	EN1861	226
export project as	663	EN60204-1,14.2.1, standard	246
export symbol as	665	EN60439-3	469
import	660	EN60617, symbol folder	226
import as symbol	663	EN61346	335
open	661	signal names according to	210
open new project by importing	662	Enter Wire Numbers Manually	399
E		Error	
Edit		general protection error	49
templates	777	Even pages	327
Edit =>		Excel	
Align	66	integration with, survey	666
Convert to lines	101	place symbols via lists	674
Copy to Clipboard	523	PLC I/O addressing via	435
Data tags	283	Explorer window	301, 839
Find	316	displaying and hiding information	302
Find next	317	move	26
Insert special	670	Selecting a symbol on a page	304
Join signal	437, 438	Export	
Links	673	DWG and DXF files	663
Next	188	PLC I/O data to Excel	435
Paste	255	project as DWG or DXF file	663
Pinswop	191	symbol as DWG or DXF file	665
Previous	188	Extend line button	90
Replace	318	Extended lines	90
Scale	298	Extensions	
Select all =>		of list files	658
All on layer	481	F	
All on page	57	FABRICATE (Show all)	576
Space	67, 631	Fabricate choice	
Variables	261	in Database menu	613
Without wire number	390	Factor, ellipses	101
Editing symbols for a building	476	Favorites	786
EIB, symbol folder	226	Field headings	655
Electrical installation	469	File =>	
Electrical installations		Close	518
in different elevations	475	Close and return to PC schematic project	671
Electrical lines	78	Export to =>	
Electrical potentials	203	DWG / DXF	663
automatic insertion of	81	PDF	331
on lower part of page	82	Modules	782
on upper part of page	82	New	38, 512, 772
settings for	83	New =>	
Electrical symbols		PC schematic project file	814
components having more	593	Open Unit	770
create automatically	199	Print	326
box symbols	200	Print =>	
circle symbols	200	Select pages =>	481
connection points	200	Save unit as	769
from database	201	Send to	490
place based on placed mechanical symbols	639	File format	655
Elements of a project	489	File type registration, dialog box	813
Elevation	482, 875	Files	875

commasep. files	
place symbols via	674
DWG/DXF	660
file names	
in database fields	622
from Object Lister	311
lists as files	649
pindata files	599
registering PC SCHEMATIC filtypes	813
wire number	401
Files =>	
Modules =>	
Scripts	7
Fill character	111
Filled areas	75
Find Next	317
Find Whole Text Only	317
Fixed digits	395
Fixed text direction	125
Flat buttons	838
FLOWCHAR, symbol folder	226
Folders	875
change symbol in	731
delete symbol from	731
most used	841
symbol folders	225
Follow connected button	91, 739
Follow connected lines	90
Font	123
Foreign languages	787
Format file	650
Formulas	
insert	531
projects	511
Frame	124
Free texts	102, 875
change	764
Ftp	
links in database fields	622
Full circle	100
Function aspects	335, 338
Functions =>	
Auto command change	57
Auto Name	176
Centre drawing	510
Component Grouping	216, 557
Coordinates	93
Count down	121
Count up	121
Database =>	
Find Article	154
Find Type	155
Insert in Database	591
Update from Database	171
Design Check	320, 323
Design Check =>	
Used more than once	
deselect	324
Generate isometry	510

Measurement	641
Mechanical load	629
Object Lister	307
Place Accessory	225
Place available Symbols	242
Replace symbol	193
Select name	238
Show available	232
Show available again	153, 232
Show conducting	79
Show signal	209
Special Functions =>	
Convert Branches to Dots	98
Convert Dots to Branches	98
Convert to Branches	99
Create CurrentPath No on all Symb	383
Exchange Article with Alt. Article on all Symbols	611
Prefix all symbol names with -	352, 838
Set all articles as approved in the database	617
Set unreferenced cable symbols invisible	225
Set unreferenced signal symbols invisible	210
Transfer data from PLCref. to PLCsymb.	430
Transfer data from PLCsymb. to PLCref.	430
Transfer Proj. Texts to Auto Compl. func.	104
Update references	372, 381
View Jumper Links	87
Wire numbers	389

G

General protection error	49
General purpose texts	102
General, tab	544
Generation names	383
German version	
PC SCHEMATIC Viewer	680
Grafisk klemplan, værktøj	781
Graphical terminal plans, tool	
Tools	
Grappical terminal plans	781
Grid	47, 875
Grid display	836
Ground plan	471
place symbols on	475
with elevation	482
Ground plan, page type	508
Group Number	560
Grouping	
components	556
layers	479
Guess	
wire numbers	392

H

Half circle	100
Heat loss calculation	
acc. to EN60439-3	469
Height	123
Help	35
Help =>	

Check for updates	851	Insert Picture	297
Http		Insert Potential	81
//www.pcschematic.com	18	Insert Reference Frame	354
Help frame	39, 839	Insert sub drawing	95
Help objects color	793	Insert new	494
Hints	838, 875	symbols or connection names	295
Home view		Insertion points	279
set User Home view	45	Install modules	782
HOMEDIR	845	Installation	
Horizontal reference	129	electrical installation	469
Horizontal, tab	380	pickmenu	475
Hot-line	8	Installations	
House drawing	471	LV installations	7
Http		instdemo.pro, demo project	475
links in database fields	622	Intelligent	
Hungarian version		copying of areas	245
PC SCHEMATIC Viewer	680	reference designations	247
Hydraulics		Interface	
place symbols via lists	674	database	608
Hydraulics, symbols	226	Internet	
I		automatic information and update	784
I/O lists		Interval in minutes	849
read	432	Invert text	116
I/O symbols		Invisible	478
create PLC I/O symbols	725	areas	70
IEC750 standard	335	circles	70
IECINST, symbol folder	226	lines	70
IECPNEU, symbol folder	226	objects	69
Ignored, page function	497	Sub drawings	71
Image		symbols	69
insert	66	texts	70
scale	298	ISO	
Image functions	42	paper size	509
Import		Isometric, page type	508
DWG and DXF files	660	Isometrical drawing	510
as page in project	661	Italic text	125
PLC I/O data from Excel	436	J	
Include Additional from Database	545	JIS	
Include cable wires	391	paper size	509
Include Lines with article data	545	Join symbols	
Index		define	741
on-line manual	37	Jokers	550
Indexed variables	872	Jumper links	86
Indexing	816	in lists and terminal plans	87
Information		K	
automatic information	784	Keep page zoom	44, 838
Insert		Keep zoom	44, 838
data fields	747	Komponenter	
image	66	rename selected	
list	538	selecting symbols	291
logo	527	L	
page for mechanical drawing	628	Language	850
unit drawing	568	Layers	478
Insert - in front of symbol names	352, 838	and drawing headers	480
Insert =>		change using the mouse	480
Insert data field	110, 855	grouping layers	479
Insert object	668, 669		

- print page with 481
- Left tool bar 40
- Left toolbar 875
- License key 850
- Licenses
 - SuperPro monitor 799
 - tanking licenses 794
 - transfer licenses by tanking 795
 - transfer licenses via license files 796
- Lines 53, 72
 - article data 79
 - Automatic line drawing 90
 - automatic wire numbering 389
 - by measures 93
 - changing elevation 482
 - color 76
 - curved 74
 - distance 76
 - electrical 78
 - extended 90
 - follow connected lines 90
 - in symbols 699
 - invisible 70
 - jumper links 86
 - Object Lister 307
 - rectangles 75
 - round 74
 - router 90
 - select elevated 482
 - slanted 74
 - straight 73
 - toolbar 73
 - transfer article data 62
 - trim 92
 - type 75
 - types for ground plans 473
 - width 76
 - wire numbers file
 - with reference designations 402
 - without wire number, deselect 390
- Links
 - edit 673
 - in database fields 622
- List files
 - buttons in the dialog box 653
 - create 650
 - criteria 654
 - example 657
 - extensions 658
 - field headings 655
 - file format 655
 - matching the system you export to 654
 - reference designations in 656
 - the contents 654
 - the file name 654
 - the format file 650
- List of drawing headers 750
- List of used objects 316

- List setup
 - terminals- cables- and PLC lists 553
- List symbols 762
- Lists
 - add to the list of drawing headers 758
 - as files 649
 - columns 546
 - company logo on 751
 - connections lists 202
 - wire numbers 404
 - create 752
 - criteria 549
 - data symbols on 736
 - delete page with list on 766
 - diagram page w. list 762
 - for each Higher level or Location 364
 - insert 538
 - jumper links in 87
 - list symbols 762
 - missing data fields 534
 - mounting lists 658
 - order in 556, 561
 - place symbols via lists 674
 - predefined 763
 - references to location of mechanical symbols .
636
 - remove form page 766
 - settings 542
 - symbols
 - sub names in lists 166
 - unit parts lists 767
 - update 541
 - using reference designations 364
 - XML format 652
- Lists =>
 - Connections List File 202
 - Connectionslist for file 403
 - List Settings 555
 - List settings 543, 544, 553
 - Parts List File 650, 657
 - Read Components List =>
 - Create from diagram page 677
 - Create from mechanical page 677
 - Load from file 674
 - Update all lists 541
 - Update lists 543
 - Update parts lists 541
 - Update Table of Contents 497
 - Update Unit Drawing 769
 - Wire number file 401
- Load defaults button 380
- Load previous project 850
- Load selected 629
- Load standard button 128
- Location aspects 335, 338
- Lock
 - after system break down 806
 - delete lock information manually 806

project files	806	MISC, symbol folder	226
project in use	806	Missing data fields	534
text types	119	Modules	781
Logo		install and uninstall	782
insert	527	start a module	783
loop diagrams	7	update and re-install	783
LV installations	7	which can you install	781
M		Monitors	
Mailto		PC with two monitors	26
links in database fields	622	Mounting correct drawing	95
Main reference, check box	380	activate	96
Manual	35	bend	96
Survey of	15	convert from previous verisons	99
Manual placing of wire numbers	398	convert to automatically	98
Mapping		Mounting lists	658
database fields	608	Mounting mode	392
Margin	329	Mouse	
Master reference, symbol type	707	change layer	480
Match Case	317	Favorites	786
Maximize	875	mouse pointer	26
MCS	800	Mouse Chasing System	8, 800
MDAC	623	automatic blocking of mouse clicks	802
Measurement	641	block for clicking on a function	801
free measurement	643	disable	802
reference points and measurement	643	Mouse pointer	876
rotate	643	Move	
Measurement objects		Explorer window	26
layers	645	origo	471
Measures		reference points	745
absolute	93	selected objects	58
polar	93	Survey window	27
relative	93	symbol	174
Measures, drawing by	93	Symbol names when move	384
Mechanical drawing		Symbol pickmenu	27
insert page	628	Toolbars	28
Mechanical load		Move button	58
from database	628	MSSQL	623
references to mechanical symbols in lists	636	Multi layer terminals	215
Mechanical symbols	201, 603, 612	on diagram pages	216
create automatically	199, 201	on mechanical pages	217
box symbols	200	Multi symbols	714
circle symbols	200	fetch	715
connection points	200	Multi-level terminals	600
from database	201	Multisignal, symbol type	707
start by drawing mechanically	639	N	
MECTYPE	593	Name format	382, 852
Menu bar	23, 38, 876	Name format types	
Menus	23, 876	change	384
choose database menu	617	Native drivers	8
direct access	48	Navigation	
Microsoft Excel		between symbols for component	170
integration with, survey	666	Negate	116
Microsoft SQL Server	8	Negate text	116
Minimize	876	NET	845
Mirror		Net	
symbols	176	tanking licenses	794
Mirror symbol horizontally button	176	Net router	837
Mirror symbol vertically button	176		

Networks
 sub drawings in 256
 New document button 38
 New versions 8
 No dot generation 719
 No visible 478
 Non conducting symbol 876
 Non conducting, symbol type 707
 Normal, page function 497
 Normal, symbol status 594
 Normal, symbol type 706
 Norms
 page norms 509
 Not yet written (New) 401
 NP 876
 Number 380
 Numbering order 393

O
 Object Lister 316
 check project texts 314
 counting on selected objects 310
 editing PLC I/O data 313
 print 316
 printing from 311
 save as file 311
 selecting objects in 309
 show used and count 313
 typing in PLC I/O data 426
 Object properties 63
 change on existing object 63
 transfer to more objects 63
 Object selection
 Cross hairs 793
 Object selection color 793
 Objects 53, 876
 align 66
 already placed 54
 invisible 69
 new/draw/place 54
 select different types of 244
 select elevated 482
 space 67
 survey 316
 transfer data between 62
 Obsolete components
 in Database menu 614
 ODBC interface 8, 623
 Odd pages 327
 OK button 25
 OLE automation 7
 OLE objects
 edit links 673
 properties 672
 On all pages 536
 On this page 536
 One line diagrams
 connection names 408

On-line manual 35
 adapt to screen 36
 buttons 36
 index 37
 index-finger 37
 open 35
 printing from 37
 return to PCschematic 37
 Table of Contents 36
 zoom 36
 Only jumper links 545
 Only show text on data 111
 Open
 DWG and DXF files 661
 new project by clicking symbol 524
 Open dialog box 514
 Open files from Tools menu 679
 Open, symbol status 594
 Open, symbol type 706
 Operator 549
 Oracle 8, 623
 Order
 in lists 556, 561
 Origo
 move 471
 Ortho 45 837
 Other programs
 copy to 523
 Output 326
 Overall planning 685
 Oversigtsprojekter 524
 Overview project 524
 Overview projects 524

P
 Page
 page setup 506
 transfer to other pages 506
 Page data 834
 data fields 110, 689
 add 692
 in standard project 687
 type in 114
 Page Data, data field type 746
 Page format 509
 Page function 843
 Page history buttons 492
 Page menu 493
 copying pages 500
 naming 500
 symbol names when changes 385
 Page numbering
 Automatic renumbering of pages 832
 Page settings 38, 471
 Page setup 843
 Page shift between Terminal Rows 553
 Page tabs 838, 876
 Page type 843

Page types	508	Place missing	393
Page zoom		Place missing and renumber	393
keep	44	Planning	685
Pages	491	PLC	418
copy	500	addressing directly from database	419
create new	495	base addressing	419
create page templates	772	choose format file for I/O file	432
drag between projects	517	contents of I/O file	433
insert drawing header	532	create PLC I/O symbols	725
Object Lister	307	create PLC ref. symbols	728
page history	492	Data on I/O symbol connection points	425
Page menu	493	displaying changes in project acc. to I/O file	434
reference designations	353	I/O addressing via Excel	435
displaying reference designation on page	354	I/O data	
remove list form	766	edit	313
Pages and layers	545	jump PLC ref. symb. / PLC I/O symb.	427
Panel Builder	469	PLC reference symbols	418
Panel Builder, tool	782	more than one	419
Panel designer	469	read I/O lists	432
Panel documentation		re-addressing	423
automatic	469	specify I/O file filename	433
panel documentation		survey over functions	417
automatic	469	symbol folder	227
Panels	469	symbol status	594
Paper and margin setup	329	symbol type	707
Paper size	509, 843	symbols	227
Parking	79, 876	synchronize PLC data	430
Parts list, page function	497	text links	136
Parts lists	111	typing in I/O data via Object Lister	426
files	649	update PLC projects from previous versions	430
order in	561	workflow	
prices	613	for PLC's not in the database	428
Parts/Components list, data field type	746	with PLC's from database	420
Password		working with PLC's	418
projects	805	PLC =>	
Path	876	PLC List File	435
PC		Read PLC I/O List	432
with two monitors	26	PLC functions	
PC SCHEMATIC		Apply addresses	424
registering filetypes	813	PLC I/O	
PC SCHEMATIC Loop	7	Object Lister	307
PC SCHEMATIC Powerdistribution	7	PLC list, data field type	746
PCSTYPE	593	PLC list, page function	497
Pencil button	72	PLC lists	
Pickmenu	39, 140	and data fields	761
attach data to symbol in pickmenu	180	create	760
enter database from pickmenu	142	data fields	111
for installations and pneumatics	475	list setup	553
place symbol in pickmenu using database	181	PLCreference	
place symbol with data	178	symbol type	707
place wire number symbol in	398	Pneumatics	
Pin names	612	pickmenu	475
PINDATA	593	Pneumatics symbols	226
Pindata	598	connection points with same name	324
Pindata files	599	place symbols via lists	674
Place		Pointer	836
available Symbols	242	Pointer/Screen	836
symbols	156	Polar	93

Polish characters	787, 850	controlling texts from project definition file	281
Polish version		copy areas between	518
PC SCHEMATIC Viewer	680	copy project into project	513
Position of name when placing symbols	131	copying areas between	255
Posno1	769	create project templates	774
Potential numbering	391	from existing project	776
Potentials	203	creating project templates	686
automatic insertion of	81	drag pages between projects	517
on lower part of page	82	elements of	489
on upper part of page	82	export as DWG or DXF file	663
settings for	83	insert logo	527
Powerdistribution	7	insert sub drawing	259
Precede page number	395	jump between several projects	305
Predefined drawing headers	751	lock file	806
Predefined lists	763	lock project in use	806
Prefix	395	open from symbol	521
Preparations	685	overview project / click symbols to open projects	524
Previous versions		524	
update PLC projects from	430	overview projects	524
Price 1	611	password	805
Primary	546	planning	685
Print	326	same project in two versions	522
batch printing	823	searching for texts via Windows	818
on net installation	823	send by e-mail	490
on stand-alone installation	823	simultaneous projects	516
chapters and dividers	328	survey of	522
from Object Lister	311	zip	825
object lists	316	Projekter	
page with layers	481	oversigtsprojekter	524
zoom	329	Properties =>	
Print pages button	326	Project data	814
Print test page	330	Protected	399
Print this page button	326		
Printer help frame	839	Q	
Printer, tab	330	Qualified guess	392
Product choice		Quarter circle	100
in Database menu	613	Question mark	550
Program settings	38		
Program toolbar	39, 877	R	
Programmable barcode scanners	624	Re-addressing	
Programs		PLCs	423
copy to other	523	Reading direction	130
Project data	831	Recovery information	849
data fields	110, 689	Re-creating files	849
add	692	Rectangle button	75, 473
in standard project	687	Rectangles	75
type in	114	with relative coordinates	94
Project Data, data field type	746	Red arrow	111
Project definition file		Redraw button	45
controlling project texts from	281	Reducing repetitive strain injuries	800
Project menu		Reference cross	380
Windows Explorer as	334	Reference crosses	129
Project oriented program	489	about	376
Project settings	38	choose	377
Project templates		jump to reference from	368
create	686	Reference designations	335
Projects		data field for specifying	363
about	489	defining	338
compressed saving	825		

displaying on page	354	drawing header	535
for a page	353	Replace All	318
in list files	656	Replace symbol texts	318
insert a data field	354	Replace symbols	193
insert auto. on signal symbols	210	connection points	195
prefix all symbol names with -	352	text properties	195
symbol names on pages with assigned	353	Reports	
symbols	349	loop reports	7
wire numbers file		Reset group numbers button	558
with reference designations	402	Restart on page shift	395
Reference frames		Restarting the program	49
Object Lister	307	Reverse direction	224
Reference letter	612	Right-click	26, 877
Reference point button	739	select	55
Reference points	839, 877	short-cuts	789
move	699, 745	symbols	176
selection	757	Rotate	
Reference symbol position	380	areas	253
Reference symbols		selected object	59
PLC reference symbols	418	texts	124
Reference system		Rotate button	59
default settings	380	Round buttons	24
setup	379	Round lines	74
Reference system setup, dialog box	379	Round lines button	74
Reference text	207	Router	89
Reference, symbol type	707	Router =>	
References	835	Active	89
appearance	372	Rows	
automatic update of ref.	367	copy rows of data fields	756
implicit references on cable wires	213	Russian characters	787, 850
implicit references on terminals	211	Russian version	
jump to ref. from ref. cross/contact mirror	368	PC SCHEMATIC Viewer	680
jump to reference	367	S	
on symbols	369	S Article no	107
symbol type and reference	370	S Function	107
Refresh	45	S Name	107
Register command	824	S Type	107
Registration		Same number through terminals	391
of PC SCHEMATIC Automation	824	Save	
of PC SCHEMATIC filetypes	813	symbol	705
Re-install modules	783	Save as default button	128
Relative	93	Save with backup	840
Relay, symbol status	594	Scale	
Relay, symbol type	706	calculate scale	253
Remove		image	298
company logo	764	scale factor	844
list from page	766	symbols	176, 708
Remove blocking	802	texts in areas	254
Remove existing	394	Screen functions	42
Remove existing and replace	394	Scripts	
Rename	479	module	782
selected symbols	289, 290	Scroll	877
symbols from Symbol menu	710	Search	
Renumber existing	394	in database	575
Repetitive strain injuries		Searchkeys	
block for clicking on a function	801	in database	577
Repetitive strain injuries, reducing	8, 800	Secondary	546
Replace			

Select	
area in all layers	481
different types of objects	244
elements in lists	24
elevated objects	482
everything on a layer	481
everything on a page	57
more objects of same kind	55
object on different layer	481
objects	55
on different layers	481
right-click	55
several objects using mouse	56
symbol by right-click	176
symbol from database	152
symbol from pickmenu	140
symbol from the symbol menu	144
symbol using k-key	153
Select all layers	481
Select Component	242
Select database file	847
Select pages, tab	327
Select set of characters	850
Selected objects	
copy	58
delete	59
move	58
rotate	59
working with	58
Selection	
reference points	757
Semi ISOmetric, page type	508
Semicolon	598
Send project by e-mail	490
Servers	850
Settings	38, 829
data base	608
lists	542
Settings =>	
Cursor/Screen =>	
Colors	792
Database =>	847
Database settings =>	
Component data	577
Database setup =>	608
Component data	577
Component search	614
Select database file	623
Directories	522, 840
Directories =>	
Database	599
Page data =>	114, 834
Reference designations	338, 353
References	379, 636
With drawing header	532
Page setup	506, 843
grid	47
page function	498
page type	508
snap	47
Page setup =>	
Reading direction	130
Pointer/Screen	44, 107, 836
Ask for names	157
grid	47
Pointer/Screen =>	
Shortcuts	790
Project data	831
Project data =>	114
Password	805
Reference designations	338
Settings	829
System	849
System =>	
Compressed saving	825
File type registration	813
License Tank	795
SuperPro monitor	799
Text/Symbol defaults	852
Drawing headers	750
References	372
Signal name	741
Signal symbols	740
symbol data fields	733
Text defaults	382
Tools	845
Settings => Page setup	
Paper size	509
Setup, tab	329
Several projects	
jump between projects	305
Short-cut keys	788
learning while working	802
user defined	790
Show	
symbol texts	705
Show all	207
Show available	232
Show available window	234
Show defaults button	128, 380
Show directory alias	841
Show from/to	207
Show layer settings	328
Show list of objects	341
Show message in (ms)	837
Show net	202
Show popup help	838
Show popup information	838
Show scrollbars	838
Show used	315
Show used and count	313
Signal buses	437
Signal names	
in single line diagrams	407
Signal symbols	203, 877
create	738
frequently used	740
frequently used names of	741

insert reference designations automatically	210	Straight lines	73
invisible	210	Straight lines button	73
place	204	Straighten lines	837
place directly	208	Strategy	685
place on parked line	208	Sub drawings	256
variant names	209	automatic placing	264
Signals		create	267
Object Lister	307	by copying	268
Simultaneous		create variablelist	283
projects	516	insert on project page	259
Single line diagrams	407	insert variables	281
connection names	408	insertion points	279
signal names	407	invisible areas	71
symbol names	408	networks	256
symbols with states	408	Sub names	
Single line symbols		on symbols	165
connection names for	598	SuperPro monitor	799
Slanted lines	74	Support	8
Slanted lines button	74	Support, symbol type	707
Snap	47, 877	Survey	
Snap button	47	of projects	522
Sort by name	68, 631	of the manual	15
Sorting order		of used objects	316
of components	558	Survey window	40, 839, 878
Space	67	move/dock	27
Spares pr. Page	395	Switch, symbol status	594
Special characters	105, 116	Switch, symbol type	706
Specify character set	787	Symbol article data =>	
Specify symbol status	603	Unit	568
SQL Server	8	Symbol article data dialog box	107
Squared grid color	792	Symbol Data Field, data field type	746
Squared grid-10 color	792	Symbol Data Fields	
Squares	24	data fields	111
Standard characters	787	Symbol defaults	852
Standard projects	511	Symbol definition	598
convert to templates	776	Symbol documentation, tool	781
copy into project	513	Symbol folders	225
Standard/Project file	514	Symbol manipulator	716
Standard/Project file, page function	498	Symbol menu	
Standards		bookmarks	148
EN1861	226	create symbol / edit existing	151
EN60204-1,14.2.1	246	enter the database from	151
EN60617	226	representation of symbols	146
EN61346	226, 335	search in	147
EN81714-2	226	select library via alias	146
IEC/EN60617	226	select library via folders	146
IEC750	335	show project symbols	146
ISO1219	226	Windows Explorer as	334
page standards	509	Symbol pickmenu	39, 839
Start number	395	dock/move	27
Start value	380	Symbol reference	129
Starting functions	821	Symbol settings button	739
load project with page open	821	Symbol texts	
load project with symbol selected	821	replace	318
loading specific pages from project	821	Symbol toolbar	
States		select symbol type in area on screen	176
symbols with states	167	Symbols	53, 137
Status bar	877	accessory symbols	225

align	66	names regain adaptability	385
alternative electrical symbols	594	names when changes in Page menu	385
alternative symbols		names when move/copy	384
PINDATA	596	navigation between	170
alternative symbols for PLC	595	Object Lister	307
attach unit drawing to	568	open project by clicking symbol	524
automatic line drawing/router	89	open project from	521
busbar symbols	724	pickmenu	
cable symbols	218	attach data to symbol in pickmenu	180
invisible	225	enter database using pickmenu	142
cables with color codes	221	place symbol in pickmenu using database	181
change in folder	731	place symbol with data	178
change names in Area	185	symbols with states	143
change names in Area, naming order	186	place and name	156
change type	731	place available symbols	242
changing symbol data for Component	242	place on ground plan	475
component having several electrical	593	place via lists	674
component level functions	228	PLC	
connection names for single line symbols	598	create PLC I/O symbols	725
connection points	187	create PLC ref. symbols	728
create	696	PLC symbols, working with	418
default text settings	716	position of name when placing symbols	131
create automatically	199	reference designations	349
box symbols	200	references on	369
circle symbols	200	rename	
connection points	200	when copying areas	245
from database	201	rename from Symbol menu	710
create data symbol	732	rename selected	289
create symbol data fields	733	direction for renaming	292
current path numbers	382	selecting symbols	291
data symbols		start number	293
with predefined values	736	terminals	293
data symbols and lists	736	which names are renamed	290
Database		replace symbols	193
update symbols from	848	save	705
delete	173	scale	176, 708
delete from folder	731	select	
Edit symbol mode		from database	152
default text settings	716	from pickmenu	140
editing for a building	476	from symbol menu	144
enter database using pickmenu	142	right-click	176
export as DWG or DXF file	665	using k-key	153
fetching symbols for component from database	230	select elevated	482
insert new symbol names	295	select symbol type in area on screen	176
invisible	69	select using Explorer window	304
join symbols, define	741	show available	232
lines in	699	show net	202
list symbols	762	show texts	705
load another symbol into	716	signal busses	437
mechanical	201, 603	signal symbols	203
create automatically	201	invisible	210
mirror	176	space	67
move	174	specify symbol status	603
move reference points	699	states	167
multi symbols	714	in single line diagrams	408
name format	382	sub names	165
names on pages with reference designations	353	functionally understood as	165
		included in lists	166
		specified from database	166
		specify	165

what is	165	Terminal plans	
survey of symbols with same Symbol Name	238	jumper links in	87
symbol documentation	781	Terminal symbols	214
symbol manipulator	716	create	717
symbol names		implicit references on terminals	211
in single line diagrams	408	Terminal, symbol status	594
position of name when placing symbols	131	Terminal, symbol type	707
symbol type and reference	370	Terminals	
terminals		graphical terminal plans, tool	781
multi layer	215	multi layer	215
on diagram pages	216	on diagram pages	216
on mechanical pages	217	on mechanical pages	217
texts connected to	107	multi-level terminals	600
toolbar	176	rename	293
transfer article data	62	Terminals list, data field type	111, 746
type in Article Number directly	154	Terminals list, page function	497
type in Symbol Type directly	155	Terminals lists	
types	706	create	759
update		list setup	553
from database	171	order in	561
when changing article number	171	Test page, print	330
update from database	231	Text	111
wire number	395	Text defaults	852
with extra connection points	715	Text fields	
Symbols button	139	place	745
Symbols text		Text link	
S Article no	107	dialog box	135
S Function	107	Text links	132
S name	107	create from Symbols command	133
S type	107	create from Texts command	132
SYSDIR	845	delete	136
System	849	edit	135
System break down		in PLC I/O lists	136
lock after	806	indication of linked texts	132
System breakdown	849	what is a text link	132
System Data		Text properties	
data fields	110	specify	122
System Data, data field type	746	Text properties button	122
T		Text translator, tool	782
Tab key	25	Text/Symbol defaults	852
Table of Contents		Texts	53, 102
create	759	align	66
data fields	111	alignment	124
On-line manual	36	appearance	122
Table of Contents, data field type	746	auto completion	103
Table of Contents, page function	497	auto text	
Templates	771	delete texts	104
create from standard projects	776	bold	125
delete	777	change	110
edit	777	change free texts	764
insertion points	279	change in cross hair	116
on networks	771	check project texts	314
page templates		connected to symbol	107
create	772	controlling from project definition file	281
project templates		counting on texts	120
create	774	counting on texts in areas	120
from existing project	776	data fields	110
template projects	511	reference designations	363

- default text settings
 - in Edit symbol mode 716
- displayed in the project (overall setting) .. 127
- fixed text direction 125
- frame 124
- free texts
 - default properties 106
- height 123
- in more than one line 116
- invert/negate 116
- invisible 70
- italics 125
- linking texts 132
- lock text types 119
- Object Lister 307
- rotate 124
- scaling texts in areas 254
- searching for texts via Windows 818
- space 67
- special characters 116
- specify text properties 122
- text links 132
 - create from Symbols command 133
 - create from Texts command 132
 - delete 136
 - edit 135
 - in PLC I/O lists 136
 - indication of linked texts 132
 - what is a text link 132
- text translator, tool 782
- toolbar 103
- type in 103
- underline 117
- underlined 125
- visible text 125
- width 123
- Texts button 102
- Tile horizontal 516
- Tile vertical 516
- Time-out 850
- Toolbars
 - arcs/circles 100
 - dock/move 28
 - lines 73
 - symbols 176
 - texts 103
- Tools => 845
 - Convert files from ver. 11 or older 12
 - Create variable list 281, 282, 283
 - Database 608
 - Mouse Chasing System 800
 - Rename symbols 289
 - Scripts 7
 - Show textfile 679
 - Singleline diagram 409
- Tools menu
 - open files from 679
- Top to bottom 438
- Transfer Group Number button 560

- Transfer information
 - from database 573
- Translate
 - database fields 608
- Translator
 - text translator tool 782
- Trim line 92
- Type in texts 103
- Types
 - line types 75
 - symbol types 706

U

- Unambiguous drawing 392
- Underline 117
- Underline text 117
- Underlined text 125
- Undo 60
- Undo button 60
- Uninstall modules 782
- Unit Drawing, page function 497
- Unit drawings
 - attach to symbol 568
 - correct 770
 - create 767
 - create using drawing of entire component 769
 - insert 568
 - open 522
 - save 769
- Unit of measurement 837
- Unit parts lists 767
 - assign article number to 768
 - correct 770
 - without unit drawings 769
- Unregister command 824
- Update 8
 - automatic information 784
 - modules 783
- Update lists 541
- Update, button 381
- User defined short-cut keys 790
- User Home view
 - set 45
- User name 849
- USERLIB, symbol folder 227

V

- Variables 871
 - create variablelist 283
 - indexed variables 872
 - insert 281
 - insertion with default value 871
 - normal variable 871
 - possible control via var. in symbol names 872
 - that inserts a value in a text 871
 - with default value 871
- Variant names
 - on signal symbols 209

Vertical reference	129	place symbol in pickmenu	398
Vertical, tab	380	place symbol manually	399
View =>		Wire numbers to file	
Activate Pickmenu	32	with reference designations	402
Clean screen	46	With reference	206
Go to page	491	With reference, symbol status	594
Goto symbol	170	With reference, symbol type	707
Home view	44	Without wire number	390
Page history back	492	Word	523
Page history forward	492	Work area	38, 40, 878
Previous zoom	44	Wrap text	111
Redraw	45	X	
Select page	30, 358, 493	X position (mm)	380
Set User Home view	45	x,y position	636
Show object below cursor	60	XML	
Show texts	705	lists in XML format	652
Survey window	46	Y	
Zoom	43	Y position (mm)	380
Zoom All	43	Z	
DWG/DXF	660	Zip	
Zoom in	43	advantages by zipping	825
Zoom out	43	on zipping	825
Visible text	125	project files	825
W		Zoom	878
Width	111, 123	keep page zoom	44
Width, lines	76	previous zoom	44
Wildcards	550	print	329
Window		Zoom button	42
several windows open	520	Zoom to page button	44
Window =>			
Arrange Icons	516		
Close All	516		
Tile	517		
Tile horizontal	518		
Windows	23		
change	25		
indexing service	816		
Windows Explorer			
as Project menu	334		
as Symbol menu	334		
Wire			
add to cable	223		
remove from cable	223		
Wire number, symbol type	707		
Wire numbers			
auto. incr. on manually placed wire no's	399		
automatic	389		
connections lists	404		
Excl. signals, tab	397		
fetch symbol	398		
files	401		
Format	395		
Functions	391		
lines without wire numbers	390		
manual placing	398		
Number symbol, tab	395		
numbering order	393		



AUTOMATION

PART 2

15/3-2013

TABLE OF CONTENTS

PART 1

PREFACE	7
UNICODE	10
SURVEY	13
QUICK SURVEY OF THE MANUAL	15
GETTING STARTED	21
WINDOWS	23
KEYBOARD SHORT-CUTS	29
THE ON-LINE MANUAL	35
THE WORK AREA	38
SCREEN / IMAGE FUNCTIONS	42
BASIC DRAWING FACILITIES	51
ON DRAWING OBJECTS	53
LINES	72
ARCS/CIRCLES	100
TEXTS	102
SYMBOLS	137
FUNCTIONS ON COMPONENT LEVEL	228
AREAS	244
SUB DRAWINGS	256
RENAME SYMBOLS	289
INSERTING IMAGES	297
SURVEY AND REFERENCE DESIGNATIONS	299
THE EXPLORER WINDOW: THE PROJECTS TAB	301
THE OBJECT LISTER	306
FIND AND REPLACE	316
DESIGN CHECK	320
PRINTING	326
WINDOWS EXPLORER AS SYMBOL/PROJECT MENU	334
REFERENCE DESIGNATIONS	335
REFERENCES	365
AUTOMATICALLY UPDATED REFERENCES	367

CONTACT MIRRORS AND REFERENCE CROSSES	374
CURRENT PATH NUMBERS / REFERENCE SYSTEM	379
WIRE NUMBERS	387
AUTOMATIC WIRE NUMBERING	389
MANUAL PLACING OF WIRE NUMBERS.....	398
WIRE NUMBER FILES.....	401
SINGLE LINE DIAGRAMS.....	405
SINGLE LINE DIAGRAMS.....	407
AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS	409
PLC FUNCTIONS.....	415
SURVEY OF PLC FUNCTIONS	417
WORKING WITH PLCS	418
READING PLC I/O LISTS	432
PLC I/O ADDRESSING VIA EXCEL	435
CONNECTION SYMBOL FOR SIGNAL BUSES.....	437
INDEX FOR PART 1 AND PART 2	443

PART 2

ELECTRICAL INSTALLATION DRAWING.....	469
CREATING A GROUND PLAN.....	471
LAYERS, ELEVATION AND SCALING.....	477
TIPS FOR INSTALLATION DRAWING.....	484
WORKING WITH PROJECTS.....	487
ON PROJECTS.....	489
PAGES AND CHAPTERS.....	491
PROJECT TEMPLATES AND STANDARD PROJECTS.....	511
COPYING FULL PROJECTS INTO OTHER PROJECTS.....	513
SEVERAL PROJECTS SIMULTANEOUSLY.....	516
OVERVIEW PROJECTS.....	524
YOU HAVE NOW ESTABLISHED A CONNECTION BETWEEN THE SYM- BOL FOR THE ELECTRICAL PANEL AND THE PROJECT THAT IT IS DOCUMENTED IN.....	525
INSERTING LOGOS IN PROJECTS.....	527
LISTS.....	529
INSERT DRAWING HEADERS/FORMULAS.....	531
LIST SETTINGS.....	542
COMPONENT GROUPING.....	556
INSERTING UNIT DRAWINGS.....	568
DATABASES.....	569
USING THE DATABASE.....	571
CREATING COMPONENTS IN THE DATABASE.....	589
DATABASE SETTINGS.....	608
CHOOSING A DATABASE.....	623
PROGRAMMABLE BARCODE SCANNERS.....	624
MECHANICAL SYMBOLS.....	625
MECHANICAL SYMBOLS - OVERVIEW.....	627
PLACING MECHANICAL SYMBOLS VIA THE DATABASE.....	628
FROM MECHANICAL TO ELECTRICAL DRAWING.....	639
MEASUREMENT OBJECTS.....	641
EXPORT / IMPORT.....	647
EXPORTING LIST FILES.....	649
DWG AND DXF FILES.....	660
INTEGRATION WITH MICROSOFT EXCEL.....	666
INSERTING FILES FROM OTHER PROGRAMS / OLE.....	667
PLACING SYMBOLS VIA LISTS.....	674
OPENING FILES FROM THE TOOLS MENU.....	679

PC SCHEMATIC VIEWER.....	680
CREATE SYMBOLS, LISTS AND PROJECTS	683
PLANNING AHEAD.....	685
PAGE DATA AND PROJECT DATA DATAFIELDS	689
CREATING SYMBOLS.....	696
CREATING DATA SYMBOLS.....	732
CREATING SIGNAL SYMBOLS.....	738
CREATING DRAWING HEADERS	742
CREATING LISTS	752
CREATING UNIT DRAWINGS	767
CREATING TEMPLATES.....	771
UPDATING, MODULES AND LICENSES	779
MODULES	781
UPDATE AND AUTOMATIC INFORMATION.....	784
ADAPTING THE SYSTEM TO YOUR NEEDS.....	786
SHORT-CUT KEYS	788
COLORS.....	792
TANKING LICENSES.....	794
PREVENTING REPETITIVE STRAIN INJURIES.....	800
ACCESS CODES, REGISTRATION E.T.C.....	803
PASSWORDS ON PROJECTS.....	805
LOCKING PROJECT FILES	806
REVISION CONTROL	808
REGISTRATION OF PC SCHEMATIC FILE TYPES	813
INDEXING.....	816
STARTING FUNCTIONS, BATCH PRINTING AND DDE.....	821
COMPRESSED SAVING / ZIPPING PROJECTS	825
APPENDICES	827
SETTINGS	829
DATAFIELDS	855
THE DATABASE FIELD PCSTYPE.....	866
THE DATABASE FIELD PINDATA.....	868
THE DATABASE FIELD PINDATA: COUNTING	869
DICTIONARY.....	873
INDEX FOR PART 1 AND PART 2.....	879



Electrical installation drawing

This section deals with the functions in PCISCHMATIC Automation, which are specially designed for electrical installation drawing.

Following topics are elaborated:

- Creating a ground plan drawing..... p. 471
- Layers, Elevation and Scaling p. 477
- Tips for installation drawing p. 484

If you want to document small switchboards and panels with heat loss calculation according to EN60439-3, then read the mini manual “Panel Builder tutorial”. You can also see how to create an electrical installation project in the mini manual “Electrical installation tutorial”.

- Both can be downloaded at www.pcschematic.com/manuals.



Other relevant chapters

- Exporting/importing DWG/DXF files p. 660
 - Inserting unit drawings p. 568
 - Creating unit partslists/drawings p. 767
-

CREATING A GROUND PLAN

This chapter is an example on how to draw a ground plan for a house.

THIS CHAPTER DESCRIBES	
— INITIAL PREPARATIONS FOR A GROUND PLAN DRAWING	P. 471
— DRAWING THE WALLS OF THE HOUSE	P. 473
— DRAWING THE INNER WALLS	P. 474
— PLACING SYMBOLS ON A GROUND PLAN	P. 475
— ELECTRICAL INSTALLATIONS IN DIFFERENT ELEVATIONS	P. 475
— EDITING SYMBOLS FOR A BUILDING	P. 476
— WORKING WITH THE DWG AND DXF FORMATS	P. 476

Please note, that it is also possible to import and export DWG and DXF files, as described in the **Tools** manual.

PREPARING FOR A GROUND PLAN DRAWING



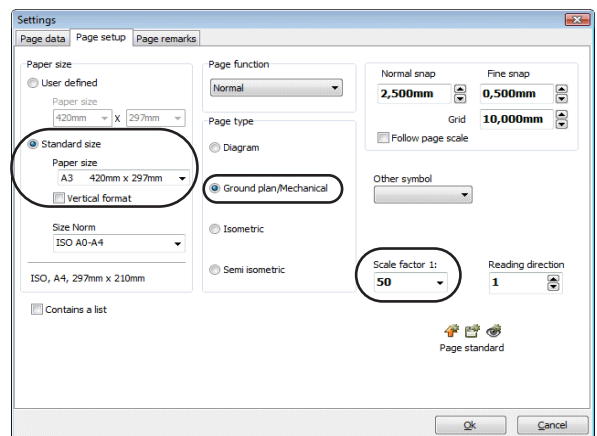
Start a new project by clicking the **New document** button.

Page settings

Choose **Settings => Page setup**, click on **Ground plan/Mechanical**, select the **Scale factor 1:50**, and choose the paper size **A3** — or another paper size, that fits your drawing. Read more about this in “Setting up the page to vertical A4 format” on page 743. Click **OK**.

Choose **Settings => Pointer/Screen** and set the unit of measurement to meters (**m**).

Click **OK**.



THE PAGE FUNCTION MUST BE SET TO NORMAL

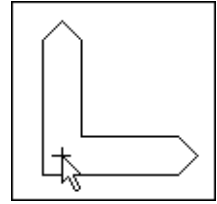
TO USE A PAGE FOR A GROUND PLAN DRAWING, THE **PAGE FUNCTION** FOR THE PAGE MUST BE SET TO **NORMAL**.

Moving the origo

Before drawing the house, you can move the origo of the page, so that it is no longer located in the lower-left corner of the page:



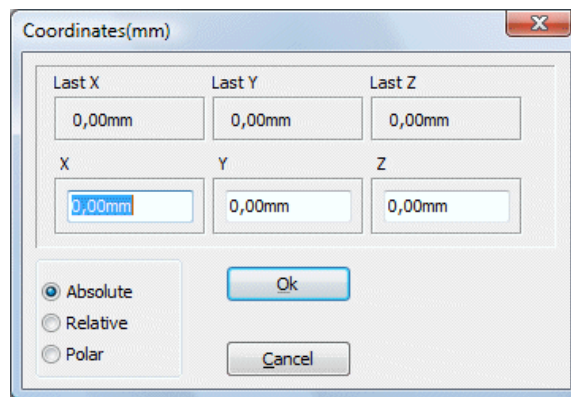
- 1) Click the **Move origo** button, which gives you the representation of the origo in the cross hairs
- 2) You now *either* click wish to position you want the origo, *or* click in the coordinates field at the lower part of the screen, where the X-Y coordinates are displayed. The coordinates field could for instance look like this:



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

The coordinates field

- 3) This dialog box will appear:



- 4) Select **Absolute**. Set **X** to 5 and **Y** to 5. You do not have to type in the unit of measurement *m*, because you have already told the program that you are working in meters

- 5) Click **OK**

The coordinates for any object placed on the page, will be calculated from the new position of the origo.



ORIGO COORDINATES CALCULATED FROM THE LOWER LEFT CORNER

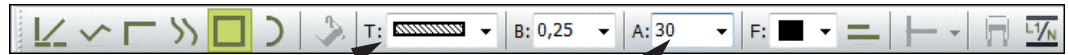
WHEN YOU PLACE THE ORIGO USING THIS DIALOG BOX, THE COORDINATES FOR THE ORIGO ITSELF ARE ALWAYS CALCULATED FROM THE LOWER LEFT CORNER OF THE PAGE.

An easier way to enter the **Coordinates** dialog box, is by using the short-cut <Ctrl+i>.

Selecting the type of line

If you are not familiar with how to draw lines, please refer to “Lines” on page 72.

When you are drawing the ground plan for a building, you should always use a line type (**T**) and line distance (**A**), that matches the inner and outer walls. Here you set the line distance to the size of the inner and outer walls in real life.



Choose this *Line type* first

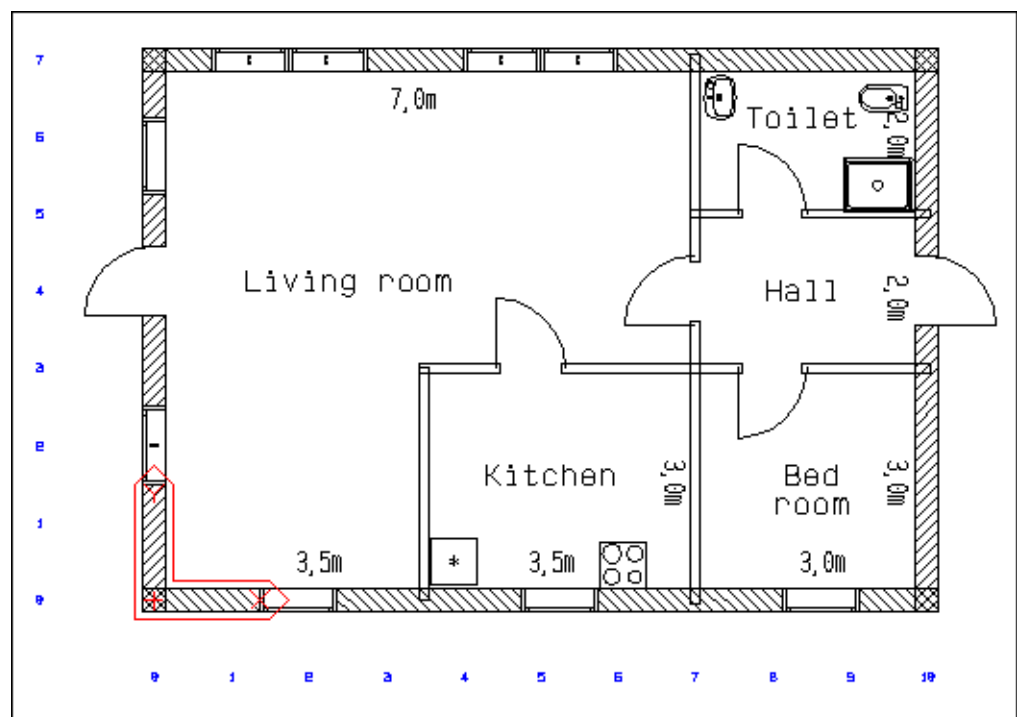
Then type in this *Line distance*

The reason you do this, is because you have already told the program that you are working in 1:50. An outer wall of 30cm will therefore have to be drawn with a line distance of 300 (mm) - or type in 30cm in the **A:** field. Please note that the Conducting lines button is automatically deselected for this type of lines.

Weather to draw the lines with **Extended line**, see “Extended lines” on page 90.

DRAWING THE WALLS OF THE HOUSE

You can now draw the walls in the house you see below:



- 1) Click the **Lines** button, set the line type (**T**) according to the instructions above, and pick an appropriate line distance (**A**)
- 2) Click the **Rectangle** button in the tool bar for lines, and activate the **Pencil** button
- 3) Press <Ctrl+i> or click the Coordinates field, select **Absolute**, set **X** and **Y** to 0
- 4) Click **OK**. You now start drawing the rectangle (the outer walls) in the point (0,0)

- 5) To set the opposite corner of the rectangle, press <Ctrl+i>, choose **Relative**, set **X** to 10 and **Y** to 7. Click **OK**

This instructs the program to draw a rectangle with the measurements 10m x 7m, starting from the origo.

You can read more about drawing with absolute, relative and polar coordinates in “Drawing by measures” on page 93.

DRAWING THE INNER WALLS

- 1) Choose a new type of line — according to the figure above — and set the line distance to 12cm
- 2) Select the **Lines** button, the **Straight lines** button and the **Pencil**
- 3) Press <Ctrl+i>, choose **Absolute**, set **X** to 7, **Y** to 0 and click **OK**.
- 4) Press <Ctrl+i> again, choose **Relative**, set **X** to 0, **Y** to 7m and click **OK**



You can also draw the walls without using the coordinate box.

DISPLAYING ABSOLUTE AND RELATIVE COORDINATES

THE ABSOLUTE COORDINATES ARE DISPLAYED CONTINUOUSLY IN THE LOWER-LEFT CORNER OF THE SCREEN. WHEN YOU ARE DRAWING A LINE OR A RECTANGLE, THE FIELD NEXT TO THE COORDINATES WILL SHOW YOU THE COORDINATES RELATIVE TO THE STARTING POINT OF YOUR LINE/RECTANGLE.

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

Absolute coordinates

Relative coordinates



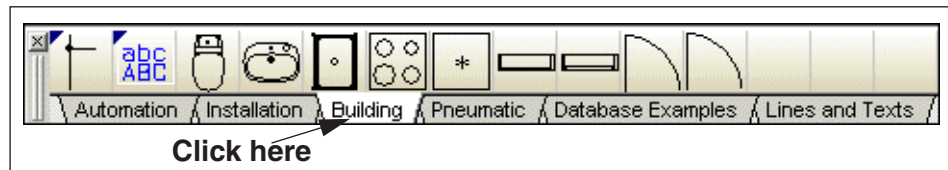
LINES OPEN UP WHEN YOU PLACE SYMBOLS

WHEN YOU DRAW THE LINES, YOU DO NOT HAVE TO CONSIDER WHERE THE DOORS AND WINDOWS ARE GOING TO BE PLACED. WHEN YOU PLACE THE WINDOW AND DOOR SYMBOLS ON THE LINES LATER, THE LINES WILL OPEN UP AND MAKE ROOM FOR THEM AUTOMATICALLY.

Now draw the rest of the house using absolute and relative coordinates where most appropriate.

PLACING SYMBOLS ON A GROUND PLAN

To place doors, windows etc. on the drawing, click on the **Building** tab in the pick-menu, until the displayed pickmenu appears.



From here you can pick the symbols you wish to place on the drawing.

Choosing a symbol

To choose a symbol from the **Symbol menu**, do the following:



- 1) Click the **Symbols** button and the **Symbol menu** button — or use the short-cut keys <s> and <F8> to enter the **Symbol menu**
- 2) In the **Symbol menu**: Select the folder/alias **Building**
- 3) Click on a symbol you wish to place on the ground plan
- 4) Click **OK**

You have now got the symbol in the cross hairs. Click where you wish to place the symbol in your ground plan. The symbol can be moved, rotated and deleted as described earlier.

The symbol can be placed on a line by clicking on the line, and space will be made for the symbol. The symbol will orientate itself according to the direction of the line. It will be rotated counter-clockwise. If the symbol is being moved, it will stick to the line (wall) it is placed on.

You can read more about symbols in “Symbols” on page 137.

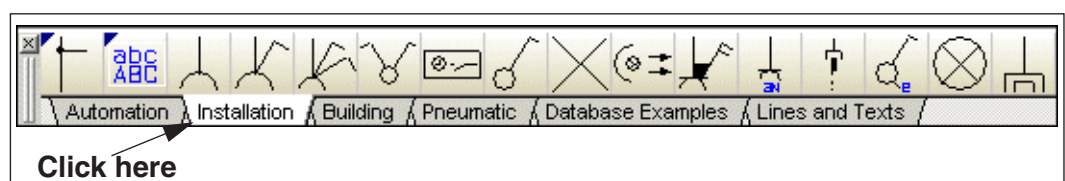
ELECTRICAL INSTALLATIONS IN DIFFERENT ELEVATIONS

If you wish to place electrical installations on the drawing, it is possible to place these in different heights/elevation. To do this you use the **Elevation** window in the upper-right corner of the screen. Symbols for electrical installations can be found in the **lecinst** alias/folder.

It is a good idea to place the ground plan on one layer and the electrical installations on another. Also, you could have a layer with sanitary installations, a layer with burglar alarms — and so on. Read more about layers in “Layers, Elevation and Scaling” on page 477.

You can see an example of this in the project **Instdemo.pro**.

By clicking on **Installation** tab in the pickmenu, you can also see pickmenus for installations and pneumatics.



USING THE GRID FOR HELP WHILE DRAWING

When you are drawing a building by measures, and the accuracy is not that important, you can use the grid/snap for drawing. If the drawing is in scale 1:50, a snap of 10mm will correspond to 0.5m. See the real coordinates at the bottom of the screen.



The cursor will then move 0.5m each time you press one of the arrow-keys.

EDITING SYMBOLS FOR A BUILDING

If you wish to design more symbols meant for ground plans, there are a few things you should know:



DESIGN SYMBOLS USING REAL MEASURES

WHEN YOU DESIGN A SYMBOL FOR A BUILDING, THE SYMBOL MUST BE DESIGNED BY SPECIFYING ITS REAL MEASURES.

This means you must choose **Settings => Page settings**, and set the scale of the page to e.g. 1:2, so that you can design the symbol specifying its real measures.

Symbols designed this way, will fit pages using any scales. Placing the symbol on for instance a 1:10 page, will give you a 1:10 symbol as well.

Windows and doors

When you design symbols that can be connected to walls/lines on a drawing, you have to be very careful where to place the connection points for the symbol:

As you might see, this symbol has a connection point on each side of the window. These connection points have to be located at exactly the same line as the reference point of the symbol. This makes it possible to open a line, and connect the symbol to the line automatically, when the symbol is placed.



The symbol must be saved as a **Non conducting** symbol. The **Selection** must be set to **Whole symbol**, when you save the symbol. — For a deeper understanding of this, please read "Creating Symbols" i PC|SCHEMATIC Automation manualen.

WORKING WITH THE DWG OR DXF FORMAT

You can load a drawing in DWG or DXF format, for instance of a building, from other CAD programs, such as AutoCAD. The drawing can be loaded as either a symbol or a project in **Automation**.

Read more about this in the **Tools** manual.

LAYERS, ELEVATION AND SCALING

In **Automation** you can work in different layers on the same project page. You can also apply scaling, and work in different elevations.

THIS CHAPTER DESCRIBES	
— ABOUT LAYERS.....	P. 477
— THE LAYER MENU	P. 478
— DRAWING HEADERS AND LAYERS.....	P. 480
— CHANGING BETWEEN LAYERS WITH THE MOUSE	P. 480
— MOVING OR COPYING BETWEEN LAYERS.....	P. 481
— SELECTING OBJECTS ON LAYERS.....	P. 481
— PRINTING OUT A PAGE WITH LAYERS	P. 481
— DRAWING WITH ELEVATION	P. 482
— SETTING THE SCALE OF AN ENTIRE PAGE	P. 483
— SCALING AN AREA	P. 483

ABOUT LAYERS

If you on a drawing want to be able to display how for instance a switchboard would look with and without a switchboard front, you have the option of doing this by placing the switchboard, the switchboard front and the components on individual layers on the drawing. You can determine what you want displayed by turning the different layers on and off.

You can copy or move elements of drawings from one layer to another, and by working on layers, you can also determine which layers/parts of the drawing you want to be included when you print out the drawing.

You can also determine to place eventual corrections on independent layers on the drawing.

The layer you are working on will always be visible, but apart from this, you can determine yourself which layers you want displayed on the screen. On each page in the project you have the option of working on 255 different layers.



LAYERS ARE ALSO APPLIED FOR CREATING SYMBOLS WITH STATES

LAYERS ARE ALSO APPLIED WHEN CREATING SYMBOLS WITH STATES, AS DESCRIBED IN “CREATING SYMBOLS WITH STATES” ON PAGE 712.

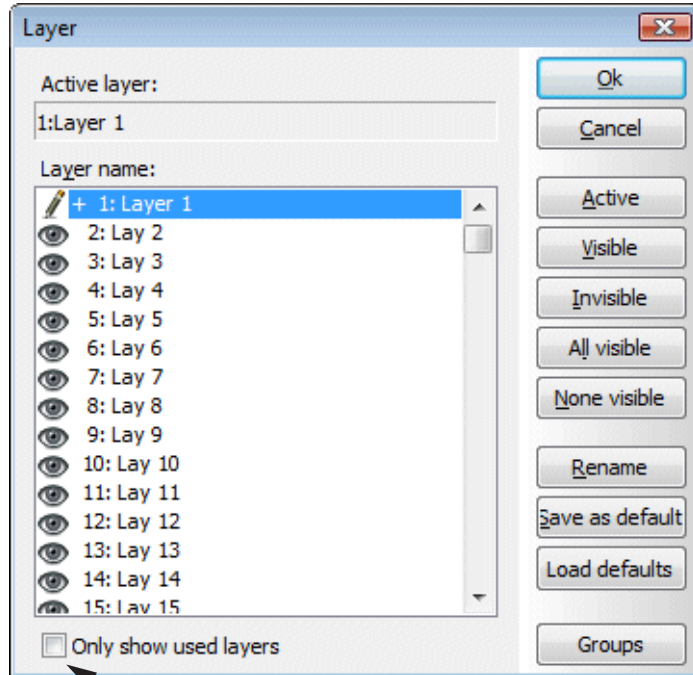
Layer information saved in the project file

Information about which layer is active, as well as which layers are visible, are saved in each project file individually.

THE LAYER MENU



To see or change the layer settings, you first have to enter the **Layer menu**, which is done by clicking on the **Layer** button on the left tool bar. Clicking on **Layer=1:Lay 1** at the bottom of the screen will have the same effect:



Click here if you only want to see layers with contents

The pencil displayed against **1:Lay1** states that layer 1 is the active layer. When a layer is active, this means you can draw and make changes on this specific layer, while you cannot make changes on the inactive layers on the drawing.

If you see a + in front of a layer, this means something has been placed on this layer.

Visible layers



The eye you see in front of all the other layers, tells you that these layers are visible. When you, like you do here, work on layer number one, you will see the other layers in the background drawn with the color grey on the diagrams.

If you want to work on layer three, you click on it, and then click on **Active**. You will now see that the pencil has moved to layer three.

Selecting more than one layer

If you want to make for instance layer four and six invisible, you click on layer four, press down <Ctrl> and click on layer six. Then you click on **Invisible**, and the eyes in front of the two names disappear.

If you then click on **No visible**, all the eyes disappear, and if you click on **All visible**, all layers have become visible again.

In order to select for instance all layers from eight to twelve, click first on layer 8, then keep the <Shift> key pressed down when clicking on layer 12.

Naming layers

If you want to insert a name or a description for a layer, first click on the layer, and then click on **Rename**. A new dialog box appears, you type in the new name and click **OK**. It is a good idea to assign names to the layers, so that you always use the same layers to the same purpose.

Saving layer changes in the project

If you regret the changes you have made, you click **Cancel**, otherwise you click **OK** to save them.

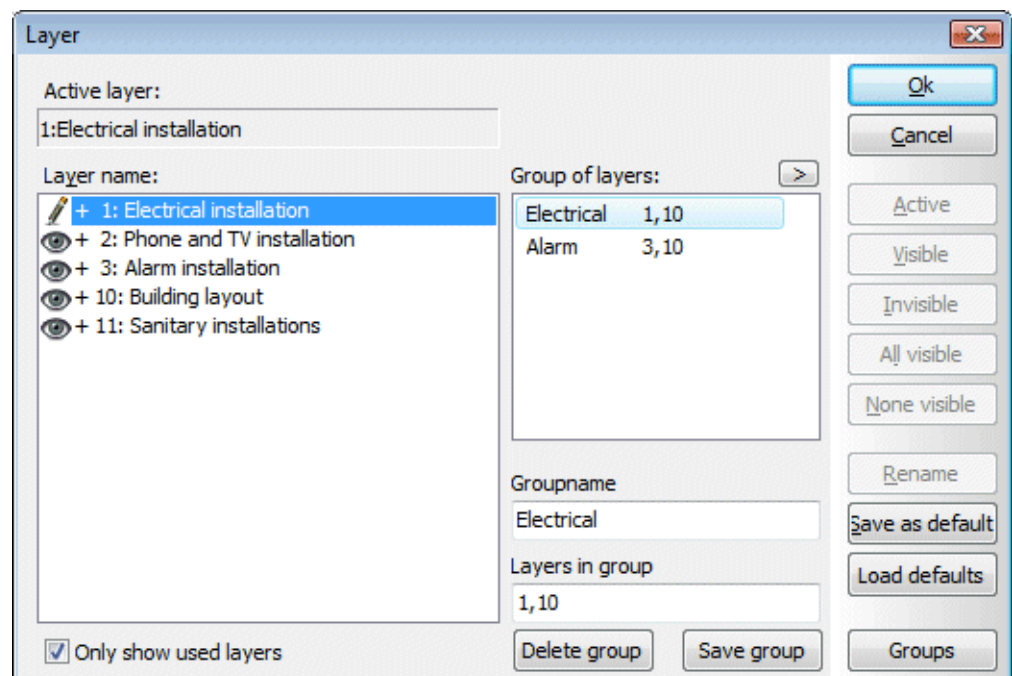
Saving layer settings as Default

If you have a specific use for the various layers, then save the settings as default by clicking on **Save as default**. Subsequently you can load the standard from the **Layer menu** in another project by clicking **Load defaults**.

Grouping layers

When you click the **Groups** button in the **Layer menu**, you have the option of grouping the layers according to which layers you wish to see at the same time. If you are working with an electrical installation project, you can for instance create a layer group containing the ground plan drawing and the electrical installations, and another layer group containing the ground plan drawing and the alarm installations.

When you click the **Group** button in the lower-left corner, the **Layer menu** is expanded, so that you can work with layer groups:



Creating a group

- 1) In the field **Group name**: Type in a name for the group.

- 2) In the field **Layers in group**: Type in the layer numbers separated by commas — or *drag* the layers from the left-hand side and place them under **Layers in group** at the bottom-right corner of the dialog box.
- 3) Click **Save group**.

The layer listed first in the group becomes the active layer, while the other layers becomes visible.

If you click on the arrow to the right of **Group of layers**, the dialog box is further expanded, so that you can see the names of the selected layers.

Deleting a group

- 1) Under **Group of layers**: Click on the group.
- 2) Click **Delete group**.

Printing layer groups

When printing you can specify that you only wish to print a specific layer group. — See “Printing layers” on page 328.

When you no longer wish to see the layer groups, just click the **Groups** button again.

Only show used layers

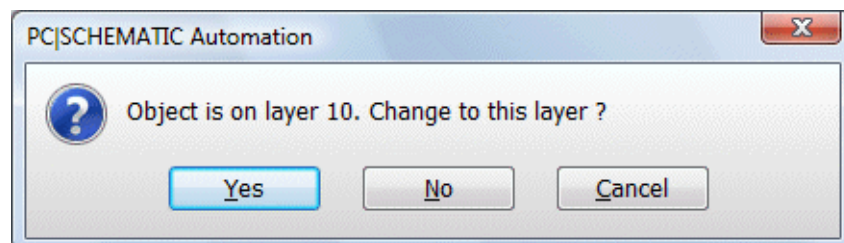
When you select **Only show used layers** at the bottom of the dialog box, the dialog box only displays the layers that are used in the project.

DRAWING HEADERS AND LAYERS

Please note, that drawing headers are always placed on layer number zero, which is always displayed. If a drawing header is placed on a page, you will always see it, no matter which settings you choose in the layer menu.

CHANGING BETWEEN LAYERS USING THE MOUSE

If you press down <Alt> and click on an object which is located on another layer, you will be asked whether to change to this layer.



Answer **Yes**, and you will change layer.

Of course the object you click on have to be of the object type you are working with for the moment. If **Symbols** is active, you can change to symbols on other layers — and so on.



MOVING OR COPYING BETWEEN LAYERS

When you want to move or copy objects from one layer to another, do this:



- 1) Enter the **Layer menu** and choose the layer you want to fetch something from
- 2) Select what you want to move/copy, right-click and choose **Move** or **Copy**
- 3) Change layer
- 4) Place the selected objects

SELECTING OBJECTS ON DIFFERENT LAYERS

The chapter “Selecting objects” on page 55 describes how to select objects in **Automation**. Below you can see the options of selecting objects in different layers.

Selecting everything on a layer

You can select everything on a layer by choosing **Edit => Select all => All on layer**. If the **Area** button is active, all objects on the page will be selected. If **Lines** is active, all lines on the layer will be selected.

Selecting an object on a different layer

If you wish to select an object located in another layer than the active, you can press down the <Alt> key while clicking on the object. You will then be told on which layer the object is located, and will get the option to change to this layer.

Selecting an area in all layers



It is possible to select everything in all layers within an area of your drawing. Select for instance **Area** and mark up an area within the page. Then right-click anywhere on the page and choose **Select in all layers**. All objects on all layers within the area are now selected.

If for instance you had selected **Symbols** in stead of **Areas**, only the symbols within the area would have been selected.

PRINTING PAGES WITH LAYERS

When you want to print out a drawing consisting of several layers, you choose **File => Print => Select pages => Show layer settings**. Hereby you enter a dialog box similar to the one you entered before, but these settings *only* apply to what you print out.

Following the same guidelines as above, you now determine which layers or layer groups should be visible, and then click **OK**. In the lower part of the dialog box, you see how the drawing would look on print.

If you do not wish to print out the entire project, please make sure that you have only checked off **Actual page** in the lower-left part of the dialog box, and then click on **Print** in the upper-right corner.

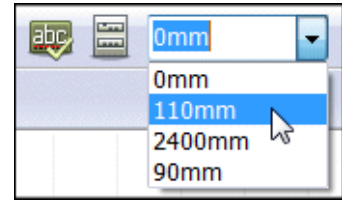
The drawing will now be printed out on your printer. Read more about printing in **Automation** in “Printing” on page 326.

DRAWING WITH ELEVATION

The **Elevation** window is used when you want to place objects in different elevations on ground plan pages, and enables you to work in three dimensions.

The **Elevation** window is only visible when you are working on a **Ground plan** page in the project.

To specify an elevation, click inside the **Elevation** window and then type in the elevation. The five latest elevations are remembered, and can be chosen by clicking on the down-arrow in the window.



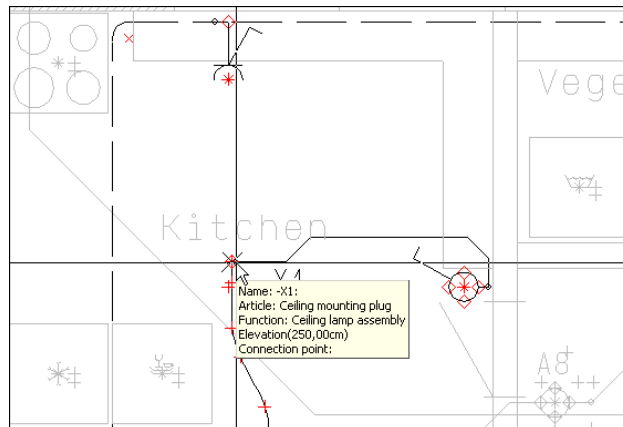
Selecting objects in another elevation

You can only select objects that are located in the elevation you see in the **Elevation** window. If you press down <Shift> while clicking on an object, you automatically change to the elevation in which the selected object is placed.

When you're drawing lines and wish to connect a line to a symbol in another elevation, the program automatically shifts to the other elevation, when you connect the line to a connection point on the symbol.

Displaying the elevation of symbols as a hint

When a symbol is not placed in the elevation **0**, the elevation is displayed as an **InfoTip** when the cross hairs is above the symbol.



Lines changing elevation

If you want to draw a line that changes elevation while drawing, you start by clicking where you want the line to start, then click in the **Elevation** window, type in the elevation for the next point of the line, and click where the line shall end.



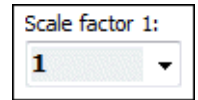
EXAMPLE ON WORKING WITH ELEVATION

SEE AN EXAMPLE ON HOW TO WORK WITH ELEVATION IN THE MINI MANUAL "ELECTRICAL INSTALLATION TUTORIAL".

YOU CAN DOWNLOAD IT FROM www.pcschematic.com/manuals.

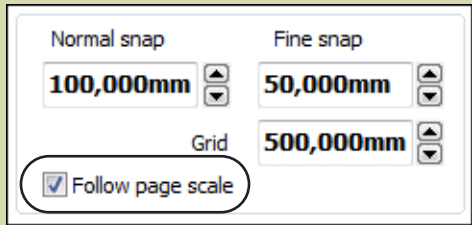
SETTING THE SCALE OF THE PAGE

The scaling on the page is set in **Settings => Page setup**. Here you determine the scale you want to apply on the page. On diagram drawings the scale will typically be 1:1, while it will typically be 1:50 on ground plan drawings.



There is no reason to scale on diagram pages, because you can choose to draw on a page with the A3 page size, and make the program scale automatically to fit the paper size when printing out the diagram. This kind of settings are set in the **Printer** dialog box, and you can read more about automatic scaling in “Setup” on page 329.

The scaling on the page has great influence on the objects you place on the page. In the following list, you will see some examples on what to keep in mind when you work on scaled pages.

EXAMPLE	PLEASE NOTE
DRAWING BY MEASURES	WHEN YOU CHANGE THE SCALING OF THE PAGE FROM 1:1 TO 1:50, ALL COORDINATES ON THE PAGE BECOMES 50 TIMES LARGER — EVEN THOUGH THE REPRESENTATION ON THE PAGE WILL REMAIN THE SAME. A PAGE OF THE A4 FORMAT OF 21.0CM X 29.7CM, WILL IN THIS CASE CORRESPOND TO 10.5M X 14.85M.
GRID/SNAP	YOUR GRID AND SNAP WILL ALSO BECOME 50 TIMES LARGER. THIS MEANS THAT A GRID ON 10MM WILL NOW CORRESPOND TO 50CM, AND A SNAP OF 2.500MM WILL CORRESPOND TO 12.5CM. TO MAKE THE GRID/SNAP FOLLOW THE PAGE SCALE, YOU CAN CLICK THE PAGE DATA BUTTON, CHOOSE THE TAB PAGE SETUP , AND SELECT FOLLOW PAGE SCALE . 
SYMBOLS	THE INFLUENCE OF THE SCALING ON A SYMBOL, DEPENDS ON WHETHER THE SYMBOL HAS AUTO SCALE OR NO SCALE ADJUSTMENT . TYPICALLY, A SYMBOL WILL HAVE SCALE ADJUSTMENT, BUT NO AUTO SCALE. READ MORE ABOUT THIS IN “SYMBOL SETTINGS: ADVANCED” ON PAGE 708.
LINE DISTANCE	WHEN YOU DRAW LINES HAVING A SPECIFIC LINE DISTANCE, IT IS IMPORTANT TO BEAR IN MIND THE SCALE OF THE PAGE. FOR INSTANCE, IF YOU DRAW A WALL OF 30CM ON A PAGE HAVING THE SCALE 1:50, YOU HAVE TO SET THE LINE DISTANCE (A) TO 30CM. THE PROGRAM WILL PERFORM THE SCALING BY ITSELF, SO THAT THE LINE IS DRAWN AS A 6MM LINE ON THE PAGE.

SCALING AREAS

It is also possible to scale an area — both on diagrams and when editing symbols. Read more about this in “Scaling areas” on page 253.

TIPS FOR INSTALLATION DRAWING

In this chapter, you'll find some tips, which you can use when you draw installation drawings. If you need general information on how to draw in **Automation**, please refer to "Basic Drawing facilities" on page 51.

You can find more options for electrical installation drawing in "Electrical installation drawing" on page 469.

THIS CHAPTER DESCRIBES	
— GROUND PLAN DRAWINGS AND DWG/DXF FILES.....	P. 484
— PRECISE POSITIONING OF MECHANICAL SYMBOLS	P. 484
— DRAWING CONNECTIONS ON INSTALLATION DRAWINGS	P. 484
— CHOOSING CABLES AND WIRES IN THE PICKMENU	P. 485
— INSTALLATION SYMBOLS IN THE PICKMENU WITH ELEVATION.....	P. 485
— DRAWING CONNECTIONS ON INSTALLATION DRAWINGS	P. 485



"ELECTRICAL INSTALLATION TUTORIAL"

AT www.pcschematic.com/manuals YOU CAN DOWNLOAD THE MINI MANUAL "ELECTRICAL INSTALLATION TUTORIAL", WHICH GUIDES YOU THROUGH CREATING AN ELECTRICAL INSTALLATION PROJECT.

GROUND PLAN DRAWINGS AND DWG/DXF FILES

When you wish to draw a ground plan drawing in **Automation**, please read "Creating a Ground Plan" on page 471. Do you need to import DWG/DXF files, please read "DWG and DXF files" on page 660. Please also refer to "Electrical installation drawing" on page 469.

PRECISE POSITIONING OF MECHANICAL SYMBOLS

See how to position mechanical symbols precisely in "Magnetic snap and Snap in objects for mechanical symbols" on page 634.

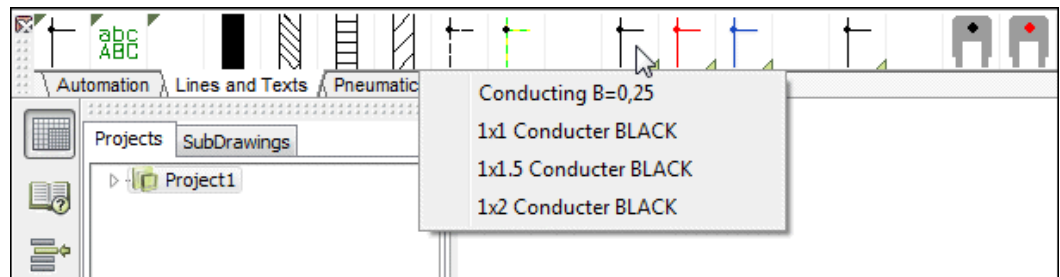
DRAWING CONNECTIONS ON INSTALLATION DRAWINGS

When symbols are placed in different elevations (for instance on an installation drawing), **Automation** automatically changes elevation when the symbols are connected.

— See also "Selecting objects in another elevation" on page 482.

SELECTING CABLES AND WIRES FROM THE PICKMENU

When you are drawing with cables, wires and jumper links, these can be selected directly from the Pickmenu.



All of these line types can be assigned line article data in the Pickmenu. To assign line article data, right-click on the cable/wire/jumper link in the Pickmenu, and choose **Component Article Data**. Just like you do for symbols.

When you're drawing cables and wires, please remember to select **Include Lines with article data** in the list settings for the parts list. — See "Which data to include" on page 544.

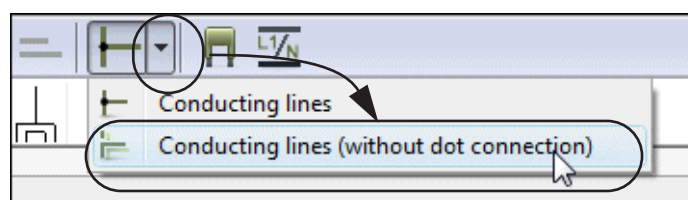
INSTALLATION SYMBOLS WITH ELEVATION IN THE PICKMENU

Installation symbols in the Pickmenu can be assigned a default elevation, which they are automatically assigned when placed on a mechanical page.

When you are working on a page of the type **Ground plan/Mechanical**, just right-click on an installation symbol in the Pickmenu, and select **Elevation**. You can then type in the default elevation for the symbol in the Pickmenu.

DRAWING LINES ON INSTALLATION DRAWINGS

When you're drawing installation drawings, you can draw lines on top of each other, without displaying connection dots.



To avoid making connection dots for parallel cables, you must choose **Conducting lines (without dot connection)**. This is only possible when you're working on a page of the type **Ground plan/Mechanical**.



Working with Projects

In this part you can see how to work with projects in PCISCHMATIC Automation.

It contains following topics:

- On projects p. 489
- Pages and Chapters p. 491
- Template and Standard projects..... p. 511
- Copying full projects into projects..... p. 513
- Working with More than one Project..... p. 516
- Overview projects..... p. 524
- Inserting logos in projects..... p. 527



Read the survey article

Read the survey article about working with projects at www.pcschematic.com/articles.

Please also refer to the Project Generator manual to see how to merge projects via Excel.



ON PROJECTS

Automation is a *project oriented* program. This means, that all information you need when working on a project, is gathered in just one file.

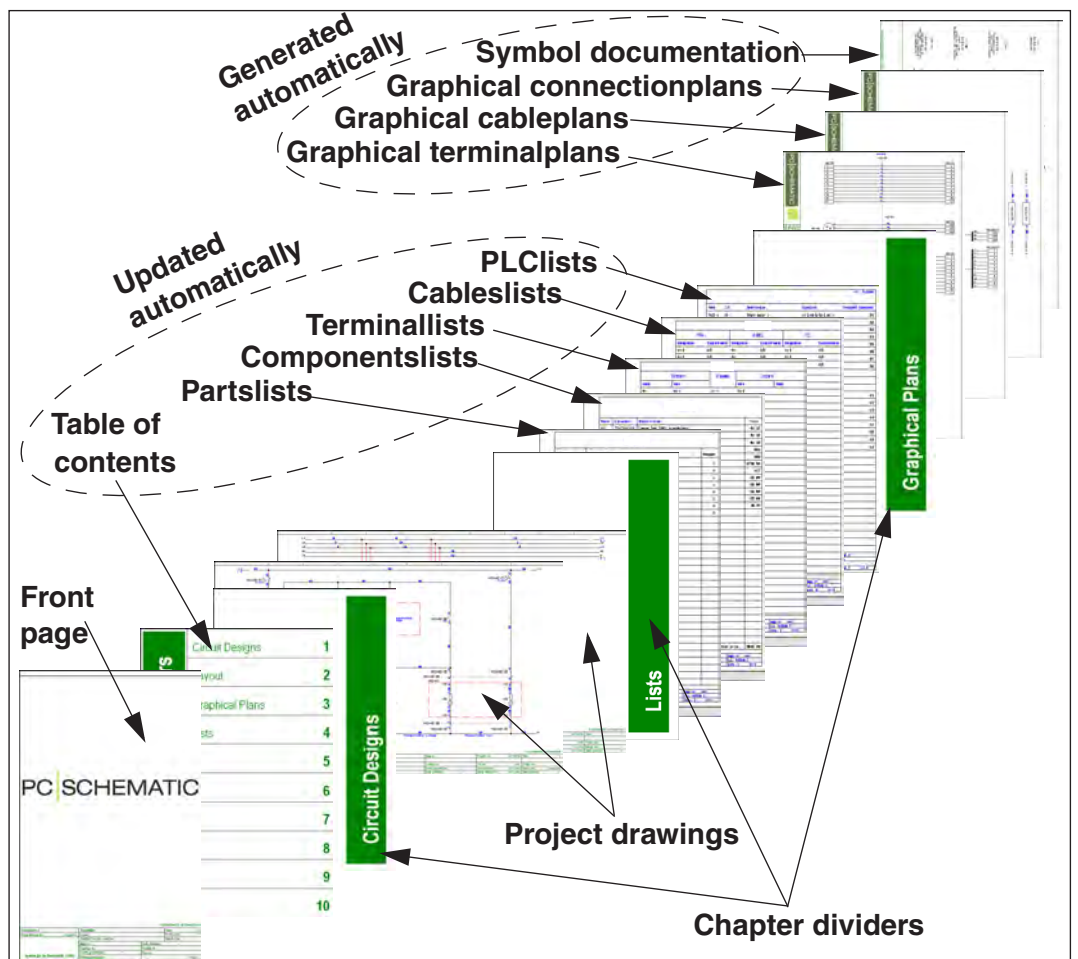
Therefore you do not need to change program to e.g. create parts lists or unit drawings, because they are located along with your diagrams inside the project file.

THIS CHAPTER DESCRIBES

THIS CHAPTER DESCRIBES THE *PROJECT ORIENTED* CONCEPT OF **Automation**.

THE ELEMENTS OF A PROJECT

A project typically consists of a front page, a table of contents, pages with diagrams, and pages containing various kinds of lists. Also, pages containing mechanical symbols for the applied components, can be part of the project. All these parts are placed on their individual page(s) inside the project. A project can contain several thousand pages.



Project diagrams

The basis of your projects are the project diagrams. Here you draw your diagrams by placing symbols, texts and lines on one or more pages in the project. And here you assign the article data to the symbols, that will be transferred to the project lists automatically.

How to draw diagrams, are described in the two previous sections “Basic Drawing facilities” on page 51 and “Electrical installation drawing” on page 469.



THIS CAN ALSO BE CREATED AUTOMATICALLY

- GRAPHICAL TERMINAL PLANS
- GRAPHICAL CABLE PLANS
- GRAPHICAL CONNECTION PLANS
- SYMBOL DOCUMENTATION

THIS IS DESCRIBED IN THE **PC|SCHEMATIC Tools** MANUAL. THERE YOU CAN ALSO SEE HOW TO TRANSLATE PROJECT TEXTS USING THE PROGRAM **PCschematic Translator**, AND SPECIFY PARAMETERS FOR DWG/DXF IMPORT/EXPORT.

This section

This section covers the topics you need to know something about when you want to create projects, containing different types of pages:

CHAPTER	CONTENTS	PAGE
"PAGES AND CHAPTERS"	HOW TO CREATE NEW PAGES IN A PROJECT, HOW TO FIND PAGES, AND WHICH TYPES OF PAGES YOU CAN CREATE. CREATING CHAPTERS.	491
"PROJECT TEMPLATES AND STANDARD PROJECTS"	HOW TO APPLY TEMPLATE PROJECTS, CONTAINING JUST THE PAGES YOU NEED EACH TIME YOU START A NEW PROJECT.	511
"COPYING FULL PROJECTS INTO OTHER PROJECTS"	HOW TO COPY FULL PROJECT INTO OTHER PROJECTS.	513
"SEVERAL PROJECTS SIMULTANEOUSLY"	HOW TO DRAG PAGES FROM PROJECT TO PROJECT, HOW TO COPY AREAS FROM PROJECT TO PROJECT, OR COPY TO OTHER PROGRAMS.	516
"INSERTING LOGOS IN PROJECTS"	HOW TO INSERT A LOGO FOR THE FULL PROJECT.	527

How to work with lists in projects is covered in the section “Lists” on page 529. In the **Project Generator** manual, you can furthermore see how to merge projects and control project data via Excel.



SENDING PROJECTS BY E-MAIL

WHEN YOU CHOOSE **FILE => SEND TO**, AN E-MAIL IS CREATED IN YOUR E-MAIL PROGRAM HAVING THE ACTIVE PROJECT AS AN ATTACHMENT. YOU CAN THEN SEND THE PROJECT BY E-MAIL.

PAGES AND CHAPTERS

When you wish to add or delete pages, and specify the type and function of a page, you typically do this in the **Page menu**. You can also *drag* pages into the active project from the **SubDrawings** tab in the **Explorer** window.

From the **Page menu** you can also divide projects into chapters, and from here you can copy several pages at a time, for insertion into other projects. You can also copy pages or projects between the open projects, just by *dragging* page or chapter tabs from one project to another.

THIS CHAPTER DESCRIBES	
— CHOOSING PAGES OUTSIDE THE PAGE MENU.....	P. 491
— THE PAGE MENU	P. 493
— INSERTING A NEW PAGE.....	P. 495
— COPYING, CUTTING AND PASTING MORE THAN ONE PAGE	P. 500
— PASTING COPIED PAGES IN OTHER PROJECTS.....	P. 501
— MOVE, COPY AND DELETE PAGES AND CHAPTERS BY USING PAGE TABS.....	P. 503
— CHAPTERS	P. 504
— NAMING PAGE TABS.....	P. 506
— SPECIFYING PAGE TYPE.....	P. 508
— SPECIFYING PAPER SIZE	P. 509
— GROUND PLAN DRAWING	P. 509
— ISOMETRIC DRAWING	P. 510

CHOOSING PAGES OUTSIDE THE PAGE MENU

When you are *not* in the **Page menu**, you can change back and forth between the project pages by clicking on the tabs on the lower part of the page.



Or you can use the <Page Up> and the <Page Down> keys to jump back and forth between the project pages. Another option is clicking the **Page history** buttons, as described in “The Page history buttons” on page 492.

Also, you can choose **View => Go to page**, and type in a page number.

When you click the chapter tabs on the right-hand side of the screen, the first page of the chosen chapter is displayed. These chapter tabs are only displayed when the project is divided into chapters. — Read more about this in “Chapters” on page 504.

Collapsed chapter tabs

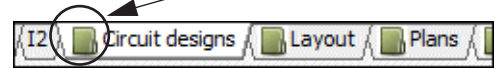
When chapter tabs appears among the page tabs at the bottom of the screen, you can choose between seeing:



a) Both the chapter tab and page tabs for all pages in the chapter:



b) Only the chapter tab:



You can change back and forth between the two views by clicking on the chapter icon on the tabs, as illustrated in the figure above.

When the active page is placed in a chapter, which is collapsed, the page number is displayed in square brackets on the chapter tab — as you can see in the figure above. — See more about chapters in “Chapters” on page 504.

The Page history buttons

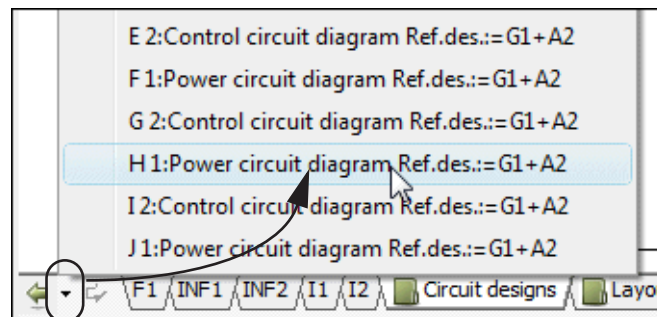


By clicking the **Page history** buttons to the left of the page tabs at the bottom of the screen, you can jump back and forth between the pages you have been working on in the project. This function can also be activated by choosing **View => Page history back** and **View => Page history forward**.

You could say that these buttons function like the **Back** and **Forward** buttons in Microsoft Internet Explorer, with the only difference that in **Automation** these are only active within the active project.

The Select page button

When you click the little down-arrow between these two buttons, you get a survey of the most recently active project pages:



When you click on one of these, you jump to this page in the project. This way you can jump to the 10 most recently active pages in the project.

— Please note that it can be an advantage also to have activated the function **Keep page zoom** — see “Keep page zoom” on page 44.

Default short-cut keys for the Page history functions

The default short-cut keys for the three functions are:

FUNCTIONS	DEFAULT SHORT-CUT KEYS
PAGE HISTORY BACK	<ALT+LEFTARROW>
PAGE HISTORY FORWARD	<ALT+RIGHTARROW>
SELECT PAGE	<ALT+DOWNARROW>

Using the Explorer window for Survey

Just to get a survey of the project pages — or for instance find a specific symbol in the project — you can also use the **Explorer** window, as described in “The Explorer window: The Projects tab” on page 301.

THE PAGE MENU



Choose the **Page menu** by clicking on the **Page menu** button, or by choosing **View => Select page**.

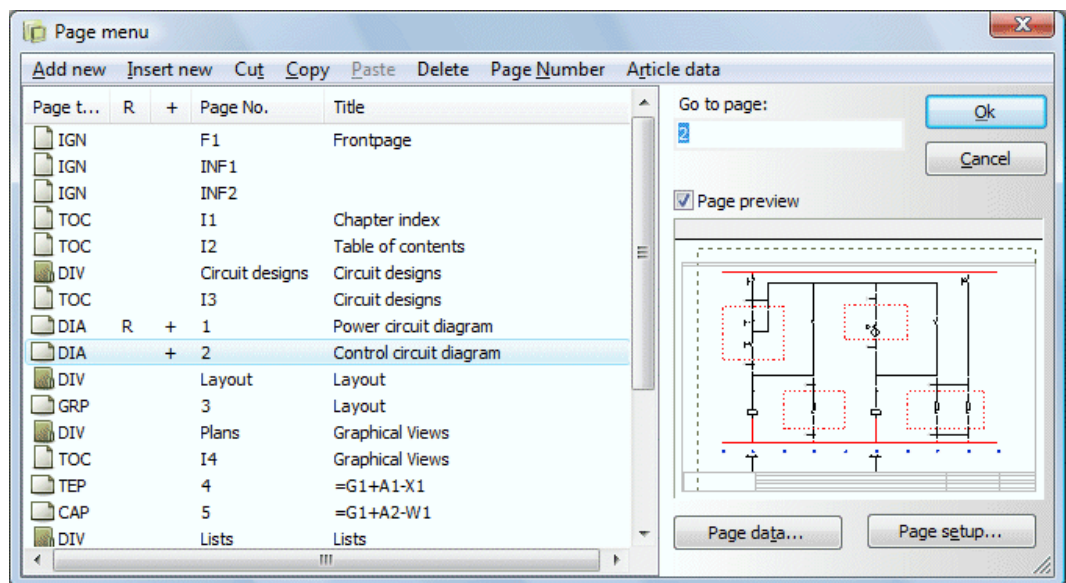
You use the **Page menu** to insert new pages into the project, copy or delete pages, or to change the settings for the pages you have created. Or you can use it to choose which page in the project you want to work on. Besides, the **Page menu** is the place to keep the survey of a huge project, because you will find the titles of all the pages listed here.

From the **Page menu** you can also load other projects into the existing project. Read how to do this in “Project templates and Standard projects” on page 511.

Options in the Page menu



The **Page menu** for a project could for instance appear like this:



In front of each page an icon indicates whether the page has a horizontal or a vertical page format. After this you see the page function, which is described in “Creating a new page” on page 495. Folders in front of pages marks the beginning of a new chapter.

Pages marked with a **+**, have been changed since the project was last saved.

Pages marked with a **!**, have been changed since the project was opened.

If there are pages marked with an **R**, this means that these pages have been changed since the start of a revision check. — See “Revision control” on page 808.

In the **Page** menu you have the following options:

CHOOSE	FUNCTION/USE
ADD NEW	ADDS A NEW PAGE IN THE END OF THE PROJECT. — SEE “CREATING A NEW PAGE” ON PAGE 495.
INSERT NEW	INSERTS A NEW PAGE <i>IN FRONT OF</i> THE SELECTED PAGE. IF THE FIRST PAGE IN THE PROJECT IS SELECTED, THE PAGE WILL BE INSERTED AS A NEW FIRST PAGE IN THE PROJECT. — SEE “CREATING A NEW PAGE” ON PAGE 495.
CUT	CLICK ON THE PAGE(S) YOU WISH TO CUT, CLICK ON CUT , AND CLICK OK TO CUT. THE PAGE(S) CAN THEN BE INSERTED WHERE YOU WANT TO, BY USING THE INSERT FUNCTION. (SHORT CUT KEY: <CTRL+X>). — SEE “COPYING, CUTTING AND PASTING MORE THAN ONE PAGE” ON PAGE 500.
COPY	SELECT THE PAGE(S) YOU WISH TO COPY, CLICK ON COPY , CLICK WHERE YOU WISH TO INSERT THE COPIED PAGES, AND CLICK ON INSERT , AS DESCRIBED BELOW. THE COPIED PAGES ARE THEN INSERTED. (SHORT CUT KEY: <CTRL+C>). THE COPIED PAGES CAN ALSO BE INSERTED INTO OTHER PROJECTS. — SEE “PAGE NAMES WHEN COPYING PAGES IN THE PAGE MENU” ON PAGE 500 AND “COPYING, CUTTING AND PASTING MORE THAN ONE PAGE” ON PAGE 500.
INSERT	IS USED AFTER CUT OR COPY . CLICK WHERE YOU WISH TO INSERT THE PAGE(S), CLICK ON INSERT , AND CLICK OK TO INSERT. (SHORT CUT KEY: <CTRL+I>).
DELETE	CLICK ON THE PAGE YOU WISH TO DELETE, CLICK ON DELETE , AND CLICK OK . — SEE HOW TO SELECT MORE THAN ONE PAGE IN “COPYING, CUTTING AND PASTING MORE THAN ONE PAGE” ON PAGE 500.
ARTICLE DATA	WHEN YOU CLICK HERE, YOU CAN EDIT THE ARTICLE DATA AND/OR REFERENCE DESIGNATIONS FOR THE PAGE. IF YOU HAVE SELECTED MORE THAN ONE PAGE, YOU EDIT ARTICLE DATA/REFERENCE DATA FOR ALL OF THE SELECTED PAGES IN ONE OPERATION. — SEE “EDIT REFERENCE DESIGNATIONS FOR MORE THAN ONE PAGE” ON PAGE 358.
PAGE NUMBER	CLICK ON THE PAGE IN THE MENU YOU WISH TO CHANGE PAGE NUMBER FOR, CLICK ON PAGE NUMBER , TYPE IN THE NEW PAGE NUMBER, AND CLICK OK . THE PAGE NUMBER IS NOW CHANGED. WHEN YOU HAVE SELECTED MORE THAN ONE PAGE, YOU ARE ASKED TO TYPE IN THE NUMBER OF THE FIRST OF THE CHOSEN PAGES. THE SUBSEQUENT PAGES ARE INCREMENTED AUTOMATICALLY. IF ANY OF THE SELECTED PAGES ARE DIVIDERS, YOU ALSO GET THE OPTION OF CHOOSING RENUMBER DIVIDERS . IF YOU SELECT RENUMBER DIVIDERS , THE SELECTED DIVIDER(S) ARE ALSO RENUMBERED. WHEN YOU DOUBLE-CLICK ON A PAGE TAB IN A PROJECT, YOU ARE ALSO ALLOWED TO CHANGE THE PAGE NUMBER.
GO TO PAGE	TYPE IN A PAGE NUMBER AND PRESS <ENTER>. THIS DISPLAYS THE CHOSEN PAGE IN THE PAGE MENU .

CHOOSE	FUNCTION/USE
PAGE SETUP..	WHEN YOU CLICK HERE, YOU ENTER SETTINGS => PAGE SETUP FOR THE SELECTED PAGE IN THE PAGE MENU . IN THIS MENU YOU SPECIFY THE PAGE TYPE, SIZE, SCALE AND SO ON. — READ MORE ABOUT THIS IN “PAGE TYPE” ON PAGE 508, “PAGE SETUP - AND TRANSFERRING THIS TO OTHER PAGES” ON PAGE 506 AND “PAGE SETUP” ON PAGE 843.
PAGE DATA	CLICKING HERE BRINGS YOU INTO THE DIALOG BOX SETTINGS => PAGE DATA FOR THE SELECTED PAGE IN THE PAGE MENU . HERE YOU CAN ASSIGN A TITLE TO THE PAGE, TYPE IN INFORMATION ATTACHED TO THIS SPECIFIC PAGE, AND PLACE A DRAWING HEADER ON THE PAGE. YOU CAN ALSO SET UP CURRENT PATH NUMBERING / A REFERENCE SYSTEM FOR THE PAGE. — SEE “INSERT DRAWING HEADERS/FORMULAS” ON PAGE 531 AND “PAGE DATA” ON PAGE 834.

The Page menu and current path numbers

If you name project symbols according to their location on page/current path, you may read more about which influence it has on these symbol names, when you make changes in the project like the ones mentioned above. Please refer to “Naming due to changes in the Page menu” on page 385.

CREATING A NEW PAGE

You can create new pages in the project in two different ways:

- By *dragging* a page into the project via the **Explorer** window — see “**Drag a new page into the project - via the Explorer window**” on page 495.
- Via the **Page** menu — see “**Creating a new project page - via the Page menu**” on page 496.

Drag a new page into the project - via the Explorer window

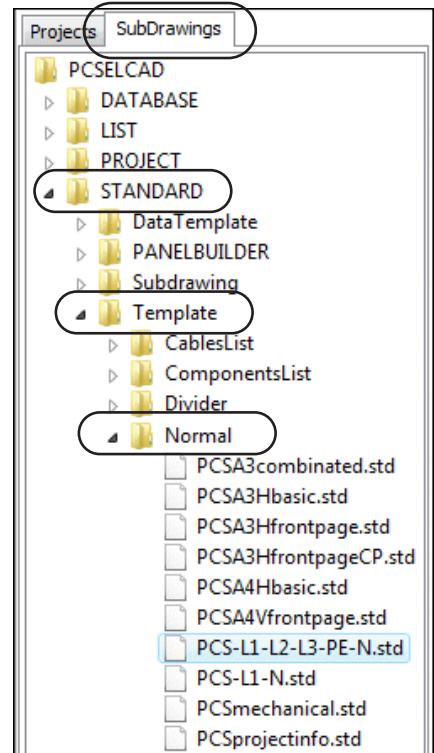
In order to *drag* a new page into the project from the **Explorer** window, this requires:

- a) A page of the desired type already is created in the **Explorer** window on the **SubDrawings** tab (see “**Sub drawings**” on page 256), or
- b) A *template page* of the desired type has been created (see “Creating Templates” on page 771).

Dragging a page into the project

To drag an existing template page into the project via the **SubDrawings** tab in the **Explorer** window, do this:

- 1) Select the page in the project, that you wish to place the new page *after* in the project — e.g. by clicking on a page tab at the bottom of the screen.
- 2) In the **Explorer** window, on the **Sub-Drawings** tab: Open the folder **STANDARD/Template** — which contains the folders with all program templates — and open e.g. the folder **Normal**, as displayed in the figure to the right.
- 3) Then *click* on one of the templates in the folder — e.g. **PCS-L1-L2-L3-PE-N.STD** — and *drag* it over to the project page in the drawing area on the screen.
- 4) When you release the mouse button, the new page is placed after the active page in the project.



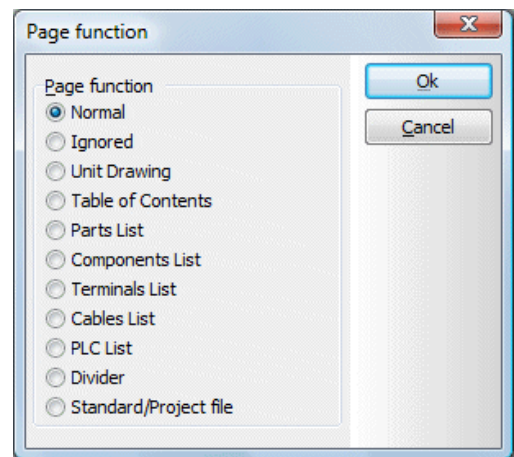
Correspondingly, you can drag other pages directly into the project from the **Sub-Drawings** tab.

Creating a new project page - via the Page menu

When you want to create a new page via the **Page menu**, you first enter the **Page menu**, and click on **Add new** or **Insert new** — as described in “The Page menu” on page 493.

If you are not in the **Page menu**, you can add a new page by going to the last page of the project, and press <Page Down>.

You then enter the **Page function** dialog box, where you specify what you are going to use the page for.



Selecting a Page function



PAGE FUNCTION NORMAL FOR SYMBOLS WITH ARTICLE DATA

WHEN YOU WISH TO INCLUDE ARTICLE DATA FOR THE COMPONENTS/SYMBOLS IN FOR INSTANCE PARTS LISTS, IT IS IMPORTANT THAT DIAGRAM PAGES HAS THE PAGE FUNCTION **NORMAL**, BEFORE PLACING SYMBOLS ON THE DIAGRAM PAGES.

Choose one of the following options and click **OK**:

PAGE FUNCTION	DESCRIPTION	NAME IN PAGE MENU
NORMAL	THE PAGE IS USED FOR DIAGRAM- OR MECHANICAL/GROUND PLAN DRAWINGS. SYMBOLS/COMPONENTS WITH COMPONENT DATA FROM PAGES OF THIS TYPE, CAN BE FILLED INTO FOR INSTANCE PARTS LISTS.	DIA (DIAGRAMS), OR GRP (GROUND PLANS)
IGNORED	THE PAGE WILL BE USED TO ENTER INFORMATION THAT SHALL NOT BE INCLUDED IN LISTS. COULD FOR INSTANCE BE MOUNTING INSTRUCTIONS, OR A FRONT PAGE FOR THE PROJECT.	IGN
UNIT DRAWING	THE PAGE IS USED FOR UNIT DRAWINGS, AND CAN FOR INSTANCE BE ASSIGNED ARTICLE DATA. YOU WILL BE ASKED TO SELECT AN EXISTING UNIT DRAWING (A .STD FILE). — READ MORE ABOUT THIS IN “INSERTING UNIT DRAWINGS” ON PAGE 568.	UNT
TABLE OF CONTENTS	THE PAGE IS USED AS A TABLE OF CONTENTS. ON PAGES OF THIS TYPE YOU CAN INSERT DATAFIELDS OF THE TYPE TABLE OF CONTENTS . PAGES OF THE TYPE TABLE OF CONTENTS CAN BE PLACED FREELY IN THE PROJECT. WHICH PART OF THE PROJECT THIS PAGE IS A TABLE OF CONTENTS FOR, IS SPECIFIED IN LISTS => LIST SETTINGS . — READ MORE ABOUT THIS IN “LIST SETTINGS FOR TABLES OF CONTENTS” ON PAGE 555. THE TABLE OF CONTENTS IS UPDATED WHEN YOU CHOOSE LISTS => UPDATE TABLE OF CONTENTS .	TOC
PARTS LIST	THE PAGE WILL BE USED AS A PARTS LIST PAGE. DATAFIELDS RELATED TO PARTS LISTS CAN BE INSERTED.	PAR
COMPONENTS LIST	THE PAGE WILL BE USED AS A COMPONENTS LIST PAGE. DATAFIELDS RELATED TO COMPONENTS LISTS CAN BE INSERTED.	COM
TERMINALS LIST	THE PAGE WILL BE USED AS A TERMINALS LIST PAGE. DATAFIELDS RELATED TO TERMINALS LISTS CAN BE INSERTED.	TER
CABLE LIST	THE PAGE WILL BE USED AS A CABLES LIST PAGE. DATAFIELDS RELATED TO CABLES LISTS CAN BE INSERTED.	CAB
PLC LIST	THE PAGE WILL BE USED AS A PLC LIST PAGE. DATAFIELDS RELATED TO PLC LISTS CAN BE INSERTED.	PLC

PAGE FUNCTION	DESCRIPTION	NAME IN PAGE MENU
DIVIDER	THE PAGE IS USED TO MARK THE START OF A NEW CHAPTER. THE TITLE OF THIS PAGE WILL BE DISPLAYED ON THE RIGHT-HAND SIDE OF THE SCREEN WHILE WORKING IN THE PROJECT, ALONG WITH THE OTHER DIVIDER TABS. A FOLDER WILL BE DISPLAYED IN FRONT OF THE PAGE TITLE IN THE EXPLORER WINDOW.	A FOLDER IS DISPLAYED
STANDARD/PROJECT FILE	APPLIED WHEN YOU WANT TO LOAD ANOTHER EXISTING PROJECT INTO THE PROJECT. THIS CAN EITHER BE AN ORDINARY PROJECT, OR A SO-CALLED <i>STANDARD PROJECT/STANDARD DRAWING</i> . — READ MORE ABOUT THIS IN “PROJECT TEMPLATES AND STANDARD PROJECTS” ON PAGE 511. IS ALSO USED FOR IMPORT OF DWG AND DXF DRAWINGS — SEE “DWG AND DXF FILES” ON PAGE 660.	NONE

OTHER PAGE FUNCTIONS

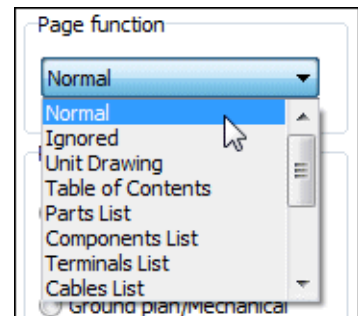
Furthermore, there are page functions for graphical terminal plans (**TEP**), graphical cable plans (**CAP**) and for graphical connection plans (**CNP**) and symbol documentation (**SYM**). Pages of this type are created in the modules for creating graphical lists / symbol documentation.

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



CHANGING THE PAGE FUNCTION

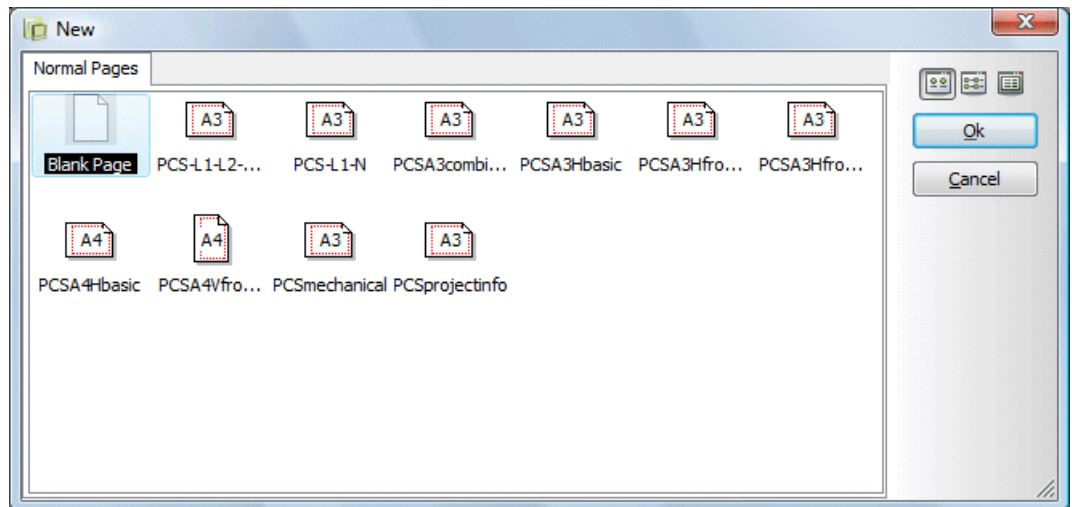
TO CHANGE THE *PAGE FUNCTION* FOR A PAGE, CHOOSE **SETTINGS => PAGE SETUP** AND CLICK THE DOWN-ARROW BELOW **PAGE FUNCTION**. A LIST OF POSSIBLE PAGE FUNCTIONS IS NOW DISPLAYED. NOW CLICK THE DESIRED PAGE FUNCTION.



You can see the *page function* of a page when you choose **Settings => Page setup**. — See “Page setup” on page 843.

Selecting a page template

When you have selected a **Page function**, you enter the dialog box **New**, from which you can choose between the created page templates, that matches the **Page function**, you selected:



Click on the desired template page, and click **OK**.

If you select the template **Blank page**, you get an empty project page with the selected page type.

**TEMPLATE PAGES FOR LISTS**

WHEN YOU HAVE SELECTED A **PAGE FUNCTION** FOR A LIST, YOU CAN, CORRESPONDINGLY, SELECT TO INSERT PAGES CONTAINING LIST DRAWING HEADERS.

— See how to create your own templates in “Creating Templates” on page 771, specially “Creating Page templates — via the Template editor” on page 772.

Click OK in the Page menu

You have now inserted the desired page, and are still in the **Page menu**.

When you click **OK** to leave the **Page menu**, you jump to the inserted project page.

COPYING, CUTTING AND PASTING MORE THAN ONE PAGE



To copy/cut and then paste more than one page in the **Page menu**, do this:

- 1) Click on the first page you wish to cut/copy.
- 2) Press down <Shift> and click on the last of the pages you wish to cut/copy.
- 3) Deselect/select further pages by pressing the <Ctrl> key when clicking on the pages.
- 4) Click on **Cut / Copy**.
- 5) When cutting: Click **OK** to cut the pages.
- 6) Click where you wish to paste the pages, and click **Paste**.
- 7) Click **OK** to paste the pages.
- 8) In the dialog box **Rename symbols**: Specify whether/how to rename the symbols on the pasted pages — see the options for this in “Intelligent copying of areas” on page 245.

Example on choices in the Page menu:

1) Click here 2) Press down <Shift> and click here (Choosing page I3-4) 3) Press <Ctrl> and click here (Deselect DIV) 4) Click on Copy and paste the pages

The copied pages can also be pasted after the last page in the project, if you click below the last page in the **Page menu** and choose **Paste**.

— See also “Pasting copied pages in other projects” on page 501.



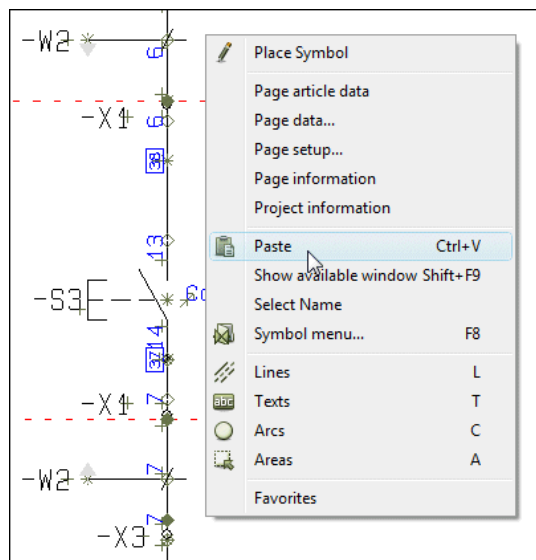
PAGE NAMES WHEN COPYING PAGES IN THE PAGE MENU

WHEN COPYING PAGES IN THE **PAGE MENU**, THE EXISTING PAGE NUMBERS ARE NOT CHANGED. THE COPIED PAGES KEEPS THEIR ORIGINAL PAGE NUMBERS, WITH AN ADDITIONAL COPY NUMBER INCLUDED IN BRACKETS. IF FOR INSTANCE PAGE **2** IS COPIED, THE COPIED PAGE IS ASSIGNED PAGE NUMBER **2 (1)**.

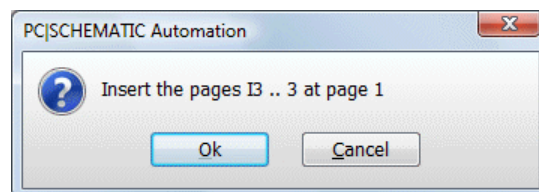
PASTING COPIED PAGES IN OTHER PROJECTS

To cut/copy pages from a project, and then paste these pages in another project, do the following:

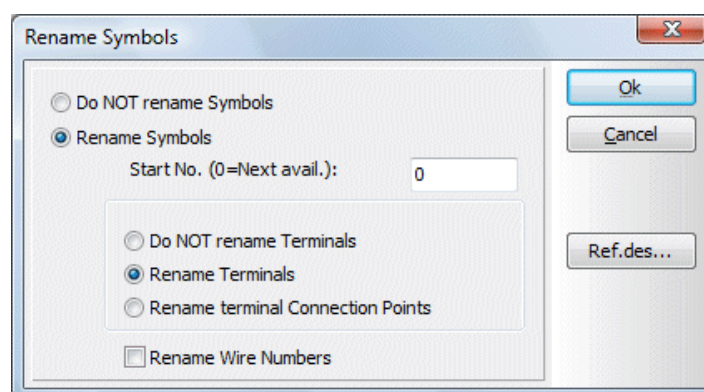
- 1) Copy the pages in the **Page menu** according to the common Windows guidelines, as described in “Copying, Cutting and Pasting more than one page” on page 500.
- 2) Open the other project.
- 3) Paste the pages via the **Page menu**, or by right-clicking on a project page, and then choose **Paste**.



- 4) Click **OK** to paste the pages:



- 5) Specify how to renumber symbols, terminals etc. and click **OK** — see “Intelligent copying of areas” on page 245:



6) The pages are now pasted into the other project, and the symbol names etc. have been updated for the pasted pages.

**COPYING CHAPTERS**

YOU CAN ALSO COPY FULL CHAPTERS USING *DRAG AND DROP*, AS DESCRIBED IN “MOVE, COPY AND DELETE PAGES AND CHAPTERS VIA PAGE TABS” ON PAGE 503.

— See “Copying full Projects into other Projects” on page 513, “Drag pages between projects” on page 517 and “Copying an area from one project to another” on page 518.

MOVE, COPY AND DELETE PAGES AND CHAPTERS VIA PAGE TABS

By using the page tabs, you can also move or copy project pages and chapters.

Moving pages by using the Page tabs

To move a single page in the project by using the page tabs, do this:

- 1) Select the page you wish to move, so that it becomes the active page in the project — e.g. by clicking its page tab at the bottom of the screen.
- 2) Click on the page tab, keep the mouse key pressed down, and *drag* the page to the page tab for the page in the project, where you wish to place the moved page. Release the mouse button.
- 3) Click **OK** to move the page.

The moved page is placed *before* the page, whose page tab you clicked on. The moved page can also be inserted after the last page in the project.

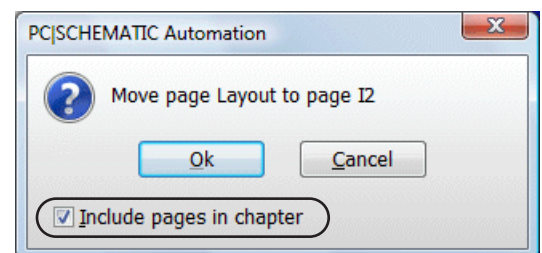
You can also move individual pages by right-clicking on the page tab, then choose **Cut**, and right-click on another page tab and choose **Insert**. You can also copy a single page by clicking on its page tab, and then choose **Copy**.

Move chapters using the page tabs

To move a chapter in the project using the page tabs, you just move the page tab for the **Divider** page, that indicates the start of the chapter.

When you answer **OK** to move the Divider page, just select **Include pages in chapter** *in chapter*:

Hereby all pages in the chapter are also moved.



Copy pages and chapters using the page tabs

You copy pages and chapters just like described above; the only difference is that you press the <Ctrl> key while doing it. When you copy pages/chapters, a small + is displayed in the cursor.

When you copy pages, you are asked whether/how to rename the symbols on the page — and change reference designations (if any) —, as described in “Intelligent copying of areas” on page 245.

Delete pages via the page tabs

You can delete an individual page by right-clicking on the page tab for the page, and choose **Delete**.

CHAPTERS

In **Automation** you can divide projects in chapters, which gives a better survey. When a project is divided in chapters, the divider/chapter tabs are displayed on the right-hand side of the screen. When you click on one of these divider tabs, the first project page in the chosen chapter is displayed.

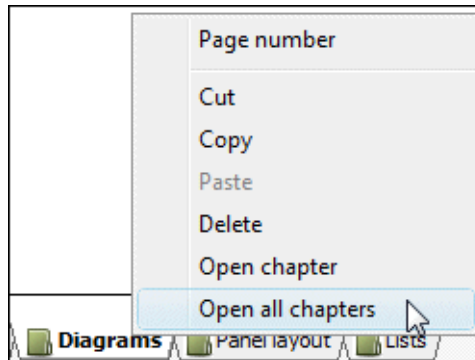
For chapters, it is possible to:

- 1) Create separate tables of contents for each individual chapter in the project — see “List settings for Tables of Contents” on page 555.
- 2) To change reference designations on chapter level — see “Editing reference designations for chapters” on page 358.
- 3) To move/copy all pages in the chapter in a single operation — see “Move, Copy and Delete Pages and Chapters via Page tabs” on page 503.



Chapter tabs at the bottom of the screen

Chapter tabs are also displayed at the bottom of the work area along with the tabs for the other project pages. When you right-click on one of these chapter tabs, you get a set of options. You can for instance choose to close all chapters, so that only the chapter tabs are displayed, or you can open all chapters, so that the individual project pages in the chapters are also displayed.



Creating a new chapter

The only thing you have to do to create a new chapter, is to insert a divider page, which indicates the start of a new chapter.

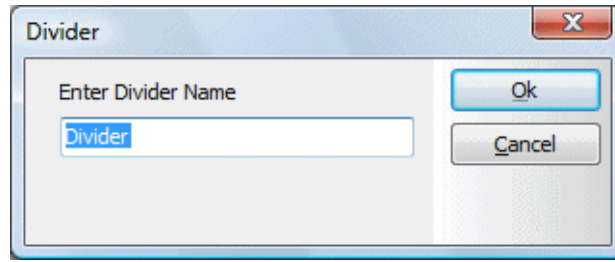
The contents of the chapter is the pages between this divider tab and the next divider tab.

To insert a divider tab, do this:



- 1) Enter the **Page menu**, click on the page in the menu where you want the chapter to start, and click **Insert**.
- 2) In the dialog box **Page function**: Click on **Divider** and click **OK**.
- 3) In the dialog box **New**: Click on a page template, and click **OK**.

4) In the dialog box **Divider**: Type in the name of the divider/chapter:



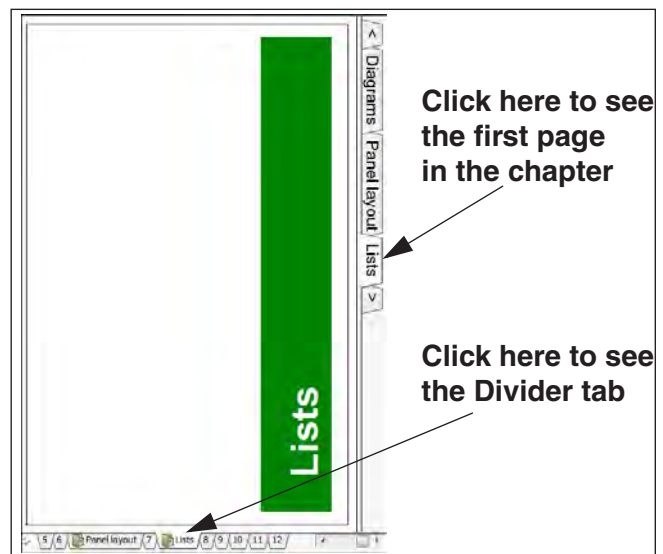
5) Click **OK**.

You have now created a new chapter.

Inserting chapter drawing headers

You can now insert a drawing header on the page, as described in “Insert drawing headers/formulas” on page 531.

Here you can either choose to use the included chapter divider drawing header **Divider** in the folder **Misc**, or you can design your own drawing header as described in “Creating Drawing Headers” on page 742.



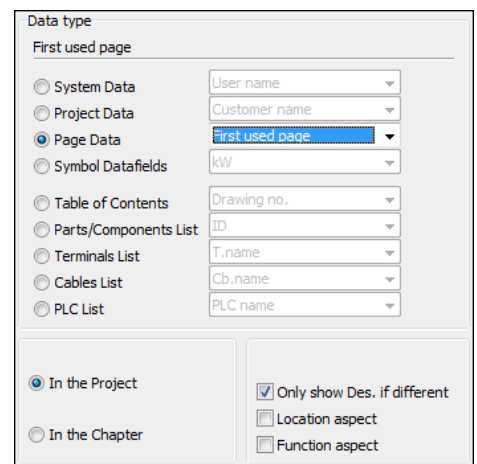
DRAG IN A CHAPTER DIVIDER PAGE FROM THE EXPLORER WINDOW

YOU CAN ALSO *DRAG* A CHAPTER DIVIDER PAGE INTO THE PROJECT FROM THE **EXPLORER** WINDOW — AS DESCRIBED IN “DRAG A NEW PAGE INTO THE PROJECT - VIA THE EXPLORER WINDOW” ON PAGE 495.

Datafields on chapter level

For the datafields **First used page**, **Last used page** and **Used pages**, you can select whether these shall cover the full project, or just the chapter pages.

— Read more about inserting datafields in “Datafields” on page 110.

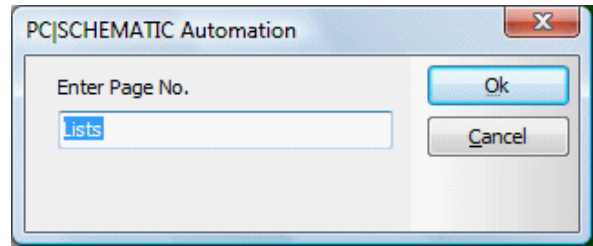


NAMING PAGE TABS

You are free to name all page tabs in a project as you please. However, you are *not* allowed to assign the same name to two different page tabs.

When you want to edit the name of a page tab or a divider tab, just double-click on the tab. You can then edit the name of the tab:

Type in the new name and click **OK**.



You can also rename one or more of the pages by selecting the page/pages in the **Page menu** and click on **Page number**, as described in “The Page menu” on page 493.

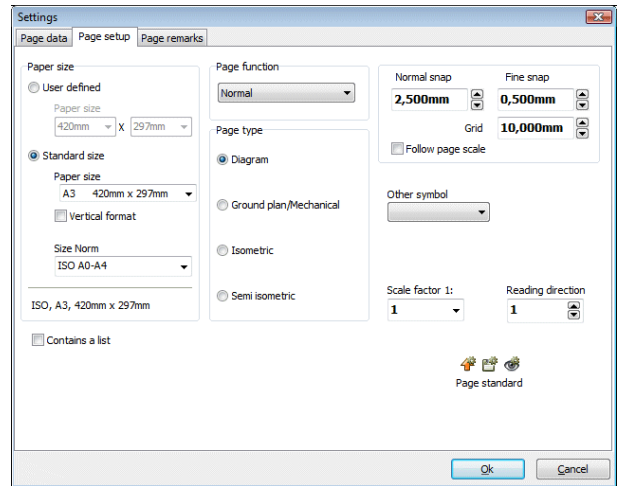
You can also rename a page by right-clicking on the page tab, and choose **Page number**.

PAGE SETUP - AND TRANSFERRING THIS TO OTHER PAGES

When you select **Settings => Page setup**, you enter the dialog box **Page setup**.

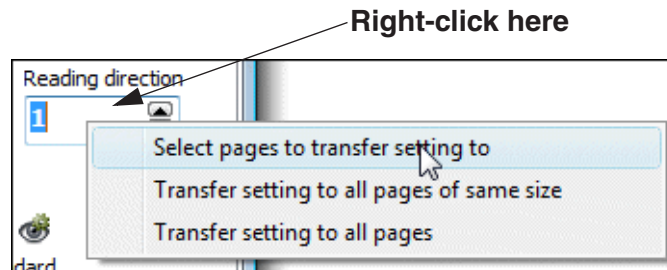
You can also enter this dialog box by right-clicking on the active project page, and choose **Page setup**.

In the dialog box **Page setup** you can for instance specify the page type and the paper size, as described in “Page type” on page 508 and in “Paper size” on page 509. — See more options in “Page setup” on page 843.

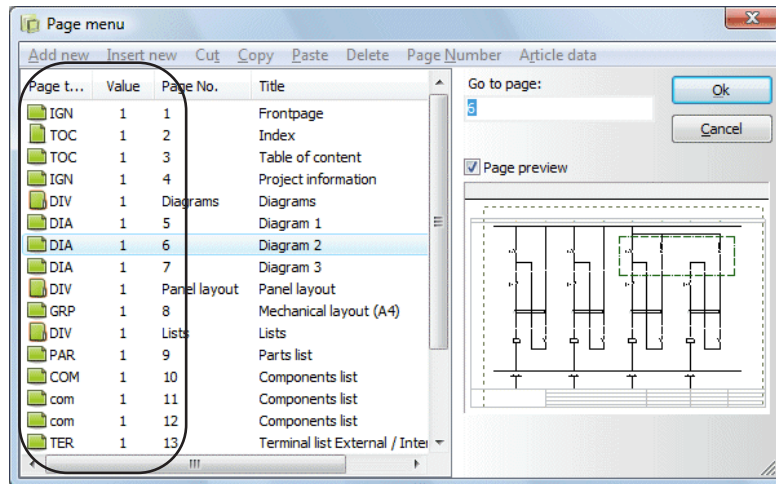


Transferring the Page setup to other project pages

To transfer a page setup from a project page to other project pages, you can right-click on some of the settings in the **Page setup** dialog box — for instance **Reading direction** as illustrated below —, which gives you the option of transferring this setting to other project pages:



If you choose **Select pages to transfer setting to**, you enter a special version of the **Page menu**, where you select which pages to transfer the setting to:



Here the green icons indicate which pages the setting can be transferred to. If an icon is dimmed, the page already has this setting — or the setting is not possible for the page. The settings for the other pages are displayed in the **Value** column. Choose the pages to transfer the setting to, and click **OK**.

Page settings that can be transferred to other pages

It is possible to transfer the following Page settings to other project pages, according to the procedure described above:

PAGE SETUP	REMARKS
READING DIRECTION	SEE THE DESCRIPTION ABOVE.
PAGE FUNCTION	PAGES, WHICH ARE NOT OF THE TYPE NORMAL , CAN HERE BE ASSIGNED THE TYPE NORMAL . THE FUNCTION IS ONLY AVAILABLE VIA THE RIGHT-CLICK MENU, WHEN A PAGE HAVE BEEN CHANGED TO THE TYPE NORMAL .
LISTTYPE	THIS FUNCTION IS ONLY AVAILABLE VIA A RIGHT-CLICK MENU, WHEN A LIST PAGE HAS BEEN CHANGED TO THE TYPE NORMAL .
CONTAINS A LIST	THE SETTING, THAT A DIAGRAM PAGE ALSO CONTAINS A LIST, CAN ALSO BE TRANSFERRED TO OTHER DIAGRAM PAGES. — SEE “DIAGRAM PAGES - CONTAINING A PARTS OR COMPONENTS LIST” ON PAGE 762,

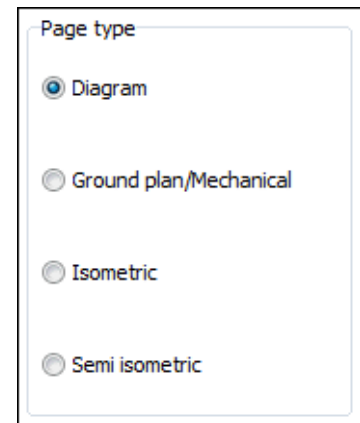
PAGE TYPE

The type of a page is determined in **Settings => Page setup** and in the lower part of the left tool bar.

— See “Direct access to menus and Tabs” on page 48.

The **Page type** can be changed any time.

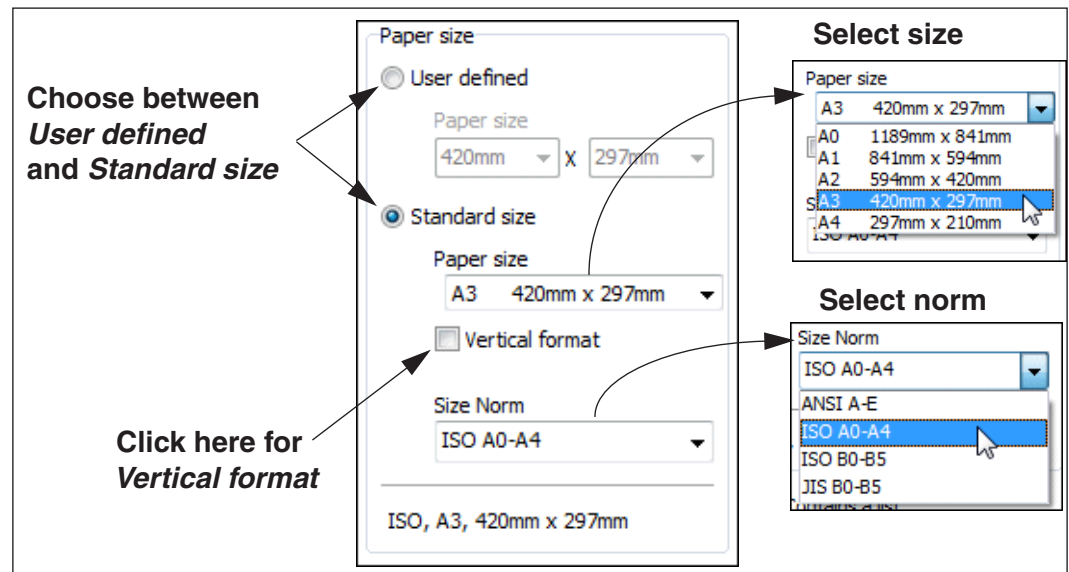
In the list below you can see what the various page types are used for:



PAGE TYPE	DESCRIPTION
DIAGRAM	THE PAGE IS A STANDARD DIAGRAM FOR DRAWING IN TWO DIMENSIONS. USED FOR ALL TYPES OF LISTS AS WELL.
GROUND PLAN/ MECHANICAL	THE PAGE IS USED FOR GROUND PLAN- AND MECHANICAL DRAWINGS, WHERE WORKING IN DIFFERENT ELEVATIONS, AND THEREBY WORK IN THREE DIMENSIONS, IS POSSIBLE.
ISOMETRIC	THE PAGE CAN BE USED FOR ISOMETRICALLY DRAWINGS, OR DISPLAY IN X, Y OR Z.
SEMI ISOMETRIC	THE PAGE IS A KIND OF ISOMETRIC PAGE, WITHOUT PURELY ISOMETRIC ANGLES.

PAPER SIZE

Under **Settings => Page setup** you can also choose the size of the page:



Basically, you can choose between **User defined**, where you type in the measures of the page, and **Standard size**, where you select a norm, and a paper size in this norm.

All pages can be either horizontal or vertical. To get a vertical page, you must select **Vertical format**.

On choosing norms for page formats

As it is, you can choose between the page format norms ANSI (American standard), ISO and JIS (Japanese standard). However, it is easy to add new page format norms to the list. Please contact PC|SCHEMATIC A/S, or your local support service, for further information.

Standard page sizes for List and Chapter tabs



If you are on a list page, and select **Settings => Page setup**, and choose **Save as default**, the size of this list page becomes the default page size for list pages.

The same is the case for chapter tabs and diagram pages.

GROUND PLAN DRAWING

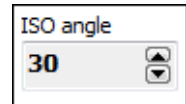
— See a practical example on how to create ground plan drawings in the “Creating a Ground Plan” on page 471.

ISOMETRIC DRAWING

Beyond drawing on an isometric page, you can also use this page type to see whether a ground plan page, where three dimensions are used, seems reasonably drawn.

If you have drawn a three dimensional ground plan, and wish to check it isometrically, you choose **Settings => Page setup**. Here you click on **Isometric** instead of **Ground plan**, and click **OK**.

When you choose **Isometric** you can also specify an **ISO angle**. Click on the up- and down arrows to change the ISO angle.



If you choose **Semi isometric** you can specify another angle accordingly.

Click **OK**.

Centre drawing

If your drawing hereby disappear completely or is halfway out of sight, you choose **Functions => Centre drawing** and get the isometric drawing placed on the middle of the screen. You can now check out whether it looks reasonable or not.

To carry on drawing on your ground plan, you choose **Ground plan** again in the **Page settings**.

If you are working on a **Ground plan** page and choose **Functions => Generate isometry**, an isometric drawing will be generated, and added to the end of the project.

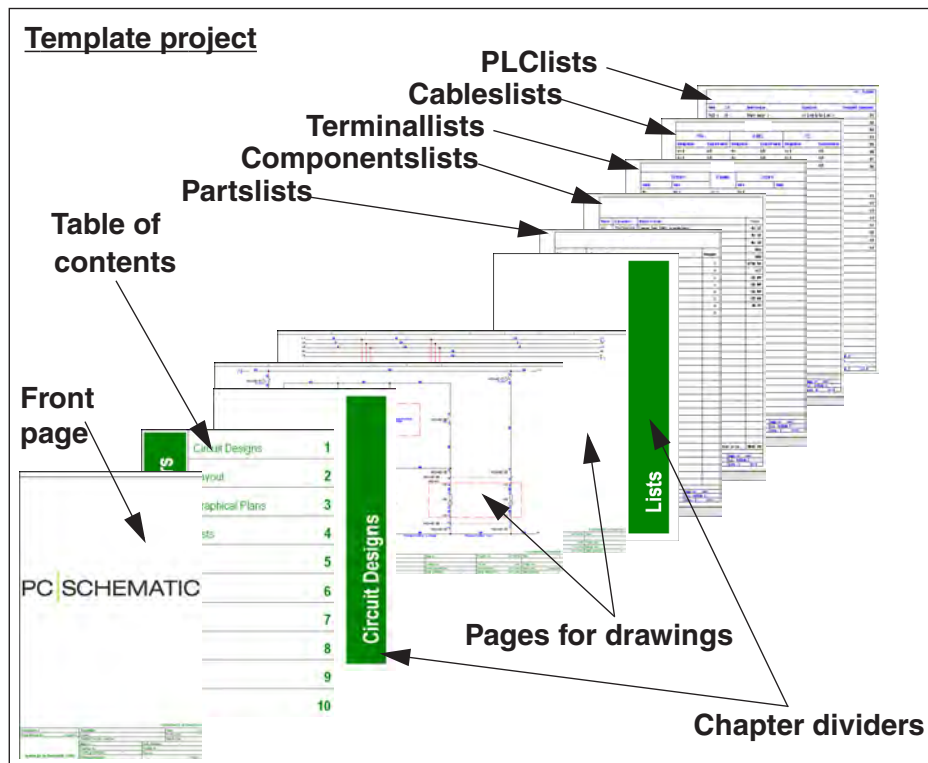
PROJECT TEMPLATES AND STANDARD PROJECTS

This chapter shows you how to start a new project using an already created standard project.

THIS CHAPTER DESCRIBES	
— WHAT IS A TEMPLATE PROJECT.....	P. 511
— USING TEMPLATE PROJECTS.....	P. 512
— ADDING LISTS TO A PROJECT USING TEMPLATE PROJECTS.....	P. 512

WHAT IS A TEMPLATE PROJECT?

To get started as quickly as possible when you create new projects, you can create a so-called template project, which contains all the pages you need every time you start up a new project.

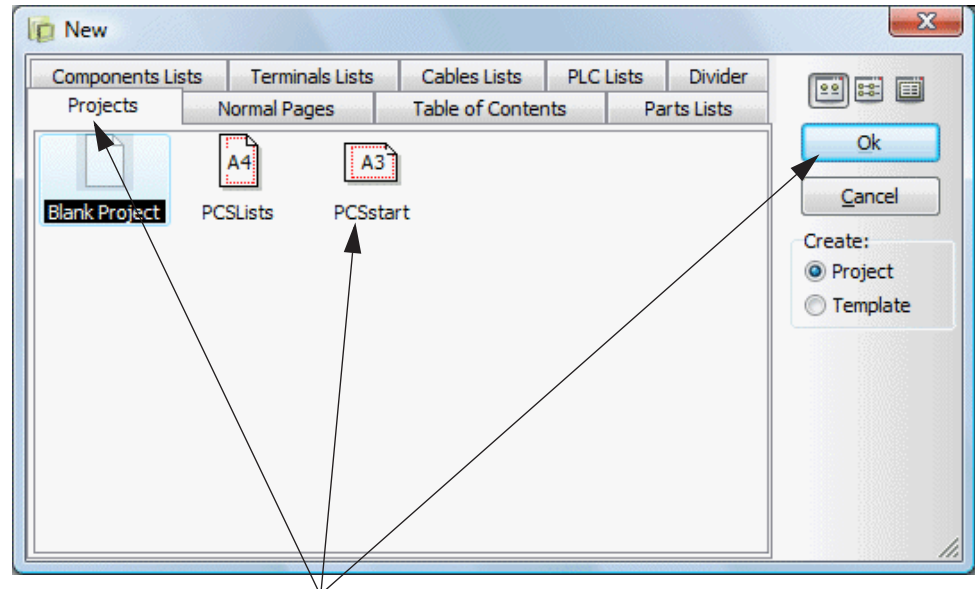


This way you can build a library of different project templates, which can be used when starting different types of projects, or for instance for projects for different customers.

USING PROJECT TEMPLATES

To create a new project based on a project template, do this:

- 1) Choose **File => New**.
- 2) In the dialog box **New**: Click on the tab for **Projects**, click on a project on the tab, and click **OK**.



Click here

- 3) You can now start working on the new based on the project template.

— See how to create project templates in “Creating Templates” on page 771.

ADDING LISTS TO PROJECTS USING STANDARD PROJECTS

When you know that there are a collection of pages — for instance a set of different types of lists — which you need to insert frequently in active projects, you can create a project containing these pages.

Such a project is called a *standard project* in **Automation**.



This standard project can be copied into the active project from the **Page menu**, as described in “Copying full Projects into other Projects” on page 513.

COPYING FULL PROJECTS INTO OTHER PROJECTS

When you are working on a project, you can load another project into this project via the **Page menu**.

THIS CHAPTER DESCRIBES	
— HOW TO COPY ONE PROJECT INTO ANOTHER	P. 513
— PROJECT DATA WHEN COPYING PROJECTS	P. 515

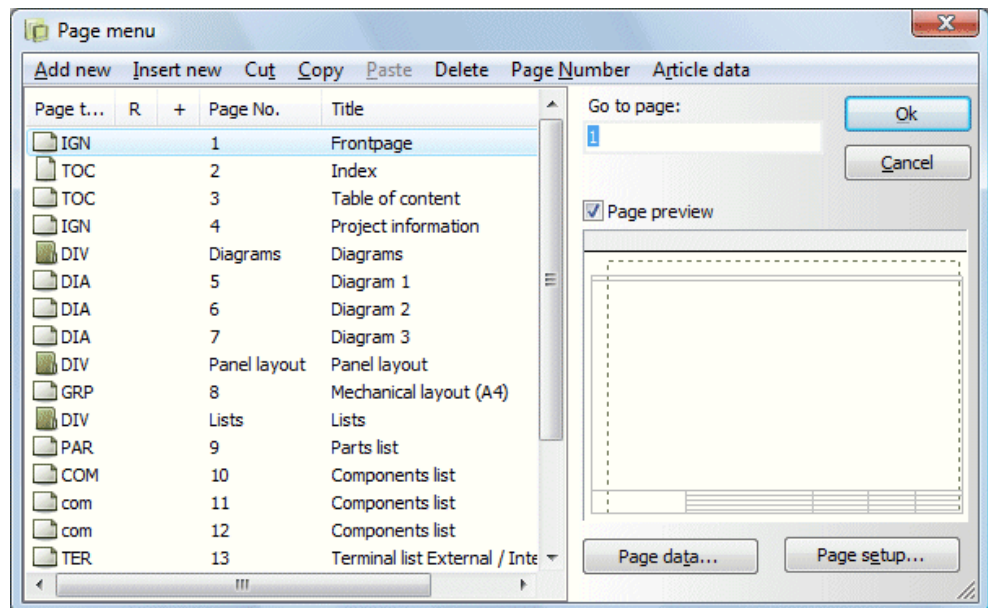
You may wish to read:

- “Project templates and Standard projects” on page 511, for information about Template projects and Standard projects, which can contain the pages you need when you start new projects.
- “Merging projects via Excel” on page 625, to see how to merge **Automation** projects via Microsoft Excel.
- “Drag pages between projects” on page 517, to see how to *drag* pages from one project to another.

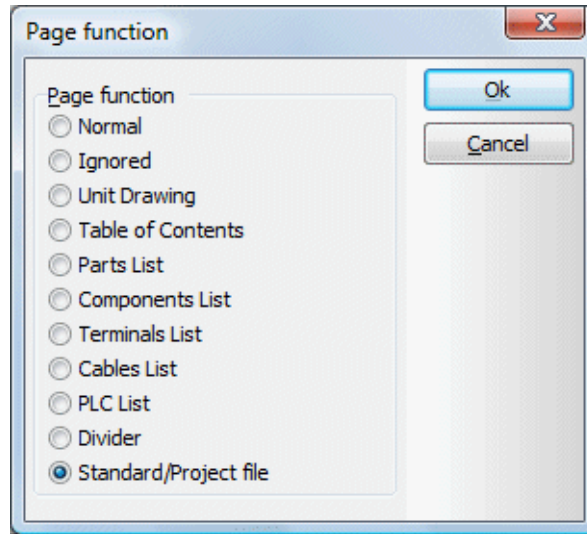
HOW TO COPY ONE PROJECT INTO ANOTHER

In **Automation** you can load any project into an existing project as a **Standard/Project file**, and then adapt the loaded project pages according to your needs in the active project.

To try how to load a project into another project, just open a project, and enter the **Page menu**.



Here you click **Add new** — to add a new project/page in the end of this project — and then click on **Standard/Project file** and click **OK**.



— Instead of **Add new** you could also click on for instance page four and then click **Insert new**. The project would then be inserted starting on page four.

You now enter the dialog box **Open** dialog box, and have to type in which project you wish to insert. In this case you could open the folder **Project**, select **plcdemo.pro**, and click **OK**.

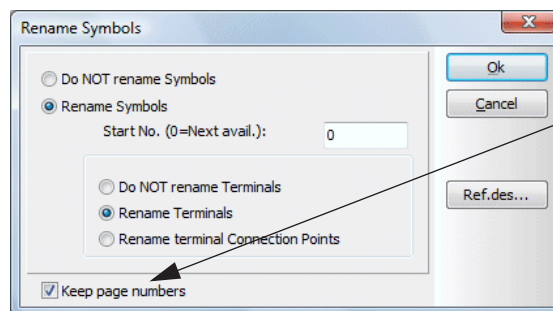
Defining new datafields when inserting projects

If the new project contains datafields, which have not been defined in the active project, you are asked whether to define these datafields in the project.

Click **Yes** to define the datafields in the project file.

Renaming symbol names and Page numbers when inserting projects

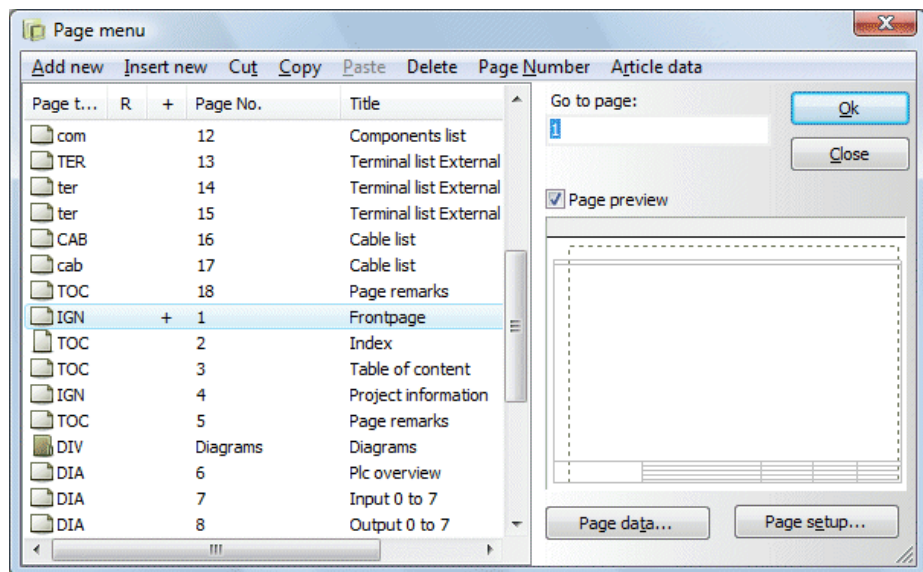
You will now be asked whether you wish to rename the copied symbols:



Click here **not** to change page numbers in the inserted project

Select **Rename symbols**, **Rename terminals**, and select **Keep page numbers**, if you wish to keep the original page numbers from the inserted project. Click **OK**.

The selected project is now inserted into the original project:



You have now loaded an entire project into another project.

If you now wish to rearrange the pages in the project, delete pages and so on, you can do this as described in “The Page menu” on page 493.

PROJECT DATA WHEN COPYING PAGES

When you load a new project into the original project, these pages will have exactly the same *Project data* as the original project. This is because the Project data only is specified in one place for the entire project.

Read more about Project data in “Project data” on page 831.

SEVERAL PROJECTS SIMULTANEOUSLY

In **Automation** you have got the option of working on several projects at the same time. In practice you should not meet any limitations on how many projects you can open; even if the projects consist of several thousand pages each.

THIS CHAPTER DESCRIBES	
— DISPLAYING MORE THAN ONE PROJECT AT A TIME	P. 516
— DRAGGING PAGES/CHAPTERS BETWEEN PROJECTS	P. 517
— COPYING AN AREA FROM ONE PROJECT TO ANOTHER	P. 518
— DRAWING HEADERS AND COPYING	P. 521
— OPENING A NEW PROJECT BY CLICKING ON A SYMBOL	P. 521
— OPENING A UNIT DRAWING	P. 522
— WORKING WITH THE SAME PROJECT IN TWO VERSIONS	P. 522
— COPYING TO OTHER PROGRAMS	P. 523

You can also see how to create overview projects, with links to other projects in “Overview projects” on page 524.

If you are interested in how to copy an entire project into another, please read “Copying full Projects into other Projects” on page 513.

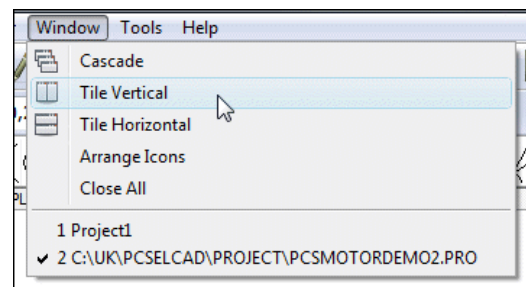
— See “Pasting copied pages in other projects” on page 501 also.

DISPLAYING MORE THAN ONE PROJECT AT A TIME

When you wish to see more than one project on the screen, first choose the menu point **Window**.

You can then choose between:

- a) **Cascade**,
- b) **Tile vertical** and
- c) **Tile horizontal**.



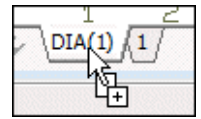
When you choose **Window => Close All**, all projects in the program are closed. If you have made changes, which have not been saved, you will get the option of saving these.

If you have minimized projects displayed on the screen, and choose **Window => Arrange Icons**, all minimized projects are gathered at the bottom of the screen.

DRAG PAGES BETWEEN PROJECTS

To *drag* a page from one project to another, you open both projects and choose **Window => Tile**, which places the projects side by side.

Then click on the page you want to copy, keep the mouse button pressed down, and *drag* the page over to the other project. You first release the mouse button when you are pointing on the tab, for the page in the project, where you want the copied page to be placed. (A small + is displayed in the cursor when you copy pages).



Then answer **Yes** to copy the page, and answer the emerging question whether — and how — to rename the symbols copied. The page copied is placed immediately *before* the page you clicked on in the project.

The copied pages can also be inserted after the last page in the project.



If necessary you can then rearrange the order of the pages in the **Page menu**. — See “The Page menu” on page 493.

For information on how to change back and forth between open projects, please refer to “Switching between windows” on page 25.

— See also “Pasting copied pages in other projects” on page 501.



COPYING FULL CHAPTERS BETWEEN PROJECTS

YOU CAN ALSO *DRAG* FULL CHAPTERS FROM ONE PROJECT TO ANOTHER, BY DRAGGING THE PAGE TAB FOR THE CHAPTER DIVIDER PAGE TO ANOTHER PROJECT — OR BY USING THE SAME PROCEDURE AS DESCRIBED ABOVE.

— SEE ALSO “MOVE, COPY AND DELETE PAGES AND CHAPTERS VIA PAGE TABS” ON PAGE 503.

COPYING AN AREA FROM ONE PROJECT TO ANOTHER

This chapter describes how to copy a drawing — or parts of a drawing — from one project to another. In this example, you copy from the project **Pcsmotordemo2** to a new project. If you will rather use your own projects, please do so.

If you want to follow the example, it is a good idea to close the project you are working on for the moment, choosing **File => Close**.

Then do the following:

- 1) Choose **File => New**, click on the project template **PCSstart**, and click **OK**.
- 2) In the dialog box **Settings**: Click **OK**.
- 3) Choose **File => Open**, open the project **Pcsmotordemo2.pro** in the **Project** folder, and click **OK**. You now have two projects open.
- 4) Select **Window => Tile horizontal** to see the projects side by side.



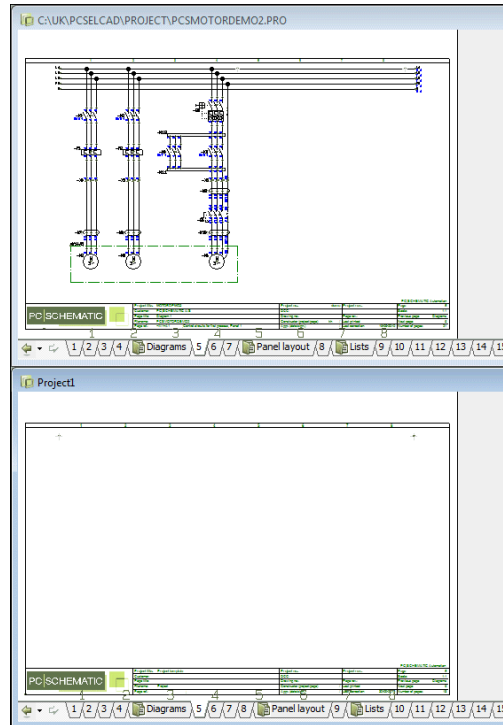
CHANGING FROM PROJECT TO PROJECT

TO CHANGE BACK AND FORTH BETWEEN THE PROJECTS, JUST CLICK SOMEWHERE WITHIN THE WINDOW YOU WANT TO WORK ON. THE BAR AT THE TOP OF THE ACTIVE WINDOW WILL THEN BECOME HIGHLIGHTED. YOU CAN ALSO CHANGE BACK AND FORTH BETWEEN THE WINDOWS BY PRESSING DOWN <CTRL> WHILE PRESSING <TAB>.

- 5) Now first click on one of the windows, and click on the page tab for the page you wish to copy *from* — in this case page **5** in the project **Pcsmotordemo2**.
- 6) Then click on the other window, and click on the page tab for the page you wish to copy *to* — here page **5** in the new project. (When there is no page to copy to in your project, you can insert a new page as described in “Pages and Chapters” on page 491).



- 7) Before copying the diagram in the **Pcsmotordemo2** project, you can, if you wish to, **Zoom** in on the area you wish to copy — see “Zoom” on page 42 if necessary.



- 8) Click on the **Area** button, and use the mouse pointer to select up the area you wish to copy — see “Selecting objects” on page 55 if necessary. — If you zoom in on the area you wish to copy, this will not effect the size of the area you insert in the other project. This is because *zoom* only has to do with what is displayed on the screen in the actual project, and does not change the measures/scaling of the objects.



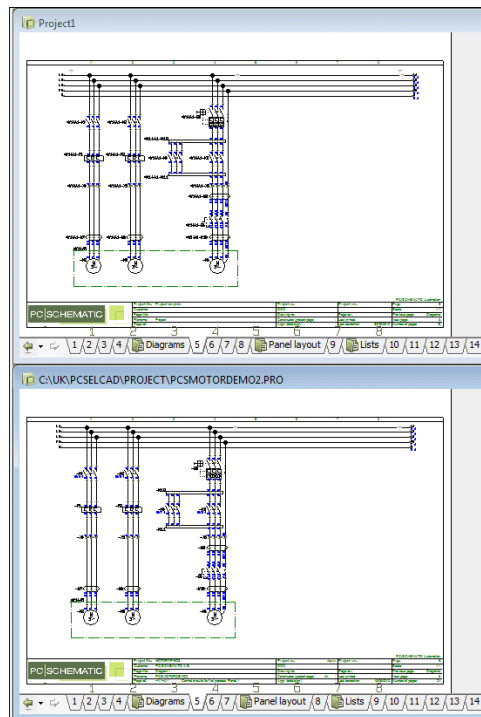
- 9) Click the **Copy** button.



- 10) Now click inside the other window, click the **Area** button if not activated, and click the **Paste** button (short-cut key <Ctrl+v>). You now have the selected area in the cross hairs in the project you wish to copy *to*. Click where you wish to insert the area on the page.

- 11) You are now asked whether you want to rename the copied symbols. Specify how you wish to rename the symbols — see “Intelligent copying of areas” on page 245 — and click **OK**.

12) The selected area is now copied from the one project into the other.



13) Press <Esc> not to place the drawing once again.

14) When you return to the project you copied *from*, you will have the selected area in the cross hairs once again. Therefore, not to place the selected area in this project, it is a good idea to change windows by clicking *on the title bar of the window*, and then press <Esc>.



SETTINGS ARE MEMORIZED FOR THE DIFFERENT PROJECTS

PLEASE NOTE THAT THE PROGRAM REMEMBERS ALL SETTINGS CONCERNING THE INDIVIDUAL PROJECTS AS YOU CHANGE BACK AND FORTH BETWEEN THE VARIOUS PROJECTS. WHEN FOR INSTANCE YOU ARE WORKING WITH SYMBOLS IN ONE WINDOW, AND TEXTS IN ANOTHER, THE PROGRAM WILL REMEMBER THIS. THUS WHEN YOU CLICK ON THE FIRST WINDOW YOU WILL SEE THE *SYMBOLS* BUTTON ACTIVE, AND WHEN YOU CLICK ON THE OTHER YOU WILL SEE THE *TEXTS* BUTTON ACTIVE.

DRAWING HEADERS AND COPYING AREAS

When you copy areas, you cannot copy the drawing headers on the page you copy from. But you can choose **Settings => Page data** any time, and determine which drawing header you want to insert on the page you copy to.

Read more about how to insert drawing headers in “Insert drawing headers/formulas” on page 531.

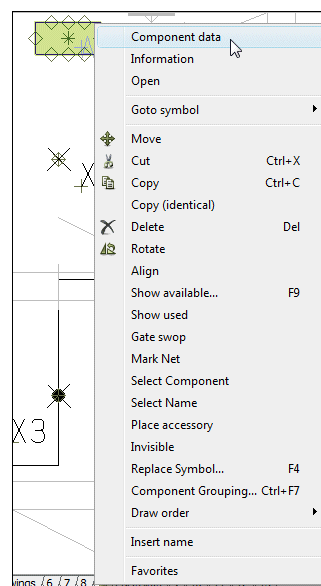
If you copy/insert an entire project into another, drawing headers will also be inserted/copied. Read more about this in “Copying full Projects into other Projects” on page 513.

OPENING NEW PROJECTS BY CLICKING ON SYMBOLS

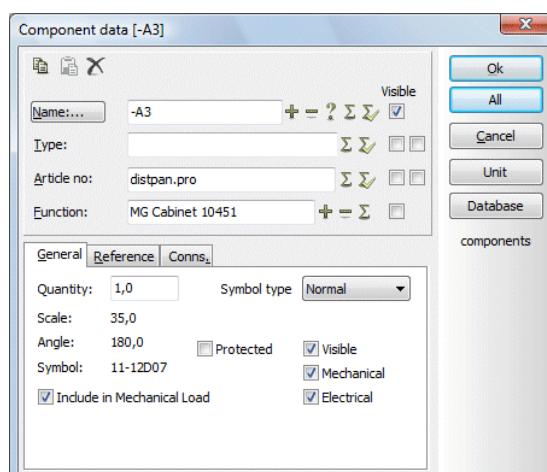
If for instance you use a distribution panel, and already have an existing project where the distribution panel is documented, it is possible to open this project by clicking on the symbol.

To achieve this, you do the following:

Place a symbol for the distribution panel on the drawing. — If you do not have a suitable symbol, you can see how to create one in “Creating Symbols” on page 696.



Right-click on the symbol, and choose **Component data**.



In the **Article no** field you type in the full path to the project in which the switchboard is documented. If the project is documented in the folder specified for projects in **Settings => Directories**, you only have to type in the name of the project file itself. Therefore it only says *distpan.pro* in the dialog box above. Read more about this in “Directories” on page 840.

Click **OK**.

You have now established a connection between the symbol for the distribution panel, and the project in which it is documented.

If you then wish to open the project, in which the distribution panel is documented, you just have to right-click on the symbol, and then choose **Open**.

Survey of your projects

You can also use this function to create a project, which you only use to get a survey of your projects. Each symbol you place in this survey, then represents an individual project.

If you attach a project to each symbol — as described above — you can then start the projects directly from this survey.

OPENING UNIT DRAWINGS

If you wish to open one of the unit drawings in your projects when you choose **Open**, then click on **Detail** in the dialog box **Component data**, and select the relevant drawing.

Read more about this in “Attach a symbol on a diagram to the unit drawing” on page 568.

WORKING WITH THE SAME PROJECT IN TWO VERSIONS

It is possible to have a project file open in several versions at the same time. If the project for instance is called **demo.pro**, the first version of the project will be named **demo.pro:1**, the second will be named **demo.pro:2** and so on.

The program will give you a warning if you open the same project twice.



KEEP TRACK OF WHICH IS THE ORIGINAL PROJECT

PLEASE NOTE, THAT IT IS *VERY IMPORTANT* TO KEEP TRACK OF WHICH PROJECT IS THE ORIGINAL, IN ORDER NOT TO SAVE THE WRONG PROJECT IN STEAD OF THE ORIGINAL.

IF YOU CLICK ON **SAVE** FOR VERSION **:2** (OR LATER) OF A PROJECT, YOU ENTER THE DIALOG BOX **SAVE AS**.

COPYING TO OTHER PROGRAMS



If you want to copy an area of the active project into another program, then click on the **Area** button, drag a window around the area, and choose **Edit => Copy to Clipboard**.

If you want to use the area you copied in for instance **Word**, you just click on the **Paste** button in Word, and the copied area will be inserted as an image.



ELECTRICAL INFORMATION IS NOT TRANSFERRED

PLEASE NOTE, THAT ELECTRICAL INFORMATION AND ARTICLE DATA IS *NOT* TRANSFERRED THIS WAY. YOU ONLY TRANSFER THE DRAWING ITSELF.



By clicking on for instance the **Symbols** button, you only copy the symbols in the area you mark up.

If you want to copy areas — or files — *from* other computer programs to **Automation**, please refer to “Inserting files from other programs / OLE” on page 667.

OVERVIEW PROJECTS

In **Automation** you can attach projects to the symbols in the diagram pages. You can use this option for creating overview projects containing links to all of the projects that are included in a specific assignment. At www.pcschematic.com/videos you can watch the video "Large projects", where these options are shown.

THIS CHAPTER DESCRIBES	
OPENING NEW PROJECTS BY CLICKING SYMBOLS.....	P. 524
OVERVIEW OF YOUR PROJECTS	P. 525
OPENING PROJECTS/PAGES FROM REFERENCE FRAMES.....	P. 526

OPENING NEW PROJECTS BY CLICKING SYMBOLS

If for instance you use an electrical panel in an electrical installation drawing, and you already have created a project for the panel, it is possible to open the panel project by clicking the panel symbol.

To make this possible, all you have to do is this:

- 1) Place a symbol for the electrical panel in the drawing. — In case you do not have an appropriate symbol, see how to create one in “Creating Symbols” on page 696.
- 2) Double-click on the symbol — or right-click and choose **Component data**.
- 3) In the dialog box **Component data**: Type in the full path to the project in the **Article no** field. However, if the project is placed in the same folder as the active project, you only need to type in the file name.
- 4) Click **OK**.

You have now established a connection between the symbol for the electrical panel and the project that it is documented in.

To open the project that the panel is documented in, all you have to do, is to right-click the panel symbol and choose **Open**.

OVERVIEW OF YOUR PROJECTS

You can also use this function for creating a project which you use exclusively for creating an overview of your projects. Each symbol that you place in this project, represents an individual project. If you then attach a project to each symbol — as described above — you can then start any of your projects directly from this overview project.

OPENING PROJECTS/PAGES FROM REFERENCE FRAMES

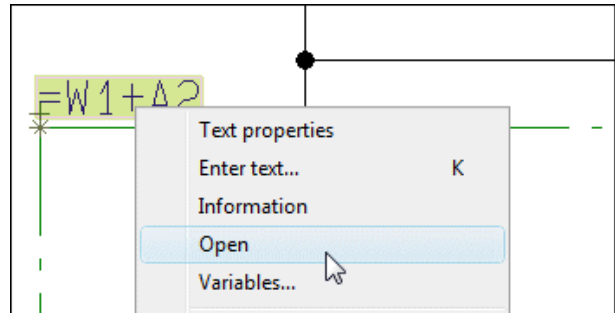
It is also possible to open projects from reference frames.

When you right-click on the *text* in a reference frame and choose **Open**, this happens:

- a) If the project contains pages with the reference designation: The first page in the project with this reference designation is opened.

- b) If the project *does not* contain pages with the selected reference designation:

Automation looks for a project with the name of the reference designation, and opens the project if it exists in the same folder as the active project. In the figure above, the project should therefore carry the name **=W1+A2.pro** in order to be found and opened.



INSERTING LOGOS IN PROJECTS

THIS CHAPTER DESCRIBES

- ON PLACING LOGOS IN PROJECTS P. 527
- INSERTING A LOGO IN A PROJECT..... P. 527

ON PLACING LOGOS IN PROJECTS

When you wish to apply a logo in a **Automation** project, you specify which logo you wish to apply for all of the project. This way the logo only takes up file space once in the project file. The logo will be saved together with the project file.

The logo must be a ***.bmp** file, a ***.jpg** file, an ***.emf** file or a ***.wmf** file.

INSERTING A LOGO IN A PROJECT

To insert logos in projects in **Automation** do this:

Specifying the logo file

1) Place the logo in an appropriate folder using the Windows Explorer.



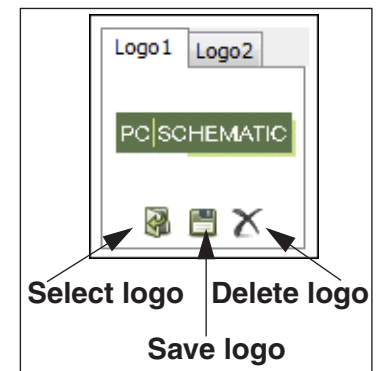
2) Click the **Project data** button.

3) In the dialog box **Project data**: Click on the tab for **Logo1** or **Logo2** in the right-hand side of the dialog box.



4) Click on the **Select logo** button under **Logo** in the right-hand side of the dialog box.

5) In the dialog box **Open**: Find the logo, specify whether to insert it directly in the project (**Insert**) or whether to create a link to the logo (**Link to file**), and click **OK**.



Place the logo using a Datafield on a project page

6) Choose **Insert => Insert Datafield**.

7) In the dialog box **Datafield**: Click on **Project Data**, choose the datafield **Logo**, and click **OK**.

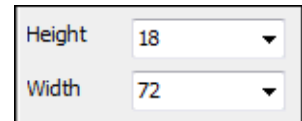
8) You get the logo in the cross hairs: Click where you wish to place the logo. Press <Esc> to remove the logo from the cross hairs.

Adapting the size of the logo



9) The logo can be very small, and its size must now be adapted: Select the logo datafield, and click on the **Text properties** button.

10) In the dialog box **Text properties**: Type in the size of the logo in the field **Height** — the width is set automatically — and click **OK**.



A screenshot of a dialog box titled "Text properties". It contains two rows of controls. The first row is labeled "Height" and has a text input field containing the number "18" and a downward-pointing arrow. The second row is labeled "Width" and has a text input field containing the number "72" and a downward-pointing arrow.

11) Press <Esc> — to deselect the logo.



Lists

In this section you can see how to use lists.

You can read about:

- Inserting drawing headers/formulas..... p. 531
- Specifying list settings p. 542
- Component Grouping..... p. 556
- Inserting unit drawings p. 568



Read the Survey article

Read the survey article on working with databases and lists at www.pcschematic.com/articles.

See also “Exporting List Files” on page 649.

INSERT DRAWING HEADERS/FORMULAS

Drawing headers are a kind of formulas consisting of text and graphics, which you can insert on the pages in your projects.

THIS CHAPTER DESCRIBES	
— WHAT A DRAWING HEADER CAN CONSIST OF	P. 532
— PLACING A DRAWING HEADER ON A PAGE	P. 532
— ADDING A NEW PAGE CONTAINING A DRAWING HEADER	P. 535
— REPLACING DRAWING HEADERS	P. 535
— INSERTING LISTS.....	P. 538
— UPDATING LISTS.....	P. 541
— CONTENTS OF LISTS WHEN NO DATABASE IS ATTACHED	P. 541

In the chapter “Creating Drawing Headers” on page 742 you can see how to create your own drawing headers.

This chapter describes how to *place* a drawing header on a project page.



TEMPLATE CONTAINING ALREADY INSERTED DRAWING HEADERS

AS AN ALTERNATIVE TO MANUALLY INSERTING A DRAWING HEADER, YOU CAN ALSO OPEN A TEMPLATE PAGE, ON WHICH THERE IS ALREADY PLACED A DRAWING HEADER.

— READ MORE ABOUT THIS IN “PAGES AND CHAPTERS” ON PAGE 491; SPECIALLY IN “CREATING A NEW PAGE” ON PAGE 495.

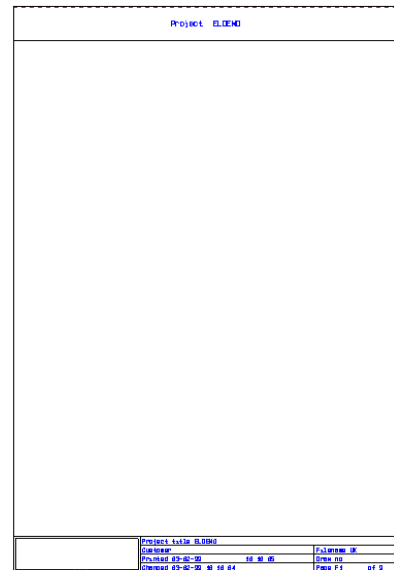
THE CONTENTS OF A DRAWING HEADER

A drawing header is a kind of formula containing text and graphics, which you can choose to insert on project pages.

A drawing header can for instance consist of a frame drawn around the entire page, with your company name and logo in the lower-left corner of the frame. Apart from this, the name and number of the individual pages, which customer the project is carried out for — and much more — can be stated in the drawing header.

When you have placed the drawing header on a page in the project, you can make the program fill in this information for you automatically.

You will typically apply different drawing headers on the pages in the project — depending on the use of the pages. — You can read more about what a drawing header is in “Dictionary” on page 873.



Please note, that you can place two drawing headers on the same page — see “Two drawing headers on the same page” on page 536.

INSERTING A DRAWING HEADER ON A PAGE IN A PROJECT



To insert a drawing header on the page you are located on in the project, you first click on the **Page data** button on the program tool bar on the top of the screen. This makes you enter the dialog box **Page data**. You can also enter this dialog box by choosing **Settings => Page data**.

The title of the drawing header is displayed here

Click here to select drawing header

Click to select a drawing header form the Symbol menu

Selecting a drawing header

To specify which drawing header is to be placed on the page, click on the down-arrow in the field below the **With drawing header** check box.

In the list that appears, you click on the name of the desired drawing header. As you scroll down the list of drawing headers, the titles of the drawing headers are displayed.

When you choose a drawing header the list disappears, and the name of the drawing header is placed in the field.

At the same time **With drawing header** is automatically selected.

You have now chosen a drawing header for the current project page.

Adding a drawing header to the list of drawing headers

If you want to add a drawing header to the list of drawing headers, please refer to "Adding drawing headers to the List Of Drawing Headers" on page 750.

Choosing a drawing header not on the list



If you want to use a drawing header that is not in the list of drawing headers in **Automation**, click on the **Browse** button next to the drawing header field. You then enter the **Symbol menu**, where you select the desired drawing header, and click **OK**.



NOT ADDED TO THE LIST OF DRAWING HEADERS

THE CHOSEN DRAWING HEADER IS *NOT* ADDED TO THE LIST OF STANDARD DRAWING HEADERS IN **Automation**.

HOWEVER, WHEN YOU HAVE USED A DRAWING HEADER IN A PAGE IN A PROJECT THIS WAY, YOU WILL SEE THE DRAWING HEADER ON THE LIST OF DRAWING HEADERS.

THIS IS BECAUSE THE LIST ALWAYS DISPLAYS THE LIST OF STANDARD DRAWING HEADERS IN **Automation** AND THE USED DRAWING HEADERS IN THE ACTIVE PROJECT. WHEN YOU START WORKING IN ANOTHER PROJECT, YOU WILL THEREFORE NO LONGER SEE THE DRAWING HEADER IN THE LIST.

Page title

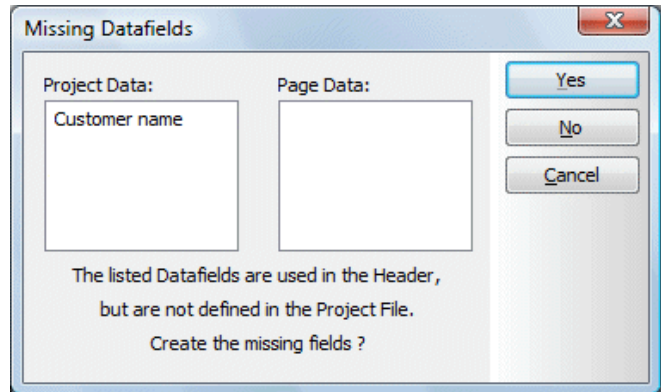
If you at the same time want to assign a title to the page, you click inside the field **Page title**. Here you for instance type in the text *Drawing*. You can also fill out the other fields with information about this particular page in the project.

Missing datafields

When you have chosen a drawing header, click **OK** on the tab **Page data**.

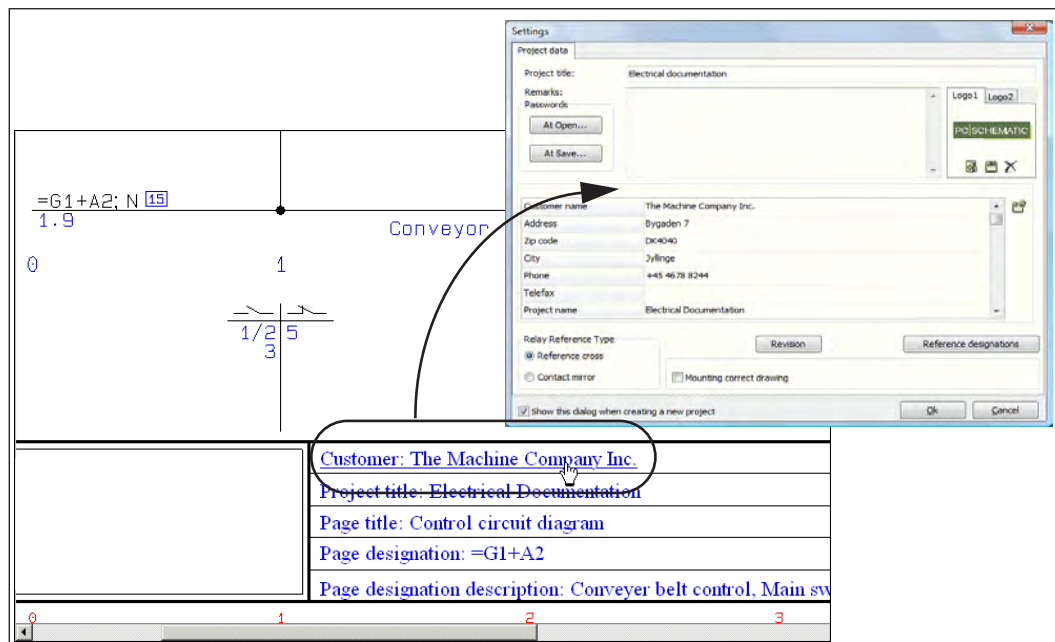
If a drawing header contains some datafields, which have not been created in the current project, you will get a message about this, as you can see in the figure to the right.

Click **Yes** to create them in your project. As you will see, the drawing header has now been placed on the page.



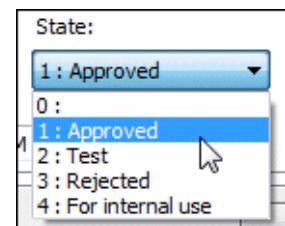
Active links from Datafields in Drawing headers to the Settings dialog box

When you double-click on a datafield, which comes from the dialog box **Settings**, the dialog box **Settings** is opened on the tab, from which the datafield collects its value. — For instance **Customer** or **Project title** in **Project data**:

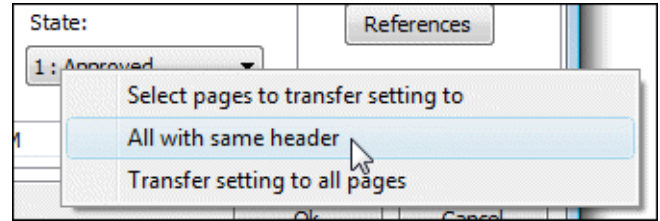


Drawing headers with States

When a drawing header has different states, you can choose the drawing header state in the lower-right corner of the dialog box:



If you wish to transfer the selected state to the other pages in the project, you can right-click on the **State** field, and specify which pages in the project to transfer the state to.



ADD NEW PROJECT PAGE, AND ASSIGN A DRAWING HEADER TO IT

If you have not created the page you want to place the drawing header on, you do this by entering the **Page menu**:



- 1) In the **Page menu**: Click on **Add**.
- 2) In the dialog box **Page function**: Click on **Normal** and click **OK**.
- 3) In the dialog box **New**: Click on the template **Blank page** and click **OK**.

If you want to know more about the **Page menu** and the various types of pages you can apply, you may read “Pages and Chapters” on page 491.

If you want to place a drawing header on the page you have selected in the **Page menu**, click on **Page data**, and then continue as described in “Inserting a drawing header on a page in a project” on page 532.

When you have inserted the drawing header, you return to the **Page menu**, and here you click **OK**.

REPLACING DRAWING HEADERS

If your company for instance has changed its phone number, you can enter Design symbol, correct the phone number in the drawing header, and save the new drawing header under the same name. — See how to change drawing headers in “Placing a Company Logo on a drawing header” on page 751.



EDITED DRAWING HEADERS ARE NOT REPLACED AUTOMATICALLY

WHEN YOU CHANGE THE *DEFINITION* OF A DRAWING HEADER, WHICH IS USED IN A PROJECT, THE DRAWING HEADER IN THE PROJECT WILL *NOT* BE REPLACED BY THE NEW VERSION OF THE DRAWING HEADER, UNTIL *YOU* SPECIFY THAT IT SHALL BE REPLACED.

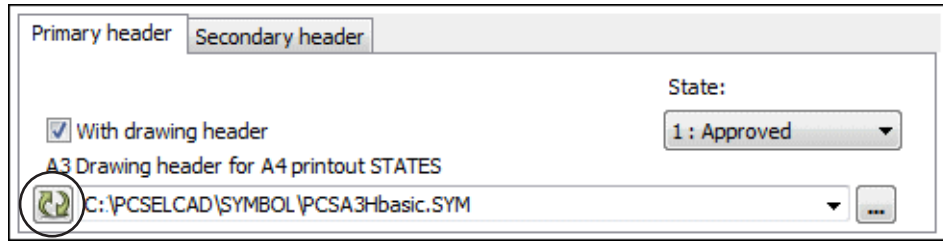
Replacing a drawing header

When you have edited a drawing header, and wish to use the new version in your project, do this:

- 1) Click on a project page using the drawing header.

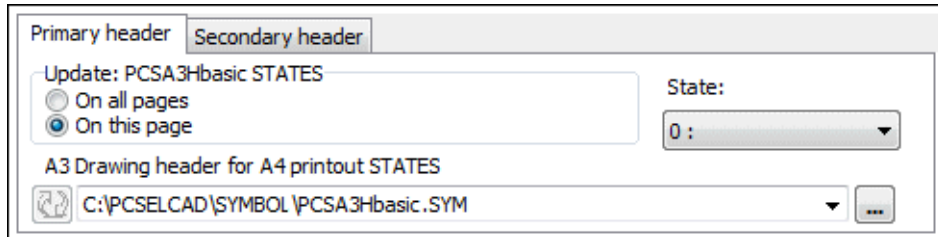


2) Click the **Page data** button.



3) At the bottom of the dialog box **Page data**: Click the **Update** button.

4) Specify whether to replace the drawing header **On all pages** or **On this page**:



If you select **On this page**: The new version of the drawing header will be inserted on this page only.

If you select **On all pages**: All pages in the project using the old version of the drawing header, will have their drawing headers replaced by the new version.

Of course you can also choose to replace a drawing header with any other drawing header from the list of drawing headers.



DRAWING HEADERS AND PAGE FORMATS

DRAWING HEADERS CAN ALWAYS BE REPLACED — ALSO WHEN THEY ARE ORIGINALLY INSERTED VIA A TEMPLATE PAGE.

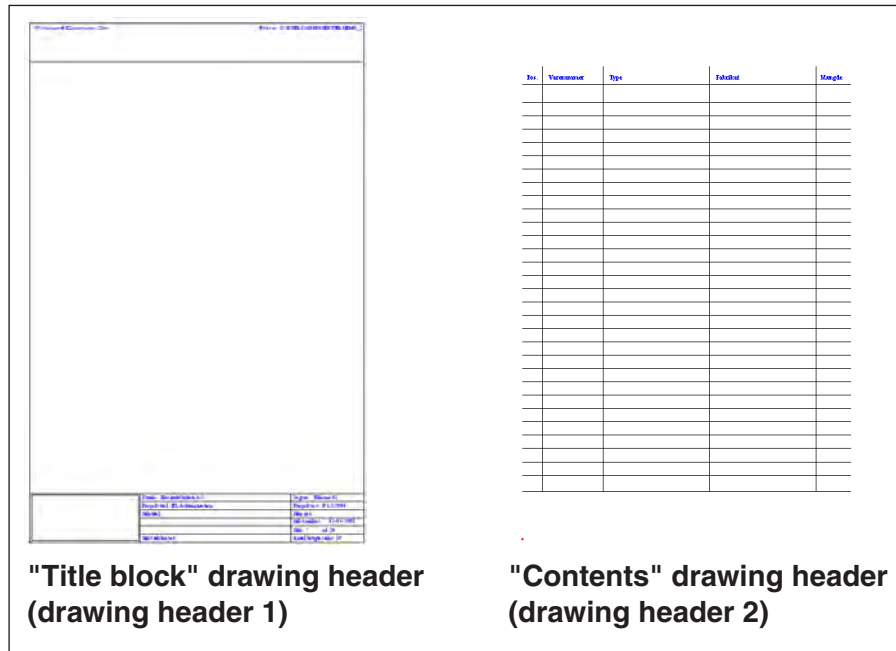
WHEN A DRAWING HEADER WITH A NEW PAGE FORMAT IS SELECTED, THE PAGE FORMAT IS CHANGED AUTOMATICALLY.

TWO DRAWING HEADERS ON THE SAME PAGE

You can place two drawing headers on the same page in the project. This can for instance be used for lists. Here you need to create:

- A drawing header containing the title block surrounding each of the list pages

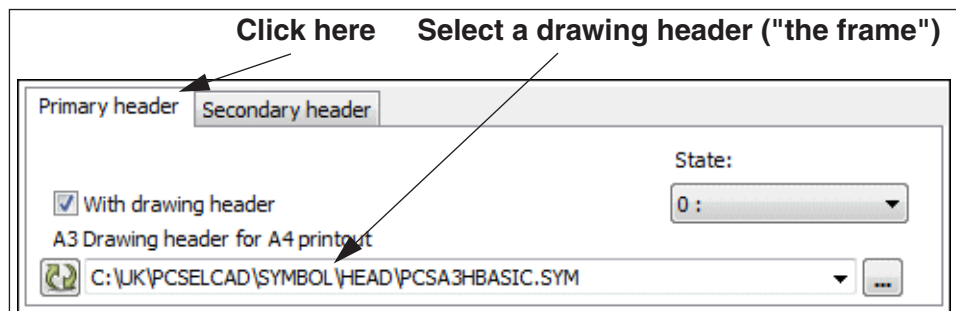
- A series of "contents" drawing headers for parts lists, components lists etc.



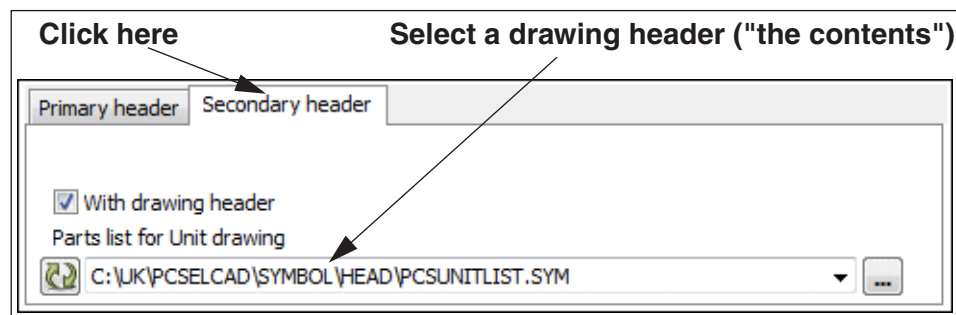
If you, at a later stage, need to change the information in the title block, you will then only need to change one single drawing header. — And not all of the various "contents" list drawing headers.

Inserting two drawing headers on the same page

In the dialog box **Settings => Page data**: Click on **Primary header**, and select a drawing header (the title block), as described in "Selecting a drawing header" on page 533:



Then click on **Secondary header**, and select the drawing header containing the "contents" for e.g. a parts list:



Apart from the page function, lists are placed as described above. — Read more about page function in “Creating a new page” on page 495.




Diagram pages can also contain lists

Diagram pages can also contain parts or components lists. Read more about this in “Diagram pages - containing a Parts or Components list” on page 762.

You can also place a list as a symbol on a diagram page, without letting the list become part of the drawing header — see “Creating a List symbol” on page 762.

Active links in lists and in tables of contents

When you double-click on datafields in lists and in tables of contents, you jump to the place in the project, from where the datafield fetched its contents — e.g. a page, a symbol or a connection point. This is the case for quite a few datafields, but not for all types.

Pos.	EAN Barcode	Article number	Type
1		<u>3389110585506</u>	AB1NE435U
2		3389110586435	AB1VV435U
3		3389110610024	XB2BA31

When the cursor is above a datafield containing an active link, the datafield is underlined, and the appearance of the cursor changes to a pointing hand. However, this is only the case if **View => Show object below cursor** is selected.

When you double-click on e.g. an article number, you jump to the first symbol for the component in the project.

Please note that this makes it possible to use the tables of contents in the project as a kind of page menus.



UPDATE ALL LISTS

IN PROJECTS ORIGINALLY CREATED IN **Automation** VERSION 8.0 OR PREVIOUS, YOU MUST FIRST CHOOSE **LISTS => UPDATE ALL LISTS** TO ACTIVATE ALL LINKS IN LISTS AND IN TABLES OF CONTENTS IN THE PROJECT.

Jumping back to the list / table of contents



When for instance you have jumped *from* an article number in a list *to* a symbol on a project page, you can return to the list by clicking the **Page history back** button to return to the list. The button is placed to the left of the page tabs at the bottom of the screen. — See “Choosing pages outside the Page menu” on page 491.

UPDATING LISTS

When for instance you have inserted a parts list on a project page, it will not be filled out with information from the project drawings right away.

Choose **Lists => Update Parts Lists** to update the parts lists in the project.

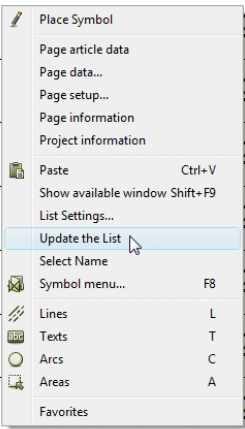
The list is now updated with information from the present project.

Updating all lists

If you want to update all the lists in the project, choose **Lists => Update All Lists**.

Updating lists by right-clicking

When the active project page contains a list or a table of contents, you can also right-click and choose **Update the List**.

Component		Part no.
=W1+A1 -F1		2250001
--- -F2		2250001
--- -K1		2250106
--- -K2		2250106
--- -K3		2250108
--- -K4		2250108
--- -K5		2250106
--- -K6		PCS2250106

If a diagram page contains a components or parts list, this list can also be updated with this function.

CONTENTS OF LISTS WHEN NO DATABASE IS ATTACHED

Parts and Components lists pages memorize all data, which were originally fetched from the database. Therefore these data are not lost when the list pages are updated, and the applied article numbers can not be found in the database.

A list will therefore not become empty when updating lists without having a database attached, or with another database attached.

If you send a project to a customer, who does not have the database attached, the lists will still be complete, if the customer chooses **Lists => Update all lists**.

LIST SETTINGS

In **Automation** you can specify which types of information you want to include on project lists.

By adjusting the settings for a list, you can for instance make separate parts lists for the various component vendors, or choose to include articles from the unit drawings in your component list.

THIS CHAPTER DESCRIBES	
— CHOOSING THE LIST IN THE PROJECT	P. 543
— UPDATING LISTS	P. 543
— WHICH PART OF THE PROJECT TO COLLECT DATA FROM.....	P. 544
— SORTING LISTS ACCORDING TO DATABASE FIELDS.....	P. 547
— SPECIFY CRITERIA FOR WHICH DATA TO INCLUDE.....	P. 549
— HOW TO HANDLE REPETITIONS IN THE LIST.....	P. 551
— HOW TO REPLACE MISSING INFORMATION.....	P. 552
— SPECIAL SETTINGS FOR CABLES, TERMINALS AND PLC LISTS	P. 553
— LIST SETTINGS FOR TABLES OF CONTENTS.....	P. 555



LIST SETTINGS FOR TEMPLATES

LIST SETTINGS CAN ALSO BE SPECIFIED ON PAGE TEMPLATES, SO THAT YOU CAN CREATE LIST TEMPLATES CONTAINING DIFFERENT LIST SETTINGS, FOR DIFFERENT PURPOSES.

— READ MORE ABOUT THIS IN “CREATING TEMPLATES” ON PAGE 771, SPECIALLY “CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR” ON PAGE 772.

In order to specify list settings, you first have to have a list in the project. Read “Insert drawing headers/formulas” on page 531 if you need to know more about how to insert lists in a project.

If you, on the other hand, want to create list *files*, you can read more about this in “Exporting List Files” on page 649.

CHOOSING THE LIST IN THE PROJECT

Enter the page you want to change the settings for, and choose **Lists => List settings**. Here you specify how you want to fill out the list you are presently working on.

Depending on which type of list you are working on, you will have different options at hand to specify the list setup.

If the components in for instance a parts list requires more than one page, the program will notify you about this, and — if you answer **Yes** — add one or more additional parts list pages immediately following the first page in the parts list.

If a list results in less pages following an update, you will also be informed about this.

UPDATING LISTS

When you are in the dialog box **List setup**, and you want to update a list, click on **Update** in the right-hand side of the dialog box. This update *only* applies for the list you are working on for the moment.

If you are content with the settings, but would like to wait until later before you update the list, you click **OK**. The update will be carried out next time you choose **Lists => Update lists**.

If you are not pleased with the new settings, you click on **Cancel**.

Please note, that information about list settings are saved in the project file itself.

Errors displayed in result window during list update

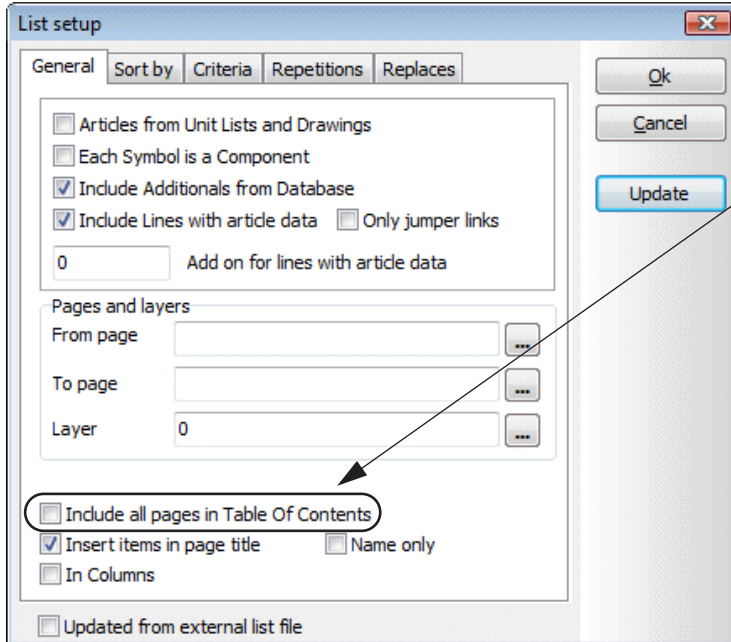
If an error occurs during a List update or during List to file operations, these errors will be displayed in a result window. These errors could for instance be unknown from/to specification in the list settings.

The window functions as for the Design check function — see “Design Check” on page 320.

SPECIFYING FROM WHERE TO COLLECT COMPONENTS

If you have not already done this, then select the relevant list page in the project and choose **Lists => List settings**.

Working with lists you have the option of specifying precisely which components you want to include in the lists, and according to which of their symbol texts you want the lists to be sorted.



Only possible for components lists

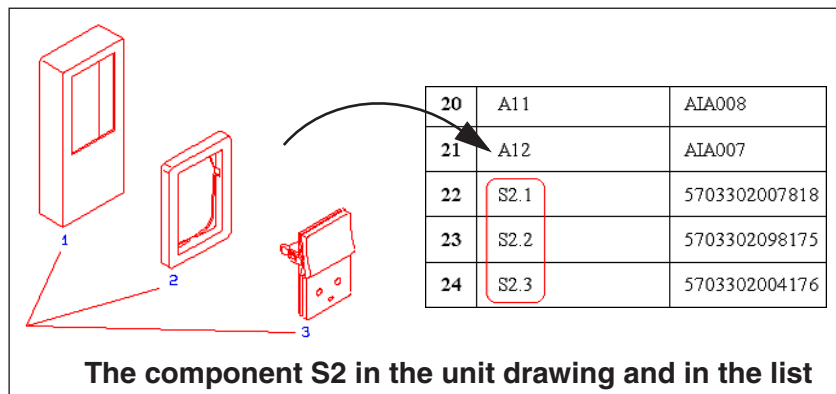


DIFFERENT TABS FOR CABLES, TERMINALS AND PLC LISTS

PLEASE NOTE THAT CONTENTS OF THE **GENERAL** TAB IS DIFFERENT FOR CABLES, TERMINALS AND PLC LISTS — AS DESCRIBED IN “LIST SETUP FOR TERMINALS- CABLES- AND PLC LISTS” ON PAGE 553.

Which data to include

If you select **Articles from Unit Lists and Drawings**, articles from unit drawings are included. The component name from the unit drawing is hereby specified as a sub name for the component in the list.



The component S2 in the unit drawing and in the list

If you select **Include Additional from Database**, the program will examine each component, and see whether they need any additional, that shall be included in the list.

When you select **Include Lines with article data**, article data from lines will be included as well, and the lengths of the lines will be calculated. If you only wish to include line data from jumper links, select **Only jumper links**.



FOR PARTS AND COMPONENTS LISTS ONLY

WHEN YOU SELECT **INCLUDE LINES WITH ARTICLE DATA**, YOU ALSO GET THE OPTION OF SPECIFYING AN **ADD ON FOR LINES WITH ARTICLE DATA**.

THIS OPTION IS MOSTLY USED FOR LINES CARRYING CABLE DATA ON **GROUNDPLAN** PAGES. PLEASE REMEMBER TO SPECIFY A UNIT OF MEASUREMENT FOR THE ADD ON — SUCH AS **100CM**.

THE SPECIFIED ADD ON IS INCLUDED IN THE DATAFIELDS TOTAL QUANTITY, QUANTITY AND LENGTH FOR LINES.

Simple parts- and components lists

If you select **Each Symbol is a Component**, each symbol will be understood as an individual component — even though other symbols with the same name exists.

Therefore, if three symbols with the name **-X1** exists, these symbols will be understood as three individual components on both parts- and components lists.

This is also the case when documenting electrical house installations, where names are not necessarily assigned to the symbols.

Choose Pages and Layers

In **Pages and layers** you specify from which project pages the parts list receive information, and from which layers information will be collected on these pages. If it says **0** in the fields, this signifies that all layers/pages are included in the parts list.



When you click the **Browse** button next to **From page** or **To page**, you enter the **Page menu**, where you select the desired page(s), and click **OK**.

Pages with reference designations

When you need to discern between two pages having the same page numbers, but different reference designations, you do this by placing the character **"/"** between the page name and the reference designation for the page.

If page **17** has the reference designation **=G1+A1**, you therefore write **17/=G1+A1**.



When the page number is selected using the **Browse** button, the reference designation is inserted automatically.

Layers



When you click the **Browse** button next to the **Layer** field, you enter the **Layer menu**, where you select the desired layer, and click **OK**.

Sorting the list

Using the fields **Primary** and **Secondary** on the tab **General**, you determine which symbol texts the list is to be sorted according to.

For Components lists only: Labels for component marking

When you under **Lists => List settings** for a **Components list** on the **General** tab choose **List only one of each name**, each name is only included once. Hereby you can for instance create labels for component marking.

Include all pages in Table of Contents

If you select **Include all pages in Table of Contents**, each page in the list is included in the Table of Contents. If not, only the first page is mentioned.

Display first and last object in the Page title for lists

When you choose **Insert name in page title**, the name of the first and the last object on the list page is displayed after the page title in each list page. If you also select **Name only**, only the name of the datafield is displayed. If it is *not* selected, the datafield also displays the connection name (or cable wire).

The page title is also displayed in the **Page menu**, in tables of contents and in drawing headers, where the page title is placed as a datafield.

Lists in Columns

In **Automation** you can create lists in columns. This setting is specified on the tab **General** in the dialog box **List settings**.

Updated from external list file

If you select **Updated from external list file**, the list will be updated / filled out via data from an external Excel or **.csv** file. When you click the **Update** button, you will then be asked to point out the external file, which is then loaded on the list page.



THIS FUNCTION IS NOT FOR CORRECTING PARTS LISTS

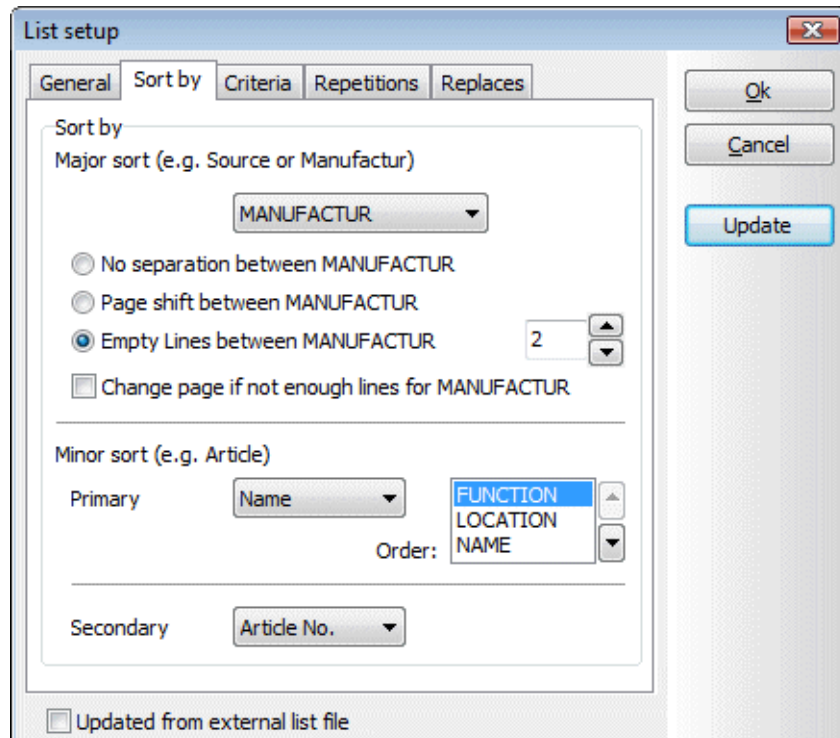
THIS FUNCTION CAN FOR INSTANCE BE APPLIED WHEN YOU HAVE A CABLING LIST BETWEEN PANELBOARDS, AND DO NOT WISH TO DRAW THE CABLING.
— OR IF YOU GENERALLY DO NOT WISH TO INCLUDE LISTS IN THE PROJECT, WHICH YOU CAN NOT FILL OUT VIA THE DRAWINGS.

THE FUNCTION IS *NOT* DESIGNED FOR CORRECTING EXISTING PARTS LISTS!

The check box can also be applied for excluding a list file from being updated.

SORTING LISTS ACCORDING TO DATABASE FIELDS

This tab only exists for parts and components lists.



On the **Sort by** tab you can specify that the list must be sorted based on a field in the database. This can for instance be applied for sorting the list by source or by manufacturer. Which datafield to sort according to, is specified in the **Major sort** field in the top of the dialog box.

Separation between e.g. manufacturers

You can furthermore specify how the different manufacturers — the selected datafield in the dialog box above — shall be separated in the list.

Here you can choose between **No separation between MANUFACTUR** and **Page shift between MANUFACTUR** — or you can select **Empty lines between MANUFACTUR**, and specify the number of blank lines between the manufacturers in the list.

When you select **Change page in not enough lines for MANUFACTUR**, the components for a new manufacturer starts on top of a new page, when there are not enough room for all components from the manufacturer on the page. — This option can not be selected when you have selected **Page shift between MANUFACTUR** above.

Minor sort

At the bottom of the tab you can furthermore specify how the components shall be sorted within each individual manufacturer.

In the above dialog box, the components will therefore primarily be grouped by manufacturer. For each manufacturer, the components will then be sorted by name, and below this by article number. — See also “Sorting on all parts of the symbol name” on page 548.

Sorting on all parts of the symbol name

Components lists can also be sorted according to all parts of the symbol name; the function aspect (=), the location aspect (+) and the product aspect / name (-). — See “Reference designations” on page 335, in particular “On reference designations” on page 335.

If for instance you want all components with the same location aspect (e.g. **+A1**) to be listed together in the same components list, you just need to move **LOCATION** to the top of the list.

You can either do this by clicking on

LOCATION and then *drag* it to the top of the list, or by clicking on **LOCATION** and then use the arrow buttons next to the list.

If you place **NAME** at the top of the list, for instance all **-K1** symbols will be placed succeedingly, regardless of their function and location aspects.

This makes it possible to create a list where e.g. all panels appears separately.

If you do not make any changes, the order will by default be **FUNCTION**, **LOCATION** and **NAME**.

SPECIFYING CRITERIA FOR THE COMPONENTS TO INCLUDE

On the tab **Criteria** you set up the conditions for which content you want on the list you are working on.

You can fill out seven lines, each containing one criterion.

Only one criterion needs to be fulfilled

When you select **Only one Criterion needs to be fulfilled** (logical OR), the text *and* changes to *or* in the dialog box.

In the dialog box above this would mean that you get all components from AEG Industry, *as well as* all components having a **K** in the symbol name.

If **Only one Criterion needs to be fulfilled** is *not* selected, the text *and* is displayed between the criteria lines. This means all the criteria have to be fulfilled at the same time. In the dialog box above, this means you only get components from AEG Industry, having a **K** in their symbol names.

Setting up a criterion

By clicking in the text fields below **Field**, you state which datafield the criterion is set for. These datafields can be any datafield from the database, or from one of the symbol texts.

If you set **Operator** to =, you can type in what the contents of the field should be identical to in the **Text** field.

If you want the contents to differ from what you type into the **Text** field, you choose the **Operator** <>.

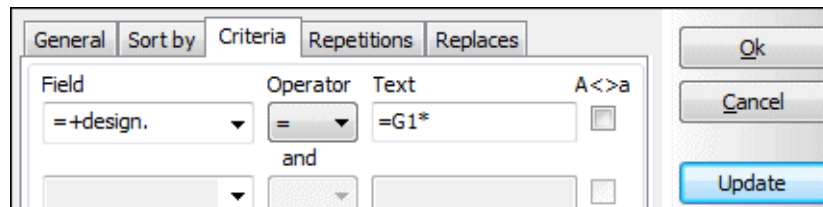
Accordingly, you can set the operator to determine that the contents must be larger than (>) or smaller than (<) what you specify in the **Text** field. Please note, that these operators examine whether the texts are placed before or after the text alphabetically.

If you select **A<>a**, you tell the program to distinguish between upper- and lower-case letters.

Data for which Criteria can be set

The data, for which you can set criteria, are:

- Datafields from the database (written in capitals)
- Component data (**Name, Type, Article no** and **Function**)
- Reference designations (the datafield **=+design.**) — see the figure below
- Symbol type — see “Symbol types” on page 706
- Symbol datafields — see “Creating symbol datafields” on page 733



Wildcards/jokers

If for instance you need all components on a list having the letter **K** in their name, you need a way to tell this to the program.

For this purpose, you can apply the so-called jokers/wildcards, which are also used in Windows.

The special characters used for this purpose are **?** and *****.

When you place a question mark (**?**) somewhere in a text, this signifies, that on the place in the text where the question mark is located, there *have to* be exactly one character. And this character could be any character.

If you place an asterisk (*****), any number of characters could be located on the place where the asterisk is located.

If you, for instance, have to find something on exactly three letters with the letter **Q** as the second character, you type in **?Q?**. Thus you state that there has to be one — and just one — character in front of **Q**, and exactly one character after as well.

If you will find a text having an **A** anywhere in the text, you type in ***A***.

— Please note, that if you use **?** there *have to* be a character, but if you use ***** this also covers cases with no characters on the location of the asterisk.

Example

If you want a list of contactors from AEG Industry, you set up as a first criteria, that **MANUFACTU** have to be equal to **AEG Industry**.

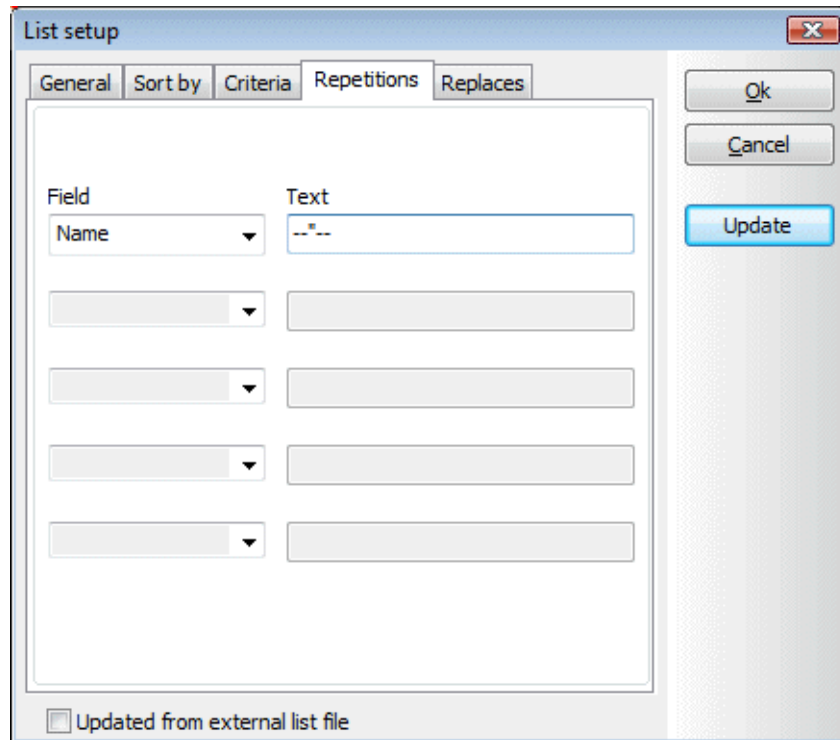
As the second criteria, you state that **Name** shall be equal to ***K***. This gives you all component names with a **K**. — You can see these settings on page 549.

By creating several different parts lists, you can create a parts list for each component vendor you order components from.

HOW TO HANDLE REPETITIONS IN LISTS

This tab only exists for parts and components lists.

On the **Repetitions** tab, you can specify how repetitions in the list shall be handled.



If you choose the datafield and type in a text, the program inserts the text for each repetition of the given datafield.

If you fill out the tab as displayed above, the list will look like this:

X1	3389110586435	Terminal 4 mm2 Uni
-- " --	3389110586435	Terminal 4 mm2 Uni
-- " --	3389110586435	Terminal 4 mm2 Uni

To empty a line on the tab, choose the blank field in the drop down box for the line.

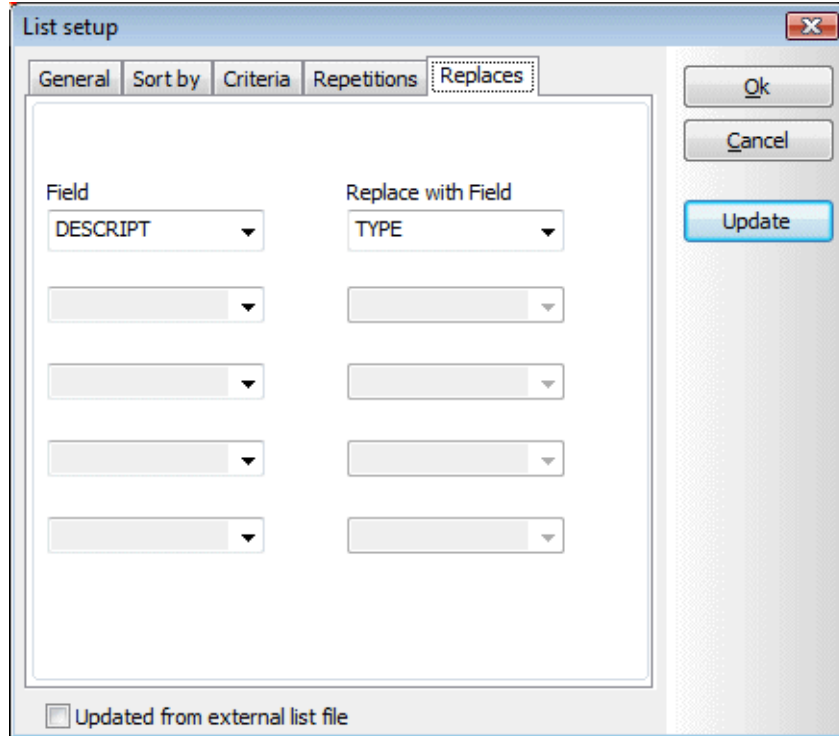
Data for which repetitions can be specified

The data, for which repetitions can be specified, are the same as the data which can be applied for **Criteria**. — See “Data for which Criteria can be set” on page 550.

HOW TO REPLACE MISSING INFORMATION

This tab only exists for parts and components lists.

If the program can not find a piece of information about a component in the datafield you have specified, you can specify another datafield where the information can be collected from.



Please note, that data is only collected from the replace field if the original datafield is empty for a component.

When the program can not find information about a components description in the database (**DESCRIPT**), you can tell the program to collect the information from the **Type** field in the **Component data** dialog box for the component on the diagram.

Please remember, that datafields written in upper-case letters represent information coming from the database.

Data for which replacements can be specified

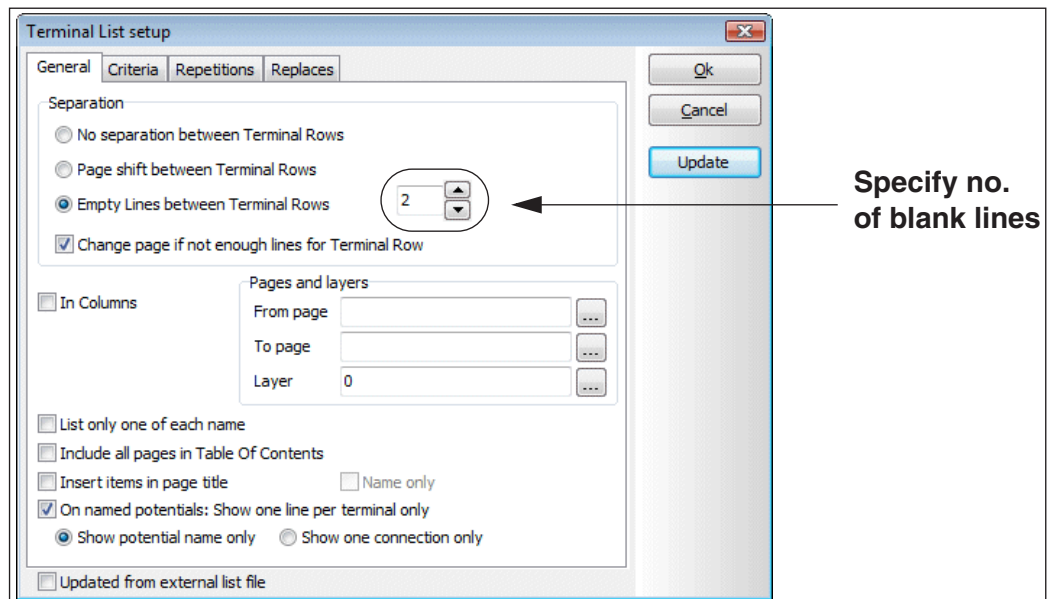
The data, for which replacements can be specified, are the same as the data which can be applied for **Criteria**. — See “Data for which Criteria can be set” on page 550.

LIST SETUP FOR TERMINALS- CABLES- AND PLC LISTS

For terminals, cables and PLC lists you have some special options on the **General** tab, and you also have the option of using symbol datafields on the **Criteria** tab. This is described in the following:

The tab General for terminals, cables and PLC lists

If you are working on a terminals, cables or PLC lists and choose **Lists => List settings**, the **General** tab looks like this:



Separation

Here you specify how to separate the terminals, cables or PLCs on the list.

Here you can choose only to have one terminal row on each page by choosing **Page shift between Terminal Rows**, or to insert empty lines between the terminal rows by choosing **Empty lines between Terminal Rows**. If you choose the latter, it is also possible to specify how many empty lines to insert between the terminals/cables/PLCs in the list.

Components starts on new page in Cables, Terminals and PLC lists

When you select **Change page if not enough lines for Terminal Row**, a component starts on a new page, when there is not enough space for it on the remains of the current list page.

Survey of cable connections

For cable- and terminals lists, it is possible to create a survey of what the cables connects. This is done by enabling **List only one of each name**, and can be applied for lists in the project as well as lists for files.

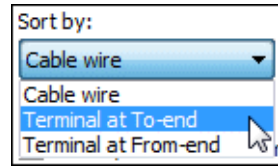
In terminals lists the first terminal in the terminal row is displayed (the lowest group number). In cable lists the first conductor in the cable is displayed.

Special for cables

For cable lists, there are two more settings:

SORTING IN CABLES LISTS

For cables lists, you can under **Sort by** specify whether to sort by **Cable wire**, **Terminal at To-end** or **Terminal at From-end**.

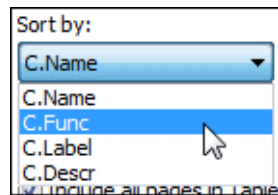


SORTING SO THAT CABLE WIRES FOR THE SAME CABLE SYMBOL ARE KEPT TOGETHER

When you select **Sort within symbol**, cable wires which are drawn next to each other in the same cable — for instance because they are under the same screen in a cable or a twisted pair — are also placed next to each other in the list.

Sort by name on PLC lists

For PLC lists you also specify according to which datafield the list is sorted.



For terminals lists only

For terminals lists you can select **On named potentials: Show one line per terminal only**. When this option is selected, there will only appear a single line in the terminals list, for the terminals that are connected to a named potential. If you then select **Show potential name only**, only the potential name appears in the list. If instead you select **Show one connection only**, one of the connections to the terminal appears in the list.

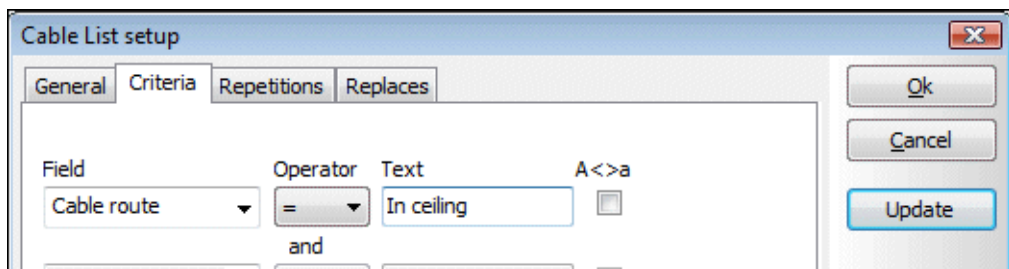
The remaining settings

Pages and layers and **Include all pages in the Table of Contents** are described in “Specifying from where to collect components” on page 544. The **Criteria** tab is described in “Specifying Criteria for the components to include” on page 549.

Symbol datafields as Criteria for terminals, cables and PLC lists

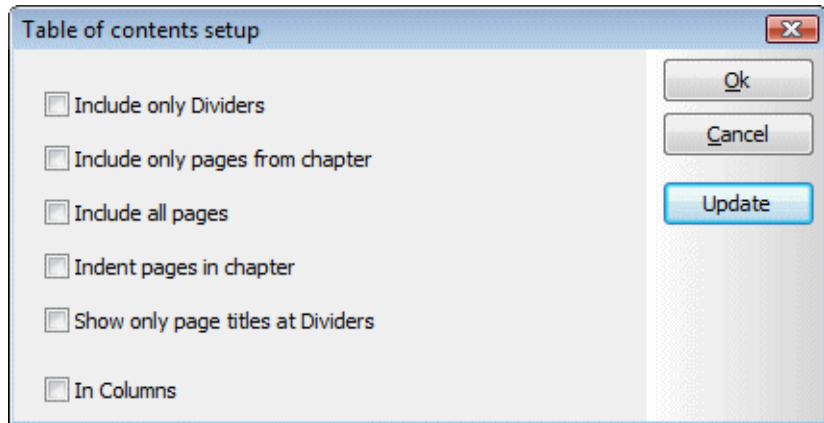
On the **Criteria** tab for terminals, cables and PLC lists, you can use user defined datafields as criteria. — See “Creating symbol datafields” on page 733 and “Symbol datafields with predefined values” on page 736.

If you therefore have created a symbol datafield **Cable route** for cables, this can be used as a criterion in cables lists, so that cables where **Cable route** has different values — such as **In ceiling** and **In wall** — can be listed side by side, or in separate lists.



LIST SETTINGS FOR TABLES OF CONTENTS

To specify the settings for a table of contents, you must activate the table of contents page and choose **Lists => List Settings**. You then enter this dialog box:



Here you specify what to include in the table of contents

Normally, a table of contents will only contain information from project pages placed *after* the table of contents in the project. If you therefore place a table of contents as the last page in the project, it will not contain any information — unless you select **Include all project pages**.

In the dialog box you have the following options:

OPTION	EFFECT
INCLUDE ONLY DIVIDERS	THE TABLE OF CONTENTS WILL ONLY CONTAIN THE DIVIDER PAGES OF THE PROJECT.
INCLUDE ONLY PAGES FROM CHAPTER	THE TABLE OF CONTENTS WILL CONTAIN THE SUCCEEDING PROJECT PAGES IN THE CHAPTER. WHEN YOU SELECT INCLUDE ONLY PAGES FROM CHAPTER , YOU ALSO GET THE OPTION OF SELECTING INCLUDE TABLE OF CONTENTS . IF YOU SELECT INCLUDE TABLE OF CONTENTS , THE PAGE(S) CONTAINING THE TABLE OF CONTENTS FOR THE CHAPTER ARE ALSO LISTED IN THE TABLE OF CONTENTS.
INCLUDE ALL PROJECT PAGES	WHEN YOU SELECT THIS OPTION, THE TABLE OF CONTENTS WILL CONTAIN INFORMATION FROM ALL PAGES IN THE PROJECT. IF YOU DO <i>NOT</i> SELECT THIS OPTION, THE TABLE OF CONTENTS WILL ONLY CONTAIN INFORMATION FROM PROJECT PAGES SUBSEQUENT TO THE TABLE OF CONTENTS.
INDENT PAGES IN CHAPTER	THE COMMON PROJECT PAGES (DIAGRAMS, LISTS ETC.) WILL BE INDENTED 15MM. COMPARED TO THE DIVIDERS.
SHOW ONLY PAGE TITLES AT DIVIDERS	FOR CHAPTER DIVIDER PAGES, IT OFTEN SAYS THE SAME IN THE FIELD SETTINGS => PAGE DATA => PAGE TITLE AND ON THE TAB. WHEN YOU SELECT THIS OPTION, ONLY THE PAGE TITLE IS DISPLAYED FOR THE DIVIDER PAGES IN THE TABLE OF CONTENTS.
IN COLUMNS	— SEE “LISTS IN COLUMNS” ON PAGE 546.

Because some of the above options mutually excludes each other, some options will be dimmed as you make selections in the dialog box.

COMPONENT GROUPING

To get the correct information transferred from the project diagrams to the lists, you sometimes need to apply the **Component grouping** function.

THIS CHAPTER DESCRIBES	
— WHEN COMPONENT GROUPING IS NECESSARY	P. 556
— THE COMPONENT GROUPING MENU	P. 557
— FINDING A COMPONENT IN THE MENU	P. 557
— THE SORTING ORDER OF THE COMPONENTS	P. 558
— CHANGING THE ORDER BY DRAG AND DROP.....	P. 558
— SPECIFYING THE COMPONENT GROUPING	P. 558
— THE ORDER OF THE COMPONENTS IN LISTS	P. 561
— SPECIFYING THE ORDER IN PARTS LISTS	P. 561
— SPECIFYING THE ORDER IN TERMINALS LISTS	P. 561
— ENDING A COMPONENT GROUPING.....	P. 562
— COMPONENT GROUPING AND COPYING	P. 562
— GRAPHICAL MANAGING OF COMPONENT GROUPING FOR TERMINALS.....	P. 563

WHEN IS COMPONENT GROUPING NECESSARY

Normally you regard all symbols with the same symbol name as the very same component. However, this component can be built up from several physical components.

Therefore, when you want to print out the parts lists, there are cases where the program cannot determine how many components of a specific type are used in the project.

This could also be the case if for instance you use three terminals, each having the same symbol name and the same article number. In this case, the program cannot tell whether they are three individual terminals, or it is a triple terminal. — Unless you have fetched the terminals directly from the database, as described in “Fetching symbols from the database” on page 152.

In this case, the program can not tell whether just one or three components with the given article number are needed on the parts list.

Therefore, the program will have to make a qualified guess, but if you are not sure that the program will guess correctly, you will have to state what is the case yourself.

To be specific about this, you have to perform a so-called *component grouping*.

Here you divide symbols having the same names into groups, by assigning the same group number to the symbols for each individual component.

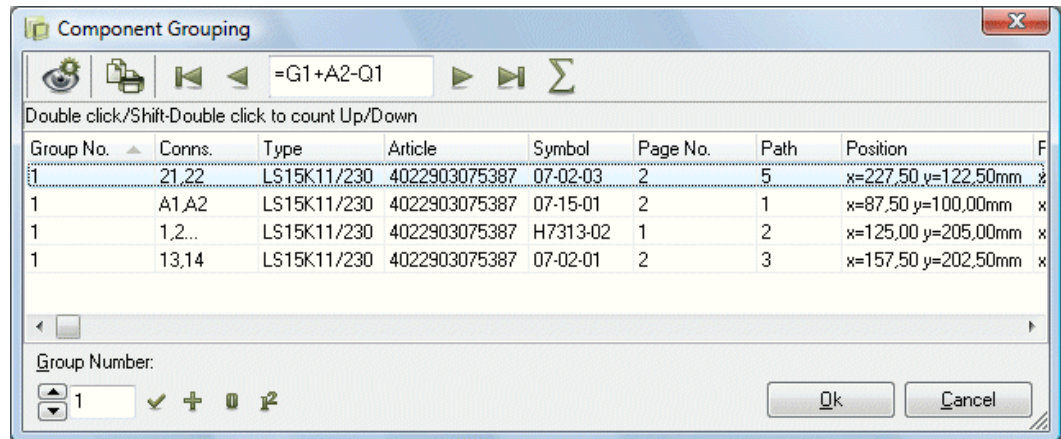
When two symbols have the same *name*, *article number* and *component group number*, the program knows they are part of the same physical component. If their group number differ, the program knows they belong to different components.

Because the component groups are printed out in numerical order (starting from the lowest group number), the group numbers have the further advantage, that you can determine the order in which components are printed out on the components list, as well as in which order terminals are printed out on terminals lists. — Read more about this in “The Order of the Components in lists” on page 561.

THE COMPONENT GROUPING MENU



When you have clicked on **Symbols**, and choose **Functions => Component Grouping** (short-cut <Ctrl+F7>), you enter this dialog box:



You can also select a symbol and choose **Functions => Component Grouping**, which makes you enter the **Component Grouping** menu for all symbols with the chosen symbol name.



Click on **Print** when you need to print out the component grouping of a component.



GRAPHICAL MANAGING OF COMPONENT GROUPING FOR TERMINALS

WHEN YOU WISH TO MAKE COMPONENT GROUPING FOR TERMINALS, THERE IS A MORE GRAPHICALLY INTUITIVE WAY TO DO THIS. THIS IS DESCRIBED IN “GRAPHICAL MANAGING OF COMPONENT GROUPING FOR TERMINALS” ON PAGE 563.

FINDING COMPONENTS

If you have not chosen the component you want to group, you can type in the name of the component in the lower-left corner of the dialog box.



Clicking on the **Used components** button gives you a list of the used symbol names in the project. Here you click on the symbol name you are interested in, and click on **OK**.



Also, you can scroll back and forth in the list of components by clicking on the **Previous component** and **Next component** buttons. Please note that the arrow-keys can be used as short-cut keys for these buttons.



Clicking on the **First component** button gives you the first component in the list, while clicking on the **Last component** button gives you the last.



When you click the **Show where the component is placed** button (or click on the symbol in the list and press <Enter>), the cross hairs points out the symbol in the project — and the **Component grouping** menu remains on screen.

THE SORTING ORDER OF THE COMPONENTS

By clicking on the column headers in the table, you determine by which criterion the components will be sorted in the dialog box. If you click on **Symbol**, they will be sorted by symbol name, if you click on **Conns.** they will be sorted by connection numbers — and so on.

You can sort in both ascending and descending order. To switch between ascending and descending order, just click the column header once again.

Conns.
21
20
19
18
17
16

Conns.
16
17
18
19
20
21

Descending/ascending order

CHANGING THE ORDER BY DRAG AND DROP

When you have selected one or more symbols in the dialog box **Component grouping**, you can move them by "drag and drop". This means:

- 1) Click on the selected symbols
- 2) Keep the left mouse button pressed down while you *drag* the symbols to the desired position
- 3) Release the mouse button

The symbols are now inserted in the desired position in the dialog box **Component grouping**.

SPECIFYING THE COMPONENT GROUPING

You can either let **Automation** guess the component grouping, or specify how it shall be yourselves:

Letting the program do the component grouping

When you need to zero all the group numbers for the chosen symbol name, do the following:

- 1) Click on the first line in the dialog box
- 2) Press down <Shift> and click on the last line in the dialog box: All lines are now selected
- 3) Click on the **Reset group numbers** button



When all group numbers for a component is set to zero, this signifies that you have not taken the grouping of the component into consideration. When you print out lists, the system will then have to guess in which order the possible parts of the

component shall be printed, as well as whether they belong to the same component or not.



GROUP NUMBER 0 FOR TERMINALS

ALL TERMINAL SYMBOLS WITH GROUP NUMBER **0** IS UNDERSTOOD AS INDIVIDUAL TERMINALS.

OTHER TYPES OF SYMBOLS WITH GROUP NUMBER **0** ARE UNDERSTOOD AS PART OF THE SAME COMPONENT.

Specifying component grouping by assigning group numbers automatically

If you wish to state that *you* have specify that all three terminals with the same symbol name are part of the same component, you therefore have to assign group number one to all of them.

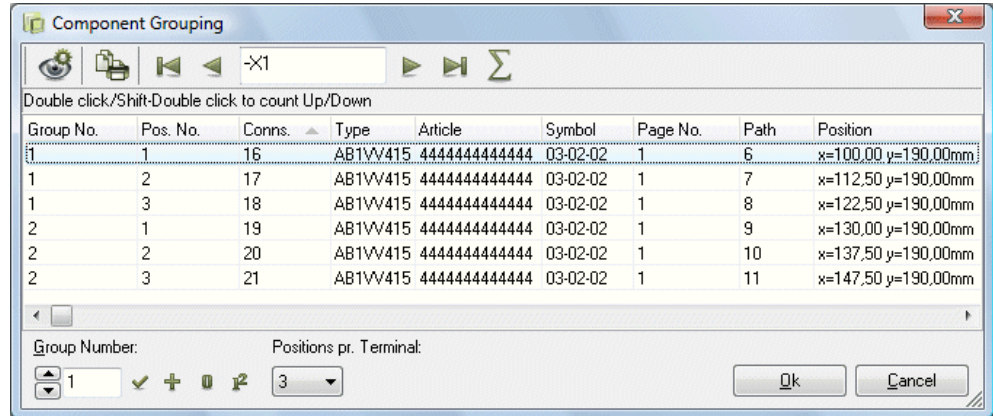
For instance you have six terminal symbols with the same symbol name, and want to state that they represent two components each containing three terminals. You therefore give three of the terminals group number **1**, while the remaining three gets group number **2**. Do the following:

- 1) Set **Position pr. terminal** to **3** (short-cut <Alt+p>). — This tells the program that three terminals are required for one part component (for instance for multi layered terminals).
- 2) Set **Groupnumber** to **1** (short-cut <Alt+g>).
- 3) Click for instance on the column **Conns.** — to sort the terminals by connection name.
- 4) Select the three terminals, and click on **Preset Group Numbers**, and the three selected terminals are assigned group number **1** and the positions **1**, **2** and **3** respectively.
- 5) Set **Group Number** to **2**.

12

12

6) Select the three remaining terminals, and click on **Preset Group Numbers**, and the three selected terminals are assigned group number **2** and the positions **1**, **2** and **3** respectively.



12



ASSIGNING GROUP- AND POSITION NUMBERS IN A SINGLE OPERATION

WHEN THE TERMINALS APPEARS IN THE DESIRED ORDER IN THE MENU, AND YOU HAVE SET **GROUP NUMBER** AND **POSITION PR. TERMINAL** CORRECTLY, YOU CAN ASSIGN ALL OF THE TERMINALS IN THE MENU CORRECT GROUP AND POSITION NUMBERS IN ONE SINGLE OPERATION, BY CLICKING THE **PRE-SET GROUP NUMBERS** BUTTON.

Assigning group numbers manually

If you want to transfer a group number to a symbol, you click the up- and down-arrows in the field below **Group Number** until the desired number is in the box.



Then select the symbols, which are part of the component, and click on the **Transfer Group Number** button, which transfers the group number to the selected symbol.



COUNTING UP AND DOWN BY DOUBLE-CLICKING

DOUBLE-CLICKING ON A COMPONENT ON THE LIST, WILL INCREASE ITS GROUP NUMBER BY ONE.

PRESSING DOWN <SHIFT> WHILE DOUBLE-CLICKING WILL DECREASE ITS GROUP NUMBER BY ONE.

THE ORDER OF THE COMPONENTS IN LISTS

When **Automation** fills out a list, the components are listed according to the following criteria:

- 1) Component name
- 2) Group number
- 3) Position number
- 4) Article number

This means, that for each symbol name, the components are printed out primarily ordered by group number, then ordered according to position number, and finally according to article number.

SPECIFYING THE ORDER IN PARTS LISTS

If you have constructed your own component, for instance a contactor with add-on contacts, you can therefore determine in which order the component parts should appear on the parts list.

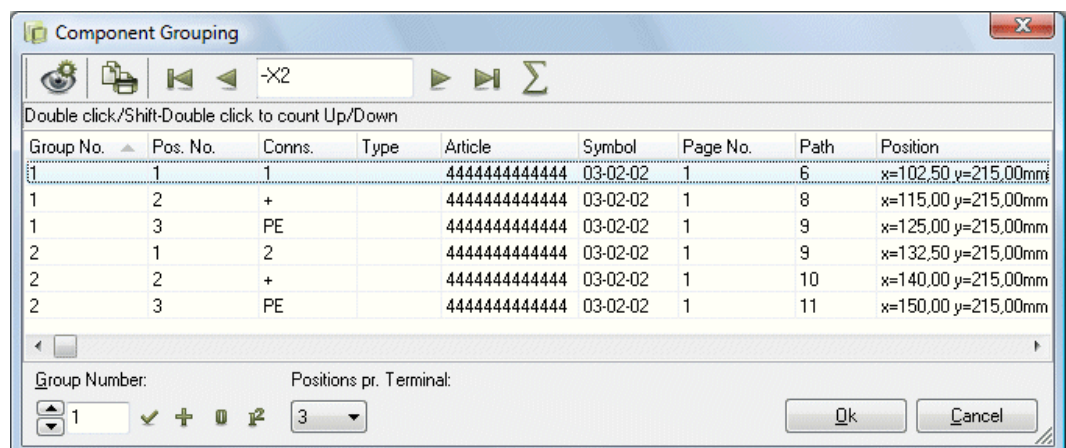
Because the lowest group numbers for the component appears first on lists, you do this by assigning the lowest group numbers to the parts that shall appear first.

In this way you can get the contactor first, followed by the auxiliary contacts.

SPECIFYING THE ORDER IN TERMINALS LISTS

The group numbers for the component also determines in which order the terminals are printed out in terminals lists.

For individual terminals you can therefore determine their order in the lists using the group numbers, so that they are printed out as they should be mounted.



— Please also see “Keep component grouping when copying terminals” on page 562.

END COMPONENT GROUPING

When you have completed the component grouping for a component, and change to another component name in the lower-left corner, you will be asked whether you want to transfer the changed group numbers to the component.

Click on **Yes**.

When you have finished grouping components, you click on **OK**, whereby you approve the component grouping you have carried out.

COMPONENT GROUPING AND COPYING

For everything which is being placed via the database, the component grouping is handled automatically by the program.

For all copying, the used component grouping is examined, and the component grouping is continued following the same directions for the inserted copies.



KEEP COMPONENT GROUPING WHEN COPYING TERMINALS

TO KEEP THE COMPONENT GROUPING WHEN COPYING TERMINALS, DO THIS: SELECT THE SYMBOLS, CHOOSE **EDIT => COPY (IDENTICAL)**, AND INSERT THE SYMBOLS. THIS WAY THE PASTED SYMBOL NAMES, CONNECTION NAMES AND GROUP NUMBERS REMAINS UNCHANGED.

WHEN YOU CHOOSE **EDIT => COPY (IDENTICAL)**, YOU CAN THEREFORE EASILY COPY TERMINAL ROWS, SO THAT THE INTERNAL SIDE IS DISPLAYED AT ONE PAGE, WHILE THE EXTERNAL IS DISPLAYED AT ANOTHER.

GRAPHICAL MANAGING OF COMPONENT GROUPING FOR TERMINALS

There also exists a more graphically oriented way of grouping terminals. To activate it, you must right-click on a terminal symbol, and choose **Show terminal row**. You then enter the dialog box **Show terminal row**:



The significance of the order of the terminals

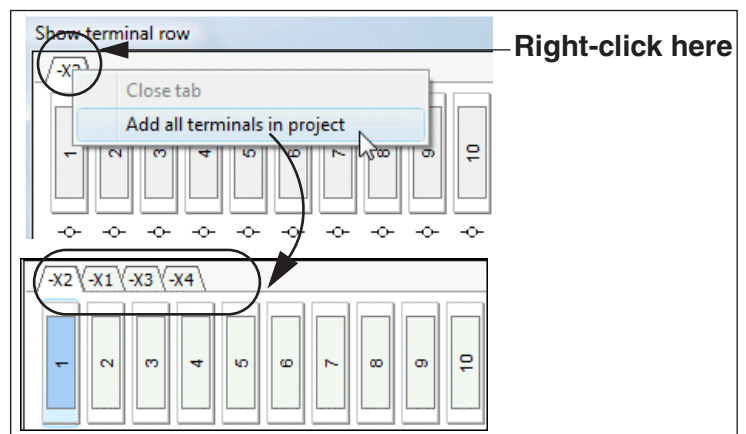
The order of the terminals in the dialog box, specifies the order, in which they subsequently appear, in parts lists, components lists, terminals lists and graphical terminal plans. When the terminals are placed using mechanical placing, this is also the order that they will appear in.

— If necessary, please refer to “When is Component grouping necessary” on page 556.

Show the terminal rows in the project

Initially, only the selected terminal row is displayed. When you right-click on the tab for the selected terminal row and choose **Add all terminals in project**, all terminal rows in the project appears as tabs in the dialog box.

To close a tab for a terminal row, just right-click on the tab, and choose **Close tab**.



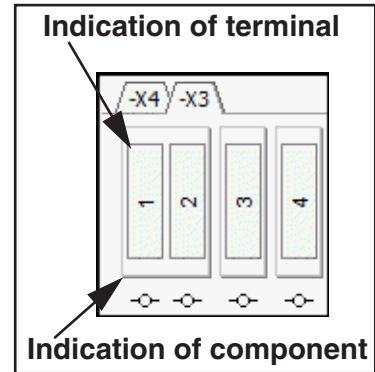
Selecting terminals

To select a terminal, just click on it in the dialog box. To for instance select the terminals 1 to 3 in the figure above, just click on terminal 1, and press <Shift> while clicking terminal 3. You can select further terminals by pressing <Ctrl> while clicking the terminals in the dialog box.

Grouping terminals in a component

In the dialog box **Show terminal row**, the individual terminal is indicated by a rectangle with a name/number inside (the connection name for the terminal symbol).

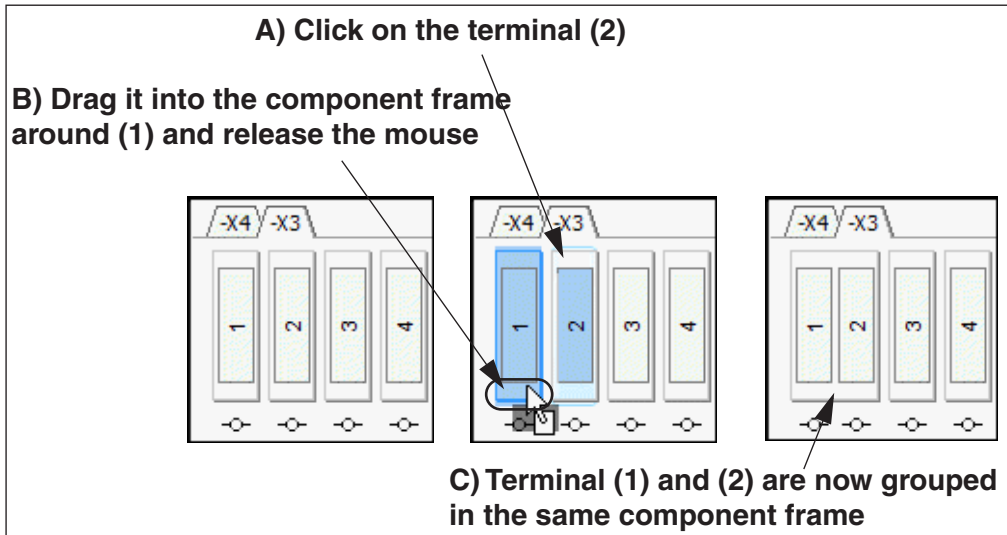
Around the individual terminals, another rectangle indicates which terminals are part of the same component. In the figure to the right **(1)** and **(2)** are therefore part of the same component.



GROUPING BY CLICK AND DRAG

To group two terminals: *Click* on a terminal, and then *drag* it into another terminal component.

Please note that you must point at the top or the bottom of the terminal when releasing the mouse button. — This meaning the frame around the component:

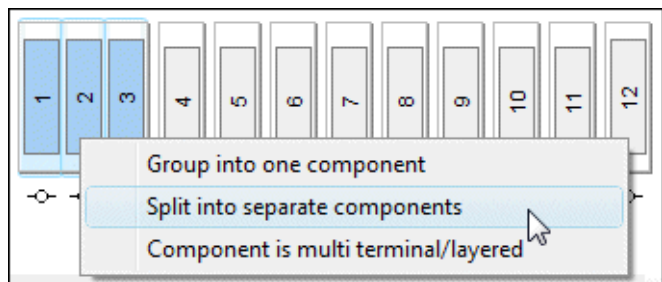


— Please also see “Changing the order of the terminals by click and drag” on page 565.

GROUPING BY RIGHT-CLICKING

You can also select the terminals you wish to group, right-click, and select **Group into one component**.

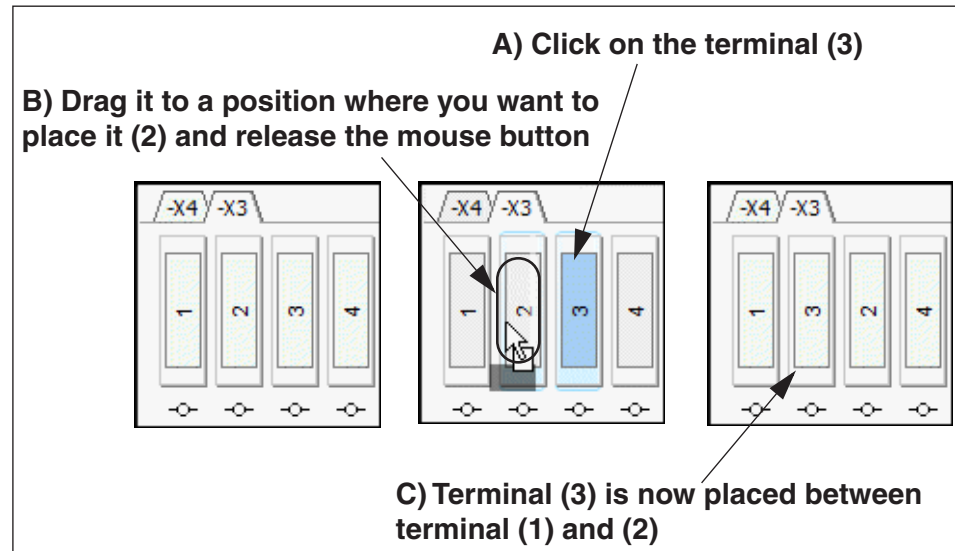
To cancel a grouping: Right-click on a grouped component and choose **Split into separate components**.



Changing the order of the terminals by click and drag

To change the order of the terminals in the dialog box: Click on the terminal you wish to move, *drag* it to the position where you wish to place it, and release the mouse button.

When you release the mouse button, you must point at a *terminal symbol* — at **(2)** in the figure below — and *not* at the top or at the bottom of the component frame:



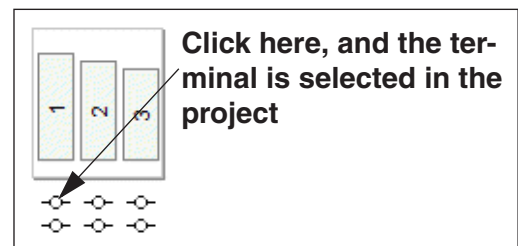
Specifying multi terminals / multi layer terminals

To specify that a grouped component is a multi terminal: Right-click on the grouped component, and choose **Component is multi terminal/layered**. This assigns position numbers to the terminals in the group.

Links to the terminals in the project

Below the terminals in the dialog box, you find a small symbol. When you click this symbol, the terminal is selected in the project.

When a terminal appears more than one place in the project — if for instance the internal and the external sides are placed on different project pages — there will be a link to each symbol for the terminal.

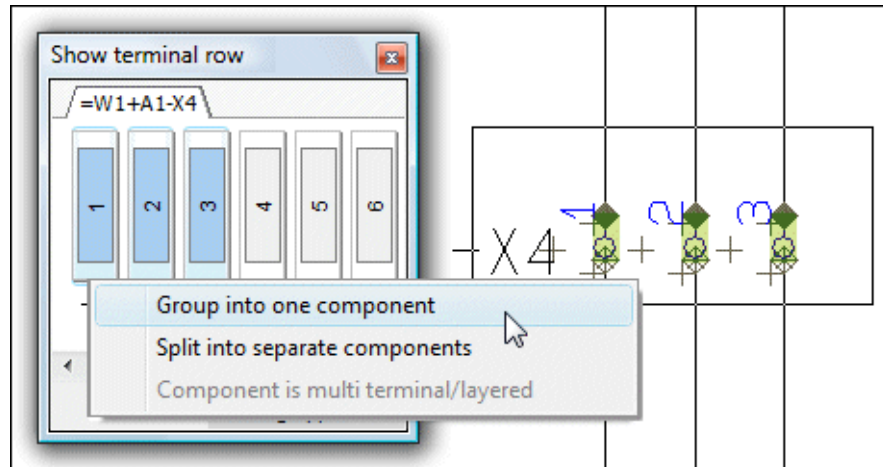


Example on graphical grouping of terminals

You can group three terminals, to become part of the same component, like this:

- 1) Open the **Show terminal row** window: Right-click on a terminal and choose **Show terminal row**.
- 2) Select the terminals in the diagram.

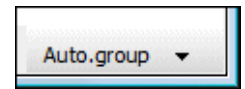
- 3) In the **Show terminal window** the terminals are now selected automatically:
Right-click on the terminals in the window and choose **Group to one component**.



- 4) The terminals are now grouped as one component.

Automatic grouping of terminals

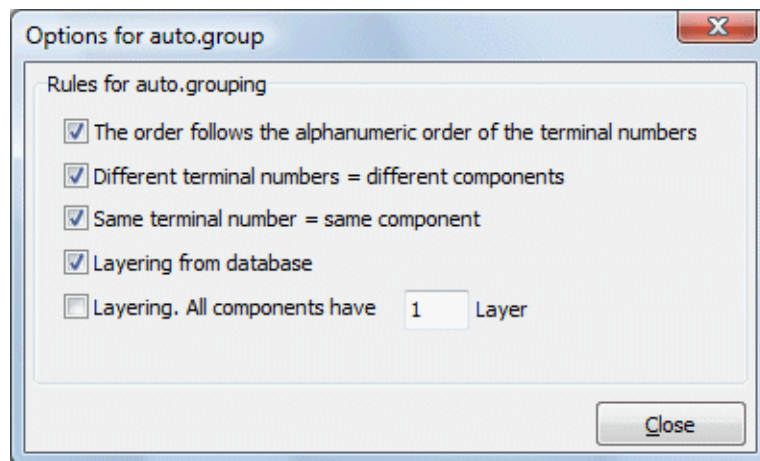
When you click the **Auto.group** button in the lower-right corner of the dialog box, the selected terminal row is grouped in the **Show terminal row** window.



When you click the down-arrow next to the **Auto.group** button and choose **Auto.group all shown terminal rows**, all terminal rows in the **Show terminal row** window are grouped.

Rules for automatic grouping

When you click the down-arrow next to **Auto.group** and choose **Options for auto.group**, you can specify how to group the terminal rows:

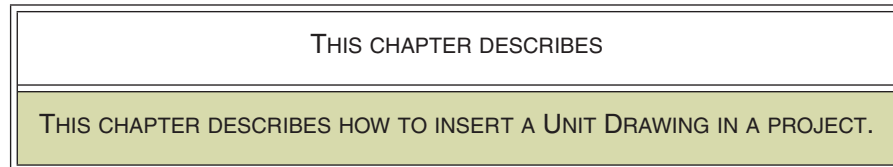


Here you have the following options:

OPTION	EFFECT
THE ORDER FOLLOWS THE ALPHANUMERIC ORDER OF THE TERMINAL NUMBERS	WHEN TERMINALS FOR INSTANCE ARE NAMED 1, 2 AND 3 , THEY ARE SORTED IN THIS ORDER. IF THEY ARE LISTED AS 3, 1, 2 IN THE WINDOW, THIS IS THEREFORE CHANGED TO 1, 2, 3 .
DIFFERENT TERMINAL NUMBERS = DIFFERENT COMPONENTS	WHEN THE TERMINALS ARE NAMED 1, 2 , AND 3 THEY BECOME THREE DIFFERENT COMPONENTS.
SAME TERMINAL NUMBER = SAME COMPONENT	WHEN THE TERMINALS ARE NAMED 1, 1 AND 1 THEY BECOME ONE COMPONENT.
LAYERING FROM DATABASE	THE LAYERING IS FETCHED FROM THE DATABASE.
LAYERING. ALL COMPONENTS HAVE _ LAYER	IF THE NUMBER OF LAYERS FOR INSTANCE ARE SET TO 3 , THE TERMINALS ARE GROUPED IN THREES. WITHIN EACH COMPONENT, THE TERMINALS GET THE LAYER NUMBERS (POSITION NUMBERS) 1, 2 AND 3 .

INSERTING UNIT DRAWINGS

In your projects you can insert the so-called *unit drawings*, specifying what an individual component consist of. It is then possible to attach the unit drawing to a symbol in the project. When you right-click on this symbol, you will then get the option to jump to the page where the unit drawing for the symbol is located.



When you use extended parts lists in your project, articles from the unit drawings are included in the parts lists. Read more about this in “Specifying from where to collect components” on page 544.

— See how to create a unit drawing in “Creating Unit Drawings” on page 767.

INSERT EXISTING UNIT DRAWING / PARTS LIST INTO A PROJECT

When you want to insert an already created unit drawing into a project, you enter the **Page menu** to create a new page, and then state that this page shall contain a unit drawing:



Click on the **Page menu** button, and then click on **Insert** or **Add**. In the emerging window you click on **Unit drawing** and click **OK**.

A new dialog box appears, and you click your way to the folder with the unit drawing, and click on the unit drawing. If you are not sure how to locate files, you can read more about this in “Windows” on page 23.

Click **OK** when you have found the unit drawing. You have now inserted the unit drawing into your project.

ATTACH A SYMBOL ON A DIAGRAM TO THE UNIT DRAWING

When you have placed the unit drawing in your project, you can enter one of your diagram pages and right-click on a symbol. Then choose **Component data => Unit**.

In the appearing list of unit drawings in the project, you select the relevant unit drawing, and click on **OK**. You then return to the **Component data** dialog box, and click **OK** again.

The program now knows which unit drawing is attached to the symbol.

When you then right-click on the symbol and choose **Open**, you automatically go to the page with the unit drawing for the symbol.



Databases

In this section you can see how to use the database in PCISCHMATIC Automation.

You can read about:

- Using the database p. 571
- Creating components in the database p. 589
- Database settings p. 608
- Choosing a database p. 623
- Programmable Barcode Scanners p. 624



Read survey article

Read the survey article about working with databases and lists at www.pcschematic.com/articles.



USING THE DATABASE

Automation has an attached database from which you can collect information about the components you use in your projects.

In **Automation** you can either use one of the component vendor databases for the program, or use the database which is installed automatically along with the program.

You can also create your own databases, by entering records in the databases by yourself, or by selecting the components you are interested in from the component vendor databases.

THIS CHAPTER DESCRIBES	
— HOW THE DATABASE WORKS (BASIC PRINCIPLES).....	P. 572
— ATTACHING COMPONENT DATA FROM THE DATABASE TO A SYMBOL	P. 573
— ABOUT THE DATABASE MENU	P. 575
— SEARCHING FOR COMPONENTS IN THE DATABASE MENU.....	P. 576
— EDITING RECORDS IN THE DATABASE	P. 582
— INTERNAL DATABASE.....	P. 584

You can also read about the database in the **PC|SCHEMATIC Database** manual, and in the following chapters.

Using the database for Lines

This chapter focus on using the database while working with symbols.



If you wish to attach article data to a line, this is also possible. To do this, activate the **Lines** button, deactivate the Pencil (press <Esc>), right-click on the line and choose **Line article data**. Then follow the same procedure as described in this chapter.

Please note that you can also apply article data to cable symbols, as described in “Working with cable symbols” on page 218.



SEE ALSO THE SURVEY ARTICLE ABOUT WORKING WITH DATABASES

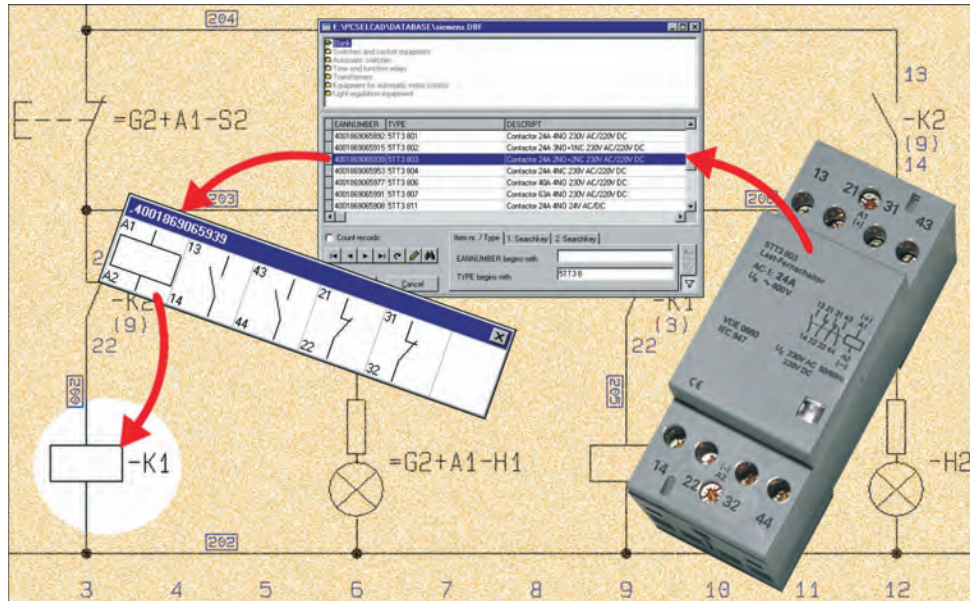
IN THE SURVEY ARTICLE IN THE BEGINNING OF THE SECTION “LISTS” ON PAGE 529, YOU CAN READ MORE ABOUT WORKING WITH LISTS AND DATABASES IN **Automation**.

HOW DOES THE DATABASE WORK?

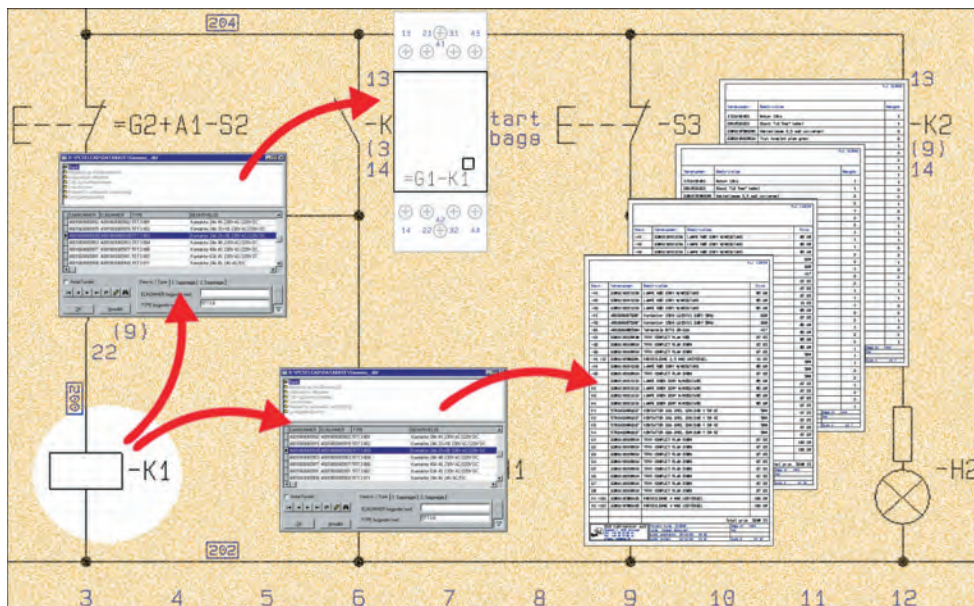
In the component vendor databases for **Automation**, the component vendors have included ordering data, as well as electrical and mechanical symbols for each component.

When you click on the component in the database, you automatically get a pick-menu containing all the electrical symbols for the component. These symbols each represent one of the functions in the component.

When you subsequently place the symbols on your diagrams, the program remembers which components these symbols represent.



When you have completed your electrical diagrams, the program can therefore automatically fill out components lists and parts lists etc. — without you having to type in any component information.



Originating from the electrical symbols on the diagrams, the program can also fetch the mechanical symbols for the components automatically. These symbols can for instance be used for mechanical layouts.

— Read more about this at www.pcschematic.com/articles.

ATTACHING INFORMATION FROM THE DATABASE TO A SYMBOL

In “Fetching symbols” on page 139 you will find a description of the many ways you can place symbols with and without information from the database.



If you want to attach information from the database to a symbol which is already placed on a diagram, you do like this: Click on **Symbols**, deactivate the Pencil (press <Esc>), right-click on the symbol, and choose **Component data**.

In the dialog box **Component data**: Click on **Database**, and you enter the database.

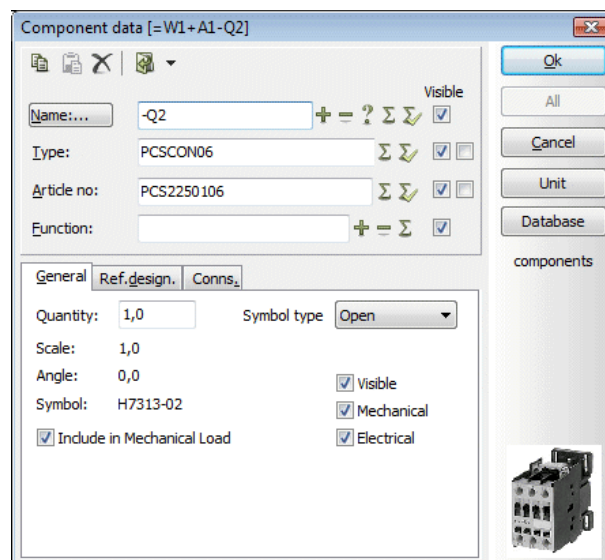
If an article number is already specified for the symbol in the **Component data** dialog box, the program locates the component when you enter the database.

If only a type text has been specified in the dialog box, all components having the same type text will be displayed in the database.

If no type text is specified, a list of components containing matching symbols will be displayed.

In the database you find the record for on the component you want to apply, and click **OK**.

You then return to the dialog box **Component data**, where you can see that the information have been transferred.

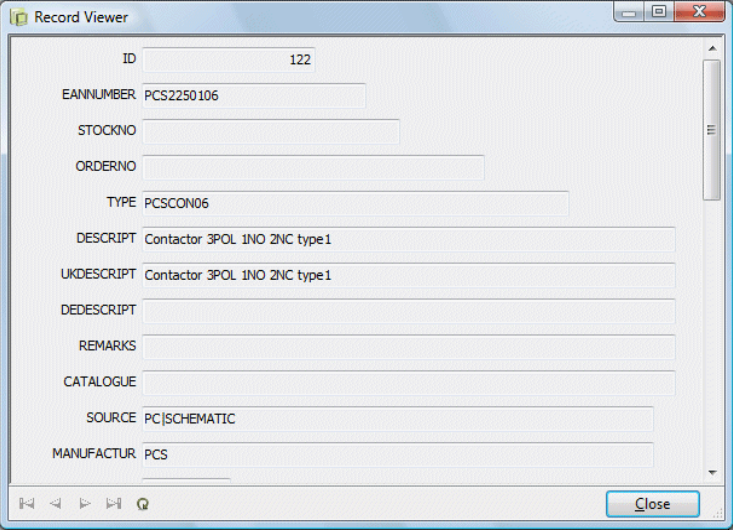


Click **OK**.

How to find the right component is described in the following.

Displaying database information for a symbol

When you want to see which information the database contains for a symbol, right-click on one of the symbols for the component in the project, and choose **Open**. You then enter the **Record Viewer**:



The Record Viewer window displays the following information:

ID	122
EANNUMBER	PCS2250106
STOCKNO	
ORDERNO	
TYPE	PCSCON06
DESCRIPT	Contactor 3POL 1NO 2NC type1
UKDESCRIPT	Contactor 3POL 1NO 2NC type1
DEDESCRIPT	
REMARKS	
CATALOGUE	
SOURCE	PC SCHEMATIC
MANUFACTUR	PCS

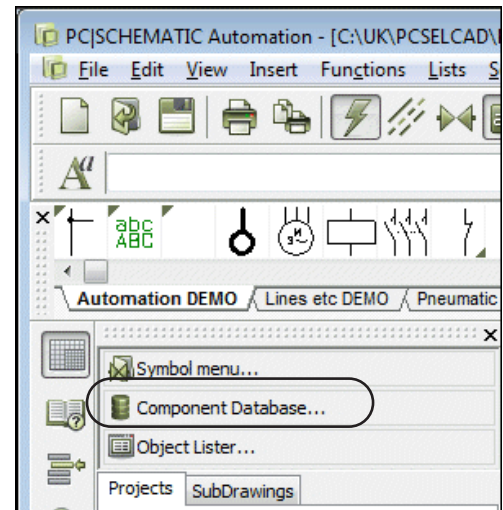
Please note that you can not change database information from here. — See how to change information in the database in “Editing records in the database” on page 582.

Click on **Close**.

ABOUT THE DATABASE MENU

When you wish to enter the **Database menu**, you can for instance press the short-cut key <d>, as described in “Fetching symbols from the database” on page 152.

You can also click the **Component Database** button in the upper part of the **Explorer** window, as displayed to the right. This button is always available, no matter which command you are working in.



The **Database menu** contains folders, each containing different groups of components.

Menu for the database

Records in database

EANNUMBER	TYPE	DESCRIPTION	SOURCE
3389110586381	AB1VW235U	Screw clamp terminal, 2,5 mm2	GRUPE SC
3389110586435	AB1VW435U	Screw clamp terminal, 4,0 mm2	GRUPE SC
3389110560039	AB1TP435U	Screw clamp terminal, 4,0 mm2, Earth	GRUPE SC
3389110585506	AB1NE435U	Screw clamp terminal, 4,0 mm2, Neutral, blue	GRUPE SC
3389110586411	AB1VW415	Screw clamp terminal, 4,0 mm2	GRUPE SC
4008190455149	WDU 2,5	Screw clamp terminal, grey	WEXO
4008190297756	WDU 2,5	Screw clamp terminal, white	WEXO
4008190297749	WDU 2,5	Screw clamp terminal, black	WEXO
4008190455248	WDU 4	Screw clamp terminal, brown	WEXO
4008190455293	WDU 4	Screw clamp terminal, grey	WEXO
4008190163440	WDU 6	Screw clamp terminal	WEXO
4008190100032	WDU 6	Screw clamp terminal, blue	WEXO

Open links (if any) for the selected record

Scroll/search among the records displayed

Specify which records to display

If you are using the **PC|SCHEMATIC Database** program you can create your own database menu. — Please refer to the **PC|SCHEMATIC Database** manual for further details.

If you are using your own database program, you can also create your own menu table there. However, it is recommended to create it in the **PC|SCHEMATIC Database** program first, and then transfer it to your own database system. This ensures you that the menu has the right format.

If you wish to change which database fields/information is displayed in the **Database menu**, please refer to “Database settings” on page 608.

SEARCHING FOR COMPONENTS IN THE DATABASE MENU

When you are in the **Database menu** (short-cut key <d>), you can find the component, that you are looking for, in various ways.

Please note that the following methods can be combined:

1. Choosing a folder in the article menu

For instance you can click on the folder in the article menu — in the upper-left corner of the dialog box — which contains the type of components you are looking for. The contents of the folder is then displayed, and you click on the desired component. However, if the folder contains many components, this will not be sufficient.

Display Fabricates with components from selected Article group only

If you have selected **Settings => Database => Component search** and have selected **Show only table menu points containing the selected fabricates components**, the **Fabricate menu** in the right-hand side only displays fabricates having components in the selected article group.

2. Selecting a Fabricate / Vendor

In the upper right-hand corner of the **Database menu** you can find a menu, from which you can choose which fabricate to display components for. — See how to activate this option in “Product choice or Fabricate choice in the Database menu” on page 613.

This fabricate menu works *in combination with* the common database menu (in which the components are grouped by function).



When clicking on a fabricate, all components being of this fabricate *and* being placed in the selected folder in the database menu, are displayed.

When for instance you click on **Automatic switches / connection material** in the menu, and on **AEG** in the fabricate menu, all components in the menu point **Automatic switches / connection material** of the fabricate **AEG** are displayed.

When you click on **FABRICATE (Show all)**, all fabricates in the selected component grouping menu point are displayed.

3. Activate search

In the lower-right corner of the dialog box, you have the option of searching in several ways.



If you activate the **Case sensitive query** button, the query will discern between upper- and lower case letters when you search for a component. (When the applied database format supports this).



If you activate the **Autorun query** button, the program will start searching each time you type in a new character in one of the search fields. If this button is not activated, the searching is not activated until you click the **Run query** button.

Specifying the EAN number or the type of the component

When you know the EAN number — or just the beginning of the number — you can click on the tab **Item No / Type** in the lower-right side of the dialog box. Here you type in the number in the field **EANNUMBER begins with**

Item No. / Type 1. Searchkey 2. Searchkey

EANNUMBER begins with

TYPE begins with

and only in Branch equipment

Click here only to search in the selected folder

with. The program then finds every component in the entire database, that matches the number. When you only wish to search in the selected folder, select **and only in Branch equipment** at the bottom of the dialog box (The menu point changes name according to the folder selected).

When working in the **Database menu**, the short-cut key <Ctrl+a> gives you direct access to this search field.

— Please note, that if you have specified the datafield **ITEMNO** as **Article number** in **Settings => Database => Database settings => Component data**, the field will be named **ITEMNO begins with** in stead.



SPECIFYING AN ALTERNATIVE SEARCH KEY

WHEN YOU CHOOSE **SETTINGS => DATABASE => DATABASE SETTINGS => COMPONENT DATA**, YOU CAN SPECIFY AN ALTERNATIVE SEARCH KEY — FOR INSTANCE THE FIELD **ORDERNO**. THE PROGRAM WILL THEN SEARCH FOR THE ALTERNATIVE SEARCH KEY, IF NONE OF THE CHOSEN RECORDS HAVE AN EAN NUMBER BEGINNING WITH THE NUMBER YOU TYPE IN.

Searching for component or type

When you know the type of the component, you can type it into the **TYPE begins with** field. When working in the Database menu, the short-cut key <Ctrl+t> gives you direct access to this search field.

Applying 1.Searchkey

When you wish to search on other fields in the database, you click on the tab **1. Searchkey**, or press the short-cut key <Ctrl+1>.

Here you choose which datafield you want to search on, an operator, and a text to compare the contents of the datafield with.

Datafield in the database Operator Text

Item No. / Type 1. Searchkey 2. Searchkey

PCSTYPE begins with 07-02-01

and TYPE contains ""

and only in Automatic switches /connection material

In the figure above the following is chosen:

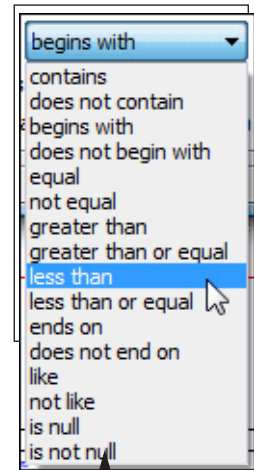
- The datafield **PCSTYPE**
- The operator ***begins with***
- The text *07-02-01*

This tells the program to find all records having a **PCSTYPE** that ***begins with*** the text *07-02-01*.

It is possible to choose from many different types of operators, which are chosen by clicking on the down-arrow in the operator field.



If the **Autorun query** button is activated, the program will find matching records, as you type in the text.



Operators

Applying 1. and 2. Search key

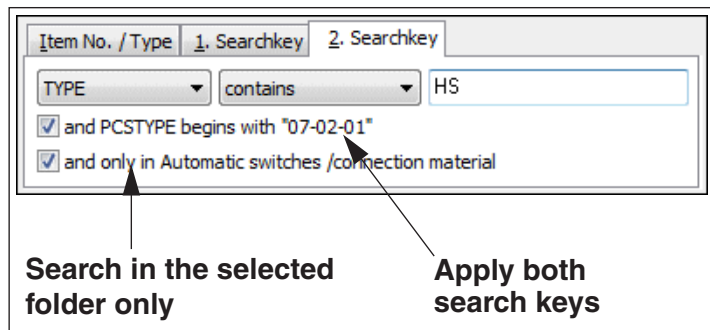
When you want to search on two datafields at the same time, you can choose to apply the first and the second search key simultaneously.

Specification of datafield, operator and text is similar to **1. Search key**. If you enable ***and PCSTYPE***

begins with "07-02-01", the program finds the records which fulfils both specified criteria. The text in this field will therefore differ, depending on the chosen search criterion on the tab **1. Searchkey**.

When you want to search in the folder chosen in the upper part of the menu, you enable ***and only in Automatic switches***. (Accordingly, the name of the folder will vary). If this function is not enabled, the search will be carried out in the entire database.

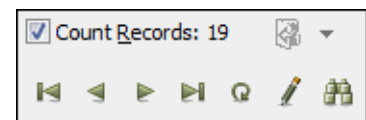
The short-cut key for the tab **2. Searchkey** is <Ctrl+2>.



Searching and editing among the found records

By applying the buttons in the lower-left corner, you can leaf through the found records, edit component information, or search among the found records.


If you enable ***Count records***, the program will automatically display how many records/components have been found, each time you have performed a new query. However, if you do not need this information, it is recommended that you disable this function, due to increased search rates.





Leafing through the found records




When you click the **First** button, you will be placed on the first component in the list.

 The **Previous** button gives you the component just above the actual chosen component.

 The **Next** button gives you the component just below the actual chosen component.

 The **Last** button gives you the last component in the list.

Update the database

 The **Refresh** button refreshes the records from the database, which are displayed on your screen. This is important when the database is located in a network, where other users can change the contents of the database while you are working with it.

Edit records



When you click the **Pencil**, you can edit, delete and add records to the database. Please read more about this in “Editing records in the database” on page 582.

Search for records



When you click the **Find** button you will enter the dialog box **Search**, where you can specify what you are searching for. You only search among the records which are listed in the **Database** dialog box for the moment — or in the selected folder.

The dialog box **Search** is elaborated in “Finding symbol texts” on page 316.

Please note that the dialog box here has an extra option: If you choose **Search all records**, the program will search in the entire database. If you do not choose this, the program will only search among the listed records.

Displaying Approved components only

When you select **Show approved components only**, only approved components are displayed in the **Database menu**.

— Read more about this in “Display only Approved components and hide Obsolete components” on page 616.



SETTING UP THE DATABASE MENU FOR APPROVED/OBSOLETE

TO WORK WITH APPROVED/OBSOLETE COMPONENTS, THE **DATABASE MENU** MUST FIRST BE SET UP FOR THIS. READ MORE ABOUT THIS IN “APPROVED AND OBSOLETE COMPONENTS IN THE DATABASE” ON PAGE 614.

Hiding Obsolete components

When you select **Hide obsolete components**, no obsolete components are displayed in the **Database menu**.

— Read more about this in “Display only Approved components and hide Obsolete components” on page 616.

Specifying that a component is Approved or Obsolete

To specify that a component is approved or obsolete, do the following:



- 1) Enter the **Database menu** (Press <d>).
- 2) In the **Database menu**: Find the component, click on it to select it, and click on the **Pencil**.
- 3) In the dialog box **Edit record**: Scroll down to the fields **Approved** and **Obsolete** and click on them, which sets a check mark in the fields.



- 4) Click the **Post changes** button, and click **Close**.
- 5) The changes are now displayed in the **Database menu**:

APPROVED	OBSOLETE	EANNUMBER	TYPE	DESCRIPTION
<input type="checkbox"/>	<input checked="" type="checkbox"/>	1122084084	LS 37.22	Contactor, 15KW, LS 37.22, 230V AC
<input checked="" type="checkbox"/>	<input type="checkbox"/>	4022903085584	B775	Overload relay, 20-32A
<input checked="" type="checkbox"/>	<input type="checkbox"/>	3389110231632	LC1D4011M7	Contactor, LC1D4011M7, 220V-40A
<input checked="" type="checkbox"/>	<input type="checkbox"/>	3389110232936	LR2D3353	Overload relay, 23-32A
<input checked="" type="checkbox"/>	<input type="checkbox"/>	4022903075387	LS15K11	Contactor, 15KW, LS15K11, 230V AC
<input type="checkbox"/>	<input checked="" type="checkbox"/>	1122001339	LS15K11	Contactor, 15KW, LS15K11, 24V AC

Show approved components only Hide obsolete components

Click here **not** to see the obsolete components
 Click here to display approved components only

Please note that it is only possible to set a check mark in the **Approved** and **Obsolete** fields if these fields are created as boolean fields in the attached database. — See “Using non-boolean fields for Approved/Obsolete” on page 615.

Open link for selected record in Database menu

When you in the **Database menu** have selected a record, which contains fields with links, you can activate these links directly from the **Database menu** — without having to open the relevant record in the **Edit record** dialog box. This can be done by clicking the **Open link** button in the **Database menu**.



When you let the mouse cursor rest over this button, it is displayed which link field that will be opened, when you click on the button.

Click here to open link for the selected record

The link is displayed here

Specify preferred link field in the database

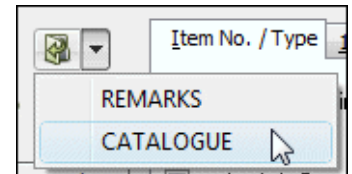
When a record for a component has more than one field containing links, the program will open the first field in the record containing a link, when you click the **Open link** button.



However, you can specify a **Preferred Link Field**, as described in “URL links” on page 620. The link from this specified field will then be opened, when you click the **Open link** button.

Choosing between several fields with links

When you click the down-button to the right of the **Open link** button, you get a survey of the fields in the selected record, that contains links. When you click on a field in the list, the link in the selected field is opened.

**SHOW PICTURES OF COMPONENTS IN THE DATABASE**

If you have pictures of your components in the database, it is possible to show these in the **Database menu**.

This requires the following:

- 1) There must be created a field in the database to specify where the pictures of the components are placed
- 2) The database must contain links to the correct pictures for the individual components. — It is recommended that you use the so-called **File Link Alias** for pictures, which you can read more about in “URL links” on page 620, especially in “Example for pictures” on page 621.
- 3) That you have selected **Show picture** in the lower-right corner of the **Database menu** dialog box.

Also displayed in the dialog box Component data

If a picture is selected for the component in the database, the picture is automatically displayed in the dialog box **Component data**, when you select a symbol for the component on a project page.

The file format for the pictures must be either **bmp, jpg, png** or **gif**.

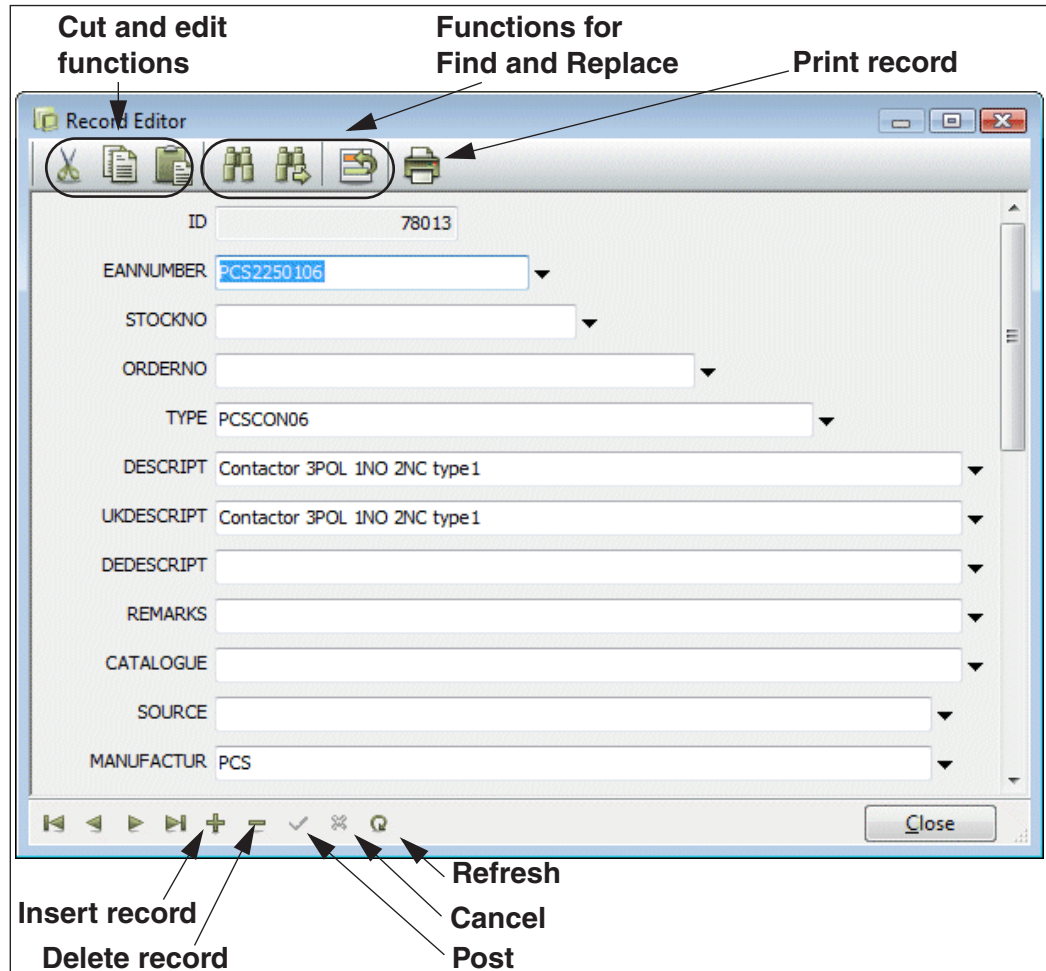
EDITING RECORDS IN THE DATABASE



When you click on the **Pencil** in the **Database menu** — see “About the Database menu” on page 575 — you enter the dialog box **Record Editor**, where you can edit the records/components, that are listed in the **Database** dialog box the moment.

If the Pencil is *not visible*, you do not have permission to alter the contents of the database.

The dialog box **Record Editor** looks like this:



When you click the **Find** button you enter the **Find** dialog box, where you specify what you are looking for. You only search among the records that are listed in the **Database** dialog box. If you click the **Find again** button, the next record fulfilling the search criteria is found. This also happens when you press <F3>.



When you click the **Replace** button, you enter the dialog box **Find & Replace**. The information is not transferred to the database until you click the **Post** button.

— Please read more about the dialog boxes **Find** and **Find & Replace** in “Find and Replace” on page 316.

Copy / Insert full records in the dialog box Edit Record



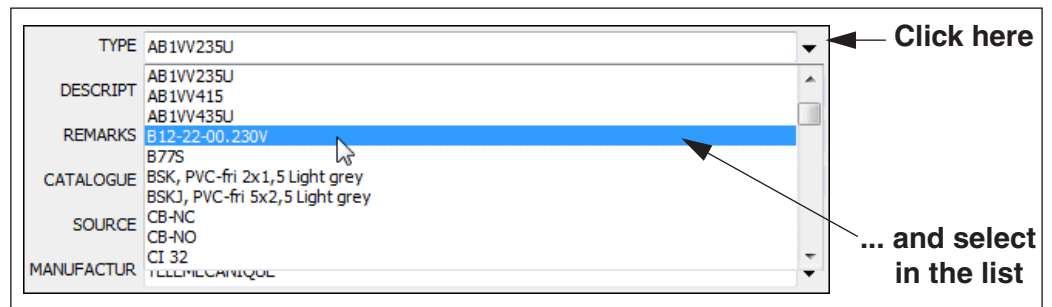
To copy a full record in the dialog box **Edit Record**, you must press the <Ctrl> key when clicking the **Copy** button.



To then insert the full record you must press the <Ctrl> key again when clicking the **Insert** button.

Drop-down lists in the dialog box **Edit Record**

In the dialog box **Edit Record** there are drop-down lists for all editable textfields, which enables you to select the existing values in the database.



PROJECT COMPONENT DATABASE

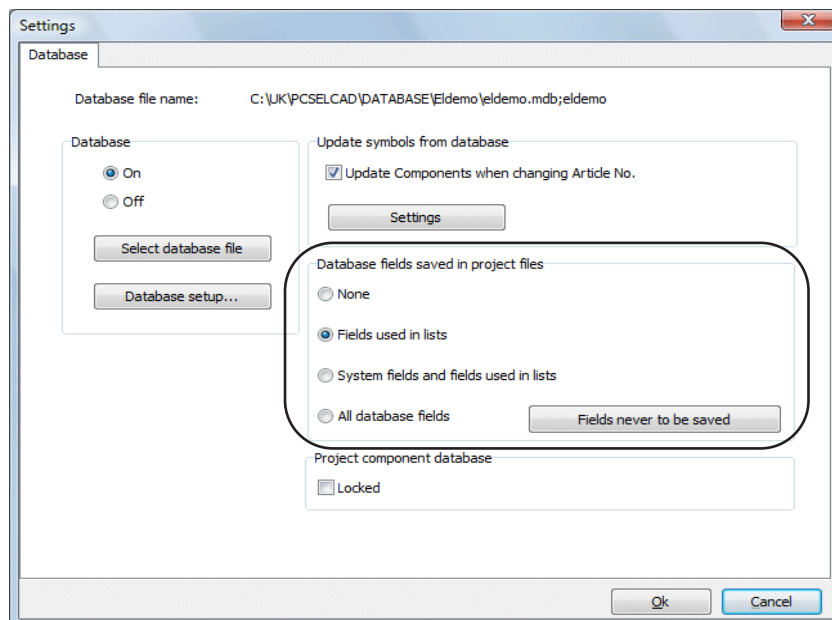
A "project component database" is a component database, which is included in the project file. The project component database contains component data for all components used in the project.

This makes it possible for you to take the project with you without losing component data, when you don't have access to your **Automation** component database. Or, you can send the project to someone else, that doesn't have any access to your **Automation** component database.

When a project has a project component database, the project fetches data from the project component database, before searching for data in the **Automation** component database. — How the component database works, depends on whether it is *locked*. Read more about this in "Locking the project component database for the active project" on page 587.

Specify what to include in the project component database

To specify which database fields to include in the project component database, select **Settings => Database**:



Below **Database fields saved in project files**, you have the following options:

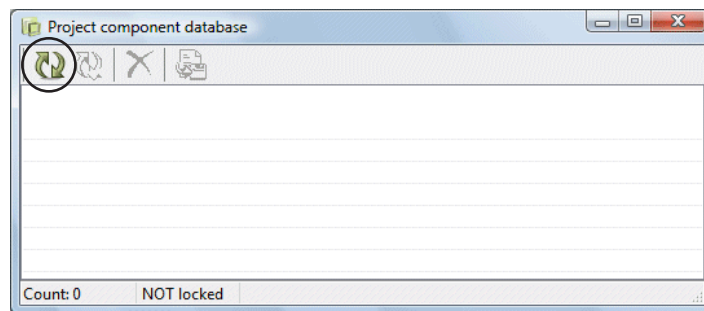
OPTION	MEANING
NONE	NO PROJECT COMPONENT DATABASE WILL BE ATTACHED TO THE PROJECT.
FIELDS USED IN LISTS	ALL DATA REQUIRED FOR UPDATING THE APPLIED LISTS, ARE INCLUDED IN THE PROJECT COMPONENT DATABASE.
SYSTEM FIELDS AND FIELDS USED IN LISTS	THE DATA MENTIONED ABOVE ARE INCLUDED, AS WELL AS THE FIELDS NEEDED BY THE PROGRAM TO DISPLAY AVAILABLE SYMBOLS ETC. FOR THE USED COMPONENTS.

OPTION	MEANING
<i>ALL DATABASE FIELDS</i>	ALL DATABASE INFORMATION FOR THE USED COMPONENTS ARE INCLUDED. PLEASE NOTE: ANY TYPE OF PRICE INFORMATION WILL ALSO BE INCLUDED.
<i>FIELDS NEVER TO BE SAVED</i>	WHEN YOU CLICK HERE, YOU ENTER A LIST, WHERE YOU DESELECT THE INFORMATION (FIELDS) THAT YOU DO NOT WISH TO INCLUDE.

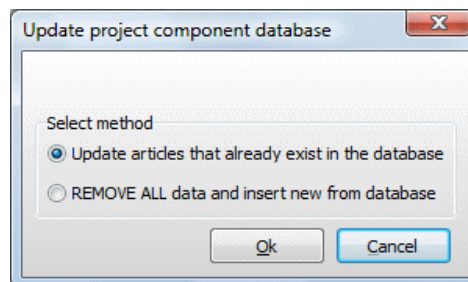
Updating the project component database

To update the project component database, do this:

- 1) Select **Functions => Database => Project component database.**



- 2) In the dialog box **Project component database**: Click the **Update** button.



- 3) In the dialog box **Update Project component database**:
 When you select ***Update articles that already exists in the database***, only article numbers existing in the project component database will be updated.
 When you select ***REMOVE ALL data and insert new from database***, all data in the project component database are deleted, and are inserted again from the program component database.
 Click **OK**.

4) The project component database is now updated:

EANNUMBER	ID	STOCKNO	ORDERNO	TYPE	DESCRIPT
1723410403	53			ZEFG45789	Motor 5KW
3389110585506	8			AB1NE435U	Screw clamp
3389110586435	6			AB1VV435U	Screw clamp
3389110610024	2			XB2BA31	Push button,
3389110610048	3			XB2BA42	Push button,
3389110611229	59			XB2BV74	Signal lamp,



ALSO UPDATED WHEN LISTS ARE UPDATED

WHEN PROJECT LISTS ARE UPDATED, THE PROJECT COMPONENT DATABASE IS ALSO UPDATED.

Editing the project component database

When you select **Functions => Database => Project component database**, you have the following options for editing the project component database:

Update selected components

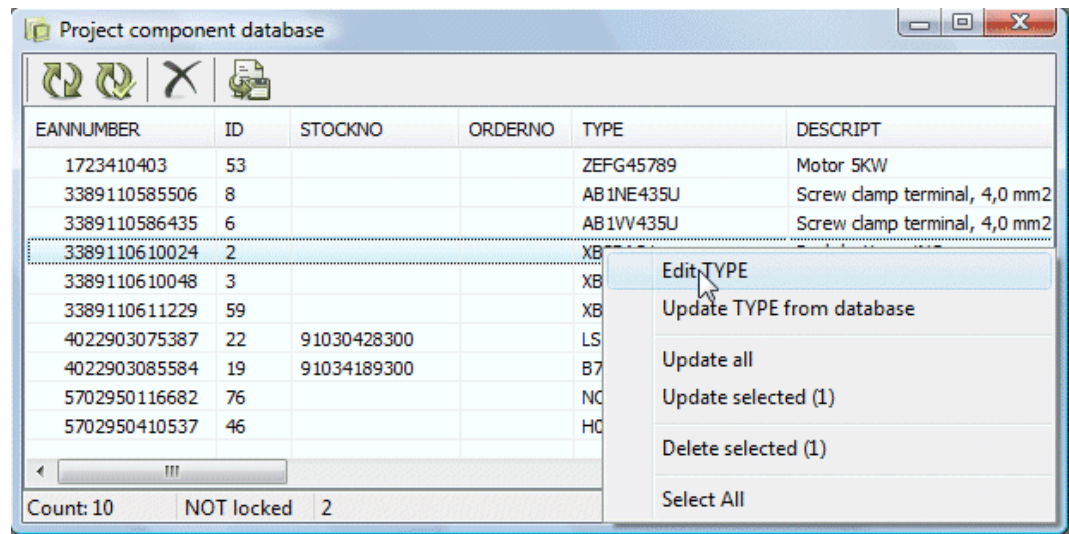
Delete selected components

Export selected components to file

Update all components

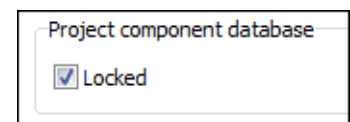
EANNUMBER	ID	STOCKNO	ORDERNO	TYPE	DESCRIPT
1723410403	53			ZEFG45789	Motor 5KW
3389110585506	8			AB1NE435U	Screw clamp terminal, 4,0 m
3389110586435	6			AB1VV435U	Screw clamp terminal, 4,0 m
3389110610024	2			XB2BA31	Push button, 1NO, green
3389110610048	3			XB2BA42	Push button, 1NC, red
3389110611229	59			XB2BV74	Signal lamp, red, 220V w/ re
4022903075387	22	91030428300		LS15K11	Contact, 15KW, LS15K11
4022903085584	19	91034189300		B77S	Overload relay, 20-32A
5702950116682	76			NOIK 19 X 1.5B mm	NOIK 19 x 1.5 mm
5702950410537	46			H07rn-F	Rubber cord, 4x2,5mm2

When you right-click on a field in the dialog box **Project component database**, you have these options:



Locking the project component database for the active project

Please note that how the project component database works, depends on whether it is **locked** or not. You specify whether the component database is locked in the lower-right corner of the **Settings => Database** dialog box.



Generally speaking, you can say that:

- When the project component database *is* locked, the project fetch component data from the project component database.
- When the project component database is *not* locked, the project fetch component data from the PC|SCHEMATIC database. When the project component database is *not* locked, it works just like in version 10 and previous versions.

When the project component database is NOT locked

- All component data are fetched from the PC|SCHEMATIC database, and the project component database is updated correspondingly, when new components are added to the project.
- When the project is saved, all unused component data are deleted from the project component database.
- If you edit the contents in the project component database: Please note that your changes may be lost when the project component database is updated by the program.

When the project component database IS locked:

- When *new* components are fetched from the **Database** menu — and these components are not represented in the project component database — their component data are still fetched from the attached PC|SCHEMATIC database.

- Component data for all *placed* project components are fetched from the project component database. Data are *only* fetched from the PC|SCHEMATIC database, when component data are *not* found in the project component database.
- When lists are updated, unknown component data are added to the project component database. After this, all data are fetched from the project component database.
- When you edit data in the project component database, you can be sure that the changes are not lost. Existing data in the project component database are never changed automatically. The only time the project component database is updated automatically by **Automation**, is when *new* component data are *added* when lists are updated.
- The datafields (columns) in the project component database are not changed by the program, unless you choose **Update** in the **Project component database** dialog box. — See “Editing the project component database” on page 586.
If for instance the **PCSTYPE** datafield is not included in the project component database, this has the consequence that you can not use the **Show available** function, until the project component database is updated with these datafields.

CREATING COMPONENTS IN THE DATABASE

In **Automation** you can create components in the database with attached electrical and mechanical symbols.

THIS CHAPTER DESCRIBES	
— GRAPHICAL CREATION OF NEW COMPONENTS.....	P. 589
— INFORMATION ABOUT THE COMPONENT IN THE DATABASE	P. 591
— WHEN THE COMPONENT ALREADY EXISTS.....	P. 592
— THE ELECTRICAL AND MECHANICAL SYMBOLS FOR THE COMPONENT	P. 593
— ACCESSORIES IN THE DATABASE.....	P. 603
— SPECIFYING THE SYMBOL REFERENCE LETTER FROM THE DATABASE	P. 605
— SPECIFYING SYMBOL SUB NAMES FROM THE DATABASE	P. 606
— CREATING PLC COMPONENTS IN THE DATABASE	P. 606

In the manual for the included database program **PC|SCHEMATIC Database** you can read more about the options for structuring your database — and much more.



YOU CAN ALSO USE THE COMPONENT WIZARD

THERE ALSO EXISTS A "COMPONENT WIZARD", WHICH GUIDES YOU THROUGH CREATING NEW COMPONENTS FOR PC|SCHEMATIC DATABASES. IT IS ACTIVATED BY CHOOSING **FUNCTIONS => COMPONENT DATABASE => COMPONENT WIZARD**, AND IS DESCRIBED IN A MINI MANUAL, WHICH CAN BE DOWNLOADED AT www.pcschematic.com/manuals.

GRAPHICAL CREATION OF NEW COMPONENTS IN THE DATABASE

If you want to create a new component in the database with attached electrical symbols, and a mechanical symbol, you first have to place all of the symbols for the component in the same project.

Placing all of the symbols for the component in the same project

If you are not already working in a project where all of the symbols for the component exists, do the following:

- 1) Create the electrical symbols and the mechanical symbol — if these does not exist in one of the existing libraries — see "Creating Symbols" on page 696
- 2) Open or create a project containing both a page of the type **Diagram** and a page of the type **Ground plan** — see "Creating a new page" on page 495
- 3) Click on the **Symbols** button

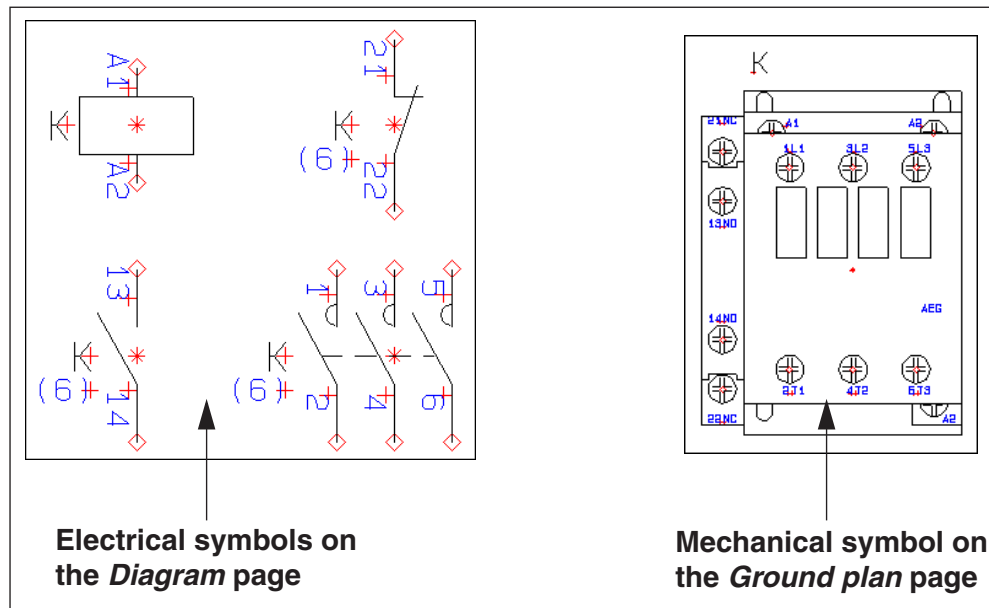


4) Place all of the electrical symbols for the component on the **Diagram** page, and the mechanical symbol on the **Ground plan** page. Assign the same name to all of the symbols

5) Right-click on one of the symbols, choose **Select Name**, and click on the **Object data** button



6) In the dialog box **Component data**: Type in **Article no** and **Type** for the symbols



You are now prepared to create the component in the database, as described below.



IN THE DATABASE THE SYMBOLS ARE ASSIGNED THE CONNECTION NAMES USED IN THE PROJECT. IT CAN THEREFORE BE NECESSARY TO CHANGE THESE BEFORE THE COMPONENT IS ADDED TO THE DATABASE.

THE SYMBOLS ARE ARRANGED AS THEY ARE PLACED IN THE PROJECT — IN THE SAME ORDER AS YOU READ A BOOK; THIS MEANS SYMBOLS IN THE TOP LEFT CORNER ARE PLACED FIRST, WHILE SYMBOLS IN THE LOW RIGHT CORNER ARE PLACED LAST.

Creating a component in the database using an existing project

When you wish to add a new component to the database from an existing project, do the following (please refer to “Placing all of the symbols for the component in the same project” on page 589 to see how to add **Article no** and **Type** information to the component):



- 1) Click on the **Symbols** button
- 2) Right-click on one of the symbols for the component
- 3) Choose **Select Component**

- 4) Choose **Functions => Database => Insert in Database** (or press down <Shift+Ctrl> and press <d>)
- 5) You now enter the dialog box **Edit Record**, from where you create the component as a record in the database — see “Information about the component in the database” on page 591

If a component with the specified article number already exists, you will be informed about this, as described in “When the component already exists in the database” on page 592.

You have now created a new component in the database.

All electrical and mechanical symbols in the project with the same symbol name and article number, will be attached to the component in the database.



SUB NAMES, ALTERNATIVE SYMBOLS, # AND SYMBOL TYPE

Automation ALSO DETECTS IF YOU APPLY SUB NAMES, ALTERNATIVE SYMBOLS, # FOR SPECIFYING MULTIPLE SYMBOLS WITH IDENTICAL NAMES, AND CHANGES IN SYMBOL TYPE COMPARED TO THE SPECIFIED TYPE IN THE DATABASE.

INFORMATION ABOUT THE COMPONENT IN THE DATABASE

When you enter the **Edit Record** dialog box, you can create new components as records in the database, as well as edit already existing records. — You can read more about editing records in “Editing records in the database” on page 582.

If you have entered the dialog box via a graphical creation of new records — as described in “Graphical creation of New Components in the Database” on page 589 — some of the datafields of the record have already been filled out.

Click and *drag* here to see more of the record

Fill out the relevant datafields with text information by clicking in the text fields, and typing in the information. Click on the **Post** button when you have completed typing in information. Then click on **Close**.



In some of the fields you have the option of specifying connection names, reference letter for the symbols and much more. — These options are described in “Electrical and Mechanical symbols for components” on page 593.

Here the use of the following datafields are elaborated:

DATAFIELD	IS DESCRIBED IN
PCSTYPE	“ELECTRICAL SYMBOLS” ON PAGE 593.
PINDATA	“CONNECTION NAMES/PINDATA” ON PAGE 598.
MECTYPE	“MECHANICAL SYMBOLS” ON PAGE 603.
ADDITIONALS	“ACCESSORIES IN THE DATABASE” ON PAGE 603.
REFID	“SPECIFYING REF. LETTER FOR THE SYMBOL IN THE DATABASE” ON PAGE 605.

WHEN THE COMPONENT ALREADY EXISTS IN THE DATABASE

If you are trying to create a component with an article number, which already exists in the database, you are informed about this. You are also told which changes this will cause in the database, if you enter the new information about the component in the database.

If you here click **OK** the component is created with the new contents, but if you click **Cancel** the previous component remains in the database.

— Please note that the program does not check whether the new component contains the same symbols as the old, arranged in a different order. The main purpose of this check is just to inform you that a component with the same article number already exists in the database.

ELECTRICAL AND MECHANICAL SYMBOLS FOR COMPONENTS

When you attach symbols to components in the database, you concentrate on three different datafields. These are the datafields **PCSTYPE**, **PINDATA** and **MECTYPE**. (However, it is possible to add extra fields for different types of mechanical symbols). These datafields contains information on the electrical symbol(s), the connection names, and the mechanical symbol (optional) for the component.

THIS CHAPTER DESCRIBES	
— ELECTRICAL SYMBOLS IN THE DATABASE	P. 593
— CONNECTION NAMES / PINDATA IN THE DATABASE	P. 598
— ADDRESSING PLC SYMBOLS VIA THE DATABASE	P. 601
— MECHANICAL SYMBOLS IN THE DATABASE	P. 603

Electrical symbols

In the database field **PCSTYPE** you specify the electrical symbol(s) that represents the component on diagrams in the **PCschematic** program.

PCSTYPE	07-02-01#3;07-02-03
PINDATA	53,54;73,74;83,84;61,62
MECTYPE	LA1-DN31

In this datafield you type in the name of the file, in which the electrical symbol is saved.

The program then searches for these files in the directory specified in **Settings => Directories => Symbols**. — Read more about this in “Directories” on page 840.

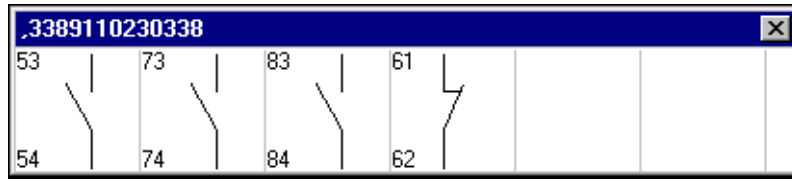
OPTIONS FOR ELECTRICAL SYMBOLS IN THE DATABASE	
— COMPONENTS WITH MORE THAN ONE ELECTRICAL SYMBOL	P. 593
— SPECIFYING THE SYMBOL TYPE	P. 594
— ALTERNATIVE ELECTRICAL SYMBOLS	P. 594
— SPECIFYING SYMBOL STATES	P. 596
— ACTIVATING REFERENCES	P. 596
— SUPPORT SYMBOLS IN THE PCSTYPE FIELD	P. 597

Components with more than one electrical symbol

If the component consists of several electrical symbols, you apply a semicolon to separate the symbols. If a component contains more symbols of the same kind, you can type in a number sign (#) followed by the number of symbols, immediately following the file name.

In the example above, **07-02-01#3** means that the component contains three electrical symbols of the type **07-02-01**. — If you need to create components with alternative electrical symbols for the functions of the component, please refer to “Specifying alternative electrical symbols via the database” on page 594.

When you are working on a **Diagram** page in **Automation**, enter the database, and select this component, you will get these electrical symbols to choose from:



The symbol **07-02-01.sym** The symbol **07-02-03.sym**

Specifying the symbol type via the database

If you want to specify a different symbol type for a symbol than the one it was created with — see “Saving the symbol” on page 705 — you can also do this in the symbol name in the **PCSTYPE** datafield. All you have to do, is to place a sign of equation and a letter following the symbol name.

Thus, if the symbol **07-01-03** should have a symbol type as for instance *Main reference*, you then write **07-01-03=M**. — However, you cannot apply this facility on multi symbols — see “Creating a symbol as a Multi symbol” on page 714 — or for symbols using # to specify that the symbol is used several times for the component.

Below you see which letters results in which symbol type.

LETTER	SYMBOL TYPE	LETTER	SYMBOL TYPE
C	CLOSE	R	RELAY
M	MAIN REFERENCE	S	SWITCH
N	NORMAL	T	TERMINAL
O	OPEN	W	WITH REFERENCE
P	PLC		

SPECIFYING SYMBOL TYPE FOR TERMINALS WITH ADDITIONAL SYMBOL TYPE

When a terminal symbol shall be assigned more than one symbol type — such as **Terminal** and **Relay** or/and **Terminal** and **Open** — you specify it like this: **07-15-01=T,R;07-02-01=T,O**

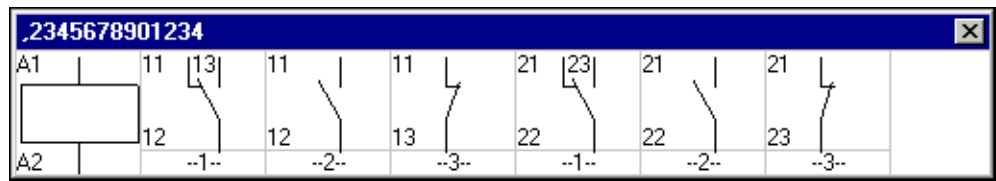
— Read more about terminal symbols with more than one symbol type in “Creating terminal symbols with additional symbol types” on page 720.

Specifying alternative electrical symbols via the database

When you fetch a component via the database — for instance from the pickmenu — it is possible to specify alternative electrical symbols for the functions of the component.

If for instance a component has relay coil and two switch functions, the switch functions can be used as both a switch, a make contact and a break contact.

When you get the pickmenu for a such component, it could appear like this:



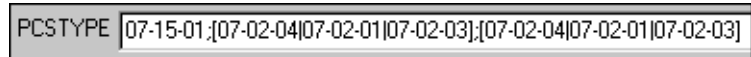
**Alternative electrical symbols
(1-2-3)**

**Alternative electrical symbols
(1-2-3)**

When you fetch one of these alternative electrical symbols, the other alternative symbols for the function disappears. — Read more about alternative electrical symbols in “Alternative electrical symbols” on page 142.

SPECIFYING ALTERNATIVE SYMBOLS

When you specify that you can choose between more than one symbol for one of a components functions, you write the options for the function in square brackets with a vertical line between the options.



The fact that you can choose between the options 07-02-04, 07-02-01 and 07-02-03 is therefore specified by writing [07-02-04|07-02-01|07-02-03]. When one of these symbols is selected, the other two disappears from the pickmenu for the component.

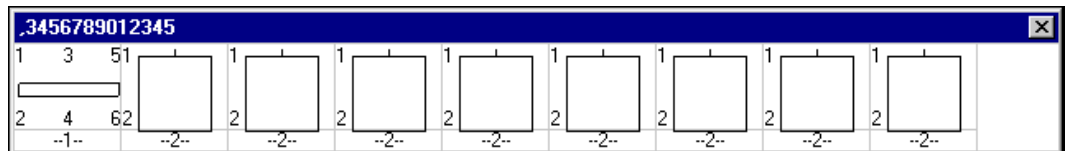
You can also specify repetitions of "or"-sections, by typing #n after the last square bracket in the section —where n represents the number of repetitions. In the figure above, the section [07-02-04|07-02-01|07-02-03] appears twice. That could therefore be specified like this: [07-02-04|07-02-01|07-02-03]#2

ALTERNATIVE SYMBOLS FOR PLCs

For PLCs you can thus choose whether to draw using one PLC module, or using single I/O symbols.



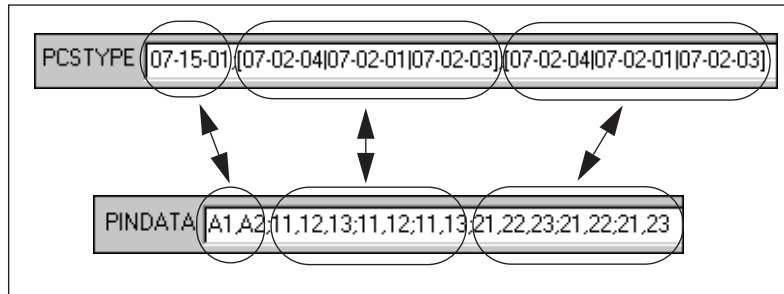
In the figure above you can choose between one PLC symbol with eight inputs, or eight symbols with one input each.



If you choose one of the symbols with one input, the symbol with the eight inputs disappears from the pickmenu. Also, the eight other symbols disappear if you choose the symbol with the eight outputs.

ALTERNATIVE SYMBOLS AND PINDATA

When you have specified the alternative **PCSTYPE**'s, you specify their connection names in the **PINDATA** field — in the same order as the symbols are listed in the **PCSTYPE** field.



— Please note that you shall not mention anything about alternative **PCSTYPE**'s in the **PINDATA** field.

Specifying symbol state for electrical symbols via the database

For electrical symbols created with states — see “Creating symbols with states” on page 712 — you can specify which state the symbol shall be in, when it is placed in the project. You specify this in the **PINDATA** field using the **§** character succeeded by the state number.

If the symbol **XXXXX** is to be placed in state **2**, you therefore write **XXXXX§2** in the **PCSTYPE** field.

— See also “Symbols with states” on page 167 and “Symbols with states and Show available” on page 167.

Activating references on electrical symbols via the database

When you do not specify information about references for a symbol in the database, the symbol references are activated/deactivated as specified in the symbols definition, when the symbol is placed in a project.

To control the activation/deactivation of references on a symbol via the database, the symbol filename in the **PCSTYPE** field must be followed by a *****, followed by either **N**, **W** or **M**.

This has the following meaning:

- 06-05-04 ***N** When the symbol is created with the reference, the reference is deactivated.
- 06-05-04 ***W** The reference will be activated, no matter what the symbol definition says.
- 06-05-04 ***M** The reference will be activated and be set to master reference, no matter what the symbol definition says.

The function has no effect on symbols, which through their symbol type already have references (e.g. **Relay**, **Open**, **Close** or **WithReference**), unless the symbol type have first been changed using the **=** function — see “Specifying symbol state for electrical symbols via the database” on page 596.

REFERENCE CONTROL COMBINED WITH OTHER OPTIONS

The control of the symbol references via the database, can be combined with the other parameters for controlling symbol properties via the database. Example:

07-02-03=T,P\$3*W The symbol gets the symbol type Terminal and PLC (=T,P), is set to state 3 (\$3) and the reference is activated (*W).

Support symbols in the PCSTYPE field

In the field **PCSTYPE** it is also possible to specify that there shall be placed one or more extra components along with the selected component — components which are not known prior to the moment of placing the component. So-called support symbols.

For a lamp this could be a blue, yellow or red glass, where you prior to the placing of the component do not know which color of the glass is needed — and, consequently, which article number to apply.

If you specify a symbol of the type **Support** in the **PCSTYPE** field, this symbol will be displayed when you select **Show available** for the component — see “The function Show available” on page 232 —, so that you will remember to place the support symbol.

If the symbol is *not* of the type **Support**, it will be conceived as a part of the component.

When placing the support symbol, you enter the dialog box **Component data**, from which you can specify name and other component data.

Please note that you can also specify additional components in the field **ADDITIONALS** — as described in “Accessories in the database” on page 603. When you specify (part)components here, these will *not* be displayed when you select **Show available**.

— Please also see “Creating Symbols” on page 696, specially “Symbol types” on page 706.

Connection names/Pindata

CONNECTION NAMES / PINDATA IN THE DATABASE	
— WHAT IS CONNECTION NAMES / PINDATA	P. 598
— CONNECTION NAMES FOR SINGLE LINE SYMBOLS	P. 598
— SYMBOLS WITH VARIABLE CONNECTION NAMES	P. 599
— CONNECTION NAMES IN FILES	P. 599
— ALREADY DESIGNED SYMBOLS AND CONNECTION NAMES	P. 599
— SPECIFYING I/O STATUS FOR CONNECTION POINTS VIA THE PINDATA FIELD	P. 600

What is connection names /pindata

When you create electrical symbols in **Automation**, you assign connection names to each of the connection points of the symbol. — See “Creating Symbols” on page 696.

In the database you have to specify which connection names the connection points have, when you place the symbol in a diagram.

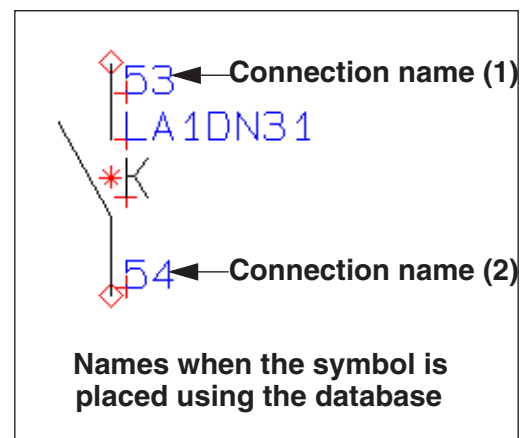
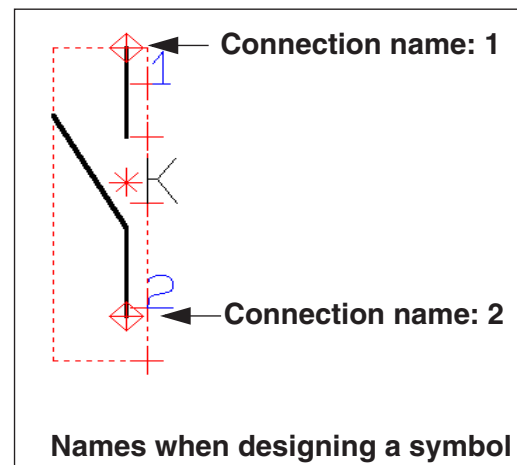
You do that in the datafield **PINDATA**. Here you type in the connection names separated by a comma.

The connection point having the smallest number in the symbol definition, is assigned the first number on the list — and so on.

If the component contains several electrical symbols, these are separated by a semicolon.

In the example above, where the first three symbols are of the same kind (**07-02-01#3**), you have to type in the connection name for each of the three symbols, separated by a semicolon. — See the **PINDATA** datafield on page 593.

When you place the symbol on a diagram, it will be assigned the connection names, you have specified in the datafield **PINDATA**.



Connection names for single line symbols

Connection names for symbols for single line drawings are specified with a ' sign around the list of connection names for a connection point.

If for instance you write *1,2,3,4,5,6;1,3,5;2,4,6* this means that the first symbol has six connection points, while the other has two connection points with three names each.

Symbols with variable connection names

If you write *?* or *** in a connection name, an extra text field appears in the dialog box **Component data**, when you place the symbol. Here you can type in a variable part of the connection name, which is the same for all of the connection points for the symbol. This is for instance handy for make, break and switch symbols, where the position is not known prior to placing the symbol in the project.

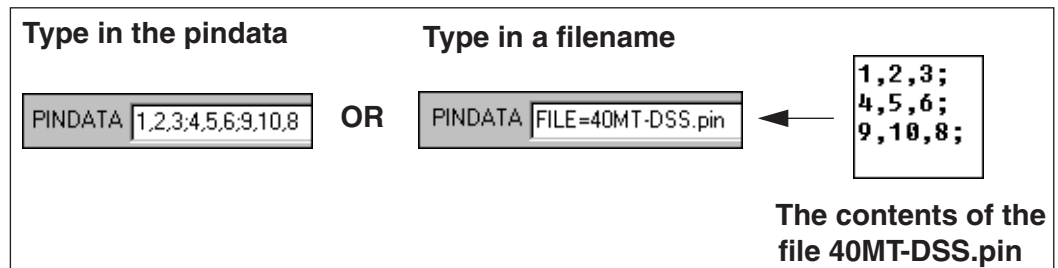
— See “Connection names with variable part in the Symbol definition” on page 162, “Connection data with a variable part in the name” on page 701 and “Wildcards/jokers” on page 550.

Connection names in files

For symbols having many connections, such as PLCs, you can choose to place the pindata information in a separate file.

This file must be a text file, where you only have information on one component on each line.

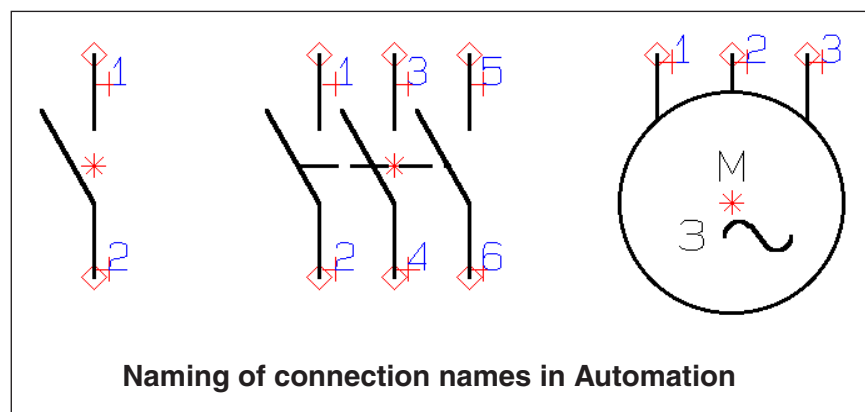
If you write *FILE=* followed by the name of the file, the program will look for the file in the library you have specified under **Settings => Directories => Database**. Alternatively, you can type in the path leading to the file, along with the file name.



— See “Creating PLC components in the database” on page 606.

Already designed symbols and connection names

In the symbol libraries of **Automation**, all symbols are designed with invisible connection names (pindata). To display in which order the connection names are specified, they are set to be visible on the figure below.



Naming of connection names in Automation

In the already designed symbols, the connection names start with the number one and is incremented following the reading direction.

When the symbols are selected using the **Automation** program, the connection names specified in the database, automatically become visible.

Specifying IO status for connection points via the PINDATA field

IO status for connection points can be specified via the **PINDATA** field in the database. The status is inserted in square brackets — as for IO addresses on PLCs (see “Addressing PLC symbols via the database” on page 601).

Syntax for specifying IO status

IO status is specified by **/I:** followed by two characters. The following options exists:

- /I:ET** External terminal connection.
- /I:IT** Internal terminal connection.
- /I:OP** Output PLC connection.
- /I:IP** Input PLC connection.
- /I:EI** External terminal connection with PLC input.
- /I:EO** External terminal connection with PLC output.
- /I:II** Internal terminal connection with PLC input.
- /I:IO** Internal terminal connection with PLC output.

EXAMPLE: TWO PLC SYMBOLS WITH ADDRESS AND IO STATUS SPECIFICATION:

PCSTYPE = PLC01;PLC11
PINDATA = 1[100.00/I:OP]; 1[100.01/I:IP]

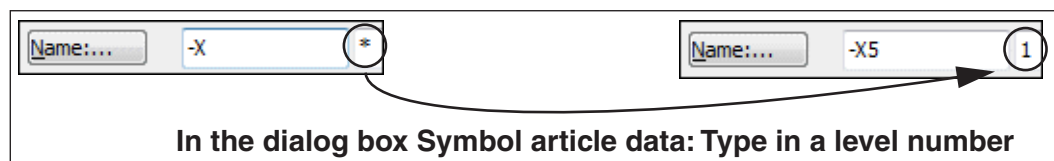
EXAMPLE: AN OPEN SYMBOL WITH IO STATUS SPECIFICATION:

PCSTYPE = 07-02-01
PINDATA = 13[I:ET],14[I:IT]

Naming levels for multi-level terminals via the database

In the PINDATA field in the database you can for multi-level terminals specify a digit or a letter for the layer. If for instance you write ***A;*B;*C**, you can, when subsequently placing the first terminal symbol, specify a value for the asterisk, which is automatically transferred to the other symbols for the terminal:

PCSTYPE	03-02-02#3
PINDATA	*A;*B;*C



This way you can e.g. name the first multi-level terminal **1A**, **1B** and **1C** respectively, while the other multi-level terminal is named **2A**, **2B** and **2C** respectively.

Activating references from the database

References can be activated and deactivated via the database, so that these can be controlled independent from the symbol definition. You can control the references by inserting **/W:1** (active reference) or **/W:0** (not active reference) in the parameter section for the connection points.

Example: **1[/W:1],2[10.00/W:1/I:OP],3[/W:0]**

This means:

- Connection **1**: Gets an **Active reference**
- Connection **2**: Gets the function text **10.00** (see “Addressing PLC symbols via the PINDATA field” on page 601), I/O status set to **Output PLC** (see “Syntax for specifying IO status” on page 600) and an **Active reference**
- Connection **3**: Gets a **Not active reference**

Addressing PLC symbols via the database

OPTIONS FOR ADDRESSING PLC SYMBOLS IN THE DATABASE	
— ADDRESSING PLC SYMBOLS VIA THE PINDATA FIELD.....	P. 601
— ENUMERATED NAMES FOR SYMBOLS WITH ONE OR MORE CONNECTION POINTP.	601
— ENUMERATED PLC ADDRESSES IN THE PINDATA FIELD	P. 602
— COUNTING FUNCTIONS FOR PLC ADDRESSES IN THE PINDATA FIELD	P. 602
— SPECIFYING A NUMBERING SYSTEM.....	P. 602
— INCREMENTING BY MORE THAN ONE	P. 602
— FIXED NUMBER OF DIGITS ON COUNTING FIELD	P. 602

— Please also see “Connection names/Pindata” on page 598 and “Connection names in files” on page 599.

Addressing PLC symbols via the PINDATA field

PLC symbols can be addressed via the database. The address (the function text) is specified in square brackets [] after the connection name in the **PINDATA** field in the database.

Example: **1[1000.00],2[1000.01], ...**

Enumerated names for symbols with one or more connection point

In the **PINDATA** field in the database, you can specify enumerated names for symbols with one connection point.

Example: **A1;...;A8;B1,...,B8**

This specifies connection names for eight one-legged symbols with the connection names **A1 - A8**, and one symbol with eight connection points **B1 - B8**. (Symbols are separated by a semicolon, connection points by a comma).

Enumerated PLC addresses in the PINDATA field

The **PINDATA** field in the database can handle enumerated PLC addresses (function texts).

Example: **A1[1000.00];...;A8;B1[2000.00];...;B8**

The address in the example is automatically incremented from 1000.00 to 1000.07, and from 2000.00 to 2000.07 for the individual connection points.

Counting functions for PLC addresses in the PINDATA field

The **PINDATA** field can contain counting functions.

Example:

1[1000.00],1G2,1V2,2[1000.01],2G2,2V2,3[1000.02],3G2,3V2,4[1000.03],4G2,4V2

Can be written using the following notation:

{++(1)[++(1000.00)],++(1)G2,++(1)V2,}#4

Here the contents of the curled brackets is repeated four times ({...}#4).

Please note that the character just before the last curled bracket (}) — in the example above a comma (,) — specifies whether the text inside the curled brackets describes one or more symbols.

If this character is a comma (,), this means that the text contains a list of pindata for one single symbol.

If the character is a semicolon (;), each repetition contains data for a new symbol.

Specifying a numbering system

Which system of notation / numbering system to count according to, can be specified after ++(. This is specified with the characters **B**, **D**, **H** or **O** — **Binary**, **Decimal**, **Hex** or **Octal**, respectively. When no system of notation is specified, the decimal numbering system is used.

Example: **{++(1)[++(O1000.00)],++(1)G2,++(1)V2,}#4**

In the example above, the PLC is addressed according to the octal numbering system, while the others are addressed according to the decimal numbering system.

Incrementing by more than one

When you wish to increment by more than one, this can be specified after ++. E.g. **++2(1)**.

Example: **{A++(0)[++2(00.00)],COM;B++(0)[++2(00.01)],COM;}#8**
A0[00.00],COM;B0[00.01],COM; A1[00.02],COM;B1[00.03],COM; ...

When you wish to decrement, ++ is replaced by --.

Fixed number of digits on counting field

When you want a fixed number of digits on a counting field, you must write : after the number, followed by a number between 1 and 9.

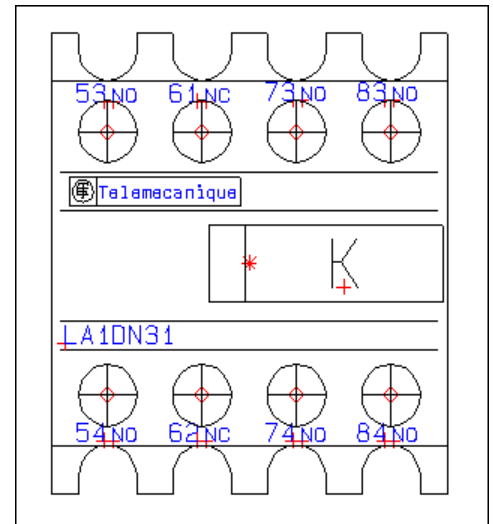
Example: **{A++(0)[00.--2(07:2)],COM;B++(0)[00.--2(01:2)],COM;}#8**

MECHANICAL SYMBOLS

You can also attach a mechanical symbol to a component. The name of the file in which the symbol is saved, should be placed in the database field **MECTYPE**.

Please note that you can also specify that other fields in the database contains mechanical symbols — see “Mechanical symbol” on page 612. Hereby you can have different mechanical symbols for the same component. The component can therefore be illustrated differently, on different mechanical pages in the project.

It is therefore possible on one page to show an operating cabinet with the front closed, so that you only see the components visible with the front closed. Another page could then show the bottom plate. You can therefore have database fields, each containing mechanical symbols for different purposes — for instance PNEU-TYPE, FRONTTYPE and MECTYPE.



Creating mechanical symbols directly from the MECTYPE field

When you want to program to automatically create a mechanical symbol of 30x50 mm, write `#x30mmy50mm` in the field **MECTYPE**. If you write `#r40mm`, the program creates a circle with a radius of 40mm as the mechanical symbol. These mechanical symbols has no connection points.

Specifying symbol state for mechanical symbols

For mechanical symbols created with states — see “Creating symbols with states” on page 712 — you can specify which state the symbol shall be in, when it is placed in the project. You specify this in the **MECTYPE** field using the **§** character succeeded by the state number.

If the symbol **XXXXX** is to be placed in state **2**, you therefore write **XXXXX§2** in the **MECTYPE** field.

— Read more about mechanical symbols in “Placing mechanical symbols via the database” on page 628, “From Mechanical to Electrical drawing” on page 639 and “Loading components from a ground plan page” on page 677.

ACCESSORIES IN THE DATABASE

To connect accessories to a component in the database, you first have to create a record for each accessory component, and place these records in the relevant

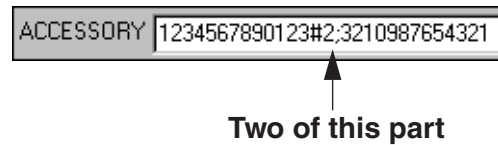
folder(s) in the database menu. These records must all contain an article number they can be identified by in the database — for instance in the **EANNUMBER** field.

SPECIFYING ACCESSORIES IN THE DATABASE	
— SPECIFYING THAT A COMPONENT HAS ACCESSORIES	P. 604
— SPECIFYING THE CONTENTS OF THE ACCESSORY FIELD IN A FILE.....	P. 604
— UNIT PARTS LIST CAN BE SPECIFIED IN THE FIELD ACCESSORY	P. 605

Specifying that a component has accessories

To specify, that a component has accessory parts, you type in the article number in the **ACCESSORY** field for the component in the database. In case a component has more than one accessory part, their article numbers must be separated by a semicolon (;) in the **ACCESSORY** field.

If you need more than one of a specific accessory part, you type # followed by the number needed, after the EANNUMBER of the accessory part in the **ACCESSORY** field.



If you for instance need two of the accessory part **1234567890123**, you type in **1234567890123#2** in the **ACCESSORY** field.

In **Automation** you can specify whether to include the accessories in your project lists. — Please refer to “Specifying from where to collect components” on page 544 for further details about this.

— See “Support symbols in the PCSTYPE field” on page 597 also.

Accessories for cables

For cables the amount of accessories can depend on two different parameters:

- 1) The cable *length*: Type in for instance **CLIPS1234#5** to specify that 5 cable clips of the type CLIPS1234 are needed for each meter of cable. (The length of the individual cables are specified in the **Quantity** field in the **Cable data** dialog box).
- 2) The *number of cables*: Type in for instance **CONNECTOR1234##2** to specify that 2 connectors of the type CONNECTOR1234 is to be included for each individual cable.

Thus you use a single # character when the accessory depends on the *length*, and two # characters when the accessory depends on the *number of cables*.

Specifying the contents of the ACCESSORY field in a file

As for connection names — see “Connection names/Pindata” on page 598 — you can specify the contents of the field **ACCESSORY** in a file. This is done by writing **FILE=** followed by the name of the file.

In this file one article number can be written on each line — such as:

3240900056666

5908326935039

1236374000001

There is no special requirements for the file name, but if you do not specify a path, the program searches for the file where the database catalog is pointing.

— Read more in “Connection names in files” on page 599.

Unit parts list can be specified in the field **ACCESSORY**

If you write *UNIT=* in the **ACCESSORY** field, this specifies that the article numbers in the **ACCESSORY** field is a unit parts list. This field therefore does not contain accessories to the components, but is a specification of what the component consists of.

If for instance you write *UNIT=1123455666;504509345;123871312* this means that the component consists of the three listed article numbers.

If you in the list setting for a parts list or a components list select **Articles from Unit Lists and Drawings** — see “Specifying from where to collect components” on page 544 — these three article numbers will be included in the list in stead of the specified article number for the component.

Typing in unit list information in the database, functions just as inserting a unit parts list in the project for the component. — Read more about this in “Inserting Unit Drawings” on page 568.

If the list of article numbers is long, you can place the article numbers in a file, and write *UNIT=FILE=* followed by the name of the file.

The article numbers must be placed on separate lines — as described in “Specifying the contents of the **ACCESSORY** field in a file” on page 604.

SPECIFYING REF. LETTER FOR THE SYMBOL IN THE DATABASE

When you type in a letter in the field **REFID**, symbols for the component will be assigned a name starting with this letter, when placing the symbols in a project.

If for instance you write *K*, the letter **K** will be suggested as default reference letter when you place the symbol in a diagram.

However, to make this work you first have to choose **Settings => Database => Database => Database setup => Component data**, and there choose **REFID** in the field **Reference letter**. If the field has no contents in the database for the chosen symbol, the reference letter (the symbol name) from the symbol definition is applied. — See “Creating Symbols” on page 696.



INSERT "-" IN FRONT OF SYMBOL NAMES AUTOMATICALLY

YOU CAN MAKE THE PROGRAM INSERT A MINUS IN FRONT OF ALL SYMBOLS BEING PLACED IN THE PROJECT, BY CHOOSING **SETTINGS => POINTER/SCREEN** AND SELECT **INSERT - IN FRONT OF SYMBOL NAMES**. — SEE “POINTER/SCREEN” ON PAGE 836.

SPECIFYING SYMBOL SUB NAMES FROM THE DATABASE

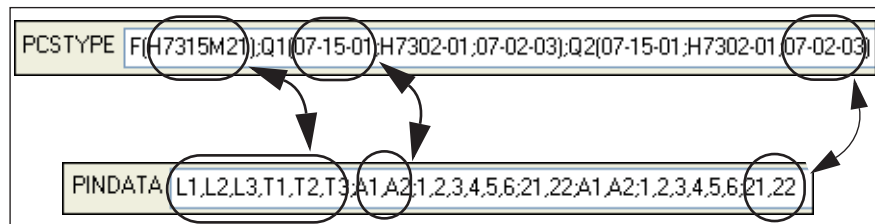
Automation can handle sub names on symbols intelligently, as described in “Sub names on symbols” on page 165. To specify sub names on symbols via the database, write the following in the field **PCSTYPE**:

```
PCSTYPE F(H7315M21);Q1(07-15-01;H7302-01;07-02-03);Q2(07-15-01;H7302-01;07-02-03)
```

Hereby the component will contain the sub names **F**, **Q1** and **Q2**.

Specifying pindata in the database for components with sub names

When you specify connection names / pindata in the database for components with sub names, you specify these in the order in which the symbols appears in the **PCSTYPE** field:



As you can see, the fact that sub names are being applied for the component, makes no difference for how to fill out the **PINDATA** field.

— See an example on this in the pickmenu:

The screenshot shows a software interface with a toolbar and a pickmenu. The toolbar includes icons for Automation, Lines and Texts, Pneumatic, Installation, Building, and PLC. The pickmenu shows a schematic diagram of a PLC component with sub names like Q.F, K.Q1, K.Q2, etc. Annotations include: "Click on this symbol (with sub names)" pointing to a symbol in the toolbar, "... and you get this pickmenu" pointing to the schematic, and "Sub names are shown here" pointing to the sub names in the schematic.

CREATING PLC COMPONENTS IN THE DATABASE

To create a PLC component in the database, do this:

- 1) Create the necessary PLC I/O symbols for the PLC, if you wish to use other PLC symbols than the ones already created in **Automation**. — See “Creating PLC I/O symbols” on page 725.
- 2) Create PLC reference symbol(s) for the PLC, if you wish to use other PLC reference symbols than the ones already created in **Automation**. — See “Creating PLC reference symbols” on page 728.
- 3) Press [d] to enter the **Database menu**, where you click on the **Pencil**, to edit records.
- 4) In the dialog box **Edit Record**: You can now either click the **Insert** button and create a new record with the PLC, or choose an existing PLC component in the database and update it. — See “Editing records in the database” on page 582.

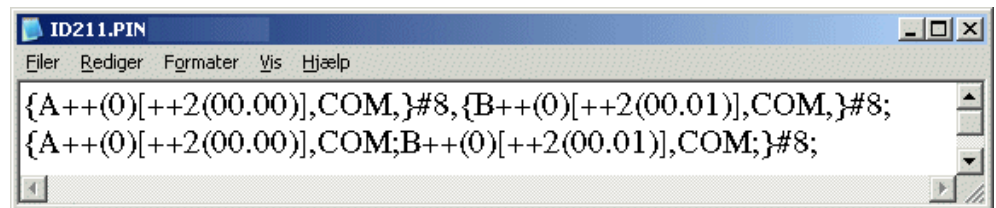


- 5) In the field **PCSTYPE** you now specify the PLC reference symbol(s) for the PLC, followed by its PLC I/O symbols. — See “Electrical symbols” on page 593, and e.g. “Alternative symbols for PLCs” on page 595.

PCSTYPE	PLCREF16I-1;PLC1IN-P#16;
PINDATA	FILE=ID211.PIN

- 6) In the **PINDATA** field you now specify the connection names for the PLC reference symbol(s) followed by the connection names for the PLC I/O symbols — see “Connection names/Pindata” on page 598.

This information most likely becomes quite comprehensive, so you will have to place it in a file, as described in “Connection names in files” on page 599:



It is recommended to address the PLCs in the database as displayed in the example above. — See “Addressing PLC symbols via the database” on page 601.



SIMPLE NOTATION FOR ADDRESSES IN PLCs

YOU DO NOT NEED TO APPLY THE COMPACT NOTATION DISPLAYED IN THE FIGURE ABOVE. YOU CAN JUST SPECIFY EACH INDIVIDUAL ADDRESS IN SQUARE BRACKETS AFTER THE CONNECTION POINTS. — SEE “ADDRESSING PLC SYMBOLS VIA THE PINDATA FIELD” ON PAGE 601.



- 7) Add the remaining relevant data for the component — such as the EAN number — and click on **Post** and then on **Close**.
- 8) The PLC component is now created/updated, and you return to the **Database menu**.

— See also “Graphical creation of New Components in the Database” on page 589, “Information about the component in the database” on page 591 and “Electrical and Mechanical symbols for components” on page 593.

DATABASE SETTINGS

This chapter deals with how the information from the attached database is represented in **Automation**.

THIS CHAPTER DESCRIBES	
— MAPPING OF THE DATAFIELDS IN THE DATABASE.....	P. 608
— TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES	P. 610
— CHOOSING PRODUCT/FABRICATE IN THE DATABASE	P. 613
— APPROVED AND OBSOLETE COMPONENTS	P. 614
— CHOOSING A DATABASE MENU.....	P. 617
— CHOOSING MENUS WHEN USING OTHER DATABASE PROGRAMS	P. 619
— URL LINKS.....	P. 620
— THE PANEL BUILDER.....	P. 621
— LINKS TO E.G. PDF FILES OR INTERNET ADDRESSES IN THE DATABASE	P. 622

The work with constructing the database itself, as well as the import of databases from component vendors, takes place in an independent database program.

This can either be the **PC|SCHEMATIC Database** program, or any other open database program you use in conjunction with **Automation** — see “Choosing a database” on page 623.

If you are using the **PC|SCHEMATIC Database** program, you can start it by choosing **Tools => Database** in **Automation**. — Please refer to the **PC|SCHEMATIC Database** manual for further details.

If you want to see how to fetch article data from components in the database, please read “Using the database” on page 571.

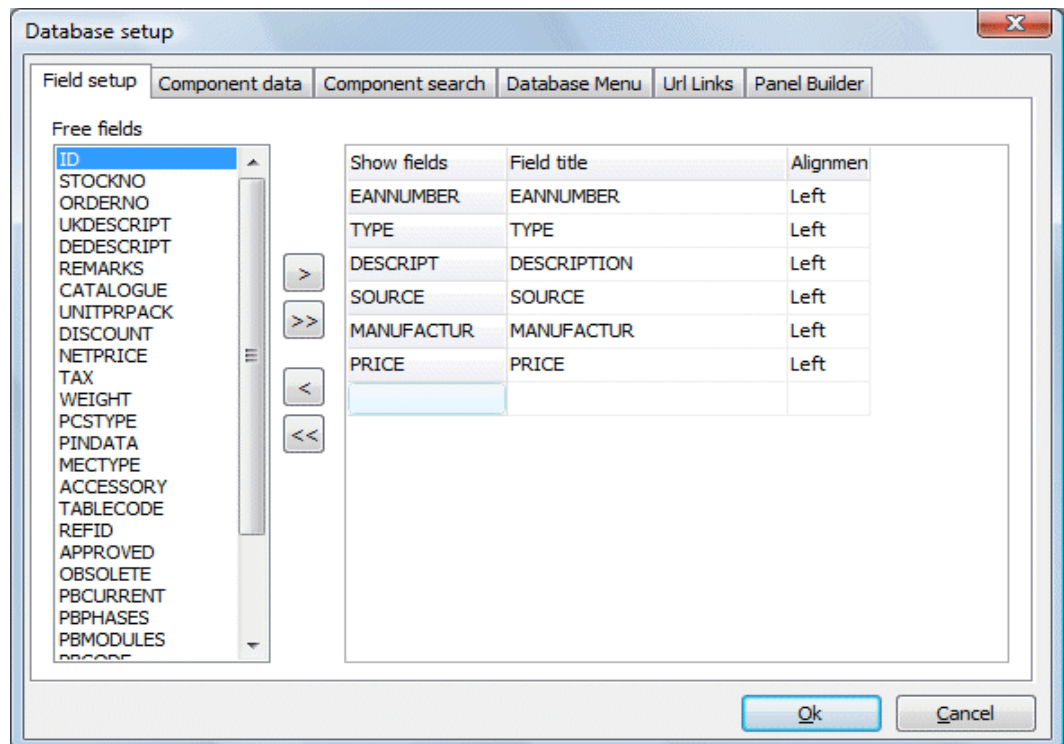
MAPPING OF THE DATAFIELDS IN THE DATABASE

When you are working with a database in **Automation**, you can determine which component information from the database will be displayed in **Automation**, and how the program shall interpret the information from the database.

To determine this you choose **Settings => Database**, and then click on **Database setup**.

You then enter the dialog box **Database setup**. Here you specify the settings you need, and click **OK** to accept the changes, or **Cancel** not to save the changes.

The **Field setup** tab looks like this:



Which database fields are displayed in Automation

On the tab **Field setup**, the datafields in the **Show fields** column are the datafields from the database, which will be displayed in **Automation**.

Here they will be displayed under the name specified in the **Field title** column. Any name in this column can be changed if you click on the name, and type in a new field name.

The order in which the field names are listed in this dialog box, is also the order in which the fields are displayed in the database menu. If for instance you want **Price** to appear first on the list, just click on **Price**, and *drag* the field to above the first field in the list. This will make these fields exchange places. The fields in between the two fields will not change places.

In the **Alignment** column you determine whether the text is to be left-, right- or centre-adjusted. If you want to change the alignment, just click on the alignment field, and choose the desired alignment in the menu that pops up.

Adding or removing fields

Under **Free fields** to the left in the dialog box, you can see the fields in the database, which are not being displayed in **Automation** at the present.



If for instance you want to display the field **TAX**, you click on the field and then click on the **Transfer** button.



To remove a field from the **Show fields** column, just click on the field, and then click on the **Revert** button.



To have all of the database fields displayed in **Automation**, you click on the **Transfer all** button.



Not to display any of the database fields, you click on the **Revert all** button. Please note that the datafields are not deleted, but transferred to the column of **Free fields**.

TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES

Click on the tab **Component data** in the upper part of the dialog box.

When you transfer article data from the database to symbols or lines, it is important that you state which database datafields will be the **Article**, **Type** and **Function** fields in the article data menus.

Field	Database Field	Field	Database Field
Article	EANNUMBER	Diagram symbol	PCSTYPE
Alt. Article		Single-line symbol	SINGLELINESYMBOL
Type	TYPE	Reference letter	REFID
Function		Pin names	PINDATA
Description	UKDESCRIPT	Mechanical sym.	MECTYPE
Price 1	PRICE	Other symbols	
Discount 1		Additional	ACCESSORY
Price 2	NETPRICE		
Discount 2			
Units/Pack	UNITPRPACK		

Use DB-Cache

Ok Cancel

This is stated in the fields **Article**, **Type** and **Function** in the dialog box.

The database field mentioned in the **Article** field, will also become the permanent search field in the bottom of the **Database** menu on the tab **Article/Type**.

In the field **Alt. article** you specify which field in the database will be used as article number, if the datafield you have chosen as **Article** has no contents for a component. This field is for instance useful when you start up using EAN numbers in stead of item numbers. If you set **Article** to **EANNUMBER** and **Alt. Article** to **ITEMNUMBER**, the symbols in the project will get the EAN number where a such exists. If no EAN number exists, they get an item number instead.

Thus you make sure that a symbol always has got an article number, without having to type in item numbers as EAN numbers in the database.



USE INDEXED FIELDS FOR ARTICLE NUMBER AND ALT. ARTICLE NUMBER

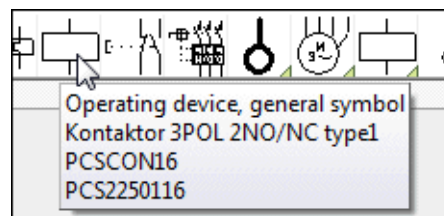
IN ORDER TO GET REASONABLE ACCESS TIMES FOR COMPONENTS IN THE DATABASE, IT IS RECOMMENDED ONLY TO USE INDEXED FIELDS FOR **ARTICLE** AND **ALT. ARTICLE**.

Replace article number with alternative article number

If you want to replace the article number with the alternative article number for all symbols in a project, choose **Functions => Special Functions => Exchange Article with Alt. Article on all Symbols**.

Description

When you select a field in the database to use as **Description**, the contents of this datafield will be displayed as pop-up information, when you let the mouse rest over the symbol in diagrams and in pickmenus. The description is also displayed in the **Information** dialog box for the symbol.



This description can also be included in lists, when you insert the datafield **Parts/Components List => Description** — Read more about creating lists in “Creating Lists” on page 752.

Parts list prices

When you insert datafields of the type **Parts list**, you can then choose to insert the datafields **Price 1** and **Price 2**.

In the lower-right corner of the dialog box, you specify which of the database fields that shall be used as **Price 1** and **Price 2**.

If a discount factor have to be included for the two prices, this is specified in **Discount 1** and **Discount 2** respectively.

Units/Pack

The quantity of the component. — See the options in the description of the **QUANTITY** datafield in “Parts/components list datafields” on page 859.

Diagram symbol

In the **Diagram symbol** field you specify which datafield in the database that contains the file name for the electrical symbol(s) for the component. This for instance makes it possible for the program to show you the electrical symbols for the component, when you choose the component from the database, and wish to place the symbols on a **Diagram** page.

— See “Electrical symbols” on page 593.

To create electrical symbols directly from the database, you type in the dimensions of the symbols in the **PCSTYPE** field for the component. — Read more about the syntax in “Syntax for automatic creation of symbols” on page 200.

Single line symbol

In the field **Single line symbol** you specify which field in the database contains the single line symbol for the component. This symbol is used for automatic generation of single line diagrams, which you can read about in “Automatic generation of single line diagrams” on page 409.

Reference letter

Here you can specify which reference letter the symbol is assigned when it is placed in a project. If this field contains a letter, this letter will be applied. If the field has no contents, the symbol will be assigned the reference letter from the symbol definition. — See “Specifying Ref. Letter for the symbol in the Database” on page 605.

Pin names

Here you specify which field in the database to use to assign names to the connection points of the electrical symbols. — See “Connection names/Pindata” on page 598.

Mechanical symbol

In the **Mechanical sym.** field you specify which field(s) in the database contains the file name of the mechanical symbol(s). These symbols displays the physical appearance of the component. This for instance makes it possible for the program to show you the mechanical symbol(s) for the component, when you choose the component from the database, and wish to place the symbols on a **Ground plan/Mechanical** page.

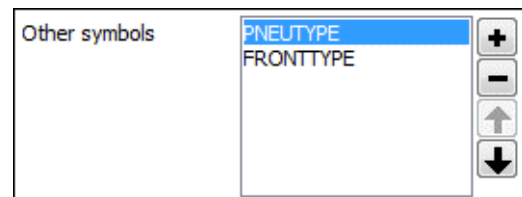
— Read more about this in “Mechanical symbols” on page 603 and “Placing mechanical symbols via the database” on page 628.

Creating mechanical symbols directly from the database

In order to create mechanical symbols directly from the database, type in the symbol dimensions in the **MECHTYPE** field for the component. — See the syntax in “Syntax for automatic creation of symbols” on page 200.

Other symbols

In the **Other symbols** field you can specify whether any other fields in the database contains other types of symbols, that you need in your projects — a part from the electrical and mechanical symbols.



When you have added these datafields in the **Other symbols** field, you subsequently have the option of placing these symbols on project pages.

Adding fields for Mechanical symbols

To add a field to the list of mechanical symbol fields, click the **Plus** button. You then enter the dialog box **Other symbols**, where you select the fields that contains the desired symbols in the attached database, and click **OK**.

Deleting fields for Mechanical symbols

To delete a field from the list of fields with other symbols, select the field in the list (see the figure), and click the **Minus** button.

Moving a field in the list of Mechanical symbols

To move a field with mechanical symbols, click on the field in the list, and then click the **Up-arrow** or the **Down-arrow** button.

Specifying that you want to apply "other symbols" on a page

When for instance you have added the datafield PNEUTYPE under **Other symbols**, you can subsequently right-click on a project page and choose **Page setup**. In the dialog box **Page setup** you can then choose the datafield PNEUTYPE under **Other symbol**.

When you work on this project page and select a component from the database, the program then shows you the symbols specified in the PNEUTYPE datafield. Correspondingly the **Show available window** will only show symbols of the type PNEUTYPE, when you work on this project page.

— You can read more about the **Show available window** in “The Show available window” on page 234.

Additional

A component can have certain additional parts attached to it, which also must be used each time the component is used, but do not have to appear anywhere but in the parts lists for the project. This could for instance be a base for a relay.

In the **Additional** field you state which field in the database contains this type of information. — See “Accessories in the database” on page 603.

Use DB-Cache

If the applied database is placed on a network, and you use the dBase database format, you can select **Use DB-Cache**, which makes the database work faster.

If you are not working on a network, this option makes no difference.

Save settings

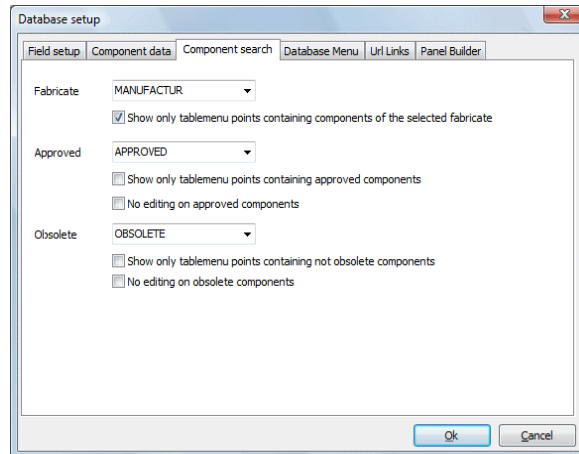
Click **OK** to save the changes you have carried out. If you click **Cancel**, your changes will not be saved.

PRODUCT CHOICE OR FABRICATE CHOICE IN THE DATABASE MENU

In the **Database menu** you can choose whether it shall be possible to show components based on their fabricate — or the like.

To choose which of the database fields to let **Automation** sort after, do the following:

1) Choose **Settings => Database => Database setup:**



2) In the dialog box **Database setup:** Click on the tab **Component search**, click on the **Down arrow** in the **Fabricate** field, and choose which database field to divide by — for instance **Manufactur**. (Choose **(None)** in order *not* to use this Fabricate menu).

3) Click **OK** to leave the dialog box **Database setup**, and click **OK** to leave the dialog box **Settings**.

— See how to apply this option in “Selecting a Fabricate / Vendor” on page 576.

Show only table menu points containing components of the selected fabricate

When you wish the table menu in the right-hand corner of the **Database menu** only to display those table menu points, which contains components of the selected fabricate, do the following:

- 1) Choose **Settings => Database => Database setup => Component search**.
- 2) Select **Show only tablemenu points containing the selected fabricates components**.



APPROVED AND OBSOLETE COMPONENTS IN THE DATABASE

In **Automation** it is possible to discern between approved and non-approved components, as well as between obsolete and non-obsolete components.

APPROVED AND OBSOLETE FIELDS IN THE COMPONENT DATABASES
IN Automation TWO SPECIAL FIELDS EXISTS IN THE COMPONENT VENDOR DATABASES FOR THE PROGRAM: AN APPROVED FIELD AND AN OBSOLETE FIELD.

APPROVED AND OBSOLETE FIELDS IN THE COMPONENT DATABASES
<p>THE DATABASE FIELD APPROVED IS USED TO GET A BETTER SURVEY OF THE DATABASE, BY MAKING IT POSSIBLE ONLY TO SHOW THOSE OF THE COMPONENTS THAT YOU TYPICALLY USE IN THE PROJECTS. — SEE “DISPLAY ONLY APPROVED COMPONENTS AND HIDE OBSOLETE COMPONENTS” ON PAGE 616.</p>
<p>THE DATABASE FIELD OBSOLETE IS USED TO MARK THE OBSOLETE COMPONENTS IN THE DATABASE. THIS MAKES IT POSSIBLE TO MAINTAIN OBSOLETE COMPONENTS IN THE DATABASE, WITHOUT THE RISK OF USING THESE COMPONENTS WHEN CREATING NEW PROJECTS.</p>

Specify which database fields to use for Approved and/or Obsolete

To specify which of the fields in the selected database that **Automation** shall conceive as **Approved** and **Obsolete**, do the following:

- 1) Choose **Settings => Database => Database setup**.
- 2) In the dialog box **Database setup**: Click on the tab **Component search**, and click on the **Down arrow** in the **Approved** and the **Obsolete** field, to choose which database field contains this information.



Approved

Show only tablemenu points containing approved components

No editing on approved components

Obsolete

Show only tablemenu points containing not obsolete components

No editing on obsolete components

Click here and choose the field in the applied database

- 3) Click **OK** to leave the dialog box **Database setup**, and click **OK** to leave the dialog box **Settings**.

Using non-boolean fields for Approved/Obsolete

In **PC|SCHEMATIC Database** it is possible to create boolean datafields. Only when one of these boolean datafields is used as **Approved/Obsolete**, you will get the option to set a check mark for **Approved** and/or **Obsolete** in the **Database menu** in **Automation**.

If you choose a non-boolean field, you get the option to specify which values in the chosen datafield that identifies approved components.

Approved Value(s)

Show only tablemenu points Multiple Approved values are separated with ";" ex. value1;value2;value3

These values must be separated by a semicolon (;).

In the component vendor databases for **Automation**, the fields **Approved** and **Obsolete** are created as boolean fields.

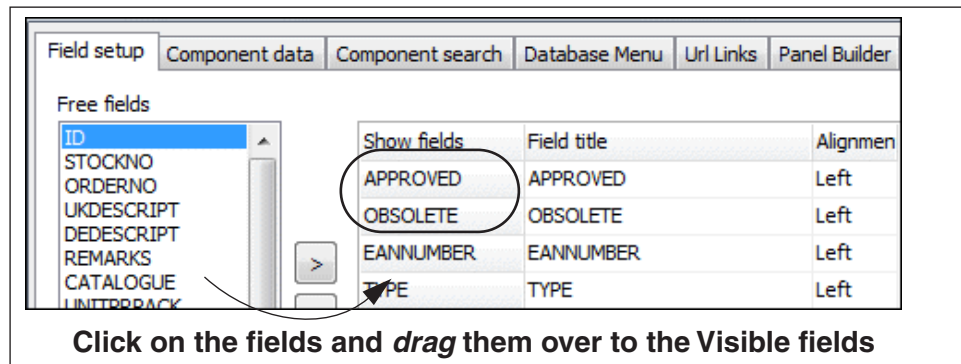
Displaying the fields **Approved** and **Obsolete** in the database menu



THE FIELDS APPROVED AND OBSOLETE DOES NOT NECESSARILY HAVE TO BE DISPLAYED IN THE DATABASE MENU.

To display the fields **Approved** and **Obsolete** in the **Database menu**, do this:

- 1) Choose **Settings => Database => Database setup**.
- 2) In the dialog box **Database setup**: Click on the tab **Field setup**.
- 3) On the tab **Field setup**: Click on the fields **Obsolete** and **Approved** in the column **Free fields** and *drag* them over to the column **Show fields**:



Display only **Approved** components and hide **Obsolete** components

When you have selected the approved components, you can set the **Database menu** to display approved components only. This makes it faster to locate the components you typically use. When you need to fetch other components, you can set the **Database menu** to display all components again.

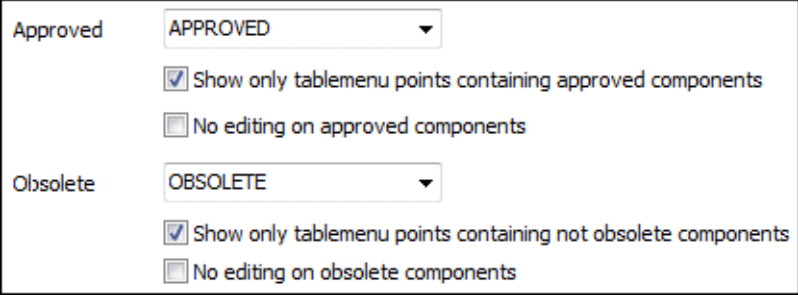
When you wish to see approved components only, choose **Show approved components only** in the **Database menu**.

Correspondingly, you can hide obsolete components by selecting **Hide obsolete components** in the **Database menu**.

Only display table menupoints containing Approved/Obsolete components

When you want the tablemenu in the upper left-hand corner of the **Database menu** only to contain menupoints, which actually contains approved components, do the following:

- 1) Choose **Settings => Database setup => Component search**.
- 2) Select **Show only tablemenu points containing approved components**.



Approved	APPROVED	<input checked="" type="checkbox"/> Show only tablemenu points containing approved components	<input type="checkbox"/> No editing on approved components
Obsolete	OBSOLETE	<input checked="" type="checkbox"/> Show only tablemenu points containing not obsolete components	<input type="checkbox"/> No editing on obsolete components

When you wish to do the same for obsolete components, select **Show only tablemenu points containing not obsolete components**.

Approving all article numbers in a project

To set all the applied article numbers in the active project to **Approved** in the database, choose **Functions => Special functions => Set all articles as approved in the database**.

No editing of approved/obsolete components

If you will not allow editing of approved and/or obsolete components, do this:

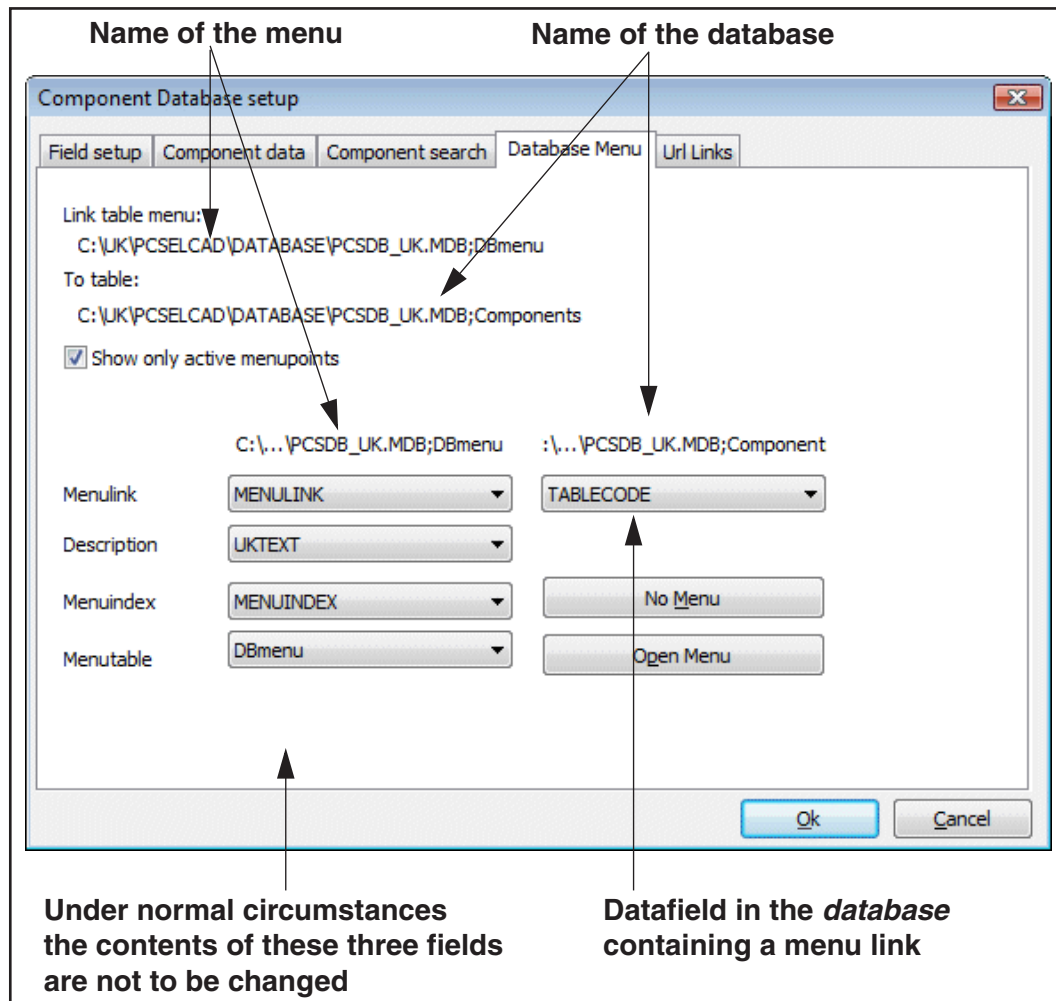
- 1) Choose **Settings => Database settings => Component search**.
- 2) Select **No editing of approved components** and/or **No editing of obsolete components**.

— See the figure in “Only display table menupoints containing Approved/Obsolete components” on page 617.

CHOOSING A DATABASE MENU

A menu structure can be attached to the database, such as the component structure menu (FAFGE), in which you can group the database components according to their use. This makes it easier to locate the desired components.

To do this, you click on the **Database menu** tab.



In the figure above, the menu **DBmenu** is attached to a database named **Components**.

If you do not wish to have a menu attached, you click the **No Menu** button in the lower-right corner.

If you want to attach an existing menu, you click **Open Menu**. You then enter the dialog box **Open**, where you choose the desired database menu.

When you choose a database menu, which have been correctly created in **PC|SCHEMATIC Database**, the fields **Table link**, **Description**, **Menu index** and **Menu table** will then be filled out correctly.

The connection between the menu and the database

The field **Table link** contains a code, which separates the different groups in the menu. When the menu and the database exchange information, it takes place using this field.

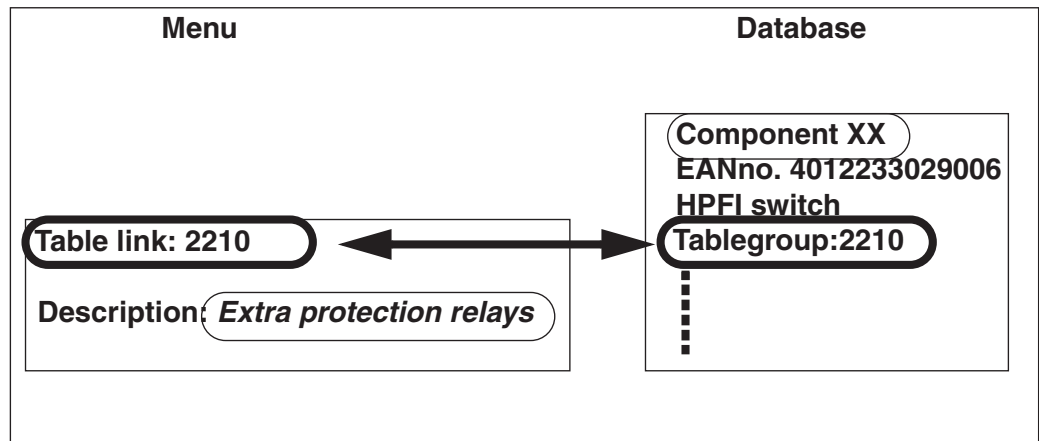
The field **Description** contains the names of the groups in the menus.

The field **Menu index** contains information on *how* the individual groups are placed within the menu.

To enable the database and the menu to communicate, there is also a datafield in the *database*, which contains a menu link. On the figure below this field is called **TABLEGROUP**.

If you have a component/record in the database, where it says *2210* in its **TABLEGROUP** field, this connects the component/record to the record, where it says *2210* in the **Menu link** field of the *menu*.

In the menu it says that the group **Extra protection relays** has got the menu link *2210*. Therefore the program knows, that the component mentioned belongs to the group **Extra protection relays**.

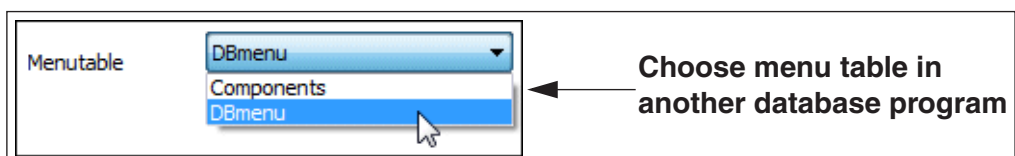


On the figure above, you can therefore see, that **Component XX** belongs to the group **Extra protection relays**.

For a more detailed explanation, please refer to the **PC|SCHEMATIC Database** manual.

CHOOSING MENUS WHEN USING OTHER DATABASE PROGRAMS

If you are using another database program than **PC|SCHEMATIC Database**, you will have an extra field on the **Database menu** tab.



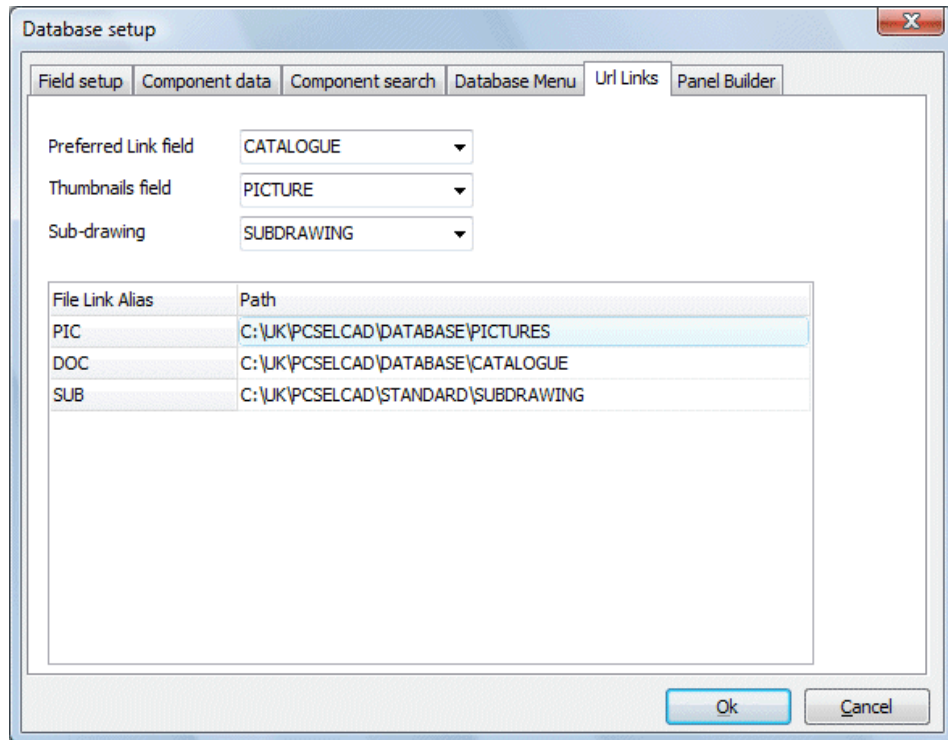
When you click on the down-arrow in the **Menu Table** field, it is displayed which tables are available for the chosen database.

Here you specify which table in the other database program, that contains the menu structure.

URL LINKS

On the tab **URL links** you can define a number of link aliases. The purpose of these are that you can specify an alias for a path to e.g. a component catalogue, which is placed as a pdf file somewhere on your PC or on the network you are attached to. This alias can then be used in the database, when you for instance wish to link to the catalogue for a specific component.

(The *idea* behind using an *alias* is explained in further detail in “The Alias concept” on page 841 — please note that the *syntax is different* for these two types of aliases).



To edit, create or delete a **File Link Alias**, right-click in the dialog box, and choose either **Edit**, **New** or **Remove**.

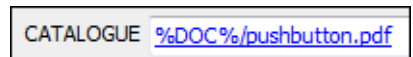
Default File link aliases

The following file link aliases have already been created during the installation of **Automation**:

FILE LINK ALIAS	USE
PIC	FOR SPECIFYING THE PATH TO THE LOCATION OF PICTURES OF COMPONENTS IN THE DATABASE.
DOC	FOR SPECIFYING THE PATH TO THE LOCATION OF DOCUMENTS, SUCH AS DATASHEETS IN THE DATABASE.
SUB	FOR SPECIFYING THE PATH TO THE LOCATION OF SUBDRAWINGS IN THE DATABASE.

Using a File Link Alias

When you are using a **File Link Alias** in the field



for a component in the database, you must place a % character before and after the name of the alias, as displayed in the figure above.

Example for subdrawings

To specify the path to a subdrawing **PCSMOTOR1.PRO** placed in the **Subdrawings** folder, you must write: **%SUB%\PCSMOTOR1.PRO**

You can see the path to the **Subdrawings** folder in the figure above.

Example for pictures

To specify the path to a picture **Con01.jpg** placed in the **Pictures** folder, you must write: **%PIC%\Con01.jpg**

You can see the path to the **Pictures** folder in the figure above.

File link aliases and other syntaxes

A **File Link Alias** can be applied in all links, unless you also use the syntax **FILE=**. (**FILE=** is for instance used for pindata / cable colors).

Please note: *Windows environment variables* can also be used in the database fields.

Create a Preferred Link field

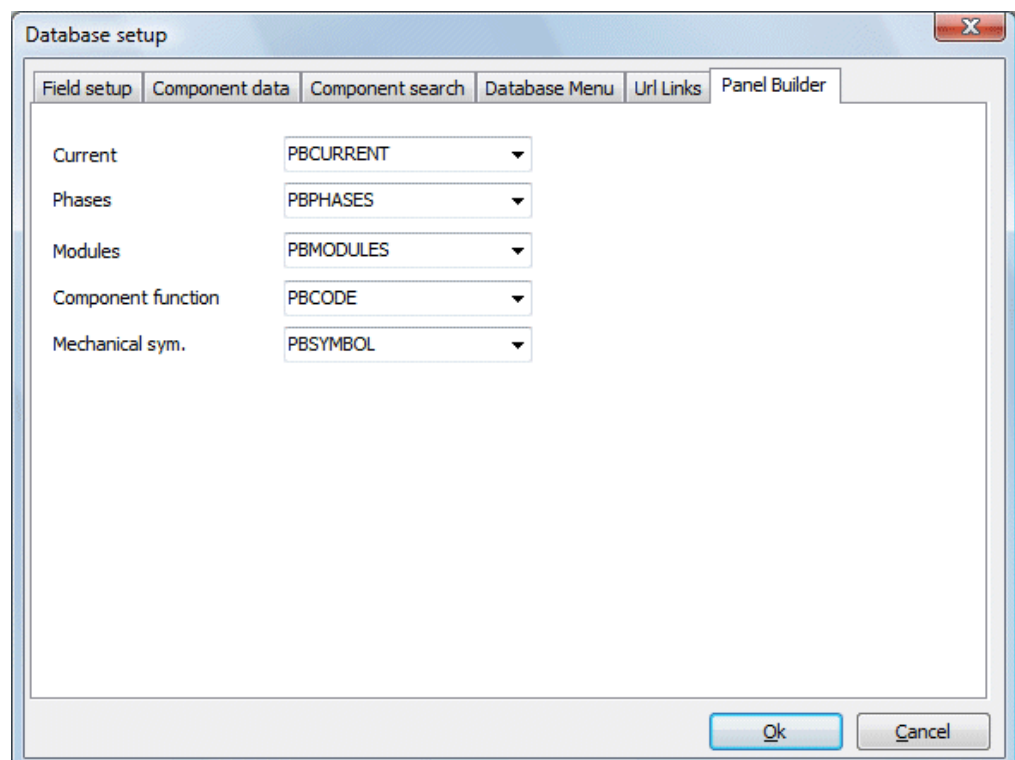
At the top of the tab **Link Alias**, you can specify a **Preferred Link field**.



When you have selected a record in the **Database menu**, where some of the fields of this record contains links, and you then click the **Open link** button, the link in the field that you have specified as **Preferred Link field** will be opened. — Read more about this in “Open link for selected record in Database menu” on page 580.

DATABASE SETTINGS FOR THE PANEL BUILDER

The settings on the tab **Panel builder** is described in “Database datafields for the Panel builder” on page 482:



LINKS IN DATABASE FIELDS

Database fields can contain links to for instance PDF files, internet addresses or e-mail addresses.

Automation will try to detect these links in the database by itself. If this does not succeed, you can write the full link by specifying the applied protocol, such as for instance:

- [http://www.pcschematic.com/...](http://www.pcschematic.com/)
- ftp://ftp.pcschematic.com/public/my_file
- <mailto:obl@company.com>

File names will be recognised when they follow the syntax `\\server\path\filename` or `x:\path\filename`.

When you are using a field in the database for this purpose, all of the contents of the field is conceived as the specification of the address. This meaning that you can not write comments or the like in the field.

CHOOSING A DATABASE

Automation supports Access databases directly. However, **Automation** is open for other database formats which are supported by either MDAC or ODBC.

Therefore you can also apply databases, which are created and maintained by other database programs. Thus you can apply MS-SQL, Oracle and most of the major database programs in conjunction with **Automation**.

CHOOSING A DATABASE

When you want to specify which database you want to attach to **Automation**, you choose **Settings => Database => Select database file**.

You then enter the dialog box **Select database**, where you click your way to the desired database, and click **Open**.

If you wish to use a database, which has not been created in Access, you must choose a so-called UDL file, by choosing **Microsoft Data Link (*.udl)**.



ON UDL FILES

UDL IS SHORT FOR UNIVERSAL DATA LINK — ALSO CALLED MICROSOFT DATA LINK. THE UDL FILE CONFIGURES THE SETTINGS FOR HOW **Automation** CREATES THE CONNECTION TO ANOTHER DATABASE SYSTEM. UDL FILES CAN BE CREATED FROM THE **PC|SCHEMATIC Database** PROGRAM.

CHOOSING THE TABLE IN THE DATABASE

When you have selected the database, you are asked to select the table in the database, which contains the component data. Select the table, click **Next**, and the database is attached to **Automation**.

— Read more about handling databases in the **PC|SCHEMATIC Database** manual.

PROGRAMMABLE BARCODE SCANNERS

If you have a programmable barcode scanner, you can use it to collect symbols from the database.

THIS CHAPTER DESCRIBES

THIS CHAPTER DESCRIBES HOW TO USE A BARCODE SCANNER IN THE PROGRAM.

When you have programmed the scanner properly, all you have to do is to scan in the barcode from the component vendor catalog. This gives you the symbol for the component directly into the cross hairs.

This requires, of course, that the component vendor have included the particular component in their database to **Automation**.

The reason why this is possible, is that there have been created short-cut keys to the search criteria in the **Database** menu.

Short-cut keys in the database menu

In the **Database** menu the following short-cuts have been created, for the search criteria tabs in the lower-right corner of the dialog box:

SHORT-CUT	FIELD	SHORT-CUT	FIELD
<CTRL+A>	ITEM NO	<CTRL+1>	1.SEARCHKEY
<CTRL+T>	TYPE	<CTRL+2>	2.SEARCHKEY

Programming the scanner

Exactly how the scanner should be programmed, depends on the applied scanner, as well as whether the scanner can send commands while scanning.

As an example, the scanner could be programmed like this: *"d"Ctrl-a<EANNR>Alt-o.*



Mechanical symbols

In this section you can see how to work with mechanical symbols in PCISCHMATIC Automation.

You can read about:

- Overview of options when working with mechanical symbols p. 627
- Place mechanical symbols, based on placed electrical symbols, via the database p. 628
- Place electrical symbols, based on placed mechanical symbols, via the database p. 639
- Measurement objects..... p. 641

MECHANICAL SYMBOLS - OVERVIEW

THIS CHAPTER DESCRIBES

HERE YOU GET A SURVEY OVER THE OPTIONS YOU HAVE WHEN WORKING WITH MECHANICAL SYMBOLS IN **AUTOMATION**.

— IF NECESSARY, YOU CAN READ MORE ABOUT THE RELATIONSHIP BETWEEN *COMPONENTS*, *ELECTRICAL SYMBOLS* AND *MECHANICAL SYMBOLS* IN "THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS" ON PAGE 229.

In the following chapters you can read about:

- How **Automation** can automatically fetch the *mechanical* symbols, that belongs to those components, for which electrical symbols have already been placed in the diagram pages. You can also read about the easiest ways to position mechanical symbols correctly. (See "Placing mechanical symbols via the database" on page 628).
- How you can start a project by creating a mechanical drawing, and then automatically fetch the *electrical* symbols, for the components that you placed the mechanical symbols for. (See "From Mechanical to Electrical drawing" on page 639).
- How to apply measurement objects on mechanical pages. (See "Measurement objects" on page 641).

Beyond this, you have the following options when working with mechanical symbols:

Creating mechanical symbols in the database

Read how to attach mechanical symbols to a component in the database in "Electrical and Mechanical symbols for components" on page 593 and "Mechanical symbols" on page 603.

See how to design your own symbols in "Creating Symbols" på side 696.

Creating mechanical symbols automatically

See how to create a mechanical symbol on the fly in "Automatic creation of symbols" on page 197, "Electrical or Mechanical symbol" on page 199 in particular.



PLACING MECHANICAL SYMBOLS VIA THE DATABASE

When you have got a database connected to **Automation**, it is possible to perform a so called mechanical load, which gives you the mechanical symbols you need to make a mechanical drawing, displaying how the components appear in real life.

THIS CHAPTER DESCRIBES	
— INSERTING A PAGE FOR A MECHANICAL DRAWING.....	P. 628
— PERFORMING THE MECHANICAL LOAD	P. 629
— PLACING MECHANICAL SYMBOLS ACC. TO REFERENCE DESIGNATIONS.....	P. 632
— PLACING MECHANICAL SYMBOLS AS BOXES.....	P. 632
— PLACE ONLY MECHANICAL SYMBOLS NOT PLACED ON OTHER PAGES.....	P. 632
— MESSAGES AND WARNINGS DURING MECHANICAL LOAD	P. 633
— SELECTING UNKNOWN MECHANICAL SYMBOLS	P. 633
— DISPLAYING CONNECTIONS FOR MECHANICAL SYMBOLS.....	P. 634
— MAGNETIC SNAP AND SNAP IN OBJECTS FOR MECHANICAL SYMBOLS	P. 634
— REFERENCES IN LISTS TO THE LOCATION OF A MECHANICAL SYMBOL.....	P. 636
— REFERENCES FROM MECHANICAL TO ELECTRICAL SYMBOLS.....	P. 637
— MEASUREMENT OBJECTS.....	P. 637
— DRAW ORDER.....	P. 638

— Read more about the relation between *components*, *electrical symbols* and *mechanical symbols* in “The relationship between Symbols and Components” on page 229.

— Read more about how the database works in “Using the database” on page 571.

INSERTING A PAGE FOR A MECHANICAL DRAWING

To generate a mechanical drawing, you first have to create a page on which to place the drawing:



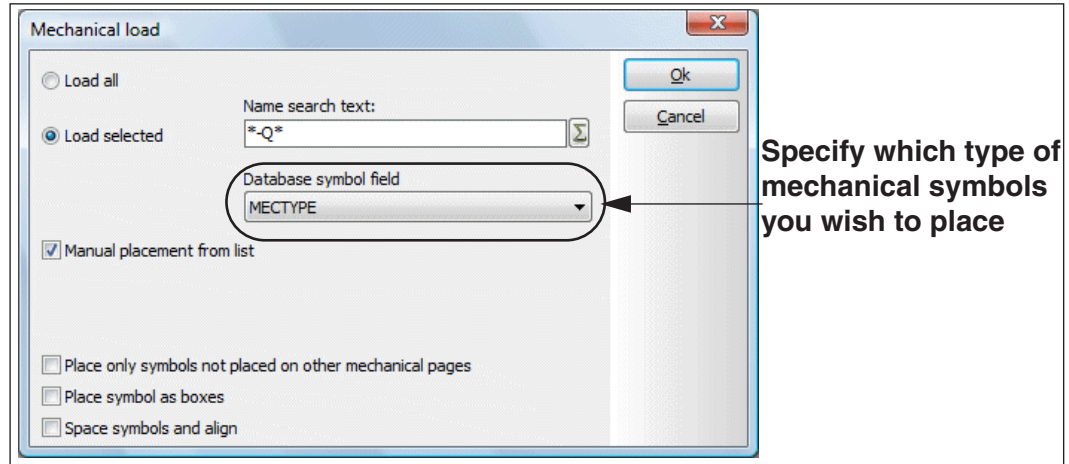
- 1) Enter the **Page Menu**, click on **Add new** and click **OK** to insert a **Normal** page.
- 2) In the dialog box **New**: Click on the template **DPSA3MECH** and click **OK**.
- 3) In the **Page menu**: Click **OK**.

PERFORMING THE MECHANICAL LOAD

After this, the mechanical symbols can be placed on the page. These will be loaded automatically from the database, in which the electrical symbols, through their EAN13 number/article number, are attached to the mechanical symbols.



Now click on the **Symbols** button, and choose **Functions => Mechanical load**.



Specify which symbols to place

In the appearing menu you click on **Load selected**, type in ***-Q*** and click on **OK**. You will now load all mechanical symbols, attached to the components in the diagram, containing the text **-Q**. — Read more about how to select the symbols in “Wildcards/jokers” on page 550.



SELECT INCLUDE IN MECHANICAL LOAD

MECHANICAL SYMBOLS ARE ONLY PLACED FOR COMPONENTS HAVING **INCLUDE IN MECHANICAL LOAD** SELECTED IN THE DIALOG BOX **SYMBOL ARTICLE DATA**.

— SEE “INCLUDE IN MECHANICAL LOAD” ON PAGE 161.

Specify which Type of mechanical symbols to place

Please note: *This is only relevant if your database contains different types of mechanical symbols* — see “Mechanical symbol” on page 612.

In the field **Database symbol field**, you can further specify which type of mechanical symbols you wish to place on the page. Please note that you can only place one type of mechanical symbols on a project page. When you specify a **Database symbol field** here, the **Database symbol field** is automatically changed under **Settings => Page setup** — see “Other symbols” on page 844.

Choose how to place the mechanical symbols

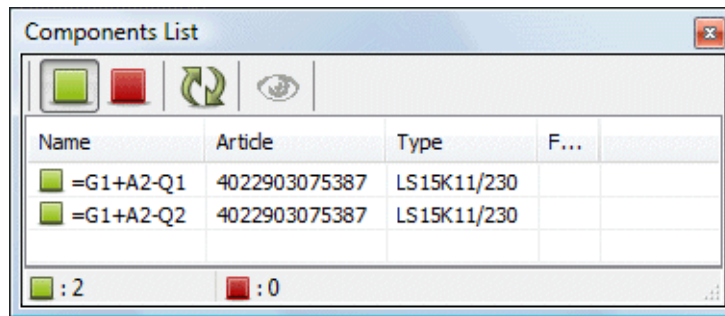
You can choose between:

- 1) Placing mechanical symbols from a list: See “Manual placing via list” on page 630

2) Placing the mechanical symbols directly on the page: See “Placing all mechanical symbols in a single operation” on page 630

1. Manual placing via list

If you select **Manual placement from list**, you get a window with a survey of the mechanical symbols you can place:

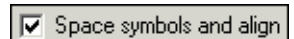


This is handled as described in “Placing symbols via lists” on page 674, “Placing the components from the Components List window” on page 675 in particular.

2. Placing all mechanical symbols in a single operation

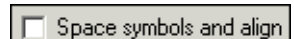
If you do *not* select **Manual placement from list**, the mechanical symbols will be placed directly on the page. You have two options for how to do this, depending on whether you choose **Space symbols and align**:

2.1 Space symbols and align

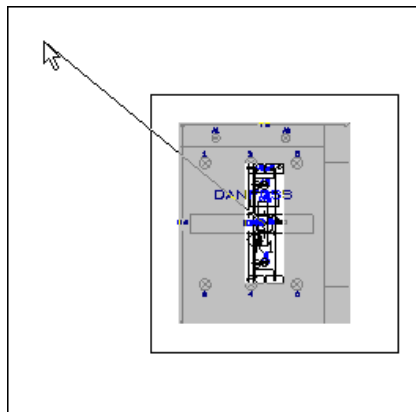


If you select **Space symbols and align**, you first get a pencil in the cross hairs, and then click on the position on the page, where you wish to place the first symbol. Then click where you wish to place the last symbol. Please note that the symbols will either be placed on a vertical or a horizontal line. If there is not enough space for the symbols within the specified area, you get a message telling you to point out a new starting position for the remaining symbols.

2.2 Place the symbols in a pile



If you do *not* choose **Space symbols and align**, you get a pencil in the cross hairs, and then click somewhere on the page.



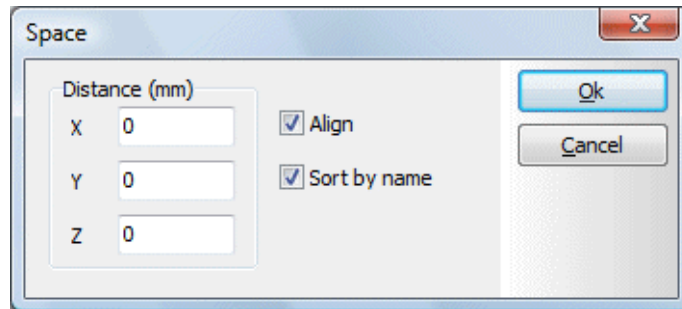
The mechanical symbols are then placed in a pile.

REARRANGING THE MECHANICAL SYMBOLS

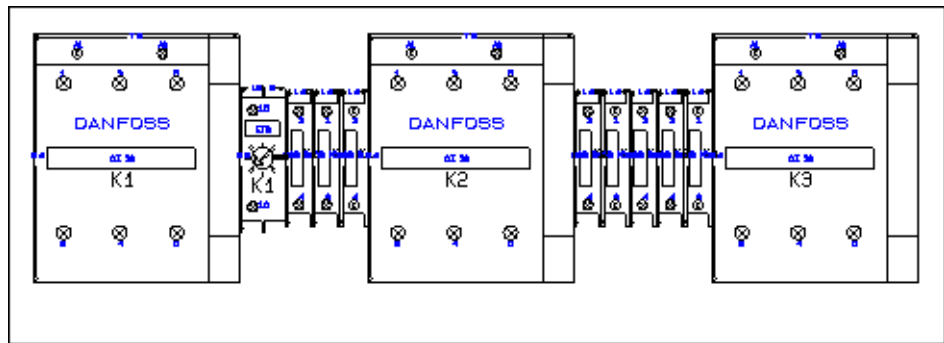
For a more appropriate arranging of the symbols, mark up a window surrounding the symbols.

Then choose **Edit => Space**. Now click where the first mechanical symbol shall be located. In the menu that appears you select **Align** as well as **Sort by name**.

Click **OK**.



Now click so that you only draw a horizontal line, which is shorter than the width of a single mechanical symbol. Hereby the mechanical symbols will be placed end to end.



By rearranging the symbols, you can then create your own switchboard.

**MAGNETIC MECHANICAL SYMBOLS**

WHEN YOU MOVE A MECHANICAL SYMBOL, IT CAN BE DIFFICULT TO POSITION IT, SO THAT IT IS PLACED EXACTLY SIDE BY SIDE WITH ANOTHER MECHANICAL SYMBOL.

THEREFORE, IF YOU ACTIVATE THE **MECHANICAL SNAP** BUTTON BEFORE POSITIONING THE SYMBOL, THE SYMBOL MAGNETICALLY CLINGS TO THE SYMBOL, THAT YOU WANT TO PLACE IT NEXT TO. READ MORE ABOUT THIS AS WELL AS OTHER OPTIONS IN "Magnetic snap and Snap in objects for mechanical symbols" on page 634.



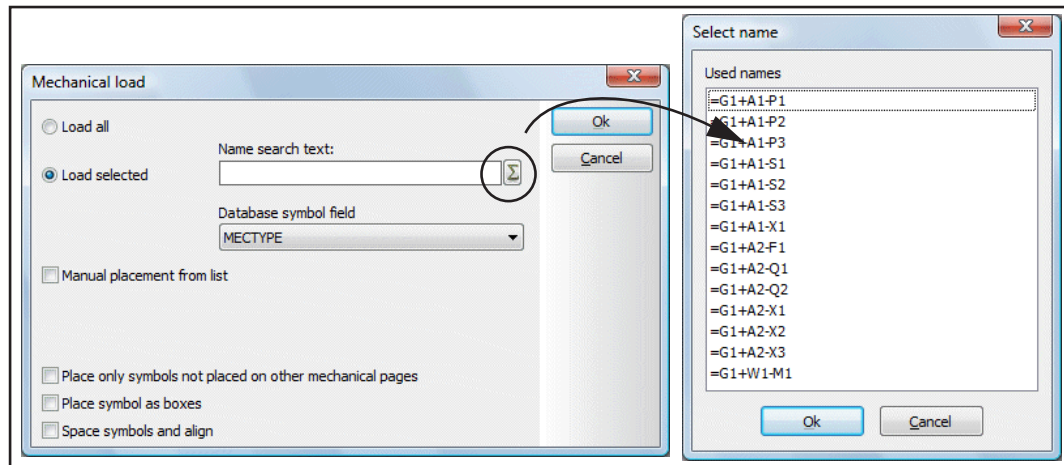
Function texts are also transferred during mechanical load

The function text for the components is also transferred to the mechanical symbols during mechanical load.



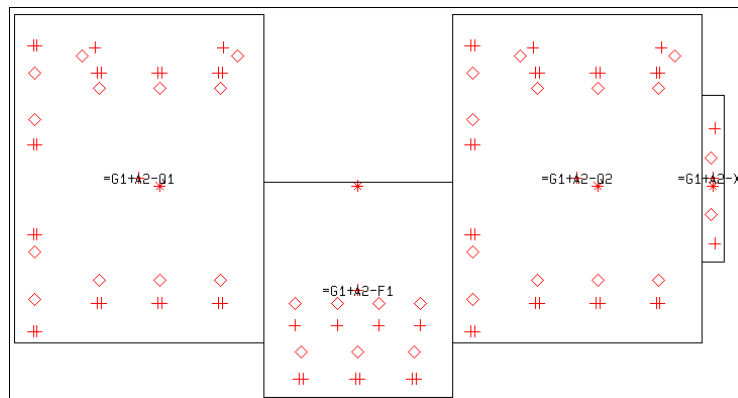
PLACING MECHANICAL SYMBOLS ACC. TO REFERENCE DESIGNATION

When you select **Functions => Mechanical Load** and click the **Used names** button, you enter the dialog box **Select name**, where you can select which reference designation to place mechanical symbols for:



PLACING MECHANICAL SYMBOLS AS BOXES

If you do not wish to use the mechanical symbols included in the component databases for **Automation**, select **Place symbol as boxes** in the dialog box **Mechanical load**.



Hereby the mechanical symbols are placed as boxes with connection points. The boxes gets the same measures as the mechanical symbols in the databases. If measures has been specified in the database, these measures are applied. Connection points are only placed if these are specified in the database.

— See “Performing the Mechanical load” on page 629.

PLACE ONLY MECHANICAL SYMBOLS NOT USED ON OTHER PAGES

If you in select **Place only symbols not used on other mechanical pages** in the dialog box **Mechanical Load**, **Automation** will only place mechanical symbols, that do not appear on other mechanical pages in the project.

MESSAGES AND WARNINGS DURING MECHANICAL LOAD

If something unexpected happens during the mechanical load, you will get messages and warnings displayed in a window.

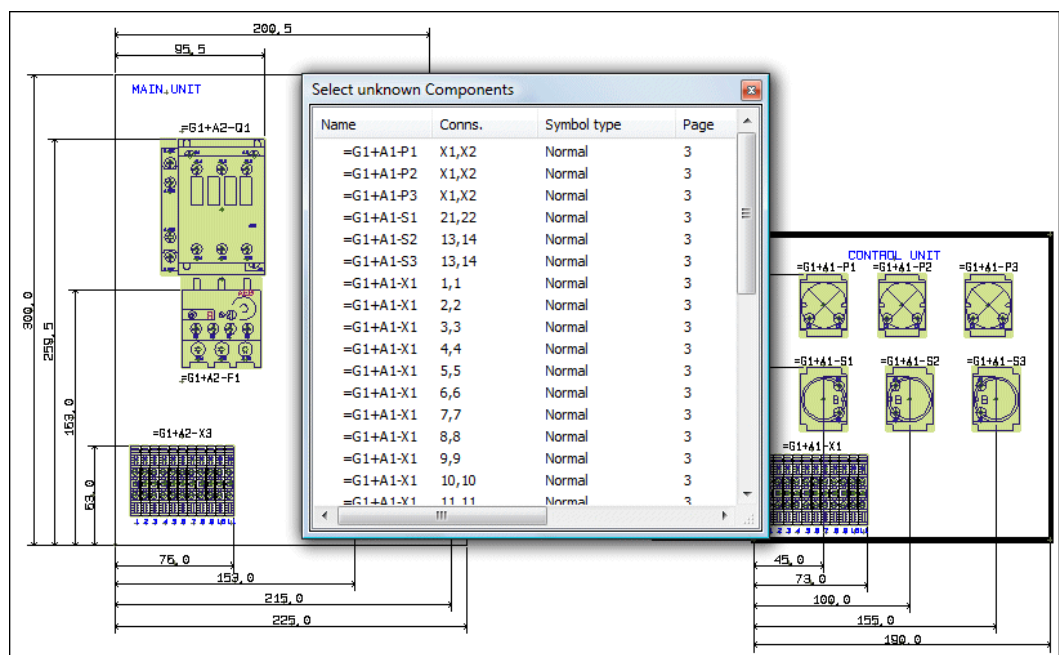
The cause can here be:

- 1) An article number have not been typed in for the electrical symbol in the project
- 2) The component does not have an attached mechanical symbol
- 3) The mechanical symbol have been removed from the hard disk

This should make it easier to make corrections succeedingly.

SELECT UNKNOWN MECHANICAL SYMBOLS

When electrical symbols are deleted in a project, this might result in mechanical pages containing mechanical symbols for components no longer applied in the project. To select these mechanical symbols, you can right-click on a mechanical page and select **Select unknown Components**:



Hereby the window **Select unknown components** is displayed, and all mechanical symbols, for which no electrical symbols exists, are selected on the page.

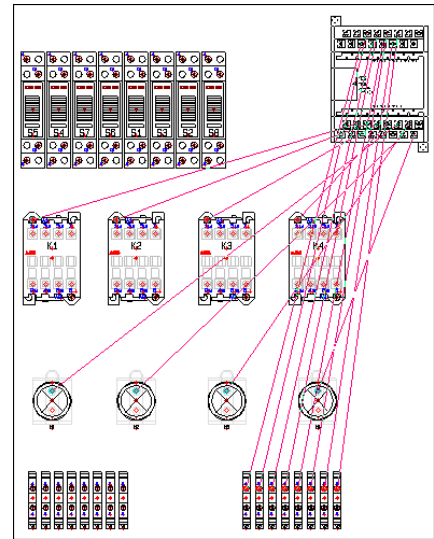
You can now delete the selected symbols, or for instance copy them to another page. The window **Select unknown components** functions like the **Select name** and the **Select components** windows, as described in “Finding symbols based on the Component Name” on page 238.

DISPLAYING CONNECTIONS FOR MECHANICAL SYMBOLS

When you move a mechanical symbol on a **Groundplan** page, it is displayed which of the other mechanical symbols are connected to the symbol — according to the information about the electrical connections on the electrical diagram pages. This makes it easier to find the optimal position for the mechanical symbol.

You get the same result when you right-click on a mechanical symbol and choose **Mark net**.

The function can be activated/deactivated by choosing **Settings => Pointer/Screen** and then click on **Net router**.



MAGNETIC SNAP AND SNAP IN OBJECTS FOR MECHANICAL SYMBOLS

When you wish to place mechanical symbols accurately relative to each other, you can either use "Magnetic snap" or "Snap in objects".

Magnetic snap



When you wish to place a mechanical symbol, so that it is placed exactly side-by-side with an already placed mechanical symbol, just activate the **Magnetic snap** button (short-cut key <m>), when placing the symbol. The mechanical symbol will then be placed precisely side-by-side with the placed symbol — with either the left side, the right side, the top or the bottom of the already placed symbol.

QUICK PLACING IN A DIN-RAIL OR IN A SPECIFIED CUT-OUT



When you wish to easily place a mechanical symbol in a panel on a DIN-rail or in a pre-specified cut-out, just activate the **Magnetic snap** button and keep the <Shift> key pressed down when placing the mechanical symbol. This places the symbol precisely in the right side or in the left side of the rail or cabinet.

For symbols without states, this requires that the area for placing components / mechanical symbols are drawn with the colour NP in layer 0 or in layer 255. For symbols with states, this area must be drawn with the colour NP in each individual state. If you wish to see the specified area on print also, you must also draw the area with a "normal" colour. To see the specified area, you must also make sure that the area drawn with the "normal" colour is placed on top of the NP colour. To do this, right-click and choose **Draw order => Bring to front**, as described in "Draw order" on page 638.

Snap in objects



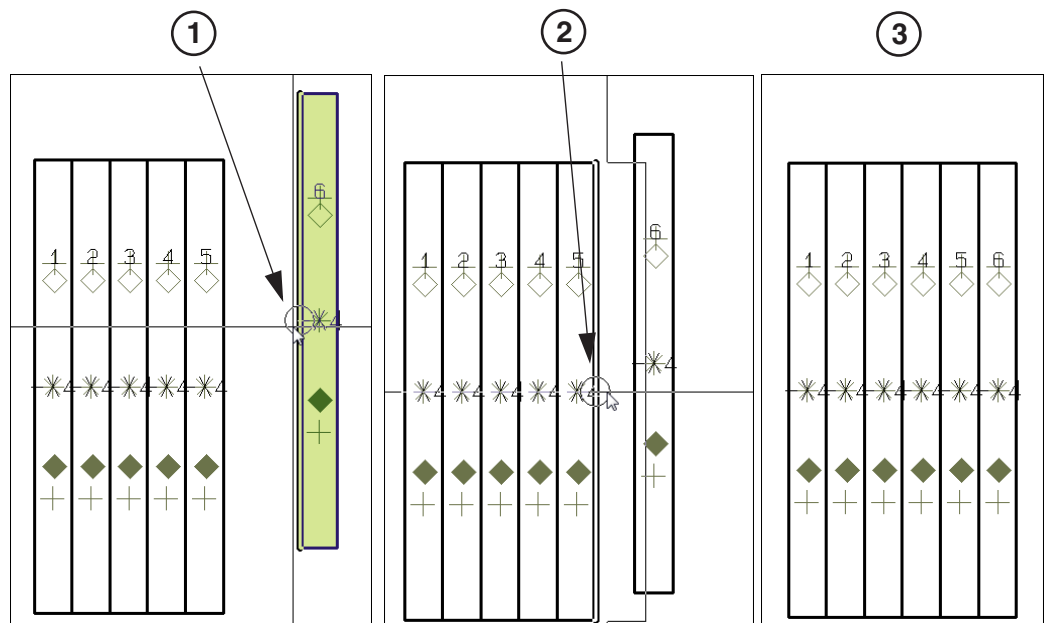
In stead of Magnetic snap, you can activate the **Snap in objects** button, which gives you the option of "snapping" based on several points on both the object you are placing, and on the object you are placing the object next to. (The "objects" are here mechanical symbols).

You can for instance snap to all object corners, and to the middle of all sides of the objects.

When the function is activated, and the cross hairs is above a possible snapping point, a small "o" is displayed in the cross hairs:



- 1) Activate the **Snap in objects** button, and click the mechanical symbol, that you wish to move. Now place the cross hairs above the point you wish to snap from. When the small "o" is displayed in the cross hairs (see the middle of the left side for the symbol in figure 1 below), press the left mouse button, and keep it pressed down.
- 2) Now *drag* the mechanical symbol over to the mechanical symbol, that you wish to place it next to. When the small "o" is displayed in the cross hairs over the point in the other symbol that you wish to snap to (the middle of the right side of the symbol in image 2 below), release the mouse button.
- 3) The mechanical symbol is now positioned precisely in relation to the other mechanical symbol.



You can also use Snap in objects, when you have selected multiple symbols.

REFERENCES IN LISTS TO THE POSITION OF MECHANICAL SYMBOLS

If a reference system have been placed on a project page containing mechanical symbols, it is also possible to refer to the position of the symbols on the page according to the reference system.

Placing the reference system

The reference system is placed on the page by choosing **Settings => Page data => Reference**, as described in the **Automation** manual. Here the reference system can be activated vertically and horizontally at the same time.

Please note, that the reference chosen to be **Main reference** is mentioned first in the datafield.

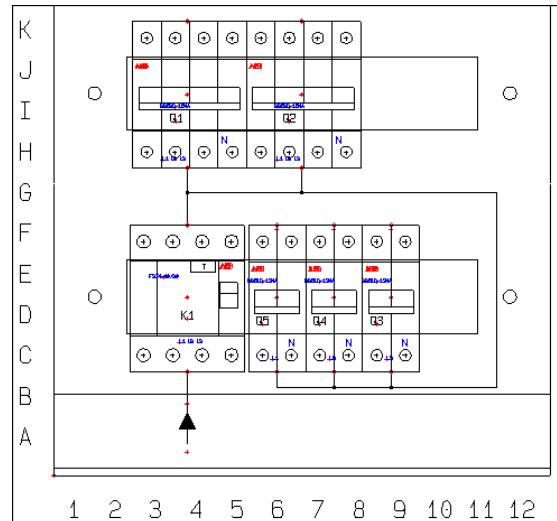
Inserting the datafield

To refer to the position of the mechanical symbols, choose **Insert => Insert Datafield**, as explained in “Datafields” on page 110. Here you click on **Parts/Components list**, and choose the datafield **Mechanical position**.

Here you can choose between **x,y position**, which gives the components coordinates in relation to the origo of the page, and **Field position**, which gives the coordinates in relation to the reference system.

In the components list for the project, you can therefore specify where in the switchboard the components are placed.

— Read more about how to create lists in “Creating Lists” on page 752.



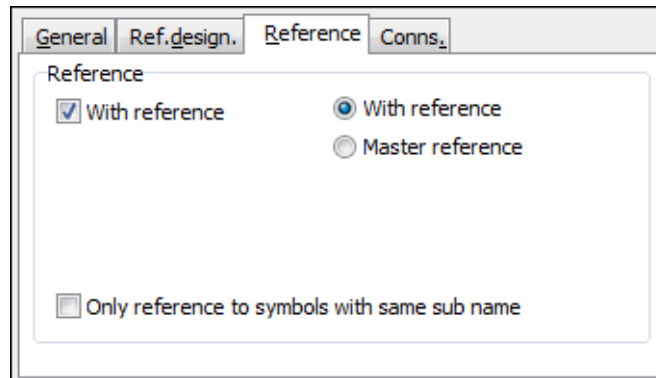
REFERENCES FROM MECHANICAL TO ELECTRICAL SYMBOLS

By default no references are displayed from mechanical symbols on the **Ground-plan/Mechanical** pages, to the electrical symbols on the **Diagram** pages.

When you wish to create a reference from a mechanical symbol to one of the electrical symbols for the component, do this:



- 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 tab **Reference**, select **With reference** and then **With reference** (in the right-hand side), and click **OK**.



You have now created a reference from the mechanical symbol. When you have selected **Symbols** and double-click on the reference for the mechanical symbol, you jump to the electrical symbol for the component, which is set to **Main reference**. If none of the electrical symbols for the component is set to this, you jump to the first found symbol for the component, which exists on a diagram page in project.



MECHANICAL SYMBOLS MUST HAVE A SIMPLE REFERENCE TYPE

THIS ONLY APPLIES FOR MECHANICAL SYMBOLS WITH A SIMPLE REFERENCE TYPE. SYMBOLS WITH **RELAY** OR **MASTER REFERENCE** MUST THEREFORE BE SET TO **NORMAL** — OR TO ANOTHER SYMBOL TYPE WITH SIMPLE REFERENCES — TO MAKE IT WORK.

MEASUREMENT OBJECTS

Read about measurement objects in “Measurement objects” on page 641.

DRAW ORDER

The *draw order* specifies which objects are in the front, and which objects are in the back on the drawing, in case of overlapping objects.

The draw order is fixed for the different object types, but within the same object type — e.g. for symbols —, it is possible to change the draw order, as described below.

The draw order is specially important on pages with mechanical drawings.

Draw order for different object types

When placing objects on a page, the objects will be drawn in the following draw order. (Please note that the objects at the *bottom* of the list will be placed at the *top* of the project drawing):

- 1) Drawing headers and current path numbers
- 2) Images and OLE objects
- 3) Datafield placed on the page with images
- 4) Lines
- 5) Circles
- 6) Free texts
- 7) Datafields placed on the page with text
- 8) Symbols

As you can see in the list, symbols are placed in the front, and they will therefore cover any other object type they might be overlapping.

Draw order for objects of the same type

To specify the draw order for objects of the same type — as for the mechanical symbol in the figure to the right — you must right-click on the object, and choose **Draw order**.

You can then choose:

- **Bring to front:** Place the object in front of all other overlapping objects of the same type.
- **Send to back:** Place the object behind all other overlapping objects of the same type.
- **Step forward:** Moves the object one position forwards, among to the overlapping objects.
- **Step backward:** Moves the object one position backwards, among to the overlapping objects.

FROM MECHANICAL TO ELECTRICAL DRAWING

When you choose to start a project by creating a mechanical drawing, you can do this by for instance placing the mechanical symbols for a motor control, such as fuses, motor starters and contactors.

When working with an attached database in **Automation**, the program can automatically fetch the *electrical* symbols, that are attached to those components, for which their *mechanical* symbols have been placed on the mechanical pages.

The electrical symbols can then subsequently be placed in the diagram pages in the project.



OTHER OPTIONS

SEE ALSO “PLACING SYMBOLS VIA LISTS” ON PAGE 674, “LOADING COMPONENTS FROM A GROUND PLAN PAGE” ON PAGE 677.

Do for instance like this:

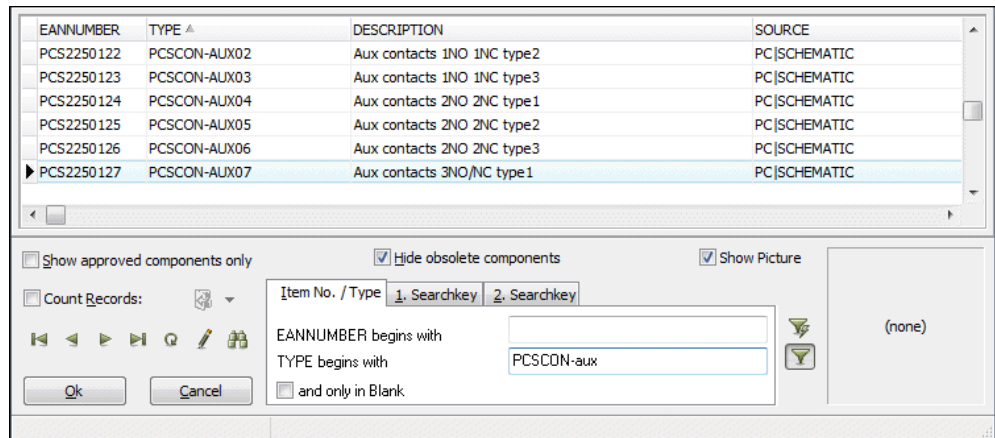
Creating a new project with a diagram page and a mechanical page

- 1) Choose **File => New**.
- 2) In the dialog box **New**: Select the template **PCSstart** and click **OK**. — You have now chosen a project containing a single page for drawing diagrams.
- 3) In the dialog box **Settings**: Click **OK**.

Placing a mechanical symbol on a Ground plan / Mechanical page

- 4) Click on the page tab for page **9**, to select a mechanical page.
- 5) While being on the mechanical page: Press <d> to enter the database.

- 6) Select a mechanical symbol: For instance write *PCSCON-AUX* in the field **Type begins with**, choose the contactor with the type *PCSCON-AUX07*, and click **OK**.



- 7) You now get the mechanical symbol for the component in the cross hairs: Click somewhere on the mechanical page to place the symbol, click **OK** in the dialog box **Component data** and press <Esc> to remove the symbol from the cross hairs.

- 8) If necessary, place more mechanical symbols on the page.

Fetch symbols automatically on the diagram page

- 9) Shift to the diagram page by clicking on the tab **5** at the bottom of the screen.
- 10) On the diagram page: Choose **Functions => Show available**.
- 11) In the dialog box **Show available**: Type in the symbol name, and click **OK**.
- 12) You now get a pickmenu containing the electrical symbols for the component, and can place these symbols as you normally do.



DISPLAY SYMBOLS NOT YET PLACED IN DIAGRAMS

WHEN YOU HAVE PLACED THE MECHANICAL SYMBOLS *BEFORE* THE ELECTRICAL SYMBOLS, AND CLICK ON THE BUTTON **USED NAMES WITH AVAILABLE SYMBOLS**, YOU GET A LIST OF ALL THE SYMBOLS, YOU HAVE NOT PLACED IN THE DIAGRAM PAGES YET. IN THIS CONNECTION THE BUTTON IS A HELP TO REMEMBER TO DRAW ALL THE CIRCUITS.



MEASUREMENT OBJECTS

THIS CHAPTER DESCRIBES	
— INSERTING MEASUREMENT OBJECTS	P. 641
— FREE MEASUREMENT	P. 643
— ROTATING MEASUREMENT OBJECTS	P. 643
— EDITING THE MEASUREMENT START AND END POINTS	P. 644
— EDITING MEASUREMENT DATA	P. 645
— MEASUREMENT OBJECTS AND LAYERS	P. 645
— DELETING MEASUREMENT OBJECTS.....	P. 645



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 508.

Inserting measurement objects

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

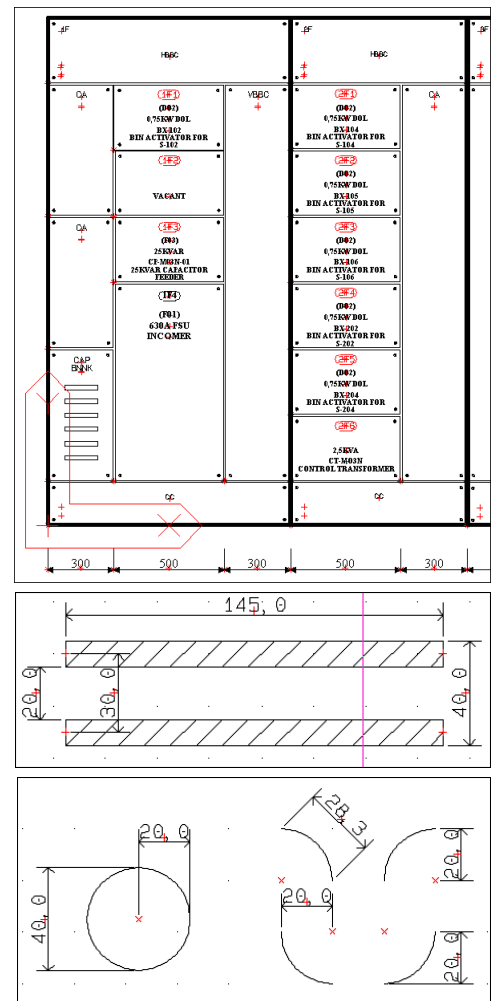
The starting point for the measurement object

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.

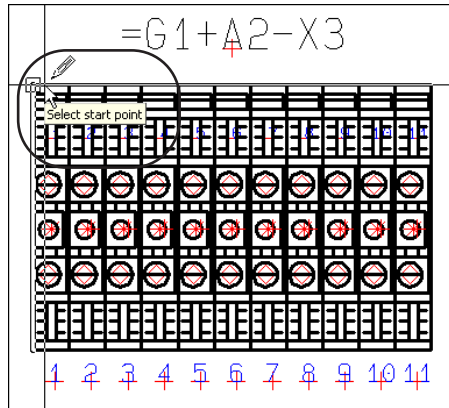
If you want to start the measurement from a symbol — e.g. from the symbol reference point or from a corner point on the symbol —, the program automatically examines the symbol definition, and finds the exact coordinates from this point.

If filled lines are applied, you can both measure from the centre 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:



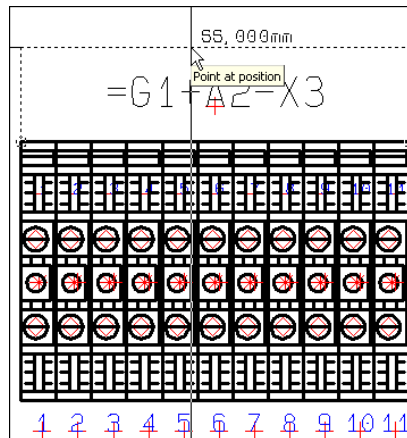
Click to point out the start point for the measurement object.

— 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 643.

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 643.



Click to place the measurement object.



MEASUREMENT OBJECTS ON OTHER LAYERS

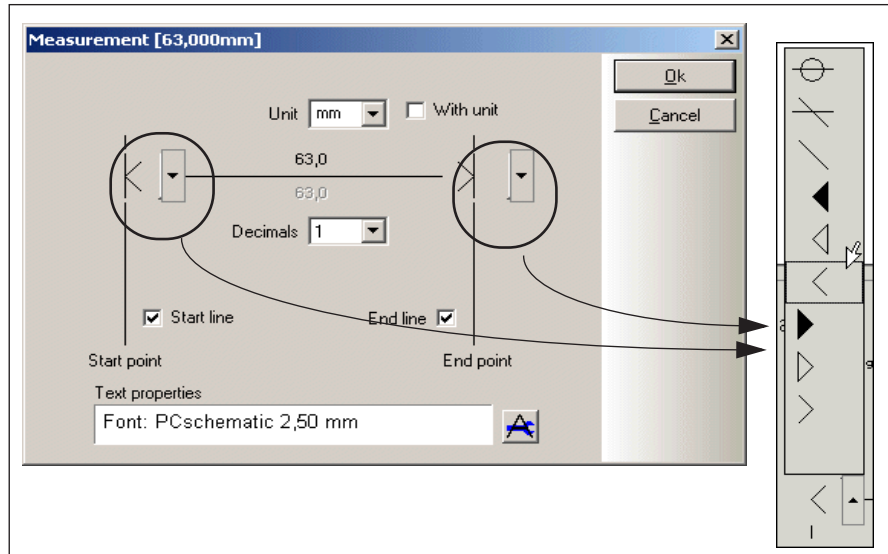
YOU CAN PLACE THE MEASUREMENT OBJECTS ON ANOTHER LAYER, THAN THE LAYER WHERE THE DRAWING IS PLACED. WHEN WORKING ON ANOTHER LAYER, JUST PRESS THE <ALT> KEY WHILE POINTING OUT THE START AND END POINTS FOR THE MEASUREMENT OBJECT.

WHEN YOU PLACE MEASUREMENT OBJECTS ON ANOTHER LAYER, YOU CAN EASILY DISABLE/ENABLE SHOWING THE MEASUREMENT OBJECTS.

READ MORE ABOUT THIS IN “LAYERS, ELEVATION AND SCALING” ON PAGE 477.

Specifying the Measurement data

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

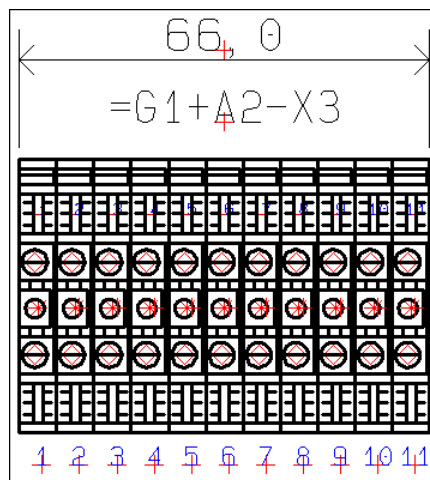


Here you can specify the appearance of the measurement object, whether to attach a unit specification to the measurement, and how many decimals to apply.



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

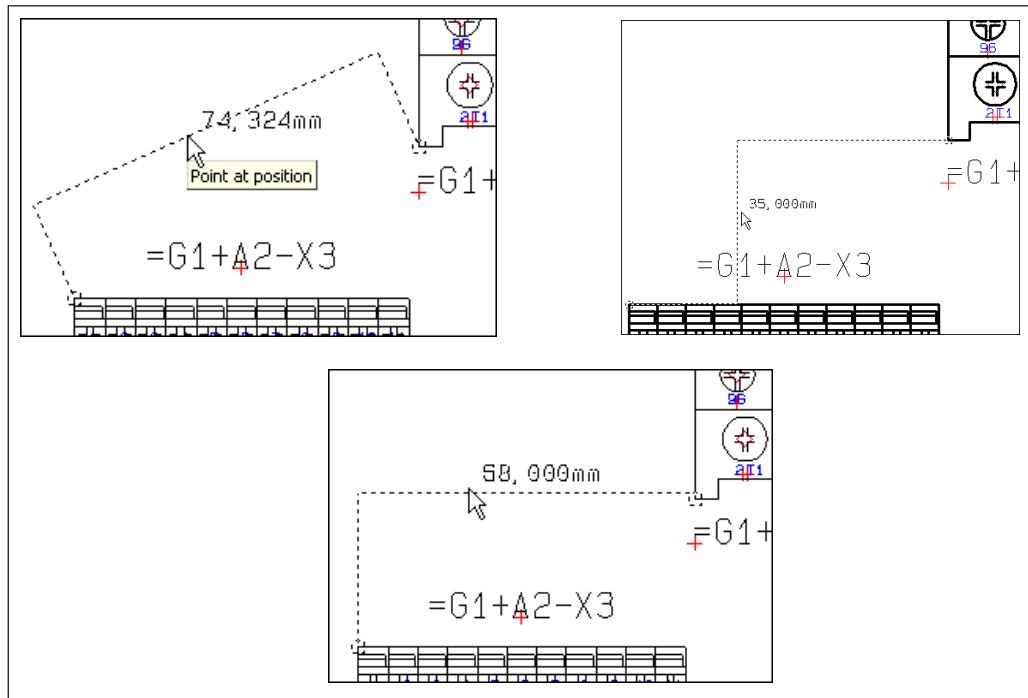
When you choose **Functions => Measurement**, the program will suggest start and end points for the measurement object, based on the objects in the diagram.

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:

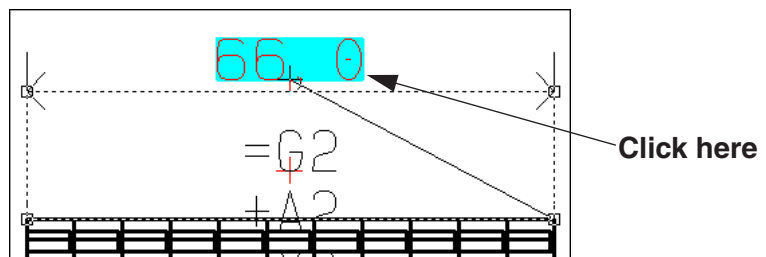


Editing the measurement start and end points

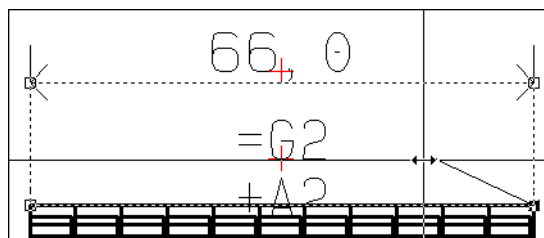
To edit the coordinates of start and/or end point of a measurement object, do the following:



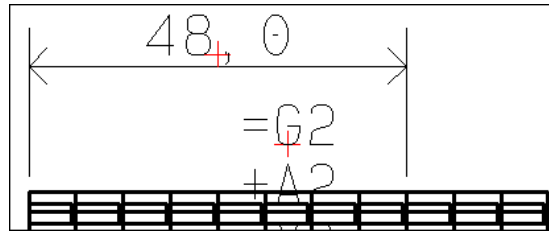
- 1) Click on the **Texts** button, and select the measurement object by clicking on the measurement text:



- 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:

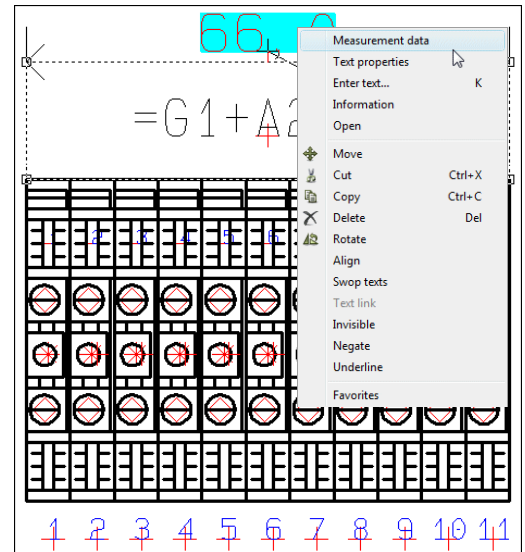


Editing Measurement data

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 643.



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 477, in particular “Printing pages with layers” on page 481.

Deleting a measurement object



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



Export / Import

In this part you can see how to import and export files.

You can read about:

- Exporting Lists as Files p. 649
- Exporting/Importing DWG/DXF files p. 660
- Integration with Excel p. 666
- OLE objects p. 667
- Placing symbols via Lists p. 674
- Opening files from the Tools menu p. 679
- The PCISCHEMATIC Viewer p. 680

EXPORTING LIST FILES

In **Automation** you can create list files — such as parts and components list files, cable lists, PLC lists, terminal lists and connection lists.

THIS CHAPTER DESCRIBES	
— LIST FILES FOR AREAS	P. 649
— HOW TO CREATE A LIST FILE	P. 650
— THE FORMAT FILE	P. 650
— LISTS AND EXCEL	P. 651
— LISTS IN XML FORMAT	P. 652
— THE BUTTONS IN THE DIALOG BOX	P. 653
— SPECIFYING THE FILE NAME	P. 654
— SPECIFYING THE CONTENTS OF THE FILE	P. 654
— HOW TO MATCH THE SYSTEM YOU EXPORT TO	P. 654
— REFERENCE DESIGNATIONS IN LISTS TO FILES	P. 656
— AN EXAMPLE.....	P. 657
— THE LIST FILES AND THEIR EXTENSIONS	P. 658
— CHOOSING LIST FILE SETTINGS FROM THE TOOLS MENU.....	P. 659

Different types of list files

This chapter describes how to create *files* with parts lists, components lists, terminals lists, cables lists, PLC lists and connections lists.

These types of files are created according to the same directions, and are created from the same type of dialog boxes.

It is possible to create comma separated files, as well as column files. These files can subsequently be loaded into other programs.

Components list files can for instance be used for labels to mechanical assemblies.

Furthermore, it is possible to create wire number files. — Please refer to “Wire number Files” on page 401 for further details.

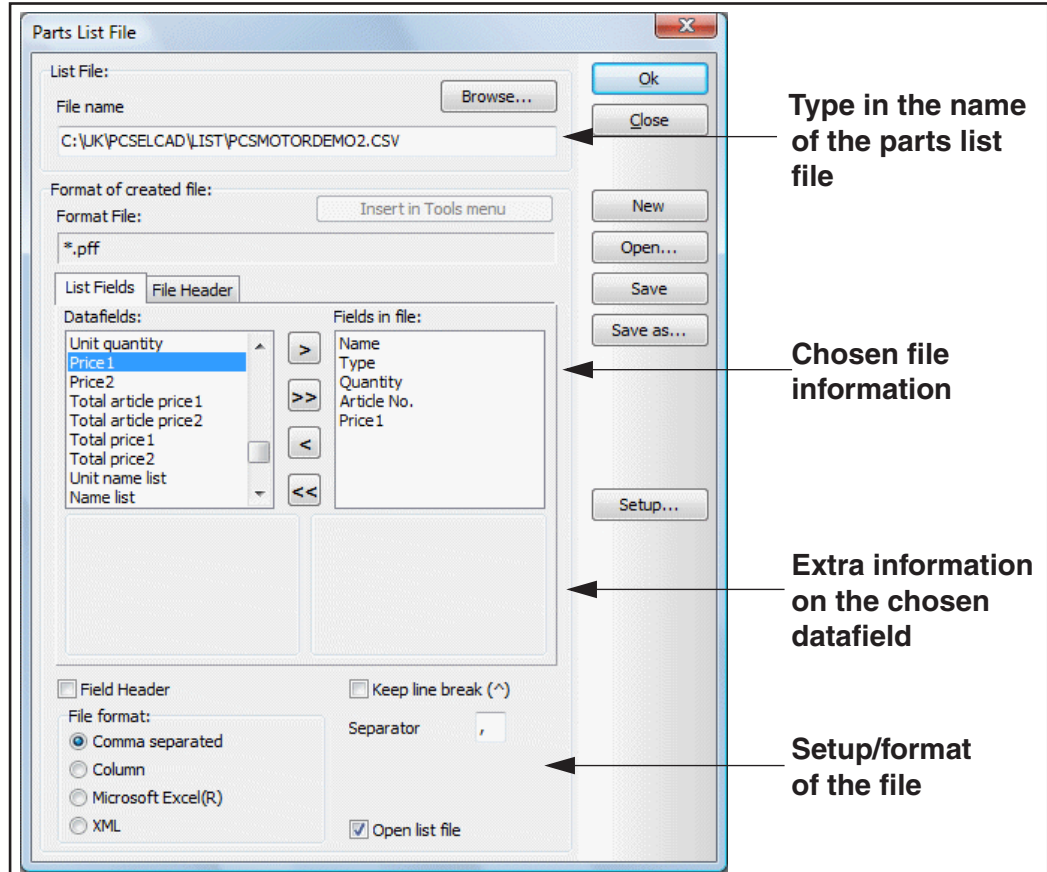
LIST FILES FOR AREAS

If you have selected an area — or just some symbols — before choosing **Lists => Parts List File**, the list will only contain information from the selected area. When n area is selected, the list will be generated for the full project.

CREATING A LIST FILE

In the following it is shown how to create a parts list file, but the other types of lists can be created correspondingly.

To create a file containing parts list information, you choose **Lists => Parts List File**. You then enter this dialog box:



THE FORMAT FILE

Along with the list file, a so-called *format file* is created, containing the information you specify in this dialog box.

It contains information on *how* the file is constructed, but no information about the individual components in the list.

Thus the format file contains information on which datafields are included in the parts list file, the order in which they appear, whether the file is comma separated ... and so on.



THE PROGRAM MEMORIZES FROM WHERE THE FORMAT FILE IS LOADED

Automation MEMORIZES FROM WHERE YOU LAST LOADED "LIST FILE" FORMAT FILES. WHEN YOU WORK ON A NETWORK, THESE FILES CAN THEREFORE BE MOVED TO A FOLDER ON THE NETWORK.

Transfer list settings for existing Project lists to List files

When you wish to create a list file with the same list settings as an existing list page in the project, do the following:

- 1) Select the page in the project.
- 2) Choose **Lists => Parts List file** — if you wish to create a parts list file.
- 3) Click on the button **As Page**. This fetches the list settings and datafields from the project list page, and the similar settings are created for the list file.
- 4) Make corrections if necessary — to save the list in e.g. Excel or XML format — and click **OK**.

Please note that the button **As Page** only appears in the dialog box when you have selected a list page in the project.

LISTS AND EXCEL

In **Automation** lists can be created directly in Excel.



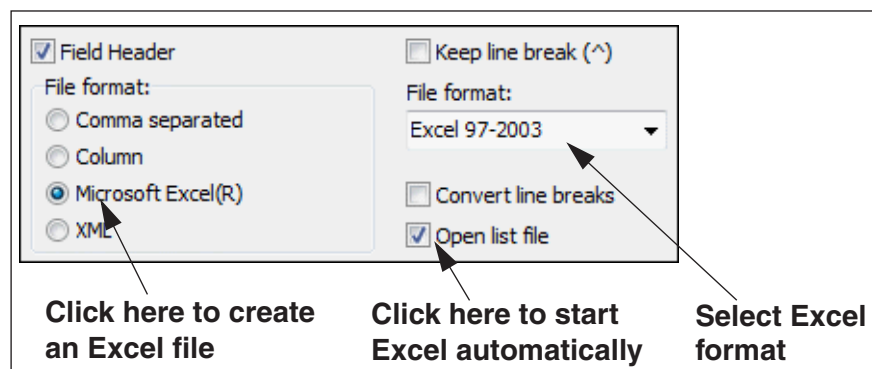
EXCEL MUST BE INSTALLED

IT IS ONLY POSSIBLE TO SAVE IN EXCEL FORMAT — OR OPEN LISTS IN EXCEL FORMAT — IF EXCEL IS INSTALLED ON THE PC. READ MORE ABOUT THIS IN “EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT” ON PAGE 666.

— See also “Lists in XML format” on page 652.

All lists can be loaded/written as Excel files

Automation can create all list files as Excel files — just like the program can load Excel files.



Starting Excel from Automation

When you create a list file, you can start Excel from **Automation** at the same time by selecting **Open list file**.

It is a good idea also to select **Field Header** to get a survey of which information is placed in which columns in the Excel file.

Converting line breaks when exporting to Excel

When you select **Convert line breaks**, the **Automation** line breaks (^) are converted to line breaks in the Excel file.

When Excel files are imported in **Automation**, all line breaks in the Excel file are always converted to the line break character (^) in **Automation**.

Placing symbols from an Excel components list

Furthermore, you can place symbols in **Automation** based on an Excel components list, as described in “Placing symbols via lists” on page 674.

LISTS IN XML FORMAT

When you, at the bottom of the dialog box **List File**, select the file format **XML**, the list is saved in the XML format.

```
<?xml version="1.0" encoding="UTF-8" standalone="yes" ?>
- <PCschematicData>
- <PARTSLIST>
- <FIELDS>
  <FIELD attrname="Name" fieldtype="string" WIDTH="3" />
  <FIELD attrname="Type" fieldtype="string" WIDTH="17" />
  <FIELD attrname="Quantity" fieldtype="string" WIDTH="2" />
  <FIELD fieldname="Article No." attrname="Article_No." fieldtype="string" WIDTH="13" />
  <FIELD attrname="Price1" fieldtype="string" WIDTH="7" />
</FIELDS>
- <ROWDATA>
  <ROW Name="-M1" Type="ZEFG45789" Quantity="1" Article_No.="1723410403" Price1="3750.00" />
  <ROW Name="-X2" Type="AB1NE435U" Quantity="1" Article_No.="3389110585506" Price1="17.40" />
  <ROW Name="-X2" Type="AB1VV435U" Quantity="28" Article_No.="3389110586435" Price1="6.15" />
  <ROW Name="-S2" Type="XB2BA31" Quantity="2" Article_No.="3389110610024" Price1="63.80" />
  <ROW Name="-S1" Type="XB2BA42" Quantity="1" Article_No.="3389110610048" Price1="63.80" />
  <ROW Name="-P3" Type="XB2BV74" Quantity="3" Article_No.="3389110611229" Price1="81.35" />
  <ROW Name="-Q1" Type="LS15K11" Quantity="2" Article_No.="4022903075387" Price1="630.00" />
  <ROW Name="-F1" Type="B77S" Quantity="1" Article_No.="4022903085584" Price1="417.00" />
  <ROW Name="-W2" Type="NOIK 19 X 1.5B mm" Quantity="1" Article_No.="5702950116682" Price1="" />
  <ROW Name="-W2" Type="H07rn-F" Quantity="2" Article_No.="5702950410537" Price1="25.00" />
</ROWDATA>
</PARTSLIST>
</PCschematicData>
```

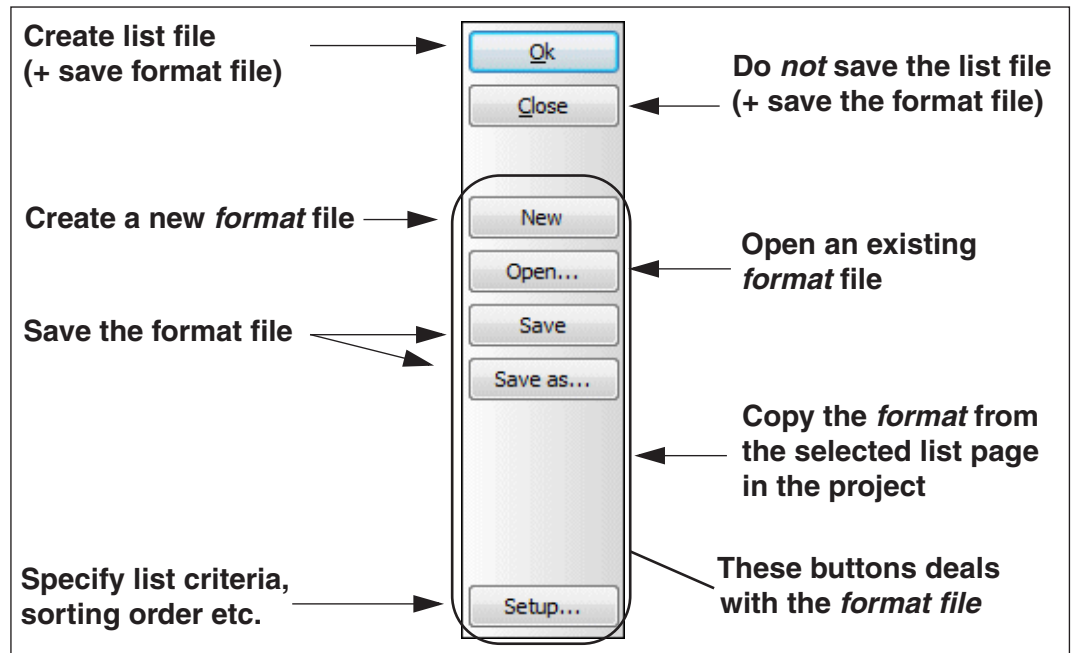
Among other options, this opens the possibility to include several lists in the same file, and to load the created lists in database programs, that handles the XML format. The created XML file can for instance be displayed in the Internet Explorer.

Please also see:

- “Lists and Excel” on page 651.
- “Other options when importing XML list files” on page 678.

THE BUTTONS IN THE RIGHT-HAND SIDE OF THE DIALOG BOX

In the right-hand side of the dialog box, you find the following buttons:



OK and Close

When you click **OK** the *list file* is created, and you will be asked whether to save the changes in the *format file*.

If you click **Close**, the list is not created, but you get the option of saving the changes in the format file.

New

Click here to create a new *format file*. You then enter the dialog box **Save as**, and do not risk overwriting the previously applied format file.

— See “The format file” on page 650 if necessary.

Open

The next time you want to create a parts list file, which shall be constructed in a similar way, you load this format file by clicking **Open**, and therefore do not have to specify the file settings again.

Save and Save as

The buttons **Save** and **Save as** are used for saving the *format* file.

As Page

Click here to fetch the settings from an existing project list page, as described in “Transfer list settings for existing Project lists to List files” on page 651.

This button only appears when you have selected a list page in the project.

Settings/Criteria

If you want to specify further criteria for how to select the components, you click on **Setup**. Hereby you enter the dialog box **List setup**, which you can read more about in “List settings” on page 542.

SPECIFYING THE FILE NAME

Under **List File** in the upper part of the dialog box — see page 650 — you specify under which name the parts list shall be saved. If the selected folder is not appropriate, you click on **Browse**, and select another folder.

The suggested extension to the file name is **.par**, because you are dealing with a parts list.

SPECIFYING THE CONTENTS OF THE FILE

To specify which datafields shall be included into the file, select the desired datafields, and click on the **Transfer information** button. Please note that you can transfer more information at the time by selecting datafields using <Shift> or <Ctrl>.



By selecting fields in the **Used fields** column and clicking on the **Revert information** button, you can correspondingly remove fields from the list of fields to be included into the file.



The information is arranged in the order in which they are transferred to the **Fields in file** column. If you therefore want the manufacturer first in the file, start by transferring the **MANUFACTUR** field, so that it is placed on top of the **Fields in file** list.

If you want to transfer all the fields, click on the **Transfer all fields** button.



If you want to empty the list, click on the **Revert all fields** button.

Please note, that you are allowed to place the same datafield more than one place in the file.

The order of the fields can be changed using *drag and drop*.



SYMBOL DATAFIELDS IN COMPONENTS LISTS

YOU CAN ALSO INCLUDE THE SYMBOL DATAFIELDS, THAT YOU HAVE CREATED BY YOURSELF, IN THE EXPORTED COMPONENTS LIST FILE. — SEE “CREATING DATA SYMBOLS” ON PAGE 732.

Extra information

For some datafields — such as **Name** in the figure on page 650 — you can determine whether extra information shall be included into the datafield. Choose the extra information you want by clicking on it.

MATCHING THE SYSTEM YOU EXPORT TO

In order to match the file format of the system you export the file to, you can also include field headings and specify the file format.

Field headings

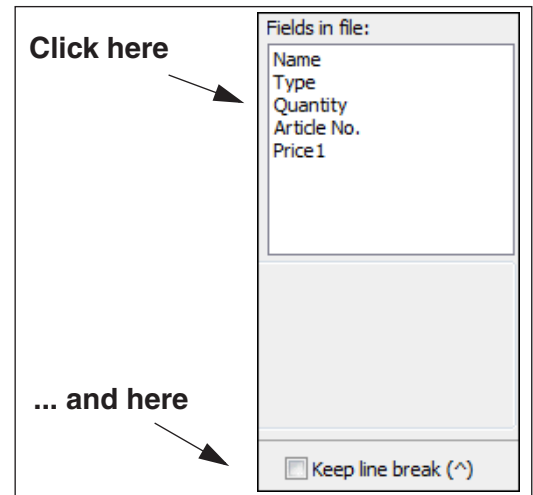
When you want the first record in the file to contain the names of the applied datafields, select **Field header**.

Whether to select **Field header** or not, depends on the program you wish to transfer the file to.

Keep line break

When you want a text to contain a line break, do the following:

- 1) Place the line break character (^) in all project texts that needs a line break — as described in “Texts in more than one line” on page 116
- 2) In the dialog box **Format file** click on the relevant field type, and select **Keep line break (^)**

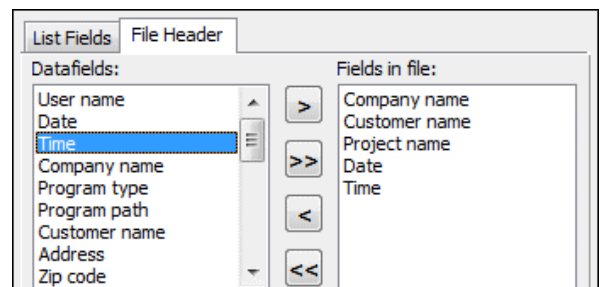


If for instance you want the **Type** field in the file to contain a line break in the file, first click on **Type**, then click on **Keep line break (^)**.

File header

When you click on the tab **File Header**, you can specify which data to include in the file header — if you wish to include a file header.

You transfer the datafields to the file as described in “Specifying the contents of the file” on page 654.

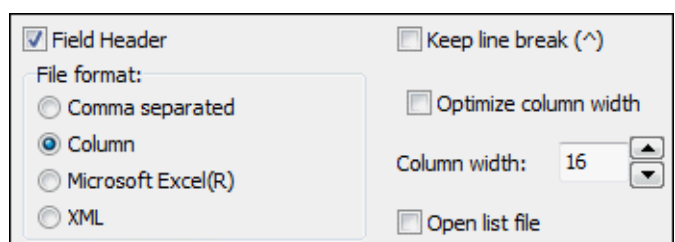


File format

The settings you specify here, depends on the system you are going to export the file to. You can choose between different file formats: Comma separated files, Column files, Excel files and XML files.

If you choose **Comma separated**, the fields will be separated by a comma. If you need another sign to separate the datafields, you type it in the **Separator** field.

If you choose **Column**, the datafields will be placed in columns. The width of the columns are then displayed in brackets for the individual datafields on the list. If you want to change the width of the column for e.g. the datafield **Type**, click on the **Type** field in the list of used fields, and type in the desired



width in the **Column width** field. If you want to optimize the column widths, select **Optimize column width**.

Thus the width of each column is set to the width of the longest datafield of the type plus one. For the field **Type** the column width is here set to **10**, because the longest field in the **Type** column is nine characters long.

Name	Type	Article no.	Quantity	Price1
M1	ZEFG45789	1723410403	1	3750.00
W1	GKS0J	3034529923	1	25.00
X1	AB1UU435U	3389110586435	5	30.75
S2	XB2BA31	3389110610024	2	127.60
S1	XB2BA42	3389110610048	1	63.80
H3	XB2BU74	3389110611229	3	244.05
K2	LS15K11	4022903075387	2	1260.00
Q1	B77S	4022903085584	1	417.00

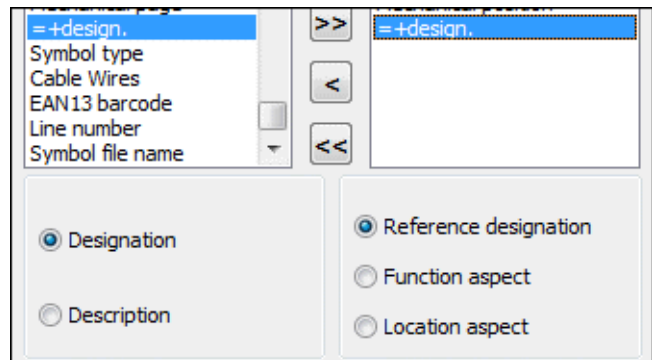
Please note, that if you have selected **Field header**, this can determine the width of the columns. On the figure above, this is the case for the **Quantity** column.

If you choose **Microsoft Excel(R)**, please refer to “Lists and Excel” on page 651.

REFERENCE DESIGNATIONS IN LIST FILES

For list files it is possible to insert the field **=+design.** and then choose **Page designation**, **Function aspect** or **Location aspect** respectively.

This makes it possible to place this information in independent columns in the file, without the component name appears in the column.



Please note that you must specify whether to place a **Designation** or a **Description**.

If you wish to include the component name, use the field **Name**, to which the reference designations can also be added.

EXAMPLE

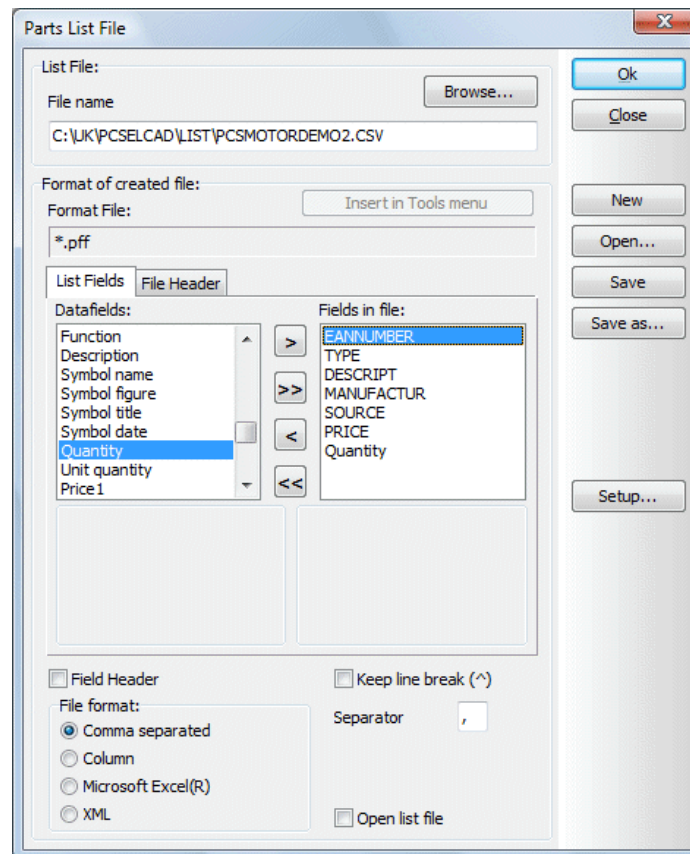
Choose **Lists => Parts List File**, and type in the name you want to assign to the parts list file in the **Fields in file** field.



In the field **Datafields** choose the datafields displayed in the dialog box below under **Fields in the file** one by one, and click on the **Transfer information** button.



If you accidentally transfer a wrong datafield, click the **Revert information** button. The datafield is then removed from the list.

**DATAFIELDS IN UPPERCASE ARE FETCHED FROM THE DATABASE**

DATAFIELDS WRITTEN IN UPPERCASE ARE FETCHED FROM THE DATABASE. THE OTHER DATAFIELDS ARE FETCHED FROM THE PROJECT DIAGRAMS.

You can change the order of appearance in the **Fields in file** column using *drag and drop*.

Click **OK** to create the file.

You will now be asked whether you want to save the changes in the setup of the parts list file. If you want to create a similar file another time, click **Yes**, otherwise click **No**.

The parts list file have now been created.

THE LIST FILES AND THEIR EXTENSIONS

When you want to create the other types of list files, this is done just like you create a parts list file.

Below you can see which extension the format files are assigned for the different file types.

LIST TYPE	FORMAT FILE
PARTS LIST	.PFF
COMPONENTS LIST	.CFF
TERMINALS LIST	.TFF
CABLE LIST	.CBF
PLC I/O LIST	.PLC
CONNECT POINTS LIST	.COF
CONNECTION LIST	.CNF

For further information about the datafields for the different list types, please refer to “Creating Lists” on page 752.

Connect points File

A *connect points file* contains all connection names for all components in the project. If for instance you create a connect point file containing the datafield **Connected**, you can generate a survey of which connection points are connected / not connected in the project.

Connections list to file

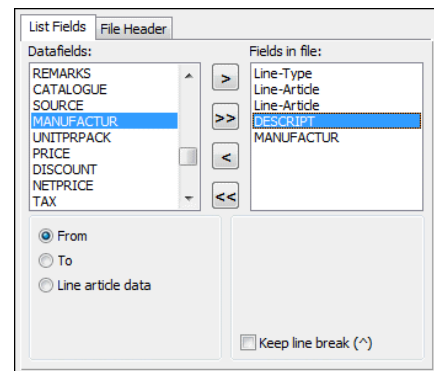
When you use fields from the database, you can for *connection list files* choose that the value is **From** or **To** the component — or that it shall be fetched from the **Line article data** for the line.

This can for instance be used for mounting lists, specifying where to mount wires.

Wire number files

It is also possible to create wire number files.

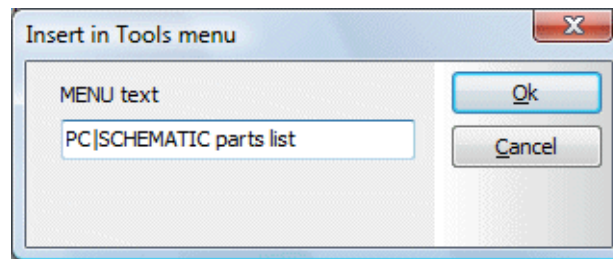
— See “Wire number Files” on page 401.



CHOOSING LIST FILE SETTINGS FROM THE TOOLS MENU

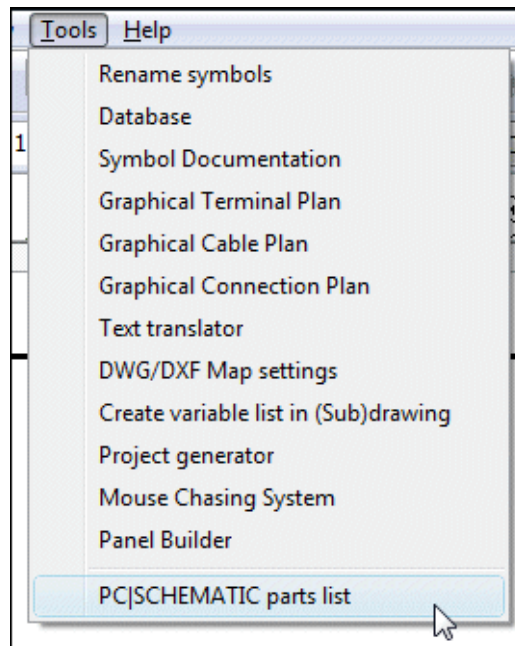
To export list files directly from the **Tools** menu, do this:

- 1) Choose **Lists => Parts List File** (or another type of list file).
- 2) In the dialog box **Parts List File**: Specify the desired settings — as explained in “Exporting List Files” on page 649.
- 3) Save the format file — as explained in “The format file” on page 650 and “The buttons in the right-hand side of the dialog box” on page 653.
- 4) Click on the button **Insert in Tools menu** at the upper part of the dialog box:



- 5) In the dialog box **Insert in Tools menu**: Type in the desired menu text, and click **OK**.

A list with the specified settings can now be created directly from the **Tools** menu:



When you select **PC|SCHEMATIC parts list**, you enter the dialog box **Parts List File**, where the settings are exactly as you have specified.

Just click **OK**, and the list is created.

DWG AND DXF FILES

In **Automation** it is possible to import DWG and DXF files, and continue working from these files in the program. It is also possible to export DWG and DXF files.

THIS CHAPTER DESCRIBES	
— THE ZOOM ALL FUNCTION.....	P. 660
— IMPORTING DWG/DXF FILES.....	P. 660
— IMPORTING A DWG/DXF FILE AS A PAGE IN A PROJECT	P. 661
— STARTING A NEW PROJECT BY IMPORTING A DWG/DXF FILE	P. 662
— IMPORTING A DWG/DXF FILE AS A SYMBOL	P. 663
— EXPORTING DWG/DXF FILES.....	P. 663
— EXPORTING A PROJECT AS A DWG/DXF FILE	P. 663
— EXPORTING A SYMBOL AS A DWG/DXF FILE.....	P. 665
— WHICH FUNCTIONS ARE TRANSFERRED?	P. 665

If you are working in layers, the layer structure will be maintained during import as well as export. — Read more about layers in “Layers, Elevation and Scaling” on page 477.

If you wish to change the parameters for the DXF import or export, please refer to the **PC|SCHEMATIC Tools** manual.

THE ZOOM ALL FUNCTION DURING DWG/DXF IMPORT

When working with imported DWG/DXF files, it might happen that objects are placed outside the normal drawing area.

During the import to **Automation**, all imported objects are placed inside the project pages in **Automation**, and the page sizes and page scales are adapted to make this possible.

If there, in the DWG/DXF file, are placed objects outside the normal drawing area, this can therefore result in, that the drawing is placed in e.g. the lower-left corner of the page. Consequently, it will not fill out much of the page in the created **Automation** project. If you subsequently change the size of the page and/or the scale of the page — in order to be able to see the drawing — this might result in that objects are placed outside the page in **Automation**.

When you wish to print such a page, this gives problems because the printing presses all objects in the work area to be placed on the page. The drawing can hereby e.g. end up in the lower-left corner of the page.

To avoid this, choose **View => Zoom All**, which displays all objects in the work area. Hereby hidden objects with negative or huge positive coordinates can be displayed — and subsequently deleted. — See “The Zoom All function” on page 43.

IMPORTING DWG/DXF FILES

There are three different ways of importing a DWG/DXF file into **Automation**:

- a) You can import a DWG/DXF file in an active project by importing it from the **Page menu** — see “Importing a DWG/DXF file as a page in a project” on page 661.
- b) You can start up a new project by importing a DWG/DXF file — see “Starting new projects by importing DWG/DXF files” on page 662.
- c) You can import a DWG/DXF file as a symbol, and then continue working with the symbol like any ordinary symbol in **Automation** — see “Importing DWG/DXF files as symbols” on page 663.

The following sections in this chapter elaborates on these options.



ADAPT DWG/DXF FILES BEFORE IMPORTING

PLEASE NOTE, THAT IT IS A GOOD IDEA TO REDUCE THE DWG/DXF FILE BEFORE IMPORTING IT, SO THAT IT ONLY CONTAINS EXACTLY WHAT YOU NEED IN YOUR PROJECT.

IMPORTING A DWG/DXF FILE AS A PAGE IN A PROJECT

If you are working in a project, and want to insert a DWG/DXF file, you first click on the **Page menu** button.



In the **Page menu** you then click where you want the page to be inserted as a new page, and click on **Insert new**. Or click on **Add new** to have it inserted as the last page in the project. — Read more about the **Page menu** in “The Page menu” on page 493.

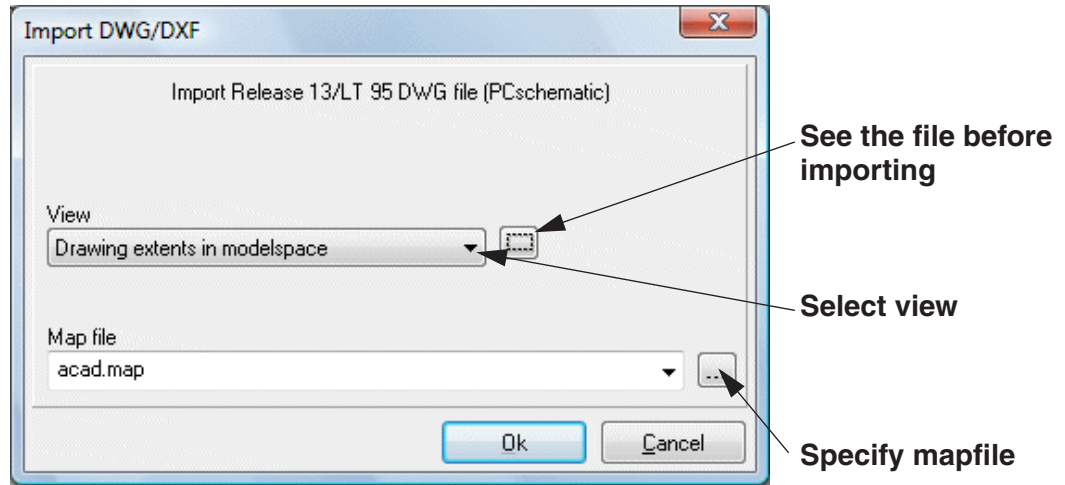
You then enter the dialog box **Page function**, where you specify which type of page you want to insert: Click on **Standard/Project file**, and then click **OK**.

Opening DWG/DXF files

In the dialog box **Open Standard** which now appear, you click your way to the folder where the DWG/DXF file is located, choose **Acad files (*.dwg)** or **DXF files (*.dxf)** as file type, click on the DWG/DXF file, and then on **Open**.

Select map file and specify View

You then enter a dialog box, where you specify which map file to use during the import:



You can also view the file in a separate window before importing, by clicking the button pointed out next to the **View** field in the figure above.

In the **View** field you specify which view you wish to import from the DWG/DXF file, if a viewpoint is defined in the file. When you select **Drawing extents in model-space**, the whole file is imported.

Click **OK**.

Opening more DWG/DXF files

You have now inserted the DWG/DXF file, but are still in the **Page menu**. If you want to insert more DWG/DXF files, you repeat this procedure for each DWG/DXF file.

When you have finished inserting DWG/DXF files, click **OK** to leave the **Page menu**.

STARTING NEW PROJECTS BY IMPORTING DWG/DXF FILES



Clicking on the **Open** button you enter the dialog box **Open**, from which you can import a DWG/DXF file as described in "Opening DWG/DXF files" above.

Hereby you create a project consisting of one page, on which the DWG/DXF file is located. Using this as a basis, you can now build up a new project.

IMPORTING DWG/DXF FILES AS SYMBOLS

It is also an option to import a DWG/DXF file as a symbol. This has the advantage that you can move the symbol in the project as you like, and at the same time you can not change the symbol by accident.

If you want to import a DWG/DXF file as a symbol, you do like this:



Click on the **Symbols** button, and then on the **Symbol menu** button, which brings you into the **Symbol menu**.

Here you click on the **Create new symbol** button, which brings you into the **Symbol editor**. Here you click on the **Open symbol** button, which leads you to the dialog box **Open**, where you choose **Acad files (*.dwg)** or **DXF files (*.dxf)** as file type.

Find the next DWG/DXF file, and click **OK**.

Returning to the **Symbol menu**, you see the DWG/DXF file displayed in the lower-right corner of the menu.

Click on **OK**, which gives you the DWG/DXF symbol in the cross hairs as a symbol. Click where you want to place the symbol.

The program automatically place the reference point for the symbol in the lower-left corner, where all symbol texts are placed as well. The symbol is a normal symbol, with selection in the reference point. — Read more about this in “Symbol types” on page 706.



If you wish to change the imported symbol — or save it in a symbol folder — you can click on the **Edit symbol** button when you are in the **Symbol menu**. Here you can edit and save as for a normal symbol.

— Read more about working with symbols in “Symbols” on page 137, and about how to edit symbols in “Creating Symbols” on page 696.

EXPORTING DWG/DXF FILES

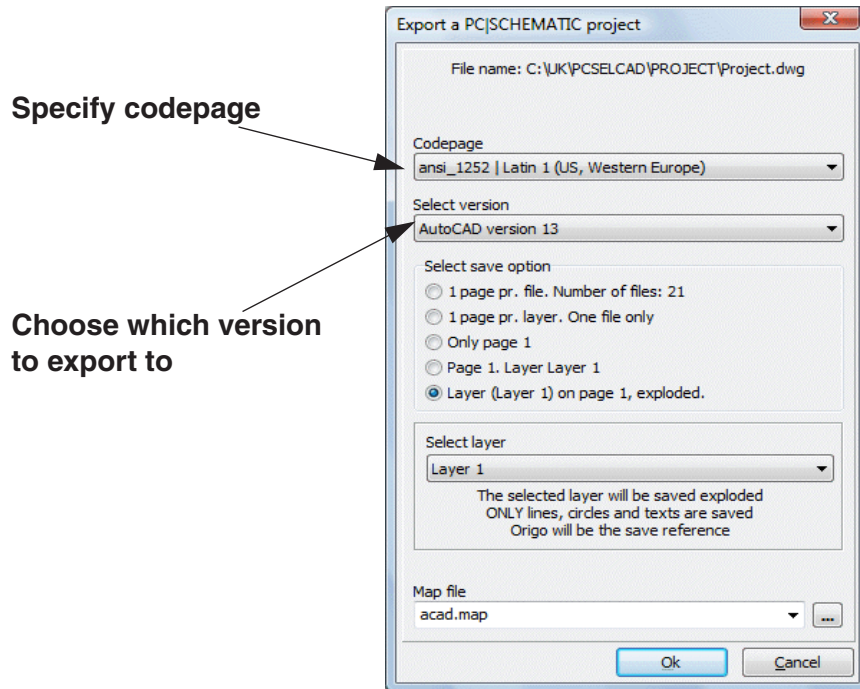
Both projects and symbols can be exported as DWG/DXF files.

EXPORTING PROJECTS AS DWG/DXF FILES

When you wish to export a project as a DWG or a DXF file, just choose **File => Export to => DWG** or **File => Export to => DXF**, specify a file name and click **Save**.

You can also choose **File => Save as**, set the file type to **Acad files (*.dwg)** or **DXF files (*.dxf)**, specify a file name, and click on **Save**.

When you save the project as a DWG/DXF file, you enter the following menu:



APPLY THE AUTOCAD VERSION 13

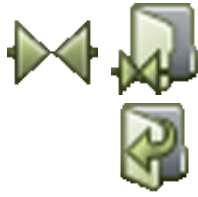
BECAUSE **Automation** DOES NOT USE AUTOCAD FUNCTIONS FROM LATER VERSIONS THAN AUTOCAD VERSION 13, IT IS RECOMMENDED TO EXPORT TO AUTOCAD VERSION 13.

THIS ALSO MAKES IT POSSIBLE TO OPEN THE FILE ON MORE PCs.

Apart from choosing which version to export to, you have the following options:

OPTION	RESULTS IN
CODEPAGE	THE PROGRAM WILL AUTOMATICALLY APPLY THE CODEPAGE, WHICH THE PC IS SET TO USE. IF YOU WISH TO APPLY ANOTHER CODEPAGE FOR THE EXPORT, SELECT THIS CODEPAGE IN THE CODEPAGE FIELD IN THE DIALOG BOX.
1 PAGE PR. FILE	EACH PAGE IN THE PROJECT IS EXPORTED AS A SEPARATE FILE.
1 PAGE PR. LAYER	EACH PAGE IN THE PROJECT IS SAVED IN A SEPARATE LAYER. THIS WAY THE ENTIRE PROJECT CAN BE SAVED IN THE SAME FILE. IF THE Automation PROJECT CONTAINS LAYERS, THE DIVISION IN LAYERS NO LONGER EXISTS.
ONLY PAGE F1	ONLY THE ACTIVE PAGE IN THE PROJECT IS EXPORTED.
PAGE 1. LAYER LAY 1	ONLY THE SELECTED LAYER ON THE ACTIVE PAGE IN THE PROJECT IS EXPORTED.
LAYER (LAY 1) ON PAGE F1 EXPLODED	WHEN YOU CHOOSE THIS OPTION, YOU CHOOSE WHICH LAYER TO EXPLODE, AND THEN EXPORT. OUTPUT CAN FOR INSTANCE BE APPLIED FOR A NUMERICALLY CONTROLLED MACHINE.

EXPORTING A SYMBOL AS A DWG/DXF FILE



Saving a symbol as a DWG/DXF file is also possible. You do this by first clicking on the **Symbols** button and then on the **Symbol menu** button, which brings you into the **Symbol menu**. — Or press the short-cut <F8> to enter the menu.

Here you find the symbol you want to save as a DWG/DXF file, click on it and click on the **Edit symbol** button. You are now in the Edit symbol mode.

Then you choose **File => Save as**, click on **OK** in the first dialog box, set the file type to **Acad files (*.dwg)** or **DXF files (*.dxf)** and type in the name in the following dialog box, and click **OK**.

Then choose **File => Close** to exit the Edit symbol mode. You now return to the **Symbol menu**, where you click on **Cancel** in order to return to the project.

Read more about editing and saving symbols in “Creating Symbols” on page 696.

WHICH FUNCTIONS ARE TRANSFERRED FOR DWG AND DXF FILES

Functions which are common for the program, you are importing from, and **Automation** will be transferred.

When importing objects which **Automation** cannot handle, it is recommended to “explode” the drawing before importing it. **Automation** cannot handle coordinates — for instance line lengths — on more than two kilometre.

During export of objects which the DWG/DXF format can not handle, **Automation** tries to use object types known to the DWG/DXF format.

Electrical properties *cannot* be transferred using the DWG/DXF format.

— Please refer to the **PC|SCHEMATIC Tools** manual.

If you are having difficulties importing DWG/DXF files, please contact your local support service.

COMMAND FILES FOR CONVERTING PROJECTS TO DWG/DXF FORMAT

It is possible to create command files, which automatically saves **Automation** projects as DWG or DXF files. Read more about this in “Command files for saving projects as PDF, DWG and DXF files” on page 822.

INTEGRATION WITH MICROSOFT EXCEL

THIS CHAPTER DESCRIBES

THIS CHAPTER IS A SURVEY OF THE VARIOUS WAYS YOU CAN USE MICROSOFT EXCEL TOGETHER WITH **Automation**.

In **Automation** you can work together with Microsoft Excel in the following ways:

- 1) Automatic project generation via Excel - see “Generating and Updating projects via Excel” on page 614.
- 2) All lists to files can be saved in Excel format. This means all reports can be saved in Excel format — see “Lists and Excel” on page 651.
- 3) PLC I/O data imported in Excel, where changes can be made, after which these changes can be imported in **Automation** — see “PLC I/O addressing via Excel” on page 435.
- 4) You can save data from the **Object Lister** in Excel format — see “The Object Lister” on page 306.
- 5) You can draw projects based on Parts/components lists from Excel, whereby **Automation** automatically fetches the applied components from the attached database — see “Placing symbols via lists” on page 674.
- 6) Furthermore, you can merge **Automation** projects via Excel — see “Merging projects via Excel” on page 625.



EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT

IT IS ONLY POSSIBLE TO SAVE AND OPEN FILES IN EXCEL FORMAT IF MICROSOFT EXCEL IS INSTALLED ON THE PC.

THIS IS BECAUSE **Automation** LETS EXCEL HANDLE THESE PROCEDURES. YOU ARE THEREFORE NOT DEPENDANT ON WHETHER **Automation** CAN HANDLE WHICHEVER VERSION OF EXCEL YOU HAVE INSTALLED.

INSERTING FILES FROM OTHER PROGRAMS / OLE

In **Automation** you can insert so-called OLE objects directly on the project pages. This means that you can work with files from other computer programs inside **Automation**.

THIS CHAPTER DESCRIBES	
— WHAT IS AN OLE OBJECT?	P. 667
— CREATING NEW OLE OBJECTS	P. 668
— INSERTING EXISTING OLE OBJECTS	P. 669
— COPYING AREAS AND INSERTING AS OLE OBJECTS	P. 670
— EDITING PLACED OLE OBJECTS	P. 670
— OPENING OTHER COMPUTER PROGRAMS VIA OLE OBJECTS	P. 671
— UPDATING PLACED OLE OBJECTS	P. 672
— INSERTING AUTOCAD PROJECTS AS OLE OBJECTS	P. 672
— HANDLING OLE OBJECTS IN OTHER COMPUTER PROGRAMS	P. 672
— PROPERTIES ON OLE OBJECTS	P. 672
— UPDATE ALL OLE OBJECTS AND OTHER LINKS	P. 673

WHAT IS AN OLE OBJECT?

OLE is a standard for communication between computer programs, which makes it possible to be in one computer program, and from there work on a document in another computer program. (OLE is an abbreviation of "Object Linking and Embedding").

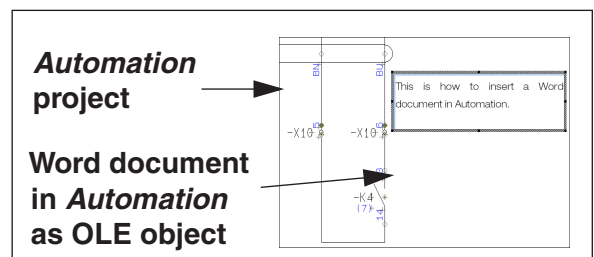
To make this possible, both computer programs have to be installed on the computer, and both computer programs have to support OLE objects.

Because **Automation** supports OLE objects, you can therefore:

- 1) Create *new* objects in other computer programs directly from a **Automation** project, and keep on working with the documents from **Automation**
- 2) Insert *existing* documents from other computer programs in **Automation**, and work on the documents from **Automation**

Thus you can insert Word or Excel documents directly on the pages in **Automation**, and keep working on them from **Automation** as if you were working in Word or Excel respectively.

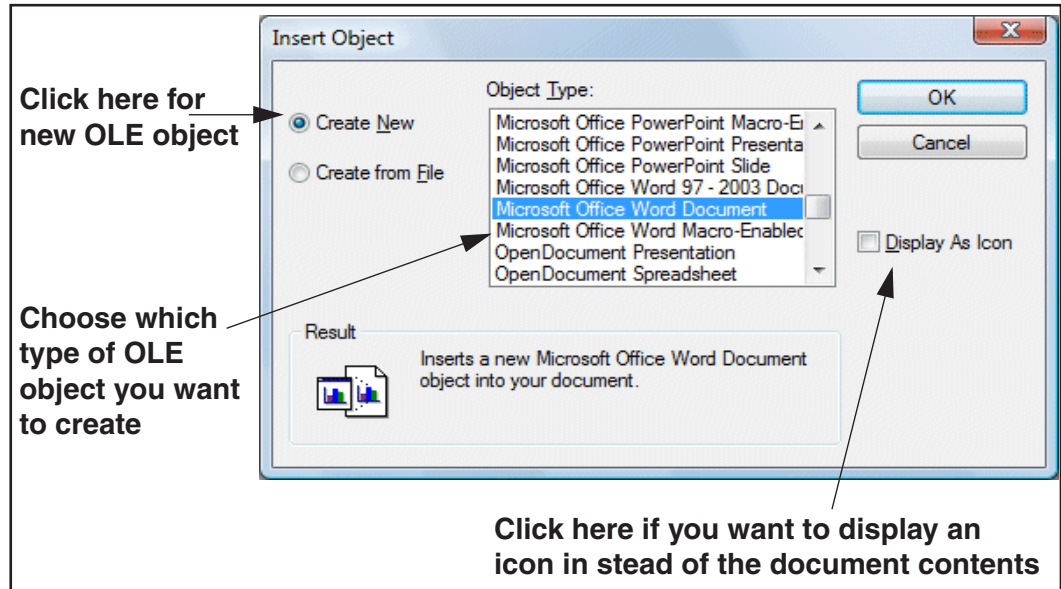
Inserting **Automation** projects as OLE objects in other computer programs is not possible.



CREATING NEW OLE OBJECTS IN Automation

To create a new OLE object — for instance a new Word document — choose **Insert => Insert object**. — If you want to insert an *existing document*, please refer to “Inserting existing objects in Automation” on page 669.

You now enter the following dialog box:



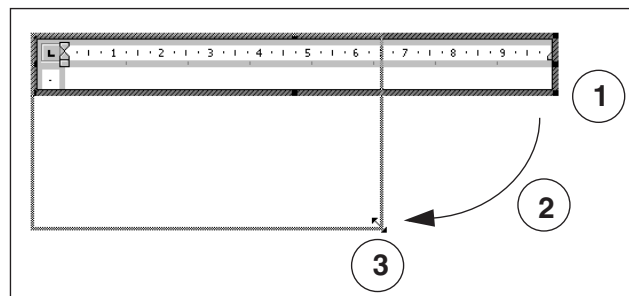
Here you click on **Create new object**, and choose which type of OLE object/document/file you want to insert in the project. The options you get on the list are those of the installed programs on your computer, which supports inserting OLE objects in other programs — for instance the Microsoft Office package.

Placing the OLE object

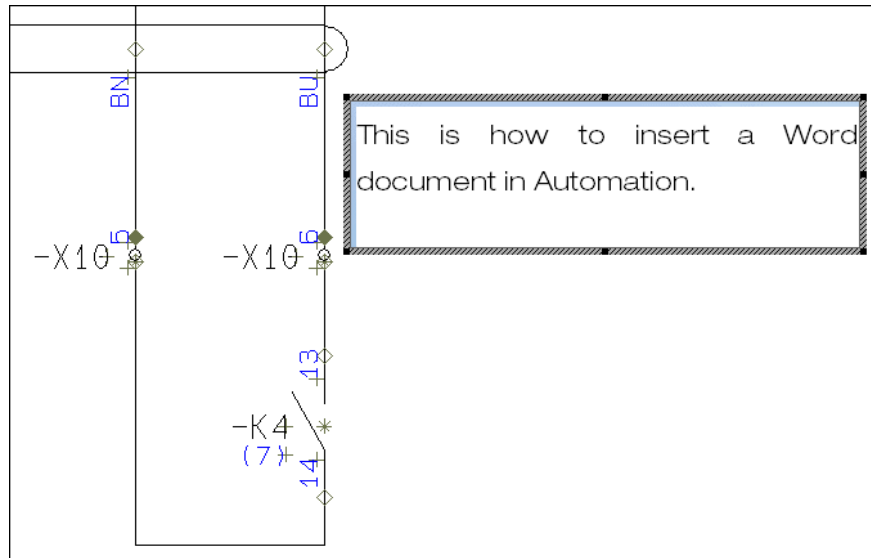
Click on **OK** when you have chosen which type of document you want to create. You are then asked to point out where on the page to place the OLE object.

Having placed the OLE object, you can change its size like this:

- 1) Place the mouse pointer over one of the corners of the OLE object, so that the cursor changes appearance to a two-way arrow
- 2) Press down the left mouse button, and *drag* the corner to another position
- 3) Release the mouse button



When an OLE object is placed — for instance a Word document — the toolbars in **Automation** are replaced with the toolbars from Word.



You can now work with the full functionality on Word inside **Automation**.

When you have finished working with the word document, you just click outside the area containing the word document. Doing this causes the **Automation** toolbars to reappear, and you can continue working in **Automation**.

The text from the Word document now appears to be an ordinary text in **Automation**.

INSERTING EXISTING OBJECTS IN Automation

When you want to insert existing documents in **Automation** — for instance an existing Word document — choose **Insert => Insert object**. — If you are interested in *creating a new file*, please refer to “Creating new OLE objects in Automation” on page 668.

You now enter the following dialog box:

Click here to insert an existing object

Specify where the object/document is placed

Insert Object

Create New

Create from File:

File: C:\UK\PCSELCAD\PROJECT\

Link

Display As Icon

Result

Inserts the contents of the file as an object into your document so that you may activate it using the program which created it.

Specify whether to create a copy, or whether you want to make changes directly in the original

Here you choose **Create from file**, and find the document you want to place in the project, by either clicking on **Browse**, or by typing in the file name in the **File** field (including the *path* — see “Dictionary” on page 873 — leading to the file).

Linking

If you choose **Link** you will enter the program in which the document is created — for instance Word — when you open the document from **Automation**. From there you make changes in the original document, and return to **Automation**.



PLEASE NOTE THAT WHEN YOU **LINK** A WORD DOCUMENT TO **Automation**, YOU CAN STILL EDIT THE DOCUMENT DIRECTLY FROM WORD WITHOUT INVOLVING **Automation**.

BECAUSE THE DOCUMENT IS **LINKED** TO THE **Automation** PROJECT, THE CHANGES IN THE DOCUMENT WILL ALSO BE DISPLAYED IN **Automation**.

When you do *not* choose **Link**, a *copy* of the document will be created in **Automation**. This means that two independent documents now exists. Changes in the one document will not be transferred to the other — and vice versa.

Click **OK**.

Placing the OLE object

You are now asked to place the OLE object, as described in “Placing the OLE object” on page 668.

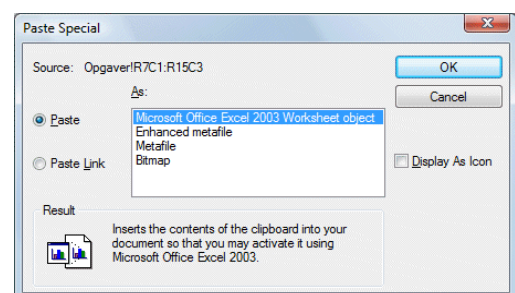
COPYING AREAS AND INSERTING THEM AS OLE OBJECTS

When you copy an area — for instance from Excel — by selecting the area and pressing <Ctrl+c>, you can insert the area in **Automation** by choosing **Edit => Paste special**.

You now get the option of choosing between **Insert** and **Insert link**.

When you choose **Insert link** changes will be transferred automatically between the project and the original file — as described in “Linking” on page 670.

When you choose **Insert** only a *copy* of the copied area will be inserted, without connection to the original file.



EDITING A PLACED OLE OBJECT

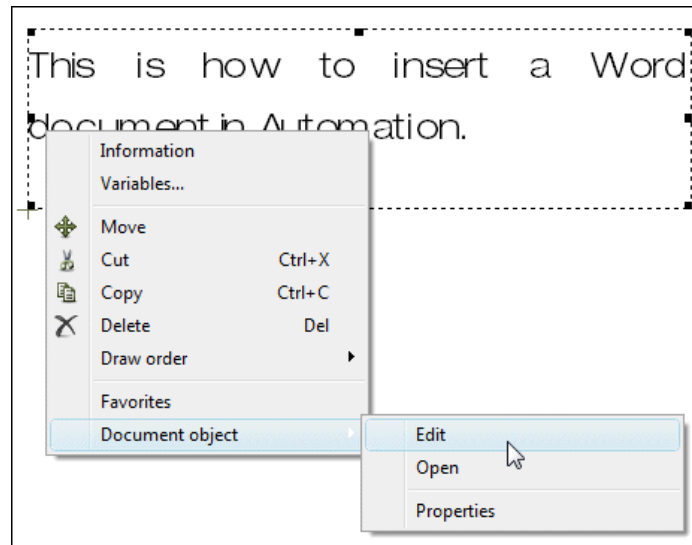
When you want to edit a placed OLE object, do like this:



1) Click the **Area** button

2) Double-click inside the area covered by the OLE object — to select the OLE object

3) Right-click on the OLE object and choose **Document object => Edit**



You have now opened the OLE object, and can edit its contents.

When you click outside the area of the OLE object, you return to working in **Automation**. The changes in the OLE object are saved automatically.



ENABLE DISPLAYING REFERENCE POINTS

IT CAN BE DIFFICULT TO SEE HOW MUCH SPACE A NON-SELECTED OLE OBJECT TAKES UP IF YOU HAVE *DISABLED REFERENCEPOINTS* IN **SETTINGS => POINTER/SCREEN**.

— PLEASE REFER TO “POINTER/SCREEN” ON PAGE 836.

OPENING OTHER PROGRAMS FROM OLE OBJECTS

When you want to open on OLE object in **Automation**, so that the computer program in which the OLE object is created fills the screen, do like this:



- 1) Click the **Area** button
- 2) Double-click inside the area covered by the OLE object to select the OLE object
- 3) Right-click on the OLE-object and choose **Document object => Open**

If for instance you have opened an Excel document, the Excel program starts, and the size of the document in Excel matches the size in has in **Automation**.

Returning to Automation

To return to **Automation**, do like this:

- 1) Choose **File => Close and return to Automation project**
- 2) Click on the **PC|SCHEMATIC Automation** button on the process line at the bottom of the screen

You have now returned to **Automation**.

UPDATING OLE OBJECTS

When the contents of an OLE object have been updated outside the control of **Automation**, you can transfer the changes to **Automation**:



- 1) Click the **Area** button
- 2) Double-click within the area covered by the OLE object to select the OLE object
- 3) Right-click on the OLE-object and choose **Document object => Update**

You have now updated the OLE object copy in **Automation**. — However, these changes will for most programs be transferred to **Automation** automatically.

INSERTING AUTOCAD DOCUMENT AS OLE OBJECT

When inserting OLE objects in **Automation**, you have the option of drawing on top of the OLE object.

When inserting a ground plan drawing, you can therefore choose to draw for instance the electrical installations on top of the AutoCAD file. This way you *only* have the electrical installations in **Automation**, and *only* have the ground plan drawing in AutoCAD — *without* needing to convert any files.

When the AutoCAD file is placed in **Automation with Link** — see “Inserting existing objects in Automation” on page 669 — the changes on the ground plan drawing will be transferred automatically from AutoCAD to **Automation**.



AUTOCAD MUST BE INSTALLED

TO INSERT AUTOCAD FILES AS OLE OBJECTS IN **Automation** YOU MUST HAVE AUTOCAD INSTALLED ON YOUR COMPUTER.

ALTERNATIVELY YOU CAN IMPORT AUTOCAD FILES IN **Automation** AS EITHER DWG OR DXF FILES. — PLEASE REFER TO “DWG AND DXF FILES” ON PAGE 660.

HANDLING OLE OBJECTS IN OTHER COMPUTER PROGRAMS

The handling of OLE objects from other computer programs, can differ from what is described above. This is because there can be a difference in how these computer programs have chosen to support OLE.

PROPERTIES ON OLE OBJECTS

It is possible to specify properties on OLE objects. This is done like this:



- 1) Click on the **Area** button.
- 2) Double-click on the OLE object.
- 3) Right-click on the OLE object, and choose **Document object => Properties**.

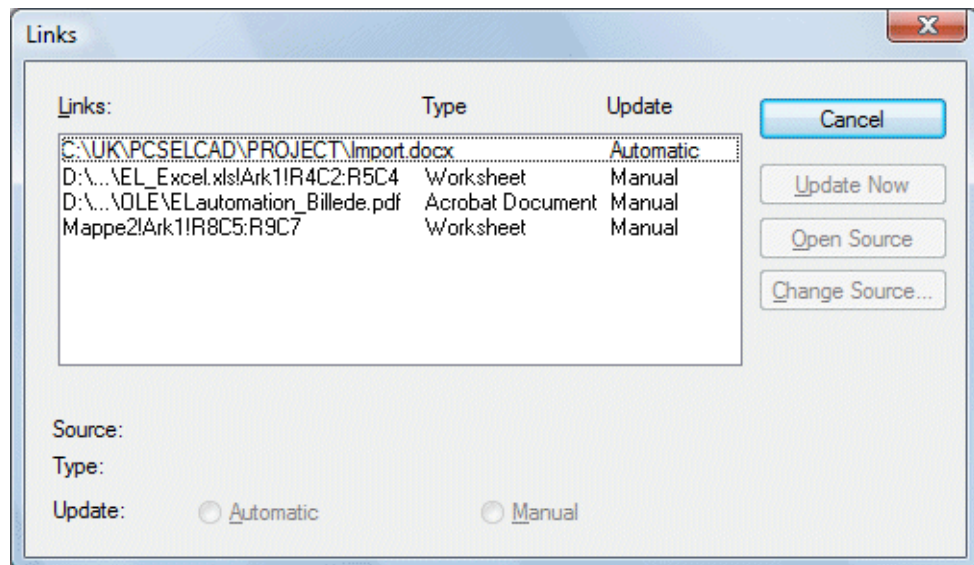
4) You can now specify properties on the OLE object.

When using linked OLE objects you can hereby change the link to another file. Furthermore you can change the size of the OLE object under **Properties**.

UPDATE OLE OBJECTS AND OTHER LINKS

When a project contains OLE objects, which have been inserted via **Insert => Insert object**, where at least one object have been **linked** to the project, you can choose **Edit => Links**.

Hereby you enter the dialog box **Links**:



If the contents of one of these OLE objects have changed, then select the object and click on **Update Now**, so that the object is updated in the project also. Please note that the OLE objects in **Automation** are only updated when you select this manually.

From the dialog box you can also open a source file — click on **Open Source** — or select another source for the OLE object — click on **Change Source**.

From this dialog box you can also handle linked pictures inserted via **Insert => Insert picture**.

PLACING SYMBOLS VIA LISTS

THIS CHAPTER DESCRIBES	
— ON PLACING SYMBOLS VIA LISTS	P. 674
— READING COMPONENTS LISTS/PARTS LISTS	P. 674
— LOADING COMPONENTS FROM A GROUND PLAN PAGE	P. 677
— LOADING COMPONENTS FROM A DIAGRAM PAGE	P. 677
— LOADING COMPONENTS / PARTS LISTS FROM MULTIPLE WINDOWS	P. 677
— UPDATING LOADED COMPONENTS / PARTS LISTS.....	P. 677
— INTERRUPTED PLACING OF COMPONENTS	P. 677
— OTHER OPTIONS WHEN IMPORTING XML LIST FILES	P. 678

ON PLACING SYMBOLS VIA LISTS

In **Automation** you can read parts and components list files containing article numbers, and use these files to place symbols in projects.

These lists will typically consist of parts for a machine, and could for instance be a components list containing sensors, motors etc.

For hydraulics/pneumatics there will typically be an extra column, where the machine responsible informs the electrical constructor which sensors to use, and how to use them.

It is also possible to change project data when reading parts list and component-list files. You can also specify where to save the **Automation** project, and where to save PDF files generated from the project — see “Other options when importing XML list files” on page 678.



POSSIBLE LIST FORMATS

THE INFORMATION IN THE LIST FILE MUST BE THE FOLLOWING: NAME, ARTICLE NO., TYPE AND FUNCTION.

THE LIST CAN BOTH BE AN EXCEL FILE AND A COMMASEPARATED FILE — FOR INSTANCE CREATED FROM A DATABASE SYSTEM.

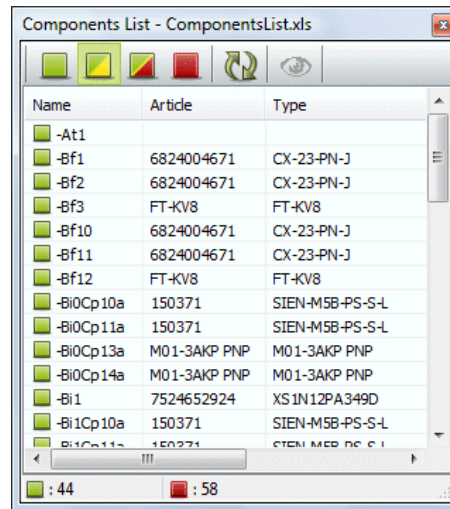
HOWEVER, IT IS ONLY POSSIBLE TO OPEN LISTS IN THE EXCEL FORMAT IF EXCEL IS INSTALLED ON THE PC. — SEE “EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT” ON PAGE 666.

READING COMPONENTS LISTS/PARTS LISTS

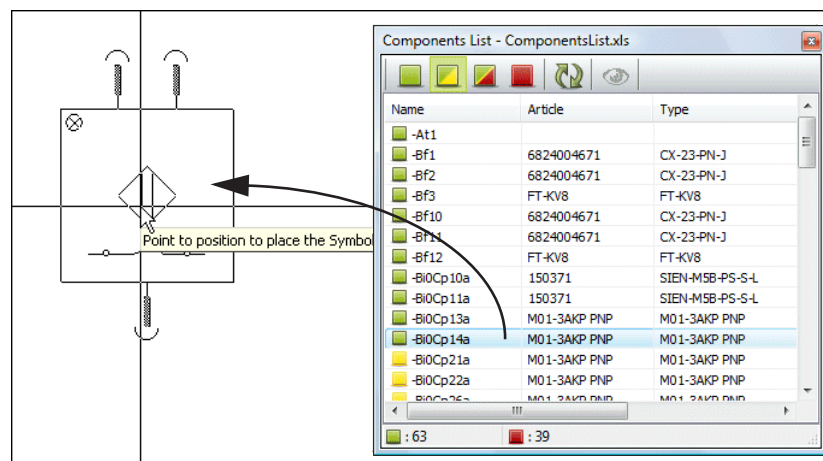
- 1) Choose **Lists => Read Components List => Load from file.**
- 2) In the dialog box **Select Components List** you select the folder where the list is placed, specify whether it is an Excel file (file type ***.xls**) or a comma separated file (file type ***.csv**), and double-click on the file.

Placing the components from the Components List window

3) A survey of the components is now displayed in **Automation**:



4) Double-click on a component in the window, and you get the component in the cross hairs. When a component contains more than one function, a symbol toolbar is displayed containing the functions for the component.



5) Place the symbol, and the components list from the file is updated in **Automation**.

6) Continue until all components are placed in the project.

The buttons in the window



When you click on the **Show not-placed** button, all components, for which all symbols are not placed, are displayed.

When you click on the **Show partly-placed** button, all components, for which all symbols are not placed — or just partly placed —, are displayed.

When you click on the **Show all** button, all placed as well as not placed components are displayed.

When you click on the **Show completely-placed** button, only the completely placed components are displayed.

When you click on the **Update list** button, the list is updated.



THE COMPONENT LIST WINDOW REMAINS VISIBLE

THE **COMPONENT LIST** WINDOW REMAINS VISIBLE WHILE WORKING WITH THE PROJECT, AND CAN BE SET TO BECOME TRANSPARENT, WHILE YOU ARE WORKING ON THE DRAWING AREA — SEE “**POINTER/SCREEN**” ON PAGE 836.

Show where the component is placed



When you wish to see where a component is placed in the project, click on the component in the list, and click the **Show where the component is placed** button.

The **Select name** window is now displayed for the component. When you click on one of the symbols in the window, the program jumps to the page in the project where the symbol is placed, and selects the symbol.

The screenshot shows a 'Components List - ComponentsList.xls' window with a table of components. The component '-Bi1Cp21a' is selected. An eye icon in the window's toolbar is also highlighted. Below it, a 'Select Name' window is open, showing a table with the selected component. Arrows indicate the sequence of actions: 1) clicking the component in the list, 2) clicking the eye icon, 3) clicking the component in the 'Select Name' window, and 4) the component being selected in the project drawing area.

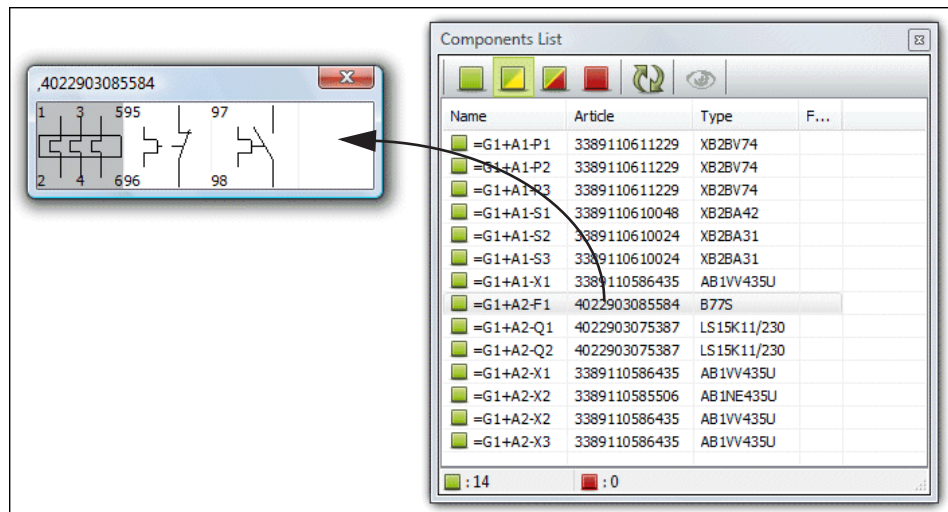
Name	Article	Type
-Bi1	7524652924	XS IN12PA349D
-Bi1Cp10a	150371	SIEN-M5B-PS-S-L
-Bi1Cp11a	150371	SIEN-M5B-PS-S-L
-Bi1Cp13a	M01-3AKP PNP	M01-3AKP PNP
-Bi1Cp14a	M01-3AKP PNP	M01-3AKP PNP
-Bi1Cp21a	M01-3AKP PNP	M01-3AKP PNP
-Bi1Cp22a	M01-3AKP PNP	M01-3AKP PNP
-Bi1Cp26a	M01-3AKP PNP	M01-3AKP PNP
-Bi1Cp27a	M01-3AKP PNP	M01-3AKP PNP
-Bi2	7524652924	XS IN12PA349D
-Bi3	7524652924	XS IN12PA349D
-Bi4	7524652924	XS IN12PA349D
-Bi5	516-119-S4-C	KIB-M12PS 002-KL3P
-Bi6	516-119-S4-C	KIB-M12PS 002-KL3P

Name	Conns	Symbol type	Page
-Bi1Cp21a	1,3...	Normal	24

- 1) Click on the component in the list
- 2) Click on the Eye
- 3) Click on the symbol in the Select name window
- 4) The symbol is selected in the project

LOADING COMPONENTS FROM A GROUND PLAN PAGE

If you wish to start by drawing the mechanical layout, you can do this, and subsequently fetch the electrical symbols for the applied components by choosing **Lists => Read Components List => Create from mechanical page**:



When you click on the component in the list, you get a pickmenu with the available electrical symbols for the component.

LOADING COMPONENTS FROM A DIAGRAM PAGE

You can also load a componentslist from a diagram page, by choosing **Lists => Read Components List => Create from diagram page**.

LOADING COMPONENTS/PARTS LISTS FROM MULTIPLE WINDOWS

The functions **Lists => Read Components List** and **Lists => Read Parts List** can open more than one window/list at the same time.

Hereby you can get access to symbols from different lists at the same time, or load the same list in different windows, and select components via different views of the same list.

UPDATING LOADED COMPONENTS/PARTS LISTS



The functions **Lists => Read Components List** and **Lists => Read Parts List** registers whether the loaded file has been changed, and displays a **Reload** button, which can reload the list file.

The button flashes red/black, and the list file is reloaded when you click the button, and subsequently answer **OK** to reload the file.

INTERRUPTED PLACING OF COMPONENTS FROM THE PARTS LIST

When placing components from the list is interrupted, and for instance continued the next day, **Automation** examines what has already been placed, and you can continue working on this basis.

OTHER OPTIONS WHEN IMPORTING XML LIST FILES

Beyond placing symbols when importing parts list and components list files, you have the following options:

Importing Project data from XML files

When you import a parts list or components list in XML format, you can transfer project data to the **Automation** project at the same time. The specified project data are hereby updated in the project, or created, if they do not already exist.

Projectdata are specified like this in the XML file:

```
- <ProjectProperties>  
  <PROPERTY name="Customer name" Value="PC|SCHEMATIC A/S" />  
  <PROPERTY name="Address" Value="Bygaden 7a" />  
  <PROPERTY name="Zip code" Value="DK4040" />  
  <PROPERTY name="City" Value="Jyllinge" />  
  <PROPERTY name="Phone" Value="+45 4678 8244" />  
</ProjectProperties>
```

Specify where to save the project and PDF files

When you import a parts list or components list in XML format, you can — in the XML file — specify in which folder to save the **Automation** project subsequently. Correspondingly, you can also specify in which folder to save PDF files generated from the project.

This type of information is specified in the XML file like this:

```
- <PATHSETUP>  
  <path name="PDF" value="C:\PCsElCad\Project1\PDF" />  
  <path name="Project" value="C:\PCsElCad\Project1\Project" />  
</PATHSETUP>
```

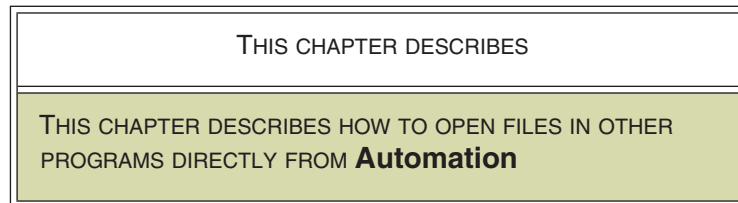
This updates the information specified under **Settings => Directories** for **Projects** and for **PDF export**. — Read more about this in “Directories” on page 840.

More XML syntax

You can read more about the syntax for XML files in “Lists in XML format” on page 652.

OPENING FILES FROM THE TOOLS MENU

Opening files in other programs directly from **Automation** is possible. This can for instance be used as a short-cut when you want to see the lists to files, which you create from **Automation**.



The #OPENFILE# command

In the dialog box **Settings => Tools**, the command #OPENFILE# can now be applied. It is used when you want to open a file in another Windows program from **Automation**.

When the command is used along with a file name, the extension of the file determines which program shall be applied for opening the program. — Please note, that it is Windows that handles which program is used for opening the file.

If it therefore says #OPENFILE#C:\tekst.doc under **Program name (full path)**, the type **doc** determines, that the file should be opened in the program Word.

— Please refer to “Tools” on page 845 for further information on the dialog box **Settings => Tools**, and which other commands can be applied there.

Display lists to files

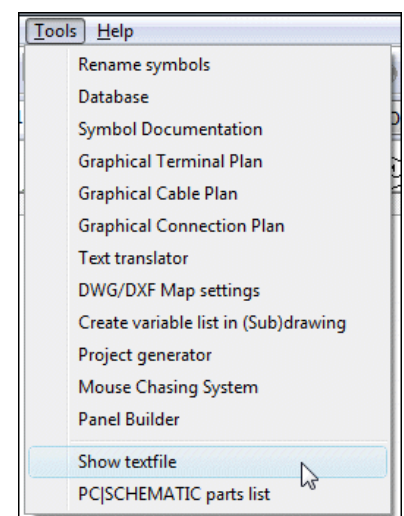
If no file name is specified, or if the file name contains the sign *, you will enter the **Open** dialog box.

This can for instance be used when you are creating lists to files. If you place the following line in the dialog box, you will be able to open the lists for files you create in **Automation** quickly:

Show textfile	#OPENFILE#%SYSDIR%\LIST*,*
---------------	-----------------------------

When you then choose **Tools => Show textfile**, you enter the dialog box **Open**, with the folder **c:\Pcscad\list** opened.

Here you click on the desired file, click **OK**, and get the file opened in the program Notepad.



PC|SCHEMATIC VIEWER

On the **Automation** CD there exists a **PC|SCHEMATIC Viewer**, which you can send to your customers for free. The Viewer can furthermore be downloaded from www.pcschematic.com in the English, German, Dutch, Polish, Czech, Hungarian, Russian, Chinese and Danish version.

Options in the PC|SCHEMATIC Viewer

When you open a project with the **PC|SCHEMATIC Viewer**, you can do the following:

- See all project pages, and zoom in all parts of the project
- See Page data and Project data
- Search for project texts and find objects via the **Object Lister**
- Choose which project layers are visible
- Print

You do *not* have the option to edit projects in the **PC|SCHEMATIC Viewer**.

Installation of PC|SCHEMATIC Viewer

The **PC|SCHEMATIC Viewer** is placed in two different ways on the **Automation** CD:

- As a file for installing **PC|SCHEMATIC Viewer**
- In a folder, from which you can run it directly from the **Automation** CD without installing the **PC|SCHEMATIC Viewer**

Installing the PC|SCHEMATIC Viewer

If you wish to install the English version of the **PC|SCHEMATIC Viewer**, enter the folder **\UKViewer** on the **Automation** CD, and double-click on the file **pcs-viewuk.exe**. This starts the installation of the viewer. During the installation process, you will be asked in which folder on your PC to install the program. Follow the self-explanatory installation procedure until **PC|SCHEMATIC Viewer** is installed.

The program is subsequently started by opening the folder **Pcsview** on your PC, and then double-click on the file **pcsview.exe**.



THE INSTALLATION PROGRAM TO **PC|SCHEMATIC Viewer** CAN BE PLACED ON A SINGLE FLOPPY DISK.

WHEN CREATING SECURITY COPIES OF PROJECTS, THE VIEWER CAN FOR INSTANCE BE PLACED ON THE CREATED CDS.

If you wish to install the German version, use the file **pcsviede.exe**. If you wish to install the Danish version, use the file **pcsviedk.exe**.

Running PC|SCHEMATIC Viewer directly from the CD

If you wish to run the English version of **PC|SCHEMATIC Viewer** directly from the **Automation** CD, enter the folder **\UK\Viewer\Pcsview** and click on the file **Pcs-view.exe**.



IN STEAD OF RUNNING THE INSTALLATION PROGRAM, YOU CAN COPY THE FOLDER \UK\VIEWER\PCSVIEW, AND PLACE IT ON YOUR PC.
THE VIEWER IS STARTED AS EXPLAINED ON THE PREVIOUS PAGE.

PC|SCHEMATIC Viewer IN OTHER LANGUAGES

The **PC|SCHEMATIC Viewer** can be found in other language versions on www.pcschematic.com. Here you can always find the latest versions of the viewer, which can be downloaded free of charge.



Create Symbols, Lists and Projects

This section deals with how to prepare your projects, and how to design the extra symbols and lists you need.

You can read about:

- Preparing Projects p. 685
- Specifying which Datafields to use p. 689
- Designing Symbols p. 696
- Designing Signal symbols..... p. 738
- Designing Drawing Headers p. 742
- Designing Lists p. 752
- Designing Unit Drawings..... p. 767
- Creating Page and Project templates..... p. 771

Reading this section saves you a lot of time in the projects to come.

PLANNING AHEAD

When you have become more familiar with the program, it is a good idea to consider how to prepare your future projects in **Automation** the best. The sooner you do this, the more time you will save — and the more you will benefit from the program facilities.

This chapter gives you a survey of what is worth considering, and contains references to the places where the various topics are elaborated.

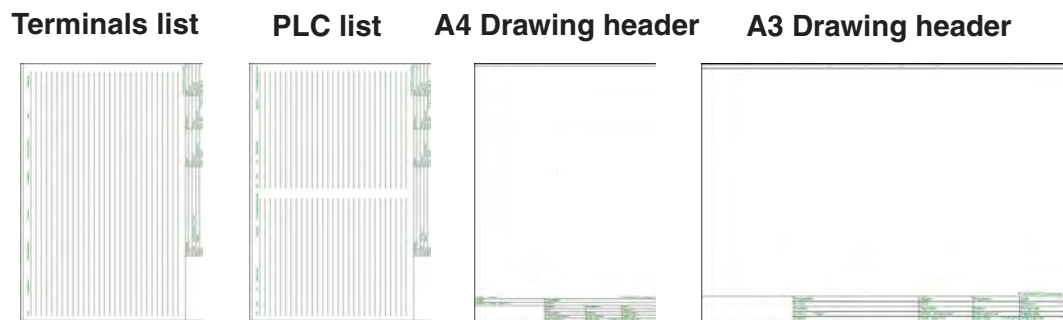
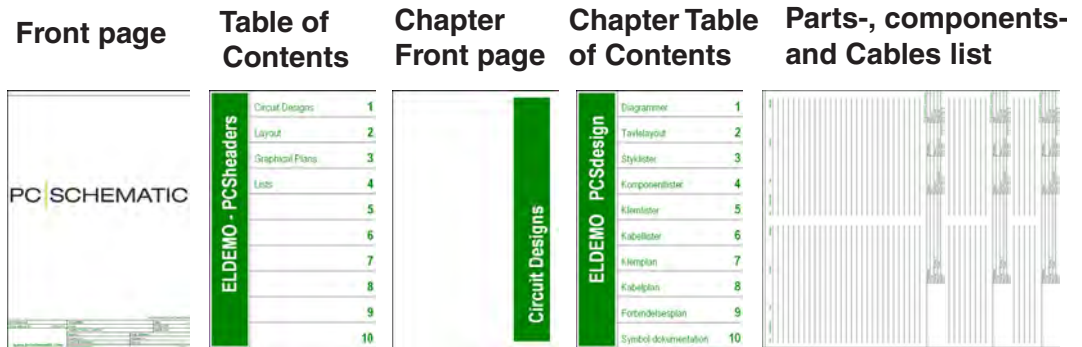
THIS CHAPTER DESCRIBES	
— CREATING DRAWING HEADERS	P. 685
— CREATING PAGE TEMPLATES	P. 686
— CREATING PROJECT TEMPLATES.....	P. 686
— CREATING SUB DRAWINGS.....	P. 687
— PROJECT AND PAGE DATA	P. 687
— CREATING SYMBOLS	P. 688
— FOLDERS CONTAINING PROJECTS.....	P. 688
— OPTIONS FOR MULTIUSER SYSTEMS	P. 688

If you have any difficulties, please contact your local support service.

CREATING DRAWING HEADERS

To adapt the program to your company needs, you will need to design your own drawing headers. These drawing headers are a kind of formulas you place on the pages in your projects, and can be used as front pages, drawing headers for drawings, or formulas for tables of contents as well as several types of lists.

Furthermore, you have got the option to create unit drawings, which are pages on which you specify what a specific component consist of.



You can either create your drawing headers from scratch, or you can create them by adapting the drawing headers, which comes along with the program, to your own needs. Read more about this in “Creating Drawing Headers” on page 742, “Creating Lists” on page 752 and “Changing an already created list/drawing header” on page 763.

If you need to create your own unit drawings, please read “Creating Unit Drawings” on page 767.

CREATING YOUR OWN PAGE TEMPLATES

When you have created these drawing headers, you can succeedingly place them on individual *page templates*, so that you, when creating new project pages, directly can select pages, on which these drawing headers are already inserted.

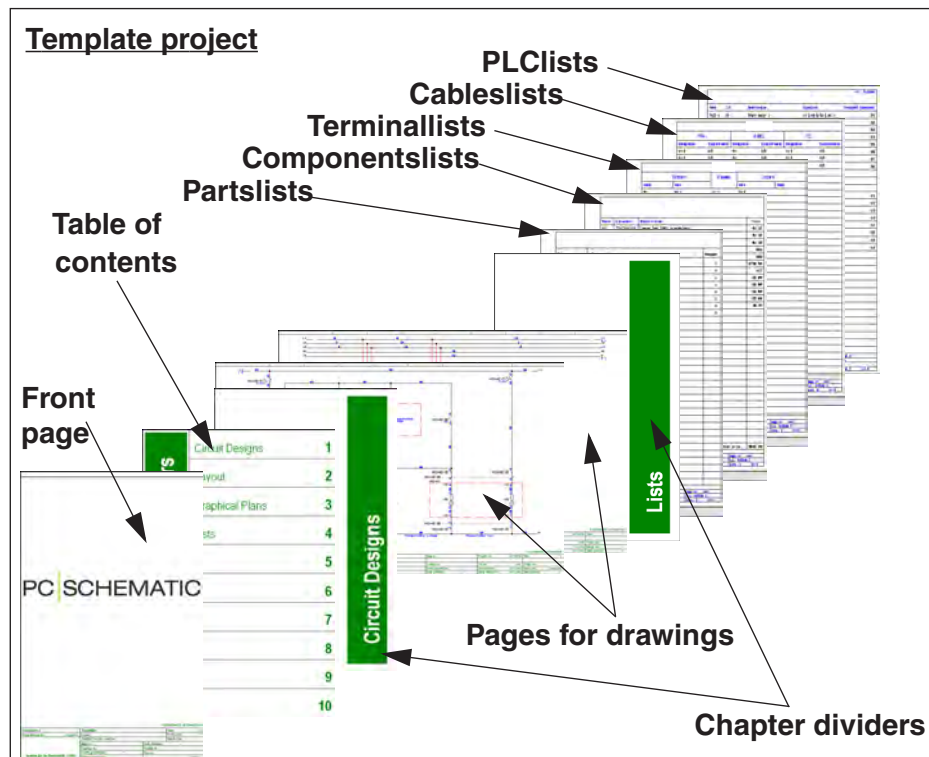
— Read more about this in “Creating Templates” on page 771, specially “Creating Page templates — via the Template editor” on page 772.

CREATING YOUR OWN PROJECT TEMPLATES

What really saves time, is if you specify a standard for how your projects shall be set up, and then create a project containing everything you need when you start up a new project.

Such a *project template* could for instance contain a front page, a table of contents, some pages for project drawings, a parts list, a components list, a terminals list and a cables list — and at the same time it contains the datafields you need.

Furthermore, you could once and for all have created the unit drawings you might need in your projects. These could then be loaded into your projects when you need them.



When you start up a new project, you start by selecting the project template, and continue working from this.

By doing the work once and for all you save time, and avoid doing the same things over and over.

Please note that you can create many different standard projects if you need to.

— Read more about this in “Creating Templates” on page 771, specially “Creating Project templates” on page 774.

CREATING SUB DRAWINGS

The next step from here is creating your own sub drawings, which you can drag directly into the project diagrams. These sub drawings can also be created based on existing diagram pages. — Read more about this in “Sub drawings” on page 256.

These sub drawings can also be applied in automatic project generation, which you can read more about in the **Project generator** manual.

PROJECT AND PAGE DATA

You can attach different types of information to a project; some is attached to the entire project, some is only attached to the individual pages in the project.

This information is typed in on a central place, and can then be inserted as *data-fields* on the project pages.

When you start **Automation**, it is already determined which datafields you can insert on your project pages. But maybe some of these datafields are superficial, while you lack other datafields.

When you have determined which information you typically need in your projects, you may therefore adapt your standard project, so that it contains exactly the datafields you need. — See how to do this in “Page data and Project data Datafields” on page 689.

CREATING SYMBOLS

Even though many symbols comes with the program, you still may need to design your own symbols, data symbols or signal symbols for your projects.

If this is the case, you may read:

- “Creating Symbols” on page 696 to see how to design symbols,
- “Creating Data symbols” on page 732 for designing data symbols, and
- “Creating Signal Symbols” on page 738 for designing signal symbols

FOLDERS CONTAINING PROJECTS

To maintain a survey of your projects, it is a good idea to create specific folders for the various types of projects.

OPTIONS FOR MULTIUSER SYSTEMS

On multiuser systems, the administrator can furthermore do several things to e.g. give every user access to a predefined set of pickmenus, to control database settings, and to ensure every user uses the same version of the program.

PAGE DATA AND PROJECT DATA DATAFIELDS

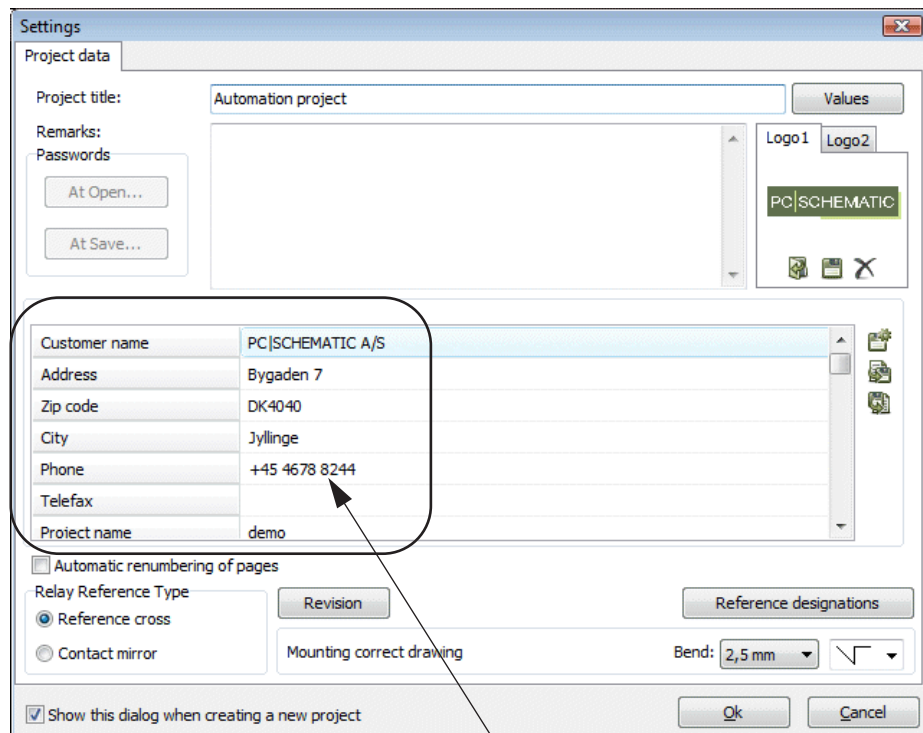
As described in “Datafields” on page 110, you can use datafields in **Automation**. It is a good idea to have a general strategy for which (user-defined) datafields you want to use in your projects.

THIS CHAPTER DESCRIBES	
— WHAT IS A DATAFIELD?.....	P. 689
— SPECIFYING WHICH DATAFIELDS SHALL BE AVAILABLE	P. 691
— ADDING NEW DATAFIELDS TO PROJECT AND PAGE DATA.....	P. 692
— DATALISTS FOR PROJECT DATA AND PAGE DATA	P. 693
— DEFAULT DATAFIELDS	P. 695

A survey of the datafields in **Automation** can be found in “Datafields” on page 855.

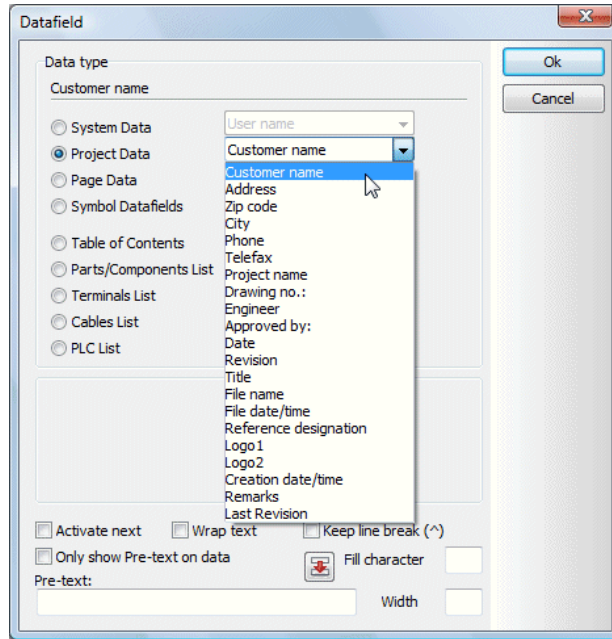
WHAT IS A DATAFIELD?

All information concerning the entire project is gathered in the dialog box **Settings => Project data**. Here you for instance type in the project title, as well as the name and address of the customer.



Project data datafields

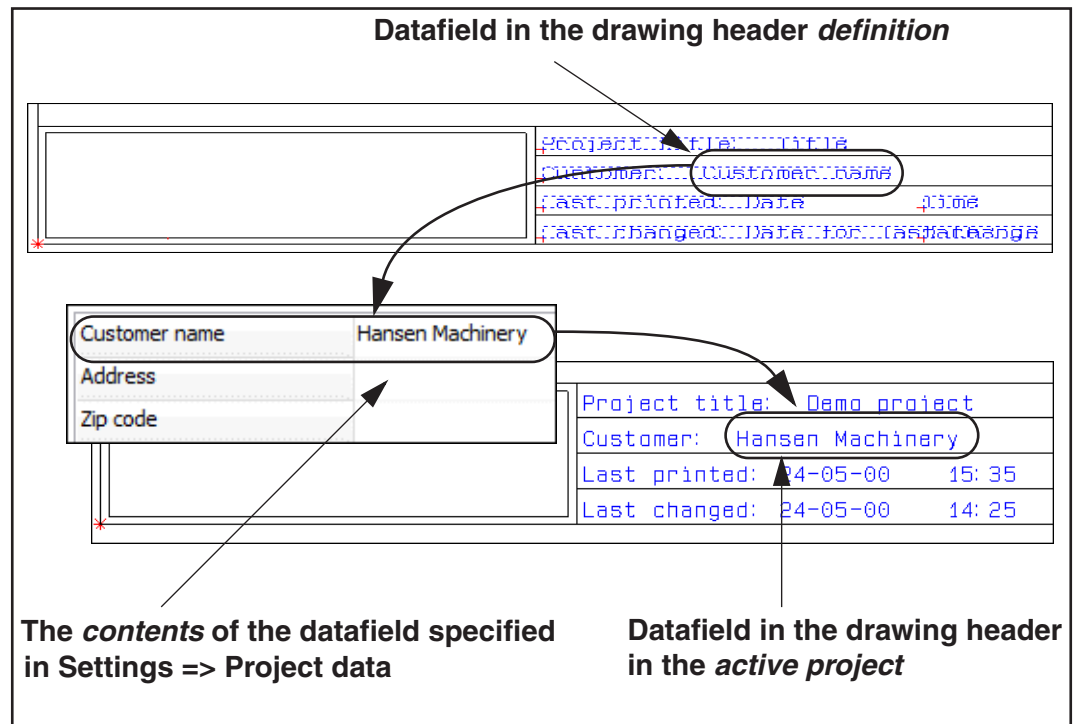
If you need to insert one of these informations in your project — for instance the address of the customer — choose **Insert => Insert Datafield**.



Here you first click on **Project Data** and then on the *datafield Customer name*.

This gives you the chosen datafield in the cross hairs, and you place it where you want it on the page.

When you update your list, the address of the customer will be read from the **Project Data** tab, and inserted on your list.



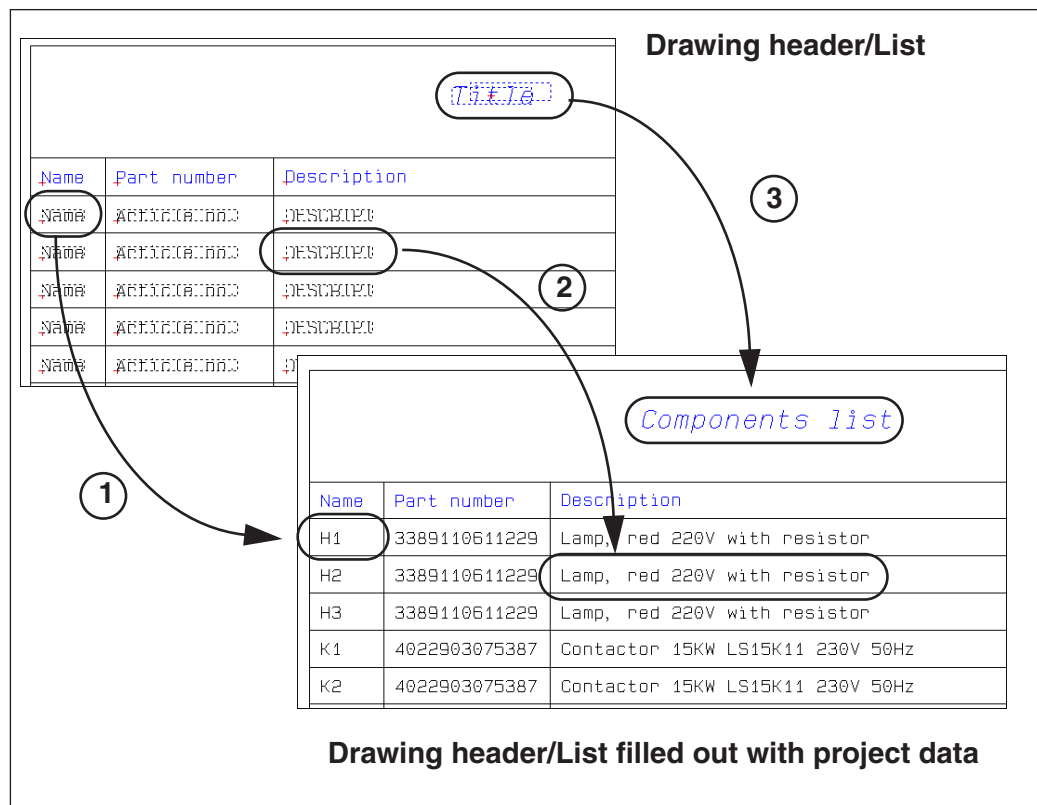
If the customer change address later on, you *only* have to change this address once in **Settings => Project data**, which automatically corrects the address on all the pages where the datafield **Customer Name** exists in the project.

Thus datafields makes it possible to have information, that need to be used several places within the project, to be located in one central place.

If one or more information need to be changed, you only have to correct it once, and the program corrects all the places the datafields appear in the project.

Datafields, which are only attached to the individual pages in the project, are located in **Settings => Page data**.

Datafields in Lists



The figure above displays the following types of datafields in lists:

- 1) The contents of the datafield **Name** collected from the diagrams
- 2) The contents of the datafield **DESCRIBE** collected from the database
- 3) The contents of the datafield **Title** collected from the **Page title** field in **Settings => Page data**

Read more about datafields, and how to place these in a project, in “Datafields” on page 110.

SPECIFYING WHICH DATAFIELDS SHALL BE AVAILABLE

When you insert datafields using **Insert => Insert Datafield**, the contents of some of these datafields, are specified in the dialog box **Settings**.

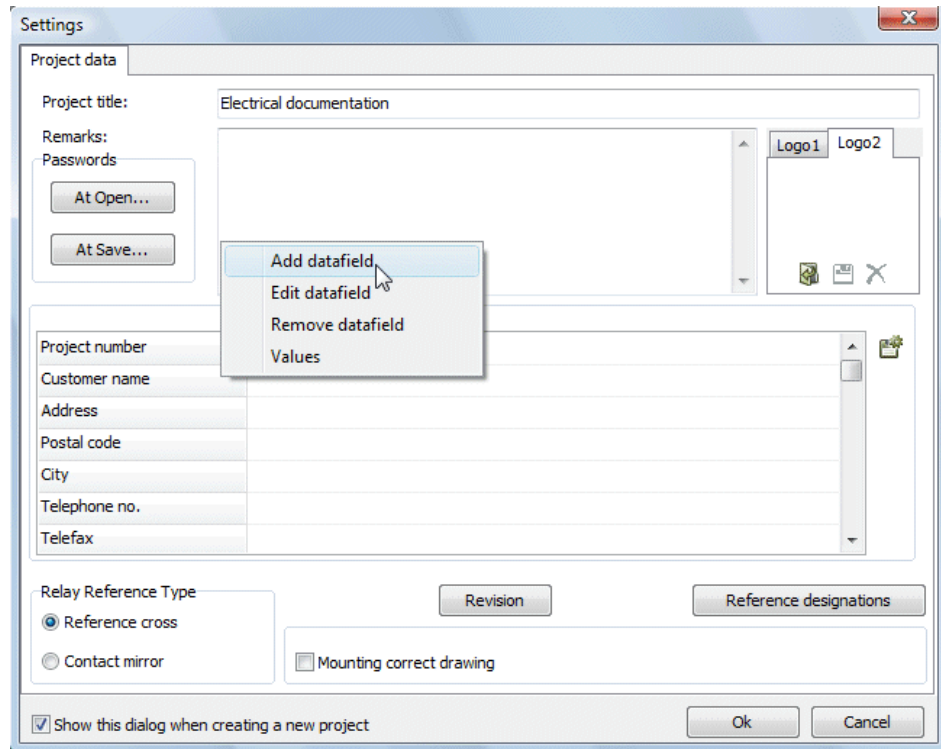
You can specifying which of datafields on the **Project data** and **Page data** tabs you want to use in your projects.

If there is a particular type of datafields you would like to be able to insert, you can create these, and if you do not need some of the other datafields, you can delete those as well.

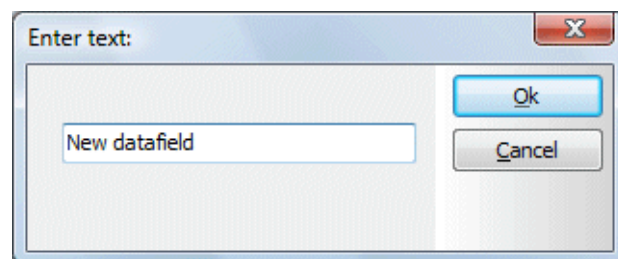
ADDING NEW DATAFIELDS TO PROJECT DATA AND PAGE DATA

To add new fields, which can be inserted as **Project data** datafields in your projects, you do like this:

Enter **Settings => Project data** and right-click on one of the project data datafields, and a menu appears:



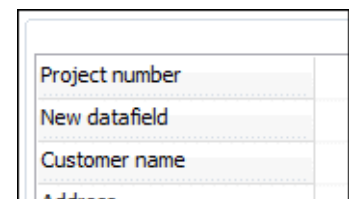
Click on **Add datafield**, and you will be asked for a name for the new datafield:



Type it in, and click **OK**.

You may now use the new datafield in your project.

Please note that the datafield will be placed where you clicked in the first place. Above there was clicked on **Customer name**, which caused this field to be moved down, and the new field to be placed on its original place.



Accordingly, you can change the name of a datafield, or remove a datafield, by clicking on the datafields and then choose either **Edit datafield** or **Remove datafield**.

Changes in **Page data** is carried out in a similar way.

If for instance you want to move the datafield **Project number** down between **Customer name** and **Subject name**, you *click* on the **Project number** field and *drag* it down there. Thus you determine the order in which the datafields appear on the two tabs.

Changing the names of used datafields not allowed

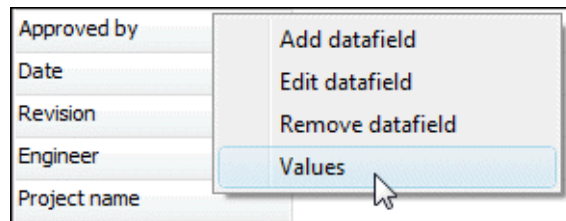
You are not allowed to change the *names* of the datafields which are being currently used in the project.

The *contents* of the datafields you can change any time.

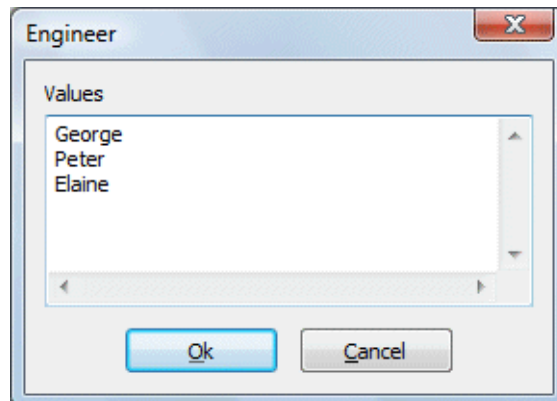
DATALISTS FOR PROJECT DATA AND PAGE DATA

It is possible to create datalists with frequently applied data values for both **Project data** and **Page data**. To e.g. create a datalist of the designers, do the following:

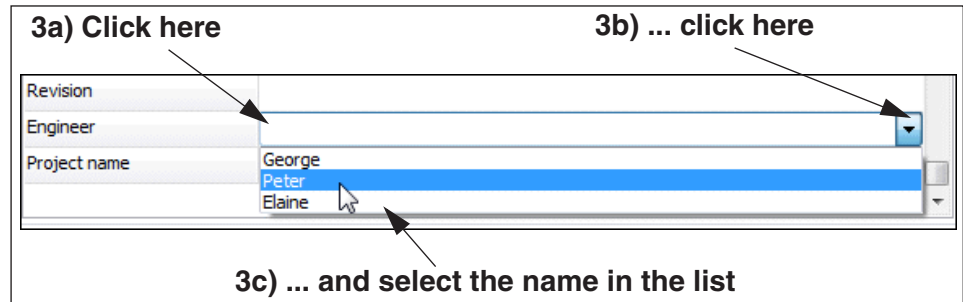
- 1) Right-click on the first of the datafields — here the datafield **Engineer** —, and select **Values**:



- 2) In the dialog box **Engineer**: Type in the names of the designers and click **OK**:



- 3) Click in the textfield next to the datafield, and a down-arrow appears in the right-hand side of the textfield. Click on the arrow, and select the desired name in the drop-down list:



- 4) The name is now selected designer is now inserted.

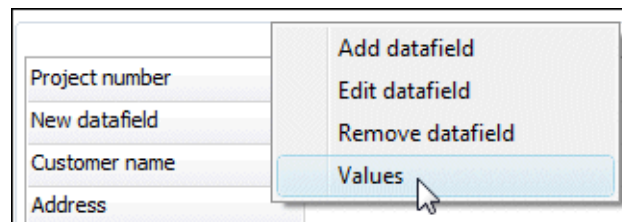
Advanced datalists

When a value in the datalist contains the characters ++ this means that the consecutive datafield(s) in **Project data / Page data** will also be filled out.

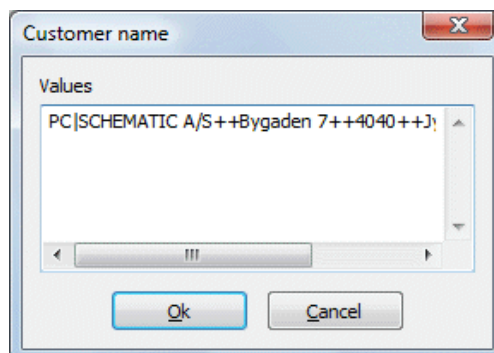
If for instance you create the data value **PC|SCHEMATIC A/S++Bygaden 7++4040++Jyllinge++4678 8244** in the list, you can hereby fill out five consecutive datafields in one operation. Do this:

Create the values in a datalist

- 1) Right-click on the first of the datafields — here the datafield **Customer name** —, and select **Values**:

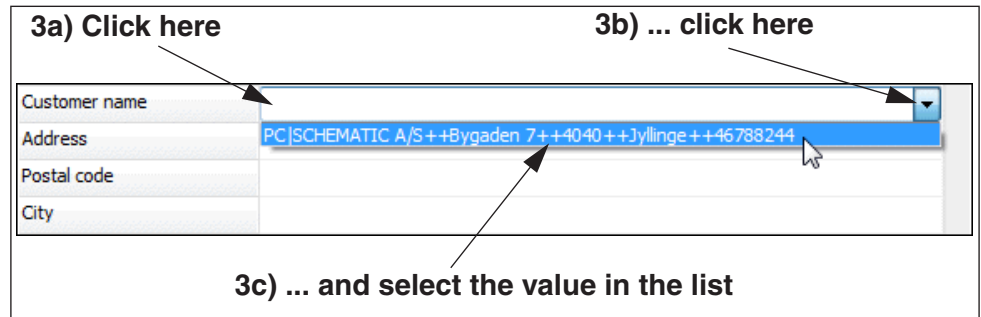


- 2) In the dialog box **Customer name**: Type in the text **PC|SCHEMATIC A/S++Bygaden 7++4040++Jyllinge++4678 8244** and click **OK**:



Select data values from the list

- 3) Click in the textfield next to the datafield, and a down-arrow appears in the right-hand side of the textfield. Click on the arrow, and select the line with the values in the drop-down list:



- 4) Hereby the datafield is filled out — as is the four subsequent datafields in the dialog box **Project data**:

Customer name	PC SCHEMATIC A/S
Address	Bygaden 7
Postal code	4040
City	Jyllinge
Telephone no.	46788244

DEFAULT DATAFIELDS



To turn your changes into a default setting for datafields, you have to click on the **Save as default** button. This have to be done on the **Project data** tab as well as on the **Page data** tab.



PROJECT DATA AND PAGE DATA DATAFIELDS ARE SAVED IN THE PROJECT FILE.

CREATING SYMBOLS

Working with **Automation**, it might happen that the program folders does not contain the specific symbol you need. Therefore, this chapter is a guide on how to create your own symbols.

THIS CHAPTER DESCRIBES	
— CREATING A NEW SYMBOL	P. 697
— THE SYMBOL TYPES.....	P. 706
— EXTRA SYMBOL DATAFIELDS IN THE SYMBOL	P. 711
— SYMBOLS WITH STATES	P. 712
— SYMBOLS WITH CONNECTION POINTS IN OTHER LAYERS	P. 713
— CREATING SYMBOLS AS MULTI SYMBOLS.....	P. 714
— SPECIFYING DEFAULT TEXT SETTINGS FOR EDIT SYMBOL MODE	P. 716
— LOADING ANOTHER SYMBOL INTO A SYMBOL	P. 716
— COPYING AREAS INTO EDIT SYMBOL.....	P. 717
— CREATING TERMINAL SYMBOLS	P. 717
— CREATING TERMINAL SYMBOLS WITH ADDITIONAL SYMBOL TYPE	P. 720
— CREATING CABLE SYMBOLS.....	P. 722
— CREATING BUSBAR SYMBOLS	P. 724
— CREATING PLC I/O SYMBOLS	P. 725
— CREATING PLC REFERENCE SYMBOLS	P. 728
— CREATING LINE CONNECTION SYMBOLS	P. 730
— OTHER OPTIONS.....	P. 731

In the following chapters you can see how to create signal symbols in “Creating Signal Symbols” on page 738, and how to create data symbols in “Creating Data symbols” on page 732.

— To see how to use the already created symbols, please read “Symbols” on page 137.



AUTOMATIC CREATION OF SYMBOLS

IF YOU WANT THE PROGRAM TO CREATE SIMPLE SYMBOLS FOR YOU AUTOMATICALLY, PLEASE READ “AUTOMATIC CREATION OF SYMBOLS” ON PAGE 197.

CREATING A NEW SYMBOL

In the example you can create a simple symbol, consisting of a rectangle with a circle inside. Furthermore, the symbol will contain a free text as well as two connection points. The precise meaning of this is follows later.

To create a symbol, you have to be working in a project. If this is not the case you have to open a new project:



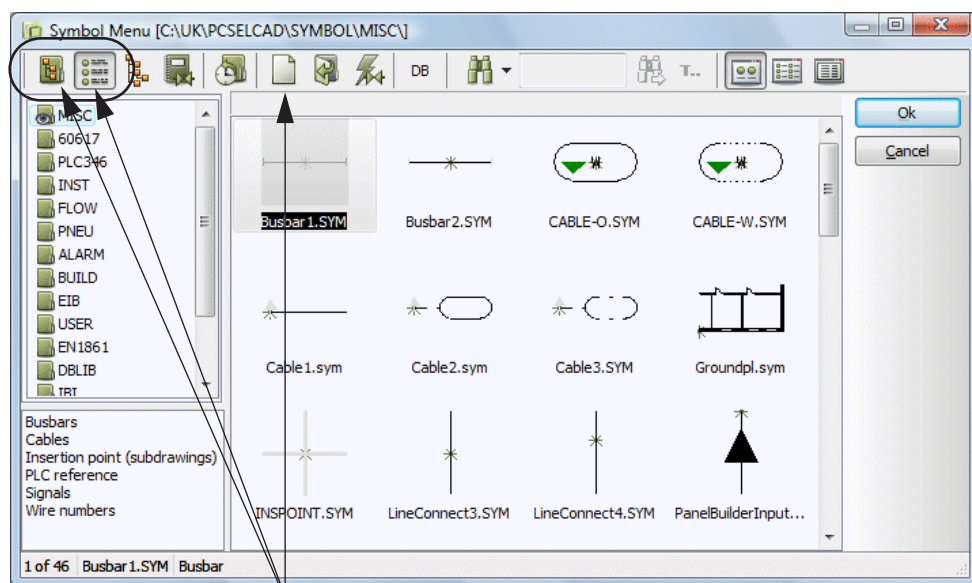
Click on the **New document** button, and leave the dialog box **Settings** by pressing <Esc> or by clicking on **Cancel**.

You are now ready to create a new symbol:



Click on the **Symbols** button, or use the short-cut <s>. Then click on the **Symbol menu** button. Also, you can press <F8> to enter the **Symbol menu** directly.

You have now entered the **Symbol menu**:



Click here



Click on the **Select Library from Folders** button or on the **Select Library from Alias** button, and select the folder/*alias* you wish to save the symbol under — e.g. the folder **MISC**.



Then click on the **Create new symbol** button, to create a new symbol.

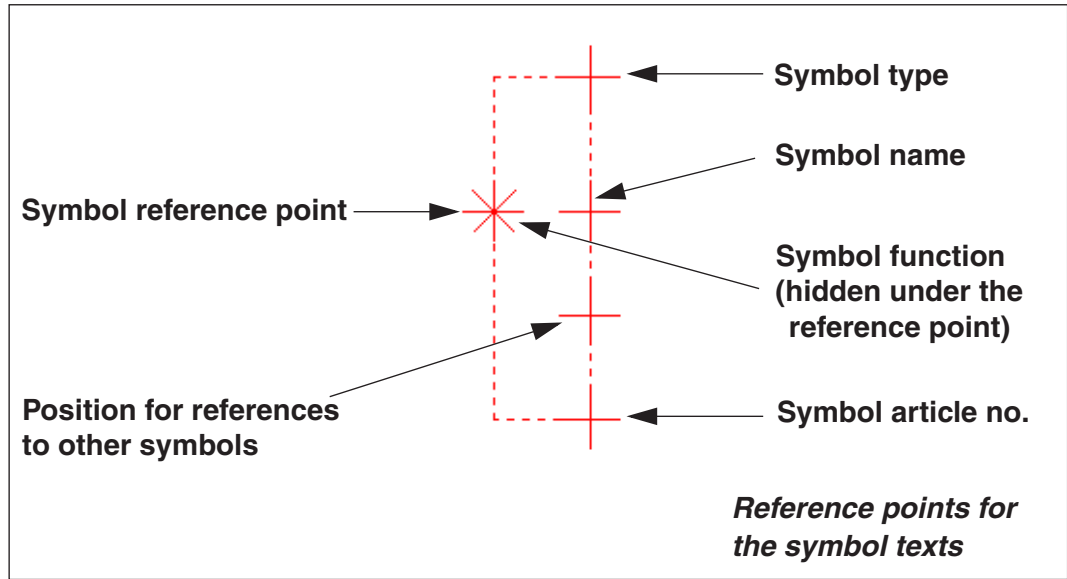
— Read more about the **Symbol menu** in “Fetching symbols using the Symbol menu” on page 144.

You now enter the *Edit symbol* mode, in which you can see **SYMB** flashing on a red background in the lower part of the left tool bar.



Make an appropriate **Zoom** <z> around the red marks in the middle of the screen. — If you are not sure to zoom, please see “Zoom” on page 42.

On the middle of the screen, you will see the following drawn with the color red:



The symbol reference point

The asterisk (*) is the reference point for the new symbol. This reference point indicates the position of the symbol, and that it is this point you rotate the symbol around, if you rotate it. A symbol has only got one reference point.

Reference points for symbol texts

The red pluses (+) are the reference points for the symbol texts. — Read more about these texts in “Texts connected to symbols” on page 107.

Position for references to other symbols

When creating new symbols, you can furthermore specify the position for references to other symbols. The position of this reference point is pointed out in the lower-left corner in the figure above.

The red plus for this reference text is always present in the *Edit symbol* mode, but does not become active for placed symbols before the symbol reference is activated — for symbols of the type **Open**, **Close** etc.

When you place a symbol of a type, for which references are not displayed — e.g. symbols of the type **Normal** —, the reference is not displayed until you have

selected **With reference** on the tab **References** in the dialog box **Component data**.



OTHER OPTIONS: USE THE SYMBOL GENERATOR AND PICKMENUS

WHEN YOU ARE IN THE **EDIT SYMBOL** MODE (WHICH YOU ARE NOW), YOU CAN ALSO USE THE **SYMBOL GENERATOR** TO CREATE A "RAW SYMBOL", WHICH YOU CAN THEN CONTINUE WORKING ON HERE.

TO DO THIS, CHOOSE **SYMBOLS** AND CLICK ON THE **SYMBOL GENERATOR** BUTTON. — READ MORE IN "AUTOMATIC GENERATION OF SYMBOLS — USING THE SYMBOL GENERATOR" ON PAGE 197.

YOU CAN ALSO CREATE YOUR OWN PICKMENUS, WHICH YOU CAN USE WHEN DESIGNING SYMBOLS. YOU CAN CREATE PICKMENUS FOR DIFFERENT GROUPS OF SYMBOLS, EACH USING THEIR SET OF ADDITIONAL SYMBOLS. THE PICKMENUS CAN CONTAIN BOTH SYMBOLS, LINE TYPES AND TEXT TYPES.



Moving the reference point

To make room to draw the symbol itself, you can now move the reference point a bit to the left. Therefore, click on the **Reference point** button. *Click* on the reference point, and *drag* it to where you want it placed. — In this case you move it 15mm to the left. See the coordinates in the lower-left corner of the screen.



Lines in symbols



Click on the **Lines** button (short-cut <I>), and click on the **Draw** button (in order to make it highlighted).

You can determine that lines in the symbol must follow the width and the color of the conducting lines, which gets connected to the symbol, when you are using it on a diagram later on. If the **Follow connected** button is not activated (highlighted), then click on it. Read more about this in "Follow connected lines" on page 90.



Drawing lines as a rectangle



To draw the rectangle in the symbol, you now click on the **Rectangle** button.

Point out the lower-left corner of the rectangle by clicking on a point 10mm below and 10mm to the left of the reference point. Then point out the upper-right corner of the rectangle by clicking on a point 10mm above and 10mm to the right of the reference point.

You have now drawn a rectangle of 20x20mm around the reference point.

Coordinates and distances are displayed in the lower-left corner of the screen.

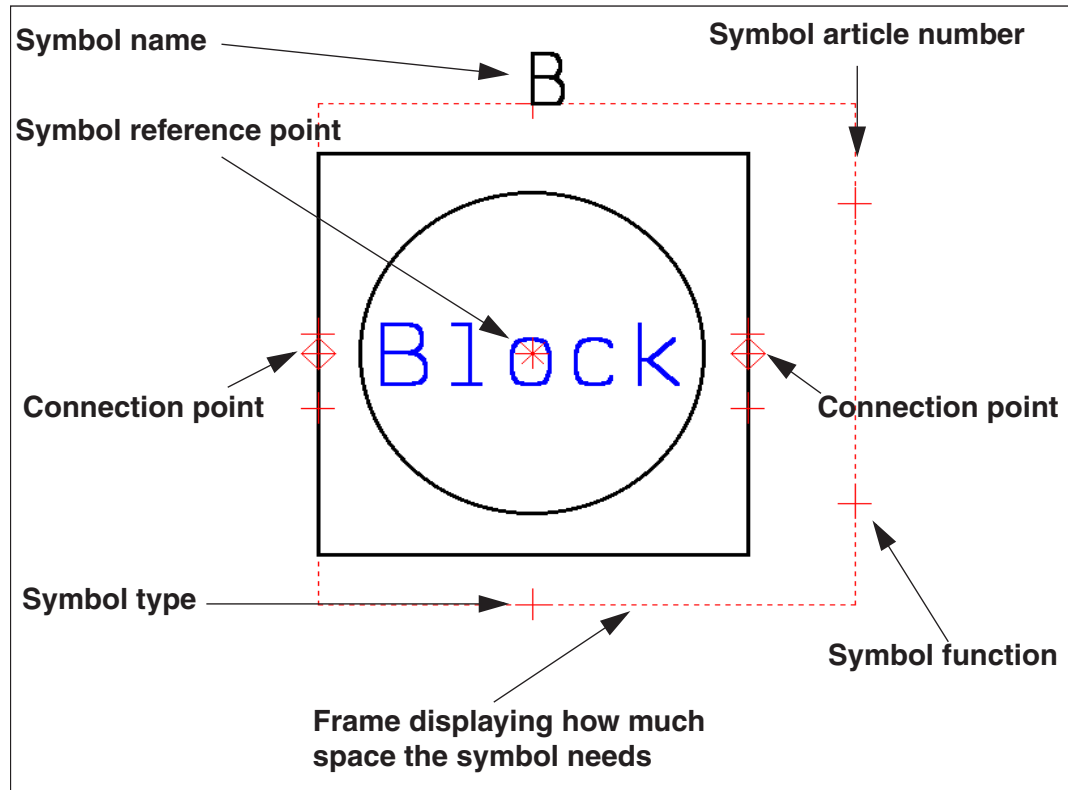
x=160,00 y=110,00mm	x=20,00 y=20,00mm
---------------------	-------------------

Place connection points

To be able to use the symbol electrically later on, there have to be some points on the symbol, to which electrical lines can be connected, when you place the symbol



in a diagram. These points are called *connection points*, and you place them by clicking on the **Symbols** button, the **Connection point** button and the **Pencil**. You then get a connection point in the cross hairs, and click where you want the two connection points located — see the illustration below.



When you then place the connection points, you will be asked about their connection point data.

A screenshot of a dialog box titled "Connection data". It has a close button (X) in the top right corner. The dialog contains several input fields and checkboxes. The "Name:" field has a text input and a "Visible" checkbox with a checkmark. The "Function:" field has a text input and a "Visible" checkbox with a checkmark. The "Label:" field has a text input and a "Visible" checkbox with a checkmark. The "Description:" field has a text input and a "Visible" checkbox with a checkmark. Below these fields is an "IO status" section with two dropdown menus: "Main type" set to "none" and "Extension" set to "none". At the bottom, there are four checkboxes: "With reference" (unchecked), "No dot generation" (unchecked), "No double check" (unchecked), and "Always dot when connected" (unchecked). On the right side of the dialog, there are "Ok" and "Cancel" buttons.

In this case, you can for instance give them the names *1* and *2* respectively, and make the names invisible on diagram pages, by deselecting the **Visible** check box for the **Name** field.

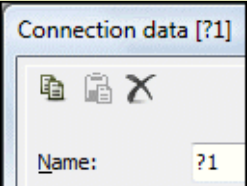
MORE OPTIONS IN THE DIALOG BOX CONNECTION DATA

Connection data with a variable part in the name

If you write *?1* and *?2*, you can — directly from the dialog box **Component data** — type in what shall be at the place of the question mark in the connection name, when you place the symbol in the project.

This can for instance be useful when placing make and break symbols, where the mechanical position is not known prior to the placing of the symbol. Correspondingly, for terminals it can be useful just to write ***, which gives you the option of typing in all of the connection name when placing the terminal in the project.

— See “Connection names with variable part in the Symbol definition” on page 162, “Wildcards/jokers” on page 550 and “Symbols with variable connection names” on page 599.



With reference

If you click here, an independent reference text is created for the connection point, which refers to connection points on symbols with the same name and connection name. This is particularly applied for PLC symbols — see “Creating PLC I/O symbols” on page 725 and “Creating PLC reference symbols” on page 728.

INACTIVE REFERENCE

When you have selected **With reference**, you then get the option of choosing **Inactive reference**. When you do so, the reference is *not* activated when the symbol is placed. But it makes it possible for you to pre-define the position of the references, if these should be activated after the symbol has been placed in a project.

No dot generation

— Regarding **No dot generation**, see “No dot generation” on page 701.

MORE OPTIONS IN THE DIALOG BOX CONNECTION DATA

No double check

When performing a *design check* — see “Design Check” on page 320 —, it is possible to examine whether the project contains symbols, which has more than one connection point with the same name. This is done by selecting **Used more than once**.

However, for some symbols — pneumatic symbols in particular — connection points with identical names allowed according to the standards.

To *disable* this check for a connection point in the symbol definition, select **No double check**. Having done this, the program will not examine whether this connection point name is used more than once for the component, when performing a design check.



NOTE FOR NO DOUBLE CHECK

BEFORE DISABLING THE DOUBLE CHECK IN THE SYMBOL DEFINITION, YOU SHOULD MAKE SURE THAT THIS IS ALLOWED FOR THIS SYMBOL TYPE ACCORDING TO DOCUMENTATION STANDARDS.

IT IS NOT POSSIBLE TO DISABLE THIS DOUBLE CHECK FOR A SYMBOL IN ANY OTHER WAY THAN VIA THE SYMBOL DEFINITION.

You have now placed the connection points for the symbol.

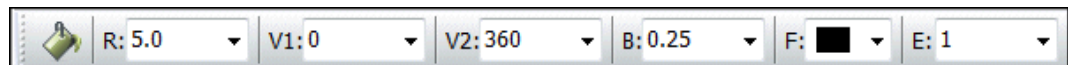
Press <Esc> to get rid of the connection point symbol in the cross hairs.

Drawing a circle



You draw the circle in the symbol by clicking on the **Pencil** and the **Circles** button (short-cut <c>).

The tool bar for circles/arcs then appear, and you get a circle in the cross hairs.



You may read more about this tool bar in “Arcs/Circles” on page 100.

Set the radius (**R:**) to 8.0, and place the circle with its centre in the symbol reference point.

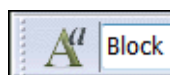
Typing in a free text

In the middle of the symbol, the *free text Block* shall appear. — See “Texts” on page 102 for further details about the various text types in **Automation**.



Therefore click on the **Texts** button, or press the short-cut key <t>.

Click in the text field in the texts tool bar, and type in the text *Block*. Press <Enter>.



You now have the text in the cross hairs.

Changing text properties

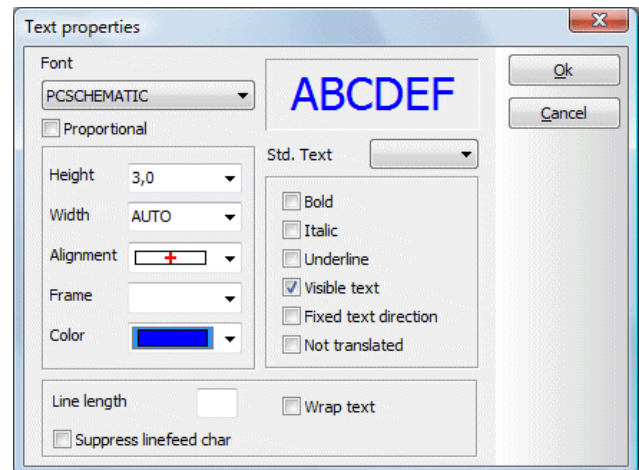


Click on the **Text properties** button. Give the text a height of *3.0mm*, and set the alignment to *centre-centre*.

Select height and alignment as displayed in the **Text properties** menu. Click **OK**.

The text in the cross hairs now has the chosen properties.

If you are not sure how to make these settings, please read “Specifying text properties / Appearance of text” on page 122.



Place the text by clicking in the middle of the symbol.

Press <Esc> or click on the Pencil in order not to place more copies of the text.

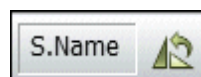
— Please note that a free text in a symbol definition can not be changed when you later place the symbol in a project. These texts can only be changed here in the programs Edit symbol mode.

Texts and text properties for symbol texts

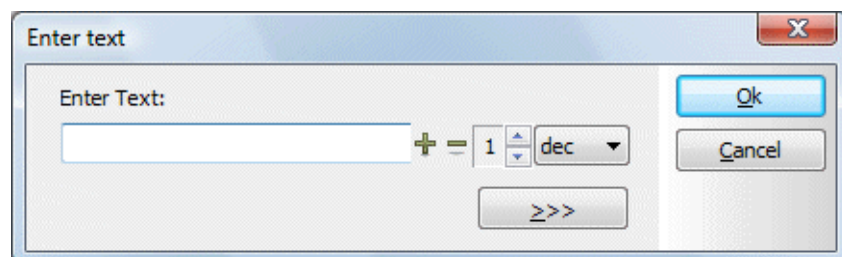


Click on the **Texts** button, if not already selected. Deselect the Pencil by pressing <Esc>. Click on the reference point for the text **Symbol name**, to select it. In the command tool bar you can see whether you have selected the right text reference point.

Here it should say **S.Name**.



If you fill out this field with the prefix *B*, this letter will be suggested as a symbol name, when you place the symbol in a project. Press <k>, and this dialog box appear:



Type in *B* and press <Enter>. — In stead of pressing on <k> you can click in the text field in the command tool bar, type in *B* and press <Enter>.

When you have selected one of the symbol texts, you can change back and forth between the symbol texts by pressing <F5>.



DO NOT WRITE "-B" IN THE SYMBOL NAME

IT IS RECOMMENDED THAT YOU DO NOT WRITE "-B" IN THE SYMBOL NAME. IF YOU CHOOSE **SETTINGS => POINTER/SCREEN** AND SELECT **INSERT - IN FRONT OF SYMBOL NAMES**, THE PROGRAM AUTOMATICALLY INSERTS A MINUS IN FRONT OF THE SYMBOL NAME WHEN YOU PLACE THE SYMBOL IN A PROJECT — SEE "POINTER/SCREEN" ON PAGE 836. IF YOU WRITE "B?" THE SYMBOL WILL AUTOMATICALLY BE ASSIGNED THE NEXT AVAILABLE SYMBOL NAME, WHEN YOU PLACE THE SYMBOL.

Changing text properties



While the text is selected on the drawing, you click on the **Text properties** button, and choose a height of 2.5mm and a *centre-centre* adjustment.

Click **OK**.

The text properties settings which are suggested for the various types of symbol texts, are set on the tab **Settings => Text/Symbol defaults**. You may read more about this in "Changing text properties" on page 128.



CHANGING SYMBOL TEXTS WITH THE SYMBOL BUTTON CHOSEN

When you have activated the **Symbols** button while designing a symbol, you get this tool bar on the screen:

Here you can also fill out the symbol texts by clicking on the various fields, and type in the relevant texts.

Moving the symbol texts

To move the *Symbol name* text on the diagram, you do like this:



Click on the **Texts** button, and deactivate the Pencil (press <Esc>). Click on the **Symbol name** text, in order to select it. Then click on the **Move** button.



Usually you work in a snap of 2.5mm. The **Snap** button is selected (highlighted).

To place the text in a finer snap, you press down <Shift>. The snap is now 0.25mm. This you can see on a red background in the lower part of the left tool bar. Now click where you want to place the text. Let go of the <Shift> key, and you return to the normal snap.

Move the other symbol texts according to the drawing of the symbol above.

Read more about *snap* and *grid* in “Screen / Image Functions” on page 42.

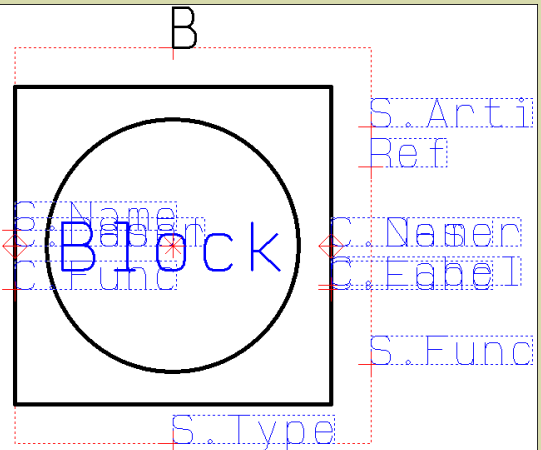
SHOW SYMBOL TEXTS

If you want to see which size the different symbol texts will have when they are filled out, which color they have, or whether they will cover the drawing of the symbol when they are filled out, you can choose **View => Show texts**.

Only texts which have not yet been filled out, will be altered.

You can change the appearance of the texts using the **Text properties** button.

Choose **View => Show texts** again, to turn the function off again.



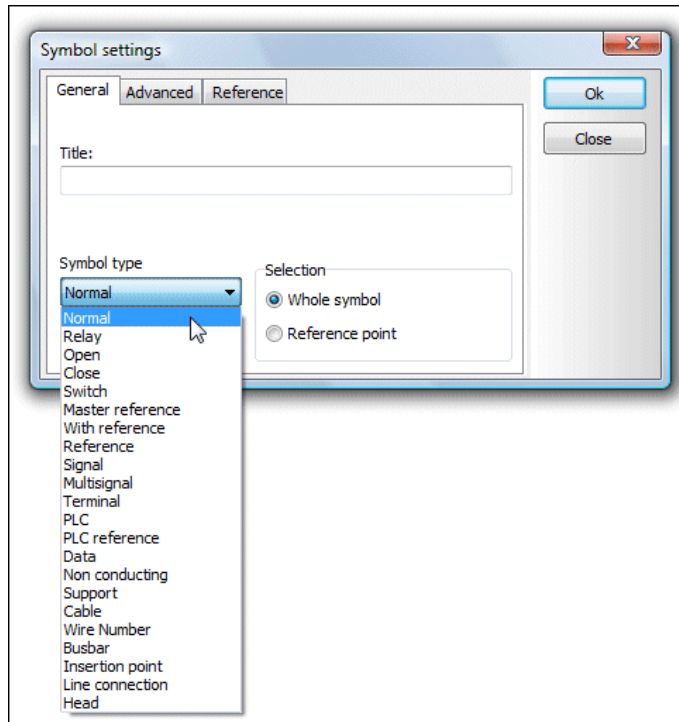


Saving the symbol



Click on the **Save** button. If you have not saved the symbol before, a dialog box for Symbol settings will appear. Give the symbol a *Title* — this means a description of the use or appearance of the symbol. This text will be displayed in the upper part of the **Symbol menu** when the symbol is selected there — and you also use it to determine which symbol to pick when you choose symbols in the pickmenu.

Click inside the field, and type in for instance *Block symbol with two connection points*.



THE SYMBOL SETTINGS BUTTON

YOU CAN ALSO ENTER THIS DIALOG BOX WHILE DESIGNING THE SYMBOL, IF YOU CLICK ON THE **SYMBOLS** BUTTON, AND THEN ON THE **SYMBOL SETTINGS** BUTTON.

Symbol types

As **Symbol type** you can choose between the following symbol types:

SYMBOL TYPE	USE
NORMAL	SYMBOLS WITHOUT A SPECIAL SYMBOL TYPE.
RELAY	SYMBOLS WHICH WILL HAVE AN ATTACHED REFERENCE CROSS WHEN PLACED ON A DIAGRAM PAGE.
OPEN	THE SYMBOL WILL BE CONCEIVED AS AN OPEN CONTACT, AND THE LOCATION OF THE SYMBOL WILL BE DISPLAYED IN THE REFERENCE CROSS, AS AN OPEN CONTACT FOR THE COMPONENT IT IS A PART OF. WILL ALSO BE ASSIGNED A REFERENCE TO THE RELAY SYMBOL FOR THE SAME COMPONENT.
CLOSE	THE SYMBOL WILL BE CONCEIVED AS A CLOSE CONTACT, AND THE LOCATION OF THE SYMBOL WILL BE DISPLAYED IN THE REFERENCE CROSS, AS A CLOSE CONTACT FOR THE COMPONENT IT IS A PART OF. WILL ALSO BE ASSIGNED A REFERENCE TO THE RELAY SYMBOL FOR THE SAME COMPONENT.
SWITCH	THE SYMBOL WILL BE CONCEIVED AS A SWITCH, AND THE LOCATION OF THE SYMBOL WILL BE DISPLAYED IN THE REFERENCE CROSS, AS AN OPEN- AS WELL AS A CLOSE CONTACT FOR THE COMPONENT IT IS A PART OF. WILL ALSO BE ASSIGNED A REFERENCE TO THE RELAY SYMBOL FOR THE SAME COMPONENT.

SYMBOL TYPE	USE
MASTER REFERENCE	THE SYMBOL WILL HAVE REFERENCES TO ALL OTHER SYMBOLS HAVING THE SAME SYMBOL NAME. FOR INSTANCE MANUALLY CONTROLLED CONTACTS.
WITH REFERENCE	REFERS EITHER TO A SYMBOL WITH MASTER REFERENCE , OR TO THE CLOSEST PRECEDING- AS WELL AS SUCCEEDING SYMBOL, WHICH IS PART OF THE SAME COMPONENT.
REFERENCE	A SYMBOL WHICH IS A REFERENCE CROSS.
SIGNAL	A SYMBOL USED AS REFERENCE FOR A SIGNAL FROM ONE ELECTRICAL POINT TO ANOTHER. READ MORE ABOUT THIS KIND OF SYMBOLS IN “CREATING SIGNAL SYMBOLS” ON PAGE 738, AND IN “SIGNAL SYMBOLS” ON PAGE 203.
MULTISIGNAL	RESERVED FOR FUTURE USE.
TERMINAL	THE SYMBOL WILL APPEAR IN TERMINALS LISTS. TERMINAL SYMBOLS HAVE THE OPTION OF HAVING AN ADDITIONAL SYMBOL TYPE — SEE “CREATING TERMINAL SYMBOLS WITH ADDITIONAL SYMBOL TYPES” ON PAGE 720.
PLC	SEE “CREATING PLC I/O SYMBOLS” ON PAGE 725. THE SYMBOL WILL APPEAR IN PLC LISTS.
PLC REFERENCE	SYMBOL, WHICH SHALL BE USED AS A PLC REFERENCE SYMBOL. — SEE “CREATING PLC REFERENCE SYMBOLS” ON PAGE 728 AS WELL AS “WORKING WITH PLCs” ON PAGE 418, SPECIALLY “ON PLC REFERENCE SYMBOLS” ON PAGE 418.
DATA	APPLIED TO ADD INFORMATION TO A COMPONENT WHICH HAVE BEEN PLACED ON THE DRAWING. THIS INFORMATION CAN ALSO APPEAR IN LISTS. READ MORE IN “CREATING DATA SYMBOLS” ON PAGE 732.
NON CONDUCTING	THE SYMBOL CAN BE CONNECTED TO A NON CONDUCTING LINE. FOR INSTANCE A SYMBOL FOR A WINDOW OR A DOOR, WHICH CAN BE PLACED ON A NON CONDUCTING LINE (A WALL).
SUPPORT	A SYMBOL BELONGING TO A SPECIFIC COMPONENT, AND THAT WILL APPEAR IN LISTS. TYPICALLY BOTH TEXTS AND LINES ARE SET TO NP (SEE “LINE COLOR: F” ON PAGE 76 AND “COLOR” ON PAGE 124), SO THEY ARE DISPLAYED ON SCREEN BUT NOT ON PRINT.
CABLE	THE SYMBOL IS A CABLE, AND WILL APPEAR IN CABLES LISTS — SEE FOR INSTANCE “WORKING WITH CABLE SYMBOLS” ON PAGE 218, AND “CREATING CABLE SYMBOLS” ON PAGE 722.
WIRE NUMBER	A SYMBOL WHICH CAN BE APPLIED AS A WIRE NUMBER SYMBOL. IS USED DURING AUTOMATIC OR MANUAL WIRE NUMBERING. THE PROGRAM SUGGESTS NEXT AVAILABLE WIRE NUMBER, WHEN YOU PLACE A WIRE NUMBER MANUALLY. — SEE “AUTOMATIC WIRE NUMBERING” ON PAGE 389.
BUSBAR	THE SYMBOL IS A BUSBAR. — SEE “CREATING BUSBAR SYMBOLS” ON PAGE 724.
INSERTION POINT	USED FOR AUTOMATIC INSERTION OF SUB DRAWINGS. — SEE “PLACING SUB DRAWINGS WITH DRAG’N DRAW” ON PAGE 264.
LINE CONNECTION	SYMBOL APPLIED IN CONNECTION WITH SUB DRAWINGS WITH MODELS. THE SYMBOL IS APPLIED FOR SPECIFYING WHERE TO CONNECT A WIRE IN A SUB DRAWING, WHEN THE SUB DRAWING IS PLACED IN A PROJECT PAGE. — SEE “CREATING LINE CONNECTION SYMBOLS” ON PAGE 730.
HEAD	SYMBOL APPLIED FOR DIRECT PLACING OF A LIST ON A PAGE. — SEE “CREATING A LIST SYMBOL” ON PAGE 762.

The symbol in this example has no specific directions, so the symbol type is **Normal**.

Selection

The **Selection** shall be set to **Whole symbol**. This means that the symbol will be selected if you click anywhere on the symbol.

If you set the **Selection** to **Reference point**, this means the symbol only will be selected when you click directly on its reference point. This facility is used for symbols on which you place other symbols, such as a switchboard symbol.

Click on **OK**.



IN PROJECTS YOU CAN ALSO CHOOSE THE REFERENCE POINTS FOR SYMBOLS WITH SELECTION SET TO **REFERENCE POINT**.

SYMBOL SETTINGS: ADVANCED

CLICK ON THE **ADVANCED** TAB FOR EXTRA OPTIONS:

Symbol settings

General **Advanced** Reference

Auto scale

No scale adjustment

Multisymbol

Mechanical

Electrical

Ok

Close

HERE YOU CAN TURN THE **NO SCALE ADJUSTMENT** FUNCTION ON AND OFF. NORMALLY IT WILL BE TURNED OFF, WHICH SIGNIFIES THAT THE SYMBOL *HAS* GOT SCALE ADJUSTMENT. THIS MEANS THAT YOU CAN USE THE SAME SYMBOL ON A PAGE WITH THE SCALE 1:1 AS WELL AS ON A PAGE WITH THE SCALE 1:50.

AUTO SCALE IS — JUST LIKE **NO SCALE ADJUSTMENT** — A FUNCTION YOU DO NOT USE IN ORDINARY DIAGRAM OR GROUND PLAN DRAWINGS. THIS IS A SPECIAL FUNCTION, WHICH MAKES THE SYMBOL SCALE ITSELF, SO THAT IT CORRESPONDS TO THE LINE TYPE YOU PLACE IT ON. YOU CAN ONLY USE THIS FUNCTION FOR SYMBOLS DESIGNED TO FIT 10MM LINES.

WHAT **MULTISYMBOL** IS USED FOR, IS DESCRIBED IN “CREATING A SYMBOL AS A MULTI SYMBOL” ON PAGE 714.

WHEN **MECHANICAL** IS SELECTED, THE SYMBOL WILL BE INCLUDED IN COMPONENTS AND PARTS LISTS. **MECHANICAL** IS SELECTED BY DEFAULT.

— SEE “SYMBOL DATA IN ELECTRICAL AND MECHANICAL LISTS” ON PAGE 161.

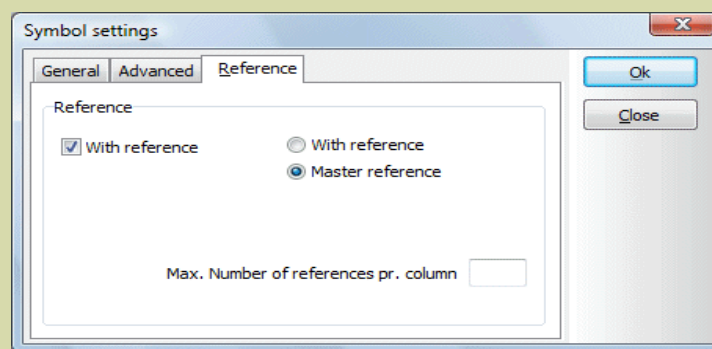
SYMBOL SETTINGS: ADVANCED

WHEN **ELECTRICAL** IS SELECTED, THE SYMBOL WILL BE INCLUDED IN ALL ELECTRICAL LISTS — SUCH AS TERMINALS, CABLES, CONNECTIONS AND PLC LISTS — WHEN THE SYMBOL CONTAINS INFORMATION TO INCLUDE IN THESE LISTS. **ELECTRICAL** IS SELECTED BY DEFAULT.

— SEE “SYMBOL DATA IN ELECTRICAL AND MECHANICAL LISTS” ON PAGE 161.

SYMBOL SETTINGS: REFERENCE

IF NECESSARY, CLICK THE **REFERENCE** TAB:



FOR SYMBOL TYPES CREATED WITHOUT REFERENCES, YOU CAN HERE SPECIFY TO CREATE THE SYMBOL WITH **WITH REFERENCE** ACTIVATED, WHEN PLACED IN A PROJECT PAGE. THE OTHER **WITH REFERENCE** SETTINGS CAN ALSO BE PREDEFINED.

— READ MORE ABOUT THIS IN “REFERENCES ON SYMBOLS” ON PAGE 369, “INSERTING REFERENCES ON SYMBOLS WITHOUT REFERENCE” ON PAGE 369 AND “SYMBOL TYPE AND REFERENCES” ON PAGE 370.

Saving the symbol

First time you save a symbol, you enter the **Save as** dialog box. Here you select the folder you want to save the symbol in, like you normally do in Windows.

Click in the **File name** field and type in the name *Block*. The symbol will automatically get the extension **.sym**.

Please refer to “Symbol folders” on page 225 to determine in which folder you will save the symbol.

Click on **Save**. The symbol has now been saved.



SYMBOL CREATION TIME IN GMT

THE TIME OF CREATION FOR SYMBOLS IS REGISTERED IN GMT. THE CREATION TIME OF ANY SYMBOL IS THEREFORE INDEPENDENT OF WHERE IN THE WORLD THIS SYMBOL IS CREATED. NO UNNECESSARY GENERATION NAMES WILL THEREFORE BE GENERATED DUE TO SYMBOLS CREATED IN DIFFERENT TIME ZONES, OR WHEN CHANGING BETWEEN SUMMER/WINTER TIME.

Printing symbols

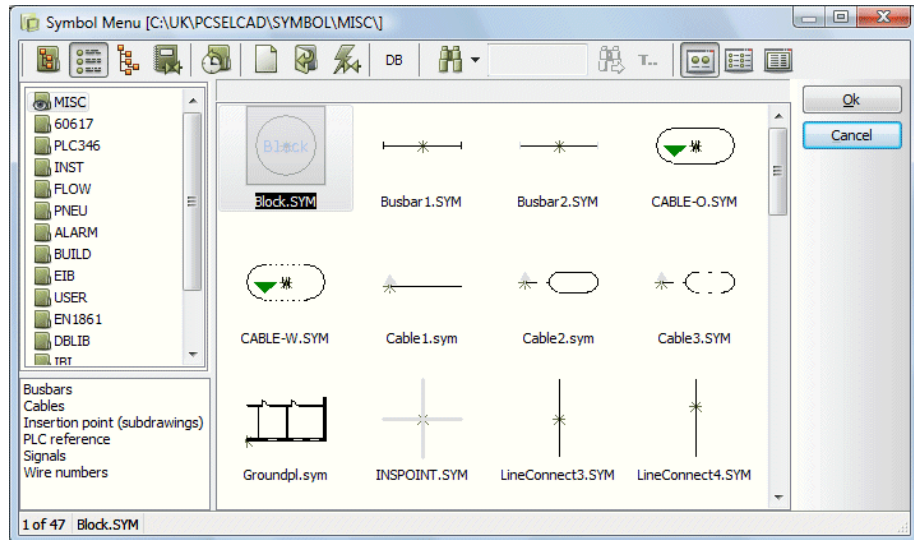


If you wish to print the symbol, just click on the **Print this page** button.

Leaving the Symbol menu

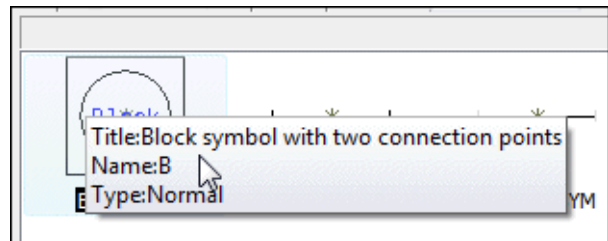
Having saved the symbol, you may now leave the Design symbol mode.

Therefore choose **File => Close**.



You now return to the **Symbol menu**, where you can find the symbol by changing to the folder you saved the symbol in.

When you place the cursor above the symbol, its **Title**, **Name** and **Type** is displayed as a hint.



If you need the symbol on your drawing, you then click on **OK**.

You then return to the project with the symbol in the cross hairs, and you can now place it in the project.

— Read more about what you then can do with the symbol in “Symbols” on page 137.

Renaming symbols

To rename a created symbol from the **Symbol menu**, do this:



- 1) Enter the **Symbol menu**
- 2) Right-click on the symbol, and choose **Rename**
- 3) Type in the new symbol name, and click **OK**

EXTRA SYMBOL DATAFIELDS IN SYMBOLS

You can also insert extra datafields in the definition of a symbol:

- 1) Create the necessary datafields — as described in “Creating symbol datafields” on page 733
- 2) Choose **Insert => Insert Datafield**
- 3) In the dialog box **Datafield**: Choose the datafield you want to insert, and place it in the symbol definition — read more about datafields in “Datafields” on page 110

You can also insert datafields of the type **Project data** and **Symbol data** in the symbol definition.

When you have placed the symbol on a project page, you type in the contents of the datafield as described in “The tab Symbol Data” on page 164.

Inserting symbol datafields from database in symbol definition

There is also a symbol datafield for all fields in the database. In your projects you can therefore place symboldata datafields, that fetches data from the database.

To place a such symboldata datafield you must do as described above, and select **Symbol Datafields** in the dialog box **Datafield**.

The datafields fetching data from the database, are named **DB[xxxxxxx]**, where **xxxxxxx** is the name on the database field. For instance will **DB[DESCRIPT]** fetch the contents from the **DESCRIPT** field for the component in the database.

Symbol datafields, which you have designed yourself, will be written as ordinary text. — See “Creating Data symbols” on page 732.

Typing in data when placing symbols with extra symbol datafields

When you have placed the symbol in a project, you type in the contents of the datafields as described in “The tab Symbol Data” on page 164.

CREATING SYMBOLS WITH STATES

When you wish to create a symbol with states — see “Symbols with states” on page 167 — you start by drawing the symbol as described in “Creating Symbols” on page 696.

How to specify the states



The only extra thing you must do, is to start by clicking the **States to layers** button, when you enter the **Edit symbol** mode.

This tells the program that the layers in the symbol now shall be understood as *states*. You can then use the **Layer** menu for specifying the symbol states:

The common parts are placed in layer 0

In layer 0 you draw what all states for the symbols have in common.

In layer 1 you draw what is to be displayed extra for the symbol in state 1. In layer 2 you draw what is special for state 2 — and so on.

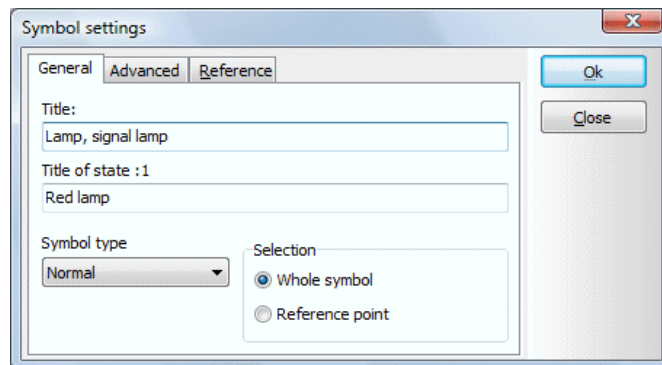


When clicking the **States to layers** button again, everything is displayed in layer 0.

Assigning titles to the states



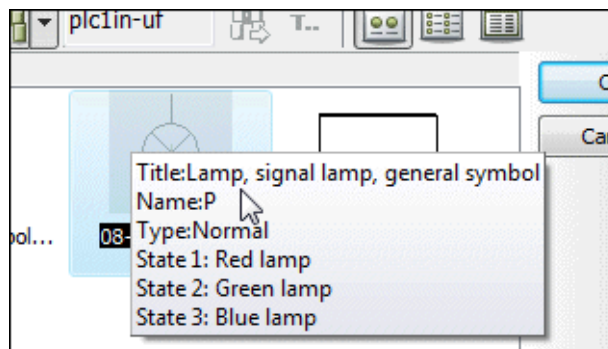
To assign a title to a state for the symbol, click the **Symbol settings** button. On the tab **General**, click in the field **Title of state:1** (displayed when you are in layer/state 1), and type in the title for this state:



To specify the title for another state, you must first change layer using the **Layer menu**, and then name the state for this layer as described above.

The state titles are displayed in the Symbol menu

When you have saved a symbol with states, you can subsequently see the names of the states in the **Symbol menu**, when you let the mouse rest over the symbol:



The states and state titles are displayed in the Component data dialog box

When you place a symbol in the project — see “Symbols with states” on page 167 — the states are ordered by state number in the dialog box **Component data**.

When you let the mouse cursor rest over one of the states, the title for the state is displayed.

States and their titles are also displayed in the **Information** dialog box for symbols.

Datafields in states

Datafields can also be placed in the individual states for a symbol. For instance you can use this to create one state where no symbol datafields are visible, and another state where some — or all — symbol datafields are visible.

— Read more about symbol datafields in “Creating symbol datafields” on page 733.

Other links

— See how to work with layers in “Layers, Elevation and Scaling” on page 477.

— See also “Symbols with states for single line diagrams” on page 408 and “Placing symbols with selected states in the pickmenu” on page 181.

— See how you from the database can control which state a symbol shall be in when placed in a the project in “Specifying symbol state for electrical symbols via the database” on page 596 and “Specifying symbol state for mechanical symbols” on page 603.

SYMBOLS WITH CONNECTION POINTS IN OTHER LAYERS

The connection points you place in the symbol definition are placed in layer 0. This makes the connection point available in the layer you place the symbol.

If you place connection points in other layers in the symbol definition, the connection points are placed in these layers — no matter which layer you place the symbol in.

— Read more about layers in “Layers, Elevation and Scaling” on page 477.

CREATING A SYMBOL AS A MULTI SYMBOL

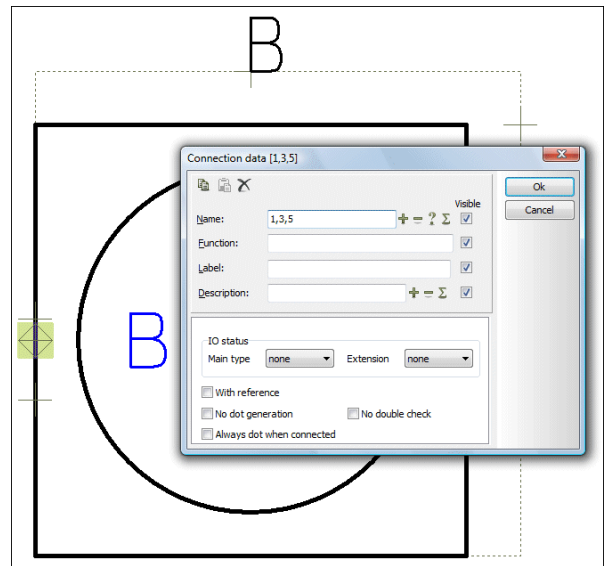
If the electrical symbols for a component have to be three independent symbols of the same kind, you can state this by saving the symbol as a so-called *multi symbol*. This facility can for instance be used for PLCs.

Enter the **Symbol menu** (press <F8>), click on the symbol you just created, and then click on **Edit symbol**.



Zoom in on the symbol, and click on the **Connection point** button.

Right-click on the connection point in the left side of the symbol, choose **Connection data** and type in *1,3,5* in the **Name** field. Click on **OK**.

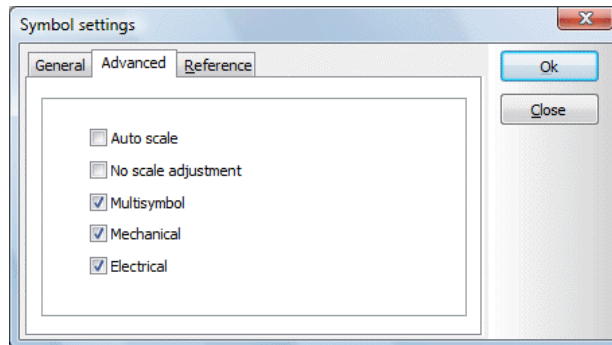


Then right-click on the connection point on the right side of the symbol, choose **Connection data**, type in *2,4,6* in the **Name** field, and click **OK**.

Hereby you state that the symbol shall consist of three independent symbols of the same name, and that the first symbol shall have the connection names **1** and **2**, the second the names **3** and **4**, and the last gets the names **5** and **6**.

Saving the multi symbol

Now choose **File => Save as**. Correct the title of the symbol if you want — please note that the **Symbol type** must remain **Normal** — and click on the tab **Advanced**.



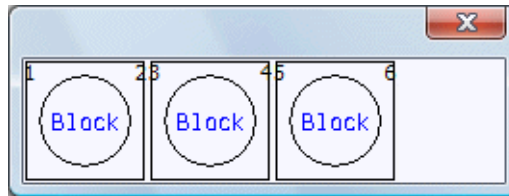
On this tab you select **Multisymbol**, and click **OK**.

Then type in the new name of the symbol — or keep the old — and click **OK**.

You have now saved the symbol as a multi symbol.

Fetching the multi symbol

When you fetch the symbol as you normally do from the **Symbol menu** (press <F8> and choose the symbol), you get a pickmenu with three symbols in it.



Click on one of the symbols, and place it on a drawing. Right-click on the symbol you have placed, and chose **Show available** to enter the pickmenu again. Read more about this in “The function Show available” on page 232.

Symbols with extra connection points

If you want one of the three symbol to have an extra connection point, you place this extra connection point on the symbol.

To determine that for instance only the third of the symbols shall have this connection point, you type in „7 in the **Name** field in its **Connection data** dialog box. Hereby you state that the two first symbols shall not have this connection point, and that the connection point in the third symbol shall have the name **7**.

If only the first of the symbols shall have the connection point, you just type in 7 in the **Name** field.

THE SYMBOL MANIPULATOR

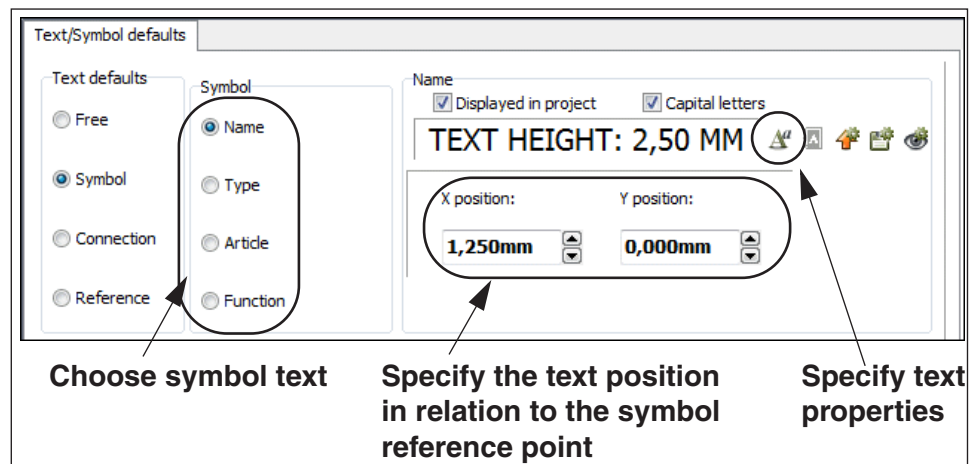
For **Automation** there also exists a so-called Symbol manipulator, which is a small software tool, that can manipulate all symbols in a symbol library. This being for instance changes in the text position, size and font.

Any use of the Symbol manipulator is at your own risk. Please remember to make a back-up, before using the Symbol manipulator. To get the Symbol manipulator, please contact PC|SCHEMATIC.

SPECIFYING DEFAULT TEXT SETTINGS FOR EDIT SYMBOL MODE

To specify where **Automation** shall position the different text reference points in relation to the symbol reference point, as well as which text properties to suggest for these texts, do the following:

- 1) When **Automation** is in the *Edit symbol* mode: Choose **Settings => Text/ Symbol defaults**.
- 2) On the tab **Text/Symbol defaults**: Click on **Symbol**.
- 3) You can then choose which symbol text to edit default settings for:



- 4) Specify the text position in relation to the symbol reference point, and click the **Text properties** button, if necessary.

- 5) In the dialog box **Text properties**: Specify the desired text properties, as described in “Specifying text properties / Appearance of text” on page 122, and click **OK**.



- 6) On the tab **Text/Symbol defaults**: Click the **Save as default** button, if the selected text settings also shall apply for other projects.

- 7) Click **OK**.

LOADING ANOTHER SYMBOL INTO THE SYMBOL

While you are designing a symbol, you have got the option to load another symbol into the symbol you are designing.



To do this, you click on the **Symbols** button, and then on the **Symbol menu** button. Please note, that when you are designing symbols, the Symbol menu button is placed to the *right* on the command tool bar.

You then enter the **Symbol menu**, click on the symbol you want to use, and click on **OK**.

You get the symbol in the cross hairs, and click where you want it placed.

When you have placed the symbol, it is no longer conceived as an independent symbol. This is because the lines are now conceived as ordinary lines in the symbol you are now designing.

The reference points and the symbol texts are not transferred in the inserted symbol, but its connection points are still connection points, and the names of the connection points remain unchanged.

You can also use **File => Open**, and find the symbol there.

Finally, you can also load a DWG or a DXF file into the symbol — see “DWG and DXF files” on page 660.

COPYING AREAS TO EDIT SYMBOL

It is possible to copy an area from a project drawing, and use this copied area when creating symbols in the Edit symbol mode.

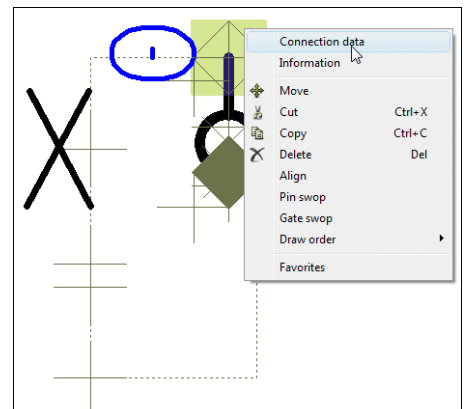
— Read more about this in “Copying areas into Edit symbol mode” on page 255.

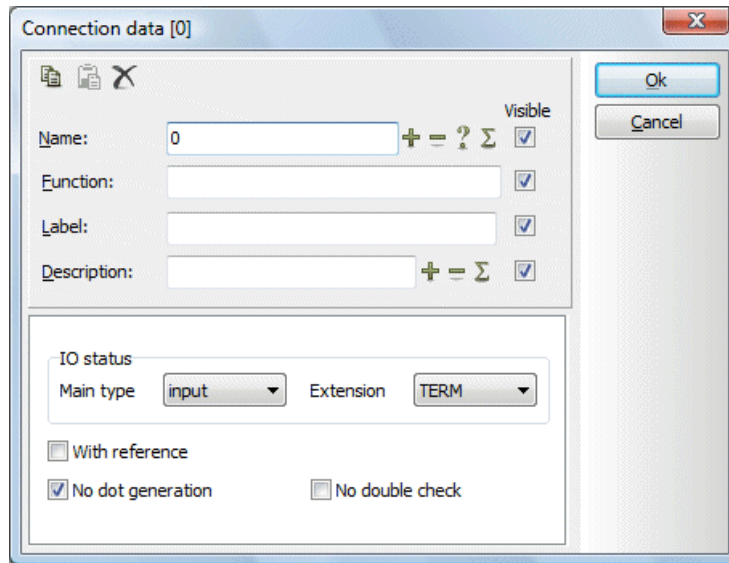
CREATING TERMINAL SYMBOLS

When you create a terminal symbol, you have to state which connection points are **Input** and which are **Output**. This you can only do *after* having placed the connection points.

When you have placed a connection point, you right-click on it, and choose **Connection data**.

By clicking on the down-arrow in the **Main type** field in the **Connection data** dialog box, you can choose whether the connection point is of the type **Input** or **Output**, which is of importance on terminals lists.





If you set a connection to the main type **Output**, the connection point will be filled out with the color red.

The type can also be changed after having placed the terminal in a project drawing. Read more about this in “Terminal symbols” on page 214.

Set the **Extension to TERM** (terminal) — see “Creating terminal symbols with additional symbol types” on page 720 also.



IDENTICAL CONNECTION NAMES FOR TERMINAL SYMBOLS

IF YOU IN THE DEFINITION OF A TERMINAL SYMBOL ASSIGN THE SAME NAME TO ALL CONNECTION NAMES, **Automation** WILL MAKE SURE THAT THESE CONNECTION NAMES REMAINS IDENTICAL, WHEN THE SYMBOL IS PLACED IN A PROJECT PAGE, AND ONE OF THE CONNECTION NAMES IS CHANGED.

— SEE “TERMINAL SYMBOLS WITH DIFFERENT CONNECTION NAMES” ON PAGE 720

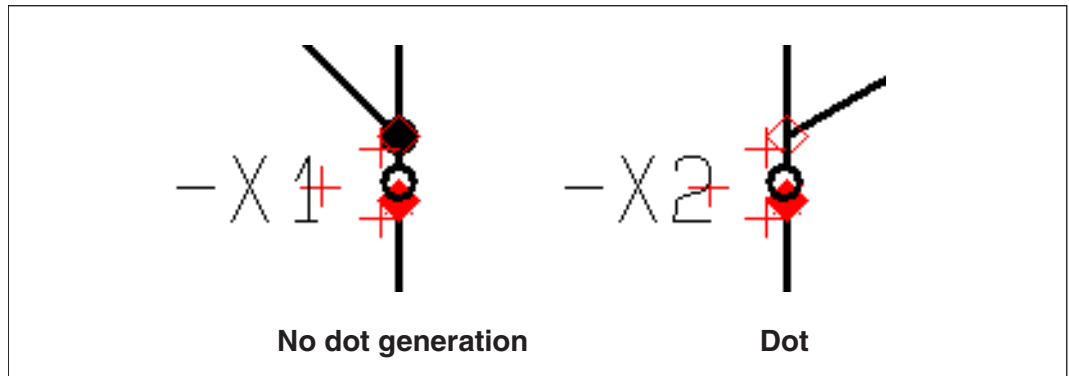
Naming the connection points

The names of both the connection points shall be set to **0**, and the **Visible** field disabled for one of the two connection names. When the connection names (terminal numbers) are inserted on the drawing, both the connection points will get the same name, but only the one will be visible.

No dot generation

When you use terminal symbols, you can let two conducting lines connect to the same connection point on the terminal.

Normally, this will result in a connection dot being placed in the connection point. If you do not want this to happen, you can select **No dot generation** in the lower part of the menu.



Save the connection symbol

Click on **OK**.

Terminal symbols are saved under the symbol type **Terminal** — see “Creating terminal symbols with additional symbol types” on page 720.

You can read more about terminal lists in “Creating a Terminals list” on page 759.

CREATING TERMINAL SYMBOLS WITH ADDITIONAL SYMBOL TYPES

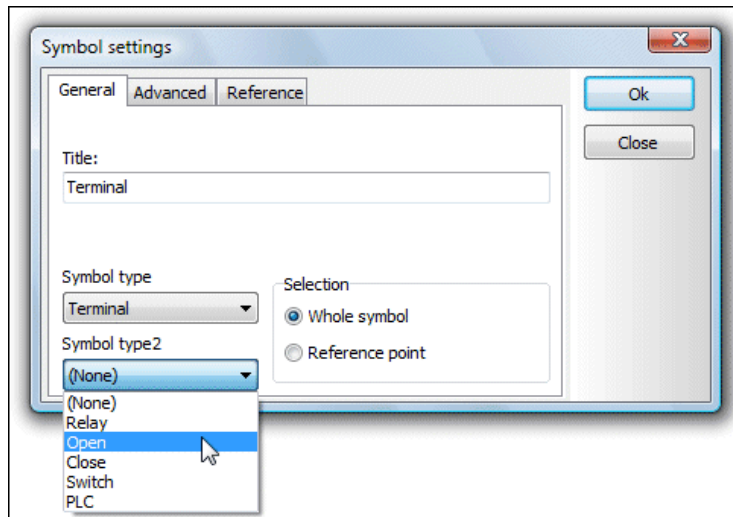
In order to support intelligent terminals and terminals with build-in functions, symbols with the symbol type **Terminal** can be assigned an additional symbol type — for instance **Terminal** and **Relay**. Terminal symbols can have the additional symbol types **Relay**, **Open**, **Close**, **Switch** or **PLC**.

Specifying additional status for terminal symbols

To specify an additional symbol type for a terminal symbol, do the following when creating the symbol in the **Edit symbol** mode:



- 1) Click the **Symbol settings** button.
- 2) In the dialog box **Symbol options**: Select the symbol type **Terminal**. You then get the option of choosing a **Symbol type2**, which is the additional symbol type for the symbol:



- 3) Select e.g. **Open** and click **OK**.

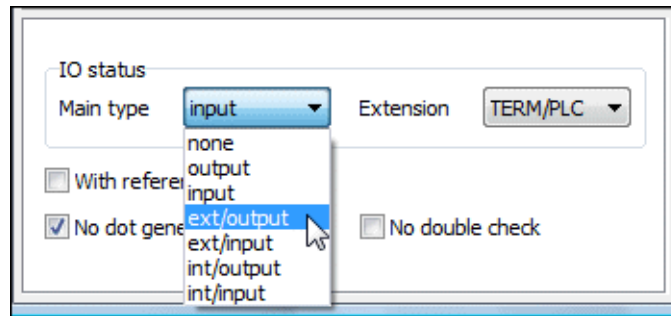
You have hereby assigned the additional symbol type **Terminal** and the additional symbol type **Open** to the symbol. The symbol will therefore function as an open contact in the diagram, with references to other symbols, and it will also be included in terminals lists.

Terminal symbols with different connection names

Because terminal symbols can behave as both terminal and relay or PLC, the connection names on all of the connections on the terminal symbols are not automatically the same. This is only the case if the symbol in **Edit symbol** is created with identical names on all connections. — See “Identical connection names for terminal symbols” on page 718.

Terminal symbols with additional symbol type PLC

For terminals with additional symbol type **PLC**, the **Extension** of the connection points must be set to **TERM/PLC**. As **Main type** you can then choose between **ext/output**, **ext/input**, **int/output** and **int/input**, which are created for this purpose.



Symbols with this additional symbol type are included in *both* terminals lists and PLC lists.

The different **Main types** under **IO status** have the following meaning:

IO STATUS MAIN TYPE	SIGNIFICANCE
NONE	THE CONNECTION POINT HAS NO MAIN TYPE. FOR INSTANCE USED FOR SUPPLY. (<i>NOT</i> AVAILABLE FOR THE EXTENSION TERM/PLC).
OUTPUT	THE CONNECTION POINT IS SET TO OUTPUT. (<i>NOT</i> AVAILABLE FOR THE EXTENSION TERM/PLC).
INPUT	THE CONNECTION POINT IS SET TO INPUT. (<i>NOT</i> AVAILABLE FOR THE EXTENSION TERM/PLC).
EXT/OUTPUT	THE CONNECTION POINT ON THE EXTERNAL SIDE OF THE TERMINAL AND THE PLC IO STATUS IS SET TO OUTPUT. (AVAILABLE FOR THE EXTENSION TERM/PLC ONLY).
EXT/INPUT	THE CONNECTION POINT ON THE EXTERNAL SIDE OF THE TERMINAL AND THE PLC IO STATUS IS SET TO INPUT. (AVAILABLE FOR THE EXTENSION TERM/PLC ONLY).
INT/OUTPUT	THE CONNECTION POINT ON THE INTERNAL SIDE OF THE TERMINAL AND THE PLC IO STATUS IS SET TO OUTPUT. (AVAILABLE FOR THE EXTENSION TERM/PLC ONLY).
INT/INPUT	THE CONNECTION POINT ON THE INTERNAL SIDE OF THE TERMINAL AND THE PLC IO STATUS IS SET TO INPUT. (AVAILABLE FOR THE EXTENSION TERM/PLC ONLY).

— See “Creating PLC I/O symbols” on page 725 also.

Specifying terminal additional symbol type via the database

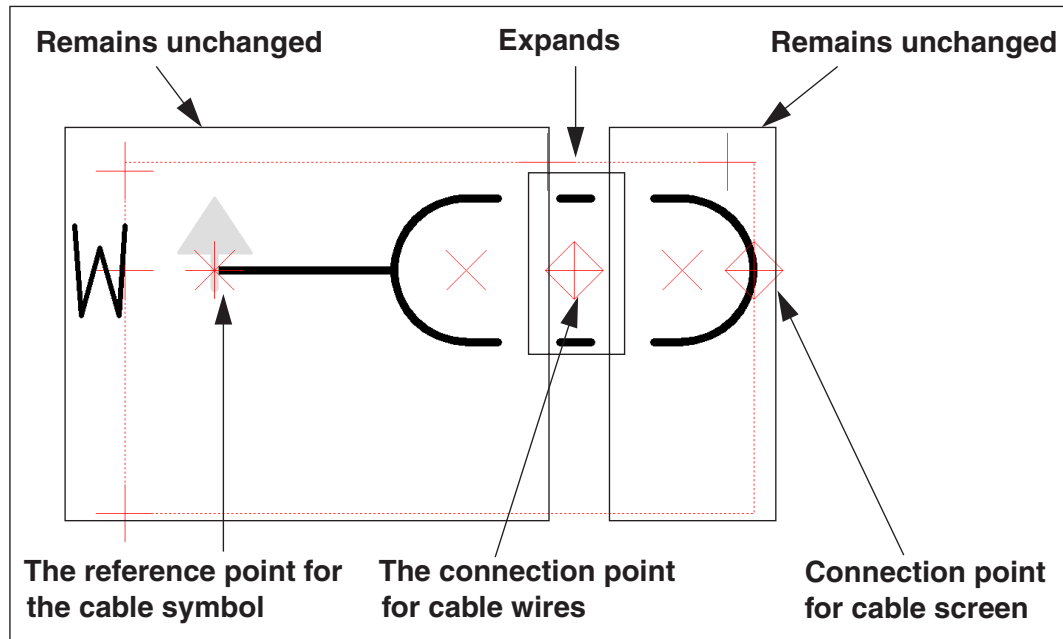
— See how to specify the additional symbol type for terminals via the database in “Specifying the symbol type via the database” on page 594, “Specifying symbol type for terminals with additional symbol type” on page 594 in particular.

CREATING CABLE SYMBOLS

Cable symbols are created as described above. However, when you design cable symbols, you only place one connection point, even though the cable for instance consist of seven wires.

— See “Creating busbar symbols” on page 724.

When you use the cable symbol on a project diagram select which wires are included into the cable, the program automatically create new connection points for each wire you add.



The connection point you place in the cable definition states the location of the first cable wire in the cable. As you add more cable wires to the cable on a diagram, the cable symbol expands.

When you design the cable symbol, it is important what you place/draw to the *left*, and what you place/draw to the *right* of the *connection point* for the cable wires.

To the *left* of the connection point you place the reference point for the cable symbol, and draw the lines you need at the start of the cable symbol.

To the *right* you draw what you need at the end of the cable symbol.

Lines going from the left side of the connection point to the right side of the connection point are *expanded* as you add new cable wires to the cable on the diagram. This means that the dotted line on the figure above will expand as you connect new cable wires to the cable on a diagram.

Connection point for cable screen

If you place two connection points for the cable symbol, the right-most connection point will be conceived as a connection point for a cable screen.

Indicate a cable wire is connected to the cable

When you connect a cable wire to the cable on a diagram, the lines in the middle part of the cable definition expands to include those, but no sign is displayed where the cable wire meets the connection point of the cable symbol.

If you want such a sign, you can place the **Function** text of the connection point on top of the connection point. Then type in the **Function** text "/" and choose the alignment *centre-centre*. — See “Specifying text properties / Appearance of text” on page 122 for further details.

Direction of the Cable symbol

In the symbol you design, *to direction* is facing upwards. If you need a small arrow displaying the direction of the cable, you have to draw it yourself. If you use the color **NP** for this arrow, it will be displayed on screen only, and not on print.

Saving the Cable symbol

When you save the symbol, you have to set the symbol type to **Cable** in the **Symbol options** menu. — See e.g. “Symbol types” on page 706.

— See how to work with cable symbols in “Working with cable symbols” on page 218.

Specifying Color codes for Cable symbols

When you are working with cable symbols you can let **Automation** control the color codes for the wires in the cable.

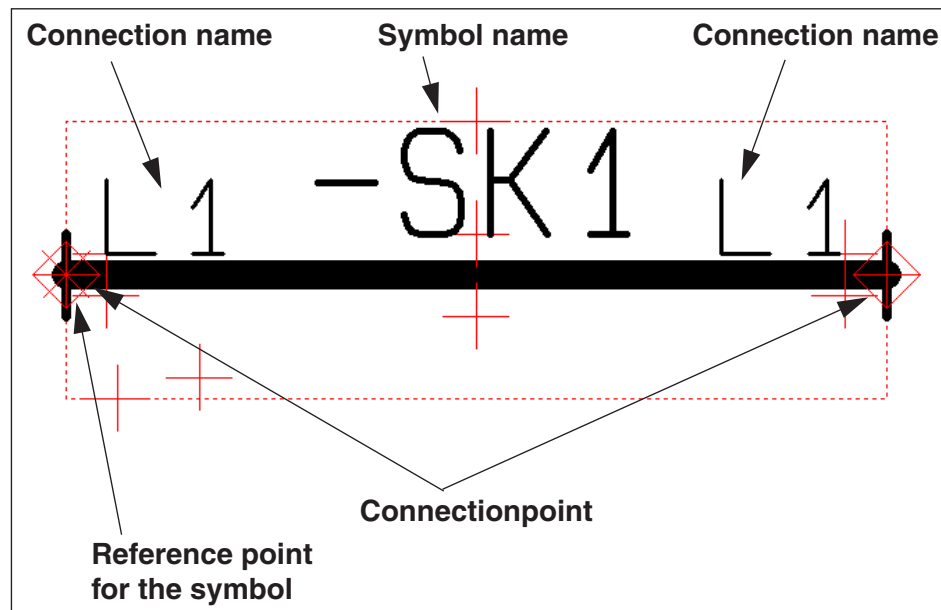
You have to specify these color codes in the record for the cable in the database. This is done like you specify connection names for symbols in general. Please refer to the **PC|SCHEMATIC Database** manual for further details.

Most used cable direction

For setting the most used cable direction, please refer to “Default Cable direction” on page 225.

CREATING BUSBAR SYMBOLS

When you create a busbar symbol, you must place two connection points, connect these with a line, and save the symbol with the symbol type *busbar*.



The connection name is applied for the name of the potential, which the busbar is connected to. Both connection points must be assigned the same connection name (**L1** in the figure above).

Please note: At the present moment there is no specific standard describing how busbar symbols should appear in electrical documentation.

Busbar symbols in projects

When you place a busbar symbol in a diagram, you can stretch it and move it just like an ordinary line.

Busbar symbol facts

There are the following advantages for using busbar symbols in stead of drawing electrical potentials:

- The busbars will appear as busbars in e.g. parts lists
- Wire numbers are not placed on busbars
- The busbars are also included in graphical plans

CREATING PLC I/O SYMBOLS

— Read about PLC I/O symbols and their use in “Working with PLCs” on page 418, “Workflow for PLCs from the database” on page 420 and “PLC data on the I/O symbol connection points” on page 425.

PLC I/O symbols are designed as other symbols — see “Creating Symbols” on page 696.

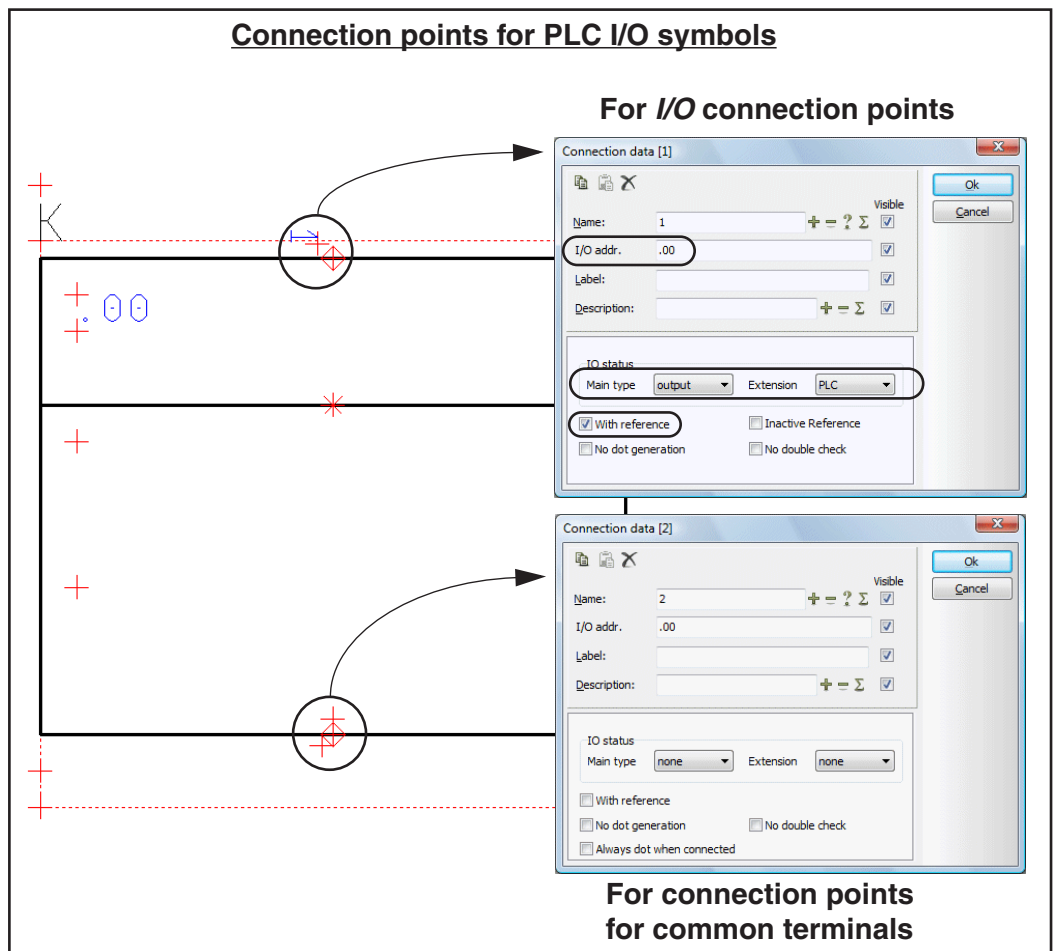
When designing PLC I/O symbols, you must furthermore do the following:

Activate With reference for PLC I/O symbol input/output connection points

When you place the connection points representing inputs/outputs on the PLC I/O symbols, you must enter the dialog box **Connection data** (e.g. by double-clicking on the connection points) and select **With reference**.

Hereby an independent reference text **Ref** is attached to the connection point. This text you can subsequently place an appropriate place in the symbol definition — see “PLC I/O symbols placed in projects” on page 727.

When you have placed the created PLC I/O symbol in the project, the **Ref** text will contain a reference to this connection point in the PLC reference symbol for the PLC in the project. — See “Jumping between PLC ref. symbols and PLC I/O symbols” on page 427.



Connection points for common terminals: Do not select With reference
For connection points with common terminals (supply/ground) you shall *not* select **With reference**.

IO status for connection points on PLC I/O symbols

Set the **Main type** for input and output connection points to *input* or *output*, depending on whether they shall refer to inputs or outputs. Furthermore, set the **Extension** for these connection points to *PLC*.

— Please refer to the figure on page 725.

Connection points for common terminals shall not be assigned Main type or Extension

For connection points for common terminals (supply/ground), both **Main type** and **Extension** shall be set to *none*.

I/O addr. in the symbol definition can specify pairs of connection points

In the field **I/O addr.** you specify an address for the connection point. This address has two functions:

- 1) The program understands connection points with the same **I/O addr.** in the symbol definition as belonging together. When you draw a PLC I/O symbol with e.g. eight connection points, which are addressed in pairs, you can therefore give the pairs the same I/O address.
- 2) When placing the PLC I/O symbol directly in the project — without first having addressed the PLC via a PLC reference symbol —, the connection point will be assigned the address which you specify here.

Please note, that the field **I/O addr.** is called **Function** for a start, and will not get the name **I/O addr.** until you have set IO status for the connection point, as described in “IO status for connection points on PLC I/O symbols” on page 726.

— Please refer to the figure on page 725.

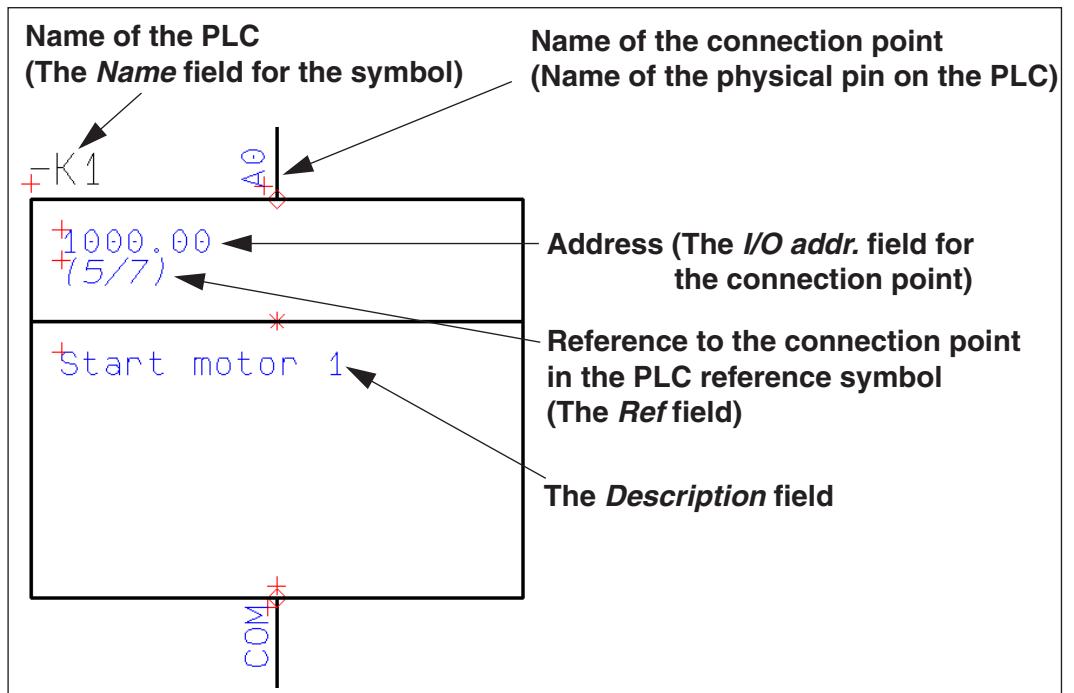
Save the PLC I/O symbol with the symbol type PLC

When you save the PLC I/O symbol — see “Saving the symbol” on page 705 and “Symbol types” on page 706 —, you must give it the symbol type **PLC**.

When you use the symbol in projects, you can therefore apply all of the automatic PLC functions in **Automation**, and the symbol will furthermore be included in PLC lists.

PLC I/O symbols placed in projects

When you have placed the PLC I/O symbol in the project, it can for instance look like this:



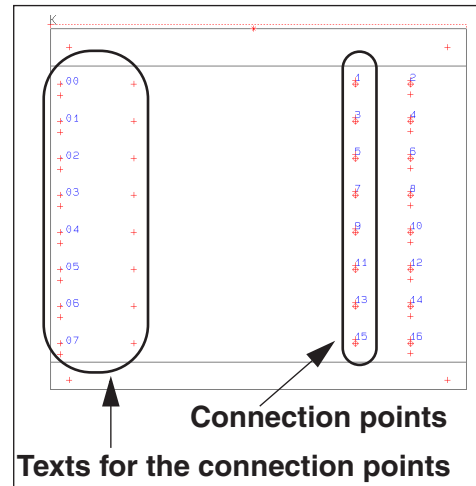
— See also “PLC data on the I/O symbol connection points” on page 425 and “Jumping between PLC ref. symbols and PLC I/O symbols” on page 427.

CREATING PLC REFERENCE SYMBOLS

— Read about PLC reference symbols and their use in “On PLC reference symbols” on page 418.

PLC reference symbols are designed as described in “Creating Symbols” on page 696.

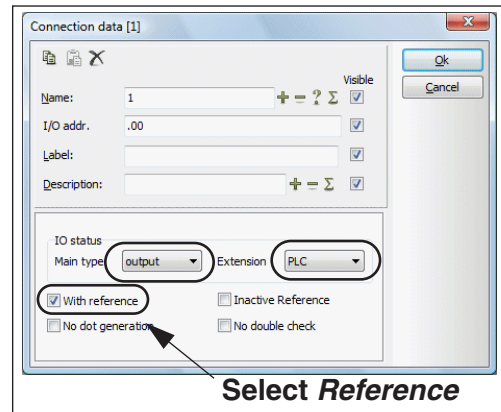
When designing PLC reference symbols, you must furthermore do the following:



Activate *With reference* for the PLC symbol I/O connection points

When you place the PLC connection points, you must enter the dialog box **Connection data** (e.g. by double-clicking on the connection point) and select ***With reference***.

This places a reference text ***Ref*** attached to the connection point. This reference text refers to connection points on PLC I/O symbols with the same name and connection point in the project. — See “Creating PLC I/O symbols” on page 725.



When you have placed the created PLC reference symbol in the project, and subsequently place the PLC I/O symbol with the connection name **1**, there will be placed a reference to the position of the PLC I/O symbol next to connection name **1** in the project. — See “Jumping between PLC ref. symbols and PLC I/O symbols” on page 427.

You shall only choose ***With reference*** for the I/O connection points.

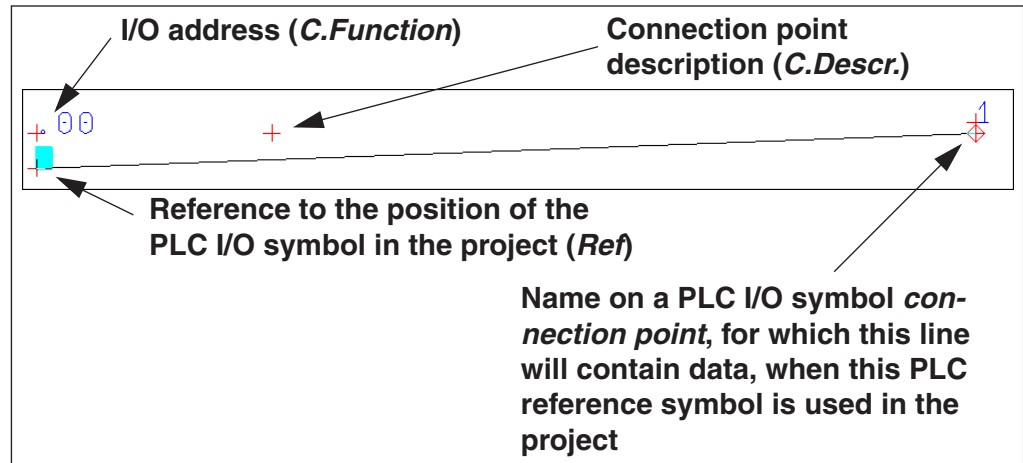
I/O status for PLC reference symbol I/O connection points

Set the **Main type** for the connection points to *output* or *input*, depending on whether the connection points shall refer to an input or an output.

Furthermore, set the **Extension** for the connection points to *PLC*.

PLC reference symbol contains connection data for I/O symbols

Data for the PLC inputs/outputs are attached to the *connection points* on the PLC I/O symbols — see “PLC data on the I/O symbol connection points” on page 425. In this example, the PLC reference symbol consists of lines with data from these connection points:



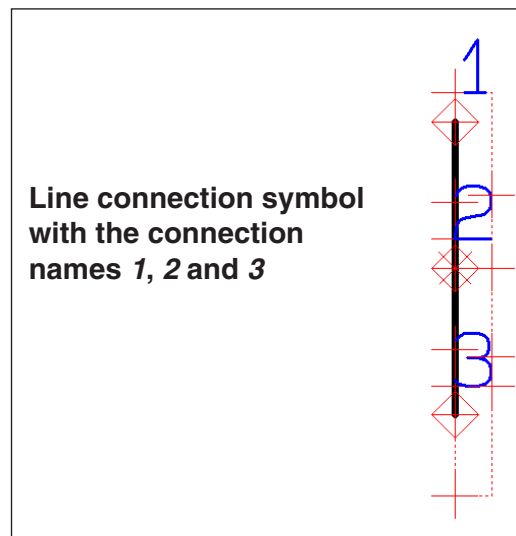
Save the PLC reference symbol with the type PLC reference

When saving the PLC reference symbol — see “Saving the symbol” on page 705 and “Symbol types” on page 706 —, you must save it with the symbol type **PLC reference**.

CREATING LINE CONNECTION SYMBOLS

The symbol type **Line connection symbol** is applied in connection with placing sub drawings using drag'n draw and in connection with sub drawings in the **Project Generator**. — See how to apply the symbol in praxis in “Sub drawings with intelligent line connection” on page 287. See the **Project generator** manual also.

The symbol is applied to specify to which wire in the diagram to connect a wire in the sub drawing, when the sub drawing is placed in a project page. The individual connection points on the symbol can for instance have the connection names **1**, **2** and **3**.



Please remember to save the symbol using the symbol type **Line connection**.

In the **MISC** folder you can find the line connection symbols **LineConnect3.sym** and **LineConnect4.sym**.

OTHER OPTIONS

Deleting symbols from folder/library

To delete a symbol from the **Symbol menu**, you have to enter the **Symbol menu** (press <F8>), right-click on the symbol you want to delete, and choose **Delete symbol** in the menu that pops up.

Changing a symbol in a folder/library



To change a symbol, you have to enter the **Symbol menu** (press <F8>), click on the symbol you want to change, and then click on the **Edit symbol** button.

Or you can right-click on the symbol you want to change, and choose **Edit symbol**.

You can now change the symbol, and change it under the same name, or under a new name.

Changing the symbol type

When a symbol is being used in a project, you can change the symbol type by right-clicking on the symbol. A menu pops up, you click on **Component data**, and then click on the down-arrow in the **Symbol type** field, which gives you a list of the possible symbol types. Pick your choice, and click **OK**.

DWG/DXF files and symbols

It is an option to save a symbol as a DWG/DXF file, as well as to load a DWG/DXF file as a symbol in **Automation**.

— See how to do this in “DWG and DXF files” on page 660.

CREATING DATA SYMBOLS

If you want to attach some extra information to a symbol you have placed in a project, you can do this by using another symbol, called a *data symbol*.

This extra information can subsequently be included into lists.

THIS CHAPTER DESCRIBES	
— CREATING YOUR OWN DATA SYMBOL	P. 732
— DESIGNING THE DATA SYMBOL	P. 733
— PLACING THE DATA SYMBOL IN A PROJECT	P. 734
— SYMBOL DATAFIELDS WITH PREDEFINED VALUES	P. 736
— DATA SYMBOLS AND LISTS.....	P. 736
— DATA SYMBOLS AND COMPONENTS LISTS FILES	P. 737

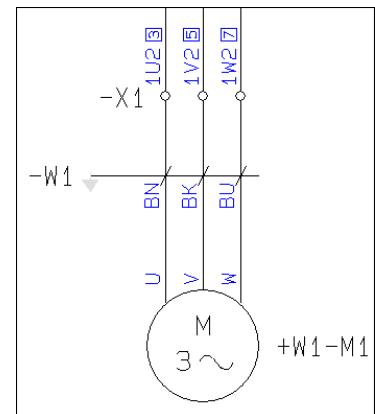
Please refer to “Creating Symbols” on page 696 for general information about how to create symbols.

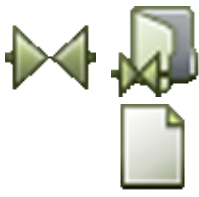
Please note that it is also possible to insert extra datafields in the definition of a symbol — see “Extra symbol Datafields in symbols” on page 711.

CREATING YOUR OWN DATA SYMBOL

In a project you use a motor, and for this motor you would like to have information about how many windings it has, as well as how many kW it is on included on your drawing and on your lists.

However, you cannot enter this information in the **Component data** menu, and you therefore have to attach them to the symbol otherwise. This is done by creating a data symbol. This symbol can for instance be drawn as a table you can fill out — as the example on page 733.





DESIGNING THE DATA SYMBOL

To design the data symbol click on the **Symbols** button and on **Symbol menu** — or press [F8> to enter the **Symbol menu**. Here you click on the **Create symbol** button.

Drawing the symbol

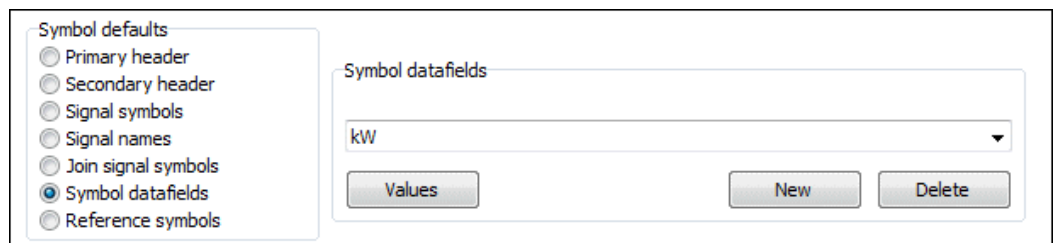


Click on the **Lines** and the **Rectangle** button, as well as the **Pencil**. Draw a rectangle like the one you can see on page 733, then click on **Straight lines** and draw the horizontal line in the middle of the symbol. — How to draw lines is described in detail in “Lines” on page 72.

Creating symbol datafields

Now you have to create two symbol datafields, to contain the information you want to insert into the data symbol.

Therefore you choose **Settings => Text/Symbol defaults**, and click on **Symbol datafields** in the left-hand side of the menu.



Here you click on **Add**, type in *Windings* in the appearing menu, and click on **OK**. Click on **Add** again, type in *kW*: and click on **OK**. You have now created the two symbol datafields **Windings** and **kW**.

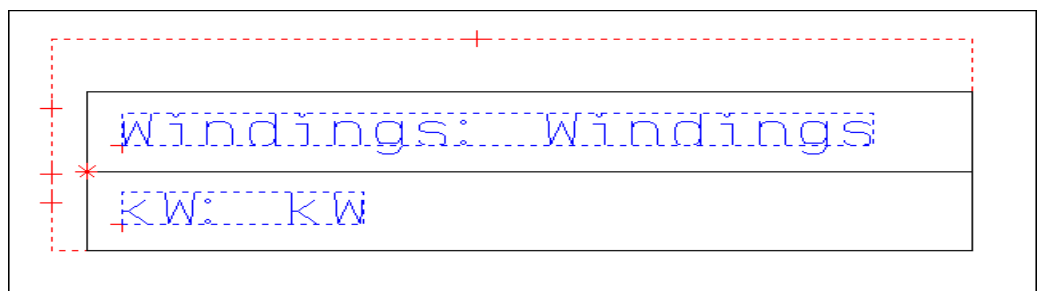
You can also create a list of predefined values for the datafields, as described in “Symbol datafields with predefined values” on page 736.

Click **OK** to exit the dialog box.

— Please note you do not have to be designing a symbol to create the symbol datafields.

Placing the symbol datafields in the data symbol

To place the symbol datafields in the data symbol, choose **Insert => Insert datafield**.



In the **Datafield** dialog box you click on **Symbol Datafield**, choose **Windings** and click on the red down-arrow in the lower part of the menu. Place a colon (:) and a space character after the transferred text in the **Text** field, and set the **Width** to 20.



You can also adapt the text size by clicking on the **Text properties** button. Read more about this in “Specifying text properties / Appearance of text” on page 122.

Place the text in the upper part of the rectangle, as displayed on the drawing.
Insert the datafield **kW** correspondingly.

Moving the symbol name

Click on the reference point for the text **Symbol name**, and place it above the data symbol by *dragging*.

Saving the symbol

You have now completed the creation of the data symbol, and are ready to save it. Click on **Save**, and you enter the dialog box **symbol settings**: Set **Selection** to **Whole symbol**, and set **Symbol type** to **Data**.

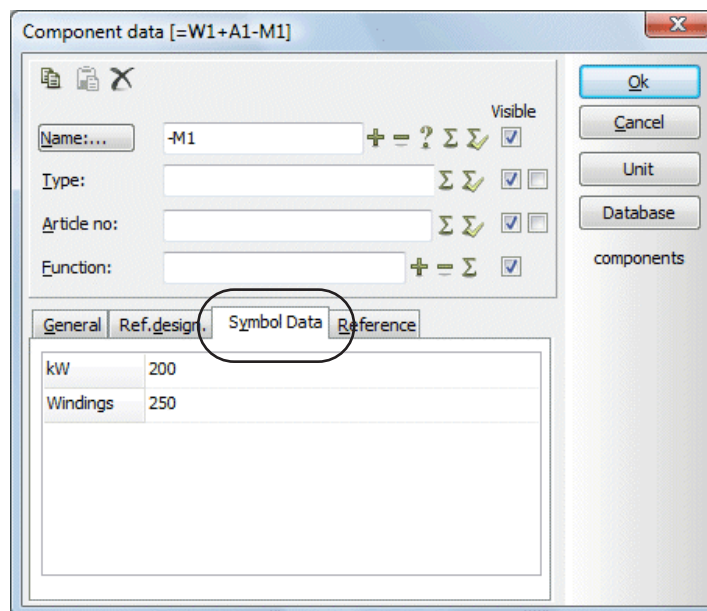
Click on **OK**, assign a name to the symbol in the following menu, and click **OK** again.

Now leave the Design symbol mode by choosing **File => Close**, and leave the **Symbol menu** by clicking on **Cancel**.

PLACING THE DATA SYMBOL IN THE PROJECT

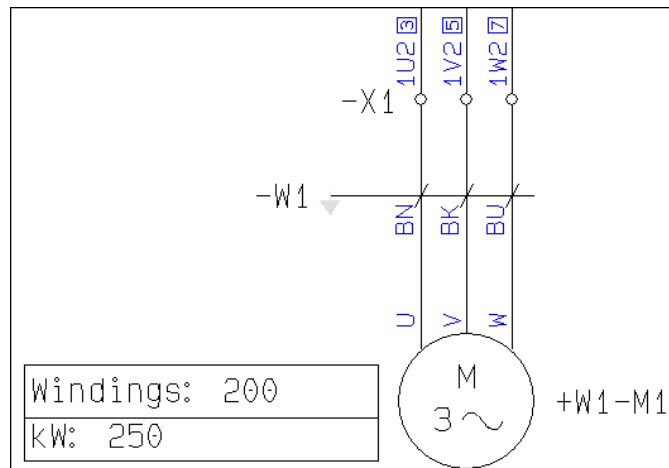
Press on [F8> to enter the **Symbol menu**, where you click on the data symbol and click **OK**. Returning to the project drawing, with the data symbol in the cross hairs, you place the symbol next to the motor symbol.

When you click to place the data symbol, you automatically enter this dialog box:



By assigning the name **+W1-M1** to the data symbol, you tell the program that the information in this data symbol, is attached to the symbol with the name **+W1-M1**.

Fill out the data symbol fields, click on **OK**, and place the symbol.

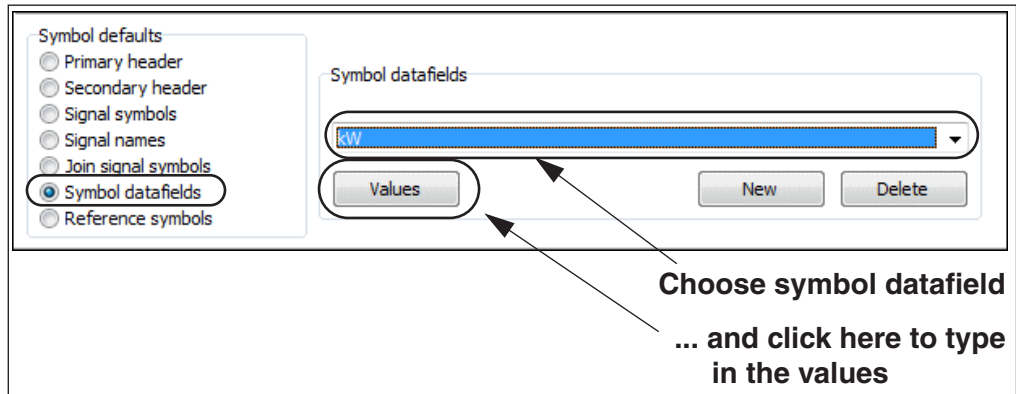


If the data symbol does not have an appropriate size compared to the motor symbol, you can scale it using the symbol scale field in the symbol tool bar. Read more about this in “Scaling symbols” on page 176.

SYMBOL DATAFIELDS WITH PREDEFINED VALUES

Symbol datafields can have a list of predefined values, consisting of typed in values for the datafield.

To create such a list of predefined values, choose **Settings => Text/Symbol defaults** and click on **Symbol datafields**. Then select the relevant symbol datafield, and click on **Values**.

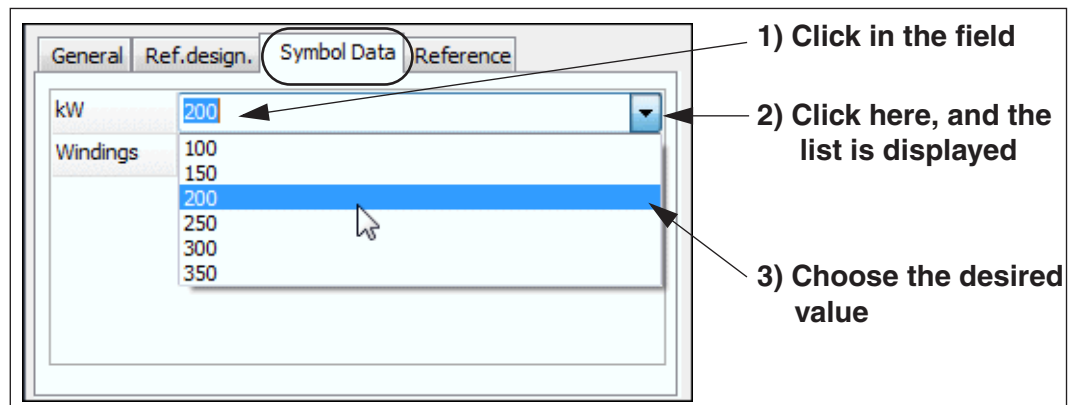


You then enter a dialog box, where you type in the values, and click **OK**.

Placing symbols containing symbol datafields

When you place a symbol containing a symbol datafield, the tab **Symbol data** is added to the dialog box **Component data**.

Here you click in the text field for the relevant datafield (1, below), and a down-arrow is displayed. When you click on the down-arrow (2), a list of the symbol data values is displayed (3).



SYMBOL DATAFIELDS AND LISTS

To include the information from the symbol datafields in your lists, you must — when you design the list — choose **Insert => Insert Datafield**, click on **Symbol Datafields**, and insert the symbol datafields you need. Read more about designing lists in “Creating Lists” on page 752.

This way you can for instance create a components list for the motors in the project, containing information about their windings and effect. Read more about sorting and list criteria in “List settings” on page 542.

DATA SYMBOLS AND COMPONENTS LIST FILES

When you create components lists files, symbol datafields for the applied data symbols — and symbol datafields from any other symbols — can be included in these lists.

— Read more about components list files in “Exporting List Files” on page 649, specially “Specifying the contents of the file” on page 654.



SYMBOL DATAFIELDS IN ALL SYMBOLS

YOU CAN INSERT SYMBOL DATAFIELDS IN ANY TYPE OF SYMBOLS IN **Automation.**

CREATING SIGNAL SYMBOLS

A special type of symbols are called *signal symbols*. They are used when you start or end a conducting line in a non-electrical point on the drawing.

THIS CHAPTER DESCRIBES	
— CREATING YOUR OWN SIGNAL SYMBOL	P. 738
— DESIGNING THE SIGNAL SYMBOL	P. 738
— FREQUENTLY USED SIGNAL SYMBOLS	P. 740
— FREQUENTLY USED SIGNAL SYMBOL NAMES	P. 741
— CREATING JOIN SYMBOLS	P. 741

Read more about signal symbols and how to use them in your projects in “Signal symbols” on page 203.

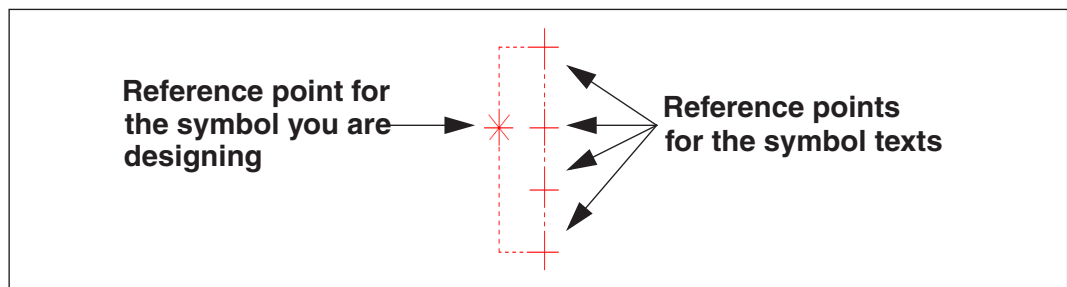
This chapter is only a brief introduction on how to design your own signal symbols. If you find it too brief, please read “Creating Symbols” on page 696, where you get a more thorough example on how to design symbols.

CREATING YOUR OWN SIGNAL SYMBOL



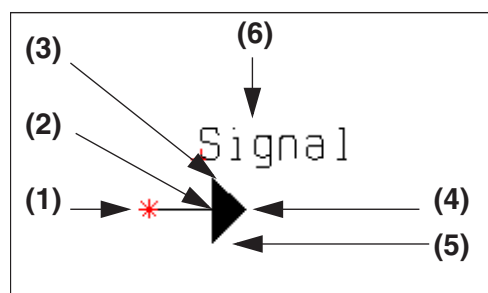
Click on the **Symbols** button and then on the **Symbol menu** button. In the **Symbol menu** you click on the **Create new symbol** button.

You now enter the Design symbol mode, and see these reference points on the middle of the screen.



DESIGNING THE SIGNAL SYMBOL

You can now design a signal symbol with the following appearance:



A right-arrow on a total of 7.5mm.

The arrowhead itself is on 45 degrees, and 5mm high.

In the name field, it could for instance say *Signal*, but you do not have to type in anything in this field, because the signal name will be filled out when you place the signal symbol on a diagram.



Drawing the signal symbol

Make an appropriate **Zoom** <z> around the symbol reference point. Click on the **Reference point** button as well as the **Pencil**, and click where you want the reference point located (1).



Click on the **Lines** button as well as the **Follow connected** button. Draw a horizontal line on 5mm (2) starting from the reference point. If you are using the 2.5mm standard snap, you do this by pressing two times on the right-arrow key. Click on the mouse and press <Esc>.

— Read more about snap in “Snap” on page 47.



Click on the **Filled areas** button, click 2.5mm above the terminal point of the line (3), click 2.5mm in extension of the horizontal line, then 2.5mm down to the left (5), in order to get a filled triangle. (The program automatically choose a slanted line of 45 degrees — if that is not so, please read more about **Ortho 45** in “Pointer/Screen” on page 836).



Click on the **Texts** button, click on the text reference point for **S.Name**, click on the **Move** button, press down <Shift> to get a finer snap, and place the text field.

Click in the white **Text** field in the texts tool bar on the top of the screen — or press <k> — and type in *Signal* and press <Enter> (6).

Text fields in signal symbols

When you save a symbol as a signal symbol, the symbol will only contain the symbol texts *Symbol Name* and *Symbol Type*.



You can see this by clicking on the **Symbols** button, and then on the **Symbol settings** button. Click on the down-arrow in the **Symbol type** field, choose **Signal** in the drop down list, and click on **OK**.

This makes the other symbol texts disappear, which also happens when you save the symbol as a signal symbol.

You should not type in anything in the *Symbol Type* field, because this field is used to contain the page reference for the signal symbol. Therefore place the symbol type field where you want the reference text to be located.

You will typically not specify a symbol name when you create a signal symbol, because you normally do not determine the name of a signal symbol until you place it in a project. — Read more about which names will be suggested in “Frequently used signal symbol Names” on page 741.

However, if you do not specify a name when you place a signal symbol in a project, the signal symbol will get the name specified in the *Symbol name* field.

Saving the symbol

Choose **File => Save as**, and assign a title to the symbol. If you have not already done this, you choose the symbol type **Signal** and click on **OK**.

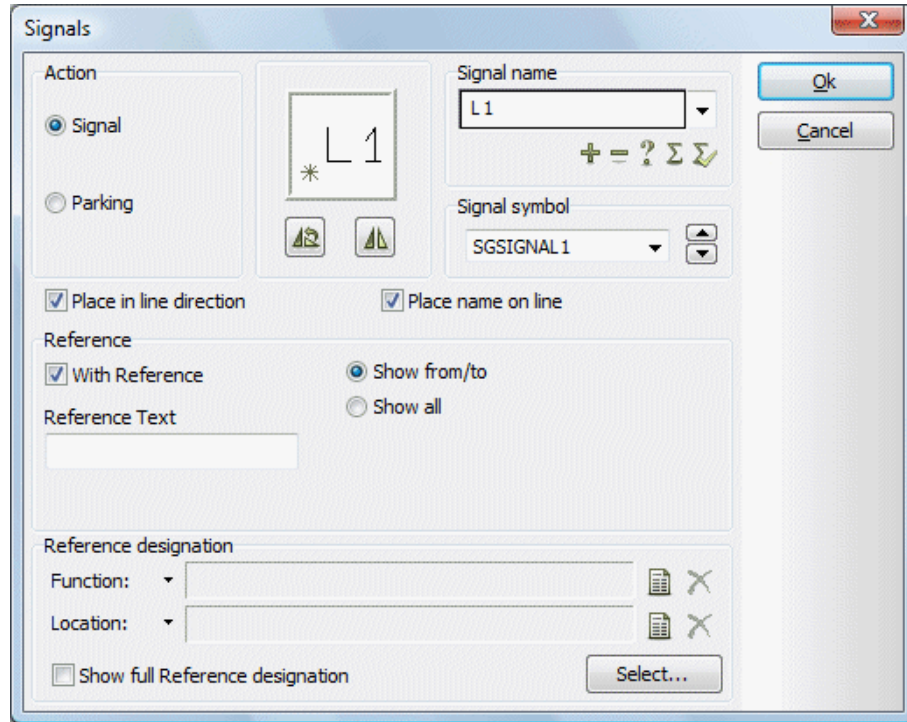
Then save the symbol, as described in “Saving the symbol” on page 709. Remember — in the **Save as** dialog box — to determine which folder you want to save the symbol in.

All **Automation** signal symbols are saved in the **MISC** folder, having names starting with SG.

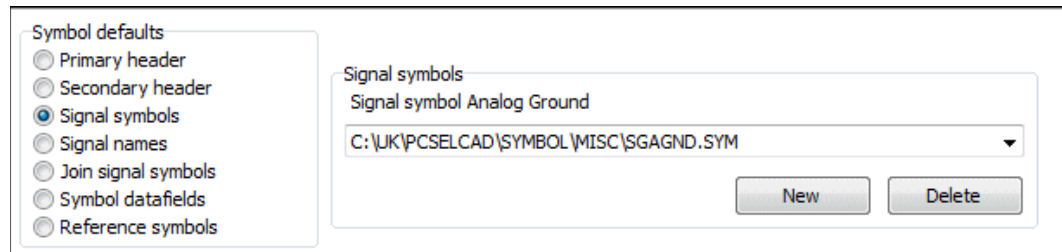
FREQUENTLY USED SIGNAL SYMBOLS

Signal symbols you use often, can be added to the list of signal symbols, that appear when you click on the up- and down-arrows below **Signal symbol** in the dialog box **Signals**, which you can see below.

You enter this dialog box automatically when you start or end a conducting line in a non electrical point in a diagram.



To add signal symbols to this list, you choose **Settings => Text/Symbol defaults**.



Here you click on **Signal symbols** in the lower-left corner of the dialog box, and click on **New**.

You now enter the **Symbol menu**, where you find and select the desired signal symbol, and click on **OK**.

Please remember to exit **Settings** by clicking **OK**.

The signal symbol is now added to the list.

If you click on **Delete** instead of **New**, you can remove signal symbols from the list.

FREQUENTLY USED SIGNAL SYMBOL NAMES

To add a signal name to the list of predefined signal names, you choose **Settings => Text/Symbol defaults**.

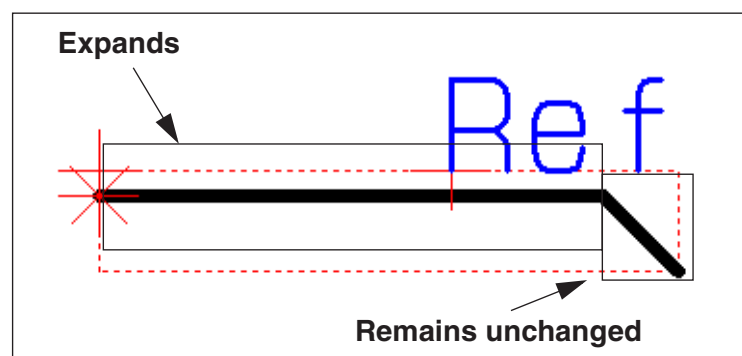
Here you click on **Signal names** and click on **Add**.

In the appearing dialog box you type in the new name, and click on **OK**.

If you want to remove a signal name, you click on the down-arrow in the field below **Signal name**, and click on the signal name you want to remove. Click on **Remove**, and the signal name is removed.

CREATING JOIN SYMBOLS

When you create join symbols, this follows the same philosophy as for creating cable symbols — see “Creating cable symbols” on page 722.



In join symbols you have to place the reference point to the left, and draw the symbol to the right.

When you place the join symbol on a diagram page, only the horizontal lines in the symbol definition are expanded. Furthermore these lines need to be on the same line as the connection point in order to be expanded.

Other lines in the symbol definition are moved according to the expansion of the horizontal line, but remain unchanged.

When you save the join symbol, choose the **Symbol type Signal**.

CREATING DRAWING HEADERS

In this chapter you can see how to create your own drawing header.

THIS CHAPTER DESCRIBES	
— WHAT IS A DRAWING HEADER.....	P. 743
— INITIAL PREPARATIONS BEFORE CREATING A DRAWING HEADER	P. 743
— DESIGNING THE DRAWING HEADER	P. 745
— ADDING THE DRAWING HEADER TO THE LIST OF DRAWING HEADERS	P. 750
— REMOVING DRAWING HEADERS FROM THE LIST OF DRAWING HEADERS	P. 750
— PREDEFINED DRAWING HEADERS	P. 751
— PLACING YOUR COMPANY LOGO ON A DRAWING HEADER	P. 751
— CHANGING DATAFIELDS IN PAGE DATA AND PROJECT DATA.....	P. 751

Please refer to “Changing an already created list/drawing header” on page 763 to see how to change one of the predefined drawing headers for your own use.

See how to place a drawing header on a project page in “Insert drawing headers/formulas” on page 531.



INSERTING DRAWING HEADERS ON TEMPLATES

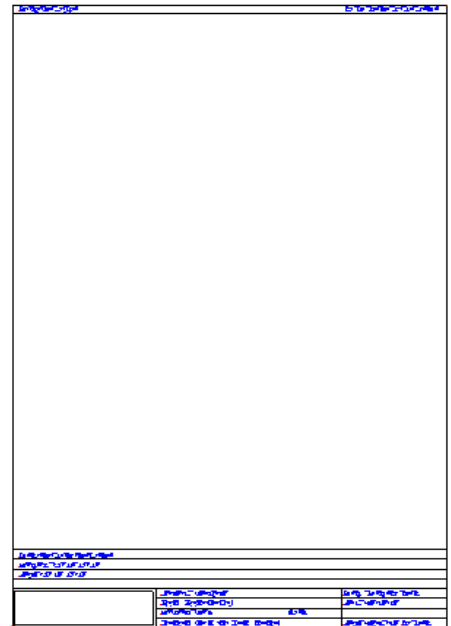
WHEN YOU HAVE CREATED A DRAWING HEADER, YOU CAN INSERT IT ON A TEMPLATE PAGE, AS DESCRIBED IN “CREATING TEMPLATES” ON PAGE 771, SPECIALLY “CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR” ON PAGE 772.

WHAT IS A DRAWING HEADER?

A drawing header is a kind of formula containing text and graphics, which you can choose to place on your project pages. For instance can a drawing header consist of a frame around the entire page, with your company name and logo placed in the lower-right corner. Apart from this the drawing header can contain information about the name and number of the individual page, which customer the project is carried out for — and much more.

Once you have created a drawing header, you can make the program fill out this information for you automatically.

This chapter contains an example on how to design your own drawing header.



INITIAL PREPARATIONS BEFORE CREATING A DRAWING HEADER

Creating a new symbol



When you create a drawing header you have to be working in a project, so if that is not the case you have to open a new project: Click on the **New document** button, and then leave the **Settings** dialog box by pressing <Esc>.



Click on the **Symbols** button and on **Symbol menu** — or press <F8> to enter the **Symbol menu**, where you click on **Create new symbol** button.

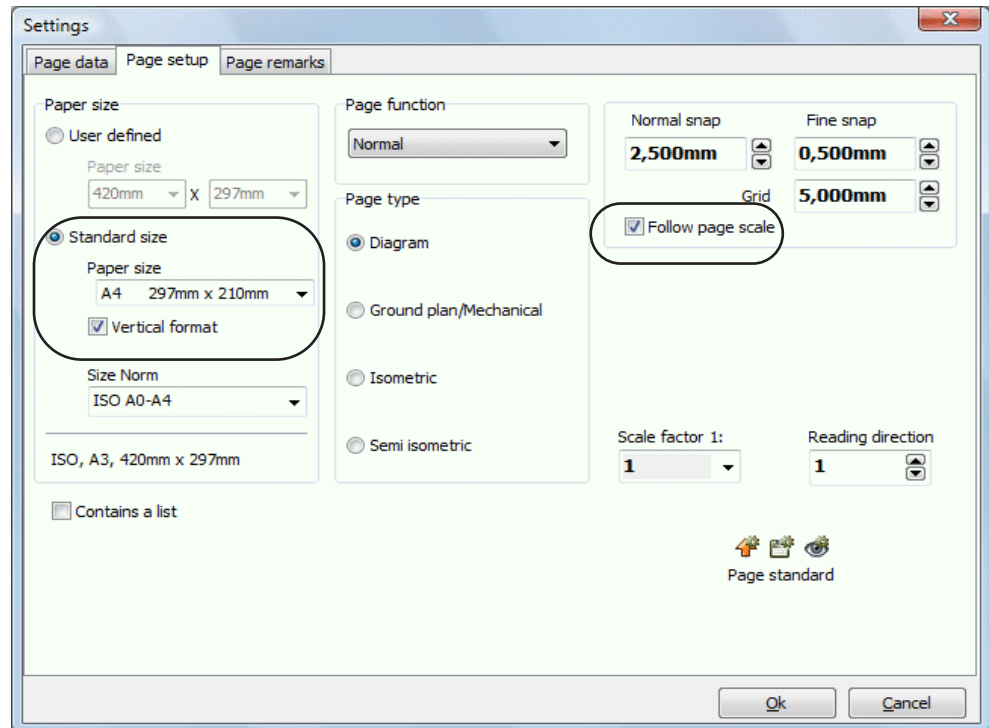
The program is now in the Design symbols mode. As long as you are here, **SYMB** will flash on a red background in the lower part of the left tool bar.

Setting up the page to vertical A4 format

The next thing you do is to make sure that the program is set to the correct page format — in this case a vertical A4 page. The page type shall also be correct, and because you are going to create a drawing header, the type must be **Diagram**:

- 1) Choose **Settings => Page setup**. — As a short-cut to enter this dialog box, you can click on the field in the lower part of the left tool bar, where the page size is displayed.
- 2) On the tab **Page setup**: Click on **Standard size**, choose the **A4** format, and select **Vertical format**. Set **Grid** to **5.00mm**. — Read more about this tab in

“Page setup” on page 843, and about *grid* and *snap* in “Screen / Image Functions” on page 42.



3) Click **OK**.

Help frame displayed on the page

Choose **Settings => Pointer/Screen**.

Click on **Help frame** in the right hand side of the dialog box.

The help frame displays the margin which is specified as the most common used standard, and is therefore only meant as a guideline for where to draw on the page. The frame is displayed with a dashed red line.

However, it is not sure that the standard corresponds to the settings on your printer. If you therefore want to see on the area on which your printer is able to print, you click on **Printer help frame**. This way you can assure yourself that your printer will be able to print out in the scale 1:1, when you print out the entire page. This help frame is displayed by a grey dash line. Deselect the printer help frame in this example.

If you only use the help frame, the program will automatically scale the drawing header to fit your printer, before you print the page.

Click on **OK**.

You now see an A4 page with help frames and a grid on 5.00mm. The reference point for the drawing header as well as the text reference points, are placed on the middle of the page. These reference points shall now be moved to a more appropriate location.

Moving the reference points

The reference point of a drawing header must be placed precisely on one of the four corners of the help frame. In this example you must place it in the lower-left corner of the frame.

When you later place the drawing header on a page in a project, the reference point of the drawing header will position itself in the corresponding corner of the help frame on the page. In this case in the lower-left corner.



Therefore, click on the **Symbol** button, and click on the **Reference point** button. Click on the reference point you see on the middle of the screen, and *drag* it until it is precisely on the lower-left corner of the help frame. In this case the coordinates should be **x=20.00 y= 5.00mm**.



Make a **Zoom** including the four text reference points *and* the lower-left corner of the page. Then click on the **Texts** button and right-click on the upper-most of the four text reference points. A rubber band starting from the symbol reference point in the lower-left corner of the screen, and ending in the text reference point, together with a menu. Choose **Move** in the menu, and then click on the symbol reference point in the lower-left corner of the screen.

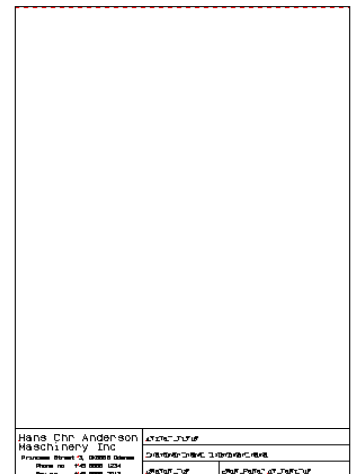
You have now placed the text reference point on the symbol reference point.

Now move the three remaining text reference points correspondingly, because you do not need any of these in a drawing header.



Click on the **Refresh** button if the page need to be refreshed.

You are now ready to design the drawing header itself.



DESIGNING THE DRAWING HEADER

Drawing the frame



Click on the **Zoom to page** button, the **Pencil**, the **Lines** button as well as the **Rectangle** button. Set the line width to **0.5mm**. Draw the frame so that it covers the entire help frame. — If you are not sure how to do this, please read “Lines” on page 72.

The fields in the drawing header is enlarged here:

Hans Chr Anderson Machinery Inc Princess Street 3, DK6000 Odense Phone no.: +45 6600 1234 Fax no.: +45 6600 7913	Address	
	Company Name	
	Page	Page

Placing the text fields



Click on the **Straight lines** button. Draw the upper-most horizontal line (See the figure above). The x- and y-coordinates are displayed in the lower-left corner of the screen. Start 30mm above the bottom of the frame (x=20, y=35) and draw the hor-

horizontal line. Let go of the line by double-clicking on the frame to the right — or click once and press <Esc>. The vertical line is drawn 70mm from the left of the frame (x=90).

Now set the line width to 0.25 and draw the rest of the lines. The small vertical line is placed in x=132.50mm.

General information on inserting datafields

When you insert datafields in a drawing header, you can make the program fill out the datafields for you. There are nine different types of datafields:

DATAFIELD TYPE	DESCRIPTION
SYSTEM DATA	GENERAL INFORMATION ABOUT USER NAME, COMPANY NAME, TIME AND DATE.
PROJECT DATA	INFORMATION REGARDING THE ENTIRE PROJECT — FOR INSTANCE THE ADDRESS OF THE CUSTOMER, CASE NUMBER, THE NAME OF THE CONSTRUCTOR AND SO ON.
PAGE DATA	RELEVANT INFORMATION FOR THE INDIVIDUAL PAGE IN THE PROJECT — SUCH AS PAGE NUMBER, THE SCALE OF THE PAGE, AND THE TIME FOR LAST CORRECTION OF THE PAGE.
SYMBOL DATAFIELD	DATAFIELDS YOU CAN DEFINE — READ MORE IN “CREATING DATA SYMBOLS” ON PAGE 732.
TABLE OF CONTENTS	SPECIAL INFORMATION NEEDED FOR A TABLE OF CONTENTS — SUCH AS A HEADING FOR THE TABLE OF CONTENTS OR SPECIFICATION OF DRAWING TYPES.
PARTS/COMPONENTS LIST	SPECIAL INFORMATION NEEDED ON PARTS- OR COMPONENTS LISTS — ARTICLE NUMBERS, PRICES, DESCRIPTIONS OF COMPONENTS AND SO ON.
TERMINALS LIST	SPECIAL INFORMATION FOR TERMINALS LISTS — THE NAME AND TYPE OF THE TERMINAL AND MORE.
CABLE LIST	SPECIAL INFORMATION FOR CABLE LISTS — THE NAME AND TYPE OF THE CABLE AND MORE.
PLC LIST	SPECIAL INFORMATION FOR PLC LISTS — THE NAME, TYPE, FUNCTION AND I/O DATA OF THE PLC.

So when you need to insert a datafield in a drawing header, you first choose which type of datafield you want to insert, and then which specific datafield of the chosen type.

Please note, that datafields written in upper case letters, contains information from the database.

To see how to fill out project- or page data, please read “Project and Page data” on page 114.

If you need to create new project- or page datafields, you can see how to do this in “Page data and Project data Datafields” on page 689.

— For a complete survey of the datafields in **Automation**, please refer to “Datafields” on page 855.

Insert the datafields

In this drawing header you need the following datafields:

Datafields from **Project Data**:- **Title**

- **Customer name**
- **Designer**

Datafields from **Page Data**:- **Page number**

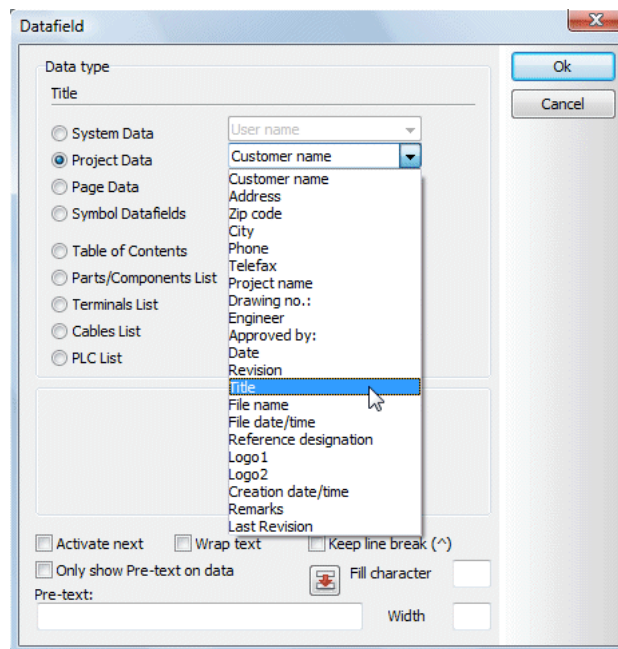
- **Last used page**



To determine the appearance of the datafields, you click on the **Texts** button, and then on the **Text properties** button. Here you set **Height** to **2.5mm**, **Width** to **AUTO** and **Alignment** to **left-down**. Click **OK**.

To insert a datafield, you choose **Insert => Insert Datafield**.

The dialog box **Datafield** is then displayed:



See an elaboration of the functions in this dialog box in “Datafields” on page 110.

Choose **Project Data**. In the field to the right of **Project Data** you click on the down-arrow. In the appearing drop-down menu you choose the datafield **Title**.



When you have chosen **Title** you can transfer the word **Title** to the **Text** field by clicking on the red arrow. The cursor is then placed inside the text field. Type in a colon (:) after **Title**. If you want space between the colon and the following title, please press the <spacebar> key.

In the **Width** field in the lower-left corner of the menu you can type in the allowed number of characters in the field. In this case, let it be **80**.

Click on **OK**. Place the datafield in the upper-right corner on the enlarged part on the drawing header.

Press <Esc> not to place the datafield again.

Continue by placing the datafield **Customer name**, which is also of the type **Project Data**.

The next field is **Designer**. It is also of the type **Project Data**, and in the **Text** field you type in *Design*. Set the width to 10. — The already transferred text is also included in this maximum number of characters.

Now click on **Page Data**, because the two last datafields **Page number** and **Last used page** are of this type.

A Choose **Page number**, type in *Page* in the **Text** field, and set the width to 10. Click on **OK**. Click on the **Text properties** button and set the alignment to *right-down* (See “Specifying text properties / Appearance of text” on page 122). Click on **OK** and place the text.

A Choose **Last used page**, type in the text *of*, set the **Width** to 10, and click on **OK**. Click on the **Text properties** button, set the **Alignment** to *left-down*, and click on **OK**. Place the text.

Free text in drawing headers

A Now click in the text field in the texts tool bar, and type in the text *Hans Chr Anderson^Maschinery Inc*. Click on the **Text properties** button, and set the text to the **Height 4.0mm**, **Width AUTO**, **Alignment left-down**, and choose **Bold text**.

The ^ character signifies line break, and will not be displayed on the screen until you have typed in a succeeding character.

Place the text like you see it on the figure above. — See more about how to type in a free text in “Type in and place a free text” on page 103.

A Now change the **Text properties** settings to the **Height 2.0mm**, **Width AUTO**, **Alignment centre-centre**. Deselect **Bold text**.

Then place the remaining texts, as you see them on the figure above.

You have now created a drawing header, and can save the final result.



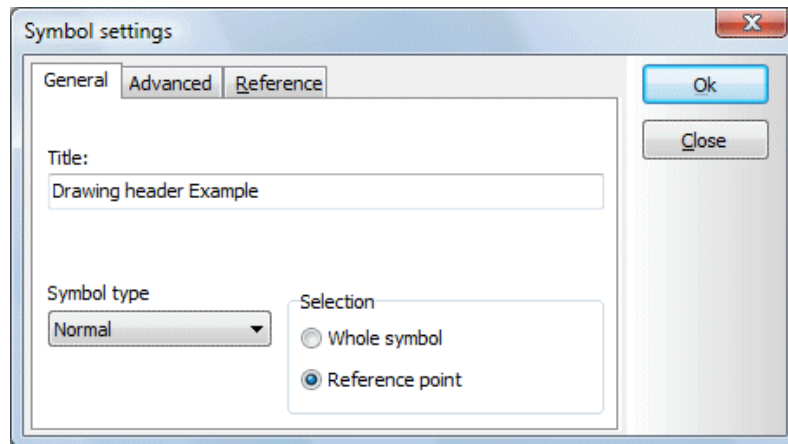
DRAWING HEADER FOR DIAGRAM PAGES - INCLUDING A LIST

IT IS ALSO POSSIBLE TO CREATE DRAWING HEADERS FOR DIAGRAM PAGES, WHICH INCLUDES A LIST ON THE PAGE. — SEE “DIAGRAM PAGES - CONTAINING A PARTS OR COMPONENTS LIST” ON PAGE 762.

Saving the symbol / the drawing header

Choose **File => Save as**.

Fill out the dialog box as you see below.



Please note, that you can also enter this dialog box while designing the drawing header, if you click on the **Symbols** button, and then on the **Symbol settings** button.

Now click on **OK**, and save the drawing header in the **MISC** folder with the file name *A4Vertic*.

You can also create your own folder for drawing headers.

Click on **OK**.

Leaving the Edit symbol mode

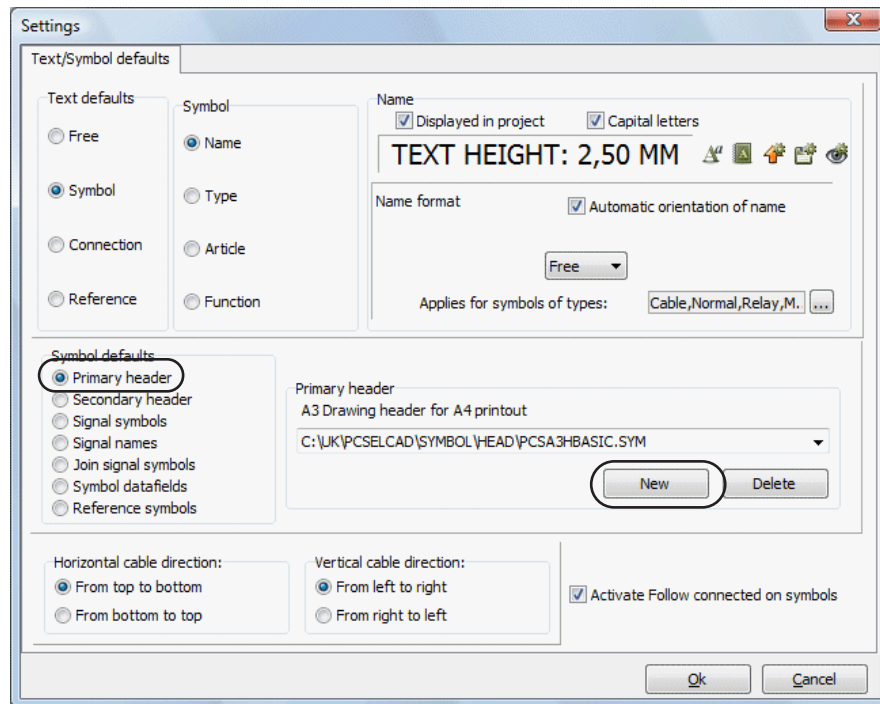
You have now saved the drawing header, and shall now make the program leave the Design/Edit symbol mode. Do this by choosing **File => Close**.

You now return to the **Symbol menu**, where you click on **Cancel** or press <Esc>.

ADDING DRAWING HEADERS TO THE LIST OF DRAWING HEADERS

The next thing you need to do, is to add the drawing header to the list, which contains all the drawing headers you can use in your projects.

Therefore choose **Settings => Text/Symbol defaults**.



Click on **Primary header** in the left side of the dialog box.

Then click on **New**, which brings you into the **Symbol menu**.

Here you click on the image of the drawing header you want to add, and the drawing header now change to the color grey.

Click on **OK**.

You have now returned to the tab **Text/Symbol Standards**, and can here see the name of the drawing header in the field **Drawing headers**. Click on **OK**.

The drawing header is now ready to be used in a project.

Inserting the drawing header in a project

After having added the drawing header to the list of drawing headers, you can use it in your project.

— See how to insert a drawing header on a project page in “Insert drawing headers/formulas” on page 531.

REMOVING DRAWING HEADERS FROM LIST OF DRAWING HEADERS

If you want to remove a drawing header from the list of drawing headers, you choose **Settings => Text/Symbol defaults**. Here you click on the down-arrow in the field below **Drawing headers**. In the appearing drop-down menu, you click on the drawing header you want to remove from the list, click on **Remove**, and answer **Yes** to remove the drawing header from the list. Click on **OK**.

The drawing header is now removed from the list of drawing headers.

PREDEFINED DRAWING HEADERS

To make things a little easier, some predefined drawing headers already exist. They are located in the folder **c:\Pcscad\Symbol\Misc**. To make use of these drawing headers in your projects, you can open them and adapt them to your needs — as described in “Changing an already created list/drawing header” on page 763.

You can drag pages containing these drawing headers into your projects from the **Sub drawings** tab in the **Explorer** window. These pages are placed in the folder **Template/Normal**.

DRAWING HEADERS WITH STATES

You can also create drawing headers with states, just like you create symbols with states. — Read more about this in “Creating symbols with states” on page 712.

— See how to select states when placing drawing headers in “Drawing headers with States” on page 534.

PLACING A COMPANY LOGO ON A DRAWING HEADER

When you wish to place a logo in a drawing header, it is recommended to do so by creating a project logo, as described in “Inserting logos in projects” on page 527.

This also makes it possible to replace the logo easily, when working for different customers.

CHANGING DATAFIELDS IN PAGE DATA AND PROJECT DATA

If you want to change the name of a datafield in **Project data** or **Page data**, please right-click on the datafield. Hereby a menu appear, in which you can edit, delete or add datafields. Read more about this in “Page data and Project data Datafields” on page 689.

CREATING LISTS

In this chapter you can see how to create your own project lists.

THIS CHAPTER DESCRIBES	
— WHAT IS A LIST	P. 753
— LISTS/DRAWING HEADERS.....	P. 753
— HOW TO CREATE A LIST	P. 754
— ADDING THE LIST TO THE LIST OF DRAWING HEADERS	P. 758
— USING DATAFIELDS.....	P. 758
— CREATING A TABLE OF CONTENTS.....	P. 759
— CREATING A COMPONENTS LIST	P. 759
— CREATING A TERMINALS LIST	P. 759
— CREATING A CABLE LIST.....	P. 760
— CREATING A PLC LIST	P. 760
— CREATING A DIAGRAM PAGE CONTAINING A PARTS OR COMPONENTS LIST ..	P. 762
— CREATING A LIST SYMBOL	P. 762
— PREDEFINED LISTS	P. 763
— CHANGING AN ALREADY CREATED LIST/DRAWING HEADER	P. 763
— OTHER OPTIONS	P. 766

If you want to create *list files* you do not need to design a list the way it is described here. Please refer to “Exporting List Files” on page 649 for further details.

In “Insert drawing headers/formulas” on page 531 you can see how to *insert* lists in projects.



INSERTING DRAWING HEADERS / LISTS IN TEMPLATES

WHEN YOU HAVE CREATED A LIST DRAWING HEADER, YOU CAN INSERT IT ON A TEMPLATE PAGE, AS DESCRIBED IN “CREATING TEMPLATES” ON PAGE 771, SPECIALLY “CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR” ON PAGE 772.

You now get the text in the cross hairs, and place it by pointing on the point **$x=20.00mm$ $y=265.00mm$** and then click once. (See the coordinates in the lower-left corner of the screen).

The text **Article number** have now been placed on the page.

Press <Esc> not to place the text again.

Now place these texts following the same procedure:

- The text *Function* in **$x=75.00$ $y=265.00mm$** and
- The text *Quantity* in **$x=177.50$ $y=265.00mm$**

Corrections



If you by accident place a text in the wrong place, then deactivate the Pencil (press <Esc>), select the text by clicking on its reference point, and then move it using the **Move** button, or delete it using the **Delete** button. Then **Refresh** the page. Also, you can choose **Undo** to undo your latest action. Read more about these options in “Working with selected objects” on page 58.

Inserting the datafields

Now the column headings have been properly placed, and it is time to place some datafields, in which information from the drawings can be inserted.

Choosing a datafield

Start by choosing **Insert => Insert Datafield**.

Click on **Parts/Components list** and then click on the down-arrow in the field immediately to the right of **Parts/Components list**.

In the appearing drop-down menu, you click on **Article no.** — Please note *not* to select ARTICLE NO written in upper case, because this will fetch its contents from the database instead. — Read more about the database in “Using the database” on page 571.

Click in the field **Width** in the lower-right corner of the dialog box, and type in *20*.

Then select **Activate next**. What this means will be elaborated in “Activate next” on page 756.

(As opposed to creating the drawing header, please do *not* click on the red arrow, because you do not need a fixed text in this datafield).

— You can read more about the **Datafield** dialog box in “Datafields” on page 110.

Click on **OK**.

Placing the datafield

You now have the datafield in the cross hairs, and place it in the point **$x=25.00$ $y=245.00mm$** .

Press <Esc> not to place it more than once.

You have now placed the first datafield.

Placing the remaining datafields

Choose **Insert => Insert Datafield**, deactivate **Activate next**, and insert the following datafields following the same procedure as above:

- The field **Function** (width 80) in **x=75.00 y=245.00mm** and
- The field **Quantity** (width 20) in **x=177.50 y=245.00 mm**

Press <Esc> not to place the last datafield again.

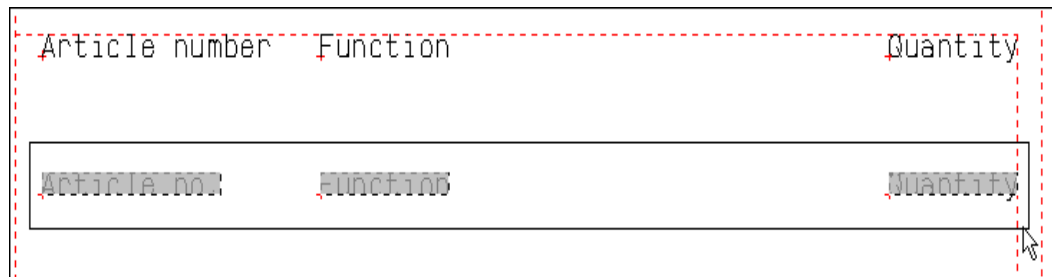


DATAFIELDS FOR EAN13 BARCODES

FOR **PARTS/COMPONENTS LISTS** YOU CAN ALSO INSERT THE DATAFIELD **EAN13 BARCODE** FOR AUTOMATIC GENERATION AND INSERTION OF EAN13 BARCODES. YOU CAN HERE CHOOSE BETWEEN THE THREE STANDARD SIZES FOR BARCODES.

Aligning the datafields

As a precaution, you can now align the datafields. Therefore mark up a window around the three datafields by clicking and dragging.



Then choose **Edit => Align**, and click on the reference point for the left-most of the texts. — Read more about this function in “The Align function” on page 66.

More rows of datafields

You have now placed the first row of datafields in the parts list. However, the parts list should consist of many rows of datafields, why you should now copy the three datafields and place them below the other datafields several times.



First make a **Zoom** including the three datafields (**Article no**, **Function** and **Quantity**) and which contains room for twelve more rows of datafields.

Then mark up an area around the three datafields (not including the column headings), so that all the datafields are selected when you release the mouse button. — If you are not sure how to mark up an area, please see “Selecting objects” on page 55.



Then click on the **Copy** button, and place the lines below each other, with a distance of for instance 10mm. This is most easily done by pressing four times on the down-arrow of the keyboard, then press <Enter>, and repeat these actions until all the rows of datafields have been placed.

Press <Esc> not to place more copies.



If you have not aligned the rows of datafields properly, then select the rows individually, and place them using the **Move** button, as described above.

Activate next

The first time the program meets a check mark in **Activate next**, you tell the program to collect information about the first component in the project, which shall be

You can for instance save the list in the folder **c:\Pcscad\Symbol\Misc**. However, it is recommended that you create your own folder for saving lists. Click on **Save** when all the settings are correct.

The parts list has now been saved.

Leaving the Create/Edit symbol mode

To make the program leave the Edit/Create symbol mode, you choose **File => Close**.

You now return to the **Symbol menu**, where you click on **Cancel** or press <Esc>.

Please note, that the text **SYMB**, which was flashing on a red background in the lower part of the left tool bar, now has disappeared.

ADDING THE PARTS LIST TO THE LIST OF DRAWING HEADERS

The next thing you have to do, is to add the parts list to the list of drawing headers. This is done as described in “Adding drawing headers to the List Of Drawing Headers” on page 750.

You will not be able to use the parts list in your project until you have done this.

See “Insert drawing headers/formulas” on page 531 to see how to insert the parts list on a page in a project, as well as how you get the list updated.

USING DATAFIELDS

Which datafields you can use, depends on which type of list you are creating.

You can always use datafields of the type **System Data**, **Project Data** and **Page Data**. But apart from these types of datafields, you can only use datafields corresponding to the type of list you are creating.

If you are creating a table of contents, you can make use of datafields of the type **Table of Contents**, but not of the type **Parts/Components List**.

CREATING A TABLE OF CONTENTS

When you serrate a table of contents, you follow exactly the same guidelines as for designing a parts list.

Table of contents	
Circuit diagram	
Circuit diagram 2001-05-14	I3
Power circuit diagram 2001-05-14	2
Control and Signal Circuit diagram 2001-06-06	3

The only differences — apart from the way you want to set up the lists yourself — is that you will need datafields of the type **Table of Contents**, and that you in the **Page menu** — as mentioned in “Insert drawing headers/formulas” on page 531 — will have to set the page function to **Table of Contents** when you insert the page in the project.

In a table of contents you can also insert a data filed displaying the reference designation of the page.

To see an example of a table of contents, take a look at **A4INDEX**, mentioned in “Predefined lists” on page 763.

CREATING A COMPONENTS LIST

You can also create a components list, containing all the components in the project. For each component you can place its name and its exact location on the project drawings, so that you can use this list to locate any component in the project.

Components List			
Name	Itemnumber	Description	Price
=G1+A1-P1	3389110611229	Signal lamp, red, 220V w/ resistor	81.35
=G1+A1-P2	3389110611229	Signal lamp, red, 220V w/ resistor	81.35
=G1+A1-P3	3389110611229	Signal lamp, red, 220V w/ resistor	81.35

For components lists, you set the page function to **Components list**, and use datafields of the type **Parts/Components list**. Components lists are created like you create parts lists.

CREATING A TERMINALS LIST

You can also create a terminals list, containing a survey of all terminals in the project, their names, and which wires enter and exit the terminals. At the same time you can let the list contain information about which cables the wires are a part of.

This way you get a survey of which wires shall be connected to which terminals.

Terminals List				
External		Terminal	Internal	
CABLE	Name		Name	CABLE
=G1+A2-W1:Brown	=G1+G1-M1:U	=G1+A2-X1:1U2	=G1+A2-F1:2	
=G1+A2-W1:Black	=G1+G1-M1:V	=G1+A2-X1:1V2	=G1+A2-F1:4	
=G1+A2-W1:Blue	=G1+G1-M1:W	=G1+A2-X1:1W2	=G1+A2-F1:6	

To get transferred special information for terminals lists, you have to set the page function to **Terminals list**, and use datafields of the type **Terminals list**. Apart from this, terminals lists are designed following the same guidelines as for parts lists.

When you place datafields for terminals lists, you have to be sure whether to click on **External** or **Internal** in the lower part of the dialog box **Datafield**. — See “Datafields” on page 110.

Correspondingly, when you place terminal symbols on a project drawing, you have to be conscious about which connection point of the terminal is the **Input** and which is the **Output**. Read more about this in “Terminal symbols” on page 214.

CREATING A CABLES LIST

Furthermore you can create a cable list containing a survey of the project cables, and from where- and to where they are to be connected.

Cables List					
FROM		CABLE		TO	
Identification	Page/path	Identification	Page/path	Identification	Page/path
=G1+A2-X1:1V2	2/5	=G1+A2-W1:Black	2/5	=G1+G1-M1:V	2/5
=G1+A2-X1:1W2	2/6	=G1+A2-W1:Blue	2/6	=G1+G1-M1:W	2/6
=G1+A2-X1:1U2	2/5	=G1+A2-W1:Brown	2/5	=G1+G1-M1:U	2/5

Set the page function to **Cable list**, use datafields of the type **Cable list**, and create the list as described for parts lists in the beginning of this chapter.

CREATING A PLC LIST

Finally, you can create a PLC list containing a survey of the inputs and outputs on the PLC. At the same time it can contain a description of which functions are

attached to the various terminals on the PLC. The list can also contain connection points labels.

PLC list				
Name	I/O	Description	Signal path	Attached component
-K1:X0	I0		-X2:1,-W1:Blue	-S1
-K1:X1	I1		-X2:2,-W1:Orange	-S2
-K1:X2	I2		-X2:3,-W1:Green	-S3

Set the page function to **PLC list**, use datafields of the type **PLC list**, and design the list as described above.

Datafields and PLC lists

When you are working with PLCs, you can place the information about the PLC labels and descriptions in two different places.



You can click on the **Symbols** button and right-click on the connection point on the PLC itself, and fill out the fields **Description** and **Label**.

Or you can right-click on the connection point on the symbol connected to the terminal in the PLC, and fill out the fields **Description** and **Label** here.

When you design a PLC list, you therefore have to be aware that you have to use different datafields depending on which of the procedures you use.

If you want to collect the information from the connection point of the PLC, you must use the datafields **C.descr** and **C.Label**. But if you want to collect the information from the connection point of the connected symbol, you have to use the datafields **Cc.C.descr** and **Cc.C.label**.

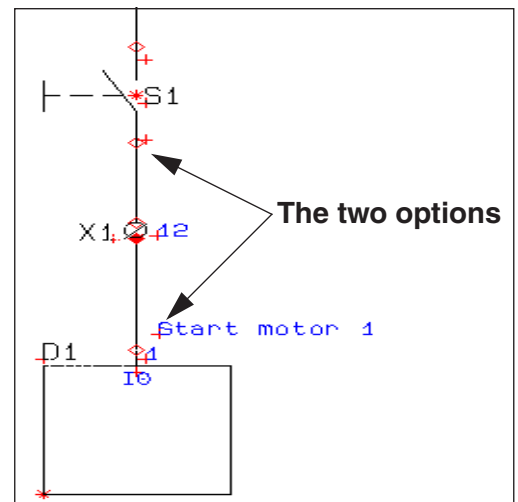


DIAGRAM PAGES - CONTAINING A PARTS OR COMPONENTS LIST

It is possible to create a drawing header for normal diagram pages, which also contains either a parts or a components list. However, these lists can only contain information from the page they are placed on.

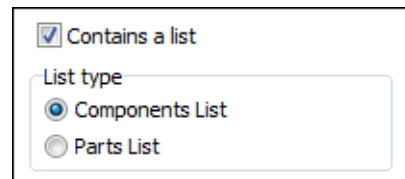
When you create the list as a part of the drawing header, the list will therefore only contain data about the components placed on the page. Furthermore, it is your responsibility to ensure that the list is long enough for the components, that are going to be placed on the page. This is because new pages will not be inserted, if this list is too short.

ID	Description	Type	Value
1	Q1	BT78	402290305584
2	Q2	LSU5K11230	4022903075387
3	Q3	LSU5K11230	4022903075387
4	TR1	1007m3	5702950416537
5	TR2	AB1VV43TU	3399110396437
6	TR3	AB1NE43TU	3399110396435
7	TR4	AB1VV43TU	3399110396435
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			
19			
20			
21			
22			
23			
24			
25			
26			
27			
28			
29			
30			

To create such a drawing header, you must do as described in “How to create a List” on page 754 — with the only difference that you make room for diagram drawings in the drawing header.

Specify that the diagram page contains a list

When you, at a later stage, insert a such drawing header on a project page, you must successfully choose **Settings => Page setup**, and select **Contains a list**, and then specify whether it is a **Components List** or a **Parts List**.



Access to List Settings

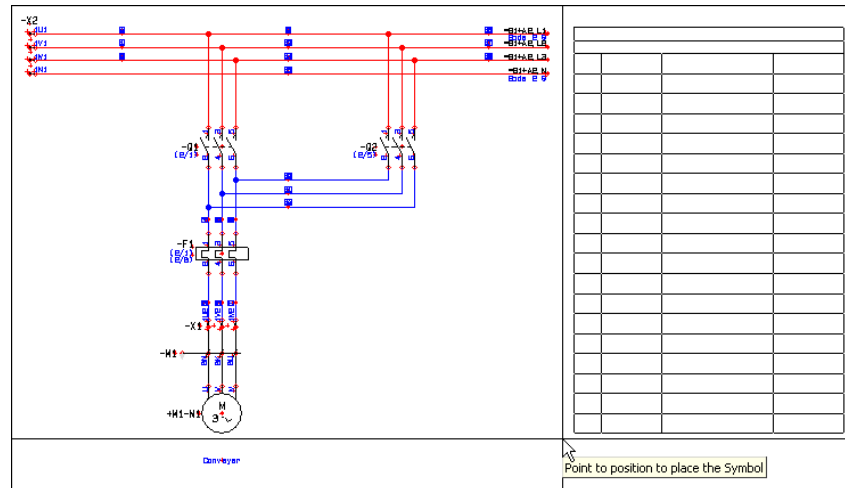
On diagram pages containing lists, you can also right-click on the page and choose **List settings**, and then specify the list settings, as described in “List settings” on page 542. For this type of lists, there will be some limitations on the settings you can specify.

CREATING A LIST SYMBOL

You can also create a symbol containing either a parts or a components list. When you place this symbol on a diagram page, it will subsequently function as a parts or

components list for the page — according to the same directions as described in “Diagram pages - containing a Parts or Components list” on page 762.

When you create a symbol of this kind, datafields etc. must be placed as described in “How to create a List” on page 754. When saving the symbol, you must set the **Symbol type** to **Head** — see “Saving the parts list” on page 757.



When you have created the symbol, you can place it on diagram pages, just like you normally place symbols. Then specify that the page contains a parts or components list, as described in “Specify that the diagram page contains a list” on page 762. The symbol will then function as a list for the page.

You can find an example on such a symbol with the name **Listsymbol.sym** in the **MISC** folder.

— See “Access to List Settings” on page 762 also.

PREDEFINED LISTS

To make things a little easier, some lists have already been defined, which you can use in your projects. They are located in the folder **c:\Pcscad\symbols\misc**. To use them in your project, you can open them and change them, until they fulfil your demands — as described in “Changing an already created list/drawing header” below.

You can drag pages containing many of these drawing headers into your projects from the **Sub drawings** tab in the **Explorer** window. These pages are placed in the sub folders under the **Template** folder.

CHANGING AN ALREADY CREATED LIST/DRAWING HEADER



If you want to adapt an already created list/drawing header, you click on the **Symbols** button, and then on the **Symbol menu** button.

You have now entered the **Symbol menu**. Here you find the folder in which the drawing header is placed, and then click on the drawing of the drawing header you wish to change.

This makes the **Edit symbol** button appear in the lower-left part of the **Symbol menu**, and you click on it. You now have the option of adapting the chosen drawing header to meet your demands.

Saving the drawing header under a new name

In order not to overwrite the already existing drawing header, you start by choosing **File => Save as** and save the drawing header under a new name. For instance you can give it the name *Newlist.sym*. — See how to do this in “Saving the parts list” on page 757.

Changing free texts and removing the company logo



Make a **Zoom** covering the lower-left corner of the list.

Click on the **Area** button, mark up an area around the cat, and press . The company logo has now disappeared.

Click on **Refresh**.

To change the company name you now click on **Texts**, and then click on the reference point for the text **PC|SCHEMATIC A/S**, which selects the text.

Now press <k> or click inside the text field in the texts tool bar. Correct the company

name to your own company name — or to for instance *H C Andersen Inc* — and press <Enter>.

As you will see, the text has now been changed.

Correct the remaining free texts correspondingly.

Please note that you can delete and move texts as you normally do. — Read more about this in “Working with selected objects” on page 58.

ARTID/ARTNR	DESCRIPTION	
	H C Andersen Inc	Project
	Bygaden 7, 4040 Jyllinge	Customer
	Tel. +45 46 78 82 44	Printed
	Fax. +45 46 73 02 15	Changed
	e-mail: dps@dps.dk	



The appearance of a selected text, can be changed by clicking on the **Text properties** button. This is elaborated in “Specifying text properties / Appearance of text” on page 122.

Changing a datafield

Also, you can change the inserted datafields on the drawing header.



Click on the **Texts** button, and click on the reference point for the datafield **Printed**. Now click on the **Data** button on the upper part of the screen, which makes you enter the **Datafield** dialog box for the datafield **Printed**.

Another way to enter the dialog box **Datafield**, is to right-click on the text/datafield, and choose **Text properties** in the pop-up menu.

Project: H C Andersen Inc	
Customer: Customer name	
Printed: Date	Date
Changed: Date	Date

Project: H C Andersen Inc	
Customer: Customer name	
Assigned by: Designer	Date
Changed: Date	Date

Here you choose the desired datafield — for instance **Designer** of the type **Project data** — and type in *Designed by:* in the **Text** field in the lower part of the dialog box. Click on **OK**.

As you can see, the datafield **Printed** has now been replaced by the datafield **Designer**.

Changing a column of datafields of the same kind

When adapting a list to meet your demands, you will also need to change an entire column of datafields of the same type, into a row of datafields of another type.



To do this, you activate the **Texts** button, and select the datafields you want to change — as described in “Working with selected objects” on page 58.



For instance you can click on the **Zoom to page** button, and mark up a window around all the datafields in the right-most column. Please be careful not to include the column header.

— You can tell that a text field is a datafield by the red dash line which is displayed around datafields.

	Items	Items
	Quantity	Price1
	Quantity	Price1
	Quantity	Price1
	Quantity	Price1
	Quantity	Price1
	Quantity	Price1



Click on the **Data** button — or right-click on the selected datafields — to enter the dialog box **Datafield** for the datafield. Now choose for instance the datafield **Price1** and click **OK**.

As you can see, all the datafields in the column have been changed to **Price1**.



When you have completed your corrections, save the list by clicking on the **Save** button.

OTHER OPTIONS

Company logo on list

If you have created your company logo as a symbol or as a DWG or a DXF file, you can place it on your list/drawing header. — See how to do this in “Inserting logos in projects” on page 527.

Delete a page with a list on



If you want to remove a page with a list on completely from a project, you have to delete the page, on which the list is placed. Therefore, click on the **Page menu** button.

In the **Page menu** you click on the page you want to delete, and choose **Delete => OK**. Click **OK** once again to close the **Page menu** dialog box.

The page has now been deleted from the project.

Remove a list from a page



If you want to remove a list, but keep the page on which the list is placed, you first click on the tab for the page. Then choose **Settings => Page data** — or click directly on the **Page data** button in the program tool bar.

In the **Page data** dialog box you deselect **With drawing header**, and click on **OK**.

The list has now been removed from the page.

Please note that the **Page function** of the page remains unchanged. If the page had the page function **Parts list**, it will still have this page function after the drawing header/list has been removed from the page.

Once a page has been created, its page function can not be changed. — Read more about this in “Creating a new page” on page 495.

CREATING UNIT DRAWINGS

If a component consists of several different parts, you can specify which parts it consists of in a unit parts list. This unit parts list can also contain a unit drawing, displaying the parts of the component, but this is not mandatory.

When you use an extended parts list (please refer to “Specifying from where to collect components” on page 544), the program examines for which components a unit drawing have been created. In the parts list, the component will then be represented by what you specified in the unit parts list.

THIS CHAPTER DESCRIBES	
— CREATING A UNIT PARTS LIST	P. 767
— INITIAL PREPARATIONS.....	P. 767
— ASSIGNING AN ARTICLE NUMBER TO THE UNIT PARTS LIST.....	P. 768
— CREATING A UNIT DRAWING	P. 768
— UNIT DRAWING FROM ALREADY CREATED SYMBOL.....	P. 769
— UNIT PARTS LISTS WITHOUT UNIT DRAWINGS.....	P. 769
— CREATING UNIT PARTS LISTS FROM A DRAWING OF AN ENTIRE COMPONENT .	P. 769
— CORRECTING A UNIT DRAWING/UNIT PARTS LIST	P. 770
— INSERTING UNIT DRAWINGS IN PROJECTS	P. 770

Unit drawings can consist of more than one page.

CREATING A UNIT PARTS LIST

The procedure for designing the unit parts list, is the same as for designing an ordinary parts list. So, if you want to design your own unit parts list, you should do this now, as specified in “Creating Lists” on page 752. Alternatively you can use an already created parts list.

However, you have to do a little more before you can use it as a unit parts list.

INITIAL PREPARATIONS



The first thing you have to do, is to click on the **New document** button.

Here you must first make sure that the page is a vertical A4 page, as described in “Setting up the page to vertical A4 format” on page 743. — Also make sure that the page function is set to **Normal**.

To place the unit parts list on the page, you now choose **Settings => Page data** and right-click on the field below **Add drawing header**. In the pop-up menu, you then click on **Add drawing header**.

You now enter the **Symbol menu**, where you can select one of the symbols mentioned below in the folder **MISC**.

Click on your own parts list, or choose the symbol **UNITLI**, and click **OK**.

You have now placed the unit parts list on the page.

ASSIGNING AN ARTICLE NUMBER TO THE UNIT PARTS LIST

To be able to use the page as a unit parts list, you have to assign an article number to the page.

Therefore, right-click somewhere on the page, and choose **Page article data**. Assign a **Name** and an **Article no** to the page, and fill out the **Type** and **Function** fields as well. Click on **OK**.

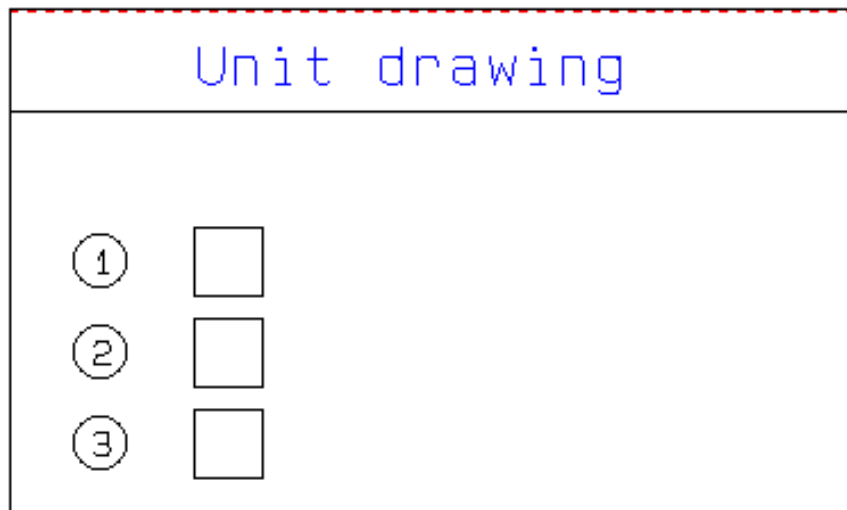
You determine the article number of the page yourself, and in the following this is the number you use, when you connect a symbol to the unit drawing.

CREATING A UNIT DRAWING



You are now ready to create the unit drawing. In this example you could for instance draw three rectangles, which illustrates the parts the component is made of. Also, you could place the page title on the top of the page.

If you are not sure how to do these things, please refer to “Lines” on page 72 as well as “Project and Page data” on page 114.



Now place the symbols (position numbers), in which you in the **Component data** specify the information, which is necessary for the parts on the unit drawing.



To do this, click on the **Symbols** button and then on the **Symbol menu** button.

Find the symbol **Posno1**, which is created as a circle, in the folder **MISC**, and double-click on the symbol.

You then get the symbol in the cross hairs, and place a symbol close to each of the three rectangles.

When you place a symbol, you enter the **Component data** menu, where you type in **1** in the **Name** field for the first symbol, and **2** and **3** in the **Name** field for the two following symbols. Type in the remaining component data yourself, or fetch them by clicking on the **Database** button. Click on **OK** when the information have been filled out correctly.

If you are not asked about the component data automatically, just right-click on the symbols one by one, and fill out the information as described. — Read more about this in “Pointer/Screen” on page 836.

If you do not want a circle around the position numbers, you can use the symbol **Posno2** in stead of **Posno1**.

Saving the unit drawing

You have now finished designing your unit drawing/parts list.

To save it, you choose **File => Save unit as**.

When you have confirmed that you wish to save the project as a unit drawing, you enter a dialog box, where you for instance type in the file name *test.std* and click **OK**.

The program now knows that the page is a unit drawing/parts list.

Therefore you can transfer the information from the symbols to the unit parts list by choosing **Lists => Update Unit Drawing**.

Save the unit drawing again.

Then choose **File => Close**.

The unit drawing is now ready to be used.

UNIT DRAWING FROM ALREADY CREATED SYMBOLS

If you already had created the parts on the unit drawing as symbols, you can place them on the unit drawing as well. — Please refer to “Creating Symbols” on page 696 to see how to create symbols.

Now place the symbols for the parts on the unit drawing.

Then right-click on the symbols — one by one — on the drawings of the parts and choose **Component data**. In stead of the symbol name, you then type in the number you want the part to have on the drawing and in the unit parts list.

In you have not assigned component information to the symbols, you can now type in this information, or find them clicking on **Database**.

Save the unit drawing as described in “Saving the unit drawing” on page 769.

UNIT PARTS LISTS WITHOUT UNIT DRAWINGS

If you do *not* want any drawings on your unit parts list, this is also an option. Still, you have to create a symbol for each part, in order to specify which parts the component is made of.

Here you may use the symbols **Posno1** or **Posno2**, as described above, or you can design a symbol for this very purpose yourself.

To these symbols you add the necessary component data, as described above.

In stead of the symbol name, you type in the number you want the part to have on the unit parts list.

CREATING UNIT DRAWINGS USING A COMPONENT DRAWING

If you have a drawing of the entire component, you can load it as a symbol from the **Symbol menu**.

If the drawing is in the DWG/DXF format, please refer to “Importing DWG/DXF files as symbols” on page 663 to see how to import the drawing via the **Symbol menu**.

In stead of drawing each individual part, you now place some new symbols (**Posno1** or **Posno2**), each pointing at an individual part on the drawing.

For each of these symbols you add the relevant article data, as described above.

The crucial part for how the unit parts list is filled out, is exclusively which information is attached to the symbols placed on the page. The *drawing* itself is no more than an illustration.

If you want to place other relevant information on the page — for instance about expenses — you just place additional symbols on the page containing this information.

CORRECTING A UNIT DRAWING/PARTS LIST

In order to change a unit drawing, you choose **File => Open Unit**.

Then you get the possibility to choose between the available unit drawings.

Pick one of them, perform the changes, and save the unit drawing as mentioned earlier.

INSERTING UNIT DRAWINGS IN PROJECTS

Please read “Inserting Unit Drawings” on page 568 to see how to insert a unit drawing in a project.

CREATING TEMPLATES

THIS CHAPTER DESCRIBES	
— ON TEMPLATES	P. 771
— CREATING PAGE TEMPLATES - VIA THE TEMPLATE EDITOR.....	P. 772
— CREATING PAGE TEMPLATES - VIA THE SUBDRAWINGS TAB	P. 773
— CREATING PROJECT TEMPLATES.....	P. 774
— CREATING TEMPLATES BASED ON EXISTING PROJECTS.....	P. 776
— CONVERTING STANDARD PROJECTS TO PROJECT TEMPLATES.....	P. 776
— EDITING EXISTING TEMPLATES	P. 777
— DELETING A TEMPLATE	P. 777
— TEMPLATE FOR CLICKING THE NEW DOCUMENT BUTTON	P. 778
— TEMPLATES IN Automation	P. 778

ON TEMPLATES

In **Automation** you can create templates for all types of project pages, as well as for full projects.

Each time you create a new project, or a new project page, you can therefore start out on the basis of an already created template.

Using templates on networks

Using templates enables you to let one person create the company templates, and make these templates available for all users via the network. This complete set of company templates therefore only needs to be placed at one location, and can be maintained by one — or more — persons.

The other users of **Automation** therefore does not have to be familiar with creating drawing headers (including lists, tables of contents and chapter tabs), or how to specify list settings.

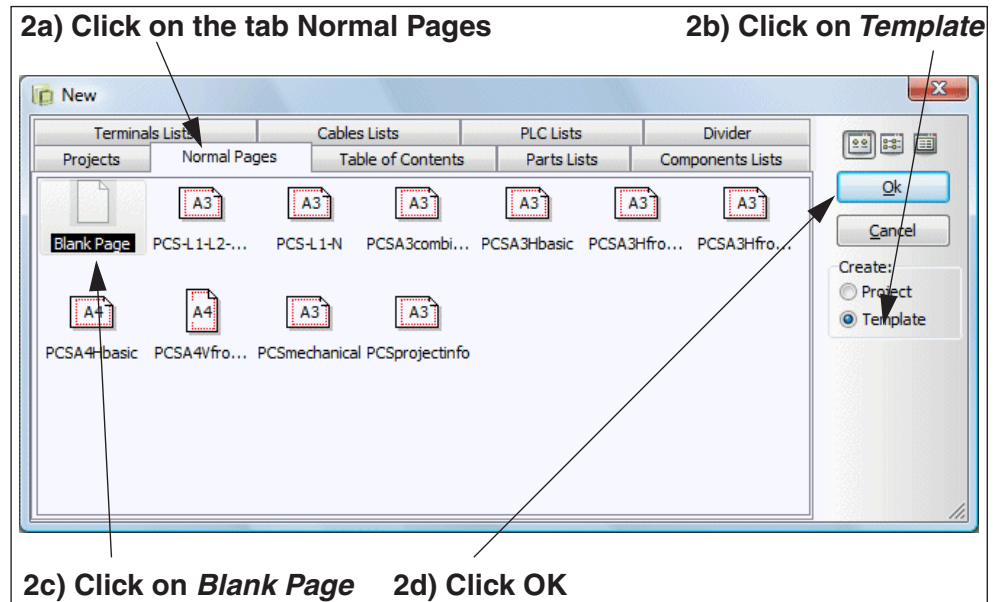
It is therefore easy to achieve a standard appearance of **Automation** projects. It is also simple to make *changes* in the complete set of template projects and pages, because these only have to be changed at one place.

CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR

In **Automation** you can create template pages for all types of project pages. The following describes how to create a template page for a normal diagram page via the **Template editor**. — See also “Creating Page templates — via the SubDrawings tab” on page 773.

To create a template for a diagram page, do the following:

- 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 — to create a new template page —, click for instance on **Blank Page**, and click **OK**.



NEW TEMPLATES FOR LISTS AND CHAPTER DIVIDERS

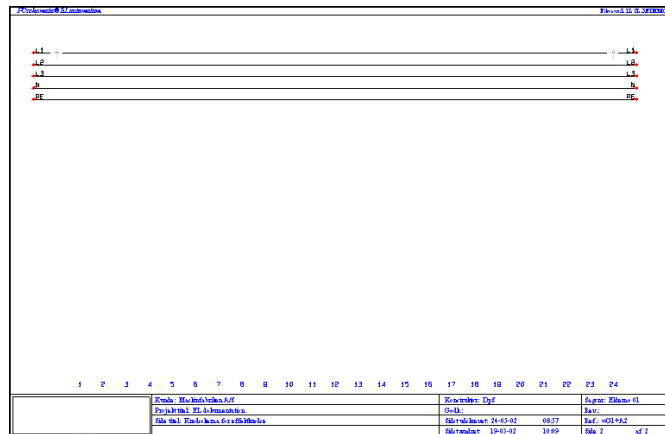
IF YOU WISH TO CREATE A NEW TEMPLATE FOR E.G. A TERMINAL LIST PAGE, SELECT THE TAB **TERMINALS LISTS** IN STEAD.

IF YOU ARE CREATING A NEW CHAPTER DIVIDER, CLICK ON THE TAB **DIVIDER**.



- 3) You now enter the **Template editor**, from where you create new templates: Click on the **Page data** button, if you wish to place a drawing header on the page.
- 4) The tab **Page data** from the dialog box **Settings** is displayed: Choose a drawing header at the lower part of the tab, type in page data if necessary, and click **OK**.
- 5) If you wish to draw/place objects on the page, do this now — exactly like when you normally work in **Automation**.

Here you can for instance draw the phases on the upper part of the page.



You can also place existing sub drawings on the page, as described in “Sub drawings” on page 256.



- 6) Click on the **Save** button, and you enter the dialog box **Save as**, where you type in a name of the template, and click **Save**.
- 7) Choose **File => Close**, and you leave the **Template editor**, and return to normal **Automation** mode.



IN PAGE TEMPLATES THE FOLLOWING IS MEMORISED

- ALL CREATED PAGE DATA DATAFIELDS, AS WELL AS THEIR CONTENTS (IF ANY)
- THE REFERENCE SYSTEM FOR THE PAGE
- FOR LISTS: THE SPECIFIED LIST SETTINGS — SUCH AS VENDOR AND REPETITIONS
- ALL OTHER PAGE SETTINGS

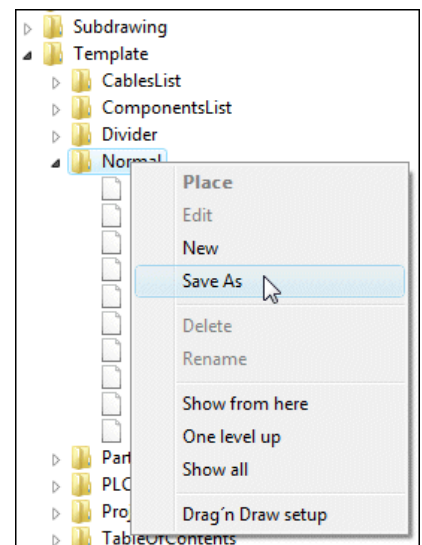
CREATING PAGE TEMPLATES — VIA THE SUBDRAWINGS TAB

When you are working in a single-page project, you can save it as a template by right-clicking on one of the folders in the **Template** folder, and choose **Save as**.

You then enter the **Save as** dialog box, where you select which folder to save the template in, assign a name to the template, and click **Save**.

Please note that the program automatically assigns the extension **.std** to the template.

If there is a drawing header on the page, the template will be understood as a full page. If not, it will behave as a normal sub drawing — see “Sub drawings” on page 256.



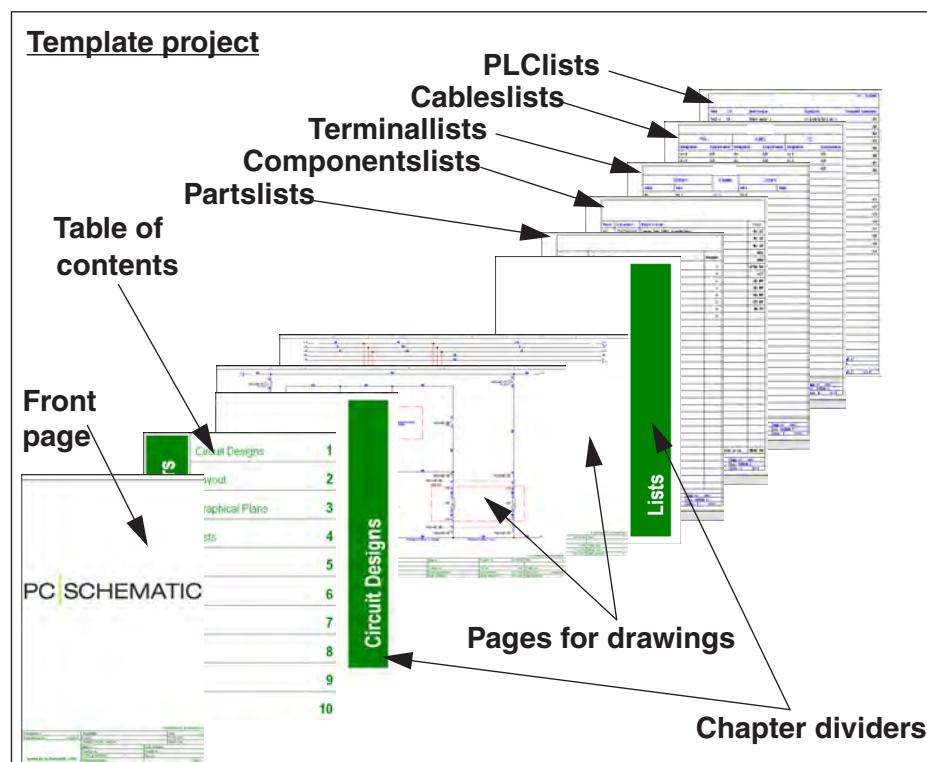
CREATING PROJECT TEMPLATES

When you wish to create templates for full projects, enter the **Template editor**, and create a full project as described in “Creating Page templates — via the Template editor” on page 772.



— PLEASE NOTE THAT THE TEMPLATE PROJECT MUST BE CREATED FROM THE TAB **PROJECTS** IN THE DIALOG BOX **NEW**.

In template projects you can place anything you need in a project:



When creating project templates, you are free to insert the already created page templates in the project template.

When you are working for different customers, you can create different project templates for each individual customer. You can also create project templates containing lists for different component vendors.

— Please read “Creating a template based on an existing project” on page 776 to see how to create a template based on an existing project.

— Please refer to “Converting Standard projects to Project templates” on page 776 to see how to use *standard projects* from previous versions of **Automation** as *templates* in this version.



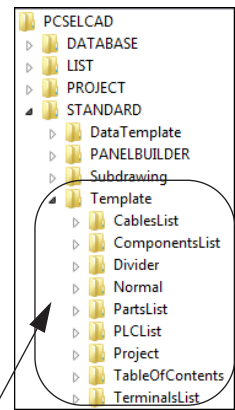
IN PROJECT TEMPLATES THE FOLLOWING IS MEMORISED

- ALL PAGE SETTINGS FOR EACH INDIVIDUAL PROJECT PAGE, AS DESCRIBED IN “IN PAGE TEMPLATES THE FOLLOWING IS MEMORISED” ON PAGE 773
 - ALL CREATED PROJECT DATA DATAFIELDS, AS WELL AS THEIR CONTENTS (IF ANY)
 - THE DEFINED REFERENCE DESIGNATIONS
 - THE APPLIED COMPANY LOGO
 - SETTINGS FOR TEXT PROPERTIES
 - SETTINGS FOR TEXT VISIBILITY
 - LAYER SETTINGS
 - ALL OTHER PROJECT SETTINGS
-

CREATING A TEMPLATE BASED ON AN EXISTING PROJECT

To create a template based on an existing project, do the following:

- 1) In the **Windows Explorer**: Right-click on the project file, and click on **Cut** — or click on **Copy** if you wish to keep the original project.
- 2) Open the folder **Pcscad\Standard\Template**: In this folder exists a folder for each type of templates.
- 3) Open the folder for the type of template you wish to create — in this case the **Project** folder —, by double-clicking on the folder.
- 4) Right-click inside the folder, and choose **Paste**.
- 5) The copied project is now inserted in the folder: Right-click on the project file, and choose **Rename**.
- 6) You can now edit the name of the file: Replace the extension **.pro** with the text **.std**, and press <Enter>.



Here you find the templates

You have now created a template based on an existing project. If you wish to edit the template, you can do this as described in “Editing templates” on page 777.


Correspondingly, you can save existing projects, containing one page only, as *page templates*. Please be careful to save the project page templates in the correct folders.

CONVERTING STANDARD PROJECTS TO PROJECT TEMPLATES

The projects called *standard projects* in version 4.0 (and previous versions) of **Automation** can be used as *templates* in this version.

To do so, you must follow the procedure described in “Creating a template based on an existing project” on page 776.

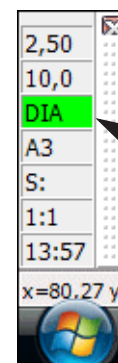
EDITING TEMPLATES

- 1) Choose **File => New**.
 - 2) In the dialog box **New**: Click on the tab on which the template is placed.
 - 3) Click on **Template** in the right-hand side of the dialog box.
 - 4) Right-click on the template, and click on **Open**.
- 
- 5) You now enter the **Template editor**, where you edit the template, and save it by clicking **Save**.
 - 6) Choose **File => Close** to leave the **Template editor**.



THE PAGE TYPE FIELD BLINKS BLUE IN THE TEMPLATE EDITOR

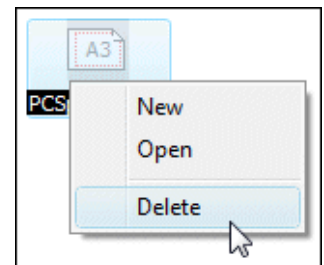
WHEN YOU ARE IN THE **TEMPLATE EDITOR**, THIS IS INDICATED BY THE **PAGE TYPE** FIELD, WHICH BECOMES BLUE AND STARTS BLINKING, IN THE LOWER-LEFT CORNER OF THE SCREEN.



Blinks green in the Template editor

DELETING A TEMPLATE

- 1) Choose **File => New**.
- 2) In the dialog box **New**: Click on the tab on which the template is placed.
- 3) Right-click on the template, and click on **Delete**.
- 4) Click **OK** to leave the dialog box.



TEMPLATE FOR CLICKING ON THE NEW DOCUMENT BUTTON



When you click on the **New document** button, you enter a project consisting of a single blank page.

However, you can create your own template project, which is used as template, when you click this button:

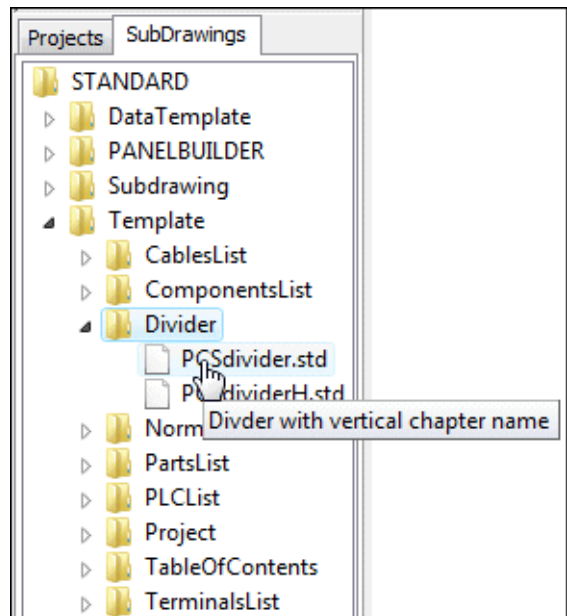
- 1) Create the project that you wish to use as project template.
- 2) Save the project in the folder **Pcscad\Standard\Template**.
- 3) In the Windows Explorer: Right-click on the project file, select **Rename**, correct the file name to **filenew.std** and press <Enter>.



Then project is now applied as template when clicking the **New document** button.

TEMPLATES IN Automation

Quite a few templates have been created in **Automation**. When you open the **Templates** folder in the **SubDrawings** tab in the **Explorer** window, just let the mouse rest an instance over a template name. This gives you a *hint* telling you what the template contains.





Updating, Modules and Licenses

In this section you get information about installing and updating program modules, and how to adapt the system to your needs.

You can read about:

- Installing new Modules..... p. 781
- Automatic program update p. 784
- Adapting the System to your needs..... p. 786
- User Defined short-cuts p. 788
- Colors..... p. 792
- Tanking licenses..... p. 794
- Reducing Repetitive Strain Injuries..... p. 800

In the appendix “Settings” on page 829 you can see how to change settings in the Settings menu.

MODULES

If your version of **Automation** allows you to work with modules, you have the following options:

THIS CHAPTER DESCRIBES	
— WHAT YOU CAN INSTALL	P. 781
— HOW TO INSTALL AND UNINSTALL.....	P. 782
— RE-INSTALLING OR UPDATING MODULES	P. 783
— STARTING A MODULE	P. 783

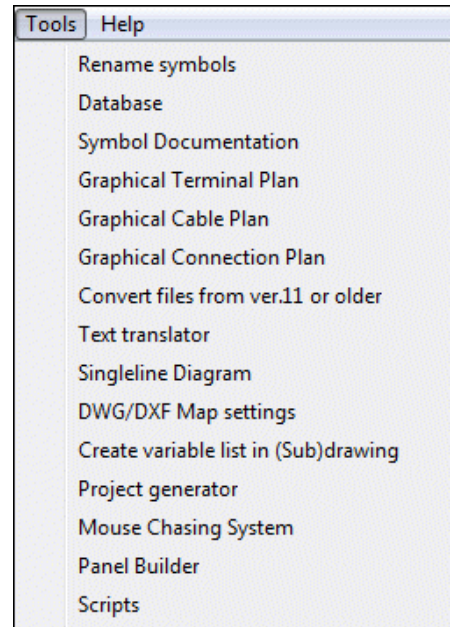
WHAT CAN YOU INSTALL

In **Automation** you find the extra tools in the **Tools** menu.

To use these tools, you must first install them as described in “How to install and uninstall” on page 782.

So, if you do not have as many options as displayed in the figure to the right, this is because some of these modules have yet not been installed.

Below you can get an overview over the different modules.

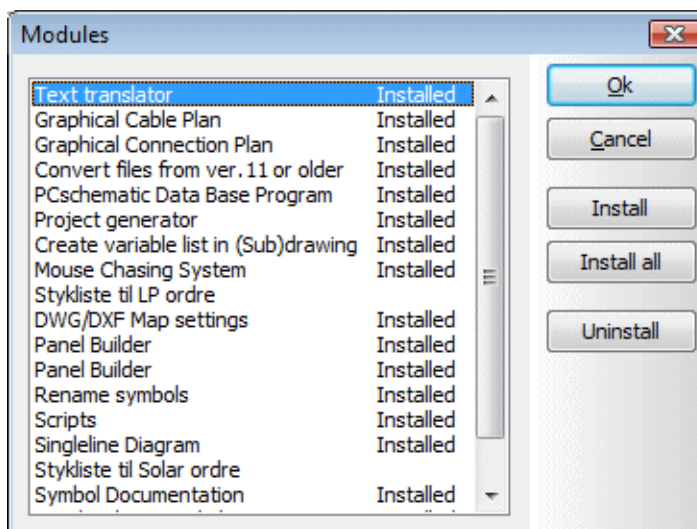


TOOL	DESCRIPTION
RENAME SYMBOLS	DESCRIBED IN “RENAME SYMBOLS” ON PAGE 289.
DATABASE	DESCRIBED IN THE PC SCHEMATIC DATABASE MANUAL.
SYMBOL DOCUMENTATION	FOR AUTOMATIC GENERATION OF SYMBOL DOCUMENTATION. DESCRIBED IN THE PC SCHEMATIC TOOLS MANUAL.
GRAPHICAL TERMINAL PLAN	FOR AUTOMATIC GENERATION OF GRAPHICAL TERMINAL PLANS. DESCRIBED IN THE PC SCHEMATIC TOOLS MANUAL.
GRAPHICAL CABLE PLAN	FOR AUTOMATIC GENERATION OF GRAPHICAL CABLE PLANS. DESCRIBED IN THE PC SCHEMATIC TOOLS MANUAL.
GRAPHICAL CONNECTION PLAN	FOR AUTOMATIC GENERATION OF GRAPHICAL CONNECTION PLANS. DESCRIBED IN THE PC SCHEMATIC GRAPHICAL CONNECTION PLANS MANUAL.
CONVERT FILES FROM VER11 OR OLDER	MODULE FOR CONVERTING PROJECTS CREATED IN VERSION 11 AND PREVIOUS VERSIONS TO UNICODE. DESCRIBED IN “UNICODE” ON PAGE 10.

TOOL	DESCRIPTION
TEXT TRANSLATOR	FOR TRANSLATING TEXTS IN PROJECTS. DESCRIBED IN THE PC SCHEMATIC TOOLS MANUAL.
SINGLELINE DIAGRAM	DESCRIBED IN "AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS" ON PAGE 409.
DWG/DXF MAP SETTINGS	FOR SETTING UP PARAMETERS FOR DWG/DXF IMPORT/EXPORT. DESCRIBED IN THE PC SCHEMATIC TOOLS MANUAL.
CREATE VARIABLE LIST IN (SUB)DRAWING	DESCRIBED IN "SUB DRAWINGS" ON PAGE 256.
PROJECT GENERATOR	DESCRIBED IN THE PROJECT GENERATOR MANUAL.
MOUSE CHASING SYSTEM	FOR REDUCING REPETITIVE STRAIN INJURIES — SEE "PREVENTING REPETITIVE STRAIN INJURIES" ON PAGE 800.
PANEL BUILDER	MODULE FOR BUILDING PANELS AUTOMATICALLY. DESCRIBED IN THE MINI MANUAL PANEL BUILDER - TUTORIAL , WHICH CAN BE DOWNLOADED AT www.pcschematic.com/manuals .
SCRIPTS	MENTIONED IN "DESIGNING YOUR OWN FUNCTIONS IN AUTOMATION" PÅ SIDE 7.

HOW TO INSTALL AND UNINSTALL

Installing and uninstalling can be done by choosing **File => Modules**, whereby you enter this dialog box:



Here you get a survey of which modules can be installed, as well as which modules are installed in your case.

In a net installation you can therefore see which modules can be installed on all of the PC's on the net, and which are installed on your machine. Opposed to this, the settings you see under **Settings => Tools** are individual settings for your PC only.

By clicking on a module on the list, you can install or uninstall the module by clicking on **Install** or **Uninstall** respectively.

If you have no need for converting **Automation** DOS files to **Automation** Windows files, you can click on **Symbol conversion, TYP to SYM**, and then click **Uninstall**.

Clicking on **Install all** will install all modules on your PC.

RE-INSTALLING OR UPDATING MODULES

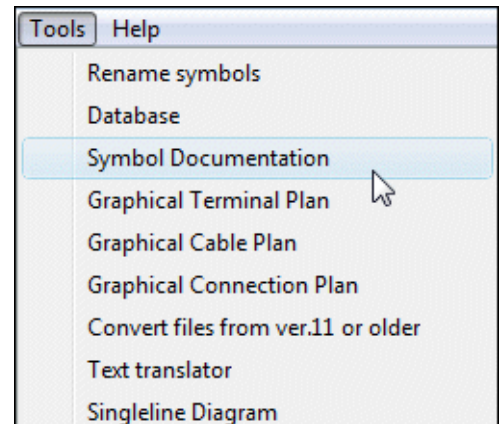
If you have uninstalled a module, you can install it again at any time, with no harm done.

If you need to update a module, you do not have to uninstall the module first. Just click on the module name, and click **Install**.

STARTING A MODULE

When a module is installed, a line is inserted in the dialog box **Settings => Tools**, and maybe also an extra button in the toolbar.

The module can then be started by first clicking on **Tools**, and then on the name of the module.



UPDATE AND AUTOMATIC INFORMATION

This chapter describes how you can update **Automation** via the internet, when new updates are created for the program version you are using. It is also described how you, via the internet, can receive information automatically from PC|SCHEMATIC A/S about **Automation**.

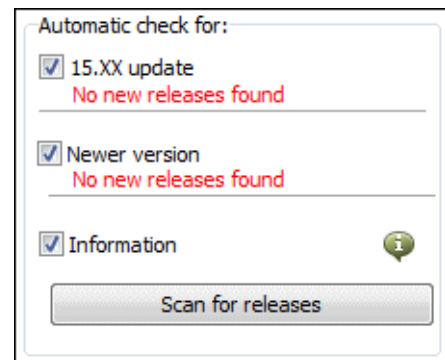
THIS CHAPTER DESCRIBES	
— UPDATING VIA THE INTERNET	P. 784
— INFORMATION FROM PC SCHEMATIC A/S VIA THE INTERNET	P. 785

UPDATING VIA THE INTERNET

When you have access to the internet from your PC, you can get **Automation** to examine automatically whether new updates exist for the version of the program, that you are working on. To do this, you must first choose **Settings => System**.

In the lower-left corner of the dialog box you can then choose:

- **15.XX update:** Hereby you automatically receive information when updates exist for the version you are using.
- **Newer version:** Hereby you automatically receive information when a new version is released.
- **Information:** Hereby you automatically get new information from PC|SCHEMATIC A/S — see “Information from PC|SCHEMATIC A/S via the internet” on page 785. When this automatic check is selected and new information is released, an information window will appear when you start the program. This will only take place the first time a new piece of information is released.
- **Scan for releases:** Hereby you can manually activate a search for new updates and new versions.



When new updates exist, this is displayed below the relevant checkbox. A button will also be displayed to the right of the checkbox. When you click this button, you enter PC|SCHEMATIC's website, from which you can download the update.

Icon for updates

When you have activated the automatic check, an icon is displayed in the lower-right corner of the screen. When you click the icon, you enter the dialog box **Settings => System**, as which is described above.

If you do not wish to see this message again, you can select **Do not show this message again**. — When a *new* patch is available, the message will pop up again.

When you right-click on the icon, a menu is displayed, which gives access to PC|SCHEMATIC's website, from which you can download the update.

INFORMATION FROM PC|SCHEMATIC A/S VIA THE INTERNET

If you have access to the internet on your PC, you can make **Automation** examine automatically whether important information about the program have been released. This information could e.g. be:

- If a critical error have been reported, which can cause loss of data. This way you can get this information the same moment the error is detected, along with a work-around on how to avoid the problem until the error is corrected.
- Information on changed release dates.

... as well as other information of this character.

Activating automatic information

The function can be set in the dialog box **Settings => System**, by selecting **Information** in the lower-right corner of the dialog box, as displayed in "Updating via the internet" on page 784.



If you click the button to the right of the check box, a list of the latest pieces of information are displayed.

You will only receive information from the program version you are working in.

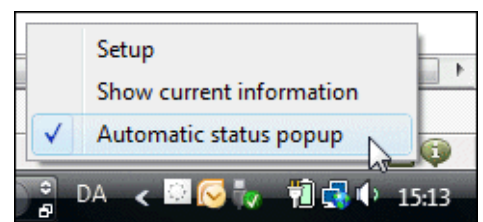
Icon for Information

When the function is activated, an icon is displayed in the lower-right corner of the screen. When you click the icon, a list of the latest pieces of information are displayed.



Automatic status popup

When you right-click on the icon and choose **Automatic status popup**, a pop-up window is displayed, when new information exist. If this function is not activated, the icon will start blinking when new information exist.



ADAPTING THE SYSTEM TO YOUR NEEDS

In **Automation** you can do quite a few things to adapt the system to your needs, in order to work faster and in a more appropriate way for you.

THIS CHAPTER DESCRIBES	
— WHERE YOU FIND THESE OPTIONS DESCRIBED.....	P. 786
— ADDITIONAL OPTIONS	P. 786

DESCRIBED IN OTHER CHAPTERS

You will find a summary of some of these options below:

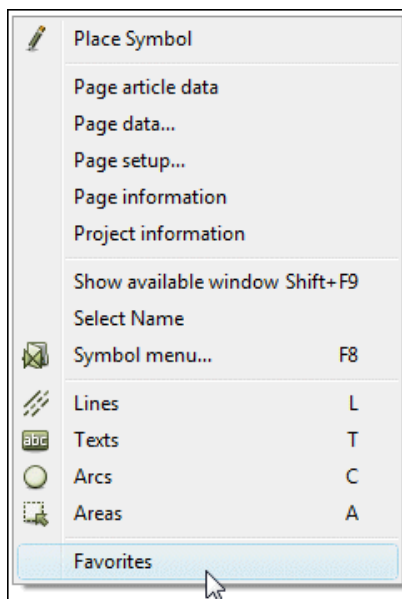
OPTION	PAGE
<i>DOCKING THE PROGRAM WINDOWS</i>	23
<i>INSTALLING EXTRA MODULES</i>	781
<i>DEFINE YOUR OWN SHORT-CUT KEYS</i>	788
<i>CHANGE THE SYSTEM COLORS</i>	792
<i>OTHER PROGRAM SETTINGS</i>	829

ADDITIONAL POSSIBILITIES

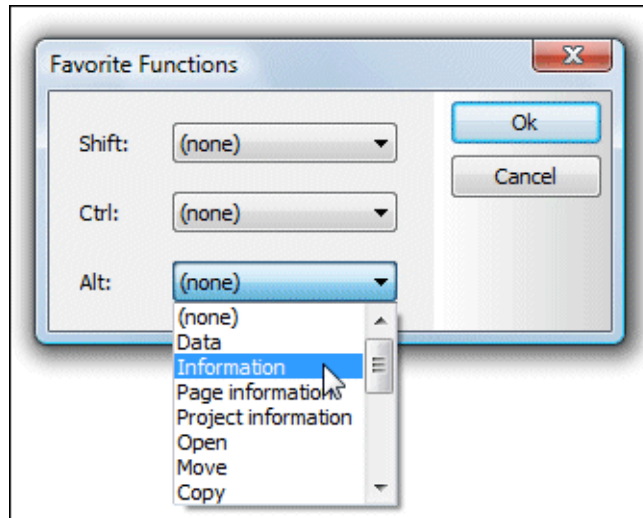
Apart from the possibilities mentioned above, you will find the following options:

Favorites

When you right-click anywhere in a project, it is now possible to choose **Favorites**.



This gives you the option to specify some favorite functions:



These functions are activated when you press down <Shift>, <Ctrl> or <Alt> and *right-click* on an object.

Specify character set

Under **Settings => System**, you can specify which character set shall be applied on the project pages.

The options are here **Standard**, **Latin2**, **Russian**, **Baltic** and **Polish**.

The information of the chosen character set is included in each project file separately.

More character sets will be created on request.

SHORT-CUT KEYS

In **Automation** you will find most commands accessible through short-cut keys. It is also possible to define your own short-cut keys in the program — as described in “User defined short-cut keys” on page 790.

FUNCTION	SHORT-CUT KEY	FUNCTION	SHORT-CUT KEY
AREAS	<A>	PRINT	<CTRL+P>
ARTICLE	<V>	RECTANGLE	<CTRL+F>
CHANGE STATE	<SHIFT + SPACEBAR>	REDRAW	<CTRL+G>
CIRCLES / ARCS	<C>	REPLACE	<CTRL+H>
COMPONENT GROUPING	<CTRL+F7>	REPLACE SYMBOL	<F4>
COORDINATES	<CTRL+ I>	ROTATE 90⁰	<SPACEBAR>
COPY	<CTRL+C>	SAVE PROJECT	<CTRL+S>
DATABASE	<D>	SELECT ALL	<CTRL+A>
DELETE		SHOW AVAILABLE	<F9>
DRAW ON/OFF	<INSERT>	SHOW AVAILABLE AGAIN	<CTRL+F9>
ELEVATION ON/OFF	<F11>	SLANTED LINES	<CTRL+K>
FIND	<CTRL+F>	SNAP, FINE	<SHIFT>
FIND ARTICLE	<V>	STRAIGHT LINES	<CTRL+R>
FIND NEXT	<F3>	SURVEY WINDOW ON/OFF	<F12>
FIND TYPE		SYMBOL MENU	<F8>
INSERT POTENTIAL	<P>	SYMBOLS	<S>
KEYBOARD ENTRY	<K>	TEXTS	<T>
LINES	<L>	TYPE	
NEXT	<F5>	UNDO	<CTRL+Z>
NEXT PAGE	<PAGEDOWN>	VIEW ARTICLE DATA	<CTRL+F2>
NEW DOCUMENT	<CTRL+N>	VIEW CONDUCTING	<SHIFT+F2>
OBJECT LISTER	<F7>	WRITE TEXT	<K>
ON-LINE MANUAL	<F1>	ZOOM	<Z>
OPEN	<CTRL+O>	ZOOM IN	<CTRL+HOME>
PRECEDING	<F6>	ZOOM, MOVE / PAN	<CTRL>+ARROW KEYS
PREVIOUS PAGE	<PAGEUP>	ZOOM OUT	<CTRL+END>
PREVIOUS SELECTED PAGE	<F10>	ZOOM TO PAGE	<HOME>

THE SHORT-CUT KEYS

Please note that using lower or upper case letters effects the short-cut function. Thus <Shift+c> will not activate the Circles command. Whether CapsLock is active makes no difference.

Right-clicking

To get further functions/options when working with the program, use right-click. For example you can right-click on a symbol, the pickmenu or on the entire page. Right-clicking in many cases causes a menu to appear, a menu containing functions and/or data dealing with whatever you are clicking on.

When in doubt, you can always try right-clicking.

Buttons with double functions

If you — without clicking — let the mouse rest for a moment on a button, a hint displaying the function of the button will be displayed in a tiny yellow rectangle below the button. If there is a plus (+) in front of the hint, this tells you that the button contains another function as well. This other function will be activated by pressing down <Ctrl> while clicking on the button.



For instance click on the **Symbol** button on the top part of the screen, and then let the mouse rest on the **Symbol menu** button on the left hand side of the screen. A yellow rectangle appear in which it says **+Symbol menu**, indicating you have got two options: You either just click the button, which brings you into the symbol menu, or if you press down <Ctrl> while clicking the button, you go directly into the database.

Or, experiment with the other buttons, and find out for yourself which other functions are included in the program buttons.

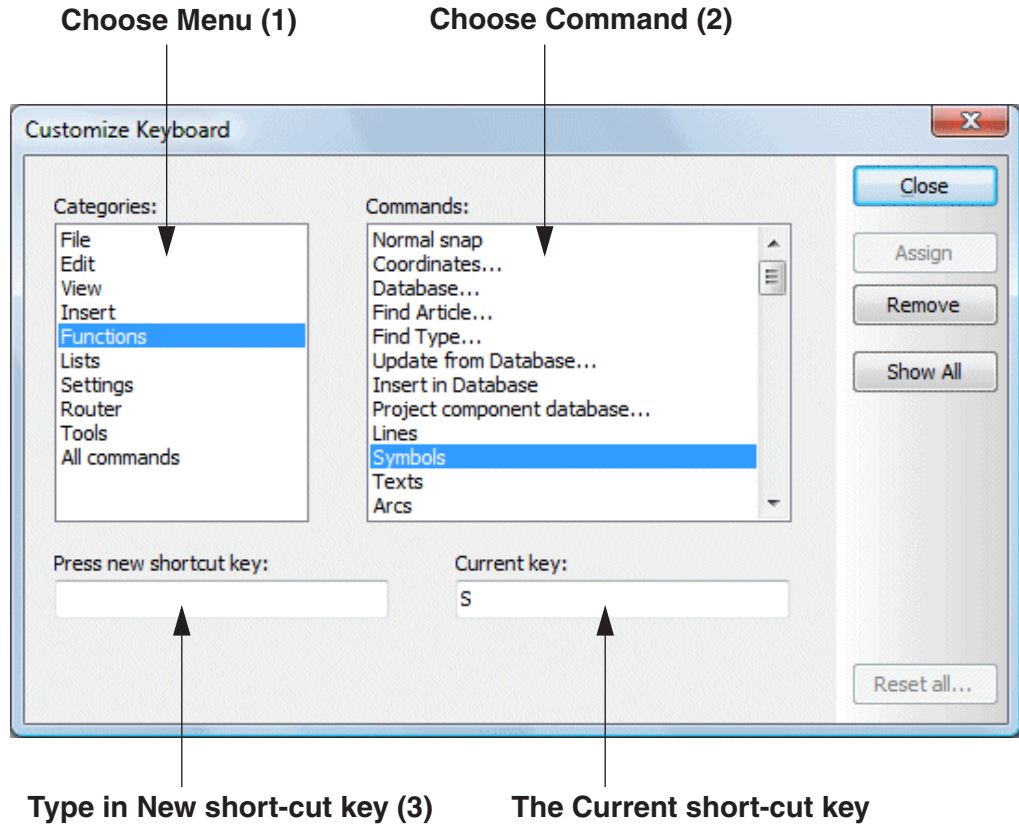
Faster short-cuts for activating the Pencil

— See “Short-cuts for activating the Pencil” on page 33.

USER DEFINED SHORT-CUT KEYS

In **Automation** you can design your own short-cuts. This is done by entering **Settings => Pointer/Screen => Shortcuts**.

You then see the following dialog box:



In this dialog box you can create short-cut keys for all commands in **Automation**. The predefined short-cut keys can also be changed.

In order to change or create a short-cut key, you do like this:

- 1) Choose the menu where the command is located
- 2) Choose the command
- 3) Click in the **Press new shortcut key** field and type in the new short-cut key
- 4) Click on **Assign**

The short-cut key is not assigned to the command before you click **Assign**.

If a short-cut key is already defined for a command, the short-cut key is displayed in the **Current key** field.

If you want to remove a short-cut key, you choose the command and click **Remove**.

Click on **Close** when you have finished your changes.

Reset all

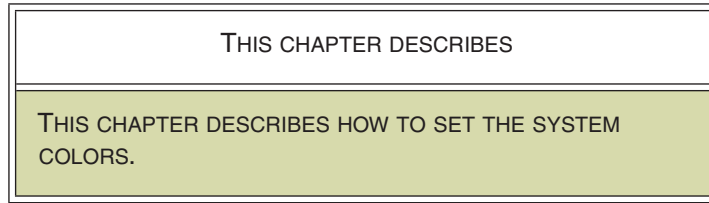
When you click **Reset all** in the lower-right corner of the dialog box, all short-cut keys are reset to their default values, which they were assigned when the program was installed. — See “Short-cut Keys” on page 788.

Displaying and printing short-cuts

When you click **Show all**, all short-cut keys are displayed in a list, which you can print by clicking on **Print**, or save it by clicking **Save**.

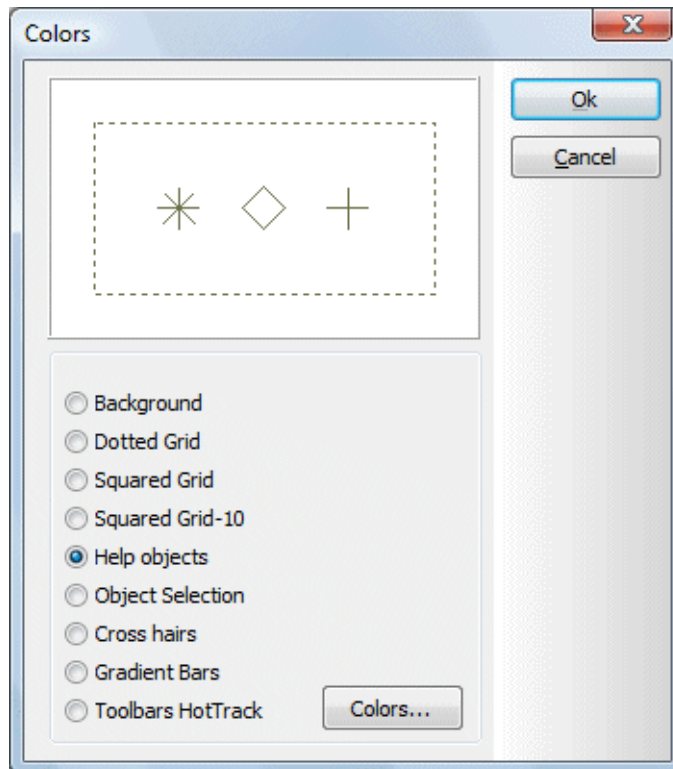
COLORS

In **Automation** you can set the system colors as you like.



To specify which colors shall be applied in **Automation**, you click on **Settings => Cursor/Screen => Colors**.

You then enter the following dialog box:



Here you determine which elements you want to change the color settings for. You get the following options:

OPTION	REMARKS
BACKGROUND	THE BACKGROUND COLOR OF THE SCREEN. IF BLACK BACKGROUND IS CHOSEN UNDER SETTINGS => POINTER/SCREEN , THE BACKGROUND COLOR CANNOT BE DETERMINED HERE.
DOTTED GRID	THE COLOR OF THE DOTS IN DOTTED GRID.
SQUARED GRID	THE COLOR OF THE LINES IN SQUARED GRID.
SQUARED GRID-10	HERE YOU CAN DETERMINE THE COLOR OF EACH TENTH LINE IN THE SQUARED GRID, FOR INSTANCE TO HIGHLIGHT THESE LINES.

OPTION	REMARKS
HELP OBJECTS	THE COLOR OF REFERENCE POINTS, THE CENTRE OF CIRCLES, THE CURRENT PATH NUMBERS, THE DISPLAYING OF PAGE MARGINS, THE APPEARANCE OF THE ORIGO E.T.C. IN SHORT: EVERYTHING WHICH IS HELP INFORMATION ON THE PAGES, BUT ARE NOT DISPLAYED WHEN YOU PRINT OUT PROJECT PAGES.
OBJECT SELECTION	WHEN YOU CHOOSE AN OBJECT IN Automation , THIS IS INDICATED BY A COLORED BOX COVERING THE OBJECT. IT IS THE COLOR OF THIS BOX YOU DETERMINE HERE.
CROSS HAIRS	HERE YOU DETERMINE THE COLOR OF THE CROSS HAIRS. PLEASE NOTE, THAT YOU HERE HAVE TO CHOOSE THE COLOR <i>COMPLIMENTARY</i> TO THE COLOR YOU WANT TO SEE ON THE SCREEN.
GRADIENT BARS	IF YOU HAVE SELECTED GRADIENT TOOLBARS UNDER SETTINGS => POINTER/SCREEN (SEE "POINTER/SCREEN" ON PAGE 836), YOU CAN HERE SPECIFY THE COLORS FOR THIS. THE THREE MOST APPLIED COLORS (FROM WINDOWS 2000, WINDOWS XP AND WINDOWS VISTA) ARE PREDEFINED UNDER USERDEFINED COLORS IN THE DIALOG BOX COLOR .

Click on **Colors** when you have chosen which color settings you want to change.

Here you either click on a basic color, or define a color in the right-hand side of the dialog box. If you have defined a color by yourself, and click on **Add to custom colors**, the color is placed under **Custom colors** in the left-hand side of the dialog box.

TANKING LICENSES

THIS CHAPTER DESCRIBES	
— WHAT IS LICENSE TANKING	P. 794
— LIMITING HOW MANY LICENCES TO TANK	P. 794
— TRANSFERRING LICENSES BY LICENSE TANK	P. 795
— TRANSFERRING LICENSES BY LICENSE FILE	P. 796
— STATUS BUTTON FOR TANK STATUS.....	P. 799
— SUPERPRO MONITOR	P. 799



NET LICENSE KEY REQUIRED (FLOATING VERSION)

TO MAKE USE OF TANKING LICENSES, THE SYSTEM, WHICH GIVES AWAY THE LICENSE, MUST HAVE A NET LICENSE KEY.

WHAT IS LICENSE TANKING

When you are using a net license key on **Automation**, you have the option of *tanking* licenses. This meaning, that you have the option of giving away a license *from* the PC on which the key is mounted, *to* a PC which has no license key. This can be done for a period of time between 10 minutes and 30 days.

This gives this PC the possibility of working independent from the net, which the PC with the license key is attached to.

Licenses can be tanked in two ways:

- 1) By tanking a license to a PC which is on the net for the moment — see “Transferring licenses by License tanking” on page 795.
- 2) By transferring a license file to a PC which is not on the net — see “Transferring licenses via license tanking” on page 796.

LIMITING HOW MANY LICENCES TO TANK

It is possible to limit how many licenses, that can be tanked from a license key, so that e.g. only 3 out of 5 licenses can be tanked. The function makes sure that there is always a predefined amount of licenses available in the company. The license keys can only be programmed by PC|SCHEMATIC A/S.

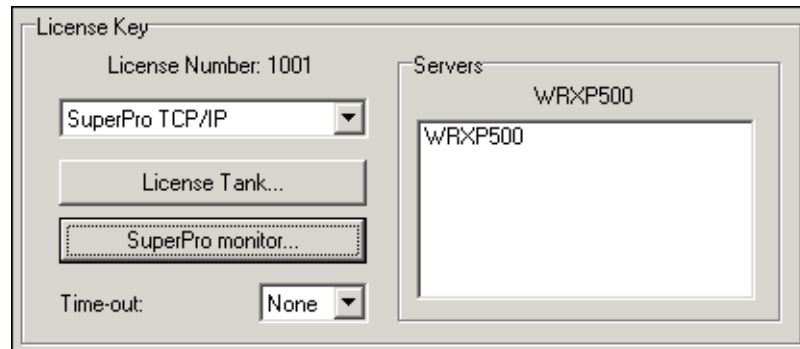
It is also possible to log the license tanking, so that the administrator can see who is tanking the licenses.

Please contact your local support service for further information about this.

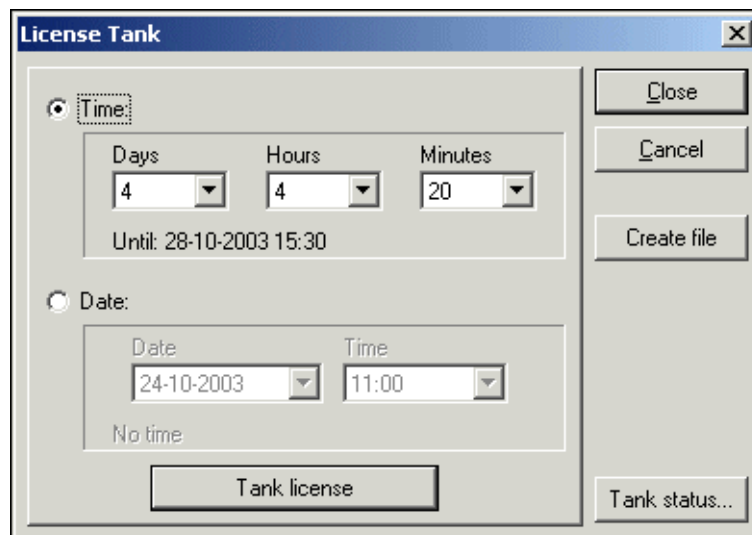
TRANSFERRING LICENSES BY LICENSE TANKING

To tank a licence *to* the PC which you are working on for the moment — *from* a net license key placed on another PC —, do the following:

- 1) Choose **Settings => System**, and the license key settings are displayed on the lower-left corner of the tab:



- 2) Click on **License Tank**:



- 3) In the dialog box **License Tank**: Select a **Time** or a **Date** and click on **Tank license**.
- 4) You are now informed whether you are allowed to tank the license: Click **OK**.
- 5) In the dialog box **License tank**: Click **Close**.

The net license key hereby loose a license for the specified period of time. When the tanking period expires, the licence automatically returns to the net license key.

Releasing a tanked license

If you wish to release a license, which has been tanked as described above, choose **Settings => System => License Tank**, and click on **Release the tanked license**.

TRANSFERRING LICENSES VIA LICENSE TANKING

When you wish to transfer a license from a PC, which you can not access via a network, do this:

- 1) Create a license file in **Automation** with information about your home PC (the PC that needs the license), for use when transferring the license. — See “Creating a license file” on page 796.
- 2) Send the file via e-mail/floppy disc to the person administrating **Automation** licenses on the system, from which you wish to tank a license.
- 3) The administrator opens a special program for transferring licenses via file/e-mail, transfers a license to the file you sent, and returns the file to you. — See “Giving away licenses via license files” on page 797.
- 4) You receive a file, load it in **Automation**, and use the license for the assigned period of time. — See “Loading a license file containing a license” on page 796.



ASSIGNED LICENSE FILES CAN NOT BE RETURNED

WHEN YOU ARE ASSIGNED A LICENSE VIA A LICENSE FILE, IT IS NOT POSSIBLE TO RETURN THIS LICENSE.

IF FOR INSTANCE YOU HAVE BEEN ASSIGNED THE LICENSE FOR A PERIOD OF 20 DAYS, YOU CAN NOT RETURN THIS LICENSE IN THESE 20 DAYS.

THE SYSTEM, FROM WHICH YOU RECEIVED THIS LICENSE, WILL THEREFORE NOT BE ABLE TO GET THIS LICENSE BACK, BEFORE THE 20 DAYS ARE OVER.

THE LICENSE CAN ONLY BE USED ON YOUR PC.

Creating a license file

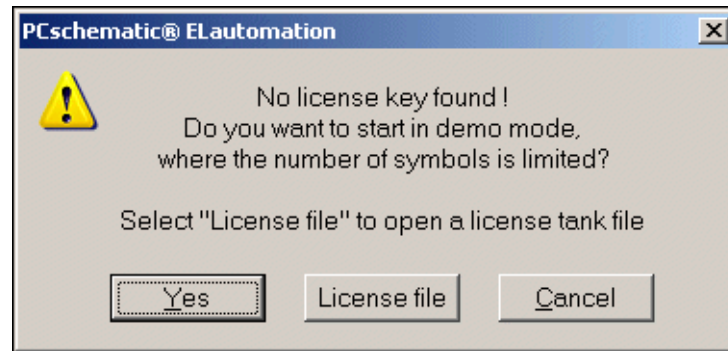
- 1) Choose **Settings => System**: The license key settings are displayed in the lower-left corner — as displayed in “Transferring licenses by License tanking” on page 795.
- 2) Click on **License Tank**.
- 3) In the dialog box **License Tank**: Click on **Create file**.
- 4) In the dialog box **Save license file**: Specify where to save the license file (the **Pcscad** folder is suggested), and click **OK**.
- 5) In the dialog box **License file**: Click on **Close**.

Now send an e-mail to the administrator for the system, from which you wish to receive a license. In this e-mail you mention for how long you want the license (max. 30 days), and attach the license file to the e-mail.

Loading a license file containing a license

When you receive a license file containing a license from the administrator, you can load the license in **Automation** like this:

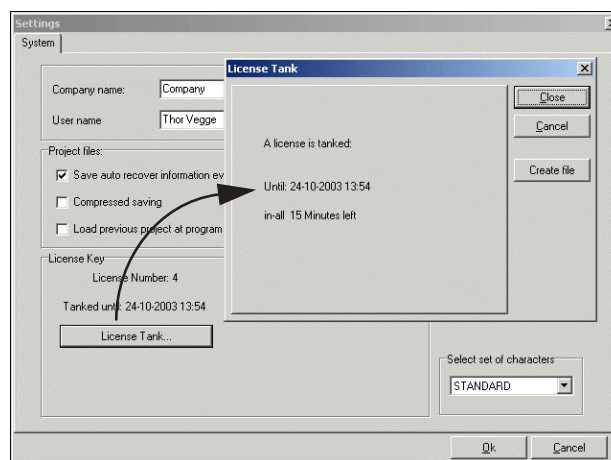
- 1) Place the license file on your harddisk — for instance in the folder **Pcselead**.
- 2) Start **Automation**. You now get the following message:



- 3) Click on **License file**.
- 4) In the dialog box **Open**: Click on the license file, and click on **Open**.
- 5) **Automation** is now started, and you can work with the program in the specified period of time.

See for how long you will keep the license

When you have tanked a license, and choose **Settings => System => License Tank**, you can see for how much longer you have the license:



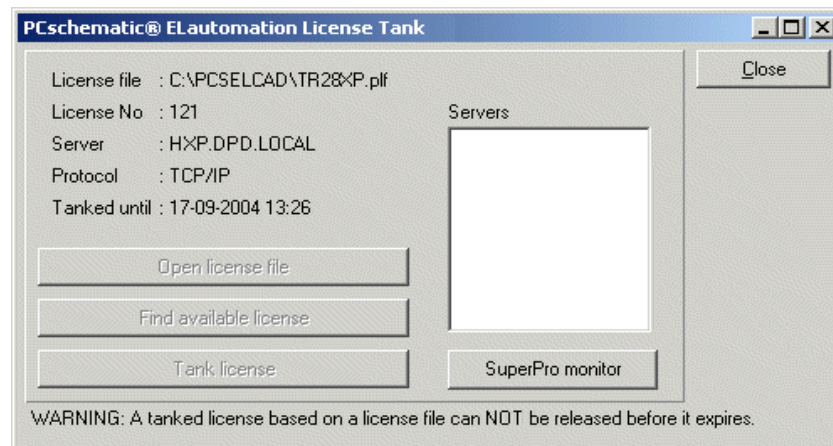
When the tanked license expires

If the PC is connected to a network when the tanked license expires, **Automation** automatically looks for an available license, and continues normally, if an available license is found.

GIVING AWAY LICENSES VIA LICENSE FILES

If you are the administrator of the licenses on a system with a net license key, and receive a license file, you can give away a license like this:

- 1) In Windows Explorer: Start the program **PCsTank.exe** — which for instance is placed on the server in the **Pcseicad** folder, or on your own PC.



- 2) In the program **Automation License Tank**: Click on **Open license file**.
- 3) In the dialog box **Open**: Click on the license file, which you received, and click on **Open**.
- 4) In the program **Automation License Tank**: Click on **Find available license**.
- 5) When an available license is found: Click on **Tank license**.
- 6) In the dialog box **License Tank**: Specify for how long to tank the license, and click on **Tank license**.
- 7) You now get a message telling for how long the license is tanked: Click **OK**.
- 8) In the program **Automation License Tank**: Click on **Close**.

The received license file has now got a license assigned for the specified period of time, and you return the license file via e-mail/floppy disk.

Please note, that *it is the same file you send back*. No new file is created for this purpose.



ASSIGNED TANKED LICENSE FILES CAN NOT BE RETURNED

WHEN YOU GIVE AWAY A LICENSE VIA A LICENSE FILE, YOU CAN IN NO WAY RETURN THIS LICENSE BEFORE THE SPECIFIED PERIOD OF TIME EXPIRES.

IF FOR INSTANCE YOU HAVE GIVEN AWAY THE LICENSE FOR A PERIOD OF 20 DAYS, THE SYSTEM CAN NOT GET THIS LICENSE BACK BEFORE THE 20 DAYS ARE OVER.

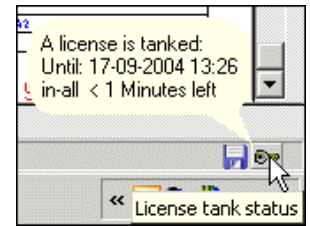
THE LICENSE CAN ONLY BE USED ON THE PC, ON WHICH THE LICENSE FILE WAS ORIGINALLY CREATED.

STATUS BUTTON FOR TANK STATUS

In the right-hand side of the status bar a **Status /Info function** button for tank status is displayed, when a license have been tanked.

When you click on the button, the remaining tanked time is displayed. The function is activated automatically when the program is started, and when half of the remaining tanked time has been used. The next time it is activated is when half of the now remaining time has been used (i.e. 3/4 of the total time) — e.t.c.

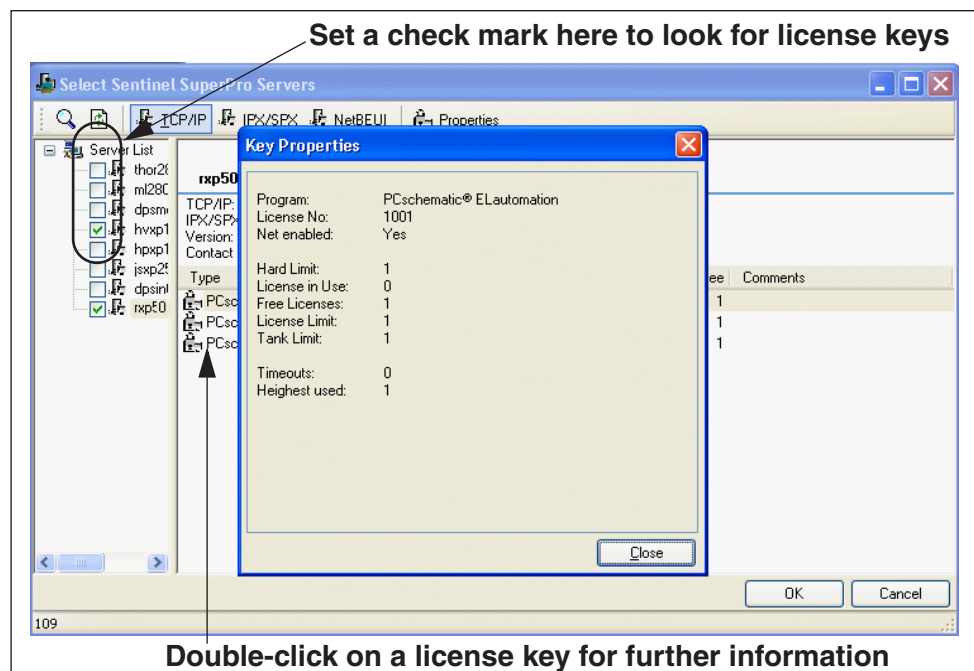
When you right-click on the button, a menu is displayed from which the automatic pop-up information can be disabled.



SUPERPRO MONITOR

If you choose **Settings => System => SuperPro monitor**, you enter a net license key monitor, where the available net license keys are displayed.

Here you can select the license server(s), on which the program shall look for **Automation** license keys.



When you select a PC in the left-hand side of the dialog box, its license keys are displayed in the right-hand side of the dialog box.

When you double-click on one of these license keys in the right-hand side, you get further information.

PREVENTING REPETITIVE STRAIN INJURIES

In order to prevent repetitive strain injuries — and at the same time save working hours — **Automation** includes the special developed program **Mouse Chasing System**, which registers which menu functions and buttons you click on during your daily work. From the basis of this information, you can then block for clicking on specific functions, and thereby learn the short-cuts to the functions gradually.

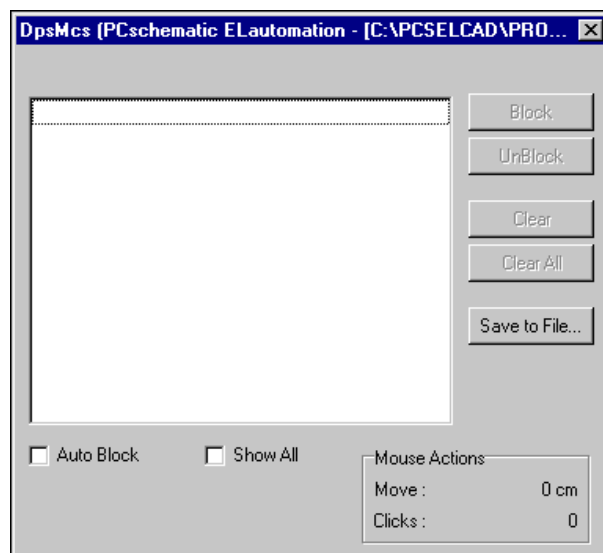
HOW TO INSTALL **Mouse Chasing System**

IF **Mouse Chasing System** IS NOT AN OPTION IN THE **TOOLS** MENU, YOU INSTALL IT LIKE THIS:

- 1) CHOOSE **FILE => MODULES**
- 2) YOU NOW ENTER THE DIALOG BOX **MODULES**: CLICK ON **Mouse Chasing System**, AND CLICK ON **INSTALL**
- 3) CLICK **CLOSE** TO LEAVE THE DIALOG BOX

STARTING THE **Mouse Chasing System**

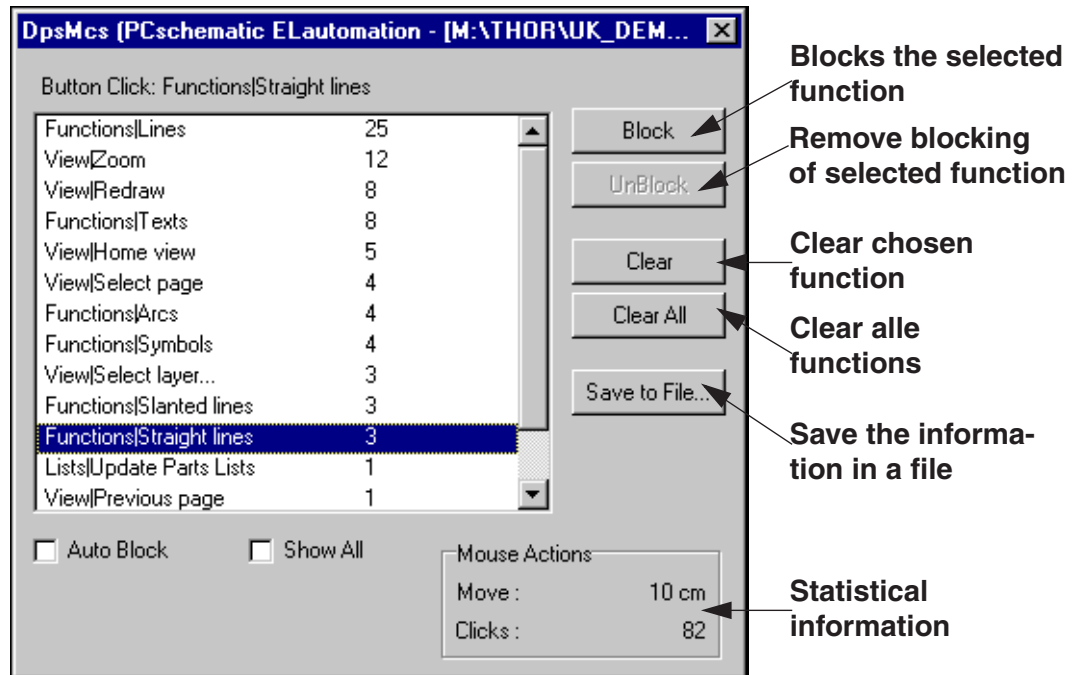
To start the **Mouse Chasing System** choose **Tools => Mouse Chasing System**. You have now started the **Mouse Chasing System**, and the following dialog box is displayed:



When you click anywhere in the **Automation** program, the dialog box disappears. Click the **Mouse Chasing System** button at the bottom of the screen when you want to see the dialog box again.

The system registers what you click on

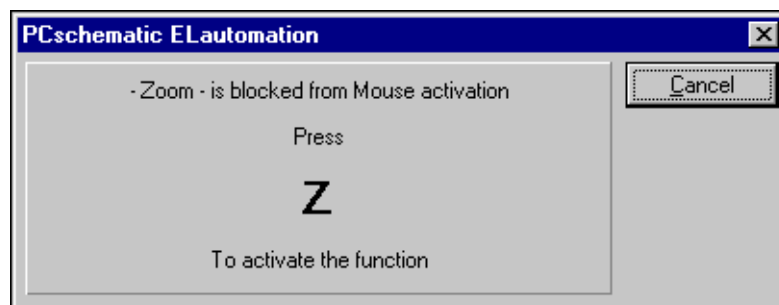
The **Mouse Chasing System** now registers all the menu selections you make and all the buttons you click on, and makes statistics about how many times you click in a menu or at a button.



When you click on a function in the dialog box, and then click on **Block**, the function is blocked when you *click* on it in **Automation**.

Clicking on a blocked function in Automation

When you click on a blocked function in **Automation** you get the following message:



This way the program informs you about the short-cut key to the function. When you press this key, the dialog box disappears automatically, and the function is activated.

If no short-cut key is defined, you can create a short-cut key as described in “User defined short-cut keys” on page 790.

Automatic blocking

When you click on **Auto Block** you can specify that the program shall automatically block all functions which have been clicked on more than for instance 100 times.

When you click on **Show all**, all menu choices in **Automation** are shown. You can then choose which functions to block.



Learning the short-cut keys by need

Because the mouse chaser tells you which functions you click on the most, you can concentrate on learning these functions. If for instance you block the three most used functions, you can learn these without using any special effort.

When you have learned the short-cut keys for these three functions, you can then block the three functions you now click on the most — and keep on doing so until you have learned the short-cut keys you need to know.

Disabling the Mouse Chasing System

You can at any time disable the blocking of the clicking on a function, by first clicking on the function in the **Mouse Chasing System** and then clicking on **Unblock**.

When you want to close the **Mouse Chasing System**, just choose **Tools => Mouse Chasing System** so that the check mark in front of **Mouse Chasing System** disappears.

Starts automatically along with Automation

When you do not close **Mouse Chasing System** before closing **Automation**, the **Mouse Chasing System** closes when you exit **Automation**. It will then start automatically the next time you start **Automation**. This way you can perform statistics on which functions you click on for a longer period of time.

THE MOUSE CHASING SYSTEM CAN BE CLOSED DOWN

When the **Mouse Chasing System** is closed down, **Automation** is informed about this, so that it knows that the **Mouse Chasing System** has been closed down.



Access codes, Registration e.t.c.

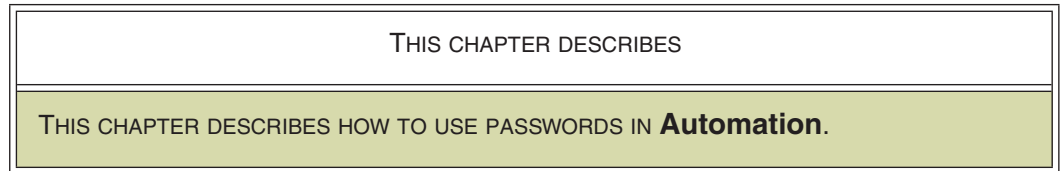
In this section you get information about network security checks, revision control, registration of file types e.t.c.

You can read about:

- Access codes on projects p. 805
- Locking project files..... p. 806
- Revision control..... p. 808
- Registering PCISCHMATIC file types p. 813
- Indexing p. 816
- Starting functions and DDE p. 821
- Compressed saving / Zipping p. 825

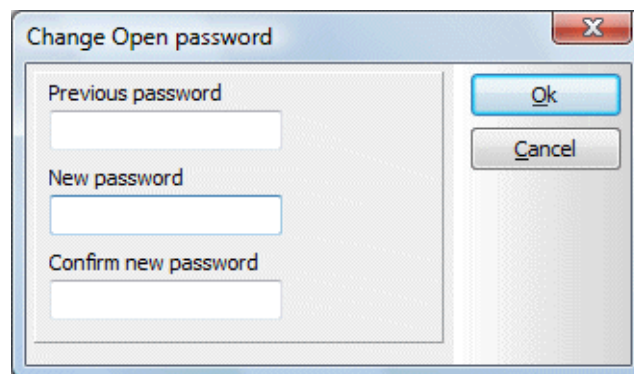
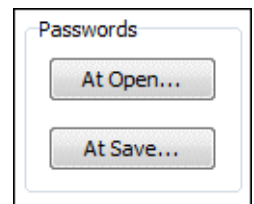
PASSWORDS ON PROJECTS

In **Automation** you can create a password for your project files, in order to prevent others from opening or saving them.



When you choose **Settings => Project data**, you can click on one of the password buttons.

You then enter this dialog box:

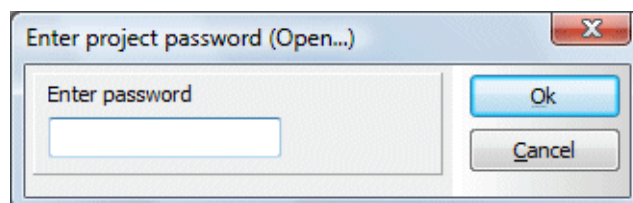


If a password already exists for the project, you first have to enter the previous password in the field **Previous password**.

Then type in the new password in **New password**, and type in the new password once more in **Confirm new password**.

Click **OK**.

The next time the project is *opened* or *saved* — depending on the password type used — you must type in the password and click **OK**.



LOCKING PROJECT FILES

Due to the security on net installations, the program control which program files are opened for the moment.

THIS CHAPTER DESCRIBES	
— LOCKING OF A PROJECT IN USE.....	P. 806
— LOCKING AFTER SYSTEM BREAK DOWN.....	P. 806
— SHARING LICENSES ON A NETWORK	P. 807

LOCKING OF A PROJECT IN USE

If you open a project file, which is being used by another user for the moment, you get a message informing you that another user is already working on the project, and gives you the name of the other user.

However, you are allowed to open a copy of the project. *But* when you want to save your copy of the project, you are only allowed to save it under another name. This takes place to prevent you from overwriting the changes made by the other user — and to prevent the other user from overwriting your changes as well.

When the original project is no longer occupied, you are can open it and make additional changes to the project.

LOCKING AFTER SYSTEM BREAK DOWN

When a project is not closed down in the authorized way — for instance due to power failure — **Automation** does not release the project.

If a new user then tries to open the project, he will be informed that the old user is still working on the project, even though this is not the case. The new user will therefore not be allowed to save the project under the original project name.

When this is the case, the old user just have to enter the project. A message is then displayed, telling the old user, that the project is already open. The old user must then answer **Yes** to re-open the project, and close it again. It is not required to save the project at this occasion.

Deleting the %pr file manually

If this procedure fails, the problem can be solved using the Windows Explorer.

Automation have placed a security file in the same folder as the project. The file has the same name as the project, but has the extension **%.pr**. If the name of the project is **example.pro**, the name of the security file will therefore be **example.pr**.

By entering the Explorer and deleting this file, the project is again accessible for other users.

Please note, that **Automation** deletes this file under normal circumstances, when the project is closed down.

SHARING LICENSES ON A NETWORK

On network installations, your license can be taken by others, if you have not worked on your PC for ten minutes. When you start working again, you get a message telling you that your license has been lost to another user.

If you then click **Yes** to search for a free license, **Automation** finds an available license for you, if possible. If no license is available, you will have to restart the program to start working again.

REVISION CONTROL

THIS CHAPTER DESCRIBES	
— WHAT IS REVISION CONTROL.....	P. 808
— STARTING A REVISION	P. 808
— THE PAGE MENU AND THE OBJECT LISTER DURING REVISION CONTROL	P. 809
— PRINTING DURING REVISION CONTROL.....	P. 809
— ENDING A REVISION	P. 810
— DELETING REVISION INFORMATION.....	P. 811
— DATAFIELD FOR DISPLAYING LAST REVISION	P. 811

WHAT IS REVISION CONTROL

When you start the revision control in **Automation**, the program keeps track of on which *pages* in the project, changes has been made.

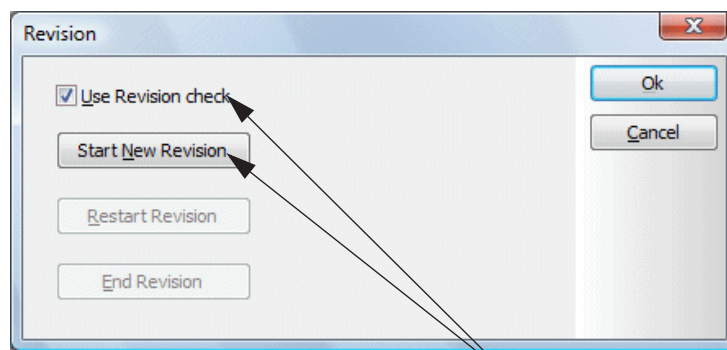
However, there are not kept track of *which* changes have been made.

It is subsequently possible to see where changes have been made — see “Page menu and Object Lister during Revision control” on page 809.

It is also possible to print the edited pages — see “Printing during revision control” on page 809.

STARTING A REVISION

- 1) Choose **Settings => Project data => Revision**



Click here

- 2) Select **Use Revision check**
- 3) Click on **Start New Revision**

The system now memorizes on which pages changes are being made subsequently.

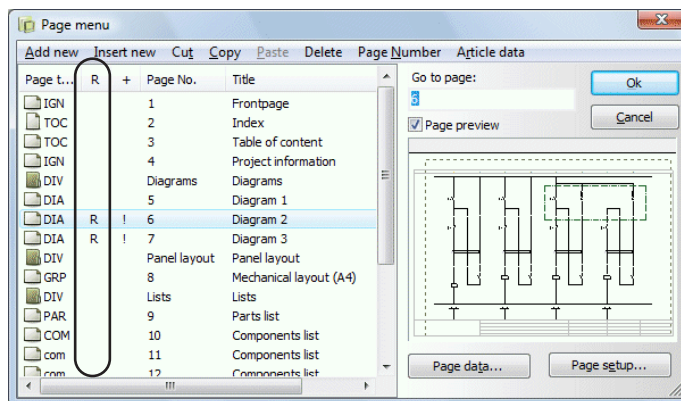
PAGE MENU AND OBJECT LISTER DURING REVISION CONTROL

When you have started a revision — see “Starting a revision” on page 808 — it is possible in the **Page menu** and the **Object Lister** to see on which pages changes have been made.

The Page menu during revision control

When you have started a revision, an extra column is displayed in the **Page menu**, named **R**. In this column an **R** is displayed for the pages, which have been changed since the start of the revision.

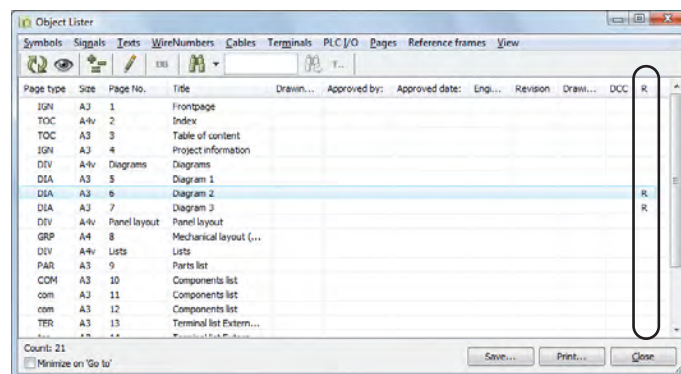
— Read more about the **Page menu** in “The Page menu” on page 493.



The Object Lister during revision control

When you have started a revision, the **Object Lister** for pages can contain an extra column, named **R**. In this column an **R** is displayed for the pages, which have been changed since the start of the revision.

You enter the **Object Lister** for pages by choosing **Object Lister => Pages**.



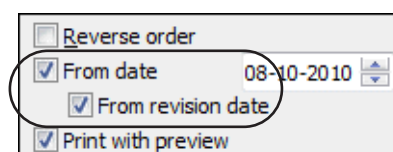
If the **R** column is not displayed, choose **View => Pages => Revised**. (Please note that the contents of the column **Revision** comes from the user defined Page data datafield of the same name). — Read more about pages in the **Object Lister** in “Editing Page data in the Object Lister” on page 315.

PRINTING DURING REVISION CONTROL

To print the pages, which have been changed since starting the revision control:



- 1) Click on the **Print pages** button
- 2) Select **All pages**
- 3) In the dialog box **Print**: Click on the tab **Settings**
- 4) Select **From date** and then select **From revision date**
- 5) Click on **OK**



The edited pages are now printed. — Read more about the dialog box **Print** in “Printing” on page 326.

ENDING A REVISION

- 1) Choose **Settings => Project data => Revision**
- 2) Choose **End Revision**

The revision is now ended. The next time you enter the dialog box **Revision**, the revision period is displayed.

If you subsequently try to edit pages in the project, you are informed that the revision has ended. However, you will get this message each time you try to make changes.



ENDING A REVISION AND DELETING REV. DATA IS NOT THE SAME

ENDING A REVISION AND DELETING REVISION DATA IS NOT THE SAME THING.
— SEE “DELETING REVISION INFORMATION” ON PAGE 811.

The Object Lister and the Page menu after ending a revision

When the revision has been ended, the **R**-column in the **Object Lister** and the **Page menu** change function: The **R**-column now displays the pages, which have been changed since the revision ended. — See “Page menu and Object Lister during Revision control” on page 809.

Creating a password after ending a revision

If you wish to make sure that no changes are made after ending a revision, you can subsequently enter a password for the project. — Read more about this in “Passwords on Projects” on page 805.

DELETING REVISION INFORMATION

- 1) Choose **Settings => Project data => Revision**
- 2) Deselect **Use Revision check**



ALL REVISION INFORMATION IS DELETED

PLEASE NOTE THAT *ALL* REVISION INFORMATION IS HEREBY *DELETED* FROM THE PROJECT. — SEE “ENDING A REVISION” ON PAGE 810 ALSO.

- 3) You are now informed that all revision settings are deleted: Click **OK**
- 4) In the dialog box **Revision**: Click **OK**
- 5) In the dialog box **Settings**: Click **OK**

DATAFIELD FOR SHOWING LAST REVISION

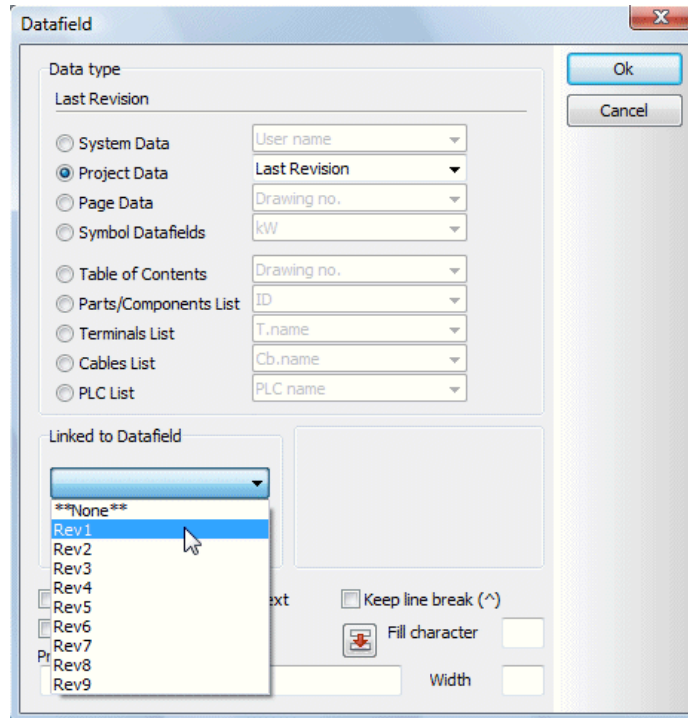
To include a history in the revision information in **Automation**, you can create consecutively numbered revision datafields in the menus **Page data** and **Project data**. These datafields can be filled out as the revisions are made.

When information about the last revision shall be placed on a project page, you place the datafield **Page revision**, which fetches the contents from the last filled out of these datafields.

Do for instance the following:

- 1) Create the datafields **Rev1**, **Rev2**, ... , **Rev9** under **Settings => Project data** — see “Page data and Project data Datafields” on page 689, specially “Adding New datafields to Project data and Page data” on page 692.

- 2) At the position where you need information about the last revision, you then insert the **Project data** datafield **Last revision**, and select one of the datafields **Rev1 ... Rev9** below **Linked to datafield** in the dialog box **Datafield**:



- 3) Fill out the datafields with revision information as the revisions are performed:

Rev1	1/7-2008
Rev2	12/8-2009
Rev3	17/9-2010
Rev4	8/10-2010
Rev5	
Rev6	
Rev7	

- 4) At the position where you placed the **Last revision** datafield in the project, the contents of the last filled out revision datafield is now displayed (In this example the contents of the datafield **Rev4**):

Last Revision: 8/10-2010



CAN ALSO ATTACH HISTORY TO OTHER DATAFIELDS

YOU CAN ALSO USE THE DATAFIELD **LAST REVISION** WITH A SEQUENCE OF OTHER DATAFIELDS WITH IDENTICAL NAMES, WHEN YOU NEED TO ATTACH HISTORY INFORMATION TO THESE DATAFIELDS.

REGISTRATION OF PC|SCHEMATIC FILE TYPES

THIS CHAPTER DESCRIBES	
— REGISTRATION OF PC SCHEMATIC FILE TYPES.....	P. 813
— REGISTRATION OF .PRO FILES	P. 814
— REGISTRATION OF .SYM FILES	P. 814

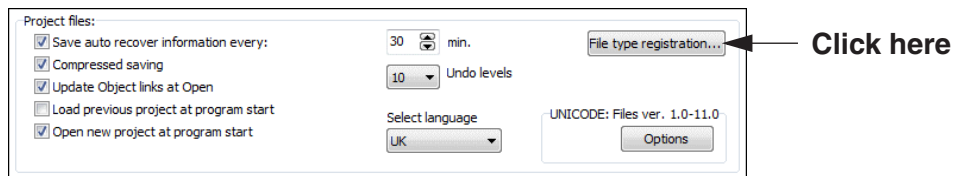
REGISTRATION OF FILE TYPES

In order to for instance start **Automation** projects directly from Windows Explorer, you must register the program file type (*.pro) in Windows. Hereby you tell Windows which program to use for starting *.pro files.

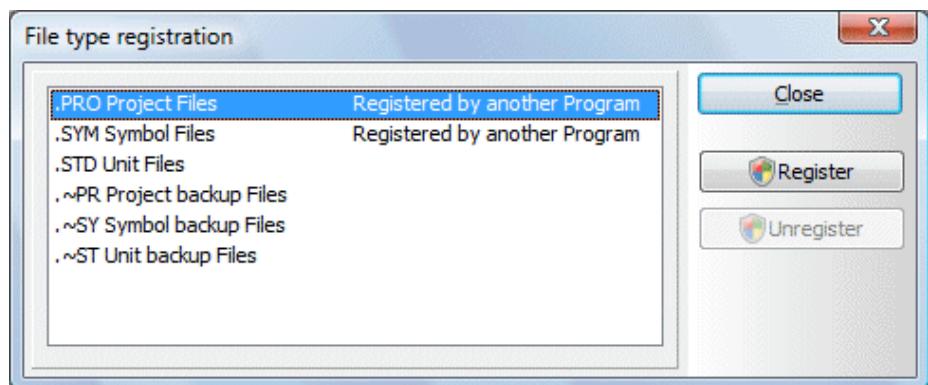
When you have done this, you furthermore get the option to apply the Windows indexing services for searching for project texts and properties — see “Indexing” on page 816.

To register the program file type in Windows, do this:

- 1) Choose **Settings => System** and click on **File type registration**:



- 2) You then enter the dialog box **File type registration**:



- 3) To register **.pro** files, first click on **.PRO Project Files** and then on **Register**. The file type is now registered. (**.sym** files are registered correspondingly).

When a file type is already registered by another program, this will be displayed in the dialog box.

When a file type is already set as **Registered** (this meaning registered by **Automation**), no harm will be done by clicking **Register** again.

Unregistering

To unregister a file type, select the file type and click on **Unregister**.

THE EFFECT OF REGISTERING .PRO FILES

When *.**pro** files are registered, this has the following effect:

- Existing **Automation** projects can be started directly from the Windows Explorer. Right-click on the **Automation** project in the Windows Explorer, and choose **Open**.
- New **Automation** projects can be created directly from the Windows Explorer. Right-click on the **Automation** project in the Windows Explorer, and choose **New**.
- Projects can be printed from the Windows Explorer. Right-click on the **Automation** project in the Windows Explorer, and choose **Print**.
- When you right-click on a project in the Windows Explorer and choose **Properties => Project data**, the project data are displayed for the selected project.
- The first five Project data datafields are displayed as a *hint* in the Windows Explorer, when the mouse cursor rests over the project file.
- A PC|SCHEMATIC icon is displayed in front of **Automation** files in the Windows Explorer.
- It becomes possible to apply the Windows Indexing services to search for project texts and properties, as described in “Indexing” on page 816.

Creating new projects from the menu in the Windows Explorer

If you wish to start creating a new project from the Windows Explorer, do this:

- 1) In the Windows Explorer: Click on the folder in which you want to create the new project
- 2) Choose **File => New => PC|SCHEMATIC project file**
- 3) In the selected folder you now type in the name of the project, and press <Enter>
- 4) Double-click on the file, and the project is opened in **Automation**.

THE EFFECT OF REGISTERING .SYM FILES

To register **.sym** files, see “Registration of file types” on page 813.

When *.**sym** files are registered, this has the following effect:

- **Automation** symbols can be opened in the **Symbol editor** directly from the Windows Explorer. Right-click on the **Automation** symbol in the Windows Explorer, and choose **Open**.
- New symbols can be created directly from Windows Explorer. Right-click on the **Automation** symbol in the Windows Explorer, and choose **New**.
- A PC|SCHEMATIC icon is displayed in front of **Automation** symbols in the Windows Explorer.

- It becomes possible to apply the Windows Indexing services to search for symbol texts, as described in “Indexing” on page 816.

INDEXING

When you use **Automation**, it is possible to search on the texts in your project and symbol files directly from Windows, via Windows indexing service — or from an html page in a web browser.

THIS CHAPTER DESCRIBES	
— WHAT IS WINDOWS INDEXING SERVICE.....	P. 816
— STARTING WINDOWS INDEXING SERVICE ON YOUR PC.....	P. 816
— SEARCHING VIA WINDOWS INDEXING SERVICE.....	P. 818
— SYNTAX FOR USING WINDOWS INDEXING SERVICE.....	P. 819



ONLY AVAILABLE ON WINDOWS XP

THIS OPTION IS ONLY AVAILABLE ON WINDOWS XP.

WHAT IS WINDOWS INDEXING SERVICE?

Windows indexing service makes it possible to search on project/document information directly from Windows — without **Automation** have to be started.

The information, that can be searched on, includes both texts in documents/projects (document contents) and the document properties — such as who has created the document.

Windows indexing service can only be used for programs, which are prepared for it — such as e.g. Microsoft Office, Adobe Acrobat and **Automation**.

When the indexing service is activated, Windows use some time to run through all documents on your PC, to create the index, which succeedingly makes the background for the search. This might take some time — as described in “Starting Windows indexing service on Windows XP” on page 816.

STARTING WINDOWS INDEXING SERVICE ON WINDOWS XP



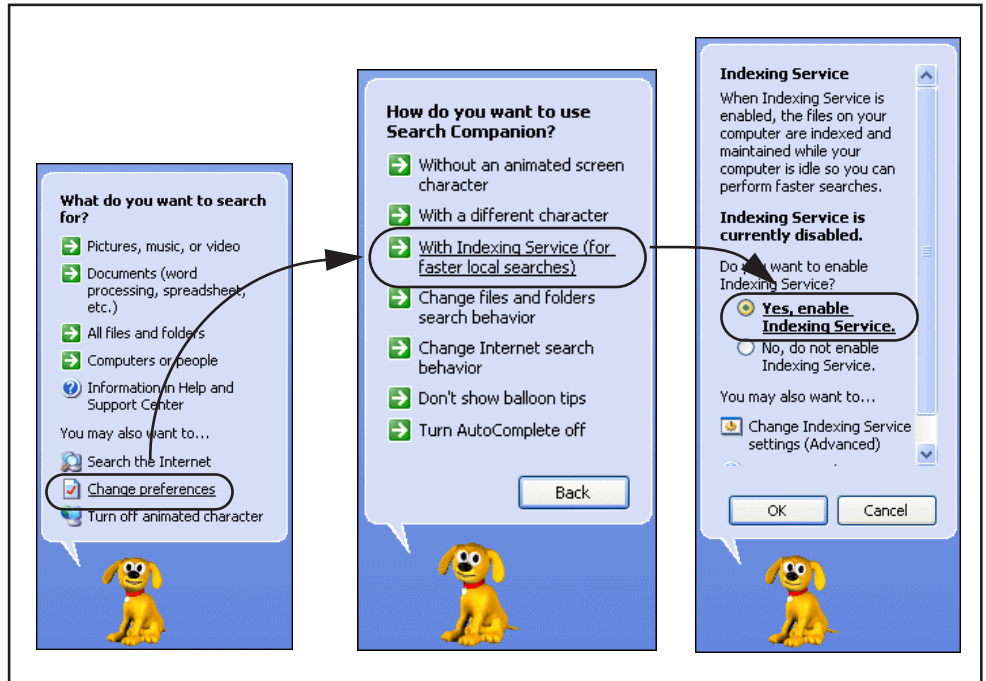
THE PC|SCHEMATIC FILE TYPES MUST BE REGISTERED

TO USE THE WINDOWS INDEXING SERVICE FOR PROJECT FILES, THE FILE TYPE **.PRO** MUST BE REGISTERED. CORRESPONDINGLY THE FILE TYPE **.SYM** MUST BE REGISTERED BEFORE YOU CAN APPLY THE WINDOWS INDEXING SERVICE FOR SYMBOLS.

— READ MORE ABOUT THIS IN “REGISTRATION OF PC|SCHEMATIC FILE TYPES” ON PAGE 813.

To start Windows indexing service, do the following in Windows:

- 1) Click on the **Start** button at the lower-left corner of the screen
- 2) Choose **Search**
- 3) Click on **Change preferences** in the lower-left corner
- 4) Click on **With Indexing Service**, and then click on **Yes, enable Indexing Service** and click **OK**



The indexing service is now activated, and Windows now spend some time to run through all documents on your PC, to create the index which is the basis for the succeeding search — as described in “Searching for project texts via Windows indexing services” on page 818.

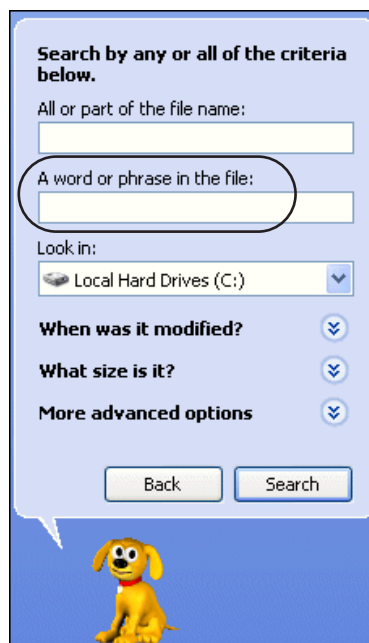
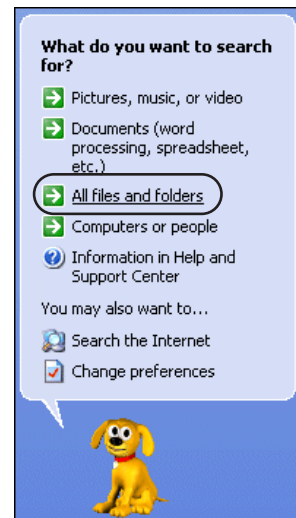
The creation of this index takes place in the background, so you can continue working on your PC while the index is created.

Please note, that if you have many files on your PC, it can be a good idea not to turn off the PC for one night while Windows creates the index.

SEARCHING FOR PROJECT TEXTS VIA WINDOWS INDEXING SERVICES

When you have activated Windows indexing service — as described in “Starting Windows indexing service on Windows XP” on page 816 —, you can search like this:

- 1) Click on the **Start** button in the lower-left corner of the screen
- 2) Choose **Search**
- 3) Click on **All files and folders** in the lower-left corner
- 4) You can now type in what you are searching for in the file **A word or phrase in the file:**

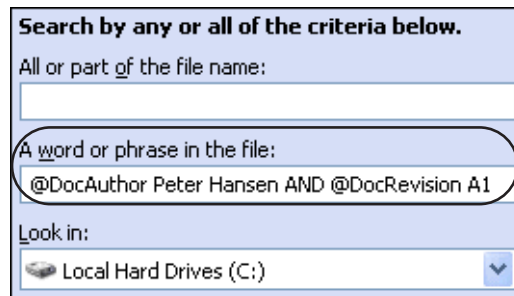


The syntax for how you can search is described in “Syntax for using Windows indexing service” on page 819.

- 5) Type in the search text, and click on **Search**.
- 6) The result of the search is now displayed in the right-hand side of the window.

SYNTAX FOR USING WINDOWS INDEXING SERVICE

When you use Windows indexing service — as described in “Searching for project texts via Windows indexing services” on page 818 — you type in a search text in the field **A word or phrase in the file:**



In this file you type in a property (e.g. *@DocAuthor*) followed by a search value (e.g. *Peter Hansen*). Hereby there are searched for whether the property e.g. contains the text *Peter Hansen*.

If more than one criterion must be fulfilled at the same time, just write **AND** between the criteria — as in the figure above.

If the one *or* the other criterion must be fulfilled, write **OR** between the criteria: *@DocTitle Eldocumentation OR @DocCompany Machinefactory*. — For further options, please refer to the Windows on-line help.

If you are only searching for the beginning of a text, or for parts of a text, you can use wildcards (e.g. *@DocTitle *document**) — as described in “Wildcards/jokers” on page 550.

You have the following options:



PROPERTY	FUNCTION
@DocTITLE	HERE YOU CAN SEARCH FOR <i>PROJECT TITLES</i> — I.E. THE CONTENTS OF THE FIELD TITLE IN THE DIALOG BOX SYSTEM => PROJECT DATA — AND <i>SYMBOL TITLES</i> — I.E. THE TITLE SPECIFIED IN THE SYMBOL EDITOR WHEN YOU CLICK ON THE SYMBOL SETTINGS BUTTON.
@DocCOMMENTS	HERE YOU CAN SEARCH FOR <i>PROJECT REMARKS</i> — I.E. THE CONTENTS OF THE FIELD REMARKS IN THE DIALOG BOX SETTINGS => PROJECT DATA .
@DocAUTHOR	HERE YOU CAN SEARCH FOR WHO HAS DESIGNED/CREATED THE PROJECT. HERE ARE ONLY SEARCHED FOR PROJECT DATA DATAFIELDS WITH THE NAMES CONSTRUCTOR, AUTHOR, CREATOR AND DESIGNER . IF YOU HAVE PLACED THIS INFORMATION IN A PROJECT DATA DATAFIELD WITH ANOTHER NAME, YOU WILL NOT BE ABLE TO FIND IT THIS WAY.
@DocREVISION	HERE YOU CAN SEARCH FOR THE REVISION NUMBER OF THE PROJECT. THERE ARE ONLY SEARCHED FOR PROJECT DATA DATAFIELDS WITH THE NAMES REVNUMBER, REVISION AND REV . IF YOU HAVE PLACED THIS INFORMATION IN A PROJECT DATA DATAFIELD WITH ANOTHER NAME, YOU WILL NOT BE ABLE TO FIND IT THIS WAY.
@DocCOMPANY	HERE YOU CAN SEARCH FOR CUSTOMERS/COMPANIES. THERE ARE ONLY SEARCHED FOR PROJECT DATA DATAFIELDS WITH THE NAMES CUSTOMER AND COMPANY . IF YOU HAVE PLACED THIS INFORMATION IN A PROJECT DATA DATAFIELD WITH ANOTHER NAME, YOU WILL NOT BE ABLE TO FIND IT THIS WAY.

PROPERTY	FUNCTION
@DocCATEGORY	HERE YOU CAN SEARCH ON SYMBOL TYPE. YOU CAN SEARCH FOR THE TYPES NORMAL, RELAY, OPEN, CLOSE, SWITCH, MASTER REFERENCE, WITH REFERENCE, TERMINAL, PLC, NON CONDUCTING OR SUPPORT .
@CONTENTS	HERE YOU CAN SEARCH FOR ALL TYPES OF TEXTS IN THE PROJECT — E.G. SYMBOL TYPES, FREE TEXTS, PAGE TITLES AND THE CONTENTS OF DATAFIELDS. IF YOU JUST WRITE A TEXT WITH NO PRECEDING PROPERTY (I.E. JUST WRITE <i>PETER HANSEN</i>), THERE WILL ALSO BE SEARCHED AMONG ALL KINDS OF TEXTS IN THE PROJECT.

STARTING FUNCTIONS, BATCH PRINTING AND DDE

It is both possible to open a **Automation** file on a specified page, and to start **Automation** files directly from the Explorer in Windows. It is also possible to start **Automation** from expert systems.

Furthermore it is possible to batch print **Automation** projects.

These options are described in the following.

THIS CHAPTER DESCRIBES	
— STARTING FUNCTIONS	P. 821
— COMMAND LINES FOR CREATING PDF, DWG AND DXF FILES.....	P. 822
— BATCH PRINTING	P. 823
— DDE FUNCTIONS.....	P. 823

STARTING FUNCTIONS

Automation contains the following starting functions, which can all be activated from the command line:

1. Loading a project with a specified page open

It is possible to open a **Automation** file on a specified project page. This is done using the command line parameter **/P**.

If you want to load the project **edemo.pro** and display page **7**, you type:

```
C:\PCSECAD\PCSECAD.EXE EDEMO.PRO /P7
```

2. Loading a project with a specified symbol selected

It is possible to open a **Automation** file with a specified symbol selected. This is done using the command line parameter **/S**.

If you want to load the project **edemo.pro** with the symbol **K1** selected, you type:

```
C:\PCSECAD\PCSECAD.EXE EDEMO.PRO /SK1
```

The first project page containing a **K1** symbol will be displayed, and all **K1** symbols in the project will be selected.

Correspondingly you can type `C:\PCSECAD\PCSECAD.EXE EDEMO.PRO /SK1:13`, which causes all **K1** symbols to be selected, and the project to be opened on the project page where **K1:13** is placed.

3. Opening only a few project pages

It is also possible only to load individual project pages. This is done using the command line parameter **/L**.

If for instance you only want to open page 3 from the **edemo.pro** project, you type:

```
C:\PCSECAD\PCSECAD.EXE /L[3] EDEMO.PRO
```

If for instance you want to open the pages 3, 5 and 7 from **edemo.pro**, you type:

```
C:\PCSECAD\PCSECAD.EXE /L[3][5][7] EDEMO.PRO
```

The name of the opened pages becomes part of the project name in the limited project which is opened. Therefore you do not risk to save the limited version of the project which replaces the original project. In the example mentioned above, the limited project will be named **edemo.[3][5][7].pro**.

4. Open template project with partslist from another system

You can open **Automation** from another software system/program, so that a template project is opened together with a partslist, which contains the components to be placed in the project.

Write for instance:

```
C:\PCSELCAD\PCSELCAD.EXE C:\PCSELCAD\STANDARD\Template.pro /IP
C:\PCSELCAD\LIST\partslist.xls
```

Here the first part is the path to the template project, that you wish to use.

The parameter /IP means "Import Partslist".

The third part is the path to the partslist containing the components, that you wish to use in the project. The partslist can both be in the formats *.xls and *.xml.

This enables you to place symbols in the project based on the partslist, as described in "Placing symbols via lists" on page 674.

COMMAND FILES FOR SAVING PROJECTS AS PDF, DWG AND DXF FILES

In **Automation** you can create command line files, which automatically saves **Automation** projects as PDF, DWG or DXF files.

Do this:

- 1) Create a text file — for instance in **Notepad** — which contains a command line for **Automation**. In "Examples on command lines for creating PDF files" on page 822 below, you can see examples on "SAVEASPDF" files of the type *.cmd.
- 2) Start **Automation** with this **cmd** file as parameter in the command line — for instance **C:\Pcscad\Pcscad.exe C:\PATH-TO-CMD-FILE\SAVEASPDF.cmd**

Examples on command lines for creating PDF files

Below you can see some examples on command lines, which you can save in a *.cmd file, so that PDF files are created automatically from **Automation** projects:

```
FILESAVEAS .PDF C:\Pcscad\Project\*.pro
```

All .pro-files in the folder **C:\Pcscad\Project** are saved as PDF files in the same folder.

```
FILESAVEAS .PDF C:\Pcscad\Project\*.pro C:\Pcscad\Project\PDFSAVE
```

All .pro-files in the folder **C:\Pcscad\Project** are saved in the folder **C:\Pcscad\Project\PDFSAVE** as PDF files.

The folder **C:\Pcscad\Project\PDFSAVE** must be created before executing the command.

```
FILESAVEAS .PDF @C:\Pcscad\Project\ProList.txt
```

All project files in the listfile **C:\Pcscad\Project\ProList.txt** are saved as PDF files in the same folder as each individual PRO file.

```
FILESAVEAS .PDF @C:\Pcscad\Project\ProList.txt C:\Pcscad\Project\PDF-  
SAVE
```

All project files in the listfile **C:\Pcscad\Project\ProList.txt** are saved in the folder **C:\Pcscad\Project\PDFSAVE** as PDF-files.

The folder **C:\Pcscad\Project\PDFSAVE** must be created before executing the command.

Command lines for creating DWG and DXF files

To create DWG or DXF files using command lines, you just replace **.PDF** with **.DWG** or **.DXF** respectively in the examples above.

BATCH PRINTING

The following function for batch printing in **Automation** exists, which can be activated from the command line, or e.g. from a schedule program:

Batch printing from stand-alone installation

To print the file **C:\batch.pro** (where **Automation** is installed in the folder **C:\Pcscad**), write this:

```
C:\PCSELCAD\PCSELCAD.EXE -xp C:\batch.pro
```

— When you need to print more than one file, write the names of these after **C:\batch.pro**, only separated by a space character:

```
C:\PCSELCAD\PCSELCAD.EXE -xp C:\batch.pro C:\batch2.pro
```

Batch printing from net installation

To print the file **C:\batch.pro** (where **Automation** is installed in the folder **F:\Pcscad**), write this:

```
F:\PCSELCAD\PCSELCAD.EXE /n:C:\PCSELNET -xp C:\batch.pro
```

Here **F:\PCSELCAD** is the system catalog, while **/n:C:\PCSELNET** is the workstation catalog.

— When you need to print more than one file, write the names of these after **C:\batch.pro**, only separated by a space character:

```
F:\PCSELCAD\PCSELCAD.EXE /n:C:\PCSELNET -xp C:\batch.pro C:\bat2.pro
```

DDE FUNCTIONS

In **Automation** exists DDE functions, which makes it possible to right-click on a **.pro** file in the Windows Explorer, and then getting the options **Open**, **New**, and **Print**.

Thus it is possible to open a file by double-clicking on it — or by right-clicking and choosing **Open**. Also a new project can be opened by right-click **New**, and a project can be printed by right-click **Print**.

Registration of Automation

From the command line **.pro** files for **Automation** are registered using the parameter /REGISTER — for instance type:

```
C:\PCSECAD\PCSECAD.EXE /REGISTER
```

The program only have to be started once this way.

The registration is removed by C:\PCSECAD\PCSECAD.EXE /UNREGISTER.

DDE registration also has the consequence that once you have started **Automation**, and opens a file from the Windows Explorer, this file is opened in the running program.

COMPRESSED SAVING / ZIPPING PROJECTS

THIS CHAPTER DESCRIBES	
— WHAT IS ZIPPING	P. 825
— SET THE PROGRAM TO AUTOMATIC ZIPPING	P. 825
— GAINS BY ZIPPING IN Automation	P. 825

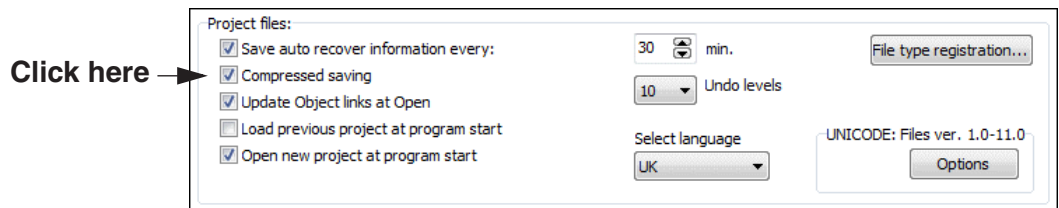
WHAT IS ZIPPING?

The Zip program is so commonly used for compressing files, that it is close to becoming an international standard for file compression.

Therefore automatic zipping of project documents have become an integrated option in **Automation**.

SET THE PROGRAM TO AUTOMATIC ZIPPING

When you wish to set **Automation** to save project files in the zip format automatically, you can choose **Settings => System** and activate **Compressed saving**:



The projects still gets the extension ***.pro**, though they are zipped.

Opening zipped project files

When a project is opened, **Automation** automatically analyses whether the project is zipped or not. When a project is saved in the zip format — from **Automation** — it is opened like any other project.

It therefore has no consequence for your working process with the program, if you choose compressed saving.

ADVANTAGES BY ZIPPING IN Automation

When you set **Automation** to automatic zipping, the size of the projects are reduced to about 8-12% of the original size, and *both* loading and saving are faster for projects on networks. For projects on local disks, there will be no time difference.

The compression follows the zip standard completely. Therefore you can choose to change the extension of a zipped project document to **.zip**, after which you can let the document be unzipped by a common unzipping program.



Appendices

In this section you can:

- Specify program Settings p. 829
- See the Datafields in the program p. 855
- Options in the PCSTYPE field..... p. 866
- Options in the PINDATA field p. 868
- Options for Variables p. 871
- Look up a term in the Dictionary p. 873
- Use the Index p. 879

SETTINGS

THIS CHAPTER DESCRIBES

IN THIS CHAPTER YOU CAN READ ABOUT WHICH TABS YOU CAN SELECT IN THE DIALOG BOX **SETTINGS**, AND GET FURTHER INFORMATION ON THE VARIOUS SETTINGS IN THE DIALOG BOX.

A SURVEY OF THE CONTENTS OF THE TABS, EXISTS IN "THE TABS IN THE SETTINGS MENU" ON PAGE 829.

CHOICES IN THE DIALOG BOX SETTINGS

The dialog box **Settings** is a menu in **Automation**. If you wish to be able to select all tabs in the dialog box, choose **Settings => Settings**. (If the tab you are looking for is not displayed in the dialog box for the moment, you can click the right- and left-arrows in the upper right-hand corner of the dialog box).

When you wish just to choose one of the menu points, you can choose this directly from the menu: When for instance you wish to access the tab **Tools** directly, you therefore choose **Settings => Tools**. Hereby the dialog box **Settings** is opened, and the tab **Tools** is selected. Please note that the tab **Tools** then becomes *the only* tab in the dialog box.

Entering Settings without an active project

It is possible to enter the **Settings** dialog box when no projects are open. Tabs in **Settings**, which are not project related, are not displayed when there are no active projects.

THE TABS IN THE SETTINGS MENU

The tabs in the dialog box **Settings** includes:

TAB	CONTENTS
PROJECT DATA <i>P. 831</i>	SPECIFICATION OF DATAFIELDS CONCERNING THE ENTIRE PROJECT. PASSWORDS, PROJECT REMARKS, REFERENCE DESIGNATIONS, MOUNTING CORRECT DRAWING, PROJECT LOGO, REVISION CONTROL AND RELAY REFERENCE TYPE.
PAGE DATA <i>P. 834</i>	SPECIFICATION OF DATAFIELDS CONCERNING THE PAGE YOU ARE WORKING ON IN THE PROJECT. PLACING A DRAWING HEADER ON THE PAGE. REFERENCE SETTINGS E.G. FOR CURRENT PATHS, ARTICLE DATA FOR THE PAGE, AND REFERENCE DESIGNATIONS FOR THE PAGE.
POINTER/SCREEN <i>P. 836</i>	SETTINGS DEALING WITH WHAT IS DISPLAYED ON THE SCREEN, AND HOW IT IS DISPLAYED. CHOOSE SYSTEM COLORS, AND USER-DEFINED SHORT-CUT KEYS.
DIRECTORIES <i>P. 840</i>	SPECIFICATION OF WHERE DIFFERENT TYPES OF FILES WILL BE SUGGESTED SAVED/LOADED FROM, AS WELL AS FOR WHICH TYPES OF FILES A BACKUP WILL BE MADE. SPECIFICATION OF THE ORDER IN WHICH THE SYMBOL FOLDERS WILL BE EXAMINED, WHEN THE PATH TO A SYMBOL IS NOT SPECIFIED (ALIAS).

TAB	CONTENTS
PAGE SETUP P. 843	SPECIFICATION OF PAPER SIZE, PAGE TYPE, PAGE SCALING, AS WELL AS GRID AND SNAP FOR THE PAGE YOU ARE WORKING ON.
TOOLS P. 845	HERE YOU SPECIFY WHICH OTHER PROGRAMS CAN BE STARTED FROM Automation .
DATABASE P. 847	CHOOSE DATABASE FILE AND SETTINGS. ACTIVATING AND DEACTIVATING THE DATABASE.
SYSTEM P. 849	COMPANY AND USER NAME. LICENSE KEY SETTINGS. FILE TYPE REGISTRATION. SPECIFICATION FOR INTERVAL BETWEEN SAVING AUTO RECOVER INFORMATION. CHOOSING CHARACTER SETS FOR OTHER LANGUAGES, AND CHOOSE LANGUAGE.
TEXT/SYMBOL DEFAULTS P. 852	SPECIFICATION OF THE STANDARD APPEARANCE OF VARIOUS TYPES OF TEXTS IN THE PROGRAM. SPECIFICATION OF WHICH DRAWING HEADERS CAN BE USED, AS WELL AS WHICH SIGNAL SYMBOLS AND SIGNAL SYMBOL NAMES THAT WILL BE SUGGESTED IN THE DIALOG BOX SIGNALS . CREATION OF DATAFIELDS FOR DATA SYMBOLS. DEFAULT CABLE DIRECTIONS.
PAGE REMARKS P. 854	HERE YOU CAN TYPE IN REMARKS FOR THE ACTIVE PROJECT PAGE.

PROJECT DATA

The screenshot shows the 'Settings' dialog box with the 'Project data' tab selected. The 'Project title' is 'Automation project'. The 'Remarks' field is empty. Under 'Passwords', there are buttons for 'At Open...' and 'At Save...'. The 'Customer name' is 'PC|SCHEMATIC A/S', 'Address' is 'Bygaden 7', 'Zip code' is 'DK4040', 'City' is 'Jyllinge', 'Phone' is '+45 4678 8244', and 'Project name' is 'demo'. There is a 'Values' button next to the project title. On the right, there are 'Logo1' and 'Logo2' buttons, and a preview of the PC|SCHEMATIC logo. Below the customer information, there is a checkbox for 'Automatic renumbering of pages' (unchecked), a 'Relay Reference Type' section with 'Reference cross' selected and 'Contact mirror' unselected, a 'Revision' button, and a 'Reference designations' button. A 'Mounting correct drawing' checkbox is checked. A 'Bend' dropdown is set to '2,5 mm'. At the bottom, there is a checked checkbox for 'Show this dialog when creating a new project' and 'Ok' and 'Cancel' buttons.

On the tab **Project data** you specify general settings for the project. You can add and fill out the Project data datafields, specify passwords for the active project, choose a project logo, choose reference designations, set up mounting correct drawing, start/stop a revision control and choose the reference type for relays.



Clicking on the **Project data** button brings you directly into this dialog box.

Passwords

For further details about passwords on projects, please refer to “Passwords on Projects” on page 805.

Project data and Project title

In **Project data** you type in information regarding the entire project, which you can read more about in “Project and Page data” on page 114.

By right-clicking on a datafield name (the grey fields to the left), you can delete, add or change the datafield names in the menu. Read more about this in “Adding New datafields to Project data and Page data” on page 692.

The field names you specify here, can be inserted into your project, when you choose **Insert => Insert Datafield** and click on **Project data**. The program then collects the information you have typed in here, and place it in the project. If you need to change an information, you therefore only have to do it here once and for all. Read more about datafields in “Datafields” on page 110.



Clicking on the **Save as default** button, saves the datafields you have specified as default datafields of the type **Project data**. When you start up a new project, these datafields will be suggested.

Export and import of Project data



When you click the **Export projekt data to file** button, you can save the project data as an Excel file or as a comma separated file (*.csv). If you click the **Import projekt data from file** button, you can import project data from an Excel file or a comma separated file.

Remarks in projects

In the **Remarks** field, you can type in remarks for the entire project. There is no limitation for the size of the remarks in the **Remarks** field.

When you have typed in a remark, you can right-click in the **Remarks** field. This gives you the opportunity to print out the remark, or to save it as an independent text file.

Logo

— See “Inserting logos in projects” on page 527.

Automatic renumbering of pages

When you select **Automatic renumbering of pages**, the program automatically renumbers the project pages to consecutive page numbering. If **Automatic renumbering of pages** is selected, and you insert a new page between project page 6 and project page 7, the new page will then get page number 7, while the previous page 7 is renumbered to page 8. If subsequent project pages exist, these will also be incremented by one — page 8 becomes page 9 etc.

The setting only applies for the active project, and it will by default *not* be activated when you start a new project.



BE VERY CAREFUL USING THIS FUNCTION!

IT IS RECOMMENDED THAT YOU DO NOT SELECT **AUTOMATIC RENUMBERING OF PAGES**, UNLESS YOU ARE ABSOLUTELY SURE ABOUT WHAT YOU ARE DOING, AND HAVE A CLEAR UNDERSTANDING OF ALL OF THE POSSIBLE CONSEQUENCES.

Relay reference type

Here you specify whether you wish to use reference crosses or contact mirrors for relay references. — See “Contact mirrors and Reference crosses” on page 374.

Revision

— See “Revision control” on page 808.

Reference designations for the project

When you click on **Reference designations**, you can specify the reference designations you want to use in the project. Please refer to “Reference designations” on page 335 for further details.

Mounting correct drawing

When you select **Mounting correct drawing**, the program supports you in creating mounting correct drawings. Please refer to “Mounting correct drawing” on page 95 for further details.

Do not display Project data when starting a new project

If you do not wish to see the tab **Project data** when creating new projects, choose **Settings => Project data**, and deselect **Show this dialog when creating a new project**. This check box is displayed at the bottom of the tab.

PAGE DATA

In this dialog box you specify various kinds of settings regarding the project page you are working on right now.



Clicking the **Page data** button brings you directly into this dialog box.

Include page in Table of Contents

If you select **Include this page in Table of Contents**, the page is included in the table of contents.

Page data and Page title

In these text fields you type in information regarding the page you are working on in the project right now.

Read more about this in “Project and Page data” on page 114.

The field names specified here, are the datafield names you can insert into your project, when you choose **Insert => Insert Datafield**, and select **Page data**. When you insert a datafield, the program collects the information you have typed in here, and insert it where you have placed the datafield. Read more about datafields in “Datafields” on page 110.

When you right-click on a datafield *name* (the grey fields to the left), you can delete, add or change the datafield names in the dialog box. Read more about this in “Adding New datafields to Project data and Page data” on page 692.



Clicking on the **Save as default** button, saves the datafields you have specified as default datafields of the type **Page data**. When you start up a new project, these datafields will be suggested.



When you click the **Export page data to file** button, you can save the page data in an Excel file or a comma separated file. When you click the **Import page data from file** button, you can load page data from an Excel file or a comma separated file.

Reference designations for the page

In **Automation** you can attach reference designations to all symbols placed on the same page.

If reference designations are already attached to the page, they are displayed, as you can see on the figure. When you wish to specify new reference designations for the page, click on **Select**.

— Please refer to “Reference designations” on page 335 for further details.

Reference designations is only visible on this tab if reference designations have been chosen for the project on the **Project data** tab. — See “Reference designations for the project” on page 833.

Drawing headers

In this part of the dialog box you specify which drawing header you want to place on the page. This is described in “Inserting a drawing header on a page in a project” on page 532.

References

When you click on the **References** button, you enter the dialog box **Reference system setup**, where you specify the settings for the reference system/current path numbers on the page. — Read more about this in “Current path numbers / Reference System” on page 379.

Article data

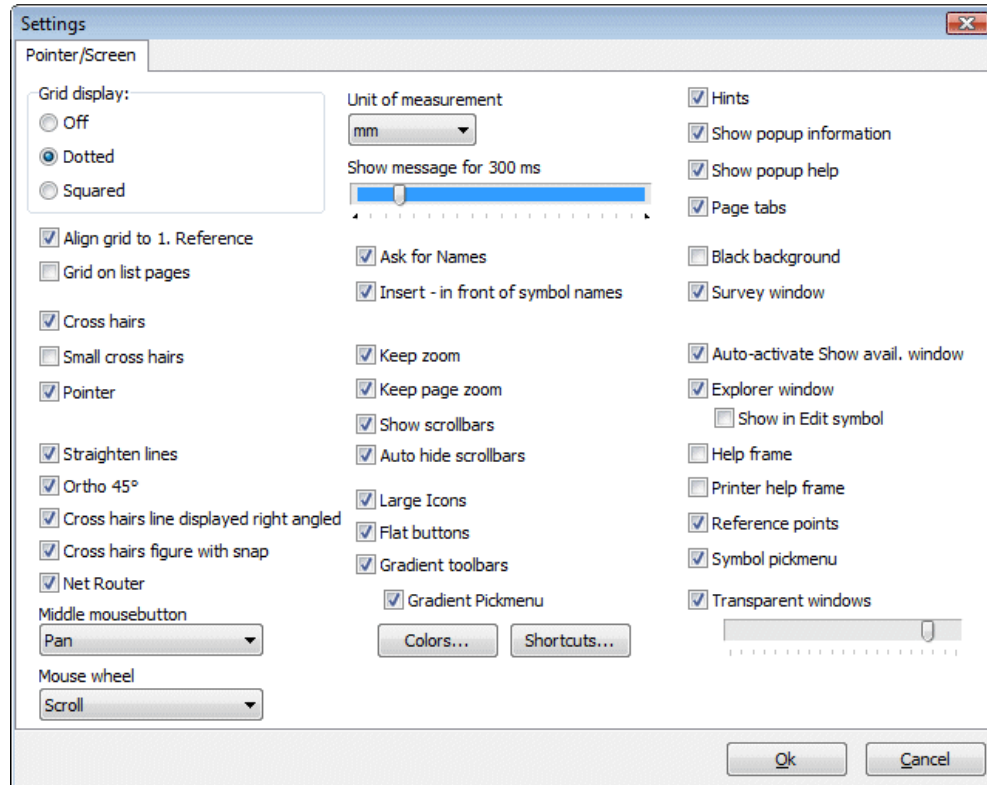
Clicking on the **Article data** button brings you into the dialog box **Page article data**.

Here you type in article data for the page if you need to.

When you right-click on the page itself, and then choose **Page article data**, you also enter this dialog box.

POINTER/SCREEN

This tab consists of a number of check boxes, in which you specify how you want the screen and pointer settings.



Change system colors

If you want to change the system colors, you click on **Colors**. Please refer to “Colors” on page 792 for further details.

Change system Short-cut keys

If you want to change the system short-cut keys, you click on **Short cuts**. Please refer to “Short-cut Keys” on page 788 for further details.

Grid display: Here you specify the appearance of the grid on the screen. Please refer to “Grid” on page 47 for further details.

Align grid to 1. Reference: If you want your grid aligned to the first reference number on the page, you select **Align grid to 1. Reference**. This goes for both vertical and horizontal reference systems.

You then have to adapt the size of the grid/snap in order to make it fit the interval between the current path numbers.

Grid on list pages: Select this option if you want grid on list pages.

Cross hairs / Small cross hairs: Here you specify whether you wish to use a cross hairs, as well as the appearance of the cross hairs. Read more about *cross hairs* in “Dictionary” on page 873.

Pointer: Specifies whether a mouse pointer is displayed on the screen. Is automatically selected if **Cross hairs** is deselected.

Straighten lines: Here you specify whether the connecting lines should be straightened when you move a symbol. Is enabled by default.



Ortho 45: When you draw **Lines**, and have chosen **Straight lines**, you are only allowed to draw horizontal and vertical lines. However, if you select this function, you are also allowed to draw diagonal lines.



Cross hairs line displayed right angled: When you are drawing **Straight lines**, a rubber band always appear, radiating from the last place you clicked towards the cross hairs.

If you have *selected* this function, you will always see the line as it will be drawn if you click in this very moment.

If you have *deselected* this function, you will see a line going from where you clicked last to the exact position of the cross hairs.

Depending on which *snap* you are using, these lines will not always be the same.

Cross hairs figure with snap: When this function is *selected*, the objects in the cross hairs follow the actual snap. This way you will continuously see where the object will be placed when you click.

If the function is *deselected*, the symbols will only follow the exact position cross hairs — but will still be placed according to the actual snap if you click. — Read more about *snap* in “Snap” on page 47.

Net router: When you move mechanical symbols, the program can display which other mechanical symbols the symbol is connected to — according to the information on the electrical pages. Read more about this in “Displaying connections for mechanical symbols” on page 634.

Middle mouse button: If you use a mouse with a middle mouse button, you here can specify how this middle button functions in **Automation**. You can choose either **Zoom** or **Pan**.

Mouse wheel: If you use a mouse with a mouse wheel between the two mouse buttons, you can here specify how this mouse wheel functions in **Automation**. You can choose either **Scroll** or **Zoom**. Read more about this in “Zoom and Scroll for mouse with a wheel” on page 43.

Unit of measurement: Here you specify according to which unit of measurement **Automation** interprets the object positions you type in. If the unit of measurement is set to **mm** and you type in *15*, this will be interpreted as 15mm.

However, you can still type in another unit of measurement together with the number — typing in *15 cm* will still be interpreted as 15cm.

Show message in (ms): Specify how long time the hints are displayed in the program.

Ask for Names: If this function is *selected*, you will enter the **Component data** dialog box each time you place a symbol in your project. If the function is *deselected*, you have to right-click on the symbols later on, choose **Component data**, and then fill out the symbol name and other information.

Insert - in front of symbol names: When there is no minus in front of the symbol name in the symbol definition, a minus will be inserted in front of the symbol name, when you place the symbol.

To insert a minus in front of already placed symbols choose **Functions => Special Functions => Prefix all symbol names with -**.

Keep zoom: Imagine you have zoomed in on an area — for instance the upper-right corner — of a project page, and change to another page in the project. If **Keep zoom** is *selected*, you will then also see the upper-right corner of the page you change to. If the function is *deselected*, you will see the entire new page.

Keep page zoom: If **Keep zoom** is selected, you also get the option to select **Keep page zoom**. If you select this function, the program remembers which zoom you are using on each individual page. When you close a project, the applied zoom settings are not memorized.

Show scrollbars: When you have zoomed in on an area, you can drag the scrollbars to move the zoom — see “The Scrollbars” on page 43. When you deselect Show scrollbars you get a larger work area on the screen. — See **Auto hide scrollbars** below.

Auto hide scrollbars: Is only displayed when **Show scrollbars** is selected. If you activate **Auto hide scrollbars**, the scrollbars are only displayed when you point the mouse pointer to the far right or to the bottom of the screen. This way you do maintain the full work area and are still able to use the scrollbars.

Flat buttons: If you want the program to apply the flat Windows 98 buttons, you select **Flat buttons**. If **Flat buttons** is not selected, the buttons will be displayed in Windows 95 style.

Gradient toolbars: If you wish to apply the gradient look, as known from e.g. the Microsoft Office programs, select **Gradient toolbars**. See how to set the colors in “Colors” on page 792.

— Please note that dialog boxes always have *gradient look*.

Gradient Pickmenu: Is only displayed when you have selected **Gradient toolbars**. Select **Gradient Pickmenu** if you also want *gradient look* in pickmenus.

Hints: If you let the mouse pointer rest for a moment, on for instance a program button, you will see the name of the button displayed in a small text field. Or, if you let the pointer rest on a symbol in the symbol pick menu, the symbol title will be displayed. This is called *hints*, and here is the place you specify whether you want hints when working with the program.

Show popup information: When selected and you let the mouse cursor rest over a symbol, a window is displayed containing information about the symbol. The window disappears when you move the mouse again.

Show popup help: When you have started a function, which requires more than one action, and you have not moved the mouse for a while, a window is displayed, telling what the program expects you to do.

Page tabs: On the bottom of each project page, you see some small tabs, displaying which project page you are working on. Clicking on these tabs enables you to change back and forth between the project pages quickly. This is the place to select whether you want tabs on your pages or not.

If the function is disabled, the drawing area on the screen will become a little larger.

Auto-activate show avail. window: Select this option to have the **Show available window** activated automatically. Read more about the **Show available window** in “The Show available window” on page 234.

Black background: Selecting this function will make the project pages turn black, while lines, symbols and so forth will become white.

Survey window: Here you enable and disable the **Survey window**. Short-cut <F12>.

Explorer window: Here you enable and disable the **Explorer** window. Please refer to “The Explorer window: The Projects tab” on page 301 for further details. In **Show in Edit symbol** you specify whether to display the Explorer window when editing symbols.

Help frame: Clicking here makes a standard margin appear on the project pages. The margin is displayed by a red dash line. The help frame is not included when you print out project pages.

Printer help frame: Clicking here will make the printable area of your printer visible. The printable area is displayed by a grey dash line, and is not displayed when you print the project pages.

Reference points: When the function is selected, the reference points of the objects are displayed on the screen. Read more about reference points in “Dictionary” on page 873. The reference points are not visible when you print out the project.

Pickmenu: Here you turn the pickmenu on and off. Read more about the symbol pick menu in “Fetching symbols from the Pickmenu” on page 140.

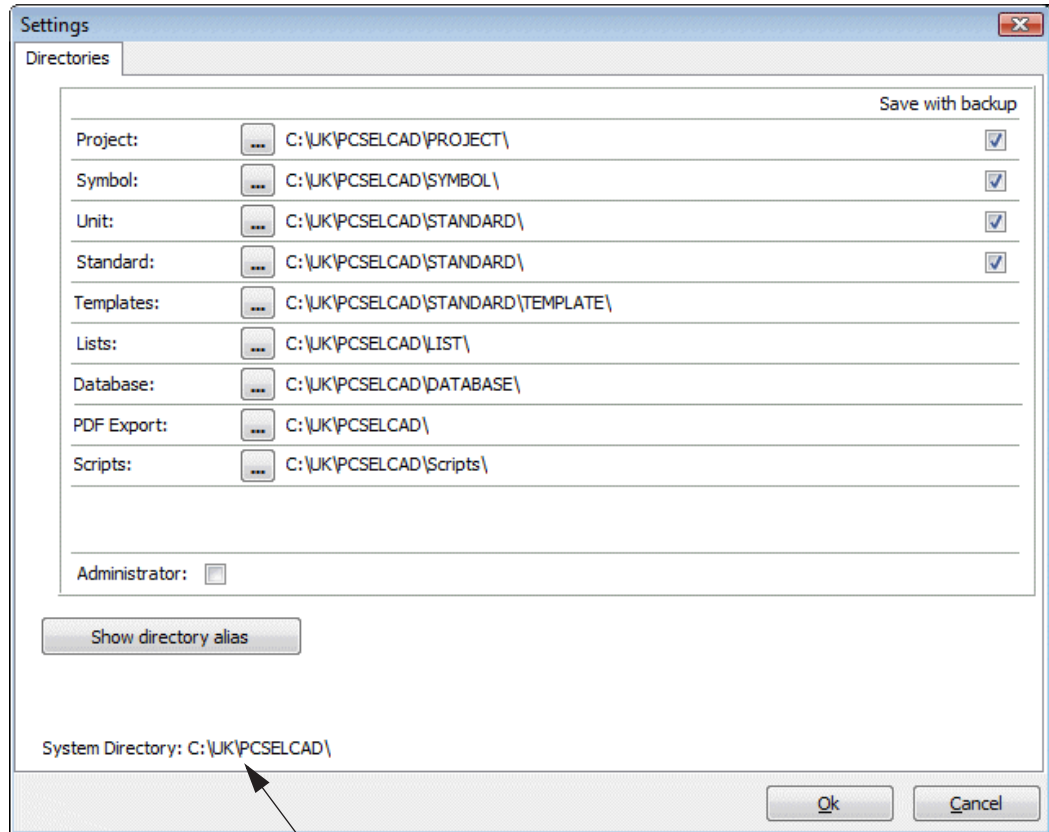
Transparent windows: Some windows can remain on screen while you are working on the drawing area in the project. These windows can be set to become transparent by selecting **Transparent windows**. You can set the degree of transparency by moving the slider below **Transparent windows**. This makes it possible to get an idea of what is behind the dialog box on the drawing area.

When the window / dialog box is activated — e.g. by clicking on it — the window becomes fully visible again. When you click on the drawing area of the screen, the window becomes transparent again.

The best way to set the degree of transparency, is to open e.g. the **Object Lister** (press <F7>), and select **Symbols**, so that the **Object Lister** contains data. Then select **Settings => Pointer/Screen**, and drag the slider below **Transparent windows** to a position, where you get the desired transparency in the **Object Lister**.

The following windows will also become transparent, when **Transparent windows** is selected: The **Show available** window, the **Insert component list** window, the **Find and Replace** window, and the result window from **Design check**.

DIRECTORIES



Here the System catalog is displayed, as well as the Workstation catalog

Directories

Here you specify in which folder/directory you want the program automatically to suggest you to save the various types of program files. If you want to save a *project*, and have chosen the setting above, the program will automatically suggest you to save it in the folder **Project**.

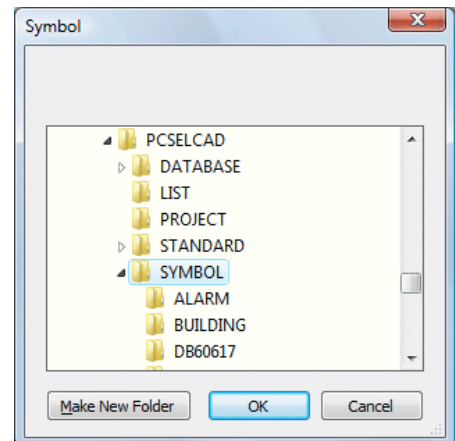


When you click on the **Browse** button in front of a directory, you get a little Explorer window, where you can locate the folder you now wish to apply as directory:

Click on the desired folder, and click **OK**.

Displaying System catalog and Workstation catalog

At the bottom of the tab **Directories** the system catalog is displayed, as well as the workstation catalog on net installations.



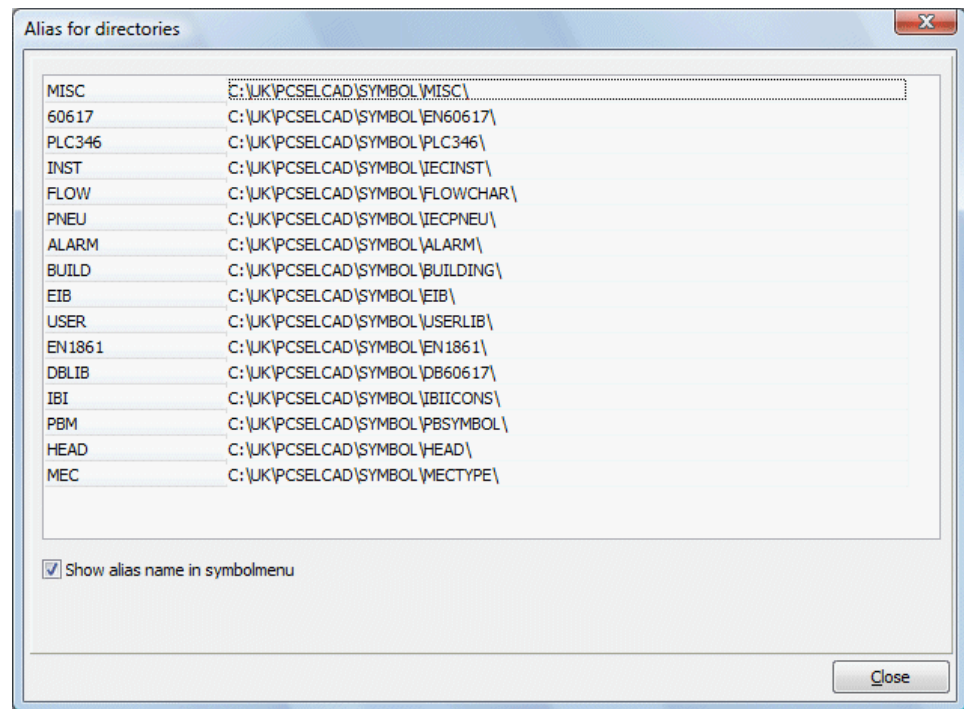
Save with backup

In the right-hand side of the dialog box you specify for which kind of files you want backups to be made. These backups are made when you save a file, and contains the previous version of the file.

The backups gets the same names as the files, but the extension of the files are changed to the character ~ followed by the first two characters in the extension of the original. Thus the backup of the project **demo.pro** will be called **demo.~pr**, while the backup of the symbol **symbol.sym** will be called **symbol.~sy**.

Show directory alias

When you have clicked on **Symbols** in the left-hand side of the dialog box, you get the option of clicking on **Show directory alias**.



Here you get the option to assign an *alias* to each symbol folder/directory.

This makes it easy for you to move the symbol directories, if this should become necessary.

The Alias concept

As displayed above, the directory **C:\PCSELCAD\SYMBOL\MISC** has the alias **MISC**. If it at some point is decided to move this directory to for instance **I:\SYMBOLSMISC**, you only have to correct the path here. In this case, the program will automatically find all the project symbols — which used to be located in the old directory — in the new directory.

You will not need to type in the alias **MISC** anywhere in **Automation**; it is merely a name you have given the program, so that it internally can distinguish between the various aliases. But off course it is a good idea to give them appropriate names, so that you yourself can distinguish between them here in the **Alias for directories** dialog box.

Specify which directories you use the most

When you fetch a symbol from the database, or load a project with no specified paths to the symbols, the program will search the symbol folders in the order you have specified here, starting from the top. The program then finds the first symbol

with the given symbol name. Please note, that the order of the directories therefore is important.

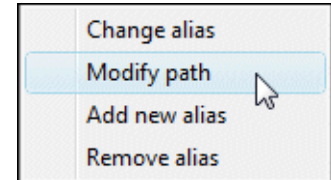
To save time, it is therefore a good idea to place the folders, you use the most, on the top of the list. You can move the aliases by *clicking* and *dragging*.

If for instance you want **User** to be on top, then click on **User**, keep the mouse button pressed down, *drag* the alias to the top of the list, and let go of the mouse button. **User** is now placed on top of the list.

Correcting directories or alias

If you want to correct an alias name or path, you first have to make sure that you are working in an empty project, and that no other projects are open.

Then right-click on the place in the list where you want to make changes. — If you are not working in an empty project, or other projects are open, you will get a warning, and will not be allowed to make any changes.



If you want to change an alias name, you then choose **Change alias**, and if you want to remove an alias, you choose **Remove alias**.

If you choose **Modify path**, you enter a dialog box, where you click your way to the desired directory, and click **OK**.

When you want to add a new alias to the list, you choose **Add new alias**. Then you type in the name of the alias, and click your way to the desired directory.

Click on **Close** when you have completed the corrections.

Example

If you are working inside an empty project — having no other projects open — and want to add a new alias, which you want to be placed as number three on the alias list, then start by choosing **Settings => Directories**.

Click on **Symbols** in the left-hand side of the dialog box, and then click on **Show directory alias**. Right-click on the lower part of the list, and choose **Add new alias**.

Now type in the alias name, click **OK**, click your way to an appropriate directory, and click **OK** again. Move the new alias to the third position on the list by *clicking* and *dragging*.

Click **OK**.

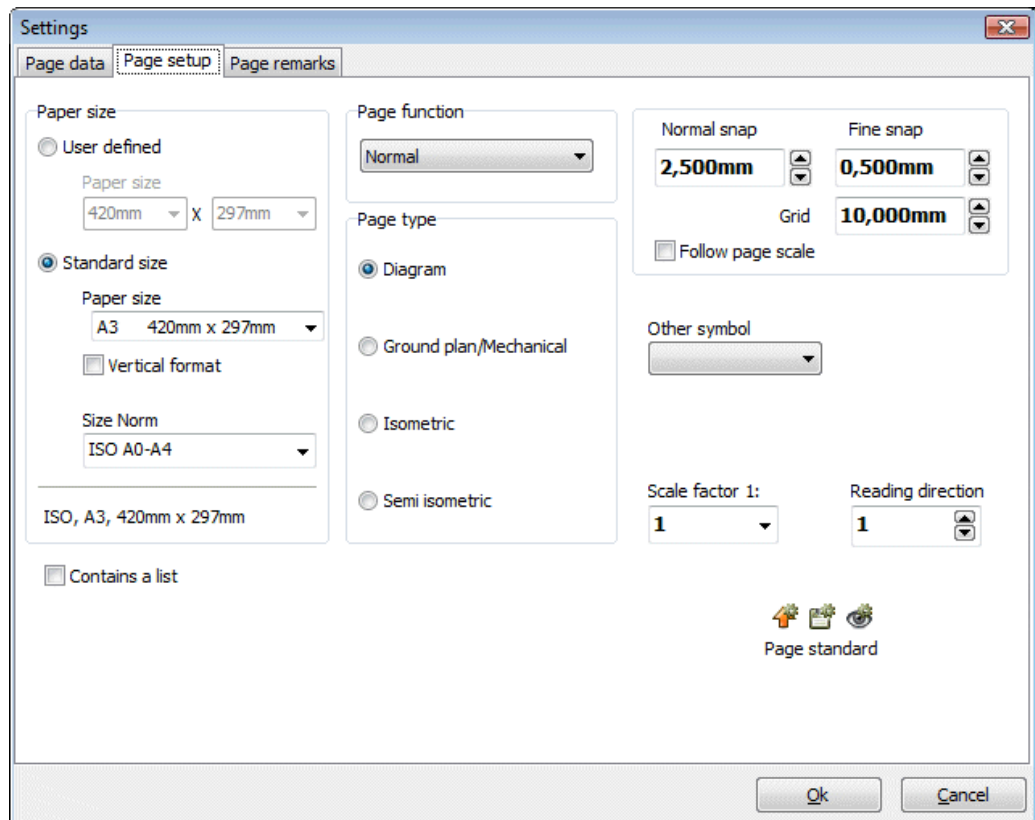
Alias when installing databases

When you install a database on **Automation**, the program automatically creates an alias to the library containing the symbols of the database.

Administrator control

If **Administrator** is selected, various system settings can be controlled by the system administrator. This function should only be activated after consulting the system administrator.

PAGE SETUP



On the tab **Page setup** you have the options mentioned below for specifying the page setup. — Page settings can also be copied between pages, as described in “Page setup - and transferring this to other pages” on page 506.

Paper size

In the upper-left corner of the tab you select the **Paper size** of the active page, as described in “Paper size” on page 509.

Page type and Page function

See how to select the **Page type** in “Page type” on page 508.

The **Page function** is also displayed here, but can not be changed. This has been specified when the page was created, and can never be changed. Please refer to “Creating a new page” on page 495 for further details.

Please note that a **Page function** can be set to **Normal**, as described in “Changing the page function” on page 498.

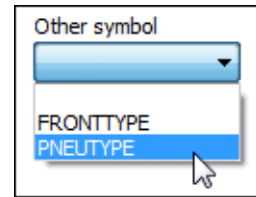
Grid and Snap

Grid and **Snap** is also specified here; read more about this in “Screen / Image Functions” on page 42.

Other symbols

Under **Other symbol**, you can choose that symbols shall be fetched from another datafield in the database, when you are working on this specific page.

To select another datafield here, you must first specify which datafields in the database, that can contain alternative symbols. Read more about this in “Database settings” on page 608, “Other symbols” on page 612 in particular.



Scale factor and Reading direction

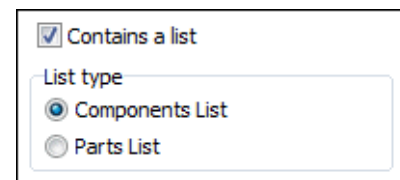
— Read more about **Scale factor** in “Setting the scale of the page” on page 483.

— **Reading direction** is described in “Specify reading direction” on page 130.

Contains a list

If you select **Contains a list**, diagram pages are allowed to contain either a **Components List** or a **Parts List** for the page.

— Read more about this in “Diagram pages - containing a Parts or Components list” on page 762.



Default settings



If you want to see the current standard settings, you click on the **Show defaults** button. To load the default settings, you click on the **Load defaults** button.



If you, on the other hand, want to save the specified settings as the future default settings, you click on the **Save as default** button.

Click **OK** when the settings are satisfactory.

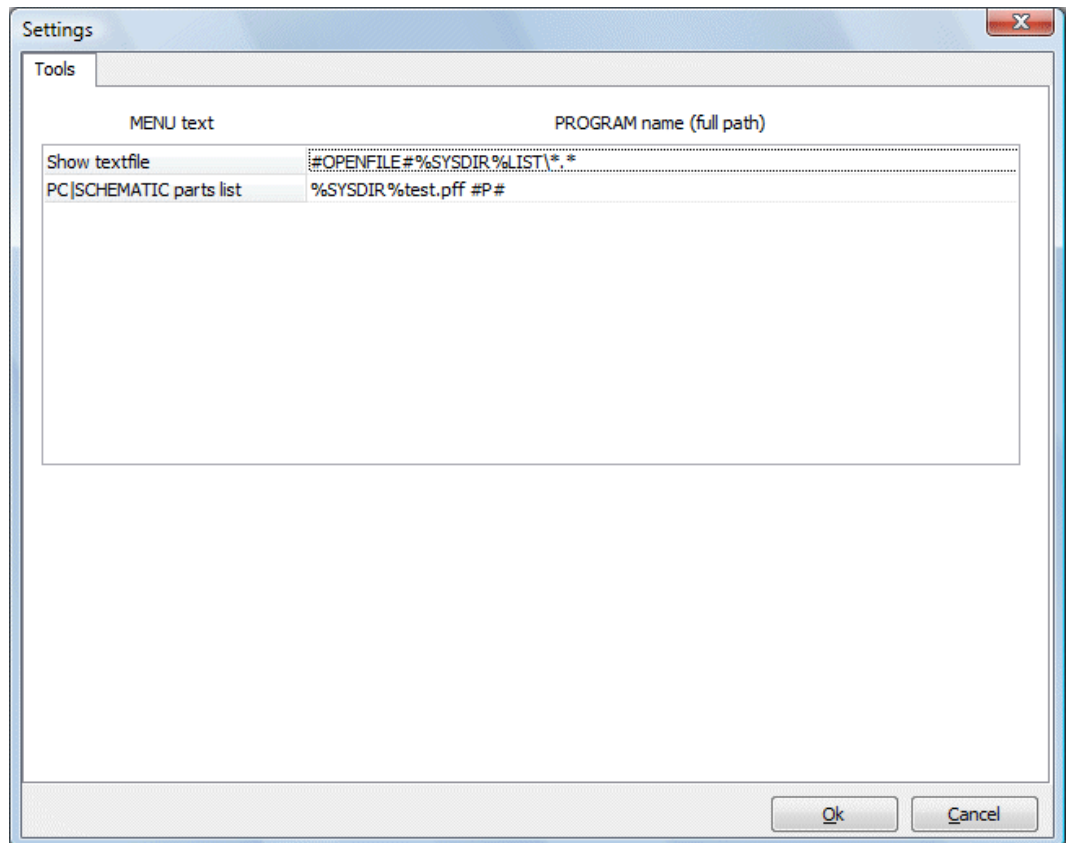
Standard settings for List pages and Chapter tabs



If you have selected a list page and choose **Settings => Page setup**, and choose **Save as default**, the page size of the selected list page becomes the default page size for list pages. This works correspondingly for chapter tabs.

— Please also see “Standard page sizes for List and Chapter tabs” on page 509.

TOOLS



In the program tool bar you can choose a menu called **Tools**. From this menu you can change back and forth between **Automation** and other programs.

Here — in **Settings => Tools** — you specify which other programs you can start directly from the **Tools** menu.

You do this by typing in the name you want to see in the **Tools** menu in the **MENU name** column, while you type in the full path and program name in the **PROGRAM name** column on the same line.

You can here use the following parameters:

%SYSDIR% here specifies the directory in which the program is installed.

%SYSDIR8% works as %SYSDIR%, but is used on net systems, which only handles eight characters in file names — or for DOS or Windows 3.11 programs.

%HOMEDIR% is the home directory. This is the same as %SYSDIR% if the program is not installed on a network server.

%HOMEDIR8% works as %HOMEDIR%, but is used on net systems, which only handles eight characters in file names — or for DOS or Windows 3.11 programs.

%DATABASE% is the name of the database attached to the program.

%NET% is used when an application needs to know whether it is running on a net installation. This is a special parameter which only works for **Automation** applications.

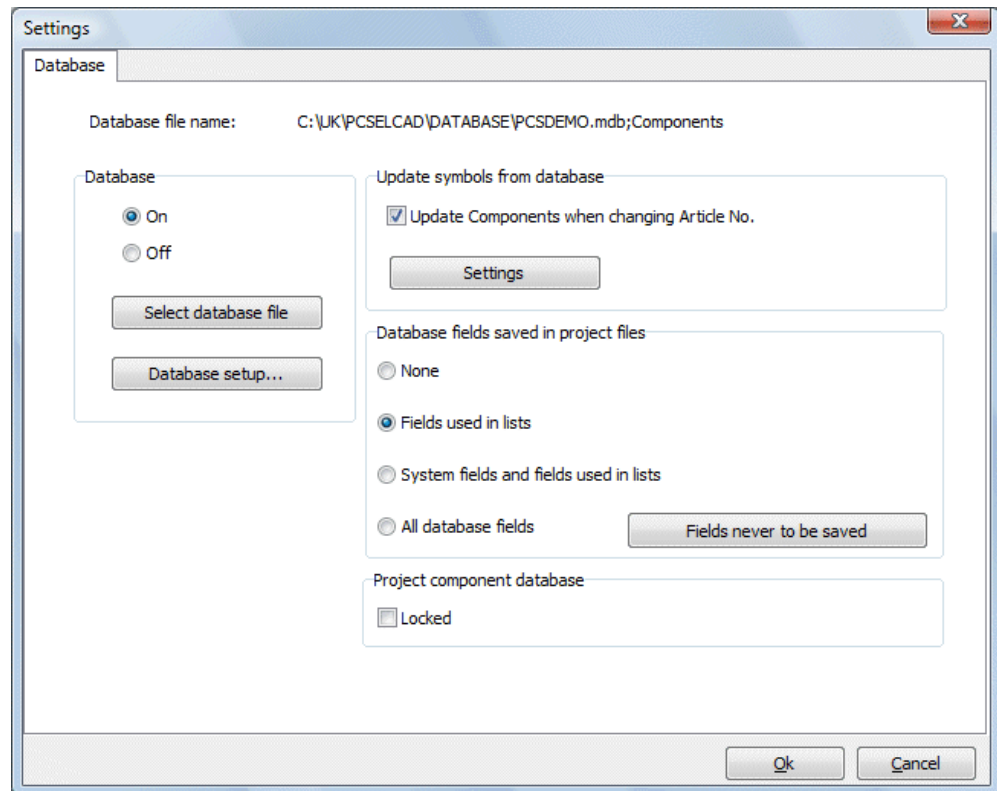
Inserting a button in the program tool bar

To make it easier to change to another program, it is also possible to insert your own buttons on the program tool bar.

If you want to add a button with the text **DOS** on it, in order to be able to change to DOS quickly, you just add **#B(DOS)#** after the program name for DOS.

If you have got a bitmap file which you want displayed on the button, you type in the name of the file (including the full path) instead of **DOS** in the example above: **#B(bitmap file)#**. The size of the bitmap file should be 20x20 pixels.

DATABASE



Here you specify whether to work in **Automation** with- or without a connected database, by clicking on **On** or **Off** respectively.

Select database file

By clicking on **Select database file** you enter a dialog box, from which you specify which database you want to work with — see “Choosing a database” on page 623.

Database setup

By clicking on **Database setup** you enter a dialog box, where you specify the database setup, as it will appear when working in **Automation**. Read more about this in “Database settings” on page 608.

You can read more about the database program in the **PC|SCHEMATIC Database** manual.

Database fields saved in project files

Settings for project component database. — See “Project component database” on page 584.

Locking the project component database

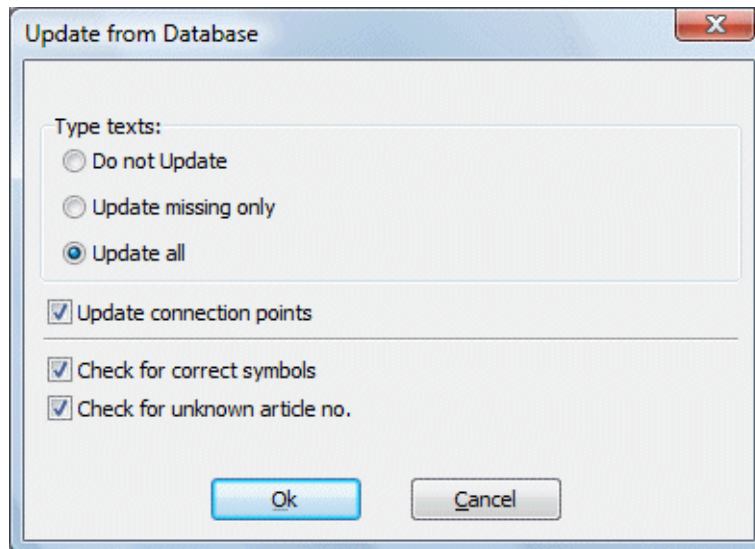
— See “Locking the project component database for the active project” on page 587.

UPDATE SYMBOLS FROM DATABASE

When selecting **Update Components when changing Article No.** you can specify whether/how **Automation** updates the other symbols for the component, when the article number is changed for one of the symbols for a component.

This update is made when the article number is changed for the symbol in the **Component data** dialog box, having **Change all symbols for the component** selected.

The function is activated by first selecting **Update Components when changing Article No.** When you subsequently click on **Settings**, you enter the dialog box **Update from Database**:



Here you have the following options:

OPTION	DESCRIPTION
DO NOT UPDATE	TYPE TEXTS FOR THE COMPONENT IS NOT UPDATED.
UPDATE MISSING ONLY	TYPE TEXTS ARE ONLY UPDATED FOR THOSE OF THE SYMBOLS FOR THE COMPONENT, WITH NO PREVIOUSLY SPECIFIED TYPE TEXT.
UPDATE ALL	ALL TYPE TEXTS FOR THE COMPONENT IS UPDATED ACCORDING TO THE DATABASE.
UPDATE CONNECTION POINTS	ALL CONNECTION NAMES FOR THE COMPONENTS ARE UPDATED ACCORDING TO THE SPECIFIED CONNECTION NAMES IN THE DATABASE.
CHECK FOR CORRECT SYMBOLS	IT IS CHECKED WHETHER THE SYMBOLS FOR THE COMPONENT HAVE THE CORRECT SYMBOL FILENAMES — ACCORDING TO THE DATABASE INFORMATION.
CHECK FOR UNKNOWN ARTICLE NO.	IT IS CHECKED WHETHER THE NEW ARTICLE NUMBER FOR THE COMPONENT EXISTS IN THE DATABASE.

SYSTEM

Company and User name

Here you type in your own name, as well as the name of your company. This information can automatically be inserted in your projects, when you choose **Insert => Insert Datafield**, and click on **System data**. The program then collects the information you have typed in here, and inserts it into the project. If you have to change this information later on, you only have to do it once right here. — Read more about datafields in “Datafields” on page 110.

Automatic recovery information

If you select **Select auto recover information every**, you will make **Automation** save automatic recovery information every 30 minutes. By clicking on the up and down-arrows you can change the contents of the **Interval in minutes** field. The recovery file is saved in the directory **Pcscad** using the same project name, but with the extension **.Spr**.

After an unauthorized closure, **Automation** will automatically start up from the automatic recovery file.



SAVE YOUR PROJECTS FREQUENTLY

PLEASE REMEMBER TO SAVE YOUR PROJECT OFTEN. THE AUTOMATIC RECOVERY FILE IS ONLY MEANT AS A MEANS OF RE-CREATING THE PROJECT FILE IF THE SYSTEM CRASHES — FOR INSTANCE BECAUSE OF A POWER SUPPLY FAILURE.

Compressed saving

— See “Compressed saving / Zipping projects” on page 825.

Load previous project

When you select **Load previous project at program start**, **Automation** automatically opens the last active project, the next time you start the program.

Undo levels

In the **Undo levels** field you can specify how many levels/actions it shall be possible to undo. You can here choose between 5 and 30 levels.

— Read more about the **Undo** function in “Undo” on page 60.

License key and Servers

The **License key** field is used for license key settings. Normally, you will not have to change the license key settings, because these are set when the program is installed. Read more about this in "License key" in the Installation & Information manual.

When a SuperPro license key is applied, the license key number is displayed under **Settings => System => License key**.

— Please see “Tanking licenses” on page 794 also.

Time-out

For the SuperPro NET key you can set a **Time-out** for the license to **None**, **10 min.**, **20 min.** or **30 min.**. If there has been no activity on the PC in the specified period of time, the license is released for other users. When you continue working with the program, **Automation** automatically searches for a free license, and continue working normally when a free license is found.

Choose language

Under **Choose language** you can choose in which language to use for the program texts in **Automation**.

It is possible to choose between English, Czech, Polish, Russian, Chinese, Dutch, Hungarian and Danish.

If these options are not available in your installation, you can have them sent to you by contacting PC|SCHEMATIC A/S.

When changing language the program must be restarted.

Select set of characters

When you want to specify which set of characters you want to use in the project, the options are **Standard** (normally used), **Latin2**, **Russian**, **Baltic** and **Polish**.

The codes for these characters (<Alt>+0xxx) follows the common Windows character sets.

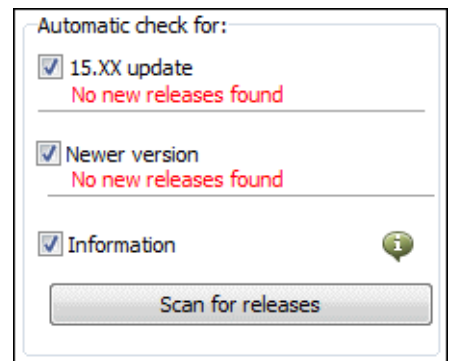
The information of the chosen character set is included in each project file separately. More character sets will be created on request.

Automatic check for Updates, New versions and Information

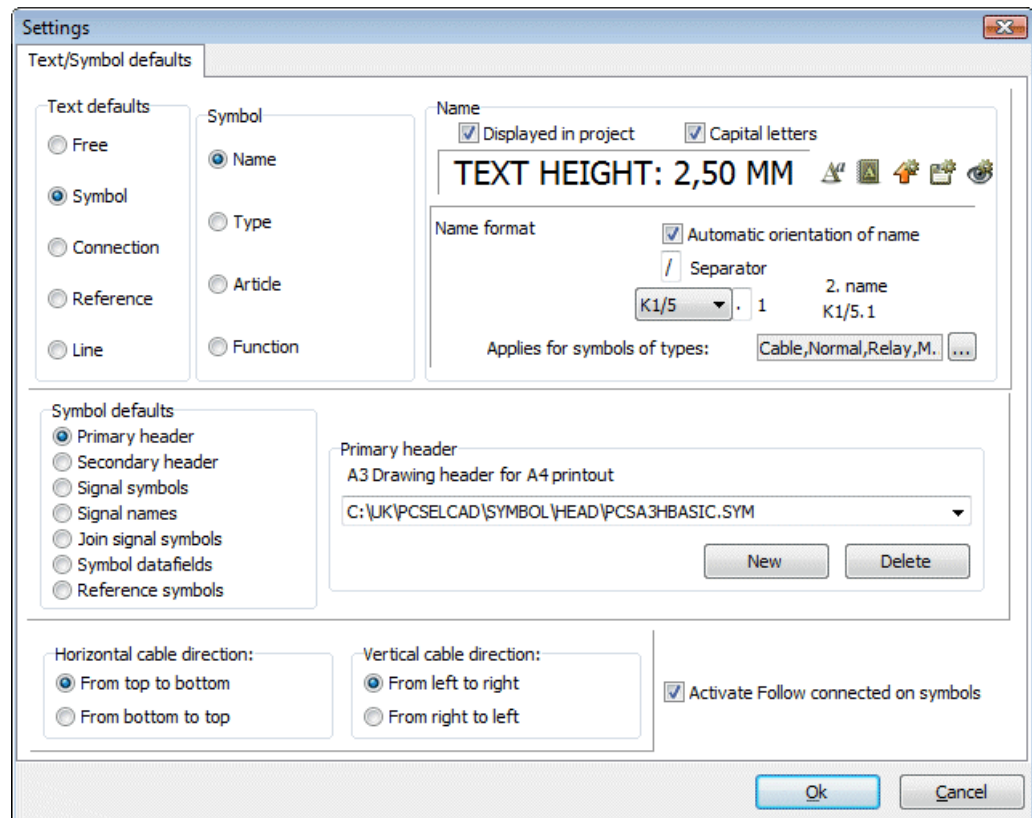
In the lower-right corner of the dialog box, you can select **15.XX update**, **Newer version** and **Information**. This makes it possible for you to receive information — via the internet — when updates or new versions are released, or to get important information about the program automatically.

— Read more about this in “Update and Automatic information” on page 784.

You can also check for new updates by choosing **Help => Check for updates** from the menu line at the top of the screen.



TEXT/SYMBOL DEFAULTS



The dialog box is divided in two parts: **Text defaults** on the upper part, and **Symbol defaults** on the lower part.

Text defaults

— Read more about this part of the dialog box in “Texts displayed in the project (general setting)” on page 127.

— If you want to change the **Name format** of the symbol names in the project, please refer to “Naming symbols due to page and current path number” on page 382.

— See also “Default text properties for free texts” on page 106, “Appearance of references” on page 372, “Changing the reference crosses” on page 381 and “Specifying default text settings for Edit symbol mode” on page 716.

Symbol defaults

The program contains some lists of which symbols can be used as for instance drawing headers or signal symbols. In this part of the dialog box you specify which symbols shall be included on these lists. (Of course you can not place a symbol on a list, before the symbol is created. Read more about creating symbols in “Creating Symbols” on page 696).

Inserting symbols in lists

To place a symbol on a list, you first specify — on the left-hand side — which type of list you want to place the symbol on. Then click on **Add**, which makes you enter the **Symbol menu**. Here you click on the symbol you want to add to the list, and click **OK**. — Read more about adding a symbol to a list in “Adding drawing headers

to the List Of Drawing Headers” on page 750.

Removing symbols from lists

Also, you can remove symbols from the lists by clicking on the down-arrow in the **Drawing headers** field, then click on the symbol you wish to remove, click on the **Remove** button, and answer **Yes** to remove the symbol from the list. — You can read more about removing a symbol from a list in “Removing drawing headers from List of Drawing Headers” on page 750.

— Read more about drawing headers in “Creating Drawing Headers” on page 742.

— Read more about signal symbols in “Creating Signal Symbols” on page 738.

— Read more about join signal symbols in “Creating Join symbols” on page 741.

— Read more about symbol datafields in “Creating symbol datafields” on page 733.

— Read more about reference symbols in “Changing the reference crosses” on page 381.

List of signal names



There also exists a list of signal names. When you place a signal symbol on a project drawing, and click on the **Used** button in the **Signals** dialog box, you will get a list of signal names to choose from. By clicking on **Signal names** in the left-hand side of the dialog box, you can specify which signal names that will be suggested in this list. Here you can add or remove signal names, as described for symbols above. — Read more about the dialog box **Signals** in “Signal symbols” on page 203, and about adding signal names to the list in “Frequently used signal symbol Names” on page 741.

Cable direction

At the bottom of the tab you can specify the default cable direction for horizontal and vertical cables.

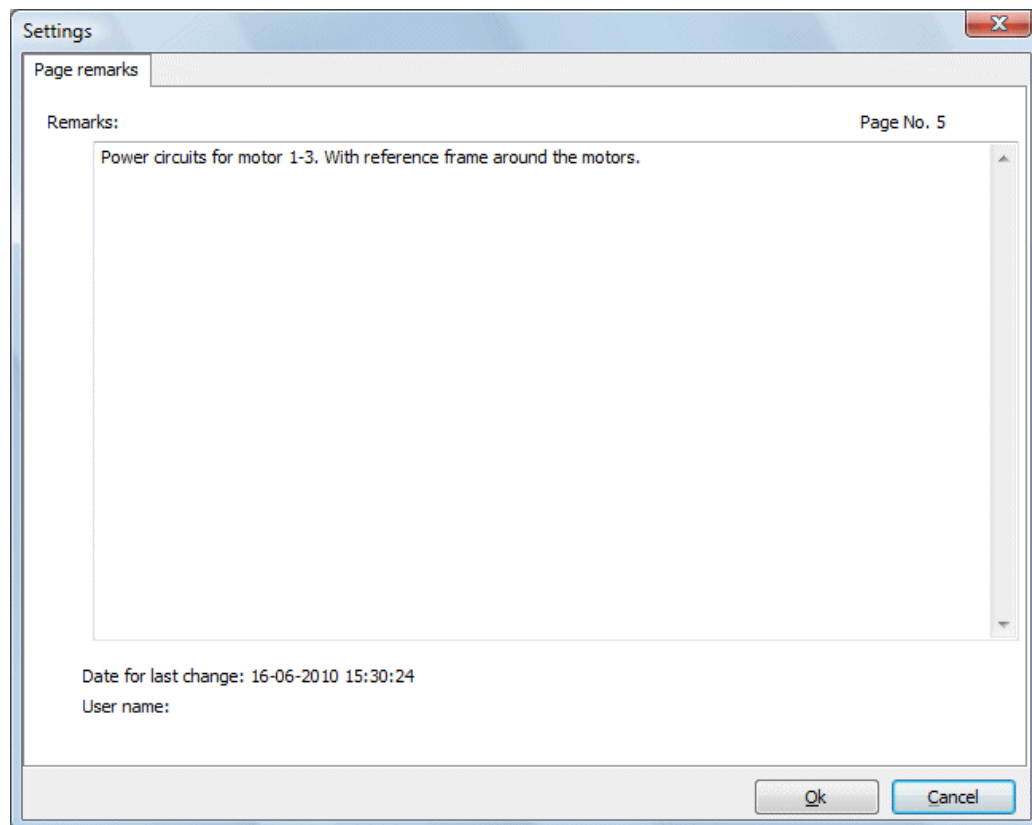
Activate Follow connected on symbols

Follow connected can be activated/deactivated as a general setting in the active project, by clicking on **Activate Follow connected on symbols**.

This setting is activated by default.

— Read more about Follow connected in “Follow connected lines” on page 90.

PAGE REMARKS



On the tab **Page remarks**, you can type in remarks about the selected page in the project.

It is possible to print all page remarks in the project as a list. For this purpose you can use the list for printing page remarks, which is installed with the program. — See the survey of lists that comes with the program in “Predefined lists” on page 763.

DATAFIELDS

In the following it is displayed which datafields can be inserted from the dialog box **Datafield**. You can enter this dialog box by choosing **Insert => Insert datafield**.

The datafields are grouped in the same order in which they appear in the dialog box.

Datafields demarcated by an asterisk (*), are user defined datafields.

DATAFIELDS	
— SYSTEM DATA	P. 855
— PROJECT DATA	P. 855
— PAGE DATA	P. 857
— SYMBOL.....	P. 858
— TABLE OF CONTENTS	P. 858
— PARTS/COMPONENTS LIST.....	P. 859
— TERMINALS LIST.....	P. 861
— CABLE LIST	P. 863
— PLC LIST DATA	P. 864

SYSTEM DATA DATAFIELDS

DATAFIELD	DESCRIPTION
USER NAME	USER NAME FROM SETTINGS .
DATE	DATE
TIME	TIME
COMPANY NAME	COMPANY NAME FROM SETTINGS .
PROGRAM TYPE	PROGRAM TYPE
PROGRAM PATH	THE FULL PATH TO WHERE THE PROGRAM IS INSTALLED.

PROJECT DATA DATAFIELDS

DATAFIELD	DESCRIPTION
PROJECT NUMBER (*)	PROJECT
REVISION (*)	REVISION.
CUSTOMER NAME (*)	CUSTOMER NAME
SUBJECT NAME (*)	SUBJECT NAME
DESIGNER (*)	DESIGNER
TITLE	TITLE
FILE NAME	FILE NAME FOR THE PROJECT. HERE YOU CAN CHOOSE TO DISPLAY ANY PART OF THE FILE NAME. YOU CAN SELECT WITH FILE PATH , PATH NAME AND/OR WITH FILE EXTENSION .

DATAFIELD	DESCRIPTION
FILE DATE/TIME	FILE DATE/TIME FOR WHEN THE PROJECT WAS LAST SAVED. CHOOSE EITHER DATE AND TIME , DATE , TIME OR YEAR .
REFERENCE DESIGNATION	REFERENCE DESIGNATION DEFINED IN THE PROJECT: CHOOSE BETWEEN ALL , FUNCTION ASPECT AND LOCATION ASPECT , AND SPECIFY WHETHER TO INSERT THE DESIGNATIONS OR THE DESCRIPTIONS .
LOGO1 OR LOGO2	THE SELECTED LOGO IS INSERTED. — SEE “INSERTING LOGOS IN PROJECTS” ON PAGE 527.
CREATION DATE/TIME	TIME AND DATA FOR WHEN THE PROJECT WAS CREATED. CHOOSE EITHER DATE AND TIME , DATE , TIME OR YEAR .
REMARKS	REMARKS FROM THE REMARKS FIELD.
LAST REVISION	THE DATAFIELD LAST REVISION IS USED FOR DISPLAYING THE MOST RECENT REVISION OF THE PROJECT. THERE CAN BE CREATED A LIST OF REVISION DATAFIELDS IN THE MENUS, WHICH CAN BE FILLED OUT AS THE REVISIONS ARE PERFORMED. — SEE “DATAFIELD FOR SHOWING LAST REVISION” ON PAGE 811.
LAST WIRE NUMBER	DISPLAYS THE NAME OF THE LAST WIRE NUMBER IN THE PROJECT.

PAGE DATA DATAFIELDS

DATAFIELD	DESCRIPTION
DRAWING NO.	DRAWING NUMBER. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
APPROVED (*)	APPROVED. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
DATE (*)	DATE. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
REVISION (*)	REVISION. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
CONSTRUCTOR	CONSTRUCTOR. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
APPROVED BY	APPROVED BY. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
APPROVED DATE	APPROVED DATE. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
PAGE INDEX	PAGE INDEX.
PAGE NUMBER	PAGE NUMBER. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
TITLE	TITLE. EITHER FROM PAGE OR FROM CHAPTER (FROM THE CHAPTER DIVIDER PAGE).
REMARKS	REMARKS FROM THE REMARKS FIELD.
DATE FOR LAST CHANGE	TIME AND DATE FOR LAST CHANGE ON THE PAGE. CHOOSE BETWEEN DATE AND TIME , DATE , TIME OR YEAR . IF YOU NEED TO SET THE TIME/DATE FOR ALL PAGES IN THE PROJECT, CHOOSE FUNCTIONS => SPECIAL FUNCTIONS => SET DATE FOR LAST CHANGE ON ALL PAGES AND ANSWER YES TO SET THE DATE FOR LAST CHANGE TO NOW.
SCALE FACTOR	SCALE FACTOR.
PRINT SCALE	PRINT SCALE.
TEXT CORNER	TEXT CORNER.
FIRST USED PAGE	FIRST USED PAGE. HERE YOU MUST EITHER CHOOSE IN THE PROJECT OR IN THE CHAPTER . YOU CAN THEN SELECT/DESELECT ONLY SHOW DES. IF DIFFERENT , LOCATION ASPECT AND/OR FUNCTION ASPECT .
LAST USED PAGE	LAST USED PAGE. HERE YOU MUST EITHER CHOOSE IN THE PROJECT OR IN THE CHAPTER . YOU CAN THEN SELECT/DESELECT ONLY SHOW DES. IF DIFFERENT , LOCATION ASPECT AND/OR FUNCTION ASPECT .
NEXT USED PAGE	NEXT USED PAGE. YOU CAN SELECT/DESELECT NOT CHAPTERS , ONLY SHOW DES. IF DIFFERENT (THE REFERENCE DESIGNATIONS ARE ONLY DISPLAYED IF THEY DIFFER FROM THE REFERENCE DESIGNATIONS ON THE PREVIOUS PAGE), LOCATION ASPECT AND/OR FUNCTION ASPECT .
PREVIOUS USED PAGE	PREVIOUS USED PAGE. YOU CAN SELECT/DESELECT NOT CHAPTERS , ONLY SHOW DES. IF DIFFERENT (THE REFERENCE DESIGNATIONS ARE ONLY DISPLAYED IF THEY DIFFER FROM THE REFERENCE DESIGNATIONS ON THE PREVIOUS PAGE), LOCATION ASPECT AND/OR FUNCTION ASPECT .

DATAFIELD	DESCRIPTION
USED PAGES	USED PAGES. HERE YOU MUST EITHER CHOOSE IN THE PROJECT OR IN THE CHAPTER . YOU CAN THEN SELECT/DESELECT ONLY SHOW DES. IF DIFFERENT , LOCATION ASPECT AND/OR FUNCTION ASPECT .
PAGE DESIGNATION	PAGE DESIGNATION. CHOOSE BETWEEN DESIGNATION AND DESCRIPTION . THEN SELECT IF IT IS FOR THE REFERENCE DESIGNATION (THE FULL REFERENCE DESIGNATION), THE FUNCTION ASPECT OR THE LOCATION ASPECT .
PICTURE	BITMAP PICTURE.
CREATION DATE/TIME	CREATION DATE/TIME. CHOOSE BETWEEN DATE AND TIME , DATE, TIME AND YEAR .
LAST REVISION	THE DATAFIELD LAST REVISION IS USED FOR DISPLAYING THE MOST RECENT REVISION OF THE PROJECT. THERE CAN BE CREATED A LIST OF REVISION DATAFIELDS IN THE MENUS, WHICH CAN BE FILLED OUT AS THE REVISIONS ARE PERFORMED. — SEE “DATAFIELD FOR SHOWING LAST REVISION” ON PAGE 811.
FIRST/LAST OBJ	THE DATAFIELD FIRST/LAST OBJ CAN DISPLAY THE FIRST AND/OR THE LAST OBJECT IN THE LIST PAGE. HERE YOU CAN CHOOSE EITHER FIRST AND LAST , FIRST OR LAST .
LAST PRINTED	TIME AND DATE FOR WHEN THE PAGE WAS PRINTED LAST. HERE YOU CAN CHOOSE BETWEEN DATE AND TIME , DATE, TIME AND YEAR .

SYMBOL DATAFIELDS

Contains only user-designed symbol datafields. — See “Creating symbol datafields” on page 733.

Symbol datafields can also be created with a list of predefined values. — See “Symbol datafields with predefined values” on page 736.

TABLE OF CONTENTS DATAFIELDS

DATAFIELD	DESCRIPTION
DATE (*)	DATE
REVISION (*)	REVISION
PAGE INDEX	PAGE INDEX
PAGE NUMBER	PAGE NUMBER
TITLE	TITLE
DATE FOR LAST CHANGE	DATE FOR LAST CHANGE
SCALE FACTOR	SCALE FACTOR
PRINT SCALE	PRINT SCALE
PAGE DESIGNATION	PAGE DESIGNATION. THIS CAN EITHER BE PAGE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT .
CREATION DATE/TIME	CREATION DATE/TIME FOR THE PAGE. CHOOSE BETWEEN DATE AND TIME , DATE, TIME AND YEAR .

DATAFIELD	DESCRIPTION
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. YOU CAN HERE CHOOSE BETWEEN ON ALL LINES AND ON USED LINES .
FIRST/LAST OBJ	THE DATAFIELD FIRST/LAST OBJ CAN DISPLAY THE FIRST AND/OR THE LAST OBJECT IN THE LIST PAGE. YOU CAN HERE CHOOSE BETWEEN FIRST AND LAST , FIRST AND LAST .
LAST REVISION	THE DATAFIELD LAST REVISION IS USED FOR DISPLAYING THE MOST RECENT REVISION OF THE PROJECT. THERE CAN BE CREATED A LIST OF REVISION DATAFIELDS IN THE MENUS, WHICH CAN BE FILLED OUT AS THE REVISIONS ARE PERFORMED. — SEE “DATAFIELD FOR SHOWING LAST REVISION” ON PAGE 811.
REMARKS	REMARKS FROM THE REMARKS FIELD FOR THE PAGE. — SEE “PAGE REMARKS” ON PAGE 854.

PARTS/COMPONENTS LIST DATAFIELDS

Datafield names written in upper-case letters, are fields from the database. These are not displayed in the survey below.

DATAFIELD	DESCRIPTION
NAME	THE NAME OF THE COMPONENT. YOU CAN HERE ALSO CHOOSE WITH QUANTITY , LOCATION ASPECT AND FUNCTION ASPECT .
TYPE	THE TYPE OF THE COMPONENT.
ARTICLE NO.	THE ARTICLE NUMBER OF THE COMPONENT. YOU CAN CHOOSE FOR COMPONENT OR FOR SUBDRAWING . THIS ALSO MAKES IT POSSIBLE TO LET ARTICLE NUMBERS FOR SUB DRAWINGS APPEAR IN THE LIST. — READ MORE ABOUT SUB DRAWINGS IN “SUB DRAWINGS” ON PAGE 256.
FUNCTION	THE FUNCTION OF THE COMPONENT
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE.
SYMBOL NAME	THE SYMBOL FILE NAME OF THE COMPONENT
SYMBOL FIGURE	THE SYMBOL FIGURE OF THE COMPONENT
SYMBOL TITLE	THE TITLE OF THE COMPONENT
SYMBOL DATE	TIME AND DATE FOR SAVING THE SYMBOL. CHOOSE BETWEEN DATE AND TIME , DATE , TIME AND YEAR .

DATAFIELD	DESCRIPTION
QUANTITY	<p>THE QUANTITY OF THE COMPONENT.</p> <p>CHOOSE BETWEEN QUANTITY ("NUMBER OF UNITS"), QUANTITY COMPL. PACKAGES (ROUNDED UP TO THE NEAREST INTEGER), OR QUANTITY IN PACKAGES (SPECIFIED WITH DECIMALS).</p> <p>WHEN YOU SELECT WITH QUANTITY (ONLY POSSIBLE FOR QUANTITY COMPL. PACKAGES OR QUANTITY IN PACKAGES), BOTH QUANTITY AND PACKAGE INFORMATION IS DISPLAYED.</p> <p>PLEASE NOTE, THAT THE INFORMATION QUANTITY PR. PACKAGE IS SPECIFIED IN THE DATABASE FIELD UNITSPERPACK. (SELECT SETTINGS => DATABASE => DATABASE SETUP, AND ASSIGN THE DATABASE FIELD UNITSPERPACK TO UNITS/PACK. — SEE "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 610).</p> <p>YOU CAN ALSO SELECT/DESELECT +(NO OF CABLES). THIS GIVES YOU THE NUMBER OF CABLES IN BRACKETS AFTER THE QUANTITY IN PARTS LISTS.</p>
TOTAL QUANTITY	<p>TOTAL QUANTITY FOR THE FULL PROJECT. HERE YOU CAN SEE HOW MANY UNITS OF THIS ARTICLE NUMBER ARE USED IN THE PROJECT. CHOOSE BETWEEN QUANTITY ("NUMBER OF UNITS"), QUANTITY COMPL. PACKAGES (ROUNDED UP TO THE NEAREST INTEGER), OR QUANTITY IN PACKAGES (SPECIFIED WITH DECIMALS).</p> <p>WHEN YOU SELECT WITH QUANTITY (ONLY POSSIBLE FOR QUANTITY COMPL. PACKAGES OR QUANTITY IN PACKAGES), BOTH QUANTITY AND PACKAGE INFORMATION IS DISPLAYED.</p> <p>PLEASE NOTE, THAT THE INFORMATION QUANTITY PR. PACKAGE IS SPECIFIED IN THE DATABASE FIELD UNITSPERPACK. (SELECT SETTINGS => DATABASE => DATABASE SETUP, AND ASSIGN THE DATABASE FIELD UNITSPERPACK TO UNITS/PACK. — SEE "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 610).</p> <p>YOU CAN ALSO SELECT/DESELECT +(NO OF CABLES). THIS GIVES YOU THE NUMBER OF CABLES IN BRACKETS AFTER THE QUANTITY IN PARTS LISTS.</p>
UNIT QUANTITY	THE NUMBER OF COMPONENTS REFERRING TO THE UNIT PAGE
PRICE1	PRICE1
PRICE2	PRICE2
TOTAL ARTICLE PRICE1	TOTAL ARTICLE PRICE1
TOTAL ARTICLE PRICE2	TOTAL ARTICLE PRICE2
TOTAL PRICE1	TOTAL PRICE1
TOTAL PRICE2	TOTAL PRICE2
UNIT NAME LIST	UNIT NAME LIST. HERE YOU CAN SELECT/DESELECT LOCATION ASPECT AND/OR FUNCTION ASPECT .
NAME LIST	LIST OF COMPONENTS HAVING THE SAME ARTICLE NUMBER
REFPOSTITON	THE PAGE/CURRENT PATH NUMBER OF THE COMPONENT. HERE CAN BE REFERRED TO FUNCTION ASPECT , LOCATION ASPECT , PAGE AND CURRENTPATH .
MECHANICAL POSITION	THE LOCATION OF THE MECHANICAL SYMBOL. CAN BE EITHER X,Y POSITION OR FIELD POSITION . PLEASE REFER TO "REFERENCES IN LISTS TO THE POSITION OF MECHANICAL SYMBOLS" ON PAGE 636 FOR FURTHER INFORMATION.

DATAFIELD	DESCRIPTION
MECHANICAL PAGE	THE PAGE NUMBER FOR THE GROUND PLAN PAGE, WHERE THE MECHANICAL SYMBOL IS PLACED. YOU CAN HERE CHOOSE WHETHER TO ADD THE FUNCTION ASPECT AND/OR THE LOCATION ASPECT ALONG WITH THE PAGE NUMBER.
=+DESIGN	THE FUNCTION AND/OR LOCATION ASPECT FOR THE COMPONENT. THIS CAN EITHER BE REFERENCE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT . YOU ALSO CHOOSE WHETHER TO SHOW THE DESIGNATION OR THE DESCRIPTION .
SYMBOL TYPE	THE MAIN SYMBOL TYPE FOR THE COMPONENT.
CABLE WIRES	CHOOSE BETWEEN WIRE COUNT , USED WIRE COUNT AND FREE WIRES COUNT .
EAN13 BARCODE	DATAFIELD WHICH AUTOMATICALLY GENERATES AN EAN13 BARCODE FOR THE COMPONENT. YOU CAN HERE CHOOSE BETWEEN DIFFERENT STANDARD SIZES.
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. CHOOSE BETWEEN ON ALL LINES OR ON USED LINES .
SYMBOL FILE NAME	THE LIBRARY FILENAME FOR THE SYMBOL. YOU CAN HERE CHOOSE WITH FILE PATH , FILE NAME AND WITH FILE EXTENSION . WHEN THERE ARE MORE THAN ONE SYMBOL FOR A COMPONENT, THE DATAFIELD INSERTS THE FILENAME FOR THE MOST SIGNIFICANT SYMBOL.

TERMINALS LIST DATAFIELDS

Datafield names written in upper-case letters, are fields from the database. These are not displayed in the survey below.

DATAFIELD	DESCRIPTION
T.NAME	THE NAME OF THE TERMINAL. YOU CAN HERE SELECT WITH SUB NAME , WITH CONNECTION NAME , LOCATION ASPECT AND FUNCTION ASPECT .
T.TYPE	THE TYPE OF THE TERMINAL.
T.ARTICLE	THE ARTICLE NUMBER OF THE TERMINAL.
T.FUNCTION	THE FUNCTION OF THE TERMINAL.
T.=+DESIGN	THE FUNCTION AND LOCATION ASPECT OF THE TERMINAL. THIS CAN EITHER BE REFERENCE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT . YOU ALSO CHOOSE WHETHER TO SHOW THE DESIGNATION OR THE DESCRIPTION .
C.NAME	THE CONNECTION NAME OF THE TERMINAL.
C.FUNCTION	THE CONNECTION FUNCTION OF THE TERMINAL. THIS CAN EITHER BE OUTPUT SIDE OR INPUT SIDE .
C.LABEL	THE CONNECTION LABEL OF THE TERMINAL. THIS CAN EITHER BE OUTPUT SIDE OR INPUT SIDE .
C.DESCR.	THE CONNECTION DESCRIPTION OF THE TERMINAL. THIS CAN EITHER BE OUTPUT SIDE OR INPUT SIDE .
C.REF	THE CONNECTION CURRENT PATH OF THE TERMINAL. YOU CAN HERE SELECT FUNCTION ASPECT , LOCATION ASPECT , PAGE AND CURRENT PATH .

DATAFIELD	DESCRIPTION
IE.NAME	INTERNAL/EXTERNAL COMPONENT NAME. THIS CAN EITHER BE EXTERNAL OR INTERNAL . YOU CAN HERE SELECT WITH SUB NAME, WITH CONNECTION NAME, LOCATION ASPECT AND FUNCTION ASPECT .
IE.TYPE	INTERNAL/EXTERNAL COMPONENT TYPE. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.ARTICLE	INTERNAL/EXTERNAL COMPONENT ARTICLE NUMBER. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.FUNCTION	INTERNAL/EXTERNAL COMPONENT FUNCTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.=+DESIGN	INTERNAL/EXTERNAL COMPONENT FUNCTION/LOCATION. THIS CAN EITHER BE EXTERNAL OR INTERNAL . CHOOSE BETWEEN REFERENCE DESIGNATION, FUNCTION ASPECT OR LOCATION ASPECT .
IE.=+DESCRIPTION	THE DESCRIPTION FOR THE INTERNAL/EXTERNAL COMPONENT FUNCTION/LOCATION. THIS CAN EITHER BE EXTERNAL OR INTERNAL . CHOOSE BETWEEN REFERENCE DESIGNATION, FUNCTION ASPECT OR LOCATION ASPECT .
IE.C.NAME	INTERNAL/EXTERNAL COMPONENT CONNECTION NAME. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.C.FUNCTION	INTERNAL/EXTERNAL COMPONENT CONNECTION FUNCTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.C.LABEL	INTERNAL/EXTERNAL COMPONENT CONNECTION LABEL. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.C.DESCR.	INTERNAL/EXTERNAL COMPONENT CONNECTION DESCRIPTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.C.REF.	INTERNAL/EXTERNAL COMPONENT CONNECTION CURRENT PATH. THIS CAN EITHER BE EXTERNAL OR INTERNAL . YOU CAN HERE SELECT WITH SUB NAME, WITH CONNECTION NAME, LOCATION ASPECT AND FUNCTION ASPECT .
IE.SIGNAL	INTERNAL/EXTERNAL SIGNAL NAME. THIS CAN EITHER BE EXTERNAL OR INTERNAL . YOU CAN ALSO CHOOSE FUNCTION ASPECT AND LOCATION ASPECT .
IE.WIRENO	INTERNAL/EXTERNAL WIRE NUMBER. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.NAME	INTERNAL/EXTERNAL CABLE NAME. THIS CAN EITHER BE EXTERNAL OR INTERNAL . YOU CAN HERE SELECT WITH SUB NAME, WITH CONNECTION NAME, LOCATION ASPECT AND FUNCTION ASPECT .
IE.Cb.TYPE	INTERNAL/EXTERNAL CABLE TYPE. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.ARTICLE	INTERNAL/EXTERNAL CABLE ARTICLE NUMBER. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.FUNCTION	INTERNAL/EXTERNAL CABLE FUNCTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.=+DESIGN.	INTERNAL/EXTERNAL CABLE FUNCTION/LOCATION. THIS CAN EITHER BE EXTERNAL OR INTERNAL . CHOOSE BETWEEN REFERENCE DESIGNATION, FUNCTION ASPECT AND LOCATION ASPECT .
IE.Cb.=+DESCRIPTION	DESCRIPTION FOR THE INTERNAL/EXTERNAL CABLE FUNCTION/LOCATION. THIS CAN EITHER BE EXTERNAL OR INTERNAL . CHOOSE BETWEEN REFERENCE DESIGNATION, FUNCTION ASPECT AND LOCATION ASPECT .

DATAFIELD	DESCRIPTION
IE.Cb.W.NAME	INTERNAL/EXTERNAL CABLE WIRE NAME. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.W.FUNCTION	INTERNAL/EXTERNAL CABLE WIRE FUNCTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.W.LABEL	INTERNAL/EXTERNAL CABLE WIRE LABEL. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.W.DESCR.	INTERNAL/EXTERNAL CABLE WIRE DESCRIPTION. THIS CAN EITHER BE EXTERNAL OR INTERNAL .
IE.Cb.W.REF.	INTERNAL/EXTERNAL CABLE WIRE CURRENT PATH. THIS CAN EITHER BE EXTERNAL OR INTERNAL . YOU CAN HERE SELECT WITH SUB NAME , WITH CONNECTION NAME , LOCATION ASPECT AND FUNCTION ASPECT .
JUMPER LINK	JUMPER LINK CONNECTION OF TERMINAL.
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. YOU CAN HERE CHOOSE ON ALL LINES OR ON USED LINES .
Pos. No.	THE POSITION NUMBER.
LINE-NAME	NAME OF THE CONNECTED LINE. CHOOSE BETWEEN OUTPUT SIDE AND INPUT SIDE .
LINE-TYPE	TYPE OF THE CONNECTED LINE. CHOOSE BETWEEN OUTPUT SIDE AND INPUT SIDE .
LINE-ARTICLE	ARTICLE NUMBER OF THE CONNECTED LINE. CHOOSE BETWEEN OUTPUT SIDE AND INPUT SIDE .
LINE-FUNCTION	FUNCTION OF THE CONNECTED LINE. CHOOSE BETWEEN OUTPUT SIDE AND INPUT SIDE .

CABLE LIST DATAFIELDS

Datafield names written in upper-case letters, are fields from the database. These are not displayed in the survey below.

DATAFIELD	DESCRIPTION
CB.NAME	CABLE NAME. THIS CAN EITHER BE WITH CONNECTION NAME , LOCATION ASPECT OR FUNCTION ASPECT .
CB.TYPE	CABLE TYPE
CB.ARTICLE	CABLE ARTICLE NUMBER
CB.FUNCTION	CABLE FUNCTION
CB.=+DESIGN.	CABLE FUNCTION AND LOCATION ASPECT. THIS CAN EITHER BE REFERENCE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT . YOU ALSO CHOOSE WHETHER TO SHOW THE DESIGNATION OR THE DESCRIPTION .
CB.W.NAME	CABLE WIRE NAME
CB.W.FUNCTION	CABLE WIRE TYPE
CB.W.LABEL	CABLE WIRE ARTICLE NUMBER
CB.W.DESCR.	CABLE WIRE FUNCTION
CB.W.REF.	CABLE WIRE CURRENT PATH. HERE CAN BE REFERRED TO FUNCTION ASPECT , LOCATION ASPECT , PAGE AND CURRENT PATH .
FT.NAME	FROM/TO COMPONENT NAME. THIS CAN EITHER BE WITH SUB NAME , WITH CONNECTION NAME , LOCATION ASPECT OR FUNCTION ASPECT .
FT.TYPE	FROM/TO COMPONENT TYPE

DATAFIELD	DESCRIPTION
FT.ARTICLE	FROM/TO COMPONENT ARTICLE NUMBER
FT.FUNCTION	FROM/TO COMPONENT FUNCTION
FT.=+DESIGN.	FROM/TO COMPONENT FUNCTION AND LOCATION ASPECT. THIS CAN EITHER BE REFERENCE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT .
FT.=+DESCRIP-TION	DESCRIPTION FOR THE FROM/TO COMPONENT FUNCTION AND LOCATION ASPECT. THIS CAN EITHER BE REFERENCE DESIGNATION , FUNCTION ASPECT OR LOCATION ASPECT .
FT.C.NAME	FROM/TO COMPONENT CONNECTION NAME
FT.C.FUNCTION	FROM/TO COMPONENT CONNECTION TYPE
FT.C.LABEL	FROM/TO COMPONENT CONNECTION ARTICLE NUMBER
FT.C.DESCR.	FROM/TO COMPONENT CONNECTION FUNCTION
FT.C.REF.	FROM/TO COMPONENT CONNECTION CURRENT PATH. HERE CAN BE REFERRED TO FUNCTION ASPECT , LOCATION ASPECT , PAGE AND CURRENT PATH .
CB.W.SIGNAL	CABLE WIRE SIGNAL NAME. YOU CAN ALSO SELECT LOCATION ASPECT AND FUNCTION ASPECT .
CB.W.WIRENO	CABLE WIRE WIRE NUMBER
CABLE WIRES	CHOOSE BETWEEN WIRE COUNT , USED WIRE COUNT AND FREE WIRES COUNT .
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. CHOOSE ON ALL LINES OR ON USED LINES .
LINE-NAME	NAME OF THE CONNECTED LINE.
LINE-TYPE	TYPE OF THE CONNECTED LINE.
LINE-ARTICLE	ARTICLE NUMBER OF THE CONNECTED LINE.
LINE-FUNCTION	FUNCTION OF THE CONNECTED LINE.
QUANTITY	QUANTITY. CHOOSE BETWEEN: QUANTITY: 3 (FROM THE QUANTITY FIELD IN THE DIALOG BOX LINE ARTICLE DATA), QUANTITY COMPL. PACKAGES: 1 (QUANTITY IN COMPLETE PACKAGES — THE SIZE OF THE PACKAGES IS SPECIFIED IN THE DATABASE FIELD UNITSPERPACK) AND QUANTITY IN PACKAGES: 0,6 .
LENGTH	DISPLAYS THE LENGTH OF FOR INSTANCE A CABLE OR A WIRE.

PLC LIST DATAFIELDS

Datafield names written in upper-case letters, are fields from the database. These are not displayed in the survey below.

For all datafields from the database it is possible to specify whether to display data for the **PLC** or the **Connected component**.

DATAFIELD	DESCRIPTION
PLC NAME	PLC NAME. THIS CAN EITHER BE WITH SUB NAME , WITH CONNECTION NAME , LOCATION ASPECT OR FUNCTION ASPECT .
PLC TYPE	PLC TYPE
PLC ARTICLE	PLC ARTICLE
PLC FUNCTION	PLC FUNCTION

DATAFIELD	DESCRIPTION
PLC =+DESIGN.	PLC FUNCTION AND LOCATION ASPECT. THIS CAN EITHER BE REFERENCE DESIGNATION, FUNCTION ASPECT OR LOCATION ASPECT . CHOOSE EITHER DESIGNATION OR DESCRIPTION .
C.NAME	CONNECTION NAME OF PLC
C.FUNCTION	CONNECTION FUNCTION OF PLC
C.LABEL	CONNECTION LABEL OF PLC
C.DESCR.	CONNECTION DESCRIPTION OF PLC
C.REF.	CONNECTION CURRENT PATH OF PLC. HERE CAN BE REFERRED TO FUNCTION ASPECT, LOCATION ASPECT, PAGE AND CURRENTPATH .
Cc.NAME	CONNECTED COMPONENT NAME. THIS CAN EITHER BE WITH SUB NAME, WITH CONNECTION NAME, LOCATION ASPECT OR FUNCTION ASPECT .
Cc.TYPE	CONNECTED COMPONENT TYPE
Cc.ARTICLE	CONNECTED COMPONENT ARTICLE NUMBER
Cc.FUNCTION	CONNECTED COMPONENT FUNCTION
Cc.=+DESIGN.	CONNECTED COMPONENT FUNCTION AND LOCATION ASPECT. THIS CAN EITHER BE PAGE DESIGNATION, FUNCTION ASPECT OR LOCATION ASPECT . CHOOSE BETWEEN DESIGNATION AND DESCRIPTION .
Cc.C.NAME	CONNECTED COMPONENT CONNECTION NAME.
Cc.C.FUNCTION	CONNECTED COMPONENT CONNECTION FUNCTION
Cc.C.LABEL	CONNECTED COMPONENT CONNECTION LABEL
Cc.C.DESCR.	CONNECTED COMPONENT CONNECTION DESCRIPTION
Cc.C.REF.	CONNECTED COMPONENT CONNECTION CURRENT PATH. HERE CAN BE REFERRED TO FUNCTION ASPECT, LOCATION ASPECT, PAGE AND CURRENT PATH .
C.SIGNAL	CONNECTED SIGNAL NAME. YOU CAN ALSO CHOOSE LOCATION ASPECT AND FUNCTION ASPECT .
C.WIRENO.	CONNECTED WIRE NUMBER
SIGNALPATH	PLC SIGNAL PATH THROUGH TERMINALS AND CABLES. THIS CAN EITHER BE WITH SUB NAME, WITH CONNECTION NAME, LOCATION ASPECT OR FUNCTION ASPECT .
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. CHOOSE BETWEEN ON ALL LINES AND ON USED LINES .
LINE-NAME	NAME OF THE CONNECTED LINE. CHOOSE BETWEEN AT PLC CONNECTION AND AT CONNECTED SYMBOL . — SEE “ARTICLE DATA ON LINES” ON PAGE 79.
LINE-TYPE	TYPE OF THE CONNECTED LINE. CHOOSE BETWEEN AT PLC CONNECTION AND AT CONNECTED SYMBOL . — SEE “ARTICLE DATA ON LINES” ON PAGE 79.
LINE-ARTICLE	ARTICLE NUMBER OF THE CONNECTED LINE. CHOOSE BETWEEN AT PLC CONNECTION AND AT CONNECTED SYMBOL . — SEE “ARTICLE DATA ON LINES” ON PAGE 79.
LINE-FUNCTION	FUNCTION OF THE CONNECTED LINE. CHOOSE BETWEEN AT PLC CONNECTION AND AT CONNECTED SYMBOL . — SEE “ARTICLE DATA ON LINES” ON PAGE 79.

THE DATABASE FIELD PCSTYPE

One symbol:

PLC1OUT-UF

Multiple symbols: ;

PLC1OUT-P;PLC1OUT-UF

Multiple symbols of the same kind: #n

PLC1OUT-UF#8

PLCREF8-;PLC1OUT-UF#8

Or-function with multiple drawing options: [A|B|C]

PLCREF8-;[PLC1OUT-UF#8|PLC8OUT]

Sub names: A()

PLCREF4I4O;O(PCSOUT-UF#4);I(PLCIN-UF#4)

Symboltype: =A or =A,B

PLCREF8-; PLC1OUT-UF=T,P#8

Options:

N	Normal	T,R	Terminal/Relay
R	Relay	T,O	Terminal/Open
O	Open	T,C	Terminal/Close
C	Close	T,S	Terminal/Switch
S	Switch	T,P	Terminal/PLC
M	MasterReference		
W	WithReference		
T	Terminal		
P	PLC		
X	PLC Reference		

States: §n

PLCREFOUT4D4A;PLC1OUTAD#4§1; PLC1OUTAD#4§2

Activating "With reference"

SYMBOLFILENAME*W **With reference** is activated for the symbol,
with the status **With reference**)

SYMBOLFILENAME*M **With reference** is activated for the symbol,
with the status **Main reference**)

FILE=

If the **PCSTYPE** field is too short, you can type in **FILE=xxxxxx.pcs**, where **xxxxxx.pcs** is the file containing the information for the **PCSTYPE** field, specified according to the syntax described above.

The file must be placed in the database folder, which you have selected in **Settings => Directories**.

Please note! In PCSTYPE files semicolon is the same as a line break:

	PLCREF8-;PLC1OUT-UF#8
Is equal to:	PLCREF8- PLC1OUT-UF#8
Or:	PLCREF8-; PLC1OUT-UF#8;

THE DATABASE FIELD PINDATA

One connection:

10

Multiple connections on the same symbol: ,

1,2,3,4,5,6

Consecutive connection names on the same symbol: ,...,

1,...,6

Separation between the connections of multiple symbols: ;

1,2;3,4;5,6

Consecutive connection names on symbols with a single connection: ;...;

1;...;6

Specification of I/O address: [adr]

1[10.00];2[10.01];3[10.02]

Specification of I/O status: /I:status

1[/I:OP];2[/I:IP]

1[10.00/I:IP];2[10.01/I:IP]

Options:

- ET** External terminal connection
- IT** Internal terminal connection
- OP** Output PLC connection
- IP** Input PLC connection
- EI** Eksternal terminal connection with PLC input
- EO** Eksternal terminal connection with PLC output
- II** Internal terminal connection with PLC input
- IO** Internal terminal connection with PLC output
- N** Changes the status to "no status"

THE DATABASE FIELD PINDATA: COUNTING

Counting on the same symbol: {++(A),}#n

Please note: There must be a comma before }

{++(1),}#16 => 1,2,3,4,5,6,.....
 {++(1)[10.++(0),}#16 => 1[10.0],2[10.1],3[10.2],.....
 {A++(1)[Y10.++(0),}#16 => A1[Y10.0],A2[Y10.1],A3[Y10.2],.....

Counting on multiple symbols with one connection each: {++(A);}#n

Please note: There must be a semicolon before }

{++(1);}#16 => 1;2;3;4;5;6;.....
 {++(1)[10.++(0);}#16 => 1[10.0];2[10.1];3[10.2];.....
 {A++(1)[Y10.++(0);}#16 => A1[Y10.0];A2[Y10.1];A3[Y10.2];.....

Counting on multiple symbols with two connections each: {++(A),++(B);}#n

{++(1),++(1)G;}#8 => 1,1G;2,2G;3,3G;.....
 {++(1)[10.++(0)],++(1)G;}#8=>1[10.0],1G;2[10.1],2G;.....

Incrementing with more than 1: {++d(A);}#n

{++2(1);}#8 => 1;3;5;7;9;.....
 {++2(1),++2(2);}#8 => 1,2;3,4;5,6;7,8;9,10;.....

Incrementing with specified numbering system: {++(DA);}#n

{++(1)[10.++(D0)];}#8 => {++(1)[10.++(0)];}#8 Decimal
 {++(1)[10.++(H0)];}#8 => Hexadecimal
 {++(1)[10.++(O0)];}#8 => Octal
 {++(1)[10.++(B0)];}#8 => Binary

Neutralize the significance of D,H,O,B: {++(/DA);}#n

Please note: There must be placed a / before D

{++(1)[++(/O0)];}#8 => 1,00;2,02;3,03;.....

Specification of field width (number of digits): {++(A:x);}#n

{++(1)[10.++(00:2)];}#8 => 1[10.00];2[10.01];3[10.02];.....

Decrementing: {--(A);}#n

{--(20),++(1);}#4 => 20,1;19,2;18,3;17,4;.....

Example

A PLC reference symbol with two connections for each PLC output and eight PLC symbols with each two connections pr. symbol:

PCSTYPE = PLCREF8-1;PLC1OUT-P#8

PINDATA = {B++(0)[++(10.00)],A++(0),}#8; {B++(0),A++(0);}#8

The incremented value of PINDATA:

B0[10.00],A0,B1[10.01],A1,.....,B7[10.07],A7; B0,A0; B1,A1;.....;B7,A7

FILE=

If the **PINDATA** field is too short, you can type in **FILE=xxxxxx.pin**, where **xxxxxx.pin** is the name of the file with the **PINDATA** contents, specified according to the syntax described above.

The file must be placed in the database folder, which you have selected under **Settings => Directories**.

Please note! In PCSTYPE files semicolon is the same as a line break:

{B++(0)[++(10.00)],A++(0),}#8; {B++(0),A++(0);}#8

Is equal to: {B++(0)[++(10.00)],A++(0),}#8

{B++(0),A++(0);}#8

Or: {B++(0)[++(10.00)],A++(0),}#8;

{B++(0),A++(0);}#8;

VARIABLES

Variable definition: /@n

/@n n is a number between 1 and 64000. Variables containing small numbers will be processed faster than variables containing larger numbers. Therefore smaller numbers are recommended in variables.

/@1 Variable 1 /@10 Variable 10

Normal variable: /@n

/@1 The variable has no default value. Fields in the project definition file, where the variable value will be typed in later, are empty.

Variable with default value: xxxx/@n

3389110610000/@1 3389110610000 is displayed in field in the project definition file, where the variable value will be typed in later.

-P1/@3 The variable has the default value -P1.
The value that is typed in later, overwrites the default value.

Variable that inserts a value in a text: xxxx</@n>

-P</@3> The text (name) has a fixed part (-P) and a variable part </@3>

</@4> out

</@4> in

Can be applied with benefit where the same value shall be inserted into texts, where the fixed part are different from text to text. Here the values could for instance be 'Weighing' in one case, and 'Casting' in another.

-P</@3>

-S</@3>

Can be applied where all — or some — names in a sub drawing always have the same serial number.
E.g. -P1 and -S1

Insertion with default value: xxxx<yyyy/@n>

-P<1/@3> The text (name) has a fixed part (-P) and a variable part </@3>, which has the default value '1'.

<Weighing/@4> out

The variable has the default value 'Weighing'.

Indexed variables: /@n[i] or /@n[i1,i2,i3,....]

/@4[1]

The variable takes the value from the first parameter in a comma separated text.

E.g.: BK,BL,BU,GR The variable gets the value BK

/@4[2]

E.g.: BK,BL,BU,GR The variable gets the value BL

/@4[2,3]

E.g.: BK,BL,BU,GR The variable gets the value BL,BU

This function is very useful for cables.

Below this is shown with cable wire colors.

< BK,BL,BU,GR/@4[1]> => BK

< BK,BL,BU,GR/@4[2]> => BL

< BK,BL,BU,GR/@4[3]> => BU

< BK,BL,BU,GR/@4[4]> => GR

Possible control via variables in symbol names

/@1=-M1

=> The symbol gets the name **-M1**

/@1=!

=> The symbol and connecting lines becomes invisible.

/@1=!C

=> The symbol and connecting lines — up to a cable symbol — becomes invisible.

/@1=?

=> The symbol name is not controlled from the variable.

Variables in reference frames

To insert a variable to control the reference designation in a reference frame:



1) Click on **Texts**

2) Double-click on the reference designation for the reference frame

3) In the dialog box **Change text**: Type in the name of the variable (e.g. /@3)

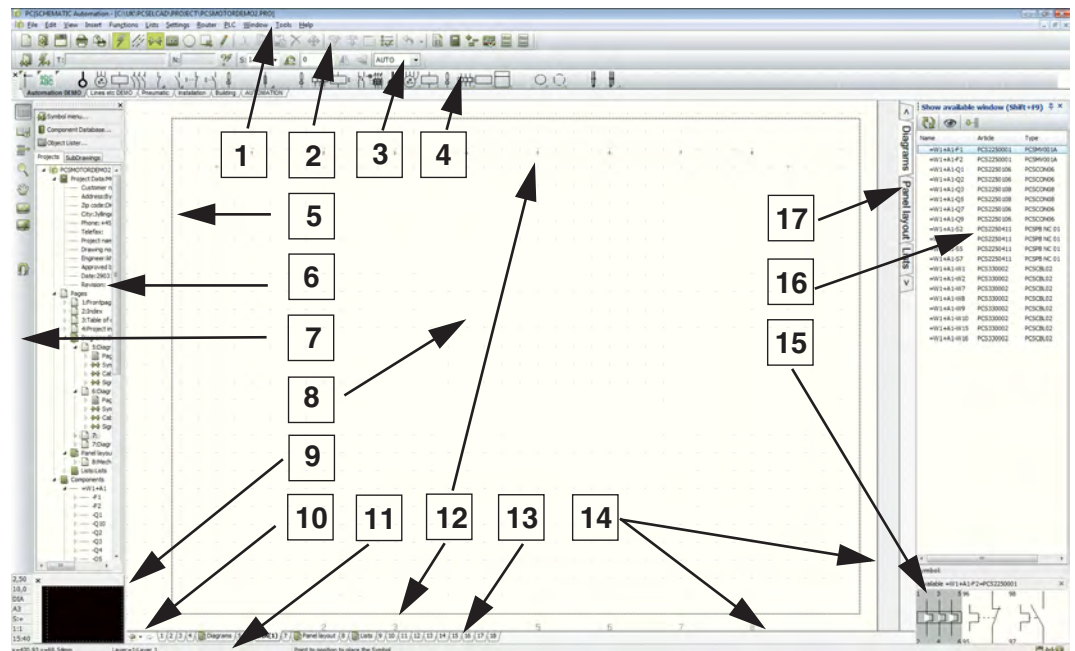
Variables in reference designations for pages

To insert a variable to control the reference designation for a page:

1) Right-click on the page and choose **Page article data**

2) In the dialog box **Page article data**: Type in the name of the variable (e.g. /@7) in the **Name** field

DICTIONARY



Activate next (in lists): The first time the program meets a check mark in an *Activate next* check box, you tell the program to collect information about the first component in the list. Subsequent datafields, such as *Description* and *Quantity*, will then be regarded as the description and quantity of the first component respectively.

The next time the program meets a check mark in the **Activate next** check box, it collects information about the following component on the list. The subsequent datafields *Description* and *Quantity*, will then be regarded as the description and quantity of the next component on the list — and so on.

It is important to note, that if you place a datafield just a bit higher on the page, than the datafield for which you enabled **Activate next**, this datafield will collect it is information from the previous component.

Please refer to “Creating Lists” on page 752 to get an example on how to use **Activate next**.

Alignment: Specifies where the reference point of a text is located. Read more about this in “**Specifying text properties / Appearance of text**” on page 122. See *reference point*.

Click: To press down the left mouse button once. Read more about the actions you can perform with the mouse in “The mouse” on page 25.

Command toolbar: Has got a varying appearance on the screen. It holds functions and editing-tools for the various drawing facilities. Point number 3 on the figure on page 873.

Connection name: The name of a *connection point*.

Connection point: The point in which a symbol can be connected to a line. Please refer to “Place connection points” on page 699 for more information.

Cross hairs: When you are working on a project drawing, the location of the mouse pointer is demarcated by the crossing of a horizontal and a vertical line. This point is called the *cross hairs*.

Datafield: All information concerning the entire project, is gathered in the dialog box **Settings => Project data**. Among others, the project title, and the name and address of the customer is specified in this dialog box.

If you need to insert one of these informations on a list in your project — for instance the address of the customer — choose **Insert => Insert Datafield**. Here you first click on **Project data** and then on the *datafield Address*. When you update your list, the address of the customer will be inserted where you placed the datafield.

If the customer changes his address later on, you *only* have to change the address in **Settings => Project data**, which causes the address to be changed automatically everywhere the *datafield Address* exists in the project.

Datafields therefore makes it possible for you to control central information — which is being applied several places in the project — in just one dialog box. If an information have to be changed, you just have to change it once, and the program will correct all the places the datafields are inserted in the project.

Datafields attached to the individual pages in the project, can be found in **Settings => Page data**.

Dimmed: If an option in a menu is written in grey, you say it is *dimmed*, and it cannot be chosen.

Directories: See *Folders*.

Docking: To move a program window or toolbar to another fixed position on the screen. Please refer to “Docking Windows and Toolbars” on page 26.

Double-click: To click two times on the left mouse-button. It is important not to move the mouse while clicking. Read more about the actions you can perform with the mouse in “The mouse” on page 25.

Drag’n draw: 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. — Read more about this in “Sub drawings” on page 256.

Drag and drop: Press down the left mouse button, and keep it pressed down while moving the mouse pointer across the screen. Is typically applied when you move an object or apply the zoom function. You are said to *drag* the object across the screen. Read more about the actions you can perform with the mouse in “**The mouse**” on page 25.

Drawing header: Apart from being what you normally understand as a drawing header — namely a frame containing information which can be placed around a project drawing — formulas for lists, tables of contents and unit drawings are also conceived as drawing headers in **Automation**.

You can say that a *drawing header* is a template with text and graphics, which you can choose to place on the various pages in your project. It can for instance contain your company logo. Read more about drawing headers, and how to create

your own drawing header in “Creating Drawing Headers” on page 742.

Read more about how to create lists in “Creating Lists” on page 752.

Read more about how to create unit drawings in “Creating Unit Drawings” on page 767.

And, finally, you can read about how to insert an already created drawing header in “Insert drawing headers/formulas” on page 531.

Drawing objects: See *objects*.

Drive: The place where all your *folders* and *files* are gathered is called a *drive*. If you have to load a *file* from a diskette, you have to contact the disk drive (often called A:), and if you have to load something from your hard disk, you for instance have to contact the c:\-drive (There can also be other drives on your hard disk).

Elevation: The height in which an object is placed on a ground plan drawing. Read more about this in “Drawing with elevation” on page 482.

Elevation window: Is used to type in the height in which you want to place an object on a ground plan drawing.

Explorer window: A window in which you can locate pages and symbols in the projects. Point number 6 on the figure on page 873. Described in further detail in “The Explorer window: The Projects tab” on page 301.

File: Anything you save on a computer, is saved in what you call a *file*. Each time you for instance save a project or a symbol, these are saved in a file. When you fetch the project or symbol again, you are said to *open* the file.

Folder: In order not to place all *files* on the computer in the same place, you collect *files*, which belong together, in the same folder. Inside this folder you can create sub folders — and so on. In “Folder structure” on page 32 you can see which folders the installation of **Automation** contains.

Free text: A text which is not connected to a symbol, a line or the like. Read more in “General purpose / Free texts” on page 102.

Grid: The dots you see placed at fixed intervals on the paper on a drawing, are called the *grid* of the drawing. You specify the distance between the dots in **Settings => Page setup**. Read more about this in “Grid” on page 47.

Hints: When you let the mouse pointer rest on for instance a program button, you will see the name of the button displayed in a small text field. Or, if you let the mouse pointer rest for a moment on a symbol in the symbol toolbar, you will get a short description of the symbol. This facility is called *hints*, and can enabled/disabled in **Settings => Screen/Pointer**.

Left toolbar: Contains various page- and zoom functions, as well as information on the page settings. Point number 7 on the figure on page 873.

Maximize: To *maximize* a window means to make it fill out the entire screen. In the upper-right corner of a window, you will find the **Maximize** button. Click this button, and you will maximize the window.



Menu: A place where you can choose which command to carry out in the program. See “**Menus**” on page 23 for further details.

Menu bar: Consists of the menus, that contains all the functions in the program. Use the mouse or the <Alt> key to choose the functions. For instance choose **File** by pressing <Alt+f>. Point number 1 on the figure on page 873.

Minimize: To *minimize* a window means to remove it from the screen temporarily, so that you cannot work on it. It will then be placed as a small button on the lower part of the screen. When you click on this button, the window is enlarged, and you will be able to work on it again.



In the upper-right corner of a window, you will find the **Minimize** button. Click this button, and you will minimize the window.

MMB Drag'n Draw: Short form of "Module and Model Based Drag'n Draw". — See *drag'n draw*.

Mouse pointer: The arrow/pointer displayed on the screen, which is controlled by the mouse. When it is above a field in which you can enter text, it is converted to a short vertical line.

Non conducting symbol: A symbol, which cannot be connected to a conducting line, without causing a warning to appear. If you connect a normal symbol to a non conducting line, you will be warned about it, but will be allowed to do so if you insist.

NP: “Non printable.” Texts/lines/circles in this color will be displayed on the screen, but not on print. Read more about this in “Specifying text properties / Appearance of text” on page 122.

Objects: A common name for the symbols, lines, texts, and circles that can be placed on a project drawing. Also called *drawing objects*. Read more about this in “On Drawing objects” on page 53.

Page tabs: On the lower part of the screen you will see some small tabs, telling you which project page you are located on. By clicking on these tabs, you can jump back and forth between the project pages. These tabs are called *page tabs*, and can be activated/deactivated in **Settings => Pointer/Screen**.

Parking: If you have not specified what a conduction line is to be connected to, you can temporarily *park* the line. You can then wait to connect it until later. Read more about this in “Conducting (electrical) lines” on page 78 and “Signal symbols” on page 203.

Path: If you want to specify the exact location of a *file* on your PC, you can specify the *path* that leads to the file. This path starts by specifying on which *drive* the file is located, and then specifies which folder or sub folder the file is located. At the end, there is the file name itself. The file **project.pro** which is saved in the **Project** folder, thus has the path: **c:\Pcscad\Project\project.pro**.

Pickmenu: A place you can place symbols for easy access while working. Point number 4 on the figure on page 873. Described in further detail in “Fetching symbols from the Pickmenu” on page 140.

Program toolbar: Contains the program buttons. Here you can choose the various functions. The common file- and print functions are located here, along with the more general drawing- and editing tools. Point number **2** on the figure on page 873.

Reference point: All symbols and texts have got a reference point. For symbols, the reference point is demarcated by a *, while the reference point for texts is +. When you want to move a symbol, it is the reference point you click on to select (get in contact with) the symbol, and if you want to rotate the symbol, it is rotated around the reference point. — This also goes for texts. Read more in “Creating Symbols” on page 696 and in “**Specifying text properties / Appearance of text**” on page 122.

Right-click: To click once on the right mouse-button. Read more about the actions you can perform with the mouse in “The mouse” on page 25.

Scroll: If you only see a smaller part of your drawing on the screen for the moment, and wish to see something that is placed to the right of the displayed section, you can gradually move the displayed section to the right on the drawing, until you see what you want on the screen. Moving the displayed section is called scrolling.

You can scroll by using the **Survey** window in the lower-right corner of the screen. *Click* on the black box in the survey window, and *drag* it until it covers the area you want to see on the full screen.

Read more about this in “Screen / Image Functions” on page 42.

Signal symbol: A symbol, which states an electrical connection, that is not displayed otherwise on the drawing. Thus, if a signal symbol with the name **L1** is placed on two different pages in a project, this means these two places are electrically connected. It is the signal *name* only, that states the electrical connection.

Therefore, a signal symbol having the *name* **L2**, has no connection to a signal symbol having the *name* **L1** — even though the same symbol is used for the two signals.

Read more about signal symbols in “Creating Signal Symbols” on page 738.

Snap: When you want to place an *object* on the screen, it is important to be able to place it exactly where you want it. To support this, you have got the option to specify that objects can only be placed at fixed intervals of, say, 2.50mm. — If you then want to place a symbol, you will only be able to move it in steps of 2.50mm at the time. This enables you to place the symbol accurately in relation to the other symbols.

If the distance between the points in which the symbols can be placed is 2.50mm, you say that *snap* is 2.50mm.



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

If <Shift> is pressed down while an *object* is in the *cross hairs*, you will change to the fine snap, and as soon as you let go of the <Shift> key, or place the object, you return to normal snap.

In **Settings => Page setup** you specify the snap settings.

Read more about snap in “Snap” on page 47.

Status bar: Contains coordinates, layer title and help text. Point number **11** on the figure on page 873.

Sub drawing: See “Sub drawings” on page 256.

Survey window: Displays the entire page, while the *zoom*, you are presently working in, is displayed by a black box. Point number **9** on the figure on page 873. Read more in “The Survey window” on page 46.

Tanking: See “Tanking licenses” on page 794.

Untick: To remove a check mark from a check box.

Work area: Reflects the size of paper you have chosen to draw on. Point number **8** on the figure on page 873.

Zoom: If you need to *zoom in* on something, this means you have to mark up a section of the drawing you have on the screen, and then the section will be enlarged.

Read more about this in “Screen / Image Functions” on page 42.

INDEX FOR PART 1 AND PART 2

A

Absolute measures 93
 Access 623
 Accessories
 for cables 604
 Accessory symbols 225
 Activate next 111, 756, 873
 Active 478
 Active, check box 380
 Actual page 481
 Adaptability
 symbol names 385
 Adapting the system to your needs 786
 Add a wire to a cable 223
 Add new 494, 514
 Additional 613
 Adjust to printer 329
 Adobe Acrobat Reader 37
 ALARM, symbol folder 226
 Alias
 correct 842
 Align 66, 631
 Align grid to 1. Reference 836
 Alignment 124
 All pages 327
 All visible 478
 Already placed objects 54
 Alternative electrical symbols 594
 Alternative symbols
 and PINDATA 596
 for PLC 595
 Ambiguous drawing 392
 ANSI
 paper size 509
 Apply addresses 424
 Approved components
 in Database menu 614
 Arcs/circles 53, 100
 toolbar 100
 Area button 53
 Areas
 change symbol names in 185
 change symbol names in, naming order 186
 choosing connection points in 189
 copy
 between projects 255
 into Edit symbol 255
 reference designations intelligently 247
 copy between projects 518
 copy intelligently 245
 counting on texts in areas 120
 filled 75
 invisible 70
 rotate 253

 scaling texts in 254
 Article data 835
 on lines 79
 transfer for symbols or lines 62
 Article number
 type in directly 154
 Articles from unit drawings 544
 Ask for Names 107, 837
 Assign article number to unit parts list 768
 Asterisk 550
 Auto hide scrollbars 838
 Autocompletion of texts 103
 Automatic
 blocking of mouse clicks 802
 insertion of electrical potentials 81
 line drawing 89
 recovery information 849
 wire numbering 389
 Automatic information 784
 Automatic renumbering of pages 832
 Automatic switches, database folder 153

B

Background color 792
 Baltic characters 787, 850
 Barcode scanners 624
 Base addressing
 for PLC's 419
 Batch printing 823
 on net installation 823
 on stand-alone installation 823
 BDE 623
 Bend 96
 Black background 839
 Bold text 125
 Bookmarks
 in Symbol menu 148
 Bottom to top 438
 BUILDING, symbol folder 226
 Busbar symbols 724
 Busbar, symbol type 707
 Busses 437

C

Cable list, page function 497
 Cable symbols 218
 accessories for cables 604
 add wire to 223
 color codes 221
 create 722
 implicit references on cable wires 213
 invisible 225
 place in one operation 222
 remove wire from a cable 223

reverse direction	224	Squared grid	792
same cable in more places	224	squared grid-10	792
Cables		Color codes for cables	221
move		Columns	
connection points	222	change column of data fields	765
Cables lists		lists in columns	546
create	760	COM/OLE automation	7
data fields	111	Combined circuit and connection diagram	95
list setup	553	Command bar	39
Calculate scale	253	Command toolbar	873
Cancel button	25	Company and User name	849
Capital letters	127	Company logo	
Cascade	516	on drawing header	751
CD		remove	764
component vendors	8	Component	
Change		insert in database	591
between projects	520	Component databases	
between windows	25	how does the database work	572
column of data fields	765	Component Grouping	
created drawing header	763	and copying	562
data field	764	menu	557
free texts	764	Component vendors	
name format types	384	CD with databases	8
symbol in folder	731	Components	
symbol names in Area	185	approved and obsolete in Database menu	614
symbol names in Area, naming order	186	changing symbol data for Component	242
symbol type	731	create in database	589
text	110	from existing project	590
text in the cross hair	116	fetching component symbols from database	230
Chapters	491	find in database	576
chapter tabs collapsed	491	functions on component level	228
Character set	787	grouping	556
choose	850	having several electrical symbols	593
Chinese version		insert new names	295
PC SCHEMATIC Viewer	680	navigation between symbols	170
Choose language	850	place available symbols	242
Circles	53, 100	rename selected	289
align	66	direction for renaming	292
convert to lines	101	start number	293
full	100	which names are renamed	290
half	100	rrename	
invisible	70	when copying areas	245
quater	100	sorting order	558
space	67	survey of symbols with same Symbol Name	238
toolbar	100	Components list, page function	497
Circles button	100	Components lists	111
Click	25, 873	create	759
Clipboard		files	649
copy to	523	Conducting (electrical) lines	78
Close, symbol status	594	Conducting lines button	78
Close, symbol type	706	Connection data	187
Codepage	664	Connection names	598, 873
Color	124, 792	in one line diagrams	408
background	792	in single line diagrams	408
dotted grid	792	insert new	295
help objects	793	Connection point button	714
lines	76	Connection point texts	109
object selection	793	Connection points	187, 873

choosing in areas	189	Cross hairs line displayed right angled	837
edit next/previous	188	Ctrl key	25
place	699	Current page	327
Connections lists	202	Current path naming	379
to file		Current path numbers	
with Line data	658	default settings	380
wire numbers	404	in symbol names	382
Contact mirrors		setup	379
about	375	Curved lines	74
and changing article numbers	375	Curved lines button	74
choose	377	Czech version	
created automatically by the program	377	PC SCHEMATIC Viewer	680
creating without using database	377	D	
get available symbols in the cross hairs via	375	Damaged files	849
jump to reference from	368	Data	
jump to the other symbols for the component	375	transfer between objects	62
Continuous	380	Data field dialog box	110
Coordinate field	472	Data fields	110, 874
Copy		and PLC lists	761
and component grouping	562	change	764
area form one project to another	518	change column of	765
areas between projects	255	copy rows of	756
areas intelligently	245	create symbol data fields	733
reference designations	247	insert	747
drawing headers	521	missing	534
project into another	513	page/project data	689
rows of data fields	756	reference designations	363
selected objects	58	types	746
symbol names when copying	384	Data symbols	
to other programs	523	and lists	736
Copy button	59	create	732
Copy properties button	63	place in project	734
Correct		with predefined values	736
directories or alias	842	Data, symbol type	707, 734
Unit drawing/parts list	770	DATABASE	845
Counting		Database	847
on texts	120	1. and 2. Searchkey	578
on texts in areas	120	1.Searchkey	577
Counting direction	380	CD from component vendors	8
Counting direction for naming	186	choose a DB alias in PC SCHEMATIC	623
Create		choose database menu	617
cable list	760	choose Menu using other DB programs	619
components list	759	choosing a database	623
diagram page w. list	762	create component	589
drawing headers	742	from existing project	590
lists	752	Database menu	
PLC lists	760	approved and obsolete components	614
signal symbol	738	Product choice or Fabricate choice	613
symbol	696	editing records	582
Table of Contents	759	enter using the pickmenu	142
terminals lists	759	fields	
unit drawing	767	file names in	622
Criteria, tab	549	links	622
Cross hairs	47, 874	find components	576
object selection	793	for lines	571
Cross hairs / Small cross hairs	836	how does the database work	572
Cross hairs figure with snap	837	insert component in database	591
		interface	608

- mechanical load 628
- place cable using color codes from database .
221
- place symbol in pickmenu using database 181
- programmable barcode scanners 624
- searching in the database 575
- settings 608
- transfer information from 573
- translate fields 608
- update symbols from 171, 231, 848
- use 571
- Database setup 847
- Datatemplates
 - create variablelist 283
- DDE functions 823
- Default vaule
 - for variable 871
- Default, tab 381
- Delete
 - page with a list on 766
 - selected objects 59
 - symbol 173
 - symbol from folder 731
 - templates 777
- Delete button 59
- DEMOMEK, symbol folder 226
- deselect wirenumbering 390
- Design
 - cable list 760
 - components list 759
 - data symbol 733
 - diagram page w. list 762
 - lists 752
 - PLC lists 760
 - signal symbol 738
 - symbols 696
 - Table of Contents 759
 - Terminals lists 759
 - unit drawings 767
- Design Check
 - dialog box 323
 - double-click on error in list 323
 - save and print 323
 - update design check list 323
- Designcheck 320
- Diagram
 - print 326
- Diagram, page type 508
- Diagrams
 - loop diagrams 7
- Dialog boxes 24
 - selecting elements in lists 24
- Dialogue 8
- Dictionary 873
- Different types of objects, select 244
- Dimmed 24, 874
- Direct access
 - to menus 48

- Directories 840, 874
 - most used 841
- Directory alias
 - show 841
- Displayed in project 127
- Distance (mm) 380
- Distance, lines 76
- Divider, page function 498
- Dock 874
 - a window 27
 - Explorer window 26
 - Symbol pickmenu 27
 - Toolbars 28
- Document object =>
 - Edit 671
 - Open 671, 672
 - Properties 672
- Dotted grid color 792
- Double-click 25, 874
- Drag
 - pages between projects 517
- Drag and drop 26, 874
- Draw
 - house walls 473
 - inner walls 474
- Draw button 72
- Drawing
 - by measures 93
 - measurement objects 641
 - print 326
 - standard projects 511
 - unambiguous 392
 - with elevation 482
- Drawing headers 835
 - add to list of drawing headers 750
 - and copying 521
 - and layers 480
 - change already created 763
 - company logo on 751
 - create 742
 - insert 531
 - predefined 751
 - remove form list of drawing headers 750
 - replace 535
- Drawing objects 53, 875
- Drawing templates
 - insert variables 281
 - insertion points 279
- Drive 875
- Drop down menu 24
- Dutch version
 - PC|SCHEMATIC Viewer 680
- DWG files 660
 - calculate scale 253
 - export 663
 - export project as 663
 - export symbol as 665
 - import 660

import as symbol	663	lines changign elevation	482
open	661	select elevated objects	482
open new project by importing	662	Elevation window	875
DXF files	660	Ellipses	101
calculate scale	253	Empty lines between Terminal Rows	553
export	663	EN1861	226
export project as	663	EN60204-1,14.2.1, standard	246
export symbol as	665	EN60439-3	469
import	660	EN60617, symbol folder	226
import as symbol	663	EN61346	335
open	661	signal names according to	210
open new project by importing	662	Enter Wire Numbers Manually	399
E		Error	
Edit		general protection error	49
templates	777	Even pages	327
Edit =>		Excel	
Align	66	integration with, survey	666
Convert to lines	101	place symbols via lists	674
Copy to Clipboard	523	PLC I/O addressing via	435
Data tags	283	Explorer window	301, 839
Find	316	displaying and hiding information	302
Find next	317	move	26
Insert special	670	Selecting a symbol on a page	304
Join signal	437, 438	Export	
Links	673	DWG and DXF files	663
Next	188	PLC I/O data to Excel	435
Paste	255	project as DWG or DXF file	663
Pinswop	191	symbol as DWG or DXF file	665
Previous	188	Extend line button	90
Replace	318	Extended lines	90
Scale	298	Extensions	
Select all =>		of list files	658
All on layer	481	F	
All on page	57	FABRICATE (Show all)	576
Space	67, 631	Fabricate choice	
Variables	261	in Database menu	613
Without wire number	390	Factor, ellipses	101
Editing symbols for a building	476	Favorites	786
EIB, symbol folder	226	Field headings	655
Electrical installation	469	File =>	
Electrical installations		Close	518
in different elevations	475	Close and return to PC schematic project	671
Electrical lines	78	Export to =>	
Electrical potentials	203	DWG / DXF	663
automatic insertion of	81	PDF	331
on lower part of page	82	Modules	782
on upper part of page	82	New	38, 512, 772
settings for	83	New =>	
Electrical symbols		PC schematic project file	814
components having more	593	Open Unit	770
create automatically	199	Print	326
box symbols	200	Print =>	
circle symbols	200	Select pages =>	481
connection points	200	Save unit as	769
from database	201	Send to	490
place based on placed mechanical symbols	639	File format	655
Elements of a project	489	File type registration, dialog box	813
Elevation	482, 875	Files	875

- commasep. files
 - place symbols via 674
- DWG/DXF 660
- file names
 - in database fields 622
- from Object Lister 311
- lists as files 649
- pindata files 599
- registering PC|SCHEMATIC filtypes 813
- wire number 401
- Files =>
 - Modules =>
 - Scripts 7
- Fill character 111
- Filled areas 75
- Find Next 317
- Find Whole Text Only 317
- Fixed digits 395
- Fixed text direction 125
- Flat buttons 838
- FLOWCHAR, symbol folder 226
- Folders 875
 - change symbol in 731
 - delete symbol from 731
 - most used 841
 - symbol folders 225
- Follow connected button 91, 739
- Follow connected lines 90
- Font 123
- Foreign languages 787
- Format file 650
- Formulas
 - insert 531
 - projects 511
- Frame 124
- Free texts 102, 875
 - change 764
- Ftp
 - links in database fields 622
- Full circle 100
- Function aspects 335, 338
- Functions =>
 - Auto command change 57
 - Auto Name 176
 - Centre drawing 510
 - Component Grouping 216, 557
 - Coordinates 93
 - Count down 121
 - Count up 121
 - Database =>
 - Find Article 154
 - Find Type 155
 - Insert in Database 591
 - Update from Database 171
 - Design Check 320, 323
 - Design Check =>
 - Used more than once
 - deselect 324
 - Generate isometry 510

- Measurement 641
- Mechanical load 629
- Object Lister 307
- Place Accessory 225
- Place available Symbols 242
- Replace symbol 193
- Select name 238
- Show available 232
- Show available again 153, 232
- Show conducting 79
- Show signal 209
- Special Functions =>
 - Convert Branches to Dots 98
 - Convert Dots to Branches 98
 - Convert to Branches 99
 - Create CurrentPath No on all Symb 383
 - Exchange Article with Alt. Article on all Symbols
611
 - Prefix all symbol names with - 352, 838
 - Set all articles as approved in the database 617
 - Set unreferenced cable symbols invisible .225
 - Set unreferenced signal symbols invisible .210
 - Transfer data from PLCref. to PLCsymb. .430
 - Transfer data from PLCsymb. to PLCref. .430
 - Transfer Proj. Texts to Auto Compl. func. .104
 - Update references 372, 381
 - View Jumper Links 87
 - Wire numbers 389

G

- General protection error 49
- General purpose texts 102
- General, tab 544
- Generation names 383
- German version
 - PC|SCHEMATIC Viewer 680
- Grafisk klemplan, værktøj 781
- Graphical terminal plans, tool
 - Tools
 - Grappical terminal plans 781
- Grid 47, 875
- Grid display 836
- Ground plan 471
 - place symbols on 475
 - with elevation 482
- Ground plan, page type 508
- Group Number 560
- Grouping
 - components 556
 - layers 479
- Guess
 - wire numbers 392

H

- Half circle 100
- Heat loss calculation
 - acc. to EN60439-3 469
- Height 123
- Help 35
- Help =>

Check for updates	851	Insert Picture	297
Http		Insert Potential	81
//www.pcschematic.com	18	Insert Reference Frame	354
Help frame	39, 839	Insert sub drawing	95
Help objects color	793	Insert new	494
Hints	838, 875	symbols or connection names	295
Home view		Insertion points	279
set User Home view	45	Install modules	782
HOMEDIR	845	Installation	
Horizontal reference	129	electrical installation	469
Horizontal, tab	380	pickmenu	475
Hot-line	8	Installations	
House drawing	471	LV installations	7
Http		instdemo.pro, demo project	475
links in database fields	622	Intelligent	
Hungarian version		copying of areas	245
PC SCHEMATIC Viewer	680	reference designations	247
Hydraulics		Interface	
place symbols via lists	674	database	608
Hydraulics, symbols	226	Internet	
I		automatic information and update	784
I/O lists		Interval in minutes	849
read	432	Invert text	116
I/O symbols		Invisible	478
create PLC I/O symbols	725	areas	70
IEC750 standard	335	circles	70
IECINST, symbol folder	226	lines	70
IECPNEU, symbol folder	226	objects	69
Ignored, page function	497	Sub drawings	71
Image		symbols	69
insert	66	texts	70
scale	298	ISO	
Image functions	42	paper size	509
Import		Isometric, page type	508
DWG and DXF files	660	Isometrical drawing	510
as page in project	661	Italic text	125
PLC I/O data from Excel	436	J	
Include Additional from Database	545	JIS	
Include cable wires	391	paper size	509
Include Lines with article data	545	Join symbols	
Index		define	741
on-line manual	37	Jokers	550
Indexed variables	872	Jumper links	86
Indexing	816	in lists and terminal plans	87
Information		K	
automatic information	784	Keep page zoom	44, 838
Insert		Keep zoom	44, 838
data fields	747	Komponenter	
image	66	rename selected	
list	538	selecting symbols	291
logo	527	L	
page for mechanical drawing	628	Language	850
unit drawing	568	Layers	478
Insert - in front of symbol names	352, 838	and drawing headers	480
Insert =>		change using the mouse	480
Insert data field	110, 855	grouping layers	479
Insert object	668, 669		

- print page with 481
- Left tool bar 40
- Left toolbar 875
- License key 850
- Licenses
 - SuperPro monitor 799
 - tanking licenses 794
 - transfer licenses by tanking 795
 - transfer licenses via license files 796
- Lines 53, 72
 - article data 79
 - Automatic line drawing 90
 - automatic wire numbering 389
 - by measures 93
 - changing elevation 482
 - color 76
 - curved 74
 - distance 76
 - electrical 78
 - extended 90
 - follow connected lines 90
 - in symbols 699
 - invisible 70
 - jumper links 86
 - Object Lister 307
 - rectangles 75
 - round 74
 - router 90
 - select elevated 482
 - slanted 74
 - straight 73
 - toolbar 73
 - transfer article data 62
 - trim 92
 - type 75
 - types for ground plans 473
 - width 76
 - wire numbers file
 - with reference designations 402
 - without wire number, deselect 390
- Links
 - edit 673
 - in database fields 622
- List files
 - buttons in the dialog box 653
 - create 650
 - criteria 654
 - example 657
 - extensions 658
 - field headings 655
 - file format 655
 - matching the system you export to 654
 - reference designations in 656
 - the contents 654
 - the file name 654
 - the format file 650
- List of drawing headers 750
- List of used objects 316

- List setup
 - terminals- cables- and PLC lists 553
- List symbols 762
- Lists
 - add to the list of drawing headers 758
 - as files 649
 - columns 546
 - company logo on 751
 - connections lists 202
 - wire numbers 404
 - create 752
 - criteria 549
 - data symbols on 736
 - delete page with list on 766
 - diagram page w. list 762
 - for each Higher level or Location 364
 - insert 538
 - jumper links in 87
 - list symbols 762
 - missing data fields 534
 - mounting lists 658
 - order in 556, 561
 - place symbols via lists 674
 - predefined 763
 - references to location of mechanical symbols .
636
 - remove form page 766
 - settings 542
 - symbols
 - sub names in lists 166
 - unit parts lists 767
 - update 541
 - using reference designations 364
 - XML format 652
- Lists =>
 - Connections List File 202
 - Connectionslist for file 403
 - List Settings 555
 - List settings 543, 544, 553
 - Parts List File 650, 657
 - Read Components List =>
 - Create from diagram page 677
 - Create from mechanical page 677
 - Load from file 674
 - Update all lists 541
 - Update lists 543
 - Update parts lists 541
 - Update Table of Contents 497
 - Update Unit Drawing 769
 - Wire number file 401
- Load defaults button 380
- Load previous project 850
- Load selected 629
- Load standard button 128
- Location aspects 335, 338
- Lock
 - after system break down 806
 - delete lock information manually 806

project files	806	MISC, symbol folder	226
project in use	806	Missing data fields	534
text types	119	Modules	781
Logo		install and uninstall	782
insert	527	start a module	783
loop diagrams	7	update and re-install	783
LV installations	7	which can you install	781
M		Monitors	
Mailto		PC with two monitors	26
links in database fields	622	Mounting correct drawing	95
Main reference, check box	380	activate	96
Manual	35	bend	96
Survey of	15	convert from previous verisons	99
Manual placing of wire numbers	398	convert to automatically	98
Mapping		Mounting lists	658
database fields	608	Mounting mode	392
Margin	329	Mouse	
Master reference, symbol type	707	change layer	480
Match Case	317	Favorites	786
Maximize	875	mouse pointer	26
MCS	800	Mouse Chasing System	8, 800
MDAC	623	automatic blocking of mouse clicks	802
Measurement	641	block for clicking on a function	801
free measurement	643	disable	802
reference points and measurement	643	Mouse pointer	876
rotate	643	Move	
Measurement objects		Explorer window	26
layers	645	origo	471
Measures		reference points	745
absolute	93	selected objects	58
polar	93	Survey window	27
relative	93	symbol	174
Measures, drawing by	93	Symbol names when move	384
Mechanical drawing		Symbol pickmenu	27
insert page	628	Toolbars	28
Mechanical load		Move button	58
from database	628	MSSQL	623
references to mechanical symbols in lists	636	Multi layer terminals	215
Mechanical symbols	201, 603, 612	on diagram pages	216
create automatically	199, 201	on mechanical pages	217
box symbols	200	Multi symbols	714
circle symbols	200	fetch	715
connection points	200	Multi-level terminals	600
from database	201	Multisignal, symbol type	707
start by drawing mechanically	639	N	
MECTYPE	593	Name format	382, 852
Menu bar	23, 38, 876	Name format types	
Menus	23, 876	change	384
choose database menu	617	Native drivers	8
direct access	48	Navigation	
Microsoft Excel		between symbols for component	170
integration with, survey	666	Negate	116
Microsoft SQL Server	8	Negate text	116
Minimize	876	NET	845
Mirror		Net	
symbols	176	tanking licenses	794
Mirror symbol horizontally button	176	Net router	837
Mirror symbol vertically button	176		

Networks
 sub drawings in 256
 New document button 38
 New versions 8
 No dot generation 719
 No visible 478
 Non conducting symbol 876
 Non conducting, symbol type 707
 Normal, page function 497
 Normal, symbol status 594
 Normal, symbol type 706
 Norms
 page norms 509
 Not yet written (New) 401
 NP 876
 Number 380
 Numbering order 393

O

Object Lister 316
 check project texts 314
 counting on selected objects 310
 editing PLC I/O data 313
 print 316
 printing from 311
 save as file 311
 selecting objects in 309
 show used and count 313
 typing in PLC I/O data 426
 Object properties 63
 change on existing object 63
 transfer to more objects 63
 Object selection
 Cross hairs 793
 Object selection color 793
 Objects 53, 876
 align 66
 already placed 54
 invisible 69
 new/draw/place 54
 select different types of 244
 select elevated 482
 space 67
 survey 316
 transfer data between 62
 Obsolete components
 in Database menu 614
 ODBC interface 8, 623
 Odd pages 327
 OK button 25
 OLE automation 7
 OLE objects
 edit links 673
 properties 672
 On all pages 536
 On this page 536
 One line diagrams
 connection names 408

On-line manual 35
 adapt to screen 36
 buttons 36
 index 37
 index-finger 37
 open 35
 printing from 37
 return to PCschematic 37
 Table of Contents 36
 zoom 36
 Only jumper links 545
 Only show text on data 111
 Open
 DWG and DXF files 661
 new project by clicking symbol 524
 Open dialog box 514
 Open files from Tools menu 679
 Open, symbol status 594
 Open, symbol type 706
 Operator 549
 Oracle 8, 623
 Order
 in lists 556, 561
 Origo
 move 471
 Ortho 45 837
 Other programs
 copy to 523
 Output 326
 Overall planning 685
 Oversigtsprojekter 524
 Overview project 524
 Overview projects 524

P

Page
 page setup 506
 transfer to other pages 506
 Page data 834
 data fields 110, 689
 add 692
 in standard project 687
 type in 114
 Page Data, data field type 746
 Page format 509
 Page function 843
 Page history buttons 492
 Page menu 493
 copying pages 500
 naming 500
 symbol names when changes 385
 Page numbering
 Automatic renumbering of pages 832
 Page settings 38, 471
 Page setup 843
 Page shift between Terminal Rows 553
 Page tabs 838, 876
 Page type 843

Page types	508	Place missing	393
Page zoom		Place missing and renumber	393
keep	44	Planning	685
Pages	491	PLC	418
copy	500	addressing directly from database	419
create new	495	base addressing	419
create page templates	772	choose format file for I/O file	432
drag between projects	517	contents of I/O file	433
insert drawing header	532	create PLC I/O symbols	725
Object Lister	307	create PLC ref. symbols	728
page history	492	Data on I/O symbol connection points	425
Page menu	493	displaying changes in project acc. to I/O file	434
reference designations	353	I/O addressing via Excel	435
displaying reference designation on page	354	I/O data	
remove list form	766	edit	313
Pages and layers	545	jump PLC ref. symb. / PLC I/O symb.	427
Panel Builder	469	PLC reference symbols	418
Panel Builder, tool	782	more than one	419
Panel designer	469	read I/O lists	432
Panel documentation		re-addressing	423
automatic	469	specify I/O file filename	433
panel documentation		survey over functions	417
automatic	469	symbol folder	227
Panels	469	symbol status	594
Paper and margin setup	329	symbol type	707
Paper size	509, 843	symbols	227
Parking	79, 876	synchronize PLC data	430
Parts list, page function	497	text links	136
Parts lists	111	typing in I/O data via Object Lister	426
files	649	update PLC projects from previous versions	430
order in	561	workflow	
prices	613	for PLC's not in the database	428
Parts/Components list, data field type	746	with PLC's from database	420
Password		working with PLC's	418
projects	805	PLC =>	
Path	876	PLC List File	435
PC		Read PLC I/O List	432
with two monitors	26	PLC functions	
PC SCHEMATIC		Apply addresses	424
registering filetypes	813	PLC I/O	
PC SCHEMATIC Loop	7	Object Lister	307
PC SCHEMATIC Powerdistribution	7	PLC list, data field type	746
PCSTYPE	593	PLC list, page function	497
Pencil button	72	PLC lists	
Pickmenu	39, 140	and data fields	761
attach data to symbol in pickmenu	180	create	760
enter database from pickmenu	142	data fields	111
for installations and pneumatics	475	list setup	553
place symbol in pickmenu using database	181	PLCreference	
place symbol with data	178	symbol type	707
place wire number symbol in	398	Pneumatics	
Pin names	612	pickmenu	475
PINDATA	593	Pneumatics symbols	226
Pindata	598	connection points with same name	324
Pindata files	599	place symbols via lists	674
Place		Pointer	836
available Symbols	242	Pointer/Screen	836
symbols	156	Polar	93

Polish characters	787, 850	controlling texts from project definition file	281
Polish version		copy areas between	518
PC SCHEMATIC Viewer	680	copy project into project	513
Position of name when placing symbols	131	copying areas between	255
Posno1	769	create project templates	774
Potential numbering	391	from existing project	776
Potentials	203	creating project templates	686
automatic insertion of	81	drag pages between projects	517
on lower part of page	82	elements of	489
on upper part of page	82	export as DWG or DXF file	663
settings for	83	insert logo	527
Powerdistribution	7	insert sub drawing	259
Precede page number	395	jump between several projects	305
Predefined drawing headers	751	lock file	806
Predefined lists	763	lock project in use	806
Prefix	395	open from symbol	521
Preparations	685	overview project / click symbols to open projects	524
Previous versions		524	
update PLC projects from	430	overview projects	524
Price 1	611	password	805
Primary	546	planning	685
Print	326	same project in two versions	522
batch printing	823	searching for texts via Windows	818
on net installation	823	send by e-mail	490
on stand-alone installation	823	simultaneous projects	516
chapters and dividers	328	survey of	522
from Object Lister	311	zip	825
object lists	316	Projekter	
page with layers	481	oversigtsprojekter	524
zoom	329	Properties =>	
Print pages button	326	Project data	814
Print test page	330	Protected	399
Print this page button	326		
Printer help frame	839	Q	
Printer, tab	330	Qualified guess	392
Product choice		Quarter circle	100
in Database menu	613	Question mark	550
Program settings	38		
Program toolbar	39, 877	R	
Programmable barcode scanners	624	Re-addressing	
Programs		PLCs	423
copy to other	523	Reading direction	130
Project data	831	Recovery information	849
data fields	110, 689	Re-creating files	849
add	692	Rectangle button	75, 473
in standard project	687	Rectangles	75
type in	114	with relative coordinates	94
Project Data, data field type	746	Red arrow	111
Project definition file		Redraw button	45
controlling project texts from	281	Reducing repetitive strain injuries	800
Project menu		Reference cross	380
Windows Explorer as	334	Reference crosses	129
Project oriented program	489	about	376
Project settings	38	choose	377
Project templates		jump to reference from	368
create	686	Reference designations	335
Projects		data field for specifying	363
about	489	defining	338
compressed saving	825		

displaying on page	354	drawing header	535
for a page	353	Replace All	318
in list files	656	Replace symbol texts	318
insert a data field	354	Replace symbols	193
insert auto. on signal symbols	210	connection points	195
prefix all symbol names with -	352	text properties	195
symbol names on pages with assigned	353	Reports	
symbols	349	loop reports	7
wire numbers file		Reset group numbers button	558
with reference designations	402	Restart on page shift	395
Reference frames		Restarting the program	49
Object Lister	307	Reverse direction	224
Reference letter	612	Right-click	26, 877
Reference point button	739	select	55
Reference points	839, 877	short-cuts	789
move	699, 745	symbols	176
selection	757	Rotate	
Reference symbol position	380	areas	253
Reference symbols		selected object	59
PLC reference symbols	418	texts	124
Reference system		Rotate button	59
default settings	380	Round buttons	24
setup	379	Round lines	74
Reference system setup, dialog box	379	Round lines button	74
Reference text	207	Router	89
Reference, symbol type	707	Router =>	
References	835	Active	89
appearance	372	Rows	
automatic update of ref.	367	copy rows of data fields	756
implicit references on cable wires	213	Russian characters	787, 850
implicit references on terminals	211	Russian version	
jump to ref. from ref. cross/contact mirror	368	PC SCHEMATIC Viewer	680
jump to reference	367	S	
on symbols	369	S Article no	107
symbol type and reference	370	S Function	107
Refresh	45	S Name	107
Register command	824	S Type	107
Registration		Same number through terminals	391
of PC SCHEMATIC Automation	824	Save	
of PC SCHEMATIC filetypes	813	symbol	705
Re-install modules	783	Save as default button	128
Relative	93	Save with backup	840
Relay, symbol status	594	Scale	
Relay, symbol type	706	calculate scale	253
Remove		image	298
company logo	764	scale factor	844
list from page	766	symbols	176, 708
Remove blocking	802	texts in areas	254
Remove existing	394	Screen functions	42
Remove existing and replace	394	Scripts	
Rename	479	module	782
selected symbols	289, 290	Scroll	877
symbols from Symbol menu	710	Search	
Renumber existing	394	in database	575
Repetitive strain injuries		Searchkeys	
block for clicking on a function	801	in database	577
Repetitive strain injuries, reducing	8, 800	Secondary	546
Replace			

Select	
area in all layers	481
different types of objects	244
elements in lists	24
elevated objects	482
everything on a layer	481
everything on a page	57
more objects of same kind	55
object on different layer	481
objects	55
on different layers	481
right-click	55
several objects using mouse	56
symbol by right-click	176
symbol from database	152
symbol from pickmenu	140
symbol from the symbol menu	144
symbol using k-key	153
Select all layers	481
Select Component	242
Select database file	847
Select pages, tab	327
Select set of characters	850
Selected objects	
copy	58
delete	59
move	58
rotate	59
working with	58
Selection	
reference points	757
Semi ISOmetric, page type	508
Semicolon	598
Send project by e-mail	490
Servers	850
Settings	38, 829
data base	608
lists	542
Settings =>	
Cursor/Screen =>	
Colors	792
Database =>	847
Database settings =>	
Component data	577
Database setup =>	608
Component data	577
Component search	614
Select database file	623
Directories	522, 840
Directories =>	
Database	599
Page data =>	114, 834
Reference designations	338, 353
References	379, 636
With drawing header	532
Page setup	506, 843
grid	47
page function	498
page type	508
snap	47
Page setup =>	
Reading direction	130
Pointer/Screen	44, 107, 836
Ask for names	157
grid	47
Pointer/Screen =>	
Shortcuts	790
Project data	831
Project data =>	114
Password	805
Reference designations	338
Settings	829
System	849
System =>	
Compressed saving	825
File type registration	813
License Tank	795
SuperPro monitor	799
Text/Symbol defaults	852
Drawing headers	750
References	372
Signal name	741
Signal symbols	740
symbol data fields	733
Text defaults	382
Tools	845
Settings => Page setup	
Paper size	509
Setup, tab	329
Several projects	
jump between projects	305
Short-cut keys	788
learning while working	802
user defined	790
Show	
symbol texts	705
Show all	207
Show available	232
Show available window	234
Show defaults button	128, 380
Show directory alias	841
Show from/to	207
Show layer settings	328
Show list of objects	341
Show message in (ms)	837
Show net	202
Show popup help	838
Show popup information	838
Show scrollbars	838
Show used	315
Show used and count	313
Signal buses	437
Signal names	
in single line diagrams	407
Signal symbols	203, 877
create	738
frequently used	740
frequently used names of	741

insert reference designations automatically	210	Straight lines	73
invisible	210	Straight lines button	73
place	204	Straighten lines	837
place directly	208	Strategy	685
place on parked line	208	Sub drawings	256
variant names	209	automatic placing	264
Signals		create	267
Object Lister	307	by copying	268
Simultaneous		create variablelist	283
projects	516	insert on project page	259
Single line diagrams	407	insert variables	281
connection names	408	insertion points	279
signal names	407	invisible areas	71
symbol names	408	networks	256
symbols with states	408	Sub names	
Single line symbols		on symbols	165
connection names for	598	SuperPro monitor	799
Slanted lines	74	Support	8
Slanted lines button	74	Support, symbol type	707
Snap	47, 877	Survey	
Snap button	47	of projects	522
Sort by name	68, 631	of the manual	15
Sorting order		of used objects	316
of components	558	Survey window	40, 839, 878
Space	67	move/dock	27
Spares pr. Page	395	Switch, symbol status	594
Special characters	105, 116	Switch, symbol type	706
Specify character set	787	Symbol article data =>	
Specify symbol status	603	Unit	568
SQL Server	8	Symbol article data dialog box	107
Squared grid color	792	Symbol Data Field, data field type	746
Squared grid-10 color	792	Symbol Data Fields	
Squares	24	data fields	111
Standard characters	787	Symbol defaults	852
Standard projects	511	Symbol definition	598
convert to templates	776	Symbol documentation, tool	781
copy into project	513	Symbol folders	225
Standard/Project file	514	Symbol manipulator	716
Standard/Project file, page function	498	Symbol menu	
Standards		bookmarks	148
EN1861	226	create symbol / edit existing	151
EN60204-1,14.2.1	246	enter the database from	151
EN60617	226	representation of symbols	146
EN61346	226, 335	search in	147
EN81714-2	226	select library via alias	146
IEC/EN60617	226	select library via folders	146
IEC750	335	show project symbols	146
ISO1219	226	Windows Explorer as	334
page standards	509	Symbol pickmenu	39, 839
Start number	395	dock/move	27
Start value	380	Symbol reference	129
Starting functions	821	Symbol settings button	739
load project with page open	821	Symbol texts	
load project with symbol selected	821	replace	318
loading specific pages from project	821	Symbol toolbar	
States		select symbol type in area on screen	176
symbols with states	167	Symbols	53, 137
Status bar	877	accessory symbols	225

align	66	names regain adaptability	385
alternative electrical symbols	594	names when changes in Page menu	385
alternative symbols		names when move/copy	384
PINDATA	596	navigation between	170
alternative symbols for PLC	595	Object Lister	307
attach unit drawing to	568	open project by clicking symbol	524
automatic line drawing/router	89	open project from	521
busbar symbols	724	pickmenu	
cable symbols	218	attach data to symbol in pickmenu	180
invisible	225	enter database using pickmenu	142
cables with color codes	221	place symbol in pickmenu using database	181
change in folder	731	place symbol with data	178
change names in Area	185	symbols with states	143
change names in Area, naming order	186	place and name	156
change type	731	place available symbols	242
changing symbol data for Component	242	place on ground plan	475
component having several electrical	593	place via lists	674
component level functions	228	PLC	
connection names for single line symbols	598	create PLC I/O symbols	725
connection points	187	create PLC ref. symbols	728
create	696	PLC symbols, working with	418
default text settings	716	position of name when placing symbols	131
create automatically	199	reference designations	349
box symbols	200	references on	369
circle symbols	200	rename	
connection points	200	when copying areas	245
from database	201	rename from Symbol menu	710
create data symbol	732	rename selected	289
create symbol data fields	733	direction for renaming	292
current path numbers	382	selecting symbols	291
data symbols		start number	293
with predefined values	736	terminals	293
data symbols and lists	736	which names are renamed	290
Database		replace symbols	193
update symbols from	848	save	705
delete	173	scale	176, 708
delete from folder	731	select	
Edit symbol mode		from database	152
default text settings	716	from pickmenu	140
editing for a building	476	from symbol menu	144
enter database using pickmenu	142	right-click	176
export as DWG or DXF file	665	using k-key	153
fetching symbols for component from database	230	select elevated	482
insert new symbol names	295	select symbol type in area on screen	176
invisible	69	select using Explorer window	304
join symbols, define	741	show available	232
lines in	699	show net	202
list symbols	762	show texts	705
load another symbol into	716	signal busses	437
mechanical	201, 603	signal symbols	203
create automatically	201	invisible	210
mirror	176	space	67
move	174	specify symbol status	603
move reference points	699	states	167
multi symbols	714	in single line diagrams	408
name format	382	sub names	165
names on pages with reference designations	353	functionally understood as	165
		included in lists	166
		specified from database	166
		specify	165

what is	165	Terminal plans	
survey of symbols with same Symbol Name	238	jumper links in	87
symbol documentation	781	Terminal symbols	214
symbol manipulator	716	create	717
symbol names		implicit references on terminals	211
in single line diagrams	408	Terminal, symbol status	594
position of name when placing symbols	131	Terminal, symbol type	707
symbol type and reference	370	Terminals	
terminals		graphical terminal plans, tool	781
multi layer	215	multi layer	215
on diagram pages	216	on diagram pages	216
on mechanical pages	217	on mechanical pages	217
texts connected to	107	multi-level terminals	600
toolbar	176	rename	293
transfer article data	62	Terminals list, data field type	111, 746
type in Article Number directly	154	Terminals list, page function	497
type in Symbol Type directly	155	Terminals lists	
types	706	create	759
update		list setup	553
from database	171	order in	561
when changing article number	171	Test page, print	330
update from database	231	Text	111
wire number	395	Text defaults	852
with extra connection points	715	Text fields	
Symbols button	139	place	745
Symbols text		Text link	
S Article no	107	dialog box	135
S Function	107	Text links	132
S name	107	create from Symbols command	133
S type	107	create from Texts command	132
SYSDIR	845	delete	136
System	849	edit	135
System break down		in PLC I/O lists	136
lock after	806	indication of linked texts	132
System breakdown	849	what is a text link	132
System Data		Text properties	
data fields	110	specify	122
System Data, data field type	746	Text properties button	122
T		Text translator, tool	782
Tab key	25	Text/Symbol defaults	852
Table of Contents		Texts	53, 102
create	759	align	66
data fields	111	alignment	124
On-line manual	36	appearance	122
Table of Contents, data field type	746	auto completion	103
Table of Contents, page function	497	auto text	
Templates	771	delete texts	104
create from standard projects	776	bold	125
delete	777	change	110
edit	777	change free texts	764
insertion points	279	change in cross hair	116
on networks	771	check project texts	314
page templates		connected to symbol	107
create	772	controlling from project definition file	281
project templates		counting on texts	120
create	774	counting on texts in areas	120
from existing project	776	data fields	110
template projects	511	reference designations	363

- default text settings
 - in Edit symbol mode 716
- displayed in the project (overall setting) .. 127
- fixed text direction 125
- frame 124
- free texts
 - default properties 106
- height 123
- in more than one line 116
- invert/negate 116
- invisible 70
- italics 125
- linking texts 132
- lock text types 119
- Object Lister 307
- rotate 124
- scaling texts in areas 254
- searching for texts via Windows 818
- space 67
- special characters 116
- specify text properties 122
- text links 132
 - create from Symbols command 133
 - create from Texts command 132
 - delete 136
 - edit 135
 - in PLC I/O lists 136
 - indication of linked texts 132
 - what is a text link 132
- text translator, tool 782
- toolbar 103
- type in 103
- underline 117
- underlined 125
- visible text 125
- width 123
- Texts button 102
- Tile horizontal 516
- Tile vertical 516
- Time-out 850
- Toolbars
 - arcs/circles 100
 - dock/move 28
 - lines 73
 - symbols 176
 - texts 103
- Tools => 845
 - Convert files from ver. 11 or older 12
 - Create variable list 281, 282, 283
 - Database 608
 - Mouse Chasing System 800
 - Rename symbols 289
 - Scripts 7
 - Show textfile 679
 - Singleline diagram 409
- Tools menu
 - open files from 679
- Top to bottom 438
- Transfer Group Number button 560

- Transfer information
 - from database 573
- Translate
 - database fields 608
- Translator
 - text translator tool 782
- Trim line 92
- Type in texts 103
- Types
 - line types 75
 - symbol types 706

U

- Unambiguous drawing 392
- Underline 117
- Underline text 117
- Underlined text 125
- Undo 60
- Undo button 60
- Uninstall modules 782
- Unit Drawing, page function 497
- Unit drawings
 - attach to symbol 568
 - correct 770
 - create 767
 - create using drawing of entire component 769
 - insert 568
 - open 522
 - save 769
- Unit of measurement 837
- Unit parts lists 767
 - assign article number to 768
 - correct 770
 - without unit drawings 769
- Unregister command 824
- Update 8
 - automatic information 784
 - modules 783
- Update lists 541
- Update, button 381
- User defined short-cut keys 790
- User Home view
 - set 45
- User name 849
- USERLIB, symbol folder 227

V

- Variables 871
 - create variablelist 283
 - indexed variables 872
 - insert 281
 - insertion with default value 871
 - normal variable 871
 - possible control via var. in symbol names 872
 - that inserts a value in a text 871
 - with default value 871
- Variant names
 - on signal symbols 209

Vertical reference	129	place symbol in pickmenu	398
Vertical, tab	380	place symbol manually	399
View =>		Wire numbers to file	
Activate Pickmenu	32	with reference designations	402
Clean screen	46	With reference	206
Go to page	491	With reference, symbol status	594
Goto symbol	170	With reference, symbol type	707
Home view	44	Without wire number	390
Page history back	492	Word	523
Page history forward	492	Work area	38, 40, 878
Previous zoom	44	Wrap text	111
Redraw	45	X	
Select page	30, 358, 493	X position (mm)	380
Set User Home view	45	x,y position	636
Show object below cursor	60	XML	
Show texts	705	lists in XML format	652
Survey window	46	Y	
Zoom	43	Y position (mm)	380
Zoom All	43	Z	
DWG/DXF	660	Zip	
Zoom in	43	advantages by zipping	825
Zoom out	43	on zipping	825
Visible text	125	project files	825
W		Zoom	878
Width	111, 123	keep page zoom	44
Width, lines	76	previous zoom	44
Wildcards	550	print	329
Window		Zoom button	42
several windows open	520	Zoom to page button	44
Window =>			
Arrange Icons	516		
Close All	516		
Tile	517		
Tile horizontal	518		
Windows	23		
change	25		
indexing service	816		
Windows Explorer			
as Project menu	334		
as Symbol menu	334		
Wire			
add to cable	223		
remove from cable	223		
Wire number, symbol type	707		
Wire numbers			
auto. incr. on manually placed wire no's	399		
automatic	389		
connections lists	404		
Excl. signals, tab	397		
fetch symbol	398		
files	401		
Format	395		
Functions	391		
lines without wire numbers	390		
manual placing	398		
Number symbol, tab	395		
numbering order	393		