PC | Automation™

PART 1

April 1st 2020

TABLE OF CONTENTS

PART 1

PC|SCHEMATIC

PART 2

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 PCSCHEMATIC A/S.

In the same family of CAD programs you will find:

- PCSCHEMATIC **Panelrouter** for automatic routing of connections in mechanical layouts and for precise calculation of wire lengths. This is an independent software, which can run as an integrated part of the **Automation** software.
- PCSCHEMATIC **Cablemanager** for cable management and cable routing. Read more about how to export cable data from **Automation** to **Cablemanager** in "Export to PCSCHEMATIC Cablemanager" on page 748.
- PCSCHEMATIC **Loop** for automatic generation of loop diagrams based on information from a database or for instance from Excel

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

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

For version 22 and later versions: More component oriented

In future versions of the **Automation** software, the structure of the database will be changed. This will result in a number of changes. Some of these changes are implemented in version 22. In future, you will be working more component oriented, once you have selected components from the database.

In version 22 we have introduced some limitations on what can be changed for components selected from the database. "Symbol type 2" is no longer an option and the options for types of connection points are changed.

When you open a project from previous versions, the connection points are converted according to the new options. You can get an overview over how they are converted in the news document for version 22 under **Help => Program news**. In this document you also find a complete overview over the changes in version 22.

For version 18: New symbol libraries

All of the 60617 symbols were updated in version 18, so that text positions and sizes follow the IEC 60617 and IEC/ISO 81714-2 (Design of graphical symbols) standards, and the new S-no has been added in the descriptions. This implicates the symbols in the folders **EN60617**, **DB60617**, **IECINST** and **ALARM**.

All of these new versions of the symbols are placed in the folder **60617_2015**.

If you choose to *update* from a version older than version 18, the folders **EN60617**, **DB60617**, **IECINST** and **ALARM** remains with unchanged contents.

If you instead choose to make a *new installation*, the folders **EN60617**, **DB60617**, **IECINST** and **ALARM** are not created.

Get an overview over the symbol folders in "Symbol folders" on page 256.

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. — Read more about this in "Scripts" on page 865.

Symbols and databases

All symbols in the program are created by PCSCHEMATIC 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 PCSCHEMATIC 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 PCSCHEMATIC agent for detailed information.

PCSCHEMATIC 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: Gives you a quick introduction to **Automation**.

- The **Project Generator** manual: Describes how to generate projects automatically in **Automation**.

- The **Component Wizard** manual: Describes how to easily create components with attached symbols for the component database in **Automation**.

- The **Tools** manual: 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 **PCSCHEMATIC Oversætter** 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: Describes how to use the **PCSCHEMATIC Database** program for creating and maintaining your database for **Automation**.

- **Guide for using Automation Service and the Mounting Assistant**: Describes how to use PCSCHEMATIC Automation Service as well as how to use the **Mounting Assistant**, which is included in all variants of **Automation**.

- The **Panelrouter** manual: Describes how to use the **Panelrouter** in **Automation**.

- The **Administrator control** manual: Describes how to perform administrator control on **Automation**. Please contact PCSCHEMATIC A/S for further information.

Tutorials

Various tutorial manuals also exist, which guides you through how to use the program for various types of projects:

- **Motor control tutorial**: Guides you through creating a motor control project in **Automation**.

- **PLC project tutorial**: Guides you through creating a motor control project in **Automation** and introduces various advanced options for working with plcs.

- **House installation tutorial**: Guides you through creating a house installation project in **Automation**.

- **Panel Builder tutorial**: Guides you through creating a Panel Builder project in **Automation**. The Panel Builder supports you in automatically documenting small switchboards and panels according to requirements in EN61439-1 and 3 concerning power dissipation calculation and more.

- **Component Wizard tutorial**: Shows you how to easily create components with attached symbols in the database.

- **Clip Project Marking integration tutorial**: Guides you through how you can easily print labels in Clip Project Marking directly from **Automation**.

All of these manuals can be downloaded at www.pcschematic.com/manuals, where you can also see the news in the latest version of **Automation**.

Videos and articles at www.pcschematic.com

At www.youtube.com/pcschematic 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.

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/License**. 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 PCSCHEMATIC 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/License** on the **System** tab.

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 PCSCHEMATIC font and Unicode

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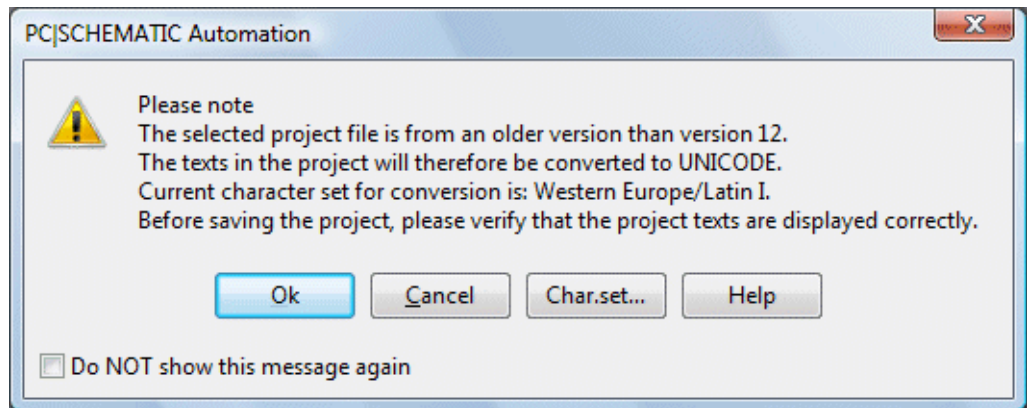
You can choose not to display the dialog box again by selecting **Do NOT show this message again** at the bottom of the dialog box.

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 selectred 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

PC|SCHEMATIC

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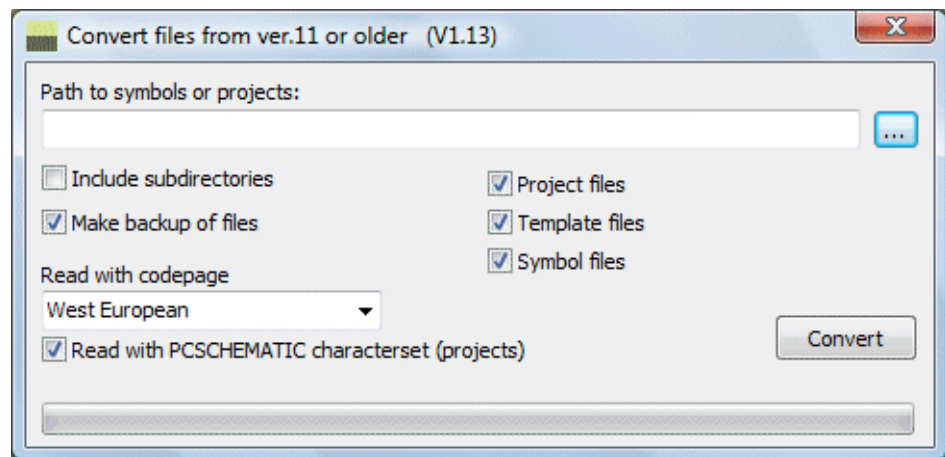
6) Select a folder and click **Convert** to convert all projects in the folder.

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.

PC|SCHEMATIC

QUICK SURVEY OF THE MANUAL

To make the program as easy accessible as possible, the user interface for **Automation** has 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 THE COMPONENT MENU OTHER FUNCTIONS ON COMPONENT LEVEL OBJECT GROUPS SUB DRAWINGS / DRAG'N DRAW RENAME SYMBOLS
"Survey and Reference Designations" (P. 343)	DESCRIPTION OF HOW TO FIND PAGES AND OBJECTS IN PROJECTS, PERFORM DESIGN CHECK AND PRINT.	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" (P. 383)	DESCRIPTION OF HOW TO USE REFERENCE DESIGNATIONS IN THE PROGRAM.	- INTRODUCTION TO REFERENCE DESIGNATIONS - REFERENCE DESIGNATION SETTINGS - HOW TO WORK WITH REFERENCE DESIGNATIONS

SECTION	DESCRIPTION	SUBJECTS
"References" (*P. 431*)	DESCRIPTION OF HOW TO USE REFERENCES IN THE PROGRAM.	AUTOMATICALLY UPDATED REFERENCES CONTACT MIRRORS REFERENCE CROSSES CURRENT PATH NUMBERS / REFERENCE SYSTEM
"Wire numbers" (*P. 453*)	DESCRIPTION OF HOW TO USE WIRE NUMBERS.	AUTOMATIC WIRE NUMBERING MANUAL PLACING OF WIRE NUMBERS WIRENUMBER FILES
"Single line diagrams" (*P. 471*)	DESCRIPTION OF HOW TO WORK WITH SINGLE LINE DIAGRAMS.	ABOUT SINGLE LINE DIAGRAMS AUTOMATIC GENERATION OF SINGLE LINE DIAGRAMS
"PLC functions" (*P. 475*)	DESCRIPTION OF PLC FUNCTIONS.	SURVEY OF PLC FUNCTIONS WORKING WITH PLCS SIGNAL BUSSES READING PLC I/O FILES PLC I/O ADDRESSING VIA MICROSOFT EXCEL

THE CONTENTS OF PART 2

SECTION	DESCRIPTION	SUBJECTS
"Electrical installation drawing" (*P. 533*)	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. 551*)	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 THAT ONE PROJECT OPEN STARTING ANOTHER PROJECT BY CLICKING
"Lists" (*P. 597*)	HOW TO INSERT DRAWING HEADERS/ LISTS.	INSERTING DRAWING HEADER/LIST LIST SETTINGS COMPONENT GROUPING UNIT DRAWINGS
"Databases" (*P. 639*)	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

PC|SCHEMATIC

SECTION	DESCRIPTION	SUBJECTS
"Mechanical pages and symbols" *(P. 701)*	GET AN OVERVIEW OVER HOW TO WORK WITH MECHANICAL SYMBOLS	HOW TO WORK WITH MECHANICAL SYMBOLS ARRANGEMENT DRAWINGS PLACE MECHANICAL SYMBOLS, BASED ON THE PLACED ELECTRICAL SYMBOLS, VIA THE DATABASE PLACE ELECTRICAL SYMBOLS, BASED ON THE PLACED MECHANICAL SYMBOLS, VIA THE DATABASE
"Export / Import" *(P. 727)*	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 PCSCHEMATIC VIEWER
"Create Symbols, Lists and Projects" *(P. 763)*	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. 859)*	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 PCSCHEMATIC FILE TYPES INDEXING DDE AND STARTING FUNCTIONS ZIPPING / COMPRESSED SAVING THE MOUSE CHASING SYSTEM
"Appendices" *(P. 905)*	FOR REFERENCE, AND FOR ADJUSTING THE PROGRAM SETTINGS.	PROGRAM SETTINGS DICTIONARY DATAFIELDS FIELDS IN THE INCLUDED DATABASE VARIABLES INDEX

HOW TO GET STARTED

Start by installing the program, as described in the independent **Information & Installation** manual.

WATCH IT ON VIDEO

YOU CAN WATCH VIDEOS ABOUT THE PROGRAM HERE:

1) AT THE PCSCHEMATIC WEBSITE: — http://www.pcschematic.com/videos

2) AT THE INTERNATIONAL PCSCHEMATIC YOUTUBE CHANNEL: — http://www.youtube.com/user/PCSCHEMATIC

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 553, 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 39.

Overall planning

In the section "Create Symbols, Lists and Projects" on page 763, 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 361.

PCSCHEMATIC website

For direct access to the PCSCHEMATIC website 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
ITALICS	TEXTS WRITTEN IN *ITALICS* ARE TEXTS YOU HAVE TO TYPE IN YOURSELF.
BOLD	**BOLD** TEXTS — NOT INCLUDING HEADINGS — ARE USED FOR PROGRAM FUNCTIONS, BUTTONS AND MENUS.
BOLD AND ITALICS	ARE APPLIED FOR FIELD NAMES OR OPTIONS YOU CAN CLICK ON IN DIALOG BOXES.
KEYS *<>* *<CTRL+G>*	ALL SHORT-CUT KEYS ON THE KEYBOARD ARE ENCLOSED IN BRACKETS — SUCH AS <DEL> 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:

Reading this section will save time using the program.

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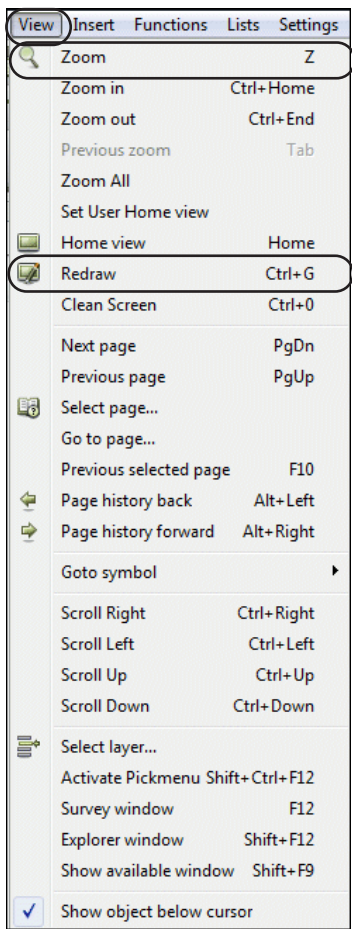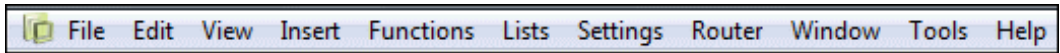
PC│SCHEMATIC

WINDOWS

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

MENUS

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

File   Edit   View   Insert   Functions   Lists   Settings   Router   Window   Tools   Help

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, **Zoom** 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

View	Insert	Functions	Lists	Settings
Zoom				Z
Zoom in				Ctrl+Home
Zoom out				Ctrl+End
Previous zoom				Tab
Zoom All				
Set User Home view				
Home view				Home
Redraw				Ctrl+G
Clean Screen				Ctrl+0
Next page				PgDn
Previous page				PgUp
Select page...				
Go to page...				
Previous selected page				F10
Page history back				Alt+Left
Page history forward				Alt+Right
Goto symbol				▶
Scroll Right				Ctrl+Right
Scroll Left				Ctrl+Left
Scroll Up				Ctrl+Up
Scroll Down				Ctrl+Down
Select layer...				
Activate Pickmenu				Shift+Ctrl+F12
Survey window				F12
Explorer window				Shift+F12
Show available window				Shift+F9
✓ Show object below cursor				

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:

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

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

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

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

SELECTING ELEMENTS IN LISTS

When elements are listed in a dialog box, these elements can be selected as described in "Selecting objects in the Object Lister" on page 354.

— See for instance "Copying, Cutting and Pasting more than one page" on page 565.

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 354.

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.

	Page article data
	Page data...
	Page setup...
	Page information
	Project information
	Variables...
	Paste Ctrl+V
	Show available window Shift+F9
	Symbol menu... F8
	Lines L
	Symbols S
	Texts T
	Arcs C
	Favorites
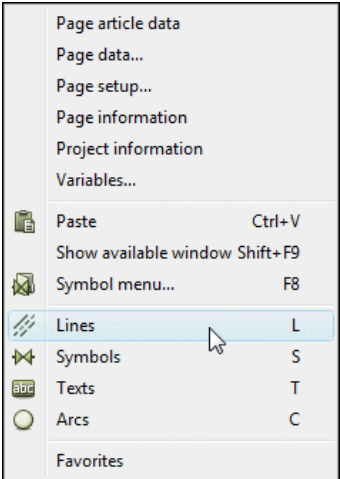
SWITCHING BETWEEN WINDOWS

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 582.

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	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 *DRAG* THE MOUSE ACROSS THE SCREEN. WHEN YOU RELEASE THE MOUSE BUTTON, THE OBJECT IS PLACED WHERE YOU POINT WITH THE MOUSE CURSOR. YOU CAN MOVE BOTH OBJECTS, PROJECT PAGES AND ELEMENTS IN LISTS BY *DRAG AND DROP*. — SEE "MOVING A PLACED SYMBOL" ON PAGE 199, "DRAGGING A REFERENCE DESIGNATION TO A NEW POSITION IN THE DIALOG BOX REFERENCE DESIGNATIONS" ON PAGE 397 AND "DRAG PAGES BETWEEN PROJECTS" ON PAGE 583.

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 **Component menu** to the secondary monitor.

Moving the Explorer, the Component list or the Symbols for component 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 **Components** and the **Symbols for component** window can be docked inside the **Component menu** on 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 **Components**
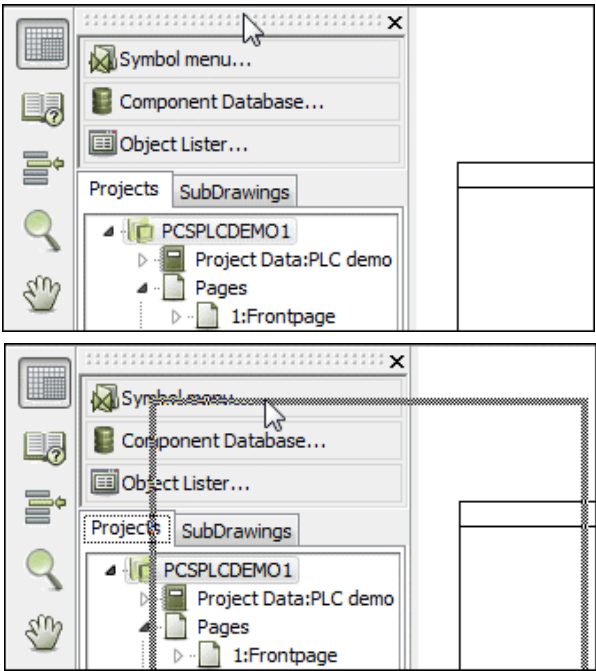
or the **Symbols for component** window you point at the title bar).

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 **Components** and the **Symbols for component** windows are by default placed inside the **Component menu** in the right-hand side of the screen. However, both windows can be *dragged* out as described above, so that you for instance can get a larger area on the screen for your diagrams by placing the **Component menu** on another screen.

Docking a window

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

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

When the mouse pointer — and *not* the edge of the grey box — is pointing 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.
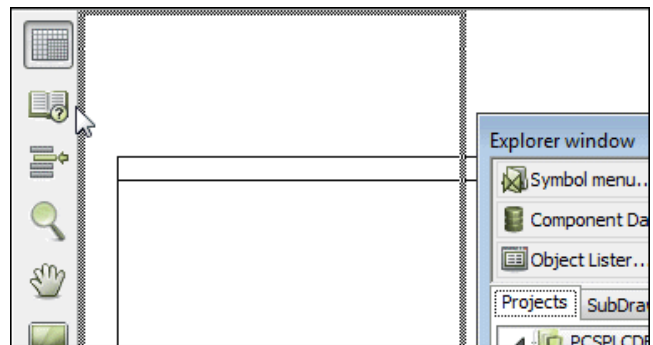
Docking the Survey window or the Symbols for component window

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 **Symbols for component** window can be docked in the upper part of the **Components** 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*.

Program toolbar   **Command toolbar (for symbols)**
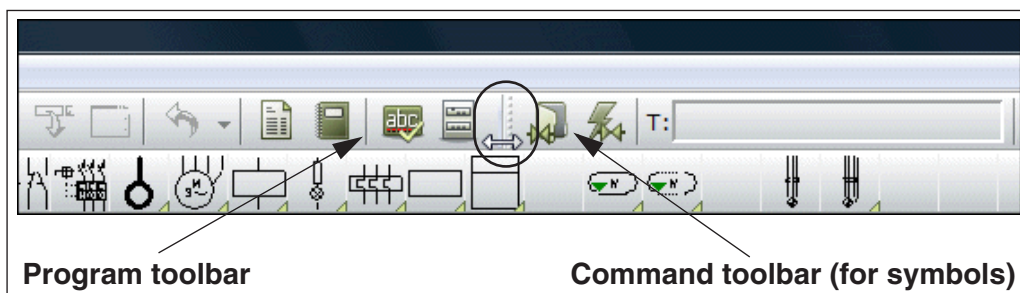
The command toolbar can not be moved/docked.

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.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— CHOOSING MENU ITEMS USING THE KEYBOARD</td><td>(P. 29)</td></tr>
<tr><td>— CHOOSING BUTTONS/FUNCTIONS USING THE KEYBOARD</td><td>(P. 30)</td></tr>
<tr><td>— NAVIGATION IN THE EXPLORER WINDOW</td><td>(P. 30)</td></tr>
<tr><td>— CHOOSING IN DIALOG BOXES WITH THE KEYBOARD</td><td>(P. 31)</td></tr>
<tr><td>— SELECTION IN AVAILABLE SYMBOLS USING THE KEYBOARD</td><td>(P. 31)</td></tr>
<tr><td>— PLACING OBJECTS USING THE KEYBOARD</td><td>(P. 32)</td></tr>
<tr><td>— CHOOSING OBJECTS USING THE KEYBOARD</td><td>(P. 32)</td></tr>
<tr><td>— CHOOSING MORE THAN ONE OBJECT WITH THE KEYBOARD</td><td>(P. 33)</td></tr>
<tr><td>— CHOOSING SYMBOLS WITH THE KEYBOARD</td><td>(P. 33)</td></tr>
<tr><td>— INCREASE/DECREASE SHORT-CUT USING THE KEYBOARD</td><td>(P. 33)</td></tr>
<tr><td>— SHORT-CUTS FOR ACTIVATING THE PENCIL</td><td>(P. 33)</td></tr>
</table>

CONCURRENT LEARNING OF SHORT-CUT KEYS
IN "PREVENTING REPETITIVE STRAIN INJURIES" ON PAGE 883, 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 ITEMS USING THE KEYBOARD

Choosing menu items using the keyboard follows the common Windows standards, as described in "Menus" on page 23.

— Please also refer to "Short-cut Keys" on page 870, 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 items 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 item 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 item **View => Select page**.

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

There are menu item for all buttons in the program.

— See "Short-cut Keys" on page 870 to see how to create keyboard short-cuts for all buttons.
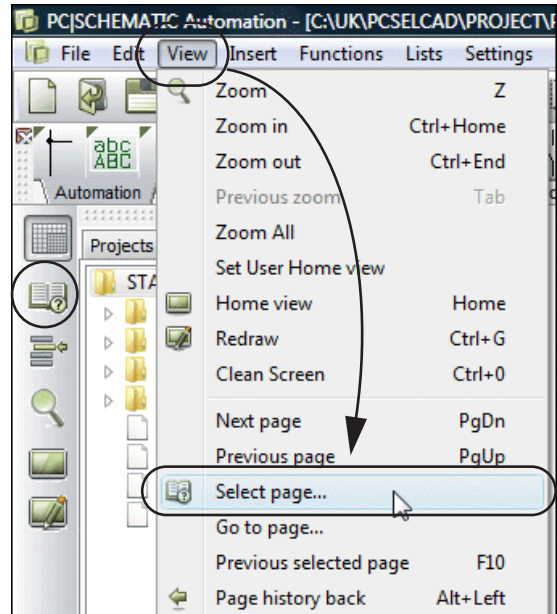
Short-cuts displayed as hints

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.
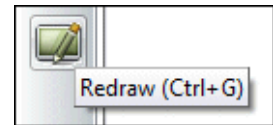
NAVIGATION IN THE EXPLORER WINDOW USING THE KEYBOARD

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.

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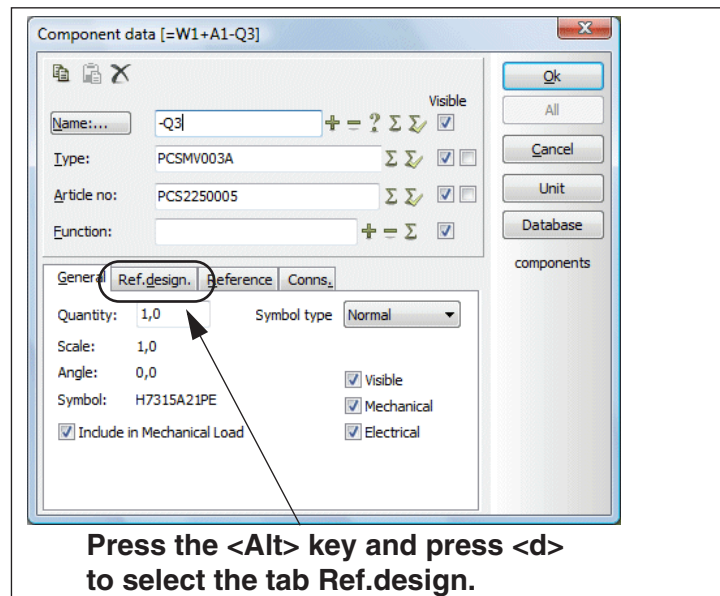
If for instance you wish to choose the tab **Ref.design.** in the dialog box **Component data** above, press down the <Alt> tab, and press the <d> button — because **d** is the underscored character on the tab.

If you wish to select **OK** in the dialog box **Component data**, press <Enter>. If you wish to select **All**, press <Ctrl+Enter>.

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 AVAILABLE SYMBOLS USING THE KEYBOARD

When you are using the **Available symbols** window for a component, the field containing the first symbol in **Available symbols** is coloured blue. 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 one of the other symbols:
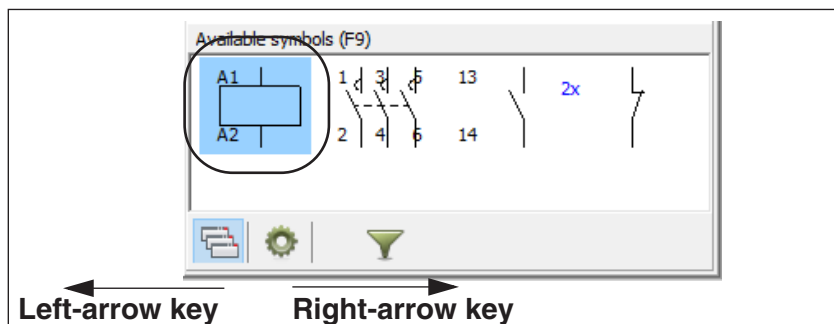
To use the keyboard to select symbols in the stationary pickmenu at the top of the screen, you must select **View => Activate Pickmenu** (Short-cut: <Shift+Ctrl+F12>).
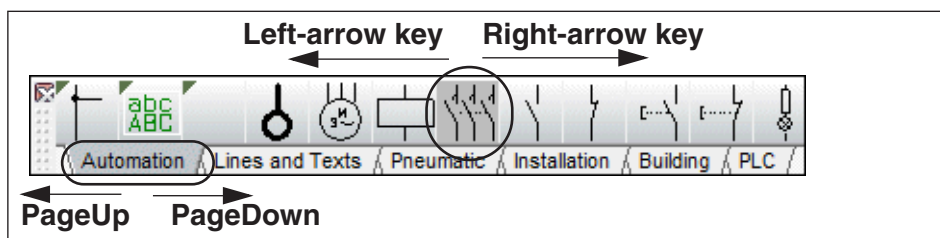
You can then select the symbols by pressing the key-arrows, and change between the different pickmenus by pressing the <PageUp> and <PageDown> keys. When you press <Enter> you get the selected symbol in the cross hairs.

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 171.

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 172.

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 173.

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 135.

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 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 or choose **File => New**. In the dialog box **New**: Double click on the page template *PCSstart*. In the dialog box **Project data**: Click **OK**.

— See more about how you select page and project templates in "Pages and Chapters" on page 556.

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 907.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES THE WORK AREA OF THE PROGRAM.
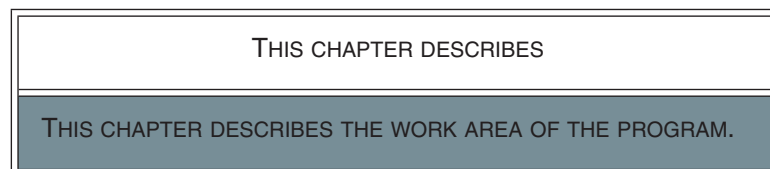
When you have opened a project, your screen should look like this:
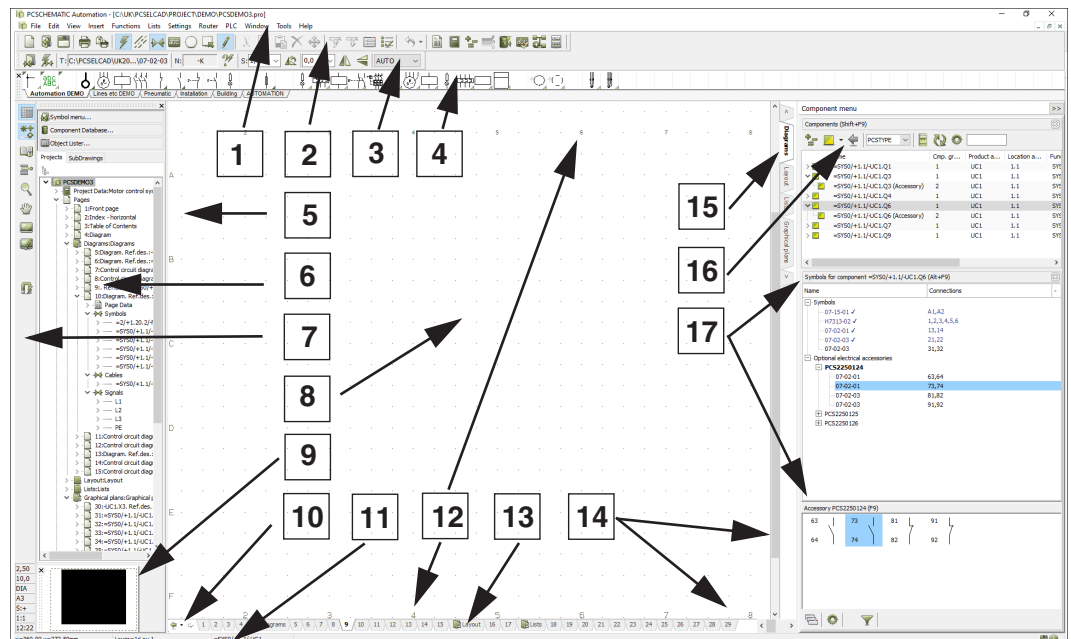
— Please note that you can change the position of the windows and the toolbars, as described in "Docking Windows and Toolbars" on page 26.

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 155, 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 918. 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 162.

- 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 170.

- 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 351.

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 345.

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 299.

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 915, or by inserting a drawing header, as described in "Insert drawing headers/formulas" on page 599.

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 557.

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 445.

PC|SCHEMATIC

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 49 and "Choosing pages outside the Page menu" on page 556.

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. Chapter tabs

When you click on a chapter tab, you jump to the first page in the selected chapter. — See "Chapters" on page 569.

16. The Component menu: The Component list window

In the **Components** window inside the **Component menu** you can for instance see all components in the project or all components in the project with available symbols (unused functions). Read more about this in "The Component menu" on page 263.

17. The Component menu: The Symbols for component window

In the **Symbols for component** window you can see the symbols for the component that you have selected in the **Components** window. The symbols and their eventual accessories are displayed in a list. When a component has available symbols these are displayed at the bottom of the window. From there you can *drag* the symbols into the diagram pages. Read more about this in "The Component menu" på side 263.

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. 39)
— THE TABLE OF CONTENTS .... (P. 40)
— THE BUTTONS IN THE ON-LINE MANUAL... (P. 40)
— ZOOMING AND ADAPTING TO YOUR SCREEN.. (P. 40)
— DIRECT ACCESS TO THE INDEX .... (P. 41)
— RETURNING TO **Automation**... (P. 41)
— PRINTING FROM THE ON-LINE MANUAL .... (P. 41)

OPENING THE ON-LINE MANUAL

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

Click to view a chapter — **Zoom** — **Fit the manual to your screen** — **About the electronical manual** — **Return** — **Up and down** — **Click directly to Index** — **Click to see the sub-chapters** — **Click to jump to the sub-chapters** — **Page number**

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>.
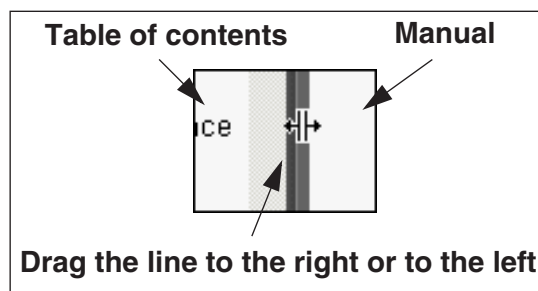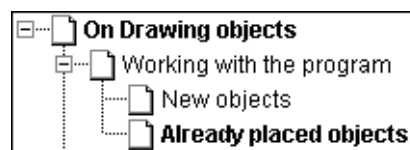
THE BUTTONS IN THE UPPER-RIGHT CORNER

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.

About the online manual

Return to the last page you read

One page up

One page down
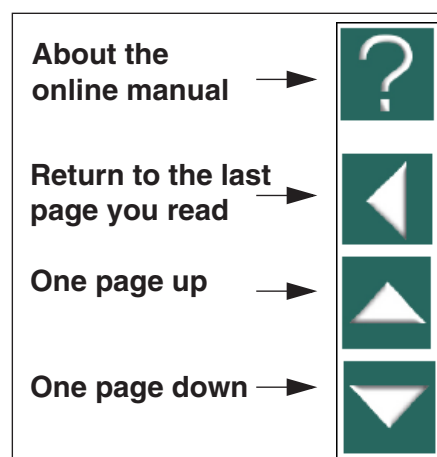
ZOOMING AND ADAPTING TO YOUR SCREEN

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.

To find an explanation about for instance 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.
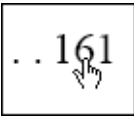
Click the **Fit in Window** button or press <Ctrl+0> to restore full page view.

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.

PC|SCHEMATIC

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.

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 870. 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
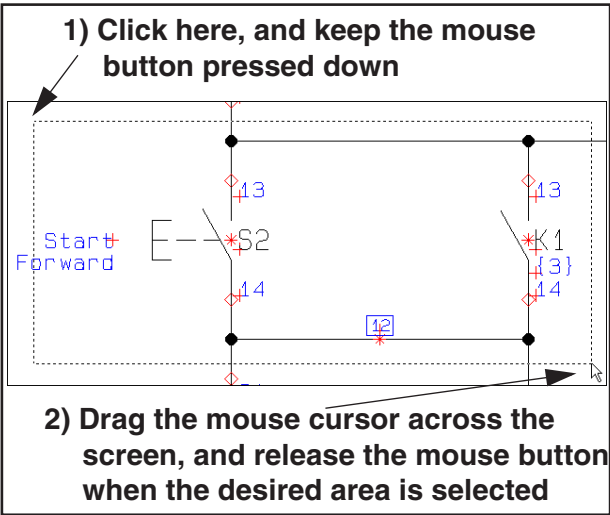
When you release the mouse button, the chosen area is now enlarged.

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>).

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 scroll-bars 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.

Scrollbar

The use of scrollbars can be activated/deactivated as described in "Pointer/Screen" on page 918.
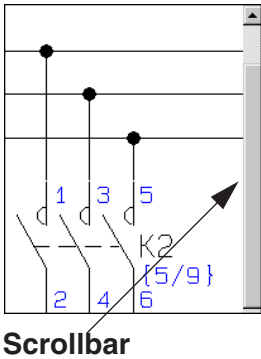
Zoom and Scroll for mouse with a wheel

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: Wheel Forward

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

PRESS	RESULTS IN
<WHEEL FORWARDS>	ZOOMS *IN* WHERE THE CROSS HAIRS IS LOCATED
<WHEEL BACKWARDS>	ZOOMS *OUT*
<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

Mouse wheel set to Zoom: Wheel Backward

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

PRESS	RESULTS IN
<WHEEL FORWARDS>	ZOOMS *OUT*
<WHEEL BACKWARDS>	ZOOMS *IN* WHERE THE CROSS HAIRS IS LOCATED
<SHIFT + WHEEL FORWARDS>	THE ZOOM IS MOVED *DOWNWARDS* ON THE PAGE
<SHIFT + WHEEL BACKWARDS>	THE ZOOM IS MOVED *UPWARDS* 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 918.

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 pro-
ject 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 spec-
ified 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.

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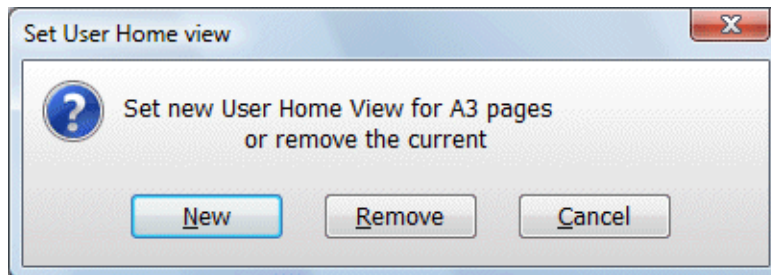
Click **Remove** to make the Home view button display the full page again, or click **New** to create a new Home view.

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.
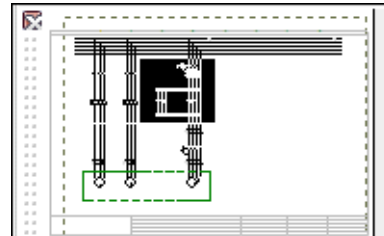
Moving the displayed area

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 Symbols for component 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 345.

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 electrical drawings.

— See "Page setup" on page 915.

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 918.

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.

On scaled (mechanical) pages you can make the snap and the grid follow the page scale factor if you choose **Settings => Page setup** and select *Follow page scale*. — See "Setting the scale of the page" on page 547.

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.
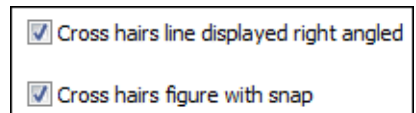
When you draw **Straight lines** (see "Straight lines" on page 73), a line will be displayed starting from where you clicked last, pointing towards the cross hairs.

☑ Cross hairs line displayed right angled

☑ Cross hairs figure with snap

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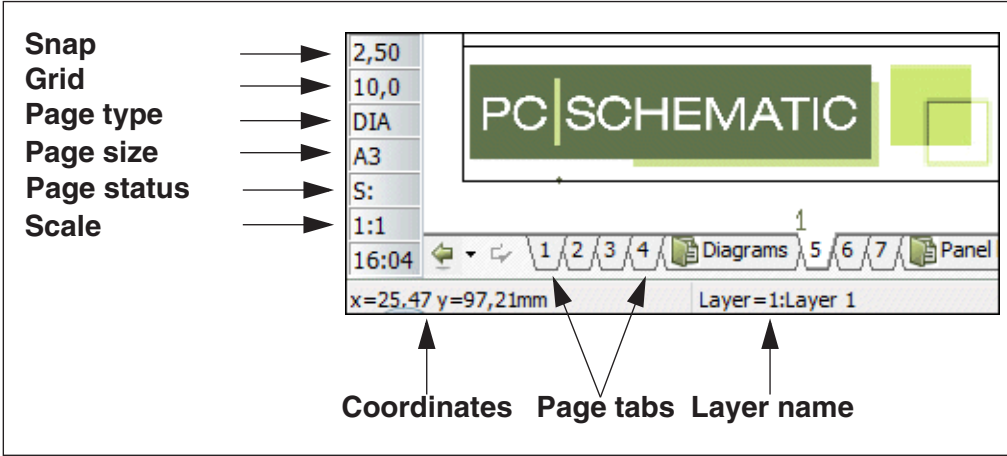
When you let the mouse cursor rest an instance over one of the fields, an explanatory text is displayed — as for **Snap** on the figure above.

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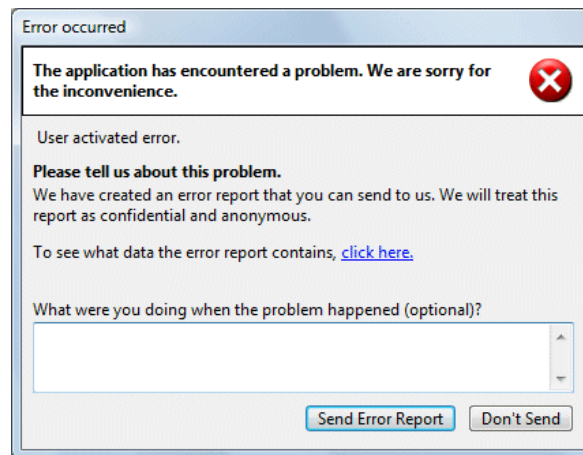
Here you can tell us what happened just before the error occurred, which we would very much like to know, so that we can correct it as soon as possible.

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:

WATCH TUTORIAL VIDEOS / READ SURVEY ARTICLE
PLEASE READ THE SURVEY ARTICLE ABOUT DRAWING FUNCTIONS AT WWW.PCSCHEMATIC.COM/ARTICLES AND WATCH VIDEOS ABOUT HOW TO CREATE PROJECTS — AND MUCH MORE — AT www.youtube.com/pcschematic.

PC | SCHEMATIC

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\>   **Texts** \<t\>   **Lines** \<l\>   **Circles** \<c\>   **Areas** \<a\>
(s. 152)   (s. 110)   (s. 72)   (s. 108)   (s. 280)

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

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:
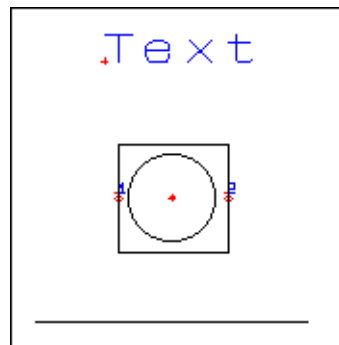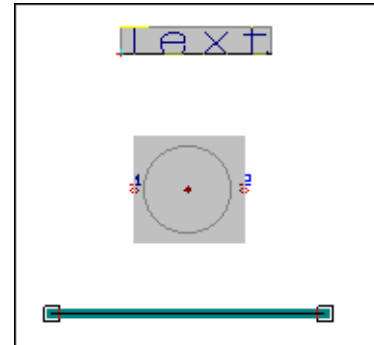
Deselected	**Selected**

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 980) 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 **Text*s*** 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

3) Place the cursor here and release the mouse button

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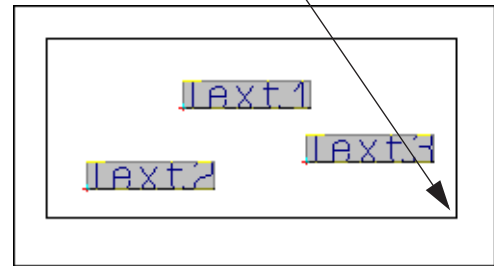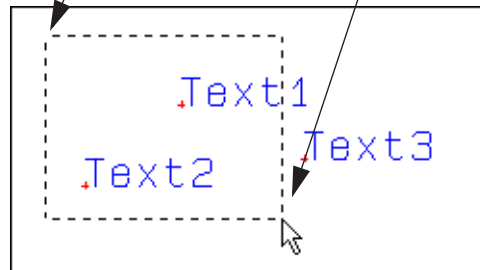
If you wish to deselect one or more objects inside the window, press down the <Ctrl> key and click on the object(s). If you also wish to select objects outside the area, also press down the <Ctrl> key, and click on these objects.

"VIEW REFERENCE POINTS" MUST BE ENABLED

IF YOU CAN NOT SEE THE REFERENCE POINTS WHEN YOU ZOOM IN ON AN AREA, THIS IS BECAUSE **VIEW REFERENCE POINTS** HAS NOT BEEN ENABLED. ENABLE IT BY CLICKING THE **REFERENCE POINTS** BUTTON IN THE LEFT TOOLBAR, OR BY SELECTING **VIEW => REFERENCE POINTS**.
— GET MORE INFO ABOUT THIS IN "SETTINGS" ON PAGE 907.

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 280.

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.

Dragging left to right   **Dragging right to left**
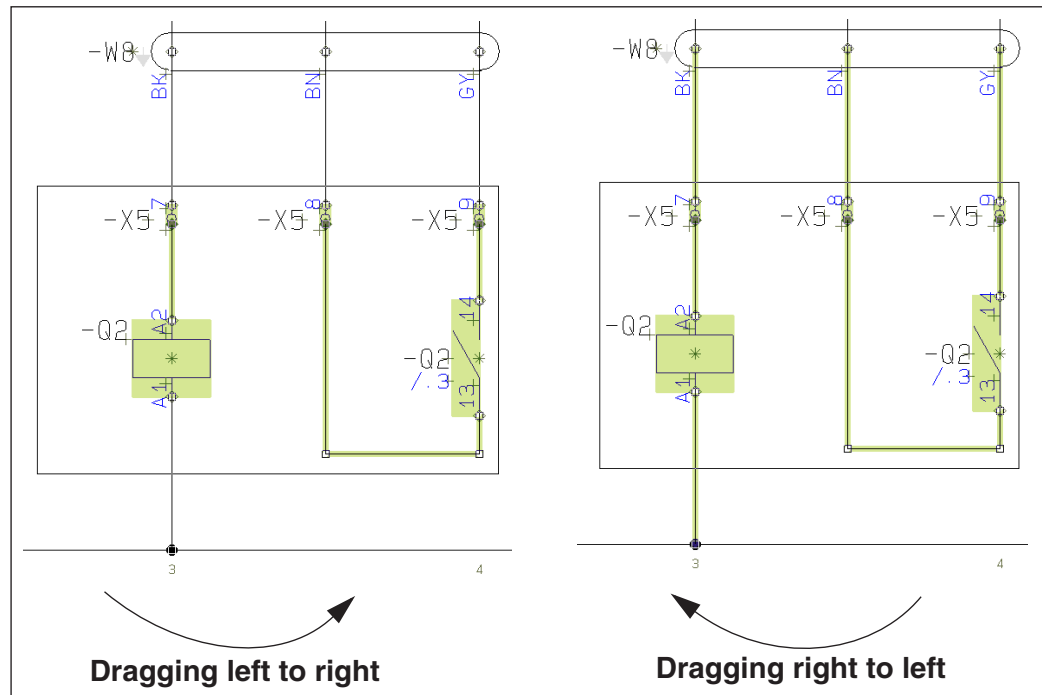
Selecting on different Layers and in different elevations

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 545, and read more about layers in general in "Layers, Elevation and Scaling" on page 541.

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 108, "Texts" on page 110 and "Symbols" on page 152.

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 <Del>. 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 <Spacebar>. The object will be rotated 90 degrees counter-clockwise. If you press <Ctrl> while pressing <Spacebar> the object is rotated 10 degrees.

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 290.

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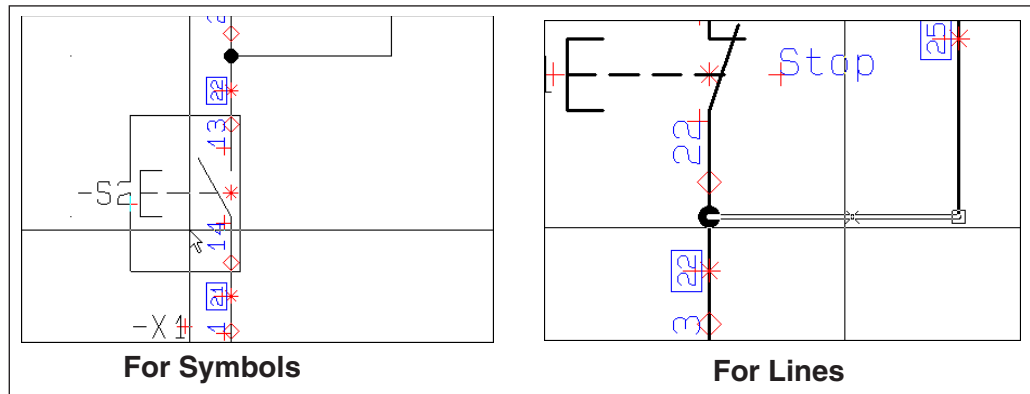
For lines it is also specified where you would connect to the line, if you clicked on it at the given moment.

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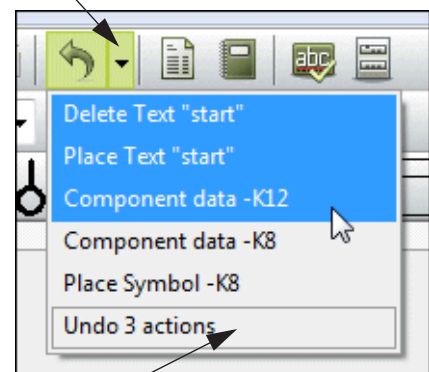
Place the cursor here, and the next points in the list are displayed

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.

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

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. This copies the contents of the fields **Name**, **Type**, **Article no** and **Function**. If symbol datafields are created for the symbol — see "Symbol datafields" on page 122 — the contents of these are also copied.
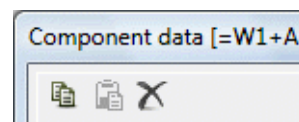
Leave the menu by pressing <Esc>.

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.

PCSCHEMATIC Automation

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**.

When you copy line data for a line, the contents for any attached line data data-fields are also copied.

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 *not* the course of the line, or if it is drawn as for instance a Slanted or Curved line. During the generation of a symbol, *Follow connected* is also transferred. See "Creating Symbols" on page 775.
Text properties	All information in the dialog box **Text properties** — see "Specifying text properties / Appearance of text" on page 136. The text itself, and the angle it might be rotated with, is *not* transferred.
Circle/arc properties	All information in the Circles toolbar — see "Arcs/Circles" on page 108.
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 se how this is done by placing the two texts *Text1* and *Text2* — each carrying different text properties — and then give both texts the same height without changing the other attributes of the texts.

Place Text1

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

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

Click on **OK** and place the text somewhere on the page.
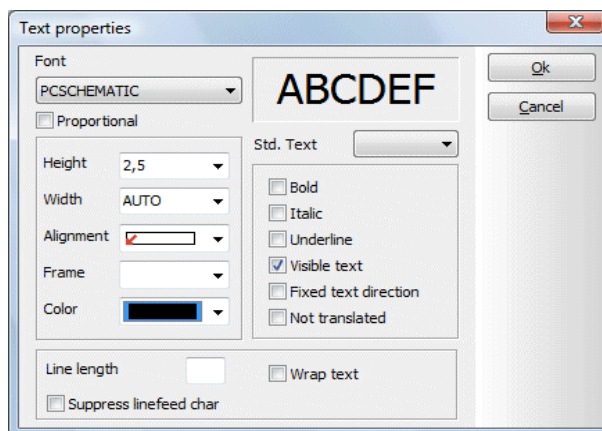
Place Text2

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

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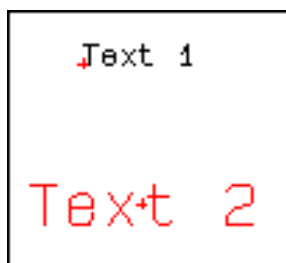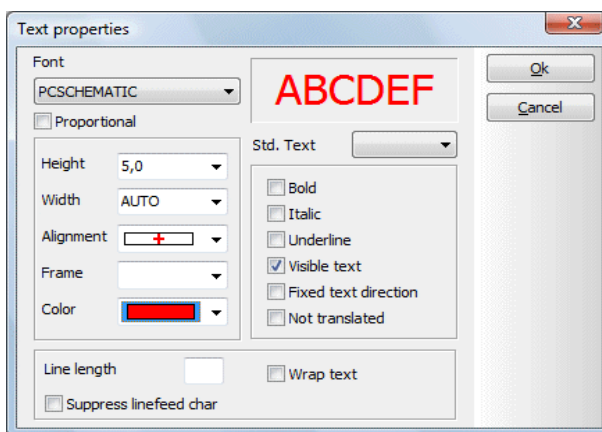
Transfer text properties

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

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

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

As you can see, the texts now have the same height, but their other attributes have not been changed.
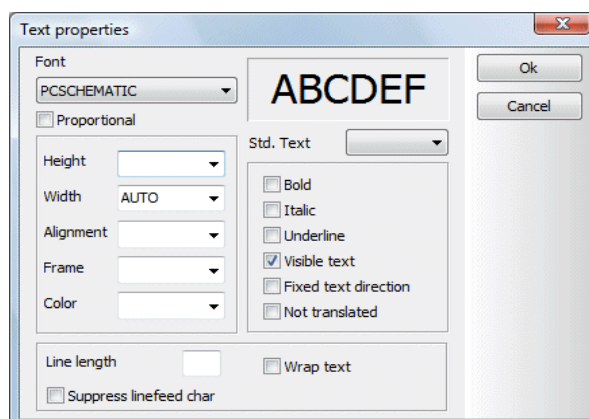
Text1 is still black and aligned *left-down*, while Text2 still is aligned *centre-centre* and red.
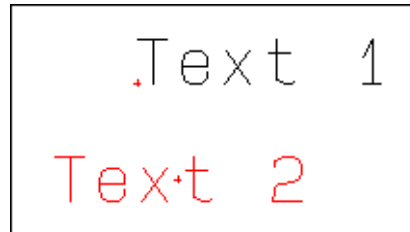
You can use the same procedure for lines and circles.

ALIGNING AND SPACING FUNCTIONS

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

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— THE ALIGN FUNCTION...</td><td>(P. 66)</td></tr>
<tr><td>— THE SPACE FUNCTION...</td><td>(P. 67)</td></tr>
</table>

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:
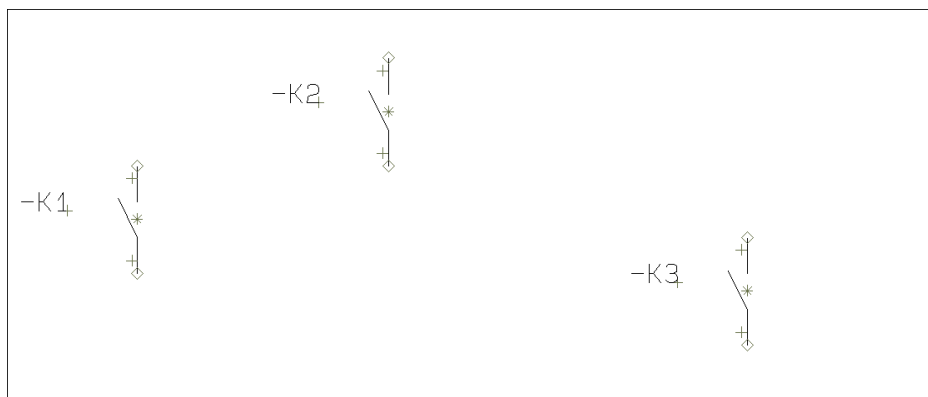
1) Make sure that the **Symbols** button is activated, then *click and drag* to select the two symbols to the right:
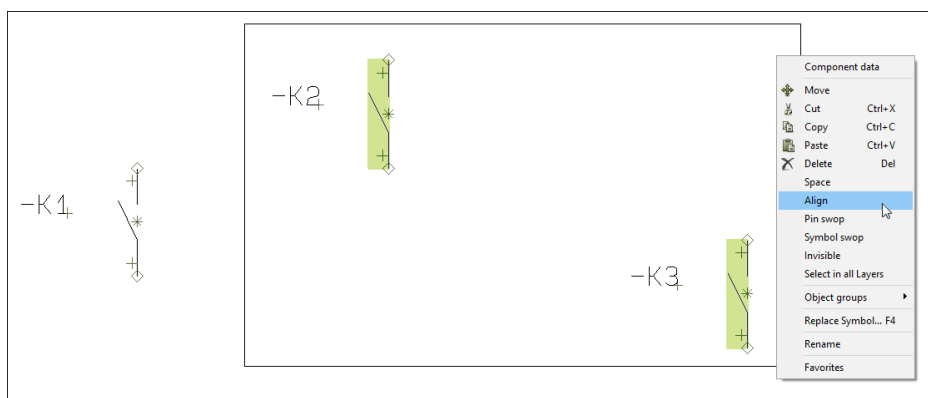
2) Select **Edit => Align**, or right-click inside the selected area and choose **Align**.

3) A rubber band originating from the symbol in the middle and ending in the cross hairs is now displayed. (See the guiding text in the bottom of the

screen). Click on the symbol to the left, and the two selected symbols are aligned with the symbol.
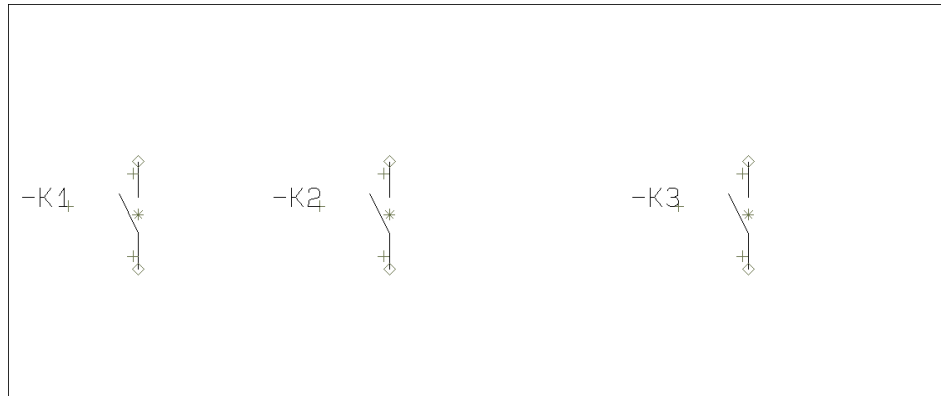
If the symbols had been closer to be arranged on a vertical line, they would have been placed on a vertical line instead.

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

ALIGN - WITH AND WITHOUT <CTRL>
WHEN YOU RIGHT-CLICK ON A SYMBOL (A) AND SELECT **ALIGN** AND THEN CLICK ANOTHER SYMBOL (B), A IS ALIGNED WITH B.
WHEN YOU PRESS DOWN <CTRL> WHILE CLICKING B, B IS ALIGNED WITH A.
THE SAME GOES FOR OTHER TYPES OF OBJECTS.

Aligning symbols - across project pages

Symbols placed on separate project pages are aligned like this:

1) Make sure that the **Symbols** button is activated, right-click on the symbol that you wish to align the other symbols to, and select **Align**.

2) Jump to another project page — for instance using the page tabs at the bottom of the screen

3) On the new project page a thin line now originates from where the selected symbol was placed on the previous project page, and ends in the cross-hairs: Now click on each individual symbol, that you wish to align to the selected symbol. This aligns them to the selected symbol.

It is not possible to do this for other types of objects.

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, the are sorted according to the first connection name. — Read more about component grouping in "Component Grouping" on page 626.
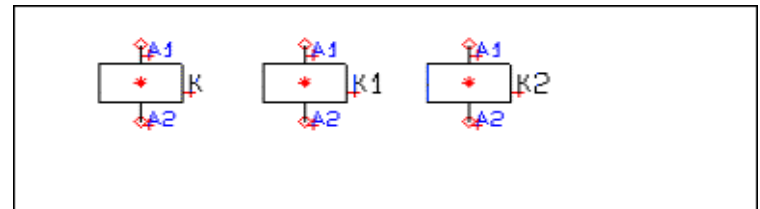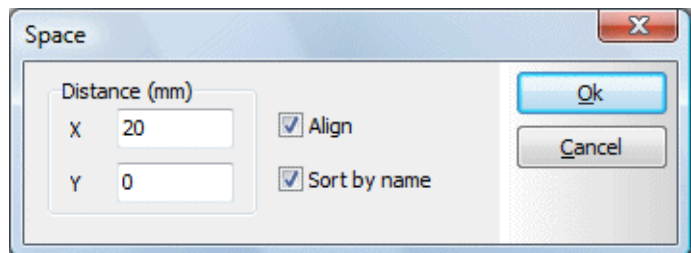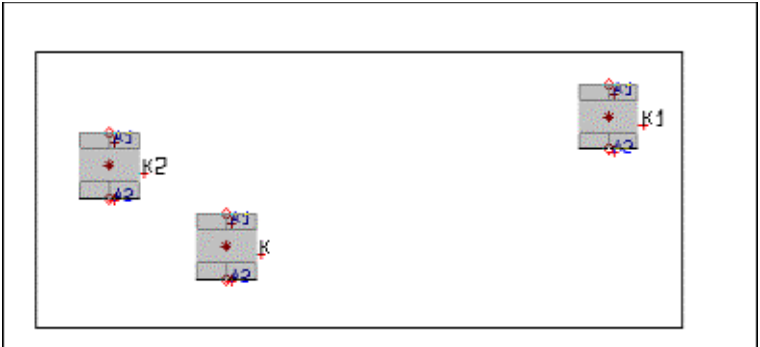
MAKING OBJECTS INVISIBLE

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.

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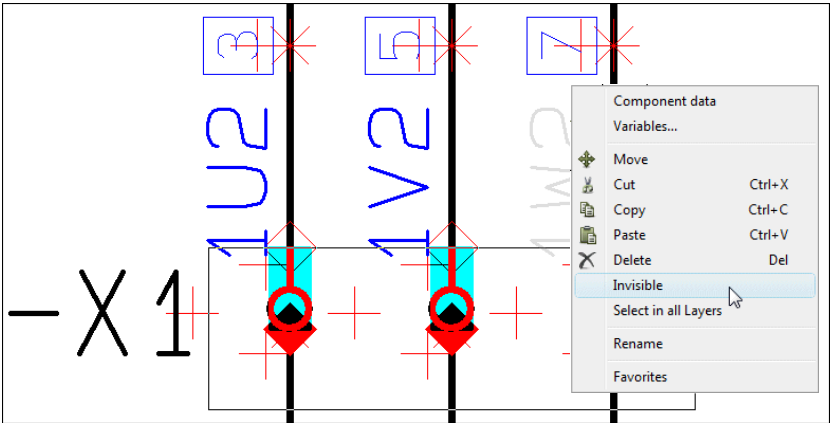
For symbols with reference crosses, the reference crosses also becomes invisible.

— Please read "Visible" on page 180 also.

Making lines invisible

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

Then do the following:

Right-click and choose Component data

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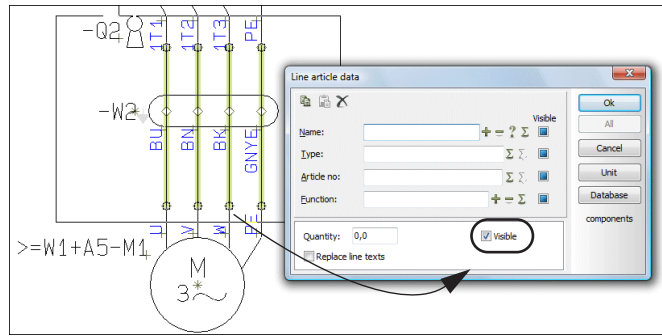
The lines are made visible again by right-clicking and then choosing **Visible**, or by entering the **Line article data** dialog box, and setting a check mark in front of *Visible*.

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 139.
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**.
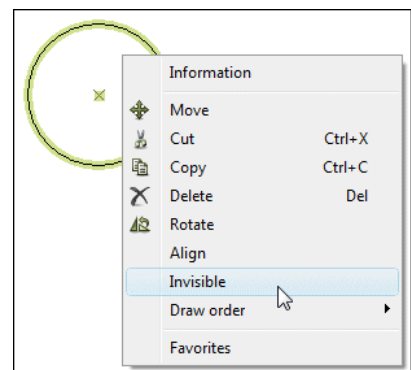
Making areas invisible

1) Click on the **Area** button and select the area — see "Areas" on page 280.

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 299.

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.

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 96.

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 87.

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 electrical connection. The first time you click the Lines button, Conducting lines is activated automatically.

Conducting lines are described in further detail in "Conducting (electrical) lines" on page 79. — Please note that you can specify how to represent crossing non-connected conducting lines on diagram pages in "Pointer/Screen" on page 918.

If Conducting lines is not activated, the lines will be general purpose (non conducting) lines that can be used anywhere within the project.

JUMPER LINKS

IT IS ALSO POSSIBLE TO DRAW JUMPER LINKS, AS DESCRIBED IN "JUMPER LINKS" ON PAGE 92.

THE LINES TOOLBAR

The toolbar for lines looks like this:
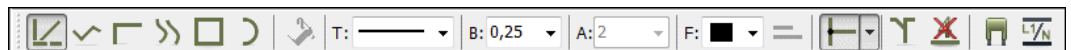
In the following you can see the functions of the buttons in the toolbar. — Please note that you can also place lines in the pickmenu, as described in "Lines in the Pickmenu" on page 79.

Straight lines

When you are drawing **Straight lines**, the lines are automatically drawn either right-angled or diagonally, then see "Pointer/Screen" on page 918).

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**.
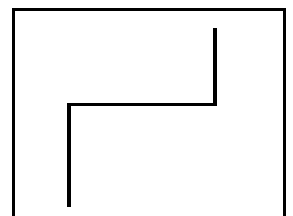
Slanted lines

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.
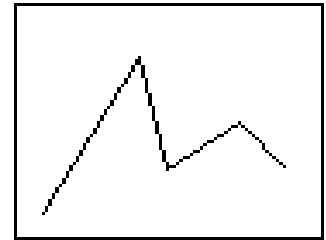
Angled lines

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 102 for further details.

Click here

... and here
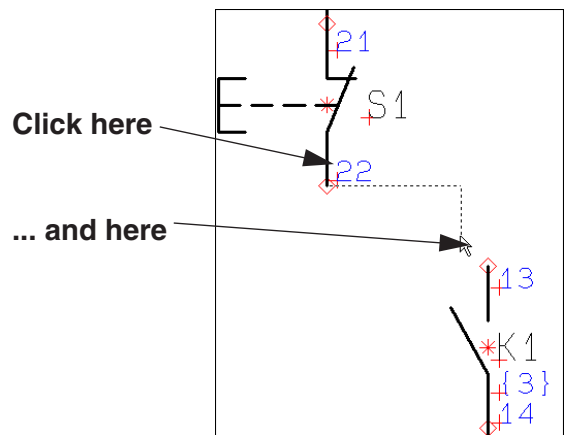
Curved lines

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.
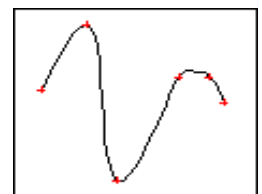
Round lines

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 (+).
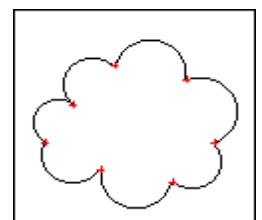
Rectangles

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.

Folding a corner on a rectangle

To fold a corner on a rectangle, do this:

1) Deselect the pencil (press <Esc>)

2) Right-click on the line close to the corner and choose **Fold corner**

3) Click to specify where to fold the corner

This function can be used for closed lines and reference frames. — See "Changing the shape of reference frames" on page 420.

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.

These line types can be used as conducting lines

These line types can not be used as conducting lines
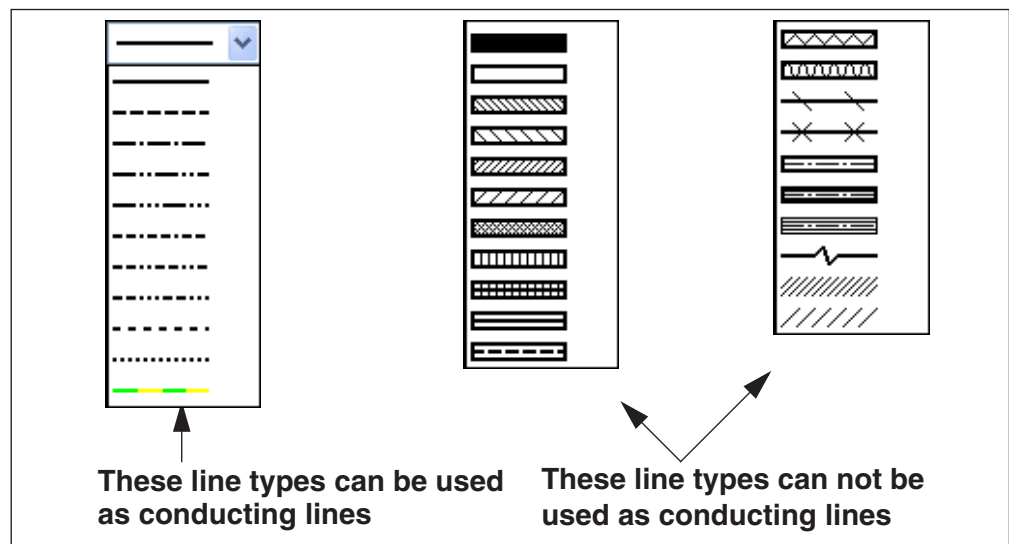
— Please note that the lower of the line types to the left is yellow/green, for PE connections.

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 distance

Line width

Line color: F

When you click here, you can choose the line color. You have these options:

Basic colors

Here you can choose fifteen predefined colors or the color **NP** (<u>N</u>on <u>P</u>rintable), which will be displayed on screen, but not on print.

Custom colors

These colors you can define yourself, as described in "Custom colors" on page 875.

Other colors

Here you can choose among the most recently used colors.

More colors

Here you can freely select a color of your own choice.
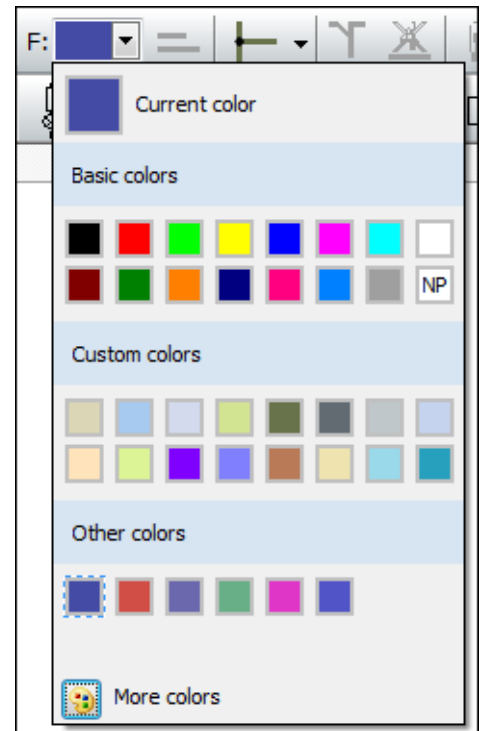
Extend line

See how to use the **Extend line** button in "Extended lines" on page 97.

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 92.

Mounting correct drawing

See how to apply **Mounting correct drawing** in "Mounting correct drawing" on page 102.

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 456.

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 97.

Insert potential

See how to use the **Insert potential button** for automatic insertion of electrical potentials in "Automatic insertion of electrical potentials" on page 87.

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.
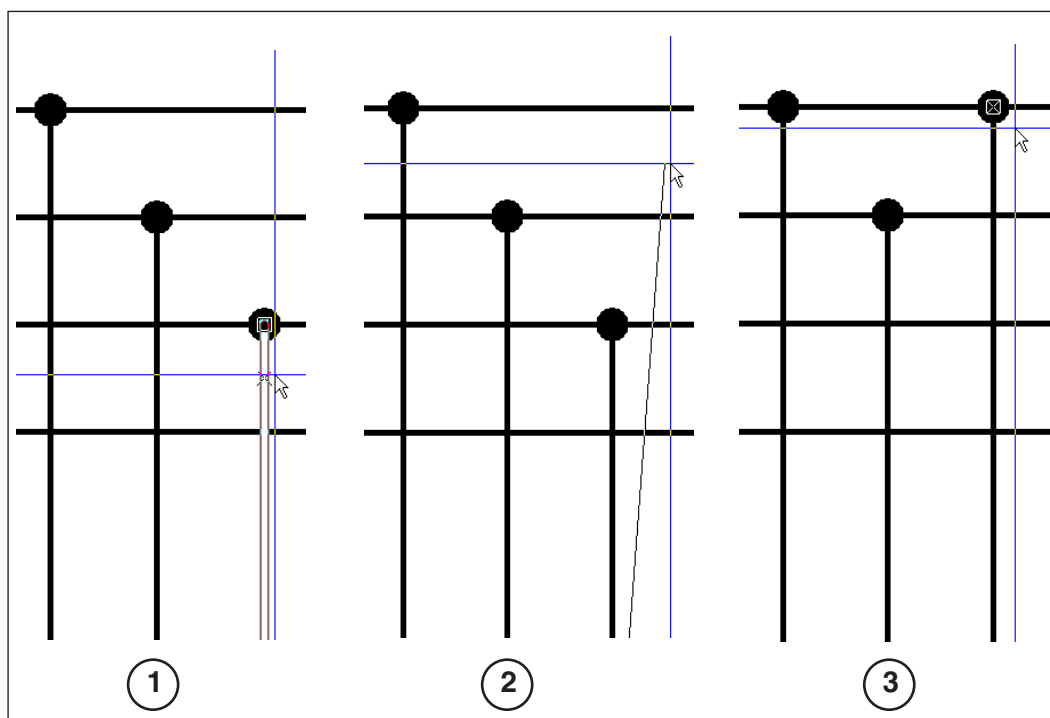
LINES IN THE PICKMENU

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.

Symbols/drawing objects displayed in all pickmenus

Click on a line, and place it in the diagram   **Select different menus in the pickmenu**
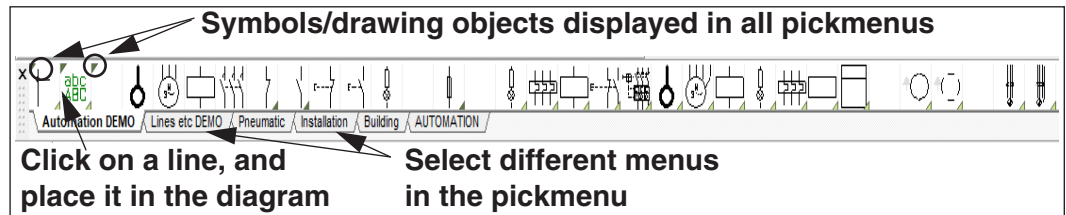
When you click on one of the *lines* in the pickmenu, the program automatically changes to working with **Lines** and the **Pencil** button is activated. You can then draw lines with the selected properties. — Read more about line properties in "Transferring Data / Properties between Objects" on page 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 155 and "Adapting the Pickmenu" on page 203.

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 234, and about reference designations on signal symbols in "Variant names on signal symbols" on page 239.
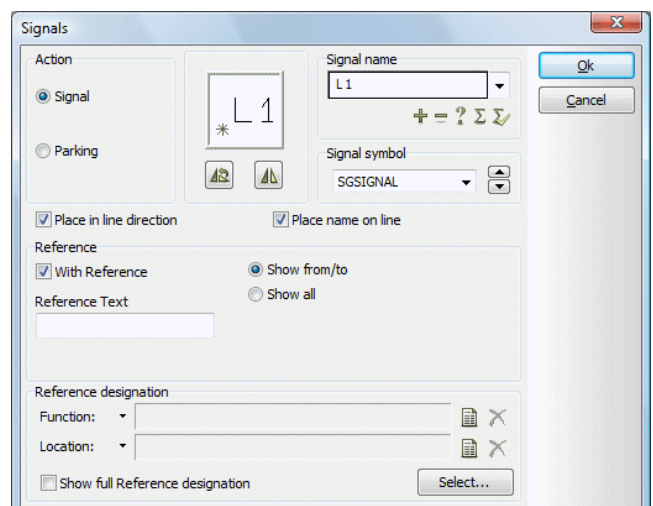
Parking

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.

When you later return to place a signal symbol, just right-click on the parked end of the line and choose **Place signal symbol**.

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.

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 641**, in particular **"Using the database for Lines" on page 641**.

— Please note that you can also attach article data to cable symbols, as described in "Working with cable symbols" on page 248.

Direct access to article data by right-clicking

When you right-click on a line with article data and choose **Open**, the dialog box **Record Viewer** opens, displaying the article data for the line in the database:
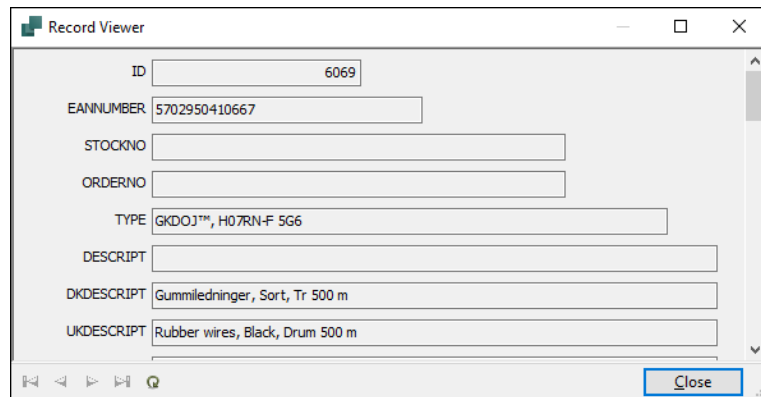
If a unit drawing is connected to the line, the unit drawing is opened in stead. — See "Connecting symbols/lines on diagrams to unit drawings" on page 638.

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.
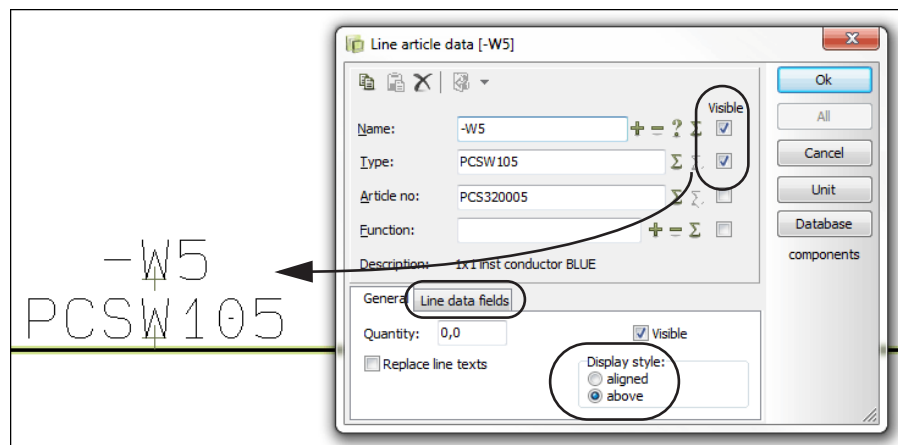
When article data for a line is set to be visible, they are placed above the middle of the longest segment on the line. — In "Text anchor for linedata" on page 82 you can see how to place them next to another segment of the line.

If this segment later disappears — e.g. by deletion or because a symbol is placed — the article data are 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 141.

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.

Text anchor for linedata

To show line data next to a specific line segment, do this:

1) Activate the **Lines** command and deselect the **Pencil**.

2) Right-click on the segment and select **Text anchor**. (If the text is already anchored to the selected segment, **Text anchor** is dimmed).

Line datafields

If *line datafields* have been created in the project, you can click the **Line data fields** tab at the lower part of the dialog box and type in the values for the line data fields. Please note that when you copy line data for a line, the contents for any attached line data datafields are also copied.

— Line datafields are created like you create symbol datafields. Read more about this in "Creating symbol datafields" on page 811.

— See how to include line datafields in lists in "Inserting datafields in lists" on page 86.

THE LINE DATAFIELD ROUTINGCLASS IS FOR THE PANELROUTER

ON THE TAB **LINE DATAFIELD** THERE IS ALREADY CREATED A LINE DATA-FIELD WITH THE NAME **ROUTINGCLASS**. THIS DATAFIELD IS USED FOR SPEC-IFYING THE ROUTING CLASS, WHICH YOU CAN READ MORE ABOUT IN THE MANUAL FOR THE **PANELROUTER**.

Calculated line datafields (Formula editor)

It is also possible to create calculated line data fields, which can be a combination of line data, texts and other line data fields. Thus you can combine multiple line data fields in a single line data field.

You can create a calculated line data field like this:

CREATE THE NECESSARY LINE DATA FIELDS

1) Choose **Settings => Text/Symbol defaults**. In the **Symbol defaults** set-tings: Click on **Line data fields**. (If you choose **Symbol data fields** in stead, you can create calculated symbol datafields).

2) Under **Line data fields**: Click the **Add** button and type in the name of the line data field — for instance *FieldA*. Also create the line data fields *FieldB* and *FormulaExample*.

CREATE A CALCULATED LINE DATA FIELD

3) Now click the **Formula editor** button:

4) You now enter the **Formula editor**, where you can specify how to create/calculate the line data field:

5) Select *Line data*, *Line data fields* and/or *Static text* and click **Add**, which adds the selected data fields / texts to the **Formula** line in the upper part of the dialog box — with the correct syntax.
Under *Result* you can see how the calculated data field will appear when inserted on the project pages.

6) When the formula is complete: Click **OK**.

EDIT THE FORMULA MANUALLY

If you need to edit the formula manually, just select *Edit formula manually* in the lower-left corner of the dialog box. When you have edited the formula manually, click the **Validate formula** button to check that the syntax is still correct.

Click **Clear formula** if you want to delete the formula.

DISPLAYING CALCULATED DATA FIELDS ON PROJECT PAGES

When you have created a calculated line data field, as in the example above, you can insert it in a project like this:

1) Double-click on a line and fill out the dialog box **Line article data** as displayed below, where you only set the line data field *FormulaExample* to be visible:

2) The line data field *FormulaExample* for the line is now displayed like this in the project:

Select name and Object Lister - for lines with line data

When working with lines, you can select lines with line data in the project by choosing **Functions => Select name**. This functions the same way as for symbols, which you can read more about in "Find all symbols with identical names" on page 274.

You can also use the **Object Lister** to work with lines with line data.
— See "The Object Lister" on page 351.

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 data

In all electrical lists — terminal, cable, connection and PLC lists — you can insert datafields with *Line-name*, *Line-type*, *Line-article no.* and *Line-function*. — See "Datafields in Automation" on page 944.

Inserting datafields in lists

When you have created a line datafield, you can subsequently insert it in parts, components, terminals, cables, PLC and connections lists. When you edit the list page, just choose **Insert => Insert Datafield**, which opens the dialog box **Datafield**. Here you can choose *Parts/Components List* and select the desired datafield at the bottom of the list of datafields.

Priority on lines

If you have set up the program to draw arcs when non-connected conducting lines cross — see *Representation of crossing non-connected conducting lines* in "Pointer/Screen" on page 918 — you can use *Priority* to specify which of the crossing lines are drawn with arcs. This is done by setting a *Low*, *Normal* or *High* priority for the lines.

The following rules applies:

- When two lines with different priorities cross, the line with the highest priority is drawn without an arc.

- When two lines with the same priority setting cross, the horizontal line has the highest priority.

AUTOMATIC INSERTION OF ELECTRICAL POTENTIALS

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:

> **The signal names that will be inserted**
>
> L1,L2,L3,PE
> ◉ Follow list from selected
> ○ Count up from selected
>
> Signal name
> L1
> L1
> L2
> L3
> PE
> N
> L1
> L2
> L3
> N
>
> **Because Number is set to 4 these four signal names are inserted from the list**

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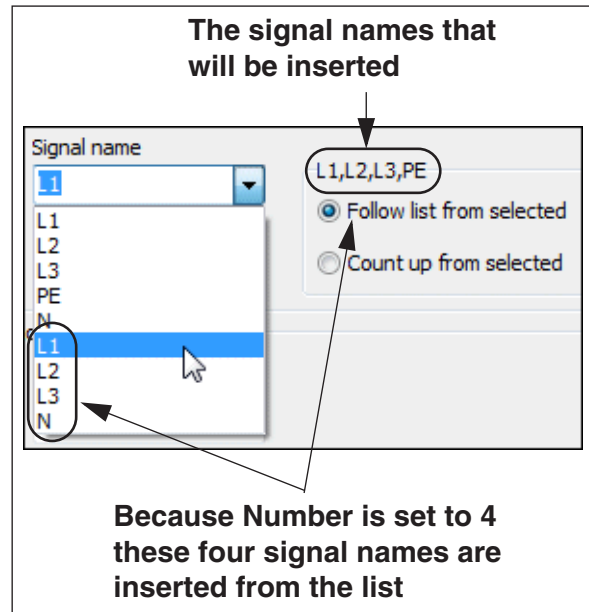
To only place electrical potentials on the upper part of the page, click the **Upper** button in the right-hand side of the dialog box.

— Please also refer to "Specify Level for the electrical potentials" on page 89, "Reference text for automatically placed electrical potentials" on page 89, "Reference designations for automatically placed electrical potentials" on page 89, "Settings for automatically placed electrical potentials" on page 89 and "Signal names on the Signal name list - for automatically inserted electrical potentials" on page 91.

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 88.

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 89, "Reference text for automatically placed electrical potentials" on page 89, "Reference designations

for automatically placed electrical potentials" on page 89, "Settings for automatically placed electrical potentials" on page 89 and "Signal names on the Signal name list - for automatically inserted electrical potentials" on page 91.

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 89.

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 413.

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 414.

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**:

Select page size for the settings

Y-position for the upper potential

Distance between potentials

Y-position for the lower potential

Click here to apply the settings

X-position for right and left side of potentials
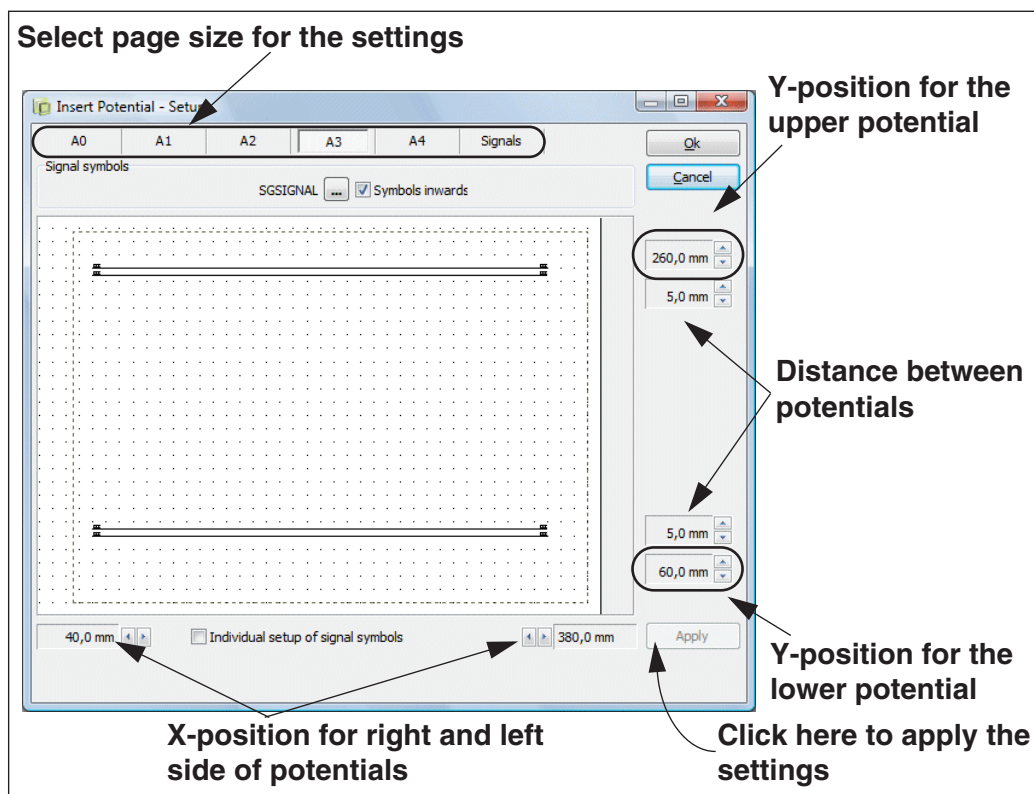
Here you choose — at the top — which page size you wish to edit the settings for, make the changes, click on **Apply**, and then click **OK**.

Symbols inward

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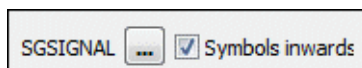
Choosing signal symbols

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:
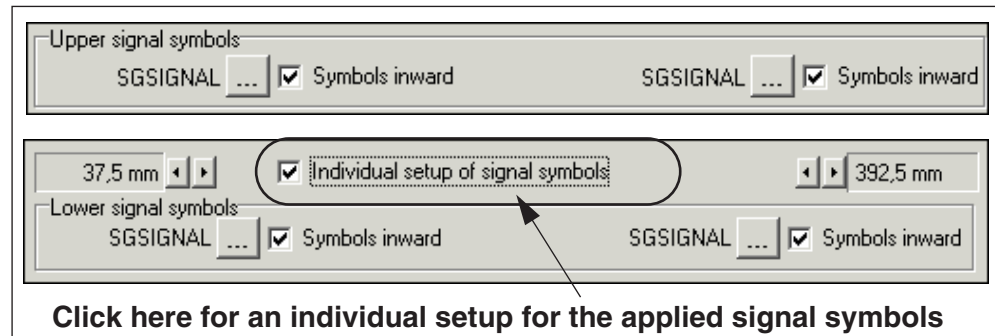
Click here for an individual setup for the applied signal symbols

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 88 —, 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**.
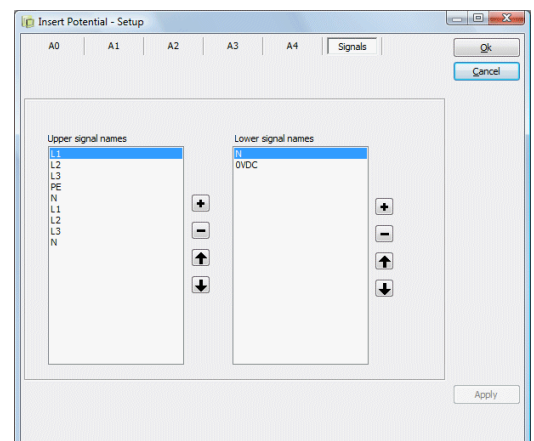
ADDING A SIGNAL NAME

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

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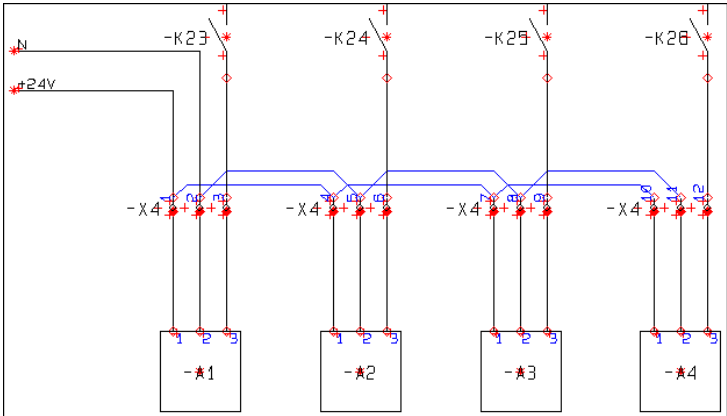
When you start and end drawing a jumper link, a short message is displayed, informing you that you are drawing a jumper link.

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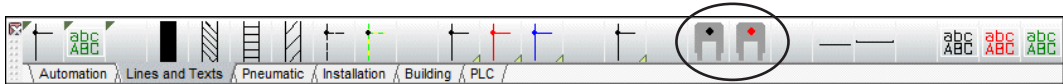
The dot in the middle of the jumper line symbols displays the colour of the jumper lines.

SPECIFYING LINE SETTINGS FOR JUMPER LINKS

UNDER **SETTINGS => BASIC SETTINGS** YOU CAN SPECIFY HOW JUMPER LINKS APPEARS IN PROJECTS. READ MORE ABOUT THIS IN "BASIC SET-TINGS" ON PAGE 938.

— 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.

oner => Vis lasker igen.

Terminals with jumper connection points

Some terminals are physically created with connections specially designed for jumpers. In the terminal symbols for these terminals, jumper connection points can be created with **Connection data** set to *Is jumper*.

To these connection points you are only allowed to draw connecting lines that are jumpers.

Jumper connection points are displayed as triangles on the screen:

JUMPER CONNECTION POINTS FOR TERMINALS

NON-FILLED TRIANGLE: JUMPER *SIDE 1 (INT)*

JUMPER CONNECTION POINTS FOR TERMINALS
FILLED TRIANGLE: JUMPER *SIDE 2 (EXT)*
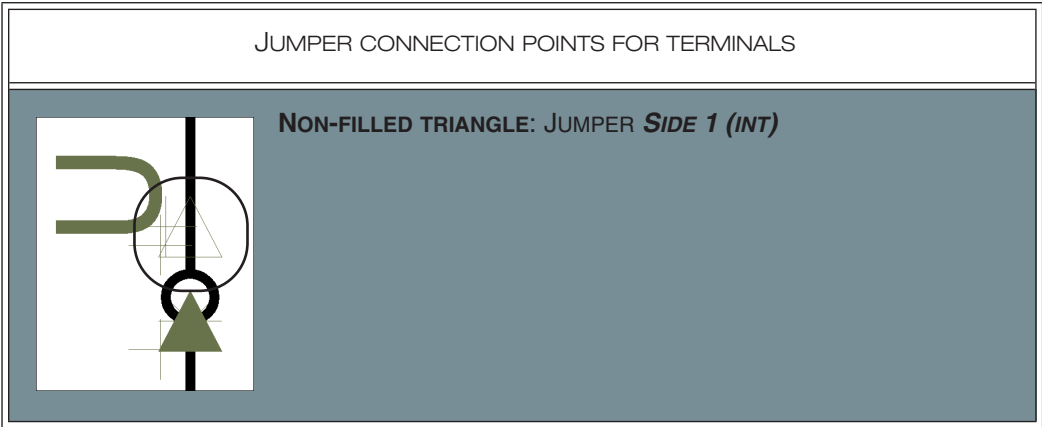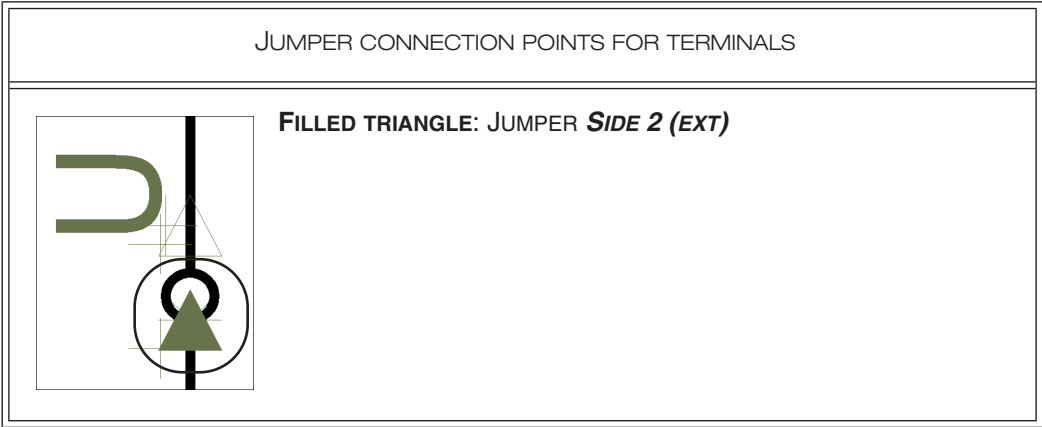
Design check and jumper connection points

When you choose **Functions => Design check**, you can click the **Settings** button and select *Illegal connections (Non conducting/Jumpers)*.

This shows you whether other types of lines than jumpers are connected to the jumper connection points in the project.

— Read more about Design check in "Design Check" on page 365.

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.

There is a similar jumper link datafield for connections lists.

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 80 and "Which data to include in the list" on page 611.

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 **PCSCHEMATIC 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 872.

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:

Point to position to place the Symbol
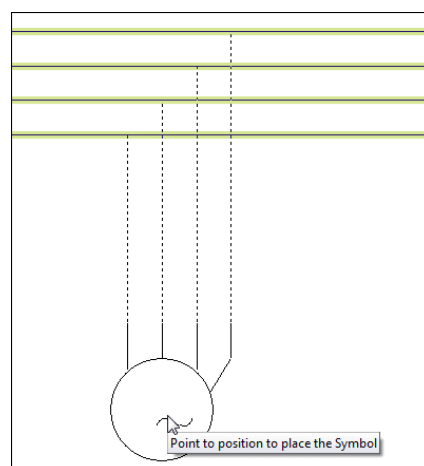
ROUTER =>	FUNCTION
ACTIVE <½>	STARTS AND STOPS THE ROUTER
ONLY NOT CONNECTED SYMBOLS BY MOVE	THE ROUTER IS ACTIVATED WHEN A SYMBOL IS MOVED, IF THE SYMBOL IS NOT CONNECTED.
ONLY IF ALL IN DIRECTION POSSIBLE	THERE WILL ONLY BE ROUTED WHEN ALL CONNECTIONS IN THE CHOSEN DIRECTION CAN BE ROUTED.
ONLY TO NEAREST DIRECTION <2>	THERE WILL ONLY BE ROUTED FROM THE SIDE OF THE SYMBOL, WHICH I 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:
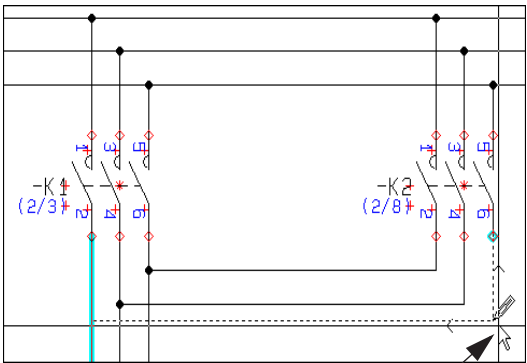
1) Place the cursor here
2) Press the <<> button until th desired line is selected
3) Click to draw the connection

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

Pressing the spacebar changes the direction of the router.

EXTENDED LINES

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

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

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

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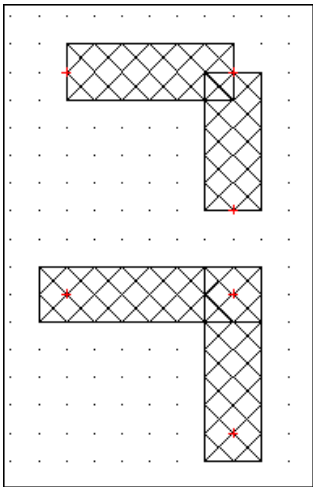
FOLLOW CONNECTED LINES

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

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

A symbol designed using Follow connected, adapt to the last line that was connected to the symbol.
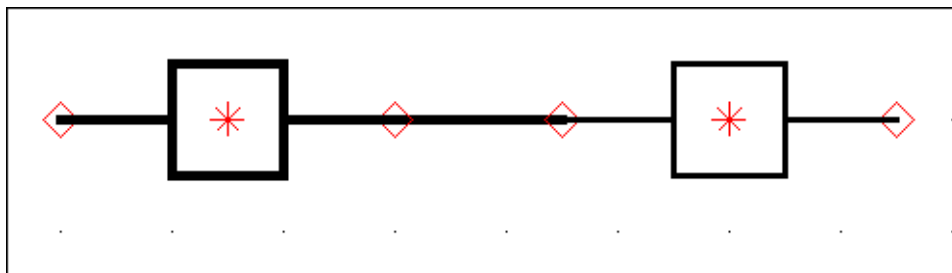
Follow connected can be deactivated as a general setting

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

PC | SCHEMATIC

THE TRIM LINE/EXTEND 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/Extend 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/Extend 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.
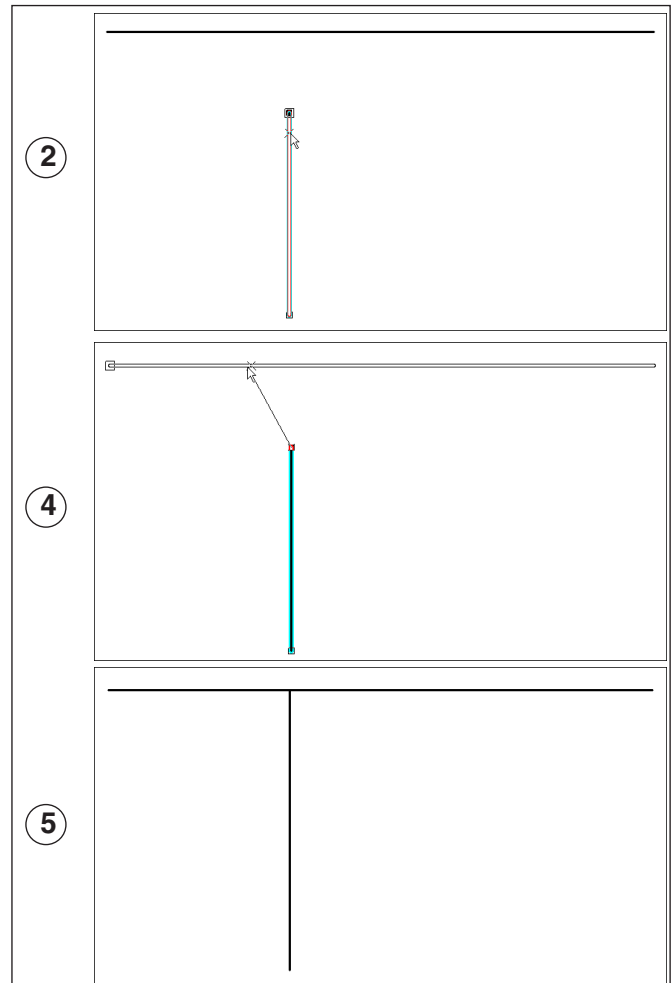
Trimming the selected end

Before applying the **Trim line/Extend line** command, please click closest to that end of the line, which you wish to trim:
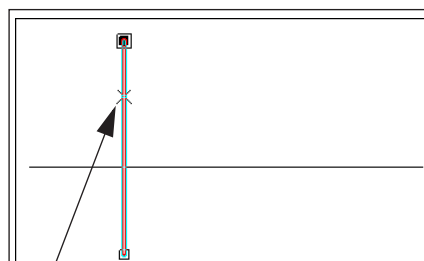
Click here before selecting Trim line, to trim this end
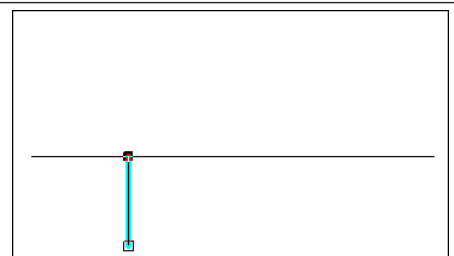
DRAWING BY MEASURES

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.

Example: Drawing a rectangle by measures

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

1) Click on the X-Y coordinate field — or press <Ctrl+i>. Choose **Absolute**.

2) Type *50* in the X field and *100* in the Y field. You do not have to specify that it is in millimetres, due to the system settings. Click **OK**. (This setting can be changed in **Settings => Screen/Pointer**. Read more about this i "Pointer/Screen" on page 918). Click **Next**.

3) Type *150* in the X field and *100* in the Y field. Click **Next**.

4) Type *150* in the X field and *150* in the Y field. Click **Next**.

5) Type *50* in the X field and *150* in the Y field. Click **Next**.

6) Type *50* in the X field and *100* in the Y field. Click **OK**.

7) Press <Esc>. The rectangle is complete.

Using Relative coordinates

1) Start the line by clicking anywhere on the screen.

2) Click on the X-Y coordinate field — or press <Ctrl+i>. Choose **Relative**.

3) Type *100* in the X field and *0* in the Y field. Click **Next**.

4) Type *0* in the X field and *50* in the Y field. Click **Next**.

5) Type *-100* in the X field and *0* in the Y field. Click **Next**.

6) Type *0* in the X field and *-50* in the Y field. Click **OK**.

7) Press <Esc>. The rectangle is complete.

Using the Rectangle command with relative coordinates

1) Click on the **Lines**, the **Rectangle** and the **Pencil** buttons. Deselect Conducting lines.

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

3) Click on the X-Y coordinate field — or press <Ctrl+i> — and choose **Relative**.

4) Type *100* in the X field and *50* in the Y field. Click **OK**.

5) 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*.

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 106.

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 96.

- 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 105.

- 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 104.

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.

Bend size

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.
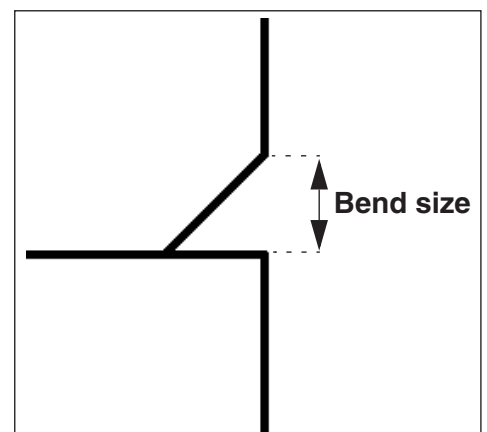
Branch figures

After the bend size, you can choose which brand figure that the program shall suggest initially, when drawing mounting correct:
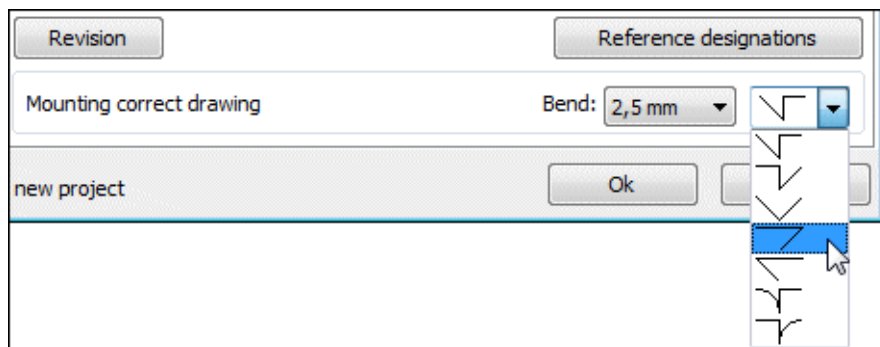
When drawing mounting correct, the program will automatically place the latest applied branch figure. When pressing the <Spacebar> repeatedly before drawing further, you can change to one of the other branch figures.

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.
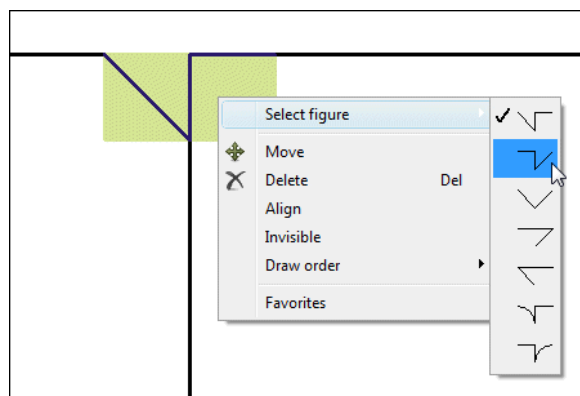
Design check for Mounting correct drawing

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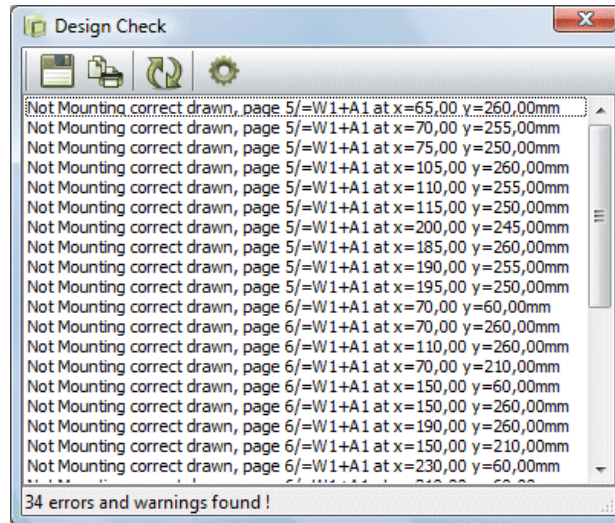
Based on this list, you can correct the project and make sure it is mounting correct. (Please remember that you can click each line in the list, to see where the project is not mounting correct).

5) — Read more about design check in "Design Check" on page 365.

Signal names in Mounting correct drawings

— See "Variant names on signal symbols" on page 239.

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.

PC|SCHEMATIC

Changing to Mounting correct drawing

You can convert diagrams to mounting correct drawing at different levels:

- For individual connections
- For all connections in an area on the page
- For pages
- For projects

Changing to mounting correct drawing - for individual connections, pages and areas

1) Click the **Lines** button in order to work with lines and press <Esc> to deactivate the Pencil.

2) Right-click on the line — or select an area using the mouse and right-click inside the area — and choose **Dot to branch**.

3) Then choose either **The selected** (for individual connections), **In the area** or **On this page**.

4) Branch figures are now inserted at the selected connections.

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

If you wish to replace a branch figure, just click the branch figure and select another figure by pressing the <Spacebar>, as described in "Changing branch figures on lines" on page 104.

Changing to Mounting correct drawing - for projects

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.

PC|SCHEMATIC

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.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO WORK WITH CIRCLES AND ARCS.

— Se 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: 5.0   V1: 0   V2: 360   B: 0.25   F: ■   E: 1
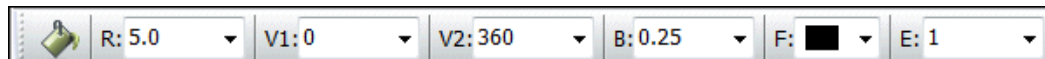
R is for the radius of the circle/arc.

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>.
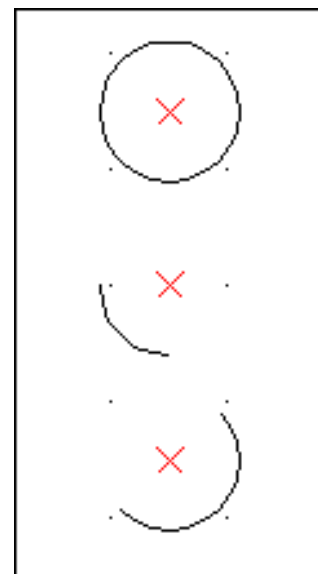
Circles / Arcs in the Pickmenu

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 79.

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.
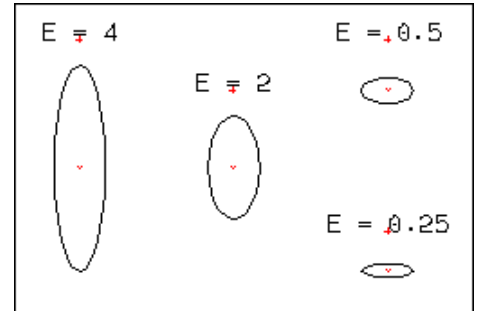
Converting circles and ellipses to lines

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.
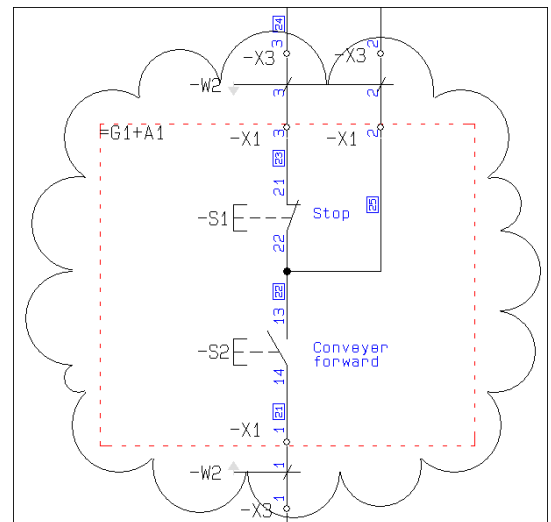
The cloud is now drawn.

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.

— 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.	111
SYMBOL TEXTS	TEXTS, THAT ARE ATTACHED TO EACH SYMBOL, CONTAINING INFORMATION ON THE COMPONENT THAT IS COVERED BY THE SYMBOL.	115
CONNECTION POINT TEXTS	TEXTS, THAT ARE ATTACHED TO EACH OF A SYMBOLS CONNECTION POINTS.	117
LINE TEXTS	TEXTS / ARTICLE DATA ATTACHED TO LINES.	80
DATAFIELDS	TEXT FIELDS THAT ARE FILLED OUT AUTOMATICALLY.	118

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 775) 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 texts is a part of a symbol definition — see "Creating Symbols" on page 775 —, 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 here   **Text type**
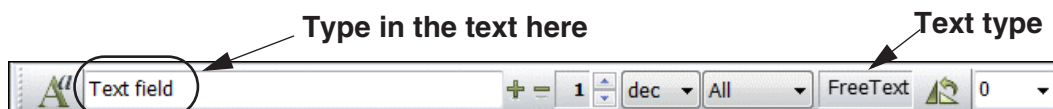
Type in the text as displayed above and press <Enter>. Please note that the toolbar informs you that you are dealing with a free text.

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.
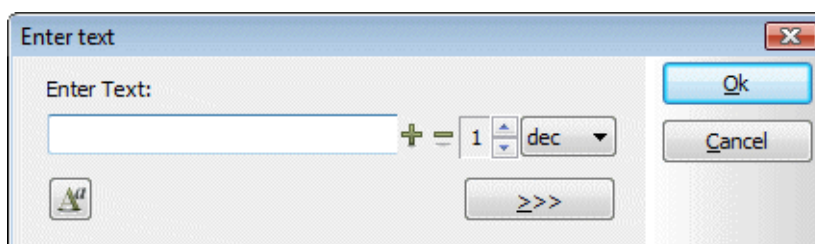
From here it is now possible to use the automatic counting functions, which can be found in the Texts toolbar. — Read more about these in "Counting on texts" on page 134.

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 359.

PC | SCHEMATIC

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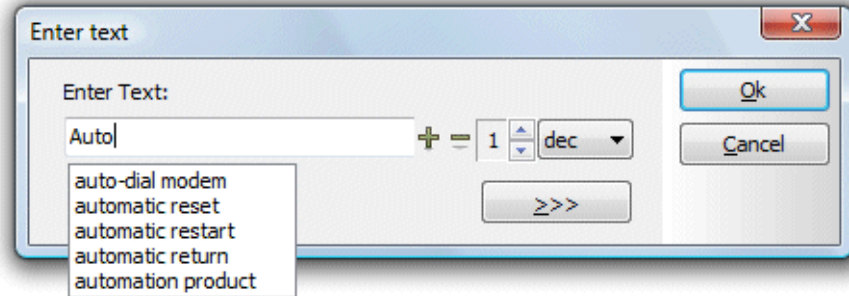
You can then choose between the suggestions in the list, or continue typing in the text.

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>.

Click here and press <Delete>
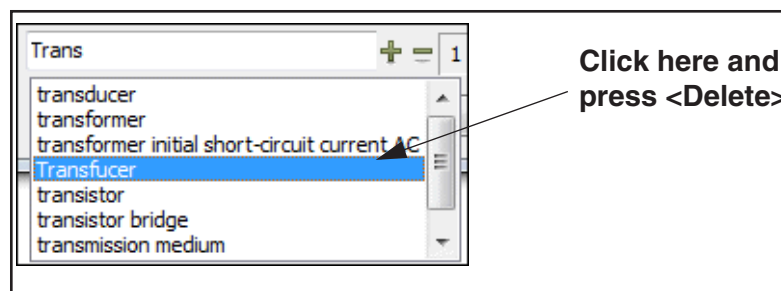
Special characters

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 e.t.c.

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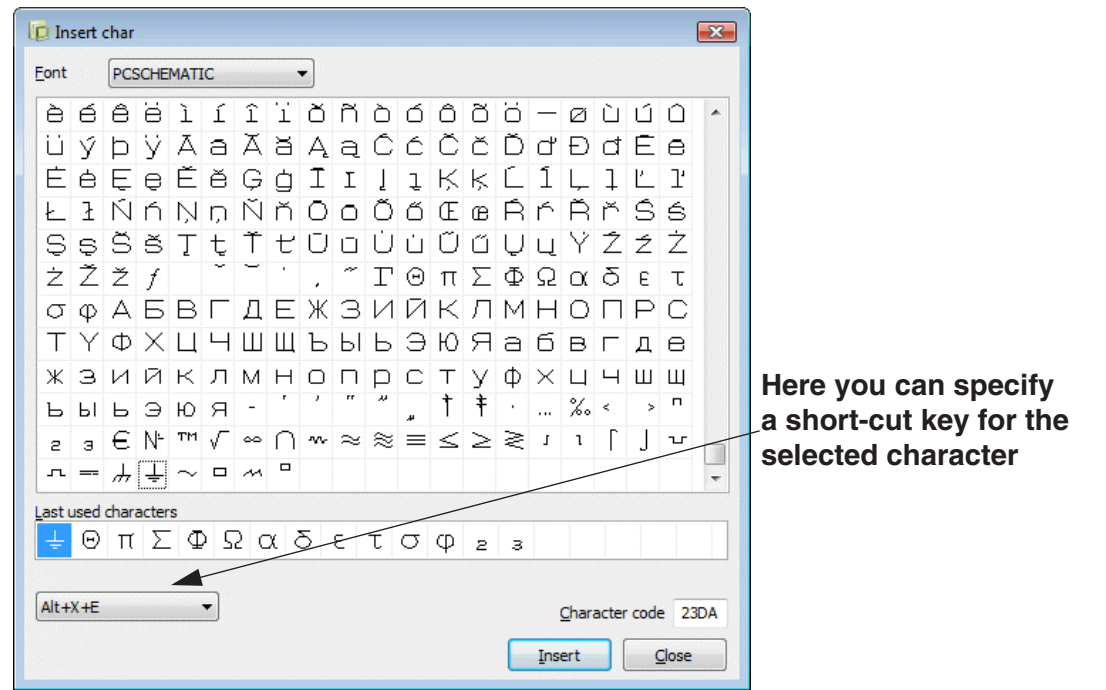
In the dialog box **Insert char** you then select the desired character, and click **Insert**.

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 130, "Negate (invert) text" on page 130 and "Underline text" on page 131.

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 **PCSCHEMATIC** FONT CONTAINS THE NECESSARY SPECIAL CHARACTERS.

Default text properties for free texts

To edit the default text properties (text size e.t.c.) 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:
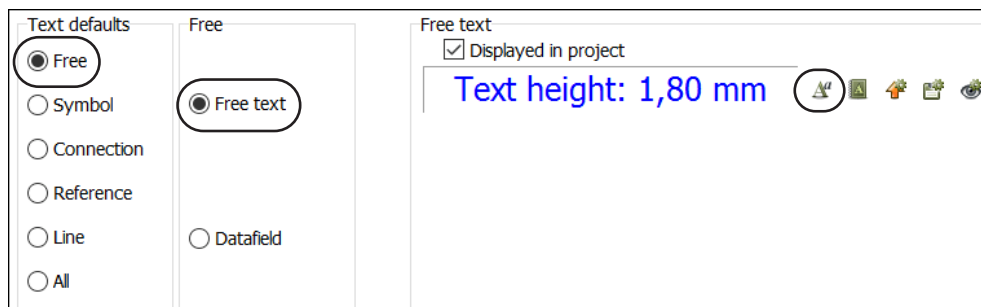
Text defaults	Free	Free text
● Free	● Free text	☑ Displayed in project
○ Symbol		Text height: 1,80 mm
○ Connection		
○ Reference		
○ Line	○ Datafield	
○ All		

3) In the dialog box **Text properties**: Specify the desired text properties, as described in "Specifying text properties / Appearance of text" on page 136, 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**.

Changing free texts in the project or on the page

When you wish to change text settings for free texts in the project — or on the active page in the project — click the **Transfer text settings to the project** button. You then get the option to choose whether to change the settings for **All Pages** or for **This Page**.

Precise line break in free texts

If you wish to specify how line breaks are made in the free texts in the project, please see "Fixed line break on texts" on page 941.

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 775.

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.

You can also create so-called symbol datafields, which can be attached to the symbols. — See "Symbol datafields" on page 122.

FETCHING DATA FROM THE COMPONENT DATABASE

COMPONENT DATA SUCH AS ARTICLE NUMBER AND TYPE CAN BE FETCHED DIRECTLY FROM THE ATTACHED COMPONENT DATABASE, AS DESCRIBED IN "ATTACHING INFORMATION FROM THE DATABASE TO A SYMBOL" ON PAGE 643.

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 918.

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 — for instance by fetching them from the attached component database — 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 174.

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:
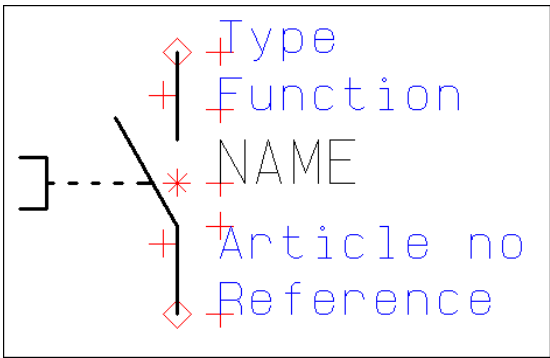
— Please note that references are only displayed when the symbol has references. See "Creating Symbols" on page 775, particularly "Position for references to other symbols" on page 778.

Clicking a symbol text

When you work with **Texts**, have disabled the Pencil (press <Esc>), and click on a symbol text, the Texts toolbar shows which type of symbol text you have clicked on.

To indicate to which symbol the symbol text is connected, a line is displayed going from the reference point of the symbol to the symbol text.

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 this 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 133 for an easier way to change symbol texts.

Double-clicking a symbol text

When you work with **Texts**, have disabled the Pencil (press <Esc>) and *double-click* on a symbol text, you enter the dialog box **Component data**, where you can edit the text.

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 115 and "Symbol Connection points" on page 212.

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 data and Page data" on page 128.

— Please refer to "Page data and Project data Datafields" on page 769 for further details.

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

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

Explanatory text about the datafield

The selected field type

Supplementary information

The selected datafield

Click here for drop-down menu
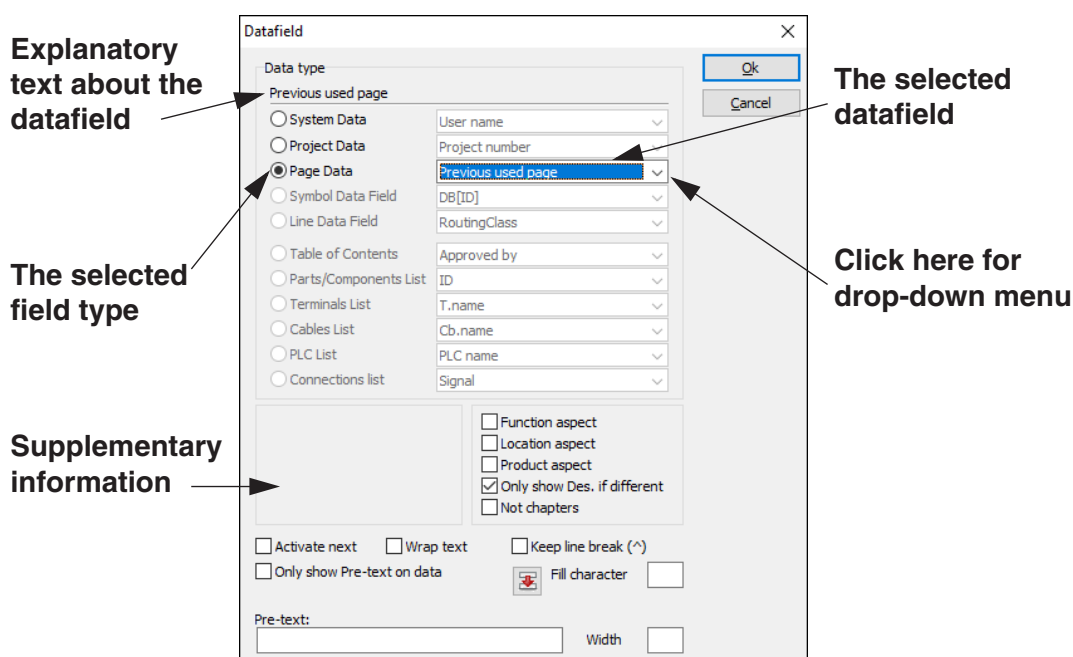
The various groups of datafields have the following contents:

DATAFIELD	CONTENTS
SYSTEM DATA	DATA ABOUT YOUR SYSTEM. FOR INSTANCE *USER NAME* WHICH IS THE NAME OF THE PERSON THAT IS TYPED IN AS USER IN **SETTINGS => SYSTEM/LICENSE** ON THE **SYSTEM** TAB. *DATE* AND *TIME* 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 DATAFIELD (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS YOU CAN DEFINE BY YOURSELF. — READ MORE ABOUT THESE IN "CREATING DATA SYMBOLS" ON PAGE 810. 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.
LINE DATAFIELD (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS FOR LINES, WHICH YOU CAN DEFINE YOURSELF — SEE "LINE DATAFIELDS" ON PAGE 83.
TABLE OF CONTENTS (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS FOR TABLES OF CONTENTS.
PARTS/COMPONENTS LISTS (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS FOR PARTS- AND COMPONENTS LISTS.
TERMINALS LIST / CABLE LIST (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	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 (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS FOR PLC LISTS.
CONNECTIONS LIST (ONLY AVAILABLE WHEN EDITING DRAWING HEADERS / LISTS)	DATAFIELDS FOR CONNECTIONS LISTS.

— See the survey of datafields in "Datafields in Automation" on page 944.

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 980.
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.

FIELD	FUNCTION
THE RED ARROW	TRANSFERS THE SELECTED DATAFIELD TEXT TO THE *TEXT* FIELD.
FILL CHARACTER	A CHARACTER THAT WILL BE FILLED IN FROM THE END OF THE TEXT AND IN THE REMAINING WIDTH OF THE DATAFIELD. FOR INSTANCE A LINE OF DOTS IN A TABLE OF CONTENTS BETWEEN THE PAGE TITLE AND THE PAGE NUMBER. (3) ON THE FOLLOWING FIGURE.
TEXT	HERE YOU CAN TYPE IN A TEXT TO PRECEDE THE CONTENTS OF THE DATAFIELD. (1) ON THE FOLLOWING FIGURE.
WIDTH	THE MAXIMUM WIDTH OF THE DATAFIELD (NUMBER OF CHARACTERS). — PLEASE ALSO SEE "FIXED LINE BREAK ON TEXTS" ON PAGE 941.

Two inserted datafields could for instance look like this:

The text in the Text field (1)

The name of datafield itself (2)

Name of customer: Name of cust_omer

Title

Fill characters (3)
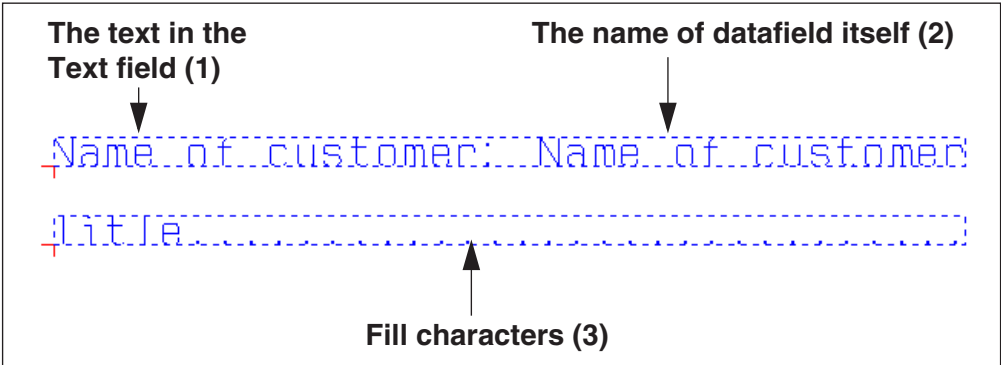
Please note that this is how datafields look like when you are in Edit symbols — for instance because you want to create a list (Read more about creating lists in "Creating Lists" on page 831). If you just insert a datafield on a page, you will not see the *name* of the datafield itself (2), but the *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.

PC|SCHEMATIC

INSERT	DESCRIPTION
%.1F	INSERTS THE CONTENTS OF THE DATAFIELD AS FLOATING POINT WITH ONE DECIMAL. YOU CAN CHOOSE FROM 0 TO 9 DECIMALS. IF YOU WRITE "PRICE: %.2F $"— AND THE CONTENTS OF THE DATAFIELD IS 25 — THIS RESULTS IN THE TEXT "PRICE: 25,00 $" IF YOU WRITE "%.3F KG." — AND THE CONTENTS OF THE DATAFIELD IS 17,65 — THIS RESULTS IN THE TEXT "17,650 KG."
%-	NO DATAFIELD VALUE IS INSERTED, AND THE TEXT IS ENDED RIGHT BEFORE THE % CHARACTER. IF FOR INSTANCE YOU WRITE "TOTALPRICE:%" THE TEXT "TOTALPRICE:" IS INSERTED IF THE CONTROLLING DATAFIELD HAS A CONTENT. 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 $" SELECT *ONLY SHOW TEXT ON DATA* FOR BOTH DATAFIELDS.

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 %".

Symbol datafields

All symbols have attached the symbol texts **Name**, **Type**, **Article no** and **Function**. When you wish to attach other information to the symbols, you can create so-called symbol datafields, which can contain this information.

Below you can see how to create these symbol datafields and how to attach the symbols datafields to symbols in the project.

This chapter describes
— How to create symbol datafields ....(p. 122)
— Symbol datafields with attached list of values ....(p. 123)
— How to add datafields to symbols..(p. 125)

How to create symbol datafields

To create symbol datafields, do this:

1) Choose **Settings => Text/Symbol defaults** and click on **Symbol data fields**.
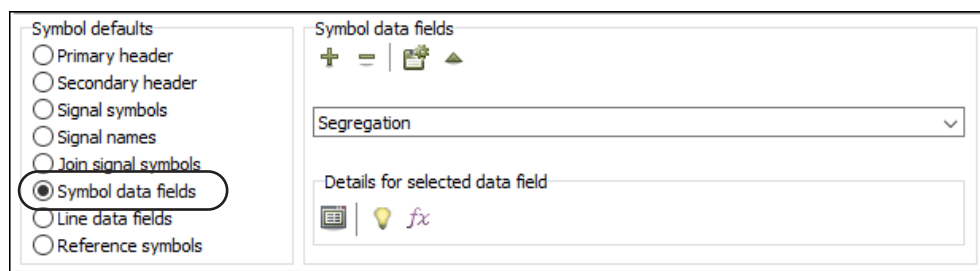
2) Click the **Add datafield** button.

3) In the dialog box **Enter text**: Type in the name of the symbol datafield and click **OK**.

4) You have now created the symbol datafield in the project. Click the **Save as default** button, if you also need to use the symbol datafield in other projects.

5) Click the **Add datafield** button again to add more symbol datafields.

You can also create a list of predefined values for the datafields, as described in "Symbol datafields with predefined values" on page 123.

Click **OK** to exit the dialog box **Text/Symbol defaults**.

Creating compound symbol datafields (Formula editor)

You can create compound symbol datafields, just like you can for line datafields. The formulas can be created based on symboldata, symbol datafields and plain text.

Read more about this in "Calculated line datafields (Formula editor)" on page 83.

LINE DATAFIELDS CREATED THE SAME WAY
WHEN YOU WISH TO CREATE LINE DATAFIELDS, THIS IS DONE THE SAME WAY. THE ONLY DIFFERENCE IS THAT YOU HAVE TO CHOOSE **LINE DATA FIELDS** IN STEAD OF **SYMBOL DATA FIELDS**.

How to delete symbol datafields

To delete a created symbol datafield, do this:

1) Choose **Settings => Text/Symbol defaults** and click on **Symbol data fields**.

2) Click on the down-arrow in the right-hand side of the dialog box, and select the symbol datafield that you wish to delete:
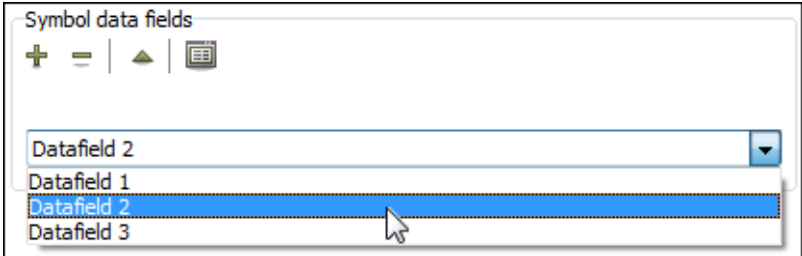
3) Click the **Delete** button.

Symbol datafields with predefined values

Symbol datafields can have an attached list of predefined values for the datafield.

To create such a data list, choose **Settings => Text/Symbol defaults** and click on **Symbol datafields**. Then choose the relevant symbol datafield and click the **Values** button.
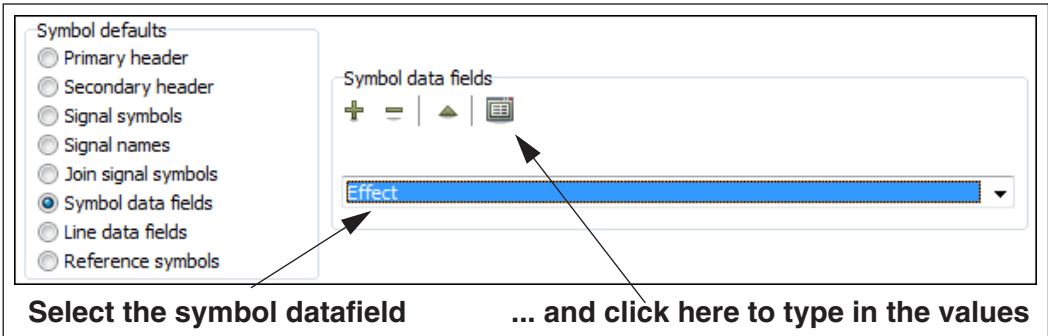
You then enter the **Values** dialog box.

The Values dialog box

In the **Values** dialog box you can maintain a list of which values/texts a specific datafield (such as for instance line datafields, tab titles as well as page and project titles — and in this case symbol datafields) can contain:

Type in the values | **Move up/down in the list**

Sort the values | **Select a value and Set as default**
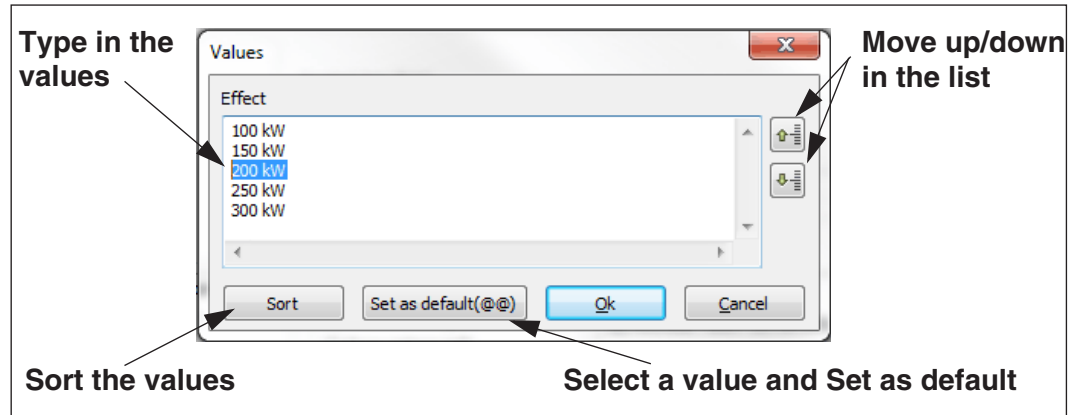
TYPE IN VALUES/TEXTS

To type in the values, click inside the **Values** field and type in the values/texts individually and press <Enter> after each value, to ensure that there is only one value on each line.

DELETE TEXTS/VALUES

To delete a value: Select the value in the **Values** field (double-click on the value) and press <Delete> or <Backspace>.

SET AS DEFAULT

When you select a value in the list and click the **Set as default** button, the selected value becomes the default value for the datafield. When a value is set as default, this is indicated by two @ characters in front of the value in the **Values** dialog box.

Click **OK** when the list is complete.

— You can also see "Project and Page data: List of values for a series of datafields" on page 772.

Lock data list for symbol datafield

When you have created a data list for a symbol datafield, you have the option of locking the data list. You can do this by clicking the lock icon to the right under **Symbol datafields**.
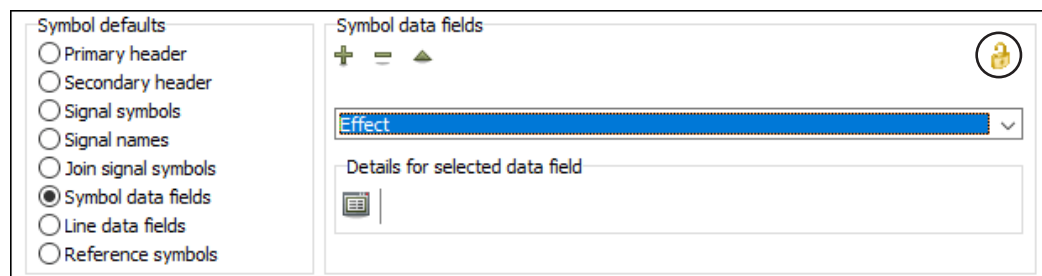
When the icon is displayed as a locked lock, it is only possible on project pages to select the values specified in the data list.

When the icon is displayed as an open lock (as above), it is also possible to type in your own values.

PC|SCHEMATIC

The lock icon is only visible when a data list is created for the selected datafield.

> **MAKE VALUES ACCESSIBLE FOR THE ENTIRE COMPANY**
> UNDER **SETTINGS => DIRECTORIES** YOU CAN SPECIFY WHICH FOLDER TO SAVE THE LIST OF VALUES IN. WHEN YOU WORK ON A NETWORK, THE CREATED LISTS OF VALUES CAN BE MADE AVAILABLE FOR THE ENTIRE COMPANY, WHEN THESE LISTS OF VALUES ARE SAVED ON THE SERVER.
> — SEE "DIRECTORIES" ON PAGE 923.

How to add datafields to symbols

To add extra datafields to a placed symbol, do this:

1) Activate the **Symbols** button and double-click on the symbol to enter the dialog box **Component data**.

2) In the dialog box **Component data**: Click the **Symbol data fields** tab.

3) On the **Symbol data fields** tab: Click **Add**. (See how to create symbol datafields in "How to create symbol datafields" on page 122).

4) In the **Datafield** dialog box: Select which datafield to attach to the symbol. Here you can choose the symbol datafields that you have created yourself as well as the datafields from the attached database. Datafields from the database are written in capital letters.

5) Select how to fetch the data:
   From component: Data is fetched from the component
   From symbol: Data is fetched from the symbol
   Symbol else component: Data is fetched from the symbol, but if no specific data is specified for the symbol, data is fetched from the component

6) Click **OK**.

The selected datafield is now attached to the symbol, and you can type in a value for the datafield.

When the datafield has a defined default value, this value is already specified for the datafield. If a list of values exists for the symbol datafield, you can choose among the values in the list like this:

1) Click inside the field

2) Click here, and the list i displayed

3) Select the desired value

Edit settings for the symbol datafield   **Remove the symbol datafield**
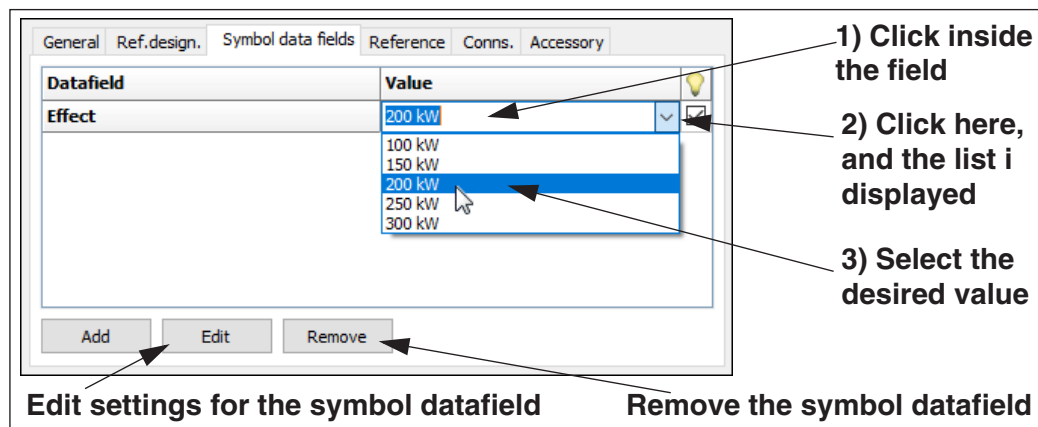
The specified values in the data list are only suggested values, so you can also type in another value in the datafield.

But if the datalist for the datafield is locked — see "Lock data list for symbol data-field" on page 124 — a lock icon is displayed next to the datafield, and you are only allowed to select one of the specified values in the list.
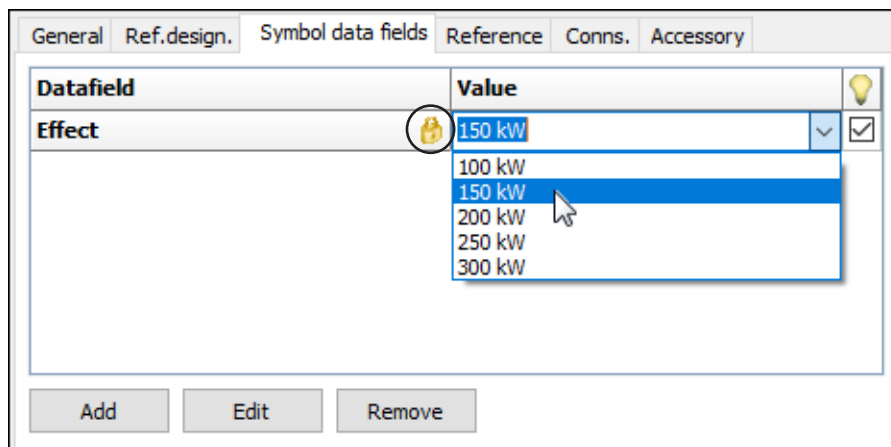
Predefined symbol datafields

Beyond the symbol datafields that you have created by yourself, there are also some predefined symbol datafields, which can always be added when you click the **Add** button. These datafields are:

- **=+-design.**: The full reference designation for the symbol.

- **Description**: The description for the reference designation.

- **Quantity**: The contents of the **Quantity** field in the tab **General** in the dialog box.

- • **_Segregation_**: The field is created so that you can assign it to cables that you want to export to **PCSCHEMATIC Cablemanager**. — Read more about this in "Export to PCSCHEMATIC Cablemanager" on page 748.

When you select **_Visible_** field next to the symbol datafields, they will be displayed on the project page next to the symbol.

PROJECT DATA 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** and for **Page settings**.

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 769.

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 820, and how to design your own lists in "Creating Lists" on page 831.

Values 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 enter the dialog box **Values**, where 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

PC | SCHEMATIC

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.

— Read more about this in "The Values dialog box" on page 124.

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 112.

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.
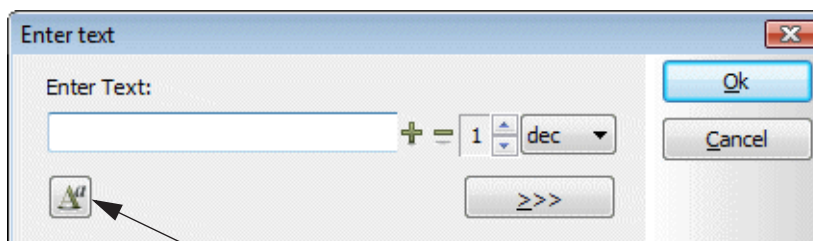
Here you can change text properties

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 138 and "Wrap text" on page 138.

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 PCSCHEMATIC FONT TO A WINDOWS FONT" ON PAGE 137.

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 PCSCHEMATIC FONT TO A WINDOWS FONT" ON PAGE 137.

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**.

Texts with text leader

To insert a text leader on a text, do this:

1) Choose **Texts** and disable the Pencil by pressing <Esc>.

2) Right-click the text and choose **With text leader**.

3) A line is now displayed from the text to the cross-hairs: Click where you want the text leader to point.

4) If you do *not* want an arrow head on the text leader: Right-click on the text again and choose **Without arrow**.

The text leader line is now drawn. If you want to reposition the text leader arrow, just — while still working in **Texts** — click on the arrow and then *drag* it to its new position.

To remove the text leader: Right-click on the *text* and choose **Without text leader**.

Information on texts

To get information about a text, do this:

1) Choose **Texts** and disable the pencil by pressing <Esc>.

2) Right-click on the text and choose **Information**.

3) You now enter the dialog box **Information**, which contains information about the text.

```
Information                           [x]

 Text: Start
 Text type: FreeText
 Font: (0) PCSCHEMATIC
 Color: 4
 Height: 1,8
 Width: 1,2
 Position: x=55,000mm y=115,000mm z=0
 Angle: 0,0
 Layer: 1
 Alignment: 0
 Bold: No
 Italic: No
 Visible: Yes
 Fixed: No
 Line length: 255
 Wrap text: No
 Bytes: 78 (5)

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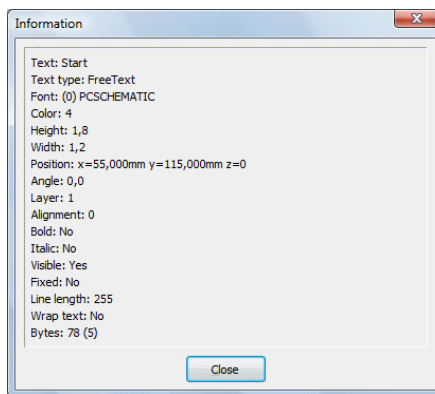
Here you find information about the text height, width, position, the used font, line length of the text, wrap text information e.t.c.

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.
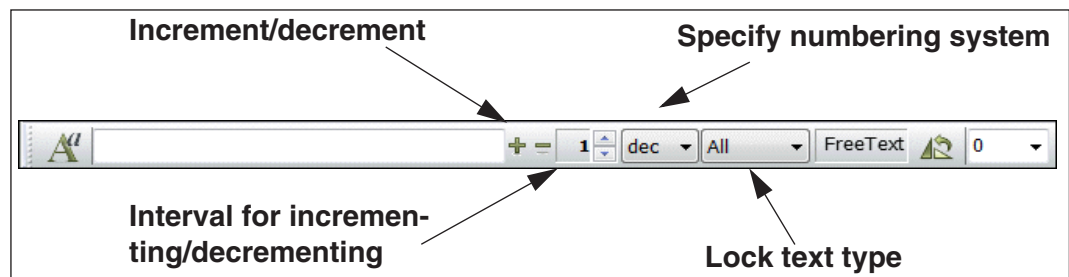
Locking a text type

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.
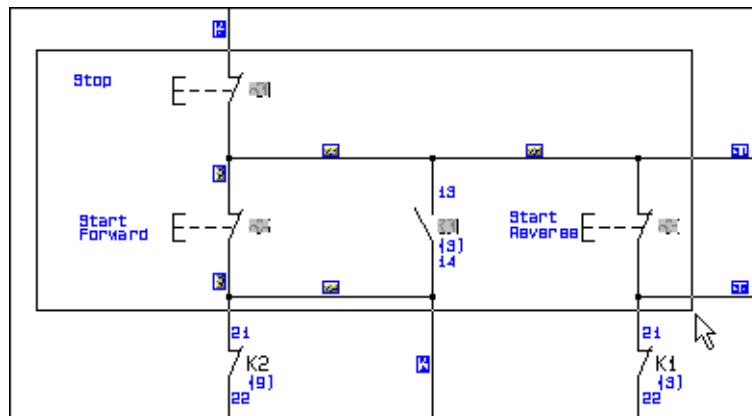
When you have locked the type to **Symbol name**, you can then assign a name to a symbol by typing in the name in the text field in the texts toolbar, press <Enter>, and click *on the symbol*.

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 133.

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 (se 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.

Click in the left-hand side first when you select the area, if you want the counting to be performed from left to right
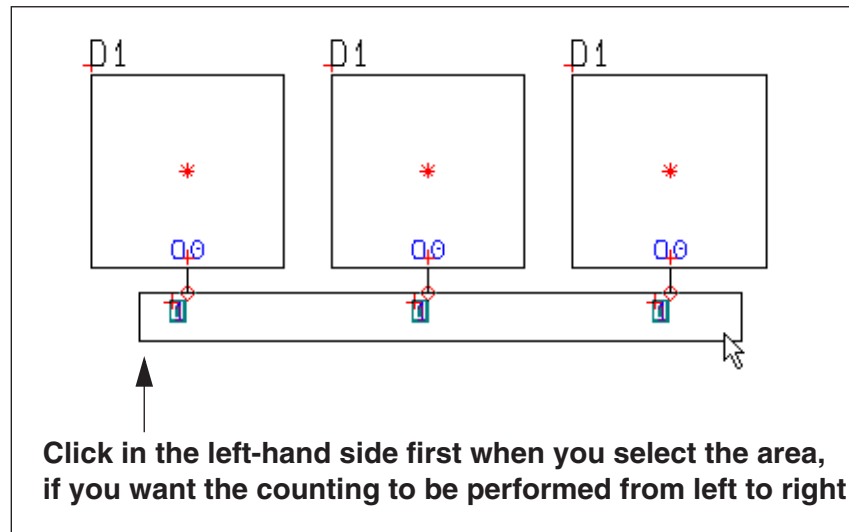
If the counting should be carried out starting from the left-hand side, you therefore start by clicking in the left-hand side, when you select the area.

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 210 to see alternative ways to apply automatic counting on symbol names, and to "Choosing connection points in Areas" on page 217 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 872.

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.

Font →
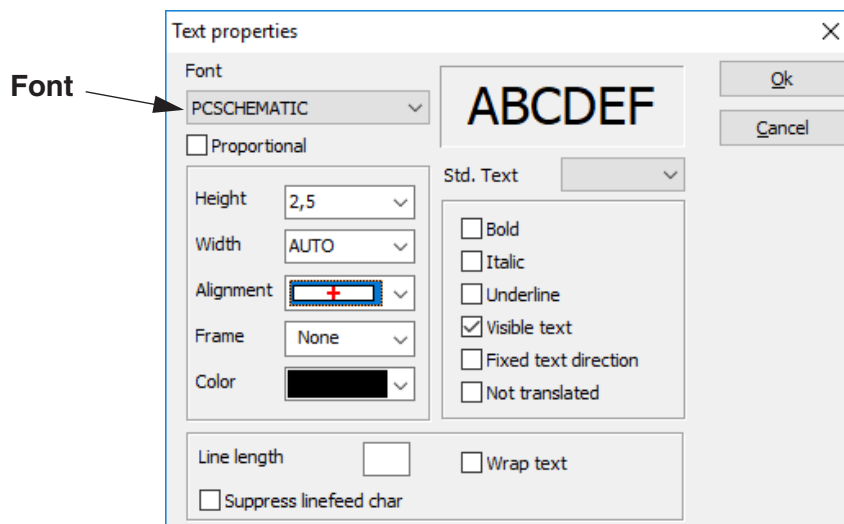
When changing the various text settings, you click the down-arrow at the left hand side of the dialog box, and click on your choice in the drop down box that emerge.

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 140.

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 **PCSCHEMATIC** 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 **PCSCHEMATIC** font follows the EN standards, and should therefore be used on diagrams.
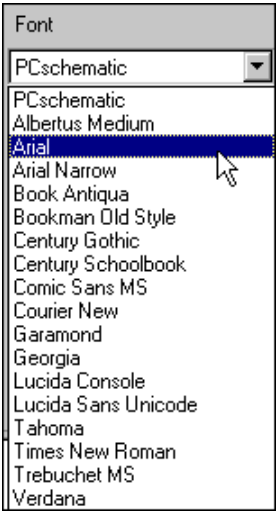
Special characters when changing the PCSCHEMATIC font to a Windows font

If you change texts written with the **PCSCHEMATIC** 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 376 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 **PCSCHEMATIC** 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 fills more than a single line, the reference point is positioned relative to the entire text block.

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 137.
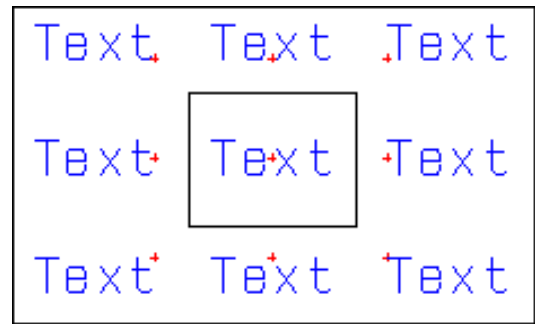
Shortcut for changing text alignment

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** (<u>N</u>on <u>P</u>rintable), you will see the text on the screen, but not on print.

— See "Visible text" on page 139 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 478.

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 147.

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 144.

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 **PCSCHEMATIC Oversætter**.

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 934 for further details on the default settings for texts.
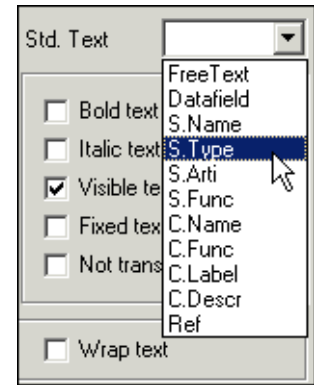
Texts in the Pickmenu

To get fast access to drawing texts with predefined properties — see "Specifying text properties / Appearance of text" on page 136 —, you can place texts with the desired properties in the pickmenu at the upper part of the screen.

Symbols/drawing objects displayed in all pickmenus

Select different menus in the pickmenu

Click on a text, and place texts with the specified properties
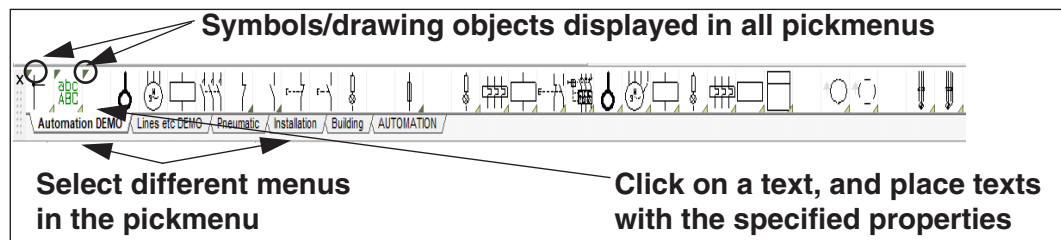
When you click on one of the *texts* in the pickmenu, the program automatically changes to working with **Texts**, and the cursor is placed in the text-field in the texts toolbar.

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 155 and "Adapting the Pickmenu" on page 203.

TEXTS DISPLAYED IN THE PROJECT (GENERAL SETTING)

Entering **Settings => Text/Symbol defaults** you can — as a general project set-
ting — 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.

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 num-
ber** 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

Transferring text settings to project texts

If you wish to transfer the text settings for the selected text type to the project — or to the active page in 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 defaults**. You are then asked whether to apply this setting for **All Pages** or for **This Page**.

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*:
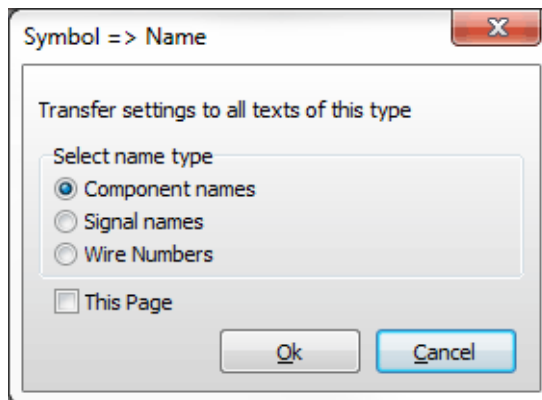
If you select **This Page** the settings are only transferred to this page — if not, the settings are transferred to the project.

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.

PC|SCHEMATIC

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 136.

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.

Corner 3	Corner 2	Corner 3	Corner 2
Corner 0	Corner 1	Corner 0	Corner 1
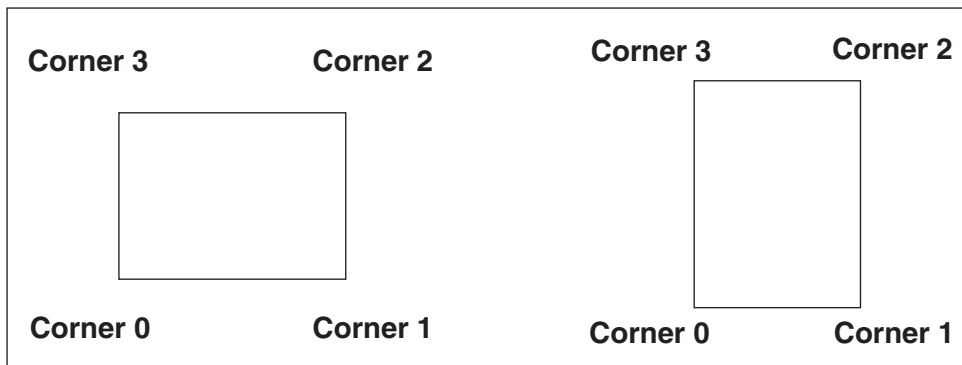
Whether the paper is in portrait or in landscape does not change this.

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.
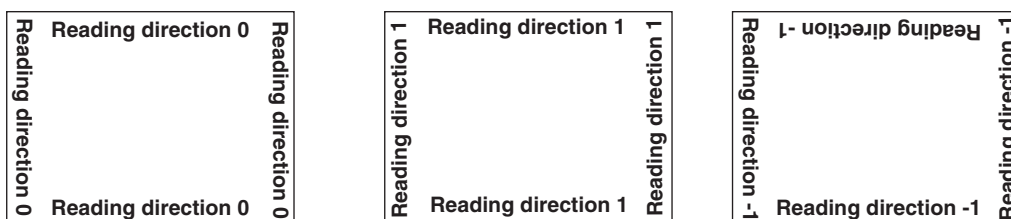
If you have got a drawing in landscape, which after printing have to be inserted into a portfolio, you have to set the reading direction to **0**.

Individual texts with fixed reading direction

You also have the option of specifying a fixed reading direction for individual texts. 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 136.

PC|SCHEMATIC

Transferring the page reading direction to other project pages

When you wish to transfer the reading direction for a page to other pages in the project, choose **Settings => Page settings** and click on the **Transfer setting** button on the right-hand side of the *Reading direction* field. You then get the following options:

- *Select pages to transfer settings to*: You now enter the **Page menu** where you select which pages to transfer the reading direction to and click **OK**.

- *Transfer setting to all pages of same page format*: The reading direction is transferred to all pages with the same page format.

- *Transfer setting to all pages*: The reading direction is transferred to all pages in the project.

Position of names when placing symbols

THIS ONLY APPLIES FOR SYMBOLS OF THE TYPE NORMAL

PLEASE NOTE, THAT THIS FUNCTION ONLY APPLIES FOR SYMBOLS OF THE TYPE *NORMAL*.

When symbols are rotated, symbol and connection names can be positioned in two different ways:

a) Names are not rotated, and they are positioned above or to the left of the symbol, or

b) Names are rotated with the symbol
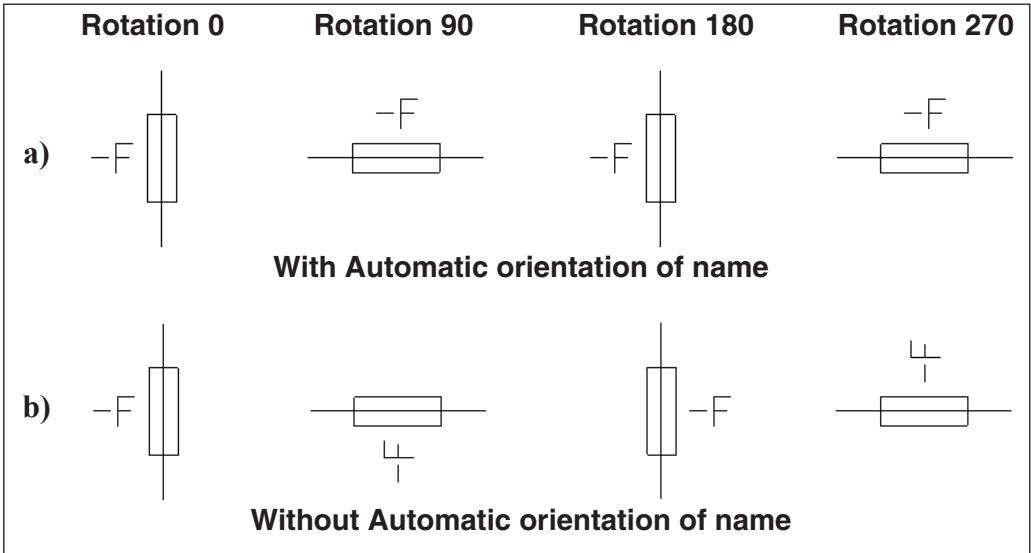
How names are positioned in the project, is specified like this:

1) Choose **Settings => Text/symbol defaults**

2) Below **Text standards**: Select *Symbol* or *Connection*

3) Select — or deselect — *Automatic orientation of name*

The setting only applies for symbols subsequently placed in the project. Already placed symbols are not affected.

By default, *Automatic orientation of name* is selected for both symbol and connection names.

ROTATION: THE RELATION BETWEEN TEXT AND SYMBOL IS FIXED
WHEN YOU ROTATE A SYMBOL, THE SYMBOL TEXTS WILL BE ROTATED WITH THE SYMBOL.

TEXT LINKS

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 151.

This is also the case when importing PLC I/O data — see "Text links and importing PLC I/O lists" on page 151.

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:
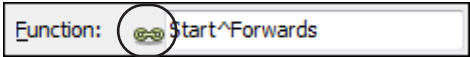
This is also the case in the **Object Lister** — see "The Object Lister" on page 351.

In diagrams linked texts are indicated by an arrow drawn through the reference point of the text:
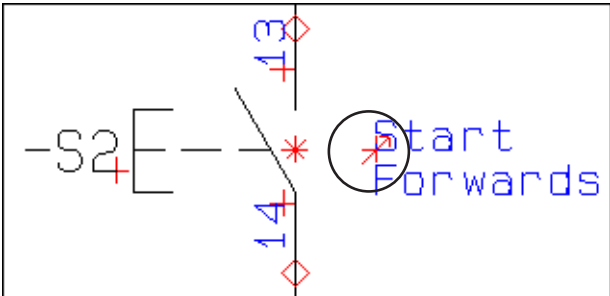
Creating a text link from the Texts command

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**:
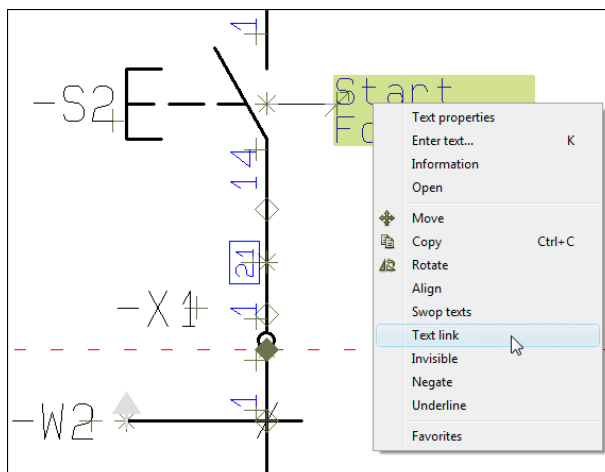
2) The dialog box **Text link** is displayed: Click on a new text in the project, and click on the **Add text to link** button to link the new text to the text in the dialog box:
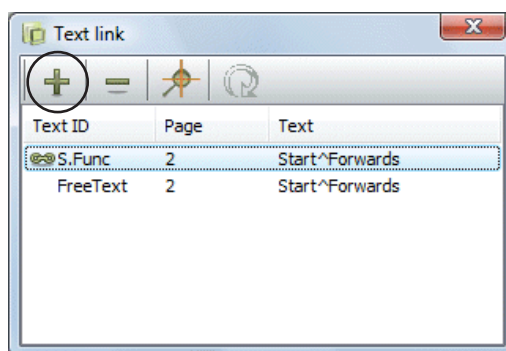
3) Continue adding texts to the text links until all relevant texts are linked.

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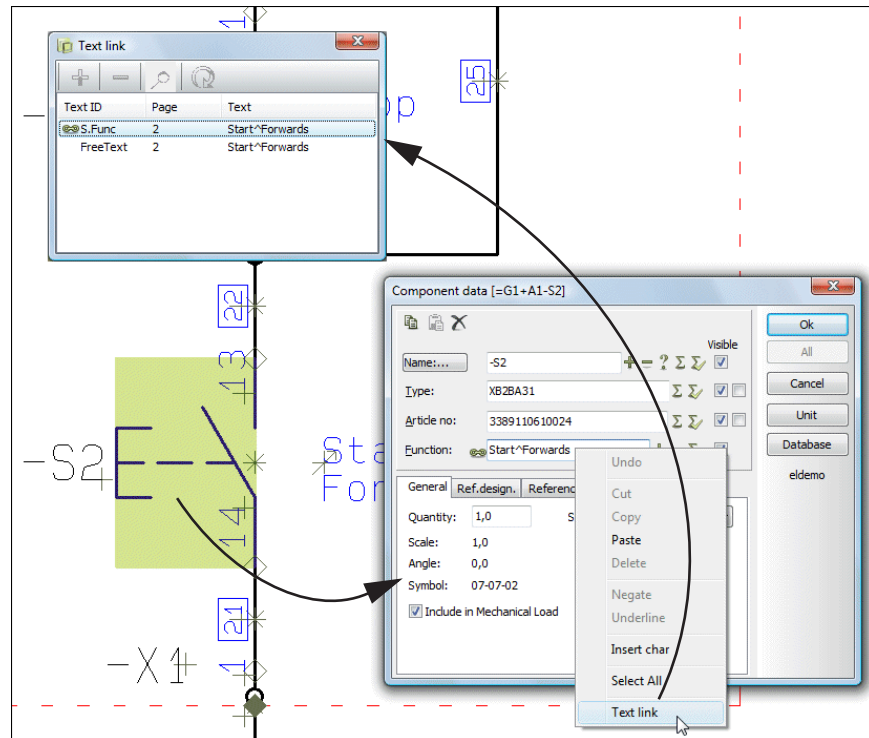
4) You now enter the dialog box **Text link**, from which you can link texts. — If the dialog box **Text link** is already open, the selected text is added to the active link list in the dialog box **Text link**).

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 147.

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.

PC|SCHEMATIC

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:
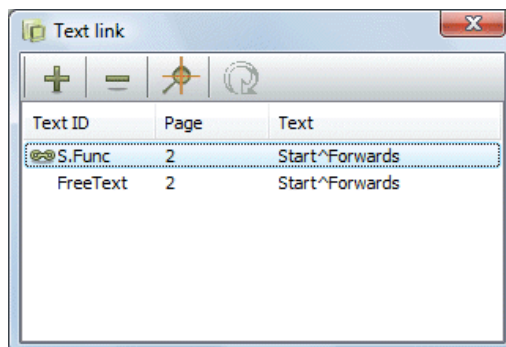
Here you have the following options:

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 147, 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 351.

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 497 — 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 148

- From the **Texts** command — see "Editing text links" on page 150

- From the **Object Lister** — see "The Object Lister" on page 351

- During import — see for instance "Text links and importing PLC I/O lists" on page 151

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 show components — or *functions in components* — in the diagrams in your projects.

Component based workflow

Automation has a *component based workflow* which encourage working with an attached component database, containing information about the components that you can use in your projects.

You can also draw - and attach article data later

But you also have the option to just place symbols in the active project and then attach article data to them at a later stage.

If you just want to place a lamp, you can just find the symbol for a lamp and insert it on the relevant diagram page — and attach article data at a later stage, if necessary. — See "Attaching information from the Database to a Symbol" on page 643.

Components with multiple functions - via the database

When you are about to place a component containing multiple functions, you can fetch it from the attached component database. This gives you a pickmenu containing a symbol for each of the functions in the project — with automatically attached article data.

Functions at symbol as well as component level

Automation contains functions for both working at symbol and component level. This chapter describes how work with symbols in **Automation**.

This chapter predominantly describes working with *electrical symbols*. If you want to know more about working with *mechanical symbols*, please see "Mechanical pages and symbols" on page 701.

> **FUNCTIONS ON COMPONENT LEVEL**
>
> MORE SYMBOL FUNCTIONS ARE DESCRIBED IN "FUNCTIONS ON COMPONENT LEVEL" ON PAGE 259, WHERE YOU ALSO WILL FIND A DESCRIPTION OF THE **COMPONENT MENU** IN THE RIGHT-HAND SIDE OF THE WORK AREA.
>
> THE RELATIONSHIP BETWEEN COMPONENTS AND SYMBOLS ARE DESCRIBED IN "THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS" ON PAGE 260.

Options for symbols described in other chapters

You can learn more about working with symbols here:

- See how to move, copy, delete and rotate symbols in the section "Working with selected objects" on page 58.

- 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 475.

- See "Creating Data symbols" on page 810 for information about how to use data symbols.

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 775, to see how you design your own symbols. — Also see "Automatic generation of symbols" on page 226.

FETCHING SYMBOLS

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.

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

WAY TO FETCH SYMBOLS	PAGE
FROM THE PICKMENU	155
FROM THE SYMBOL MENU	162
FROM THE DATABASE	170
TYPING IN A SYMBOL FILENAME	171
TYPING IN AN ARTICLE NUMBER	172
TYPING IN A SYMBOL TYPE	173
DIRECT GENERATION OF SYMBOLS	226
FROM WINDOWS EXPLORER	381
USING A BARCODE SCANNER	700

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 96.
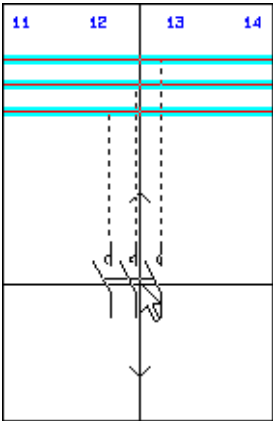
1. Fetching symbols from the Pickmenu

When working in **Automation**, you have the **Pickmenu** available on the upper part of the screen. Here you place the symbols, components, sub drawings and other drawing objects that you use the most.

Symbols/drawing objects displayed in all pickmenus

Select different menus in the pickmenu

Click on a symbol, and place it in the diagram
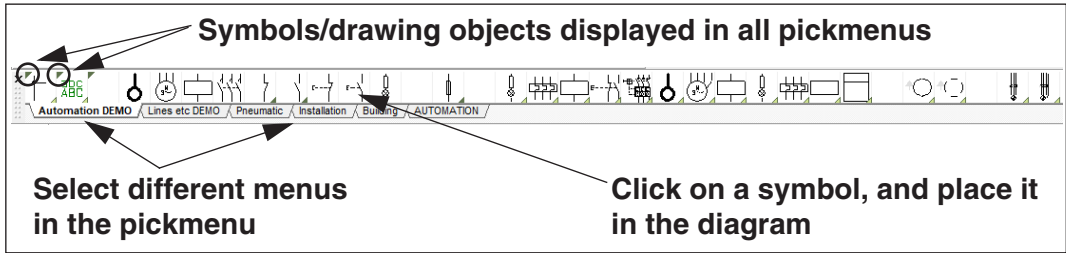
When you click on a *symbol* in the pickmenu, you get the symbol in the cross hairs, and can place it in the project. The program changes automatically to work with **Symbols**, and the **Pencil** is activated.

When you choose a symbol with attached article data (a *component*), the symbols for the component are displayed under **Available symbols** in the lower right-hand corner of the screen, if the component has more than one attached symbol. — See "Components in the Pickmenu (Symbols with attached article data)" on page 156.

When you choose a sub drawing, you get the sub drawing in the cross hairs, and can place it in the project. — Read more in "Sub drawings in the Pickmenu" on page 160.

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

OTHER POSSIBILITIES WHEN USING THE PICKMENU
— CHANGING BETWEEN MENUS IN THE PICKMENU .... (P. 156) — SYMBOLS / DRAWING OBJECTS DISPLAYED IN ALL PICKMENUS .... (P. 156) — COMPONENTS IN THE PICKMENU .... (P. 156) — ENTERING THE DATABASE USING THE PICKMENU.................................... (P. 157) — ALTERNATIVE ELECTRICAL SYMBOLS .... (P. 157) — SYMBOLS WITH STATES IN THE PICKMENU .... (P. 160) — POSSIBILITIES AND LIMITATIONS WHEN USING THE PICKMENU .... (P. 161)

The pickmenu can also contain line types, text types and arcs. — Read more about the pickmenu in "Lines in the Pickmenu" on page 79, "Texts in the Pickmenu" on page 140 and "Circles / Arcs in the Pickmenu" on page 108.

Selecting the different Pickmenus

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

**Click to select a new menu
in the pickmenu**
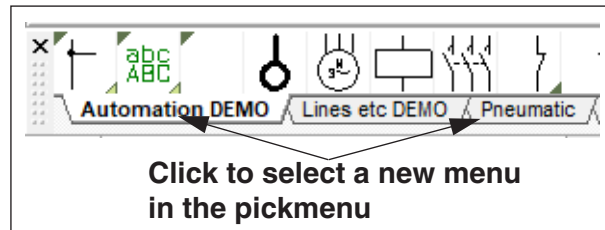
— See how to create new menus in the pickmenu in "Creating new menus in the Pickmenu" on page 205.

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 203.

**Displayed in
all pickmenus**

**Displayed in selected
pickmenu only**
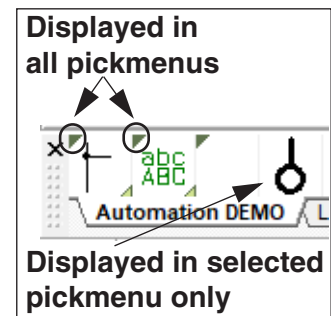
Components in the Pickmenu (Symbols with attached article data)

The symbols in the Pickmenu can be attached to components in the database.

When you place symbols for a component on a project page, article data are automatically attached to the symbols in the diagram.

This way you automatically attach component data to the symbols when you place them in diagrams.

When the component contains more than one function, the symbols for the component are displayed in the **Symbols for component** window, at the bottom of the **Component menu**, when you click on the component in the Pickmenu.
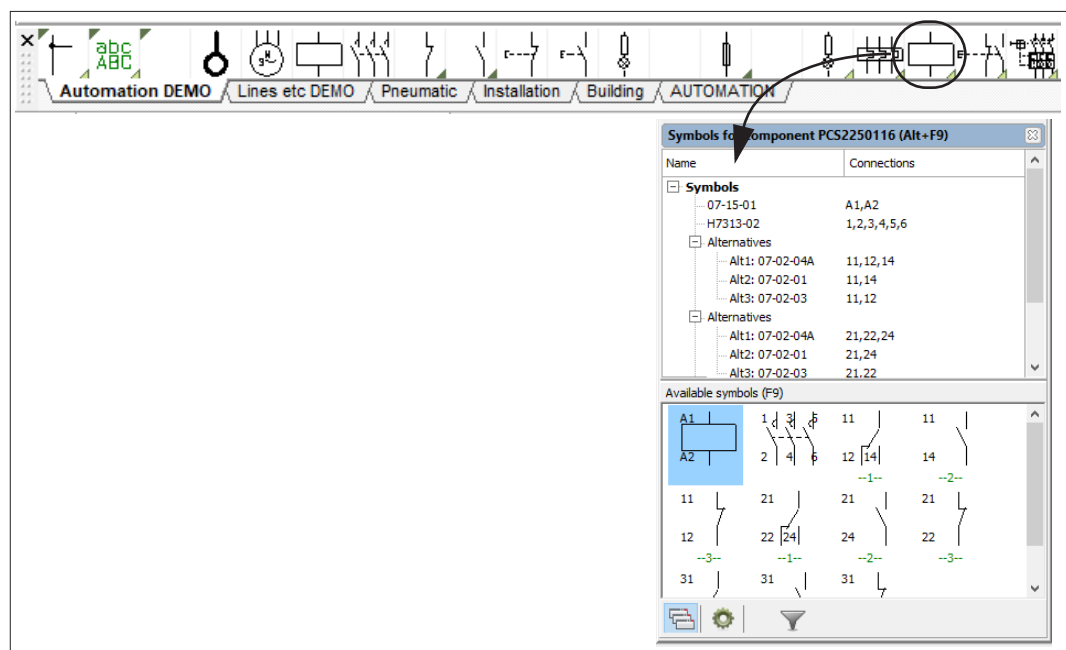
Here you can click one of the symbols under **Available symbols** and place it in the project. This way you include the component in the project. If you do not place any of the symbols for the component on a project page, you have *not* included the component in the project.

— Read more about this in "The Component menu" on page 263.

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 submenus in the pickmenu" on page 204.
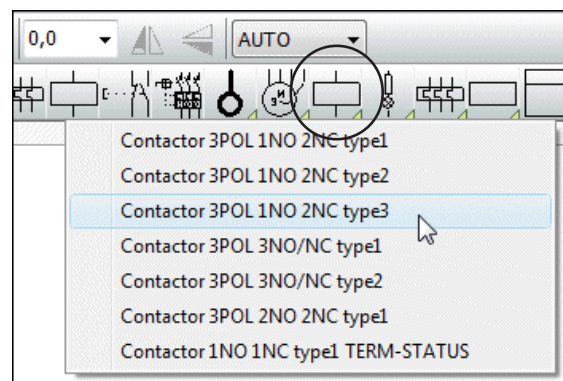
You can see that a symbol cell contains component data, by the light-blue rectangle in the lower-right corner of the cell.

— 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 205.

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 682 and/or the **PCSCHEMATIC Database** manual.

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

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 Pickmenu — 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 three switch functions, the switch functions can be used as either a switch, a make contact or a break contact.

When such a component is displayed under **Symbols for components** in the lower right-hand corner of the screen, the three sets of alternative symbols could look like this:

Two displays of the same set of alternatives
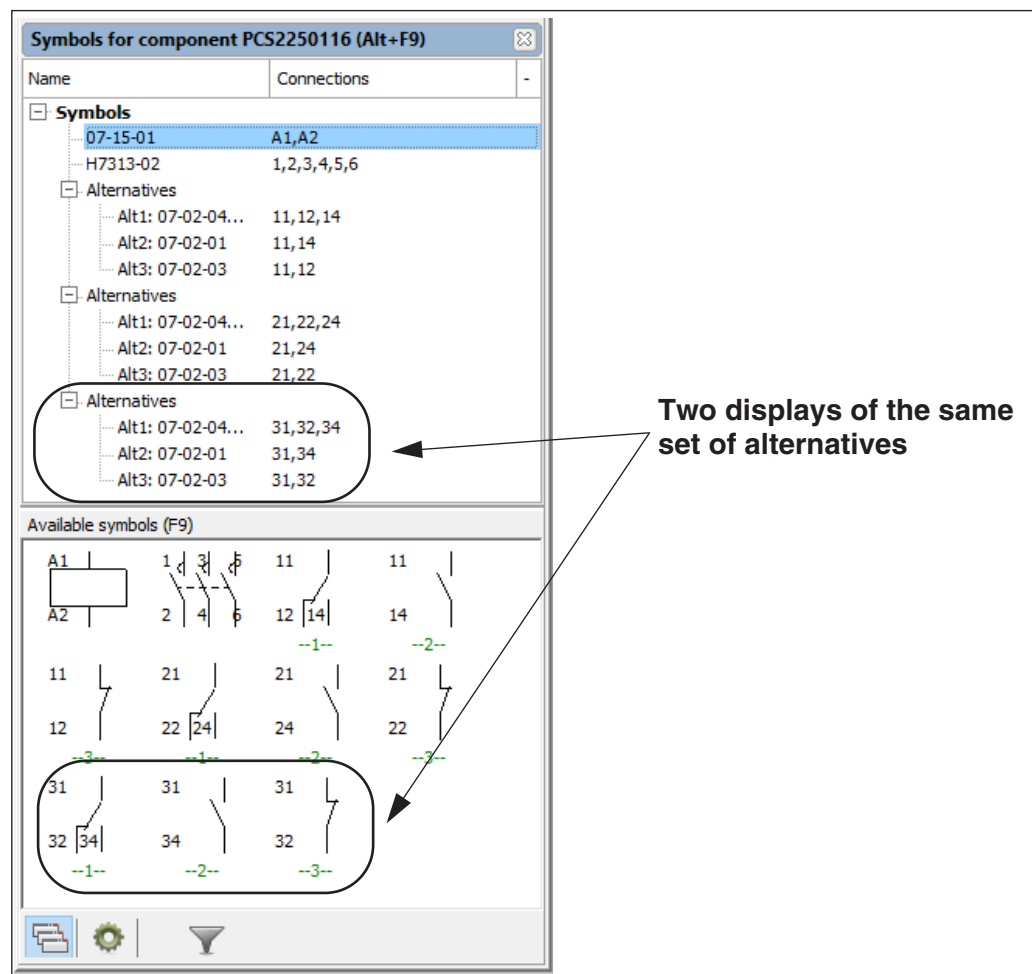
When you choose one of these alternative symbols, the other alternatives for the function disappears.

In the tree structure a checkmark is set next to the selected symbol, which is also highlighted by blue text while the other symbols are dimmed.
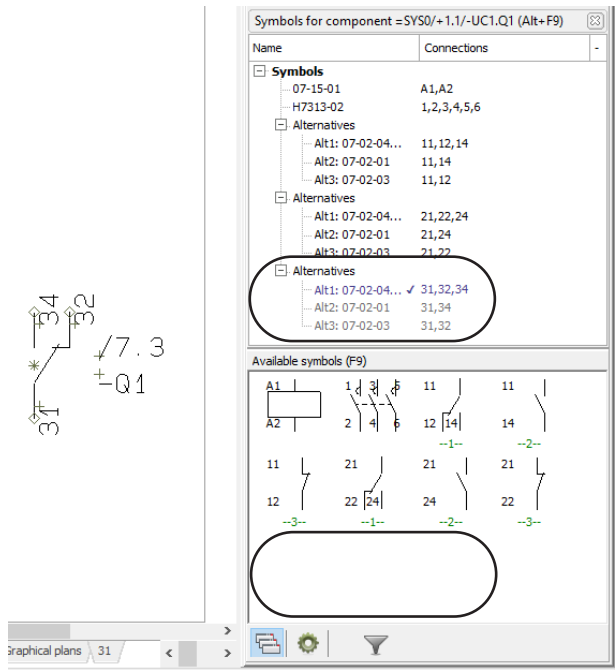
SYMBOLS WITH MULTIPLE LAYERS OF ALTERNATIVES

It is also possible to create components containing "alternatives containing further alternatives". For instance you could first choose between different relay coils, and then — depending on this choice — choose between different sets of contacts. Thus, the first layer of alternatives is the selection of the relay coil, while the second layer of alternatives is the selection of contact sets.

Below you can see how a such component could be presented in the tree structure in the **Symbols for component** window, as well as four examples on how this component could be displayed in the diagram.
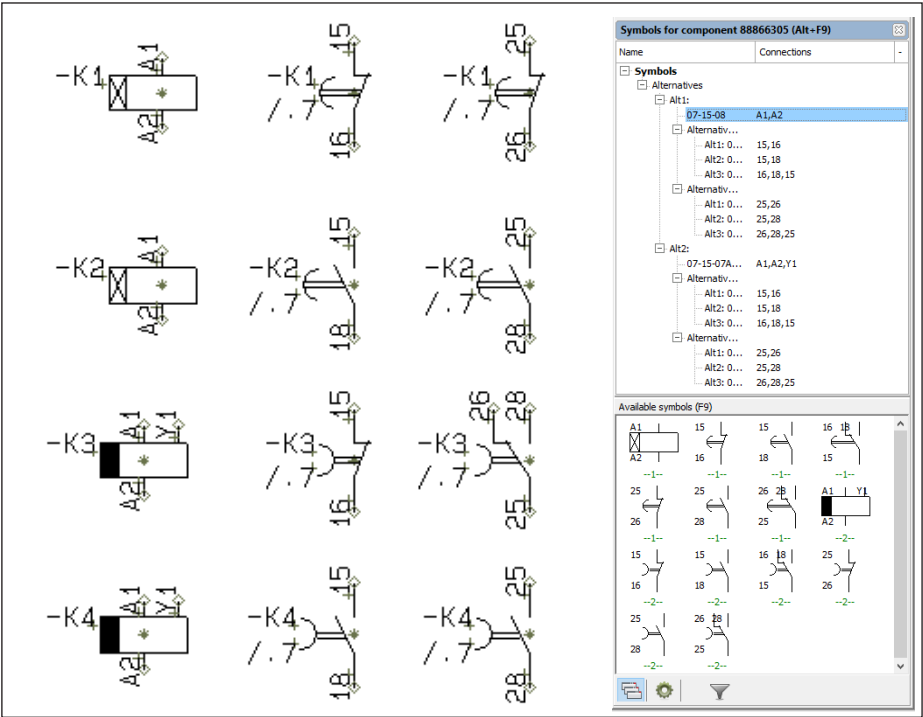
— Please also see "Symbols with states" on page 160.

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 664.

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 204. 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 190, "Symbols with states for single line diagrams" on page 474 and "Creating symbols with states" on page 791.

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:
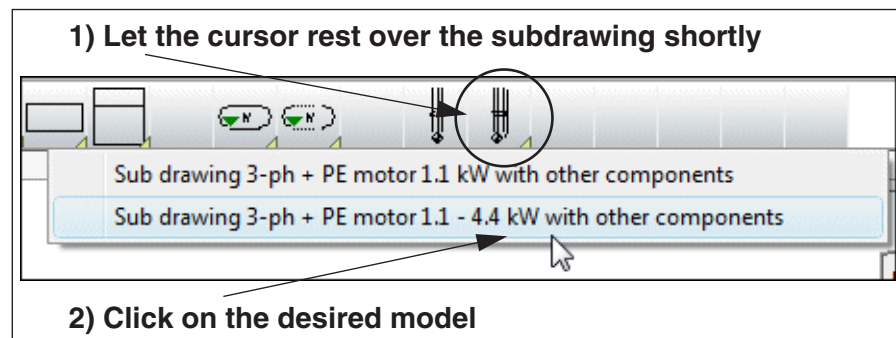
When you choose to place the sub drawing on a mechanical page, the mechanical symbols for the sub drawing are placed on the page.

— See how to place sub drawings in the Pickmenu in "Placing sub drawings in the Pickmenu" on page 207.

THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS

IN "MANUAL PLACING OF SUB DRAWINGS" ON PAGE 302 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 299.

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 203.

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**.
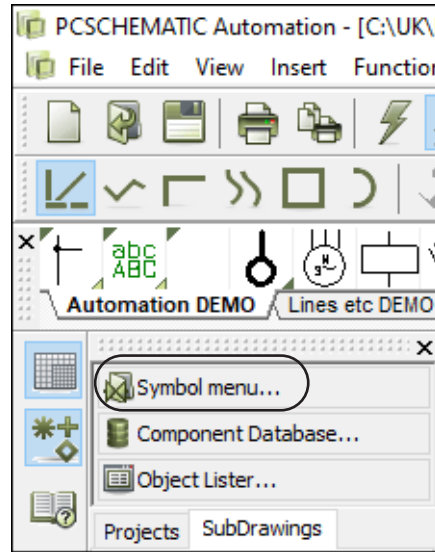
In the **Symbol menu** you then pick the symbol you need:
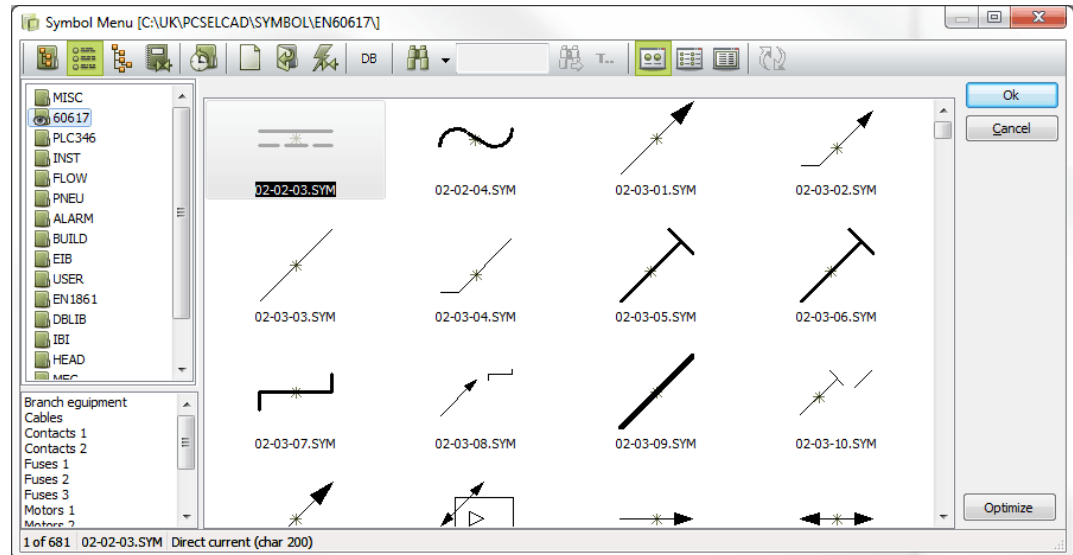
SHOW SYMBOLS FASTER IN THE SYMBOL MENU

WHEN THE SYMBOLS IN YOUR SYMBOL LIBRARIES ARE *NOT* PLACED ON YOUR OWN PC, BUT ARE LOCATED SOMEWHERE ON YOUR NETWORK, THIS CAN RESULT IN A SLOW DISPLAY OF SYMBOLS IN THE DIALOG BOX.
— SE HOW TO PREVENT THIS IN "SHOW SYMBOLS FASTER IN THE SYMBOL MENU" ON PAGE 169.

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.

<table>
<tr><td colspan="1">POSSIBILITIES IN THE SYMBOL MENU</td></tr>
<tr><td>— SYMBOL LIBRARIES / PROJECT SYMBOLS IN THE SYMBOL MENU .... (P. 163)
— REPRESENTATION OF SYMBOLS IN THE SYMBOL MENU............................ (P. 164)
— SEARCHING IN THE SYMBOL MENU .... (P. 165)
— BOOKMARKS IN THE SYMBOL MENU .... (P. 166)
— ENTERING THE DATABASE VIA THE SYMBOL MENU .... (P. 169)
— CREATING NEW SYMBOLS FROM THE SYMBOL MENU............................... (P. 169)
— SHOW SYMBOLS FASTER IN THE SYMBOL MENU...................................... (P. 169)</td></tr>
</table>

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 **Pcselcad** 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 256.

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 "Alias for directories" on page 924.

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*.
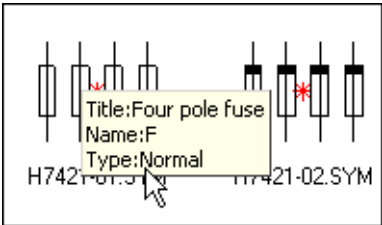
Symbol names displayed in a List

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*.
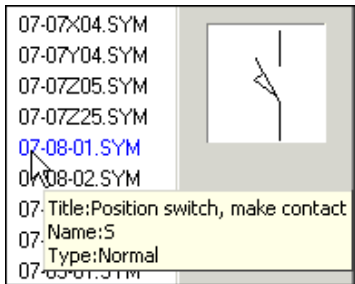
Symbol names displayed with Details

When you click on the **Detail** button, a list of the symbol file names is displayed, containing detailed information on the selected symbols.

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>.
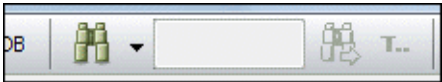
A search field is then displayed. Here you can type in the text you are searching for.

Click on the down-arrow to the right of the Search button to specify which type of information you are searching for:
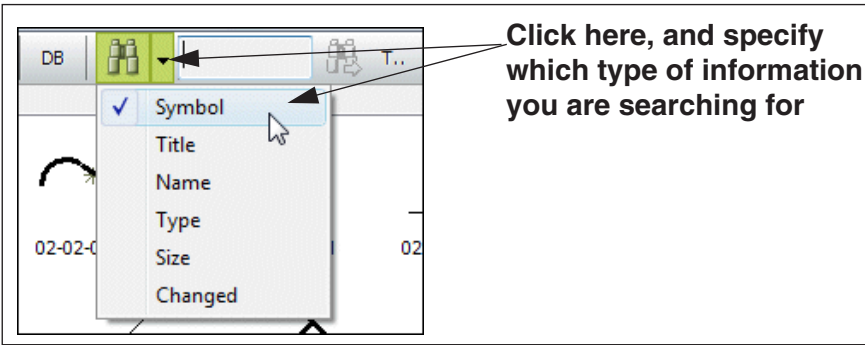
Then type in the symbol text you are searching for in the search field. As you type in the text, the program will select the first symbol in the symbol library, having a symbol text beginning with the text you have typed in so far. To jump to the next symbol matching the text, click the **Search again** 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 spe

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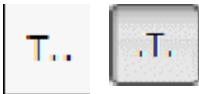
When this happens, the program searches for the text fragments anywhere in the selected text types.

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.
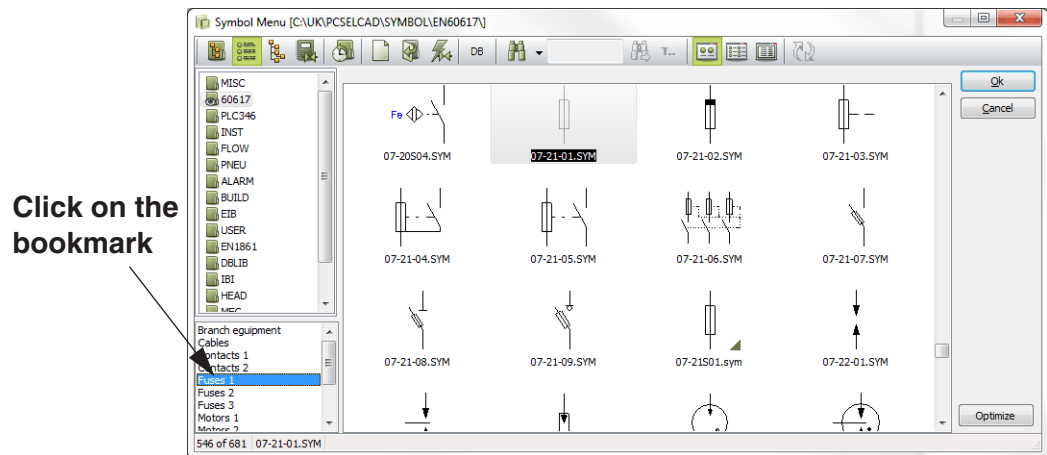
Creating a bookmark in the Symbol menu

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**:
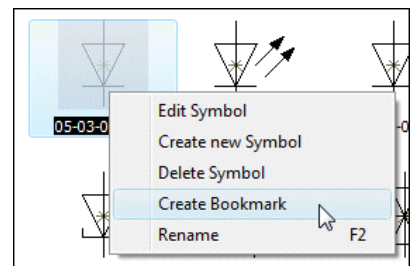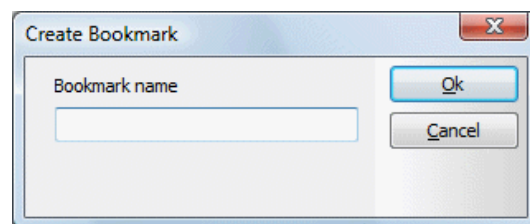
Type in the name of the bookmark, and click **OK**.

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 166 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 **Pcselcad\Symbol**:
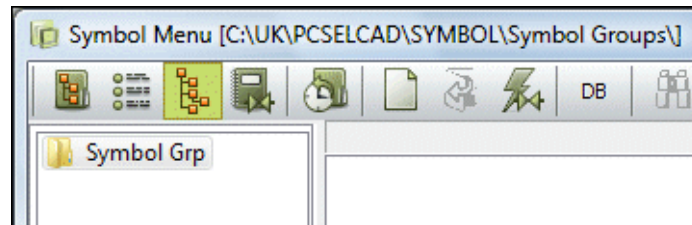
Here you can create your own grouping of the symbols, which you already have access to.

This is done by first creating a structure of folders with relevant names. In these folders, you create the so-called *groups*, and it is in these groups that you can place the symbols. Please note, that it is *not* possible to place the symbols in a *folder*, they can only be placed in a *group*.

Example on creating a hierarchical structure

1) Create the first folder: Right-click on the folder **Symbol Grp**, and select **Add folder**. Type in the folder name **Contacts**, and click **OK**.

2) Create sub folders: Right-click on the folder **Contacts**, and create the sub folder **Make**, just like you created the first folder. Under the **Contacts** folder, you also create the sub folders **Break** and **Switch**.

3) Create a group: Right-click on the folder **Switch** and select **Add group**. Type in the group name **1P**, and click **OK**. In this group **1P**, you can now place symbols.
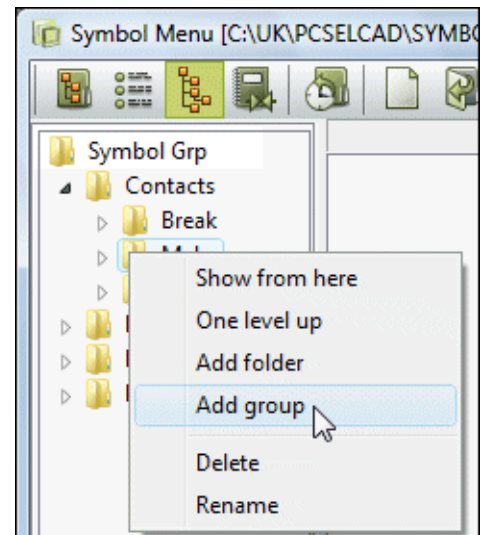
SYMBOLS ARE NOT PLACED DIRECTLY IN THE GROUPS

THE SYMBOLS ARE NOT PLACED DIRECTLY IN THE CREATED GROUPS. IN STEAD, 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**:
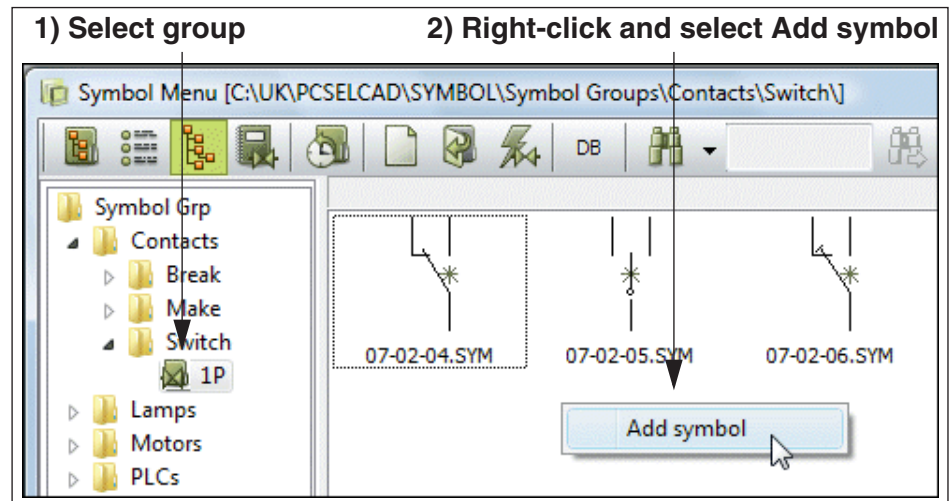
3) The dialog box **Get symbol** is now displayed: Select the folder where the symbol is placed, by clicking on the folder in the left side of the window.

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:
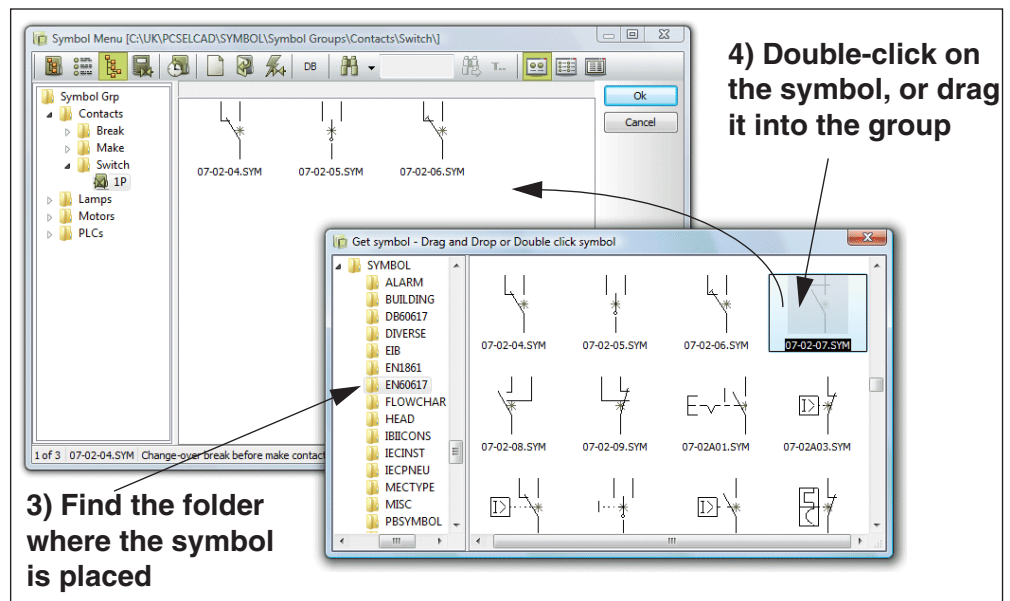
Remove symbol from group

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 682 and/or the **PCSCHEMATIC Database** manual.

— Read more about fetching symbols via the database in "Fetching symbols from the database" on page 170.

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 775.

Show symbols faster in the Symbol menu

When the symbols in your symbol libraries are not placed on your own PC, but are located somewhere on your network, this can result in a slow display of symbols in the dialog box.

To prevent this on slow networks, do this:

1) Click the **Optimize** button in the lower-right corner of the **Symbol menu** dialog box

2) In the dialog box **Optimize symbol view**: Select *Include subfolders*, click the **Show default symbol library** button and click **Start**

MAY TAKE A LONG TIME ON SLOW NETWORKS!

DEPENDING ON THE SPEED OF THE NETWORK — AND THE NUMBER OF SYMBOLS IN THE FOLDERS — THIS MAY TAKE A LONG TIME. EVEN HOURS. BUT ONCE IT'S DONE, YOU WILL NEVER NEED TO DO IT AGAIN.

---

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 641 for how to turn a database on). You can also press the short-cut key <d>.

Select component group

Components

Searching and navigation

Choose e.g. Fabricate
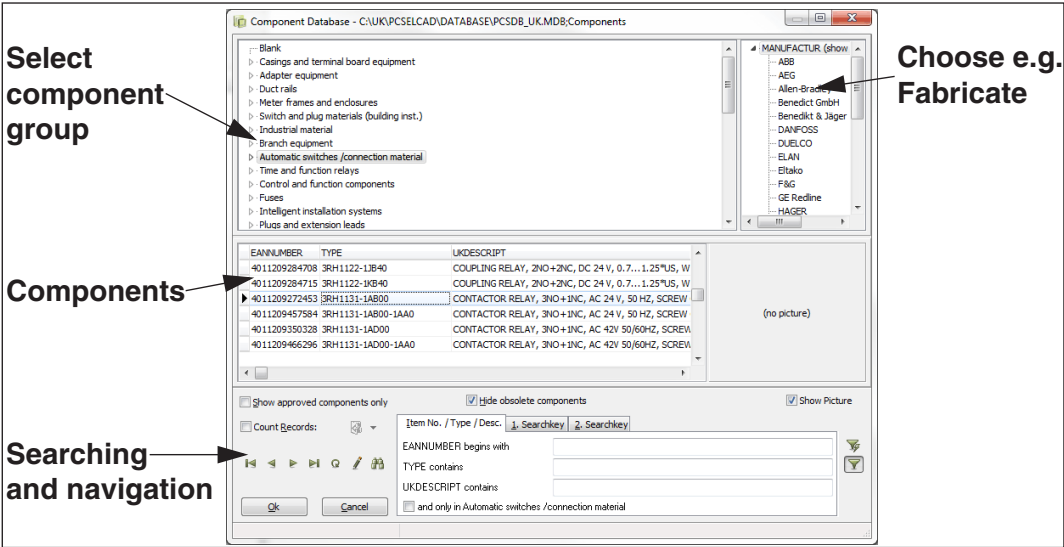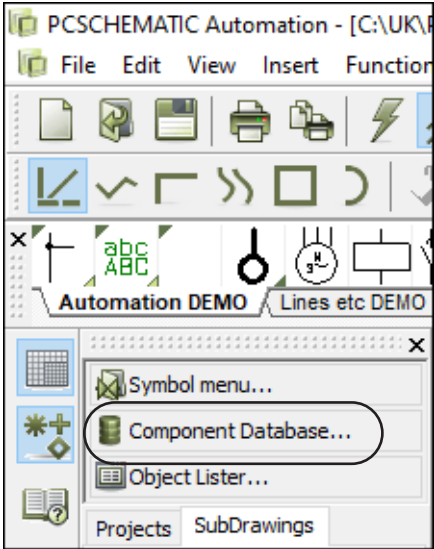
OPTIONS IN THE DATABASE MENU

— SELECTING A SYMBOL IN THE DATABASE .... (P. 171)

— PLACING SELECTED SYMBOLS FROM THE DATABASE .... (P. 171)

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 641.

Placing the selected symbols from the database

A special pickmenu, containing all the electrical symbols for the component, now appears in the lower part of the **Symbols for component** window inside the **Component menu** in the right-hand side of the screen:
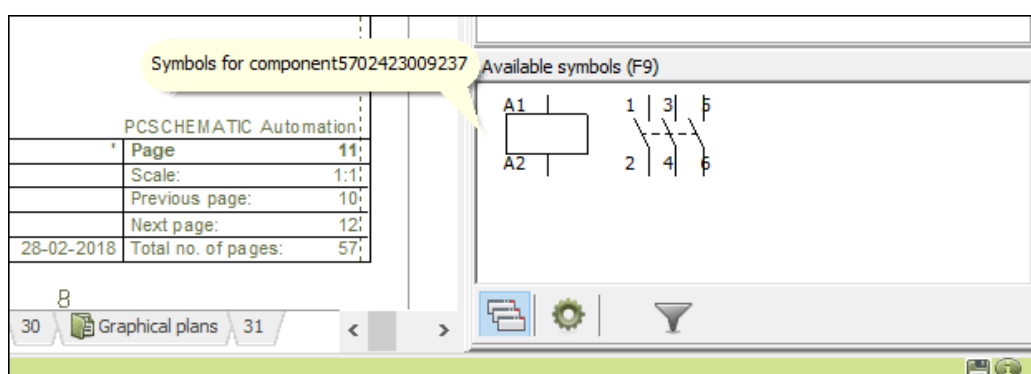
If the component consists of just one symbol, you automatically get this symbol in the cross hairs.

Click on the symbol you want to use, and place it where you want it.

4. Typing in the symbol file name directly

When you have activated **Symbols** and press the short-cut key <k>, you enter this dialog box:
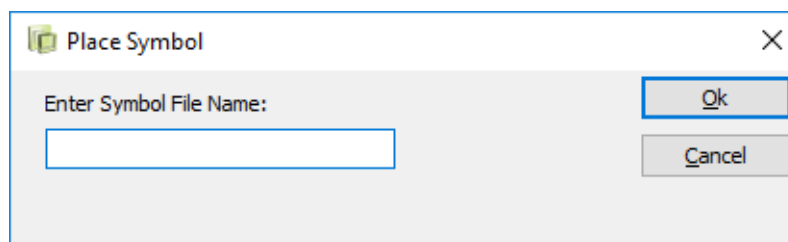
Type in the file name of the symbol you want, and press <Enter>. The program now search the contents of the folder you used last time you used the **Symbol menu**. If the program does not find the symbol there, the program searches the symbol libraries with *alias* — as described in "Alias for directories" on page 924.
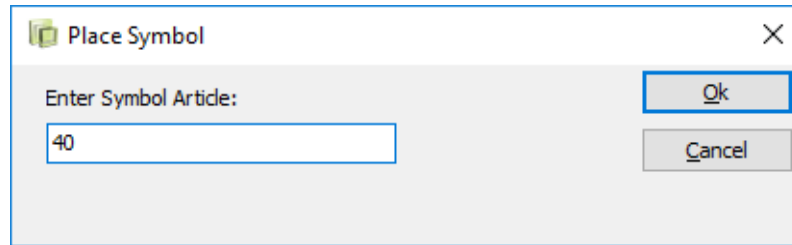
If the program can not find a matching symbol, you will enter the **Symbol menu**. If there exists a symbol in the **Symbol menu** whose name starts with the text you typed in, it will be selected.

— In "Automatic generation of symbols" on page 226 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 <b> — 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**.

If the component consists of a single symbol, you now get the symbol in the cross hairs. If the component contains more than one symbol, these are displayed under **Available Symbols** in the lower-right corner of the screen.

Click on the symbol you want to use, and place it in a diagram.

If you type in a complete article number in the **Place symbol** dialog box, 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 <v> — you enter this dialog box:

Place Symbol	×
Enter Symbol Type:	Ok
	Cancel
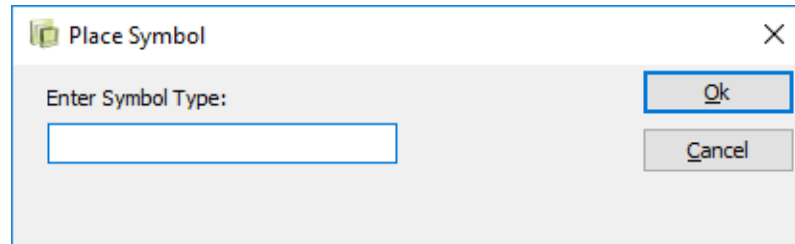
Here you type in as much you can remember of the symbol type, and click **OK**.

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 is displayed under **Available symbols** — in the lower-right corner of the screen — 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 154 — 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:
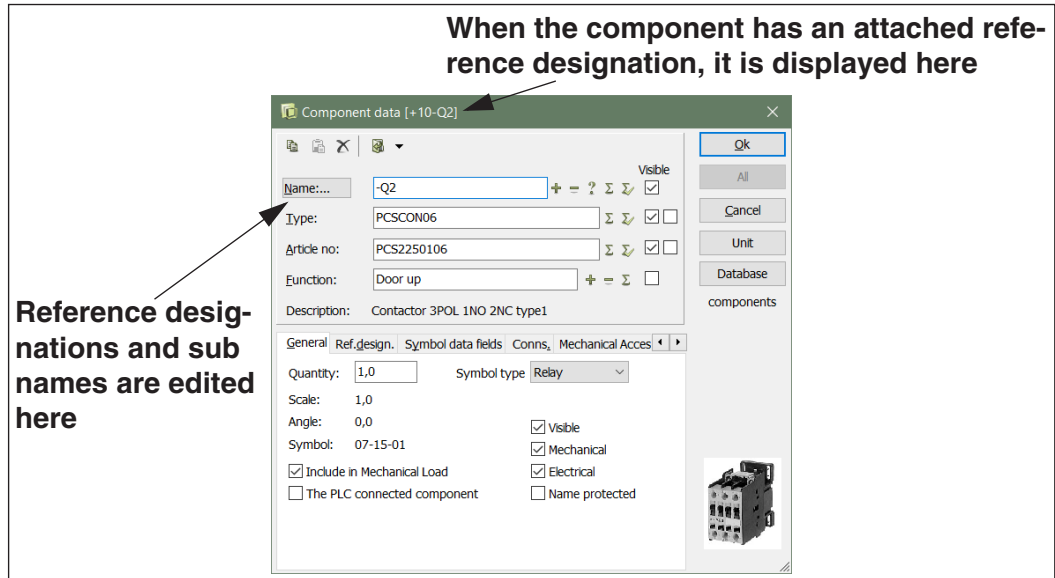
In the dialog box **Component data** above you type in the component name, and other relevant information. — Please refer to "Texts connected to symbols" on page 115 for details about the contents of these text fields.

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.
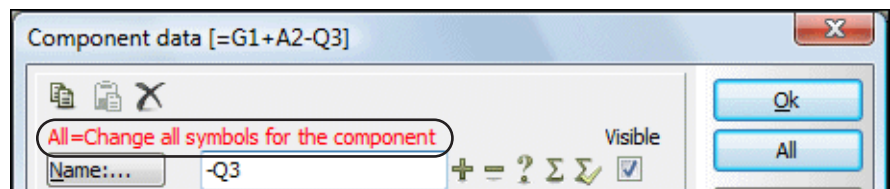
IF YOU THEN CLICK THE **ALL** BUTTON IN THE DIALOG BOX, THE CHANGES IN THE FIELDS ***NAME***, ***TYPE***, ***ARTICLE NO***, ***FUNCTION*** AND THE REFERENCE DESIGNATIONS ARE TRANSFERRED TO ***ALL*** THE OTHER SYMBOLS FOR THE COMPONENT. AT THE SAME TIME ALL SYMBOLS FOR THE COMPONENT ARE UPDATED FROM THE DATABASE, AS DESCRIBED IN "UPDATE FROM DATABASE WHEN CHANGING SYMBOL ARTICLE NO" ON PAGE 279.

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 201.

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 918.X

OPTIONS IN THE DIALOG BOX COMPONENT DATA
— SPECIFYING THE COMPONENT NAME .... (P. 176)
— DISPLAYING/EDITING THE PARTS OF A COMPONENT NAME .... (P. 177)
— SPECIFYING TYPE AND ARTICLE NO .... (P. 178)
— SPECIFYING FUNCTION .... (P. 179)
— VISIBLE/INVISIBLE TEXTS .... (P. 179)
— THE GENERAL TAB .... (P. 179)
— THE REF.DESIGN. TAB .... (P. 181)
— THE REFERENCE TAB .... (P. 182)
— CHANGING CONNECTION DATA .... (P. 182)
— THE ACCESSORY TAB .... (P. 184)
— THE SYMBOL DATA TAB .... (P. 186)
— COPYING COMPONENT DATA .... (P. 186)
— COLLECTING INFORMATION FROM THE DATABASE .... (P. 186)
— ATTACHING UNIT DRAWINGS TO SYMBOLS .... (P. 187)

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.
Σ	Gives you a list, where you can choose between the already used symbol names.
Σ	Gives you a selected list of symbol names for all relevant symbols in the actual situation.

INSERT SYMBOL NAMES - OR RENAME SYMBOLS

IF YOU NEED TO INSERT NEW/AVAILABLE SYMBOL NAMES — OR CONNECTION NAMES — IN THE PROJECT, YOU CAN SEE HOW TO DO THIS IN "INSERT NEW SYMBOL NAMES OR CONNECTION NAMES" ON PAGE 338.

IN "RENAME SELECTED SYMBOLS" ON PAGE 332 YOU CAN ALSO SEE HOW TO RENAME ALL OF THE SYMBOLS IN THE PROJECT — OR JUST SELECTED SYMBOLS.

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 210.

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 116.

More advanced ways to change texts directly on diagrams are described in "Locking text types" and "Counting on texts" on page 134.

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 448.

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:

Click here and choose between the applied reference designations

In this dialog box you can see and edit the different parts of the symbol name: **Name** (-), **Function** (=), **Location** (+), **Product** (-) and **Sub name** (.). Only active aspects are displayed in the dialog box.

— Please also see "The Ref.design. tab" on page 181 and "Sub names on symbols" on page 188.

Code letters from 81346-2

When you click the down-arrow in the **Code letter** field, you can choose the code letter for the component from the list of all code letters from IEC ISO 81346-2.

This new edition of the standard from 2019 contains code letters on both 1, 2 and 3 letters.
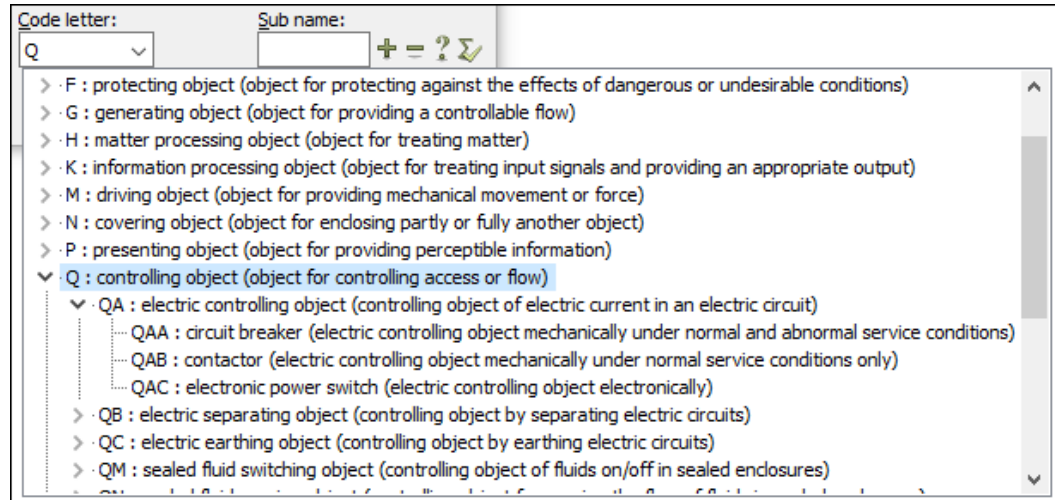
When you insert a symbol where the code letter for instance is **Q**, the letter **Q** is highlighted in the list. If you then for instance select the code letters **QAB**, the program automatically finds the next available number.

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.
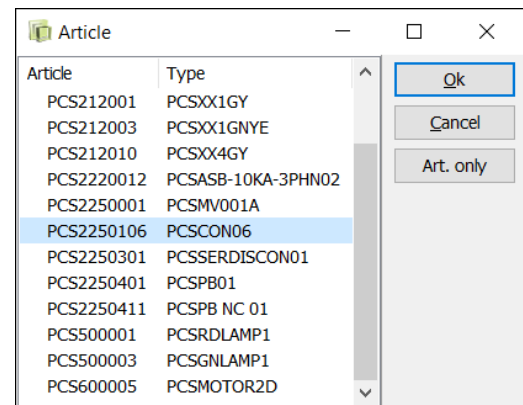
— Article number and type data can also be transferred automatically when choosing components via the database, as described in "Fetching symbols from the database" on page 170.

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 279.

— See also "Updating symbols from the database" on page 196.

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 115.

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 141.

The General tab

On the **General** tab 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.

The Quantity field in the dialog box Component data

The field *Quantity* in the dialog box **Component data** specifies how many occurrences of the component the symbol represents.

For cable symbols you can use the *Quantity* field for *either* calculating the cable lengths in the entire project *or* for counting the number of cables / cable symbols in the project. In the first case the cable lengths are specified in meters, so that it is possible to calculate how many meters of cable are needed in the project. — Also see "Predefined symbol datafields" on page 126.

The field Symbol type in the dialog box Component data

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 785.

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 787.

Visible

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

Name protected

When you select **Name protected**, the symbol name is protected against changes. This has the following consequences:

- When copying and inserting symbols for the component: The symbol name is not changed when the copied symbol is inserted. This also applies when symbols for the component are part of a copied area.

- When using **Insert name** and **Tools => Rename symbols** (see "Rename selected symbols" on page 332) the symbol is not renamed.

- It is not possible to change the symbol name from the **Object lister**.

IN THE DIALOG BOX COMPONENT DATA: CLICK ON "ALL"

FOR COMPONENTS WITH MORE THAN ONE ATTACHED SYMBOL, PLEASE BE AWARE OF THIS: WHEN YOU SELECT **NAME PROTECTED** FOR ONE OF THE SYMBOLS FOR THE COMPONENT, YOU MUST SUBSEQUENTLY CLICK THE **ALL** BUTTON (AND *NOT THE OK BUTTON*) IN ORDER TO PROTECT THE NAME FOR ALL SYMBOLS FOR THE COMPONENT.

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 708.

The PLC connected component

When **The PLC connected component** is selected, the component is understood as the connected component when generating PLC lists.

— See "The PLC connected component" on page 955.

The Ref.design. tab

When you use reference designations in the active project, the **Ref.design.** tab will also appear in the dialog box **Component data**.
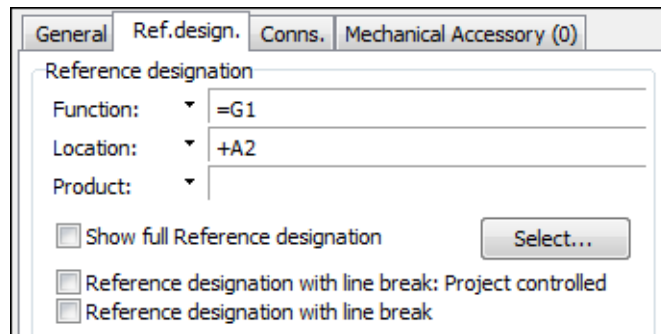
From here you choose the reference designations for the symbol, as described in "Reference designations for symbols" on page 412. — Another way of selecting reference designations for the symbol, is described in "Button for Displaying/Editing the parts of a Component name" on page 177.

— See also "Show full Reference designation" on page 414 and "Individual control of reference designations with line breaks from Component data" on page 414.

The Reference tab

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**.
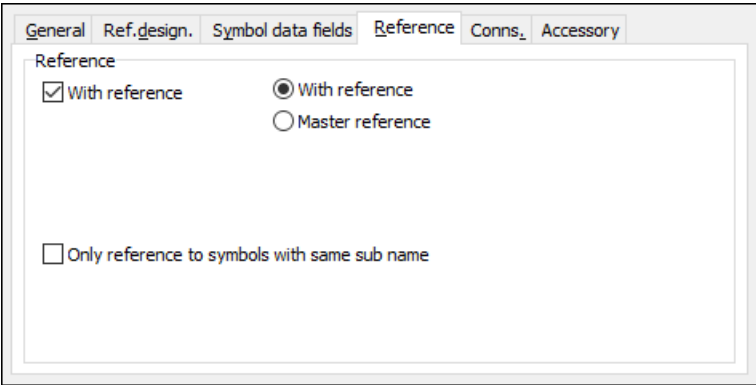
— Read more about the use of this tab in "References on symbols" on page 435.

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.
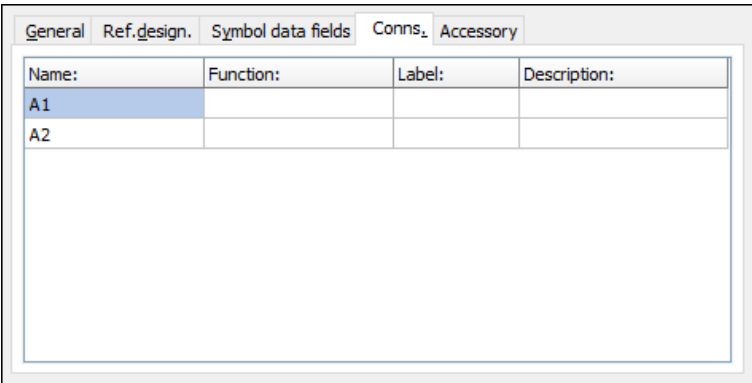
If the connection names are changed in this dialog box, they are set to be visible automatically.

This can also be done from the dialog box **Connection data**, as described in "Symbol Connection points" on page 212.

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 780 —, there will exist an extra text field in the dialog box **Component data**.

Type in the variable part, and the connection names are updated
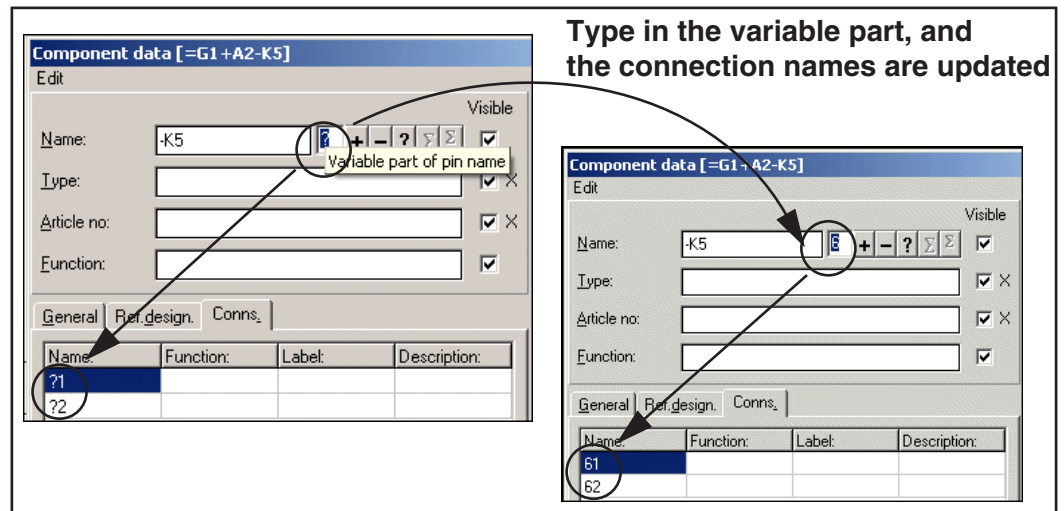
Here you can type in the variable part of the connection name. If there is a question mark, this represents one single character. If there is an asterisk, you can type in all of the connection name. This can for instance be applied for terminal symbols. — See "Wildcards/jokers" on page 618 for further 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:

Click the + button
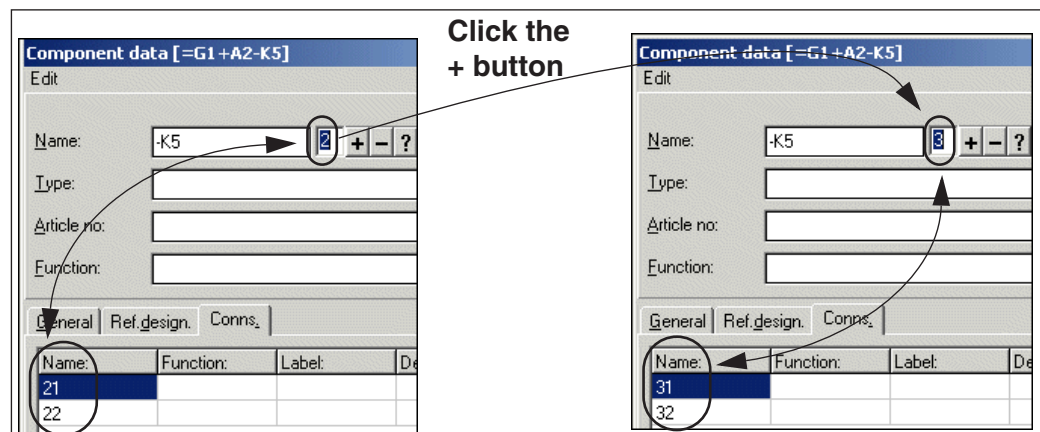
These button functions as described in "Specifying the Component name" on page 176.

— Please refer to "Symbols with variable connection names" on page 670 and "Connection data with a variable part in the name" on page 780.

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 779.

If the connection names are changed in the **Component data** dialog box, they are automatically set to be *Visible*.

The Accessory tab

Attached mechanical accessories for components can be handled on the **Accessory** tab in the dialog box **Component data**:
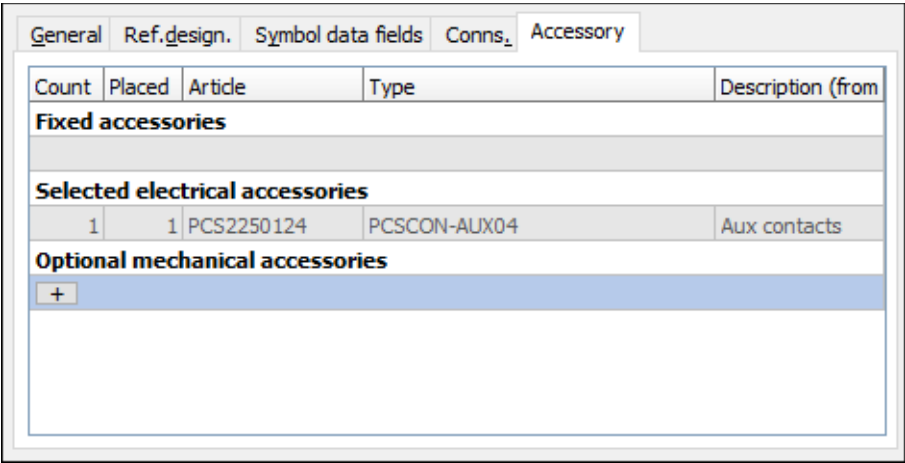
— If you need to know more about the different types of accessories, please read "Accessory types" på side 676.

On the **Accessories** tab you have the following options:

The dialog box Component data: Fixed accessories

Under **Fixed accessories** you can see whether fixed accessories are defined for the component in the database. This is information only, and you can not change it from here, because fixed accessories are defined in the database — see "Accessories in the database" on page 676.

The dialog box Component data: Selected electrical accessories

Here you can see which electrical accessories are selected for the component in the **Component menu**. This is information only, and you can not change it from here.

— See how to select electrical accessories for the component in "Optional electrical accessories in Symbols for component" on page 271.

The dialog box Component data: Optional mechanical accessories

To add optional mechanical accessories to the component, you must first click the **+** button under ***Optional mechanical accessories***. You can then specify which component that you wish to add:

Specify quantity   Specify Article no / Type
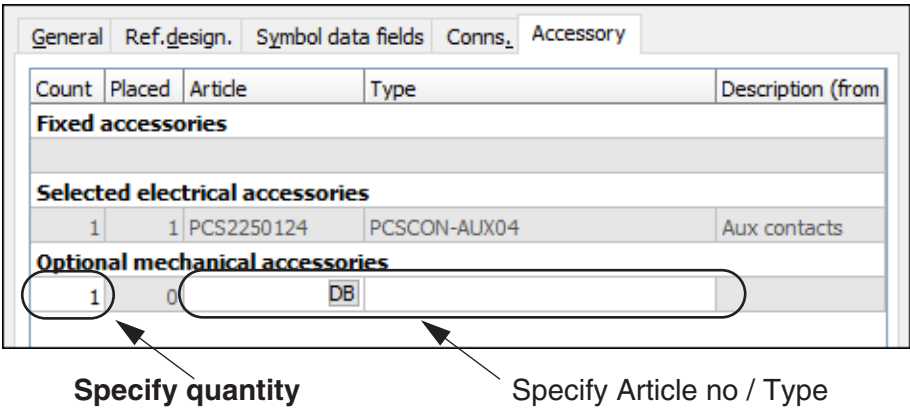
You can then click the **DB** button in the Article number field to look up the component in the database, or type in an article number. You can also specify a type.

When you have added an accessory, you get the option of adding another accessory:
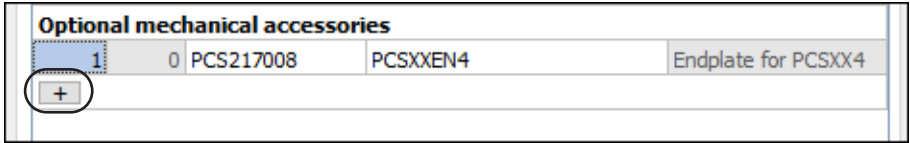
When the accessory has an attached image in the database, the image is displayed to the right in the dialog box, when the mouse cursor is placed above the accessory.

Displaying fields from the database on the Accessory tab

When you right-click on the column headings on the **Accessory** tab in the dialog box **Component data**, you can choose which of the fields from the database to display on the tab.

Right-click here
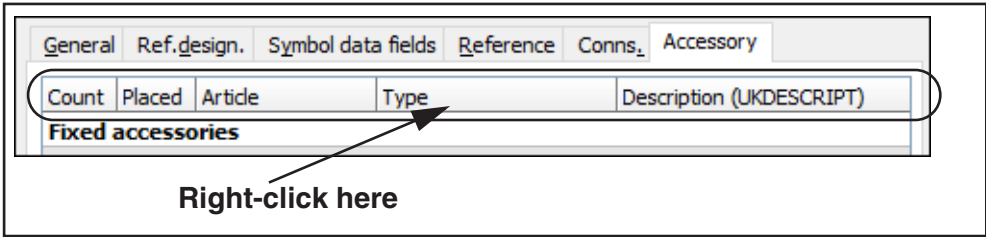
Data from the fields that you choose here are also displayed as pop-up information, when you let the mouse cursor rest over the component in the project. Thus you can for instance insert links to the manufacturers website in a database field, and choose to display it on the **Accessory** tab. You can then open these links directly from the **Accessory** tab.

Fast access to the Accessories tab

You can jump directly from a diagram page to the **Accessory** tab like this: Right-click on the symbol in the diagram and choose **Accessory**.

In parts lists mechanical accessories appear with specified quantity and data from the database (if any), and in components lists accessories appear together with the main component.

Accessories in the database

To determine which field in the database to use for specifying accessories, please read "Transferring article data to symbols and lines" on page 684, particularly "Optional accessories - Mechanical" on page 689.

The tab Symbol data fields

When a symbol is created with extra symbol datafields, you can type in data for these datafields on the **Symbol data fields** tab. — See "Extra symbol Datafields in symbols" on page 790.

You can also add extra symbol datafields to the symbol and type in values for these symbol data fields. Furthermore, you can edit and remove these extra symbol datafields. — See "Symbol datafields" on page 122 and "How to add datafields to symbols" on page 125.

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 641.

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 "Connecting symbols/lines on diagrams to unit drawings" on page 638.

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**:
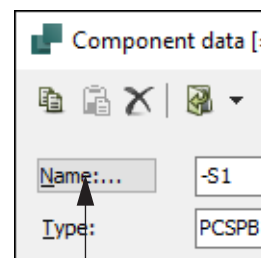
2) Click here

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 680.

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 118.

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 791.

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.

Select the state to display for the symbol

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 474 and "Placing symbols with selected states in the pickmenu" on page 206.
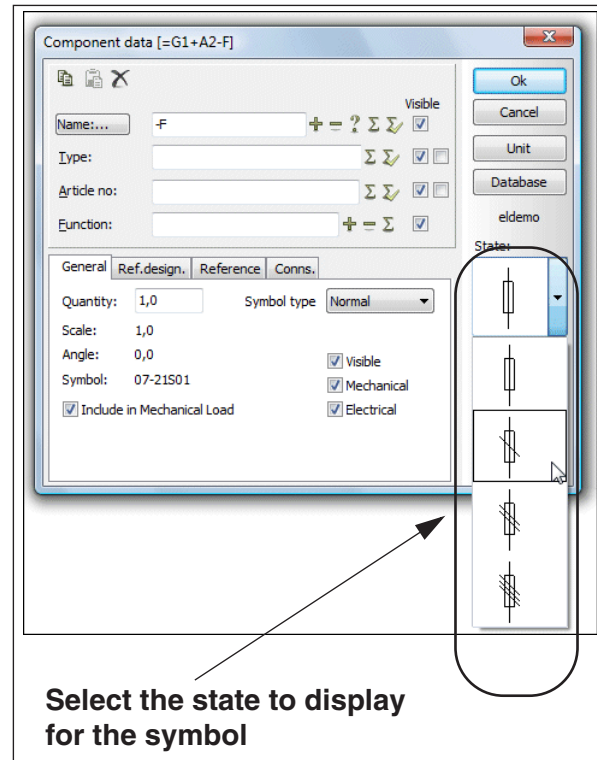
> **SHIFT STATE WITH <SHIFT + SPACEBAR>**
> WHEN YOU HAVE SELECTED A SYMBOL CONTAINING STATES, YOU CAN SHIFT BETWEEN THE STATES BY PRESSING <SHIFT> WHILE PRESSING THE <SPACEBAR>. THIS IS ALSO THE CASE WHEN YOU HAVE THE SYMBOL IN THE CROSS HAIRS.

Symbols with selected states and the Component menu

If it is defined in the database that a symbol is to be displayed with a specific state selected, it is not possible to change the state of the symbol subsequently. When no state is specified for a symbol in the database, the state can be changed when the symbol is placed in the project or at a later stage.

— See how to specify states for symbols in the database in "Specifying symbol state for electrical symbols via the database" on page 666.

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:

**Example:
Place terminal
symbols here**
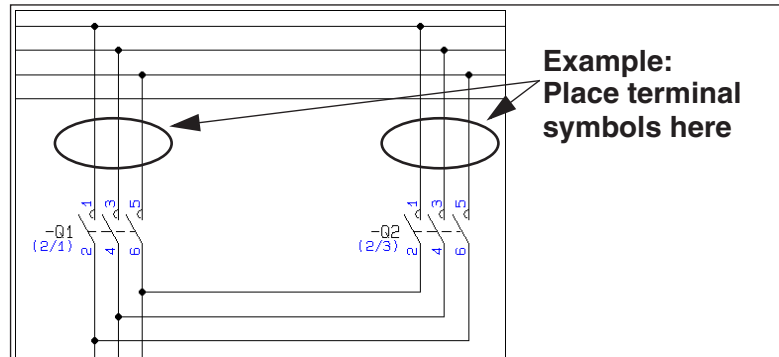
1) Select a symbol, e.g. a terminal from the pickmenu.

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 193.
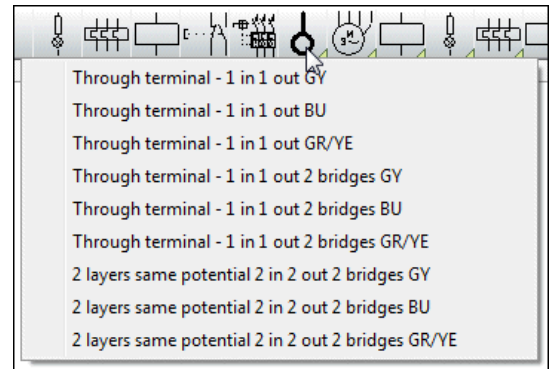
4) Right-click on the page, and select **Execute Multiplacement**.

2) Right-click here   **4) Right-click here**
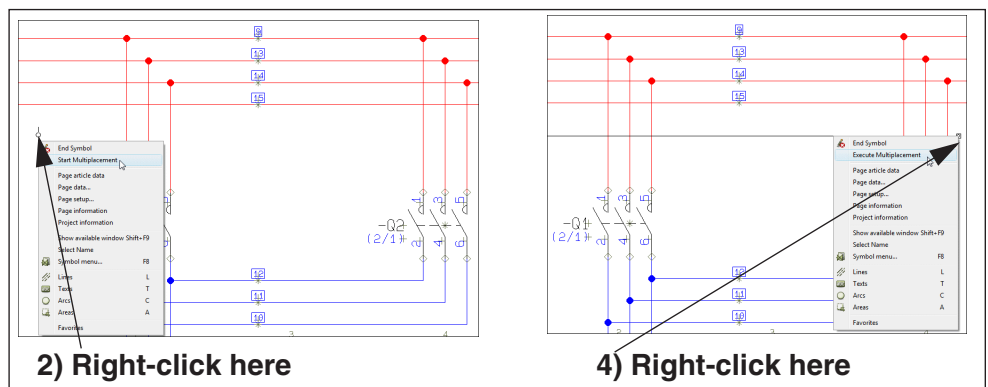
5) In the dialog box **Component data**: Specify symbol names — for instance type -X4:0, press <Ctrl> while clicking the **+** button and click **OK**:
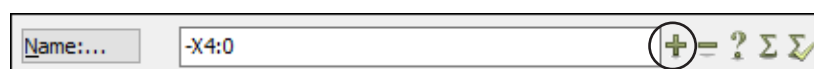
— See also "Placing and Naming symbols" on page 174 for how to name connection names for terminals directly from the **Component data** dialog box.

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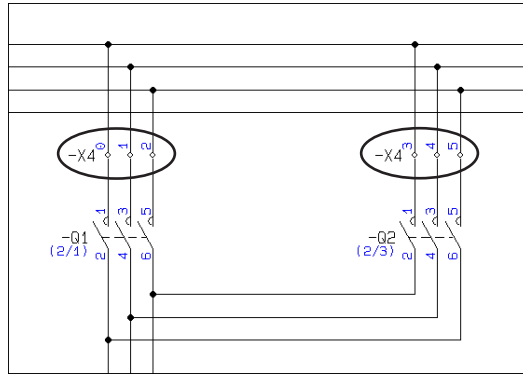
Multiplacing motor terminals with UVW

You can also multiplace motor terminals with UVW.

When specifying the component name above in the **Component data** dialog box, do this:

Type in *-X1:1U*, *-X1:1U1* or *-X1:1U2* and subsequently press <Ctrl> while clicking the **+** or the **?** button on the right-hand side of the **Name** field.

The program automatically detects if a PE is needed.

If there are only two conductors, they will be named U and V, and the naming starts over again.
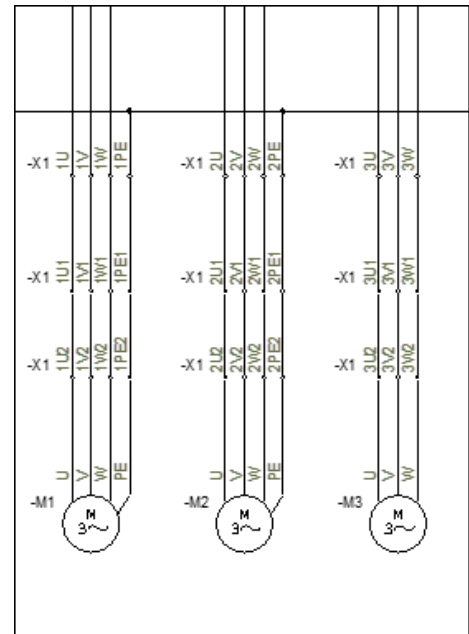
MARKING TERMINALS FOR INCOMING AND OUTGOING CONDUCTORS (EN 60445:2017)

TERMINALS FOR **INCOMING** CONDUCTORS SHOULD BE MARKED AS *U1*, *V1* AND *W1*.

THIS SHOWS THAT THE THREE TERMINALS *U1*, *V1* AND *W1* ARE MEANT FOR CONNECTING TO THREE POWER SUPPLY PHASES.

TERMINALS FOR **OUTGOING** CONDUCTORS MUST BE MARKED AS *U2*, *V2* AND *W2*.

THIS SHOWS THAT THE THREE TERMINALS *U2*, *V2* AND *W2* ARE MEANT FOR CONNECTING TO THREE OUTGOING PHASES OF CIRCUIT SUPPLIED FROM THE CONTROL PANEL.

IF THERE ARE SEVERAL INCOMING OR OUTGOING CONDUCTORS/WIRES, YOU MUST USE SEQUENTIAL NUMBERING OF TERMINALS.
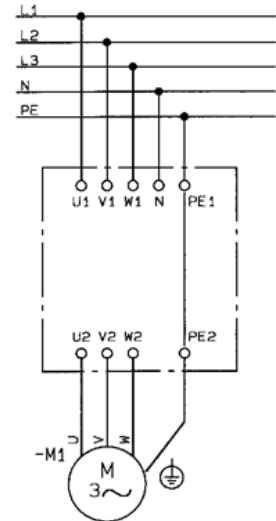
TERMINALS FOR INCOMING CONDUCTORS:
1U1, *1V1*, *1W1* ... *3U1*, *3V1*, *3W1*

TERMINALS FOR OUTGOING CONDUCTORS:
1U2, *1V2*, *1W2* ... *3U2*, *3V2*, *3W2*

Skipping line segments when multiplacing symbols

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

The two lines in the middle are skipped

①   ②   ③   ④
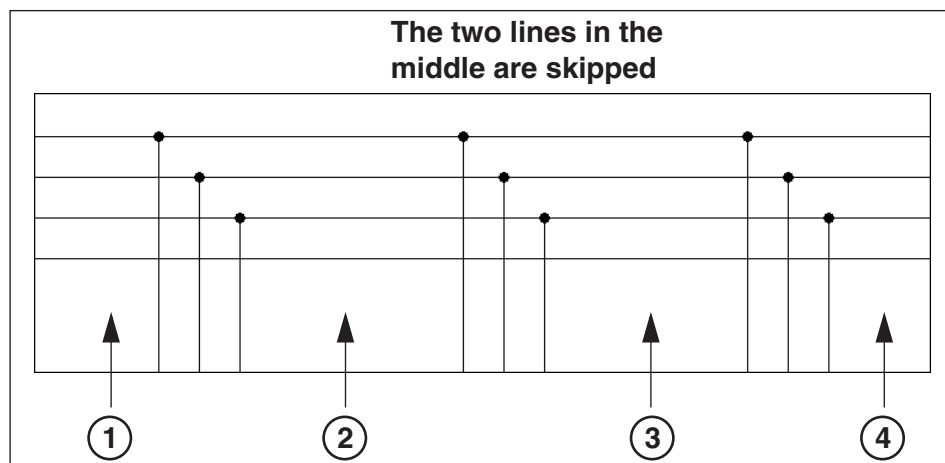
1) Select a symbol, e.g. a terminal, from the pickmenu. Right-click, and select **Start Multiplacement** — see the figure above.

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**.
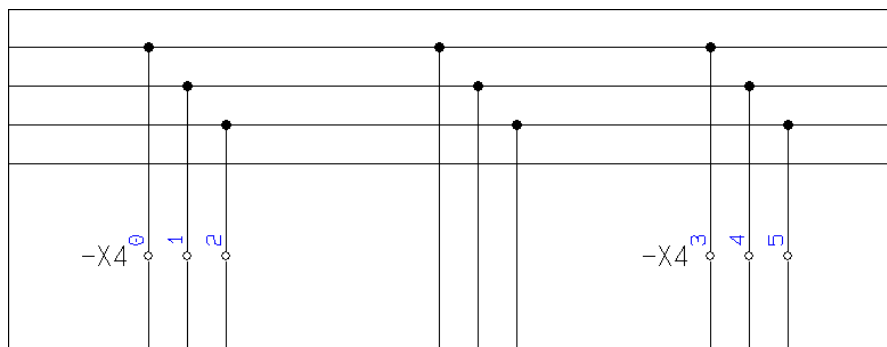
— See also "Multiplacing symbols" on page 191.

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:
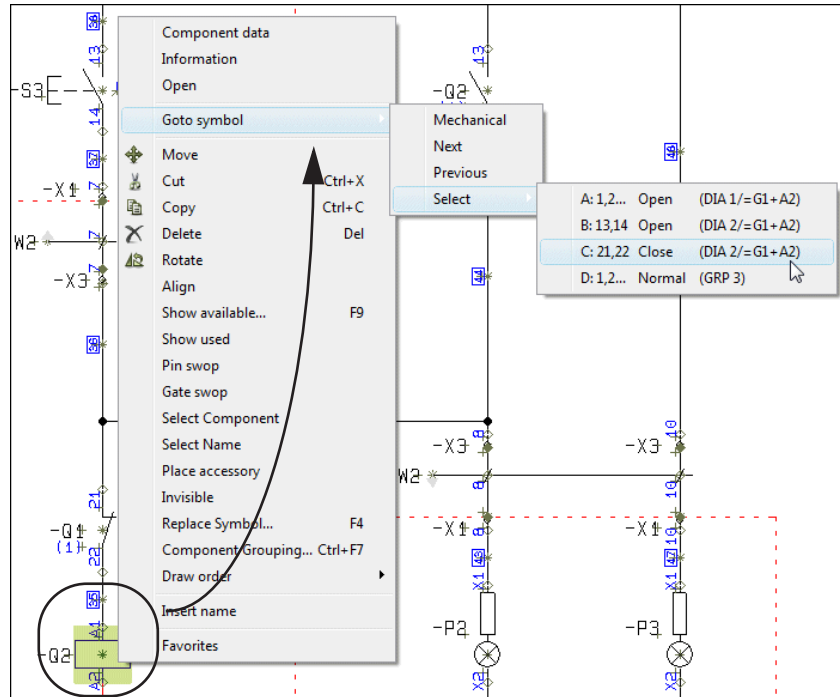
In the **Goto symbol** function, you have the following options:

- **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 433, IN PARTICULAR "AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK" ON PAGE 433. 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 279 AND "UPDATE FROM DATABASE WHEN CHANGING ARTICLE NUMBER" ON PAGE 178.

---

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 => Component Database => Update from Component Database**.

3) You now enter the dialog box **Update from Component 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 *NOT* 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 684).
UPDATE CONNECTION POINTS	UPDATES ALL CONNECTION POINTS, SO THAT THEY ARE ASSIGNED THE CONNECTION POINT NAMES DESCRIBED IN THE DATABASE.
UPDATE CABLE WIRES	UPDATES ALL CABLE WIRES WITH THE NAMES SPECIFIED IN THE COMPONENT 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 288.

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:

Here you see how many components are found with this name in the project

The number of symbols that will be deleted, if you choose *Name*

The number of symbols that will be deleted, if you choose *Component*
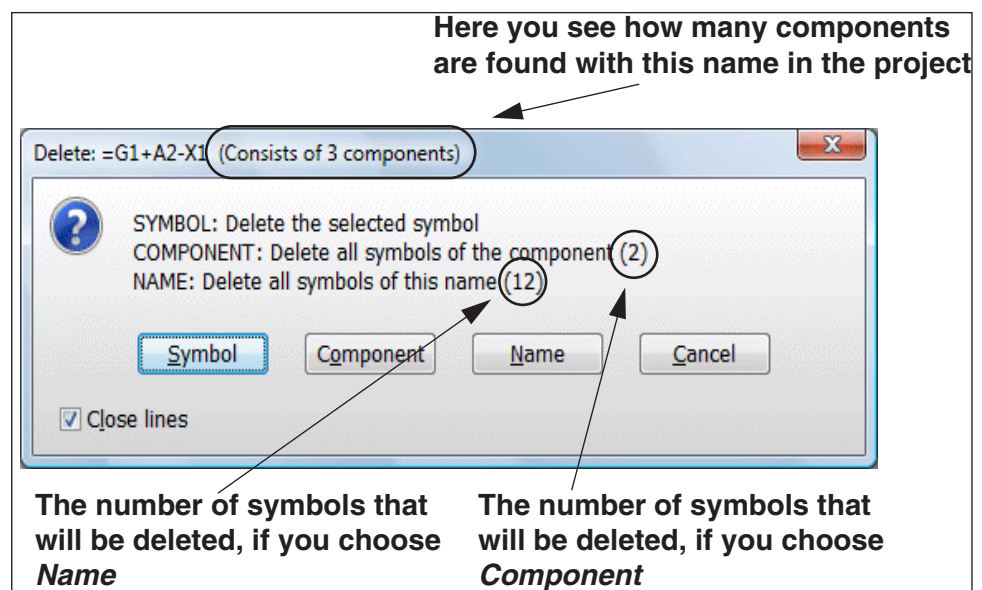
3) You now have the following options:
Click on **Symbol**: Only the selected symbol is deleted.
Click on **Component**: All symbols for the selected component is deleted (in the figure above, you can see that the physical terminal, from which the symbol is selected, consists of two symbols).
Click on **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.

a) Click on the Delete button

b) Press down <Ctrl> and click the Delete button
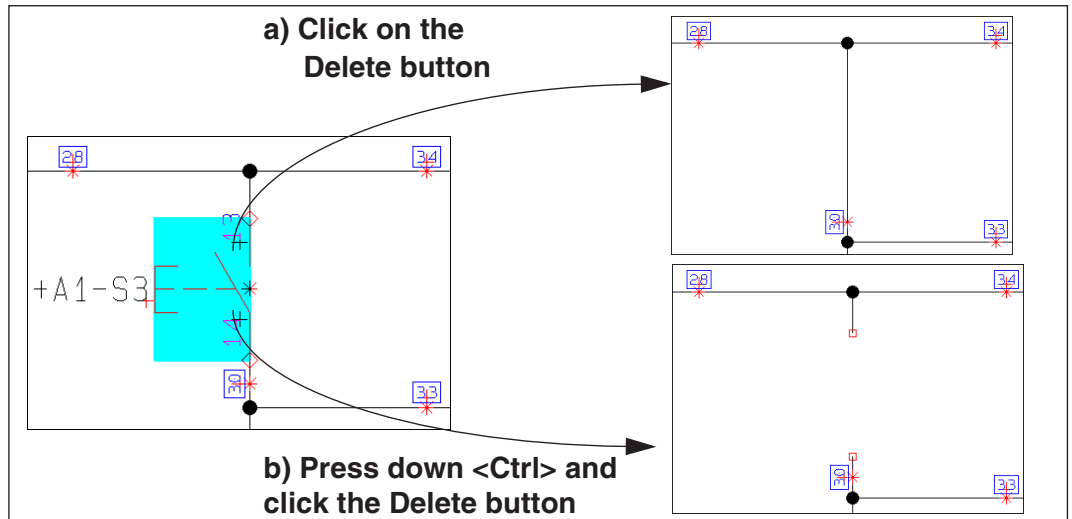
Deleting more than one symbol

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 210 — 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.
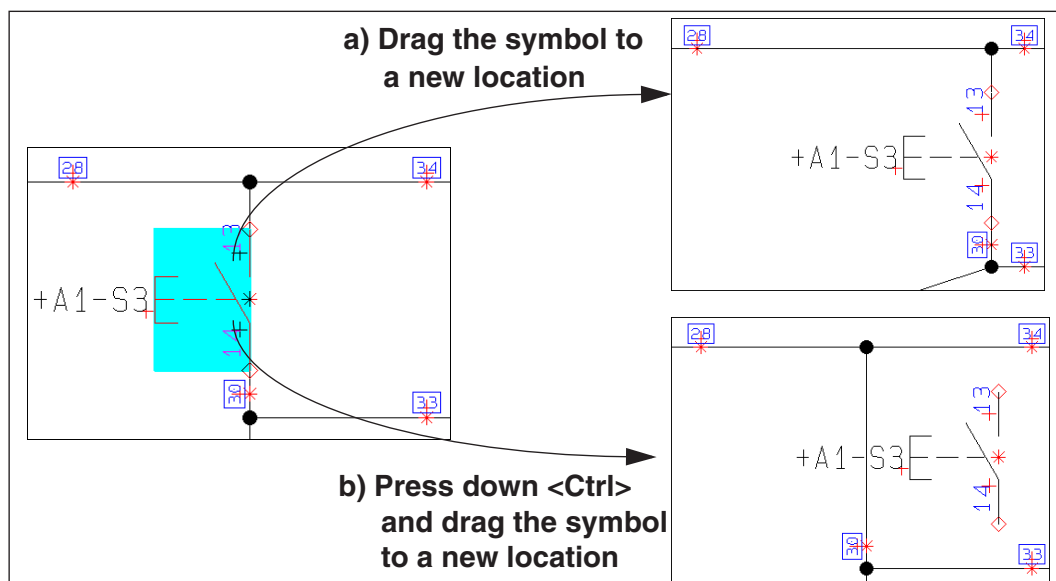
Moving more symbols at the same time

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 210 — 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).
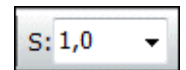
Read more about scaling symbols in "Symbol settings: Advanced" on page 787.

Mirroring symbols

Select a symbol, click the **Mirror symbol vertically** button, and the symbol will 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

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

2a) Click here

2b)Press down the mouse button and drag the mouse cursor over the screen
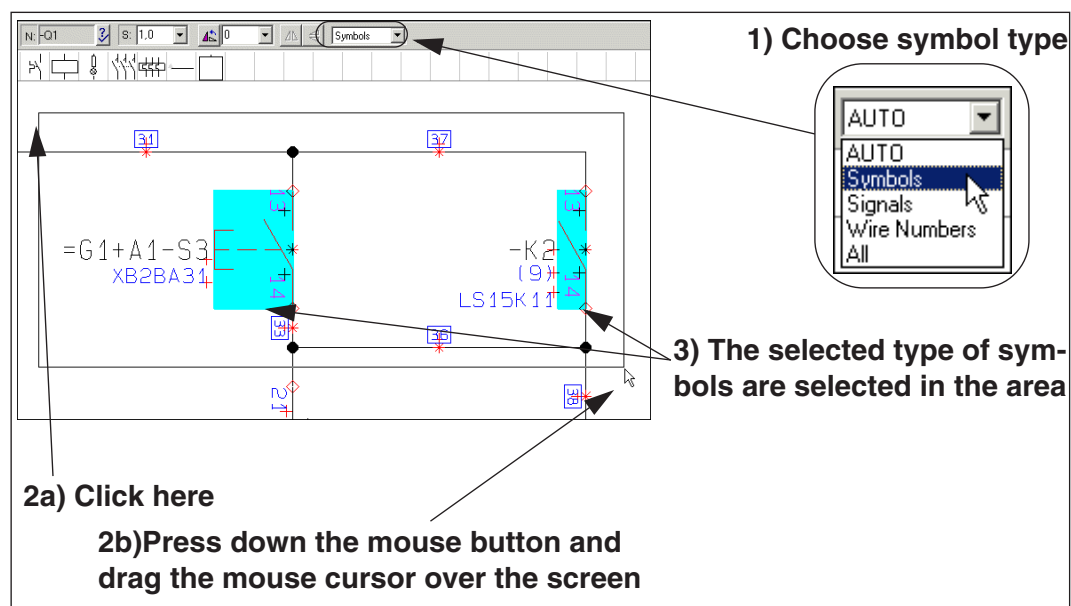
Selecting symbols using Select all or by selecting in areas

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 155.

The pickmenu can also contain line types, text types and arcs. — Read more about this in "Lines in the Pickmenu" on page 79, "Texts in the Pickmenu" on page 140 and "Circles / Arcs in the Pickmenu" on page 108.

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:

Displayed in all pickmenus

Displayed in selected pickmenu only
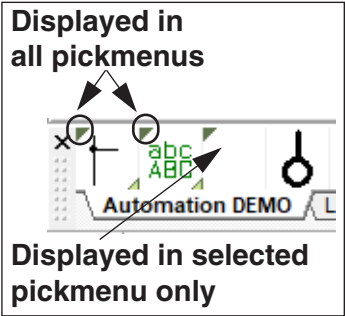
1) Right-click on the pickmenu, select **Properties**, and you enter the dialog box **Pickmenu properties**:

2) Correct the number of fixed cells
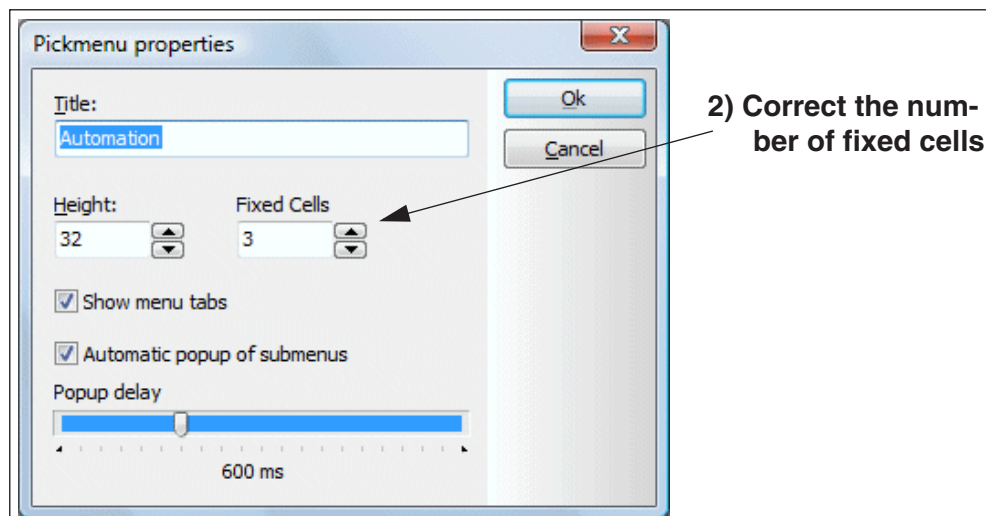
2) Set the number of fixed cells, click **OK**, and the new cell(s) are displayed:

Place a new symbol / drawing object
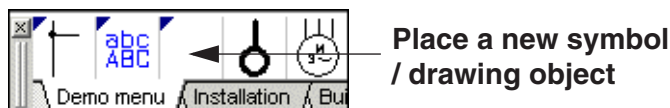
3) Place a new symbol / drawing object in the cell, as described in "Placing symbols in the pickmenu" on page 203 or "Include component data when inserting a symbol in the pickmenu" on page 203.

4) The symbol is now represented in all pickmenues.

Removing fixed cells from the pickmenues

To remove a number of symbols from being displayed in all pickmenues, 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 190) or for symbols with multiple sets of article data attached (see "Symbols with multiple sets of data in the Pickmenu" on page 157), when you let the mouse rest over the symbol in the pickmenu.

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

Under *Popup delay* you can specify whether the popup menu appears instantly, when the mouse cursor rests over a symbol with states / sets of article data, or
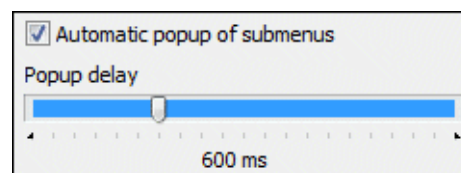
whether to insert a delay before the popup menu appears. To insert a delay has the advantage that submenus are not displayed when the mouse cursor just passes the pickmenu, in order to make selections in other of the program menus or tool-bars.

When **Automatic popup of submenus** is not acti-vated, small buttons are displayed in the pickmenu in the relevant cells. When you click this button, the sub-menu is displayed.

Click for submenu
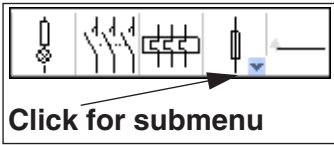
Creating new menus in the Pickmenu

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 pick-menu.

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:
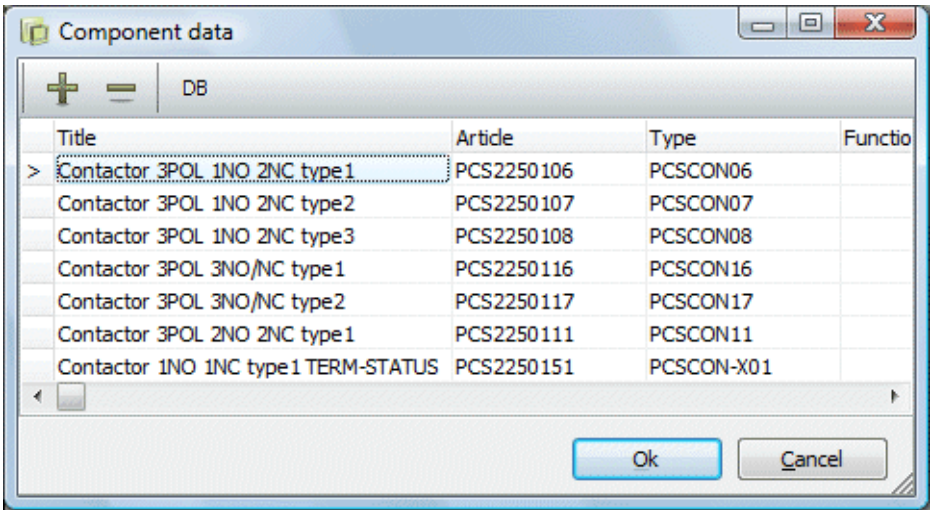
Here you have the following options:

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 645.

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.

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 190 and "Creating symbols with states" on page 791.

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 696, particularly "Example for sub-drawings" on page 697.

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 160.

THE RECOMMENDED PROCEDURE FOR SUB DRAWINGS

YOU CAN READ MORE ABOUT SUB DRAWINGS IN THE CHAPTER "SUB DRAW-INGS" ON PAGE 299. THERE YOU CAN ALSO READ ABOUT THE RECOM-MENDED 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 Cell
Delete Cell

Place Symbol...
Place Line
Place Text
Place Arc

Clear cell
Component Article Data...
Elevation
Information...
Database

Go to Menu...
Create new Menu
Delete Menu

Properties...
```

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.
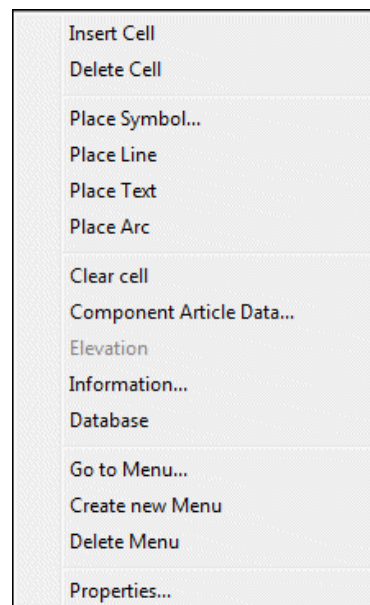
Information about the symbol

Right-click on the symbol in the pickmenu and choose **Symbol information**.

```
Information                                          X

  Symbol
  Name: K
  Type:
  Article:
  Function:
  Symbol: C:\UK\PCSELCAD\SYMBOL\EN60617\07-02-01.SYM
  Symbol type: Open
  Bytes: 2086

                         Close
```
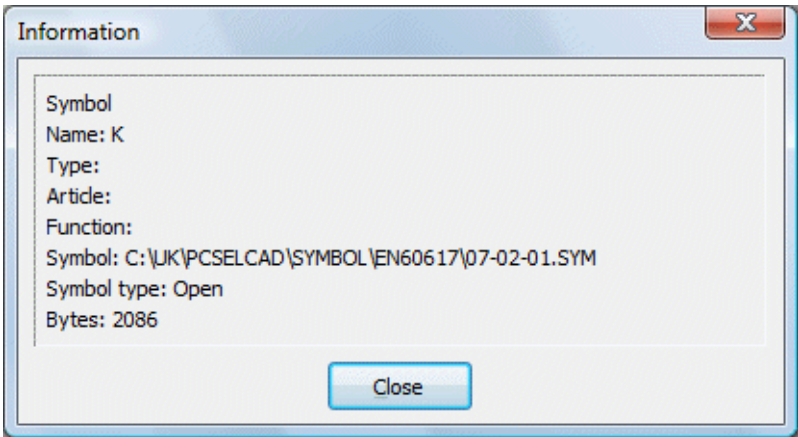
CHANGING SYMBOL NAMES IN AN AREA

THIS CHAPTER DESCRIBES
— CHANGING MORE THAN ONE SYMBOL NAME .... (P. 210)
— THE NAMING ORDER OF SYMBOLS .... (P. 211)
— OTHER OPTIONS.. (P. 211)

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**.
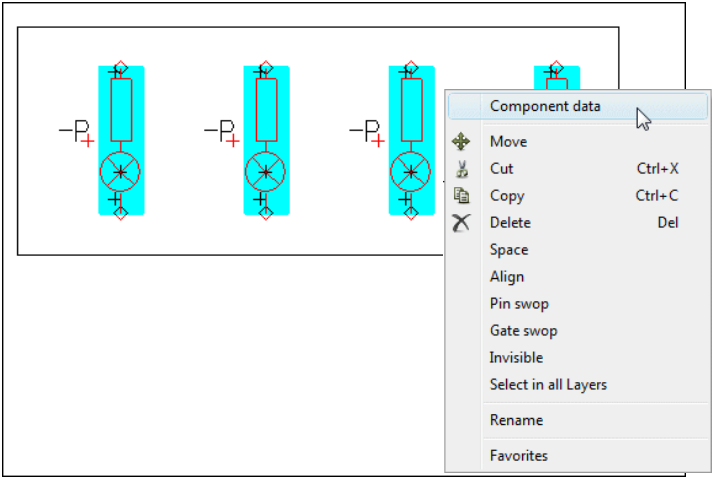
You then enter the **Component data** dialog box.
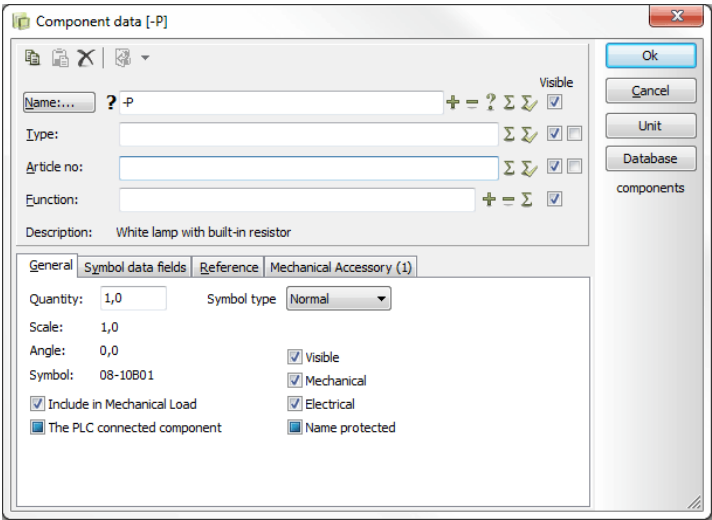
In the **Component data** dialog box you apply the automatic counting facility by typing in the name of the first symbol, pressing <Ctrl> while clicking on **?**, and then

click **OK**. — Please refer to "Automatic counting in the Component data dialog box" on page 176 for further details.
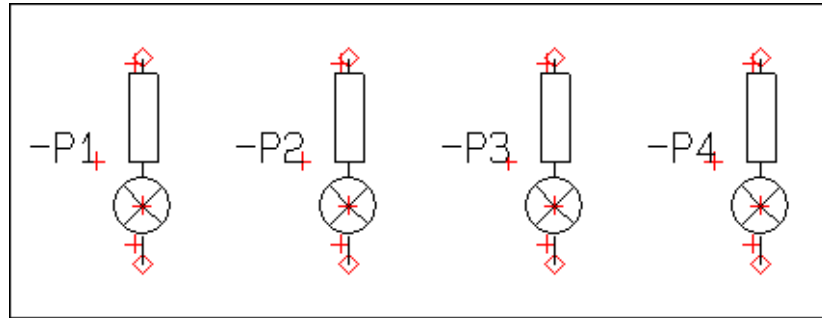
The symbol names are now changed. Click outside the selected area to deselect the symbols.

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 217.

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 116.

Other advanced ways to change texts directly on diagrams are described in "Locking text types" on page 133 and "Counting on texts" on page 134.

RENAME SYMBOLS - OR INSERT SYMBOL NAMES

IN "RENAME SELECTED SYMBOLS" ON PAGE 332 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 338.

SYMBOL CONNECTION POINTS

Connection points for symbols contains data

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

On the figure below, a connection point of a terminal symbol is selected by right-clicking on the connection point.
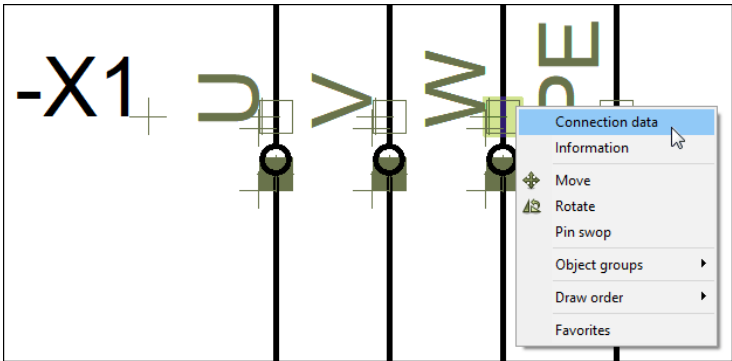
Specifying Connection data

When you are working with **Symbols** and 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.

Connection data

In the dialog box **Connection data** 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 117.

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 141 for further details.

The function of the four buttons in the dialog box, is described in "Specifying the Component name" on page 176.
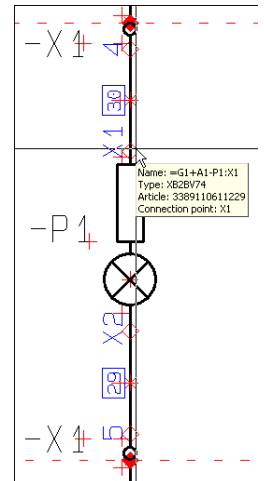
Connection details

Under **Connection details** you can specify whether the connection point is included in terminals lists or not. Connection points are displayed differently in the project depending on whether they are included in terminals lists or not.

The options may be different depending on the symbol type.

— See "Appearance of connection points on symbols (terminals lists)" on page 214.

With reference

— See "With reference" on page 781.

No dot generation

— See "No dot generation" on page 781.

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.
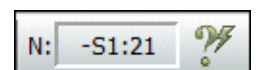
Editing connection texts from the Component data box

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 182.

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.

Appearance of connection points on symbols (terminals lists)

While you work on a project, the connection points for the symbols appears differently, depending on the choices you have made. This enables you to see whether for instance a connection point appears in terminal lists or whether a connection point on a terminal is a connection point for jumpers only.

General appearance of connection points on symbols (except terminals)
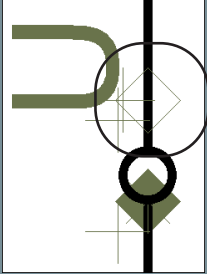
For all symbol types except terminals the following applies:

APPEARANCE OF CONNECTION POINTS FOR SYMBOLS (EXCEPT TERMINALS)

NON-FILLED RHOMBUS: THE CONNECTION POINT IS NOT INCLUDED IN TERMINALS LISTS.

FILLED SQUARE: THE CONNECTION POINT IS INCLUDED IN TERMINALS LISTS.

PC|SCHEMATIC
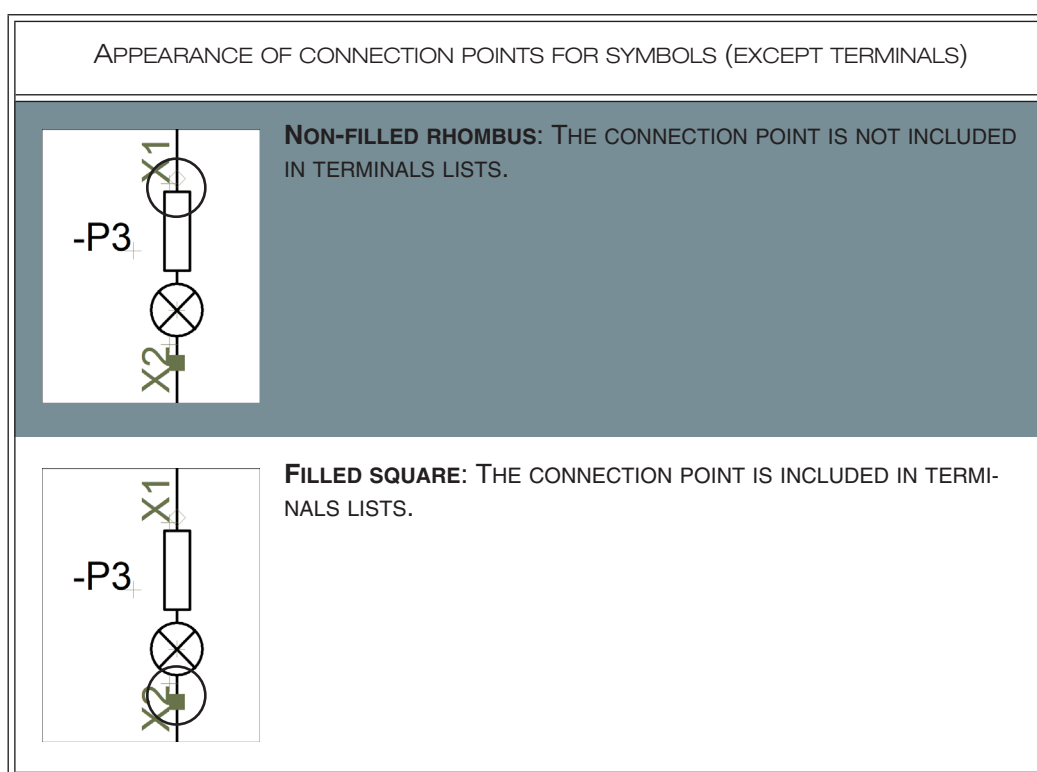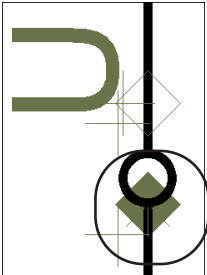
Appearance of connection points for terminals

When connection points on terminals are displayed as rhombuses/diamonds, this means that they are not included in terminals lists. When they appear as squares, this means that they are included in terminals lists:
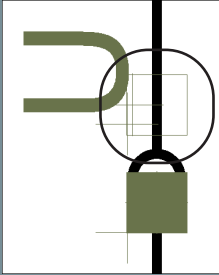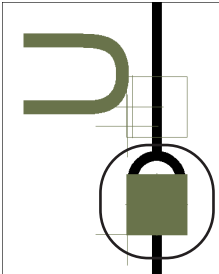
APPEARANCE OF CONNECTION POINTS FOR TERMINALS
NON-FILLED RHOMBUS: TERMINAL SIDE *SIDE 1 (INT)*. THE CONNECTION POINT IS *NOT* INCLUDED IN TERMINALS LISTS.
FILLED RHOMBUS: TERMINAL SIDE *SIDE 2 (EXT)*. THE CONNECTION POINT IS *NOT* INCLUDED IN TERMINALS LISTS.
NON-FILLED SQUARE: TERMINAL SIDE *SIDE 1 (INT)*. THE CONNECTION POINT IS INCLUDED IN TERMINALS LISTS.
FILLED SQUARE: TERMINAL SIDE *SIDE 2 (EXT)*. THE CONNECTION POINT IS INCLUDED IN TERMINALS LISTS.

Appearance of jumper connection points on terminals

Jumper connection points on terminals appears as triangles:
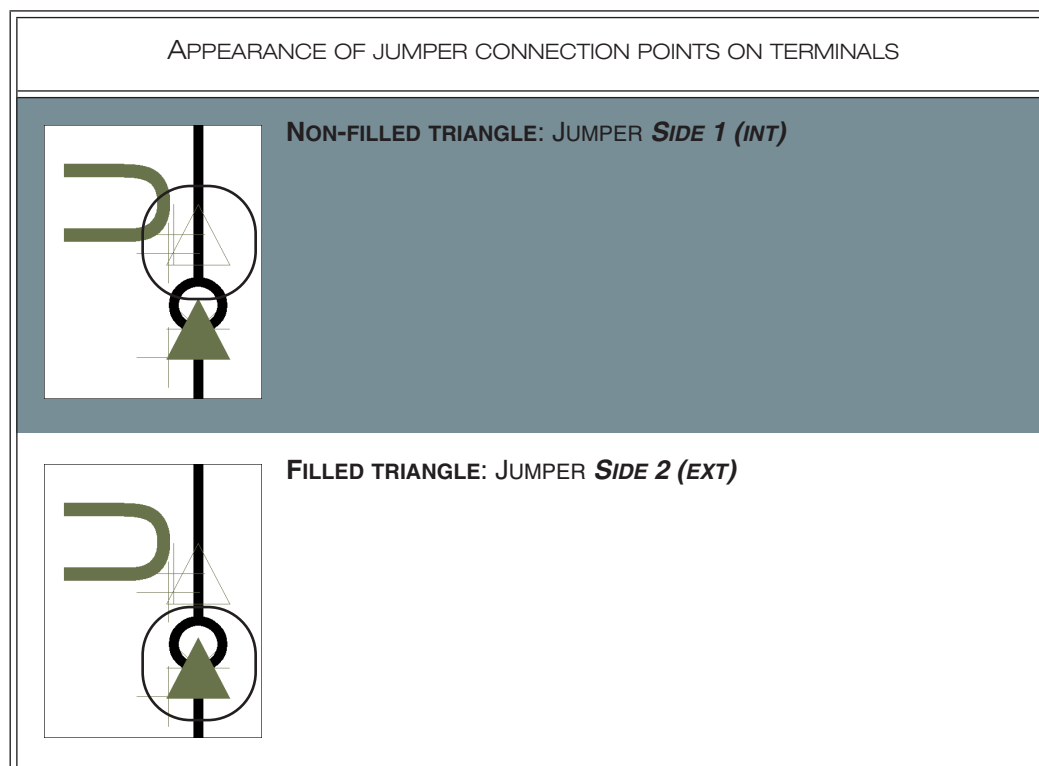
APPEARANCE OF JUMPER CONNECTION POINTS ON TERMINALS
NON-FILLED TRIANGLE: JUMPER *SIDE 1 (INT)*
FILLED TRIANGLE: JUMPER *SIDE 2 (EXT)*

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.
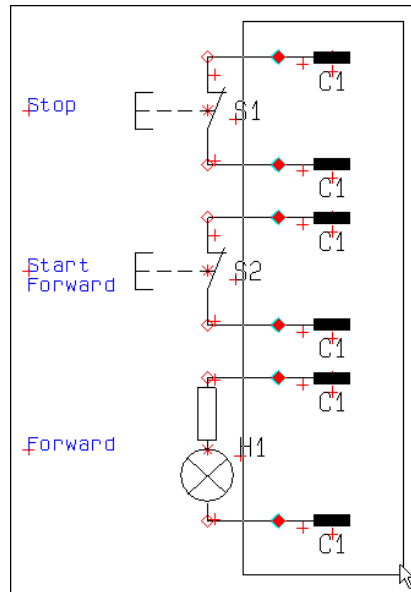
Select the connection points by pressing <Ctrl> when you release the mouse button
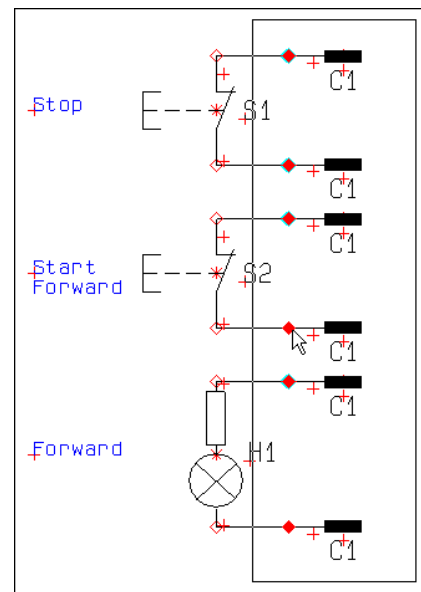
Press <Ctrl> and click on the unwanted connection points in the area

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 176 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.
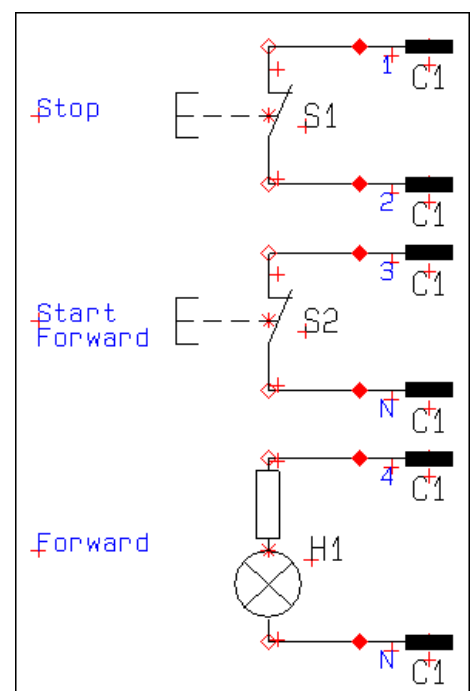
PIN SWOP

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.

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.
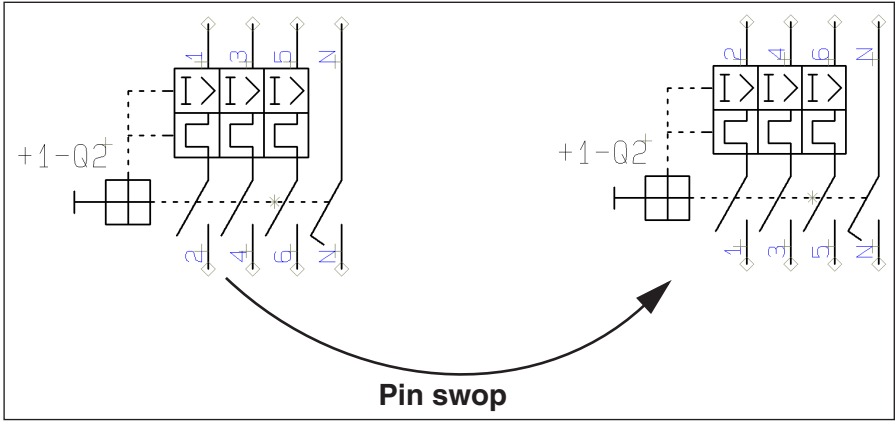
Pin swop

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.
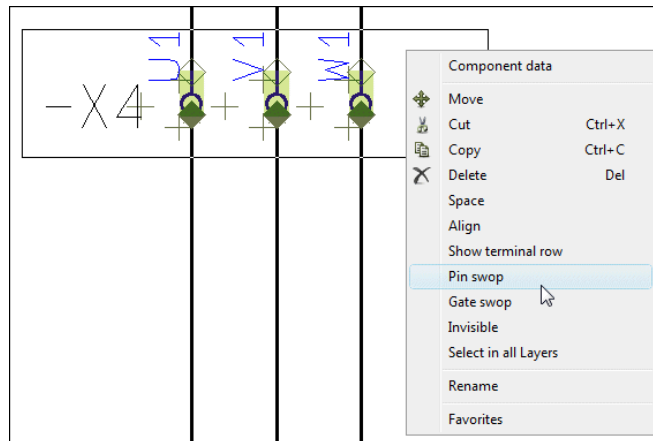
In this way you can for instance easily change a terminal row, from having the output side turning upwards to turning downwards. — Please refer to "Creating terminal symbols" on page 798 for further details about the input- and output side of terminals.

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

Swopping 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.
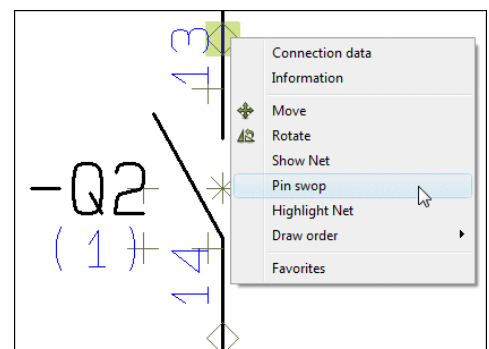
Now choose **Pin swop**, and you get a rubber band in the cross hairs. Then click on the connection point you want to swop with, and the connection points swop places.

Please note, that this swopping can be performed within the same symbol, or from one symbol to another.
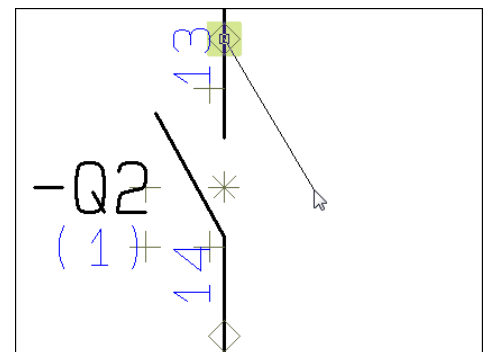
Moving connection points

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

Replacing symbols automatically

In **Automation** you can have all occurrences of a specific symbol replaced automatically. (Also see "Replacing all symbols in a project" on page 224).

If for instance you decide to use 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:

Click here to keep the position of the function text

Click here to re-place all symbols in the project or on the page

Choose a symbol in the Symbolmenu

Click here to select the symbol
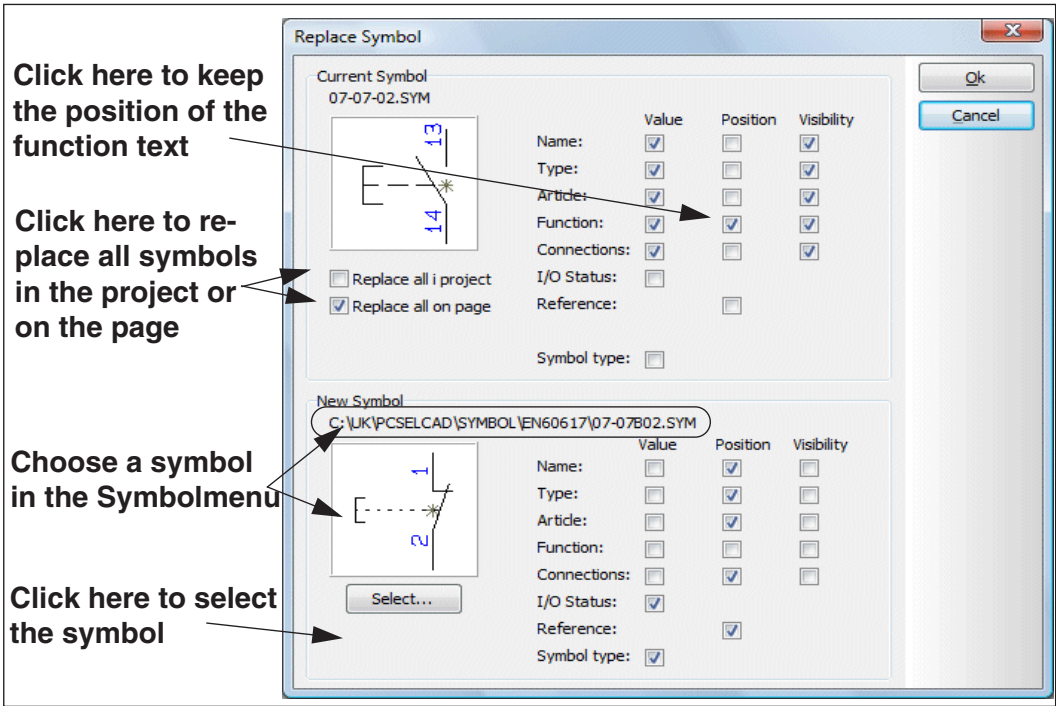
When you enter the dialog box, it is presumed that you wish to keep the contents/value of the symbol texts from the *old* symbol, while the positions of the texts follows the text-positions for the *new* symbol.

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.

IGNORE SYMBOL PATH

When you select **Replace all in project** or **Replace all on page**, you also get the option of selecting **Ignore symbol path**.

When you select **Ignore symbol path**, all symbols with the selected symbol file name are replaced — no matter where the symbols were fetched from originally.

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*.
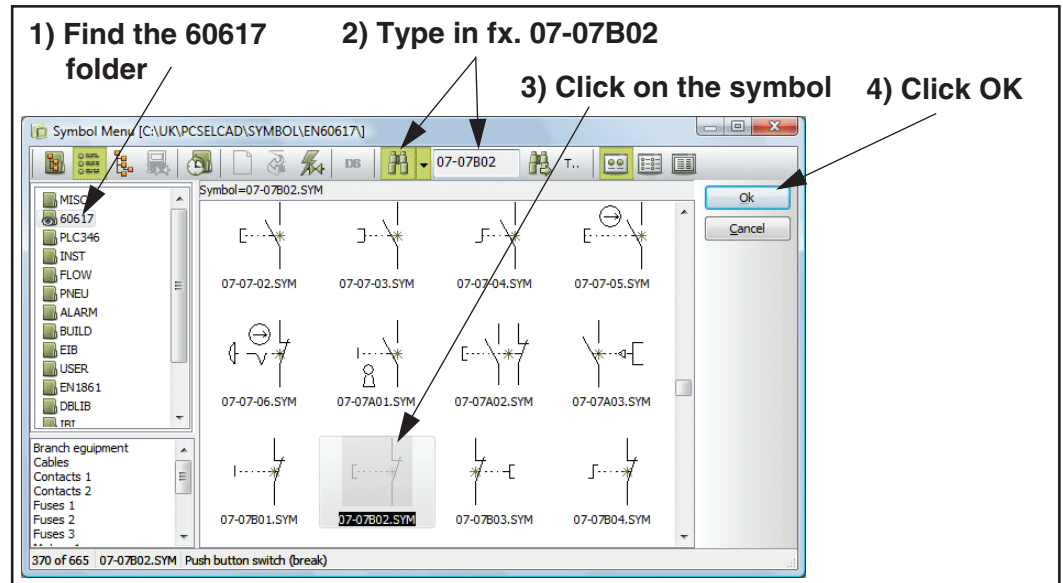
Click **OK**. — See "Fetching symbols using the Symbol menu" on page 162.

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:
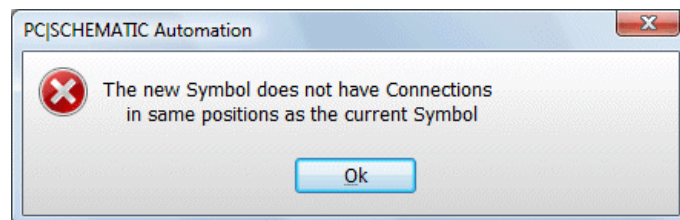
Click **OK**. However, you can still choose to replace the symbol(s) by clicking **OK** in the **Replace Symbol** dialog box.

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 an older version exists of a symbol in the project, if it has a question mark (**?**) in its symbol file name.

If multiple versions of the selected symbol exists in the project, the field **All versions(?)** is displayed in the left-hand side of the dialog box. If you select this option, all versions of the symbol on the page or in the project are replaced.

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 775.

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 221.

Replacing all symbols in a project

When a symbol is placed in a project, the symbol is saved under the *symbol file name* for the symbol (e.g. *Contactor1.sym*). Where the symbol originally came from then makes no difference for the program.

However, several symbols with the same symbol file name can occur in a project.

You can therefore use the function **Functions => Special functions => Replace all symbols**. It reloads all symbols in the active project based on the order of symbol libraries specified under **Alias for directories**. — See "Alias for directories" on page 924.

Replacing all symbols with symbols from another symbol directory

This special function also enables you to replace all symbols in a project with simi-lar symbols — symbols with the same *symbol file name* — that are placed in another symbol directory.

This requires that you change the order of the symbol libraries under **Alias for directories** and then choose **Functions => Special functions => Replace all symbols**.

AUTOMATIC GENERATION 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.

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 226. (If you enter the **Symbol generator**, you can enter the *Edit symbol* mode from there, and continue working on the symbol as described in "Creating Symbols" on page 775).

b) By manually typing in the syntax for the appearance of the symbol: Read "Automatic generation of electrical and mechanical symbols - By syntax" on page 228.

Automatic generation of symbols — using the Symbol generator

You can enter the **Symbol generator** in four different ways:
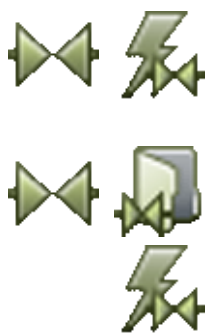
a) Select **Symbols** and click the **Symbol generator** button.

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 present settings. The syntax for the generated symbol is displayed in the title bar for the dialog box:

Here the syntax for the gene-rated symbol is displayed

Here the symbol is displayed based on the specified settings

Click here to continue working with the symbol design in the Edit symbol mode
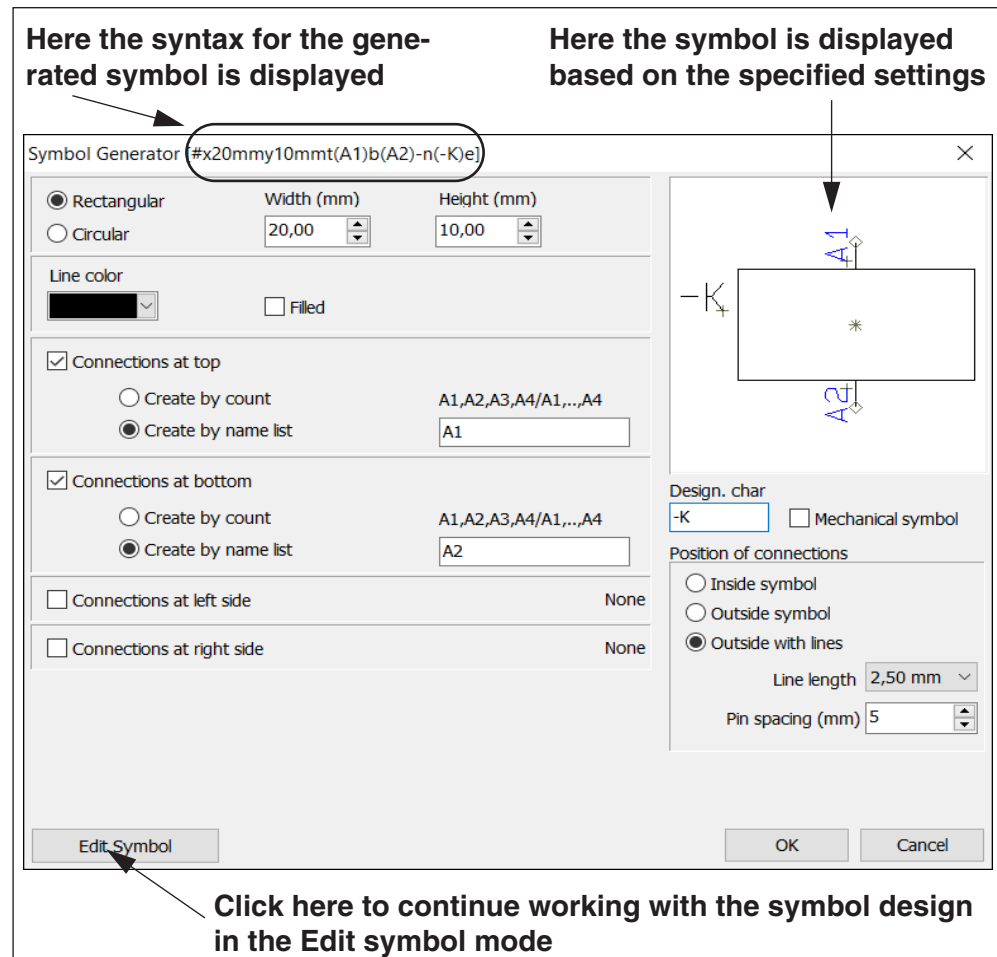
SELECT RECTANGULAR OR CIRCULAR

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.

FONTS AND TEXT SIZES IN THE SYMBOL GENERATOR

THE TEXTS IN THE GENERATED SYMBOLS ARE ASSIGNED THE SAME TEXT SIZES AND FONTS, WHICH ARE USED IN THE ACTIVE PROJECT WHEN THE SYMBOL GENERATOR IS STARTED.

CONNECTIONS: NUMBER, NAMING AND POSITION

You can place connections to the symbol on the top, at the bottom, and at the right and the left side of the symbol.

All of these places, you can either specify a number of connections (which names the connections 1,2,3,...), or specify the names by typing in a name list. — Please remember to see the selected connections in the upper-right corner of the dialog box.

SPECIFYING A DESIGN CHARACTER

In the right side of the dialog box, you can specify a designation character for the symbol. Please note that it is not necessary to place a dash (-) before the symbol name, because the program can handle this for you automatically for all symbols in the project. — See "Preceding "-" in symbol names" on page 415.

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**, and can specify the spacing between the pins in the field **Pin spacing (mm)**.

Automatic generation 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:

Place Symbol	
Enter Symbol File Name:	**Ok**
#x40mmy50mml2r2	**Cancel**
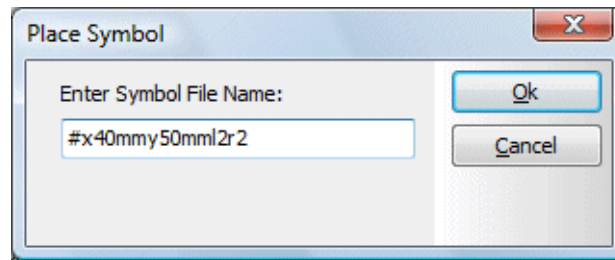
2) Type in the dimensions of the symbol, and the number of connection points — see "Syntax for automatic generation of symbols" on page 229.

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.
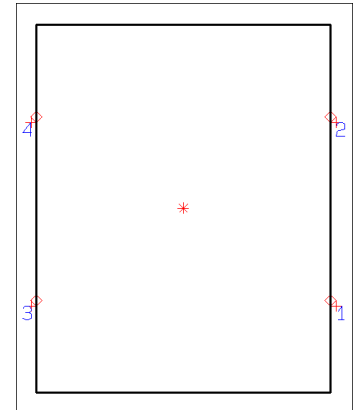
Symbol type for automatically created symbols

Symbols, which are created automatically, are assigned the symbol type **Normal**.

SYNTAX FOR AUTOMATIC GENERATION 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.
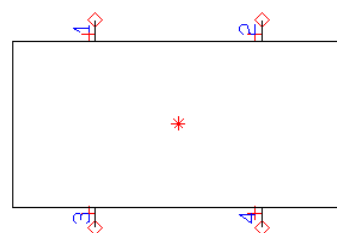
Placing the connection point names

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 generation 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 generation of symbols" on page 229.

Automatic generation 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 in stead write **#r40mm**, the program creates a circle with radius40 mm as a mechanical symbol. Read more about this in "Syntax for automatic generation of symbols" on page 229.

— Read more about mechanical symbols in "Mechanical symbols in the database" on page 674.

NET NAVIGATOR

You can use the **Net Navigator** to get an overview over nets (potentials) and connections (wires, cable wires and jumper links) in the active project. Despite its tiny appearance this is a powerful tool for navigating the nets and connections in the active project.

Using the Net Navigator

Start the **Net Navigator** by choosing **Tools => Net Navigator** — or by clicking the **Net Navigator** button in the program toolbar.

You then enter the **Net Navigator**:
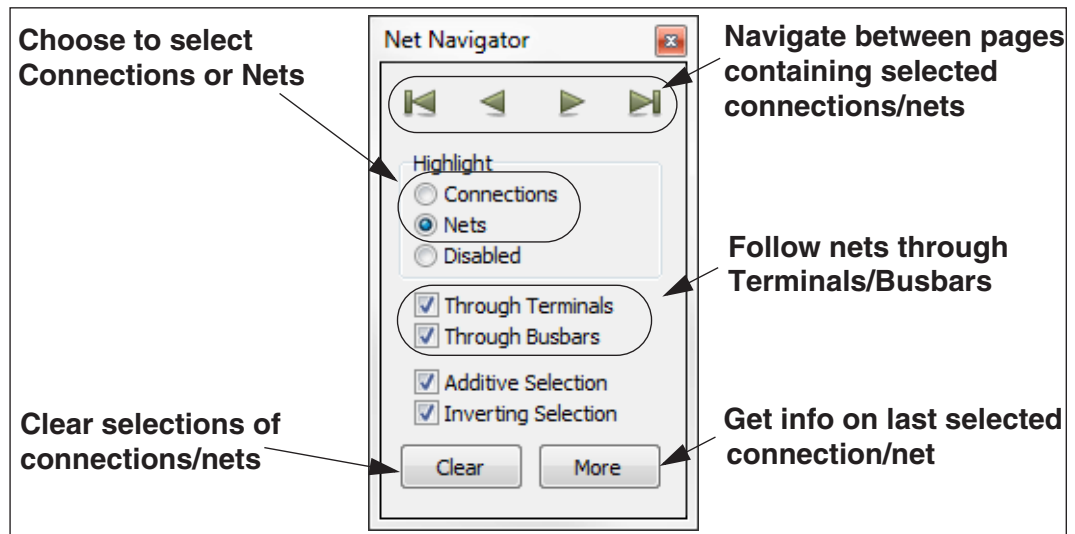
How to select connections/nets in the Net Navigator

1) Specify whether to select **Connections** or **Nets**

2) Make sure that **Additive Selection** *is selected* (if you wish to select multiple connections/nets) and make sure that **Inverting Selection** is *not* selected

3) Click on a line in the project which is a part of the net/connection that you wish to select. The connection/net is now selected in the project and displayed in the **Net Navigator**

4) More connections/nets can be selected correspondingly

You can now navigate among the pages containing the selected connections/nets by clicking the navigation buttons in the **Net Navigator**.
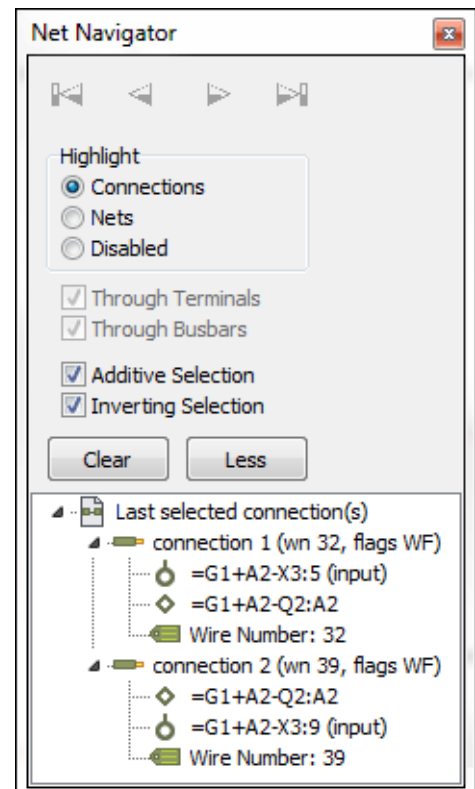
How to remove selections in the Net Navigator

This is how to remove individual selections of nets/connections:

> 1) In the **Net Navigator**: Select *Inverting Selection*

> 2) Click on the selected nets/connections, that you wish to deselect

Click **Clear** if you wish to deselect all selections in the project.

The Net Navigator remains on the screen

The **Net Navigator** remains on the screen until you close it.

When you click on *Deactivated*, you can let the **Net Navigator** remain on the screen without the risk of accidentally selecting or deselecting more connections/nets.

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 729.

— See also "Connections list for wire numbering" on page 468.

SIGNAL SYMBOLS

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 87.

USE THE NET NAVIGATOR

YOU CAN ALSO USE THE **NET NAVIGATOR** TO EASILY MAINTAIN THE OVERVIEW OVER NETS AND CONNECTIONS IN THE PROJECT.

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**.
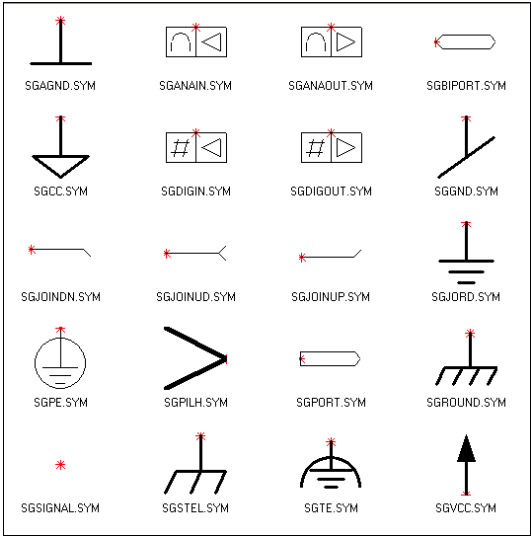
Designing signal symbols

See how to design your own signal symbols in "Creating Signal Symbols" on page 816.

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**:
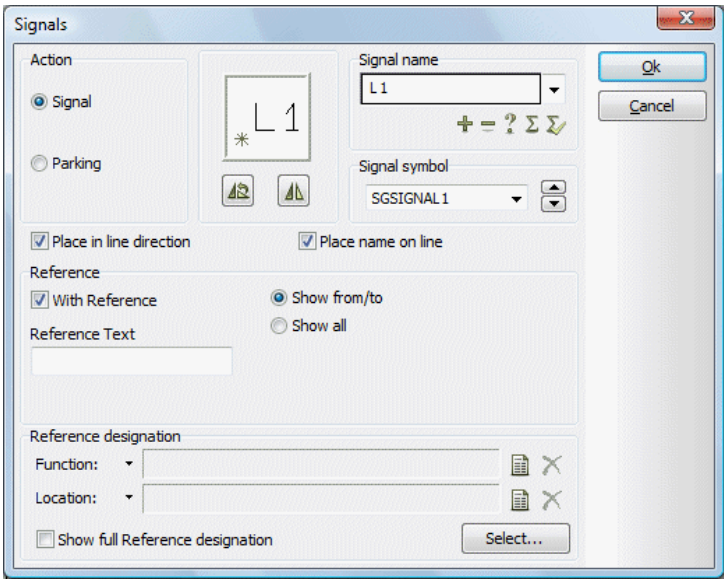
Parking

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.

When you later want to place a signal symbol, just right-click on the parked end of the line and choose **Place signal symbol**.

— See how to locate parkings in a project in "Design Check" on page 365.

Placing the signal symbol

If you, on the other hand, wish to place a signal symbol, click on **Signal**.

SPECIFY SIGNAL NAME

In the field **Signal name** you can either type in a name, or use one of the following options:

BUTTON	FUNCTION
	INCREASE THE NUMBER IN THE SIGNAL NAME BY ONE.
	DECREASE THE NUMBER IN THE SIGNAL NAME BY ONE.
	GIVES YOU THE NEXT AVAILABLE SIGNAL NAME OF THE SYMBOL TYPE CHOSEN.
	GIVES YOU A LIST, WHERE YOU CAN CHOOSE BETWEEN THE SIGNAL NAMES ALREADY USED IN THE PROJECT.
	GIVES YOU A LIST OF USED VARIANT NAMES FOR THE SELECTED SIGNAL SYMBOL — SEE "VARIANT NAMES ON SIGNAL SYMBOLS" ON PAGE 239.

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 819 to see how you can add names to this list by yourself.

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, IF A FUNCTION TEXT HAS BEEN SPECIFIED FOR THE CONNECTION POINT, FROM WHICH THE LINE IS BEING DRAWN. WHEN YOU CLICK THIS BUTTON, THE SIGNAL NAME IS FETCHED FROM THE *FUNCTION TEXT* FROM THE CONNECTION POINT, FROM WHICH THE LINE STARTED:
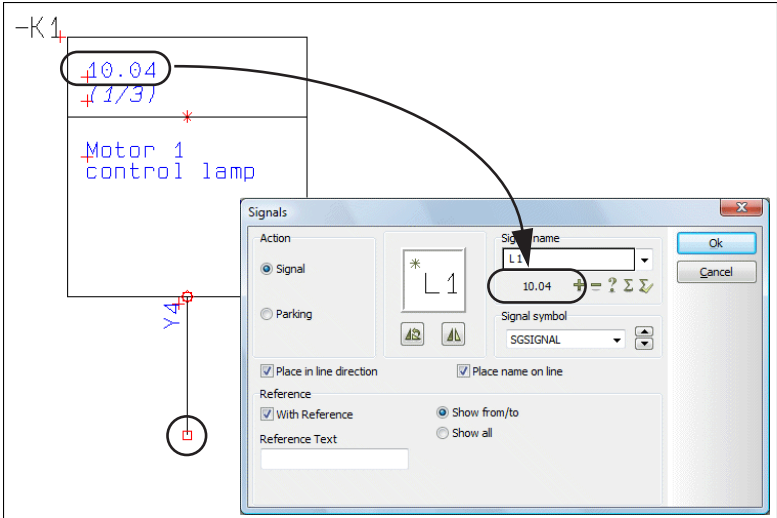
WHEN FOR INSTANCE THE LINE STARTS FROM A PLC CONNECTION POINT — AS IN THE FIGURE ABOVE —, YOU CAN THEREFORE EASILY ASSIGN THE ADDRESS FOR THE PLC CONNECTION TO THE SIGNAL NAME. THE SHORTCUT KEY FOR THIS FUNCTION IS <ALT+S>.

WHEN A FUNCTION TEXT HAS *NOT* BEEN SPECIFIED FOR THE CONNECTION POINT FROM WHICH THE LINE IS BEING DRAWN, EITHER *CONNECTION NAME* OR *COMPONENT NAME:CONNECTION NAME* CAN BE ASSIGNED AS THE SIGNAL NAME.
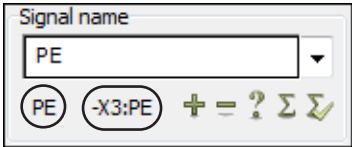
SELECT SIGNAL SYMBOL

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 818.

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.
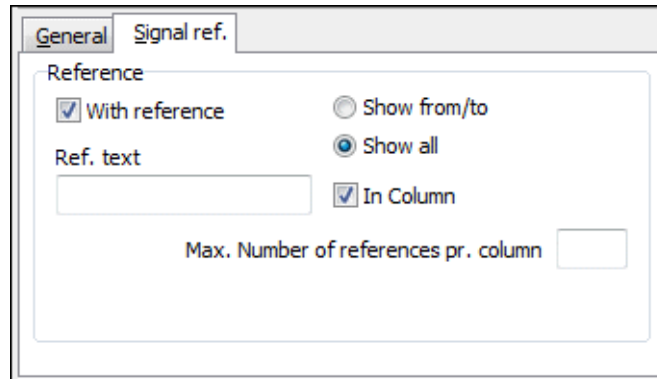
When you select **Show all** and **In Column**, you can specify the maximum number of references in each column in the field **Max. Number of references pr. column**.

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 412, or:

b) Click on the button **Insert current page functional/locational aspect** for the fields **Function** and/or **Location**. This assigns the Functional/Locational aspect to the signal name.

If you click the **Delete** button for the fields **Function** / **Location**, the displayed functional/locational aspect is deleted.
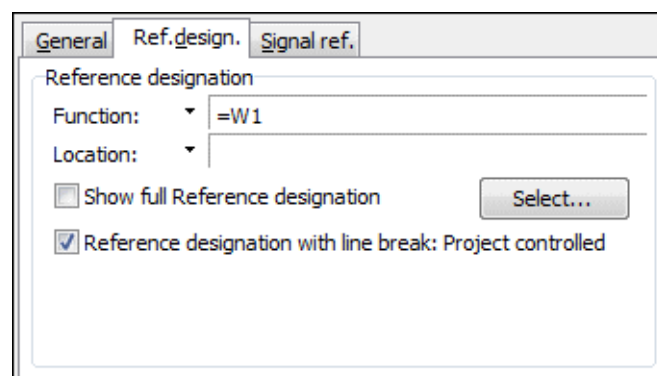
If you click on **Show full Reference designation**, the full reference designation is displayed for the symbol. If not, only parts of the reference designation different from the page reference designation, are displayed.

When you are contempt with the settings, click **OK**, otherwise click **Cancel**.

Placing signal symbols on parked lines

To get the signal symbol in the cross hairs, click the **Symbols** button, and fetch a signal symbol — just like you would fetch any other symbol. Read more about this in "Fetching symbols" on page 154.

When you have the signal symbol in the cross hairs, you can place it on a parked line. — See "Parking" on page 235.

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.

Reference to other project page (double-click to jump)

Supply panel

Using Variant names, you can follow the signal branches in the project
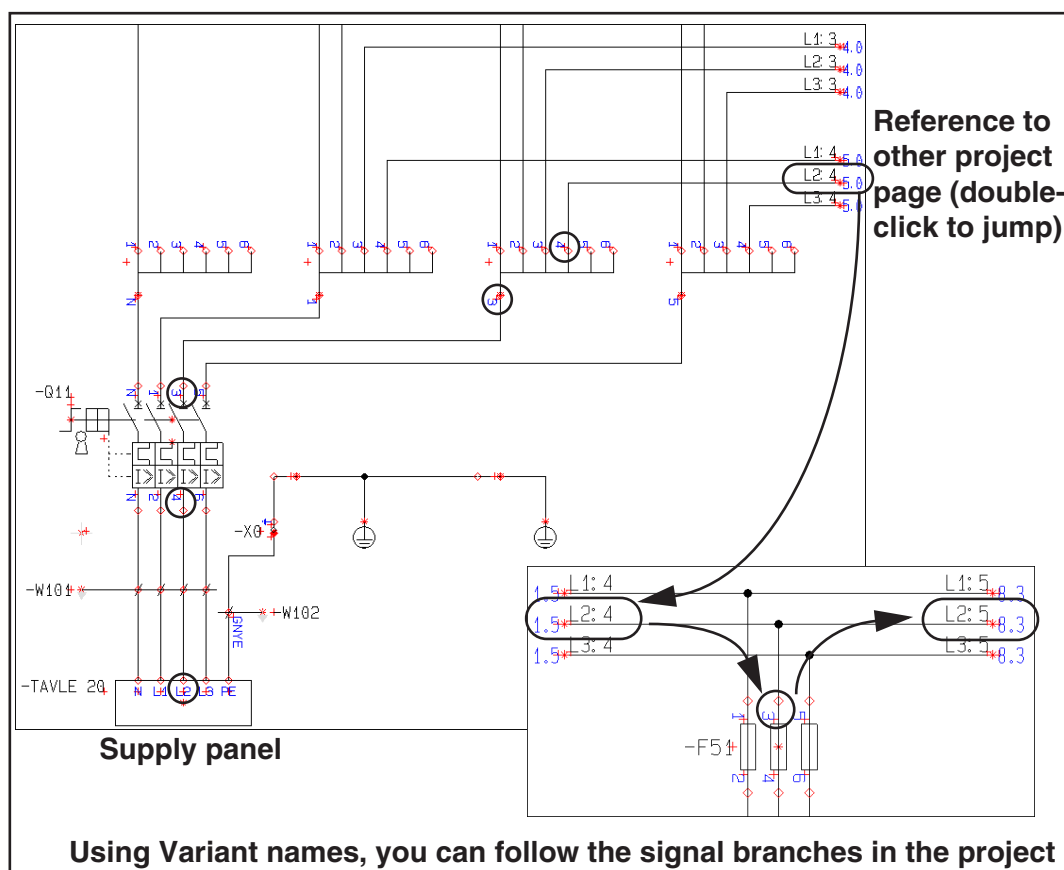
Comments on variant names

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 274.

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*.

☑ Insert Reference designation on Signals
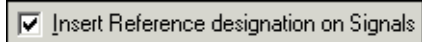
When you place signal symbols on *pages* with assigned reference designations, the signal symbol will then get these reference designations.

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 255.

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:
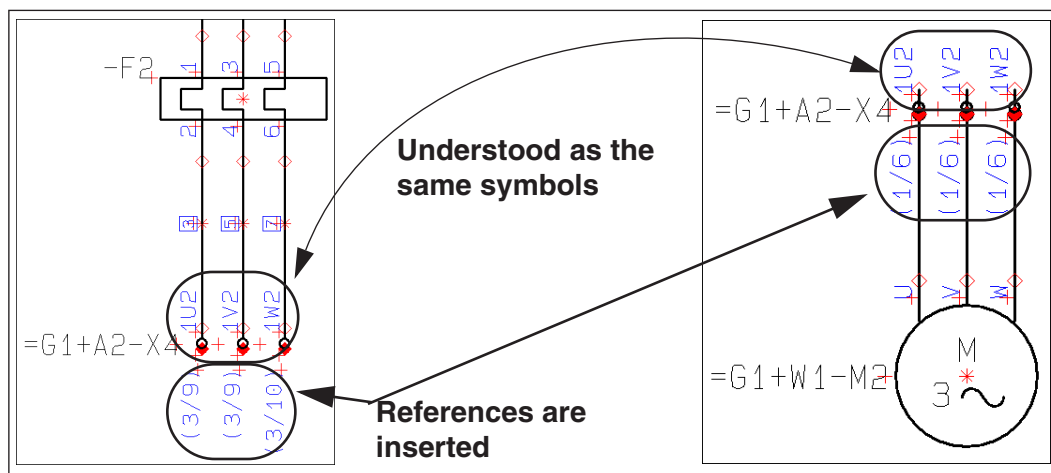
The terminal symbols in the figure above are understood as representations of the same symbols, because:

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
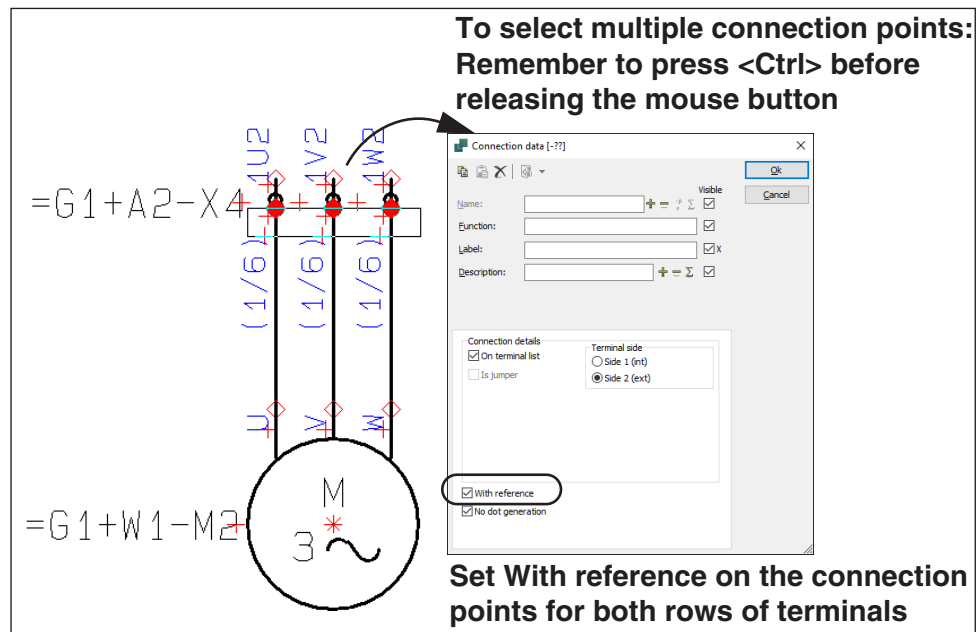
Implicit signals on cable wires

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.

Understood as the same cable wires

Inserted
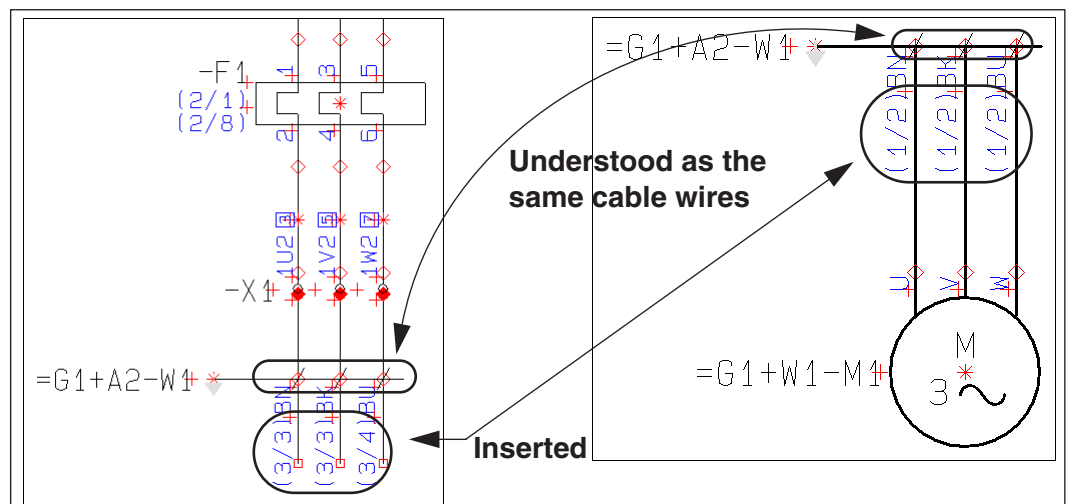
The cable wires in the figure above are understood as the same symbols, because:

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

PC SCHEMATIC

3) Both sets of cable wires are only connected in one direction

Set to With reference on the connection points for both cables
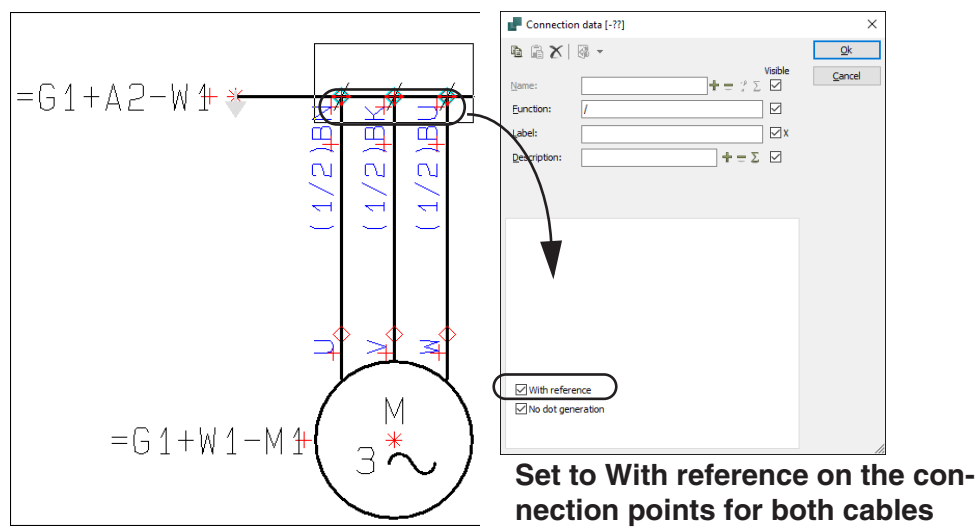
TERMINAL SYMBOLS

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 839.

— See "Multi layer terminals" on page 246.

— See "Multiplacing symbols" on page 191.

Input and Output on terminal symbols

On terminal symbols you identify the *output* connection point (called **Side 2**) by the fact that the connection point is filled with color, which the input connection point (called **Side 1**) is not.
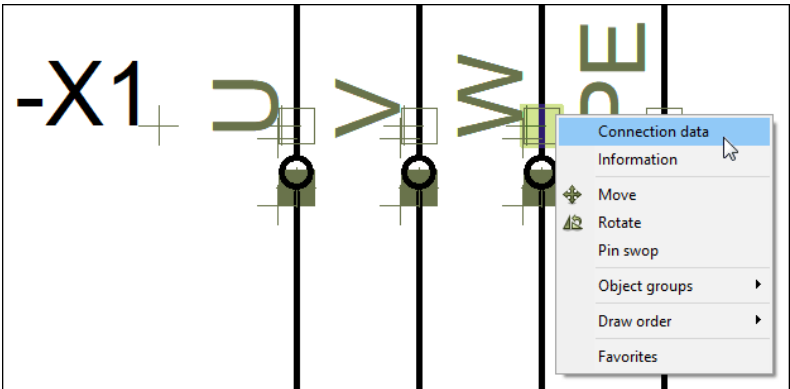
If you want to change whether a connection point is *input* or *output*, you click the **Symbols** button, right-click on the connection point, and choose **Connection data**.

Under **Connection details** you either choose **Side 1 (int)** for input or **Side 2 (ext)** for output.
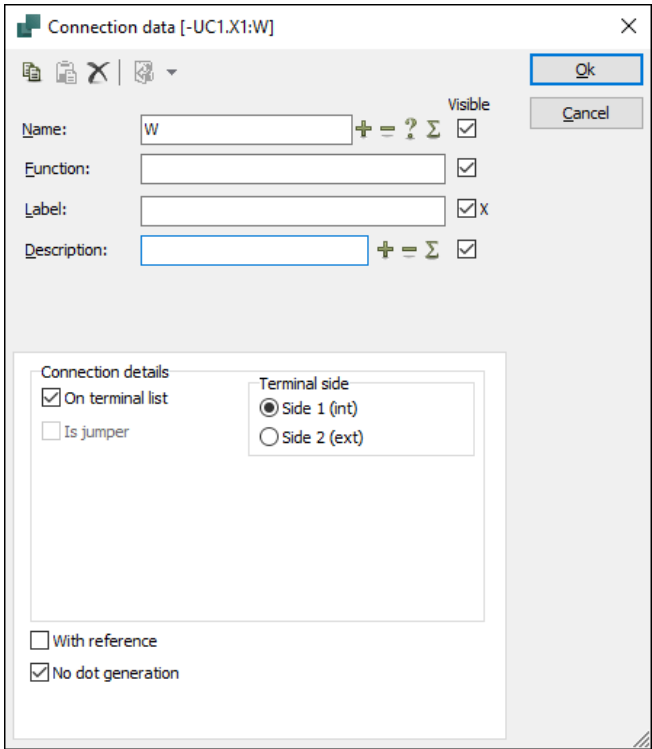
If you accidentally select the entire symbol instead of the connection point, then right-click on the connection point once more.

Connection points for terminal symbols on terminals lists

To include a connection point on a terminal symbol in terminals lists, do this:

1) Open the dialog box **Connection data**, as explained above

2) Select ***On terminal list***

You can see directly on the diagrams that connection points are included in terminals lists. This is visualized by displaying the connection points as squares, which you can see in "Appearance of connection points for terminals" on page 215.

— See more about the appearance of connection points for other symbol types in general as well as for jumper connection points in "Appearance of connection points on symbols (terminals lists)" on page 214.

Changing connection names for terminal rows

— See "Renumbering pinnumbers for a terminal row using Select name" on page 276.

Designing terminal symbols

See how to design terminal symbols in "Creating terminal symbols" on page 798.

MULTI LAYER TERMINALS

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 626.

Multi layer terminals on diagram pages

When you fetch multi layer terminals from the database, the first layer is displayed first in the **Symbols for component** window in the lower part of the **Component menu**.

(In version 19 and previous versions, it was *the order* in which the symbols were placed in project, that determined the layer. This is no longer the case.)

Layer 1   **Layer 2**
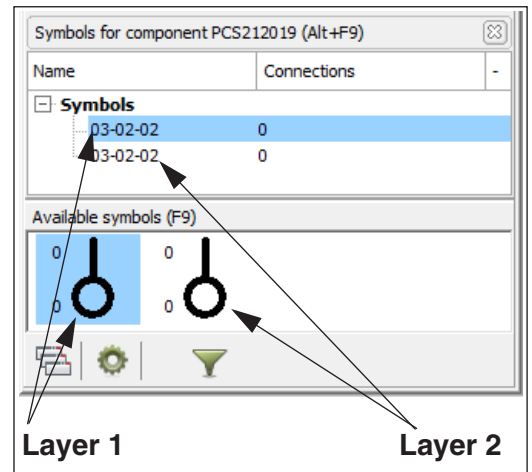
The position numbers can be specified/changed in **Functions => Component Grouping**.

— Please also see "Graphical managing of component grouping for terminals" on page 633.

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.
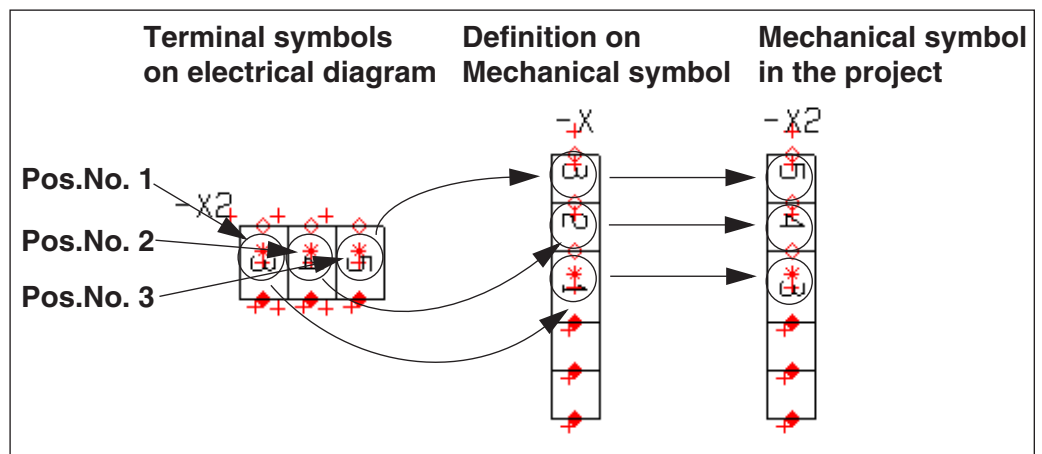
WORKING WITH CABLE SYMBOLS

This section deals with how to apply cable symbols on your diagrams.

— See "Creating cable symbols" on page 800, "Creating busbar symbols" on page 802 and "Export to PCSCHEMATIC Cablemanager" on page 748.

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*.

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:

Specify the number of wires in the cable
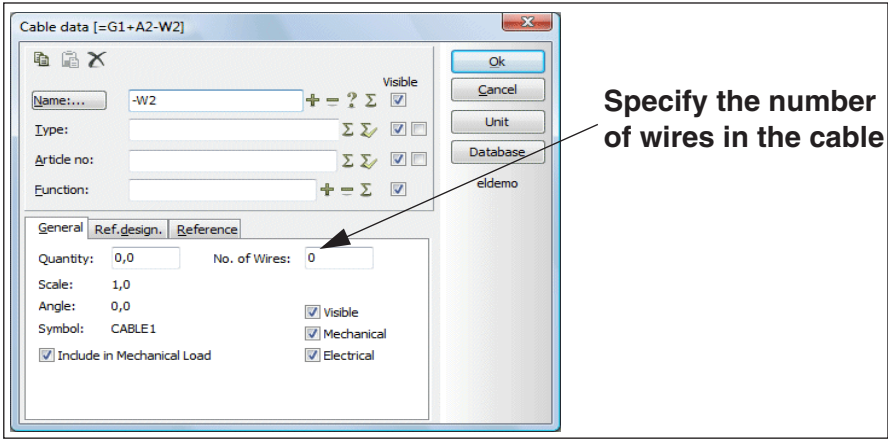
Number of cable wires in the cable

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 174. Click **OK**.

The illustration below is a part of a larger drawing, where the three vertical lines are wires.
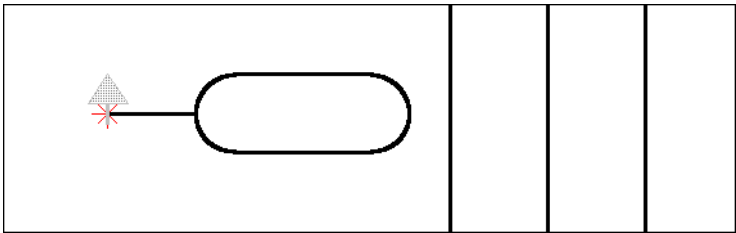
When you place the cable, a rubber band now starts in the reference point of the symbol, and ends up in the cross hairs. This indicates that you can now point out which wires to include in the cable.

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**.
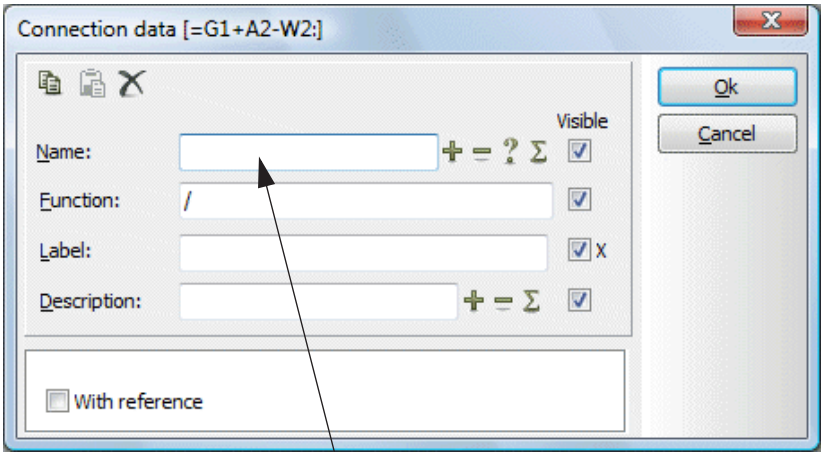
Double-click in the field to select Name from a list of cable wire names according to standards

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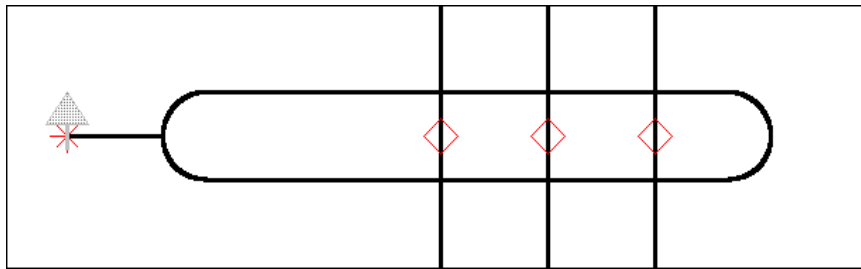
When you do not wish to include more wires into the cable you press <Esc>, and the "rubber band" disappears. If the number of cable wires is specified in the dialog box **Cable data**, the "rubber band" disappears when the specified number of cable wires have been placed in the project.

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:
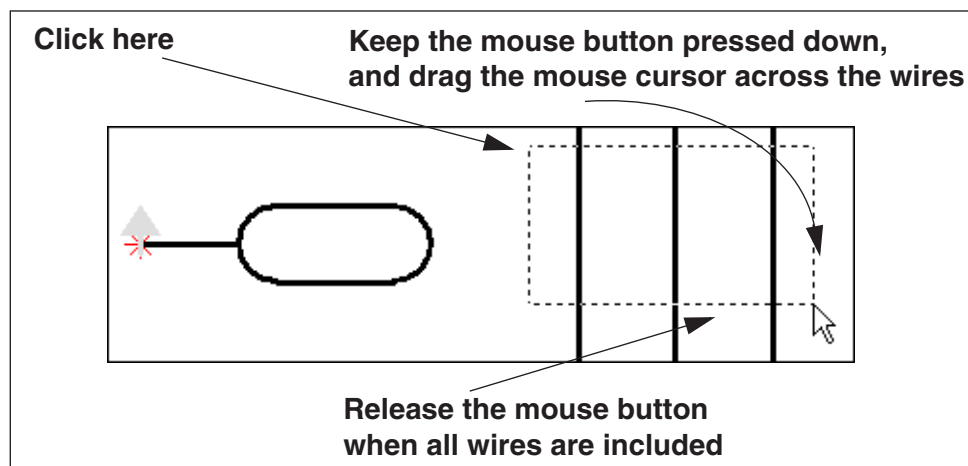
When you release the mouse button, you enter the dialog box **Connection data**, from where you can name cable wires — for instance by using automatic counting:
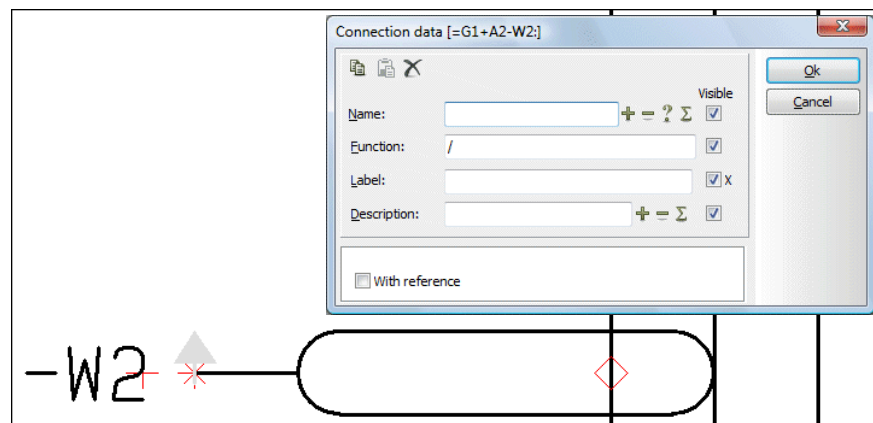
If for instance you write *1* in the ***Name*** field, press down <Ctrl> while clicking on **+**, and then click **OK**, the three wires in the example will get the names *1*, *2* and *3* respectively. — See e.g. "Automatic naming of symbols" on page 201.

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 248.

Only, when you select a cable wire, you enter this dialog box instead:
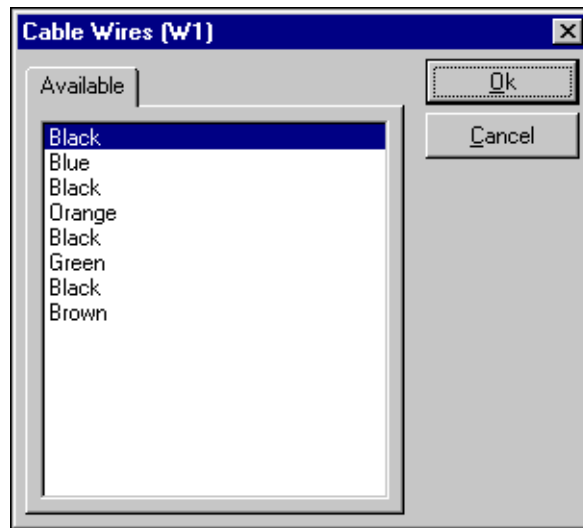
Here you choose which of the cable conductors you wish to apply, and click **OK**.

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:
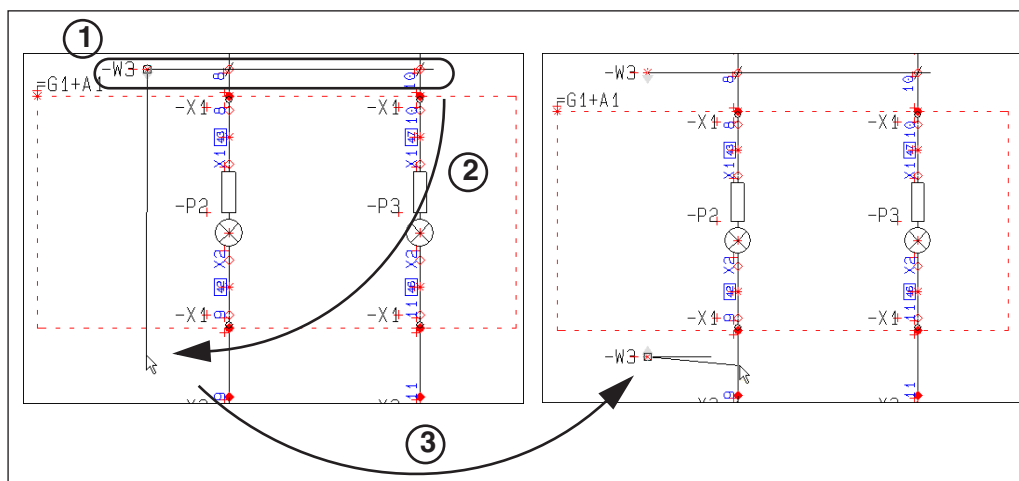
To do as in the figure above, do the following:

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.
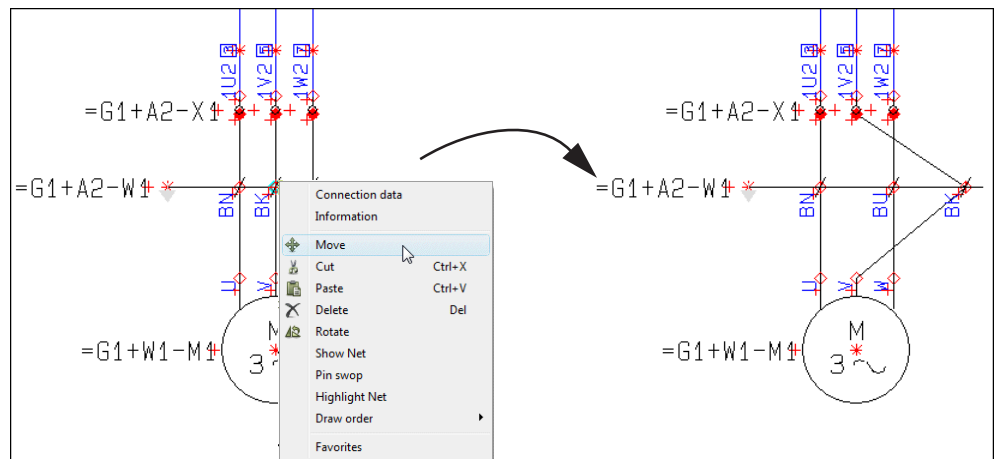
Moving connection points for cable wires in areas

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 251.

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 250.

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 801.

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:
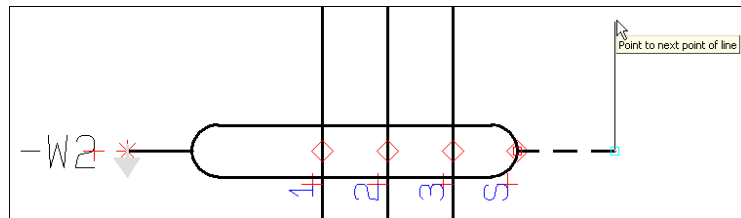
5) Change line type if necessary, and connect the line to the desired symbol.

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**. If you have selected multiple cables, the direction is reversed on all selected cables.

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 241.

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 BUSBAR SYMBOLS

— See "Creating busbar symbols" on page 802.

USING ACCESSORY SYMBOLS

To use an accessory symbol 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.

Support symbol for -Q1

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.
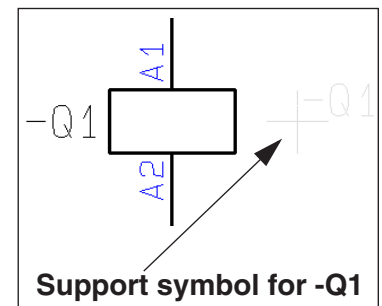
Data from accessory symbols are included in the lists, but the accessory symbol itself is not included when printing the diagram pages. Accessory symbols do not necessarily have to be attached to other symbols in the documentation.

RECOMMENDED TO INCLUDE ACCESSORIES IN OTHER WAYS
STARTING FROM VERSION 17, EASIER WAYS TO INCLUDE ACCESSORY INFORMATION HAVE BEEN INTRODUCED. PLEASE READ MORE ABOUT THIS IN "COMPONENT ACCESSORIES" ON PAGE 688 AND "OPTIONAL ACCESSORIES - MECHANICAL" ON PAGE 689.

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:\Pcselcad\SYMBOL**. Here is a survey of the contents of these folders:

FOLDER	CONTENTS
ALARM	SYMBOLS FOR ALARM SYSTEMS. NOT STANDARD. (THIS FOLDER IS NOT CREATED FOR NEW INSTALLATIONS OF VERSION 18 AND SUBSEQUENT VERSIONS. INSTEAD, THE DESCRIBED CONTENTS IS PLACED IN THE **60617_2015** FOLDER).
ANSI	SYMBOLS FOLLOWING THE AMERICAN ANSI 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. (THIS FOLDER IS NOT CREATED FOR NEW INSTALLATIONS OF VERSION 18 AND SUBSEQUENT VERSIONS. INSTEAD, THE DESCRIBED CONTENTS IS PLACED IN THE **60617_2015** FOLDER).
DEMOMEC	EXCLUSIVELY MECHANICAL SYMBOLS FOR SWITCHBOARD DRAWINGS. AT PRESENT THE FOLDER ONLY CONTAINS EXAMPLES USED IN "AUTOMATION PROJECT TUTORIAL" IN THE ***PCschematic Tutorial*** 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.

FOLDER	CONTENTS
EN60617	THE FOLDER CONTAINS SYMBOLS ACCORDING TO THE EUROPEAN STANDARD IEC/EN 60617, GRAPHICAL SYMBOLS FOR DIAGRAMS. THE SYMBOLS FOLLOWS IEC/ISO 81714-2, CONSTRUCTION OF GRAPHICAL SYMBOLS. THE SYMBOL NAMES FOLLOWS THE EN 61346 STANDARD FOR LETTER CODES. PCSCHEMATIC HAS A LICENSE FOR USING THE IEC STANDARDS. THE SYMBOLS ARE SAVED UNDER THEIR SERIAL NUMBER, FOR INSTANCE 04-02-14. 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. 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. 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 918. 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. (THIS FOLDER IS NOT CREATED FOR NEW INSTALLATIONS OF VERSION 18 AND SUBSEQUENT VERSIONS. INSTEAD, THE DESCRIBED CONTENTS IS PLACED IN THE **60617_2015** FOLDER).
60617_2015	*SPECIAL INFORMATION FOR UPDATING TO VERSION 18*: ALL 60617 SYMBOLS HAVE BEEN UPDATED SO THAT THE POSITION AND SIZES OF THE TEXTS FOLLOWS THE STANDARDS IEC 60617 AND IEC/ISO 81714-2 (SYMBOL DESIGN), AND ARE ALSO ADDED THE NEW S-NO IN THE DESCRIPTION. THIS INCLUDES THE SYMBOLS IN THE FOLDERS **EN60617**, **DB60617**, **IECINST** AND **ALARM**. ALL OF THESE NEW VERSIONS OF THE SYMBOLS FROM THESE FOLDERS ARE PLACED IN THE *60617_2015* FOLDER. IF YOU HAVE CHOSEN TO *UPDATE* TO VERSION 18 THE FOLDERS **EN60617**, **DB60617**, **IECINST** AND **ALARM** REMAINS WITH UNCHANGED CONTENTS. IF YOU HAVE CHOSEN TO MAKE A *NEW INSTALLATION* OF VERSION 18, THE FOLDERS **EN60617**, **DB60617**, **IECINST** AND **ALARM** ARE NOT CREATED.
FLOW-CHAR	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. (THIS FOLDER IS NOT CREATED FOR NEW INSTALLATIONS OF VERSION 18 AND SUBSEQUENT VERSIONS. INSTEAD, THE DESCRIBED CONTENTS IS PLACED IN THE **60617_2015** FOLDER).
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.

Folder	Contents
MISC	Special symbols, which are not official standard, but are included in **Automation**: Signal symbols, cable symbols, logos, PLC reference symbols etc. (This folder is not created for new installations of version 18 and subsequent versions. Instead, the described contents is placed in the **60617_2015** folder).
MISC_2015	Contains the most recently updated version of the MISC folder.
PBSYM-BOL	Symbols for the Panelbuilder. In order to read more about the Panelbuilder, please download the tutorial for the Panelbuilder at www.pcschematic.com/manuals
PLC346	PLC symbols are created in cooperation with a board of PLC vendors. Not standard.
USERLIB	Folder for your own symbols.

FUNCTIONS ON COMPONENT LEVEL

Automation has a component based workflow which encourages working with an attached database containing information about the components that you use in your projects.

This means that when you work 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 how to work at component level in **Automation**. In "Symbols" on page 152 you can see more about how to work at symbol level.

- See how to create your own components in the database in "Creating components in the database" on page 659.

- There is also a **Component Wizard** which guides you through creating new components. It is activated by clicking the **Component Wizard** button in the program toolbar, or by choosing **Functions => Component Database => Component Wizard**. It is described in an independent mini manual, which you find under **Help => Component Wizard manual**.

RENAME COMPONENTS / SYMBOLS

IN "RENAME SELECTED SYMBOLS" ON PAGE 332 YOU CAN SEE HOW TO RENAME SELECTED COMPONENTS/SYMBOLS — OR HOW TO RENAME ALL COMPONENTS/SYMBOLS IN THE PROJECT.

THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS

In order to understand the component based workflow in **Automation**, it is important to understand the relationship between *components* and *symbols*.

Components have Electrical and Mechanical symbols

In **Automation** *symbols* are used in the documentation as a graphical representation of the physical components — *electrical* symbols on diagram pages and *mechanical* symbols on mechanical pages.

A *component* in **Automation** has a unique component name. All symbols having the same component name are therefore understood as being a part of the same component. The components can have both electrical an mechanical symbols and article data attached.

The components have attached *one* electrical symbol for each function in the component. Components containing more than one function therefore have multiple electrical symbols attached.

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

Components can also have one or more mechanical symbols attached.

It is also possible to attach other kinds of symbols, such as pneumatic symbols, to the components. Read more about this in "The page database field for symbols" on page 266.

Specifying that symbols are part of the same component

In order to specify that symbols belongs to the same component, they must have the same component name:

**Symbols for -K1, placed on different project pages
(Two electrical symbols, and one mechanical)**
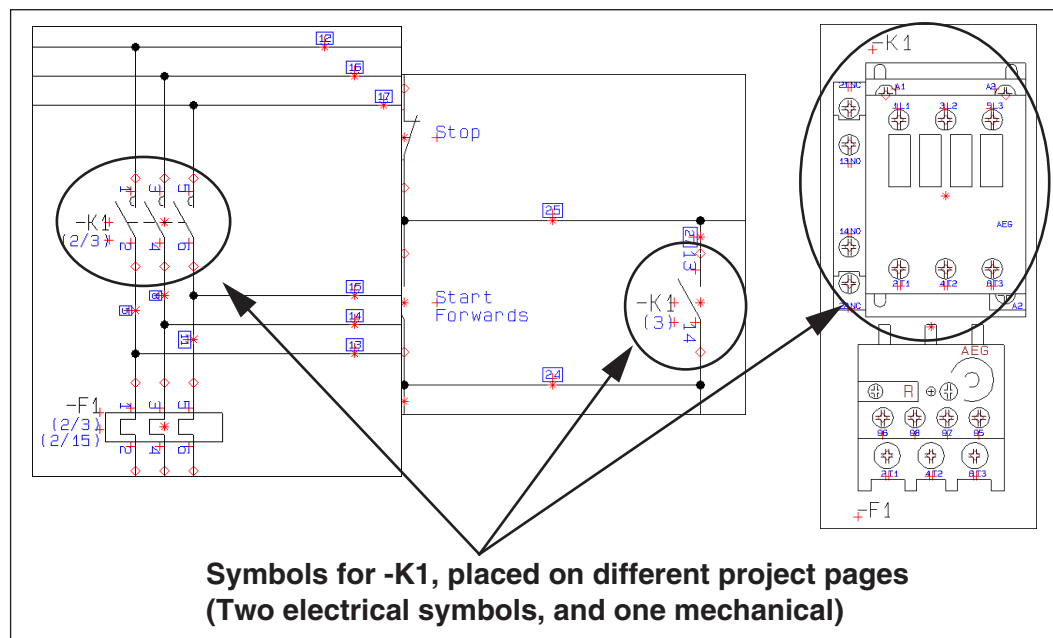
When you type in the same symbol name for two electrical symbols in a project, this specifies that they are part of the same component — please also see "Function assemblies - with multiple article numbers" on page 261.

When you are on a diagram page and choose a component via the *database*, the electrical symbols for the component are displayed under **Available symbols** at

the bottom of the **Component menu**. From there you can place the symbols in the project diagrams. — See "Fetching symbols for a Component from the Database" on page 262.
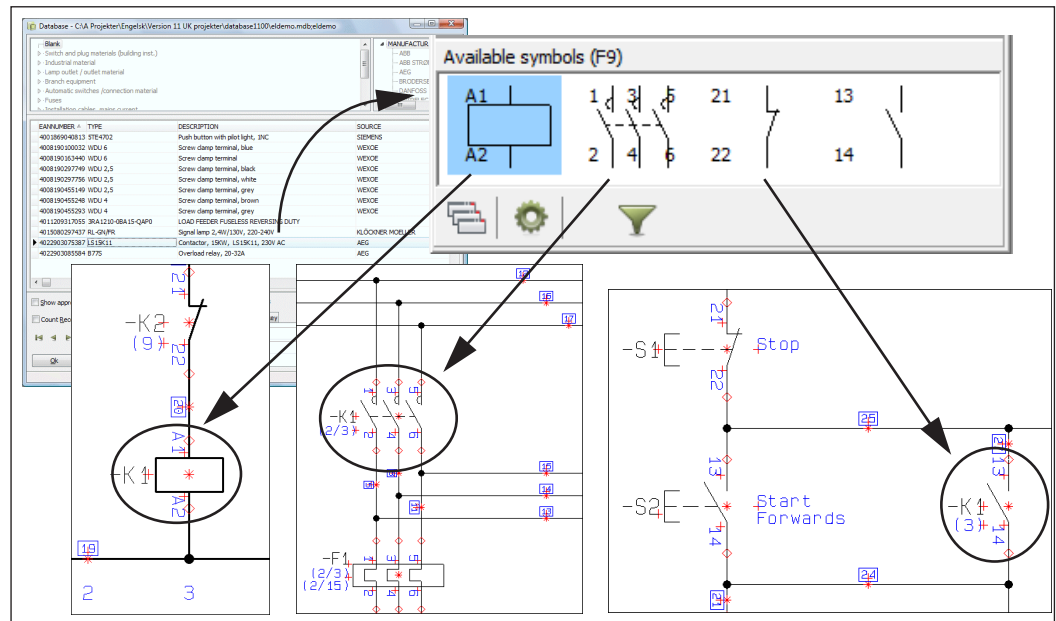
If you are working on a *mechanical* page, you get the *mechanical* symbol(s) for the component in stead, which you can then place on the mechanical page.

— See "Update from Database when changing symbol Article no" on page 279 also.

Function assemblies - with multiple article numbers

When you are working with advanced contactors, the symbols 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 188 also.

Component data, 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 641.

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 708.

FETCHING SYMBOLS FOR A COMPONENT FROM THE DATABASE

When you select a component via the *database*, the electrical symbols for the components are displayed under **Available symbols** at the bottom of the **Component menu** in the right-hand side of the screen. These symbols can then be placed in the project diagrams.
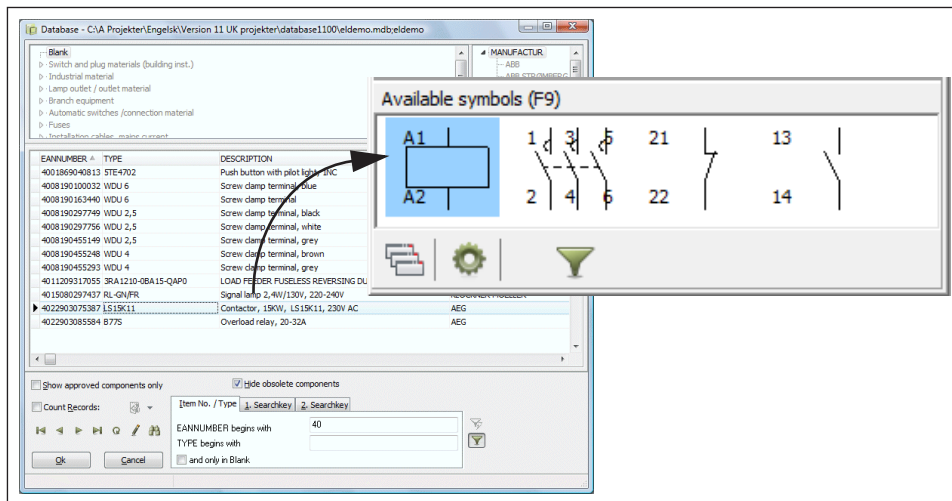
This is described in further detail in "Fetching symbols from the database" on page 170.

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 641.

THE COMPONENT MENU

The **Component menu** is placed in the right-hand side of the work area, and gives you an overview over the applied components in the project.

The **Component menu** only displays symbols from the symbol category (such as electrical, mechanical, pneumatic or hydraulic), which are selected for the active page in the project:

- When you work on a *Diagram page*, only components with *electrical* symbols are displayed.

- When you work on a *Mechanical page*, only components with *mechanical* symbols are displayed.

- If you work with another symbol category on the page, only symbols from this category are displayed. — See "The page database field for symbols" on page 266.

The **Component menu** contains two independent windows:

- The **Components** window, which contains a list of the applied components in the project (see "The Components window" on page 265) and

- The **Symbols for component** window, which contains a tree structure displaying all of the symbols and optional accessories for the selected component, as well as a pickmenu containing the available symbols for the component (see "The Symbols for component window" on page 269).

PC|SCHEMATIC
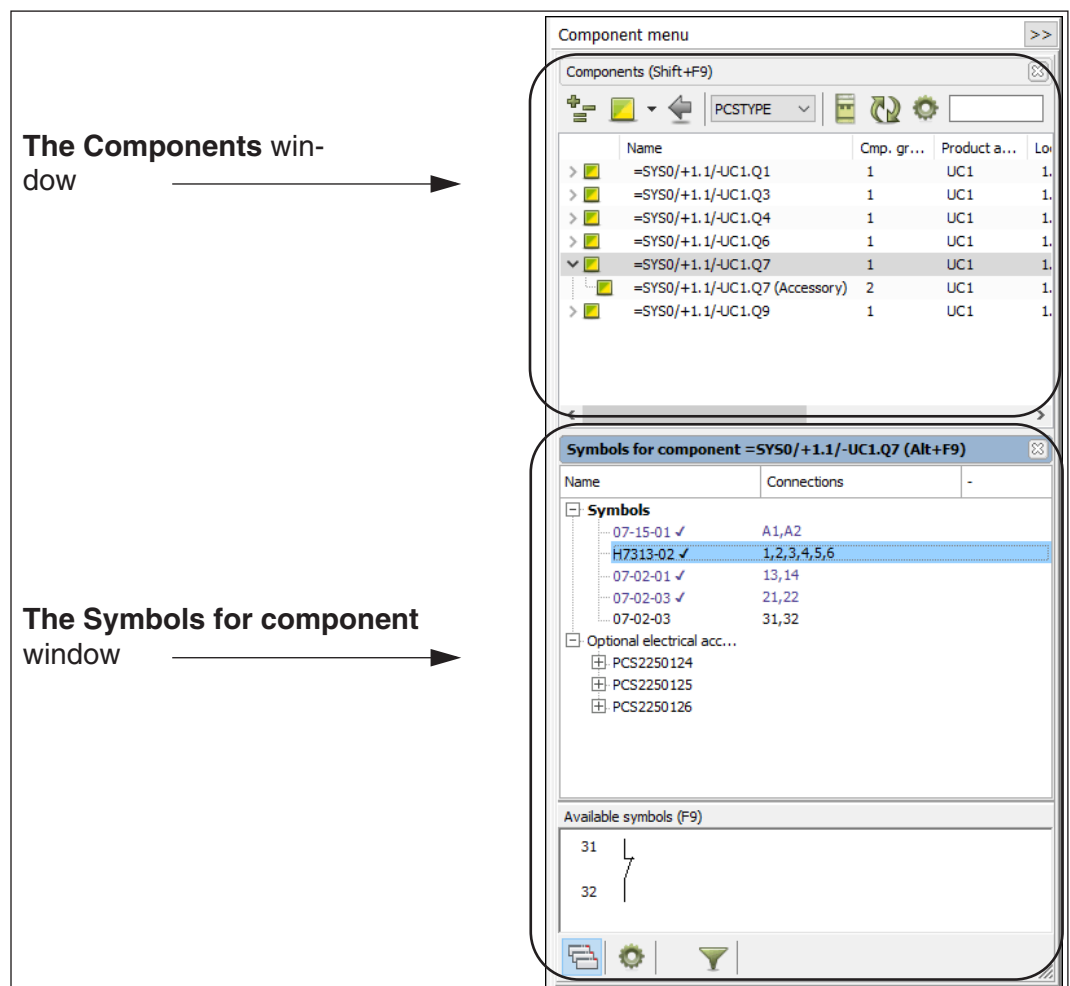
The **Component menu** can be hidden by clicking the **>>** button in the upper right-hand corner of the window.

In order to view the window again, just click the **<<** button on the **Component menu** tab at the far right of the screen.

Furthermore, there is the option of *deactivating* the **Component menu** in case you do not want the window to be refreshed while you work. — See "Pointer/Screen" on page 918.

Moving the Components and the Symbols for component windows

Both the **Components** and the **Symbols for component** windows can be moved. In order to move one of the windows, you just need to click on the title bar for the window and *drag* the window to the desired position — for instance on another screen, if you work with multiple screens.

Using the Component menu window for different kinds of symbols

You can for instance read more about how to apply the **Component menu** when working with different kinds of symbols here:

PCSCHEMATIC Automation

PC|SCHEMATIC

The Components window

The **Components** window is the upper window inside the **Component menu**. Here you can see a list of selected components in the project. When you click on a component in the **Components** window, the contents of the **Symbols for component** window is updated, so that it contains information about the selected component. — See "The Symbols for component window" on page 269.

At the top of the **Components** window you can specify the criteria for which components in the project, that are displayed. This can either be one specific criterion, or a combination of different types of criteria: Reference designations, component names and placing status.

Filtering functions
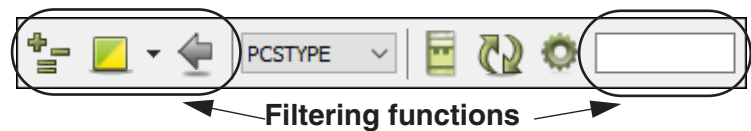
Filter functions in the Components window: Reference designations

When you click the **Reference designations** button, you enter the dialog box **Reference designations**, where you can specify which reference designations to show.

— Read more about reference designations in "Reference designations" on page 383.

Filter functions in the Components window: Texts for the component

When you click in the **Search in selected column** field, you can search among the texts for the components in the project. First you click on the column that you want to search in — the column *Name* in the figure below — and then click in the search field in the upper-right corner. Then type in the characters that you search for in the component name. The component names matching the search criterion are now displayed in the **Components** window:
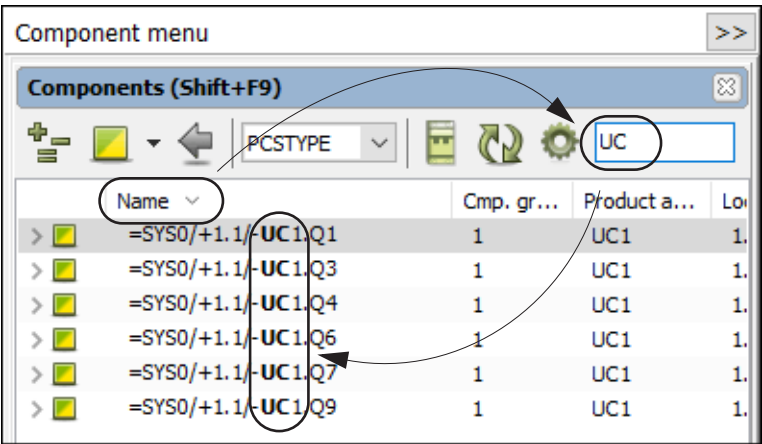
Filter functions in the Components window: Placing status

You can also specify which placing status of the components:

- When you click the **Show all** button, all components in the project are displayed

- When you click the **Completely-placed** button, only components with all symbols placed in the project are displayed.

- When you click the **Partly-placed** button, only components where not all symbols have been placed are displayed.

- When you click the **Not-placed** button, only components with no placed symbols are displayed. This option can for instance be handy if you have placed all electrical symbols for a component, and now work on a mechanical page and are about to place its mechanical symbol(s) — or the other way around.
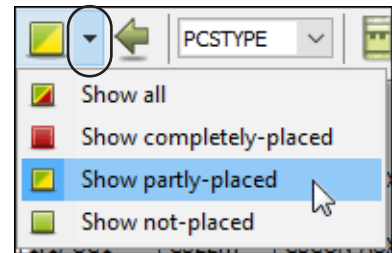
DIMMED RED SQUARE

If a dimmed red square is displayed next to a component in the list, this means that the symbols for the main component are placed in the project, but the component still has accessories, which are not placed yet.
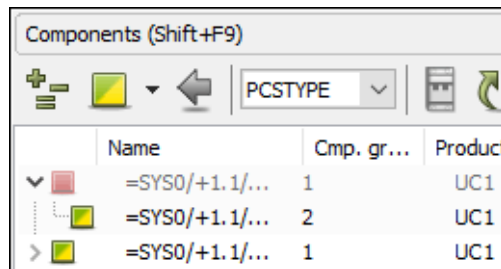
GREEN/YELLOW CORRESPONDS TO THE "SHOW AVAILABLE" WINDOW

IF YOU CHOOSE THE **SHOW PARTLY-PLACED** BUTTON, YOU WILL GET THE SAME RESULT AS YOU GOT IN THE "SHOW AVAILABLE" WINDOW, WHICH WAS USED IN **Automation** UNTIL VERSION 19.

Show Last selected component

When you click the **Last selected component** button, the last component selected in the **Components** window, is displayed again.

You may need to use this button, because the **Components** window is updated when you click on symbols in the project. When you click this button, the **Components** window once again shows the last component, that you clicked on inside the **Components** window.

The page database field for symbols

Normally, you work with electrical symbols on diagram pages and mechanical symbols on mechanical pages.

However, in the field **Page database field for symbols**, you can specify that you want to work with a different category of symbols on the page:
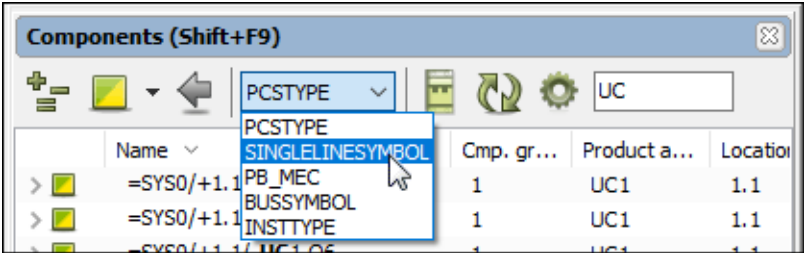
If you work with another category of symbols on the page, only symbols of the specified category are displayed in the **Component menu**.

— See how to change the settings in the program, which makes this possible, in "Other symbols" on page 687.

Select component and Select component name

When you click on a component in the **Components** window and then click the **Select component** button, the **Select component** window is opened for the selected component.

— See "Select Component" on page 278.

When you right-click on a component in the **Components** window and choose **Select component name**, all components with the same component name are selected in the **Components** window.

Update list (from database)

When you click the **Update list** button, the components in the **Components** window are updated with information from the database. It is only relevant to click this button if you know that changes have been made in the database.

Shown columns

When you click the **Shown columns** button, you can specify which information to display in the **Components** window.

Please note that you can also move the columns in the **Components** window, so that they appear in an order that suits you. To move a column just *click* on the column header and *drag* it to the desired position.
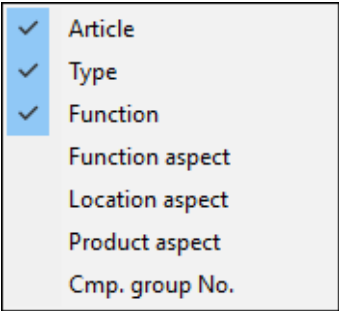
Displaying Accessories in the Components window
When accessories are chosen for a component, the accessories are displayed in a tree structure below the main component in the **Components** window:
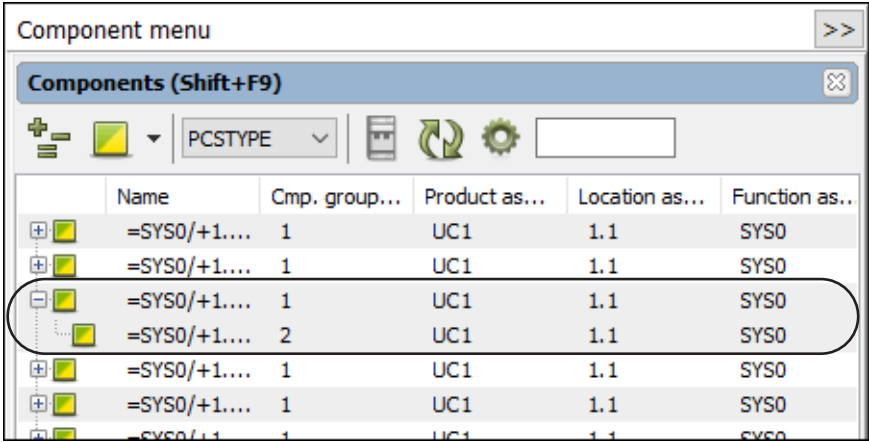
Fixed accessories for a component are not displayed until they are placed in the project.

The Symbols for component window

The **Symbols for component** window is the window at the bottom of the **Component menu**.

When you have selected a component in the **Components** window (see "The Components window" on page 265), the **Symbols for component** window displays a tree structure containing the symbols and optional accessories for the selected component, as well as a pickmenu containing the available symbols for the component.
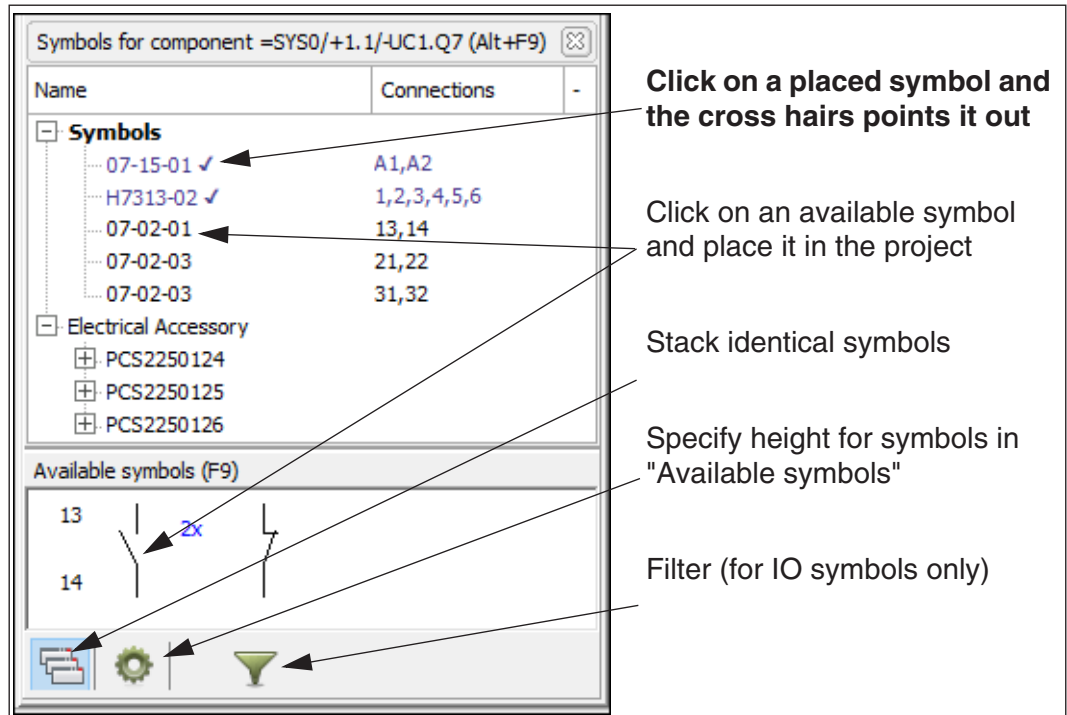
Symbols for the component, that are placed in the project, have a check mark after the symbol name in the tree view. A check mark is not set for accessories, because accessories can be placed multiple times. The selected accessory is displayed below the component in the **Components** window. When you double-click on a placed symbol in the **Symbols for component** window, the cross hairs points out the symbol on the project page.

When you double-click on an available symbol in the upper part of **Symbols for component**, or click on a symbol under **Available symbols**, you get the symbol in the cross-hairs, and can subsequently place it in the project.

Stack similar symbols

When a component has multiple available symbols of the same type, these can be displayed in two different ways under **Available symbols**:

- When the **Stack similar symbols** button is activated: A single symbol is displayed along with an indication of the number of available symbols of this type (i.e. *2x*)

- When the **Stack similar symbols** button is *not* activated: The symbols are displayed individually

Filtering and other special options for PLCs

For components with many attached symbols, there can be tabs at the bottom of **Available symbols**, which makes it easier to find the symbols that you need.

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

The **Stack similar symbols** button is also handy when handling many similar I/O symbols.

Tabs for PLCs   Filtering
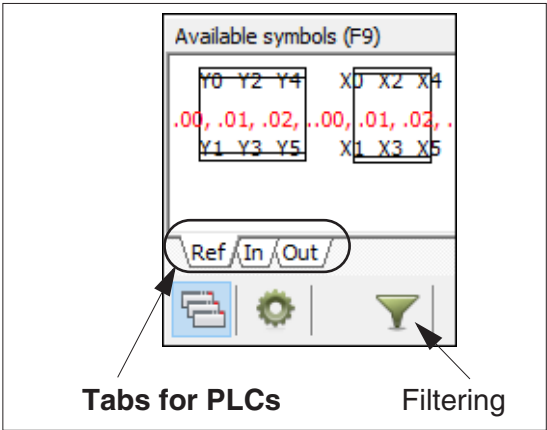
FILTERING FOR SHOWING PLC SYMBOLS UNDER AVAILABLE SYMBOLS

There is also a **Filter** button, which is used for filtering which symbols to show under **Available symbols**. The filter is activated/deactivated by clicking the **Filter** button. The filter settings is specified by clicking the **Filter** button, which gives you the option of specifying which address interval to show.

You can see that the filter is active when the background color on the **Filter** button is green.

— Learn more about working with PLCs in "Working with PLCs" on page 478.

DISPLAYING COLUMNS FOR PLC SYMBOLS

When working with PLC symbols, you can specify which columns to display for the symbols in the **Symbols for component** window:

1) Click the **Settings** button in the lower part of the window

2) Select which of the columns to display

Please note that you only have the option of changing these settings when working with PLC symbols.
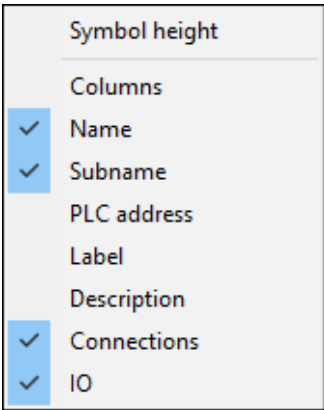
Optional electrical accessories in Symbols for component
When a component has optional electrical accessories, the optional electrical accessories are displayed in the **Symbols for component** window:

Click here ...

... and the symbols for the accessory is displayed here
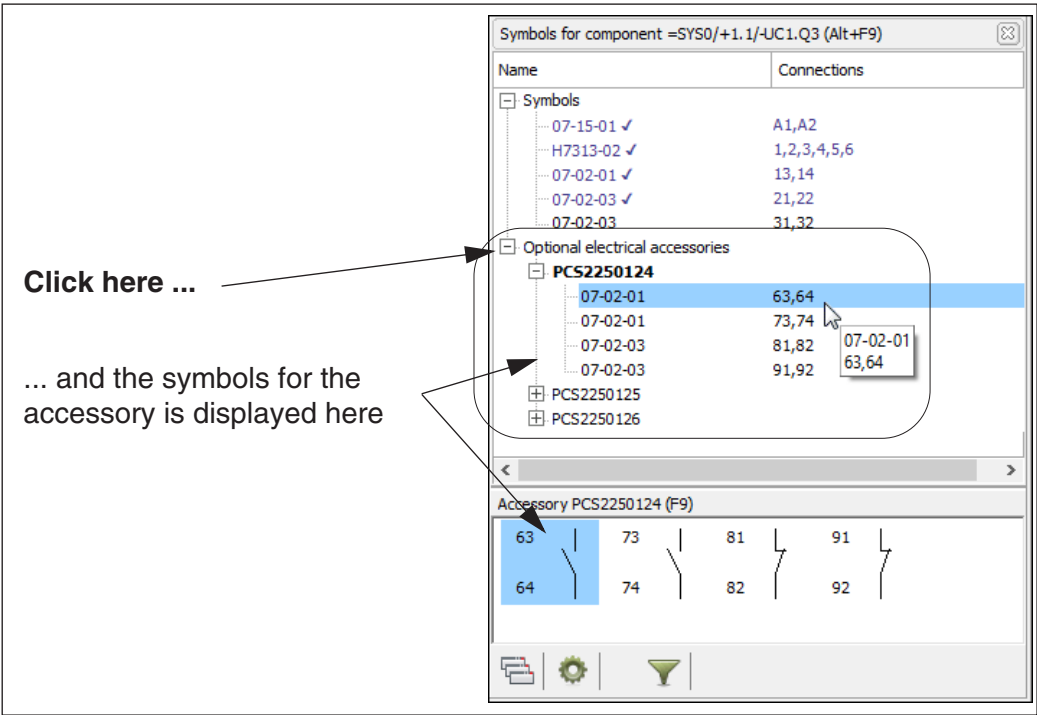
When you click on the **+** in front of one of the optional accessory components in the upper part of the **Symbols for component** window (*PCS2250124* in the figure above), the symbols for the accessory component are displayed in list form in the upper part of the window while the symbols for the component are displayed in the lower part of the window.

From there they can be placed in the project, like any other symbol. Contrary to the symbols for "the main component", a check mark is not set for the accessory symbols, because these can be placed more than once in the project. The selected accessories are displayed below the component in the **Components** window.

When one of the symbols for an accessory is placed in the project, the accessory will also appear in the **Components** window in the upper part of the **Component menu**, where the accessory is indented below the main component:
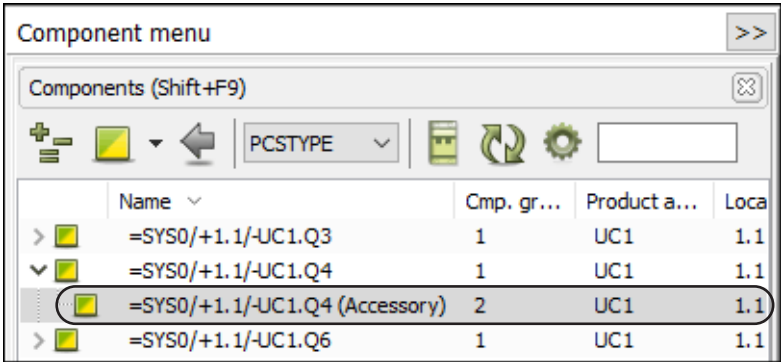
The selected electrical accessories are also displayed in the **Component data** dialog box on the **Accessory** tab — see "The Accessory tab" on page 184.

Placing an available symbol for a component

When you click on a component in the list in the **Components** window, you can see the available symbols for the component in the lower part of the **Symbols for component** window under **Available symbols**.

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.

Placing all available symbols for a component

You can also right-click on one of the symbols for the component under **Available symbols** in the lower part of the **Component menu**, and choose **Place 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.

Place 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
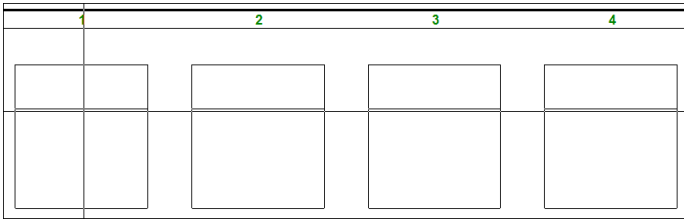
Place all - on mechanical pages

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

When for instance you use **Place 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.
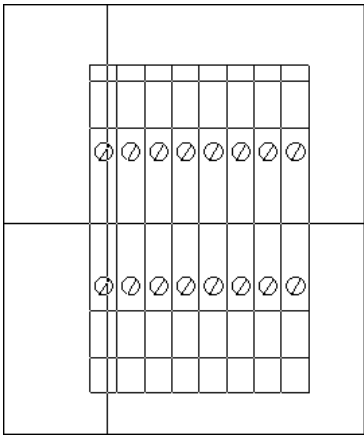
SHORT-CUTS IN THE COMPONENT MENU

You can use the following short-cut keys in the **Component menu**:

- Short-cut <Shift + F9>: **Functions => Go To: Components** — here you return to the **Components** window for the last component that you selected in the **Component menu**.

- Short-cut <Alt + F9>: **Functions => Go To: Symbols for component** — here you return to **Symbols for component** window for the last component that you selected in the **Component menu**.

- Short-cut <F9>: **Functions => Go to: Available symbols** — see "Go to: Available symbols <F9>" on page 273.

- Short-cut <Ctrl + F9>: **Functions => Go To: Available symbols again** — here you return to **Available symbols** for the last component that you selected in the **Component menu**.

Go to: Available symbols <F9>

When you have selected a symbol in a diagram and press <F9> or choose **Functions => Go to: Available symbols**, you enter the **Available symbols** window at the bottom of the **Component menu**, where the first available symbol for the component is highlighted:
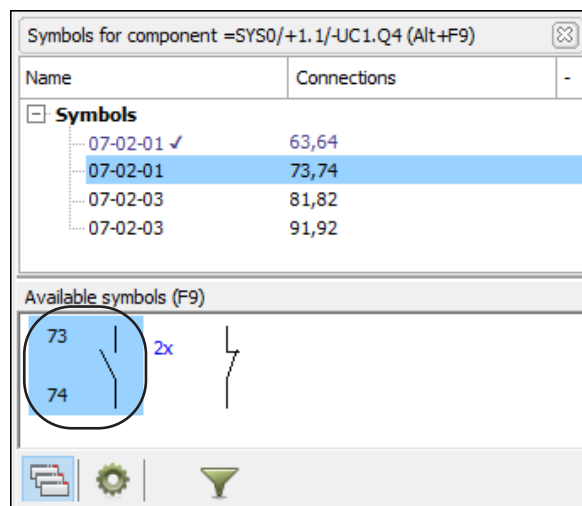
FINDING SYMBOLS BASED ON THE COMPONENT NAME

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

— See "Select Component" on page 278 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:
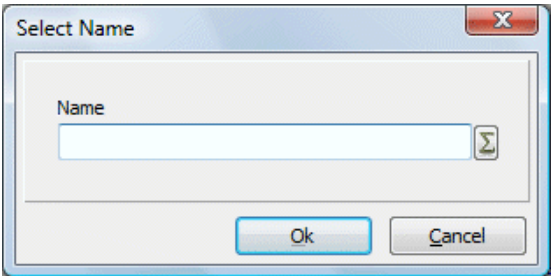
When you wish just to find the symbols for a specific component, type in the component name, and press <Enter>.

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 354.

The contents of the window can be sorted as described in "The sorting order of the data in the Object Lister" on page 354.

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 618.

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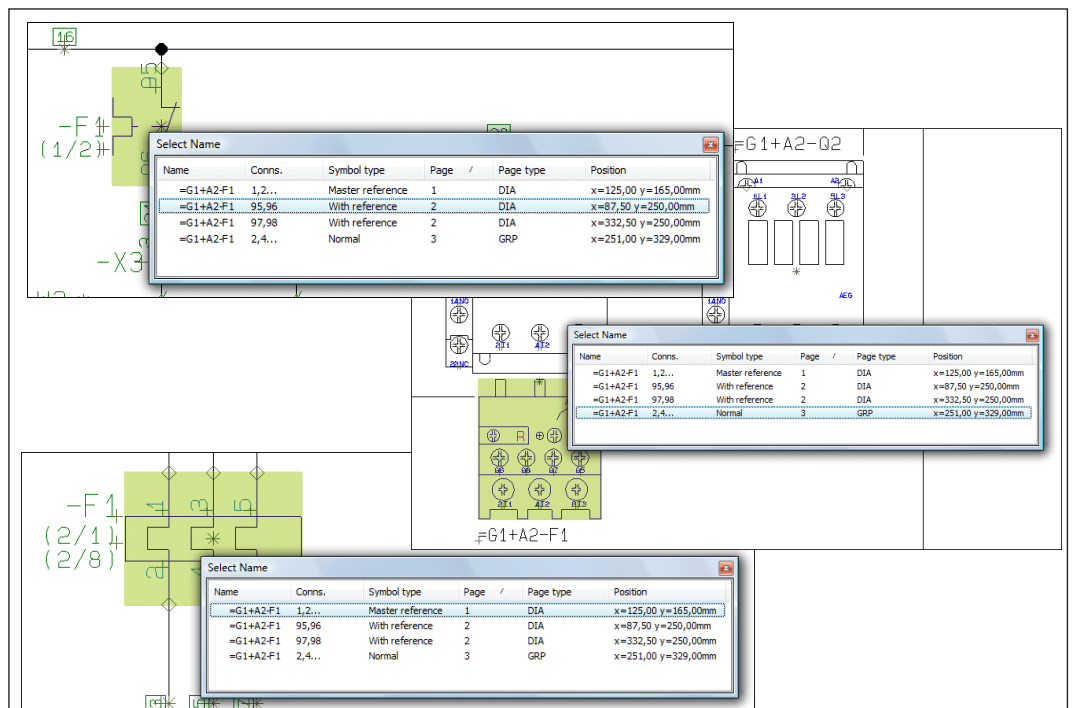
You can also activate the function by right-clicking on a symbol, and then choose **Select Name**.

PC | SCHEMATIC

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 275) and click **OK**.

Name	Conns.	Symbol type	Page	Page type	Position
=G1+A1-X1	1	Terminal	2	DIA	x=87,50 y=160,00mm
=G1+A1-S2	13,14	Normal	2	DIA	x=87,50 y=177,50mm
=G1+A1-S1	21,22	Normal	2	DIA	x=87,50 y=202,50mm
=G1+A1-X1	3	Terminal	2	DIA	x=87,50 y=220,00mm
=G1+A1-X1	2	Terminal	2	DIA	x=105,00 y=220,00mm
=G1+A1-X1	5	Terminal	2	DIA	x=157,50 y=82,50mm
=G1+A1-P1	X1,X2	Normal	2	DIA	x=157,50 y=100,00mm
=G1+A1-X1	4	Terminal	2	DIA	x=157,50 y=125,00mm
=G1+A1-X1	7	Terminal	2	DIA	x=227,50 y=182,50mm

3) In the window **Select Name**: Remove the symbols from the list, which you do *not* wish to change the reference designations, and click on the **Component data** button in the toolbar for symbols.

4) In the dialog box **Component data**: Click on the tab **Ref. design.** and click on **Select**.

5) In the dialog box **Reference designations**: Select the new reference designation — for instance ***+A2*** —, and click **OK**.

6) In the dialog box **Component data**: Click **OK**.

7) In the window **Select name** the new reference designations are now displayed. Click the **Close** button in the upper right-hand corner to close the window.

Renumbering pinnumbers for a terminal row using Select name

Do you wish to renumber pinnumbers for a terminal row, so that the pinnumbers increase from the start to the end of the project, do the following:

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

Σ

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:

Select Name					
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.

Position /
x=20,00 y=122,50mm
x=20,00 y=127,50mm
x=20,00 y=132,50mm

Click here

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.

Page /
2
2
2

Click here

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).

Select Name					
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 component 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 274 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 component 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 618.

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 178.

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 like this:

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 Component 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 196.

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 196.

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

INSERTING SUB DRAWINGS

PLEASE NOTE THAT YOU CAN ALSO INSERT SUB DRAWINGS — "SUB DRAW-INGS" ON PAGE 299.

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**.

This can for instance be handy when working with arrangement drawings — see "General arrangement drawings" on page 705.

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 280.

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
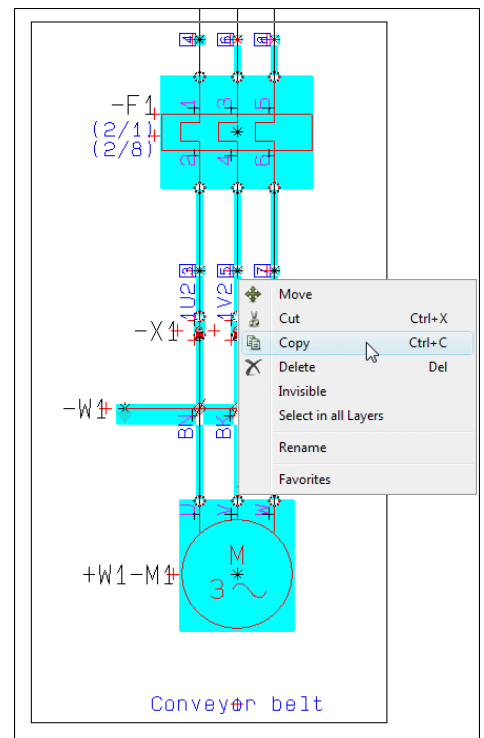
When you place the copied area, you enter a dialog box, where you specify the directions for renaming the names in the copied area:

NAME PROTECTED COMPONENTS ARE NOT RENAMED
READ MORE ABOUT THIS IN "NAME PROTECTED" ON PAGE 180.

The options in this dialog box depends on which types of symbols exists in the copied area.

FULFILLING STANDARDS WHEN COPYING AREAS

ESPECIALLY IN CONNECTION WITH NAMING OF TERMINALS, DETAILED RULES APPLIES FOR NAMING/NUMBERING. THUS MOTOR TERMINALS, SUPPLY TERMINALS AND INPUT- AND OUT-PUT TERMINALS ARE NUMBERED ACCORDING TO DIFFERENT PRINCIPLES.

WHEN YOU COPY AN AREA THE PROGRAM FIRST EXAMINES WHETHER THE AREA IS DRAWN ACCORDING TO THE RULES IN *EN60204-1,14.2.1 CONDUCTOR AND CABLE RUNS*. IF THIS IS THE CASE, THE PROGRAM THEN ANALYSE WHICH TERMINAL TYPES IT CONTAINS, AND RENAMES THESE ACCORDING TO THE DIFFERENT PRINCIPLES.

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.

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. — IF YOU CHOOSE **DO NOT RENAME TERMINALS**, AND ALL TERMINALS IN THE COPIED AREA CARRY THE SAME NAME, THE TEXT IS CHANGED TO **KEEP OR EDIT -X4** (WHEN THE TERMINAL NAME IS *-X4*). HERE YOU GET THE OPTION OF RENAMING THE TERMINALS IN THE AREA BY TYPING IN A NEW TERMINAL NAME.
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 282.

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 455.

Include mechanical accessories

If you select **Include mechanical accessories (Only when renaming)**, any mechanical accessories for the symbols will also be copied. As stated it requires that this is done when renaming the symbols. Otherwise it would not be possible to keep track of where the accessories belong.

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 286.

ASSIGNING REFERENCE DESIGNATIONS WHEN COPYING AREAS

If reference designations are used in the area you copy, the button **Ref.des.** is displayed in the dialog box **Rename symbols**. This gives you the option to assign other — or new — reference designations to the copied area.

When you click the **Ref.des.** button, you enter this dialog box:

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

REF. DESIGNATIONS FOR SUB DRAWINGS, PAGES AND FULL PROJECTS

THIS CHAPTER DESCRIBES HOW TO EDIT REFERENCE DESIGNATIONS WHEN COPYING AREAS. YOU HAVE THE EXACT SAME OPTIONS WHEN COPYING FULL PAGES AND WHEN INSERTING STANDARD PROJECTS.

See symbols using the listed reference designations

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:
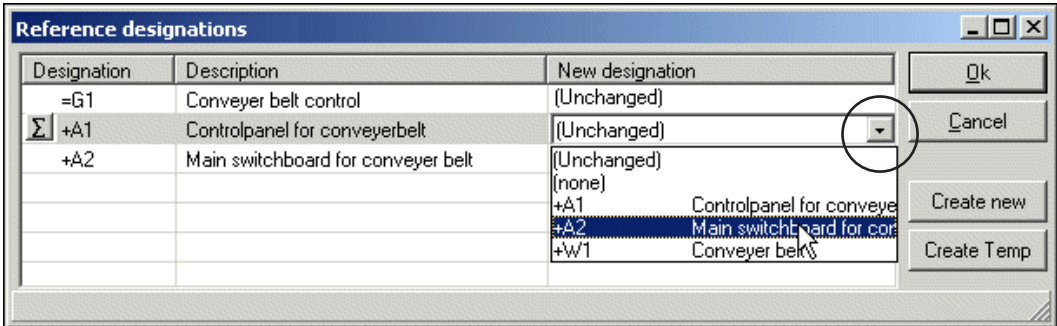
By default **(Unchanged)** is selected, but you get the option of choosing by all applied reference designations in the project. You can also choose **(none)**, to delete this reference designation from the symbols in the copied area.

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 project reference designations" on page 393**, in particular "Adding new reference designations" on page 395.

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:
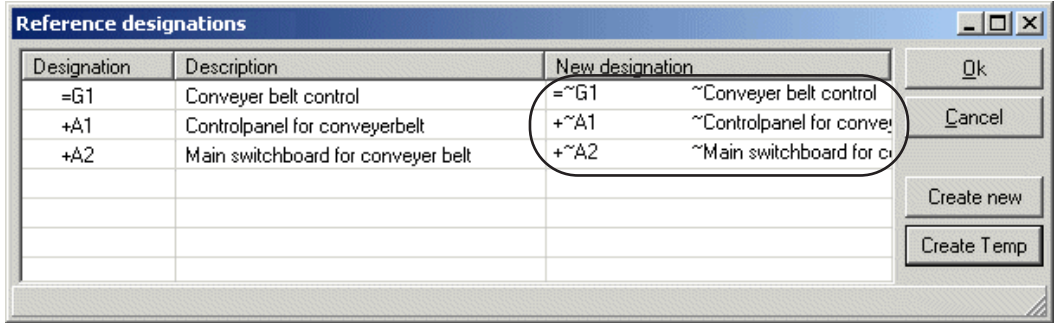
These temporary reference designations are assigned a ~ sign after the reference designation prefix. This temporary reference designation is created from the initial reference designation, so that *=G1* becomes *=~G1*. If this temporary value is already used in the project, the temporary reference designation 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 286.

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 584 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.
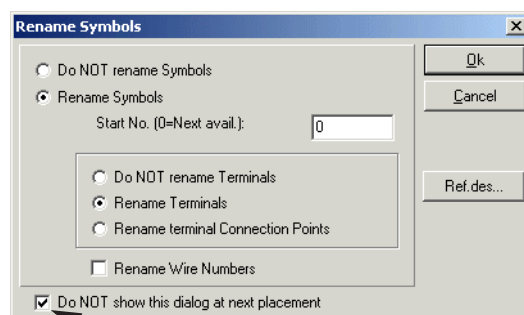
Click here NOT to show the dialog box again

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 626.

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 584.

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**.

| Select the area and choose Cut (or Copy) | The area is deleted and the connected lines are parked |
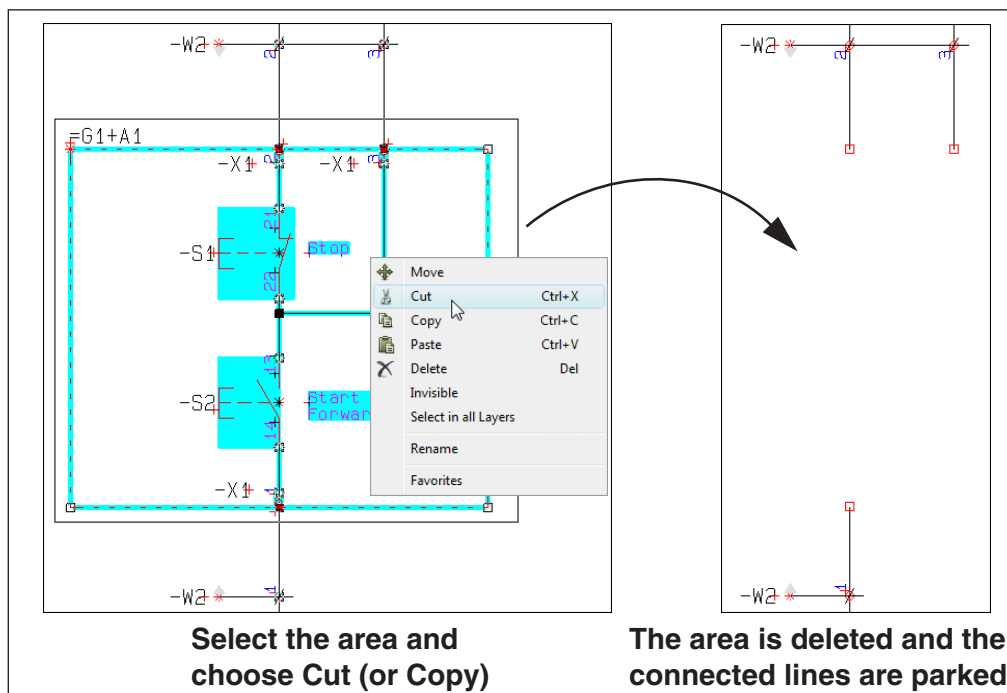
3) The area is deleted: Shift to another page — if necessary — and click the **Paste** button. You now get the area in the cross hairs.

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 281. 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.

Parked connection point

=G1+A1

Connected to existing line

Point to position to place the Area

Click to place the area, and rename the symbols
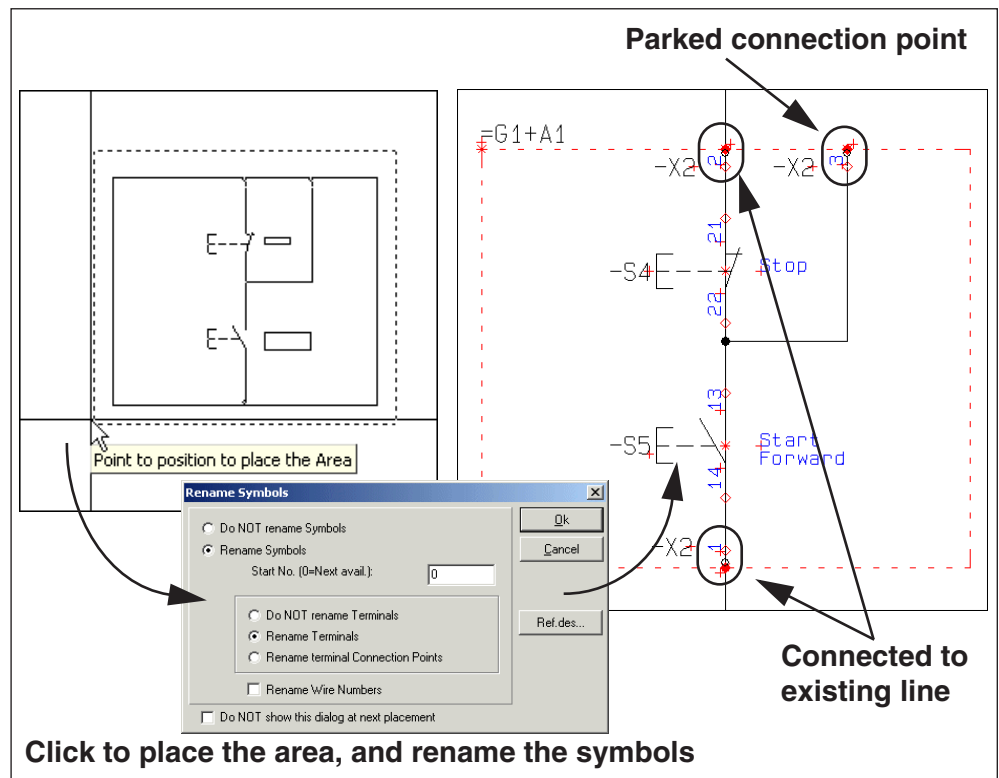
Lines are cut along the edges of areas when copying

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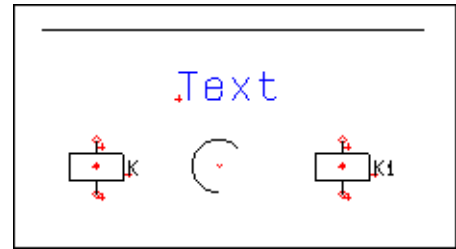
Click inside the area, keep the mouse button pressed down and move the mouse. Press <spacebar>. Hereby the area is rotated around the point the mouse is pointing at.

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.
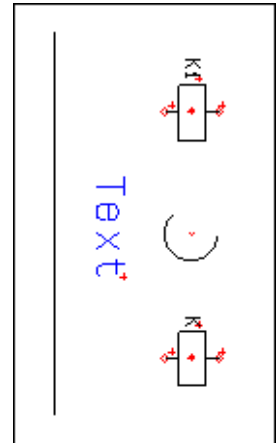
SCALING AREAS

You can use scaling when you make drawings as well as when you create/edit symbols (see "Creating Symbols" on page 775).

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**
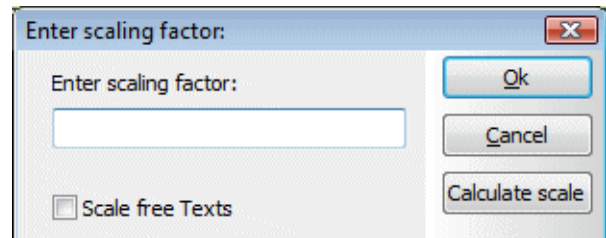
The area have now been scaled. If you are editing a symbol, this can require adaptation of texts, and moving of the reference point, because they are now placed in another distance to the scaled objects.

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 830.

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:
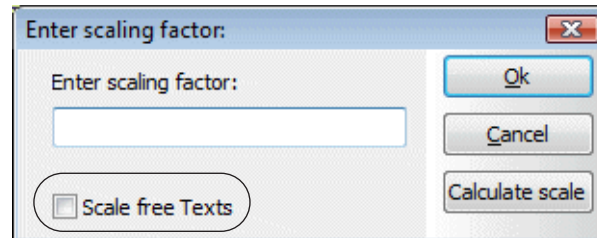
Scaling everything on a page

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 547.

COPYING AREAS INTO EDIT SYMBOL MODE

To copy an area from a project drawing into a symbol, do the following:
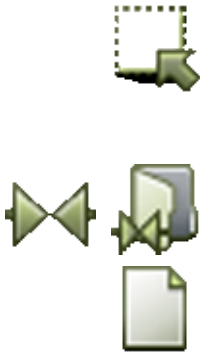
1) Click the **Area** button, and select the area you wish to copy.

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 e.t.c. as described in "Creating Symbols" on page 775.

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
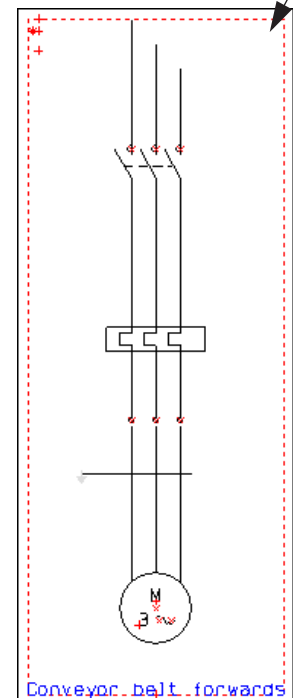
OBJECT GROUPS

In **Automation** you can group drawing objects (symbols, lines, circles and texts) and images, which are placed on the same project page, in so-called *object groups*. It then becomes easier to work with them subsequently, and you can copy, move, delete or replace a group of objects in a single operation.

Please note that *object groups are created on page level.* — Read more about this in "Object groups and sub drawings" on page 297.

CREATING OBJECT GROUPS

To create an object group on a project page, do this:

1) Select the project page where you wish to create an object group.

2) Activate the **Area** button and use the mouse to select an area on the page.

3) Right-click inside the selected area and choose **Object groups => Edit object group**.

4) In the dialog box **Object groups**: Click **Create**.

5) In the dialog box **Create Object group**: Type in a name for the object group and click **OK**.

You have now created an object group. When you then right-click on the page and choose **Object groups => Show object groups**, you can see that it appears in the list of object groups on the page.

— If you wish to remove/deselect objects from the created object group, please see "Removing objects from an object group" on page 296.

— If you wish to add objects to the created object group, please see "Adding an area to a created object group" on page 296.

OBJECT GROUP NAME IN LISTS

WHEN A COMPONENT IS PART OF AN OBJECT GROUP, YOU CAN HAVE THE NAME OF THE OBJECT GROUP DISPLAYED IN PARTS AND COMPONENTS LISTS BY INSERTING THE DATAFIELD *OBJECT GROUP NAME*. — SEE "PARTS/COMPONENTS LIST DATAFIELDS" ON PAGE 949.

EDITING OBJECT GROUPS

To edit an object group, do this:

1) Select the project page where the object group is created.

2) Right-click on the project page and choose **Object groups => Show object groups**.

You now enter the dialog box **Object groups**:
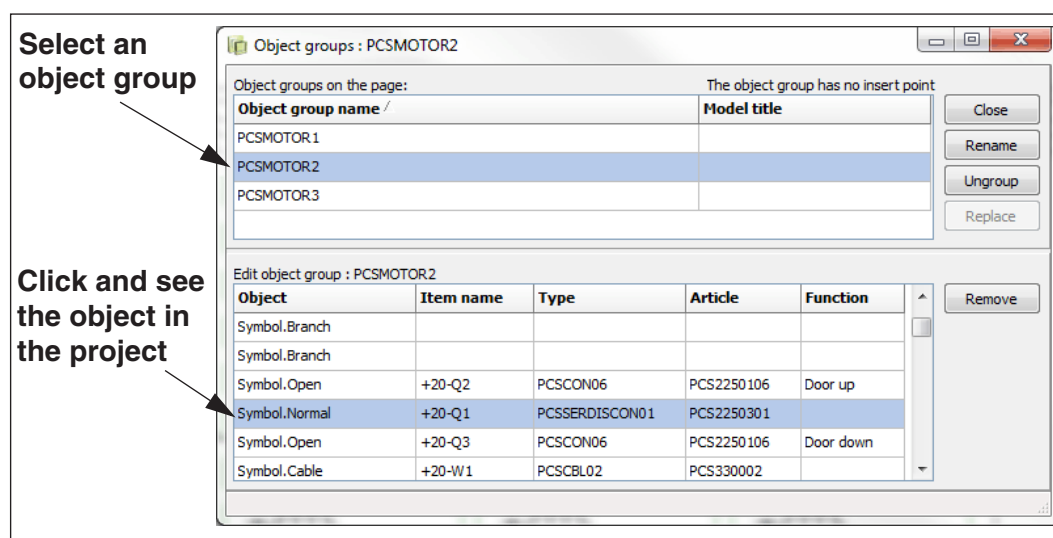
When you select an object group here, it is also selected in the project and is still selected in the project when you close the dialog box. This is also the case if you select a number of objects in an object group.

You can subsequently continue working with the selected objects in the project.

In the dialog box **Object groups** you also have the following options:

Renaming an object group

In the dialog box **Object groups** you rename an object group like this:

1) Click on the name of the object group in the upper part of the dialog box.

2) Click **Rename** in the right-hand side of the dialog box.

3) In the dialog box **Rename object group**: Type in the new name and click **OK**.

Ungrouping an object group

In the dialog box **Object groups** you ungroup an object group like this:

1) Click on the name of the object group in the upper part of the dialog box.

2) Click **Ungroup** in the right-hand side of the dialog box, and answer **Yes** to ungroup the object group.

The object group is now ungrouped. The placed symbols, lines and more, which were part of the object group on the project page, are still placed on the project page. They are just no longer part of an object group.

Replacing an object group

When an object group contains insertion points — for instance because it was originally placed as a sub drawing — it is possible to replace the object group.

— Get additional information in "Object groups and sub drawings" on page 297 and "The function of the insertion points" on page 322.

This is how you replace an object group containing insertion points:

1) In the dialog box **Object groups**: Click on the name of the object group in the upper part of the dialog box.

2) Click on **Replace** (dimmed if the selected object group has no insertion points) in the right-hand side of the dialog box, and the dialog box **Replace object group** is displayed:
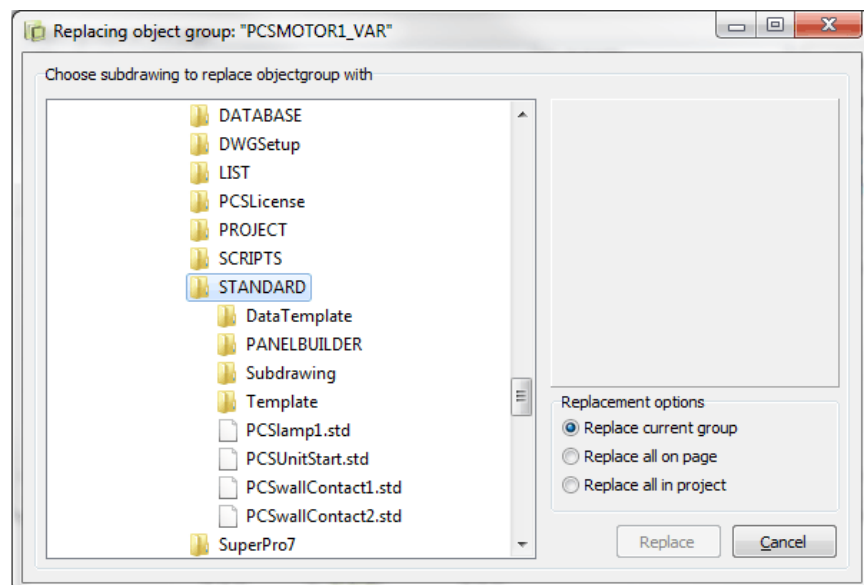
3) In the dialog box **Replace object group**: Choose the *sub drawing* that you wish to replace the object group with.

4) Select one of the following three options:
   Replace current group: Only the selected object group is replaced.

Replace all on page: All object groups of the same name on the project page is replaced.
Replace all in project: All object groups with the same name *in the project* are replaced.

5) Click **Replace**.

6) In the dialog box **Rename symbols**: Specify how to rename symbols and terminals in the selected sub drawing, choose model if possible and click **OK**. — Read more about the options in this dialog box in "Intelligent copying of areas" on page 281.

The object group is now replaced with the selected sub drawing.

Removing objects from an object group

In the dialog box **Object groups** you remove objects from an object group like this:

1) Click on the name of the object group in the upper part of the dialog box. The objects in the selected object group are now displayed in the lower part of the dialog box.

2) In the lower part of the dialog box: Click on the object that you wish to remove from the object group, and click **Remove** in the right-hand side of the dialog box. (Multiple objects can be selected at the same time using <Ctrl> or <Shift> following normal Windows guidelines).

3) Answer **Yes** to confirm removing objects from the group.

The objects are now no longer part of the object group. They are still placed on the project page, they are just no longer part of an object group.

ADDING AN AREA TO A CREATED OBJECT GROUP

This is how you add an area to a created object group:

1) Select the project page where the object group is placed.

2) Activate the **Area** button and use the mouse to select an area on the page.

3) Right-click inside the selected area, choose **Object groups => Add to object group** and then select the name of the object group, that you wish to add the area to.

The area is now added to the object group.

You can also right-click on an object group — e.g. a symbol — and choose **Object groups => Add to object group** and then select the name of the object group, that you wish to add the area to.

INSERTING AN OBJECT GROUP ON ANOTHER PAGE IN THE PROJECT

This is how you copy an object group placed on a project page, and insert it on another page in the project:

1) Select the project page where the object group is placed.

2) Activate **Symbols**, right-click on a symbol in the object group and choose **Object group => Select object group**.

3) Click the **Copy** button — or right-click on the selected object group and choose **Copy**. You now have the object group in the cross hairs.

4) Find the relevant project page and click to place the object group on the page.

5) Fill out the dialog box **Rename symbol** and click **OK**.

6) Press <Esc> in order not to place the object group again.

The object group is now placed.

DOES NOT BECOME AN OBJECT GROUP
PLEASE NOTE THAT THE COPIED OBJECT GROUP DOES *NOT* BECOME AN OBJECT GROUP AUTOMATICALLY ON THE NEW PAGE.

OBJECT GROUPS AND RIGHT-CLICKING

When you right-click on a drawing object in an object group — for instance a symbol — and choose **Object group**, you get the following options:

- **Remove from object group**: The object is removed from the object group.
- **Select symbols in the object group**: All symbols in the object group are selected.
- **Select object group**: The object group is selected.
- **Edit object group**: You enter the dialog box **Object groups**, where the actual object group is selected.

OBJECT GROUPS AND SUB DRAWINGS

Please note that object groups are created at *page level*. In the dialog box **Object groups** you therefore only see the object groups, which are created on the page that you are presently working on.

Sub drawings are created at *program level*, and are accessible in any project you are working on.

Object groups across project pages

Even though object groups are only specified for individual project pages, you do have some options to use them across project pages:

- Copy object groups from one project page to another
  —see "Inserting an object group on another page in the project" on page 296.
- Replace all object groups with the same name in the project with a *sub drawing* — *see* "Replacing an object group" on page 295.

Sub drawings are converted to object groups

When a sub drawing is inserted in a project, it is automatically created as an object group. This means that the inserted sub drawing can then use all of the options, that are available for working with object groups.

The created object group automatically gets the name of the sub drawing.

— Read more about sub drawings in "Sub drawings" on page 299.

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 **Projekt generator** manualen.

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 345.

— Read more about the **Explorer window** in "The Explorer window: The Projects tab" on page 345.
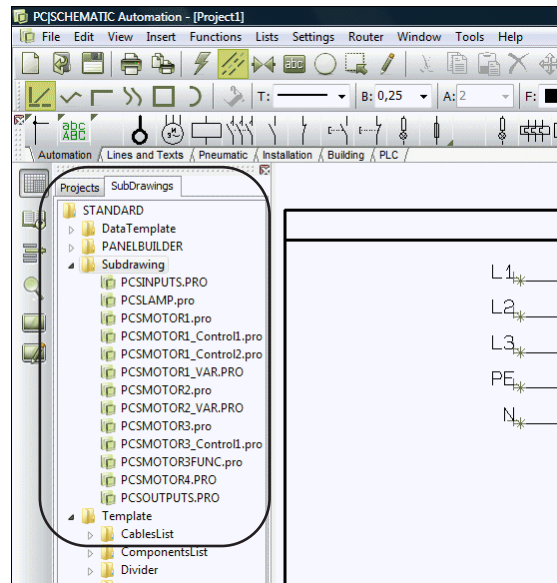
Specifying what to display on the SubDrawings tab

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

You can then choose between:

Show from here: The **SubDrawings** tab displays the contents of the selected folder.

One level up: The **SubDrawings** tab displays the contents of the folder one level above the selected folder.

Show all: The **SubDrawings** tab displays everything you have access to from your PC.
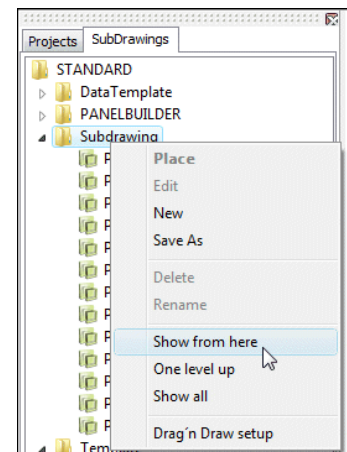
Other options in the right-click menu

Beyond this, you can choose between:

Place: Hereby you get the selected sub drawing in the cross hairs, and can place it in the active project, as described in "Manual placing of sub drawings" on page 302.

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 311.

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 311.
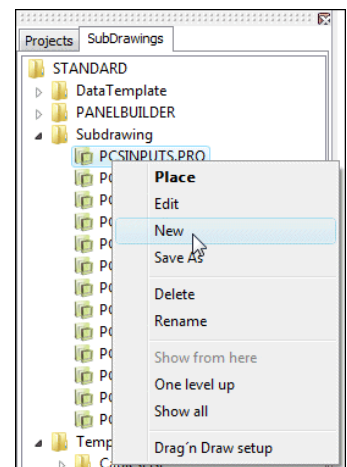
(Click on Folder)

(Click on Sub drawing)

Delete: Hereby the selected sub drawing is deleted. It is not possible to delete folders from the **SubDrawings** tab.

Rename: Hereby you can rename the selected sub drawing.

Drag'n Draw setup: — See "Drag'n Draw setup" on page 308.

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 302.

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 307.

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 314.

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 **Projekt generator** manualen.

> **SUB DRAWINGS ARE CONVERTED TO OBJECT GROUPS**
> WHEN YOU INSERT A SUB DRAWING IN A PROJECT, IT IS AUTOMATICALLY CONVERTED TO AN *OBJECT GROUP*. — READ MORE ABOUT THIS IN "OBJECT GROUPS AND SUB DRAWINGS" ON PAGE 297, PARTICULARLY "SUB DRAWINGS ARE CONVERTED TO OBJECT GROUPS" ON PAGE 298.

MANUAL PLACING OF SUB DRAWINGS

To insert a sub drawing *manually* — see "On inserting/placing sub drawings" on page 301 — 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.

**1) Click here   2) Click on the sub drawing, and   3) Click where you
                      drag it into the project page   want to place
                                                              the sub drawing**

4-6) The symbols in the sub drawing are renumbered automatically
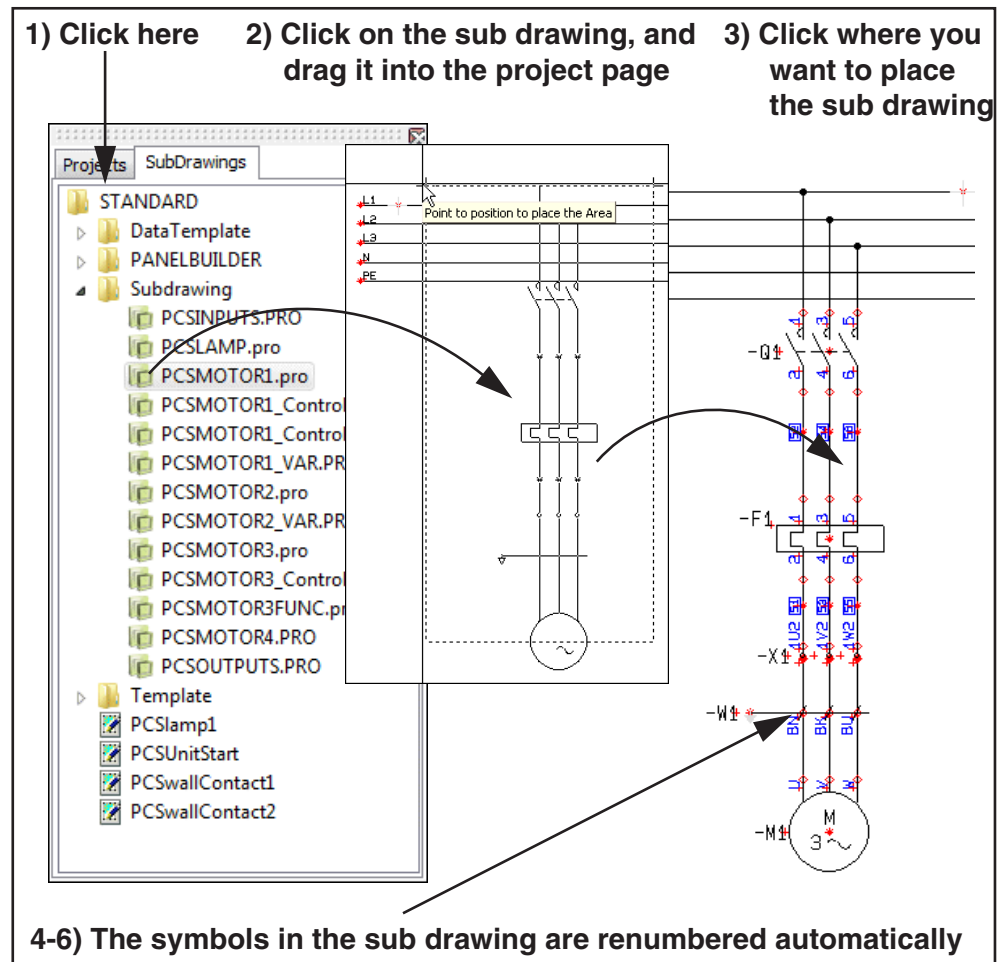
If the sub drawing contains Model data

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 303.

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

PCSCHEMATIC Automation

setup" on page 308)
— 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 315 and see how to apply sub drawings for automatic project generation in the **Projekt generator** manualen.

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.
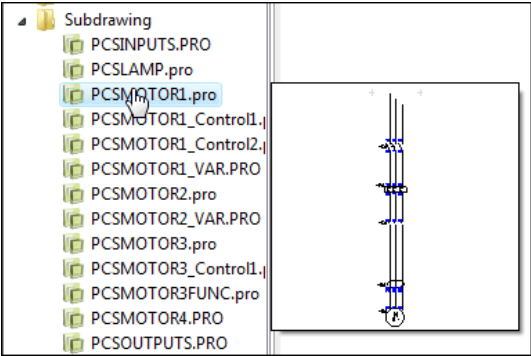
Inserting sub drawings: With model data

When you insert a sub drawing — see "Manual placing of sub drawings" on page 302 —, 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:
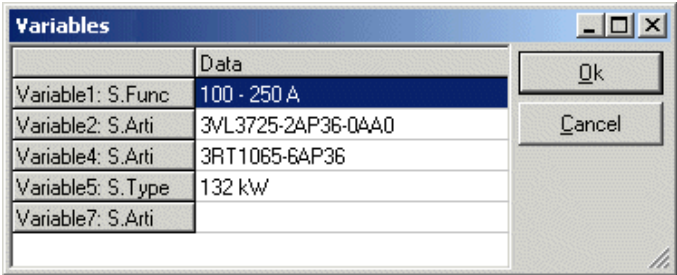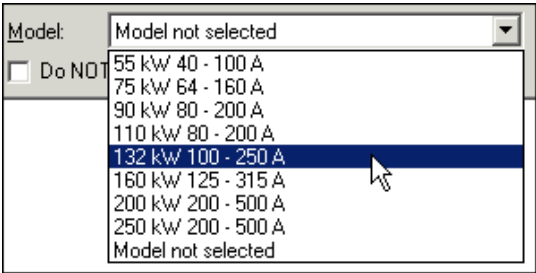
Here you can edit data — if necessary — by double clicking on a field and then editing the text. When you have placed the sub drawing in the project, you can edit the texts in via this dialog box, as described in "Editing variables for inserted sub drawings" on page 304.

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 308, "Drag'n Draw settings: For model data" on page 308 in particular.

— See also how to create models for a sub drawing in "Creating Model data for sub drawings" on page 327.

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 324, PARTICULARLY "INSERTING VARIABLE MARKERS ON A SUB DRAWING (OR ON A DRAWING TEMPLATE)" ON PAGE 324.

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**.
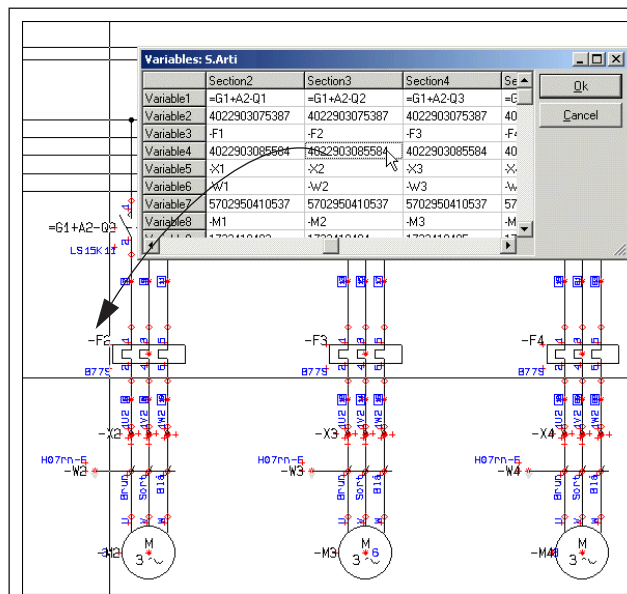
Click on the variable field, and the cross hairs points out its position on the page

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.

PCSCHEMATIC Automation

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**:
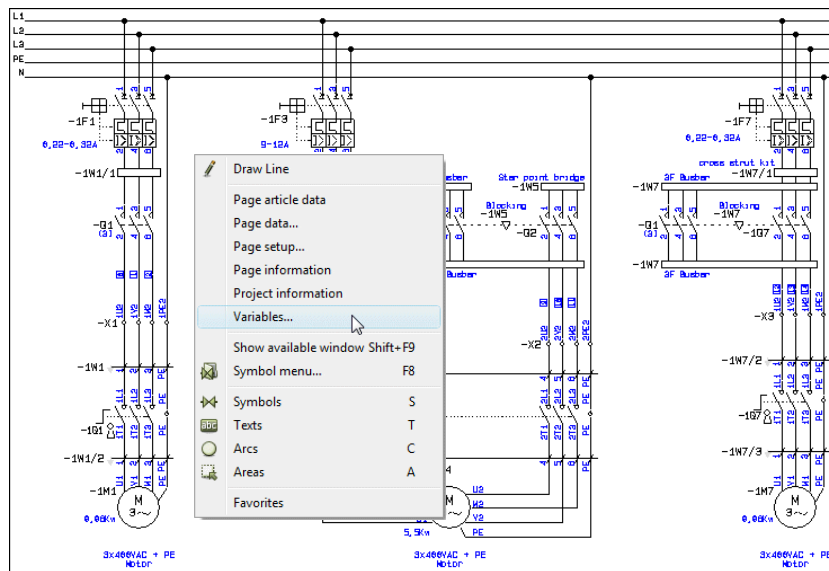
2) In the dialog box **Variables**: Right-click on the column header for the section (sub drawing), for which you wish to change model.

3) You now see all of the models for the sub drawing. Click on the desired model:
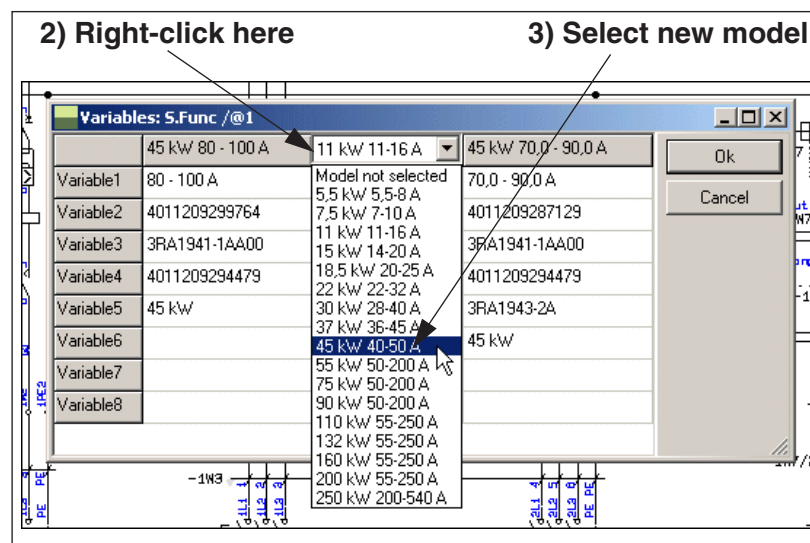
The model data for the sub drawing are now changed.

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 314.

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:

1) Click here

2) Click on the sub drawing, drag it over the project page, and release the mouse button

The sub drawing is placed automatically, and connected to the existing e.g. electrical potentials. The symbols in the sub drawing are renamed automatically.
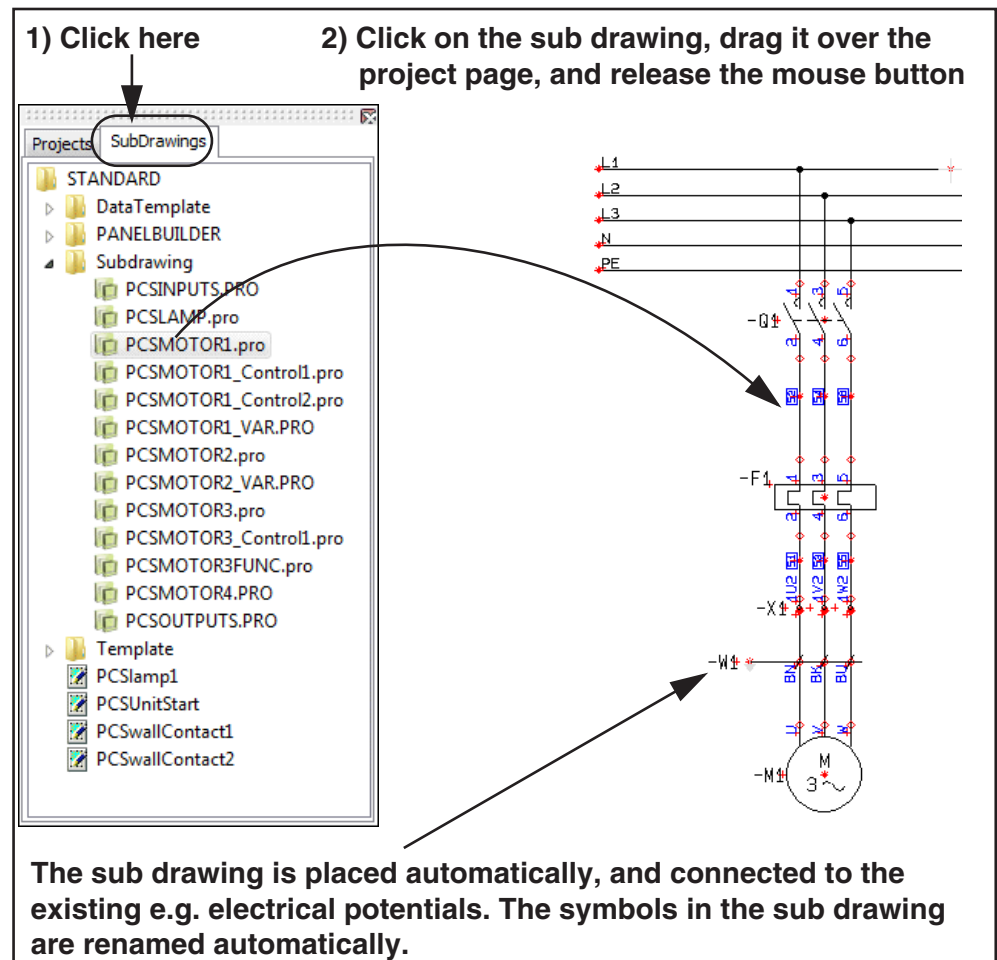
If variables are placed in the sub drawing, or if you can select different sets of model data for the sub drawing, you now get the option of specifying which data to insert in the sub drawing. You can read more about this by continuing the explanation under "If the sub drawing contains Model data" on page 302.

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**:
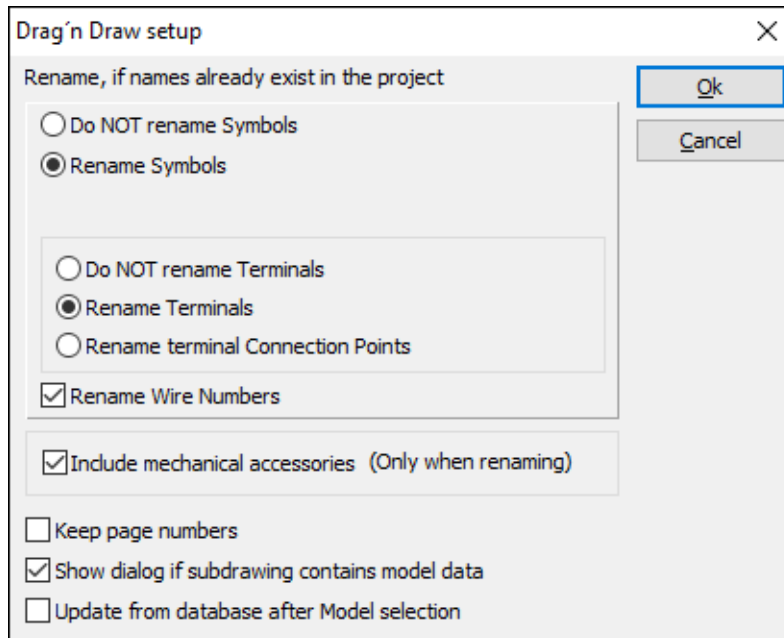
Here you specify whether to rename symbols in the sub drawings you insert, and how to rename these symbols. — Read more about this in "Intelligent copying of areas" on page 281.

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 302, "Inserting sub drawings: With model data" on page 303 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 **Settings** button appears in the dialog box: Click on it, and you enter the dialog box **Update from Component database**.

Here you specify how to perform the update from the database. These options are described in "Updating symbols from the database" on page 196.

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 311.

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**:
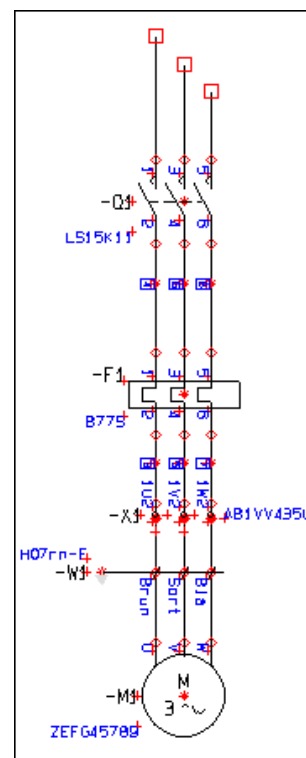
2) **Automation** now changes to the **Edit sub drawing** mode, which you can see because the **Page type** field at the bottom of the left toolbar blinks green. You can now create a new sub drawing.

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 311.

4) Continue as described in "Inserting e.g. variable markers and/or model data" on page 312.
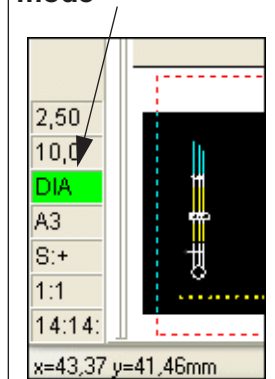
LAYERS ARE FLATTENED TO ONE LAYER

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 310.

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
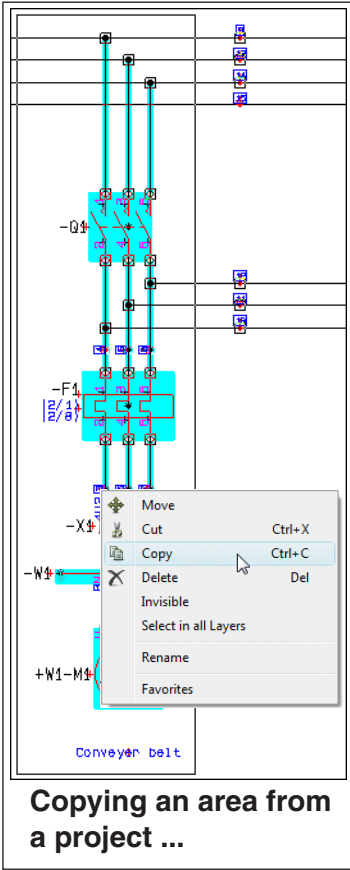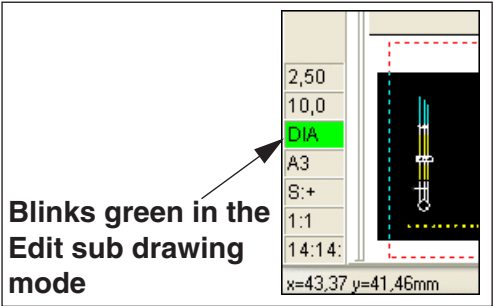
💡 **SAVE COPIED AREAS USING SAVE AS**

IN STEAD 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 **SUBDRAWINGS** 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:
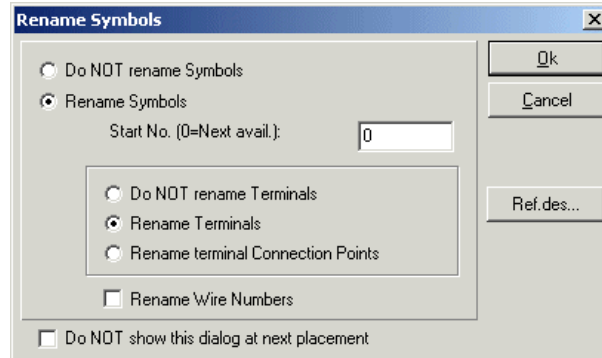
— See which options you have in "Intelligent copying of areas" on page 281.
Select the desired settings and click **OK**.

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.

Inserting a copied area when creating a new sub drawing
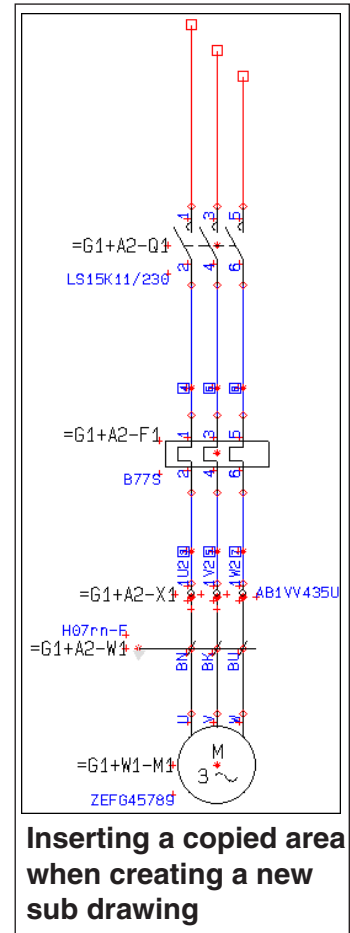
Adapting the sub drawing

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 324, and you can create different sets of *model data* for the sub drawing, as described in "Creating Model data for sub drawings" on page 327.

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 322. (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 **Projekt generator** manualen).

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 314 AND "PREPARING SUB DRAWINGS FOR DRAG'N DRAW / AUT. PLACING" ON PAGE 320.

REPLACING SUB DRAWINGS

See how to replace sub drawings in "Replacing an object group" on page 295.

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 320. 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.

| **Project page prepared for automatic placing of sub drawings by drag'n draw** | **+ 4 sub drawings for drag'n draw** | **= Automatic filled out project page** |
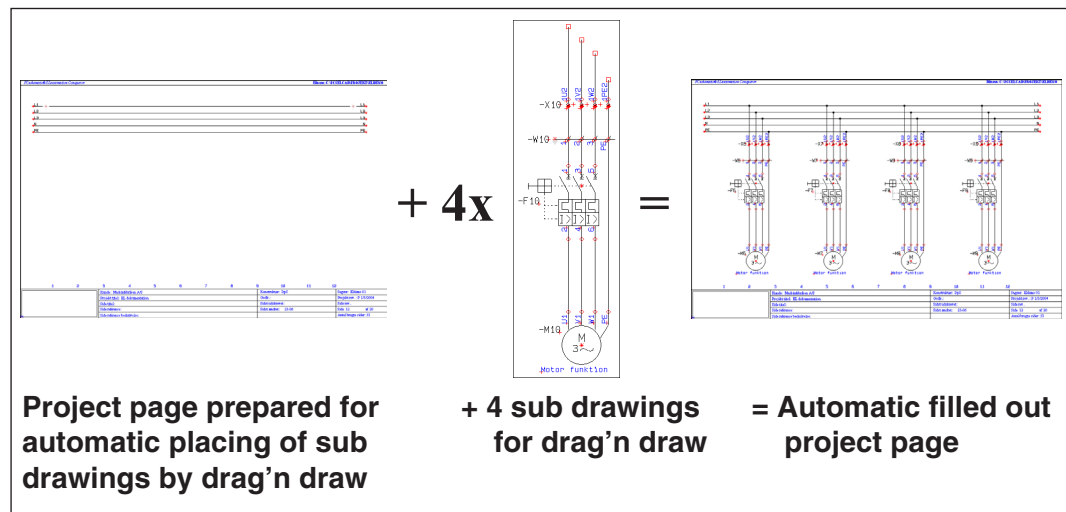
Example

An example on creating a drawing template can be found in "Creating Project pages for aut. placing of sub drawings" on page 315.

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** or click the **New document** button, and you enter the dialog box **New**:

2a) Click on the tab Normal pages

2b) Click on Template to create/edit a page template

Or: Click on Project when creating drawing templates

Terminals Lists	Cables Lists	PLC Lists	Divider
Projects	Normal Pages	Table of Contents	Parts Lists	Components Lists

Blank Page · PCS-L1-L2-L3-PE-N · PCS-L1-N · PCSA3combi... · PCSA3Hbasic · PCSA3Hfro... · PCSA3Hfro...

PCSA4Hbasic · PCSA4Vfro... · PCSmechanical · PCSprojectinfo

Ok · Cancel · Create: Project / Template

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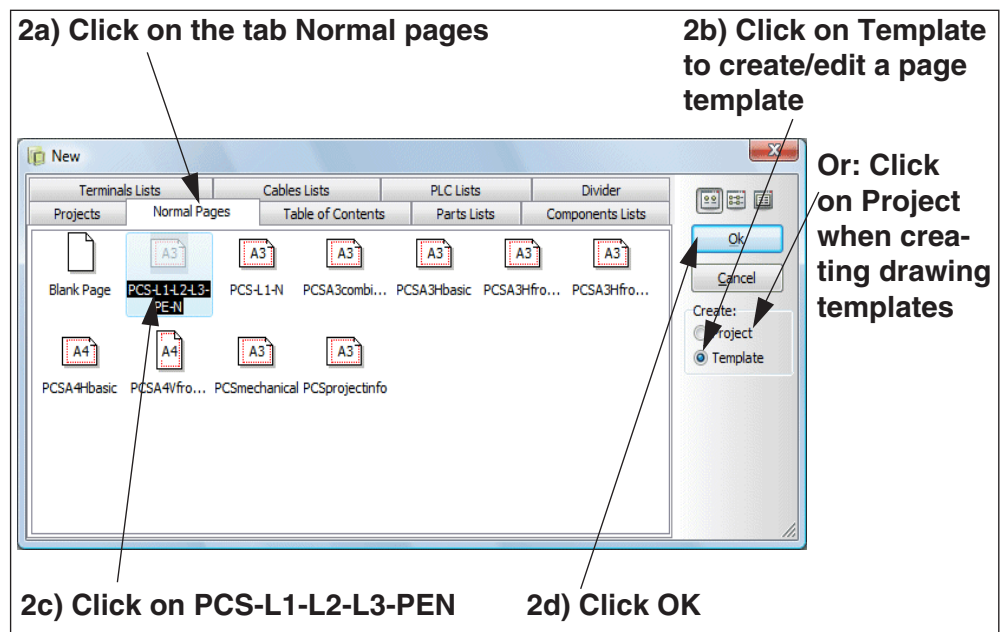
2) Click on the tab **Normal pages**, click on *Template* in the right-hand side, click on the page **PCS-L1-L2-L3-PEN**, and click **OK**.

> **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 **Projekt generator** manualen — 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 **Projekt generator** manualen.

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.
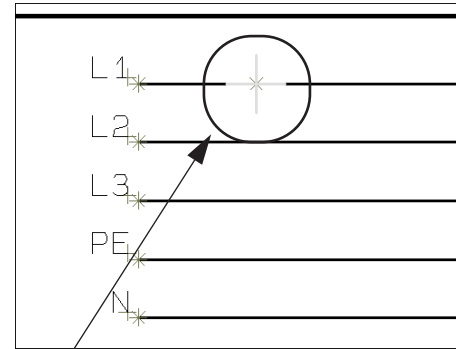
4) In the other end of the electrical potentials, there is placed another insertion point symbol. These two symbols specifies the area in which sub drawings can be placed automatically. It is important that these two symbols are aligned.

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 322.
— SEE "Points to consider in Project generation" IN THE **Projekt generator** manualen 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 315. 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 **INSPOINT.SYM**, and click **OK**. — Read more about the **Symbol menu** in "Fetching symbols using the Symbol menu" on page 162.

5) Place the insertion point symbol — for instance at the *right*-hand side of *L1*.
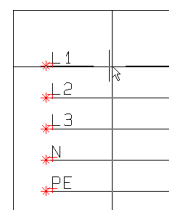
6) Insert another insertion point symbol — for instance at the *left*-hand side of *L1* — and press <Esc> to remove the symbol from the cross hairs. It is important that the two insertion point symbols are aligned.

7) Click on the **Save** button.

8) In the dialog box **Save**: Write for instance the name *Example.std*, select the folder **Pcselcad\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 **Projekt generator** manualen).

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 IN STEAD *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 **Projekt generator** manualen.

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** or click the **New document** button. In the dialog box **New**, click on the tab **Normal pages**, and double-click on **PCS-L1-L2-L3-PEN**:
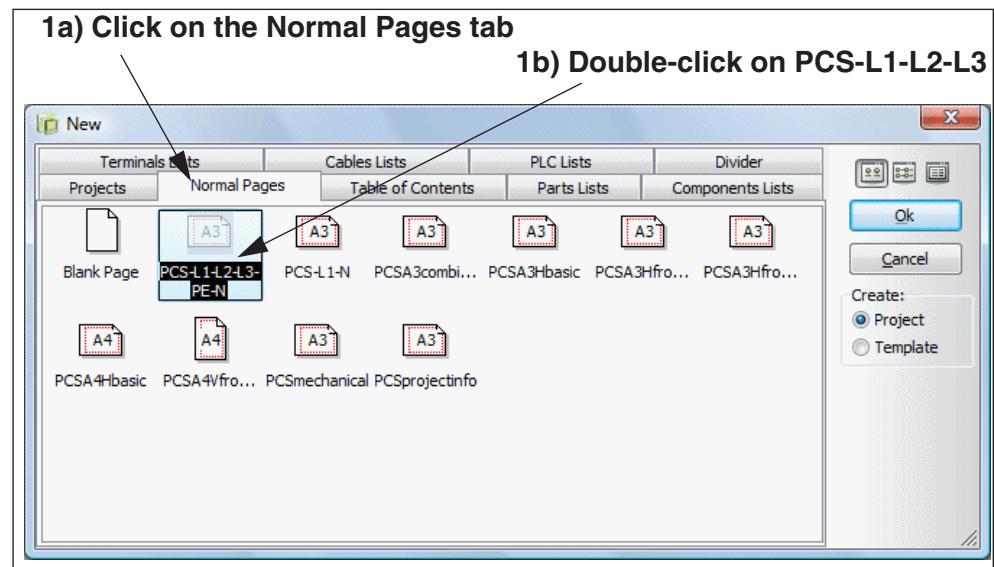
2) When you have clicked **OK** in the **Settings** dialog box, you can see the two insertion points on *L1*, as described in "Page Templates prepared for automatic placing of sub drawings" on page 315.

3) In the **Explorer window**: Click on the **SubDrawings** tab, open the folder **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 320), *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*:
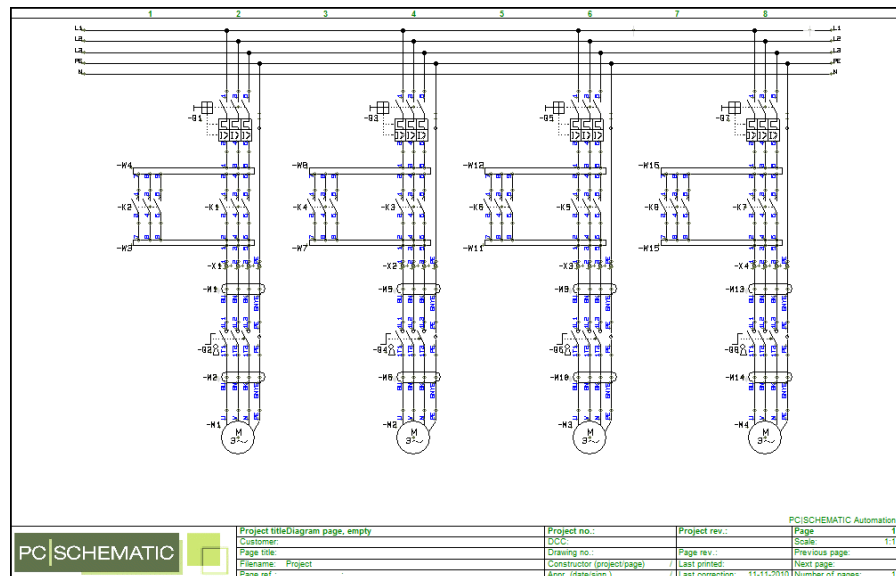
— If you could only get room to place the sub drawing three times on the page, this is because the insertion points on the page are placed too close to make room for four sub drawings on the page. For the future use of the page, you might therefore consider to make room for exactly four sub drawings on the page. — See e.g. "Points to consider in Project generation" in the **Projekt generator** manualen, "Creating sub drawings as Modules" in particular.

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 304.

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 307.

— 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 315.

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 **Projekt generator** MANUALEN. 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 310, 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 311.

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 316 or read more about the **Symbol menu** in "Fetching symbols using the Symbol menu" on page 162.

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 322 (and see "Points to consider in Project generation" in the **Projekt generator** manualen).

The left insertion point

The right insertion point
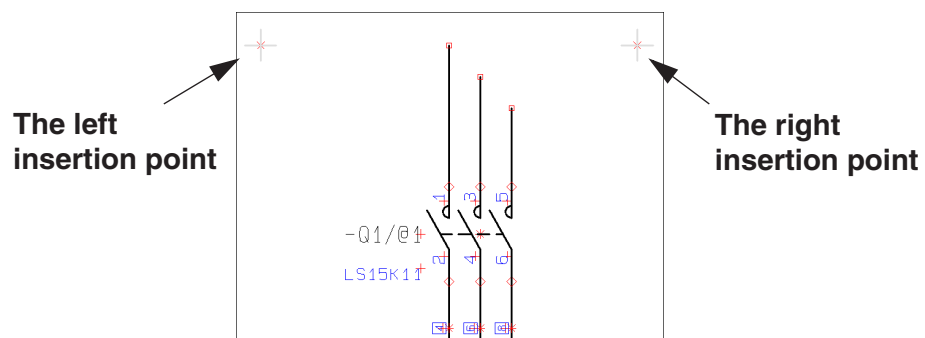
5) *If* you also wish to control the *texts* in the sub drawing (e.g. for automatic project generation — see the **Automation**): Insert some so-called *variable markers*, which makes it possible to control the texts of the sub drawing when placing the sub drawing (or to control the texts from a *project definition file* during automatic project generation). Read how to do this in "Inserting variables in Sub drawings, Pages and Projects" on page 324.

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 327.

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 **Projekt generator** manualen.

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 307. 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 322.

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 314).

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.

The left insertion point of the project page before inserting the sub drawing

The left insertion point of the project page after inserting the sub drawing
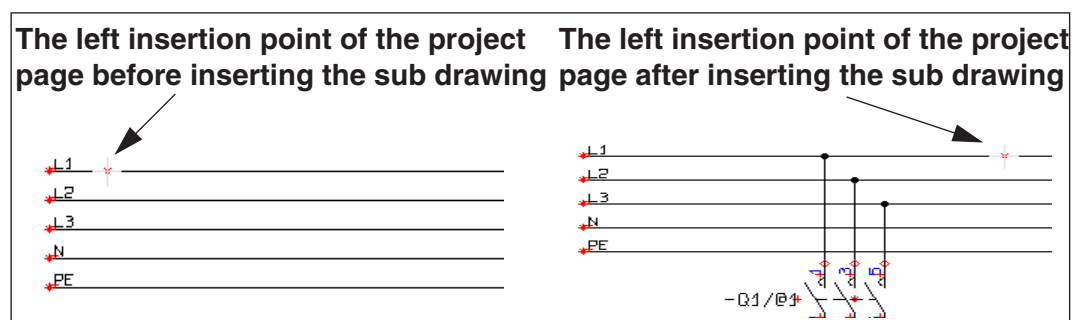
A sub drawing can not be placed automatically if there is not enough room for this sub drawing between the insertion points on the project page.

Insertion points and automatic project generation
During automatic project generation — see "Generate and Update projects auto-matically" in the **Projekt generator** manualen — 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.

| **Project page** | **Sub drawing** |
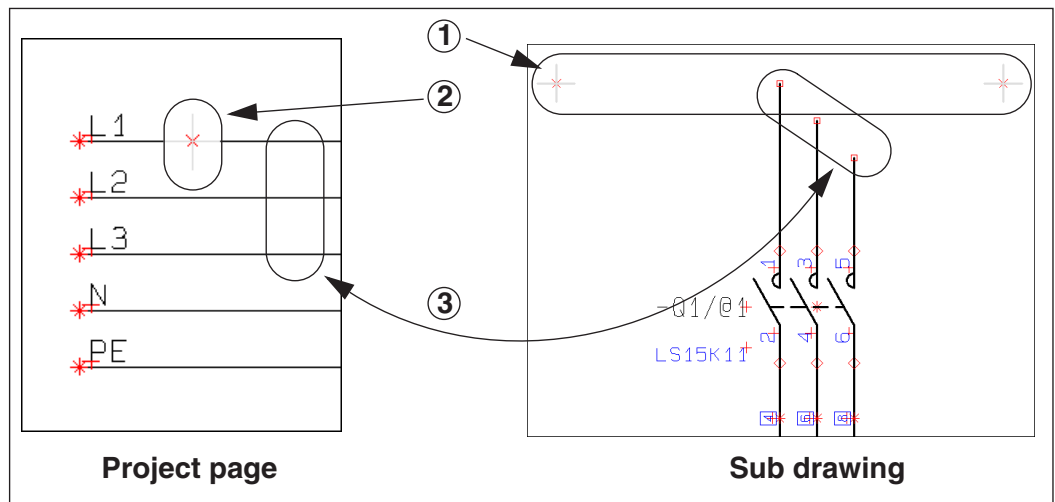
It is important to make sure of the following (observe the figure above):

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 gen-eration.

— Please also read "Points to consider in Project generation" in the **Projekt gene-rator** manualen. 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 302.

To do this you must:
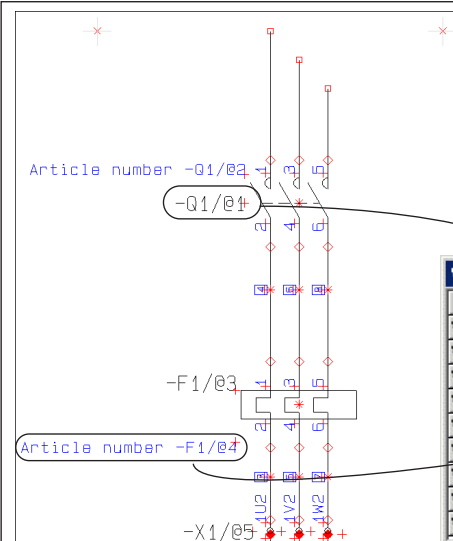
1) Insert *variables* in the sub drawing, as described in "Inserting variable markers on a sub drawing (or on a drawing template)" on page 324.

2) Create a *variable list*, as described in "Creating a variable list" on page 326.

When applying sub drawings for automatic project generation, you can control *all* texts and names in the project. — See the **Projekt generator** manualen.

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 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 304 and "Manual placing of sub drawings" on page 302.

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 327.

VARIABLES ON ORDINARY PROJECT PAGES

YOU CAN ALSO INSERT VARIABLES ON ORDINARY PROJECT PAGES.
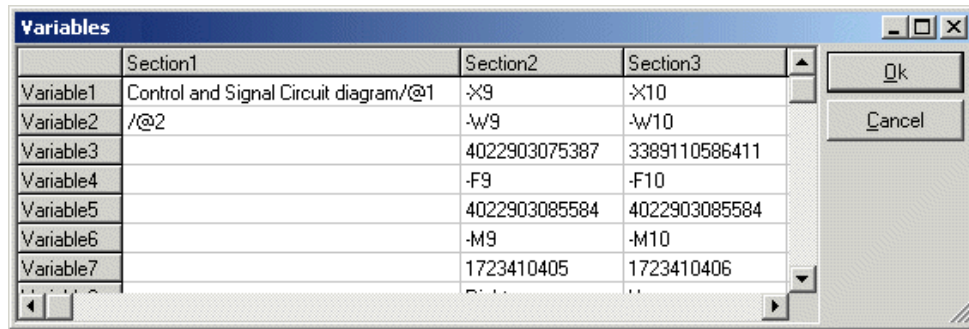
Variables for Page title and Page data

On drawing templates — and in ordinary project pages — you can also insert variables on all **Page data** datafields, and in the **Page title**.

When you have inserted these variables in the pages, and have executed **Tools => Create variable list**, you can choose **Edit =>**

Variables. Here the created variables for the page will always be placed in the first section:

Variables				
	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

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.

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 790 —, you can also insert variables on these.

Variables in reference designations

You can also insert variables in reference designations for symbols, reference frames and pages. Read more about this in "Variables in reference designations" on page 976.

Creating a variable list

This applies for sub drawings / drawing templates / project pages:

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.

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 310 or "Creating sub drawings by Copying from projects" on page 311. — 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 320.

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 324.

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**:

Variables			
	Data		Ok
Variable1: S.Func	40 - 100 A/@1		
Variable2: S.Arti	4011209481039/@2		Cancel
Variable4: S.Arti	4011209272521/@4		
Variable5: S.Type	0,06Kw/@5		Add model
Variable7: S.Arti	/@7		
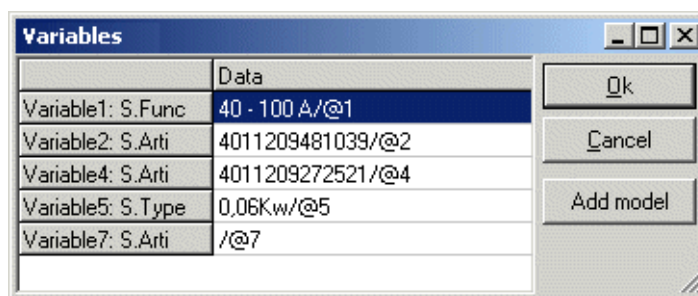
4) In the dialog box **Variables**: Click on **Add model**:

New model	
Enter model title	Ok
55 kW 40 - 100 A	Cancel
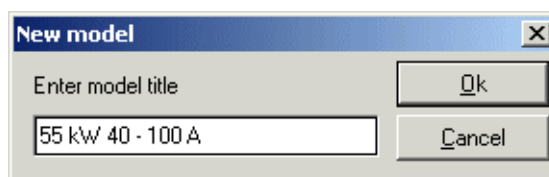
5) In the dialog box **New model**: Type in the name on the new model you wish to create, and click **OK**.

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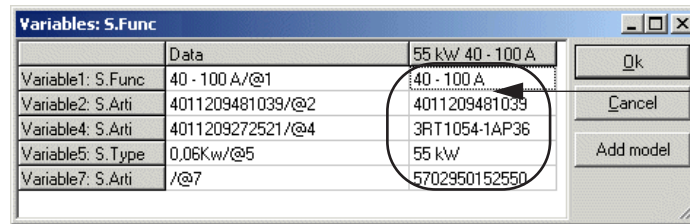
Double-click in the fields, and type in the data

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**:
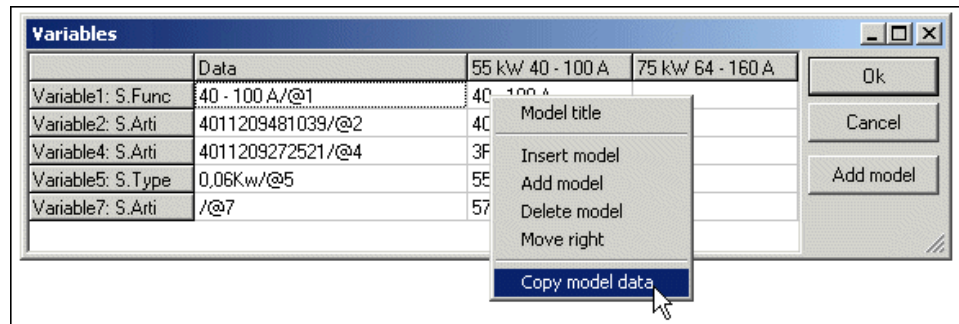
9) Right-click on the new model, and choose **Paste model data**:
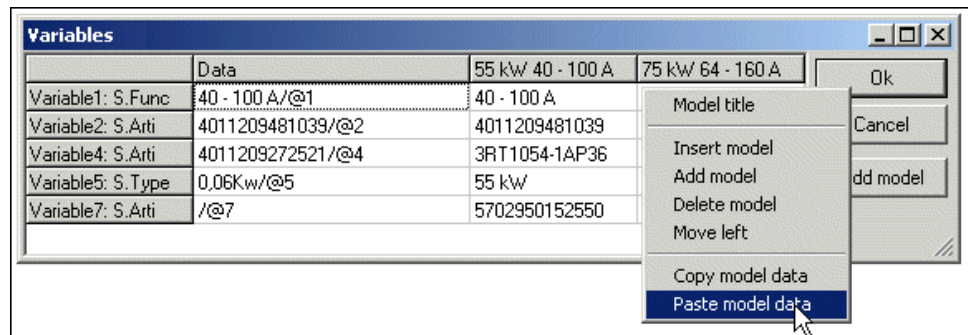
10) Answer **OK** to overwrite the model data for the new model (do *not* select to insert the title), and the data are inserted as data for the new model.

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.
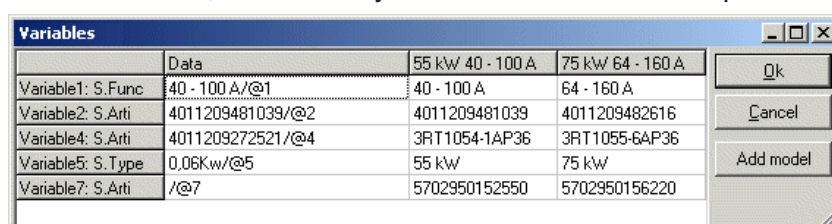
12) Click **OK**.

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 302, "Inserting sub drawings: With model data" on page 303 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 **Projekt generator** manualen.

Model data: Options when right-clicking

When you right-click in the dialog box **Variables** when creating model data, you get the following options:

Database: Gives you the option of choosing components from the database. Appears when right-clicking on an **S.Article** or a **S.Type** variable.

Model title: You can edit the title for the selected model.

Model variable: See "Creating a model variable" on page 329.

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 330.

Move right: Moves the selected model one position to the right. — See also "Drag and drop Models in the dialog box" on page 330.

Copy model data: Copies the values for the selected model — see "Creating a new model by copying data" on page 328.

Insert model data: Inserts the copied model values — see "Creating a new model by copying data" on page 328.
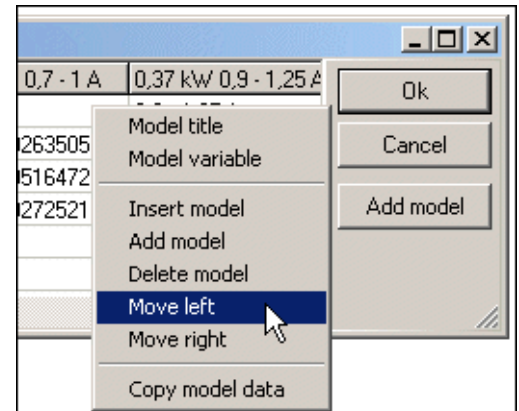
Drag and drop Models in the dialog box

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 808.

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

Insert e.g. the line connection symbol here
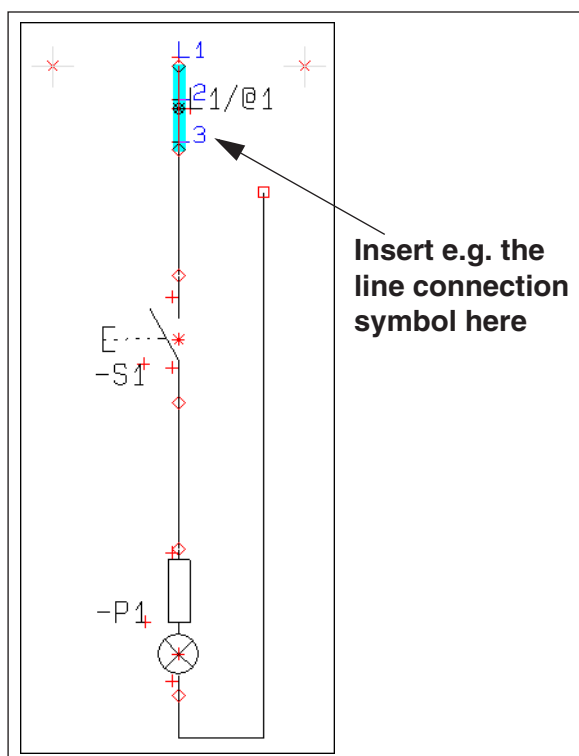
Controlled via variable marker in the symbol name

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:
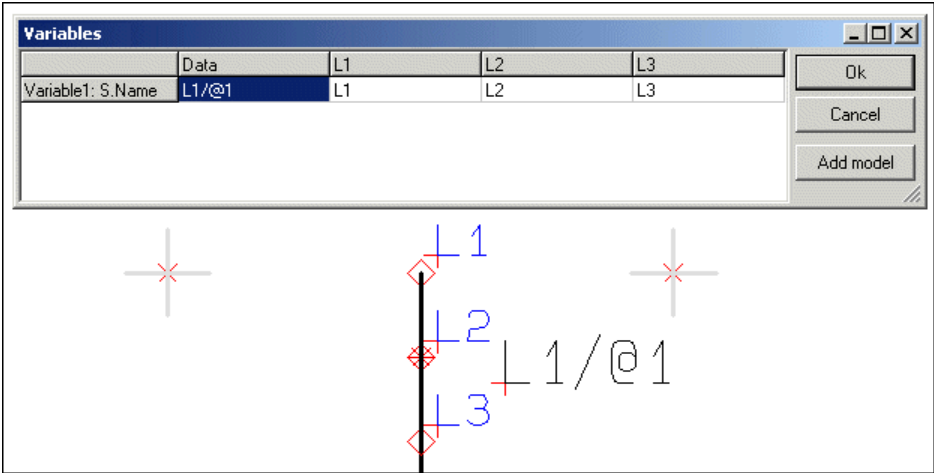
When you place the sub drawing in a project, you can therefore get the following result by placing the three different models:

Model L1 selected   Model L2 selected   Model L3 selected
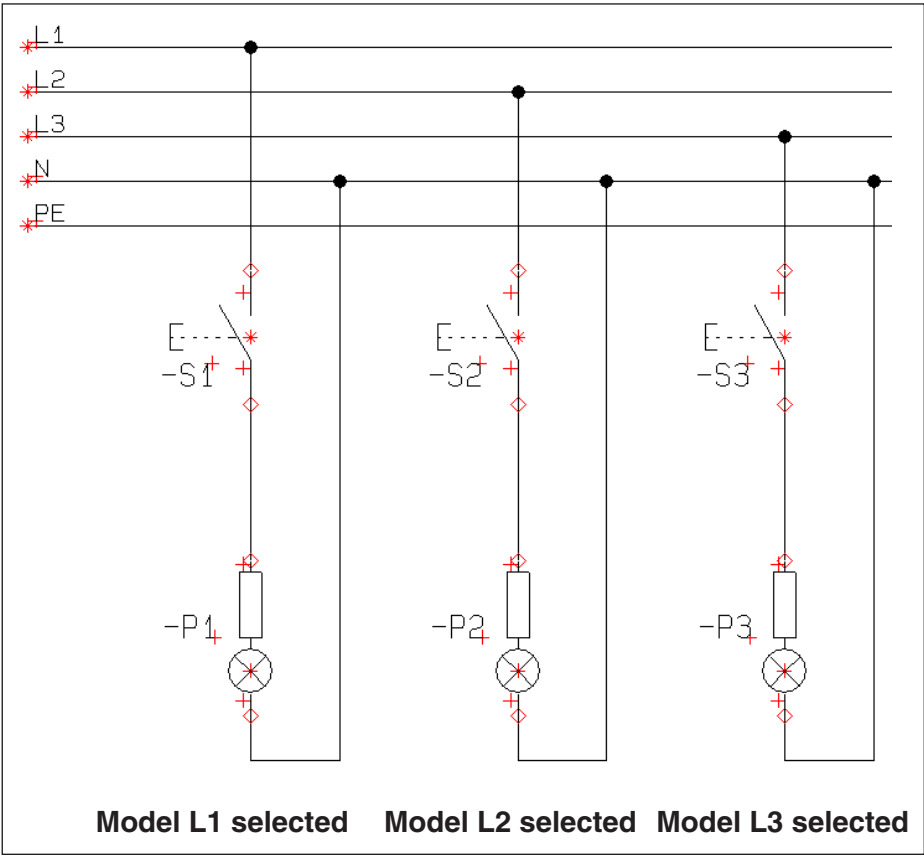
When the sub drawing is placed on the project page, the line connection symbol is converted to an ordinary line, and is therefore no longer a symbol.

RENAME SELECTED 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.

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 338.

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 334.

2) Specify a direction for renaming the symbols — see "Direction for renaming symbols" on page 335.

3) Specify how to rename terminals (if any) — see "Naming when renaming terminals" on page 336.

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 336.

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 MEANING 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.

"NAME PROTECTED" COMPONENTS ARE NOT RENAMED

READ MORE ABOUT THIS IN "NAME PROTECTED" ON PAGE 180.

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 354.

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*)

=G1+A1-P*

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**:

    5a) 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.

5b) Right-click and select **Rename** (see the figure below) — or select **Tools => Rename symbols**.
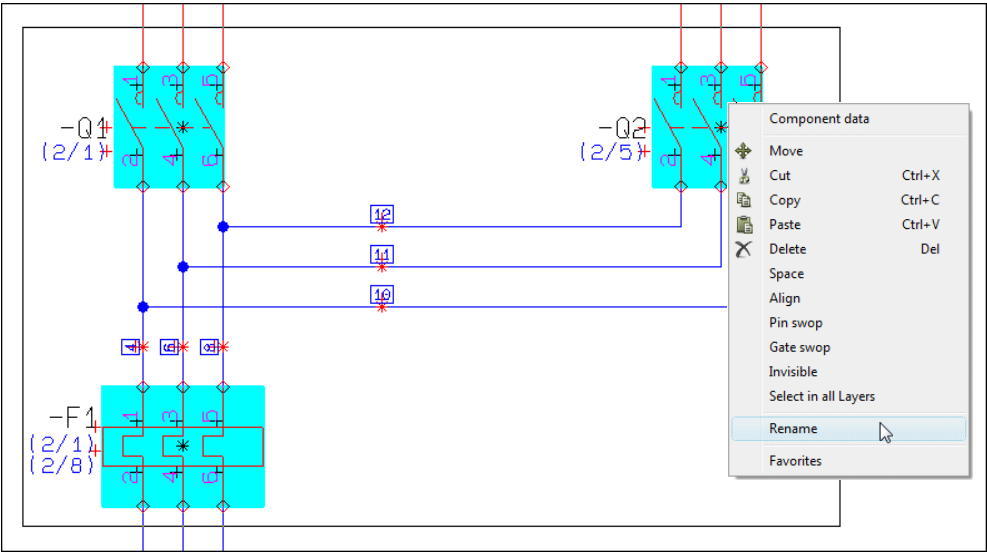
5c) You now enter the dialog box **Rename selected symbols**, where only the selected symbols are displayed:

Name	Pinno	Type	New Name	New Pinno	Page no.
=G1+A2-F1	1,2..(6)	B77S			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**.
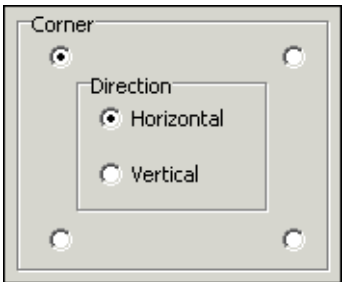
What is important for which number a component is assigned, is therefore the first time a symbol for the component appears in the project.

— 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 *NOT* RENAMED.
RENAME TERMINAL CONNECTION POINTS	THE TERMINAL CONNECTION POINTS ARE RENAMED. TERMINAL NAMES ARE NOT RENAMED. — SEE "FULFILLING STANDARDS WHEN COPYING AREAS" ON PAGE 282.

— 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 281.

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	AB1VV435U	=G1+A1-X1	6	4
=G1+A1-X1	9	AB1VV435U	=G1+A1-X1	8	4
=G1+A1-X1	8	AB1VV435U	=G1+A1-X1	9	4
=G1+A1-X1	7	AB1VV435U	=G1+A1-X1	6	3
=G1+A1-X1	8	AB1VV435U	=G1+A1-X1	9	3
=G1+A1-X1	6	AB1VV435U	=G1+A1-X1	7	3
=G1+A1-X1	6	AB1VV435U	=G1+A1-X1	7	4

Start value: 0

— See also "Insert new Symbol names or Connection names" on page 338.

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:
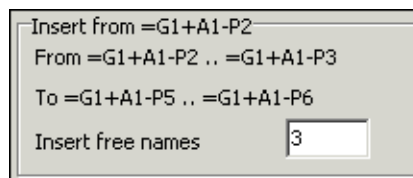
1. Insert new symbol names

To insert e.g. three new symbol names starting from **-P2** in the project, do this:

6d) Select **-P2** in the project.

6e) Select **Functions => Rename symbols** — or right-click on **-P2** and select **Insert name**.

6f) 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        3
```

6g) 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.
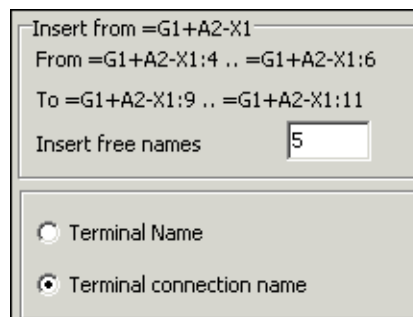
2. Insert new connection names on terminals

To insert e.g. five new connection names in terminal **-X1**, starting from connection **4**, do this:

7h) Select the terminal which has the connection **4** in the project.

7i) Select **Functions => Rename symbols**. (Or right-click directly on the terminal symbol and select **Insert name**).

7j) 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        5


  ○ Terminal Name
  ● Terminal connection name
```

7k) 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 338.

SORTING ACCORDING 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 draw-
ing objects you are working with.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO INSERT AN IMAGE IN A PROJECT.

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

Here you can choose between inserting the image with or without a link. — Se
more about this on the figure above.

The file types you can open from here
are either *.bmp* files, *.jpg* files, meta-
files, enhanced metafiles, *.gif* files or
.png files.
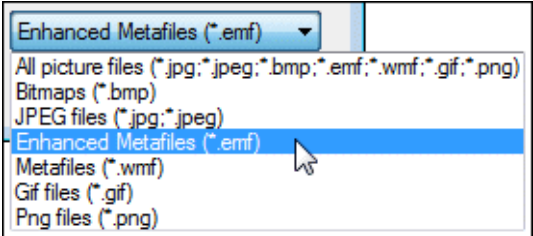
If you need to insert other file types/
image formats, it is possible that you
can insert the image as an OLE object,
which is described in "Inserting files
from other programs / OLE" on page 749.

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.

MAXIMUM SIZE FOR IMAGES 1600X1200 PIXELS
PLEASE NOTE, THAT WHEN INSERTED IMAGES ARE LARGER THAN
1600X1200 PIXELS, THEY WILL BE DOWNSCALED PROPORTIONALLY.

Move, delete, copy or rotate inserted image

If you want to select an image, you click the **Area** button, and then click on the ref-
erence 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

Point at the corner, and drag until the desired size is achieved

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 290.

SURVEY AND REFERENCE DESIGNATIONS

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

PC | SCHEMATIC

THE EXPLORER WINDOW: THE PROJECTS TAB

The **Explorer** window contains two different tabs: The **Projects** tab and the **Sub-Drawings** tab. This chapter describes the **Projects** tab, as well as the general options in the **Explorer** window. The **SubDrawings** tab is described in "Sub draw-ings" on page 299.

There are also three buttons at the top of the **Explorer window**, which are described else-where: The **Symbol menu** button (see "Fetching symbols using the Symbol menu" on page 162), the **Component database** button (see "Fetching symbols from the database" on page 170) and the **Object Lister** button (see "The Object Lister" on page 351).
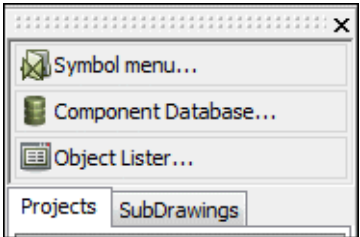
In the **Projects** tab in the **Explorer** window, you can display all types of information about the active projects, and can navigate back and forth between the project pages in all of the open projects. Furthermore, you can edit Project data, Page Data and component data directly from the **Explorer** window.

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 item which shows/hides the **Explorer** win-dow, 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.

PC | SCHEMATIC

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.

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.

Click here to open
Click here to close
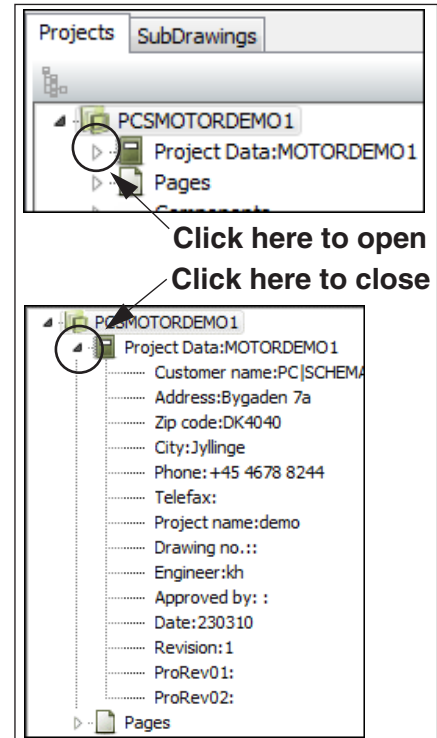
CHANGING DATA FROM THE PROJECTS TAB

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 data and Page data" on page 128.

Make the changes and click **OK**.
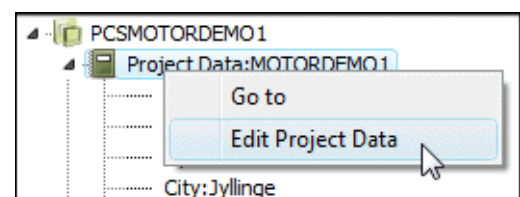
Editing the contents of a single Project data datafield

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 346.

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 360.

Editing Component data from the Projects tab in the Explorer window

When you want to edit the Component data for at symbol, this can be done by right-clicking on the symbol in the **Projects** tab in the **Explorer** window, and choosing **Edit Symbol Data**.
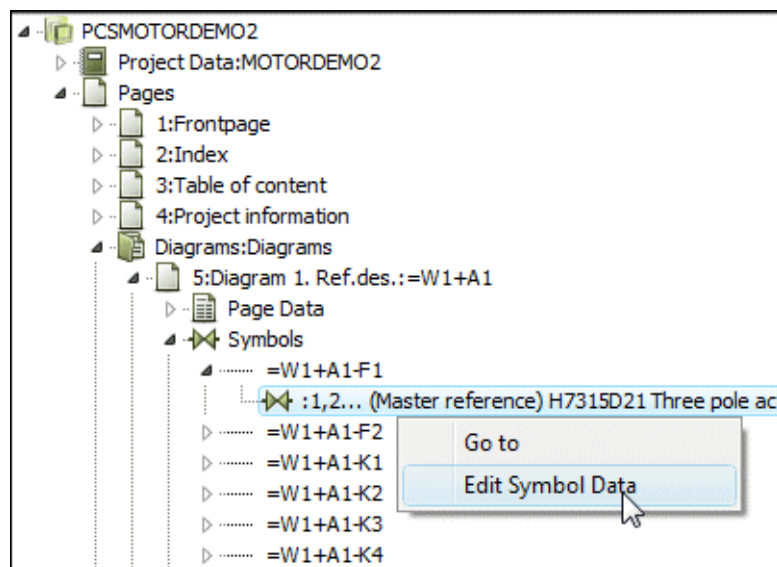
This brings you into the dialog box **Component data**, where you edit the component data and click **OK**.

You can also make changes in article data for cables and signals from the **Projects** tab in the **Explorer** window.

Component data and cable article data can also be changed directly from the **Object Lister** (short-cut key <F7>), as described in "Check and edit component data and cable data" on page 358.

THE EXPLORER WINDOW: SELECTING SYMBOLS

When you wish to navigate between the symbols in the project using the **Projects** tab in the **Explorer** window, you have two options:

- You can select a symbol in the active project, and then have it selected in the **Explorer** window

- You can select a symbol in the **Explorer** window, and then have it selected in the project

Select a symbol in the project - and see it in the Explorer window

When you select a symbol in the project and then click the **Show selected symbol** button at the top of the **Projects** tab in the **Explorer** window, the symbols for the components are displayed on the tab:
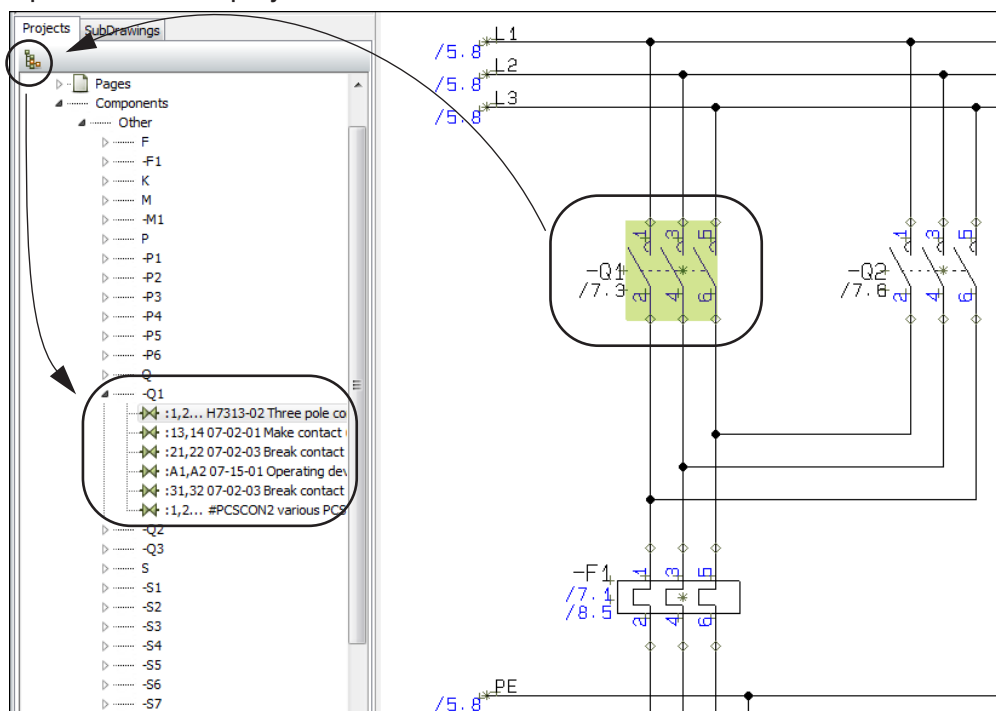
This also gives you the possibility to jump to the other symbols for the component by double-clicking them in the **Explorer** window.

— If reference designations are used in the project, please see "Reference designations in the Explorer window" on page 411.

Select a symbol in the Explorer window - and see it in the project

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:
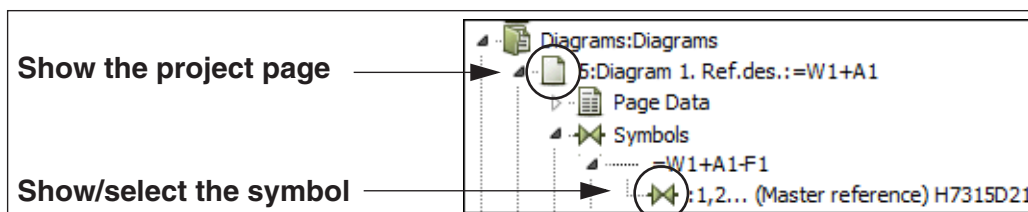
You can for instance do this in the **Projects** tab (see the picture below): Click the triangle icon next to the project name and click the triangle icon next to **Pages**. You now get a survey of all the pages in the project.

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*, *Symbols*, *Cables* and *Signals* on the page.

Open *Symbols*, and then open *=W1+A1-Q1*.

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:
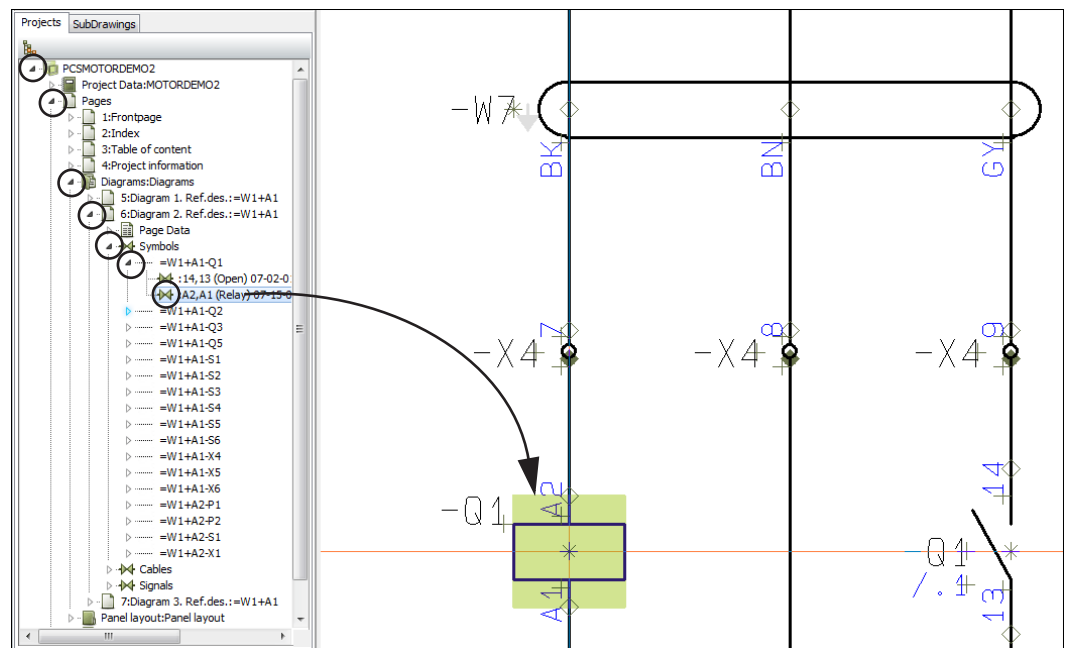
When you then click on the symbol icon next to the make contact, the cross-hairs then points out the make contact for you.

The line you clicked on can contain information about the connection names for the symbol, the type of the symbol, the symbol file name 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.
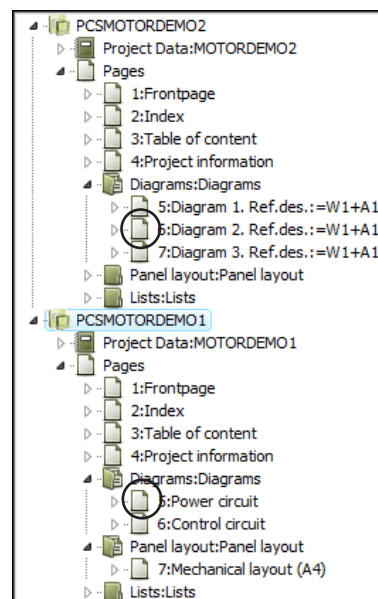
UPDATING THE EXPLORER WINDOW

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**.
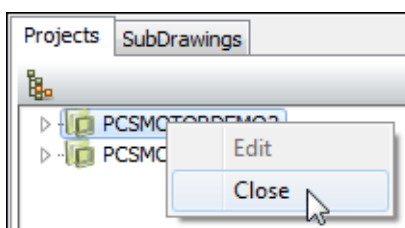
You can also right-click on a project in the **Projects** tab in the **Explorer** window, and choose **Go to**. This activates the last page you worked on in the project.

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, symbol names, reference designations, PLC I/O data, page data, free texts and many other texts in 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 345 and "Find and Replace" on page 361.

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 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 texts/data for **Symbols** (components), **Signals**, **Wire numbers**, **Cables**, **Terminals**, **PLC I/O**, **PLC I/O Ref**, **Texts**, **Reference frames**, **Pages** or **Lines** (only lines with article data are displayed).

THE OBJECT LISTER REMAINS ON SCREEN

THE **OBJECT LISTER** REMAINS ON SCREEN WHILE YOU WORK WITH THE PROJECT. WHEN THE **OBJECT LISTER** DISPLAYS DATA, THESE DATA ARE UPDATED AUTOMATICALLY, WHEN YOU MAKE CHANGES IN THE PROJECT.
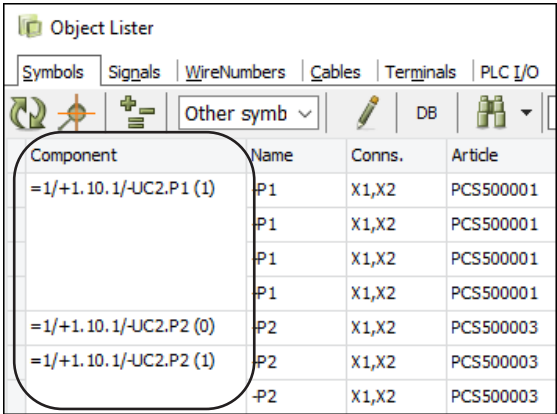
— THE **OBJECT LISTER** CAN BE SET TO BECOME TRANSPARENT, WHEN YOU ARE WORKING ON THE DRAWING AREA, WHILE YOU LET THE **OBJECT LISTER** REMAIN ON THE SCREEN — SEE "POINTER/SCREEN" ON PAGE 918.

COMPONENTS IN THE OBJECT LISTER

If components are not displayed in the **Object Lister**, you can activate displaying components as described in "Displaying information in the Object Lister" on page 356.

When a component contains more than a single symbol, the component name is only displayed for the first symbol for the component — as you can see in the figure to the right.

Brackets after the component name specifies the component group number — see "Component Grouping" on page 626.

SYMBOLS IN THE OBJECT LISTER

When you have selected the **Symbols** tab in the **Object Lister**, you will get the project symbols listed in alphabetical order — independent from which page that you are working on.

You can specify which symbol types to display by selecting the relevant symbol types in the toolbar in the **Object Lister**.
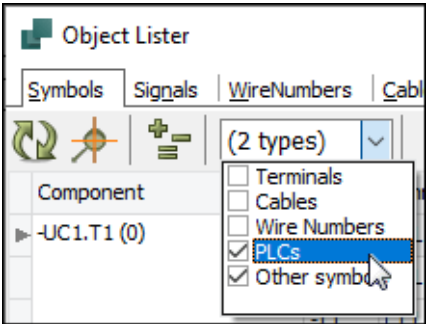
NAME PROTECTED SYMBOL NAMES CAN NOT BE CHANGED HERE

IF A SYMBOL NAME IS **NAME PROTECTED**, THE SYMBOL NAME CAN NOT BE CHANGED FROM THE **OBJECT LISTER**. HOWEVER, OTHER TEXTS FOR THE SYMBOL CAN BE CHANGED. — READ MORE ABOUT THIS IN "NAME PROTECTED" ON PAGE 180.
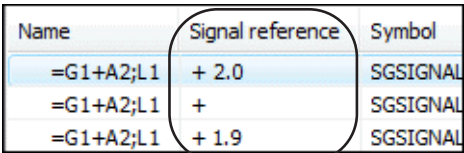
Signals in the Object Lister

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.

If you right-click on the line with field names, you can also choose to display individual columns with *Ref.Design. Function*, *Ref.Design. Location* and/ or *Ref.Design. Product*.You can then easily sort the list by mechanical assembly (position) or function.

Only objects with this reference designation are displayed

Right-click here to select columns for displaying reference designations

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 the **Print** button 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 357.

This also applies when you click the **Save as file** button to save a file. — See "Save as File from the Object Lister" on page 357.

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 or I/O addresses.

When for instance you wish to change the 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 I/O addresses for the symbol that you want to change the addresses for — see "Selecting objects in the Object Lister" on page 354 —, and press <Enter> or <F2>:
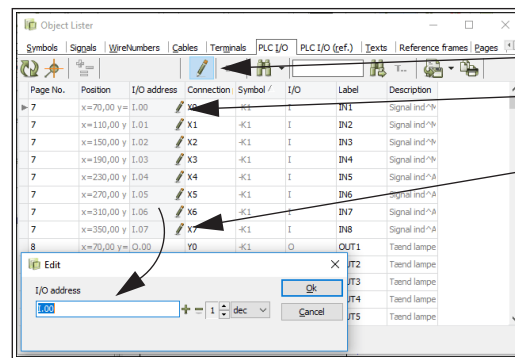
2) Click the Pencil

3a) Click here

3b) Press down
     <Shift> and
     click here

3c) Press <F2>
     or <Enter>

4) You now get the option to change the selected texts automatically, as described in "Changing a text" on page 130. Click **OK**.

DISPLAYING INFORMATION IN THE OBJECT LISTER

When you right-click on the row with column headings, you can choose which information to display 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.

If for instance components are not displayed in the **Object Lister**, you can select **Component**, and the column will now appear in the **Object Lister**.

If you want the component name to appear as the first column, you can just *click* on the column header **Component** and *drag* it to the desired position.

— See "Components in the Object Lister" on page 353.
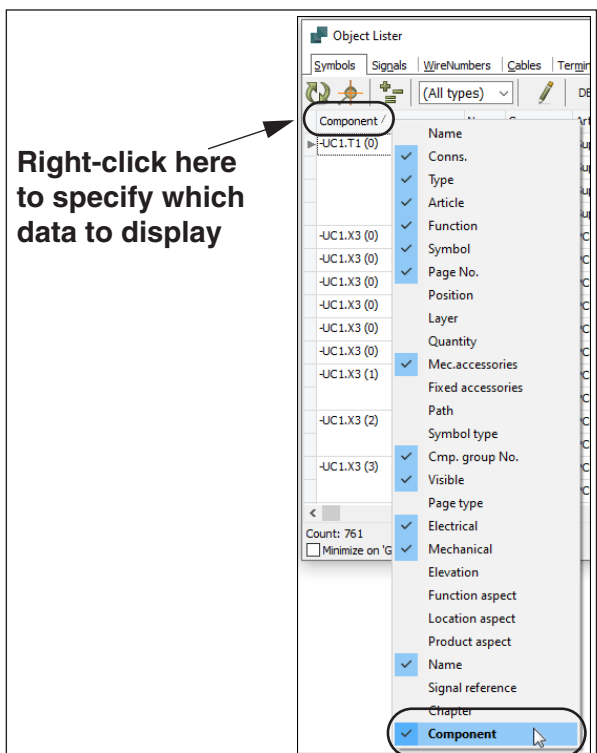
**Right-click here
to specify which
data to display**

PRINTING FROM THE OBJECT LISTER

When you have selected more than one object, only the selected objects are printed, when you click the **Print** button. 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 354.

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 the **Save as file** button. 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 354.

When you click on **Save** you enter the dialog box **Save**. Here you can save the selected objects in three different ways:

Click here for a short- cut to save in Excel

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)** or **Microsoft Excel (*.xlsx)**: The file is saved in Microsoft Excel format.
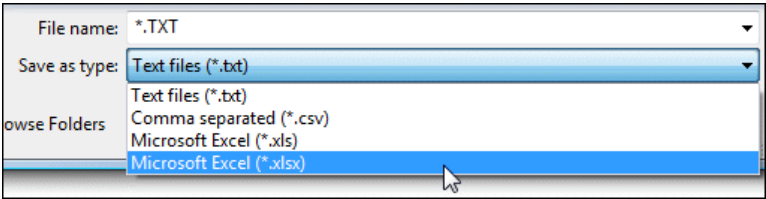
Select file type, type in a file name, and click **OK**.

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 DEPENDENT 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 354.

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*.

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.

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**.

If for instance you want to change the symbol type, click in the *Type* field for the relevant symbol, and make the change.
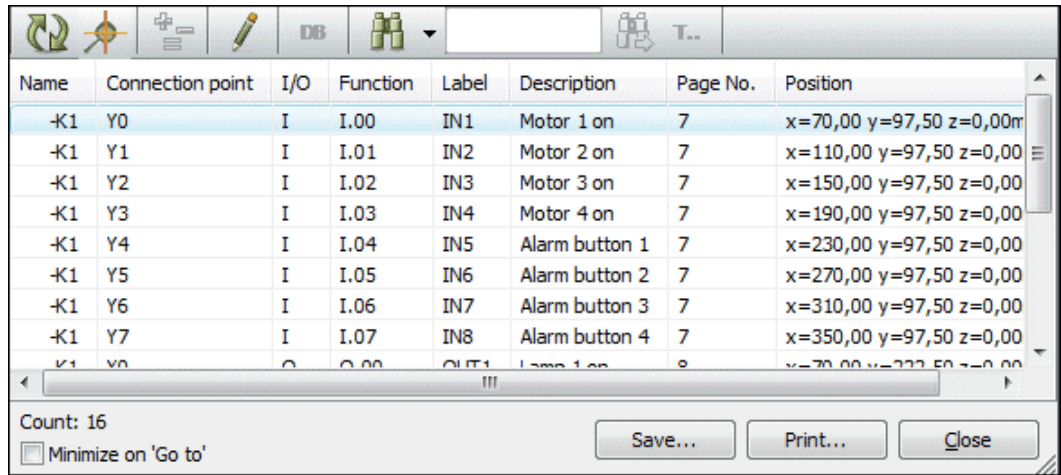
When you click on **Cables** you can make similar changes for **Cable data**.

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 359.

EDITING PLC I/O DATA VIA THE OBJECT LISTER

To edit PLC I/O data in **Automation**, click on **PLC I/O** in the **Object Lister**.

You can then choose either **Used I/O** — which gives you all used PLC I/O symbols in the project — or **From reference symbols** — which gives you all mentioned connection points in the PLC reference symbols in the project.

Name	Connection point	I/O	Function	Label	Description	Page No.	Position
-K1	Y0	I	I.00	IN1	Motor 1 on	7	x=70,00 y=97,50 z=0,00m
-K1	Y1	I	I.01	IN2	Motor 2 on	7	x=110,00 y=97,50 z=0,00
-K1	Y2	I	I.02	IN3	Motor 3 on	7	x=150,00 y=97,50 z=0,00
-K1	Y3	I	I.03	IN4	Motor 4 on	7	x=190,00 y=97,50 z=0,00
-K1	Y4	I	I.04	IN5	Alarm button 1	7	x=230,00 y=97,50 z=0,00
-K1	Y5	I	I.05	IN6	Alarm button 2	7	x=270,00 y=97,50 z=0,00
-K1	Y6	I	I.06	IN7	Alarm button 3	7	x=310,00 y=97,50 z=0,00
-K1	Y7	I	I.07	IN8	Alarm button 4	7	x=350,00 y=97,50 z=0,00
K1	Y0	O	O.00	OUT1	Lamp 1 on	8	x=70,00 y=222,50 z=0,00

Count: 16

☐ Minimize on 'Go to'

[Save...] [Print...] [Close]

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 358.

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 356.

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 357.

CHECKING AND EDITING PROJECT TEXTS VIA THE OBJECT LISTER
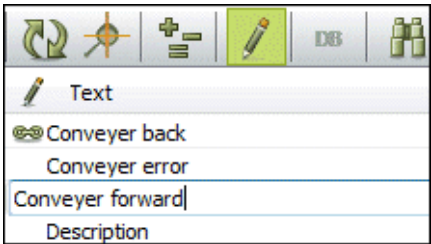
It is a good idea to check the project texts by clicking on **Texts**. If you need to make corrections, click the **Pencil**, and click in the text field you wish to edit.

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 356.

EDITING PAGE DATA IN THE OBJECT LISTER

When you click on **Pages**, you get a survey of all the project pages.

When you then click the **Pencil**, you get the option of editing the page titles.

OBJECT LISTER FOR SELECTED SYMBOL

If you select a symbol when you are working in the project — for instance **-F1** — and press <F7>, you will get a survey of all the places **-F1** is used in the project.

Alternatively you can right-click on a symbol, and choose **Show used**.

You then get the following survey of the symbol:

These functions are also available for signals and wire numbers.

Because the **Object Lister** remains on the screen, you can subsequently select another symbol in the project, and click the **Update** button in the **Object Lister**. This updates the **Object Lister** with information from the component, you have selected now.

When you want the list on print, click on **Print**. Click **Save** to save it as a file.

Click on the **Pencil** to correct the component data, or click the **DB** button to find the symbol in the database.

TEXT LINKS IN THE OBJECT LISTER

Information about linked text is also displayed in the **Object Lister**.

— See "Text links" on page 147.

FIND AND REPLACE

To find objects in projects quickly, you either use the *Object Lister* — see "The Object Lister" on page 351 — 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

 — Other ways to find *symbols* quickly are described in "Functions on Component Level" on page 259.

FINDING SYMBOL TEXTS

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

Click here to see a list containing project texts of the chosen Text type
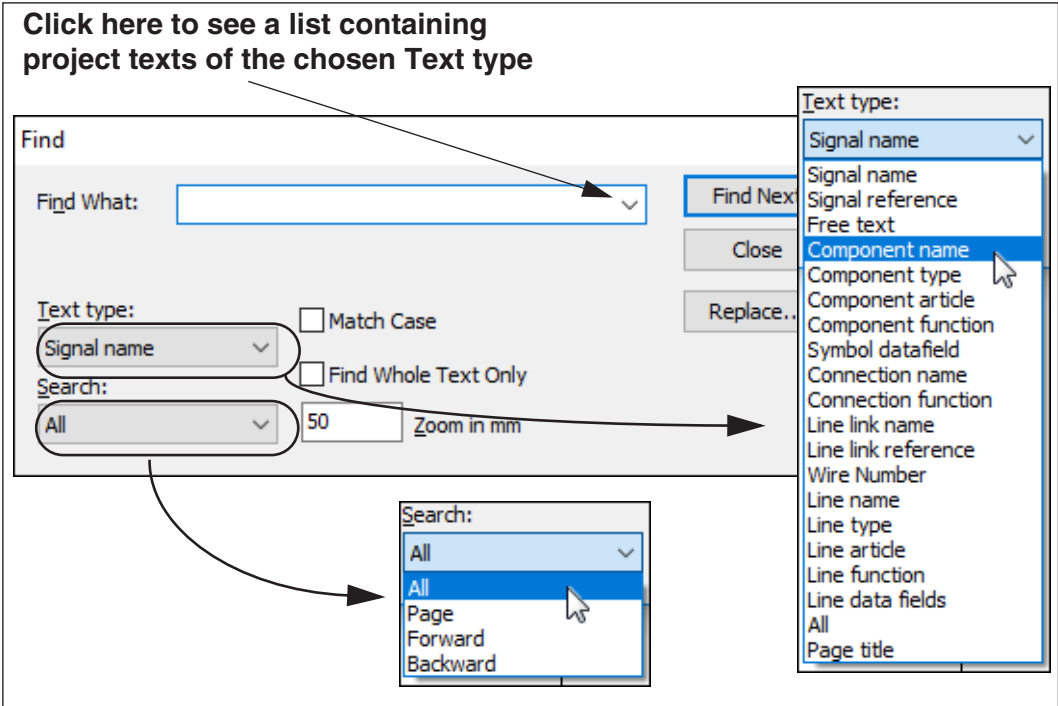
Here you type in the type of text you want to find, and state which type of symbol text you want to find, and in which direction you want the search carried out. The search direction determines whether you search the full project, the page you are working on, or in the direction forwards or backwards. — 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.

Found text

Find (=W1+A1-Q1)

Find What: -Q1
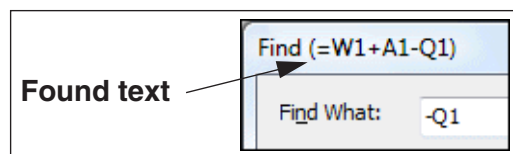
Text recognition

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:

Click here to see a list containing project texts of the chosen Text type
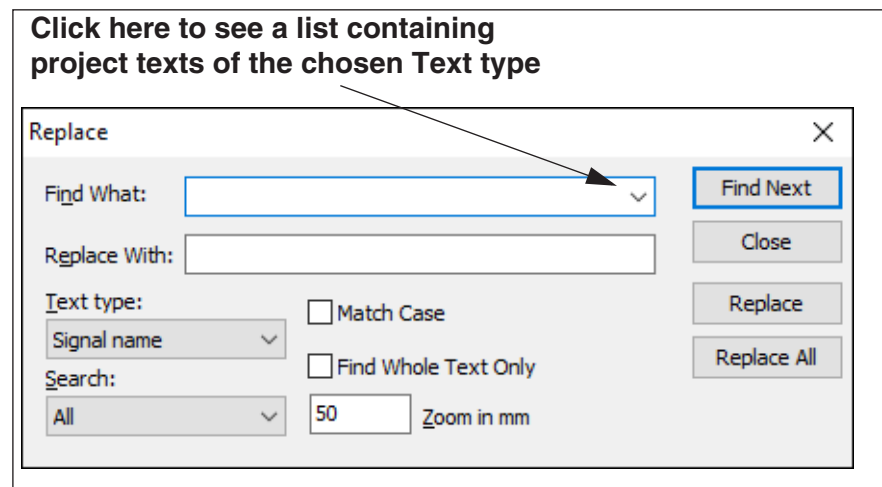
You specify what you want to *find* as explained above, and in the **Replace with** field you type in the text you want to replace.

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.
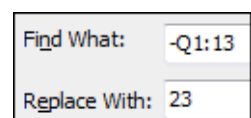
Find and Replace in selected area

This is how to use **Find** or **Replace** in selected areas:

1) Choose the **Texts** command

2) Use the mouse to select the area

3) Choose **Edit => Find** or **Edit => Replace**

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 918.

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**.
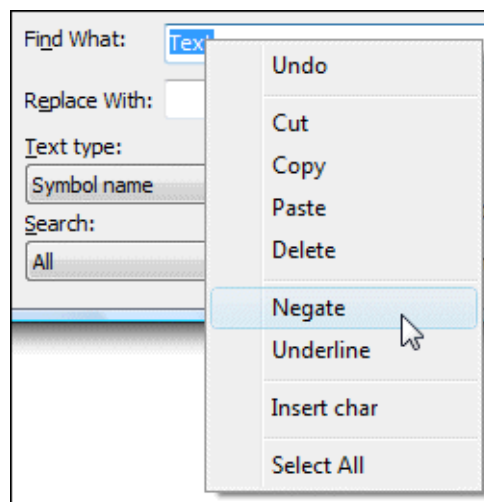
CHANGING FONTS AND NEGATION
PLEASE NOTE, THAT IF YOU CHANGE FONT, THIS CAN HAVE AN EFFECT ON THE NEGATION OF TEXTS. — SEE "SPECIAL CHARACTERS WHEN CHANGING THE PCSCHEMATIC FONT TO A WINDOWS FONT" ON PAGE 137.

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.

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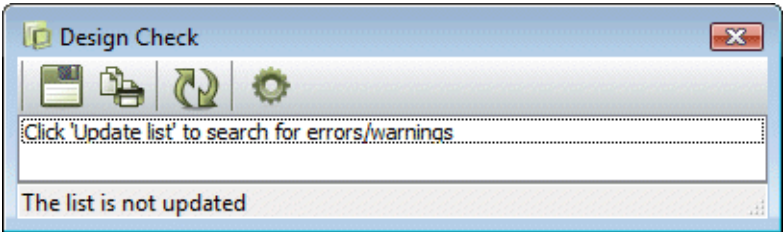
2) In the **Design Check** dialog box: Click the **Design Check settings** button.

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

Select what to check for

Here you select whether to execute the design check online (ie while drawing)

Activate Online check here

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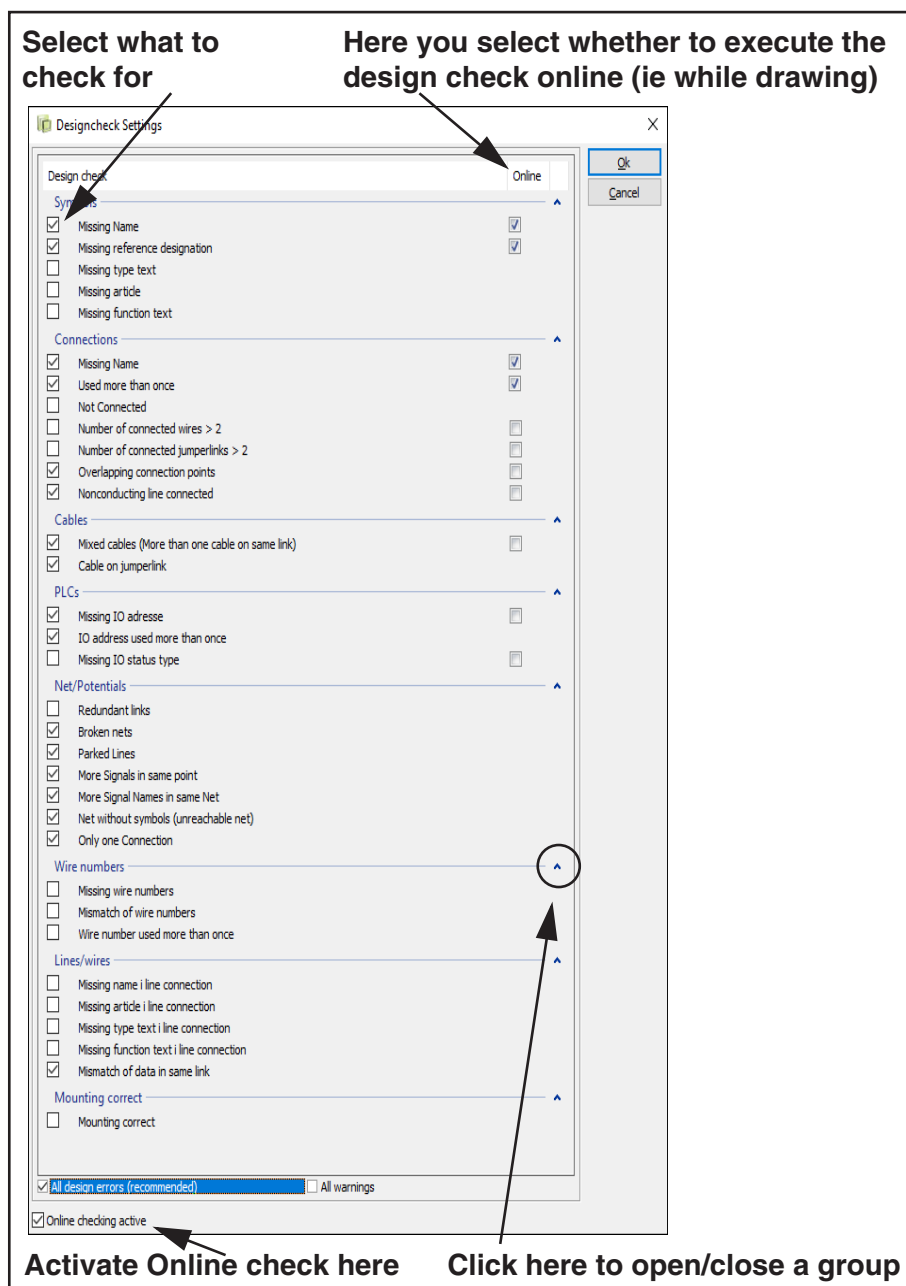
4) Select which errors to check the project for. To maintain the overview, it may be a good idea just to check the project for one type of errors at a time.
   If you select *All design errors (recommended)* at the bottom of the dialog box, all checks for design errors are selected in the dialog box.

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 OPTI-
> MIZED. 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 set-tings" on page 369.

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 368. 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**.
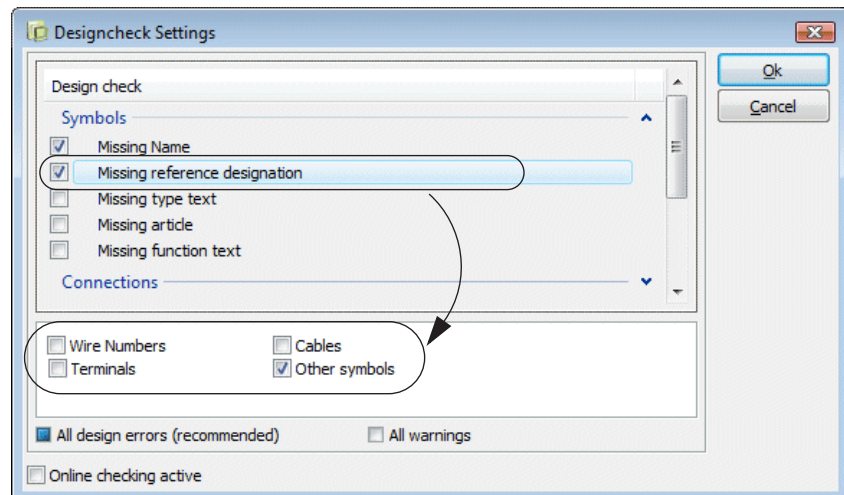
THE DESIGN CHECK DIALOG BOX

When you choose **Functions => Design Check**, and click **OK** to execute a design check, you enter the dialog box **Design Check**:
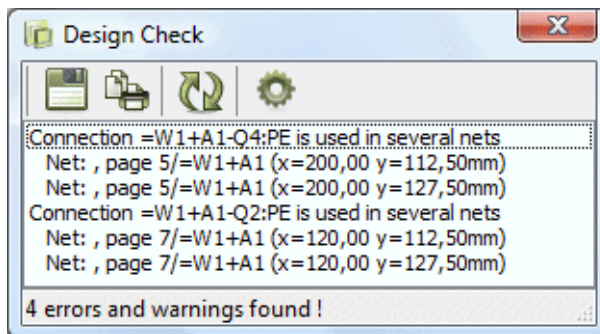
This dialog box remains on screen while you are working in the project. — The dialog box can be set to be transparent, as described in "Pointer/Screen" on page 918.

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 — particularly for pneumatics symbols — is allowed to have more connection points with the same name.

— Read more about this in "No double check" on page 781.

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 *WITHOUT* ACTUALLY CONNECTING. GRAPHICALLY THEY *APPEAR TO BE* 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.
	ILLEGAL CONNECTIONS (NON-CONDUCTING/JUMPERS)	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. IT IS ALSO CHECKED THAT JUMPER CONNECTION POINTS ON TERMINALS ARE CONNECTED USING JUMPER LINES.

CABLES	MIXED CABLES (MORE THAN ONE CABLE ON SAME LINK)	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/ POTEN- TIALS	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 VER- SIONS.
	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 SYM- BOLS (UNREACHABLE NET)	BOXES AND THE LIKE DRAWN WITH CONDUCTING LINES. THESE WILL BE CONSIDERED AS NETS BY THE PRO- GRAM, BUT THEY WILL BE UNREACHABLE.
	ONLY ONE CONNEC- TION	A NET WITH ONLY ONE CONNECTION CAN NOT HAVE ELECTRICAL FUNCTIONS.
WIRE NUM- BERS	MISSING WIRE NUM- BERS	CONNECTION WITHOUT WIRE NUMBER.
	MISMATCH OF WIRE NUMBERS	DIFFERENT WIRE NUMBERS APPEARS ON THE SAME CONNECTION.
	WIRE NUMBERS USED MORE THAN ONCE	THE SAME WIRE NUMBER APPEARS ON DIFFERENT CON- NECTIONS.
LINES/ WIRES	MISSING NAME IN LINE CONNECTION	CHECK FOR LINES WITHOUT SPECIFIED NAME.
	MISSING ARTICLE IN LINE CONNECTION	CHECK FOR LINES WITHOUT SPECIFIED ARTICLE NUM- BER.
	MISSING TYPE IN LINE CONNECTION	CHECK FOR LINES WITHOUT SPECIFIED TYPE.
	MISSING FUNCTION TEXT IN LINE CON- NECTION	CHECK FOR LINES WITHOUT SPECIFIED FUNCTION TEXT.
	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 SEG- MENTS 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 e.t.c.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— OPENING THE PRINT DIALOG BOX</td><td>(P. 371)</td></tr>
<tr><td>— THE RIGHT-HAND SIDE OF THE DIALOG BOX</td><td>(P. 372)</td></tr>
<tr><td>— SELECT PAGES</td><td>(P. 372)</td></tr>
<tr><td>— SETUP</td><td>(P. 374)</td></tr>
<tr><td>— CHOOSE PRINTER</td><td>(P. 376)</td></tr>
<tr><td>— BATCH PRINTING</td><td>(P. 376)</td></tr>
<tr><td>— IF DIAGRAMS ARE MINIMIZED WHEN PRINTING</td><td>(P. 376)</td></tr>
<tr><td>— EXPORTING PDF FILES</td><td>(P. 376)</td></tr>
<tr><td>— EXPORTING DWG/DXF FILES</td><td>(P. 376)</td></tr>
</table>

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>.
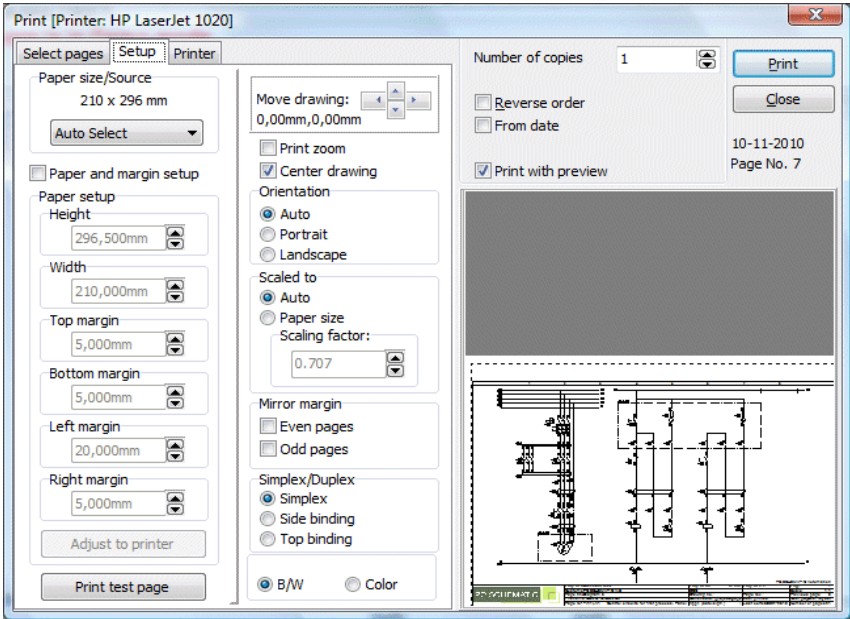
THE RIGHT-HAND SIDE OF THE PRINTER DIALOG BOX

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 892 — 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.
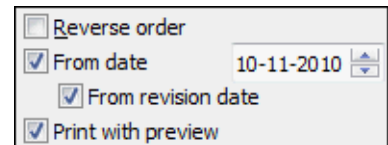
— See "Printing during revision control" on page 893 also.

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.
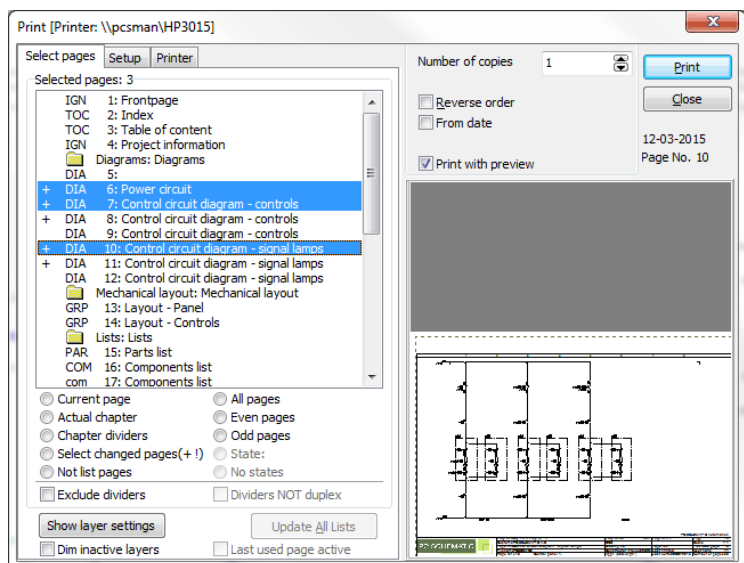
Pages marked with a preceding **!** character, have been changed since the project was opened.

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

When you select ***Select changed pages(+ !)***, all pages that have either changed since the project was last printed, or since the project was last saved, become selected.

If states are selected on project pages, the names of these states are displayed in brackets after the page title.

Last used page active

When a project page is marked as "Last used page", a checkmark will be set in the ***Last used page active*** checkbox at the bottom of the dialog box. This way you can for instance print all project pages until the page which is marked as the last used page. When you wish to print all project pages, just remove the checkmark in the ***Last used page active*** checkbox.

— To mark the active project page as the last used page choose **Settings => Page data** and select ***Last used page***.

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.
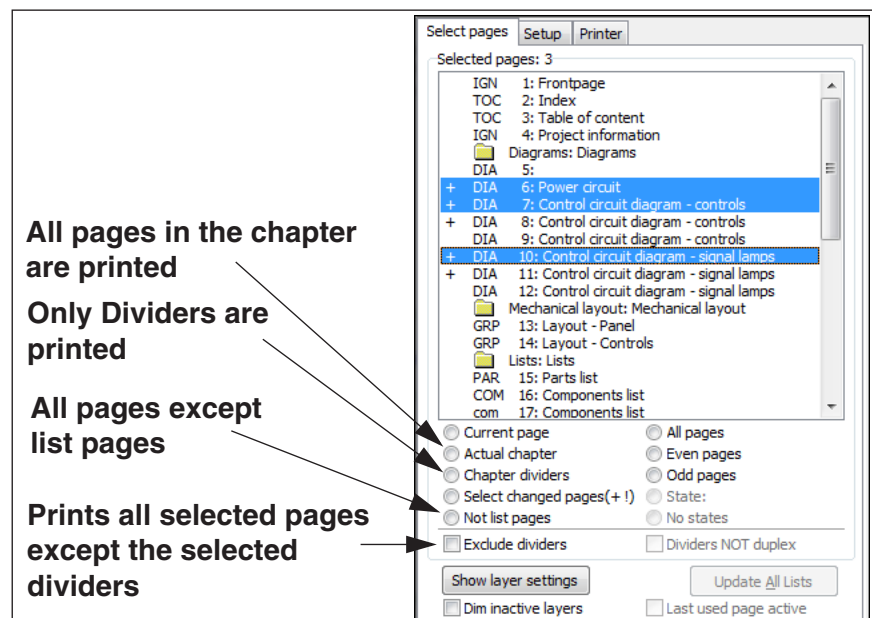
When you are printing on a duplex printer, you can also specify not to print dividers on duplex, by clicking on ***Dividers NOT duplex***. This check box is dimmed if the selected printer can not handle duplex.

Printing pages with states

When pages in the active project are using states, you can select all project pages with a specific state selected, by doing this:

1) Click on **State** in the lower-right part of the dialog box

2) You now get the option of choosing any of the applied states in the project: Choose a state

3) All project pages using the chosen state are now selected. The name of the state is also displayed in the **State** field, and the selected state is listed in brackets after the page titles in the dialog box

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 545.

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 542.

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.

Under **Settings => Special settings** you can choose to display a reminder to remember to update the project lists before printing. — See "Special settings" on page 940, in particular "Printer and PDF dialog" on page 941.

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.
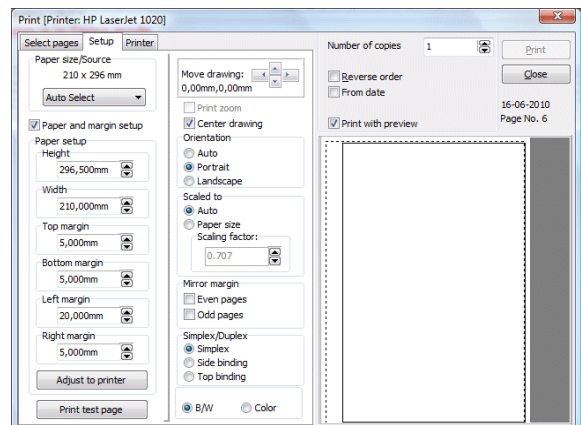
If you want a margin with room for holes in the paper, you must set the left margin to 20mm.

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.

Move drawing:	◀ ▲ ▶
0,00mm,0,00mm	▼
☑ Print zoom	
☐ Center drawing	

Click here to move the chosen zoom

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

Print test page

If you are having difficulties printing the drawing on your printer, click **Print test page**. This will make the printer print out a test page, displaying the printable area of your printer, drawn with a thin line along the edges. The right, left, top and bottom margins will also be printed in figures. These are given using one thousandth of a millimetre, making *5865um* equal to *5.865mm*. These settings are specified in your printer driver, and therefore are not changeable from the **Automation** program. If the test print does not include all four lines, you have not chosen the correct printer driver for your printer.

A biased left or right margin, suggests that the printer is not feeding the paper correctly. If the printable area is biased towards the left- or the right hand side, placing the paper differently in the tray might help.

If you are having further difficulties, please contact your local supplier.

CHOOSE PRINTER

The last option is the **Printer** tab, 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 PCSCHEMATIC 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 379.

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*.
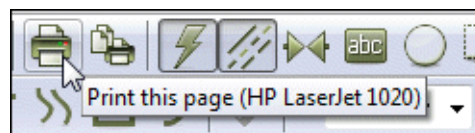
BATCH PRINTING

Read more about this in "Batch printing" on page 902.

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 741 and "The Zoom All function" on page 43.

EXPORTING PDF FILES

— See "Intelligent PDF files" on page 377.

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 741.

INTELLIGENT PDF FILES

In **Automation** you can export your projects as PDF files, where links for instance are generated between symbols for the same component on the diagram pages.

EXPORT A PDF FILE

You can start the export of the active project as a PDF file in three different ways:

   a) Choose **File => Export to => PDF**, or

   b) Choose **File => Save as** and choose ***PDF files (*.pdf)*** below ***File type*** at the
       bottom of the dialog box, or

   c) Press <e>

This opens the **Setup** tab in the dialog box **PDF export**:

**Specify the reso-
lution in the PDF file**

**Select which layer to print
via the Layer menu**

**Page size
in the PDF file**

**Select which
pges to export**

**Update all lists
before printing**
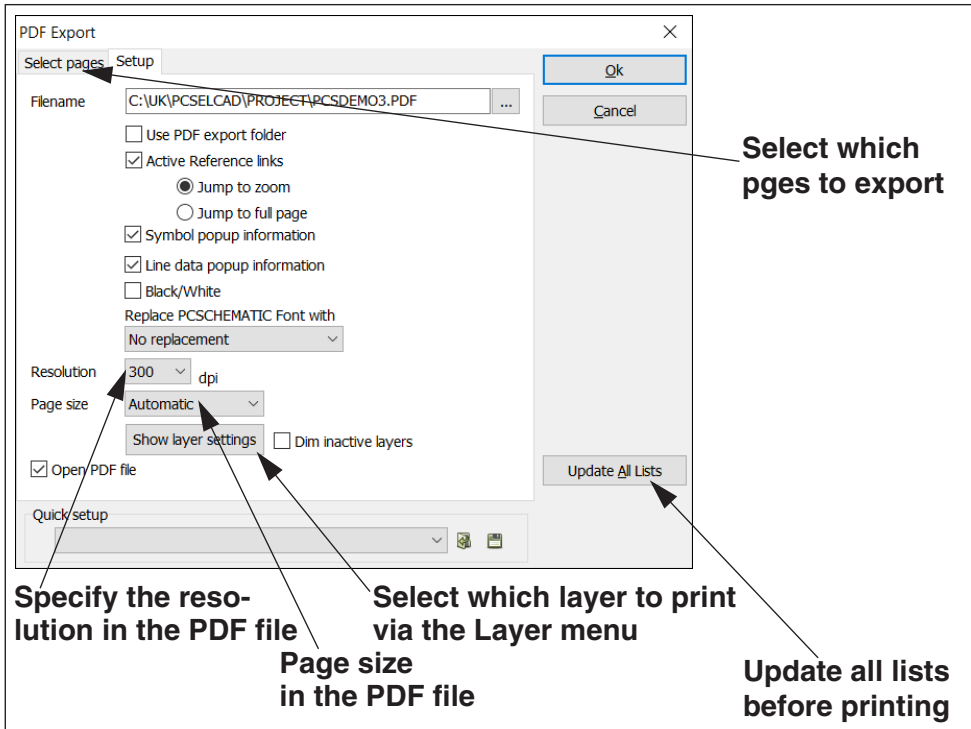
On the **Setup** tab you have the following options:

Use PDF export folder

When you select **Use PDF export folder**, the PDF file is saved in the folder speci-
fied for *Directories* for PDF files. Read more about this in "Directories" on
page 923. 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.

Furthermore, you can specify how to zoom when clicking on a link in the PDF file:

- **Jump to zoom**: The zoom is on the selected symbol on the page.

- **Jump to full page**: The full page, where the symbol is placed, is displayed.

Symbol popup information

When you select **Symbol project information**, the generated PDF file will contain
popup information for the symbols in the project pages.

Line data popup information

When **Line data popup information** is selected, the PDF file will contain popup
information for line data on the project pages.

Quick setup

At the bottom of the **Setup** tab in the dialog box **PDF export** you can save the
selected settings by clicking the **Save** button.

If quick settings are already created, you can select these by clicking on them in
the list at the bottom of the tab or by clicking on the **Open** button.

Quick-settings for command line files

When you have created a set of quick-settings for PDF export, these can also be
applied, if you create PDF files via command line files.

— See "Command line files for creating PDF files via quick-settings" on page 902.

Fonts when creating PDF files

When exporting/printing a project as a PDF file, all **PCSCHEMATIC** fonts will be
turned into graphics. Due to the search facilities in the **Acrobat** program, it is pos-
sible 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 accord-
ing to the standards*.

PC|SCHEMATIC

APPLYING OTHER FONTS WHEN PRINTING/EXPORTING PDF FILES

When printing/exporting PDF files, it is possible to substitute the **PCSCHEMATIC** 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 **PCSCHEMATIC** 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 **PCSCHEMATIC** font.

To do this, you must select the *Arial* font in the **Alternative PCSCHEMATIC font** field. You also have the option of choosing other fonts.

SPECIAL CHARACTERS IN PDF FILES

If you change texts written with the **PCSCHEMATIC** 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 PCSCHEMATIC font to a Windows font" on page 137.

> **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 **PCSCHEMATIC** FONT CONTAINS THE NECESSARY SPECIAL CHARACTERS.

Specifying which pages to include in the PDF file

When you click the **Select pages** tab in the dialog box **PDF Export**, you can select which pages to include in the PDF file:
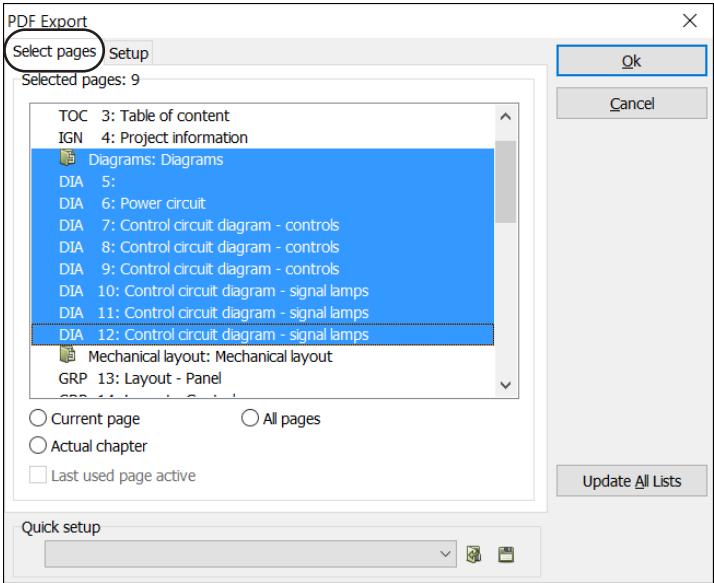
This is done as in the Print dialog box, which is described in "Select pages" on page 372.

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 901.

WINDOWS EXPLORER AS SYMBOL/PROJECT MENU

THIS CHAPTER DESCRIBES
— USING THE WINDOWS EXPLORER AS A PROJECT MENU .... (P. 381)   — USING THE WINDOWS EXPLORER AS A SYMBOL MENU............................. (P. 381)

USING THE WINDOWS EXPLORER AS A PROJECT MENU

In **Automation** it is possible to use **Windows Explorer** as a kind of *Project menu*.

When you have opened Windows Explorer, you can therefore *drag* project files from the Explorer to the **Automation** program, whereby the projects are opened.

When you select several projects at a time, all of the selected projects will be opened in **Automation**:
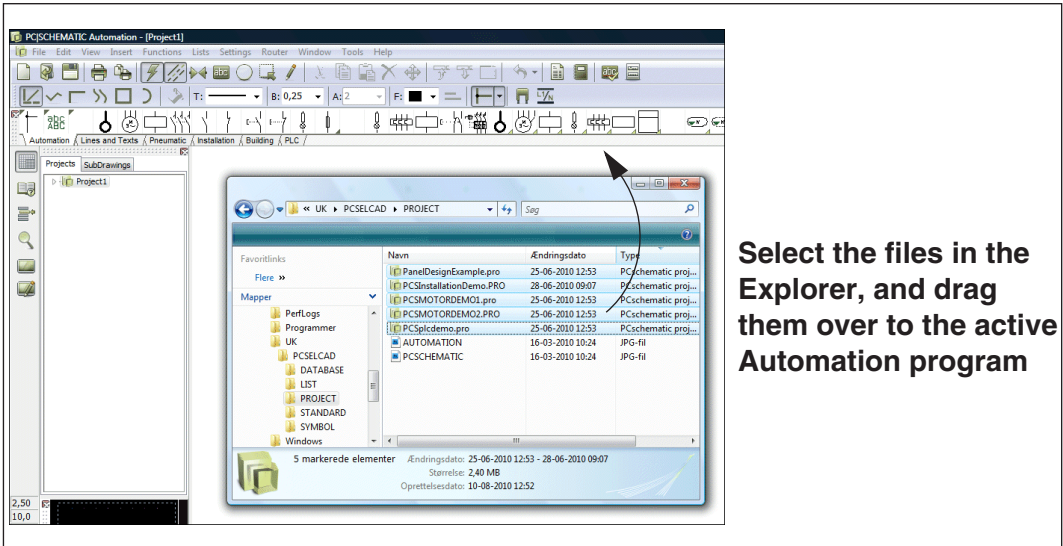
Select the files in the Explorer, and drag them over to the active Automation program

USING THE WINDOWS EXPLORER AS A SYMBOL MENU

In **Automation** it is possible to apply **Windows explorer** as a kind of *Symbol menu*.

As described for projects, you can apply the Windows Explorer to drag *symbols* into an open project. Hereby you get the symbol directly in the cross hairs in **Automation**. You can therefore apply the Windows Explorer as an alternative symbol menu, which you can keep open while working with **Automation**.

You can only fetch one symbol at the time from the Windows Explorer.

If you open two copies of Windows Explorer, you can apply the one to open projects from, and the other to fetch symbols from.

REFERENCE DESIGNATIONS

In PCSCHEMATIC Automation you can apply reference designations in the naming of your symbols/components. Reference designations can be defined freely, and are not tied to any specific standard, but the program makes it easy to follow the ISO/IEC/EN 81346 standard.

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 PCSCHEMATIC A/S.

PC | SCHEMATIC

ON REFERENCE DESIGNATIONS

Reference designations are used in **Automation** for naming objects — this being symbols/components and groups of these.

Reference designations are constructed hierarchically, which makes it easy to understand how an object with an assigned reference designation is part of a larger picture.

Reference designations for a component are placed next to its symbol(s) in the diagrams, and on / next to the physical component, where it is placed/mounted.

Automation supports reference designations according to the ISO/IEC/EN 81346 standard, but is also open for using reference designations in other ways. It is therefore both possible to use an internal company standard and to follow changing international standards.

ADVANTAGES WHEN USING REFERENCE DESIGNATIONS

When you use reference designations, you for instance get the following advantages:

- You get a clearly structured documentation, where it is easy to identify the individual objects and which contexts they are parts of (constituents of).

- You make it easier for different professions to talk about the same issues seen from their individual perspectives. — See "Aspects" on page 386.

- It makes it easier for different subcontractors to contribute to the same project / the same documentation.

- It makes it easier to reuse the structure from other parts of the documentation.

- It makes it easier to generate documentation with selected/deselected options. — See "Projects with selectable options" on page 592.

- If for instance you use reference designations for describing a full production plant, you can subsequently easily generate a list containing all products, another containing all functions and yet another containing all positions.

Before defining the reference designations, you should have an overview over which functions as well as which switchboards/racks and other positions you will need in your project.

ASPECTS

A reference designation in **Automation** can contain up to three different aspects:

- The function aspect
- The position aspect
- The product aspect

You can understand an aspect as "a pair of glasses" that you can put on, when you look at the objects (symbols e.t.c.) in a project from a specific perspective. When you put on another pair of glasses (use another aspect), you look at the project in another way.

Depending on which aspect you use, you can therefore look at the objects as *functions* (the function aspect), as *positions* (the position aspect) or as *products* (the product aspect).

Aspects and designations

It is important to be aware that you use the term *aspect* when you talk about different ways that you can look at the documentation / the project from an overview position. However, when you talk about what an aspect *contains* for a specific object, you use the term *designation* instead.

If for instance it says -P1 in the product aspect for a component, you say that the component has the *product designation* -P1.

Separate aspects or Combined aspects

When you use reference designations for identifying objects, you have to choose whether to work with the aspects as *separate aspects* or as *combined aspects*.

Separate aspects (Recommended)

You can choose that *each individual aspect identifies an object*. This is called *separate aspects*. This way you can let the function aspect, the position aspect and the product aspect *each* identify the objects.

This can give you the following advantages:

- Different involved parties can talk about the same object from their individual understanding, and they can all identify the objects from their own perspective. When some talk about *functions* others can talk about *positions* and others again about *products* — and they can all identify what they are talking about, from their own perspective.

- Separate aspects typically makes it easier to get an overview over complex assignments. Here you can for instance just look at the functions or products, depending on where your focus is.

- See an example in "Example: Separate aspects" on page 392.

Combined aspects

You can also choose to *identify an object by combining aspects*, which is called *combined aspects*. Here it is for instance the function aspect and the product aspect which *in combination* identifies the object.

— See an example in "Example: Combined aspects" on page 391.

ARE COMBINED ASPECTS MAKING IT UNNECESSARILY COMPLICATED?
PLEASE BE AWARE THAT YOU CAN END UP MAKING IT MORE COMPLICATED FOR YOURSELF, IF YOU IN THIS WAY MAKES THE ASPECTS DEPEND ON EACH OTHER. FOR IS IT ALWAYS COMPLETELY CLEAR WHICH ASPECT DEPENDS ON WHICH? IS IT FOR INSTANCE THE POSITION THAT DEPENDS ON THE FUNCTION — OR IS IT THE OTHER WAY AROUND?

PREFIXES

Each individual aspect have an assigned *prefix*, which is a character that you place in front of each aspect, that makes it easy to identify the aspect:

- The standard prefix for the Function aspect is: **=**

- The standard prefix for the Location aspect is: **+**

- The standard prefix for the Product aspect is: **-**

ASPECT LEVELS

A reference designation for an aspect contains of a prefix, which typically is placed in front of a capital letter, which is followed by an integer. For instance **-M01**.

Because reference designations are constructed hierarchically, each aspect can contain multiple levels. These levels are typically specified by a separator, but it is also allowed according to the standards to omit a separator. According to the ISO/IEC/EN 81346 standard, the following options are allowed:

SEPARATOR BETWEEN LEVELS IN ASPECTS	EXAMPLE
PREFIX AS SEPARATOR	-A1-B1-C1
DOT AS SEPARATOR	-A1.B1.C1
SPACE AS SEPARATOR	-A1 B1 C1
NO SEPARATOR	-A1B1C1

Please note that the option **No separator** above requires that the individual levels consists of a letter followed by an integer.

REFERENCE DESIGNATIONS ON ONE OR MORE LINES

When you display different aspects in the reference designation on individual lines, this means, according to the standards, that each individual line identifies the object (separate objects).

According to the standards the character **/** can also be used for specifying line breaks.

This gives the following options for representing reference designations:

THE SET OF REFERENCE DESIGNATIONS CONTAINS	REFERENCE DESIGNATIONS ON INDIVIDUAL LINES	REFERENCE DESIGNATIONS IN A SINGLE LINE
THREE REFERENCE DESIGNATIONS (SEPARATE ASPECTS)	=A2 -B2 +C3	=A1/-B2/+C3
A SINGLE REFERENCE DESIGNATION (COMBINED ASPECTS)	=A1-B2+C3	=A1-B2+C3
TWO REFERENCE DESIGNATIONS (HYBRID)	=A1-B2 +C3	=A1-B2/+C3

Please note that quite a few projects created in version 17 and previous versions will be of the type *A single reference designation (combined aspects)*.

We now recommend to use separate aspects, which also is the default setting for **Automation**. — See "Separate aspects or Combined aspects" on page 386.

CHARACTER SPECIFYING FULL REFERENCE DESIGNATION

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 unambiguously identifies the component.

Please note, that this character can therefore also appear when no reference designation has been specified for a component.

REFERENCE DESIGNATIONS ON PAGES AND REFERENCE FRAMES

In **Automation** you can also specify reference designations for pages and for areas on individual pages (reference frames).

Reference frames on pages

It is possible to specify that all symbols on a project page e.g. belongs to Pumping1 (**=M1**) and are located in Hall2 (**+R2**).

If a symbol has the reference designation **=M1/+R2/-K5**, and you have specified the reference designation for the page as **=M1/+R2**, only the remaining part of the reference designation — this being **-K5** — will be displayed next to the symbol.

If a symbol does not have the same reference designation as the page, **Automation** automatically displays those of the designations that deviate from the reference designations of the page, preceded by the **>** character to specify that this is the full reference designation for the symbol.

Reference designations for areas (reference frames)

You can create *reference frames* on project pages, which can make it easier to get an overview over the circuit diagrams. A reference frame specifies that all objects placed inside the reference frame have a specific reference designation.

It is also possible to specify other reference designations for individual objects inside the reference frame, as shown in "Example: Reference designation with three separate aspects" on page 390.

Example: Reference designations with a single aspect

Here you see the lamp **-S1.P3** and the relay coil **-S1.K7**:
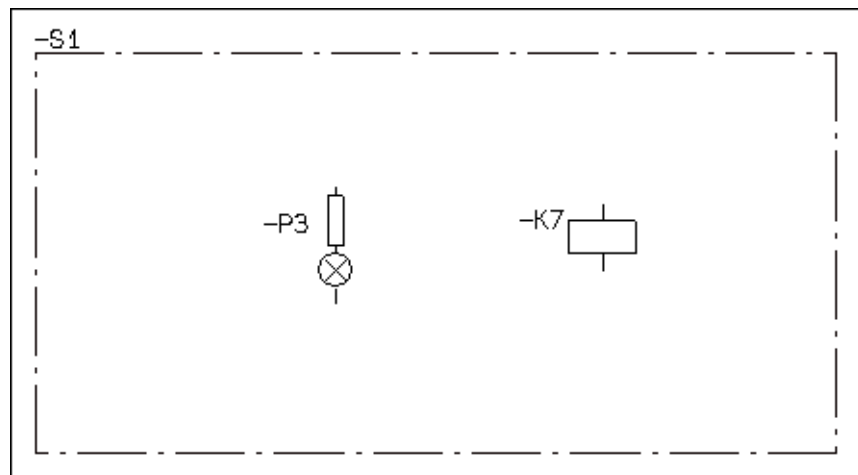
For both symbols inside the reference frame, you can see that the reference frame specifies the first level in the product aspect (**-S1**), and that the dot is the separator between the levels.

Example: Reference designation with two combined aspects

Here you see the lamp **+S1-P2** and the relay coil **+S1-K6**:
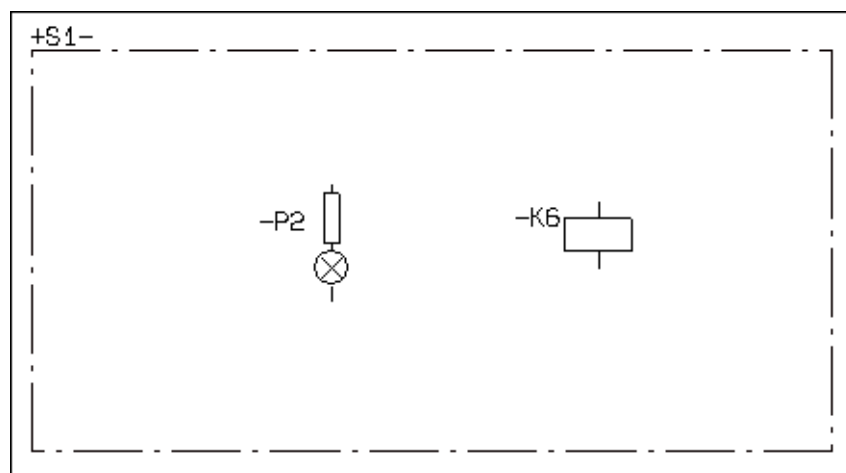
In this case we are working with *combined aspects*. Therefore a **-** character is placed after the reference designation for the reference frame (a so-called "trailing prefix"). This specifies that the reference designation for the reference frame is to

be put together with the designation for the symbol, even though it seems to be part of another aspect.

Example: Reference designation with three separate aspects

Here you see the relay coil **=A1.C1/+C3.B3/-B2.S1.K2** and the lamp **=A2/+C3.C5/-P4**:
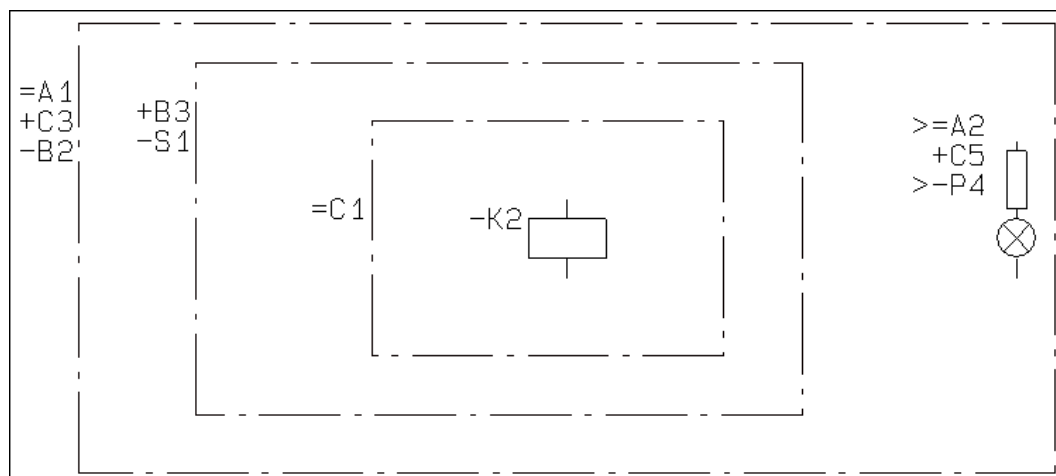
In this case we are working with *separate aspects.*

Please note the following:

- The relay coil is assigned the designations from all of the surrounding reference frames. (The aspects are read individually).

- The **>** character in front of the function designation and the product designation for the lamp specifies that this is the *full* designation for these aspects. The location designation does not contain this character, because the location aspect is **+C3.C5** (where **+C3** comes from the outermost reference frame).

Please note that the outermost reference frame in this example could just as well have been the reference designation specified for the project page.

EXAMPLE: COMBINED ASPECTS

Below you can see an example of *combined aspects*. Read more about combined aspects in "Separate aspects or Combined aspects" på side 386.

For combined aspects:

For a symbol you can see the following reference designation:

$$=M1+R2-K5$$

**Function designation
(function aspect)**

**Location designation
(location aspect)**

**Component name
/ Product designation
(product aspect)**
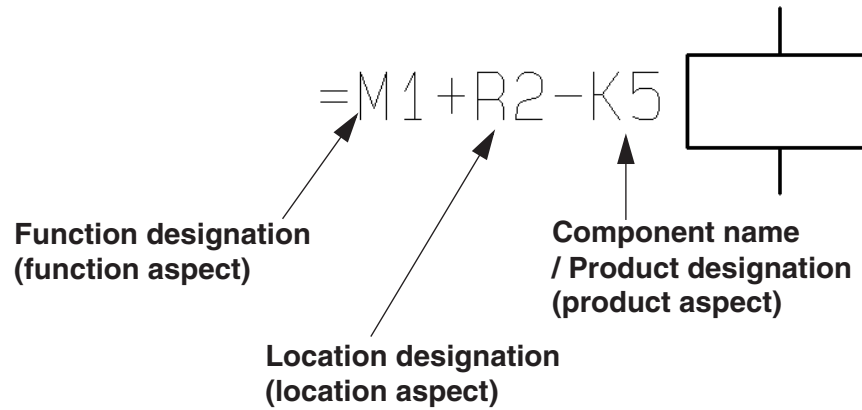
The reference designation =M1+R2-K5 is interpreted like this:
The product/component -K5 is part of the location +R2, which is a part of the function =M1.
These three aspects in combination identifies the component,
because this is combined aspects.

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** could for instance be "Hall2".

The product designation **-K5** is used to identify the component name itself.

PRODUCT DESIGNATION / SYMBOL NAME

INSTEAD OF THE *PRODUCT DESIGNATION* FOR A SYMBOL, THE TERM *SYMBOL NAME* IS USED IN THE REST OF THIS MANUAL.

EXAMPLE: SEPARATE ASPECTS

Below you can see an example of *separate aspects*. Read more about separate aspects in "Separate aspects or Combined aspects" på side 386.

For separate aspects:

For a symbol you can see the following reference designation:

Function designation
(function aspect) ➝ =M 1

Location designation ➝ +R2
(location aspect)

Component name ➝ -K5
/ Product designation
(product aspect)

The reference designation *=M1/+R2/-K5* is interpreted like this:
The product/component *-K5* is placed in the location *+R2*
and is attached to the function *=M1*.
These three aspects each identify the component,
because they are separate aspects.

Please note that the character **/** for specifying line breaks will be used for reference designations in lists.

DEFINING PROJECT REFERENCE DESIGNATIONS

To create reference designations in your project, you either click the **Reference designations** button or choose **Settings => Project data** and click **Reference designations**.
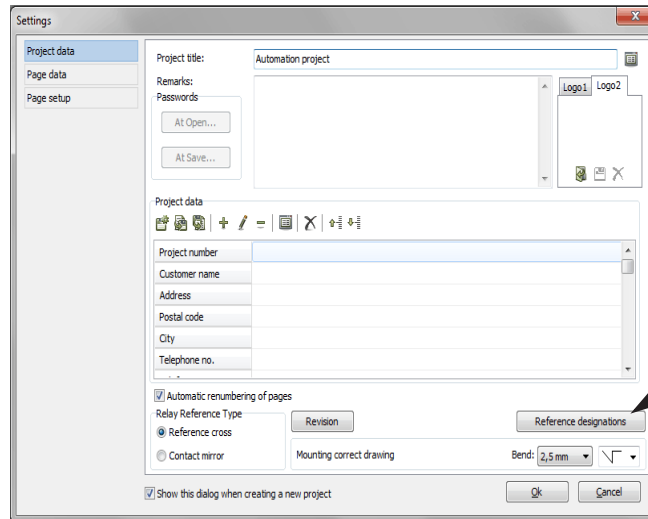
You then enter the dialog box **Reference designations**:
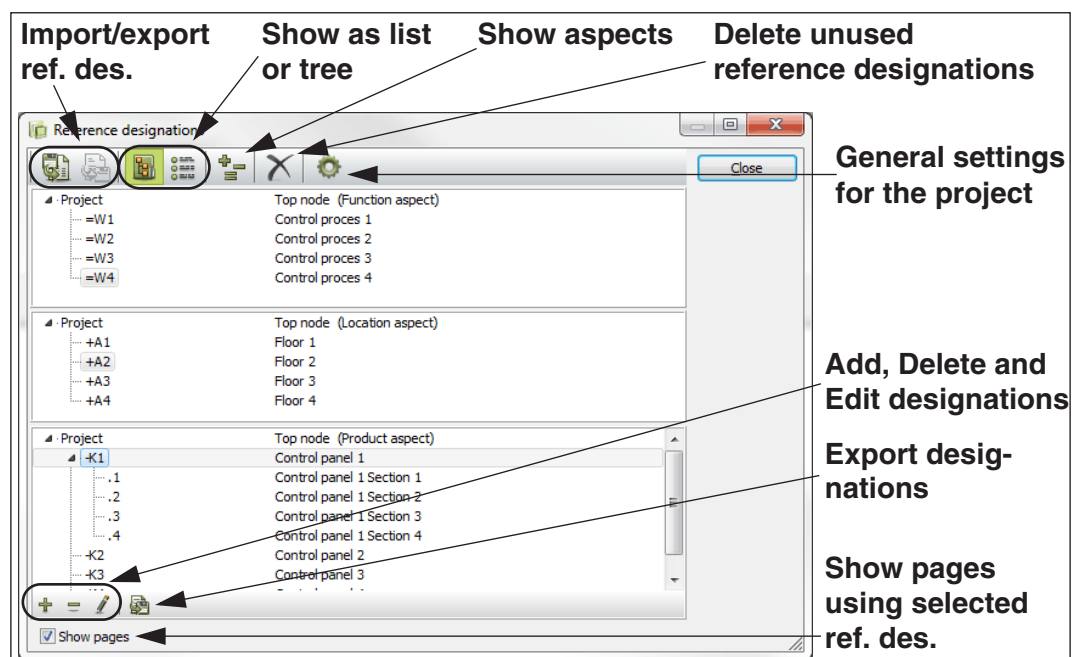
In the dialog box **Reference designations** you can create and edit the **Function**, **Location** and **Product** aspects for the reference designations.
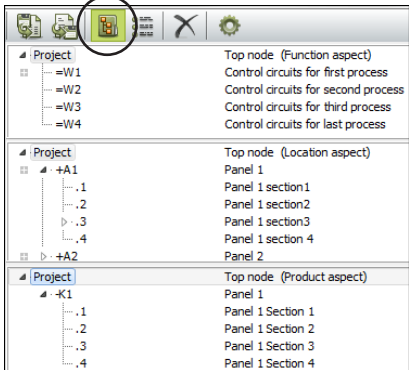
— Please note that reference designations can also be imported into the project and exported from the project, as described in "Import / export of reference designations" on page 408.

The first time you enter the dialog box, no reference designations will be specified.
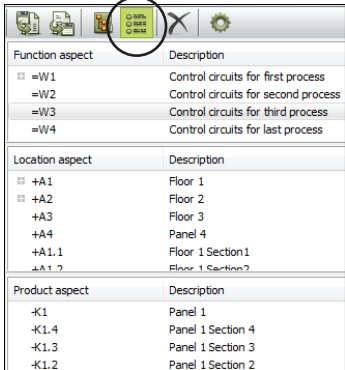
OPTIONS IN THE DIALOG BOX REFERENCE DESIGNATIONS

DISPLAYING REFERENCE DESIGNATIONS IN THE DIALOG BOX REFERENCE DESIGNATIONS

In the upper-left part of the dialog box **Reference designations** you can specify how to display the reference designations in the dialog box:

Hierarchial display of reference designations

List display of reference designations

Display reference designations as hierarchy / Tree

When you click the **Show as tree** button, the reference designations are displayed as a hierarchical structure — see above.

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 e.t.c.

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 above.

Please note that it is not possible to edit reference designations in list view.

ADDING NEW REFERENCE DESIGNATIONS

This is how to add a new function reference designation in the *Function aspect* section in the dialog box **Reference designations**. (Location and Product reference designations are added accordingly in the *Location aspect* and *Product aspect* section):

1) Click on the function reference designation that you wish to create a new function reference designation *under*. If no function reference designations has been created yet, just click inside the *Function aspect* section.

2) Click the **New** button at the bottom of the section — or press <Insert>.

3) In the dialog box **New**: Type in a new function designation, a *Description* for the function designation and a *Description 2* (which for instance can contain a description in another language or a number referring to an ERP system). In the dialog box the **=** character is suggested as the prefix for the reference designation, but you are free to use another prefix. (See "Settings according to standards" on page 402 for non-standard prefixes).
If a function reference designation is selected in the dialog box, the dialog box **New** will contain the data from this reference designation. This way you will not need to type in the same initial part of the description text more than once.

4) Click **OK** when you have typed in the function aspect and its description.

IEC / EN STANDARD

THE CHARACTERS **=**, **+** AND **-** ARE IEC STANDARD PREFIXES FOR THE FUNCTION, LOCATION AND PRODUCT ASPECTS.

SEE USED REFERENCE DESIGNATIONS

The reference designations currently applied in the project are marked with an **Applied reference designation** icon next to the reference designation.

Function aspect	Description
=W1	Control circuits for first process
=W2	Control circuits for second process
=W3	Control circuits for third process

Delete all unused reference designations

If you wish to delete all unused reference designations in the project, click the **Delete unused** button at the top of the dialog box.

See on which page the reference designations are used

If you wish to see which pages a reference designation is used on, 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 (you can also double-click on the page):

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.

FOR LIST VIEW: 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 394.

In the dialog box **Reference designations** you can sort the reference designations alphabetically based on the columns **Function aspect**, **Location aspect** and **Position aspect** as well as **Description** for the reference designations.

This is done as described in "The sorting order of the data in the Object Lister" on page 354.

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 reference designation is placed just *below* the reference designation, that you point on when you release the mouse button.

If you *drag* a reference designation beyond the top or the bottom of the list, the list starts scrolling.

The last used reference designation will be memorized for the next time the dialog box is opened.

EDITING REFERENCE DESIGNATIONS

If you wish to change a reference designation, click on it and click the **Edit** button at the bottom window for the aspect — or click on the reference designation and press the <F2> key.

If you change a reference designation with underlying 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.

DESCRIPTIONS FOR UNDERLYING ASPECTS ARE NOT CHANGED!

PLEASE NOTE THAT ONLY *DESIGNATIONS* FOR UNDERLYING ASPECTS ARE CHANGED. IF FOR INSTANCE YOU CHANGE *+A1* TO *+A3* AND ALSO CHANGE THE *DESCRIPTION* FROM *FLOOR 1* TO *FLOOR 3*, THIS CHANGE IN THE DESCRIPTION IS NOT TRANSFERRED TO THE UNDERLYING ASPECTS, WHEN THE *+A1* ASPECTS ARE CHANGED TO *+A3*.

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 423.

Changing reference designations for Chapters

Changing reference designations for a full chapter, see "Editing reference designations for chapters" on page 423.

DELETING REFERENCE DESIGNATIONS

To delete a reference designation, click on it and click the **Delete** button at the bottom of the aspect window— or press the <Delete> key.

MOVE A REFERENCE DESIGNATION TO A NEW POSITION

To move a designation to a new position — for instance from being placed under **=W3** to be placed under **=W4** — do this:

1) In the dialog box **Reference designations**: Right-click on the designation and select **Cut**

2) Right-click on the designation, that you wish to place the designation under, and select **Insert**

The designation has now been moved.

If the designation has any sub designations, these are also included, when you move the designation. If the designation and its sub designations are used in the project, these are automatically renamed for all relevant objects in the project.

REFERENCE DESIGNATIONS IN THE PAGE MENU

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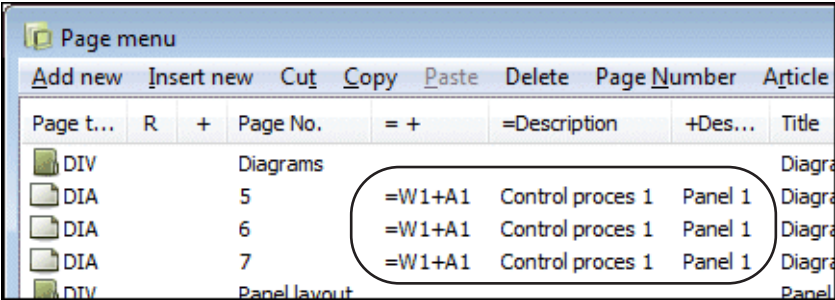
SETTINGS FOR REFERENCE DESIGNATIONS

When you click on the **Settings** button at the top of the dialog box **Reference designations**, you enter the dialog box **Reference designations - Settings**. In the dialog box you have the following tabs:

TABS IN THE DIALOG BOX "REFERENCE DESIGNATIONS - SETTINGS"
— FORMAT: FORMAT FOR REFERENCE DESIGNATIONS .... (P. 399)
— STANDARD: SETTINGS WITH RESPECT TO STANDARDS .... (P. 402)
— IMPORT: IMPORT SETTINGS FOR REFERENCE DESIGNATIONS .... (P. 405)
— GENERAL: GENERAL SETTINGS FOR REFERENCE DESIGNATIONS .... (P. 407)

FORMAT FOR REFERENCE DESIGNATIONS

On the **Format** tab in the dialog box **Reference designations - Settings** you specify the format for the reference designations in the project.

In the upper-right corner of the tab, you can see an illustration of the format that will be applied using the current settings.
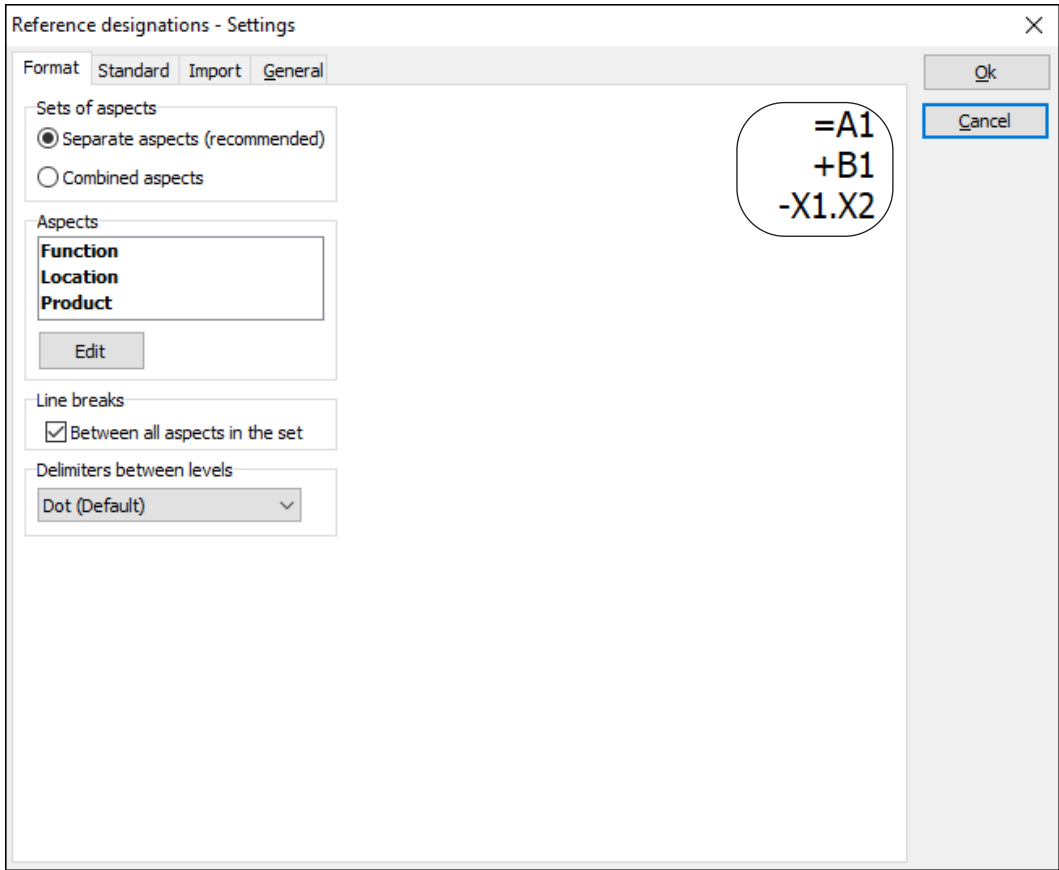
Sets of aspects

Under *Sets of aspects* you specify — as an overall setting — whether to work with **Separate aspects** or **Combined aspects** — see "Aspects" on page 386, particularly "Separate aspects or Combined aspects" on page 386.

The aspects are displayed differently
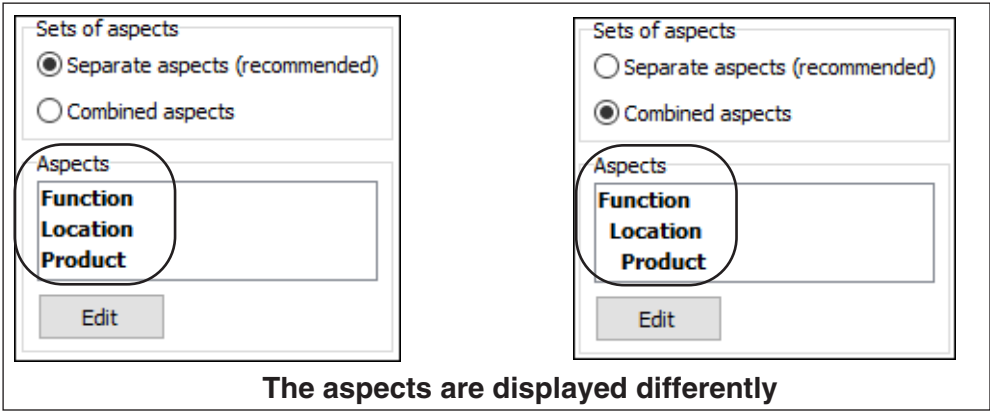
Please note that the aspects are displayed differently under **Aspects**, which clarifies that combined aspects are hierarchical, while separate aspects consists of more than one aspect.

Click **Edit** to select/deselect aspects or to change their order. This brings you into the dialog box **Edit aspects**, where you have the following options:

Move the selected aspect up/down   **Select/deselect aspects**
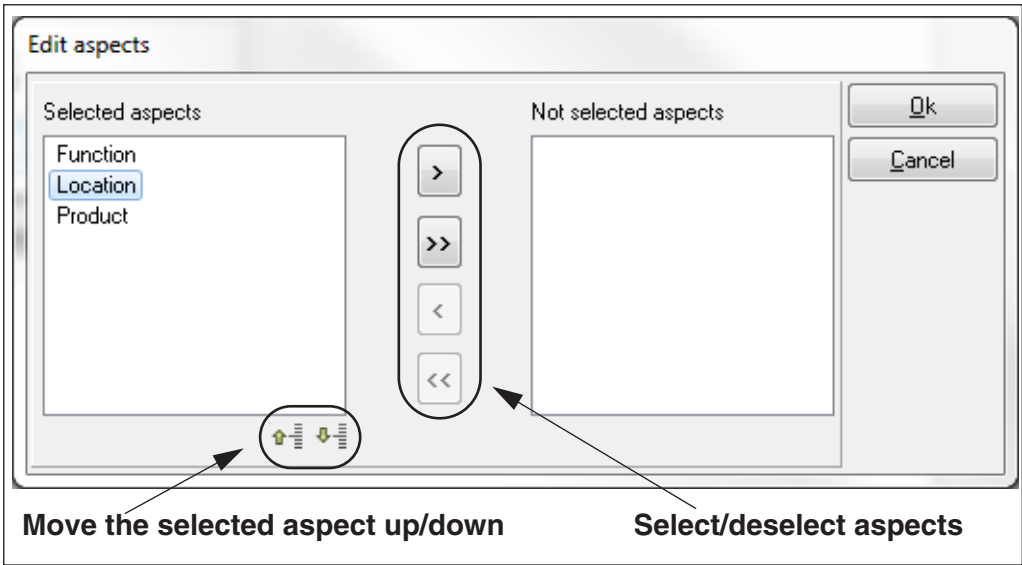
The settings here applies for the graphical representation of reference designations on diagram pages and in lists.

Line breaks

This option normally only appears for **Separate aspects**, but can also appear for **Combined aspects**.

Line breaks for Separate aspects

If you select **Between all aspects in the set**, the aspects are positioned on individual lines in the project. If not, the aspects are written in the same line, separated by the **/** character.

If you have selected **Allow unauthorised line breaks between individual aspects** on the **Standard** tab, you get extra options under **Line breaks**. Here you can choose either **Between all aspects in the set**, **After product**, **After function** and/or **After location**.

Line break - for combined aspects

If you have selected **Allow line break between aspects** on the **Standard** tab, you get extra options under **Line breaks**. Here you can choose either **After product**, **After function** and/or **After location**.

Delimiters between levels

Here you specify which delimiter to use between the levels within each aspect. You have these options, which all follows the IEC 81346 standard:

OPTION	DESCRIPTION
NO DELIMITER (-A1B1C1)	THE LEVELS ARE WRITTEN AS A LONG STRING OF CHARACTERS WITH NO DELIMITER TO SEPARATE THEM.
DOT (DEFAULT) (-A1.B1.C1)	THE LEVELS ARE SEPARATED BY A DOT. THIS IS THE DEFAULT CHOICE.
SPACE (-A1 B1 C1)	THE LEVELS ARE SEPARATED BY A SPACE CHARACTER.
PREFIX (-A1-B1-C1)	THE LEVELS ARE SEPARATED BY THE CHOSEN PREFIX FOR THE ASPECT.

Prefixes

This option only appears when you have selected **Allow non-standard prefixes** on the **Standard** tab. If you choose non-standard prefixes here, this should be documented in the project, in order to follow the standards.

For **Function**, **Location** and **Product** you can here choose either **None/User defined**, **Standard** or **Space**.

None/User defined can be used for either *not* using a prefix, or for typing in a prefix yourself. The latter is not recommended because **Automation** does not contain automatic functions for supporting user defined prefixes.

SETTINGS ACCORDING TO STANDARDS

When you click the **Settings** button at the top of the dialog box **Reference designations**, you enter the dialog box **Reference designations - Settings**. On the **Standard** tab in the dialog box you can specify *nonconformities* to the ISO/IEC/EN 81346 standard.

On the **Standard** tab you have different options depending on whether you have chosen *Combined aspects* or *Separate* aspects on the **Format** tab.

When you select options on the **Standard** tab, this can also cause additional options to occur on the **Format** tab.
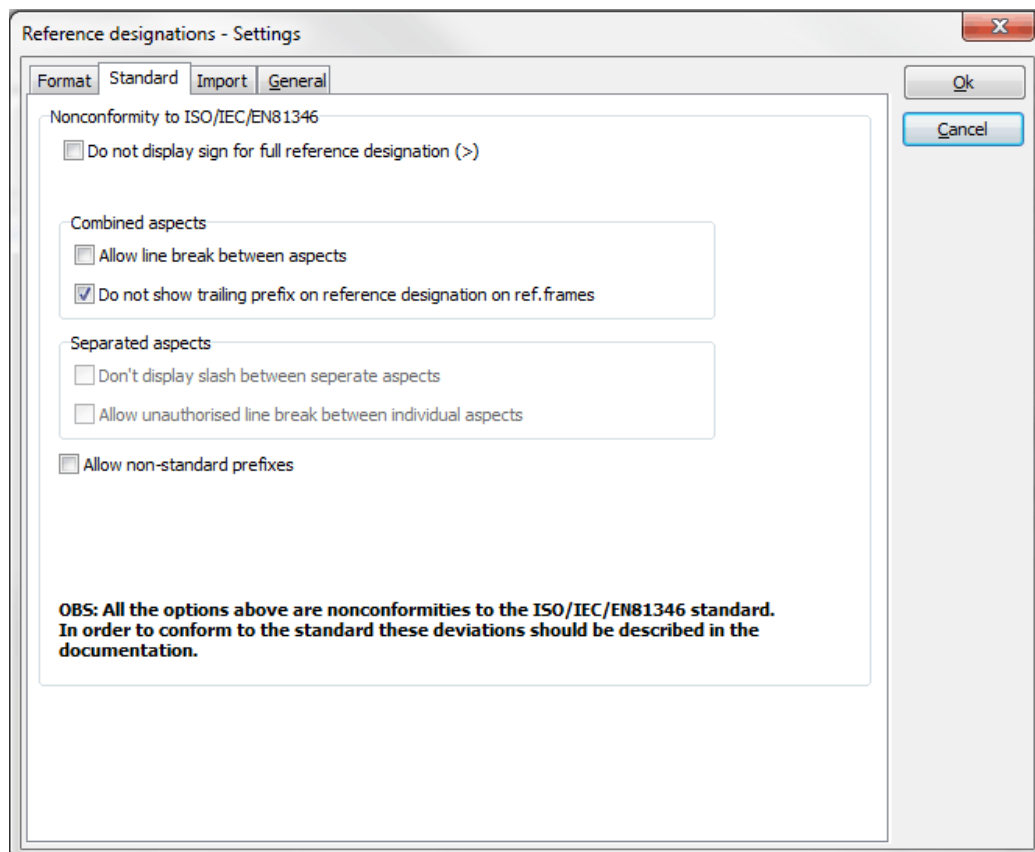
PROJECTS FROM VERSION 17 AND PREVIOUS VERSIONS

IDEALLY NOTHING SHOULD BE SELECTED ON THE **STANDARD** TAB, BECAUSE *ALL SELECTIONS HERE ARE NONCONFORMITIES TO THE STANDARD*.

WHEN YOU OPEN A PROJECT FROM VERSION 17 OR PREVIOUS VERSIONS, SOME OF THE OPTIONS WILL MOST LIKELY BE SELECTED. THIS REFLECTS THAT THE REFERENCE DESIGNATION SETTINGS IN THE PROJECT HAVE NOT BEEN FULLY CORRECT ACCORDING TO THE STANDARDS.

IF/WHEN YOU DESELECT THE SELECTED OPTIONS ON THE **FORMAT** TAB AND CLICK **OK**, THE PROJECT IS CORRECTED SO THAT THE REFERENCE DESIGNATIONS NOW FOLLOW THE STANDARDS. IF YOU DON'T CHANGE THESE SETTINGS, THE PROJECT REMAINS UNCHANGED, AND YOU CAN CONTINUE WORKING AS BEFORE.

Do not display sign for full reference designation (>)

When the character > appears as the first character for a reference designation, it specifies means that it is *the full reference designation* for the component. The displayed reference designation next to the symbol is therefore not meant to be put together with either the reference designation for the page or any reference designations for any reference frames, which the symbol is placed inside. If it therefore says **>+A1-P2** next to a symbol, this means that **+A1-P2** is the *full* reference designation for the symbol.

If you select ***Do not display sign for full reference designation (>)***, you make it much harder to interpret the reference designations in the project correctly — and you deviate from the standard.

Please note: For separate aspects a > character can appear in front of each of the aspects.

Options for Combined aspects

When you have selected ***Combined aspects*** on the **Format** tab, you have the following options:

Allow line break between aspects

When you are working with *Combined aspects* and select ***Allow line break between aspects***, it will appear like you are working with *Separate aspects*. This will be misleading in the documentation, and is therefore *not* recommended.

If you select ***Allow line break between aspects***, you will get additional options on the **Format** tab, where you can specify how to insert the line break. — See "Line breaks" on page 400.

Do not show trailing prefix on reference designations on ref. frames
When you are working with *Combined aspects*, the reference designation for a reference frame must, according to the standard, contain a trailing prefix for the succeeding aspect.

You can deselect this by choosing ***Do not show trailing prefix on reference designations on ref. frames***.

Options for Separate aspects

When you have selected ***Separate aspects*** on the **Format** tab, you have the following options:

Don't display slash between separate aspects

In lists, and for individual settings for symbols and reference frames, there must, according to the standards, be a slash character separating the aspects.

You can deselect this by choosing ***Don't display slash between separate aspects***.

Allow unauthorised line break between individual aspects

When you choose ***Allow unauthorised line break between individual aspects***, you get the option of specifying how to insert the line break on the **Format** tab.
— See "Line breaks" on page 400.

Allow non-standard prefixes

When you choose ***Allow non-standard prefixes***, you get the option of specifying which prefixes you want to use, at the bottom of the **Format** tab.

— See "Prefixes" on page 401.

IMPORT SETTINGS FOR REFERENCE DESIGNATIONS

When you click the **Settings** button at the top of the dialog box **Reference designations**, you enter the dialog box **Reference designations - Settings**. On the **Import** tab in this dialog box you can specify which settings applies for importing reference designations.

In the upper-right corner of the tab, you can see an example of the reference designation format specified by the current settings.

Format as defined on the Format tab

When you select *Format as defined on the Format tab*, the settings specified on the **Format** tab are used when importing (recommended setting).

When you select this option you make it easier for yourself to import the exported file again, if you later make changes in the file using for instance Excel — for instance for IO lists for PLC's.

Delimiters between levels

If you do *not* choose *Format as defined on the format tab*, you can specify which *Delimiter between levels* to apply:

Letter followed by number

When **Letter followed by number** is selected, the program interprets letters followed by a number as an individual level in an aspect:

- *A1*, *BB2* and *ABC3* will therefore be interpreted as a designation containing a single level.

- **A1BB2** will be interpreted as a designation containing two levels.

When **Letter followed by number** is *not* selected, the program will not interpret letters followed by numbers as the specification of levels in reference designations.

Dot

When **Dot** is selected, the program understands dots in reference designations as specifications of levels:

1, *A* and *112* will be interpreted as a designation in one level.

- **A.B**, *1.A* and *1.12* will be interpreted as a designation in two levels.

Space

When **Space** is selected, space characters will be interpreted as specifications of levels.

Support of KKS codes

On the **Import** tab you can also specify that you wish to import KKS codes, and make these behave hierarchically.

When the KKS codes appears in the project, these can also be presented without a separator, which you can specify on the **Format** tab.

You have these two options:

  - 1-letter codes/numbers (KKS)
  - 1-letter codes/numbers (KKS) w/leading 3-letter code

GENERAL SETTINGS FOR REFERENCE DESIGNATIONS

When you click the **Settings** button at the top of the dialog box **Reference designations**, you enter the dialog box **Reference designations - Settings**. On the **General** tab you can specify whether/how to insert reference designations in the project for specific symbol types:
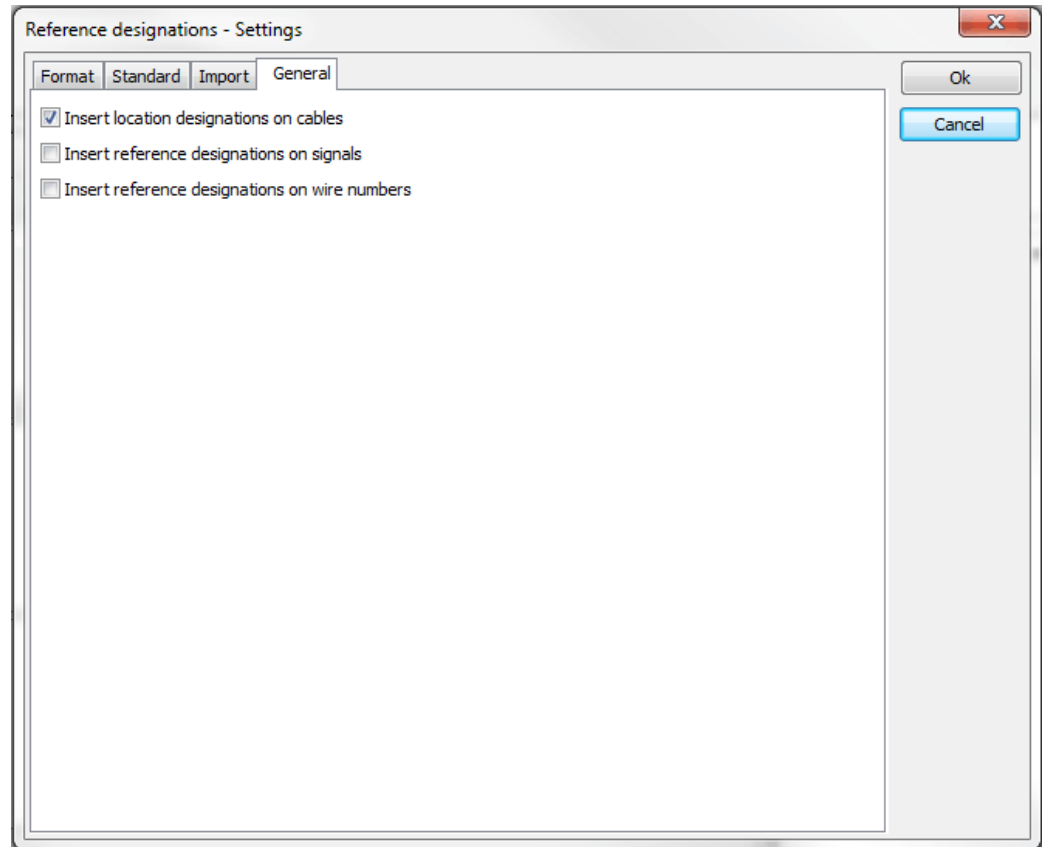
Insert location designations on cables

Because location reference designations are not typically assigned to cables (a cable runs from one location to another), you can here specify whether to **Insert location designations on cables**.

This setting specifies whether to automatically insert a location reference designation for a cable, when the cable is placed on a page with reference designations.

Insert reference designations on signals

To assign signal references to signals: Select **Insert reference designations on signals**. — Please see "Insert Reference designations on Signals automatically" on page 241.

Insert reference designations on wire numbers

To include reference designations in wire numbers: Select **Insert reference designations on wire numbers**.

IMPORT / EXPORT OF REFERENCE DESIGNATIONS

Reference designations can be exported/imported to/from Excel and to/from comma separated files (*.csv* files).

ON IMPORT AND EXPORT OF REFERENCE DESIGNATIONS

IMPORTING REFERENCE DESIGNATIONS

When importing reference designations via an Excel file, the reference designations must be created as texts in the Excel file. During the import the reference designations are created automatically in the active project based on their prefixes (=, + and -).

In "Import settings for reference designations" on page 405 you can see how to specify how **Automation** understand the reference designation files.

— See an example of an Excel file with reference designations in **Pcselcad/Lists/ ref-designations_uk.xlsx**.

Import the reference designations

When you wish to import reference designations, click the **Import** button at the top of the dialog box **Reference designations** — or right-click in the dialog box and select **Import**. You the get the option of choosing which file you wish to import reference designations from.

When you have selected the file, you enter this 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:
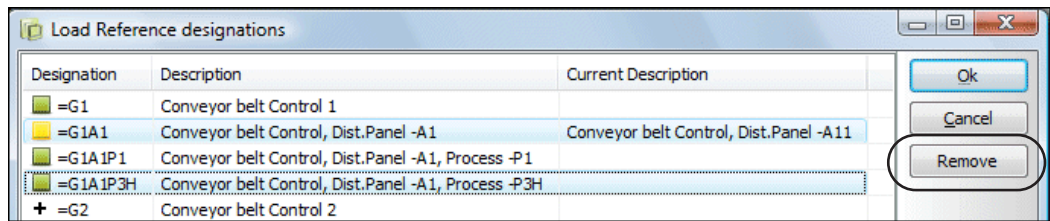
Then import the remaining reference designations by clicking **OK**.

Import reference designations for individual aspects

To import reference designations for just one of the aspects — for instance the **Location aspect** — do one of the following:

    a) Right-click in the **Location aspect** area and select **Import Location designations** — or

    b) Click in the **Location aspect** area and click the **Import Location designations** button at the bottom of the area for the location aspect.

Then continue as described in "Import the reference designations" on page 408.

PREFIX NOT INCLUDED DURING THE IMPORT

WHEN YOU IMPORT REFERENCE DESIGNATIONS FOR AN INDIVIDUAL ASPECT, IT IS NOT TAKEN INTO CONSIDERATION WHICH PREFIX IS USED — OR *WHETHER* A PREFIX IS USED — IN THE FILE THAT YOU IMPORT FROM.

IN OTHER WORDS: YOU HAVE TO MAKE SURE THAT IT IS THE RIGHT ASPECT THE FILE IS IMPORTED TO!

EXPORTING REFERENCE DESIGNATIONS

When you want to export the project reference designations, you have two options: You can export *all reference designations in the project*, or just *export the reference designations for one of the aspects*.

    • To export *all reference designations in the project*: Click the **Export all** button at the top of the dialog box **Reference designations**.

    • To export *all reference designations for one of the aspects*: Click inside the window for the relevant aspect (for instance the location designation) and click the **Export Location designations** button at the bottom of the window — or right-click in the window and choose **Export Location designations**.
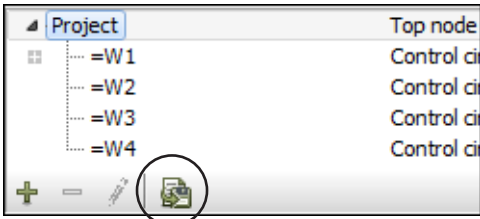
Specify the file format for the reference designation file

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:

REFERENCE DESIGNATIONS IN THE EXPLORER WINDOW

When you have created reference designations in the active project, these are displayed on the **Projects** tab in the **Explorer window** at the left side of the screen. Here you find the reference designations under **Components**.

When you click on a symbol in the project, and then click the **Show selected symbol** button at the upper part of the **Projects** tab, the reference designations for the component are displayed on the tab:
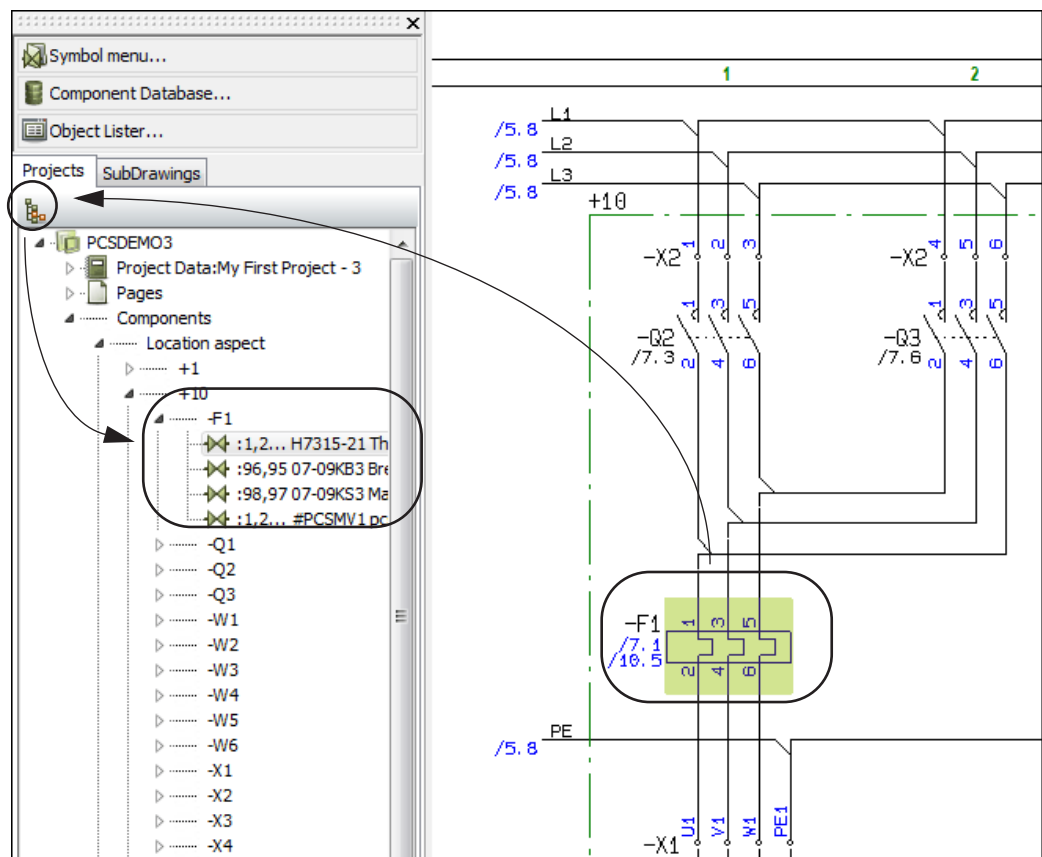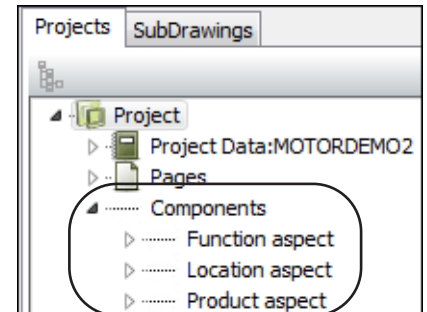
This also makes it possible to jump directly to the other symbols for the components on the project pages by double-clicking on them in the **Explorer window**.

— You can read more about the **Projects** tab in the **Explorer window** in "The Explorer window: The Projects tab" on page 345.

REFERENCE DESIGNATIONS FOR SYMBOLS

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.
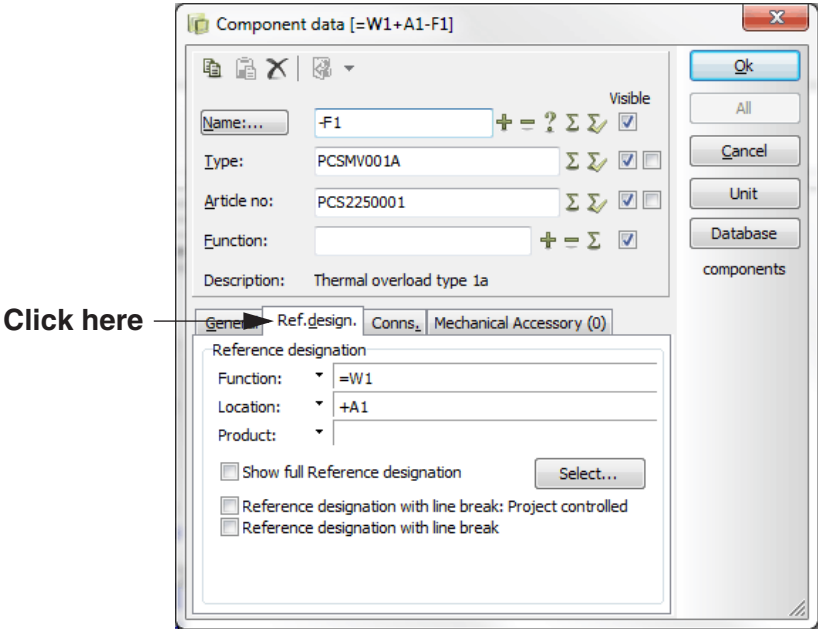
Choosing reference designations when placing symbols

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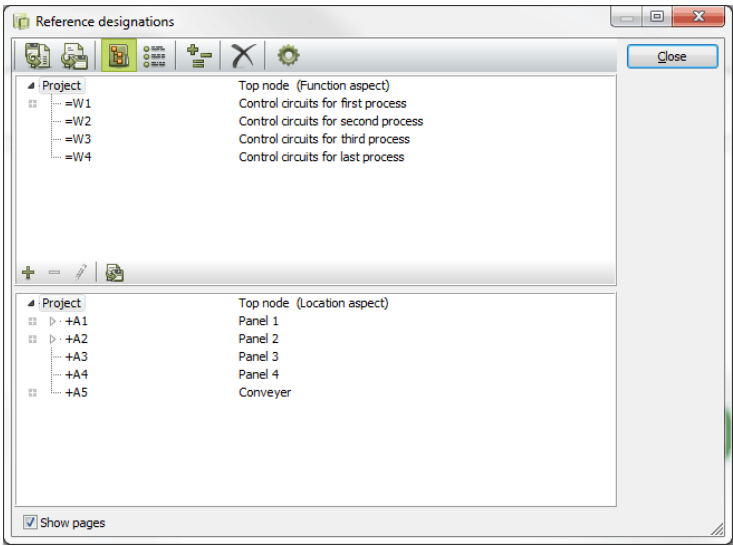
ONLY CREATED ASPECTS DISPLAYED

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

IF FOR INSTANCE YOU HAVE NOT CREATED A PRODUCT ASPECT, THIS PART OF THE DIALOG BOX IS NOT DISPLAYED, AND THE FUNCTION AND LOCATION ASPECT WINDOWS FILL 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 "Adding new reference designations" on page 395.

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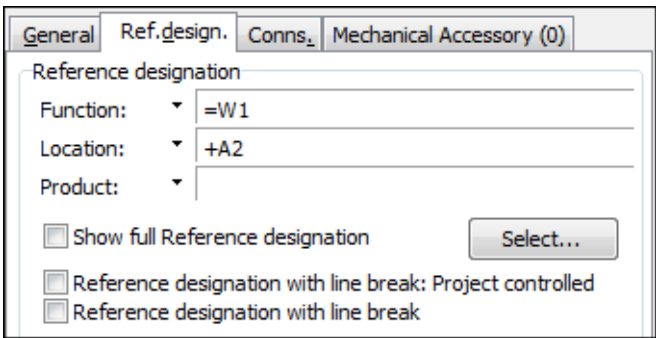
Displaying the page reference designation and the most recently applied reference designations

When you click on the down-arrow next to the *Location* field, the location 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 location reference designation for the page and the most recently applied location reference designation are displayed in the top of the list.
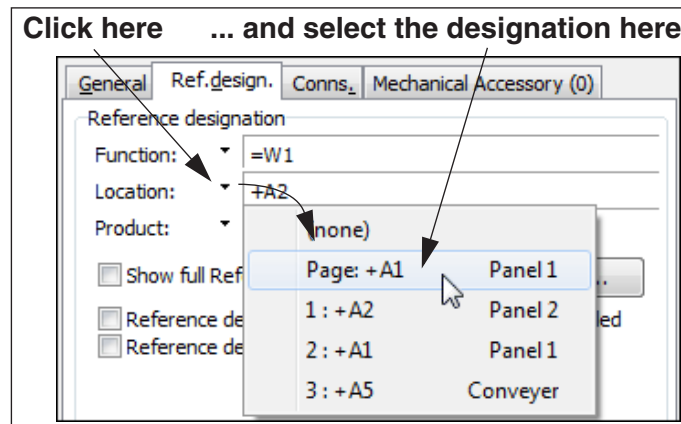
You can do just the same for function and product reference designations.

Short-cut keys

These functions have the short-cut keys <Ctrl+f> for function reference designations, <Ctrl+l> for location reference designations and <Ctrl+p> for product 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***. However, *this is not recommended because the reference designation hereby can be misinterpreted!*

— See "Character specifying full reference designation" on page 388 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:
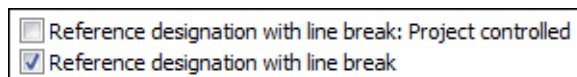
You can then select/deselect **Reference designation with line break** for the selected symbol, independent of the general project settings. However, *this is not recommended because it will be misleading according to the standards!*

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 SELECT NAME

— See "Changing the symbol reference designations using Select name" on page 276.

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.

☑ Insert - in front of symbol names
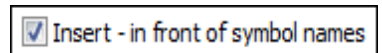
Adding "-" in already placed symbols

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 438.

REFERENCE DESIGNATIONS FOR AREAS, PAGES & CHAPTERS

CONSIDERATIONS ABOUT REFERENCE DESIGNATIONS

When you specify reference designations for a project, it is recommended to start by specifying the reference designations at the top level. This makes it much easier to maintain an overview over the structure of the reference designations.

It is therefore recommended to start by specifying the reference designations for chapters and pages, and then specify reference designations for reference frames — if any — on individual project pages.

If you use nested reference frames, it is recommended to start by placing the outer reference frames first.

REFERENCE DESIGNATIONS FOR PAGES

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 420.

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 417.

Click **OK**.

Symbol names on pages with 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 form the reference designation of the page.

Using the Component menu on pages with reference designations

When you via the **Component menu** 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**.

The page reference designation
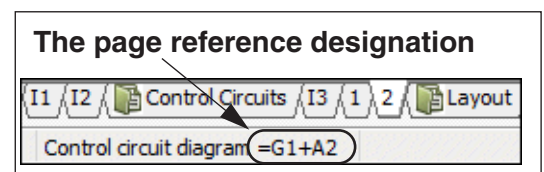
Page reference designation displayed as popup information

When you let the mouse cursor rest over a page tab, the page reference designation is displayed in the popup information.
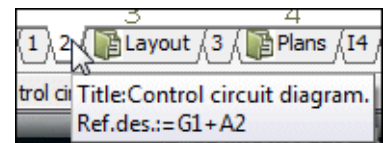
Inserting a drawing header displaying reference designations

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 428.

Please refer to "Creating Drawing Headers" on page 820 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 (REFERENCE FRAMES)

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.
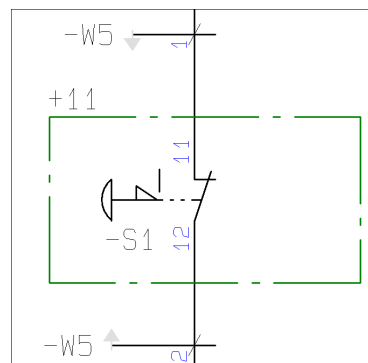
When you have drawn the frame, you enter this dialog box:
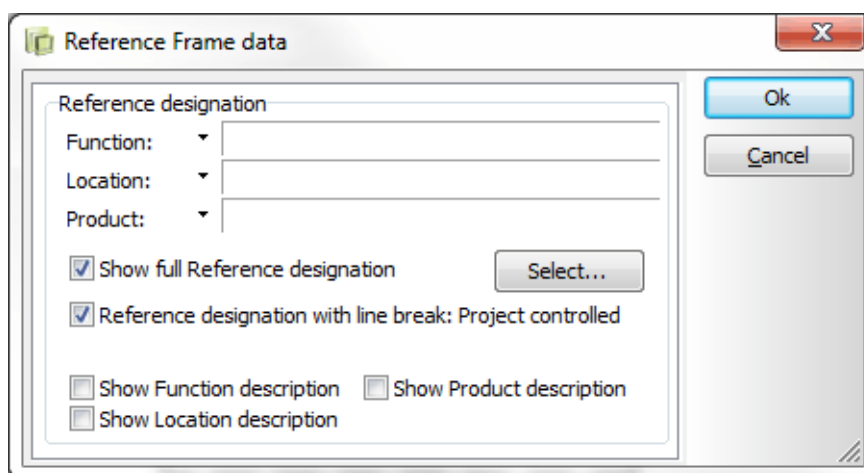
Here you select the relevant reference designation as described in "Selecting Reference designations when placing symbols" on page 412, and click **OK**.

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.

🚫 **DO NOT DISPLAY THE FULL REFERENCE DESIGNATION FOR THE REFERENCE FRAME!**

WHEN YOU WANT THE FULL REFERENCE DESIGNATION TO BE DISPLAYED, YOU SELECT *SHOW FULL REFERENCE DESIGNATION*. HOWEVER, IT IS STRONGLY RECOMMENDED NOT TO DO THIS, BECAUSE IT MAY RESULT IN THAT THE REFERENCE DESIGNATIONS WILL BE MISINTERPRETED!

Show Function, Location and/or Product description

If you select **Show Function description**, **Show Location description** and/or **Show Product description** the selected descriptions for the reference frame are displayed.

The description for the function reference designation is positioned in the upper-left corner of the frame, while the location reference designation description is positioned in the lower-left corner of the frame with the description of the product reference designation description positioned above it.
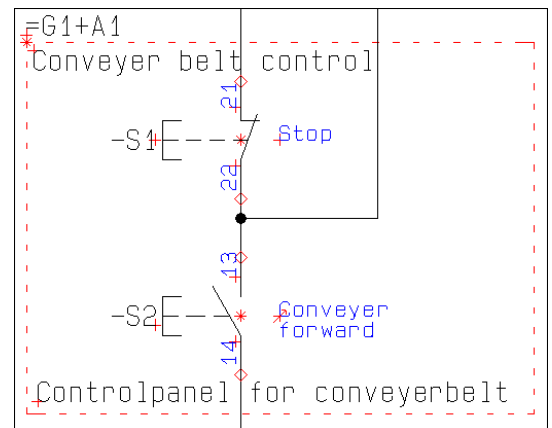
The texts can be moved freely, and their text data can be edited, as for other texts in the program. — See "Moving symbol texts on a Diagram page" on page 117 and "Specifying text properties / Appearance of text" on page 136.

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 414.

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 420.

SPECIFY SETTINGS FOR LINES, TEXTS AND REFERENCE FRAMES

UNDER **SETTINGS => BASIC SETTINGS** YOU CAN SPECIFY HOW LINES ARE DRAWN, WHEN YOU CHOOSE **INSERT REFERENCE FRAME**.

IN THE SAME DIALOG BOX YOU CAN ALSO SPECIFY THE TEXT PROPERTIES FOR THE REFERENCE DESIGNATION TEXTS DISPLAYED NEXT TO THE REFERENCE FRAME.

— READ MORE ABOUT LINES SETTINGS IN "THE LINES TOOLBAR" ON PAGE 73.

Moving a reference frame

1) Disable the Pencil (press <Esc>).

2) Click on the upper-left corner of the reference frame and *drag* the reference to the desired position.

Changing the shape of reference frames

When you have placed a reference frame, you can change the shape of the reference frame the following ways:

Changing a reference frame to a rectangle with a different shape

Click on one of the corners of the reference frame and *drag* until the reference frame has the desired shape.

Please note: You can not change the shape of a reference frame by dragging the upper-left corner. This is because the upper-left corner is used for *moving* the reference frame.

Insert Line Point

Right-click on the reference frame and choose **Insert Line Point**. Then click to specify the point where the line breaks.

Fold corner

Right-click on one of the corners of the reference frame and choose **Fold corner**. Then click to specify how to fold the corner.

Please note: You can not fold the upper-left corner of a reference frame.

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 (Reference frames)" on page 418. 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.
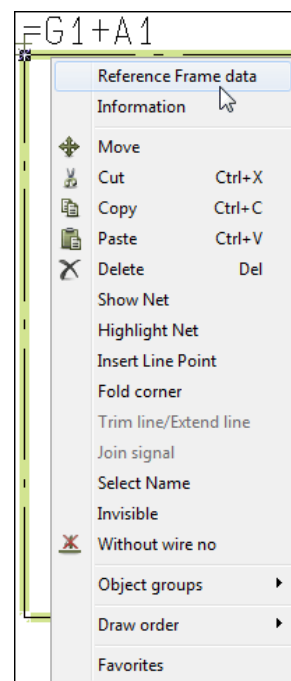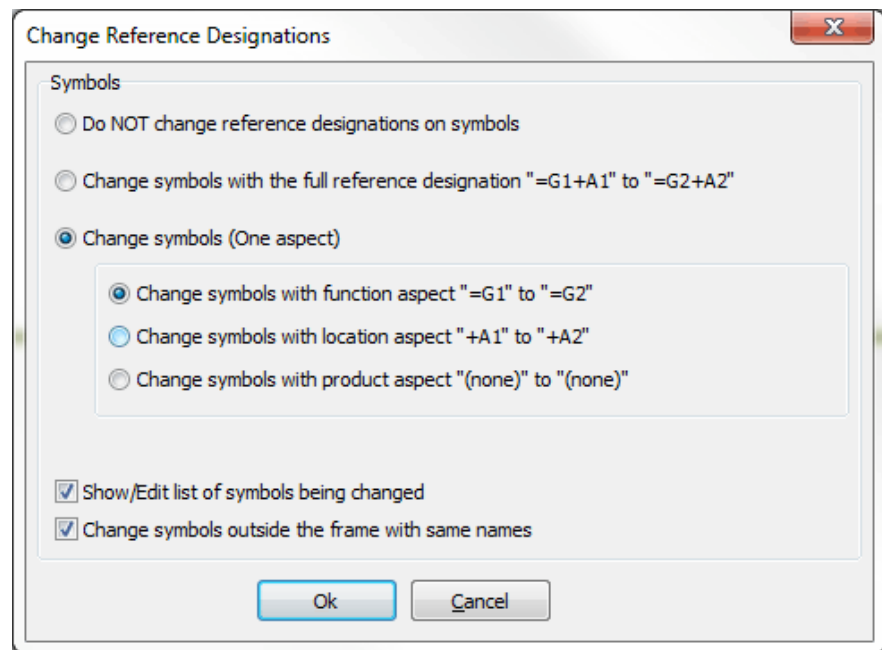
Specifying which symbols inside the frame to assign new reference designations to

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 the full reference designation "=G1+A1" to "=G2+A2"**: Symbols with the same full reference designation as the reference frame, are updated

- **Change symbols (One aspect)**: Here you get the option of specifying which aspect to change for symbols with matching reference designations for the 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 427.

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 351.

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 423 —, 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 PAGES" ON PAGE 416.

— Please refer to "Change reference designations via chapter Divider pages" on page 426 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 **Page menu** button, or select **View => Select page.**

2) In the **Page menu**: Select the pages for which you wish to change reference designations — e.g. 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*, *LOCATION* AND/OR *PRODUCT* ARE DIMMED, THIS MEANS THAT THE SELECTED PAGES DO NOT HAVE IDENTICAL REFERENCE DESIGNATIONS.

Reference designation
Function:
Location:
Product:
Select...

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*:
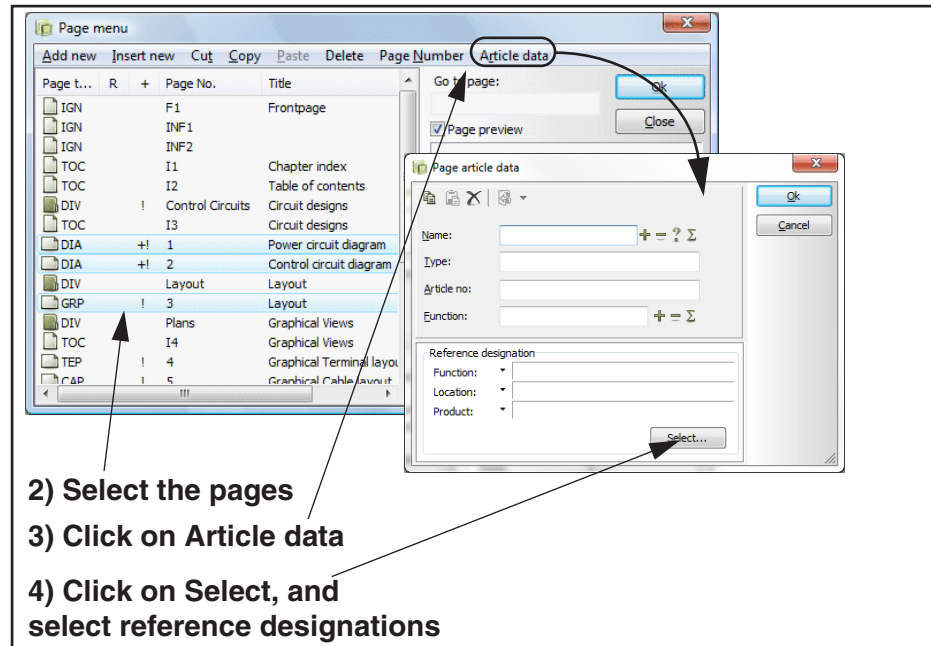
2) Select the pages

3) Click on Article data

**4) Click on Select, and
select 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 the relevant reference designation(s) to confirm that you wish to select this/these reference designation(s). This causes *Function*, *Location* and/or *Product* 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 423).
Click **OK**.

**Click here to confirm the
selected designations**
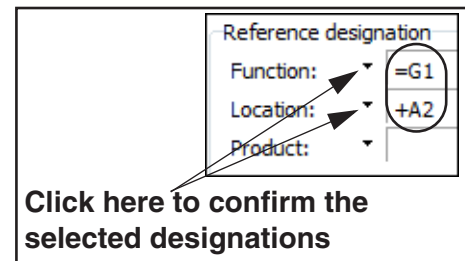
If the pages did have identical reference designations

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:
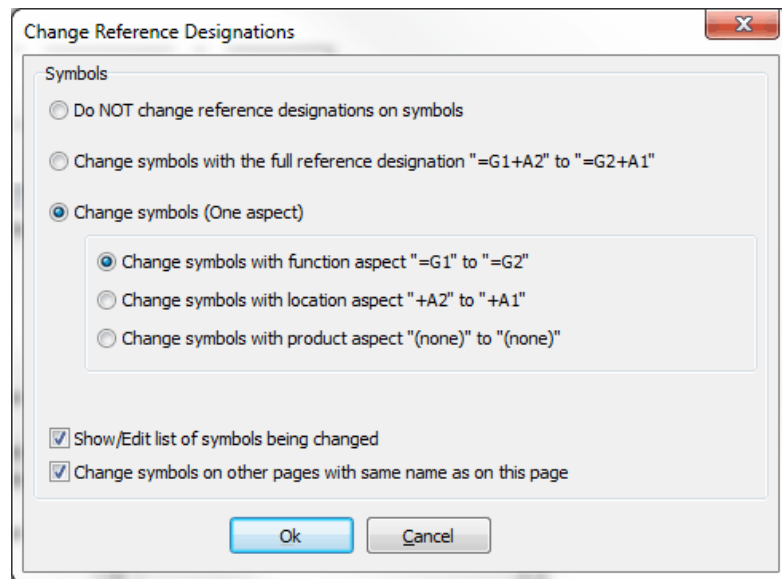
— Read about the options in this dialog box in "Options in the dialog box Change Reference Designations" on page 426.
Choose how to change reference designations for the involved symbols, and click **OK**. If you have chosen ***Show/Edit list of symbols being changed***, please also see "Show/Edit list of symbols being changed" on page 427.

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** in the *Reference designations* section.

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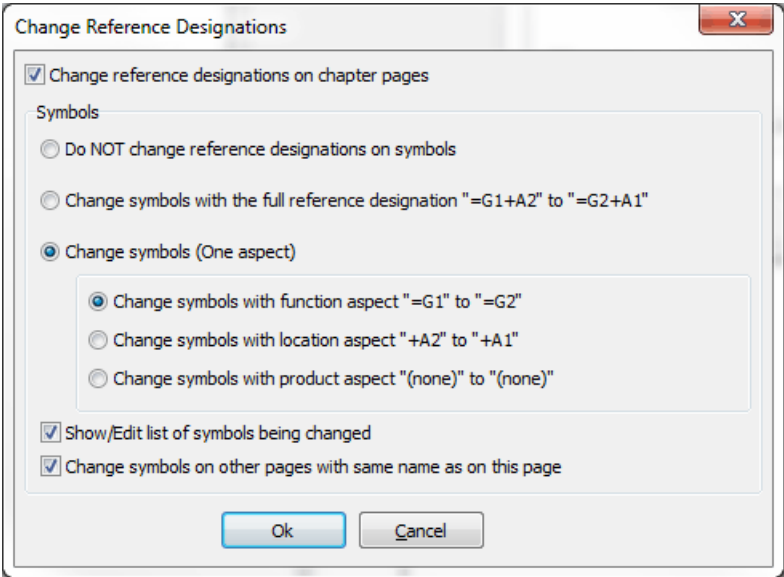
Here you are asked whether to change reference designations on the pages in the chapter having the same reference designations as the divider page, and which directions to apply for changing the symbols. You have the following options:

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 THE FULL REFERENCE DESIGNATION "X" TO "Y"	CLICK HERE TO CHANGE THE FULL REFERENCE DESIGNATION.

OPTION	EFFECT
CHANGE SYMBOLS (ONE ASPECT)	CLICK HERE — AND THEN CHOOSE ONE OF THE OPTIONS — TO CHANGE *REFERENCE DESIGNATIONS FOR ONE OF THE ASPECTS*.
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
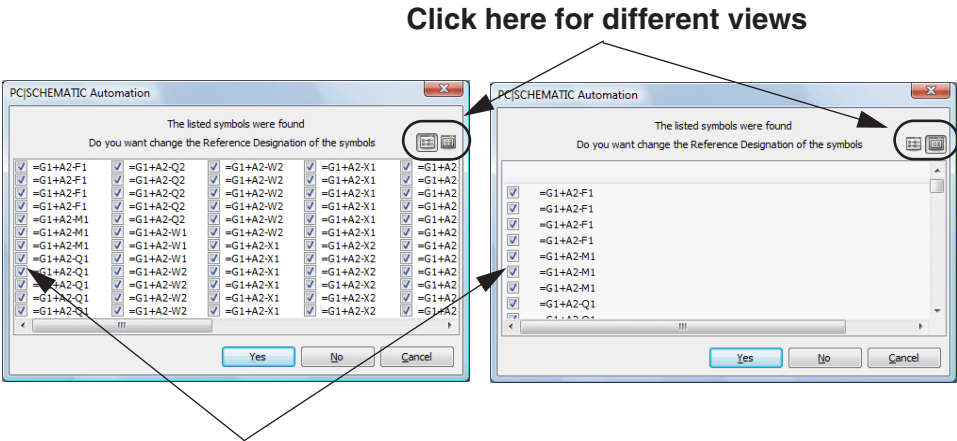
Remove the check mark in front of the symbols, that shall not have their reference designations changed, and click **OK**.

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

You have now changed reference designations for the desired pages and symbols.

REFERENCE DESIGNATIONS: DATAFIELDS AND LISTS

THIS CHAPTER DESCRIBES

DATAFIELD FOR REFERENCE DESIGNATIONS IN PROJECTS

When you wish to place a data-field 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**.

The Reference designation is displayed
The attached description is displayed

The full ref. designation is displayed
One of the aspects is displayed

> **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.

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 ACCORDING TO REFERENCE DESIGNATIONS

When you apply reference designations, it is possible to create separate lists for each Function, Location or Product 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* reference designation **=P1** — which for instance could be a pump — you fill out the **Criteria** tab like this:

General	Sort by	Criteria	Repetitions	Replaces

Field	Operator	Text	A<>a
Name	=	=P1*	☐

Or fill out the **Criteria** tab like this to get a list of the assembly **+A1**:

General	Sort by	Criteria	Repetitions	Replaces

Field	Operator	Text	A<>a
Name	=	=*A1*	☐

— Please refer to "Specifying Criteria for the components to include" on page 617 for further details on criteria on lists.

SELECT/DESELECT OPTIONS USING REFERENCE DESIGNATIONS

When you have created a project with reference designations, this makes it possible to select/deselect options in the project based on the reference designations, and then generate a new project only containing the selected options. — Read more about this in "Projects with selectable options" on page 592.

REFERENCES

This section describes the use of references in PCSCHEMATIC Automation:

PC│SCHEMATIC

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.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— INSERTING A REFERENCE SYSTEM</td><td>(P. 433)</td></tr>
<tr><td>— AUTOMATIC JUMP BETWEEN REFERENCES ON DOUBLE-CLICK</td><td>(P. 433)</td></tr>
<tr><td>— REFERENCES ON SYMBOLS</td><td>(P. 435)</td></tr>
<tr><td>— APPEARANCE OF REFERENCES</td><td>(P. 438)</td></tr>
<tr><td>— REFERENCE DESIGNATIONS ON SYMBOL AND SIGNAL REFERENCES</td><td>(P. 438)</td></tr>
<tr><td>— REFERENCES FOR PLCS</td><td>(P. 439)</td></tr>
</table>

— Please see to "Contact mirrors and Reference crosses" on page 440.

— Please also see "Navigation between symbols for the same component" on page 195.

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 445.

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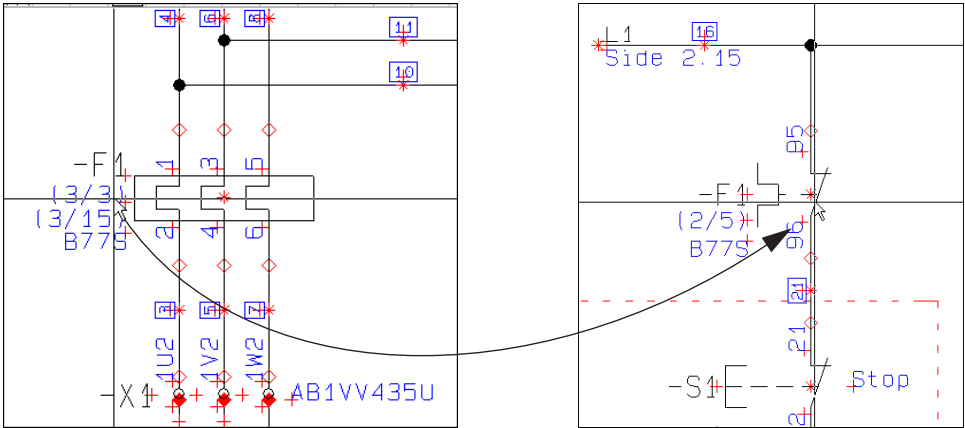
Double-click on the reference, and the referred symbol is selected

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:

**Double-click here to return
to the relay coil symbol**

**Double-click on the reference in the reference cross
and the referred symbol is selected**
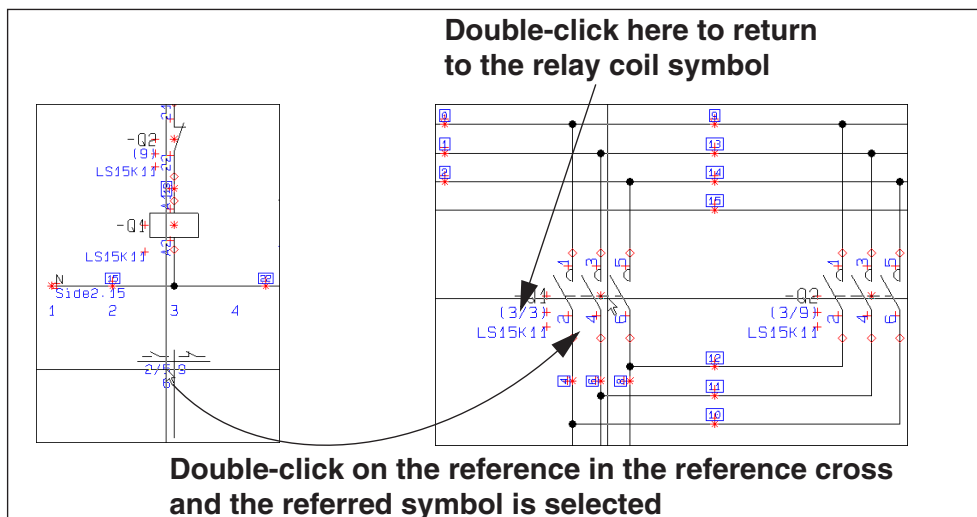
When you double-click on the reference for a symbol in the reference cross/contact mirror, you return to the symbol with the symbol having the symbol type *Relay*.

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 775; see "Symbol types" in particular. You can not subsequently change the symbol type.

However, you can see 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 the **Symbol type** is displayed:

The following symbol types exist:

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.
TERMINAL	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
PLC	THE SYMBOL IN NOT ASSIGNED REFERENCES — UNLESS SPECIAL ACTION IS TAKEN.
PLC REFERENCE	SYMBOL WHICH IS TO BE USED AS A PLC REFERENCE SYMBOL. — SEE "CREATING PLC REFERENCE SYMBOLS" ON PAGE 806 AS WELL AS "WORKING WITH PLCS" ON PAGE 478 AND "ON PLC REFERENCE SYMBOLS" ON PAGE 478.
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.
HEAD	SYMBOL THAT ARE USED FOR PLACING OF A LIST ON A PAGE. — SEE "CREATING A LIST SYMBOL" ON PAGE 842.

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 775*, "SYMBOL SETTINGS: REFERENCE" ON PAGE 788 IN PARTICULAR.

APPEARANCE OF REFERENCES

To specify the appearance of the reference, do the following:

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

2) Click on *Reference*.

3) Then click on *Symbol reference* to specify the appearance of the symbol references.

4) You now get the following options:
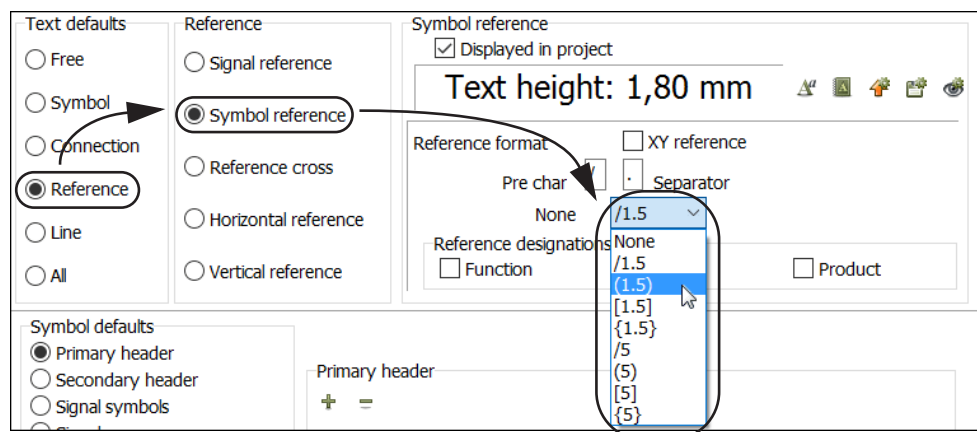
5) Now choose *Reference format*, *Pre char* and *Separator* and brackets if necessary. If you use a reference system with both horizontal and vertical references, then select *XY reference* (see "Horizontal and vertical references" on page 446).

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 136.

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 *Reference*.

3) Then click on *Symbol reference* to specify the appearance of the references for symbols.

4) Now choose which reference designations to include in the references, 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

CONTACT MIRRORS AND REFERENCE CROSSES

When displaying references for symbols with the type **Relay** — see "References on symbols" on page 435 — you can either use *reference crosses* or *contact mirrors*.

This chapter describes how to use reference crosses and contact mirrors in the program.

— Please read "Setting up the current paths" on page 445 to see how to set the reference system on the project pages.

— Please also read "Automatically updated references" on page 433 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 443 — and place a symbol with an article number from the attached database, the other symbols for the component are displayed at the bottom of the page.

Thus contact mirrors are a graphical representation of the other symbols for the component. The contact mirror is placed under the symbol for the component, which has the symbol type *Relay*.

When for instance you place a relay coil symbol via the database, the other symbols for the component are displayed below the relay coil symbol at the bottom of the page — as displayed on the figure to the right. In the contact mirror for the relay coil symbol you see a three-poled open contact, a close contact and an open contact.

Contact mirrors and changes in the article number

When you change the article number for a symbol, the contact mirror will be changed, so that it matches the information about the article number in the database.

Contact mirror for the symbol

Symbol with the type Relay
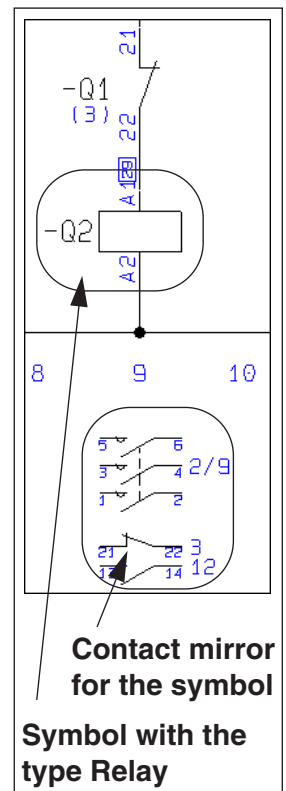
Direct jump to the other symbols for the component

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 433.

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.

Double-click on an available symbol, and you get it in the cross hairs
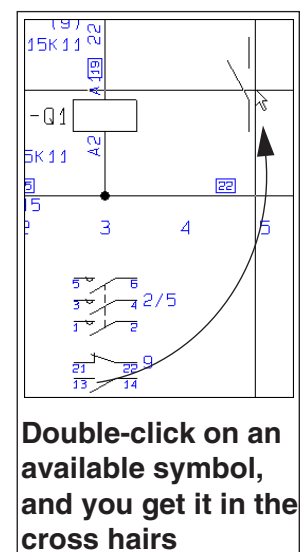
On reference crosses

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 433.

Click here, and you jump directly to the symbol, which is referred to

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 441.

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" on page 188, particularly "Functionally understood as different components" on page 188.

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.
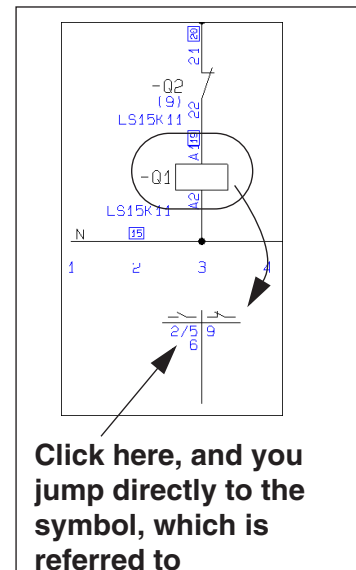
CHOOSING BETWEEN CONTACT MIRRORS AND REFERENCE CROSSES

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**.
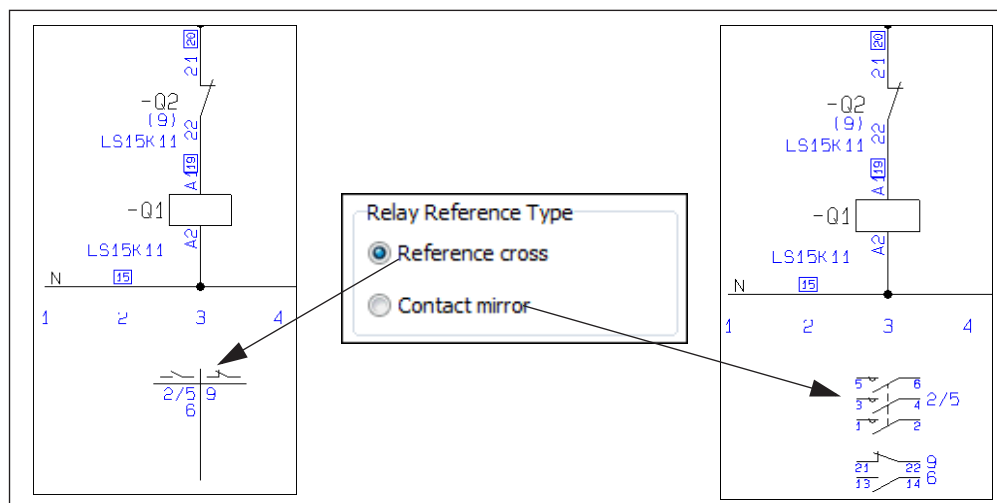
You can, at any time, choose whether you wish to use contact mirrors or reference crosses.

If **Contact mirror** is activated without an attached database, the contact mirrors only contains the symbols, which are used in the project.

When you have chosen **Contact mirror**, you can subsequently scale the size of the contact mirror by clicking the up or down arrow under **Scale**.
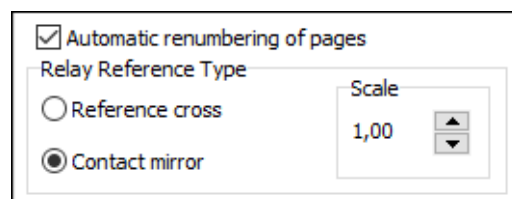
SWITCHING BETWEEN CONTACT MIRRORS AND REF. CROSSES

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.

Moving contact mirrors or reference crosses

To move a contact mirror / reference cross with respect to the relay coil symbol, do this:

1) Right-click the relay coil symbol and choose **Move reference symbol**.

2) Click to point out the new position of the reference symbol.

Skip reference cross

If you do not want a reference cross or a contact mirror displayed for a relay coil symbol, do this:

1) Double-click on the relay coil symbol.

2) In the dialog box **Component data**, on the **General** tab: Select *Skip reference cross*.

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 682).

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.

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:
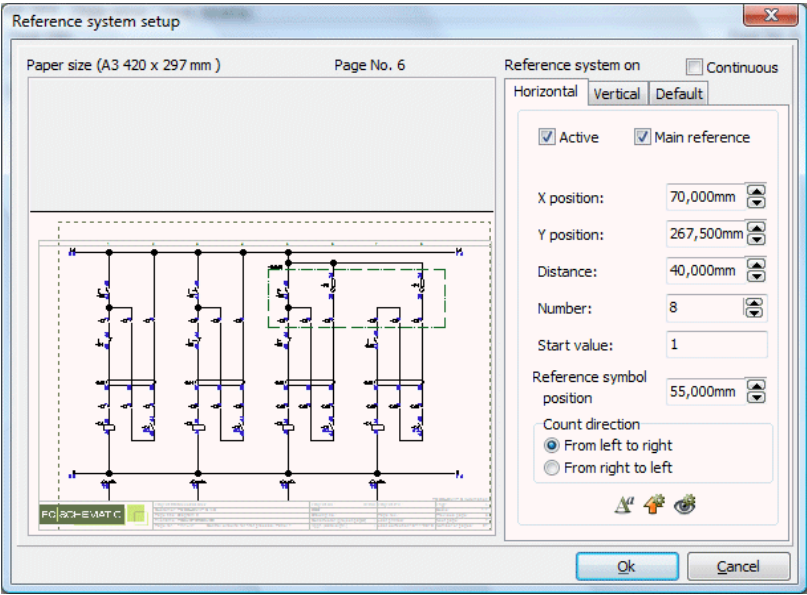
HORIZONTAL REFERENCE SYSTEM

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 785). 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.

HORIZONTAL AND VERTICAL REFERENCES

When you wish to use both horizontal and vertical references in projects, do this:

1) In **Settings => Page data => References**: Select *Active* for both the **Horizontal** and the **Vertical** reference system.

2) If you want to list the horizontal reference first, select *Main reference* on the **Horizontal** tab. — And if you want to list the vertical reference first, select *Main reference* on the **Vertical** tab.

3) Click **OK**.

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 556.
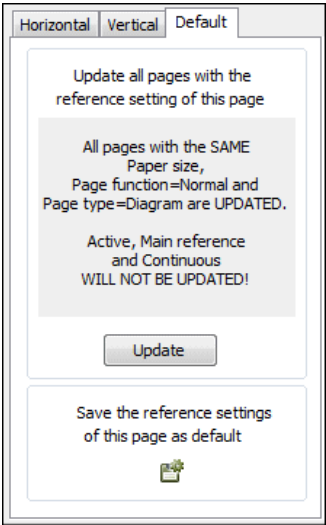
CHANGING THE REFERENCE CROSSES

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 775.

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 775.

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:
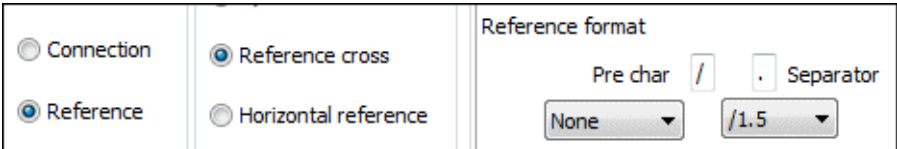
If you use a reference system with both horizontal and vertical references, then select *XY reference* (see "Horizontal and vertical references" on page 446).

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 *succeedingly*.

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 141.

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 451.

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 560).

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 loose 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*.

— Se 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:

PC | SCHEMATIC

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.

— Please refer to "Manual placing of wire numbers" on page 464 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 464.

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.
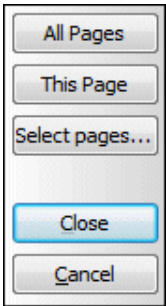
When you click on **Select pages**, you enter a special version of the **Page menu**:

These pages can
be selected →
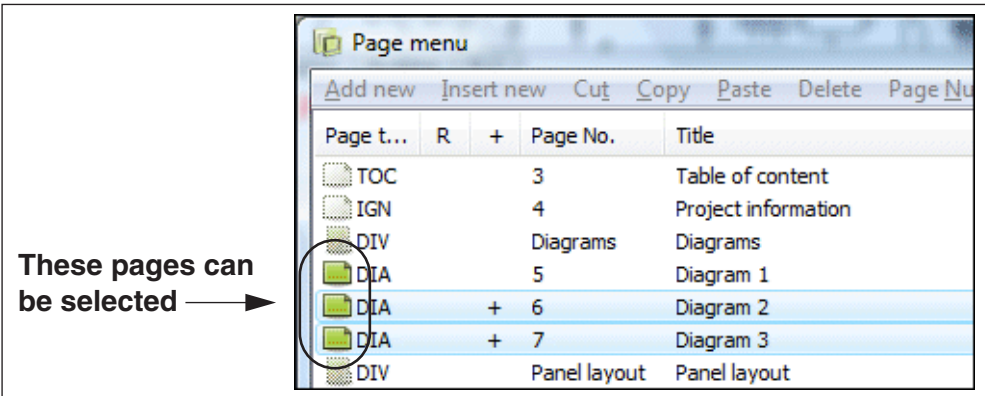
The pages where it is possible to perform wire numbering, are indicated by a green page icon. Select the pages where you wish to assign wire numbers, and click **OK**.

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 351.

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
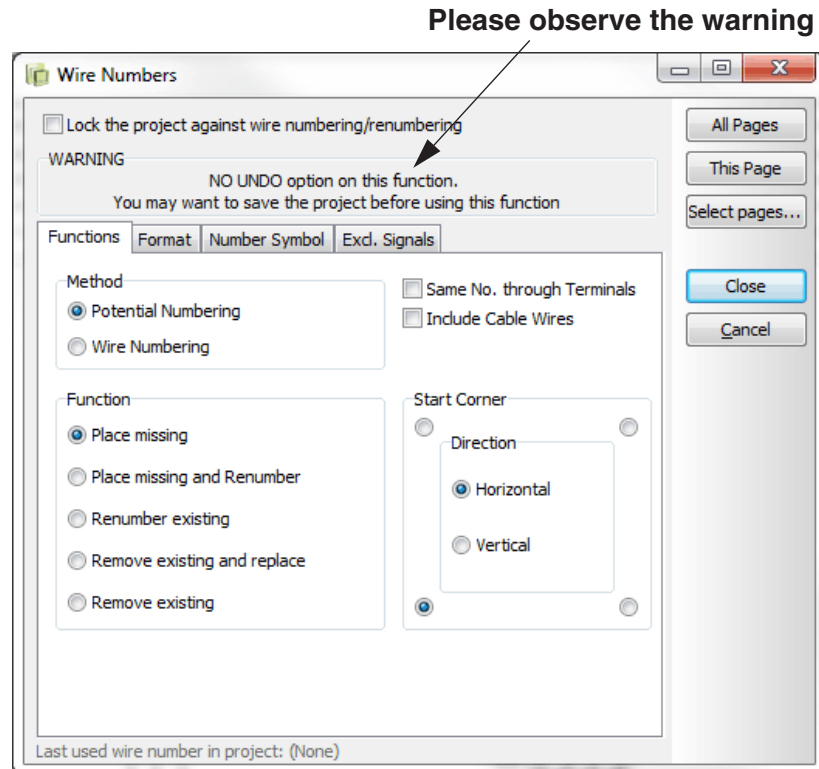
Basically, there are two different principles for performing the wire numbering, namely potential numbering and wire numbering.

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 234) — 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**.

Wire numbering

If you want to assign a unique number to each wire in the entire project, you check off **Wire Numbering** in the upper-left corner of the dialog box.

THE WIRE NUMBERING IS A QUALIFIED GUESS

IF YOU CANNOT TELL FROM THE DRAWING HOW THE WIRE NUMBERS SHOULD BE PLACED, YOU CANNOT ASSUME THAT THE PROGRAM CAN GUESS WHERE THE WIRE NUMBERS SHOULD BE LOCATED!

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

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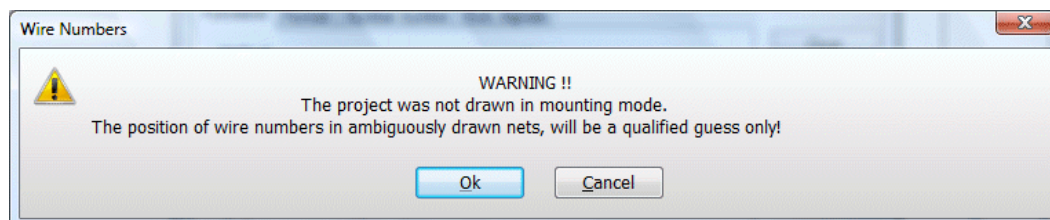
Unambiguous drawing

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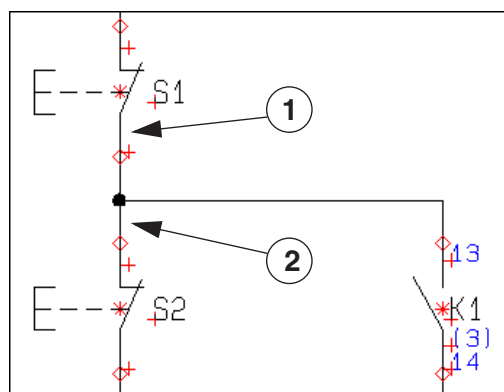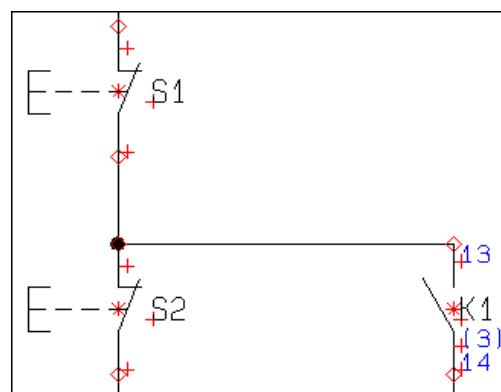
Ambiguous   **Unambiguous**

On the ambiguously drawn figure above, it is not clear whether the horizontal wire connects **K1** to **S1**, or from **K1** to **S2**.

On the other hand, the unambiguously drawn figure clearly states that the wire connects **K1** to **S2**.

It only makes sense to place a wire number on a drawn line, which only represents one wire. However, on the ambiguous drawing above, it is not clear whether the

line from **S1** to the connection dot (**1**), or the line from the connection dot to **S2** represents both wires. On the ambiguous drawing, the symbols could be connected in two different ways:

Option 1

Option 2
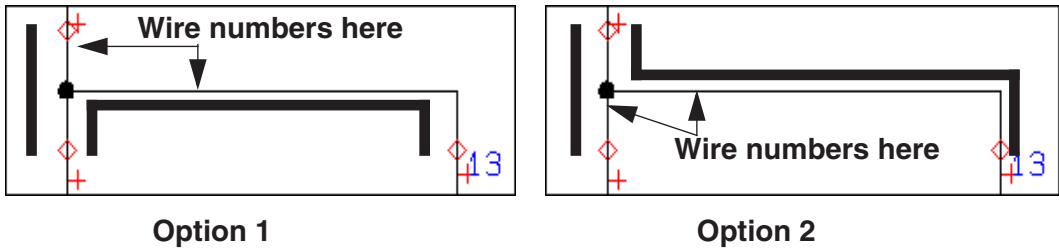
In this case, the program must make a qualified guess.

The program guess

Automation will make a guess which polyline represents more than one wire. These are not assigned a wire number.

— In this context a polyline is the line drawn *from* the connection point of a symbol *to* the connection dot in which the line meets another line. On the figure, one of the polyline is the line starting in the connection point *13*, and ending in the connection dot.
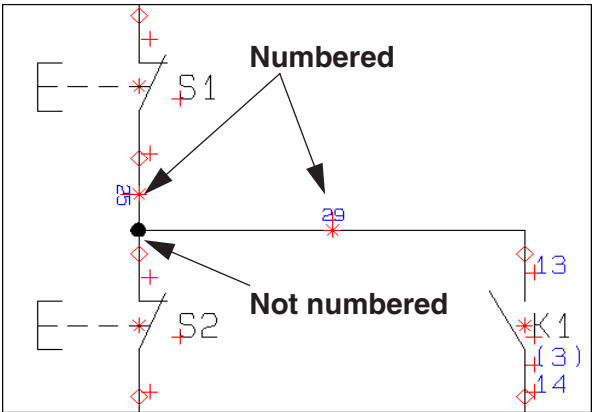
In an electrical potential with for instance three connected symbols, two lines will be included. The program will then make a guess, that the shortest polyline represents both wires. Please bear this in mind when you draw diagrams.

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**.
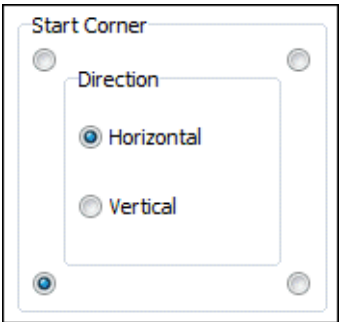
Function

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 *NOT* 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 *POSITION* ON THE DIAGRAMS, BUT ALL WIRE NUMBERS ARE RENUMBERED.

FUNCTION	ACTION
RENUMBER EXISTING	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.
REMOVE EXISTING AND REPLACE	ALL EXISTING WIRE NUMBERS ARE REMOVED COMPLETELY, AND NEW WIRE NUMBERS ARE PLACED. THE EARLIER POSITION OF THE WIRE NUMBERS ARE NOT REMEMBERED.
REMOVE EXISTING	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
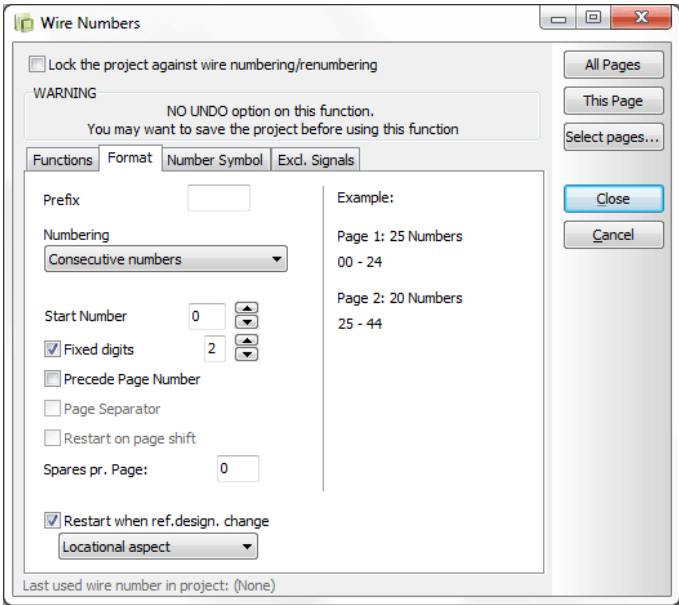
Here you determine the numbering principles for the wire numbers.

FORMAT	ACTION
PREFIX	HERE YOU CAN PLACE A PREFIX, WHICH IS PLACED AT THE START OF THE WIRE NUMBERS.
NUMBERING	ONLY FOR POTENTIAL NUMBERING: HERE YOU GET THE POSSIBILITY TO CHOOSE CONNECTED COMPONENT/CONNECTION. HEREBY THE WIRE NUMBER IS ASSIGNED BASED ON THE NAME OF A CONNECTION POINT AMONG ONE OF THE CONNECTED COMPONENTS IN THE ELECTRICAL POTENTIAL — FOR INSTANCE -Q1:2.
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.
RESTART WHEN REF.DESIGN. CHANGE	HERE YOU CAN SPECIFY THAT THE WIRE NUMBERING RESTARTS WHEN THE REFERENCE DESIGNATION CHANGES. CHOOSE EITHER LOCATIONAL ASPECT, FUNCTIONAL ASPECT OR FULL REFERENCE DESIGNATION.

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.
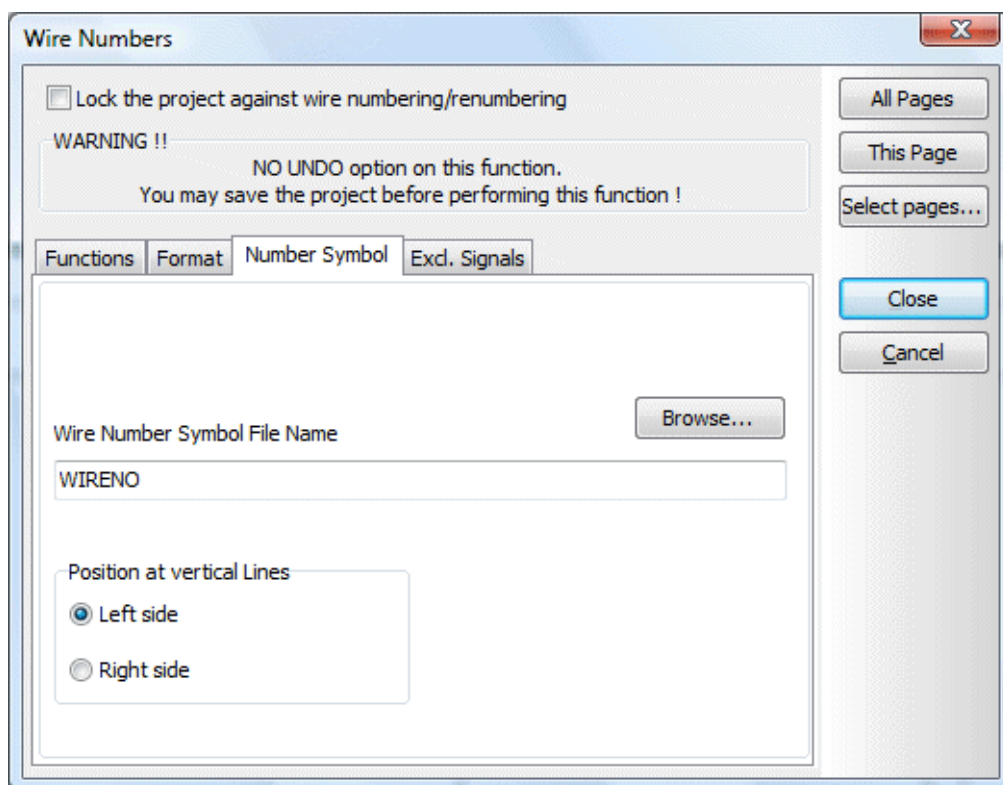
If you want to change which symbol is being used, you click on **Browse**, whereby you enter the **Symbol menu**. Here you click on the desired wire number symbol, and click **OK**.

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 775.

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.
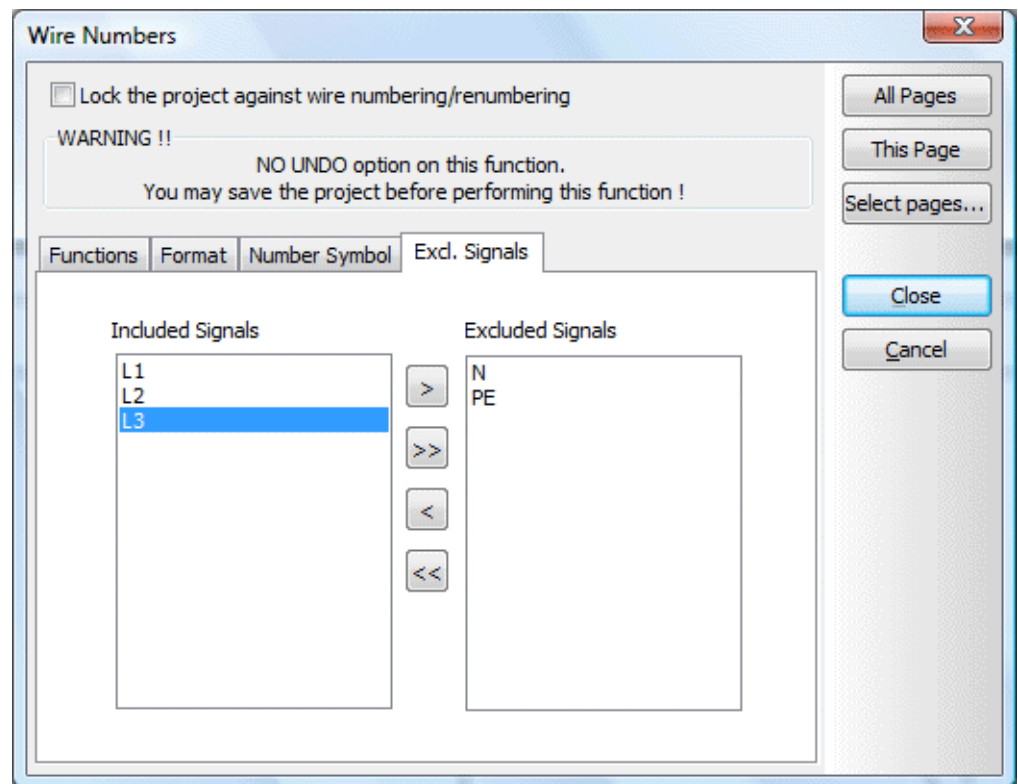
Select the signals you want to transfer and click on the **Transfer** button, or click on the **Transfer all** button to exclude all signals from the automatic wire numbering.

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. 464) — PLACING A WIRE NUMBER SYMBOL .... (P. 465) — AUTOMATICALLY INCREMENTING OF WIRE NUMBER NAMES .... (P. 465)

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 455.

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 203.

This speeds up the manual placing of the symbol.

Please refer to "Fetching symbols" on page 154 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:
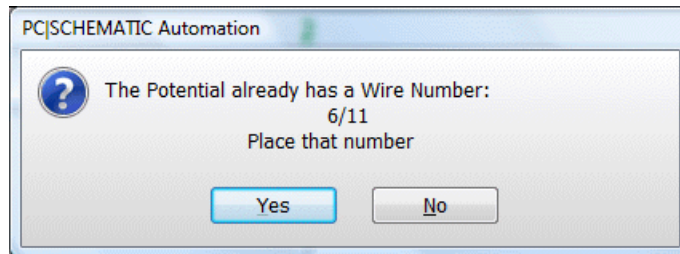
If you want to place another wire number, click **No**, and you enter the dialog box **Wire number data**.
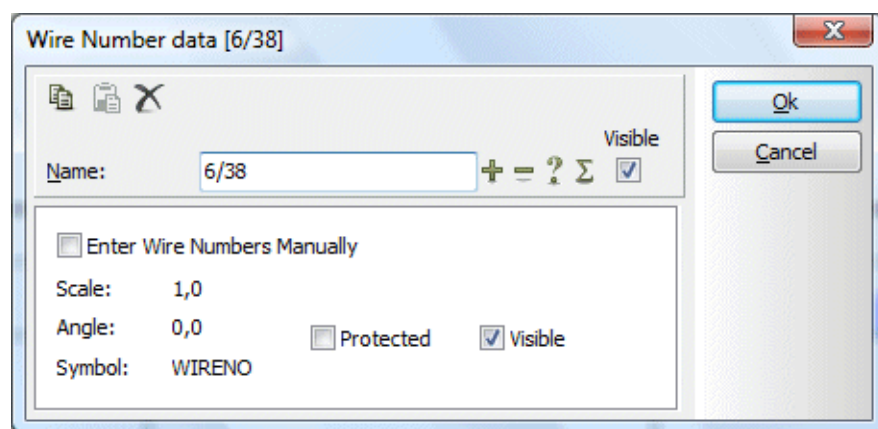
If *Enter Wire Numbers Manually* is *not* selected, the program automatically suggest the next available wire number — according to the wire numbering principle specified in **Functions => Wire Numbers**.

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 174.

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

When you want have your wire numbers saved in a file, you choose **Lists => Wire number file**.

You then enter this dialog box:
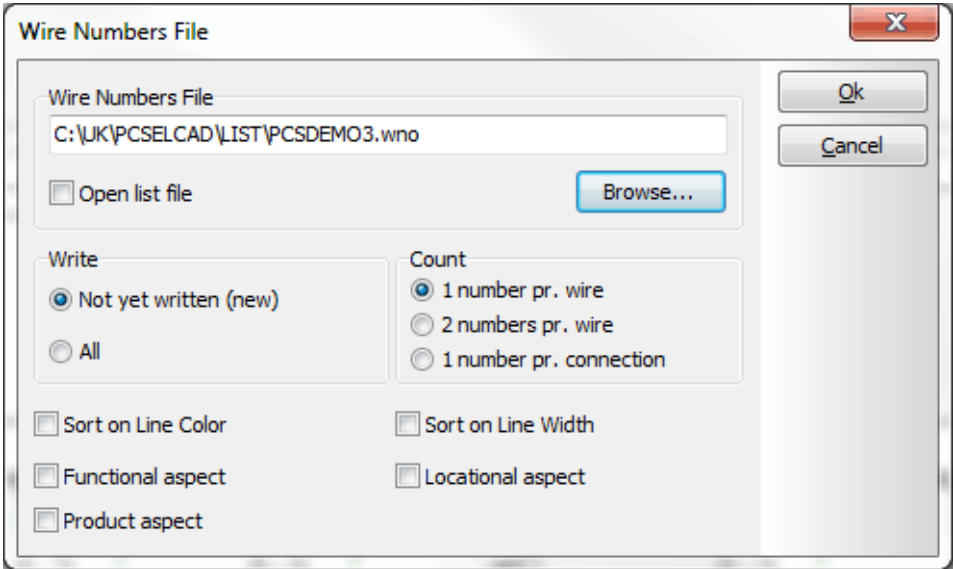
Here you have the following options:

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 acc. to Function, Location and Product aspect

If you use reference designations on wire numbers — see "Insert reference designations on signals" on page 617 —, you can also specify whether to include the *Function aspect*, the *Location aspect* and/or the *Product 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 => Connections List File**, and you enter this dialog box:
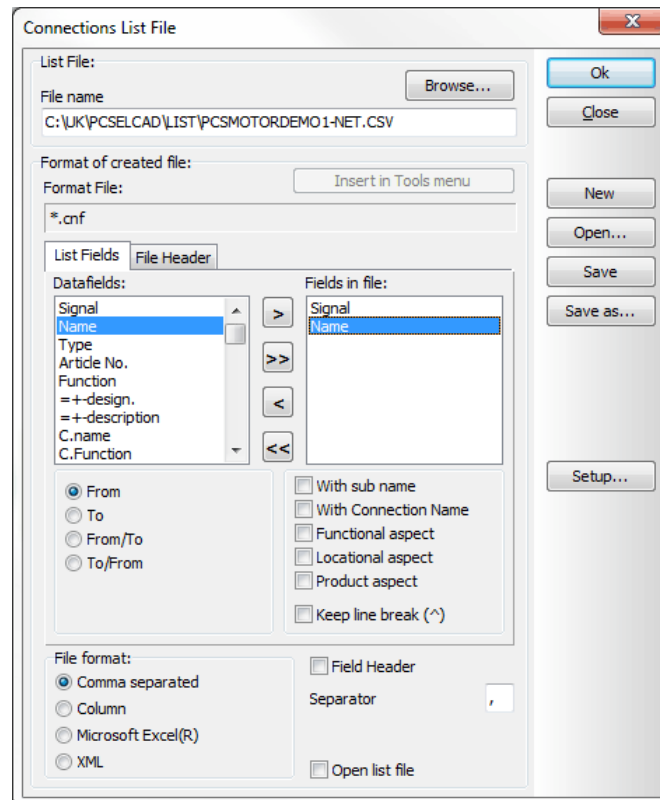
If you create a connections list file, which only contains component names with connections for all of the electrically connected components, just choose the datafield **Name** twice.

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

You can also select **Name** once and choose either **From/To** or **To/From**.

— Read more about the options in this dialog box in "Specifying the contents of the file" on page 735, and see "The buttons in the right-hand side of the dialog box" on page 734 also.
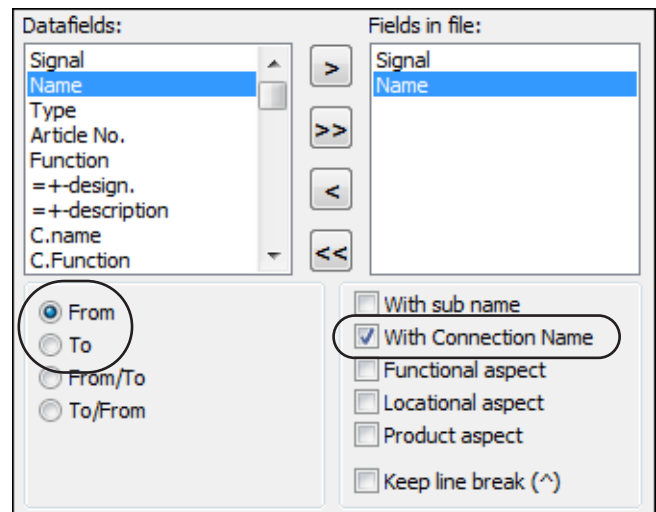
Settings for Connections lists for file

If you click on **Setup**, you enter the following dialog box:
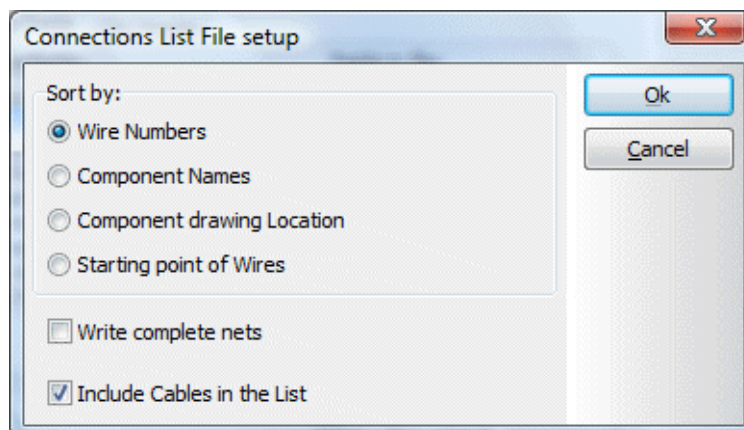
Here you have the following options for specifying how to sort the list:

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 PCSCHEMATIC A/S.
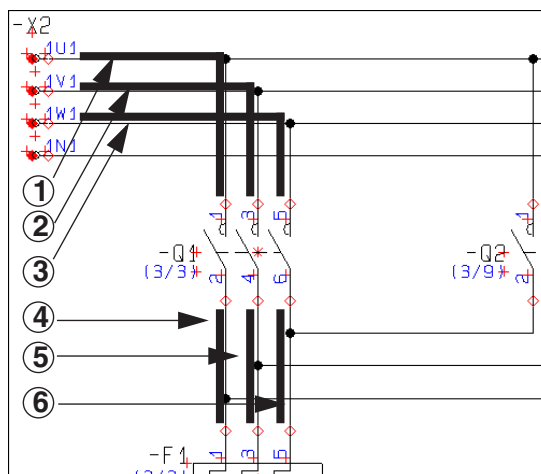
WRITE COMPLETE NETS

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 729.

SINGLE LINE DIAGRAMS

This section describes functions for single line diagrams:

PC|SCHEMATIC

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.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>

</td></tr>
</table>

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**.
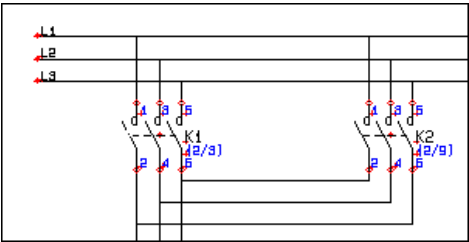
Figure 1
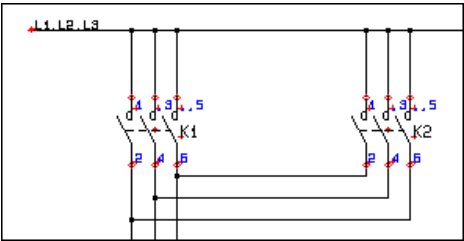
Figure 2

CONNECTION NAMES IN SINGLE LINE DIAGRAMS

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.
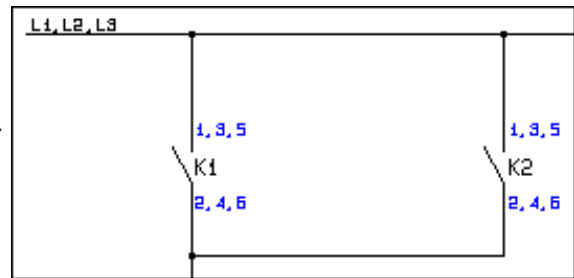
Figure 3

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.
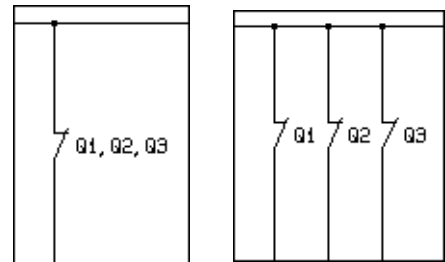
SYMBOLS FOR SINGLE LINE DIAGRAMS IN PICKMENU

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 664.

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 190.

— See how to create symbols with states in "Creating symbols with states" on page 791.

PLC FUNCTIONS

This section describes:

VIDEOS, TUTORIALS AND SURVEY ARTICLE

WATCH THE VIDEO SERIES ABOUT WORKING WITH PLCS ON OUR YOUTUBE CHANNEL AT www.youtube.com/PCSCHEMATIC.

READ THE SURVEY ARTICLE ON WORKING WITH PLCS AT WWW.PCSCHEMATIC.COM/ARTICLES.

AT WWW.PCSCHEMATIC.COM/MANUALS YOU CAN ALSO DOWNLOAD THE MINI MANUAL "PLC TUTORIAL", WHICH GUIDES YOU THROUGH CREATING A PLC PROJECT.

PC | SCHEMATIC

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 478.

2) Working with PLCs and sub drawings: Read the mini manual "PLC tutorial". Here you for instance learn how to use the function **PLC => Transfer PLC addresses** to transfer PLC addresses between PLC reference symbols and PLC I/O symbols, and the **PLC => Readdress PLC reference symbols** which gives you the option of creating addresses on multiple PLC reference symbols in a single operation. These options are not described further in this manual.

3) Exporting/Importing PLC I/O data: See "PLC I/O addressing via Excel" on page 500.

4) Controlling PLC related texts for other symbols: See "Text links" on page 147, particularly "Text links and importing PLC I/O lists" on page 151.

5) Editing PLC I/O data via the Object Lister: See "Typing in PLC I/O data via the Object Lister" on page 490 and "Editing PLC I/O data via the Object Lister" on page 359.

6) PLC I/O addressing via Excel: "PLC I/O addressing via Excel" on page 500.

7) Addressing PLC symbols via the database: See "Addressing PLC symbols via the database" on page 672.

8) Applying alternative PLC symbols: See "Alternative symbols for PLCs" on page 665.

9) Connection symbol for signal buses: See "Connection symbol for signal buses" on page 502.

WATCH THE PLC VIDEOS / READ THE "PLC TUTORIAL"

TO GET AN INTRODUCTION TO WORKING WITH PLCS, IT IS RECOMMENDED THAT YOU WATCH THE VIDEO SERIES ABOUT WORKING WITH PLCS ON OUR YOUTUBE CHANNEL AT www.youtube.com/PCSCHEMATIC.

YOU CAN ALSO 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

— Please also refer to "PLC I/O addressing via Excel" on page 500, in particular "Export of PLC I/O data to Excel" on page 500.

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 481.

PLC reference symbol

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:

I/O address for the node

Reference to the I/O symbol in the project (Double-click to jump to the I/O symbol)

Connection name on the PLC (connect-point on the PLC)

Another connection name (Here for the supply)
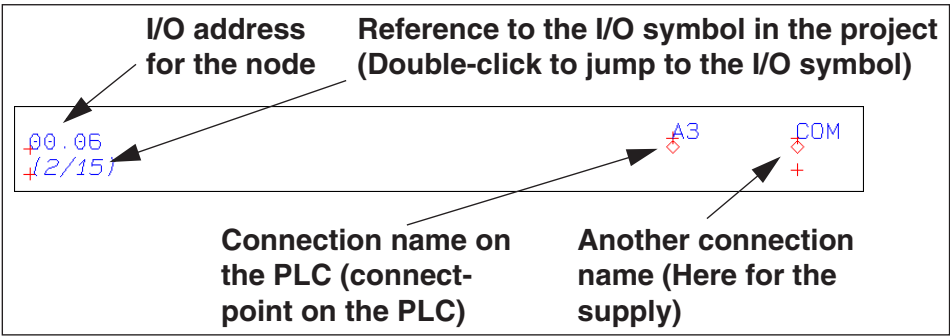
Each of these nodes on the PLC can be represented as individual PLC I/O symbols in the project drawings. You can also apply PLC symbols representing e.g. eight nodes.

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 486.

Creating PLC reference symbols

See how to create PLC reference symbols in "Creating Symbols" on page 775, in particular "Creating PLC reference symbols" on page 806.
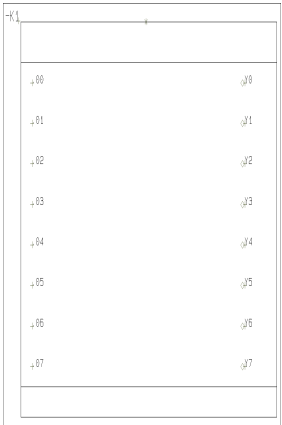
MORE PLC REFERENCE SYMBOLS FOR THE SAME PLC COMPONENT

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 486.

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 659, particularly "Addressing PLC symbols via the database" on page 672.

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

PLC addressed directly from the 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.

VIDEO: CREATE YOUR OWN PLC COMPONENTS IN THE DATABASE EASILY

TO SEE HOW TO EASILY CREATE YOUR OWN PLC COMPONENTS WITH ATTACHED SYMBOLS IN THE DATABASE, PLEASE WATCH THE PLC VIDEOS IN THE "CREATE YOUR OWN COMPONENTS EASILY" PLAYLIST ON OUR YOUTUBE CHANNEL AT WWW.YOUTUBE.COM/PCSCHEMATIC.

THERE IS ALSO A "CREATE YOUR OWN COMPONENTS EASILY (COMPONENT WIZARD)" TUTORIAL, WHICH YOU CAN DOWNLOAD AT www.pcs-chematic.com/manuals.

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 659 and "Addressing PLC symbols via the database" on page 672 —, 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

- Press <d> to enter the database and select a PLC, 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 can now see the symbols for the PLC in the **Symbols for component** window in the lower-right corner of the screen:

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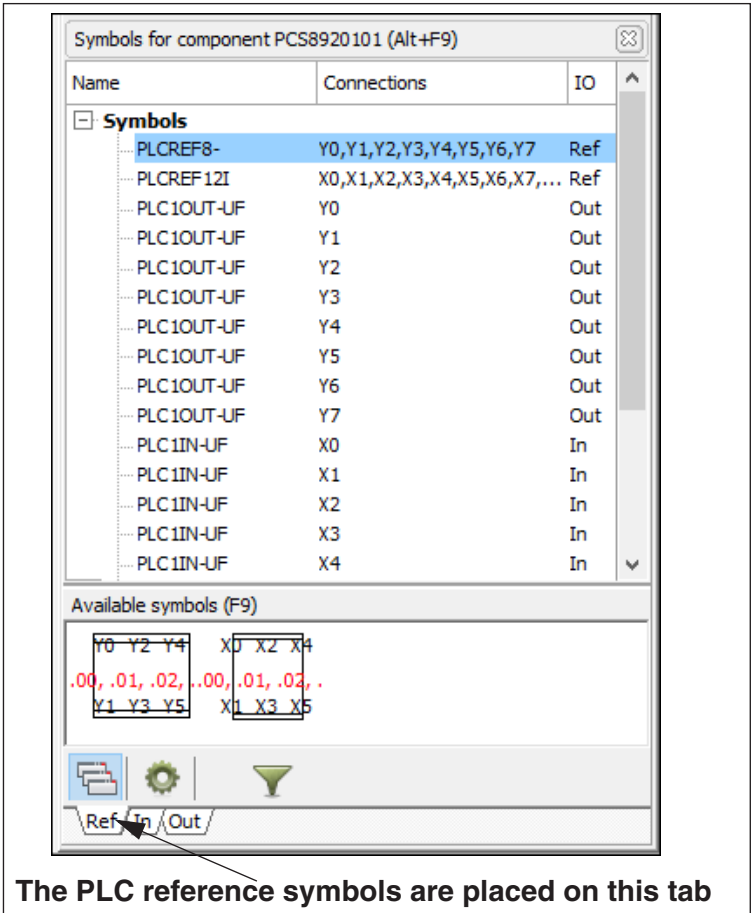
- Choose the first of the PLC reference symbols (in the figure above: the reference symbol for the outputs), place it on a project page, and assign a name to the component in the dialog box **Component data**.

The reference symbol is now placed on the project page:
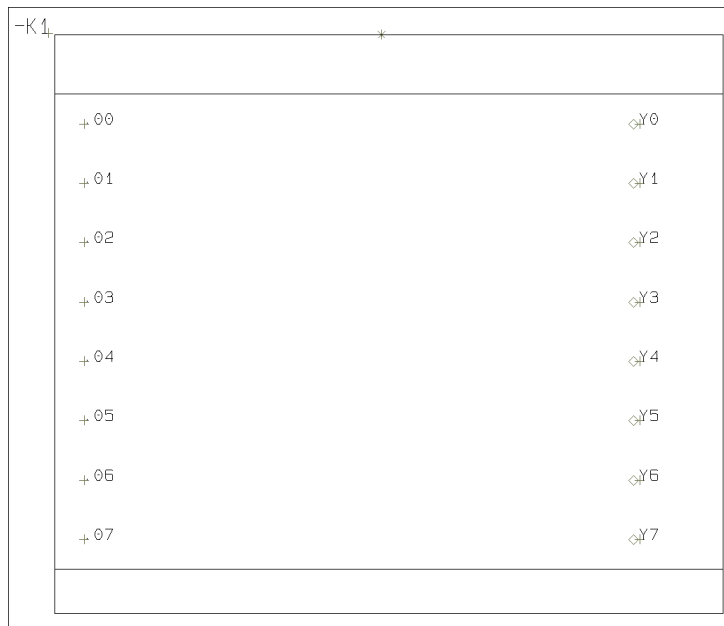
At the same time the PLC is included in the **Components** window in the upper-right corner of the screen (If necessary, see "The Component menu" on page 263). Here you will always be able to find the PLC in the project:
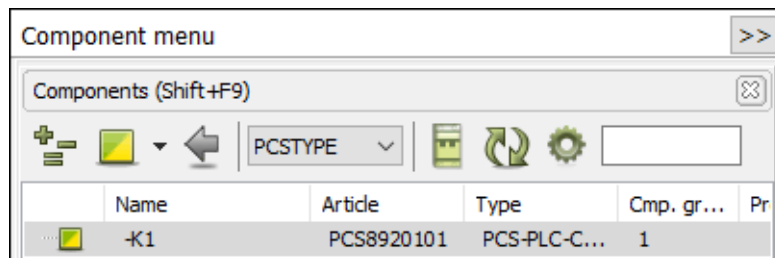
- From the PLC reference symbol, you can now control all data for the PLC in the project. In each individual line in the PLC reference symbol, you can see information about an individual node in the PLC, represented by I/O symbols for the PLC in the project — see "Each line in the PLC reference symbol represents a node" on page 478.

2. Loading the PLC I/O list
- 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 497. Load e.g. the file **Pcsexcelio.xls** in the **Lists** folder (apply the format file *ExcelIO.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 490. (Please also read "Re-addressing PLCs" on page 486).

3. Placing PLC I/O symbols in projects
- You can now place the I/O symbols for the PLC in the project: Click on the tab for the output symbols under **Available symbols** and select an I/O symbol — or select the symbol in the list in the upper part of the **Symbols for compo-**

nent window — and place it in the project.

Under **Available symbols** the address for each symbol is written in red. Connection names are displayed as for other symbols. — See more about PLC symbols in **Available symbols** in "Filtering and other special options for PLCs" on page 270.

- Now place the remaining PLC I/O symbols.

Other options for working with PLCs

There are a couple of options under **Available symbols**, which can be handy when working with PLCs containing many I/O symbols:

Place all

You can right-click on an I/O symbol and choose **Place all**. This makes it possible to for instance place all outputs on the diagram in a single operation.

If page/current path numbers are applied, the symbols are placed with a spacing that corresponds to the distance between the current paths, and there is placed no more than one symbol for each current path on the page.

Stack similar symbols

When working with PLCs with many I/O symbols, you can choose to click the **Stack similar symbols** button. This way, similar symbols are displayed as a single

symbol along with a specification of how many symbols it covers (12 in the example below).
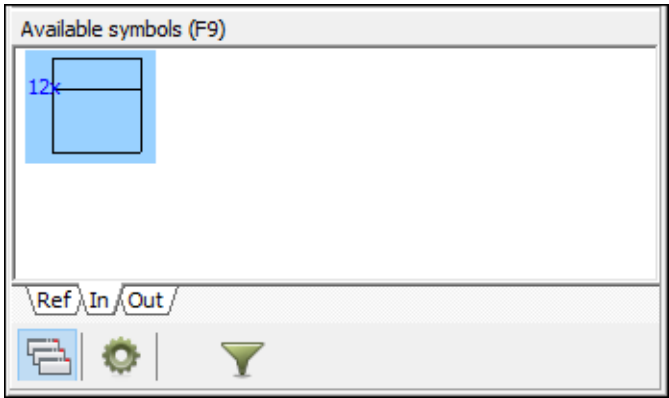
When you click the "stacked" symbol, a list containing all the names, that you can choose the individual I/O symbols from, is displayed. You also get the option of choosing **Place all**.
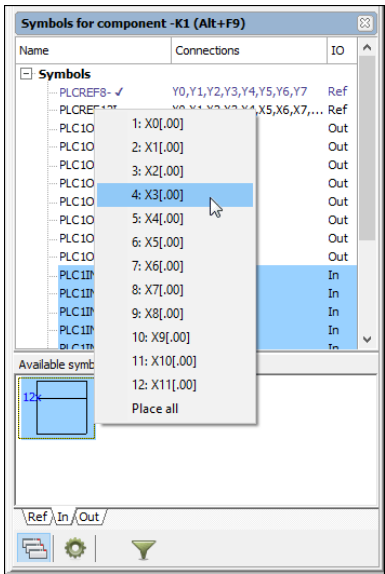
Filter on addresses

When you click the **Filter** button under **Available symbols**, you can specify an interval of addresses for which of the symbols for the component to display — for instance from *O.02* to *O.07*:
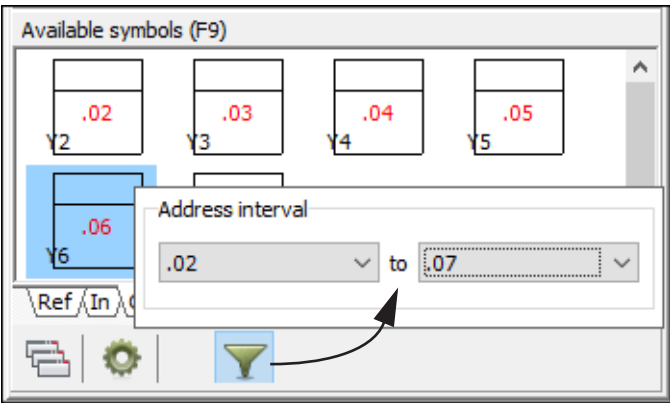
Please also refer to:

- "More PLC reference symbols for the same PLC component" on page 479.

- "Re-addressing PLCs" on page 486.

- "PLC data on the I/O symbol connection points" on page 488.

- "On PLC reference symbols" on page 478.

- 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.pcs-chematic.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**:
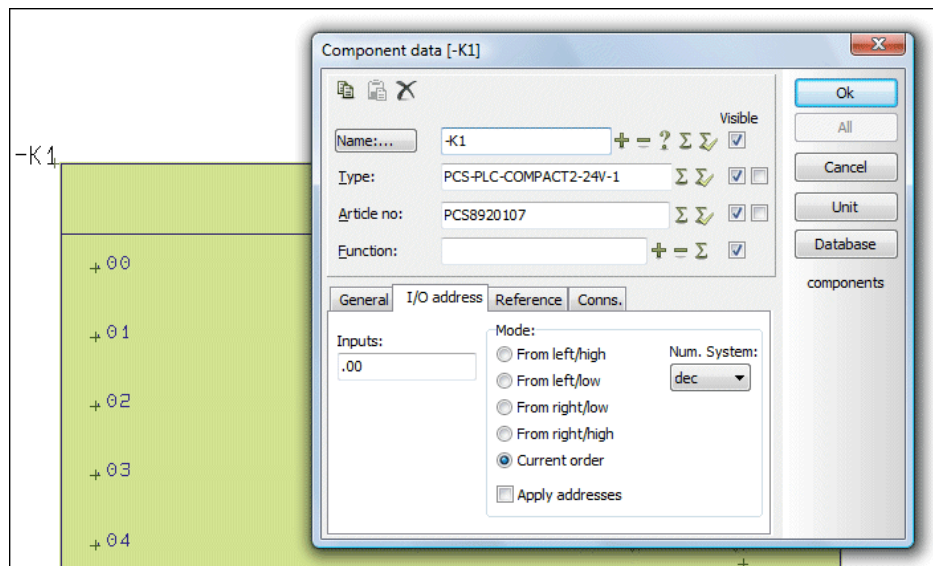
2) Type in a new start address for the PLC in the left-hand side, click on **From left/high** — or pick another corner to start numbering from —, choose **Number system**, and click **OK**. (See "Options on the tab I/O address in the Component data dialog box" on page 486).

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.

Exporting PLC I/O data

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 PLCs 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, and see PLC data for the node

Or: Right-click on the symbol, and choose PLC I/O data ...

... and edit data via the Object Lister

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 sclick **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.

PLC connection points in terminal lists

In the dialog box **Connection data** you can specify whether the connection point is included in terminal lists by selecting **On terminal list**.

PLC I/O connection and I/O status type

You can also specify whether the connection point is an I/O connection by selecting **Is I/O connection**, and specify whether the **I/O direction** is **Input** or **Output**.

The **I/O status type** can be set to **Analog**, **Digital** or **Generic**. If you choose **Generic**, you can then choose any of the PLC I/O status types, that you have created under **PLC => Define PLC I/O status types**.

> **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 486.

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 490.

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 PLCS
IF YOU WISH TO RE-ADDRESS A PLC, PLEASE READ "RE-ADDRESSING PLCS" ON PAGE 486.

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.
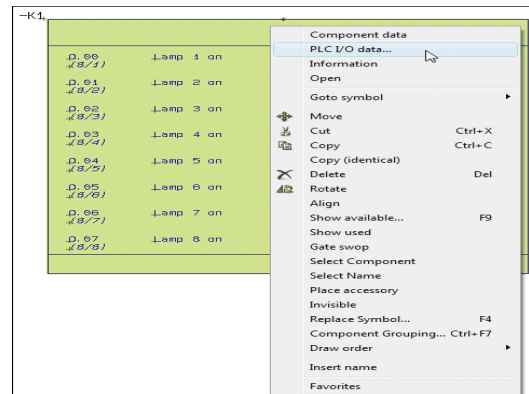
3) In the **Object Lister**: Click on the **Pencil**, double-click on the relevant fields (or click and press <F2> / <Enter>), and type in the new data.
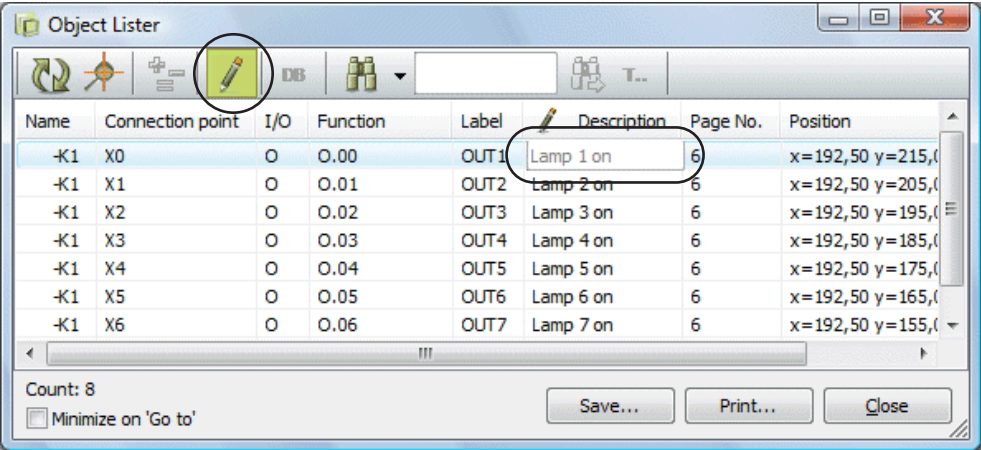
4) Click **OK**, and the change(s) are transferred to the PLC.

> **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.

Other PLC options in the Object Lister

When you choose **PLC => List PLC I/O symbols** you enter the **Object Lister**, where all of the PLC I/O symbols in the project are displayed.

When you choose **PLC => List PLC reference symbols** you enter the **Object Lister**, where all of the PLC reference symbols in the project are displayed.

Read more about the **Object Lister** in "The Object Lister" on page 351. 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 497 also.

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:

PLC reference symbol

PLC I/O symbol
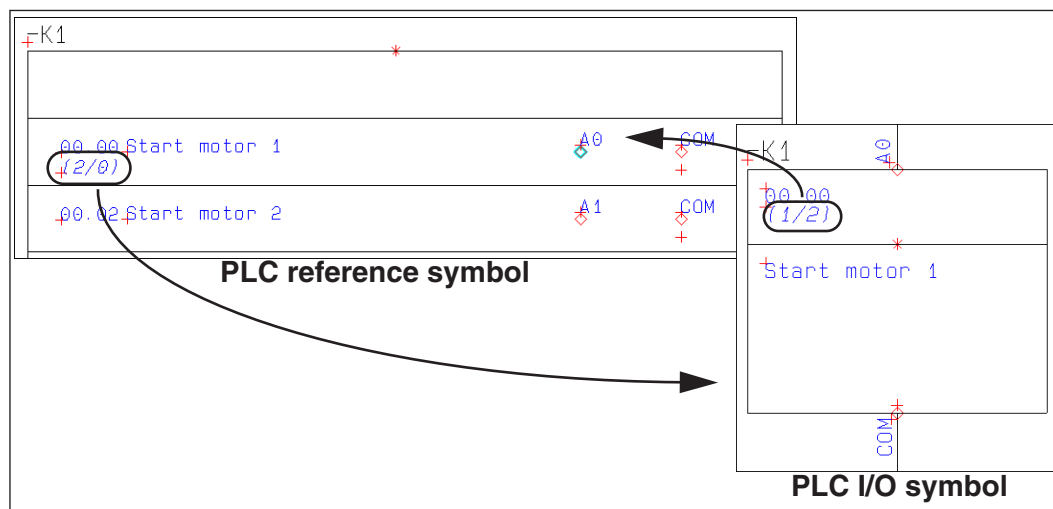
— Read more about references in "Automatically updated references" on page 433.

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 PLCs TO THE DATA-BASE. 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 479.
>
> IT IS ALSO RECOMMENDED TO USE THE BUILT IN **COMPONENT WIZARD** FOR CREATING PLCs IN THE DATABASE. THE MANUAL FOR THE COMPONENT WIZARD MANUAL CAN BE OPENED AT **HELP => COMPONENT WIZARD MANUAL** AND CAN BE DOWNLOADED AT www.pcschematic.com/manuals.

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

5a) 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:

PLC reference symbol for inputs   **PLC pickmenu**

PLC reference symbol for outputs
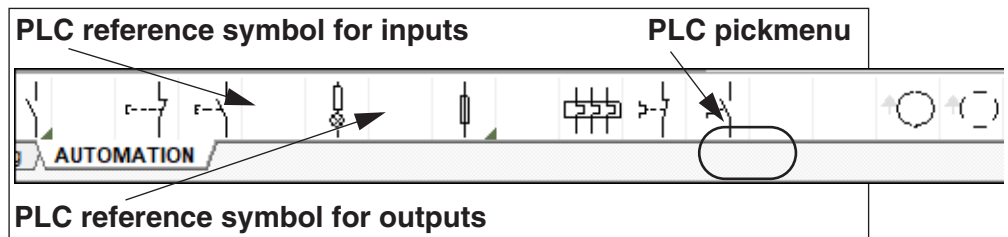
— If you need another already created PLC reference symbol, these can be found via the **Symbol menu** in the folder **PLC346**, where the PLC reference symbol names starts with *PLCRef*.

5b) 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.

5c) Then address the PLC reference symbol(s) as described in "Re-addressing PLCs" on page 486, so that the addresses matches this PLC.

2. Type in data for the PLC

5d) Type in data for the PLC, as described in "Typing in PLC I/O data via the Object Lister" on page 490.

3. Place the PLC I/O symbols and choose PLC I/O data

- Place the PLC I/O symbols for the PLC by selecting them in the Automation pickmenu at the upper part of the screen:

PLC I/O symbol for output

PLC I/O symbol for input
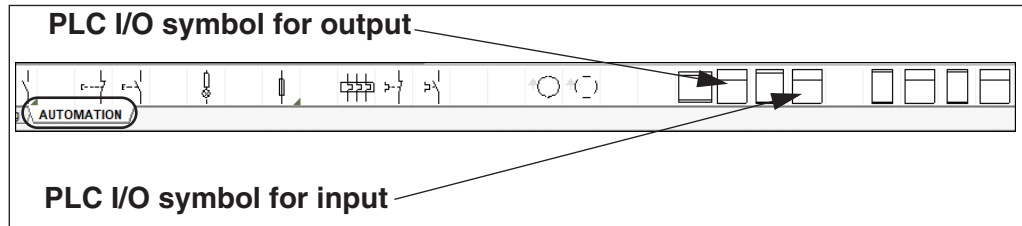
Select an I/O symbol, and click to place it on the project page.
— If you need another already created PLC I/O symbol, these can be found via the **Symbol menu** in the folder **PLC346**.

- When you place the symbol you enter the dialog box **Component data**. Here you click the **I/O addr** button in the lower-right corner of the dialog box.

- You then enter the dialog box **Inputs** (or **Outputs**):

1) Select component

Used Inputs are marked by a + (if any)

2) Select input/output
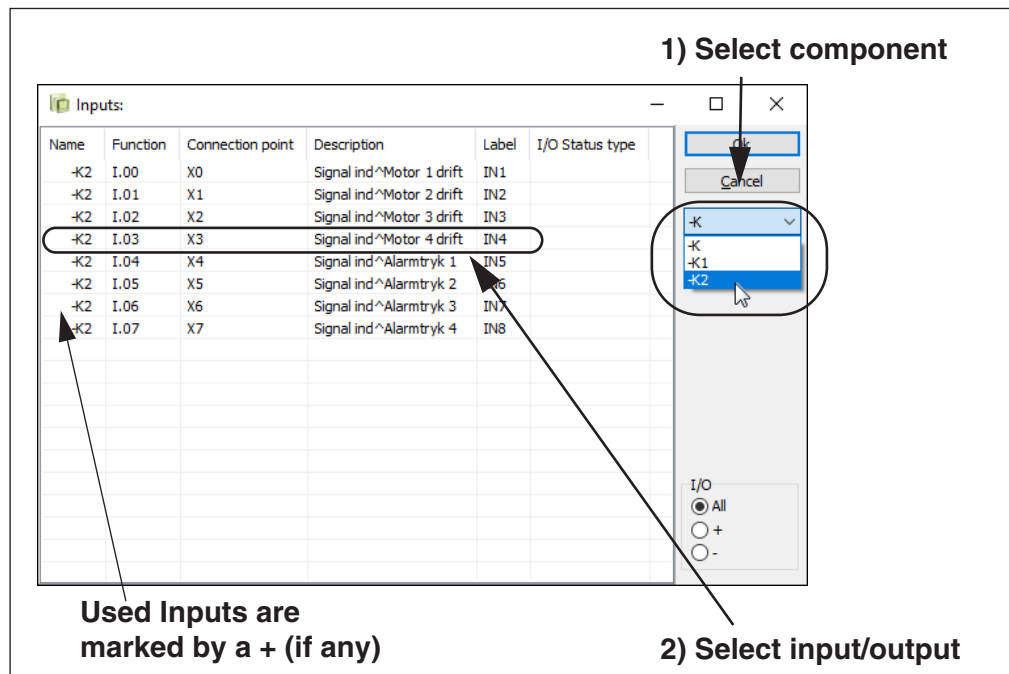
- Select the desired PLC in the right-hand side of the dialog box, and then select the desired input (or output) in the list, and click **OK**.
Please note that used inputs/outputs are marked by a preceding **+**.
Under **I/O** in the lower-right corner of the dialog box you can choose either to show all inputs and outputs in the list (**All**), to show all used inputs and outputs (**+**) or only to show the available inputs and outputs (**-**).

CHOOSE INPUTS/OUTPUTS FOR ALL PLCS IN THE PROJECT

IF YOU CHOOSE -K IN THE **NAME** FIELD IN THE RIGHT-HAND SIDE OF THE DIALOG BOX, INPUTS/OUTPUTS FOR ALL PLCS IN THE PROJECT WITH NAMES BEGINNING WITH -K ARE DISPLAYED.

PCSCHEMATIC Automation

- 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:
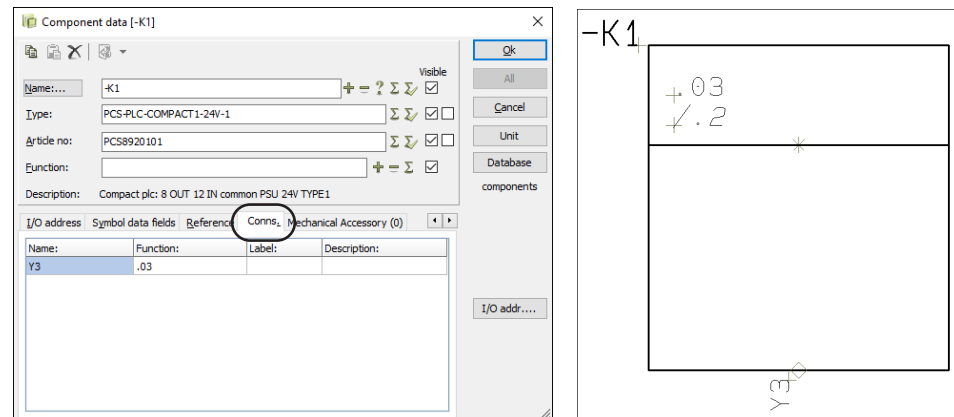
- Continue like this, until you have placed the relevant symbols.

Please also refer to:

— "Workflow for PLCs from the database" on page 481.

— "Re-addressing PLCs" on page 486.

— "PLC data on the I/O symbol connection points" on page 488.

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:

- **PLC => Transfer data from PLC reference symbols to PLC symbols**, or

- **PLC => 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 803.

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 221.

PC|SCHEMATIC

3) Create PLC reference symbols for all PLCs in the project. — See "Creating PLC reference symbols" on page 806.
(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 680.
Because the PLCs most likely are already addressed in the existing project, it is not strictly necessary to address the PLCs 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 479.

5) Right-click on one of the PLC I/O symbols in the project.

6) In the **Component menu** under **Available symbols**: 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.

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.

Importing symbol datafields for PLCs

If you have created your own symbol datafields for the PLCs in the project, you can also import data for these symbol datafields, when importing a PLC I/O list. If datafields appear in the imported PLC I/O list, which are not created for the PLCs in the project, these datafields will not be imported.

— Read more about this in "Symbol datafields" on page 122.

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**.
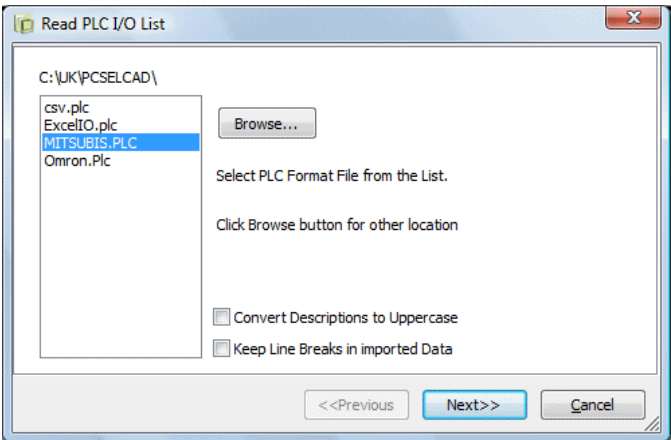
Click **Next**.

— For further information about format files, please refer to "Format files" on page 730.

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 off 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**.
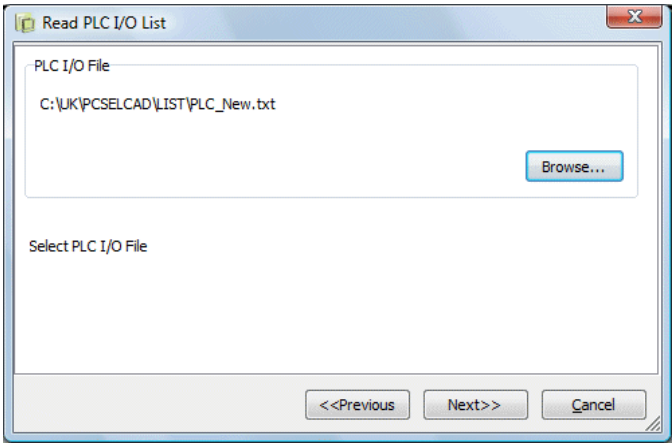
THE CONTENTS OF THE FILE

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.
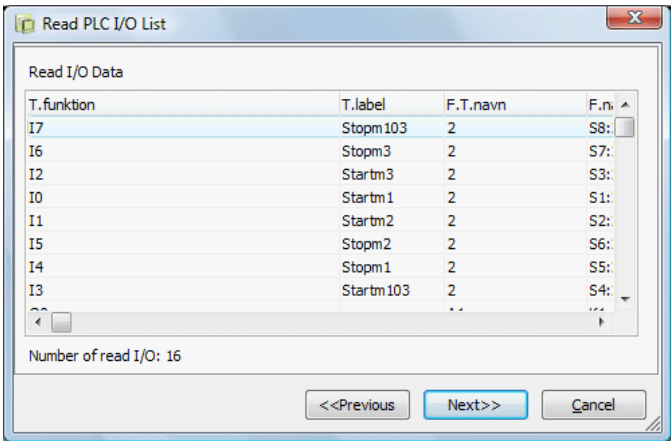
> **CONNECTION NAMES AS PART OF PLC NAMES**
> DURING LOAD OF PLC I/O LISTS, THE CONNECTION NAMES OF THE PLC I/O'S CAN BE A PART OF THE PLC NAME, SO THAT YOU DO NOT NEED A SEPARATE COLUMN FOR CONNECTION NAMES IN THE PLC I/O FILE. (E.G. **K1:1**).

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:
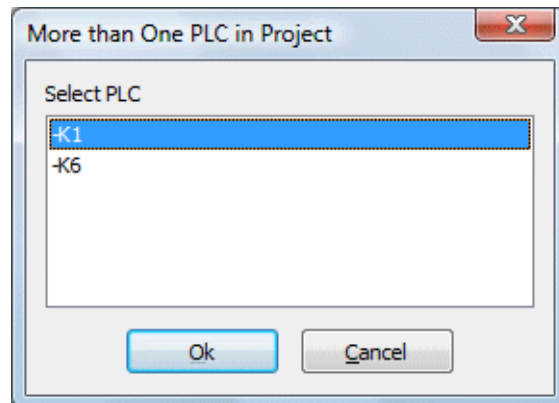
Click on the PLC, and click **OK**.

DISPLAYING THE CHANGES IN RELATION TO THE PROJECT

It is then displayed which changes the PLC program-mer have made, compared to the information in the pro-ject.

When there are ## charac-ters 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 esti-mates 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 col-umn 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.
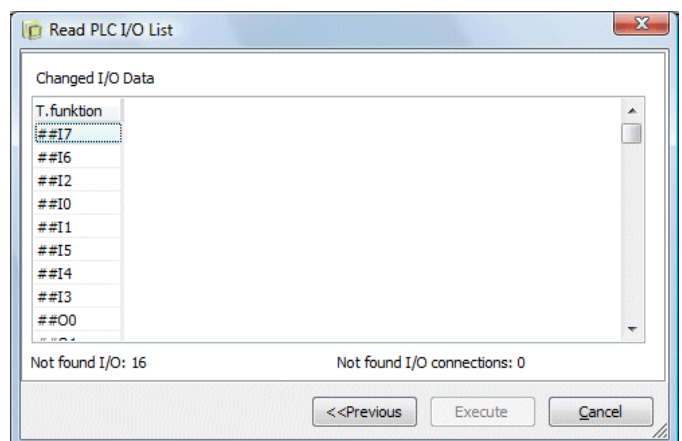
PLC I/O ADDRESSING VIA EXCEL

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

For further details about the options in the individual dialog boxes, please refer to "Reading PLC I/O lists" on page 497.

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 747.

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**:
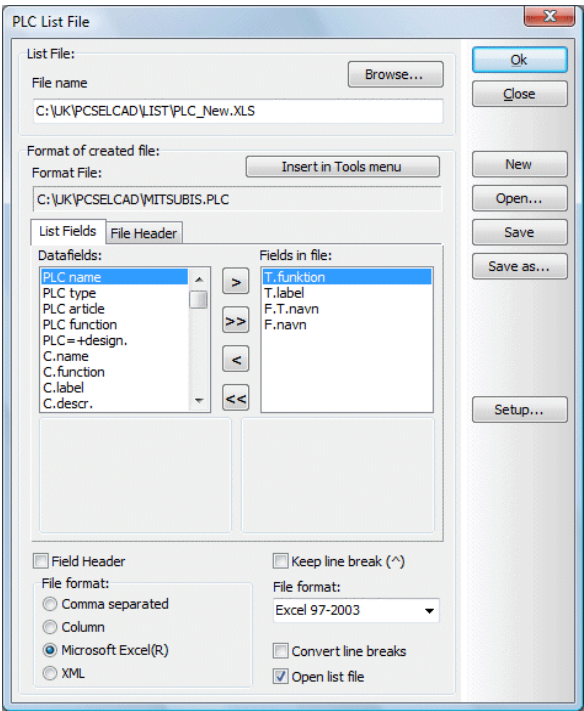
2) Click on **Open** — to open a format file specifying which information to include in the generated file. (You can also specify this information in the dialog box manually).

3) In the dialog box **Open**: Select the format file *ExcelIO.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 PLCs — 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

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 819 to see how to create join symbols.
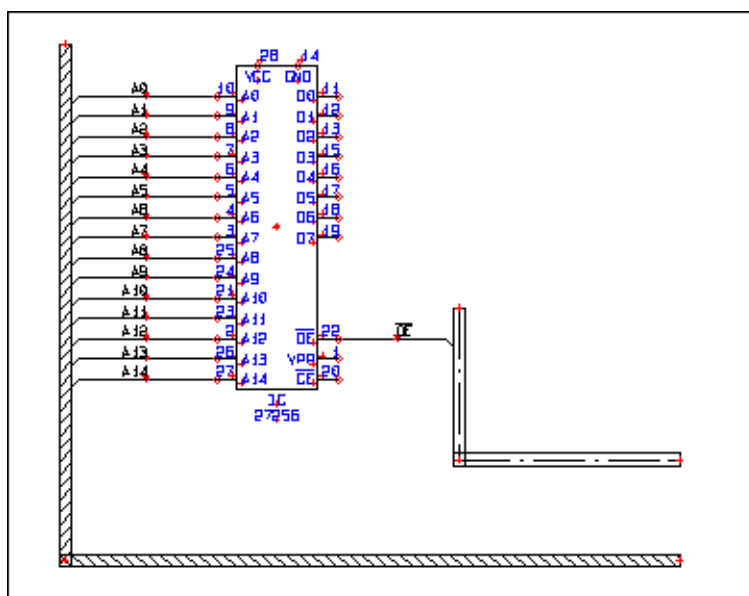
To specify a signal bus as displayed above, do this:

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** — or right-click on the line and choose **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:
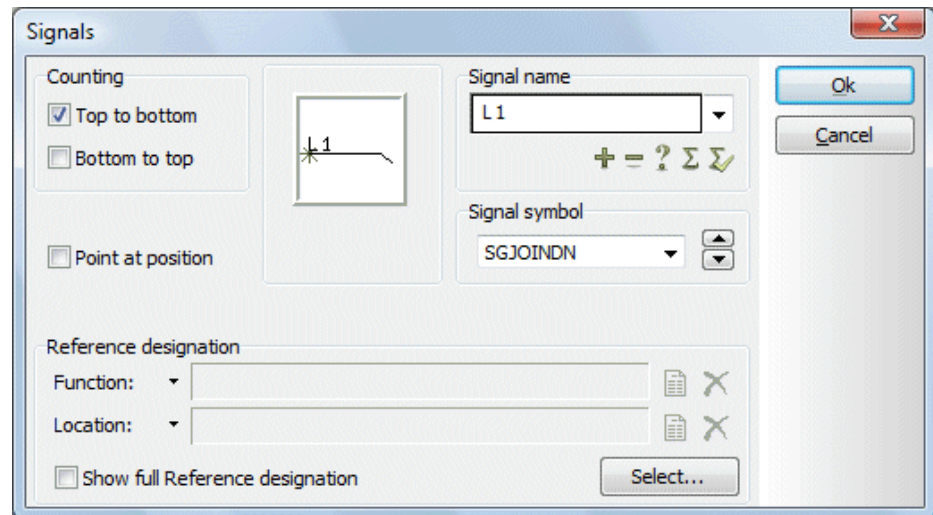
9) Below *Signal name* you either type in a name, or specify the name using the buttons under the *Signal name* field, as described in "Specify Signal name" on page 236.
If you click on the up- or down-arrow in the right hand side of the dialog box, you can choose between the signal symbols, that are 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

PC|SCHEMATIC

INDEX FOR PART 1 AND PART 2

Automation

Automation

PC | Automation™

PART 2

April 1st 2020

PC|SCHEMATIC

TABLE OF CONTENTS

PART 1

Automation

PC|SCHEMATIC

PART 2

Automation

ELECTRICAL INSTALLATION DRAWING

This section deals with the functions in PCSCHEMA-TIC Automation, which are specially designed for electrical installation drawing. Following topics are elaborated:

If you want to document small switchboards and panels with heat loss calculation according to EN61439-3, 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.

You can also see how to make House installation drawings on our YouTube channel at www.youtube.com/PCSCHEMATIC.

OTHER RELEVANT CHAPTERS

PC | SCHEMATIC

CREATING A GROUND PLAN

This chapter is an example on how to draw a ground plan for a house.

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 821. Click **OK**.

Choose **Settings => Pointer/ Screen** and set the unit of measurement to meters (**m**).

Click **OK**.
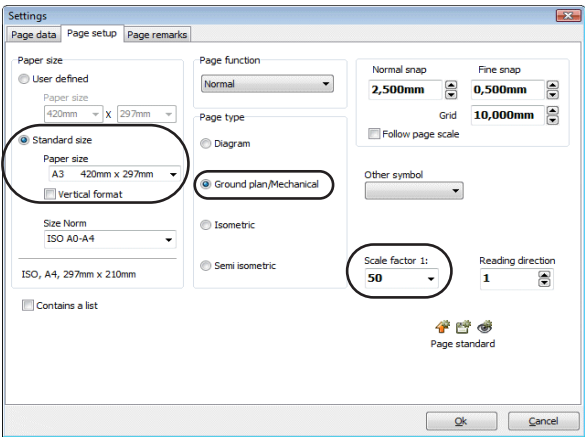
> **THE PAGE FUNCTION MUST BE SET TO NORMAL**
> TO USE A PAGE FOR A GROUND PLAN DRAWING, THE **PAGE FUNCTION** FOR THE PAGE MUST BE SET TO **NORMAL**.

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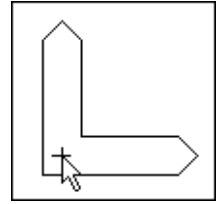
x=82,50 y=117,50mm	x=37,50 y=20,00mm

The coordinates field

3) This dialog box will appear:

Coordinates(mm)

Last X	Last Y	Last Z
0,00mm	0,00mm	0,00mm

X	Y	Z
0,00mm	0,00mm	0,00mm

- ⦿ Absolute
- ◯ Relative
- ◯ Polar

Ok

Cancel
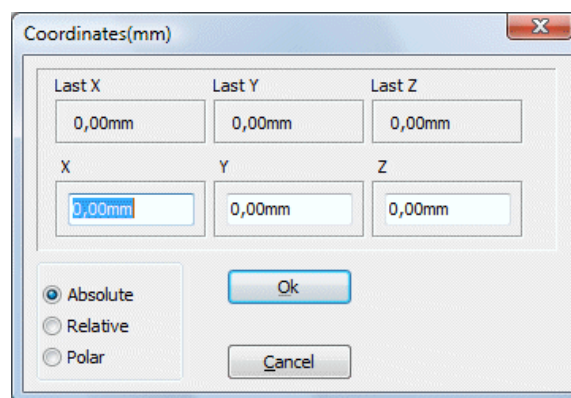
4) Select **Absolute**. Set **X** to *5* and **Y** to *5*. You do not have to type in the unit of measurement *m*, because you have already told the program that you are working in meters

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.
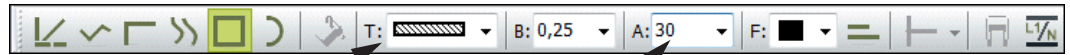
Choose this Line type first   **Then type in this Line distance**

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 97.

DRAWING THE WALLS OF THE HOUSE

You can now draw the walls in the house you see below:
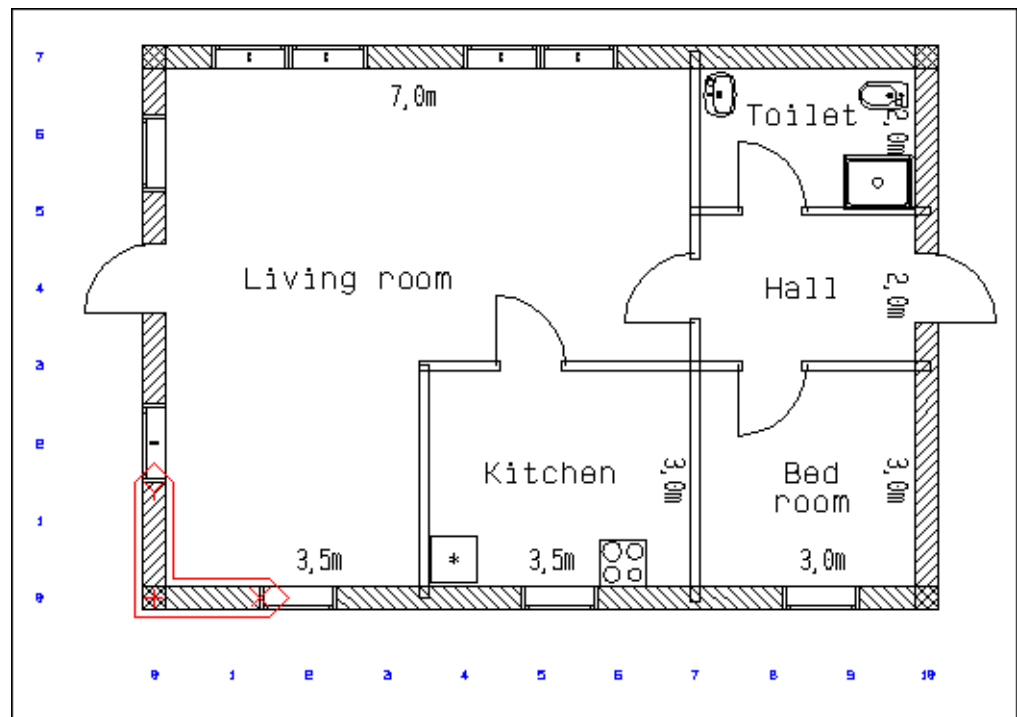
1) Click the **Lines** button, set the line type (**T**) according to the instructions above, and pick an appropriate line distance (**A**)

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 100.

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 COR-NER 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.

Click here
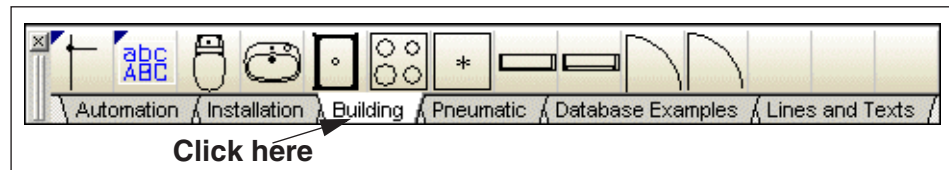
From here you can pick the symbols you wish to place on the drawing.

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 152.

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 541.

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.

Click here
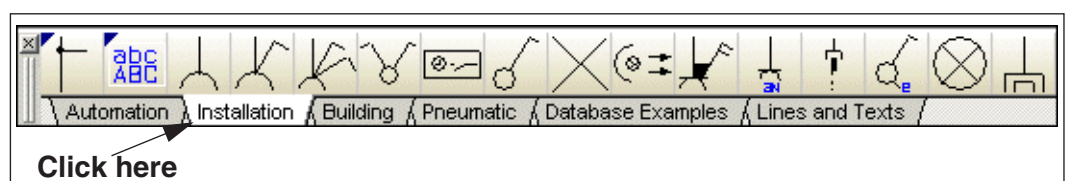
USING THE GRID FOR HELP WHILE DRAWING

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.

x=500,00 y=850,00cm	x=0,00 y=500,00cm

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.
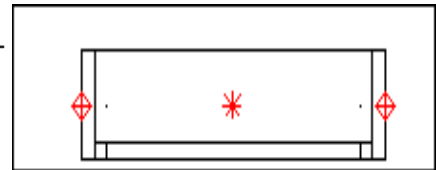
The symbol must be saved as a **Non conducting** symbol. The **Selection** must be set to **Whole symbol**, when you save the symbol. — For a deeper understanding of this, please read "Creating Symbols" on page 775.

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.

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 791.

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:
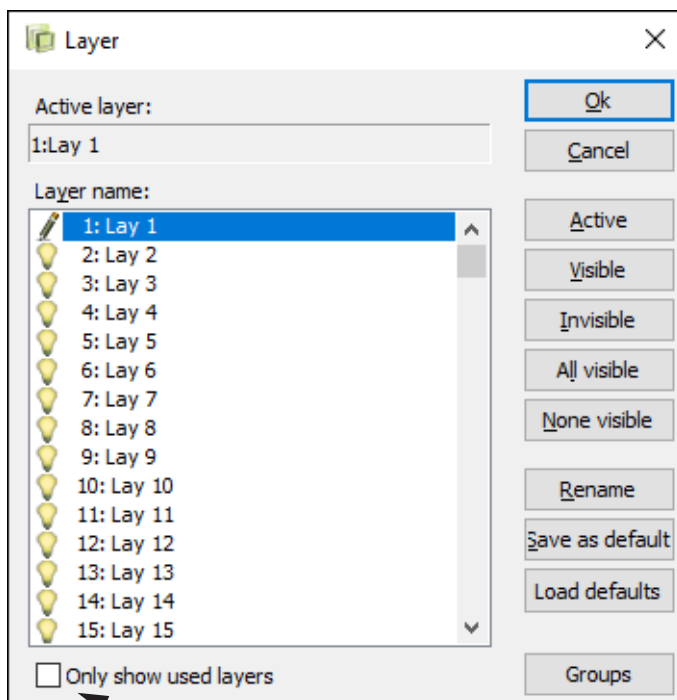
Click here if you only want to see layers with contents

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 light bulb that 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:
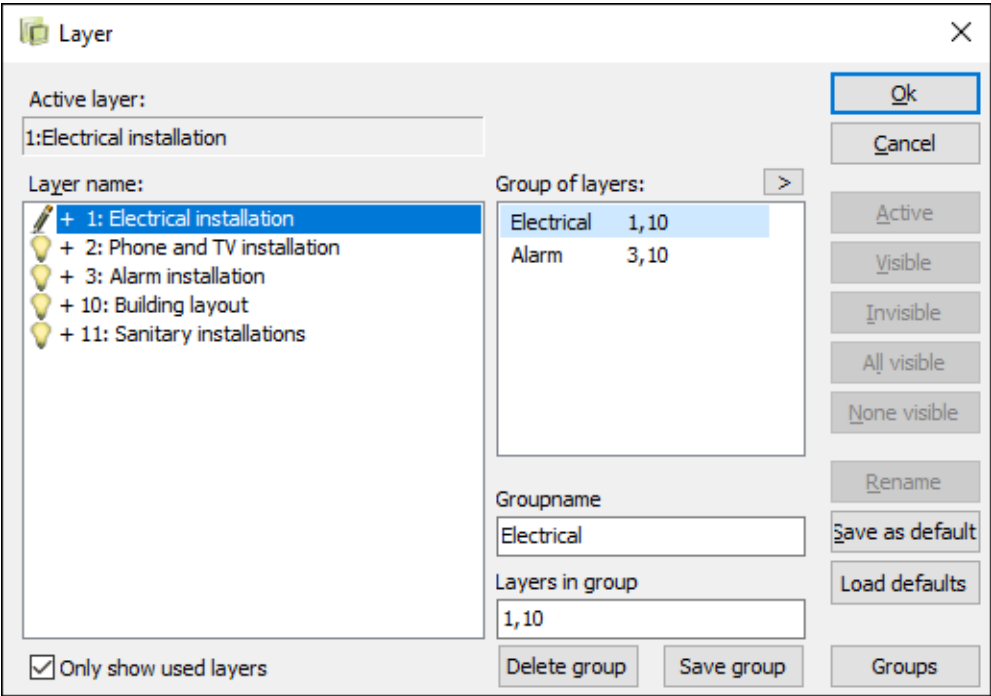
Creating a group

    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 374.

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.

PCSCHEMATIC Automation ✕

❓ Object is on layer 10. Change to this layer ?

[Yes] [No] [Cancel]
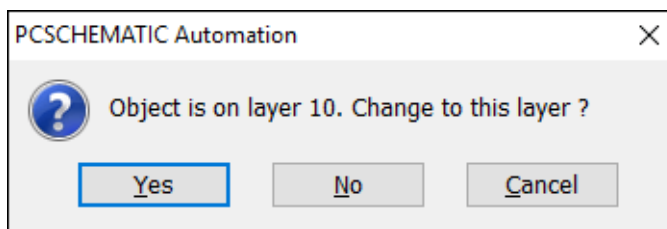
Answer **Yes**, and you will change layer.

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 371.

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.
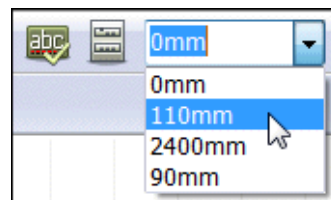
Selecting objects in another elevation

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.

Changing elevation on selected objects

To change elevation on an object, do this:

1) Select the object

2) Click the **Copy properties** button.

3) Select / type in the desired elevation in the **Elevation** window

4) Click the **Transfer properties** button

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.
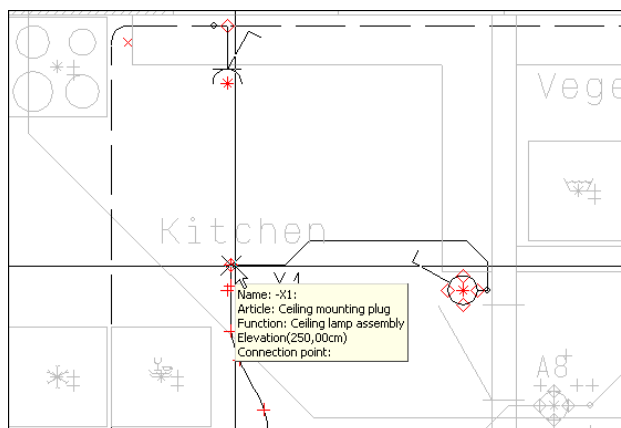
Lines changing elevation

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.
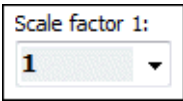
There is no reason to scale on diagram pages, because you can choose to draw on a page with the A3 page size, and make the program scale automatically to fit the paper size when printing out the diagram. This kind of settings are set in the **Printer** dialog box, and you can read more about automatic scaling in "Setup" on page 374.

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* .

THIS SETTING ONLY APPLIES FOR THE SELECTED PAGE(S) IN THE PROJECT.

PC|SCHEMATIC

EXAMPLE	PLEASE NOTE
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 787.
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 290.

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 533.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— GROUND PLAN DRAWINGS AND DWG/DXF FILES ....</td><td>(P. 549)</td></tr>
<tr><td>— PRECISE POSITIONING OF MECHANICAL SYMBOLS ....</td><td>(P. 549)</td></tr>
<tr><td>— DRAWING CONNECTIONS ON INSTALLATION DRAWINGS............................</td><td>(P. 549)</td></tr>
<tr><td>— CHOOSING CABLES AND WIRES IN THE PICKMENU ....</td><td>(P. 550)</td></tr>
<tr><td>— INSTALLATION SYMBOLS IN THE PICKMENU WITH ELEVATION ....</td><td>(P. 550)</td></tr>
<tr><td>— DRAWING CONNECTIONS ON INSTALLATION DRAWINGS............................</td><td>(P. 550)</td></tr>
</table>

"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.

AT www.youtube.com/pcschematic YOU CAN WATCH IT ON VIDEO.

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 535. Do you need to import DWG/DXF files, please read "DWG and DXF files" on page 741. Please also refer to "Electrical installation drawing" on page 533.

PRECISE POSITIONING OF MECHANICAL SYMBOLS

Se how to position mechanical symbols precisely in "Magnetic snap and Snap in objects for mechanical symbols" on page 715.

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 546.

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.
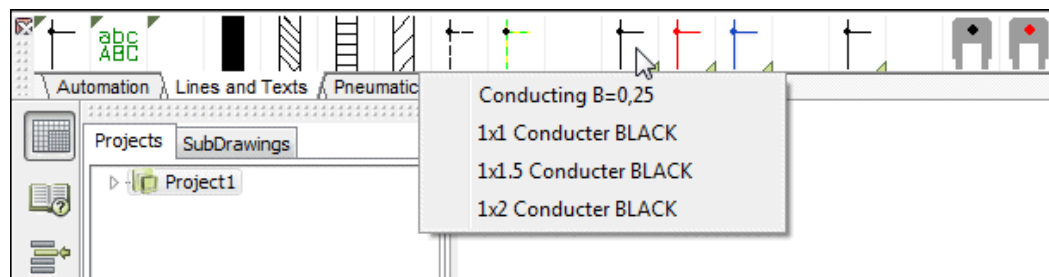
All of these line types can be assigned line article data in the Pickmenu. To assign line article data, right-click on the cable/wire/jumper link in the Pickmenu, and choose **Component Article Data**. Just like you do for symbols.

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 in the list" on page 611.

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.
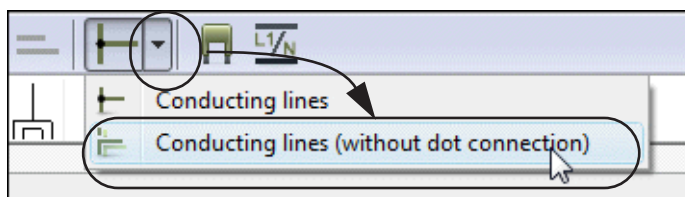
To avoid making connection dots for parallel cables, you must choose **Conducting lines (without dot connection)**. This is only possible when you're working on a page of the type *Ground plan/Mechanical*.

WORKING WITH PROJECTS

In this part you can see how to work with projects in PCSCHEMATIC Automation. It contains following topics:

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.

PC | SCHEMATIC

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.
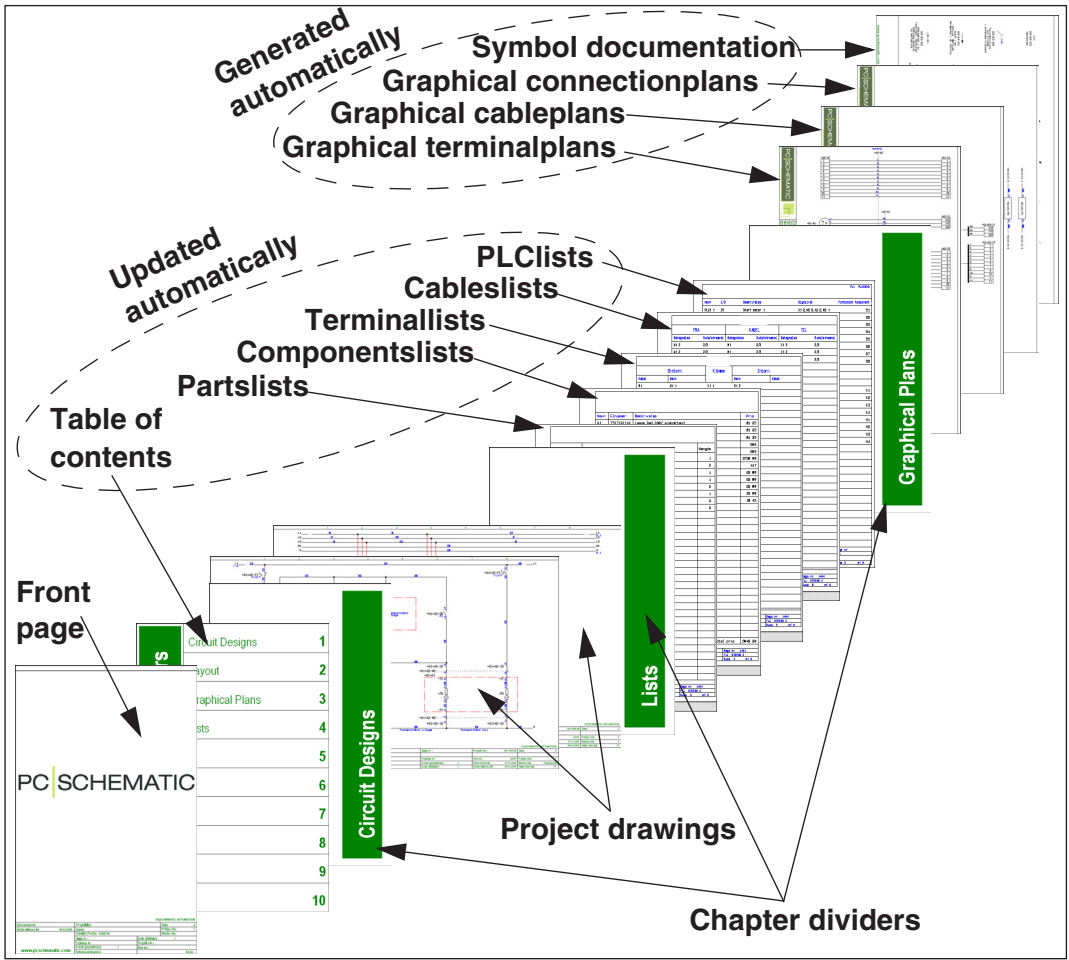
Project diagrams

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 533.

THIS CAN ALSO BE CREATED AUTOMATICALLY

— GRAPHICAL TERMINAL PLANS

— GRAPHICAL CABLE PLANS

— GRAPHICAL CONNECTION PLANS

— SYMBOL DOCUMENTATION

THIS IS DESCRIBED IN THE **PCSCHEMATIC 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.	556
"Project templates and Standard projects"	HOW TO APPLY TEMPLATE PROJECTS, CONTAINING JUST THE PAGES YOU NEED EACH TIME YOU START A NEW PROJECT.	577
"Copying full Projects into other Projects"	HOW TO COPY FULL PROJECT INTO OTHER PROJECTS.	579
"Several Projects Simultaneously"	HOW TO *DRAG* PAGES FROM PROJECT TO PROJECT, HOW TO COPY AREAS FROM PROJECT TO PROJECT, OR COPY TO OTHER PROGRAMS.	582
"Overview projects"	DESCRIBES HOW TO CREATE AN OVERVIEW PROJECT WITH LINKS TO ALL RELEVANT PROJECTS IN AN ASSIGNMENT.	590
"Projects with selectable options"	DESCRIBES HOW YOU CAN CREATE PROJECTS, WHERE YOU CAN SELECT/DESELECT OPTIONS, AND THEN GENERATE A NEW PROJECT ONLY CONTAINING THE SELECTED OPTIONS.	592
"Inserting logos in projects"	HOW TO INSERT A LOGO FOR THE FULL PROJECT.	595

How to work with lists in projects is covered in the section "Lists" on page 597. 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.

Project information

When you want technical information about the active project, you can right-click on any project page and choose **Project information**.

Here you get information about when the project was originally created, when it was last saved, which file name the project is saved under and see how many symbols and connection points are used in the project.

The latter information is calculated according to directions for calculating limitations in Automation 40 and other limited Automation versions. Therefore symbols and connection points on the first mechanical project page are not included in the calculation.

Information

Title: My First Project - 3
Pages: 39
File name: C:\UK\PCSELCAD\PROJECT\PCSDEMO3.pro
Saved: 08-01-2014 11:08:26
Creation date/time: 02-11-2011 09:03:08
Project status: Normal (Normal, PC|SCHEMATIC Automation)
Bytes: 2175893
Symbols = 135
Connection points = 375

Close
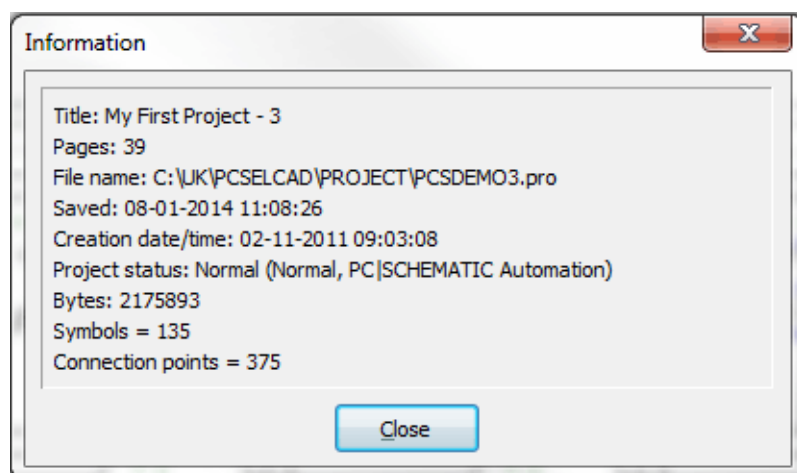
PAGES AND CHAPTERS

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.

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.
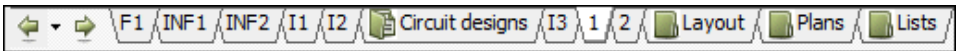
Or you can use the <Page Up> and the <Page Down> keys to jump back and forth between the project pages. Another option is clicking the **Page history** buttons, as described in "The Page history buttons" on page 557.

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 569.

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:

Click here to change

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 569.

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 work-
ing 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:
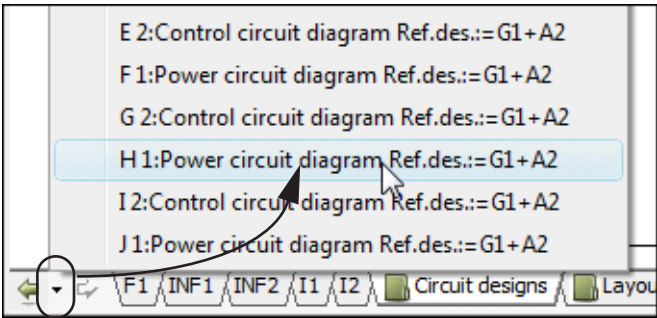
When you click on one of these, you jump to this page in the project. This way you
can jump to the 10 most recently active pages in the project.

— 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 345.

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 577.

Options in the Page menu

The **Page menu** for a project could for instance appear like this:
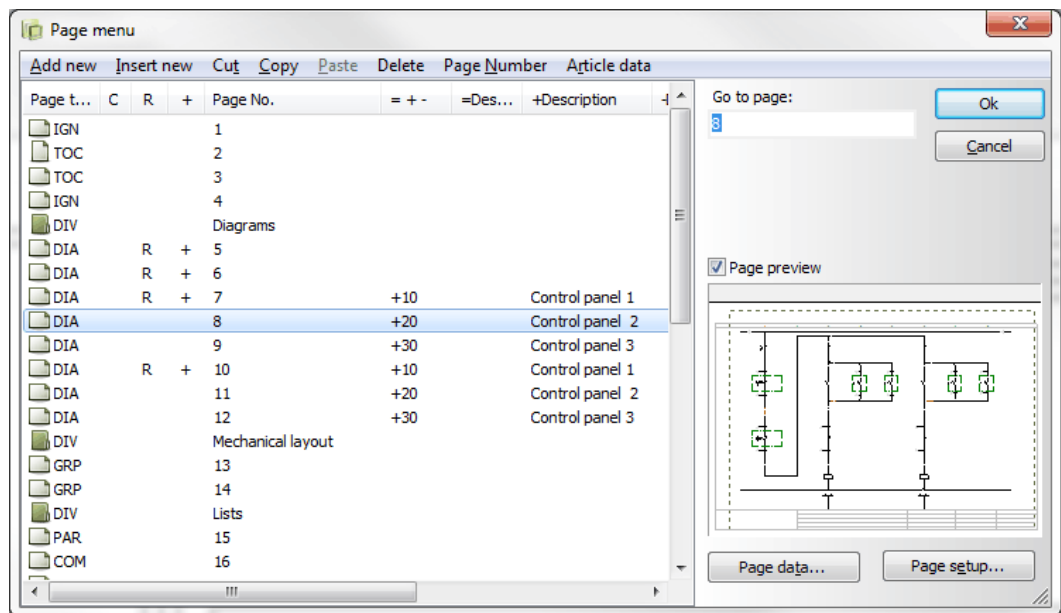
In front of each page an icon indicates whether the page has a horizontal or a vertical page format. After this you see the page function, which is described in "Creating a new page" on page 560. Folders in front of pages marks the beginning of a new chapter.

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 892.

When colors are used for page tabs — see "Colors on page tabs" on page 571 — these are displayed in the **C** column next to the page type. If you right-click on a page here in the **Page menu**, you can also change the color of the page tab for the page.

When states are selected on project pages, the selected states are displayed in brackets after the page titles.

In the **Page menu** you have the following options:

"SELECT ALL PAGES" OR "SELECT CHAPTER"

WHEN YOU RIGHT-CLICK ON A PAGE IN THE **PAGE MENU**, YOU CAN CHOOSE **SELECT CHAPTER** TO SELECT ALL PAGES IN THE CHAPTER. YOU CAN ALSO CHOOSE **SELECT ALL PAGES** TO SELECT ALL PAGES IN THE PROJECT.

CHOOSE	FUNCTION/USE
ADD NEW	ADDS A NEW PAGE IN THE END OF THE PROJECT. — SEE "CREATING A NEW PAGE" ON PAGE 560.
INSERT NEW	INSERTS A NEW PAGE *IN FRONT OF* 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 560.
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 565.
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 565 AND "COPYING, CUTTING AND PASTING MORE THAN ONE PAGE" ON PAGE 565.
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 565.
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 423.

CHOOSE	FUNCTION/USE
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. IF YOU RIGHT-CLICK ON A PAGE IN THE **PAGE MENU** AND CHOOSE **PAGE NUMBER**, YOU CAN ALSO CHANGE THE PAGE NUMBER. 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**.
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 574, "PAGE SETUP - AND TRANSFERRING THIS TO OTHER PAGES" ON PAGE 572 AND "PAGE SETUP" ON PAGE 915.
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 599 AND "PAGE DATA" ON PAGE 912.

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 451.

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 560**.**

- Via the **Page menu** — see "Creating a new project page - via the Page menu" on page 561.

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 299), or

b) A *template page* of the desired type has been created (see "Creating Templates" on page 850).

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.
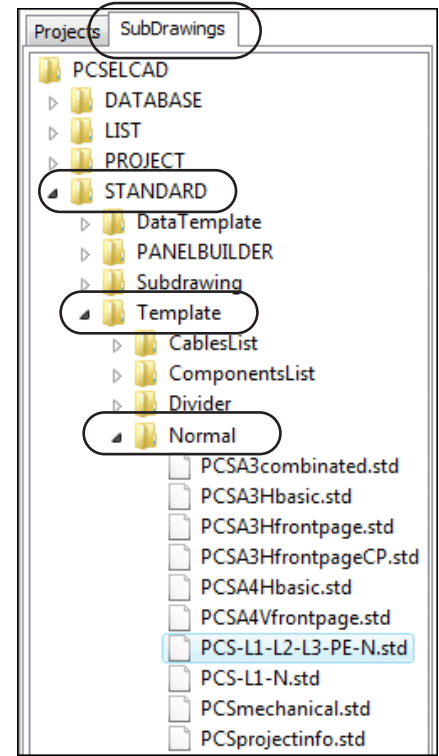
Correspondingly, you can drag other pages directly into the project from the **SubDrawings** tab.

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 558.

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.
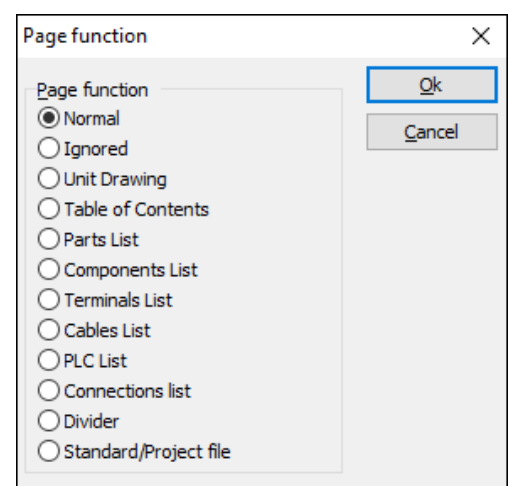
Selecting a Page function

PAGE FUNCTION NORMAL FOR SYMBOLS WITH ARTICLE DATA
WHEN YOU WISH TO INCLUDE ARTICLE DATA FOR THE COMPONENTS/SYM-
BOLS 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 (DIA-GRAMS), OR GRP (GROUND PLANS)
IGNORED	THE PAGE WILL BE USED TO ENTER INFORMATION THAT SHALL NOT BE INCLUDED IN LISTS. COULD FOR INSTANCE BE MOUNT-ING INSTRUCTIONS, OR A FRONT PAGE FOR THE PROJECT.	IGN
UNIT DRAW-ING	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 638.	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*. — SEE THE DATAFIELDS IN "TABLE OF CON-TENTS DATAFIELDS" ON PAGE 948. 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 624. 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. — SEE THE DATAFIELDS IN "PARTS/COMPONENTS LIST DATAFIELDS" ON PAGE 949.	PAR
COMPO-NENTS LIST	THE PAGE WILL BE USED AS A COMPONENTS LIST PAGE. DATA-FIELDS RELATED TO COMPONENTS LISTS CAN BE INSERTED. — SEE THE DATAFIELDS IN "PARTS/COMPONENTS LIST DATA-FIELDS" ON PAGE 949.	COM
TERMINALS LIST	THE PAGE WILL BE USED AS A TERMINALS LIST PAGE. DATA-FIELDS RELATED TO TERMINALS LISTS CAN BE INSERTED. — SEE THE DATAFIELDS IN "TERMINALS LIST DATAFIELDS" ON PAGE 951.	TER
CABLE LIST	THE PAGE WILL BE USED AS A CABLES LIST PAGE. DATAFIELDS RELATED TO CABLES LISTS CAN BE INSERTED. — SEE THE DATAFIELDS IN "CABLE LIST DATAFIELDS" ON PAGE 954.	CAB

PAGE FUNCTION	DESCRIPTION	NAME IN PAGE MENU
PLC LIST	THE PAGE WILL BE USED AS A PLC LIST PAGE. DATAFIELDS RELATED TO PLC LISTS CAN BE INSERTED. — SEE THE DATAFIELDS IN "PLC LIST DATAFIELDS" ON PAGE 955.	PLC
CONNECTIONS LIST	THE PAGE WILL BE USED AS A CONNECTIONS LIST PAGE. DATAFIELDS RELATED TO CONNECTIONS LISTS CAN BE INSERTED. — SEE THE DATAFIELDS IN "CONNECTIONS LIST DATAFIELDS" ON PAGE 958.	NET
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. WHEN YOU INSERT A DIVIDER TAB, YOU ARE ASKED TO TYPE IN THE NAME OF THE TAB. HERE YOU CAN CHOOSE TO CLICK ON **VALUES**, WHICH GIVES YOU THE OPTION OF CREATING A LIST OF POSSIBLE NAMES FOR DIVIDER TABS. WHEN YOU SUBSEQUENTLY NEED TO SPECIFY THE NAME OF A DIVIDER TAB, YOU CAN CHOOSE AMONG THESE NAMES FROM A DROP-DOWN LIST.	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 *STANDARD PROJECT/STANDARD DRAWING*. — READ MORE ABOUT THIS IN "PROJECT TEMPLATES AND STANDARD PROJECTS" ON PAGE 577. IS ALSO USED FOR IMPORT OF DWG AND DXF DRAWINGS — SEE "DWG AND DXF FILES" ON PAGE 741.	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 **PCSCHEMATIC Tools** manual.

CHANGE PAGE FUNCTION & TRANSFER PAGE FUNCTION TO OTHER PAGES

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.

IN ORDER TO TRANSFER THE SELECTED PAGE FUNCTION TO OTHER PAGES IN THE PROJECT, CLICK THE **TRANSFER SETTINGS** BUTTON.
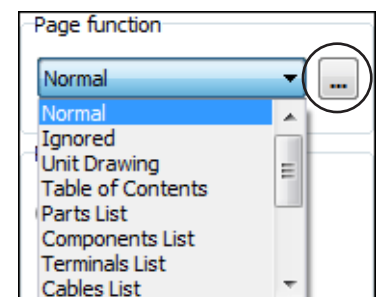
You can see the *page function* of a page when you choose **Settings => Page setup**. — See "Page setup" on page 915.

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:
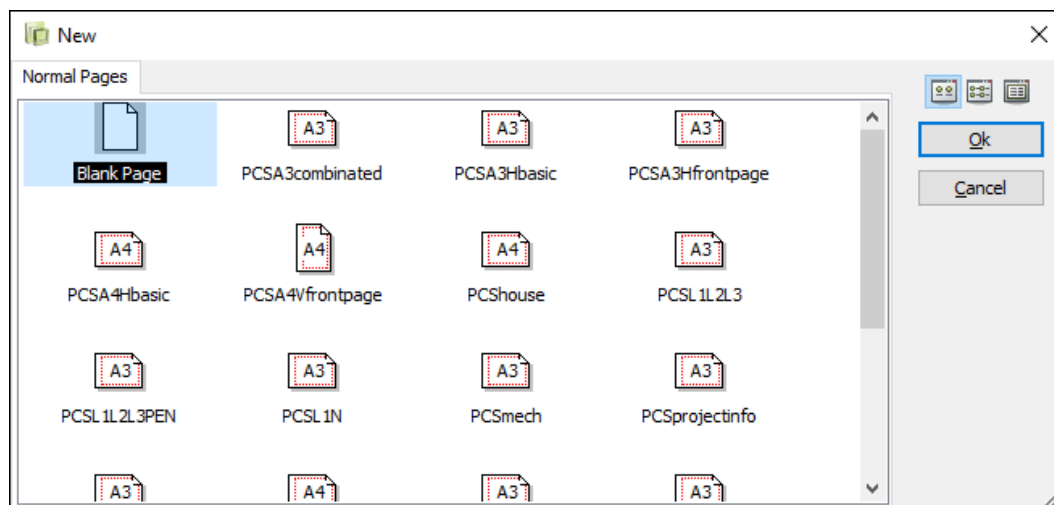
Click on the desired template page, and click **OK**.

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, COR-RESPONDINGLY, SELECT TO INSERT PAGES CONTAINING LIST DRAWING HEADERS.

— See how to create your own templates in "Creating Templates" on page 850, particularly "Creating Page templates — via the Template editor" on page 851.

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.

Selecting language on predefined drawing headers

In the predefined drawing headers included in the program, it is possible to choose between all of the languages that the program can handle. Read more about this in "Drawing headers in multiple languages" on page 829.

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 281.

Example on choices in the Page menu:
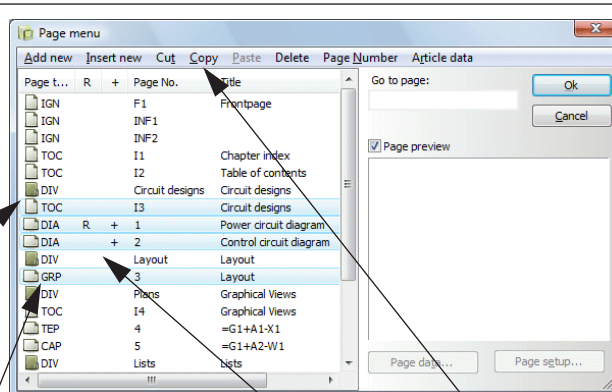
1) Click here 2) Press down <Shift>   3) Press <Ctrl>   4) Click on Copy
                   and click here   and click here   and paste
                   (Choosing page I3-4)   (Deselect DIV)   the pages

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 566.

> **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 565.

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**.
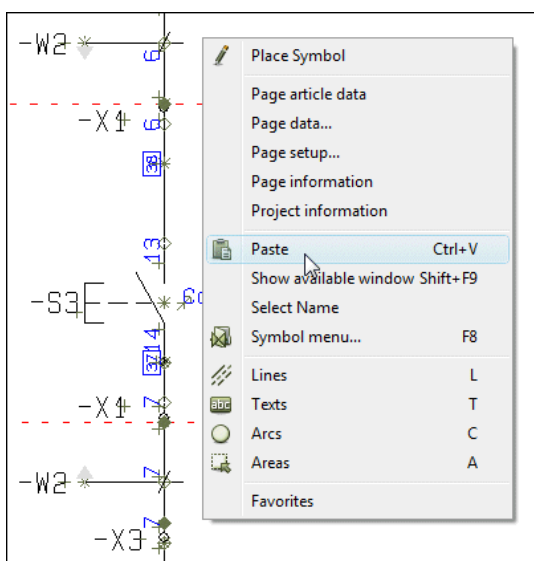
4) Click **OK** to paste the pages:
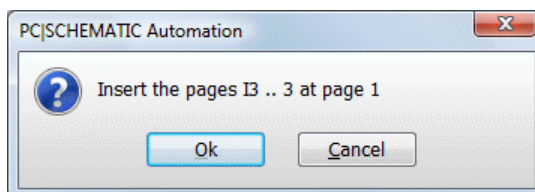
5) Specify how to renumber symbols, terminals etc. and click **OK** — see "Intelligent copying of areas" on page 281:
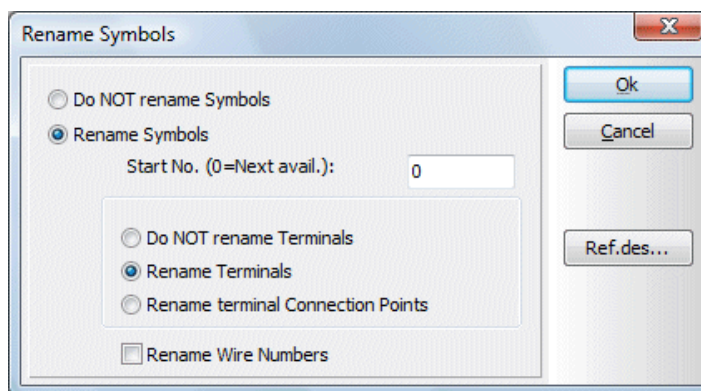
6) The pages are now pasted into the other project, and the symbol names etc. have been updated for the pasted pages.

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 568.

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**:

Hereby all pages in the chapter are also moved.
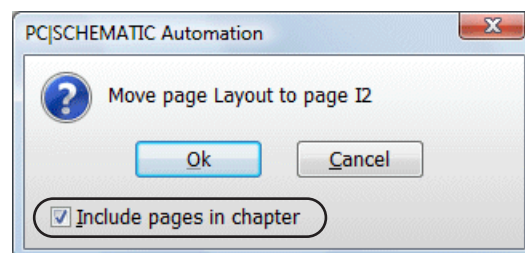
Copy pages and chapters using the page tabs

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 281.

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 — se "List settings for Tables of Contents" on page 624.

2) To change reference designations on chapter level — see "Editing reference designations for chapters" on page 423.

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 568.

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.
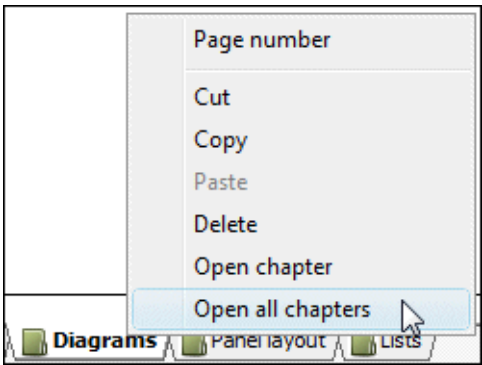
Creating a new chapter

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 new**.

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:
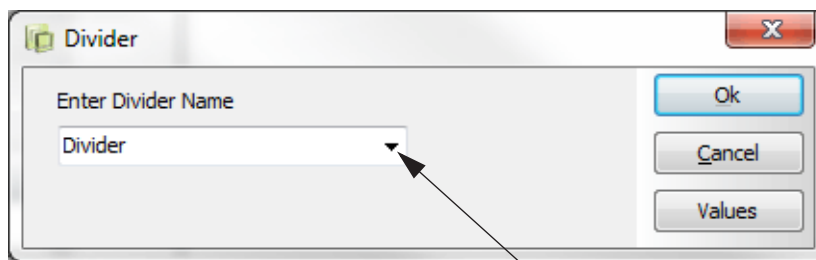
**Select a chapter title by clicking here, if a
Values list as been created for dividers**

If you click the **Values** button in the dialog box above, you can create a list of chapter titles that you typically use in projects. — See "Values list for Project and Page title" on page 128.

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 599.

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 820.

Click here to see the first page in the chapter

Click here to see the Divider tab

Lists
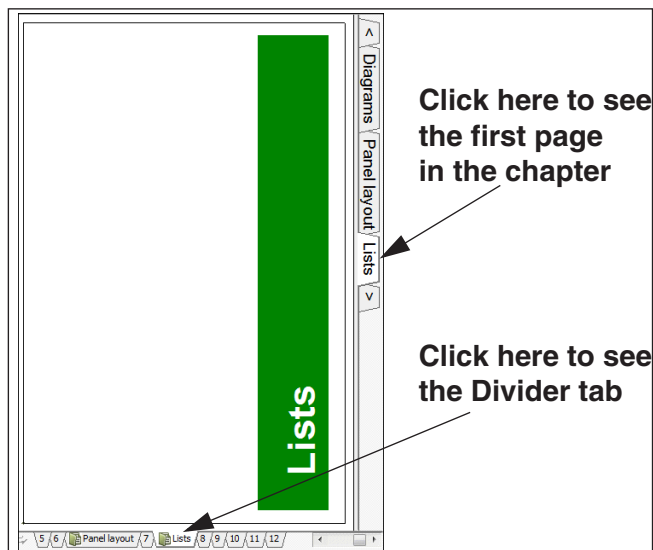
DRAG IN A CHAPTER DIVIDER PAGE FROM THE EXPLORER WINDOW

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 560.

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 118.

— Please also see "Last used page active" on page 373.

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.

Names on page tabs at the bottom of the sceeen

To edit the name of a page tab at the bottom of the screen, do this:

1) Double-click on the tab.

2) 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 558.

You can also rename a page by right-clicking on the page tab, and choose **Page number**.

Names on chapter tabs at the right-hand side of the screen

The texts in the divider tabs at the right-hand side of the screen shows the *Page title* for the divider tabs.

To quickly change the text on a divider tab in the right-hand side of the screen, do this:

1) Double-click the tab.

2) In the dialog box **Page data**: Type in the new name in the **Page title** field and click **OK**.

COLORS ON PAGE TABS

To change the color of a page tab at the bottom of the screen, do this:

1) Right-click on the tab and choose **Color**

2) Click on **Select**

3) In the dialog box **Color**: Select the desired color

To remove the color again simply right-click the tab and choose **Color => None**.

The selected color is also displayed in the **Page menu** — see "Options in the Page menu" on page 558.

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 574 and in "Paper size" on page 575. — See more options in "Page setup" on page 915.
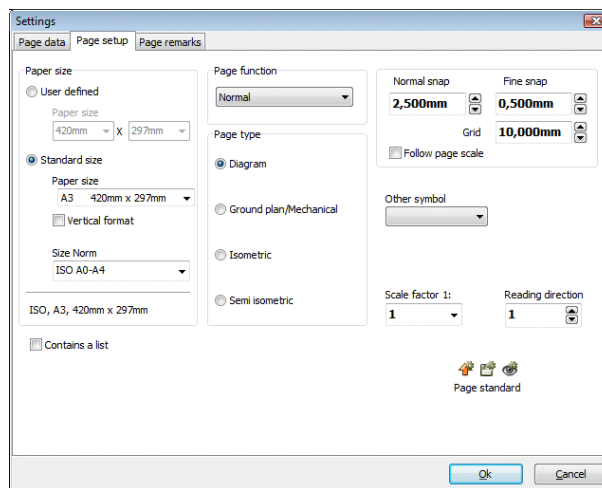
Transferring the Page setup to other project pages

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:
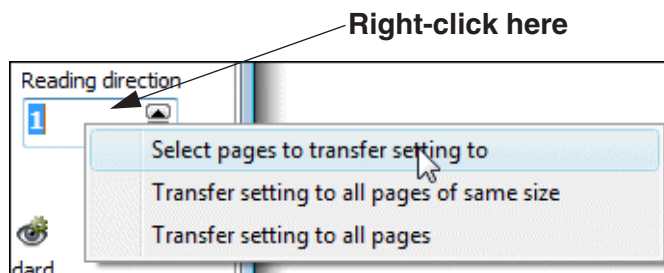
If you choose **Select pages to transfer setting to**, you enter a special version of the **Page menu**, where you select which pages to transfer the setting to:
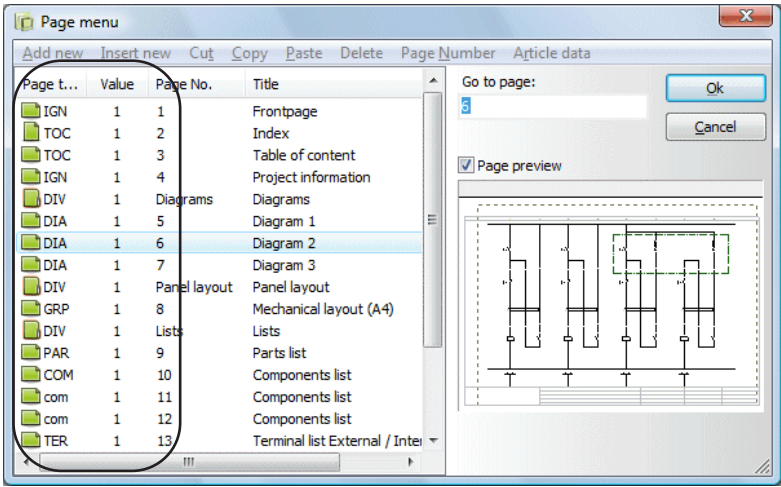
Here the green icons indicate which pages the setting can be transferred to. If an icon is dimmed, the page already has this setting — or the setting is not possible for the page. The settings for the other pages are displayed in the **Value** column.

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 841,

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 49.

The *Page type* can be changed any time.

In the list below you can see what the various page types are used for:
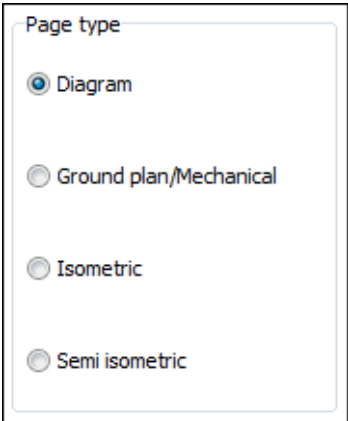
PAGE TYPE	DESCRIPTION
DIAGRAM	THE PAGE IS A STANDARD DIAGRAM FOR DRAWING IN TWO DIMENSIONS. USED FOR ALL TYPES OF LISTS AS WELL.
GROUND PLAN/ MECHANICAL	THE PAGE IS USED FOR GROUND PLAN- AND MECHANICAL DRAWINGS, WHERE WORKING IN DIFFERENT ELEVATIONS, AND THEREBY WORK IN THREE DIMENSIONS, IS POSSIBLE.
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:

Choose between User defined and Standard size

Click here for Vertical format

Select size

Select norm
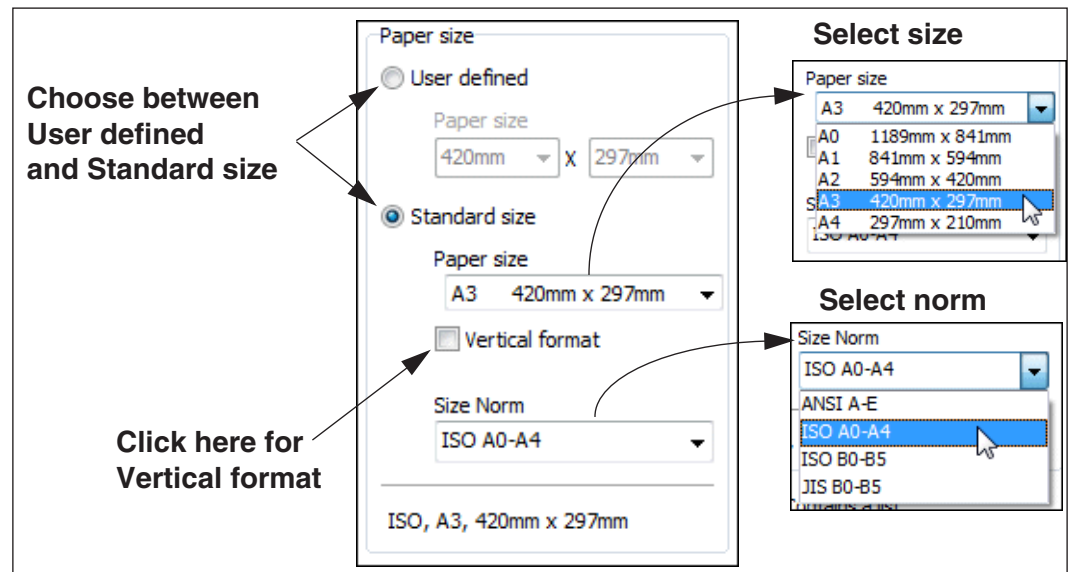
Basically, you can choose between **_User defined_**, where you type in the measures of the page, and **_Standard size_**, where you select a norm, and a paper size in this norm.

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 PCSCHEMATIC 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

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.
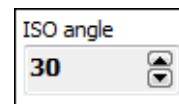
If you choose *Semi isometric* you can specify another angle accordingly.

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.

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.
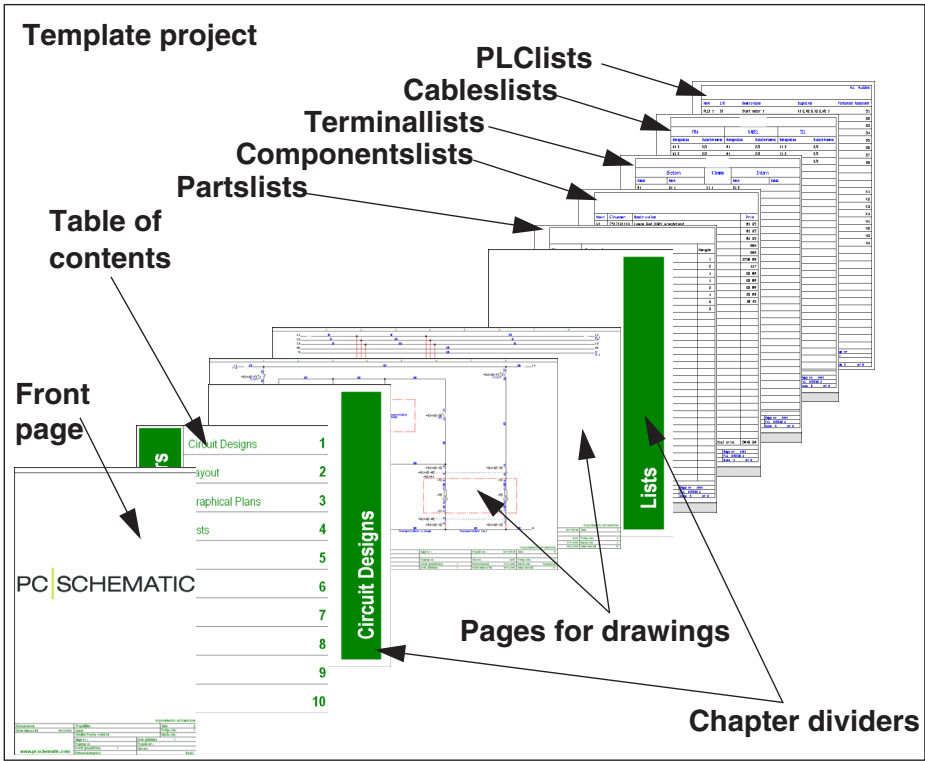
This way you can build a library of different project templates, which can be used when starting different types of projects, or for instance for projects for different customers.

PC|SCHEMATIC

USING PROJECT TEMPLATES

To create a new project based on a project template, do this:

1) Choose **File => New** or click the **New document** button.

2) In the dialog box **New**: Click on the tab for **Projects**, click on a project on the tab, and click **OK**.
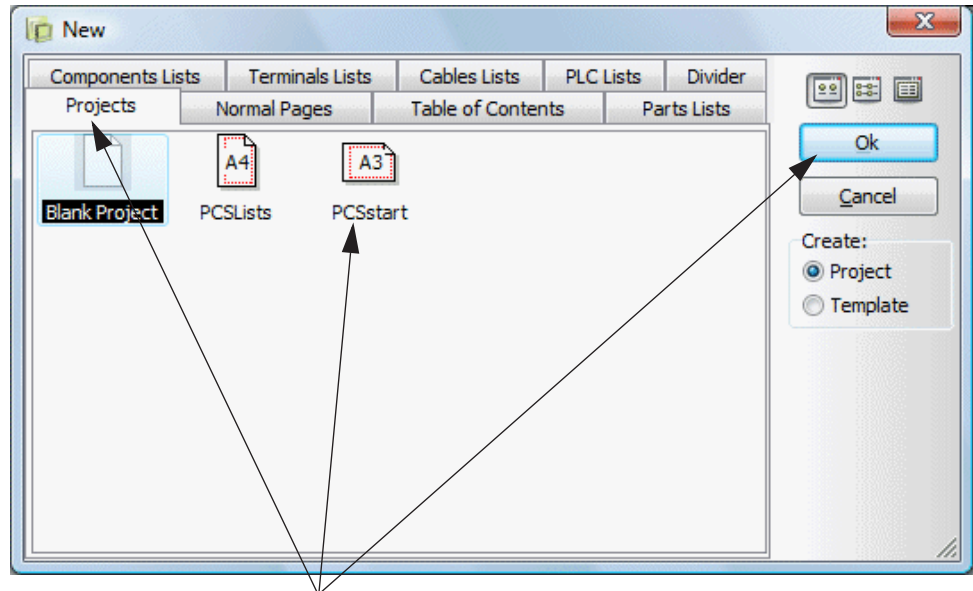
Click here

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 850.

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 579.

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**.

You may wish to read:

- "Project templates and Standard projects" on page 577, 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 583, 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.**
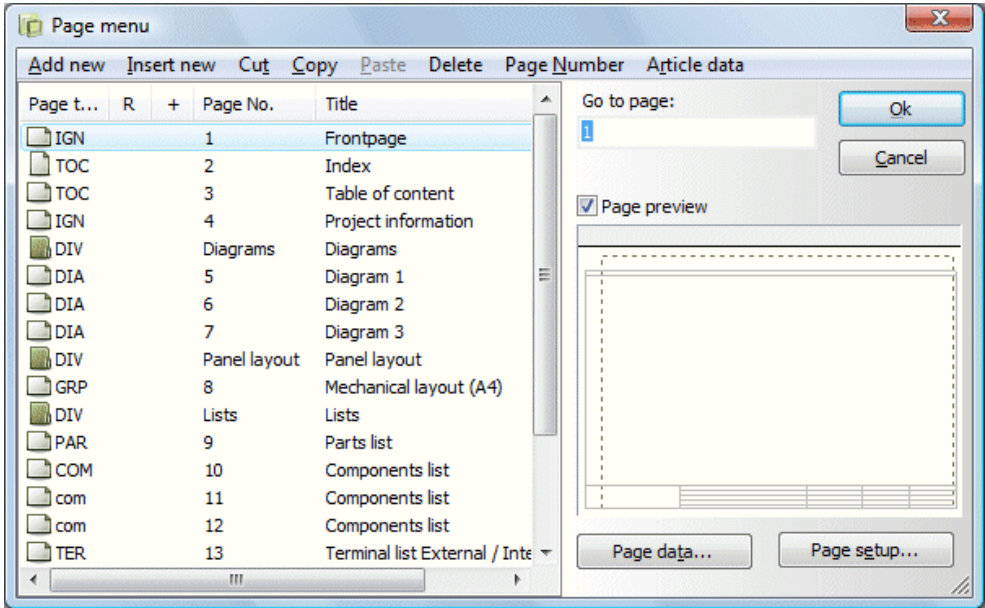
Here you click **Add new** — to add a new project/page in the end of this project — and then click on ***Standard/Project file*** and click **OK**.
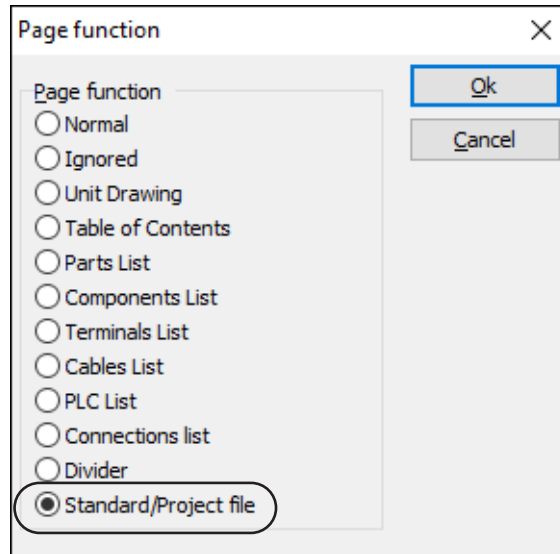
— Instead of **Add new** you could also click on for instance page four and then click **Insert new**. The project would then be inserted starting on page four.

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 **plc-demo.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:
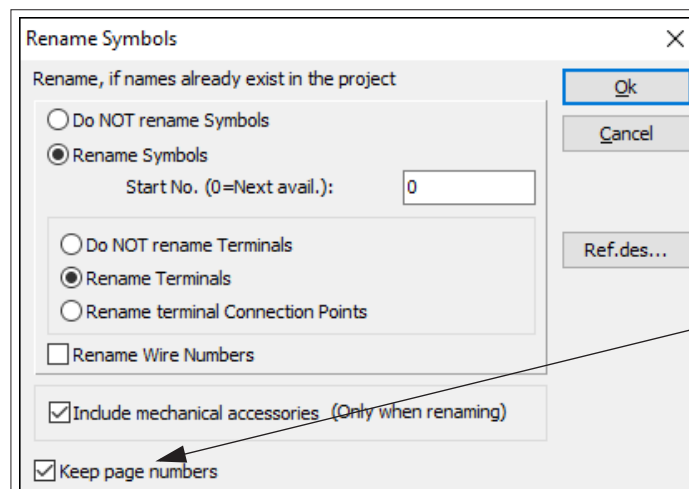
Click here not to change page numbers in the inserted project

Select ***Rename symbols***, ***Rename terminals***, and select ***Keep page numbers***, if you wish to keep the original page numbers from the inserted project. — If necessary see "Intelligent copying of areas" on page 281.
Click **OK**.

The selected project in now inserted into the original project:
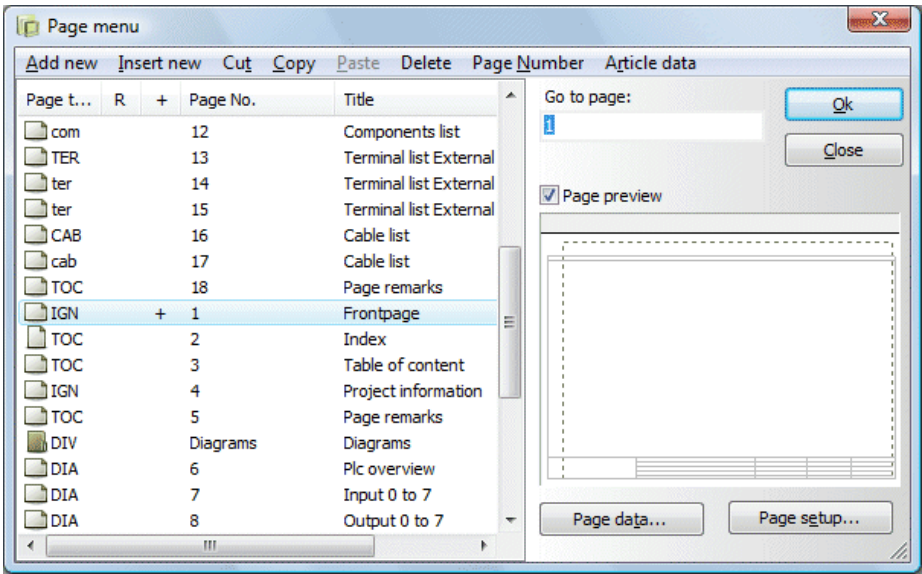
You have now loaded an entire project into another project.

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 558.

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 909.

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.

You can also see how to create overview projects, with links to other projects in "Overview projects" on page 590.

If you are interested in how to copy an entire project into another, please read "Copying full Projects into other Projects" on page 579.

— See "Pasting copied pages in other projects" on page 566 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 item **Window**.

You can then choose between:

a) **Cascade**,

b) **Tile vertical** and

c) **Tile horizontal**.
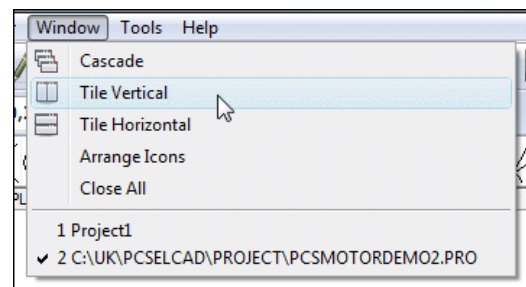
When you choose **Window => Close All**, all projects in the program are closed. If you have made changes, which have not been saved, you will get the option of saving these.

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).
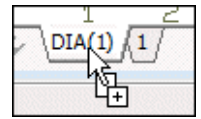
Then answer Yes to copy the page, and answer the emerging question whether — and how — to rename the symbols copied. The page copied is placed immediately *before* the page you clicked on in the project.

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 558.

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 566.

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 568.

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** or click the **New document** button, 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 SOME-WHERE 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 556).

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.
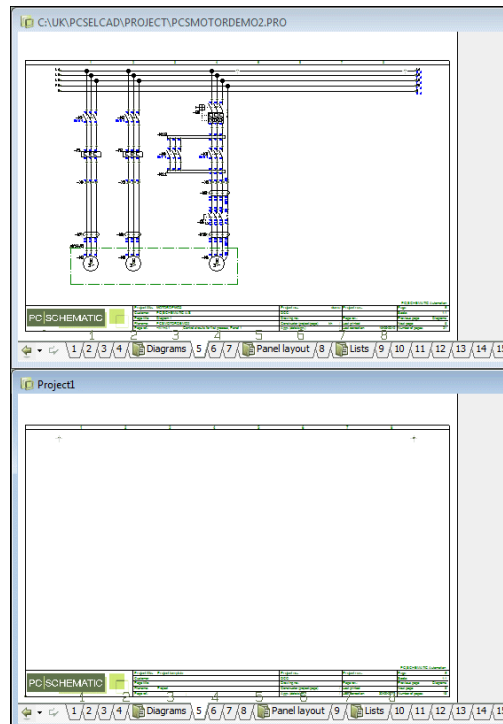
8) Click on the **Area** button, and use the mouse pointer to select up the area you wish to copy — see "Selecting objects" on page 55 if necessary.
— If you zoom in on the area you wish to copy, this will not effect the size of the area you insert in the other project. This is because *zoom* only has 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 281 — and click **OK**.

12) The selected area is now copied from the one project into the other.
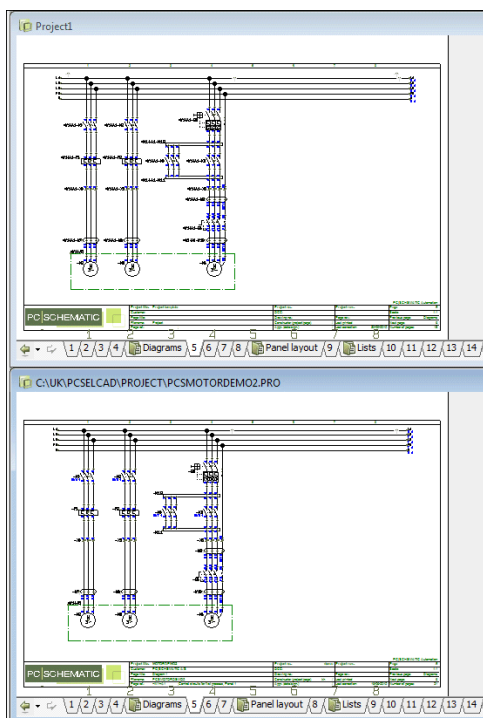
13) Press <Esc> not to place the drawing once again.

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 599.

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 579.

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 775.
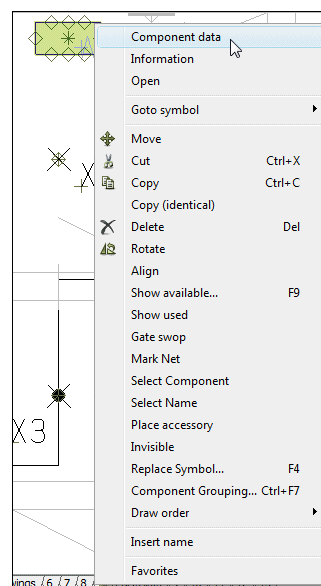
Right-click on the symbol, and choose **Component data**.
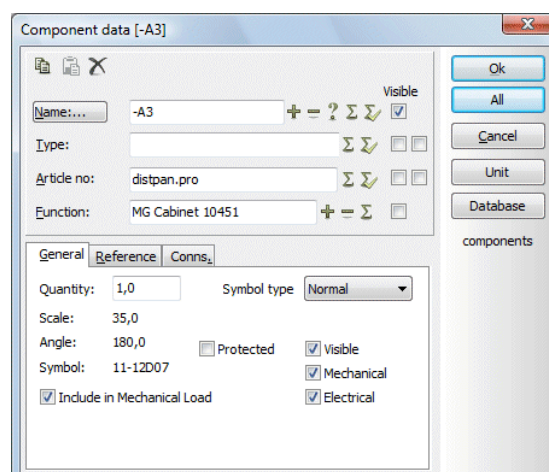
In the *Article no* field you type in the full path to the project in which the switch-board is documented. If the project is documented in the folder specified for projects in **Settings => Directories**, you only have to type in the name of the project file itself. Therefore it only says *distpan.pro* in the dialog box above. Read more about this in "Directories" on page 923.

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 "Connecting symbols/lines on diagrams to unit drawings" on page 638.

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 749.

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.

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 775.

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.
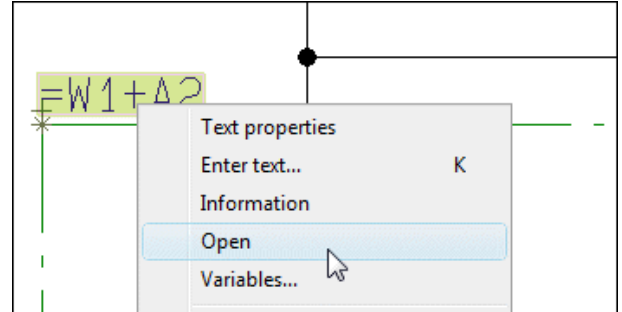
PROJECTS WITH SELECTABLE OPTIONS

If for instance you produce a product where you can select/deselect various options, it is not necessary to create individual documentation projects for each combination of options for the product.

In stead you can create a single (master) project where all of the options are documented. Based on this project you can then automatically generate documentation projects, where you specify exactly which options to include in the individual projects.

This chapter describes how to automatically generate projects with the selected options, based on an already created master project.

INSTALL THE MODULE

To generate projects with selected options, you must first install the module **Generate project from selected options**:

1) Choose **File => Modules**.

2) In the dialog box **Modules**: Click *Generate project from selected options* in the list, and click **Install**.

3) Click **Close**.

The module is now ready for use.

START GENERATE PROJECT WITH SELECTED OPTIONS

Before activating the module you need to create a "master" project, where you apply reference designations to specify the options in the project. This project can then be used to generate other projects from.

When you generate a new project from the "master" project, you apply the reference designations to specify which options — or parts of the "master" project — to include in the generated project.

Please note that this "master" project is just an ordinary project, which is created and saved just like any other project.

— Read more about reference designations in "On reference designations" on page 385.

Generating a project from the master project

To generate a new project based on the "master" project, start by opening the "master" project. Then start the module by choosing **Tools => Generate project with selected options**.

You then enter the dialog box **Generate project with selected options**, where you generate a new project from the "master" project.

In the dialog box you specify the desired settings on the tabs **Reference designations** and **Overview symbol**. Then click **Generate**, and the new project is generated. Please note that the "master" project is closed when the new project is generated.

THE REFERENCE DESIGNATIONS TAB

On the tab **Reference designations** you choose which of the created function and/or location aspects, that is to be included in the new project.

Select which aspects to include in the generated project
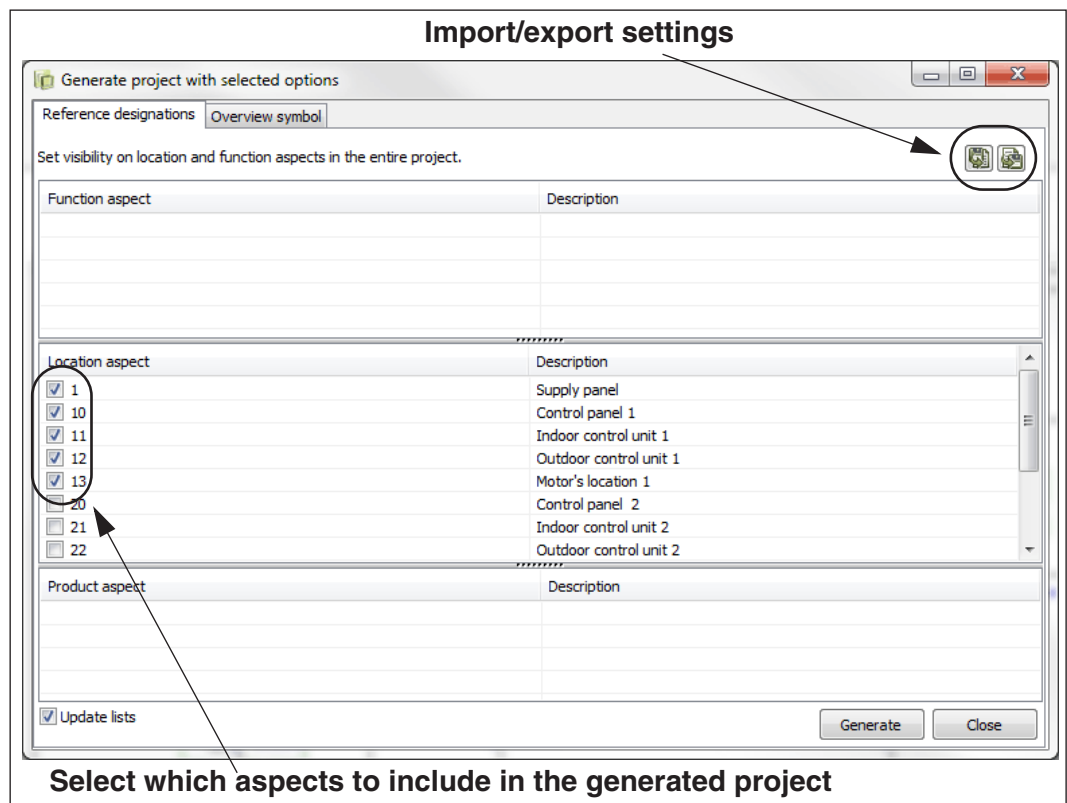
If you want to update the lists in the generated project, so that they match the contents on the project pages, please remember to select *Update lists* at the bottom of the dialog box.

Updating the generated project

If you are not satisfied with the generated project, just choose **Tools => Generate project with selected options** again, select/deselect options, and click the **Update** button to update the project according to the changes. (The **Update** button is located where the **Generate** button was originally placed).

THE SURVEY SYMBOL TAB

On the **Survey symbol** tab you specify whether to include an overview symbol in the generated project, which gives an overview over the selected options.

Click here to insert an overview symbol
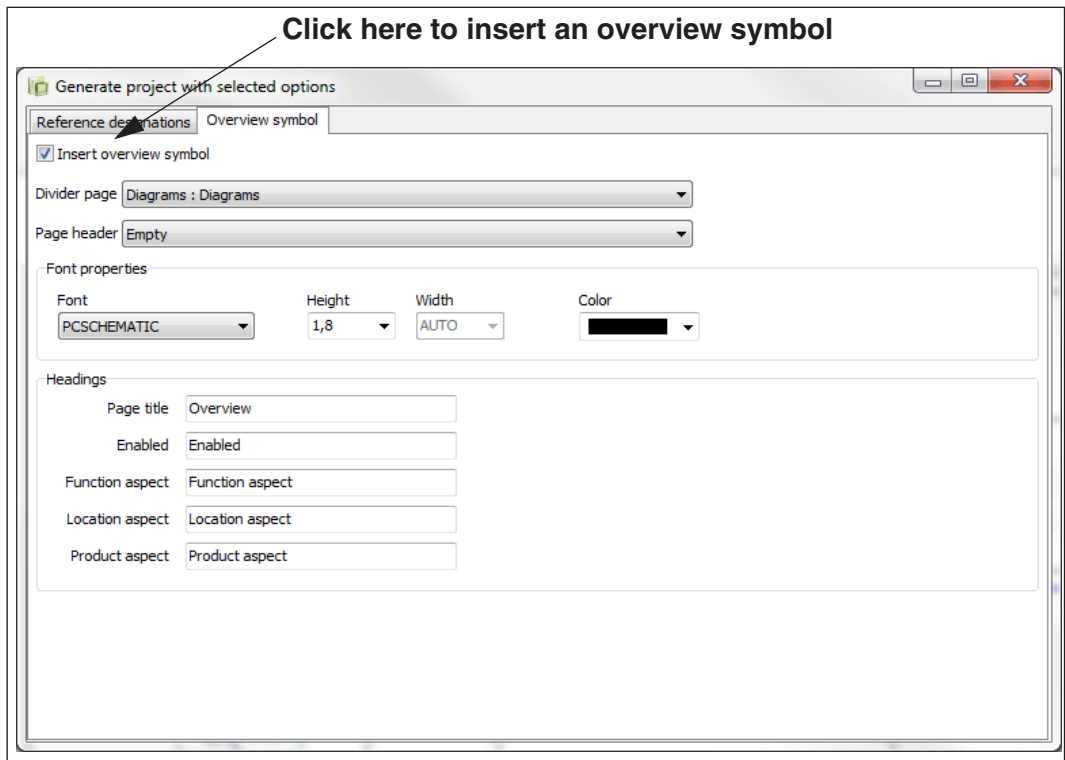
When you have selected **_Insert overview symbol_** at the top of the dialog box, you get the option of specifying:

- Which drawing header to insert on the page containing the overview symbol
- The text properties for the texts in the overview symbol
- The heading texts for the columns in the overview symbol

Enabled	Location aspect	Description
☑	+1	Supply panel
☑	+10	Control panel 1
☑	+11	Indoor control unit 1
☐	+12	Outdoor control unit 1
☑	+13	Motor location 1

INSERTING LOGOS IN PROJECTS

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**.
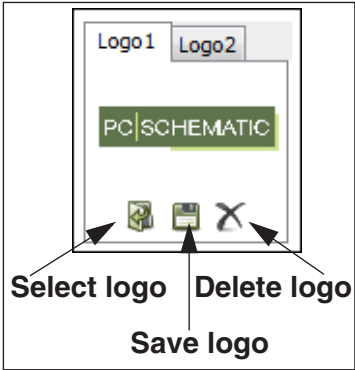
Place the logo using a Datafield on a project page

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.
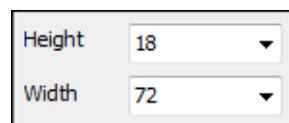
Adapting the size of the logo

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**.

| Height | 18 |
| Width | 72 |

11) Press <Esc> — to deselect the logo.

LISTS

In this section you can see how to use lists.
You can read about:

READ THE SURVEY ARTICLE

READ THE SURVEY ARTICLE ON WORKING WITH DATABA-
SES AND LISTS AT WWW.PCSCHEMATIC.COM/ARTICLES.

PC|SCHEMATIC

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.

In the chapter "Creating Drawing Headers" on page 820 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 556 AND "CREATING A NEW PAGE" ON PAGE 560.

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 980.

Please note, that you can place two drawing headers on the same page — see "Two drawing headers on the same page" on page 605.
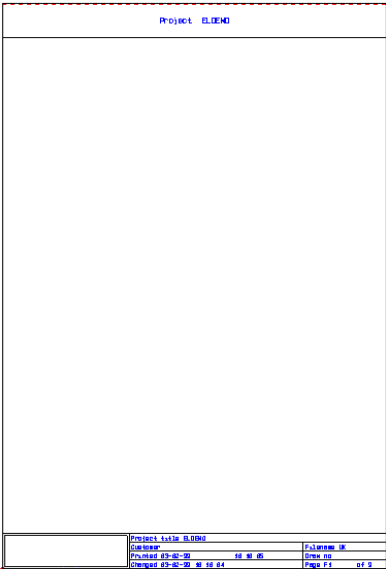
INSERTING A DRAWING HEADER ON A PAGE IN A PROJECT

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 from 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 828.

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.
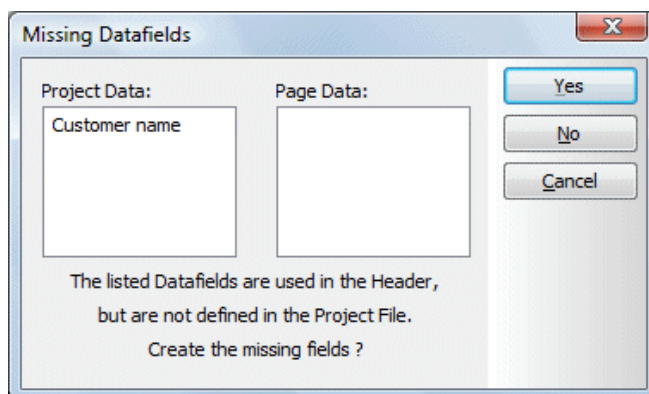
Active links from Datafields in Drawing headers to the Settings dialog box

When you double-click on a datafield originating from **Project data** or **Page data**, the dialog box **Settings** is opened with focus on the specific field, from which the datafield collects its value. — For instance *Customer* or *Project title* in **Project data**:

PCSCHEMATIC Automation
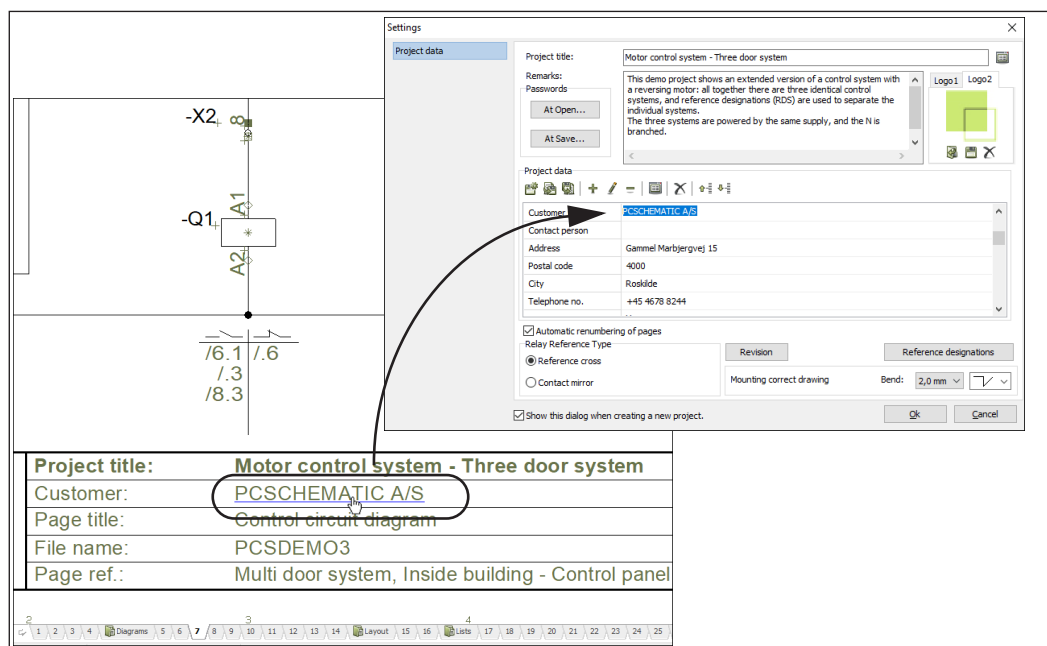
Drawing headers with States

When a drawing header contains different states, you can choose the drawing header state in the lower-right corner of the dialog box **Page data**:
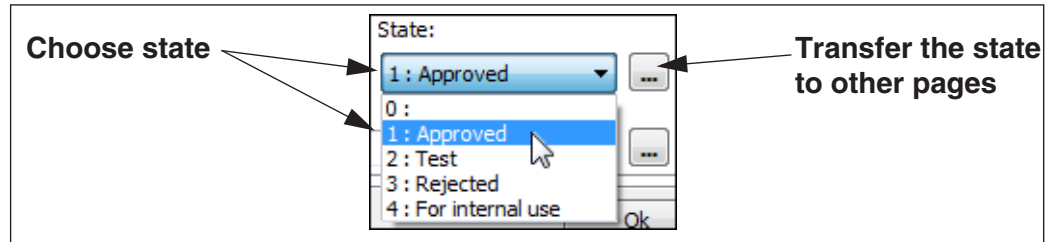
If you wish to transfer the selected state to the other pages in the project, click the **Transfer setting** button on the right-hand side of the **State** field. You then get the following options:

- **Select pages to transfer setting to**: You now enter the **Page menu**, where you select the pages and click **OK**.

- **Transfer setting to all pages with same drawing header**: The setting is transferred to all project pages using the selected drawing header.

- **Transfer setting to all pages**: The state is transferred to all project pages of the same page type.

> USING STATES IN DRAWING HEADERS TO CHANGE LANGUAGE
>
> YOU CAN USE STATES IN DRAWING HEADERS TO CHANGE LANGUAGE FOR THE TEXTS IN THE DRAWING HEADER.
>
> — READ MORE ABOUT THIS IN "DRAWING HEADERS IN MULTIPLE LANGUAGES" ON PAGE 829.

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 556.

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 600.

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 draw-

ing header under the same name. — See how to change drawing headers in "Placing a Company Logo on a drawing header" on page 830.

> **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.
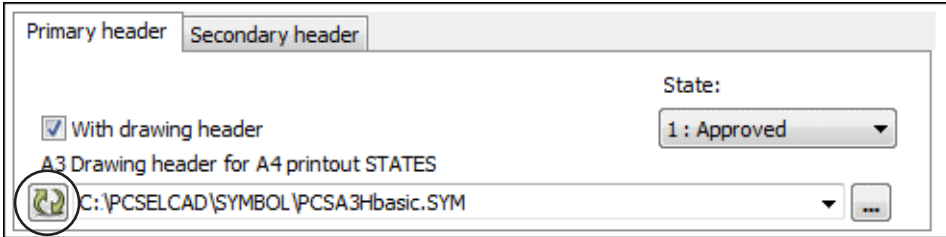
Replacing a drawing header

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.
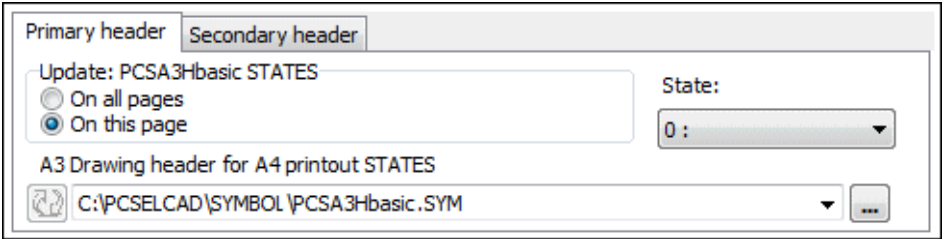
2) Click the **Page data** button.

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.
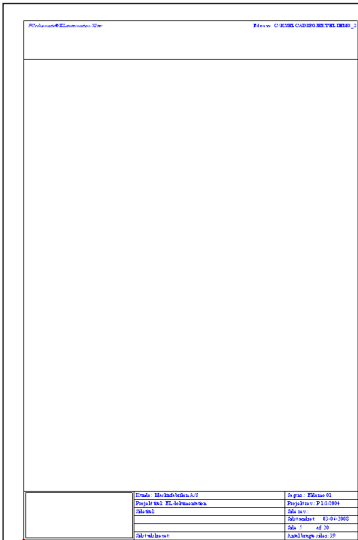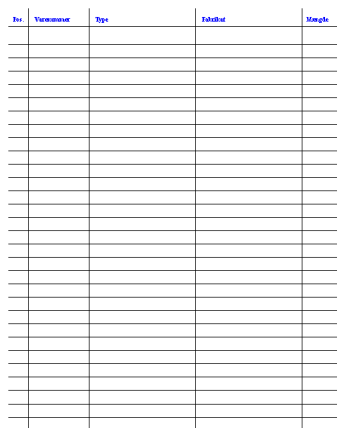
TWO DRAWING HEADERS ON THE SAME PAGE

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.

"Title block" drawing header (drawing header 1)   **"Contents" drawing header (drawing header 2)**

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 601:

Then click on **Secondary header**, and select the drawing header containing the "contents" for e.g. a parts list:
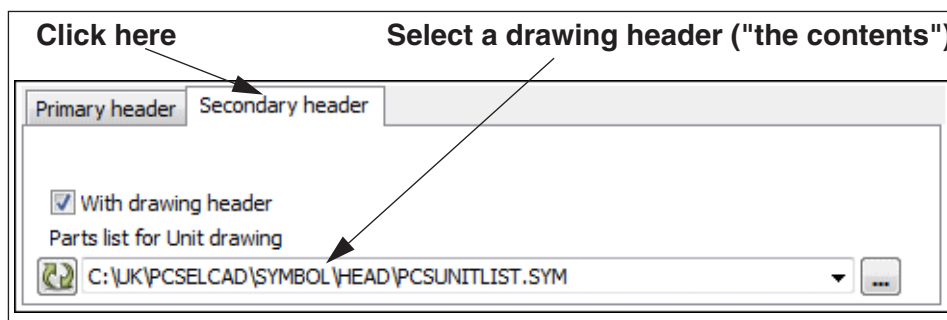
Both drawing headers are now inserted on the page, and appears just like any other drawing header:

INSERTING A LIST

Lists are a special type of drawing headers, which you can place in your projects.

There are several types of lists in **Automation**. These are for instance parts lists, components lists, terminals lists, cables lists, PLC lists and tables of contents.

The parts lists collect information from the symbols/components which have been placed on the project drawings, while the table of contents pages collect information about the various project pages.

Because lists are a special kind of drawing headers, they have to be updated to have all information filled out correctly.

To tell the program which type of information have to be filled into a list, you have to make sure that the *page*, on which the list is placed, is set to the right *page function*.

If you want to insert a parts list, the page function has to be **Parts list**, and if you want to insert a table of contents, the page function must be **Table of Contents** — as opposed to above, where the page function was set to **Normal**.

🚫 **THE CORRECT PAGE FUNCTION MUST BE SELECTED FOR LISTS**

IF YOU DO NOT CHOOSE THE CORRECT PAGE FUNCTION FOR A LIST, THE LIST WILL NOT BE FILLED OUT WHEN UPDATED.

Apart form the page function, lists are placed as described above. — Read more about page function in "Creating a new page" on page 560.

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 841.

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 842.

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, a connection point or a line. This is the case for quite a few datafields, but not for all types.

Pos.	EAN Barcode	Article number	Type
1	3 389110 585506	3389110585506	AB1NE435U
2	3 389110 586435	3389110586435	AB1VV435U
3	3 389110 610024	3389110610024	XB2BA31

When the cursor is above a datafield containing an active link, the datafield is underlined, and the appearence 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.

TO ACTIVATE LINKS FROM LINES IN YOUR LISTS, YOU NEED TO UPDATE LISTS FOR PROJECTS CREATED IN **Automation** VERSION 21.0 OR EARLIER.

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 556.

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

Right-click menu shown over the list:
Place Symbol
Page article data
Page data...
Page setup...
Page information
Project information
Paste   Ctrl+V
Show available window Shift+F9
List Settings...
Update the List
Select Name
Symbol menu...   F8
Lines   L
Texts   T
Arcs   C
Areas   A
Favorites

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.

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 850, PARTICULARLY "CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR" ON PAGE 851.

In order to specify list settings, you first have to have a list in the project. Read "Insert drawing headers/formulas" on page 599 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 729.

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 contempt 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 365.

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 622.

Which data to include in the list

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 fixed accessories from Database**, the program will examine each component, and see whether they need any accessories, which must be included in the list. — Please note that "Optional electrical accessories" and "Optional mechanical accessories" are always included in lists. See "Accessories in the database" on page 676, in particular "Accessory types" on page 676.

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 or layer group, 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 IN PARTS AND COMPONENTS LISTS

The tab **Lists => List setup => Sort by** only exists for parts and components lists.

Here you can specify how to sort the contents in the list. You can specify a *Major sort* at the upper section of the dialog box and a *Minor sort* in the lower section:
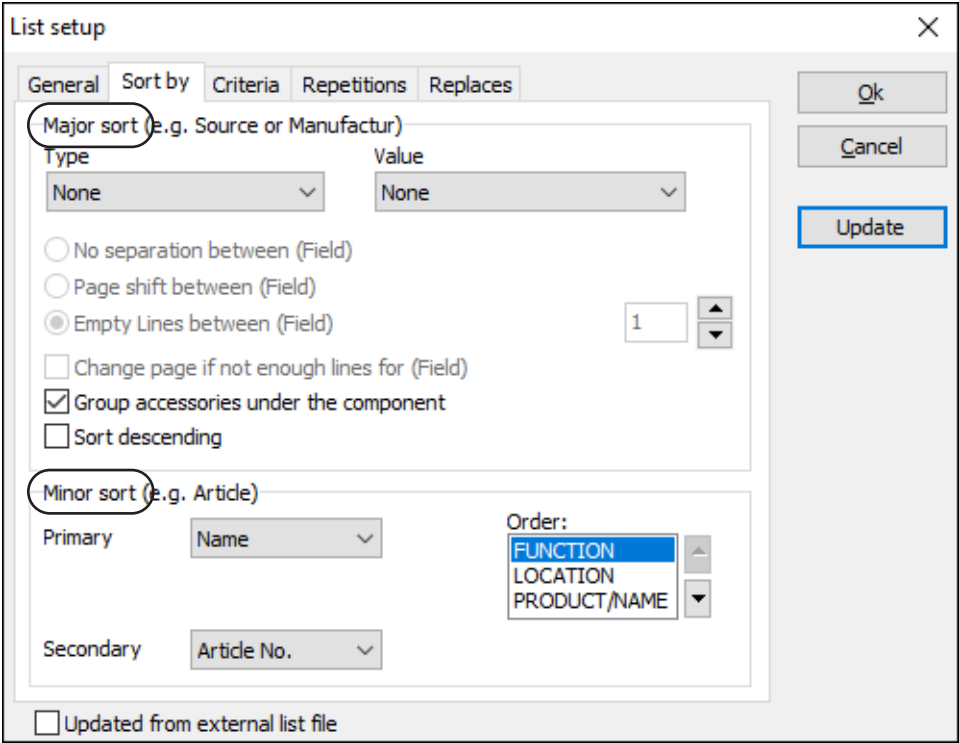
Major sort

Under *Major sort* in the upper section of the tab, you have the following options in the *Type* field:

Major sort: None

The sorting performed at specified under *Minor sort*.

Major sort: Database fields

When you have selected the type *Databasefields*, you can specify which field in the database to sort from in the *Value* field. This can for instance be applied for sorting the list by source or by manufacturer.
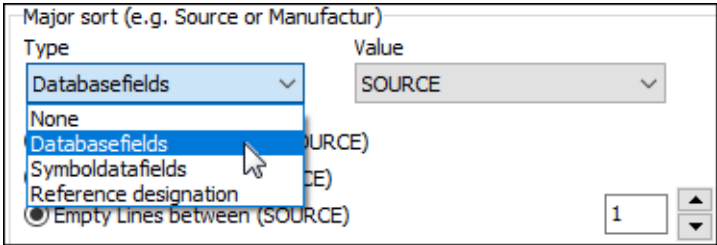
The option of sorting according to database fields are only displayed when working with an attached database. — Read more about databases in "Databases" on page 639.

Major sort: Symbol datafields

The option of sorting by **Symboldatafields** is only displayed when symbol data-fields are created in the project. — Read more about symbol datafields in "Symbol datafields" on page 122.

Major sort: Reference designations

When you have selected **Reference designation**, you get the option of specifying whether to sort based on the **Full reference designation**, **Function aspect**, **Location aspect** or **Product aspect**.

The option of sorting by reference designation is only available when reference designations are defined in the project. — Read more about reference designations in "Reference designations" on page 383.

Separation between e.g. manufacturers

Furthermore, you can specify how to separate the different manufacturers in the list — if you chose **Databasefields** and the value **Manufactur**.

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 (MANU-FACTUR)** above.

You can also choose **Group accessories under the component** and **Sort descending**.

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 615.

Sorting on all parts of the symbol name

Regardless of what you have specified as **Major sort**, components lists can also be sorted according to all parts of the symbol name in the **Minor sort** section; the function aspect (=), the location aspect (+) and the product aspect / name (-). — See "On reference designations" on page 385.

If for instance you want all components with the same locational aspect (e.g. *+A1*) to be listed together in the same components list, you just need to move *LOCA-TION* 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 *PRODUCT/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*, *LOCA-TION* and *PRODUCT/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.
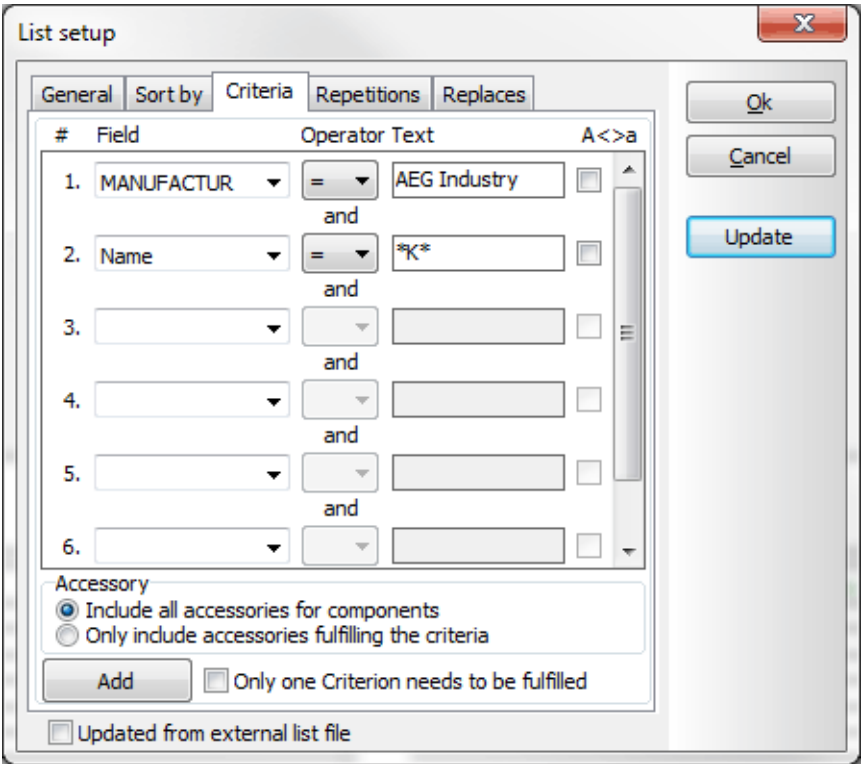
You can fill out seven lines, each containing one criterion. If you need to add more criteria, click **Add**.

Criteria for accessories

The settings on the **Criteria** tab specifies the criteria for components in the project, and not for their accessories. Accessories are therefore only included in lists, when the main component fulfil the criteria.

If you select **Include all accessories for components**, all accessories for components fulfilling the criteria, are included in the list.

If you select **Only include accessories fulfilling the criteria**, accessories are only included when they also fulfil the specified criteria.

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.
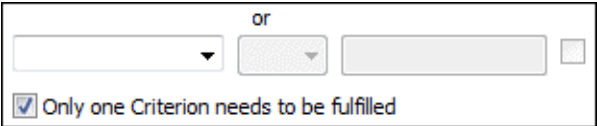
In the dialog box above this would mean that you get all components from AEG Industry, *as well as* all components having a **K** in the symbol name.

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** — as well as your own symbol datafields)
- Reference designations (the datafield **=+design.**) — see the figure below
- Symbol type — see "Symbol types" on page 785
- Symbol datafields — see "Creating symbol datafields" on page 811
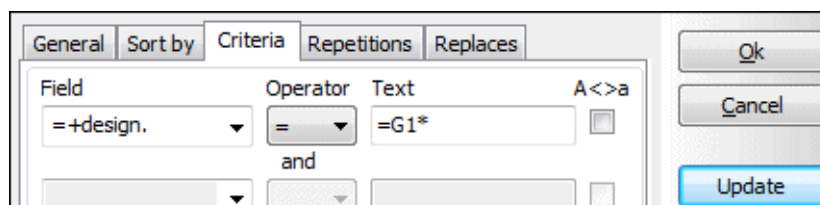
If you choose **Name**, **Type**, **Article no** or **Function** a list is displayed containing the applied names, types, article numbers or functions in the project. If you choose **=+-design.** you enter the dialog box **Reference designations**, where you specify which reference designation to use as criterion.

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 617.

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:

To empty a line on the tab, choose the blank field in the drop down box for the line.

On the tab you can specify 7 repetitions. Click the **Add** button at the bottom of the dialog box, if you need to specify more.

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 618.

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 data-field 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 (*UKDESCRIPT*), 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.

On the tab you can specify 7 replacements. Click the **Add** button at the bottom of the dialog box, if you need to specify more.

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 618.

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:

Specify no. of blank lines
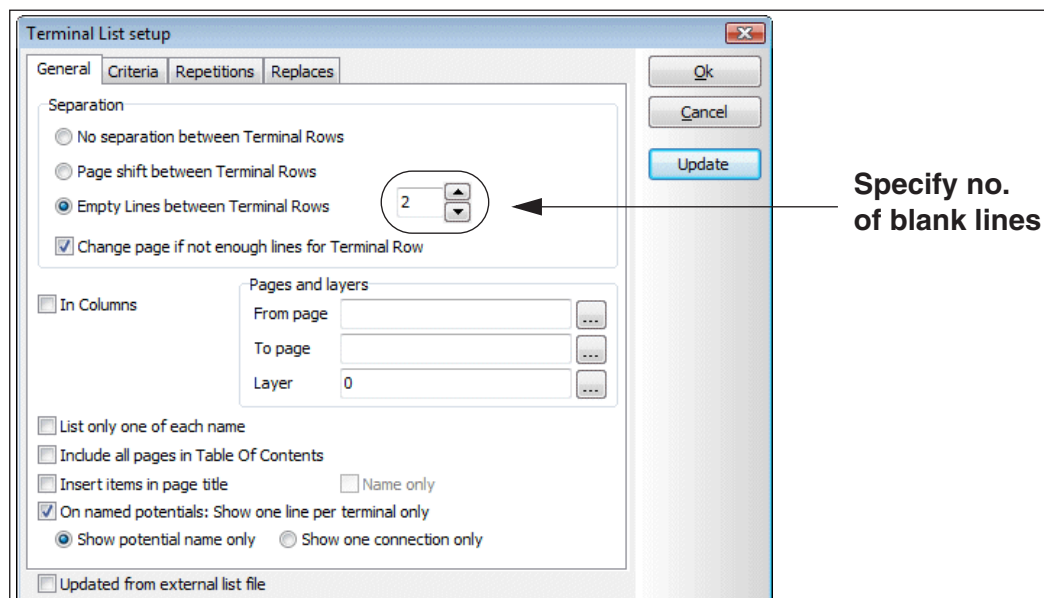
Separation

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**.
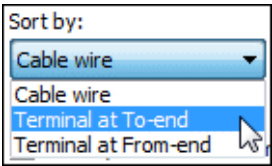
SORTING SO THAT CABLE WIRES FOR THE SAME CABLE SYMBOL ARE KEPT TOGETHER

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.
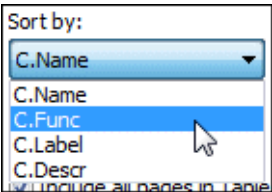
For terminals lists only

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 in stead 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 611. The **Criteria** tab is described in "Specifying Criteria for the components to include" on page 617.

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 811 and "Symbol datafields with predefined values" on page 813.

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.
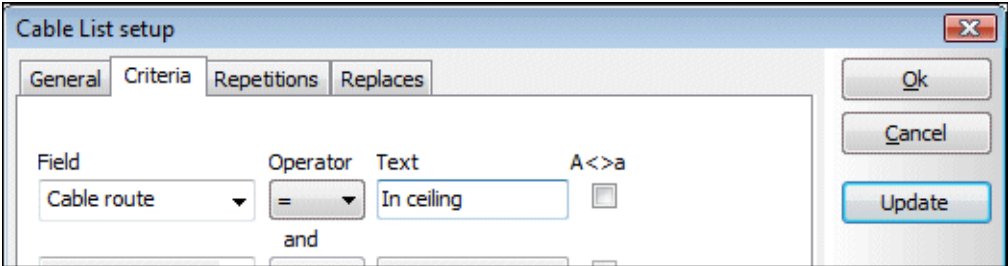
LIST SETTINGS FOR CONNECTIONS LISTS

To specify the settings for a connections list, you must activate the table of contents page and choose **Lists => List Settings**. You then enter this dialog box:
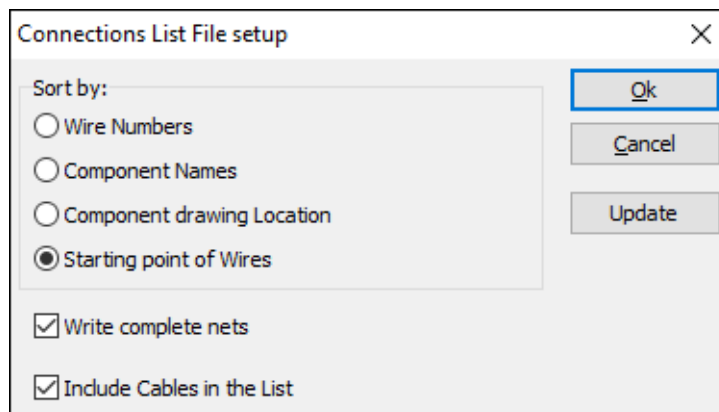
Here you can specify whether to sort the connections list by **Wire Numbers**, **Component names**, **Component drawing location** or **Starting point of wires**.

You can also specify whether to **Write complete nets** and/or whether to **Include cables in the list**.

LIST SETTINGS FOR TABLES OF CONTENTS

To specify the settings for a table of contents, you must open the table of contents page and choose **Lists => List Settings**. You then enter this dialog box:
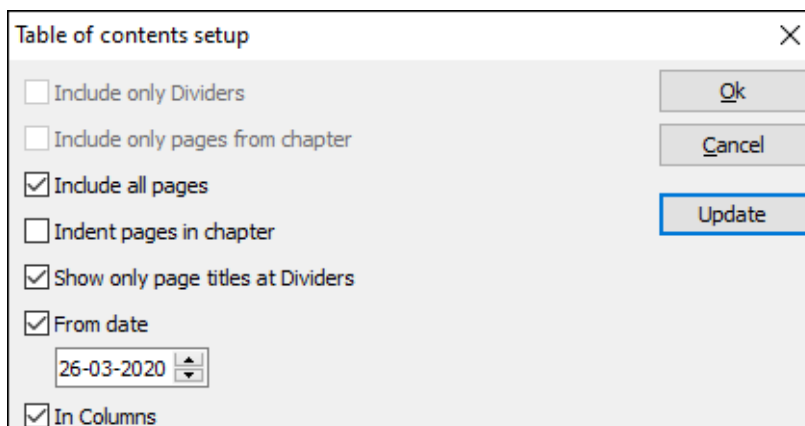
Here you specify what to include in the table of contents

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 *NOT* 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 E.T.C.) 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.
FROM DATE	HERE YOU HAVE THE OPTION OF QUICKLY CREATING A LIST OVER THE MOST RECENTLY CHANGED PAGES.
IN COLUMNS	— SEE "LISTS IN COLUMNS" ON PAGE 613.

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.

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 170.

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 631.

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 633.

Component grouping on mechanical pages

When working with projects containing pages of the type **Ground plan/Mechanical**, you can click on **Mechanical** at the bottom of the dialog box, which makes it possible to perform the component grouping on the mechanical symbols in the project.

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**.
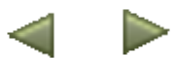
Also, you can scroll back and forth in the list of components by clicking on the **Previous component** and **Next component** buttons. Please note that the arrow-keys can be used as short-cut keys for these buttons.

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.

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) Type in the component group number *0* and click the **Transfer 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.

PLEASE NOTE: TERMINALS WITH COMPONENT GROUP NUMBER 0 CAN NOT HAVE ACESSORIES. *STARTING FROM VERSION 21 WE GENERALLY RECOMMEND NOT TO ASSIGN COMPONENT GROUP NUMBER 0 TO TERMINALS*.

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*.

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.
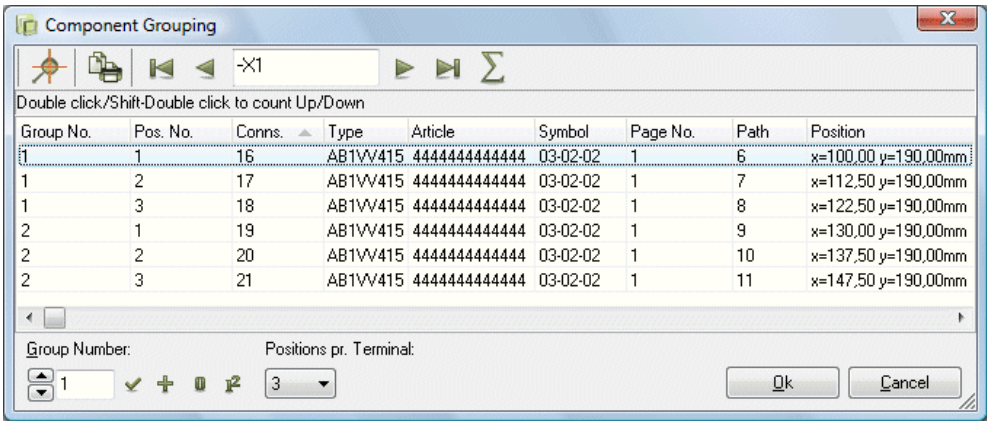
ASSIGNING GROUP- AND POSITION NUMBERS IN A SINGLE OPERATION

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.
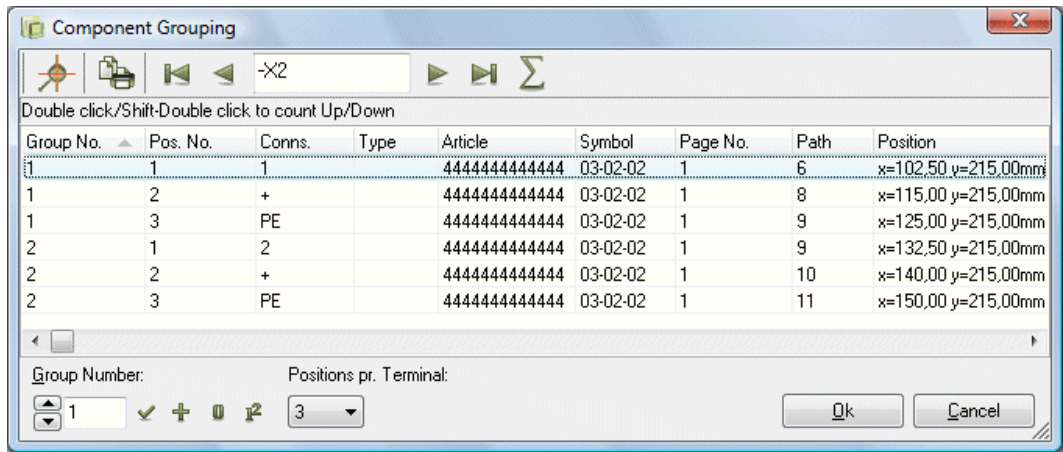
— Please also see "Keep component grouping when copying terminals" on page 632.

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**:
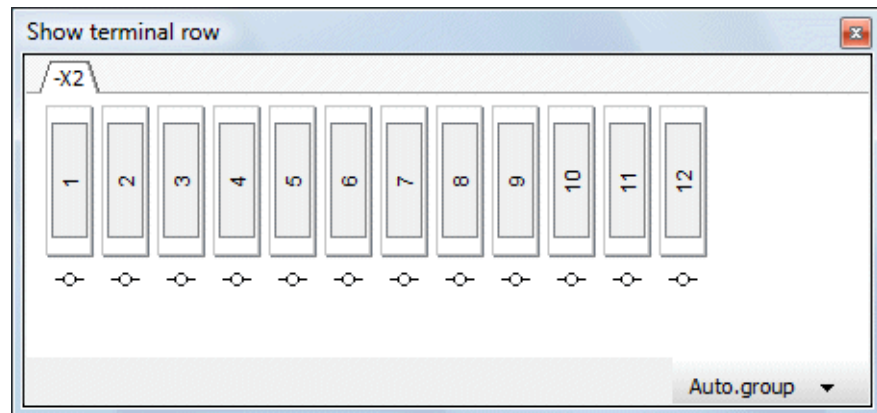
The significance of the order of the terminals

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 626.

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**.

Right-click here
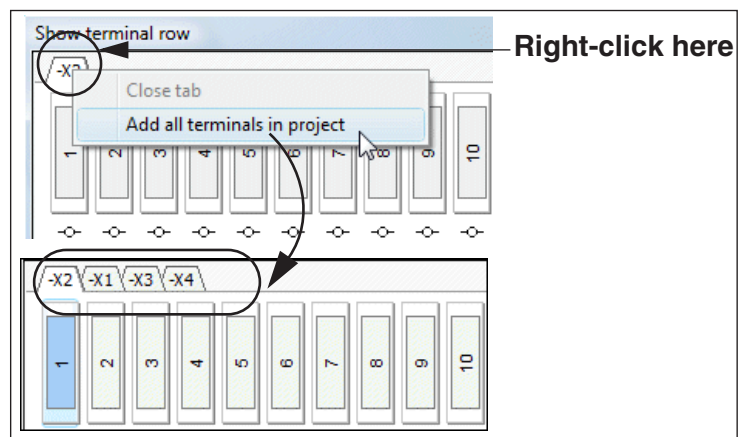
Selecting terminals

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.
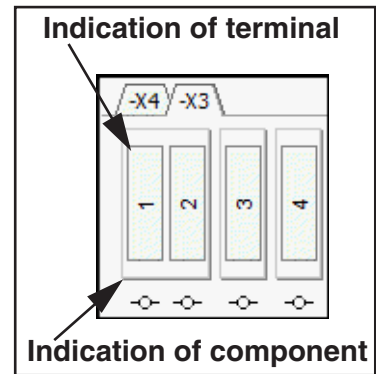
GROUPING BY CLICK AND DRAG

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:

A) Click on the terminal (2)

B) Drag it into the component frame around (1) and release the mouse

C) Terminal (1) and (2) are now grouped in the same component frame
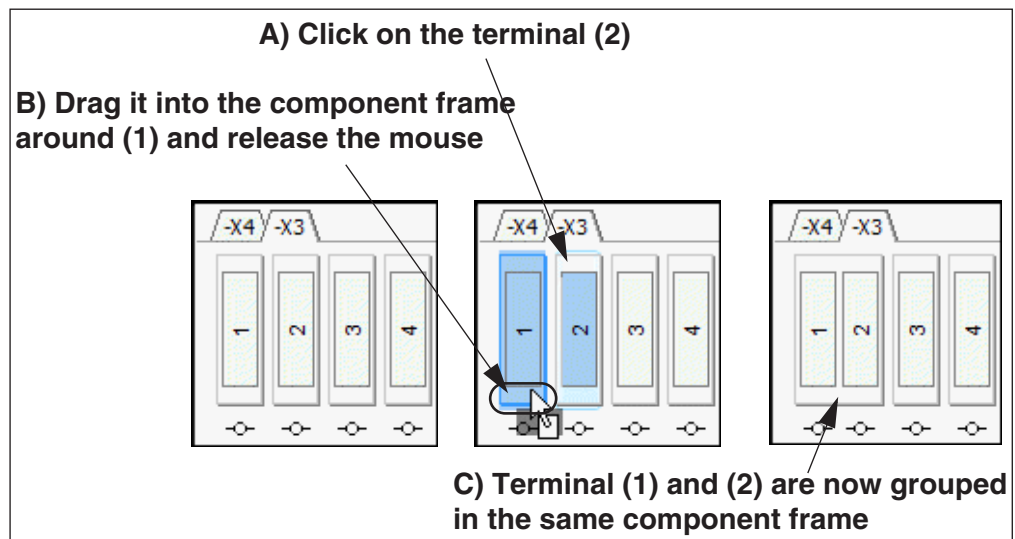
— Please also see "Changing the order of the terminals by click and drag" on page 634.

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**.
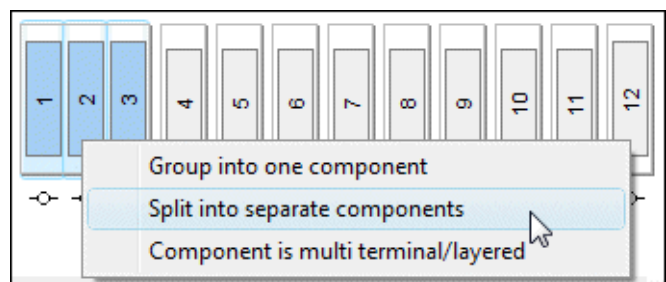
Changing the order of the terminals by click and drag

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:

A) Click on the terminal (3)

B) Drag it to a position where you want to place it (2) and release the mouse button

C) Terminal (3) is now placed between terminal (1) and (2)
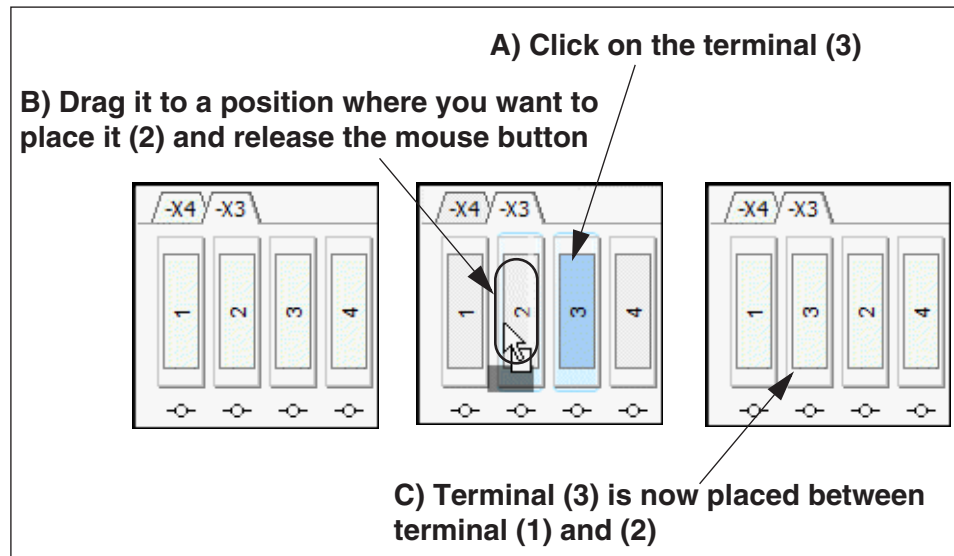
Specifying multi terminals / multi layer terminals

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.

Click here, and the terminal is selected in the project
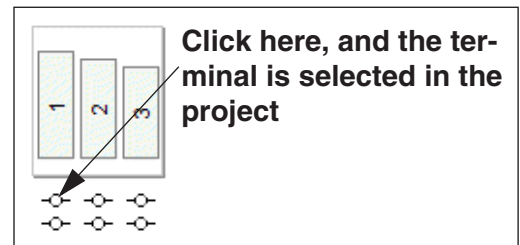
Example on graphical grouping of terminals

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 com-
ponent**.
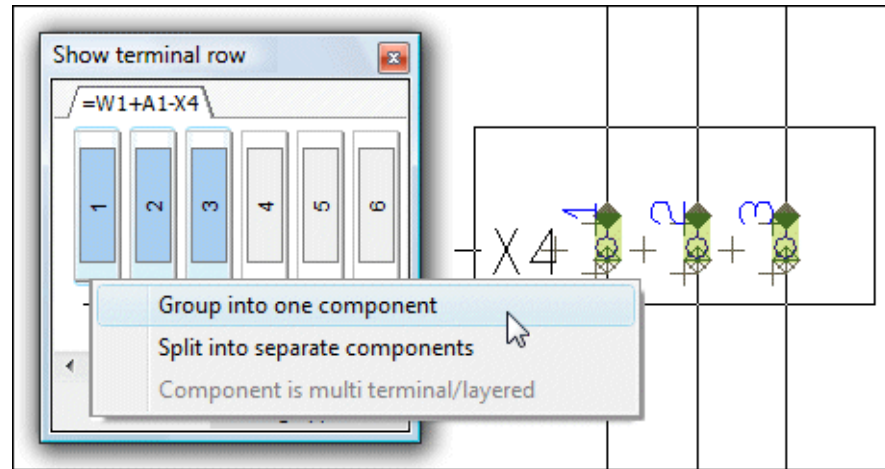
4) The terminals are now grouped as one component.

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.
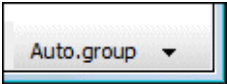
When you click the down-arrow next to the **Auto-group** button and choose **Auto.group all shown terminal rows**, all terminal rows in the **Show terminal row** window are grouped.

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:
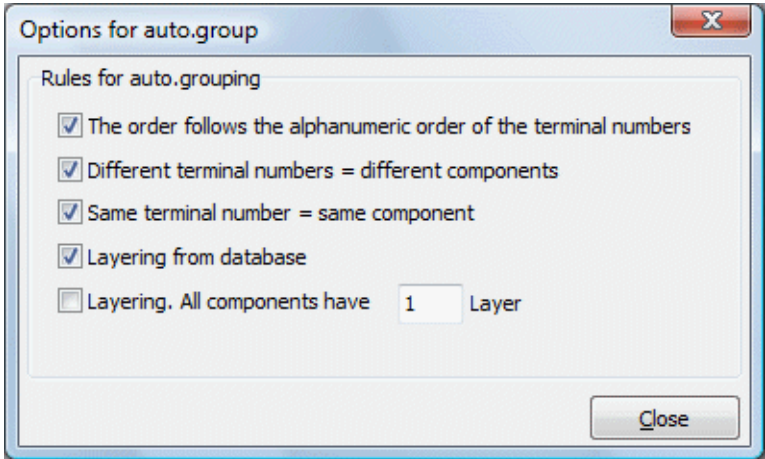
Here you have the following options:

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 symbols or lines in the project, so that you by right-clicking on the symbol or line get the option of jumping to the unit drawing.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO INSERT UNIT DRAWINGS IN PROJECTS, AND HOW TO ATTACH SYMBOLS AND LINES TO THESE.
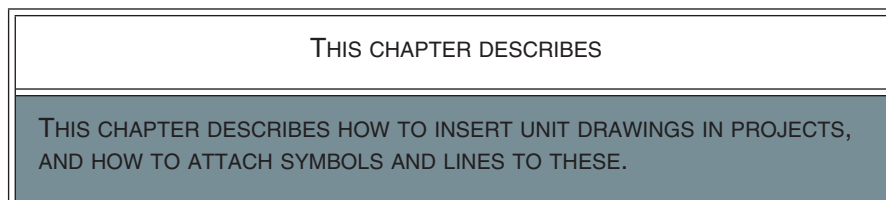
When you use extended parts lists in your project, articles from the unit drawings are included in the parts lists. Read more about this in "Specifying from where to collect components" on page 611.

— See how to create a unit drawing in "Creating Unit Drawings" on page 846.

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.

CONNECTING SYMBOLS/LINES ON DIAGRAMS TO UNIT DRAWINGS

When you have created a unit drawing in your project, you attach it to a component in the project like this:

1) Activate **Symbols** and double-click on a symbol for the component.

2) In the dialog box **Component data**: Click the **Unit** button.

3) In the dialog box **Select unit page** all unit drawings in the project are displayed: Click on the relevant unit drawing and click **OK**.

You have now attached the unit drawing to the symbol.

When you then right-click on the symbol and choose **Open**, you will automatically go to the page with the unit drawing for the symbol.

If in stead you start by activating **Lines**, you can do the same for lines.

DATABASES

In this section you can see how to use the database in PCSCHEMATIC Automation.

You can read about:

READ SURVEY ARTICLE - WATCH VIDEOS

READ THE SURVEY ARTICLE ABOUT WORKING WITH DATABASES AND LISTS AT WWW.PCSCHEMATIC.COM/ ARTICLES.

WATCH THE VIDEOS ABOUT HOW TO CREATE YOUR OWN COMPONENTS AT THE PCSCHEMATIC YOUTUBE CHANNEL:

www.youtube.com/PCSCHEMATIC

PC | SCHEMATIC

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.

You can also read about the database in the **PCSCHEMATIC 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 248.

SEE ALSO THE SURVEY ARTICLE ABOUT WORKING WITH DATABASES

IN THE SURVEY ARTICLE IN THE BEGINNING OF THE SECTION "LISTS" ON PAGE 597, 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.
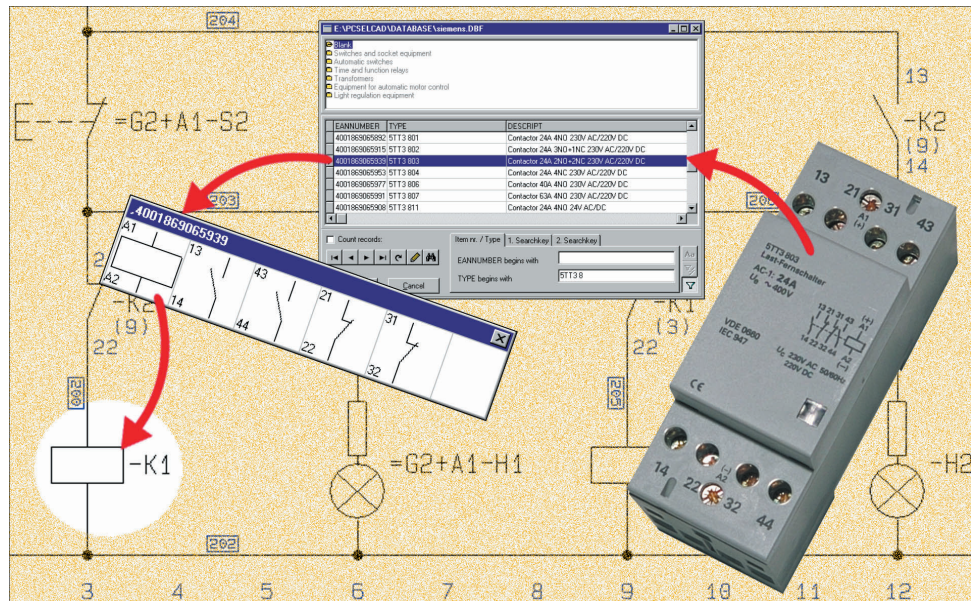
When you have completed your electrical diagrams, the program can therefore automatically fill out components lists and parts lists etc. — without you having to type in any component information.
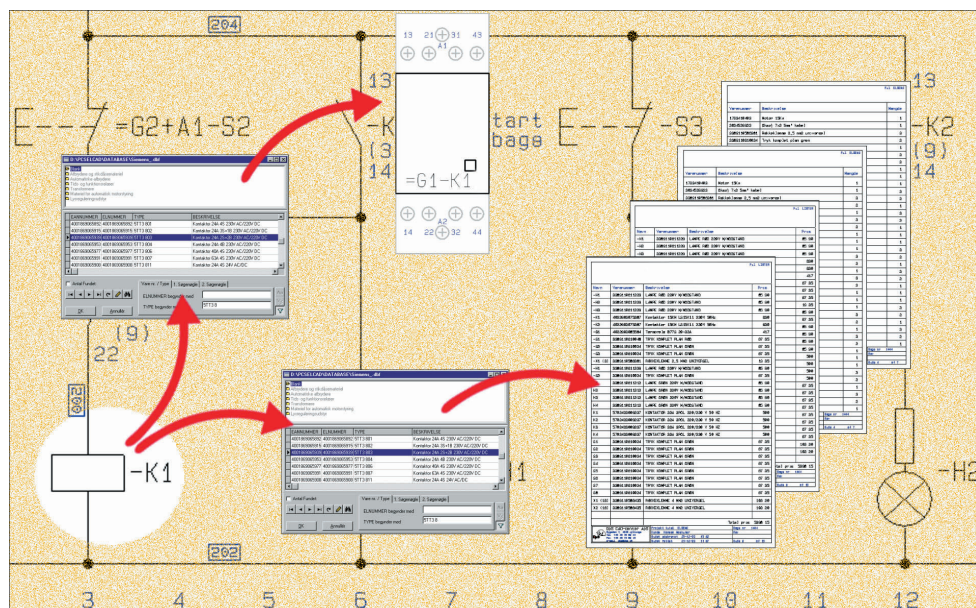
Originating from the electrical symbols on the diagrams, the program can also fetch the mechanical symbols for the components automatically. These symbols can for instance be used for mechanical layouts.

— Read more about this at www.pcschematic.com/articles.

ATTACHING INFORMATION FROM THE DATABASE TO A SYMBOL

In "Fetching symbols" on page 154 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.
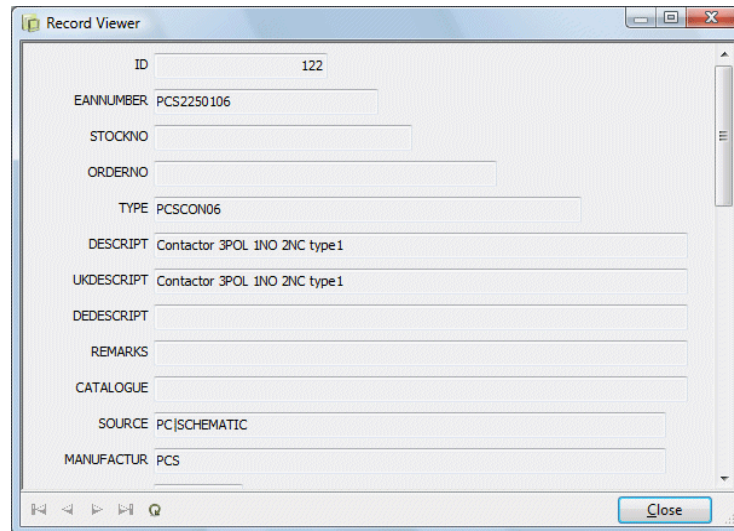
Click **OK**.

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**:

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 652.

Click on **Close**.

APPENDIX FOR DATAFIELDS IN THE COMPONENT DATABASE

IN "DATAFIELDS IN THE COMPONENT DATABASE" ON PAGE 969 YOU CAN SEE WHICH DATAFIELDS ARE CREATED IN THE COMPONENT MANUFACTURER DATABASE, WHICH IS DELIVERED WITH THE **Automation** SOFTWARE.

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 170.

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.
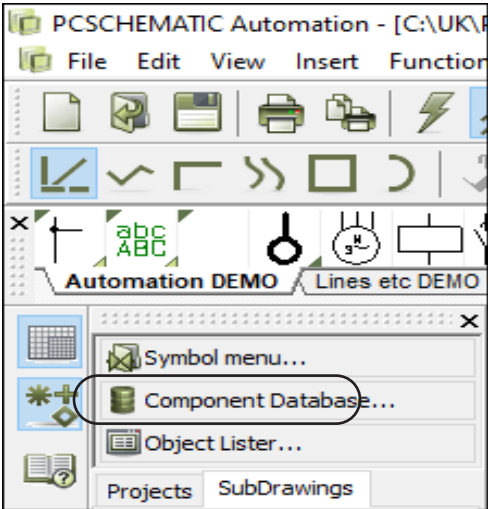
The **Database menu** contains folders, each containing different groups of components.

Menu for the database — **Records in database** — **Open links (if any) for the selected record** — **Scroll/search among the records displayed** — **Specify which records to display**
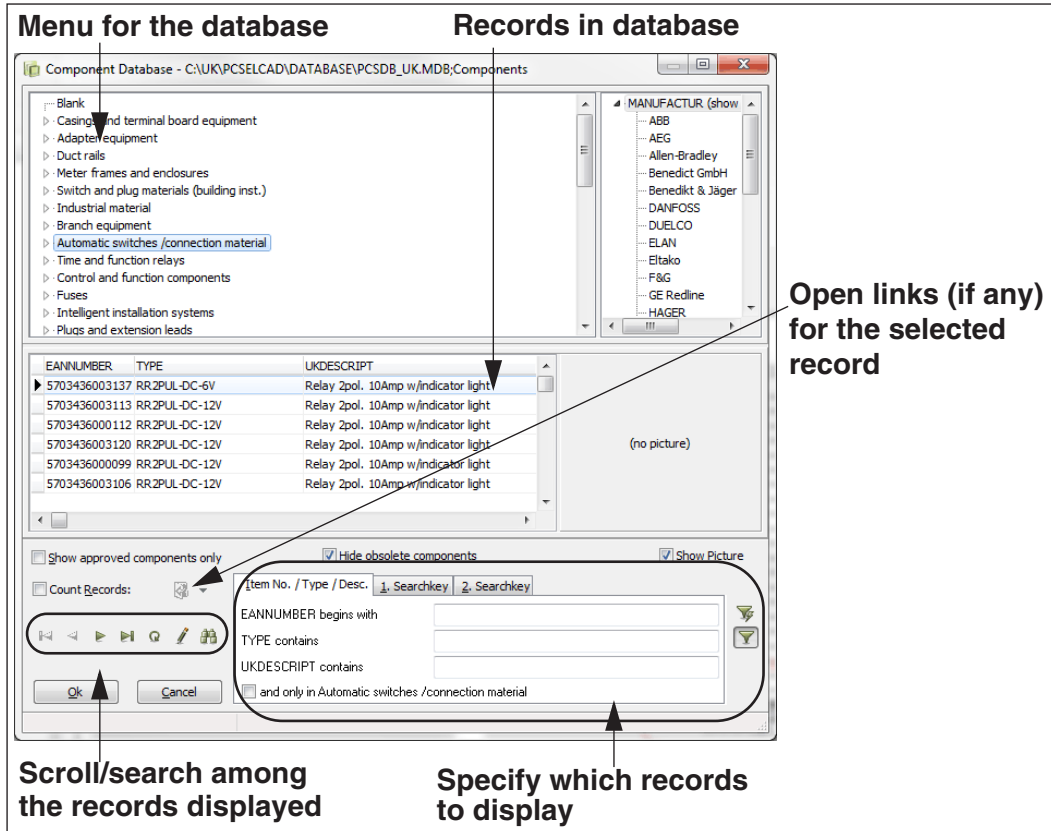
If you are using the **PCSCHEMATIC Database** program you can create your own database menu. — Please refer to the **PCSCHEMATIC Database** manual for further details.

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 **PCSCHEMATIC 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 682.

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 tablemenu 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 689.

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 item ***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 item 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 / Desc.** in the lower-right side of the dialog box. Here you type in the number in the field ***EANNUMBER begins***

**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 item 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.
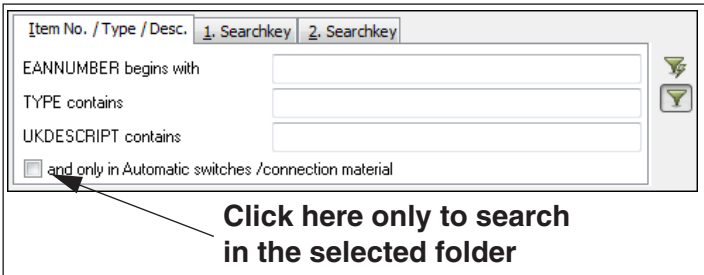
SPECIFYING AN ALTERNATIVE SEARCH KEY

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>.

Datafield in the database   **Operator**   **Text**

Here you choose which datafield you want to search on, an operator, and a text to compare the contents of the datafield with.
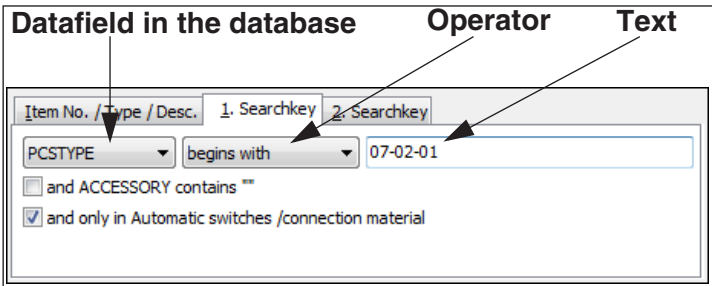
In the figure above the following is chosen:

- The datafield **PCSTYPE**

- The operator *begins with*

- The text *07-02-01*

Operators

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.
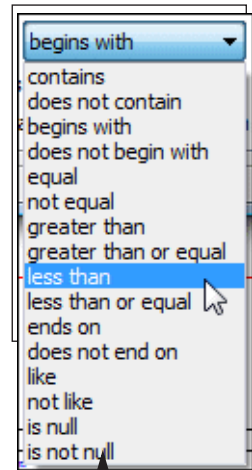
Applying 1. and 2. Search key

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.

Search in the selected folder only

Apply both search keys

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*.
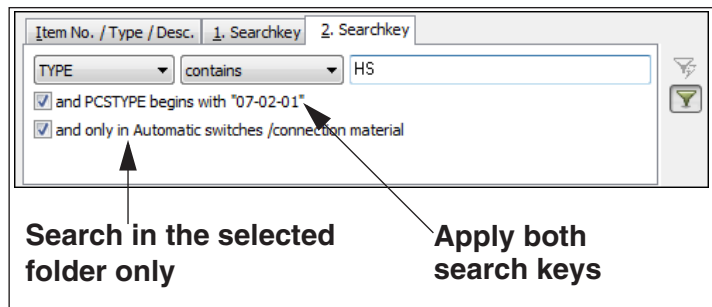
When you want to search in the folder chosen in the upper part of the menu, you enable *and only in Automatic switches*. (Accordingly, the name of the folder will vary). If this function is not enabled, the search will be carried out in the entire database.

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.
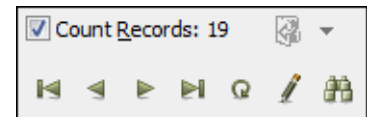
Leafing through the found records

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 652.

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 361.

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 692.

> **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 691.

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 692.

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
	✔	1122084084	LS 37.22	Contactor, 15KW, LS 37.22, 230V AC
✔		4022903085584	B77S	Overload relay, 20-32A
✔		3389110231632	LC1D4011M7	Contactor, LC1D4011M7, 220V-40A
✔		3389110232936	LR2D3353	Overload relay, 23-32A
✔		4022903075387	LS15K11	Contactor, 15KW, LS15K11, 230V AC
	✔	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 691.

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.

Item No. / Type   1. Se

CATALOGUE ANNUMBER begins w
TYPE begins with

Cancel   ☐ and only in Automat

%Danfoss%\CI32.pdf

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 696. 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.
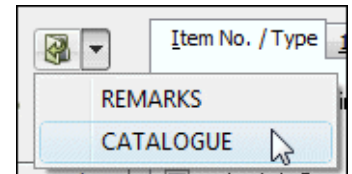
SHOW PICTURES OF COMPONENTS IN THE DATABASE

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 696, particularly in "Example for pictures" on page 697.

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 645 — 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:
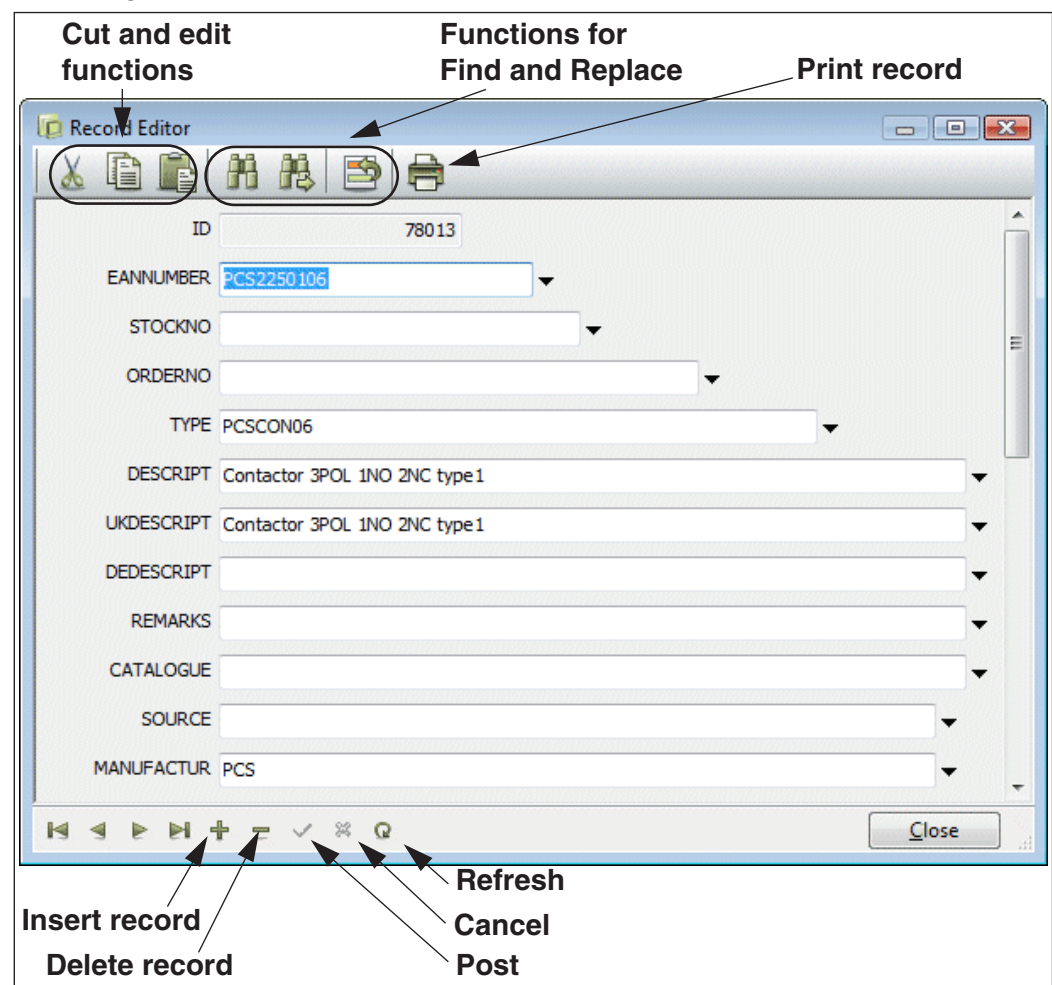
Cut and edit functions   Functions for Find and Replace   Print record

Insert record
Delete record
Refresh
Cancel
Post

APPENDIX FOR DATAFIELDS IN THE COMPONENT DATABASE

IN "DATAFIELDS IN THE COMPONENT DATABASE" ON PAGE 969 YOU CAN SEE WHICH DATAFIELDS ARE CREATED IN THE COMPONENT MANUFACTURER DATABASE, WHICH IS DELIVERED WITH THE **Automation** SOFTWARE.

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 361.

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 loosing 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 657.

Specify what to include in the project component database

To specify which database fields to include in the project component database, select **Settings => Database**:
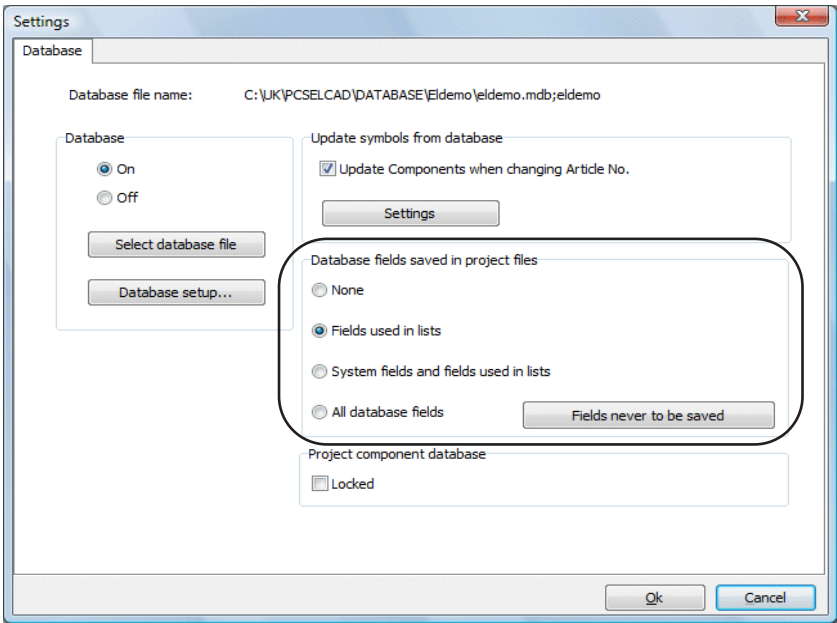
Below ***Database fields saved in project files***, you have the following options:

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
ALL DATABASE FIELDS	ALL DATABASE INFORMATION FOR THE USED COMPONENTS ARE INCLUDED. PLEASE NOTE: ANY TYPE OF PRICE INFORMATION WILL ALSO BE INCLUDED.
FIELDS NEVER TO BE SAVED	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**.
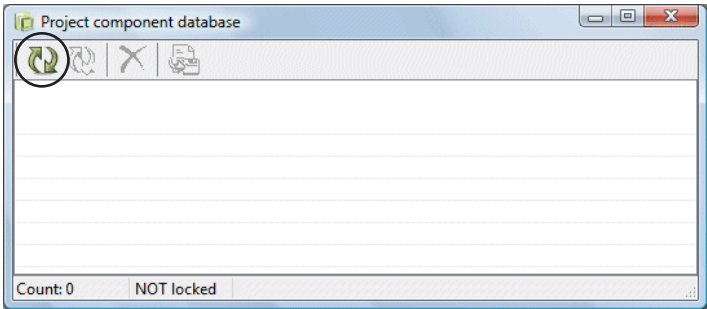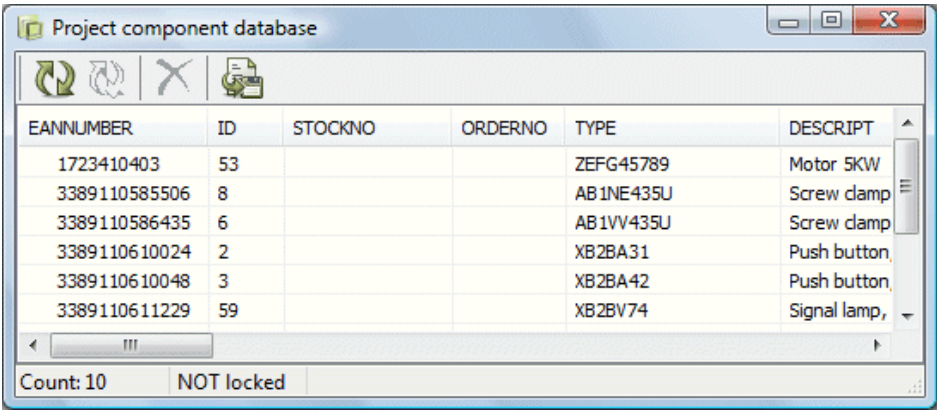
2) In the dialog box **Project component database**: Click the **Update** button.
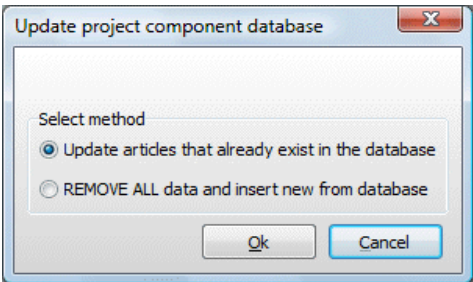
3) In the dialog box **Update Project component database**:
When you select ***Update articles that already exists in the database***, only article numbers existing in the project component database will be updated.
When you select ***REMOVE ALL data and insert new from database***, all data in the project component database are deleted, and are inserted again from the program component database.
Click **OK**.

PC | SCHEMATIC

4) The project component database is now updated:
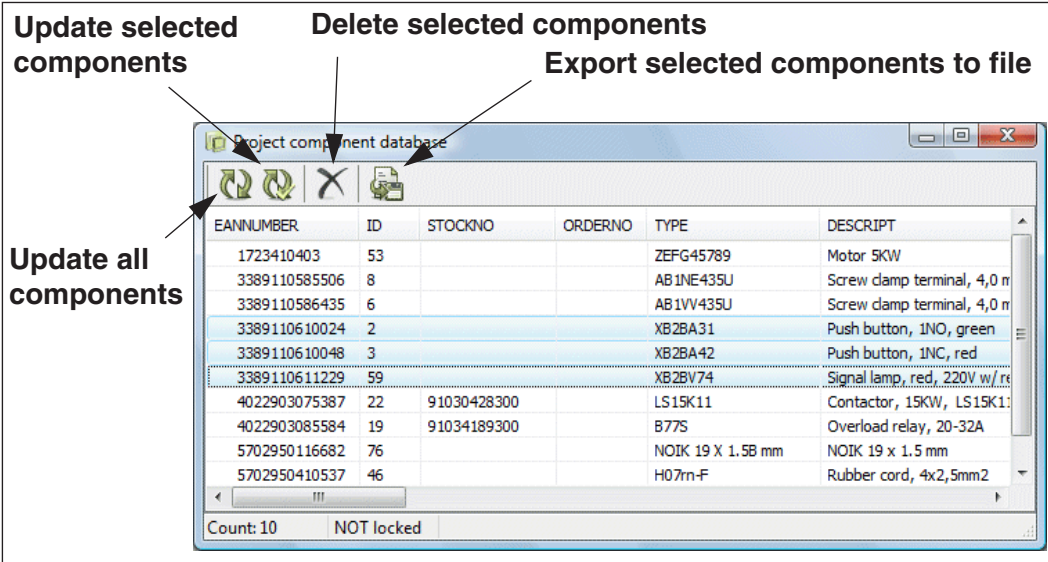
ALSO UPDATED WHEN LISTS ARE UPDATED

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:

When you right-click on a field in the dialog box **Project component database**, you have these options:
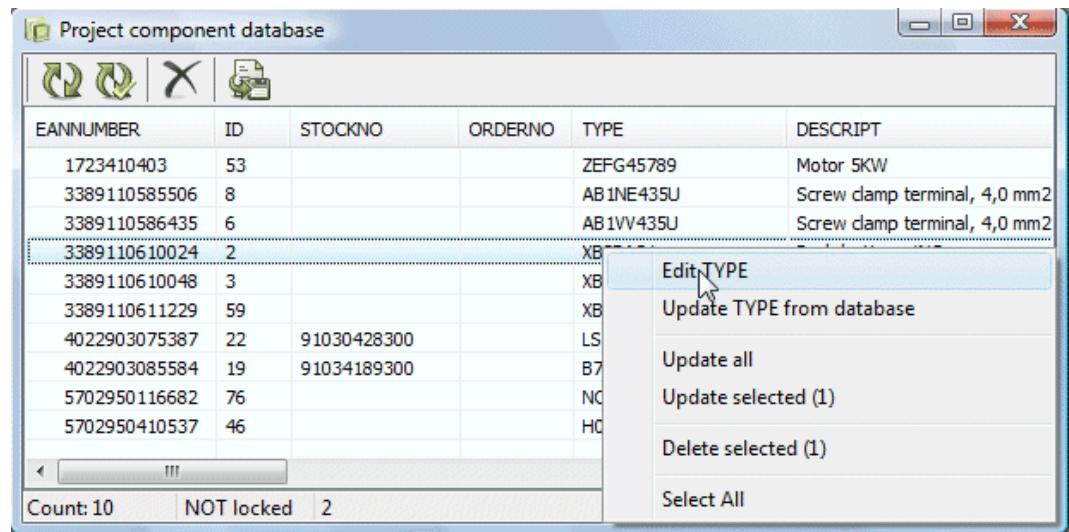
Locking the project component database for the active project

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.
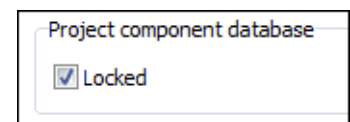
Generally speaking, you can say that:

- 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 PCSCHEMATIC 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 PCSCHEMATIC 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 PCSCHEMATIC database.

- Component data for all *placed* project components are fetched from the project component database. Data are *only* fetched from the PCSCHEMATIC 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 656.
If for instance the **PCSTYPE** datafield is not included in the project component database, this has the consequence that you can not use the **Available symbols** function in the **Component menu**, 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.

<table>
<tr><td align="center">THIS CHAPTER DESCRIBES</td></tr>
</table>

In the manual for the included database program **PCSCHEMATIC Database** you can read more about the options for structuring your database — and much more.

> **YOU CAN ALSO USE THE COMPONENT WIZARD - WATCH VIDEOS!**
> THERE ALSO EXISTS A "COMPONENT WIZARD", WHICH GUIDES YOU THROUGH CREATING NEW COMPONENTS FOR PCSCHEMATIC DATABASES. IT IS ACTIVATED BY CHOOSING **FUNCTIONS => COMPONENT DATABASE => COMPONENT WIZARD**, AND IS DESCRIBED IN A MINI MANUAL, WHICH YOU CAN FIND UNDER **HELP => MANUAL FOR COMPONENT WIZARD**.
> WATCH THE VIDEOS ABOUT HOW TO CREATE YOUR OWN COMPONENTS AT THE PCSCHEMATIC YOUTUBE CHANNEL:
> WWW.YOUTUBE.COM/PCSCHEMATIC

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 775

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 560

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

Electrical symbols on the Diagram page

Mechanical symbol on the Ground plan page
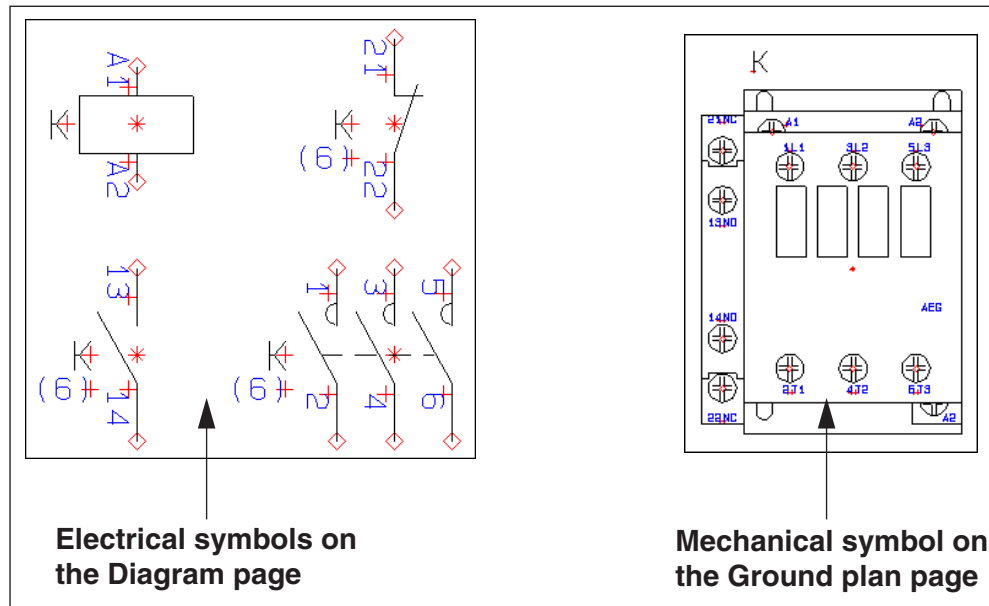
You are now prepared to create the component in the database, as described below.

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 MEANING 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 659 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 661

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 662.

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 652.

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 659 — 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**.

PC|SCHEMATIC

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 663.

Here the use of the following datafields are elaborated:

DATAFIELD	IS DESCRIBED IN
PCSTYPE	"ELECTRICAL SYMBOLS" ON PAGE 663. (SEE "THE DATABASE FIELD PCSTYPE IN THE COMPONENT DATABASE" ON PAGE 961).
PINDATA	"CONNECTION NAMES/PINDATA" ON PAGE 669. (SEE "THE DATABASE FIELD PINDATA IN THE COMPONENT DATABASE" ON PAGE 963 AND "THE DATABASE FIELD PINDATA: COUNTING" ON PAGE 965).
MECTYPE	"MECHANICAL SYMBOLS IN THE DATABASE" ON PAGE 674. (SEE "THE DATABASE FIELD MECTYPE" ON PAGE 967).
ACCESSORY	"ACCESSORIES IN THE DATABASE" ON PAGE 676.
REFID	"SPECIFYING REF. LETTER FOR THE SYMBOL IN THE DATABASE" ON PAGE 679.

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 **MEC-TYPE**. (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 field **PCSTYPE** you specify the electrical symbol(s) that represents the component on diagrams in **Automation**.

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 923.

PCSTYPE	07-02-01#3;07-02-03
PINDATA	53,54;73,74;83,84;61,62
MECTYPE	LA1-DN31

OPTIONS FOR ELECTRICAL SYMBOLS IN THE DATABASE

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 664.

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 under **Available symbols** in the lower-right corner of the screen:
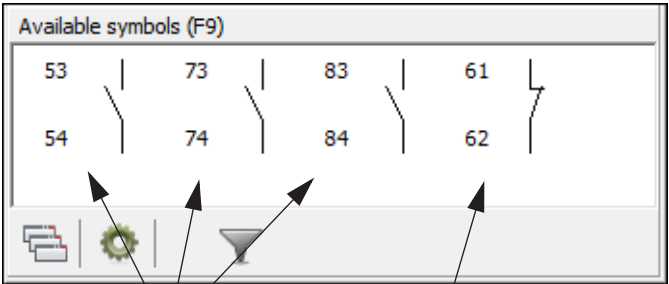
The symbol 07-02-01.sym   The symbol 07-02-03.sym

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, 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 *Open*, you then write **07-01-03=O**. — However, you cannot apply this facility on multi symbols — see "Creating a symbol as a Multi symbol" on page 794 — 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
N	NORMAL	S	SWITCH
O	OPEN	T	TERMINAL
P	PLC		

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 under **Available symbols** in the lower-right corner of the screen:
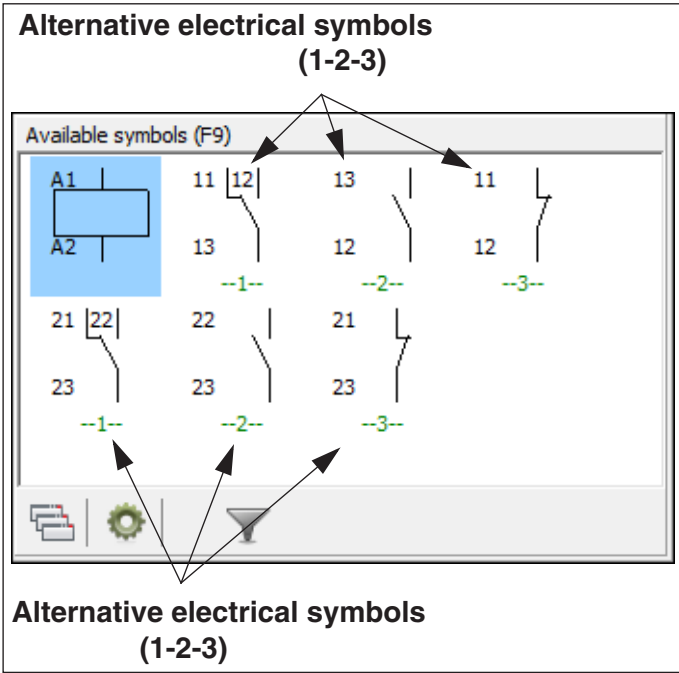
When you fetch one of these alternative electrical symbols, the other alternative symbols for the function disappears. — Read more about alternative electrical symbols in "Alternative electrical symbols" on page 157.

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.
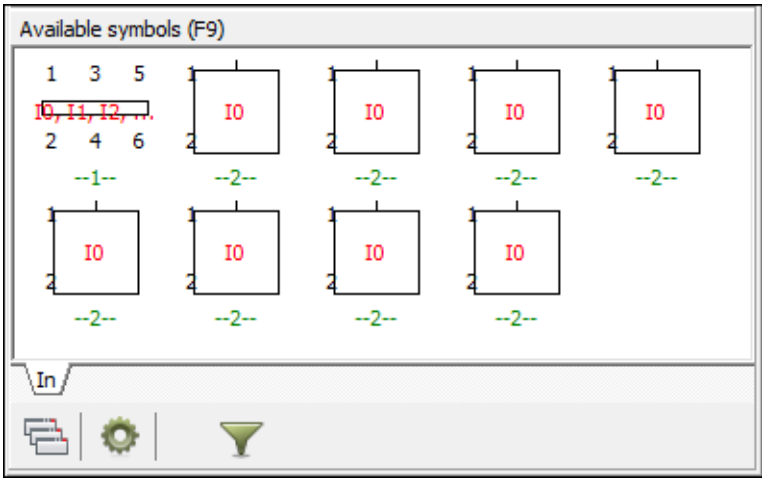
If you choose one of the symbols with one input, the symbol with the eight inputs disappears from the pickmenu. Also, the eight other symbols disappear if you choose the symbol with the eight outputs.

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.
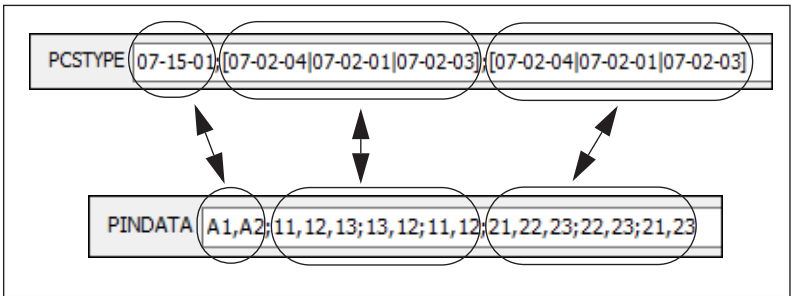
— Please note that you shall not mention anything about alternative **PCSTYPE**s in the **PINDATA** field.

FETCHING PINDATA DIRECTLY FROM THE SYMBOL DEFINITION

If pindata for a symbol is to be fetched directly from the symbol definition — and *not* from the database — just type a semicolon in the PINDATA field, and continue specifying pindata for the subsequent symbols. Thus, if for instance the first specified symbol under PCSTYPE should be assigned pindata from the symbol definition, simply let the contents of the PINDATA field start with a semicolon, and then specify the pindata for the remaining symbols.

Specifying symbol state for electrical symbols via the database

For electrical symbols created with states — see "Creating symbols with states" on page 791 — 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 190 and "Symbols with selected states and the Component menu" on page 190.

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**, **Switch**, **Open** or **Close**), unless the symbol type have first been changed using the **=** function — see "Specifying symbol state for electrical symbols via the database" on page 666.

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**).

PC|SCHEMATIC

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 under **Available symbols** in the **Component menu** when you are working with the component, 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 additionals to components in the field **ADDI-TIONALS** — as described in "Accessories in the database" on page 676. When you specify (sub)components here, these will be displayed as accessories in the **Component menu** under **Symbols for component**.

— Please also see "Creating Symbols" on page 775, particularly "Symbol types" on page 785.

SUPPORT SYMBOLS ARE BEING PHASED OUT - USE ACCESSORIES!

EVEN THOUGH YOU ARE ABLE TO USE SUPPORT SYMBOLS IN THE PROGRAM, THEY WILL BE PHASED OUT IN THE LONG RUN.

IN STEAD, WE RECOMMEND THAT YOU SPECIFY ACCESSORIES TO COMPONENTS IN THE FIELD **ACCESSORY** — AS DESCRIBED IN "ACCESSORIES IN THE DATABASE" ON PAGE 676. IF YOU SPECIFY (SUB)COMPONENTS HERE, THESE WILL BE DISPLAYED AS ACCESSORIES IN THE **COMPONENT MENU** UNDER **SYMBOLS FOR COMPONENT**.

— ALSO SEE "CREATING SYMBOLS" ON PAGE 775, IN PARTICULAR "SYMBOL TYPES" ON PAGE 785.

Connection names/Pindata

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 775.

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.

Connection name: 1

Connection name: 2

Names when designing a symbol

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 663.

When you place the symbol on a diagram, it will be assigned the connection names, you have specified in the datafield **PINDATA**.

Connection name (1)

Connection name (2)

Names when the symbol is placed using the database
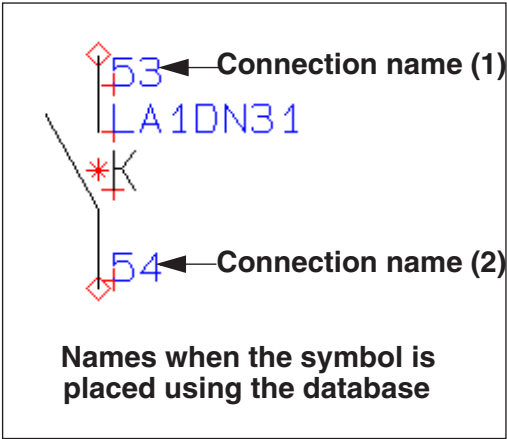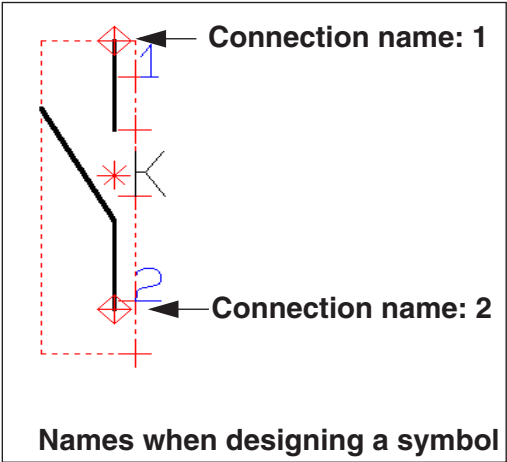
Connection names for single line symbols

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 182, "Connection data with a variable part in the name" on page 780 and "Wildcards/jokers" on page 618.

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 680.

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
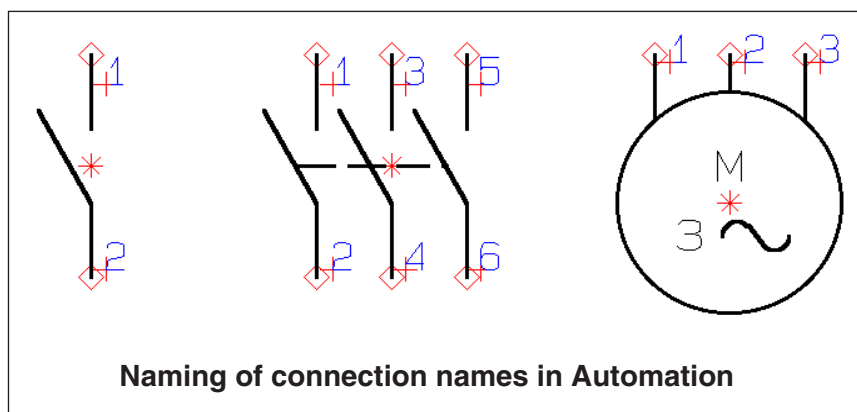
In the already designed symbols, the connection names start with the number one and is incremented following the reading direction.

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 672).

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.
/I:N	Sets IO status to **None, None** (deletes IO status)

EXAMPLE: TWO PLC SYMBOLS WITH ADDRESS AND IO STATUS SPECIFICATION:

PCSTYPE = PLCO1;PLCI1

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

In the dialog box Symbol article data: Type in a level number

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 672), I/O status set to ***Output PLC*** (see "Syntax for specifying IO status" on page 671) 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. 672)
— ENUMERATED NAMES FOR SYMBOLS WITH ONE OR MORE CONNECTION POINT(P. 672)
— ENUMERATED PLC ADDRESSES IN THE PINDATA FIELD .... (P. 673)
— COUNTING FUNCTIONS FOR PLC ADDRESSES IN THE PINDATA FIELD ..... (P. 673)
— SPECIFYING A NUMBERING SYSTEM .... (P. 673)
— INCREMENTING BY MORE THAN ONE .... (P. 673)
— FIXED NUMBER OF DIGITS ON COUNTING FIELD .... (P. 673)

— Please also see "Connection names/Pindata" on page 669 and "Connection names in files" on page 670.

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 in the database

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** — for instance *PCSMV1*.

Please note that you can also specify that other fields in the database contains mechanical symbols — see "Mechanical symbol" on page 687. 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 PNEUTYPE, FRONTTYPE and MECTYPE.

lige formål — f.eks. PNEUTYPE, FRONTTYPE og MEKTYPE.
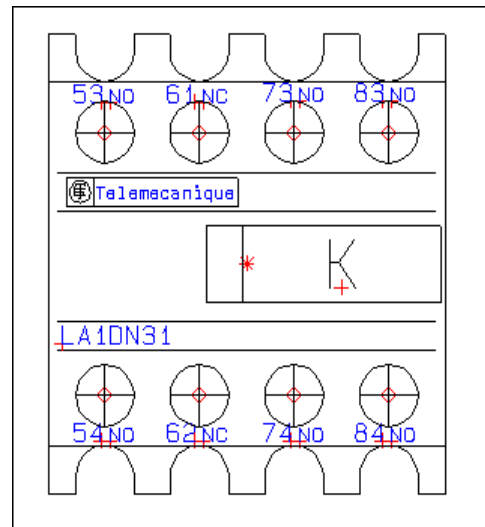
Specifying multiple mechanical symbols for a component

If a component contains multiple mechanical symbols, these are specified separated by a semicolon: *PCSMV1;PCSMV5*

When the component contains multiple mechanical symbols of the same type, this is specified by typing the symbol name followed by a # character, which again is followed by the number of mechanical symbols. To specify two symbols of the type PCSMV1, write this: *PCSMV1#2*

Specifying optional mechanical symbols

To specify that you can choose either PCSMV1 *or* PCSMV5, write this: *[PCSMV1|PCSMV5]*

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 791 — 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 708 and in "From Mechanical to Electrical drawing" on page 720.

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.

When you work with accessories, it is recommended that all accessories are created in the database.

SPECIFYING ACCESSORIES IN THE DATABASE

Accessory types

It is possible to specify these types of accessories:

- **Fixed accessories**: Accessories attached to the "main component" without placing any electrical or mechanical symbols for the accessory in the project. When the main component is placed in the project, the fixed accessories for it is included in component and parts lists.
  — See "Specifying that a component has fixed accessories" on page 676.

- **Optional mechanical accessories**: Can be fetched from the database when selected. When electrical symbols for the main component are placed on a *Diagram* page, the mechanical symbol for the accessory can be placed on *Ground plan/Mechanical* pages. Optional mechanical accessories are always included in component and parts lists.
  — See "Optional accessories - Mechanical" on page 678.

- **Optional electrical accessories**: Can be fetched from the database when selected. The electrical symbol(s) are placed on *Diagram* pages in the project. A mechanical symbol will typically also be attached to the accessory component. Optional electrical accessories are always included in component and parts lists.
  — See "Optional accessories - Electrical" on page 679.

Specifying that a component has fixed accessories

In the **Automation** program: Under **Settings => Database => Database setup => Component accessories** in the field *Fixed accessories* you can specifywhich field in the attached database that contains fixed accessories. In normal installations of the software, the database field **ACCESSORY** is used for this purpose.

In the database: To specify, that a component has fixed 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 fixed accessory part, you type *#* followed by the number needed, after the EAN-NUMBER of the accessory part in the **ACCESSORY** field.

Two of this part

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 fixed accessories in your project lists. — Please refer to "Specifying from where to collect components" on page 611 for further details about this.

— See "Support symbols in the PCSTYPE field" on page 668 also.

Fixed accessories for cables

For cables the amount of fixed 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 669 — 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 670.

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=112345566;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 611 — 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 638.

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 677.

Optional accessories - Mechanical

Optional mechanical accessories are mechanical accessories for a component which can be fetched from the database when necessary. — This being for instance end plates and dividers for terminal rows.

When the electrical symbol for the main component is placed on a **Diagram** page, the optional mechanical symbol(s) for the accessories can subsequently be placed on **Ground plan/Mechanical** pages.

Optional mechanical accessories are always included in component and parts lists.

Until version 16, this type of mechanical accessories were specified using support symbols, which still is an option — even though it is not recommended. Read more about this in "Support symbols in the PCSTYPE field" on page 668.

— Please also see "Component accessories" on page 688, in particular "Optional accessories - Mechanical" on page 689.

Add mechanical accessories

In "The Accessory tab" on page 184 you can see how to add mechanical accessories to a component directly from the active project.

Optional accessories - Electrical

Optional electrical accessories are electrical accessories for a component which can be fetched from the database when necessary. — This being for instance auxiliary contacts.

Optional electrical accessories appears in the **Component menu** under **Symbols for component** when working on **Diagram** pages (electrical symbols) and when working on **Ground plan/Mechanical** pages (mechanical symbols).

When you have selected optional electrical accessories for a component, the accessories are always included in electrical and mechanical lists.

When you let the mouse cursor rest over the optional electrical accessory in **Symbols for component**, the Type and Description are displayed in a pop-up window.
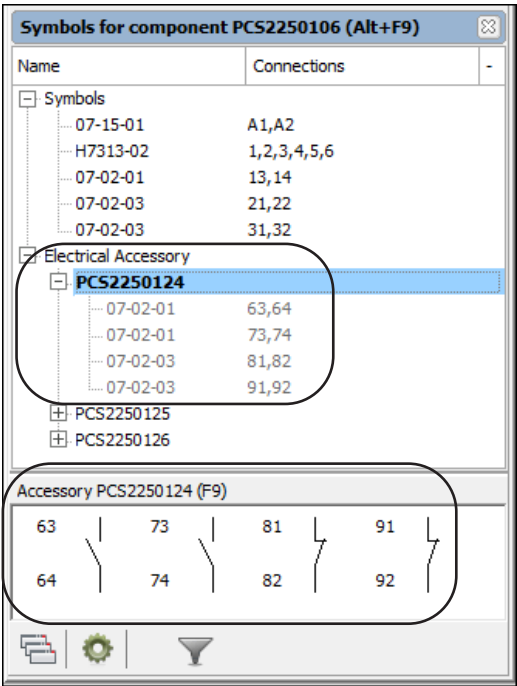
— Please also see "Component accessories" on page 688, in particular "Optional accessories - Electrical" on page 689.

Add electrical accessories

To add electrical accessories to a component directly from the active project, right-click a symbol for the component and choose **Electrical accessory**. You then enter the database, where you select the accessory.

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 775.

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 918.

SPECIFYING SYMBOL SUB NAMES FROM THE DATABASE

Automation can handle sub names on symbols intelligently, as described in "Sub names on symbols" on page 188. 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:
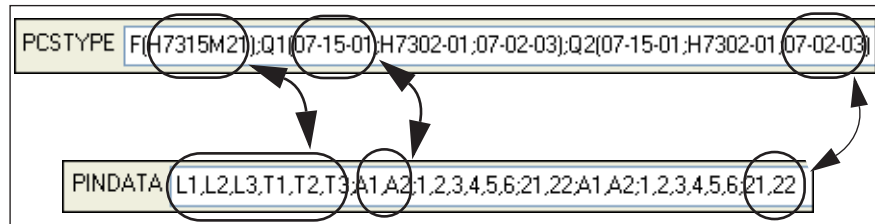
As you can see, the fact that sub names are being applied for the component, makes no difference for how to fill out the **PINDATA** field.

— See an example on this in the pickmenu:
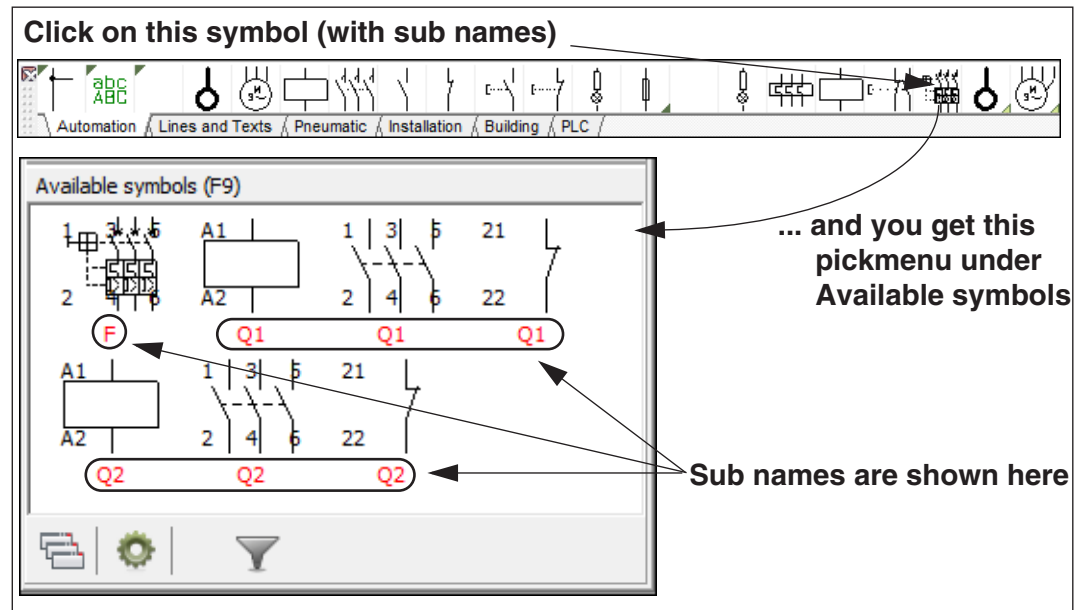
CREATING PLC COMPONENTS IN THE DATABASE

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 803.

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 806.

PCSCHEMATIC Automation

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 652.

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 663, and e.g. "Alternative symbols for PLCs" on page 665.

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 669.
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 670:

ID211.PIN

Filer Rediger Formater Vis Hjælp

{A++(0)[++2(00.00)],COM,}#8, {B++(0)[++2(00.01)],COM,}#8;
{A++(0)[++2(00.00)],COM;B++(0)[++2(00.01)],COM;}#8;

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 672.
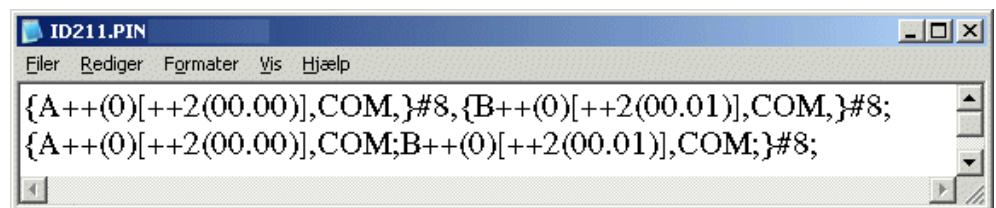

SIMPLE NOTATION FOR ADDRESSES IN PLCS
YOU DO NOT NEED TO APPLY THE COMPACT NOTATION DISPLAYED IN THE FIGURE ABOVE. YOU CAN JUST SPECIFY EACH INDIVIDUAL ADDRESS IN SQUARE BRACKETS AFTER THE CONNECTION POINTS. — SEE "ADDRESSING PLC SYMBOLS VIA THE PINDATA FIELD" ON PAGE 672.

---

7) Add the remaining relevant data for the component — such as the EANnumber — 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 659, "Information about the component in the database" on page 661 and "Electrical and Mechanical symbols for components" on page 663.

DATABASE SETTINGS

This chapter deals with how the information from the attached database is represented in **Automation**.

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 **PCSCHEMATIC Database** program, or any other open database program you use in conjunction with **Automation** — see "Choosing a database" on page 699.

If you are using the **PCSCHEMATIC Database** program, you can start it by choosing **Tools => Database** in **Automation**. — Please refer to the **PCSCHEMATIC 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 641.

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.

PC | SCHEMATIC

The **Field setup** tab looks like this:
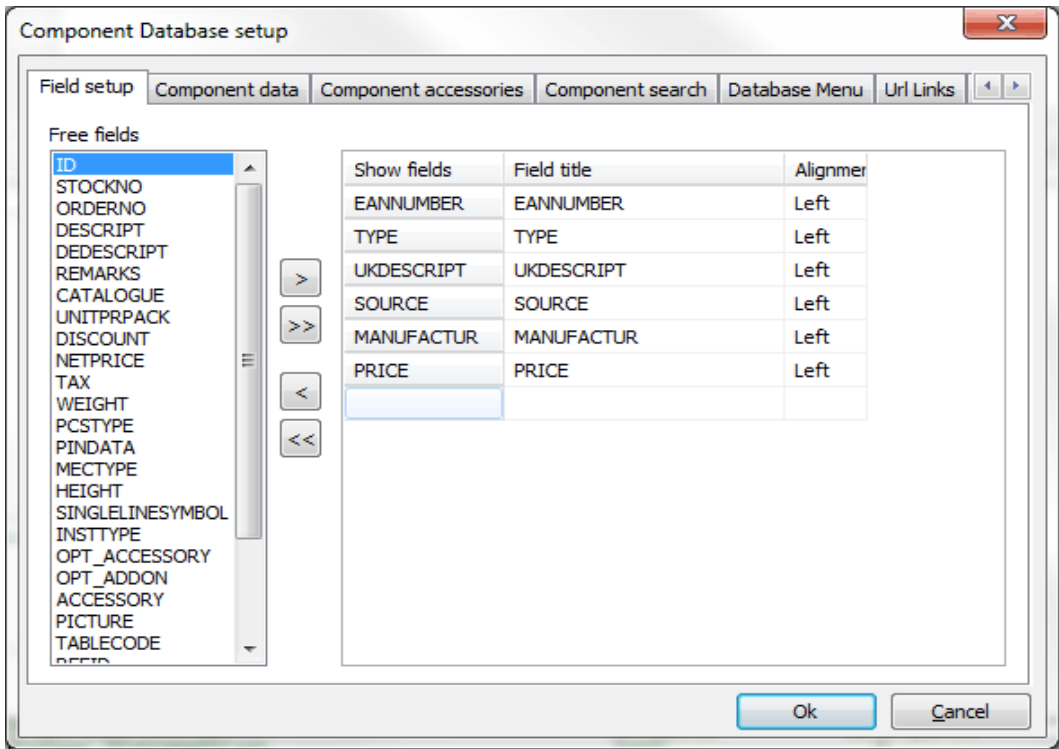
Which database fields are displayed in Automation

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.

In order to transfer article data from the database to the symbols or lines in projects, it is important that you specify which *fields* in the database are transferred as **Article**, **Type** and **Function** article data to the symbols and lines in the projects.

This is specified in the fields **Article**, **Type** and **Function** on the **Component data** tab in the **Component Database setup** 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 **Item no./Type/Desc.**.

— See an overview over the used field names in the database, which is installed with the **Automation** software in "Datafields in the component database" on page 969.

Alternative article number

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.

Operating device, general symbol
Kontaktor 3POL 2NO/NC type1
PCSCON16
PCS2250116
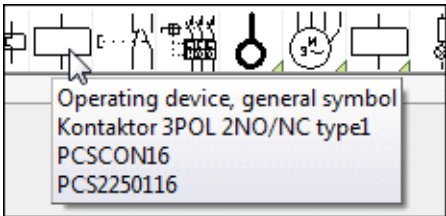
You can for instance choose to map to the database field **UKDESCRIPT** (default setting) to have the English texts for the symbol displayed, or map to the database field **DEDESCRIPT** for German texts — or use the field **DESCRIPT** for another language.
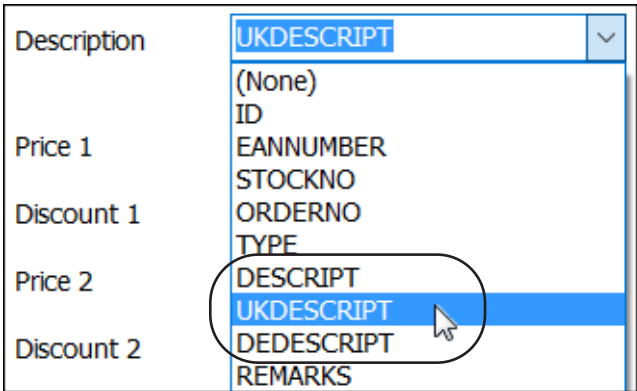
You can also create your own database fields in other languages, and map to these.

Descriptions displayed in lists

This description can also be included in lists, when you insert the datafield **Parts/Components List => UKDESCRIPT**. — Read more about this in "Creating Lists" on page 831.

Description displayed for symbols

The description can also be inserted as a datafield in the definition of a symbol, so that you can have the description text for the symbol displayed in different lan-

guages, depending on which database field you choose to map to. — See "Inserting symbol datafields from database in symbol definition" on page 791.

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 949.

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 663.

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 generation of symbols" on page 229.

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.

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 679.

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 669.

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 in the database" on page 674 and "Placing mechanical symbols via the database" on page 708.

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 generation of symbols" on page 229.

Other symbols

In the *Other symbols* field you can specify whether any other fields in the database contains other symbol categories, that you need in your projects — a part from the electrical and mechanical symbols.
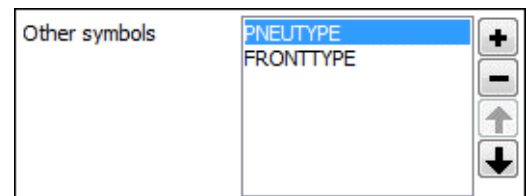
When you have added these datafields in the *Other symbols* field, you subsequently have the option of placing these symbols on project pages.

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 symbols of the desired symbol category 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 select this datafield for the active project page in the **Page database field for symbols** — see "The page database field for symbols" on page 266.

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 **Component menu** will only show symbols of the type PNEUTYPE, when you work on this project page.

— You can read more about the **Component menu** in "The Component menu" on page 263.

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.

COMPONENT ACCESSORIES

In **Settings => Database => Database setup => Component accessories** you specify how the program understands ("maps") the fields in the database, which contains accessories.
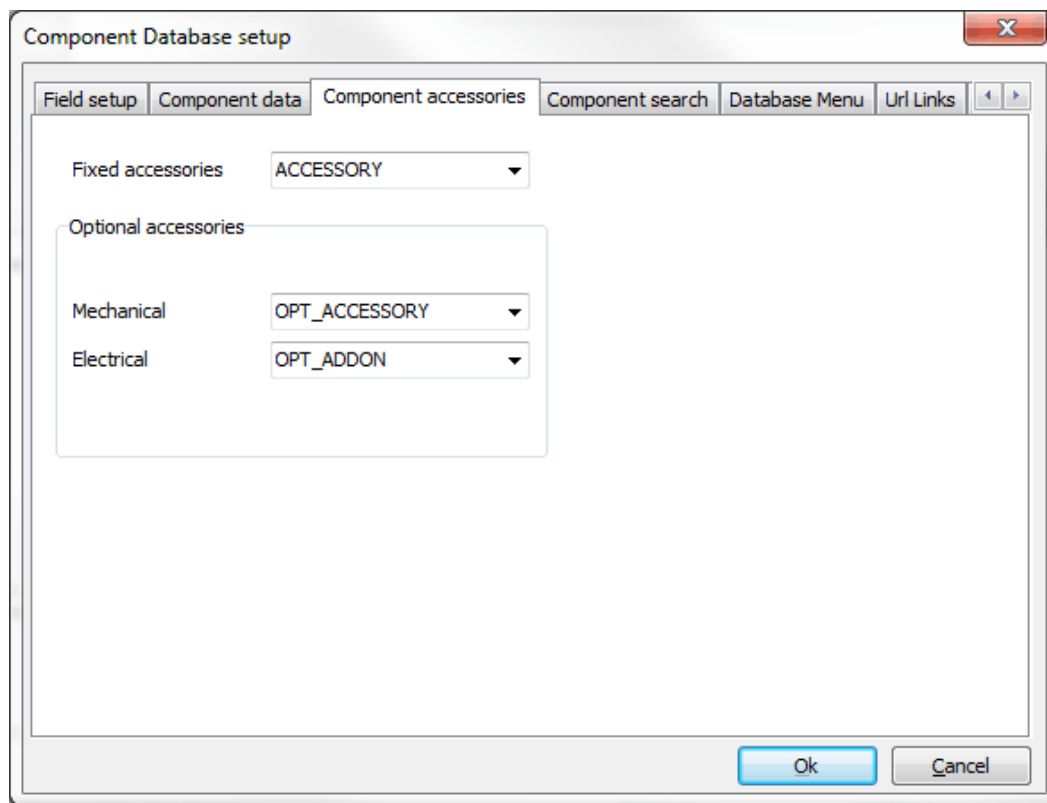
There are three types of accessories:

- Fixed accessories
- Optional mechanical accessories
- Optional electrical accessories

Read more about these types of accessories in "Accessories in the database" on page 676 and "Accessory types" on page 676.

Fixed accessories

A component can have specific additional parts attached to it, which also must be used each time the component is used, but not necessarily need to appear anywhere else than in the parts lists for the project. This could for instance be a base for a relay.

In the *Fixed accessories* field you specify which field in the database contains this accessory. — See "Datafield: ACCESSORY" on page 972.

If a mechanical symbol is defined for the accessory, you can place this mechanical symbol on mechanical pages in the project. However, whether you place the mechanical symbol for the accessory in the project, makes no difference for whether the accessory is included in the project lists. This is exclusively specified in the settings for the individual lists.

In "Specifying from where to collect components" on page 611 and in "Which data to include in the list" on page 611 you can see how to specify whether to include accessories in lists.

— Read more about fixed accessories in "Specifying that a component has fixed accessories" on page 676.

Optional accessories - Mechanical

Under **Optional accessories** in the field *Mechanical* you can specify which field in the database, that contains the article numbers which are presented as optional mechanical accessories in the dialog box **Component data**.

In the PCSCHEMATIC database the field OPT_ACCESSORY can be used for specifying optional accessories. — See "Datafield: OPT__ACCESSORY" on page 971.

— See "Optional accessories - Mechanical" on page 678 and "The Accessory tab" on page 184.

Optional accessories - Electrical

Under **Optional accessories** in the field *Electrical* you can specify which field in the database, that contains the article numbers which are presented as optional electrical accessories in the dialog box **Component data**.

In the PCSCHEMATIC database the field OPT_ADDON can be used for specifying optional electrical accessories. — See "Datafield: OPT_ADDON" on page 972.

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**:
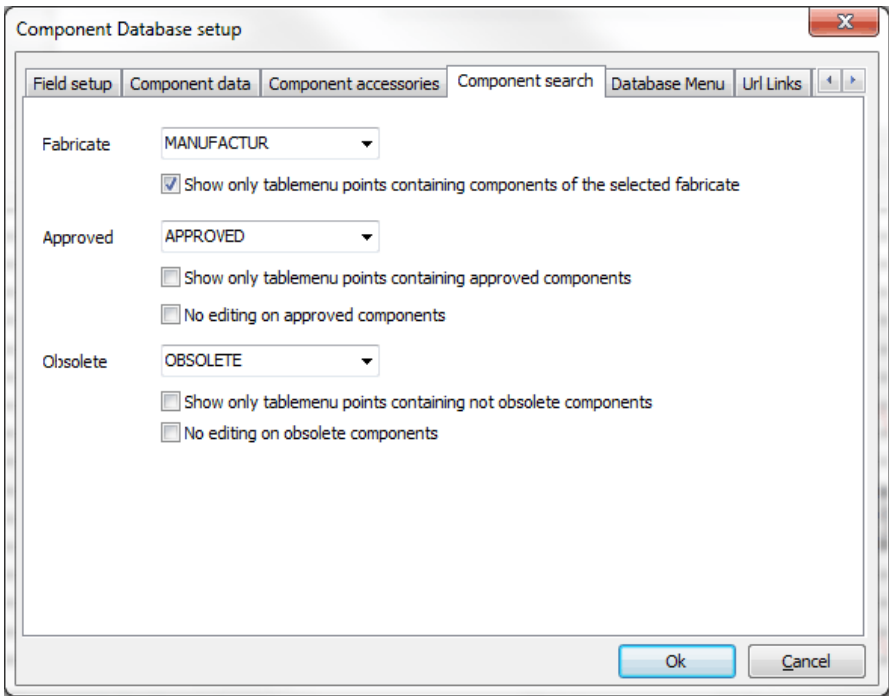
2) In the dialog box **Database setup**: Click on the tab **Component search**, click on the **Down arrow** in the *Fabricate* field, and choose which database field to divide by — for instance *Manufactur*. (Choose *(None)* in order *not* to use this Fabricate menu).

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 646.

Show only table menu items 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 items, 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.

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 692.

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.

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.

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 **PCSCHEMATIC 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.

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**:

Click on the fields and drag them over to the Visible fields
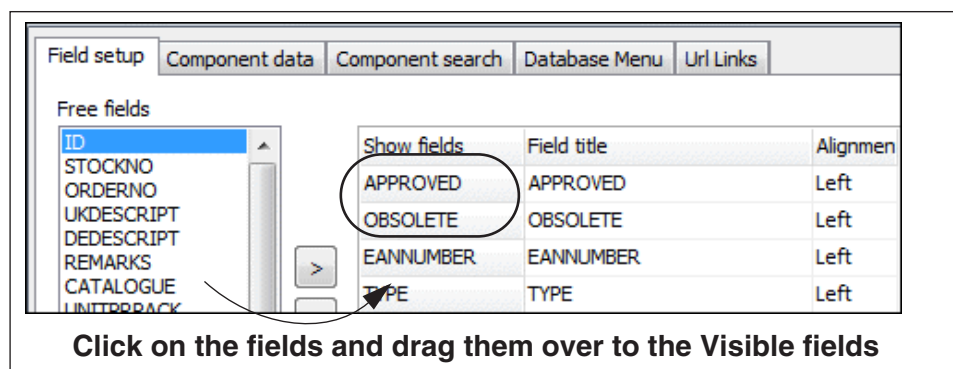
Display only Approved components and hide Obsolete components

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**.

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 693.

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.

Name of the menu　　　　**Name of the database**

Component Database setup

| Field setup | Component data | Component accessories | Component search | Database Menu | Url Links |

Link table menu:
  C:\UK\PCSELCAD\DATABASE\PCSDB_UK.MDB;DBmenu
To table:
  C:\UK\PCSELCAD\DATABASE\PCSDB_UK.MDB;Components
☑ Show only active menupoints

　　　　　　　　　　C:\...\PCSDB_UK.MDB;DBmenu　　　:\...\PCSDB_UK.MDB;Component

Menulink	MENULINK	TABLECODE
Description	UKTEXT	
Menuindex	MENUINDEX	No Menu
Menutable	DBmenu	Open Menu

Ok　　Cancel

**Under normal circumstances
the contents of these three fields
are not to be changed**　　　　**Datafield in the database
containing a menu link**

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 **PCSCHEMATIC Database**, the fields *Table link*, *Description*, *Menu index* and *Menu table* will then be filled out correctly.

If you select *Show only active menupoints*, menu items are only displayed when they contain components.

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**.
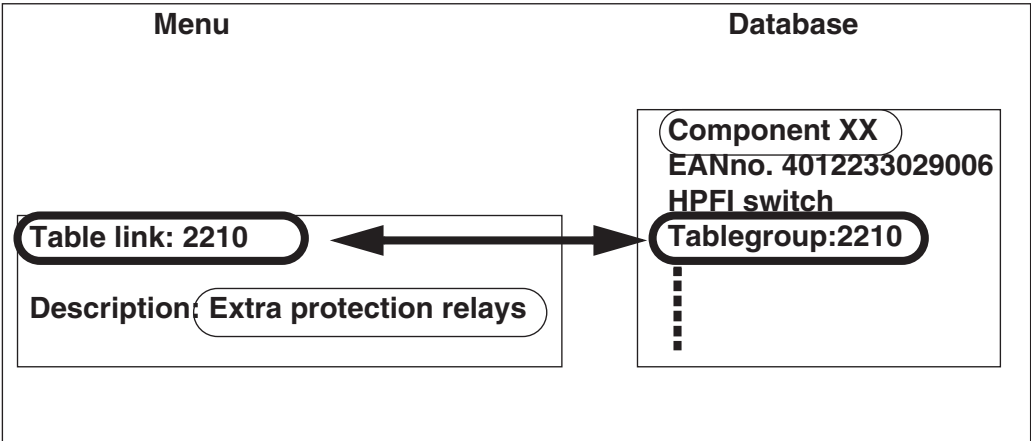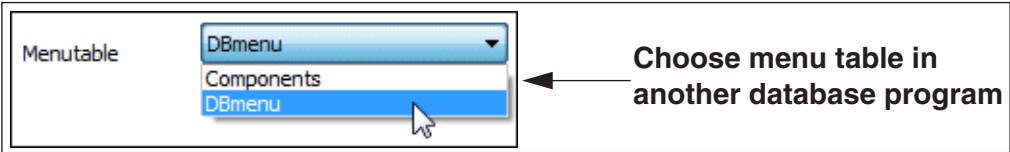

On the figure above, you can therefore see, that **Component XX** belongs to the group **Extra protection relays**.

For a more detailed explanation, please refer to the **PCSCHEMATIC Database** manual.

CHOOSING MENUS WHEN USING OTHER DATABASE PROGRAMS

If you are using another database program than **PCSCHEMATIC 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, or when you want to create a zip-file containing all data sheets attached to the project.

(The *idea* behind using an *alias* is explained in further detail in "The Alias concept" on page 924 — please note that the *syntax is different* for these two types of aliases).
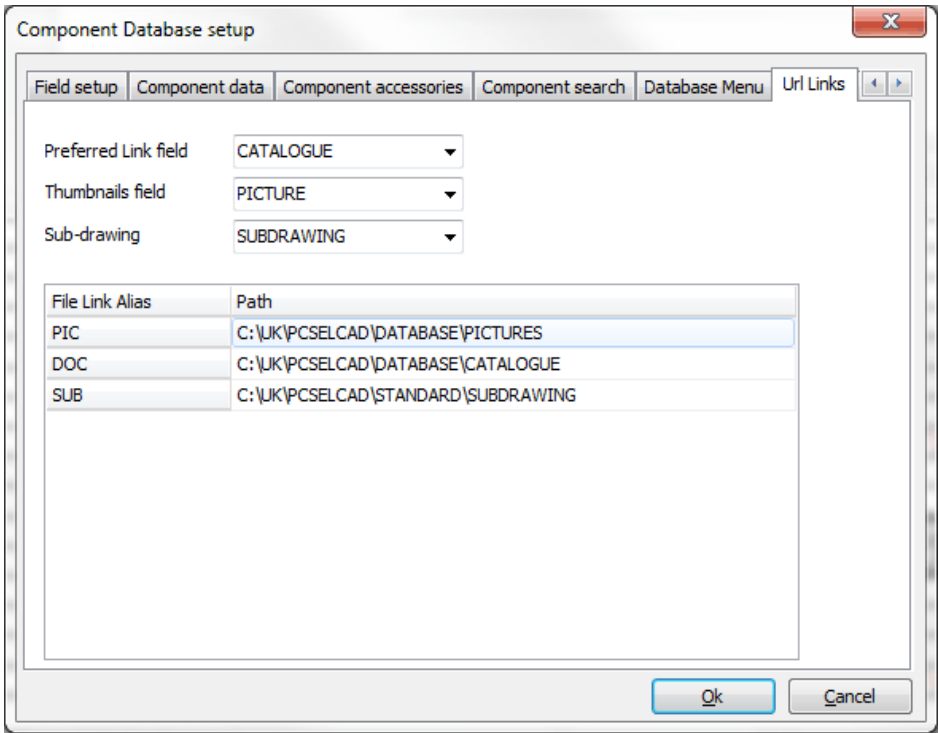
To edit, create or delete a **File Link Alias**, right-click in the dialog box, and choose either **Edit**, **New** or **Remove**.

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

CATALOGUE %DOC%/pushbutton.pdf

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.

Creating 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 650.

Creating a zip-file containing all attached data sheets

To create a zip-file containing all of the data sheets that are linked to from all components in the project, choose **Tools => Create zip package of linked files**.

The zip file is saved in the same folder as the project, with **_documents.zip** added to the file name.

This means that if the project is named **Project1.pro**, the file then gets the name **Project1_documents.zip**.

When the zip file is generated, you get a list over all components with no attached data sheets, and get the option of saving and/or printing the list.

Please note that the data sheets are fetched via the database field, that are specified under *Preferred Link field* — see "Creating a Preferred Link field" on page 697.

When links are directed to data sheets located on web addresses, these data sheets are not included in the zip file.

COMPONENT WIZARD

On the tab **Settings => Database => Database setup => Component Wizard** you can specify which database fields you want to work with in the **Component Wizard**, which helps you to create components in the database easily.

The **Component Wizard** is activated by clicking the **Component Wizard** button in the program toolbar — or by choosing **Functions => Component database => Component Wizard** — and is described in an independent mini-manual, which you can download at www.pcschematic.com/manuals.

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:

- https://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 **PCSCHEMATIC Database** PRO-GRAM.

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 **PCSCHEMATIC 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 PAGES AND SYMBOLS

In this section you can see how to work with mechanical symbols in PCSCHEMATIC Automation.

You can read about:

GROUND PLAN PAGES

PAGES OF THE TYPE *GROUND PLAN/MECHANICAL* ARE APPLIED FOR MECHANICAL SYMBOLS, AND ARE ALSO USED FOR GROUND PLAN DRAWINGS. READ MORE ABOUT THIS IN "ELECTRICAL INSTALLATION DRAWING" ON PAGE 533.

PC | SCHEMATIC

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 260.

In the following chapters you can read about:

- In "General arrangement drawings" on page 705 you can read more about how to create general arrangement drawings on mechanical pages in **Automation**.

- In "Placing mechanical symbols via the database" on page 708 you can see how **Automation** can automatically fetch the *mechanical* symbols, that are attached 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.

- In "From Mechanical to Electrical drawing" on page 720 you can see how to 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.

- In "Measurement objects" on page 722 you can see how to apply measurement objects on mechanical pages.

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 663 and "Mechanical symbols in the database" on page 674.

See how to design your own symbols in "Creating Symbols" på side 775.

Creating mechanical symbols automatically

See how to create a mechanical symbol on the fly in "Automatic generation of symbols" on page 226 and in "Electrical or Mechanical symbol" on page 228.

Displaying mechanical symbols on screen

When working with mechanical symbols, these are drawn with thin lines on the screen, which should increase the readability. If this is not the case on your monitor, you can disable the setting under **Settings => Special settings**.

— See "Settings for the page type Ground plan / Mechanical" on page 941.

3D/isometrical symbols

It is also possible to create 3D/isometric symbols, where all connection points are placed on the correct position with attached data.

These symbols can only be created using the **Component Wizard**, which is described in an independent manual, which is available under **Help => Component Wizard manual**.

These symbols can be used together with the built-in **Panelrouter**, which for instance can generate wiring lists, from which you can have wire bundles / harnesses produced.

The **Panelrouter** is described in an independent manual.

You can see an example of a page containing this category of symbols in the demo project **Pcsdemo2.pro**, where you can open the first layout page, choose **Settings => Page setup**, set the page type to *ISOmetric* and click **OK**:
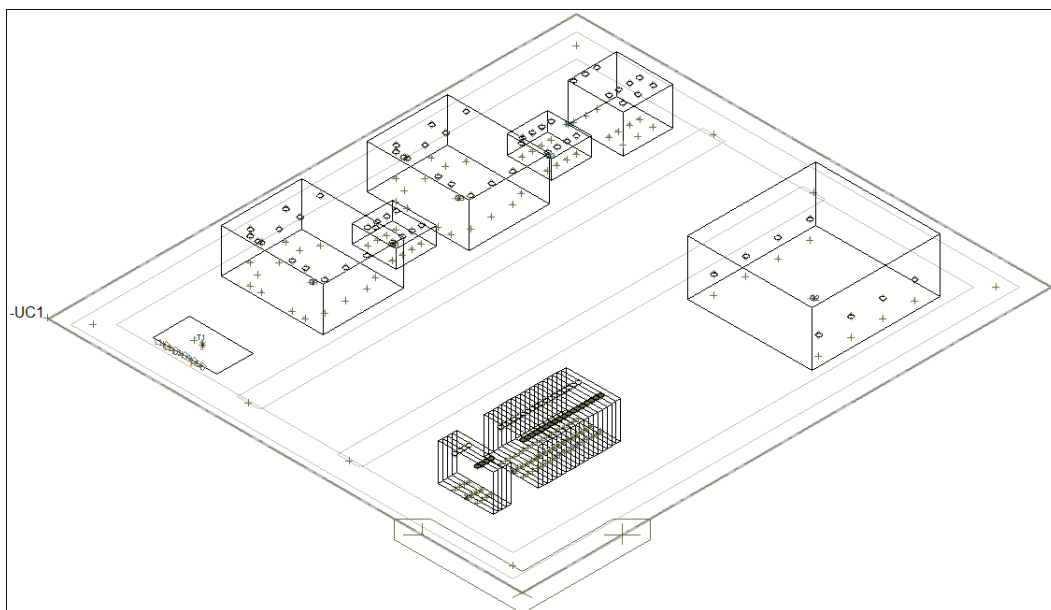
GENERAL ARRANGEMENT DRAWINGS

THIS CHAPTER DESCRIBES
YOU CAN WORK WITH ARRANGEMENT DRAWINGS AT MANY LEVELS IN **Automation** — RIGHT FROM CREATING A QUICK OVERVIEW TO GENERATING PRECISE OUTPUT FOR WIRE ASSEMBLY MACHINES. THIS CHAPTER GIVES ADVICE ON HOW TO ESTABLISH AN EFFECTIVE WORK FLOW FOR GENERATING ARRANGEMENT DRAWINGS ON MECHANICAL PAGES.

Basic information on general arrangement drawings

General arrangement drawings are created on pages of the type *Ground plan/ Mechanical*, and can for instance be used for creating a quick overview over whether there is room enough for placing the necessary components in a specific panel. — See how to create these pages and place symbols on them in"Placing mechanical symbols via the database" on page 708.

All objects that you place on these pages are dimensionally stable. You can put measures on mechanical pages using the measurement command, as described in "Measurement objects" on page 722.

You can also place conduits, and include the conduits in parts lists, if you have specified article data for the conduits.

GET MORE INFORMATION IN THE PANELROUTER MANUAL

IN THE MANUAL FOR THE **PANELROUTER** YOU CAN SEE HOW TO CREATE A CLEAR AND UNAMBIGUOUS DOCUMENTATION, THAT SPECIFIES EXACTLY HOW TO MOUNT THE PANELS — SO THAT YOU ARE SURE THAT THERE ARE ENOUGH ROOM FOR THE WIRES IN THE CONDUITS, THAT INCOMPATIBLE WIRE CLASSES ARE NOT PLACED IN THE SAME CONDUITS E.T.C.

BASED ON THE CREATED DOCUMENTATION YOU CAN SUBSEQUENTLY GENERATE COMPLETE PRODUCTION LISTS CONTAINING AUTOMATICALLY CALCULATED WIRE LENGTHS, WHICH CAN BE SENT DIRECTLY TO SELECTED WIRE PREFABRICATION MACHINES.

In the following you will get some tips on how to create your general arrangement drawings appropriately:

1. Apply layers for different parts of the panel
In order to make it easier to make changes in the different parts of the panel, you can benefit from creating *layers* on the mechanical page, so that the mounting plate is on one layer, DIN rails on another, conduits on a third — and so on.

— Read more about layers in "Layers, Elevation and Scaling" on page 541.

2. Create template pages matching the module measures of the panels

When you are working with commonly used panel types, you can benefit from creating template pages, where the settings in **Page setup** for *snap* and *grid* match the module measures of the panels.

Also select ***Follow page scale***, so that the specified grid will match the applied measures on the page. — See "Setting the scale of the page" on page 547.

— Read more about snap and grid in "Snap" on page 47 and "Grid" on page 48.

— Read more about creating page templates in "Creating Templates" on page 850.

3. Module based symbols in general arrangement drawings

You can also create mechanical symbols, that fits the measures of the modules. If the measure of the modules are for instance 18mm, you can design the mechanical symbols, so that they fit module measures of 18mm — namely 18mm, 36mm, 54mm e.t.c.

You can also create symbols for DIN rails, where the distance between the rails is fixed and match the module measures of the panels. Here you can for instance create a symbol with states, where each state contains one, two or three rows of rails.

In order to make this work, it is crucial to place the reference points for the symbols consequently in the same corner of the modules.

The small panel to the left in the figure below is pieced together by symbols for socket, door, DIN rails, sides — left and right — and top.

It is important that the sizes of all the mechanical symbols fits the module measures of the page, which again fits the module measures of the applied panel type. It is equally important that the corners are positioned identically — so that the reference points are placed on identical positions, which makes the individual parts fit like legos.

Panel put together by modules   Socket   Door   DIN rails   Sides   Top
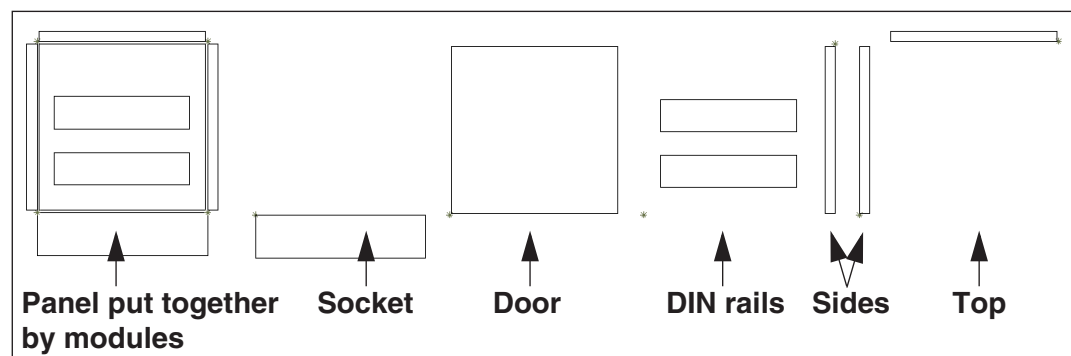
— See how to create symbols with states in "Creating symbols with states" on page 791.

4. Easy positioning in DIN rails and in specified cut-outs

In order to make the first symbol in a DIN rail "snap" to the rail, you need to create the rail as a rectangle drawn using the color **NP**.

— Read more in "Quick placing in a DIN-rail or in a specified cut-out" on page 715.

5. Apply the Mechanical load function

When you place multiple components / mechanical symbols on a mechanical page, they are frequently placed next to each other. When you use the function **Mechanical load**, this will happen automatically.

— See "Placing mechanical symbols via the database" on page 708, especially "Placing mechanical symbols using Mechanical load" on page 710.

When you place the mechanical symbols via the **Component menu**, you have to make sure that the components are placed next to each other. Here it can help to apply the **Magnetical snap** function.

— See "Magnetic snap and Snap in objects for mechanical symbols" on page 715.

6. Place terminals quickly

If you want to be able to fetch an entire terminal row from the **Component menu**, so that you can subsequently place them in correct order, this requires that the terminals are component grouped correctly based on the article number — for instance that double terminals have two layers.

This is most easily handled using the **Show terminal row** function, which is in the right-click menu for terminals on diagram pages. — Read more about this in "Graphical managing of component grouping for terminals" on page 633.

— Read more about component grouping in "Component Grouping" on page 626.

7. Moving areas with a new anchor point

When you move objects using the **Area** command, you can right-click inside the selected area to specify a new anchor point for the moved area, which makes it easier to place it correctly in relation to the other objects on the page.

— Read more in "Reference points when selected areas are moved/copied" on page 281.

8. Importing DWG/DXF files

Creating a mechanical page / general arrangement drawing can also be done by importing a DWG/DXF file.

This could for instance be relevant when you apply one of the various tools, where you start by generating the mounting plate, with the intention of being able to drill holes e.t.c. for the final production.

— Get more information in "DWG and DXF files" on page 741.

9. Exporting to DWG/DXF

When the mounting plate is ready, it can be exported as a DWG/DXF file. Here it is possible to export one layer per drawing. Thus you can place mounting holes on specific layers, which can be exported to the DWG/DXF file, which can then be used for the final production.

— Get more information in "DWG and DXF files" on page 741.

PLACING MECHANICAL SYMBOLS VIA THE DATABASE

When you have 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.

You can also place mechanical symbols individually using the **Component menu**.

— Read more about the relation between *components*, *electrical symbols* and *mechanical symbols* in "The relationship between Symbols and Components" on page 260.

— Read more about how the database works in "Using the database" on page 641.

INSERTING A PAGE FOR A MECHANICAL DRAWING

To generate a mechanical drawing, you first have to insert a page of the type **Ground plan/Mechanical**, 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**.

PLACING MECHANICAL SYMBOLS VIA THE COMPONENT MENU

When you are working on a page of the type *Ground plan/Mechanical*, you can place mechanical symbols on the page, and hereby for instance create general arrangement drawings, which you can read more about in "General arrangement drawings" on page 705.

You can place mechanical symbols on mechanical pages using the **Component menu**. In the following the special options for working with mechanical symbols in the **Component menu** are described. For more general information about the **Component menu**, please refer to "The Component menu" on page 263.

You can also place multiple mechanical symbols in a single operation, which is described in "Placing mechanical symbols using Mechanical load" on page 710.

Placing terminal rows on mechanical pages

When working with terminals in the **Component menu** on a mechanical page, you can place a full terminal row like this:

1) Right-click on one of the terminal components in the upper part of the **Component menu** and choose **Select component**. (Or: Double-click on one of the components).

2) Under **Available symbols**: Right-click on one of the symbols and choose **Place all**.

3) You now get the full terminal row in the cross hairs: Click where you want to place the terminal row on the page.
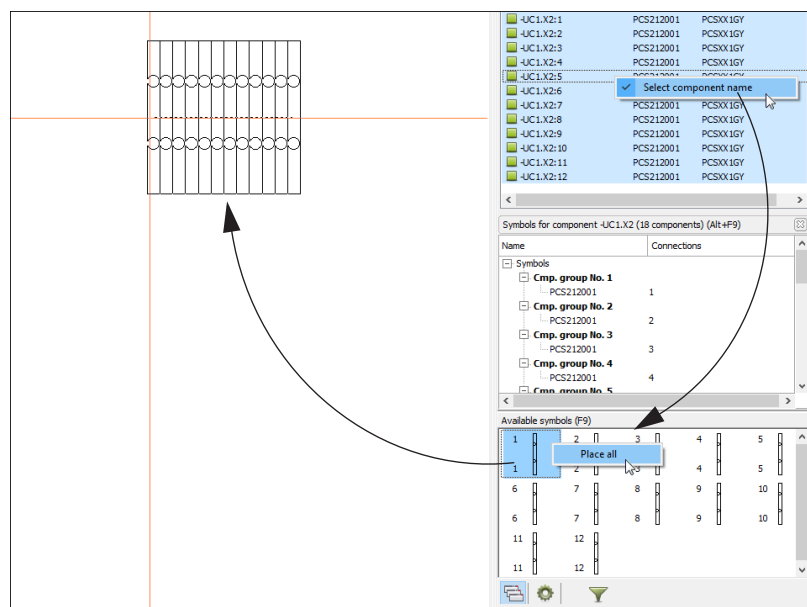
Placing mechanical symbols for electrical accessories

When an electrical accessory for a component is selected, the list in the upper part of the **Component menu** contains an extra line with the same component name — for the mechanical symbol for the electrical accessory.

Here you can also choose **Select component** and then **Place all**.

Placing mechanical accessories

Fixed mechanical accessories are also displayed as mechanical symbols, which can be placed from the **Component menu** on mechanical pages.

Optional mechanical accessories can also be placed from the **Component menu**.

— Read more about fixed mechanical accessories in "Accessories in the database" on page 676, especially "Specifying that a component has fixed accessories" on page 676.

— Read more about optional mechanical accessories in "Optional accessories - Mechanical" on page 678.

PLACING MECHANICAL SYMBOLS USING MECHANICAL LOAD

You can place multiple mechanical symbols in a single operation by **Mechanical load**. This function fetches the mechanical symbols for the components with electrical symbols placed in the project. The mechanical symbols are loaded automatically from the database, in which the electrical symbols, through their EAN13 number/article numbers, are connected to the mechanical symbols.

Click the **Symbols** button, and choose **Functions => Mechanical load**, or right-click on the project page and choose **Mechanical load**.
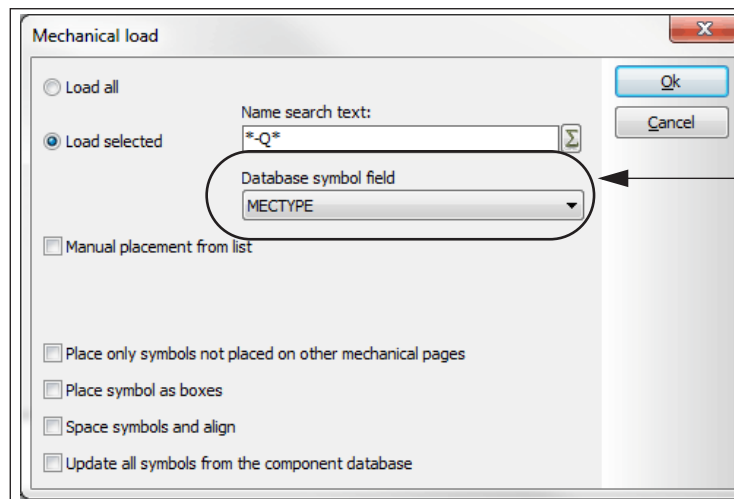
Specify which type of mechanical symbols you wish to place

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 618.

> **SELECT INCLUDE IN MECHANICAL LOAD**
> MECHANICAL SYMBOLS ARE ONLY PLACED FOR COMPONENTS HAVING **_INCLUDE IN MECHANICAL LOAD_** SELECTED IN THE DIALOG BOX **COMPONENT DATA**.
> — SEE "INCLUDE IN MECHANICAL LOAD" ON PAGE 180.

Specify which Type of mechanical symbols to place

Please note: *This is only relevant it your database contains different types of mechanical symbols* — see "Mechanical symbol" on page 687.

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 687.

Place the mechanical symbols on the page

When you click **OK** in the dialog box **Mechanical load**, the mechanical symbols are placed directly on the page. You have two options for how to do this, depending on whether you choose **Space symbols and align**:

1 Space symbols and align

☑ Space symbols and align
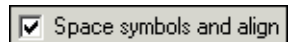
If you select **Space symbols and align**, you first get a pencil in the cross hairs, and then click on the position on the page, where you wish to place the first symbol. Then click where you wish to place the last symbol. Please note that the symbols will either be placed on a vertical or a horizontal 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 Place the symbols in a pile

☐ Space symbols and align
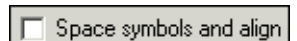
If you do *not* choose **Space symbols and align**, you get a pencil in the cross hairs, and then click somewhere on the page.
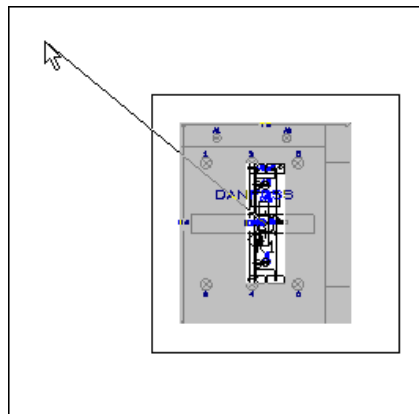
The mechanical symbols are then placed in a pile.

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**.
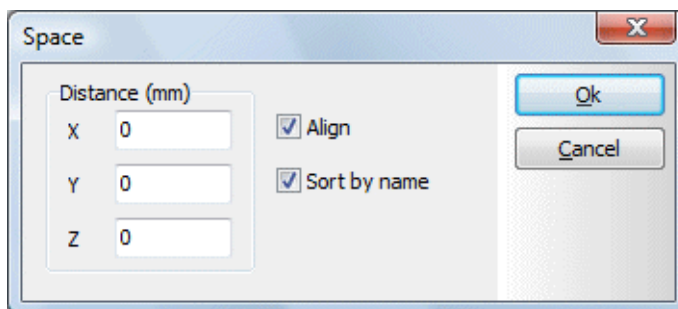
Now click so that you only draw a horizontal line, which is shorter than the width of a single mechanical symbol. Hereby the mechanical symbols will be placed end to end.
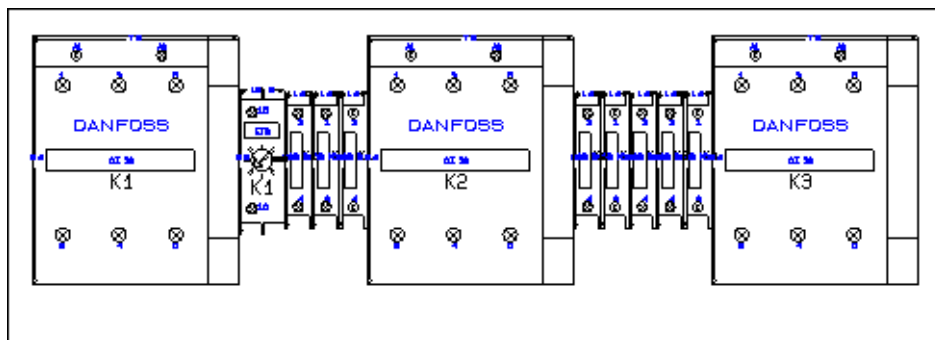
By rearranging the symbols, you can then create your own switchboard.

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 715.

Function texts are also transferred during mechanical load

The function text for the components is also transferred to the mechanical symbols during mechanical load.

Convert to boxes / Convert to mechanical symbols

At the bottom of the **Symbols for component** window there is a button which gives you the option to toggle between *Convert to boxes* and *Convert to mechanical symbols*. This means that you can change how mechanical symbols are represented on the page *after* they are placed on the project page:
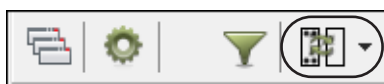
PLACING MECHANICAL SYMBOLS ACC. TO REFERENCE DESIGNATION

Σ

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:
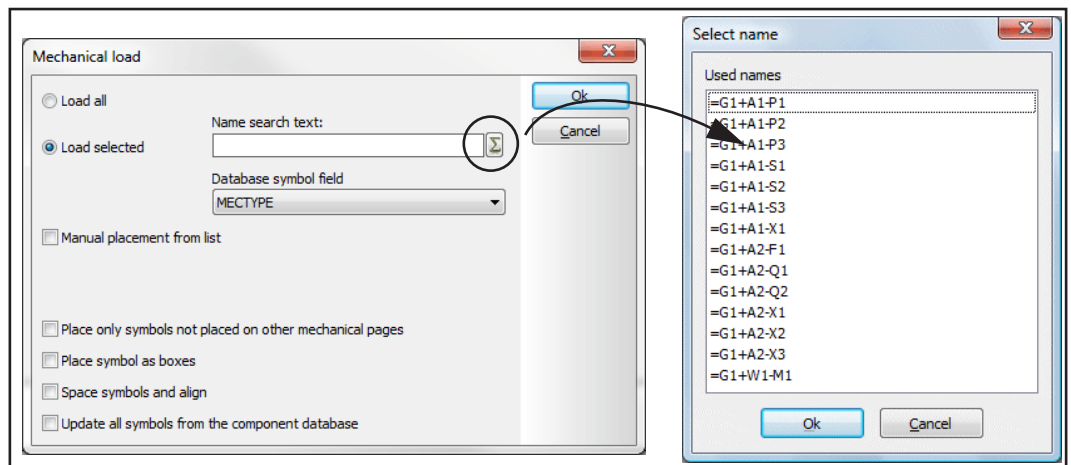
PLACING MECHANICAL SYMBOLS AS BOXES

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**.
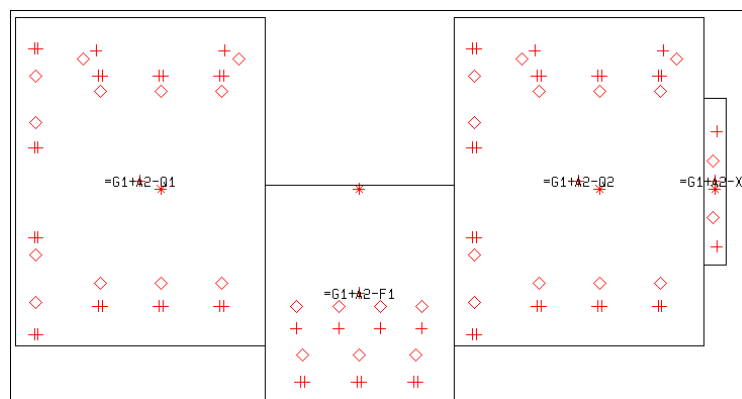
Hereby the mechanical symbols are placed as boxes with connection points. The boxes gets the same measures as the mechanical symbols in the databases. If measures has been specified in the database, these measures are applied. Connection points are only placed if these are specified in the database.

— See "Placing mechanical symbols using Mechanical load" on page 710.

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.

UPDATE ALL SYMBOLS FROM THE DATABASE

This is how to update all mechanical symbols on a project page, so that they match the definitions for the mechanical symbols in the database:

1) Select the project page where the mechanical symbols are placed and choose **Functions => Mechanical load**

2) In the dialog box **Mechanical load**: Select *Update all symbols from the component database* and click **OK**.

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**:
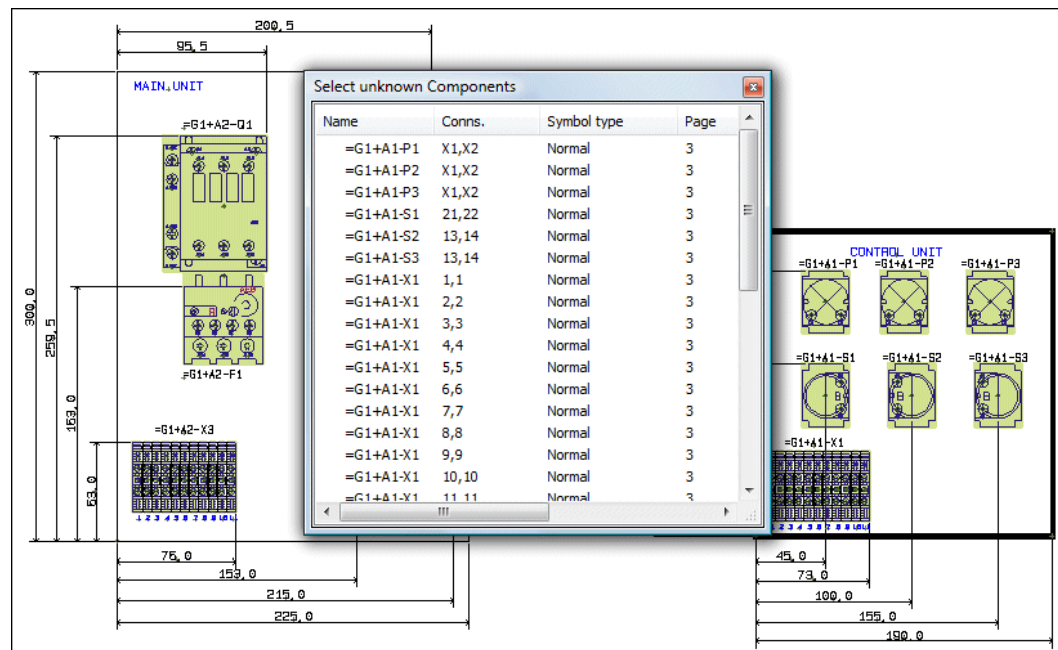
Hereby the window **Select unknown components** is displayed, and all mechanical symbols, for which no electrical symbols exists, are selected on the page.

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 274.

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*.
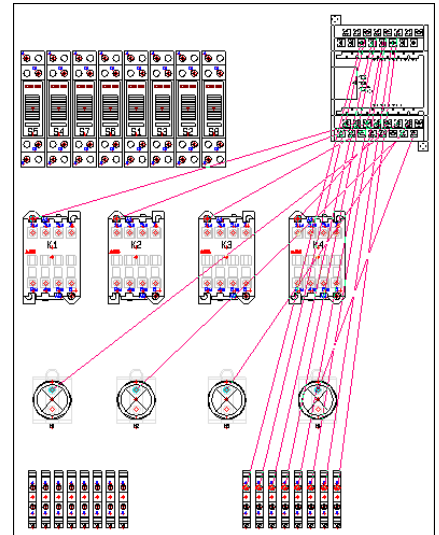
MAGNETIC SNAP AND SNAP IN OBJECTS FOR MECHANICAL SYMBOLS

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 719.

— See more tips for working with general arrangement drawings in "General arrangement drawings" på side 705.
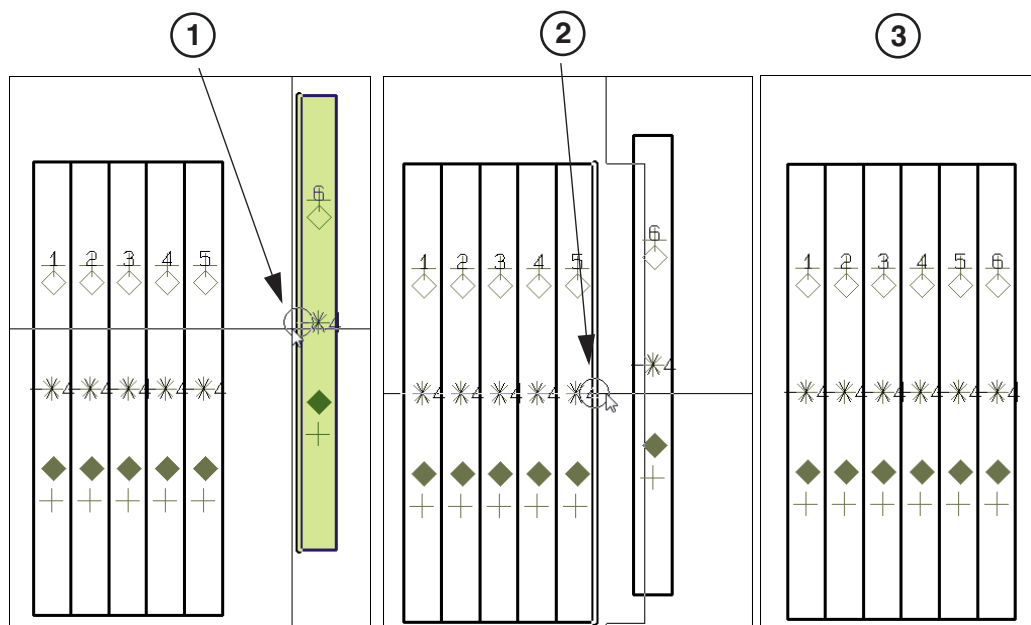
Snap in objects

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 118. 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 831.
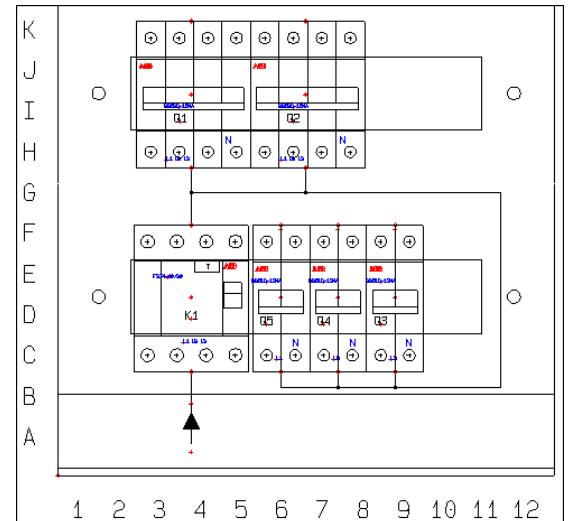
REFERENCES FROM MECHANICAL TO ELECTRICAL SYMBOLS

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**.

General	Ref.design.	Reference	Conns.

Reference
- ☑ With reference
- ◉ With reference
- ○ Master reference

☐ Only reference to symbols with same sub name
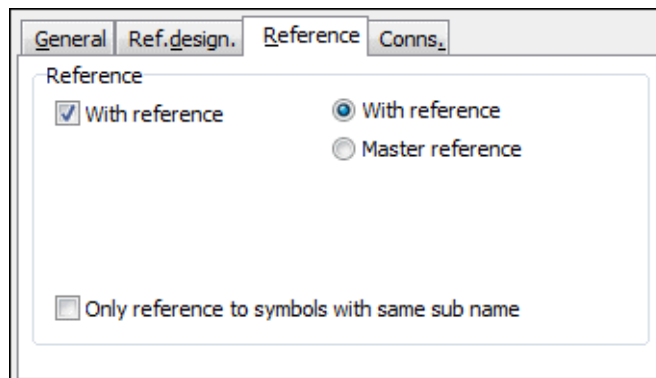
You have now created a reference from the mechanical symbol. When you have selected **Symbols** and double-click on the reference for the mechanical symbol, you jump to the electrical symbol for the component, which is set to **Main reference**. If none of the electrical symbols for the component is 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 722.

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
ALSO SEE "PLACING MECHANICAL SYMBOLS VIA THE COMPONENT MENU" ON PAGE 709.

Do for instance like this:

Creating a new project with a diagram page and a mechanical page

1) Choose **File => New** or click the **New document** button.

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 **10**, 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 contains***, choose the contactor with the type ***PCSCON-AUX07***, and click **OK**.
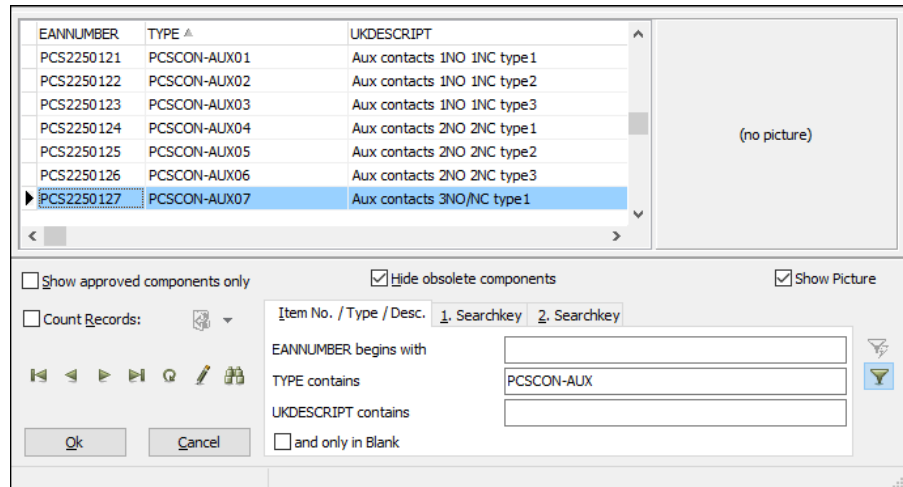
7) You now get the mechanical symbol for the component in the cross hairs: Click somewhere on the mechanical page to place the symbol, click **OK** in the dialog box **Component data** and press <Esc> to remove the symbol from the cross hairs.

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: In the upper part of the **Component menu** you can now select which component to place electrical symbols for.

11) The electrical symbols for the component are now displayed under **Available symbols** at the bottom of the **Component menu**, and you can now 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 **SHOW PARTLY-PLACED** BUTTON IN THE **COMPONENTS** WINDOW, 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.
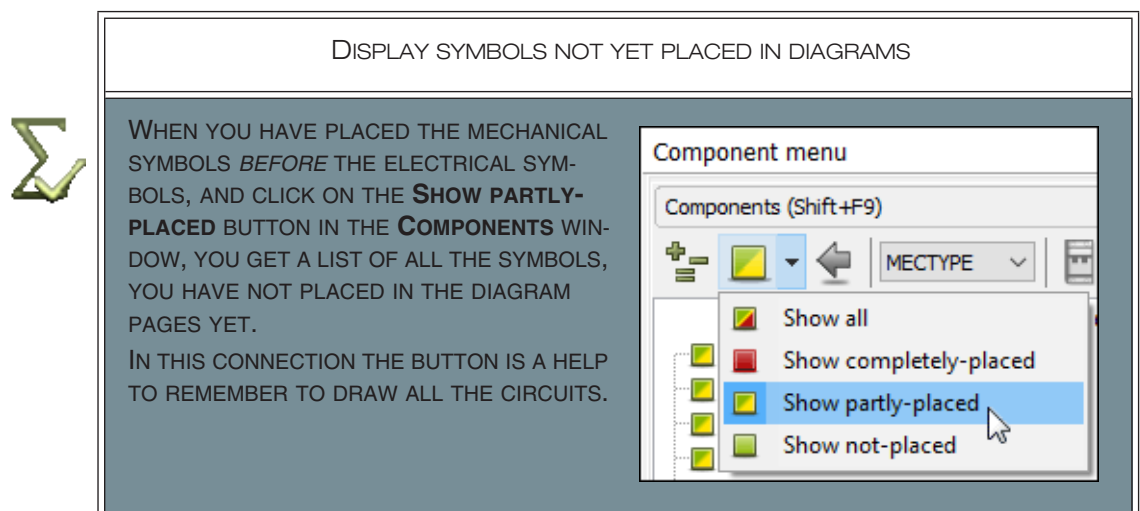
MEASUREMENT OBJECTS

<table>
<tr><td align="center">THIS CHAPTER DESCRIBES</td></tr>
</table>

MEASUREMENT ON GROUND PLAN/MECHANICAL AND DIAGRAM PAGES
YOU CAN INSERT MEASUREMENT OBJECTS ON PAGES WITH PAGE TYPE *GROUND PLAN / MECHANICAL* AND ON PAGES WITH PAGE TYPE *DIAGRAM*. IN THE FOLLOWING YOU CAN SEE HOW TO USE MEASUREMENT OBJECTS ON PAGES WITH PAGE TYPE *GROUND PLAN / MECHANICAL*.

Inserting measurement objects

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

The starting point for the measure-ment 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 refer-ence 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.
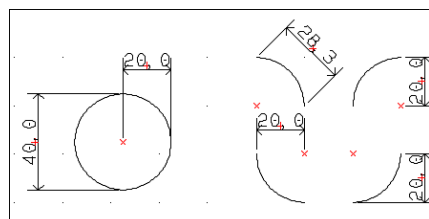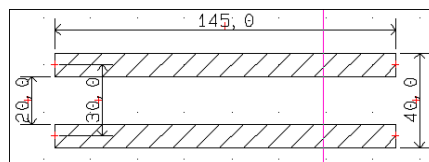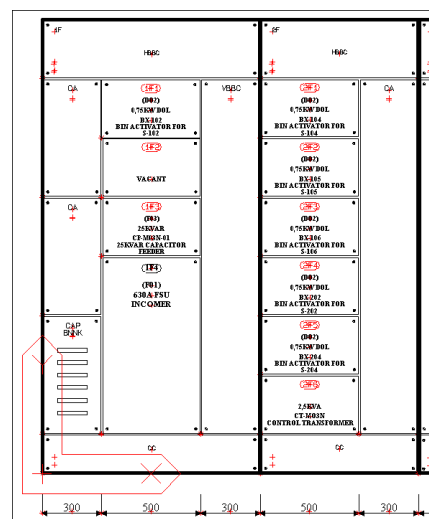
As displayed in the figure below, the program shows exactly which point you are selecting:
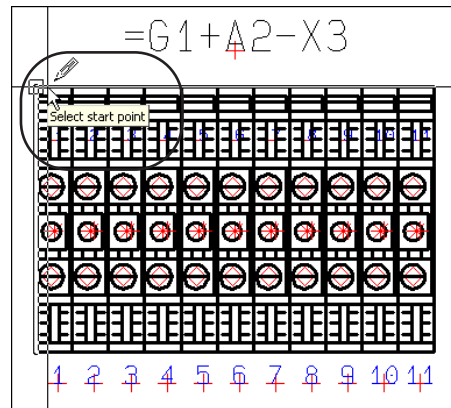
Click to point out the start point for the measurement object.

— 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 724.

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 725.
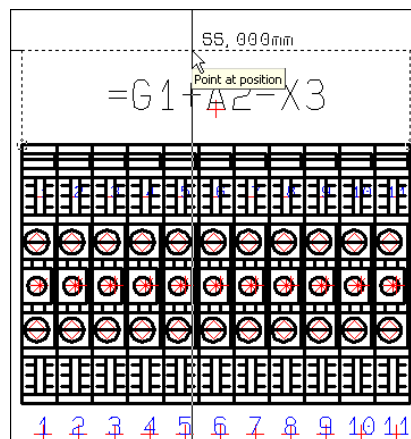
Click to place the measurement object.

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 541.

Specifying the Measurement data

You now enter the dialog box **Measurement**:
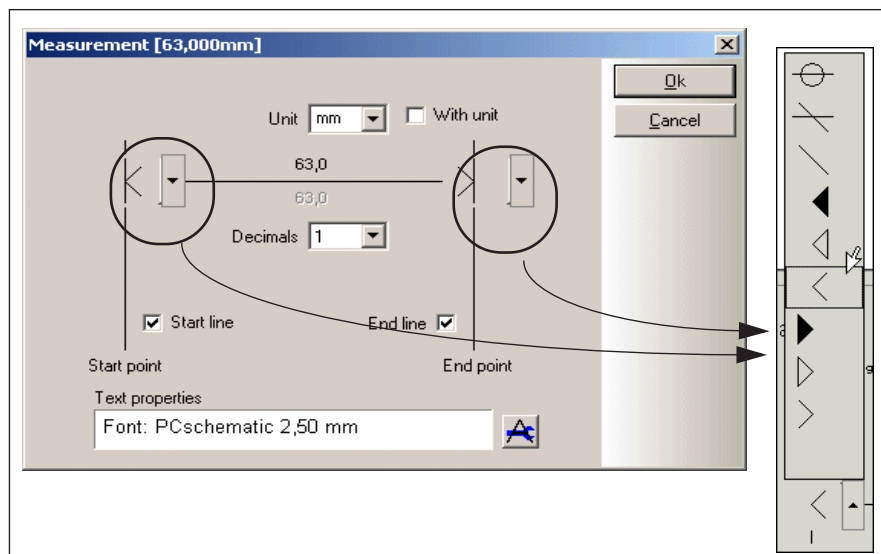
Here you can specify the appearance of the measurement object, whether to attach a unit specification to the measurement, and how many decimals to apply.

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

BASIC SETTINGS: TEXT PROPERTIES FOR MEASUREMENT OBJECTS

THE BASIC SETTINGS FOR TEXTS IN MEASUREMENT OBJECTS ARE SPECIFIED IN **SETTINGS => BASIC SETTINGS**. — SEE "BASIC SETTINGS" ON PAGE 938 AND "REFERENCE FRAME, CLOUD AND MEASUREMENT TEXTS" ON PAGE 938.

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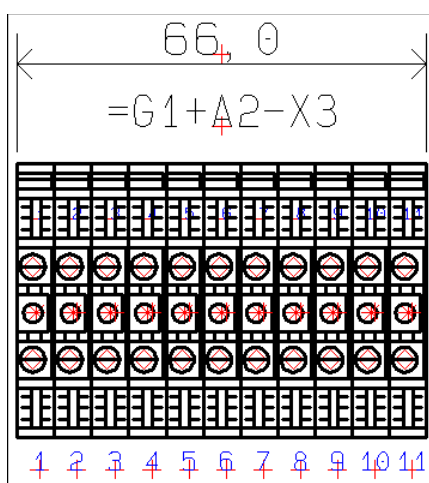
Free measurement

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:
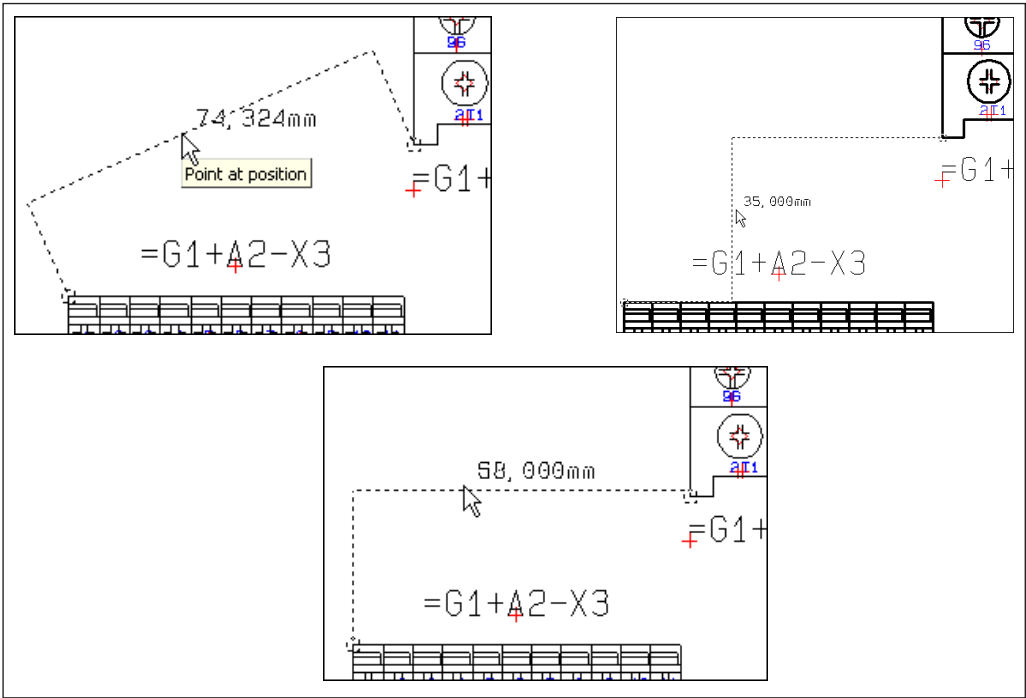
Editing the measurement start and end points

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:
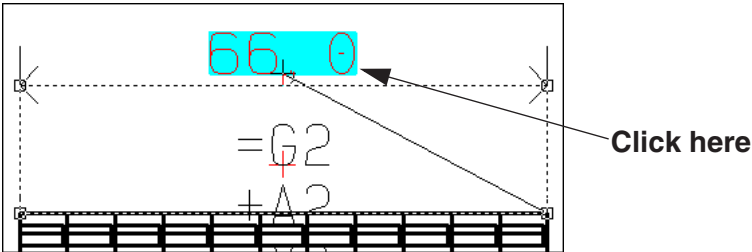
2) Now click on the measurement start/end point, and *drag* it to the desired position:
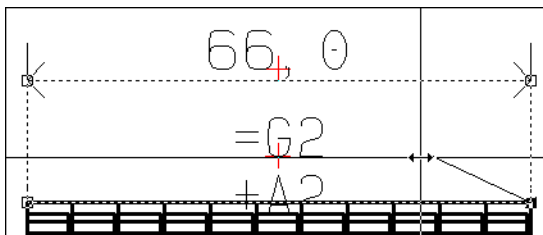
3) Release the mouse button, and the position of the start/end point(s) have now been changed:
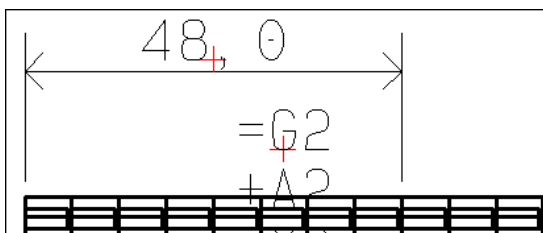
Editing Measurement data

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 724.
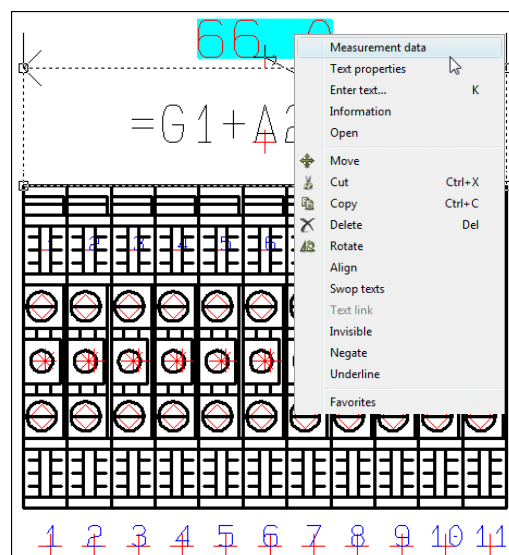
Measurement objects and Layers

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 541, in particular "Printing pages with layers" on page 545.

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:

PC | SCHEMATIC

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.

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 467 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:
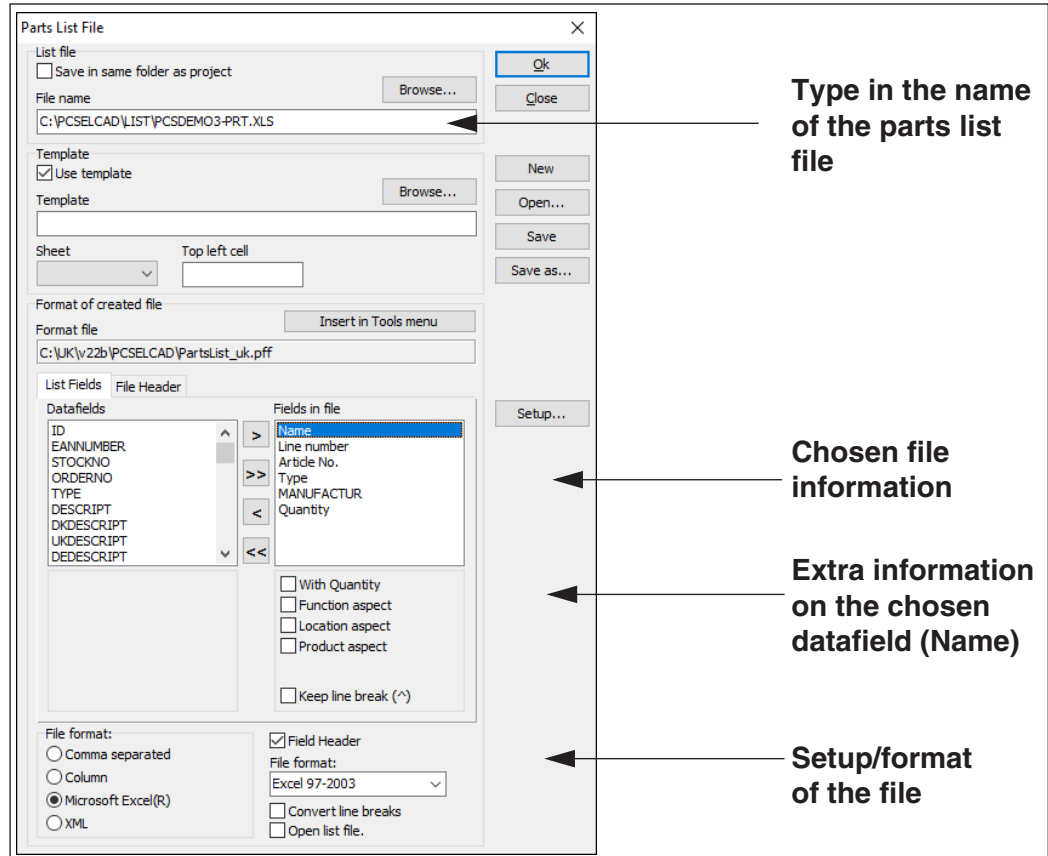
— See "The buttons in the right-hand side of the dialog box" on page 734.

FORMAT FILES

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.

Format files are independent of languages, which means that you can use the same format files for your English projects as for your projects in other languages — as long as the format file exclusively contains the predefined PCSCHEMATIC datafields. When the format file contains datafields, that you have defined yourself,

you must be aware to give these datafields the same name in any language that you use.

MAKE FORMAT FILES AVAILABLE FOR ALL OF THE COMPANY

UNDER **SETTINGS => DIRECTORIES** YOU CAN SPECIFY IN WHICH FOLDER TO SAVE THE FORMAT FILES FOR THE PROJECT. WHEN YOU WORK ON A NET-WORK, THESE FILES CAN THEREFORE BECOME AVAILABLE TO ALL OF THE COMPANY, IF THEY ARE SAVED IN A FOLDER WHICH IS ACCESSIBLE FOR ANY-ONE IN THE NETWORK.

— READ MORE IN "DIRECTORIES" ON PAGE 923.

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.

Placing fixed list formats in the Tools menu

When you frequently use a specific list format, you can make it directly accessible from the **Tools** menu at the upper part of the screen:

1) In the dialog box **Parts list file**: Set the desired settings for the file and click the **Insert in Tools menu** button:
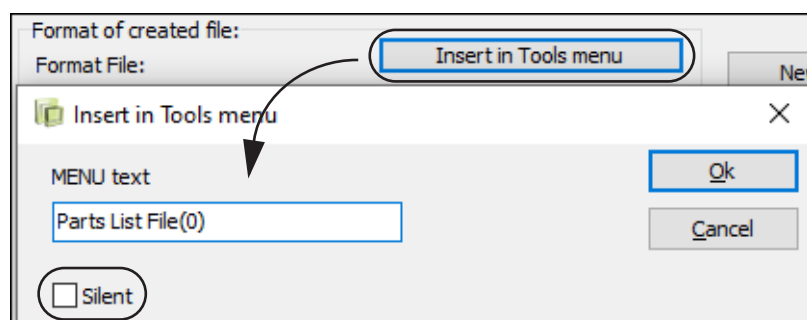
2) In the dialog box **Insert in Tools menu**: Type in an appropriate name and click **OK**.

When you subsequently choose **Settings => Parts List File(0)**, the dialog box **Parts List File** is opened with the specified settings. Click **OK** and the list file is generated.

If you want the list to be generated without entering the dialog box **Parts List File**, just select *Silent* in the lower-left corner of the dialog box.

— Read more about the **Tools** menu in "Tools" on page 926.

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 IN INSTALLED ON THE PC. READ MORE ABOUT THIS IN "EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT" ON PAGE 747.

— See also "Lists in XML format" on page 733.

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.

Click here to create an Excel file **Click here to start Excel automatically** **Select Excel format**
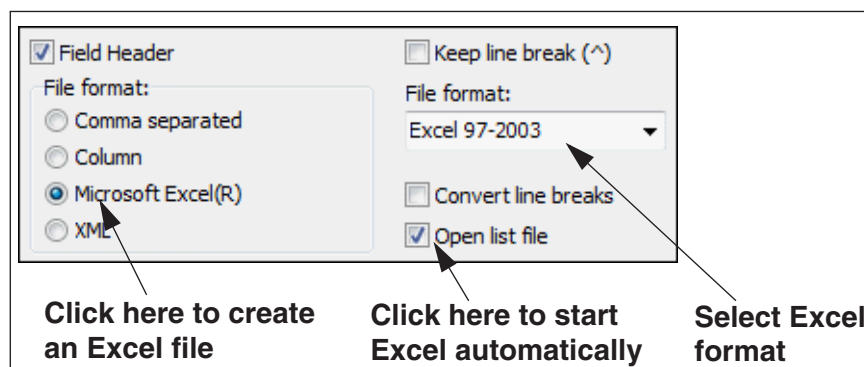
Starting Excel from Automation

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 756.

LIST EXPORT TO EXCEL TEMPLATE FILE

When you export lists to Excel, you can export to an existing Excel file / template. Here you can specify the name of the sheet in the file that you want to export to, as well as in which cell to start inserting data:
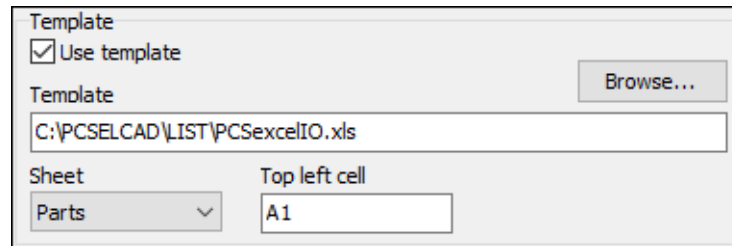
This way you can include different types of lists on each sheet in the same Excel file.

LISTS FOR EXTERNAL PROGRAMS

If for instance you have installed an interface for ClipProject or MPrintPro, you have the option in the **File format** section of choosing **External program**. You can then specify which program you wish to export the file for.
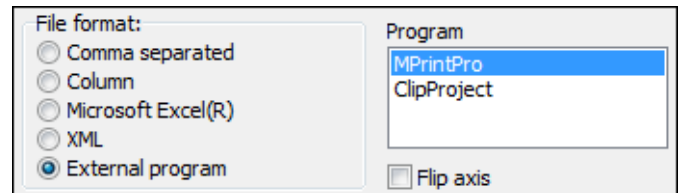
Interfaces for other programs can be downloaded at the PCSCHEMATIC website at www.pcschematic.com/en/support/automation-updates/#drivers.

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.


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 732.

— "Other options when importing XML list files" on page 759.

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:
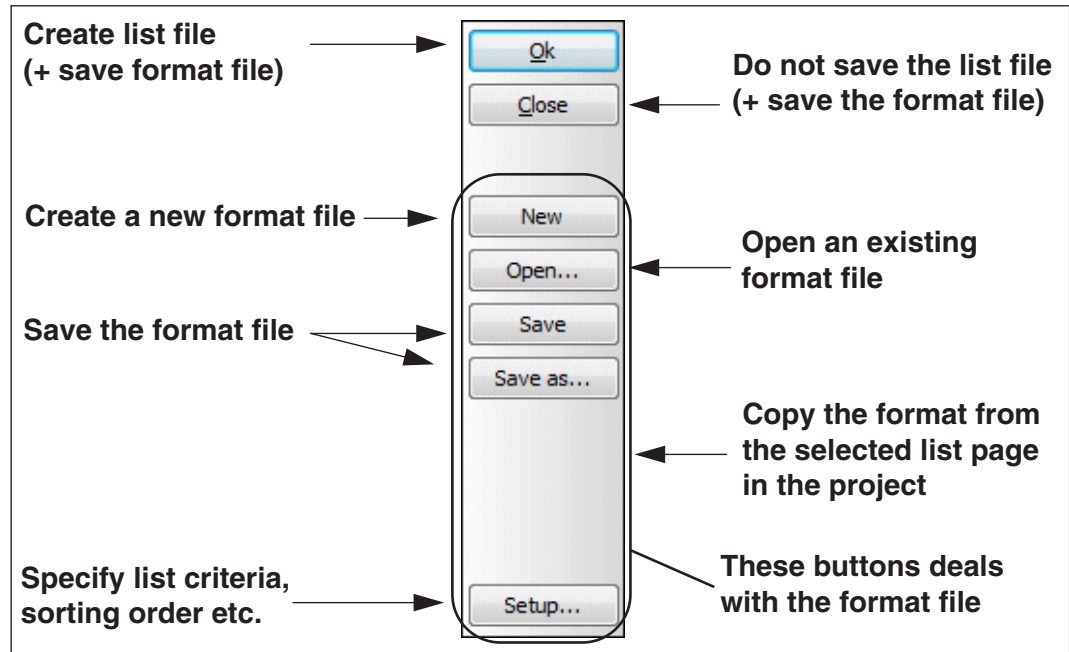
OK and Close

You specify where to save the file in the **File name** field. If you select **Save in same folder as project**, the file is saved in the same folder as the project.

When you then 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 "Format files" on page 730 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 731.

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 609.

SPECIFYING THE FILE NAME

Under *List File* in the upper part of the dialog box — see page 730 — 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 810.

Extra information

For some datafields — such as **Name** in the figure on page 730 — 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 130

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 (^)**.
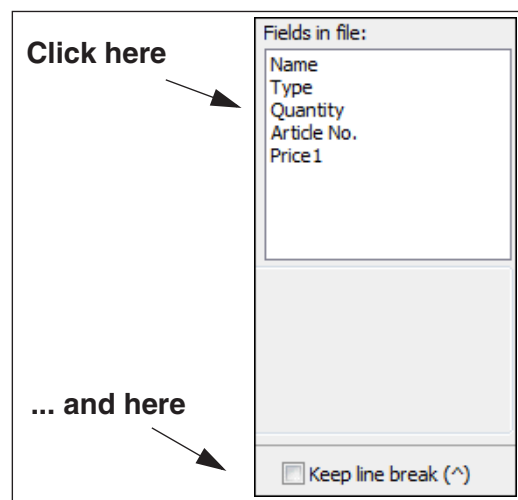
File header

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 735.
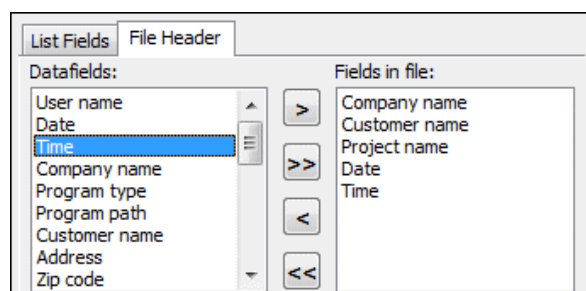
File format

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 data-

field **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.

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 732.

REFERENCE DESIGNATIONS IN LIST FILES

For list files it is possible to insert the field **=+design.** and then choose **Page designation**, **Functional aspect** or **Locational 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.
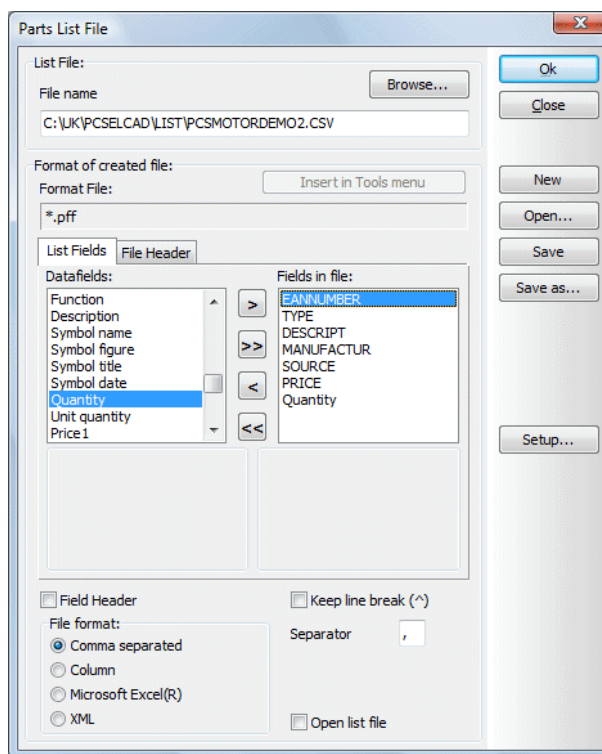
DATAFIELDS IN UPPERCASE ARE FETCHED FROM THE DATABASE
DATAFIELDS WRITTEN IN UPPERCASE ARE FETCHED FROM THE DATABASE. THE OTHER DATAFIELDS ARE FETCHED FROM THE PROJECT DIAGRAMS.

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
CABLES LIST	.CBF
PLC I/O LIST	.PLC
CONNECT POINTS LIST	.COF
CONNECTIONS LIST	.CNF

For further information about the datafields for the different list types, please refer to "Creating Lists" on page 831.

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.
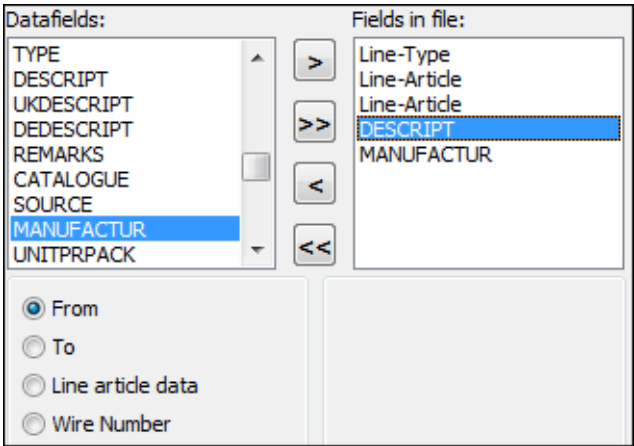
Connections list to file

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, or from the attached **Wire Number**.

This can for instance be used for mounting lists, specifying where to mount wires.

If symbol datafields have been created, you can here choose between **From**, **To** and **Wire Number**.

It is also possible to apply connections list files for specific types of wire numbering, as described in "Creating a connections list for wire numbering" on page 469.

Wire number files

It is also possible to create wire number files.

— See "Wire number Files" on page 467.

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 729.

3) Save the format file — as explained in "Format files" on page 730 and "The buttons in the right-hand side of the dialog box" on page 734.

4) Click on the button **Insert in Tools menu** at the upper part of the dialog box:
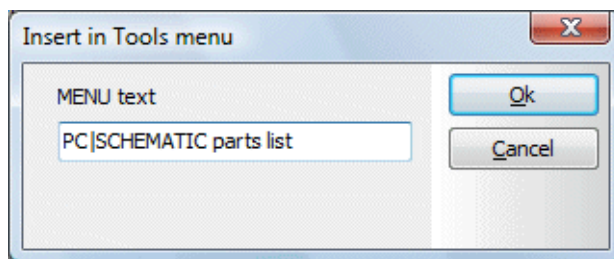
5) In the dialog box **Insert in Tools menu**: Type in the desired menu text, and click **OK**.

A list with the specified settings can now be created directly from the **Tools** menu:
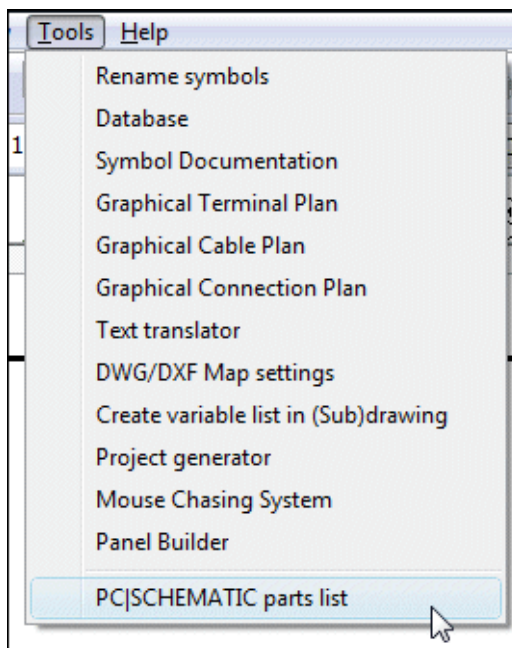
When you select **PCSCHEMATIC parts list**, you enter the dialog box **Parts List File**, where the settings are exactly as you have specified.

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.

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 541.

If you wish to change the parameters for the DXF import or export, please refer to the **PCSCHEMATIC 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 742.

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 743.

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 744.

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 558.

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:
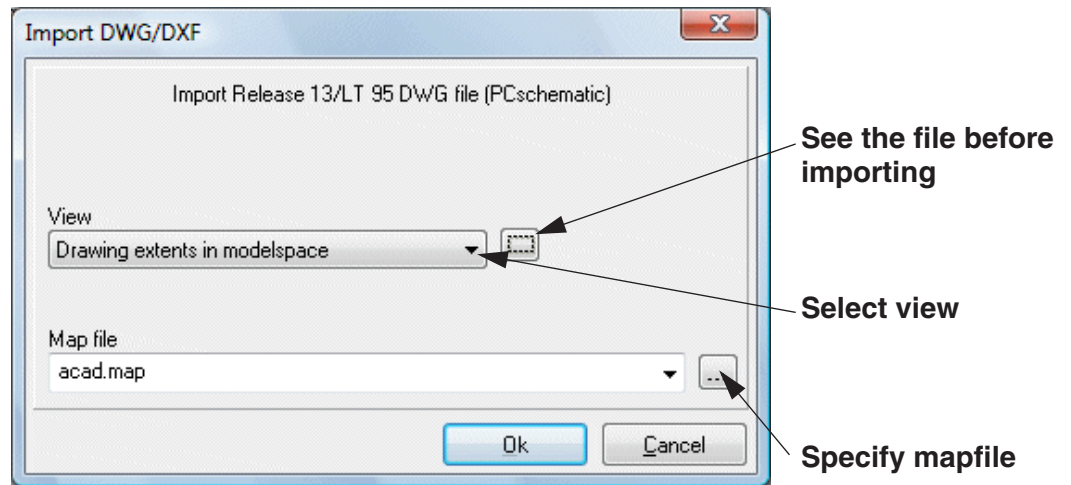
You can also view the file in a separate window before importing, by clicking the button pointed out next to the **View** field in the figure above.

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 785.

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 152, and about how to edit symbols in "Creating Symbols" on page 775.

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:

Specify codepage

Choose which version
to export to

Change layer settings
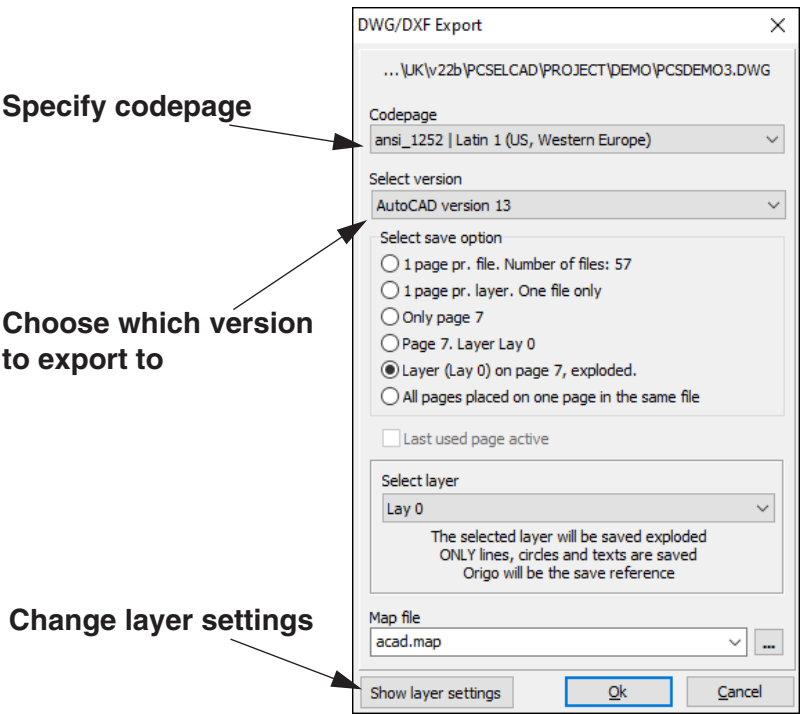
Apart from choosing which version to export to, you have the following options:

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 7	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 0) ON PAGE 7 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.
ALL PAGES PLACED ON ONE PAGE IN THE SAME FILE	ALL PROJECT PAGES ARE PLACED ON THE SAME LAYER ON A SINGLE PROJECT PAGE.

OPTION	RESULTS IN
LAST USED PAGE ACTIVE	IF YOU ON ONE OF THE PROJECT PAGES HAVE ENTERED SETTINGS => PAGE DATA AND SELECTED MARK AS LAST USED PAGE (SEE "MARK AS LAST USED PAGE" ON PAGE 913), YOU CAN USE THIS WHEN EXPORTING DWG/DXF FILES. IN THE DIALOG BOX DWG/DXF EXPORT: - IF YOU SELECT LAST USED PAGE ACTIVE, ONLY THIS PAGE AND PRECEDING PROJECT PAGES ARE EXPORTED. - IF YOU DO NOT SELECT LAST USED PAGE ACTIVE, ALL PROJECT PAGES ARE EXPORTED.
SHOW LAYER SETTINGS	WHEN YOU CLICK HERE, YOU ENTER THE LAYER MENU, WHERE YOU CAN CHANGE THE LAYER SETTINGS BEFORE EXPORTING IF NECESSARY.

EXPORTING A SYMBOL AS A DWG/DXF FILE
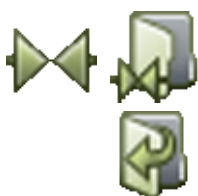
Saving a symbol as a DWG/DXF file is also possible. You do this by first clicking on the **Symbols** button and then on the **Symbol menu** button, which brings you into the **Symbol menu**. — Or press the short-cut <F8> to enter the menu.

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 775.

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 **PCSCHEMATIC 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 901.

INTEGRATION WITH MICROSOFT EXCEL

<div style="text-align:center">

THIS CHAPTER DESCRIBES

THIS CHAPTER IS A SURVEY OF THE VARIOUS WAYS YOU CAN USE MICROSOFT EXCEL TOGETHER WITH **Automation**.

</div>

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 732.

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 500.

4) You can save data from the **Object Lister** in Excel format — see "The Object Lister" on page 351.

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 756.

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.

EXPORT TO PCSCHEMATIC CABLEMANAGER

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO EXPORT CABLE DATA FROM **PCSCHEMATIC AUTOMATION** TO **PCSCHEMATIC CABLEMANAGER**.

The software **PCSCHEMATIC Cablemanager** keeps track of cables and cable raceways. If you need to know more about the program, you can read more about **PCSCHEMATIC Cablemanager** on our website www.pcschematic.com.

In **PCSCHEMATIC Automation** you can export data to **PCSCHEMATIC Cablemanager**.

Preparation for export to PCSCHEMATIC Cablemanager

Before you can export data to **PCSCHEMATIC Cablemanager**, you must first fill out the data field *Segregation* with relevant data on all of the cables in the project, that you wish to export data from. If you haven't filled out the *Segregation* data field for a cable, the cable will not be included when exporting.

— Read more about it in "Symbol datafields" on page 122, specificly "Predefined symbol datafields" on page 126.

Exporting to PCSCHEMATIC Cablemanager

All you then have to do, is simply to click the **Export to Cablemanager** button at the upper part of the screen:

An Excel file with the relevant information is then generated exactly as **PCSCHEMATIC Cablemanager** needs it. If Excel is installed on your PC, the file is opened in Excel automatically.

You can then import the file from **PCSCHEMATIC Cablemanager**.

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**.

<table>
<tr><td colspan="2">THIS CHAPTER DESCRIBES</td></tr>
</table>

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.

Automation project

Word document in Automation as OLE object
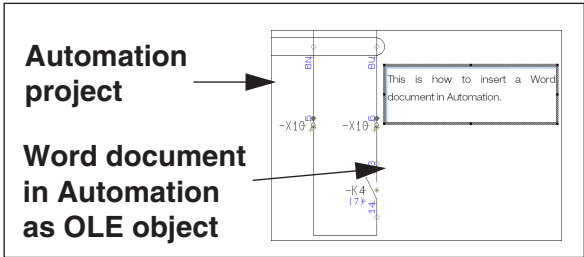
CREATING NEW OLE OBJECTS IN Automation

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 751.

You now enter the following dialog box:

Click here for new OLE object

Choose which type of OLE object you want to create

Click here if you want to display an icon in stead of the document contents
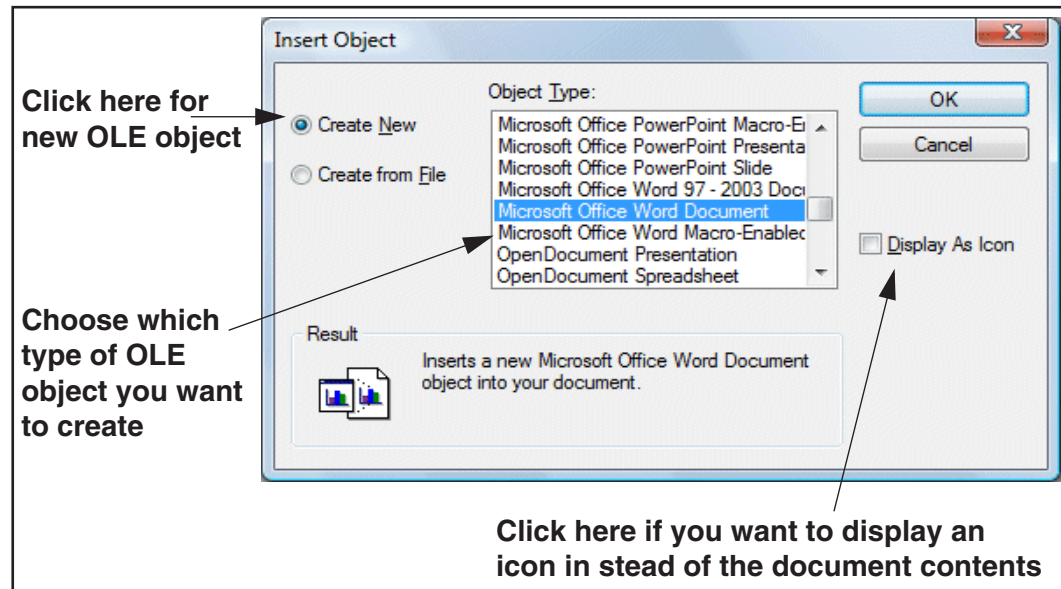
Here you click on *Create new object*, and choose which type of OLE object/document/file you want to insert in the project. The options you get on the list are those of the installed programs on your computer, which supports inserting OLE objects in other programs — for instance the Microsoft Office 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
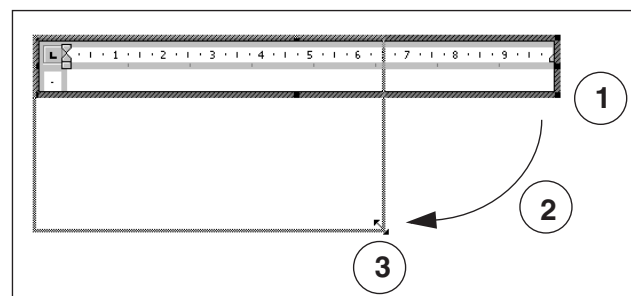
When an OLE object is placed — for instance a Word document — the toolbars in **Automation** are replaced with the toolbars from Word.
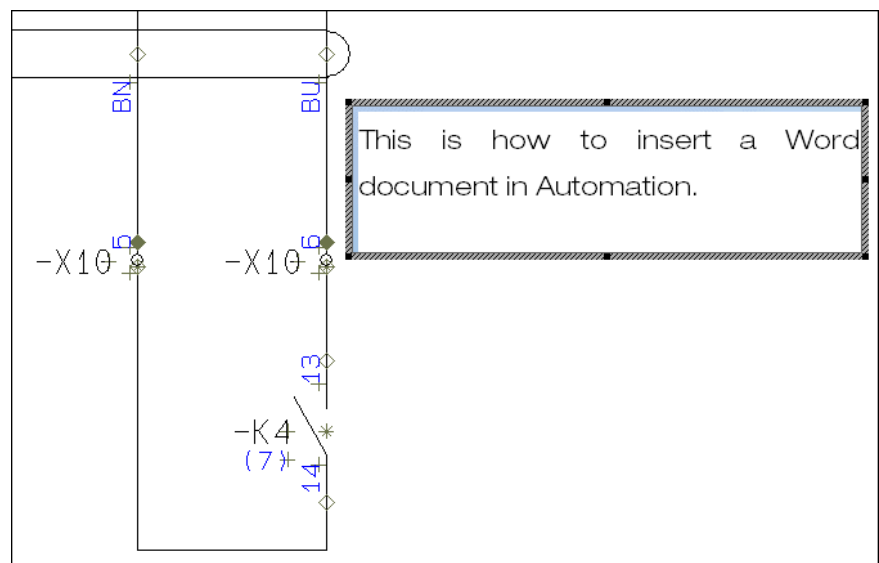
You can now work with the full functionality on Word inside **Automation**.

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 750.

You now enter the following dialog box:

Click here to insert an existing object

Specify where the object/document is palced

Specify whether to create a copy, or whether you want to make changes directly in the original
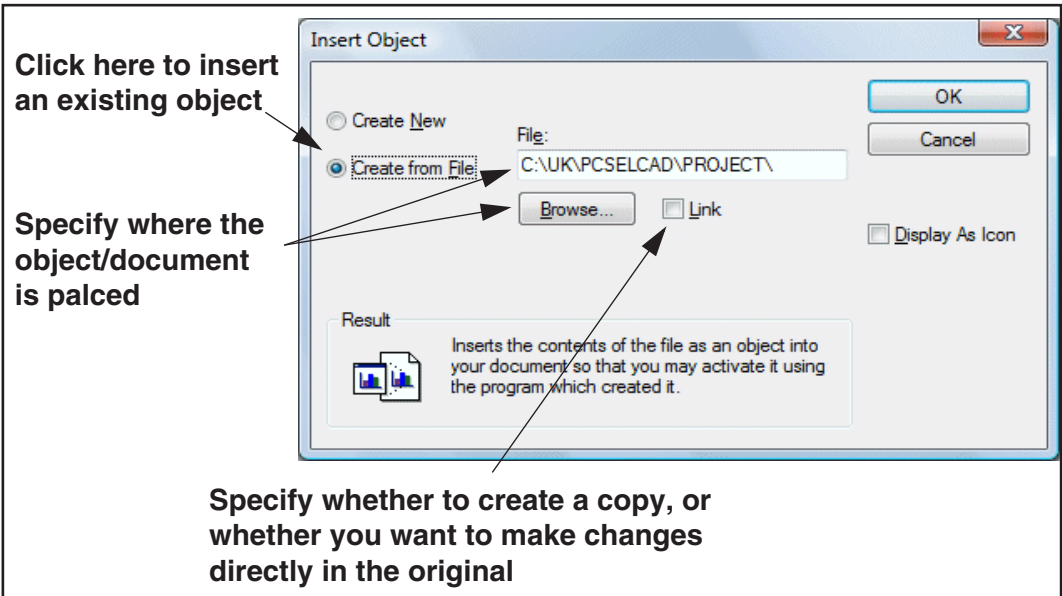
Here you choose *Create from file*, and find the document you want to place in the project, by either clicking on **Browse**, or by typing in the file name in the *File* field (including the *path* — see "Dictionary" on page 980 — leading to the file).

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 750.

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 752.

When you choose *Insert* only a *copy* of the copied area will be inserted, without connection to the original file.
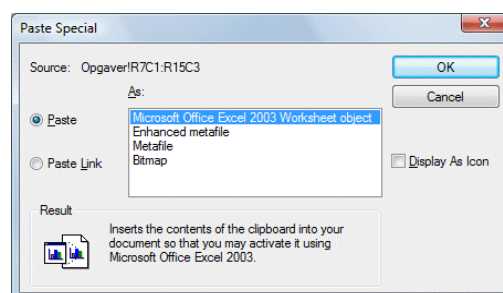
EDITING A PLACED OLE OBJECT

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**
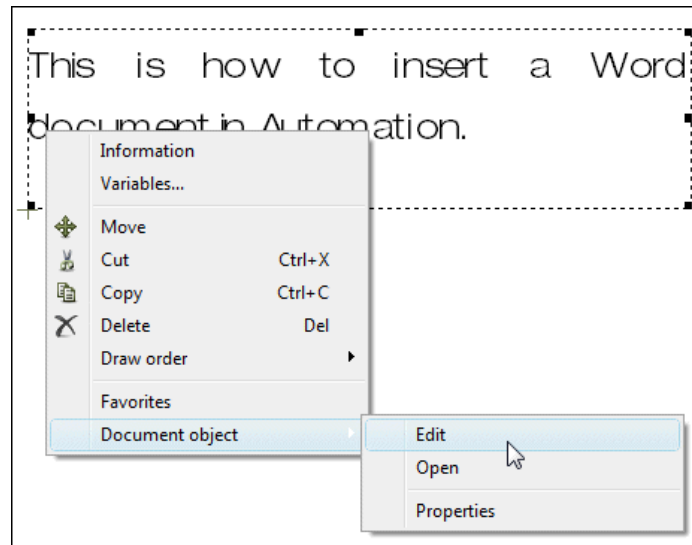
You have now opened the OLE object, and can edit its contents.

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** OR BY CLICKING THE **REFERENCE POINTS** BUTTON IN THE LEFT TOOLBAR.

— PLEASE SEE TO "POINTER/SCREEN" ON PAGE 918.

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 **PCSCHEMATIC 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 751 — 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 741.

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**:
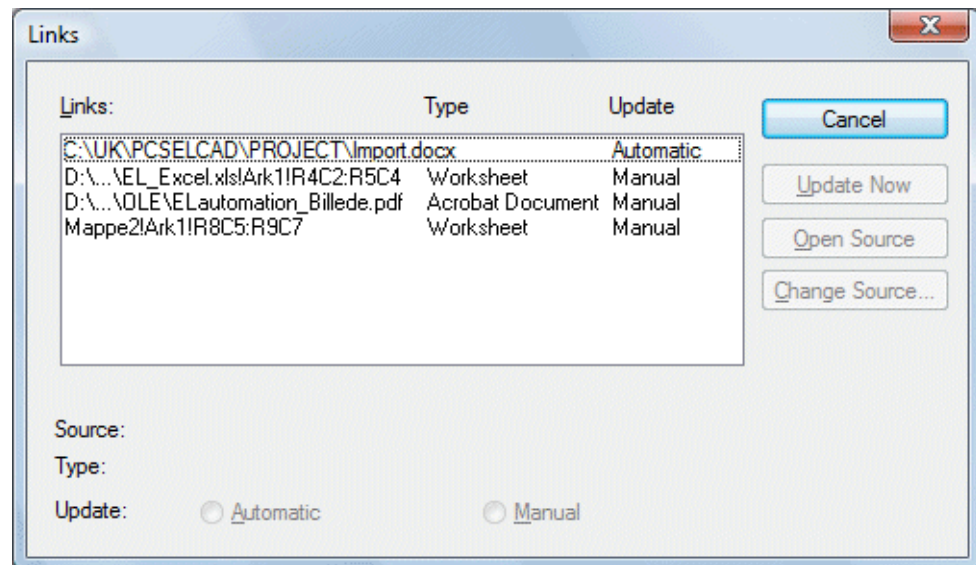
If the contents of one of these OLE objects have changed, then select the object and click on **Update Now**, so that the object is updated in the project also. Please note that the OLE objects in **Automation** are only updated when you select this manually.

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

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 can also contain symbol datafields.

These lists will typically consist of parts for a machine, and could for instance be a components list containing sensors, motors e.t.c.

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 partis list and component-slist 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 759.

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.
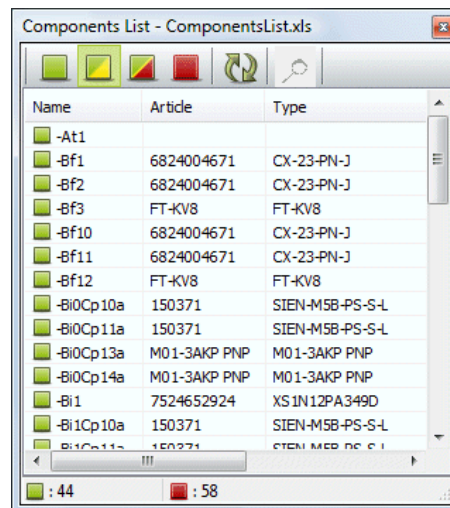
HOWEVER, IT IS ONLY POSSIBLE TO OPEN LISTS IN THE EXCEL FORMAT IF EXCEL IS INSTALLED ON THE PC. — SEE "EXCEL MUST BE INSTALLED TO PERFORM EXCEL IMPORT/EXPORT" ON PAGE 747.

READING COMPONENTS LISTS/PARTS LISTS

1) Choose **Lists => Read Components List** (or **Lists => Read Parts List**).

2) In the dialog box **Select Components List**: 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.

3) In the dialog box **Select Format file**: Select the format file which is to be used — for instance **ComponentsList.cff**.

Placing the components from the Components List window

4) A survey of the components is now displayed in **Automation**:

5) 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, the symbols for all functions in the component are displayed under **Available symbols** in the lower-right corner of the screen.
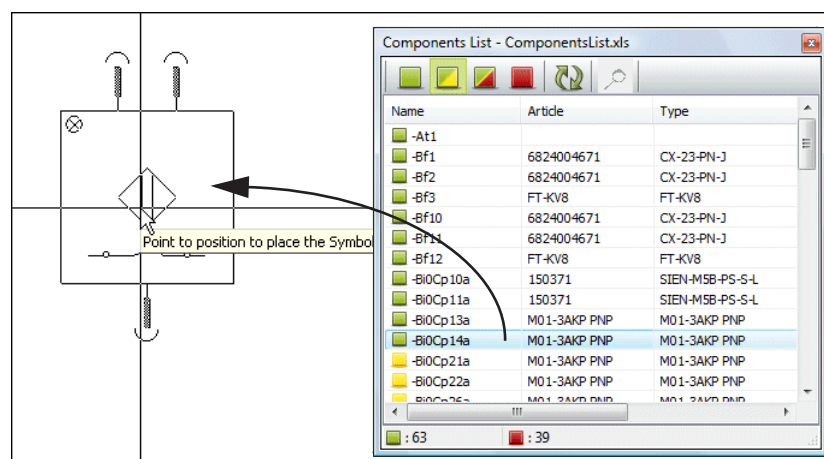
6) Place the symbol, and the components list from the file is updated in **Automation**.

7) 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 918.

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.

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/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:

```xml
- <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 923.

More XML syntax

You can read more about the syntax for XML files in "Lists in XML format" on page 733.

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**.

<table>
<tr><td align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>THIS CHAPTER DESCRIBES HOW TO OPEN FILES IN OTHER PROGRAMS DIRECTLY FROM Automation</td></tr>
</table>

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 926 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:\Pcselcad\list** opened.

Here you click on the desired file, click **OK**, and get the file opened in the program Notepad.

Tools	Help
Rename symbols	
Database	
Symbol Documentation	
Graphical Terminal Plan	
Graphical Cable Plan	
Graphical Connection Plan	
Text translator	
DWG/DXF Map settings	
Create variable list in (Sub)drawing	
Project generator	
Mouse Chasing System	
Panel Builder	
Show textfile	
PC|SCHEMATIC parts list	
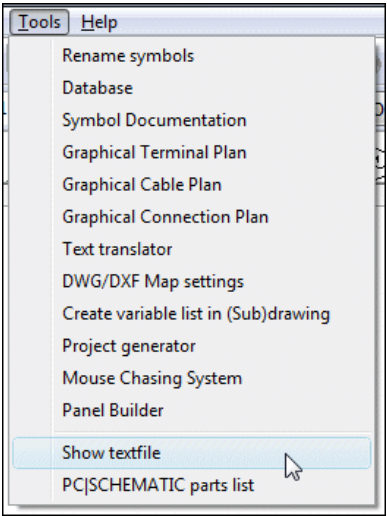
PCSCHEMATIC VIEWER AND PCSCHEMATIC SERVICE

PCSCHEMATIC Viewer

For the **Automation** software there exists a **PCSCHEMATIC Viewer**, which you can send to your customers for free. The Viewer can be downloaded from www.pcschematic.com/viewer in the English, German, Polish, Czech, Hungarian, Russian, Chinese, Swedish, Norwegian and Danish version.

Options in PCSCHEMATIC Viewer

When you open a project with the **PCSCHEMATIC 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 out on paper

You do *not* have the option of editing projects in the **PCSCHEMATIC Viewer**.

PCSCHEAMTIC Service has more options

For service and maintenance **PCSCHEMATIC Service** (commercial version) gives you more options. Beyond the options in the viewer, you can also:

- Apply the **Mounting Assistent**
- Export production list files
- Export to PDF (with active links)
- Navigate in the project with the **Net Navigator**
- Change object type and hereby specify which type of objects you are working with
- Right-click on objects, and hereby for instance navigate in projects starting from the individual symbols
- Get information from the attached database
- Create short-cut keys

You do *not* have the option of editing projects in **PCSCHEMATIC Service**.

At www.pcschematic.com/manuals you can download a short introduction to the options that you have in **PCSCHEMATIC Service** — including how to use the **Mounting Assistent**.

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:

Reading this section saves you a lot of time in the projects to come.

VIDEOS ABOUT CREATING YOUR OWN SYMBOLS

WATCH THE VIDEOS ABOUT HOW TO CREATE YOUR OWN SYMBOLS AT THE PCSCHEMATIC YOUTUBE CHANNEL:

www.youtube.com/PCSCHEMATIC

PC|SCHEMATIC

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.

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.
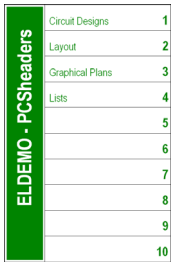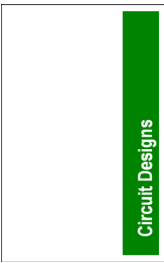
Front page	Table of Contents	Chapter Front page	Chapter Table of Contents	Parts-, components- and Cables list
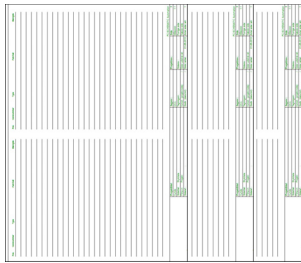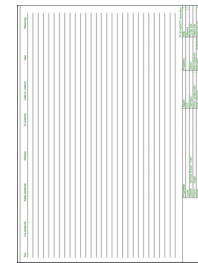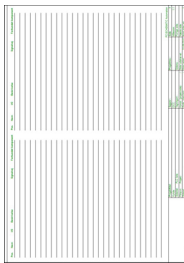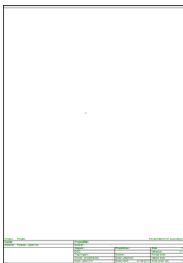
Terminals list	PLC list	A4 Drawing header	A3 Drawing header

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 820, "Creating Lists" on page 831 and "Changing an already created list/drawing header" on page 843.

If you need to create your own unit drawings, please read "Creating Unit Drawings" on page 846.

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 850, particularly "Creating Page templates — via the Template editor" on page 851.

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.
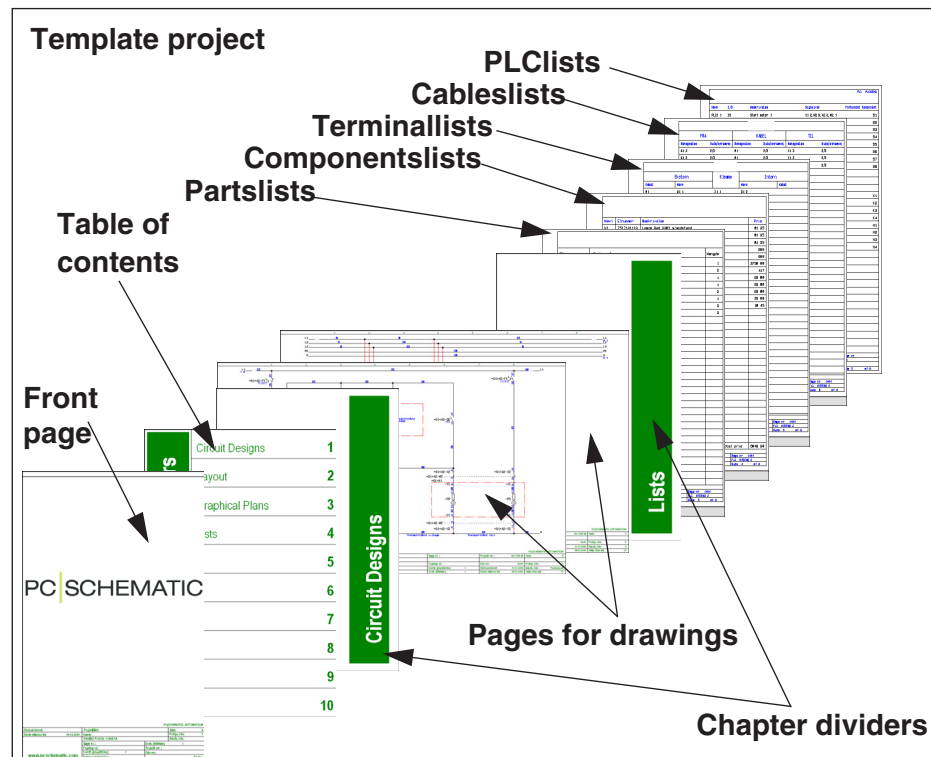
When you start up a new project, you start by selecting the project template, and continue working from this.

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 850, particularly "Creating Project templates" on page 853.

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 299.

These sub drawings can also be applied in automatic project generation, which you can read more about in the **Projekt generator** manualen.

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 769.

CREATING SYMBOLS AND SYMBOL DATAFIELDS

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 775 to see how to design symbols,
- "Symbol datafields" on page 122 about how to create your own symbol datafields,
- "Creating Data symbols" on page 810 for designing data symbols, and
- "Creating Signal Symbols" on page 816 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 118, 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.

A survey of the datafields in **Automation** can be found in "Datafields in Automation" on page 944.

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.
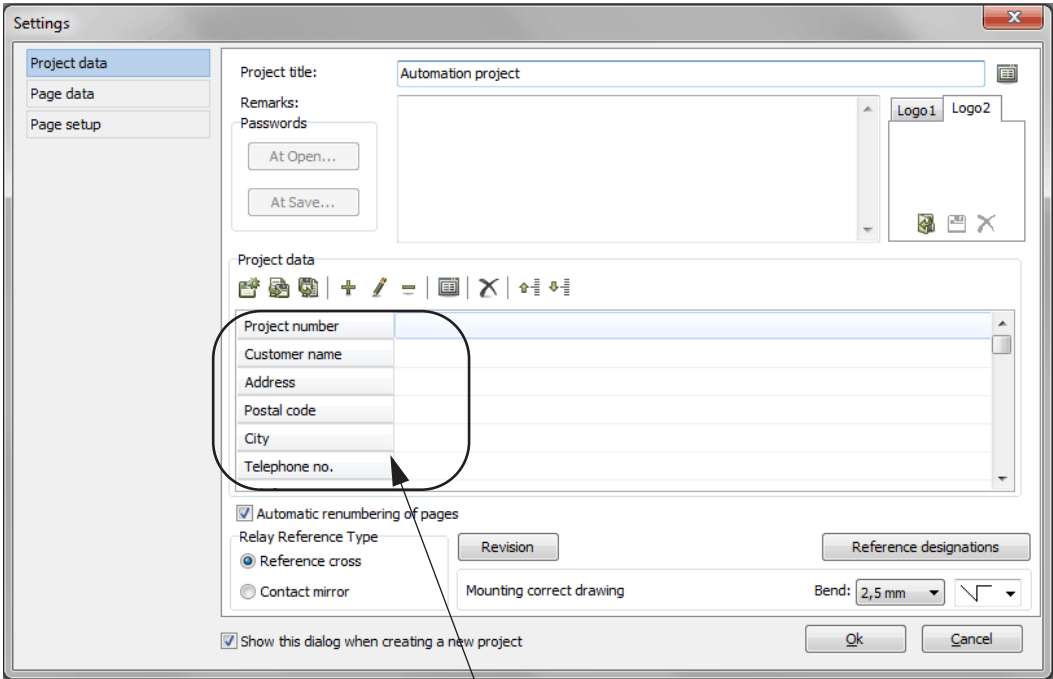
Project data datafields

If you need to insert one of these informations in your project — for instance the name of the customer — do this:

PC|SCHEMATIC

1) Choose **Insert => Insert Datafield**.

2) In the dialog box **Datafield**: Click on *Project Data* and then on the *datafield Customer name*.

3) You now get the chosen datafield in the cross hairs: Click to specify where to place it on the page.

When you update your list, the name of the customer will be read from the **Project Data** tab, and inserted on your list.

Datafield in the drawing header definition

Customer name — Hansen Machinery
Address
Zip code

Project title: Demo project
Customer: Hansen Machinery
Last printed: 24-05-00   15: 35
Last changed: 24-05-00   14: 25

The contents of the datafield specified in Settings => Project data

Datafield in the drawing header in the active project
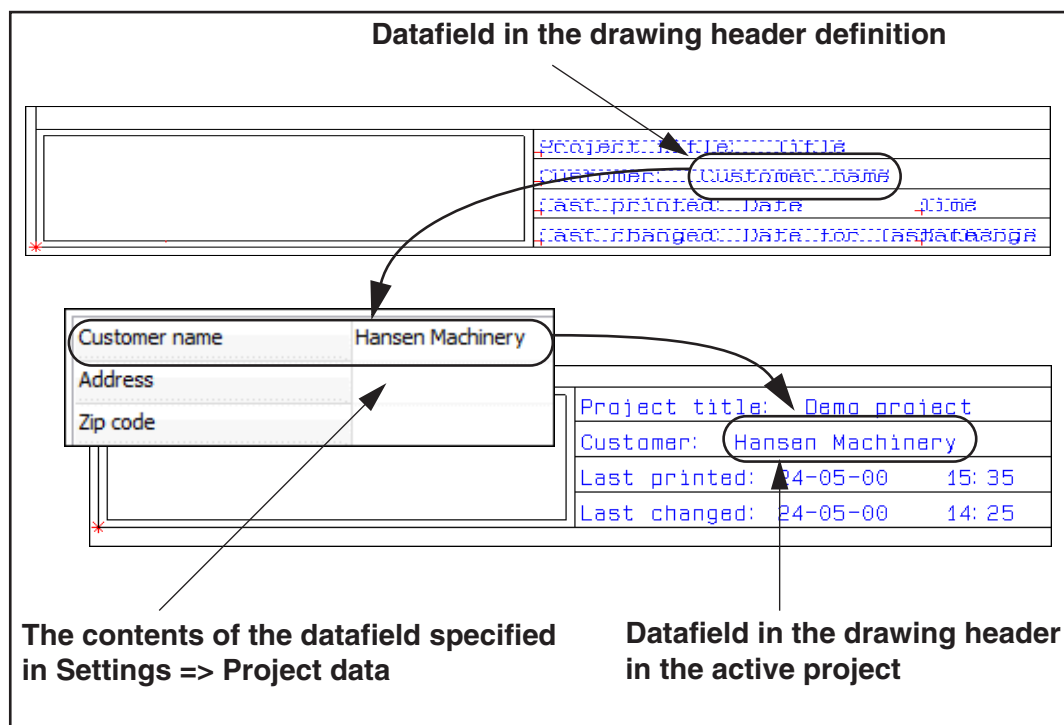
If the customer change address later on, you *only* have to change this address once in **Settings => Project data**, which automatically corrects the address on all the pages where the datafield *Customer Name* exists in the project.

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

Drawing header/List filled out with project data
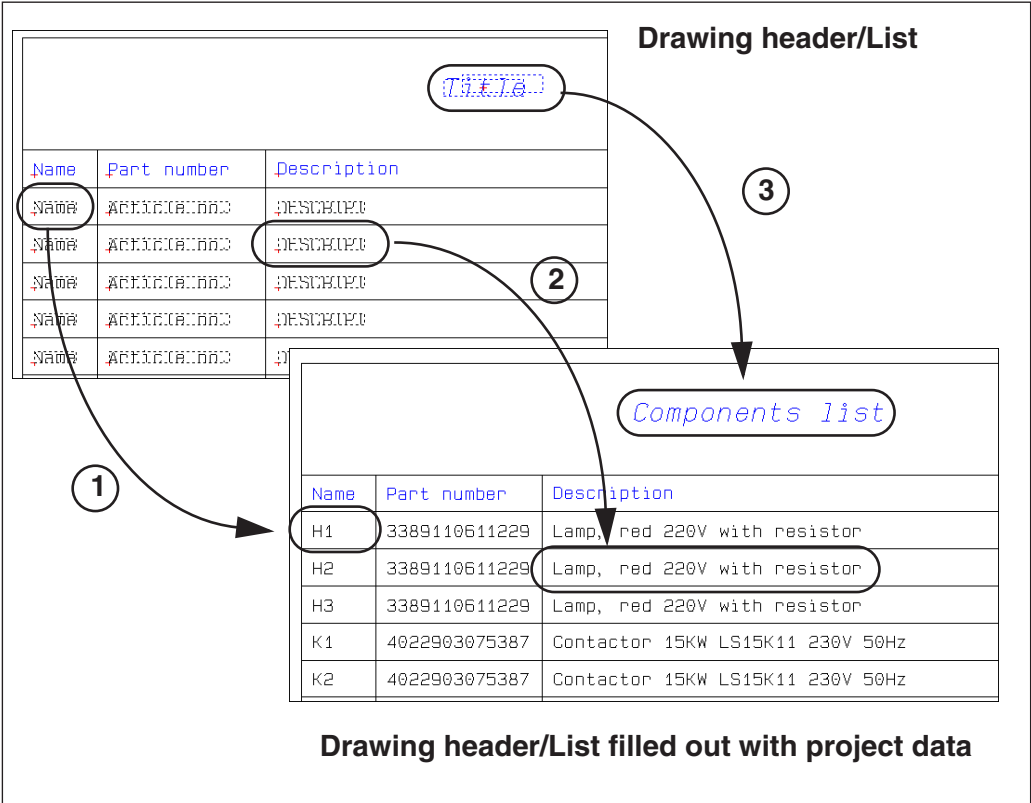
The figure above displays the following types of datafields in lists:

1) The contents of the datafield *Name* collected from the diagrams

2) The contents of the datafield *DESCRIPT* 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 118.

SPECIFYING AVAILABLE DATAFIELDS

When you insert datafields using **Insert => Insert Datafield**, the contents of some of these datafields, are specified in the dialog box **Settings**.
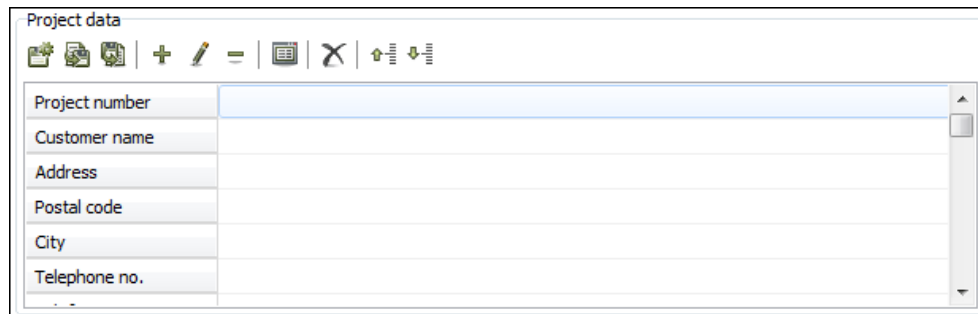
You can specifying which of datafields on the **Project data** and **Page data** tabs you want to use in your projects.

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 datafields, that can be inserted as **Project data** datafields in your projects, do this:

1) Enter **Settings => Project data**:

2) Click on the name of the project data datafield that is placed where you wish to insert the new project data datafield and click the **Add datafield** button. (You can also right-click on the datafield and choose **Add datafield**).

3) In the dialog box **Enter text**: Type in the name of the new datafield and click **OK**.

You can now use the new Project data datafield in the project.

Please note that the new datafield is positioned in the list where you clicked initially. If for instance you clicked the **Address** field, this field is moved one position downwards, and the new datafield has taken its place.

You can also delete and edit datafields, change the position of the datafields in the dialog box, remove unused datafields etc. Read more about this in "Buttons for handling Project and Page data datafields" on page 910.

Changes in the datafields for **Page data** is made correspondingly.

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.

PROJECT AND PAGE DATA: LIST OF VALUES FOR A SERIES OF DATAFIELDS

In lists of values for **Project** and **Page data** datafields it is possible to fill out a series of datafields in a single operation — for instance all information attached to a specific customer.

When a value in the list of values contains the characters **++** this specifies that the consecutive datafield(s) in **Project data** / **Page data** will also be filled out with the text entered after the **++** characters.

If for instance you enter the value
PCSCHEMATIC A/S++Bygaden 7++4040++Jyllinge++4678 8244
five consecutive datafields will be filled out in one operation. Do this:

Create the values in the list of values

1) Click on the first of the datafields — here the datafield **Customer name** —, and click the **Values** button.

2) In the dialog box **Values**: Type in the text ***PCSCHEMATIC A/S++Bygaden 7++4040++Jyllinge++4678 8244*** and click **OK**:
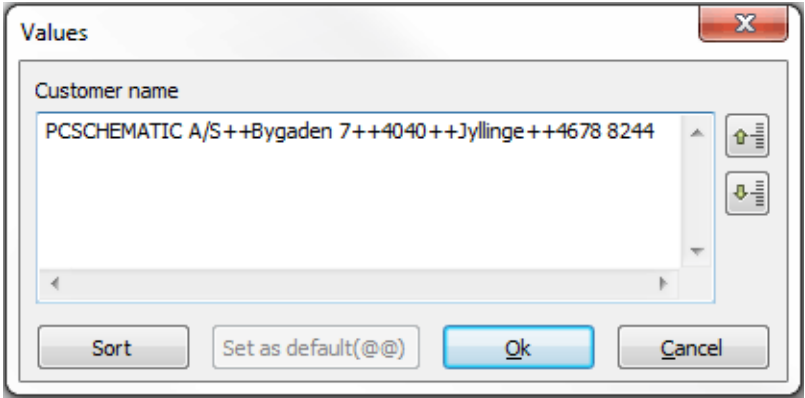
Select values from the list

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:
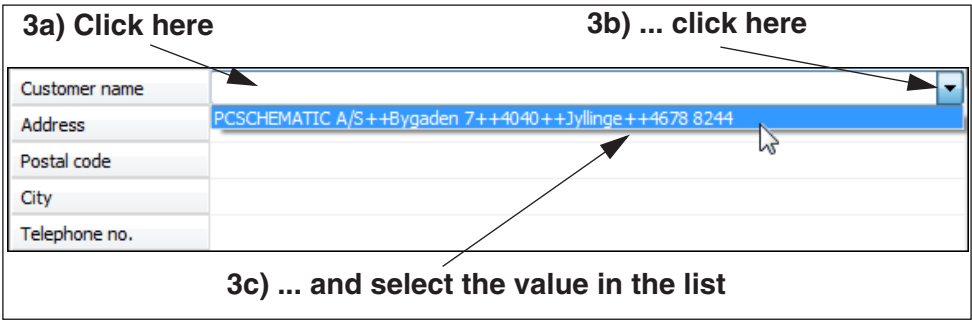
4) Hereby the datafield is filled out — as is the four subsequent datafields in the dialog box **Project data**:

Customer name	PCSCHEMATIC A/S
Address	Bygaden 7
Postal code	4040
City	Jyllinge
Telephone no.	4678 8244

Series of datafields can be filled out the same way for **Page data**.

— Read more about lists of values in "The Values dialog box" on page 124.

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

When you are 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.

In the following chapters you can see how to create signal symbols in "Creating Signal Symbols" on page 816, and how to create data symbols in "Creating Data symbols" on page 810.

— To see how to use the already created symbols, please read "Symbols" on page 152.

AUTOMATIC CREATION OF SYMBOLS

IF YOU WANT THE PROGRAM TO CREATE SIMPLE SYMBOLS FOR YOU AUTO-MATICALLY, PLEASE READ "AUTOMATIC GENERATION OF SYMBOLS" ON PAGE 226.

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**:
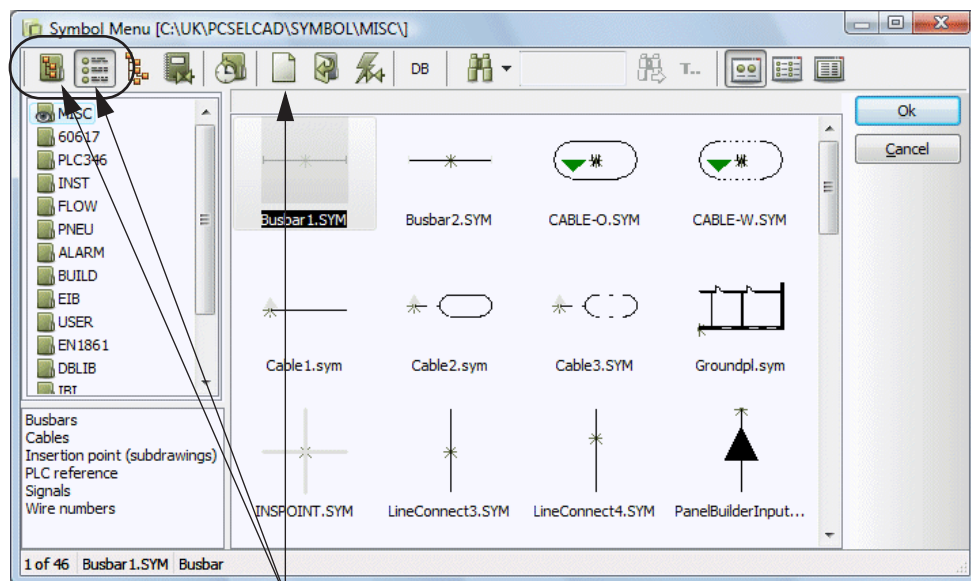
Click here

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 162.

You now enter the dialog box **Symbol settings**:
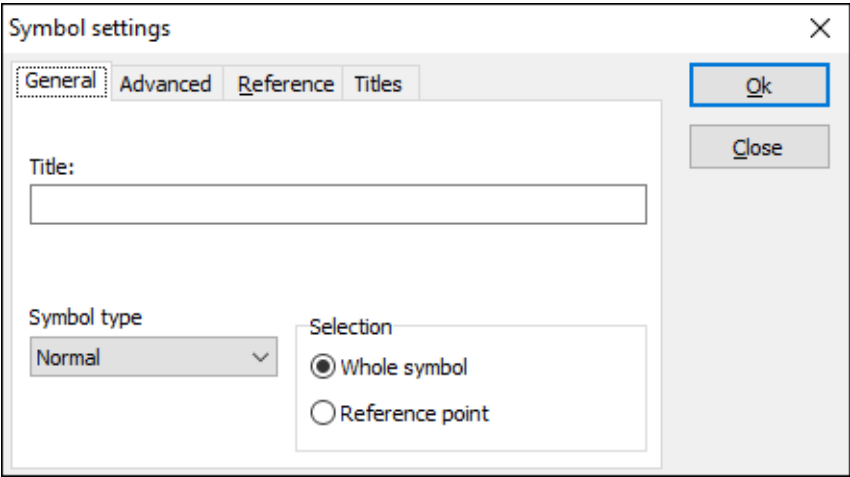
Here you specify under **Symbol type** which type of symbol that you wish to create. Because we are about to create a symbol of the type *Normal*, just click **OK**.

The program is now in 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:
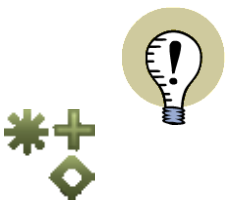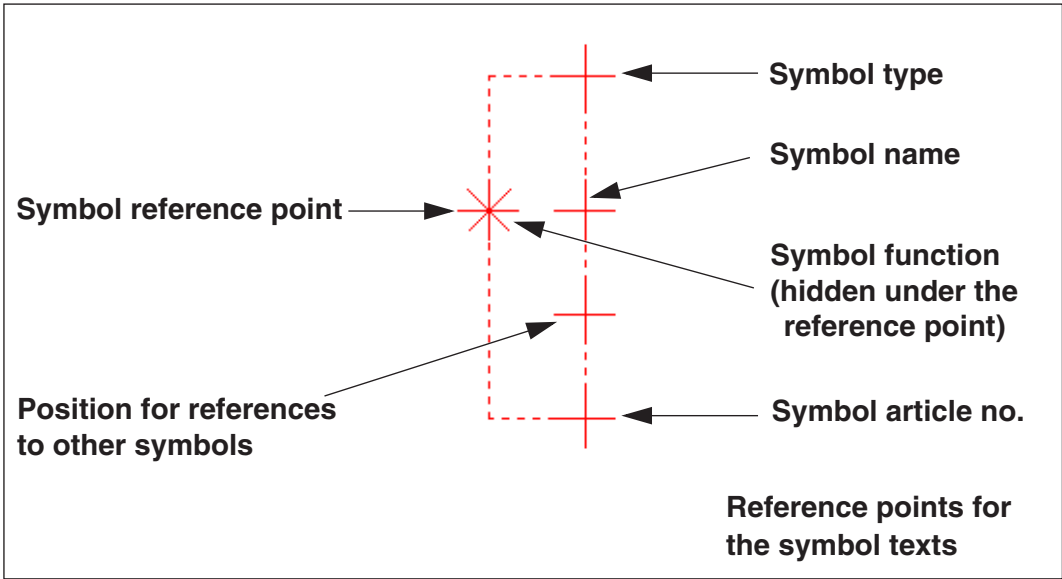
"VIEW REFERENCE POINTS" MUST BE ENABLED

IF YOU CAN NOT SEE THE REFERENCE POINTS WHEN YOU ZOOM IN ON AN AREA, THIS IS BECAUSE **VIEW REFERENCE POINTS** HAS NOT BEEN ENABLED. ENABLE IT BY CLICKING THE **REFERENCE POINTS** BUTTON IN THE LEFT TOOLBAR, OR BY SELECTING **VIEW => REFERENCE POINTS**.
— GET MORE INFO ABOUT THIS IN "SETTINGS" ON PAGE 907.

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 115.

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 226.

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 <l>), 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 97.

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.

Frame displaying how much
space the symbol needs
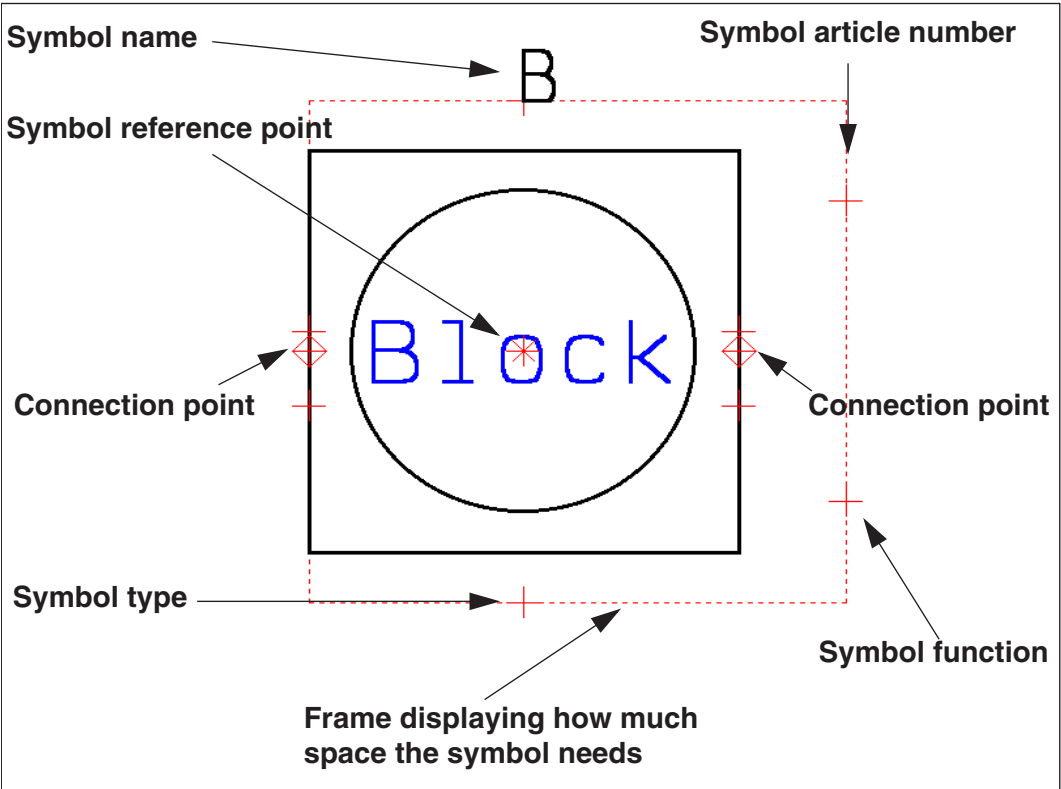
When you then place the connection points, you will be asked about their connection point data.
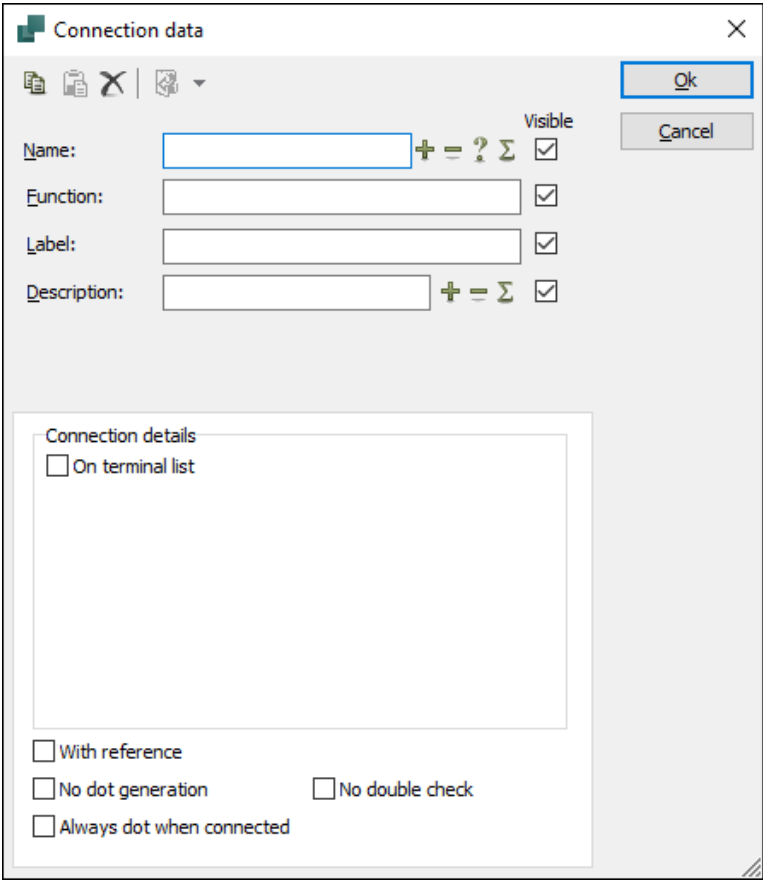
In this case, you can for instance give them the names *1* and *2* respectively, and make the names invisible on diagram pages, by deselecting the **Visible** check box for the **Name** field.

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 182, "Wildcards/jokers" on page 618 and "Symbols with variable connection names" on page 670.

MORE OPTIONS IN THE DIALOG BOX CONNECTION DATA

On terminal list

When you select **On terminal list**, the connection point is included in terminal lists.

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 803 and "Creating PLC reference symbols" on page 806.

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 781.

No double check

When performing a *design check* — see "Design Check" on page 365 —, 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.

| R: 5.0 ▼ | V1: 0 ▼ | V2: 360 ▼ | B: 0.25 ▼ | F: ■ ▼ | E: 1 ▼ |
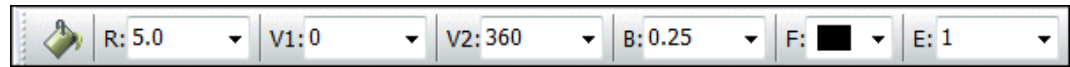
You may read more about this tool bar in "Arcs/Circles" on page 108.

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 110 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>.

| Aᵃ | Block |
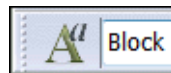
You now have the text in the cross hairs.

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 136.

Text properties

Font: PCSCHEMATIC

☐ Proportional

ABCDEF

Std. Text

Height 3,0
Width AUTO
Alignment ▣ +
Frame
Color ■

☐ Bold
☐ Italic
☐ Underline
☑ Visible text
☐ Fixed text direction
☐ Not translated

Line length
☐ Suppress linefeed char
☐ Wrap text

Ok   Cancel
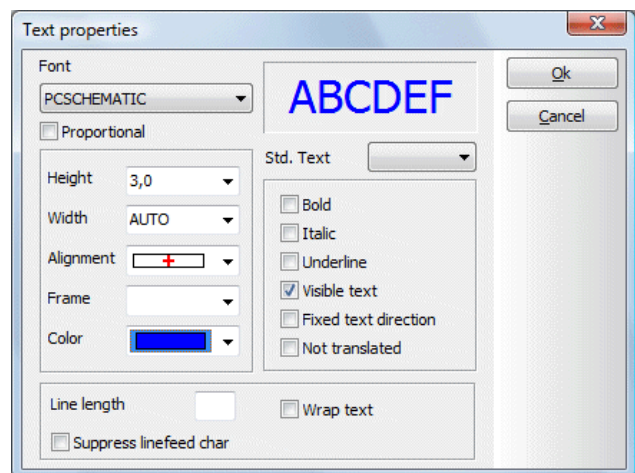
Place the text by clicking in the middle of the symbol.

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:
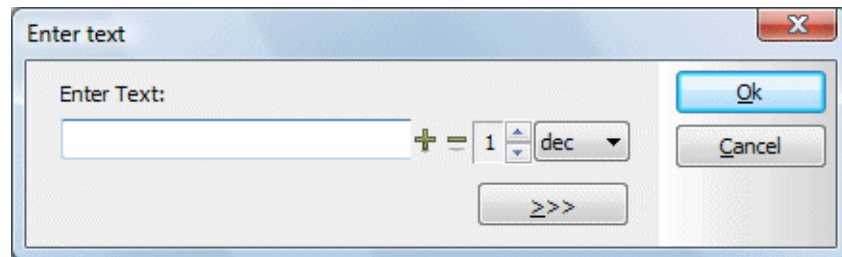
Type in *B* and press <Enter>. — In stead of pressing on <k> you can click in the text field in the command tool bar, type in *B* and press <Enter>.

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 918.

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 143.

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: ✳ ◇ N: Name   T: Type   V: Article no   F: Function   ⚙ 🗔 ⚔ 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.

Symbol settings

Now activate **Symbols** and click the **Symbol settings** button. You now enter the dialog box **Symbol settings**.

Here you give 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*.
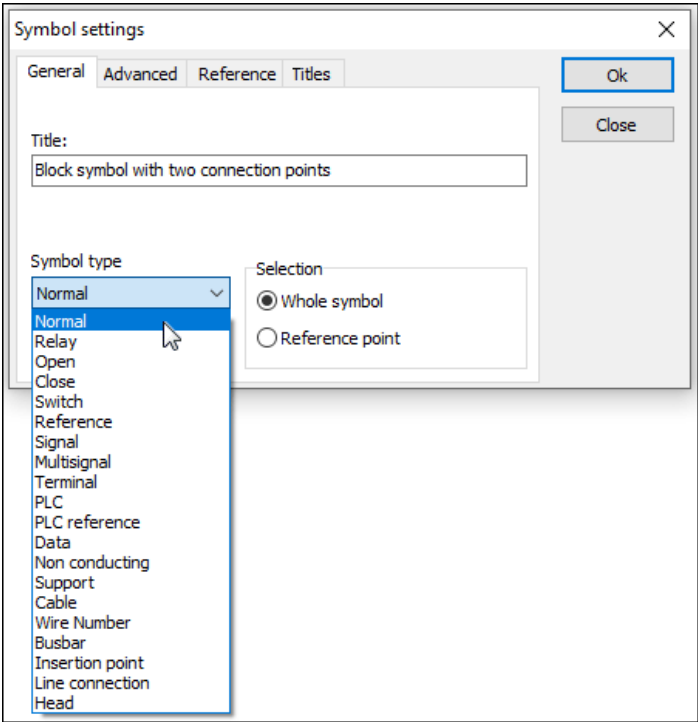
THE SYMBOL SETTINGS BUTTON

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

Under *Symbol types* in the dialog box **Symbol settings** you can choose between the following symbol types. When you have assigned a symbol type to a symbol, it is not possible to change the symbol type subsequently:

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.

Symbol type	Use
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.
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 816, and in "Signal symbols" on page 234.
Multisignal	Reserved for future use.
Terminal	Unless you take other action, the symbol will appear in terminals lists.
PLC	See "Creating PLC I/O symbols" on page 803. The symbol will appear in PLC lists.
PLCreference	Symbol, which shall be used as a PLC reference symbol. — See "Creating PLC reference symbols" on page 806 as well as "Working with PLCs" on page 478, particularly "On PLC reference symbols" on page 478.
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 810.
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 77 and "Color" on page 138), 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 248, and "Creating cable symbols" on page 800.
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 455.
Busbar	The symbol is a busbar. — See "Creating busbar symbols" on page 802.
Insertion point	Used for automatic insertion of sub drawings. — See "Placing sub drawings with drag'n draw" on page 307.

SYMBOL TYPE	USE
LINE CONNEC-TION	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 808.
HEAD	SYMBOL APPLIED FOR DIRECT PLACING OF A LIST ON A PAGE. — SEE "CREATING A LIST SYMBOL" ON PAGE 842.

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 | Titles

Ok

Close

☐ Auto scale
☑ No scale adjustment
☐ Multisymbol
☑ Mechanical
☑ Electrical

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.

SYMBOL SETTINGS: ADVANCED

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 794.

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 179.

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 179.

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 435, "INSERTING REFERENCES ON SYMBOLS WITHOUT REFERENCE" ON PAGE 435 AND "SYMBOL TYPE AND REFERENCES" ON PAGE 436.

SYMBOL SETTINGS: TITLE

IF NECESSARY, CLICK TIL **TITLE** TAB:

Symbol settings	✕
General Advanced Reference **Titles**	Ok
1033 UK Block symbol with two connection points	
	Add a language title
	Remove a language title

HERE YOU CAN ADD AND DELETE TITLES IN DIFFERENT LANGUAGES.

WHEN YOU RIGHT-CLICK THE TAB, YOU CAN CHOOSE EITHER **ADD A LANGUAGE TITLE** OR **REMOVE A LANGUAGE TITLE**. WHEN YOU CHOOSE TO ADD A LANGUAGE TITLE, YOU WILL SUBSEQUENTLY BE ASKED TO SPECIFY A LANGUAGE CODE FOR THE TITLE.

WHEN YOU PLACE THE SYMBOL IN A PROJECT, THE SELECTED LANGUAGE FOR THE WHOLE INSTALLATION DETERMINES WHICH LANGUAGE TITLE IS DISPLAYED FOR THE SYMBOL.

— SEE "CHOOSE LANGUAGE" ON PAGE 931.

Saving the symbol

Click the **Save** button to save the symbol.

The 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 256 to determine in which folder you will save the symbol.

Click on **Save**. The symbol/symbol definition has now been saved under the symbol file name *Block.sym*.

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**.
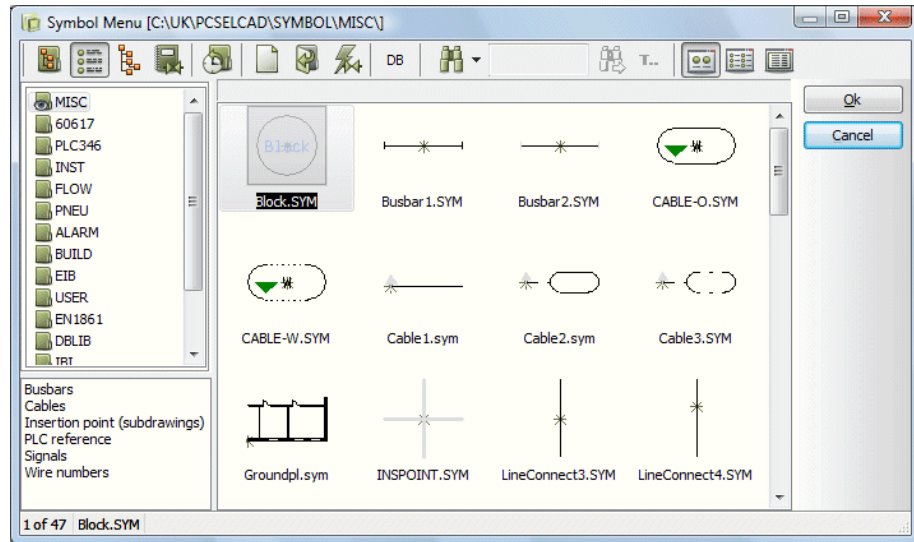
You now return to the **Symbol menu**, where you can find the symbol by changing to the folder you saved the symbol in.

Here the *symbol file name* is displayed below the symbol in the menu. When you place the cursor above the symbol, its **Title**, **Name** and **Type** is displayed as a hint.
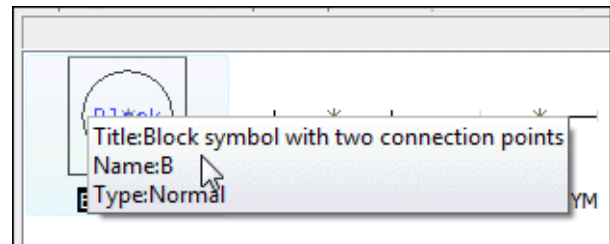
If you need the symbol on your drawing, you then click on **OK**.

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 152.

Renaming symbol file names

To rename the symbol file name of 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 file name, and click **OK**

EXTRA SYMBOL DATAFIELDS IN SYMBOLS

You can also insert extra datafields in the definition of a symbol. To do this you must first create the relevant symbol datafields, and then insert these in the symbol definition. Read more about this in "Symbol datafields" on page 122.

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 fields" on page 186.

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 in the definition of a symbol, do this when you are in the **Edit symbol** mode:

1) Choose **Insert => Insert Datafield**.

2) In the dialog box **Datafield**: Click on *Symbol datafield* and select one of the datafields, that fetches information from the database. (The datafields fetching data from the database, are named **DB[xxxxxx]**, 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 810.

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 fields" on page 186.

Inserting datafields with reference designations in the symbol definition

To insert a reference designation for a symbol in the symbol definition, you must specify that you want to insert a *Symbol datafield*, as described above, and insert the symbol datafield *=+-design.*.

— See more about the options for the datafield in "Symbol datafields" on page 947 in the subsection "Edit symbol: Description and reference designation as extra Symbol datafield" on page 947.

Descriptions in multiple languages in the symbol definition

You can also insert the datafield *Symbol Datafield => Description* in the symbol definition, which also makes it possible to control in which language the description of the symbol is written. — See "Description" on page 685.

CREATING SYMBOLS WITH STATES

When you wish to create a symbol with states — see "Symbols with states" on page 190 — you start by drawing the symbol as described in "Creating Symbols" on page 775.

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:
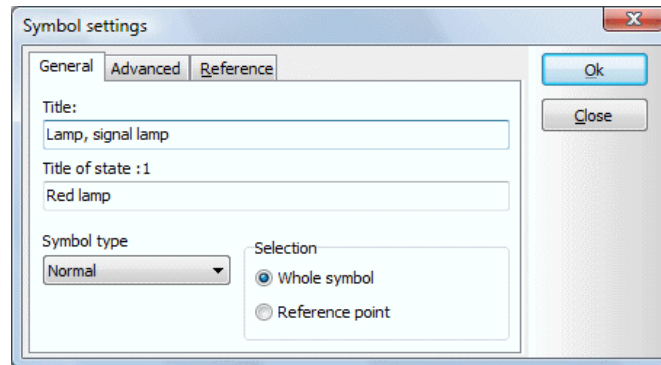
To specify the title for another state, you must first change layer using the **Layer menu**, and then name the state for this layer as described above.

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:
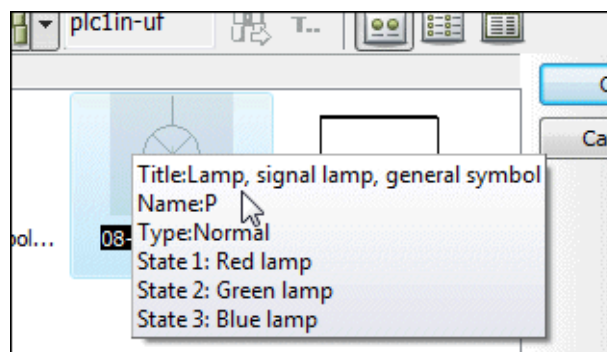
The states and state titles are displayed in the Component data dialog box

When you place a symbol in the project — see "Symbols with states" on page 190 — 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 811.

Inserting states and Removing empty states

When working with multiple symbols for which you want to apply the same state numbers for identical states, it may become necessary to adapt which layers are used for specific purposes for the individual symbols. Therefore, you can right-click on a layer in the **States** dialog box and choose **Insert state** or **Remove empty state**, which makes it easier to adapt the state numbers for the individual symbols.

FOR THE DESIGN PHASE ONLY
PLEASE NOTE, THAT THIS IS MEANT FOR THE DESIGN PHASE ONLY. IF YOU, AT A LATER STAGE, REPLACE A SYMBOL IN A PROJECT WITH A SYMBOL WHERE THE STATE NUMBERS / LAYERS HAVE OTHER MEANINGS, THE APPLICATION OF THE STATES WILL NO LONGER MAKE SENSE.

Other relevant links

— See how to work with layers in "Layers, Elevation and Scaling" on page 541.

— See also "Symbols with states for single line diagrams" on page 474 and "Placing symbols with selected states in the pickmenu" on page 206.

— 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 666 and "Specifying symbol state for mechanical symbols" on page 674.

— See how to use mechanical symbols with states in order to easily create module based general arrangement drawings in "Module based symbols in general arrangement drawings" on page 706.

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 541.

PC|SCHEMATIC

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**.
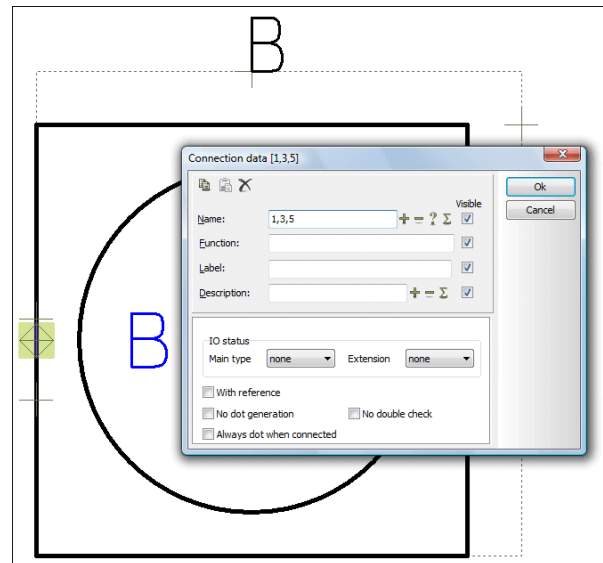
Then right-click on the connection point on the right side of the symbol, choose **Connection data**, type in *2,4,6* in the *Name* field, and click **OK**.

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**.
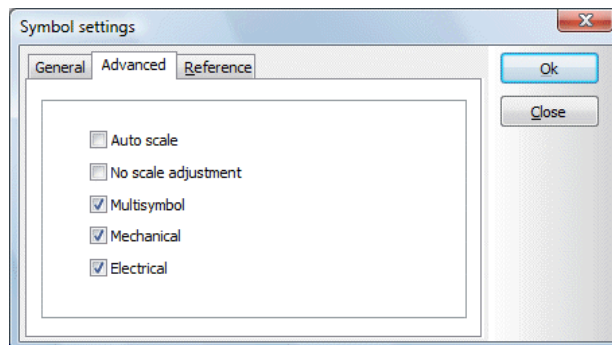
On this tab you select *Multisymbol*, and click **OK**.

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), three identical symbols are displayed under **Available symbols** at the bottom of the **Component menu**.

Click on the symbols, and place them individually on the drawing.

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 PCSCHEMATIC.

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:

Choose symbol text   **Specify the text position in relation to the symbol reference point**   **Specify text properties**
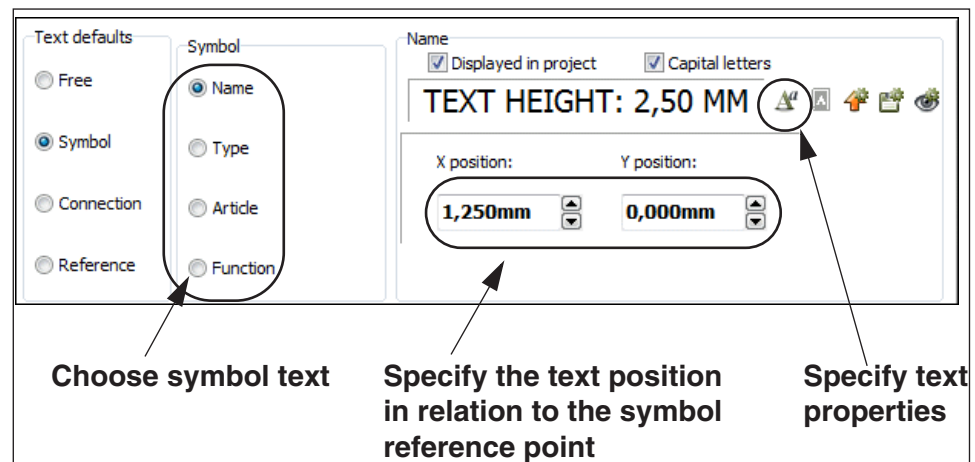
4) Specify the text position in relation to the symbol reference point, and click the **Text properties** button, if necessary.

5) In the dialog box **Text properties**: Specify the desired text properties, as described in "Specifying text properties / Appearance of text" on page 136, 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 741.

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 292.

CREATING TERMINAL SYMBOLS

When creating terminal symbols (with the symbol type **Terminal**) the following applies:

- Symbols with the symbol type **Terminal** are always through terminals. This means that there is the same electrical potential on both sides of the symbol.

- All connection points in the symbol have the same name.

- There are at least one connection on each side of the terminal symbol (**Side 1** and **Side 2** respectively).

- All connections appears on terminals lists by default.

CHANGED SYMBOL TYPES FOR TERMINALS IN VERSION 22

THE WAY YOU WORK WITH SYMBOL TYPES HAS CHANGED IN VERSION 22 OF AUTOMATION. YOU CAN GET AN OVERVIEW OVER THE CHANGES IN THE NEWS DOCUMENT FOR VERSION 22.

CHOOSE **HELP => PROGRAM NEWS** — OR VISIT WWW.PCSCHEMATIC.COM/ MANUALS — TO FIND THE NEWS DOCUMENT.

When you wish to create a terminal symbol and enter the dialog box **Symbol settings**, you must set **Symbol type** to **Terminal** and click **OK**.

When you then place a connection point, you enter the dialog box **Connection data**.

Under **Terminal side** you specify whether the connection point is for input — **Side 1 (int)** — or output — **Side 2 (ext)**. This has significance in terminals lists.

Connection points set to **Side 2 (ext)** are displayed on the screen as filled with color, while connection points set to **Side 1 (int)** are not filled.

When you select **On terminal list** the connection points are included in terminals lists, and the connection point is displayed as a square on the screen. When **On**

terminal list is *not* selected, the connection point is displayed as a rhumbus/diamond. — See "Appearance of connection points for terminals" on page 215.

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.

| **No dot generation** | **Dot** |
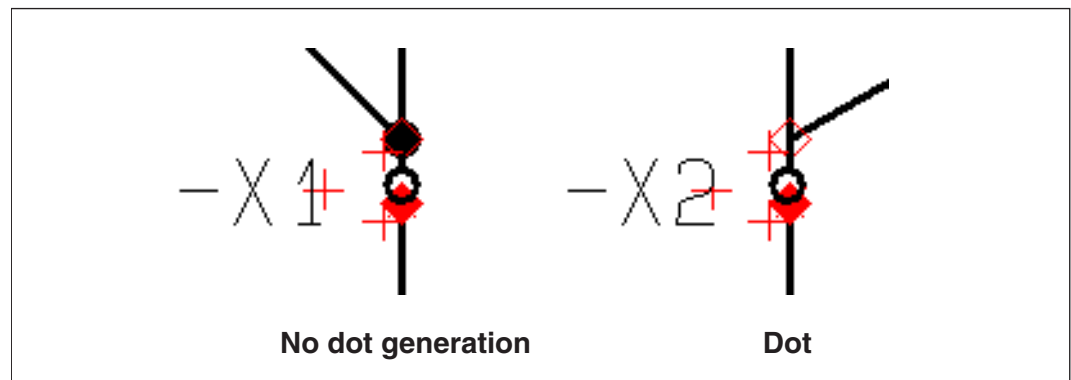
Save the connection symbol

In the dialog box **Connection data**: Click **OK**.

Click the **Save** button when the symbol is ready.

You can read more about terminal lists in "Creating a Terminals list" on page 839.

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.

Create the cable symbol with the type Cable

When you start creating a cable symbol and enter the dialog box Symbol settings, you must set the Symbol type to Cable. — See "Symbol types" on page 785 if necessary.

— See "Creating busbar symbols" on page 802.

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.
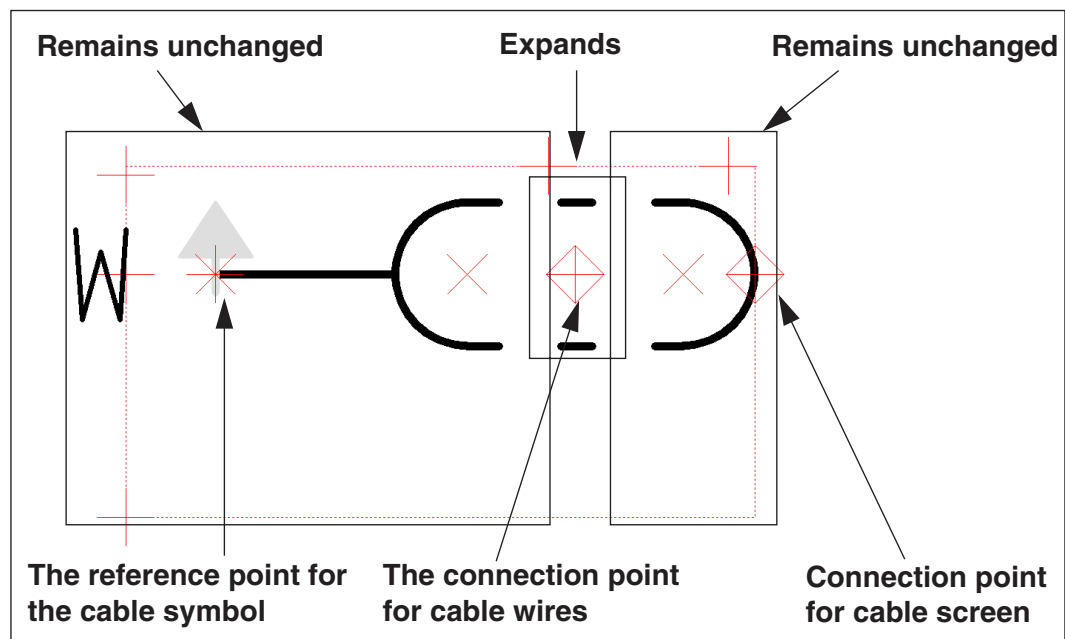
The connection point you place in the cable definition states the location of the first cable wire in the cable. As you add more cable wires to the cable on a diagram, the cable symbol expands.

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 136 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 785.

— See how to work with cable symbols in "Working with cable symbols" on page 248.

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 **PCSCHEMATIC Database** manual for further details.

Most used cable direction

For setting the most used cable direction, please refer to "Default Cable direction" on page 255.

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*:
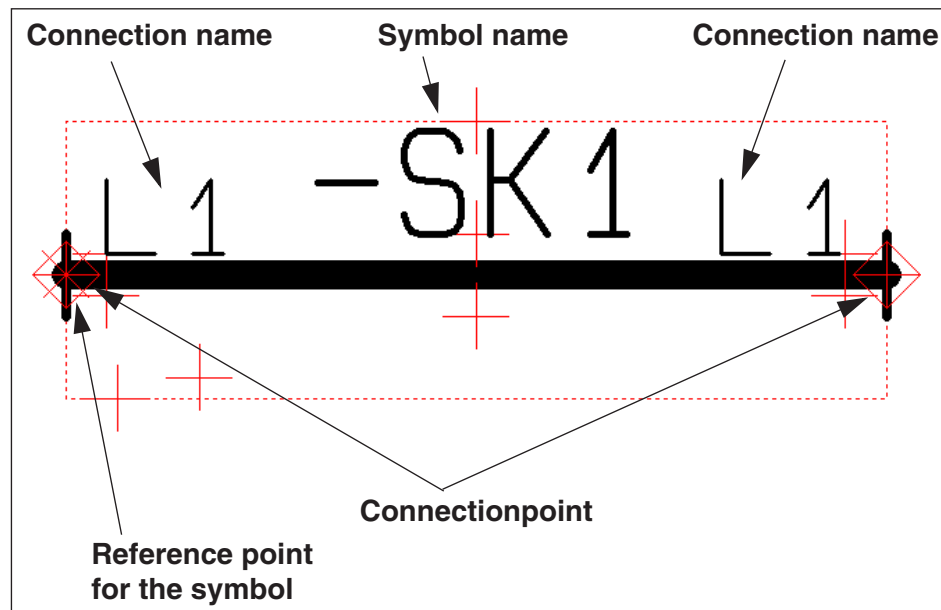
The connection name is applied for the name of the potential, which the busbar is connected to. Both connection points must be assigned the same connection name (*L1* in the figure above).

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 478, "Workflow for PLCs from the database" on page 481 and "PLC data on the I/O symbol connection points" on page 488.

Create the PLC I/O symbol with the type PLC

When you start creating a PLC I/O symbol and enter the dialog box Symbol settings, you must set the Symbol type to PLC. — See "Symbol types" on page 785 if necessary.

When you use the symbol in projects, you can then use all of the automatic functions for PLC symbols, and the symbol will be included in PLC lists.

PLC I/O symbols are designed as other symbols — see "Creating Symbols" on page 775.

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 805.

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 491.
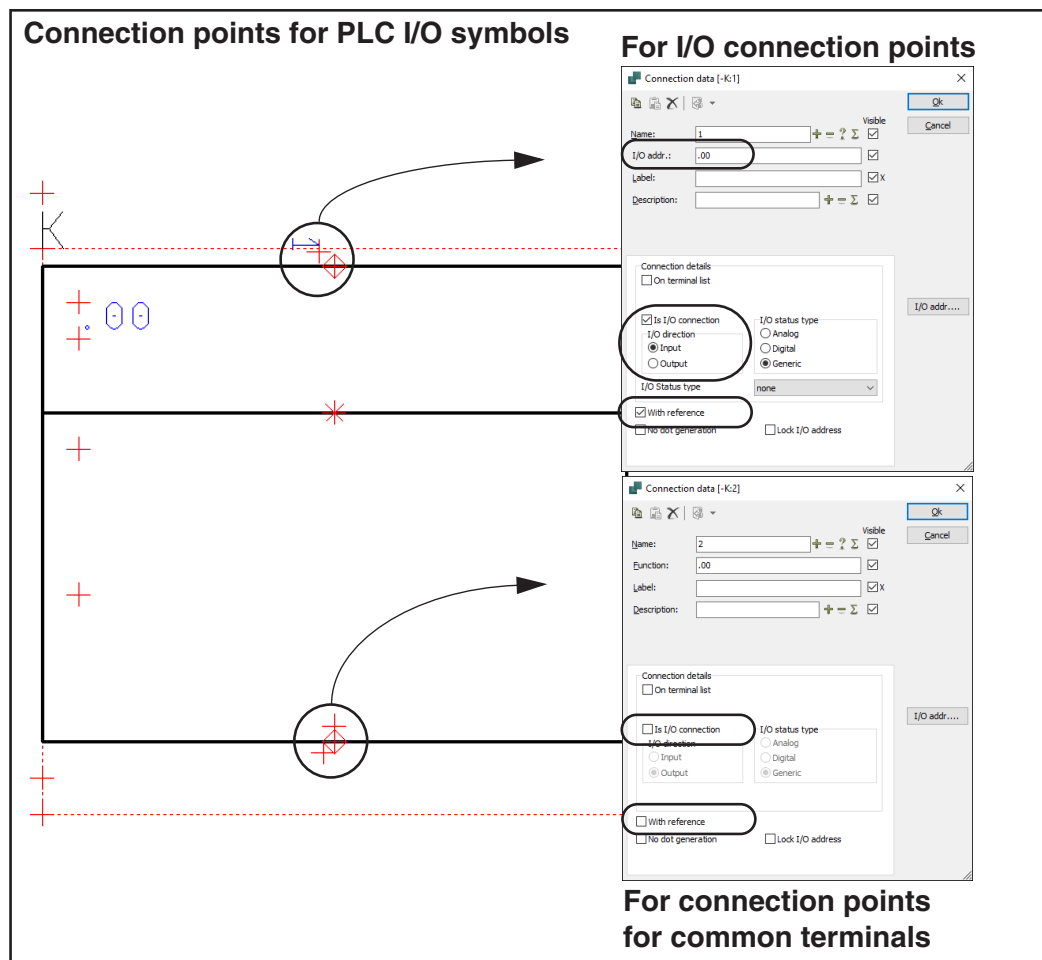
Connection points for common terminals: Do not select With reference

For connection points with common terminals (supply/ground) you shall *not* select **With reference**.

IO status for connection points on PLC I/O symbols

For input and output connection points, you select **Is I/O connection** and then specify whether it is **Input** or **Output**. You can further set the **I/O status type** to **Analogue**, **Digital** or **Generic**. When you choose **Generic** you can subsequently choose any of the options that you have defined under **PLC => Define PLC I/O-status types**.

— Please refer to the figure on page 804.

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 804.

Save the PLC I/O symbol with the symbol type PLC

When you save the PLC I/O symbol — see "Symbol types" on page 785 —, 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:
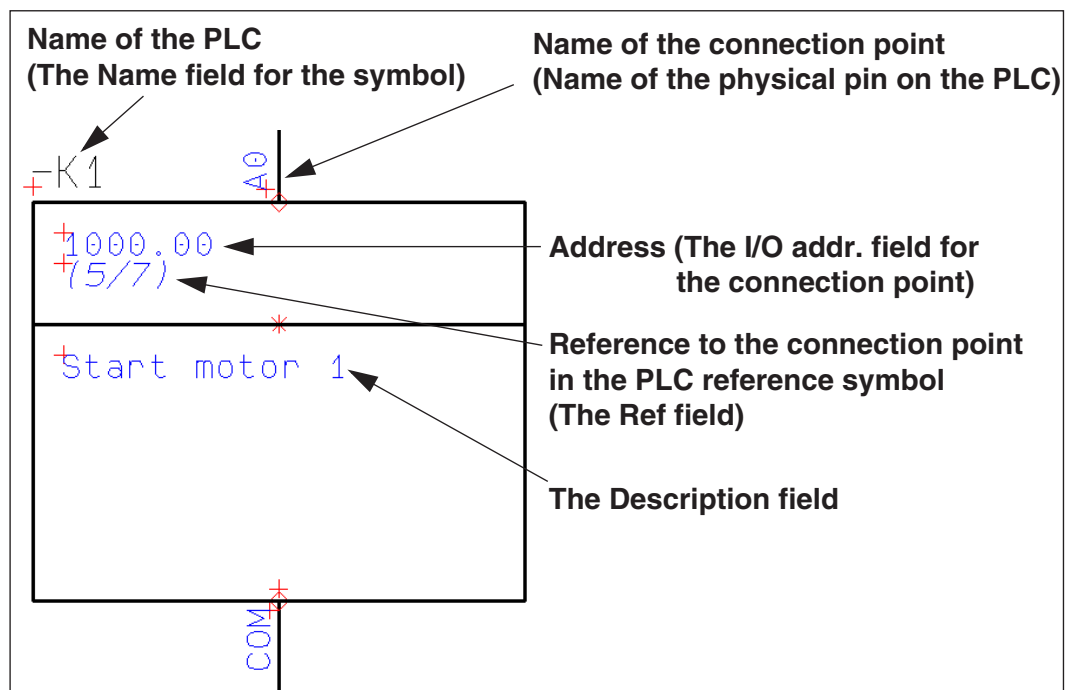
— See also "PLC data on the I/O symbol connection points" on page 488 and "Jumping between PLC ref. symbols and PLC I/O symbols" on page 491.

CREATING PLC REFERENCE SYMBOLS

— Read about PLC reference symbols and their use in "On PLC reference symbols" on page 478.

Create the PLC reference symbol with the symbol type PLC reference

When you start creating a PLC reference symbol and enter the dialog box Symbol settings, you must set the Symbol type to PLC reference. — See "Symbol types" on page 785 if necessary.

Connection points

Texts for the connection points
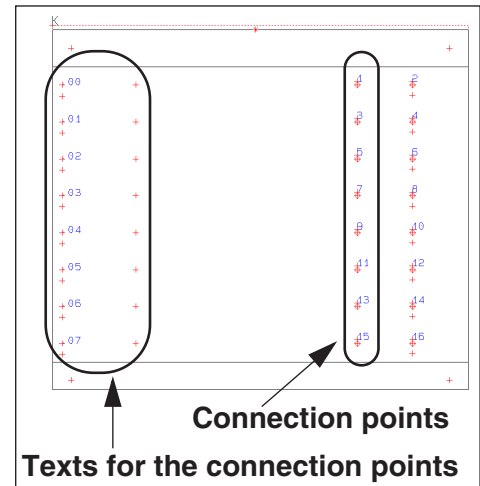
PLC reference symbols are designed as described in "Creating Symbols" on page 775.

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 803.

When you have placed the created PLC reference symbol in the project, and sub-

Select Reference

sequently 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 491.

You shall only choose *With reference* for the I/O connection points.
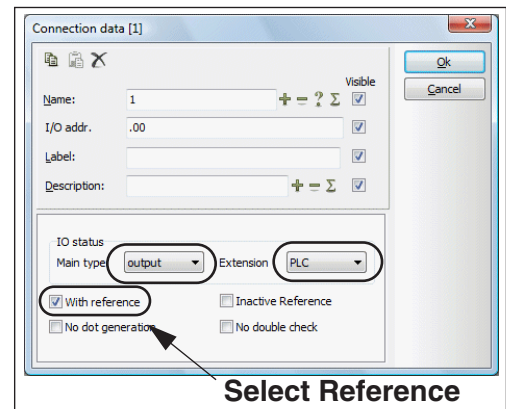
I/O status for PLC reference symbol I/O connection points

Select *Is I/O connection* and choose *Input* or *Output*, depending on whether the connection points is used as an input or an output.

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 488. In this example, the PLC reference symbol consists of lines with data from these connection points:
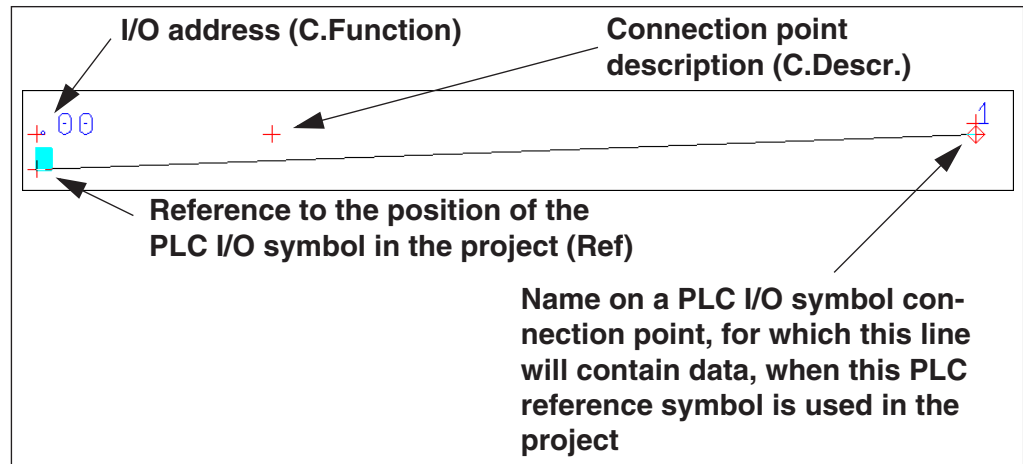
Save the PLC reference symbol with the type PLC reference

When saving the PLC reference symbol — see "Symbol types" on page 785 —, 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 330. See the **Projekt generator** manualen 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**.

Line connection symbol with the connection names 1, 2 and 3
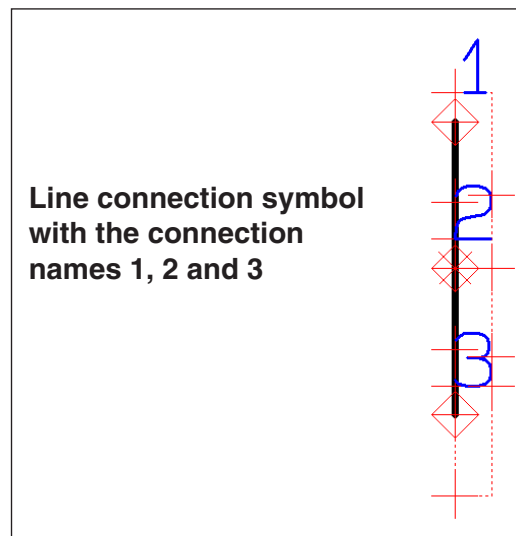
Please remember to save the symbol using the symbol type **Line connection**.

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.

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 741.

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.

<table>
<tr><td colspan="2" align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>— CREATING YOUR OWN DATA SYMBOL</td><td>(P. 810)</td></tr>
<tr><td>— DESIGNING THE DATA SYMBOL</td><td>(P. 811)</td></tr>
<tr><td>— PLACING THE DATA SYMBOL IN A PROJECT</td><td>(P. 812)</td></tr>
<tr><td>— SYMBOL DATAFIELDS WITH PREDEFINED VALUES</td><td>(P. 813)</td></tr>
<tr><td>— DATA SYMBOLS AND LISTS</td><td>(P. 814)</td></tr>
<tr><td>— DATA SYMBOLS AND COMPONENTS LISTS FILES</td><td>(P. 814)</td></tr>
</table>

Please refer to "Creating Symbols" on page 775 for general information about how to create symbols.

INSERT YOUR OWN SYMBOL DATAFIELDS IN THE SYMBOL DEFINITION

PLEASE NOTE THAT IT IS ALSO POSSIBLE TO INSERT YOUR OWN SYMBOL DATAFIELDS IN THE DEFINITION OF A SYMBOL, WHICH IN MOST CASES WILL BE THE BEST SOLUTION — SEE "EXTRA SYMBOL DATAFIELDS IN SYMBOLS" ON PAGE 790.

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 812.
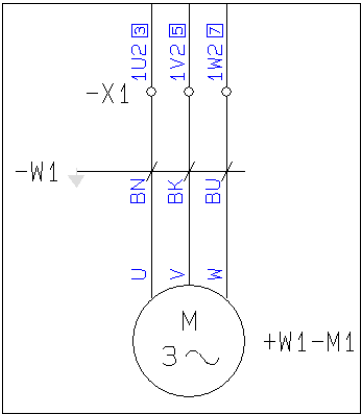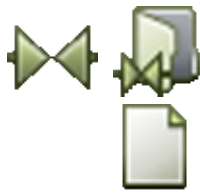
DESIGNING THE DATA SYMBOL

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 812, 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 data fields* in the left-hand side of the menu.
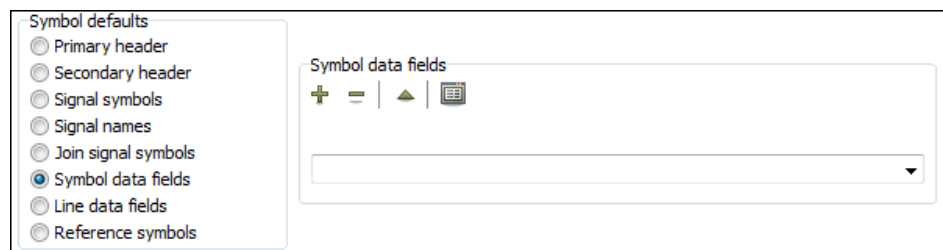
Here you click the **Add** button, type in *Windings* in the appearing menu, and click on **OK**. Click the **Add** button again, type in *kW:* and click on **OK**. You have now created the two symbol datafields ***Windings*** and ***kW***.

You can also create a list of predefined values for the datafields, as described in "Symbol datafields with predefined values" on page 813.

Click **OK** to exit the dialog box.

— Please note you do not have to be designing a symbol to create the symbol datafields.

LINE DATAFIELDS ARE CREATED THE SAME WAY

WHEN YOU WISH TO CREATE LINE DATAFIELDS THIS IS DONE THE SAME WAY. IN STEAD OF *SYMBOL DATA FIELDS*, YOU JUST CHOOSE *LINE DATA FIELDS*.

Placing the symbol datafields in the data symbol

To place the symbol datafields in the data symbol, choose **Insert => Insert data-field**.
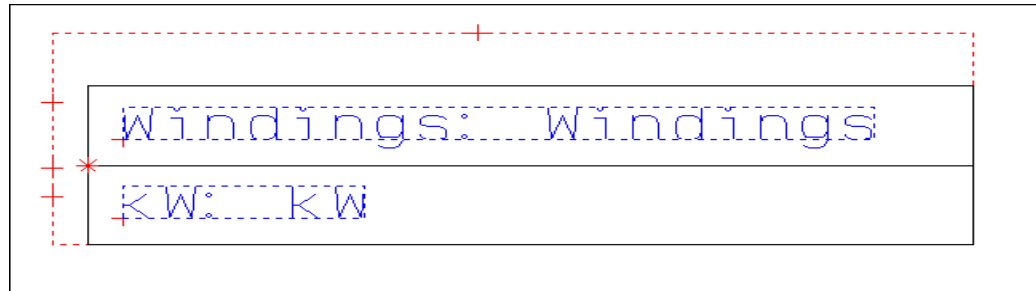
In the **Datafield** dialog box you click on *Symbol Datafield*, choose *Windings* and click on the red down-arrow in the lower part of the menu. Place a colon (:) and a space character after the transferred text in the *Text* field, and set the *Width* to *20*.

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 136.

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:
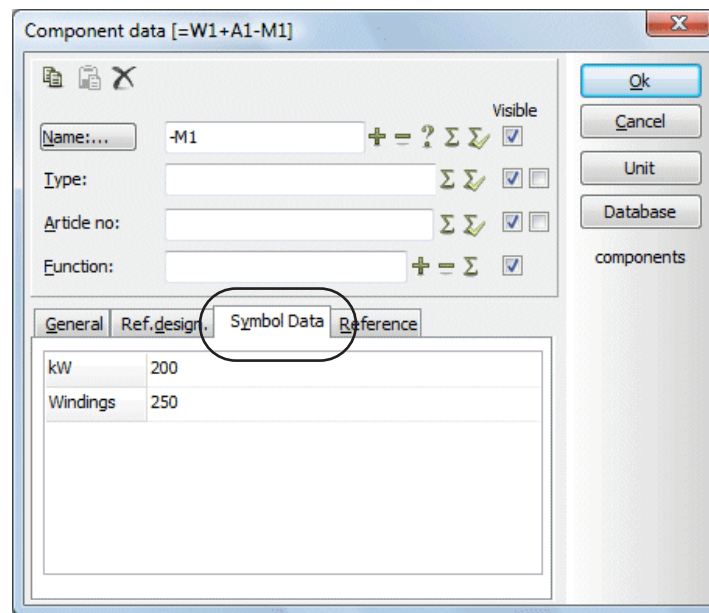
By assigning the name **+W1-M1** to the data symbol, you tell the program that the information in this data symbol, is attached to the symbol with the name **+W1-M1**.

Fill out the data symbol fields, click on **OK**, and place the symbol.
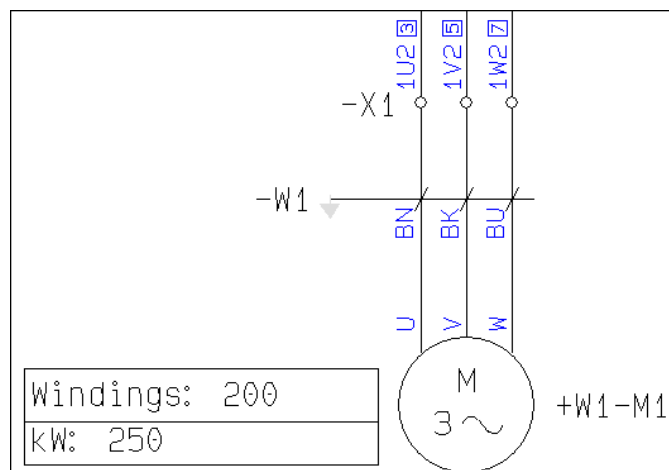
If the data symbol does not have an appropriate size compared to the motor symbol, you can scale it using the symbol scale field in the symbol tool bar. Read more about this in "Scaling symbols" on page 201.

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 data-field, and click on **Values**.

Choose symbol datafield

... and click here to type in the values
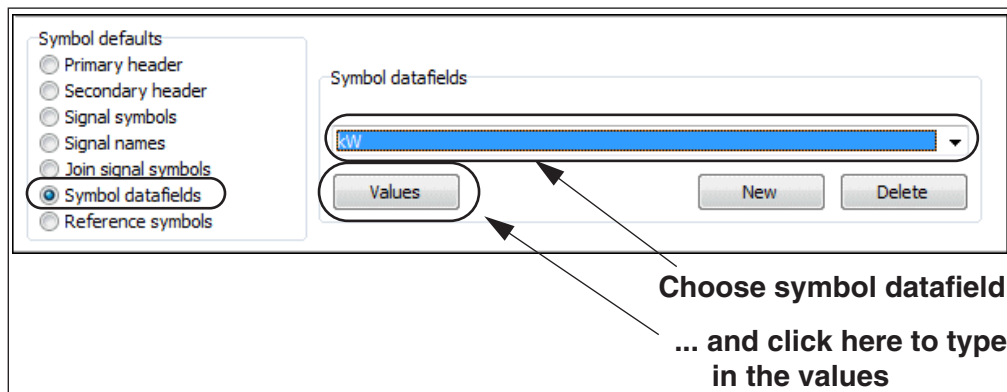
You then enter a dialog box, where you type in the values, and click **OK**.

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 symboldata values is displayed (3).

1) Click in the field

2) Click here, and the list is displayed

3) Choose the desired value
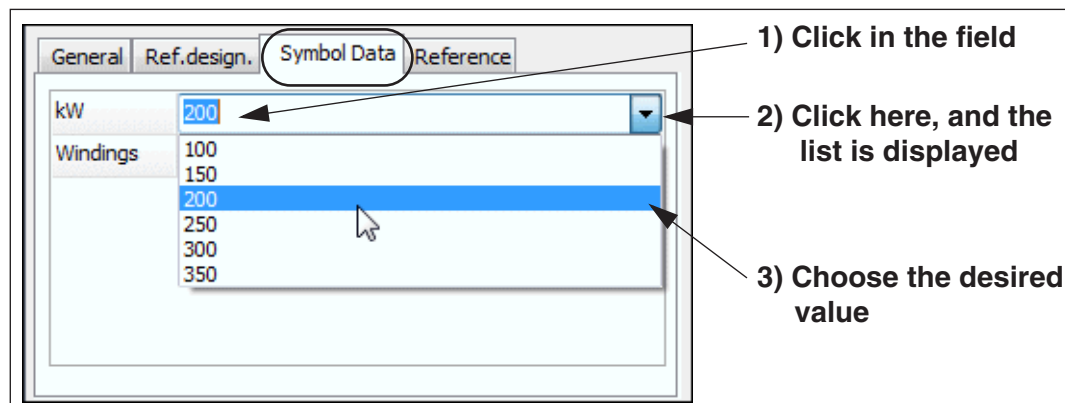
SYMBOL DATAFIELDS AND LISTS

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 831.

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 609.

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 729, particularly "Specifying the contents of the file" on page 735.

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

Read more about signal symbols and how to use them in your projects in "Signal symbols" on page 234.

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 775, 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.
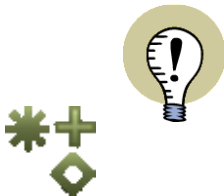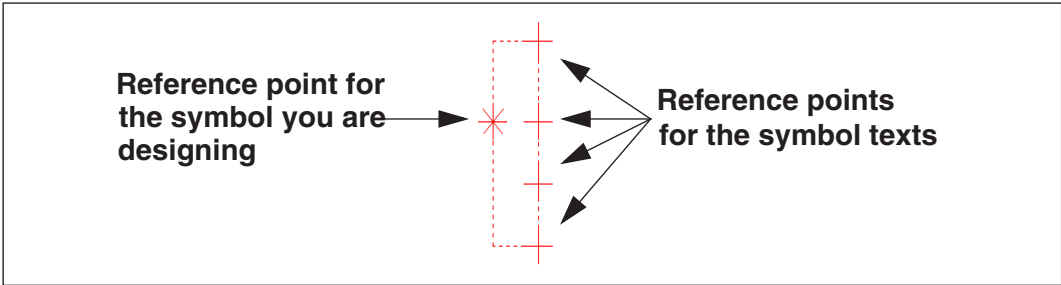
Reference point for the symbol you are designing

Reference points for the symbol texts

"VIEW REFERENCE POINTS" MUST BE ENABLED

IF YOU CAN NOT SEE THE REFERENCE POINTS WHEN YOU ZOOM IN ON AN AREA, THIS IS BECAUSE **VIEW REFERENCE POINTS** HAS NOT BEEN ENABLED. ENABLE IT BY CLICKING THE **REFERENCE POINTS** BUTTON IN THE LEFT TOOLBAR, OR BY SELECTING **VIEW => REFERENCE POINTS**.

— GET MORE INFO ABOUT THIS IN "SETTINGS" ON PAGE 907.

DESIGNING THE SIGNAL SYMBOL

You can now design a signal symbol with the following appearance:
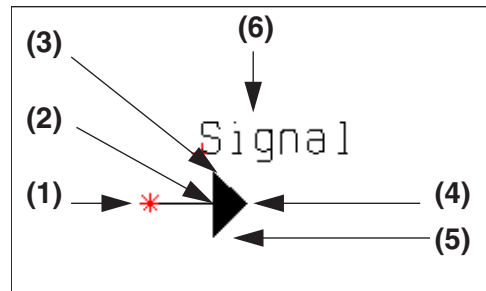
A right-arrow on a total of 7.5mm.

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 918).

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 819.

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 789. 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.
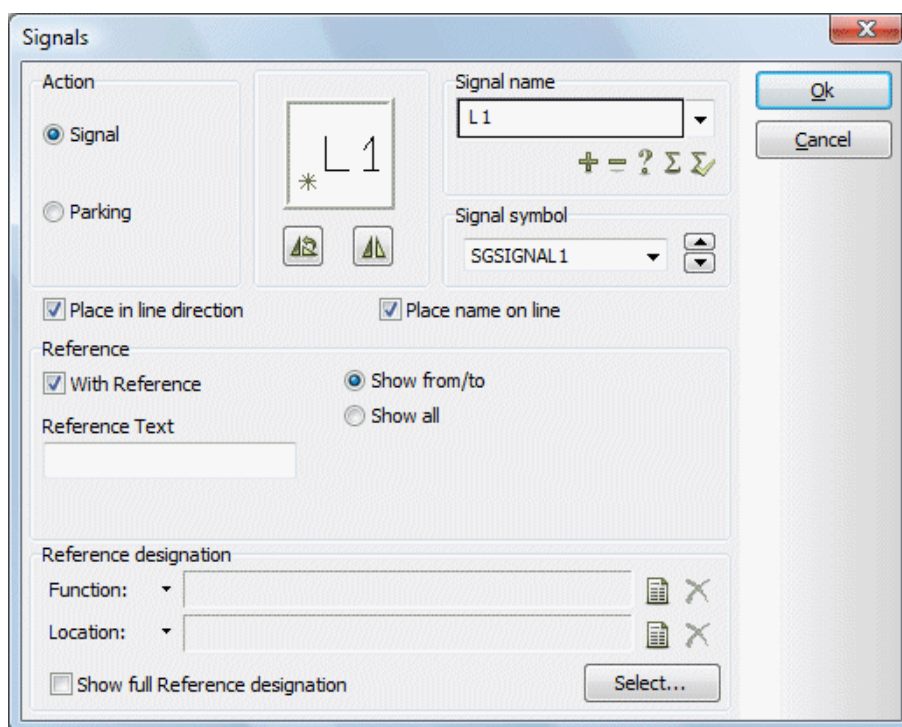
To add signal symbols to this list, you choose **Settings => Text/Symbol defaults**.
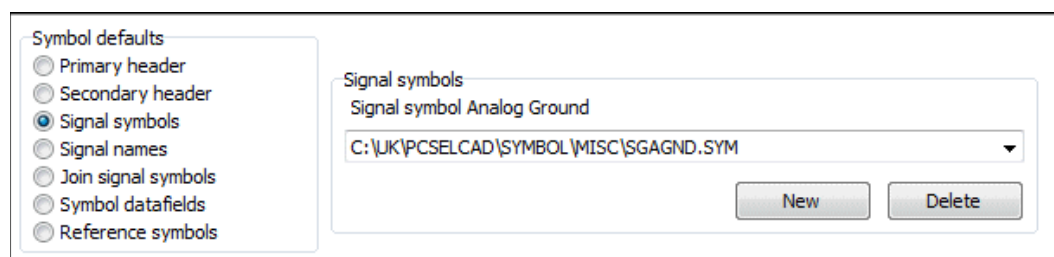
Here you click on *Signal symbols* in the lower-left corner of the dialog box, and click on **New**.

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 800.
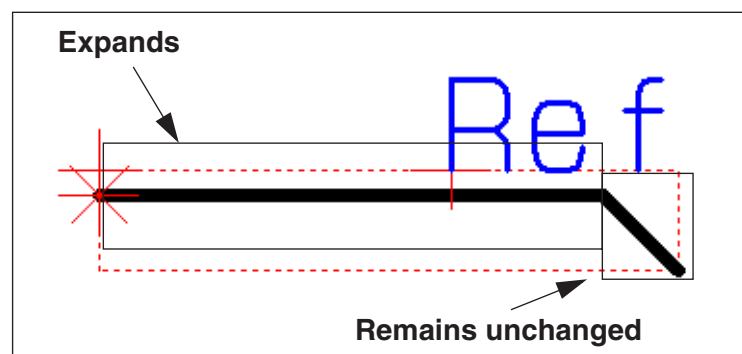
In join symbols you have to place the reference point to the left, and draw the symbol to the right.

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.

Please refer to "Changing an already created list/drawing header" on page 843 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 599.

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 850, PARTICULARLY "CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR" ON PAGE 851.

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.
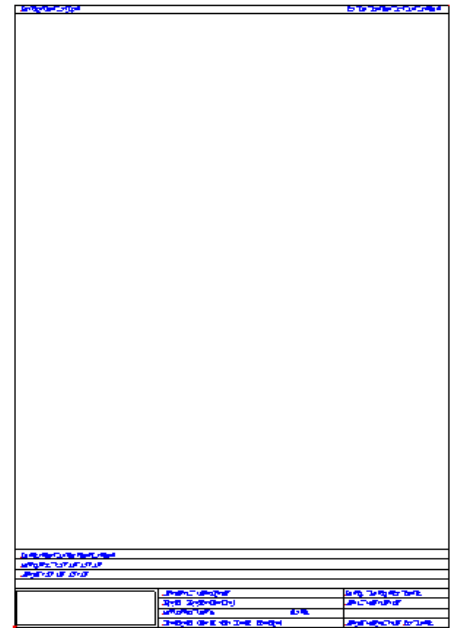
INITIAL PREPARATIONS BEFORE CREATING A DRAWING HEADER

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 915, and about *grid* and *snap* in "Screen / Image Func-tions" on page 42.
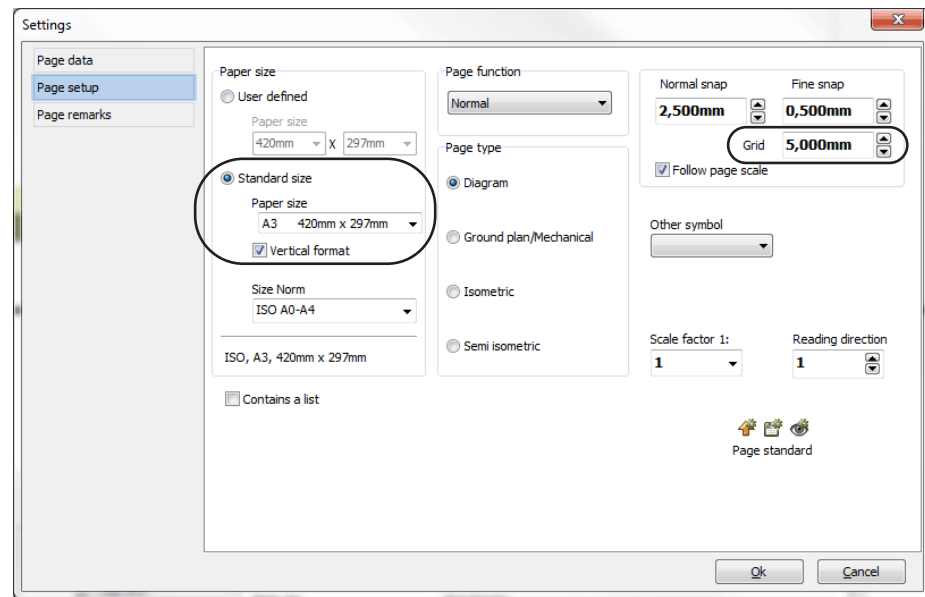
3) Click **OK**.

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 you 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 to 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 appro-priate location.
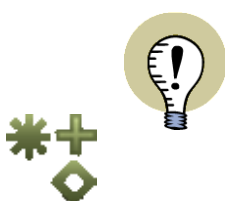
"View reference points" must be enabled

If you can not see the reference points when you zoom in on an area, this is because **View reference points** has not been enabled. Enable it by clicking the **Reference points** button in the left toolbar, or by selecting **View => Reference points**.

— Get more info about this in "Settings" on page 907.

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 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.
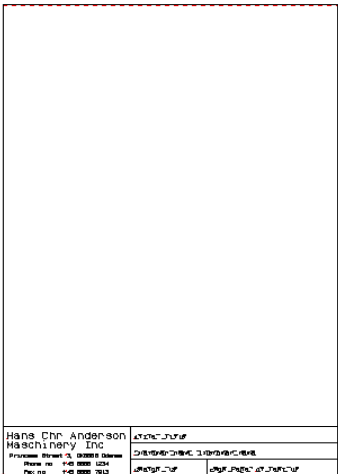
DESIGNING THE DRAWING HEADER

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	Customer name Customer name	
Phone no.: +45 6600 1234		
Fax no.: +45 6600 7913	Design line	Page Page of Last no

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 hori-

zontal 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 "SYMBOL DATAFIELDS" ON PAGE 122.
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 data and Page data" on page 128.

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 769.

— For a complete survey of the datafields in **Automation**, please refer to "Datafields in Automation" on page 944.

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:
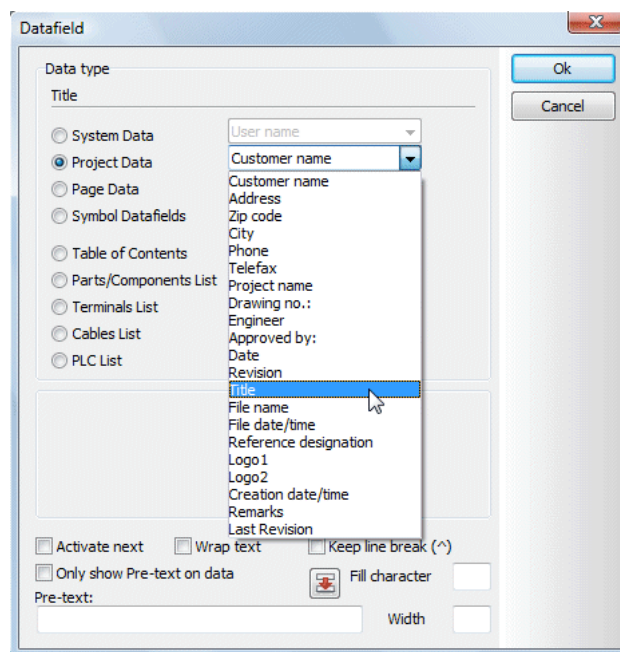
See an elaboration of the functions in this dialog box in "Datafields" on page 118.

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 filed. 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.

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 136). Click on **OK** and place the text.

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

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 111.

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 841.

Saving the symbol / the drawing header
Choose **File => Save as**.

Fill out the dialog box as you see below.
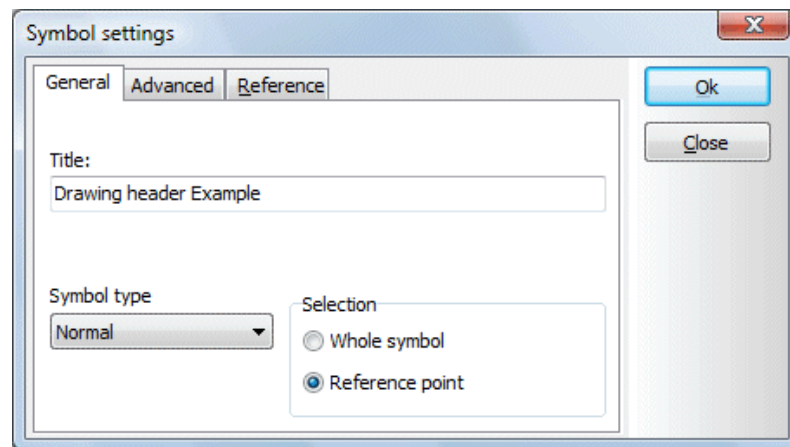
Please note, that you can also enter this dialog box while designing the drawing header, if you click on the **Symbols** button, and then on the **Symbol settings** button.

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**.
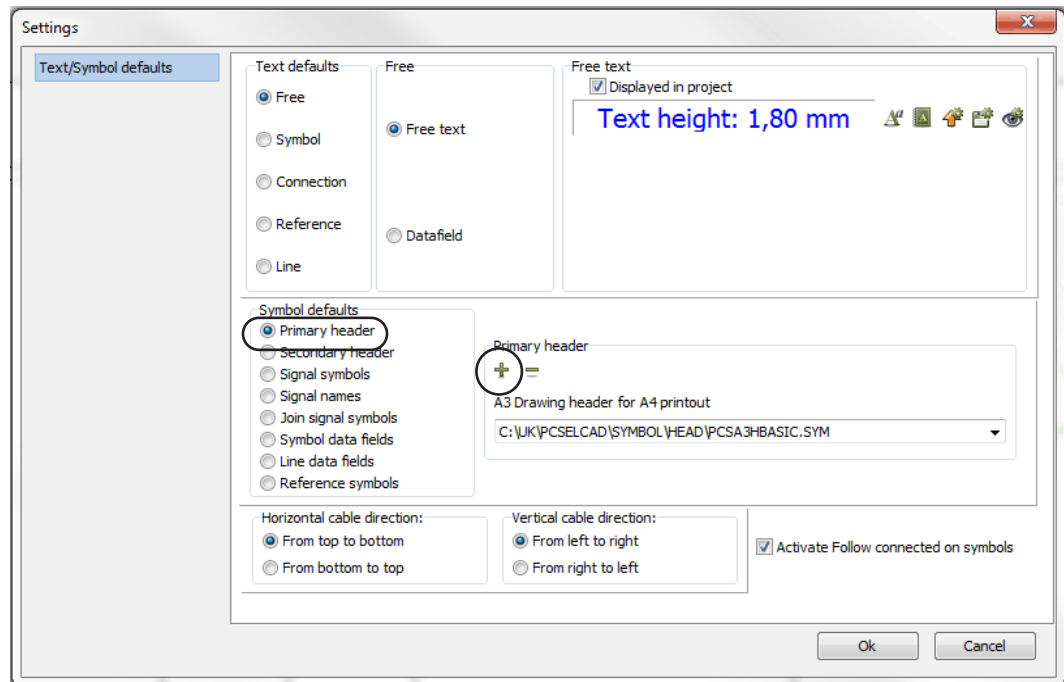
Click on *Primary header* in the left side of the dialog box.

Then click the **Add** button, 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 599.

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 the **Remove** button, 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.

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 791.

— See how to select states when placing drawing headers in "Drawing headers with States" on page 603.

DRAWING HEADERS IN MULTIPLE LANGUAGES

You can use states in drawing headers for controlling which language is used in the drawing headers.

How it works

1) Create a drawing header, which contains states with attached language codes. The first letters in the title for the state must contain the language code — for instance UK for English. To make this work, it requires that you, for the relevant states, have translated the texts to the desired language(s).
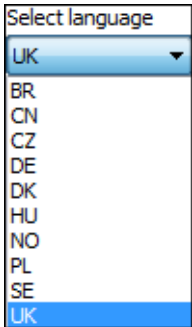
2) Place the drawing header on a page in the project: The program now automatically activates the first state, which is created for the language, that you have chosen to use in the program. If for instance you work in English, the first state with the English language code is selected.
(You choose language under **Settings => System**, where you can also see the language codes — see "System" on page 930, particularly "Choose language" on page 931).

3) If you wish to change other drawing headers to the same state, you can do this by clicking the **Transfer settings** button to the right of the *State* field — see "Drawing headers with States" on page 603. This for instance makes it possible to change language in all drawing headers in the project.
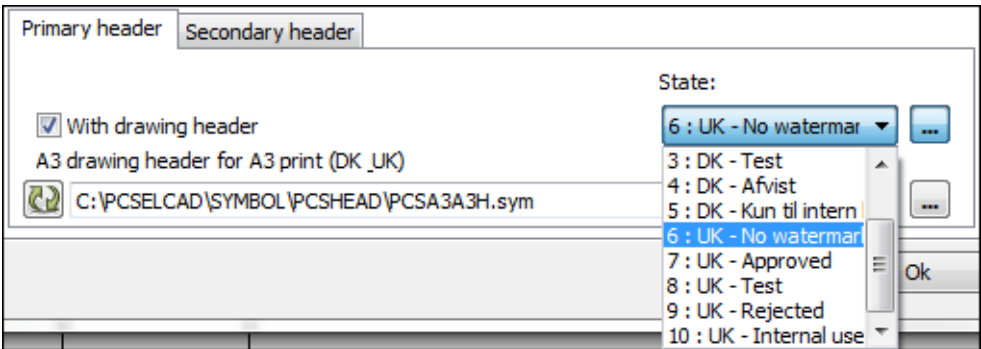
Examples on the application

The drawing headers that are included in the program (see "Predefined drawing headers" on page 830 and "Predefined lists" on page 843) are created so that you can choose whether to use English or Danish texts. You change language by changing state for the drawing header.

PREDEFINED DRAWING HEADERS

To make things a little easier, some predefined drawing headers already exist. They are located in the folder **c:\Pcselcad\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 843.

Drawing headers in this folder are created so that you can choose whether they should use English or Danish texts. You switch between the languages by choosing state for the drawing header. — See "Drawing headers with States" on page 603.

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**.

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 595.

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 769.

CREATING LISTS

In this chapter you can see how to create your own project lists.

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 729 for further details.

In "Insert drawing headers/formulas" on page 599 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 850, PARTICULARLY "CREATING PAGE TEMPLATES — VIA THE TEMPLATE EDITOR" ON PAGE 851.

What is a List?

In your projects you may need to create various types of lists, such as parts lists, components lists, terminals lists, cable lists, PLC lists or tables of contents.

This chapter is a thorough example on how to create a parts list, followed by brief descriptions of how to create the other types of lists.

The procedure how to create lists, are basically the same for all types of lists; the only difference is which datafields you insert on the various types.

Basically it is all about placing text fields and datafields, which collects information from the other project pages, onto a drawing header page. You determine how to place the texts and datafields, and you can also draw lines and place a company logo on the page.

When you have created a list and inserted it in your project, you can make the program fill out the list with project information automatically.

This way you can for instance create a table of contents, with information about which drawings and lists are placed on the project pages, as well as a parts list containing information about all the components you need to carry out the project in praxis.

Lists/Drawing headers

It is important to note, that in **Automation**, lists are conceived as drawing headers. As a consequence of this, you create lists the same way as you create drawing headers. Also, the functions in the program which can be used for drawing headers, can be used for lists as well.

However, when working with lists, you have to *update* the lists to have them filled out with project information. This will be elaborated later.

If a filled out list needs more than just one project page, the program will automatically insert the required extra amount of pages.

You are now ready to start creating a parts list.

How to create a List

The following is an example on how the create a parts list.

Initial preparations

Because a list is also a drawing header, all initial preparations are the same for lists as for drawing headers.

Now perform the initial preparations as described in "Initial preparations before creating a drawing header" on page 821.

You Are now ready to design the parts list.

The upper part of the list you are about to design in this example looks like this:

SELECT DATAFIELDS AND PLACE TEXTS AS YOU WISH

YOU ARE FREE TO PLACE THE TEXTS DIFFERENTLY, TO ASSIGN OTHER NAMES TO THE COLUMNS OR TO USE OTHER DATAFIELDS THAN THE ONES CHOSEN HERE.

Column headings

You Are now ready to make the column headings for the parts list. The texts you shall place in this example are:

- ***Article number***
- ***Function***
- ***Quantity***

Click on the **Texts** button, and then on the **Text properties** button.

The **Text properties** menu appear. Here you set ***Height*** to *3.0*, ***Width*** to *AUTO*, and ***Alignment*** to *left-down*. If you are not sure how to do this, please read "Specifying text properties / Appearance of text" on page 136.

Click on **OK**. You have now determined the appearance of the text, and have returned to the page.

Click inside the field to the right of the Text properties button, type in *Article number* and press <Enter>.

PC|SCHEMATIC

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 641.

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 835.

(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 118.

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.
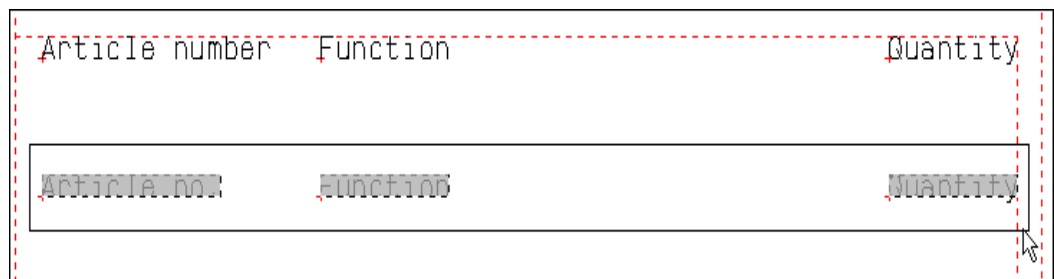
Then choose **Edit => Align**, and click on the reference point for the left-most of the texts. — Read more about this function in "The Align function" on page 66.

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

included in the parts list. The following datafields — such as **Function** and **Quantity** — will then contain the function and quantity of this first component.

The next time the program meets **Activate next**, it collects information about the next component on the project drawings. The datafields **Description** and **Amount** now contains information about the second component in the project — and so forth.

DATAFIELDS FOLLOWING ACTIVATE NEXT MUST BE ALIGNED

IT IS IMPORTANT TO NOTE, THAT IF YOU PLACE A DATAFIELD JUST A BIT HIGHER THAN THE DATAFIELD IN WHICH YOU HAD SELECTED **ACTIVATE NEXT**, THIS DATAFIELD WILL COLLECT ITS INFORMATION FROM THE COMPONENT ABOVE.

THEREFORE IT IS RECOMMENDED TO LET DATAFIELDS, WHICH FOLLOWS **ACTIVATE NEXT**, BE ALIGNED BEFORE YOU START COPYING.

Creating a fancier parts list

Now the parts list is completed, and ready to be saved, but you can still choose to do a few things to make it look better:

a) You can draw lines on the parts list — see "Lines" on page 72.

b) You can insert the datafields and lines, which were inserted in "Creating Drawing Headers" on page 820. In that case follow the directions given in "Designing the drawing header" on page 823.

c) You can insert a heading for the entire parts list. (The datafield on the figure is **Page Data**, **Title**).

d) Place a company logo on the page — see "Placing a Company Logo on a drawing header" on page 830.

Saving the parts list

To save the parts list, you do like this:

Choose **File => Save as**.

You now enter the dialog box **Symbol settings**: Type in a **Title** for the symbol, set the symbol type to **Normal** and set the selection to **Reference point**.

The symbol **Title** is the description of the symbol, which will be displayed in the **Symbol menu**. Click **OK**.

You now enter the dialog box **Symbol settings**: Type in a file name — such as **Testlist.sym** — and select which folder to save the symbol in.

You can for instance save the list in the folder **c:\Pcselcad\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 828.

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 599 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**.

See which datafields you can use in parts lists in "Parts/components list datafields" on page 949.

CREATING A TABLE OF CONTENTS

When you serrate a table of contents, you follow exactly the same guidelines as for designing a parts list.

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 599 — 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 843.

See which datafields you can use in tables of contents in "Table of Contents datafields" on page 948.

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.

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.

See which datafields you can use in components lists in "Parts/components list datafields" on page 949.

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 118.

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 245.

See which datafields you can use in terminals lists in "Terminals list datafields" on page 951.

See how to specify which connection points to include in project lists in "Symbol Connection points" on page 212, specifically "Appearance of connection points on symbols (terminals lists)" 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**	
Indentification	Page/path	Indentification	Page/path	Indentification	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.

If you want cable lengths to appear in the cables list, please see "Cable direction and Default cable quantity" on page 936 and "The Quantity field in the dialog box Component data" on page 179.

See which datafields you can use in cables lists in "Cable list datafields" on page 954.

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.

See which datafields you can use in PLC lists in "PLC list datafields" on page 955.

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.

The two options

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*.
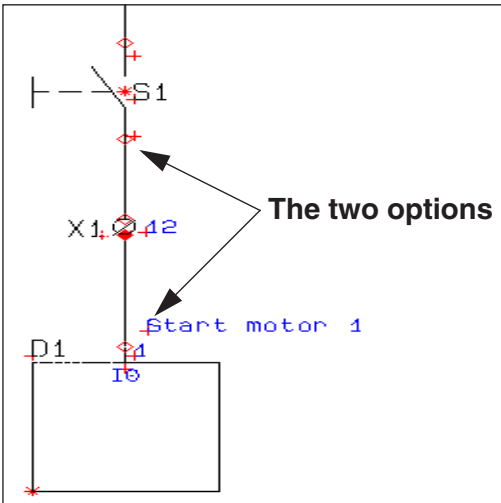
Specifying The PLC connected component

When you have selected a symbol on a project page and enter the **Component data** dialog box, you can specify that the selected symbol shall be understood as the component connected to the PLC, by selecting *The PLC connected component*.

CREATING A CONNECTIONS LIST

You can also create a connections list containing an overview over the connections in the project. It is basically the same as creating a connections list file, which is described in "Connections list to file" on page 739.

Set the page function to **Connections list**, insert datafields of the type **Connections list**, and create the list as described above.

See which datafields you can use in connections lists in "Connections list datafields" on page 958.

DIAGRAM PAGES - CONTAINING A PARTS OR COMPONENTS LIST

It is possible to create a drawing header for normal diagram pages or ground plan/mechanical 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 to short.
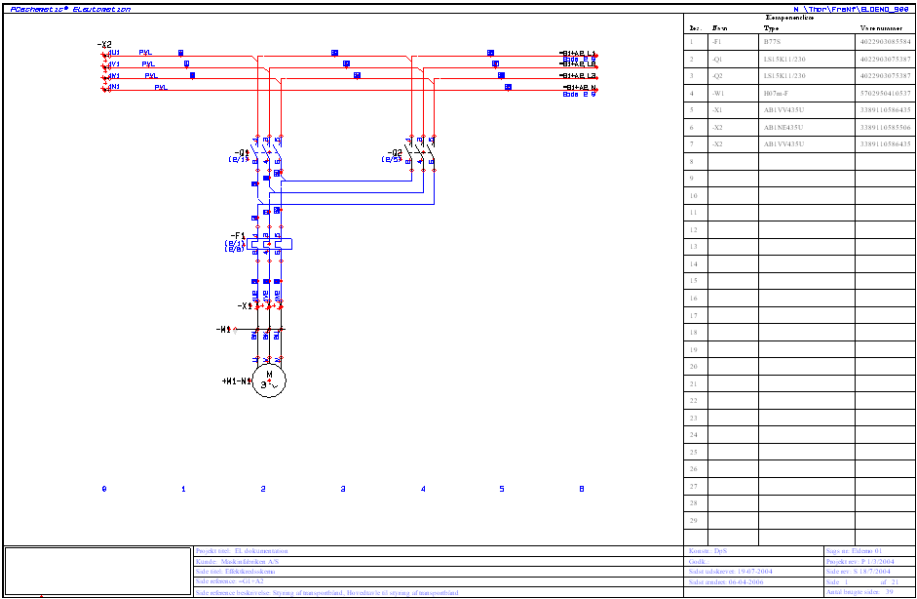
To create such a drawing header, you must do as described in "How to create a List" on page 833 — with the only difference that you make room for diagram drawings in the drawing header.

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 succeedingly choose **Settings => Page setup**, and select **Contains a list**, and then specify whether it is a **Components List** or a **Parts List**.
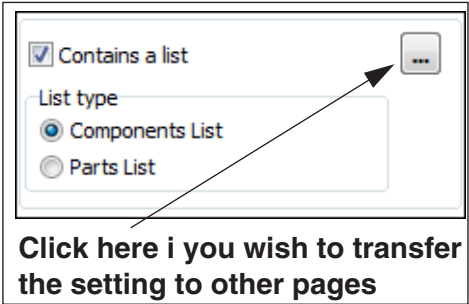
Click here i you wish to transfer the setting to other pages

If you want the list to contain both vertical and horizontal references to the components on the page, do this:

1) Set the reference system for the page to vertical and horizontal references — see "Horizontal and vertical references" on page 446

2) Set the reference format for symbols to **XY reference** — as described in "Appearance of references" on page 438

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 609. 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 841.

When you create a symbol of this kind, datafields e.t.c. must be placed as described in "How to create a List" on page 833. When saving the symbol, you must set the **Symbol type** to **Head** — see "Saving the parts list" on page 836.
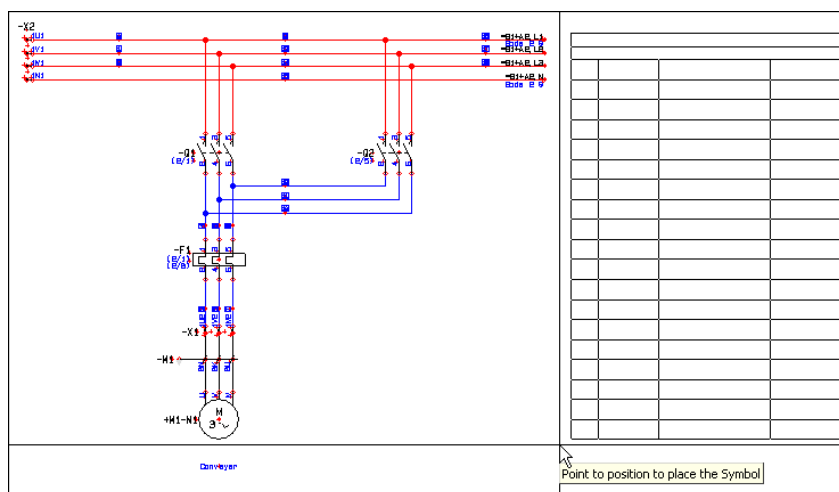
When you have created the symbol, you can place it on diagram pages, just like you normally place symbols. Then specify that the page contains a parts or components list, as described in "Specify that the diagram page contains a list" on page 841. The symbol will then function as a list for the page.

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 842 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:\Pcselcad\symbol\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 836.

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 <Del>. 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 **PCSCHE-MATIC 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.

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 136.

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.

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.

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 595.

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 form 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 560.

Use states in drawing headers / lists to change language

You can use states in drawing headers / lists to change language for the applied texts.

— See "Drawing headers in multiple languages" on page 829.

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. Unit drawings / parts lists can also be attached to symbols and lines in the project.

When you use an extended parts list (please refer to "Specifying from where to collect components" on page 611), 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.

<table><tr><td>THIS CHAPTER DESCRIBES</td></tr></table>

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 831. 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 821. — 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 data and Page data" on page 128.
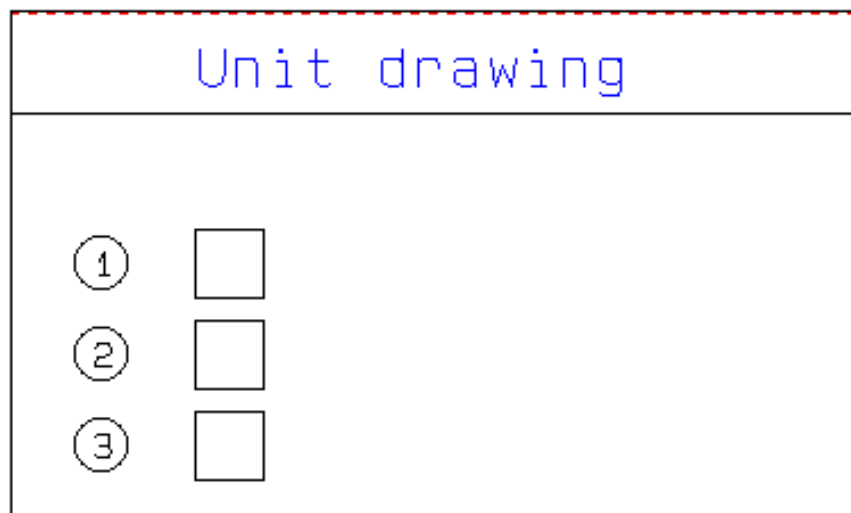
Now place the symbols (position numbers), in which you in the **Component data** specify the information, which is necessary for the parts on the unit drawing.

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 918.

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 are asked to specify **Page data** (for instance type in the name *Test*), and then 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.

INSERTING LINKS TO COMPONENTS IN THE UNIT DRAWING

On the unit drawing itself, you can also insert a list of the components in the project that applies the unit drawing:

1) Choose **Insert => Datafield**

2) In the dialog box **Datafield**: Click on ***Parts/Components list***, choose the datafield ***Unit name list*** — type in a pre text if necessary — and click **OK**.

When you use the unit drawing in a project, you can then double-click on the displayed component names in the unit drawing, you can then jump directly to the component in the project.

— Read more about datafields in "Datafields" on page 118.

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 775 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 848.

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 744 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 638 to see how to insert a unit drawing in a project.

CREATING TEMPLATES

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 852.

To create a template for a diagram page, do the following:

1) Choose **File => New** or click the **New document** button, and you enter the dialog box **New**:

2a) Click on the tab Normal Pages   **2b) Click on Template**

2c) Click on Blank Page   2d) Click OK
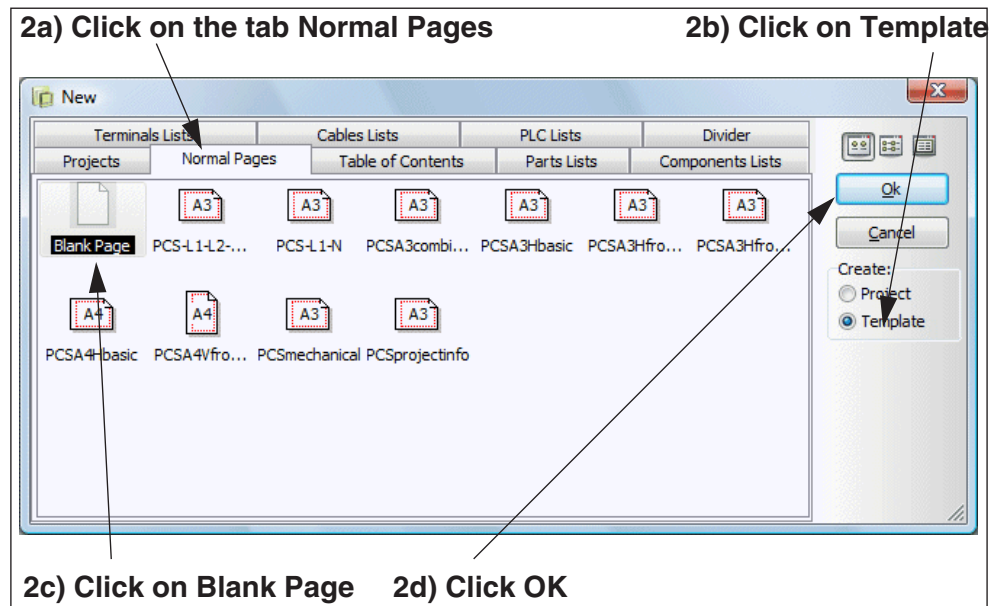
2) Click on the tab **Normal Pages**, click on *Template* in the right-hand side — to create a new template page —, click for instance on *Blank Page*, and click **OK**.

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.
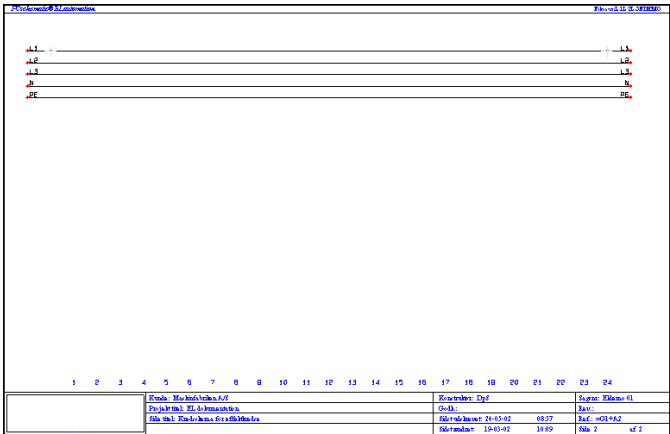
You can also place existing sub drawings on the page, as described in "Sub drawings" on page 299.

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 299.
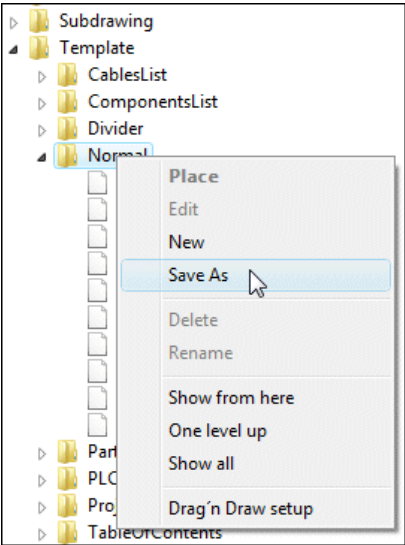
CREATING PROJECT TEMPLATES

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 851.

— 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:
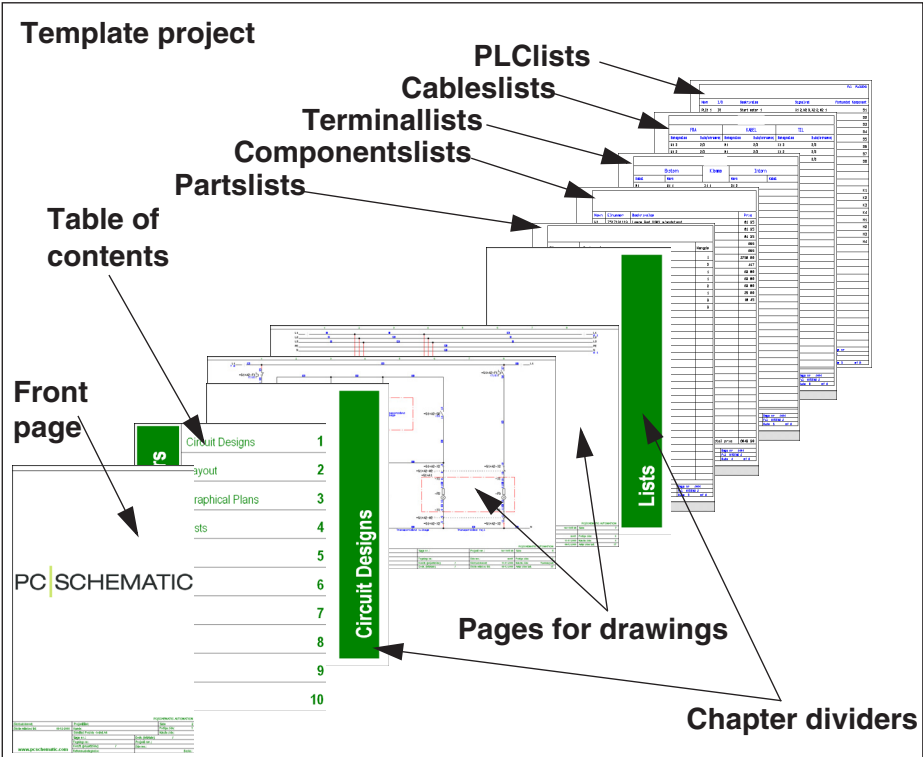
When creating project templates, you are free to insert the already created page templates in the project template.

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 855 to see how to create a template based on an existing project.

— Please refer to "Converting Standard projects to Project templates" on page 855 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 852
— 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:

**Here you find
the templates**

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 **Pcselcad\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>.

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 856.

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 subsequent versions.

To do so, you must follow the procedure described in "Creating a template based on an existing project" on page 855.

EDITING TEMPLATES

1) Choose **File => New** or click the **New document** button.

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.
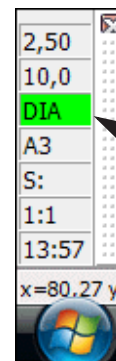
Blinks green in the Template editor

DELETING A TEMPLATE

1) Choose **File => New** or click the **New document** button.

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.
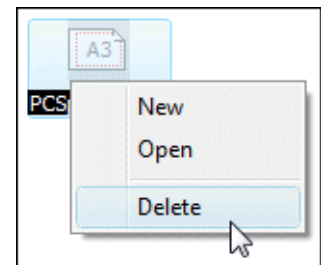
TEMPLATE FOR CLICKING ON THE NEW DOCUMENT BUTTON

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 **Pcselcad\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.
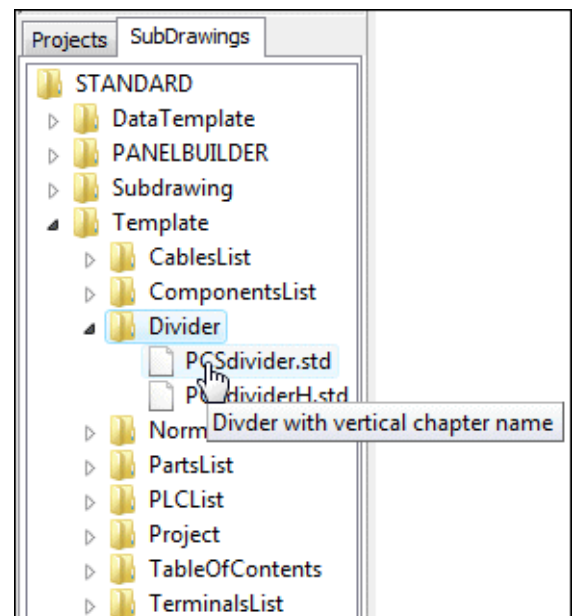
UPDATING, MODULES AND LICENSES

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:

PC | SCHEMATIC

Modules and scripts

This chapter gives you an overview over which modules you can install and uninstall in **Automation**, and gives a short description of what it means to apply scripts.

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 863.

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.

Tools	Help	
	Rename symbols	
	Database	
	Symbol Documentation	
	Graphical Terminal Plan	
	Graphical Cable Plan	
	Graphical Connection Plan	
	Edit symbols datafields	
	Convert files from ver.11 or older	
	List Compare	
	Text translator	
	Generate project from selected options	
	Export/Import ClipProject	
	Eksport/Import RailDesigner	
	Singleline Diagram	
	Net Navigator	
	Panelrouter	Shift+R
	DWG/DXF Map settings	
	Create variable list in (Sub)drawing	
	Project generator	
	Mouse Chasing System	
	Panel Builder	
	Scripts	

TOOL	DESCRIPTION
RENAME SYMBOLS	DESCRIBED IN "RENAME SELECTED SYMBOLS" ON PAGE 332.
DATABASE	DESCRIBED IN THE **PCSCHEMATIC DATABASE** MANUAL.
SYMBOL DOCUMENTATION	FOR AUTOMATIC GENERATION OF SYMBOL DOCUMENTATION. DESCRIBED IN THE **PCSCHEMATIC TOOLS** MANUAL.
GRAPHICAL TERMINAL PLAN	FOR AUTOMATIC GENERATION OF GRAPHICAL TERMINAL PLANS. DESCRIBED IN THE **PCSCHEMATIC TOOLS** MANUAL.
GRAPHICAL CABLE PLAN	FOR AUTOMATIC GENERATION OF GRAPHICAL CABLE PLANS. DESCRIBED IN THE **PCSCHEMATIC TOOLS** MANUAL.

Tool	Description
Graphical connection plan	For automatic generation of graphical connection plans. Described in the **PCSCHEMATIC Graphical connection plans** manual.
Edit symbol datafields	Module which gives you the option of exporting data for all component names with attached symbol datafields in the active project to Excel. The symbol datafields can then be edited in Excel and can subsequently be imported back into the project.
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.
List Compare	Module that can compare parts or components lists from two different projects.
Text translator	For translating texts in projects. Described in the **PCSCHEMATIC Tools** manual.
Generate project from selected options	Described in "Projects with selectable options" on page 592.
Export/Import Clip-Project	This module can be downloaded at www.pcschematic.com/EN/download-menu/other/download-other.htm. Download a tutorial at www.pcschematic.com/manuals.
Export/Import RailDesigner	This module can be downloaded at www.pcschematic.com/EN/download-menu/other/download-other.htm.
Singleline diagram	For automatic generation of single line diagrams.
Net Navigator	See "Net Navigator" on page 232.
Panelrouter	The Panelrouter can automatically draw the connections between the connection points on the mechanical symbols on mechanical pages, based on the information specified on the diagram pages. The **Panelrouter** is described in a separate manual, which you can download at www.pcschematic.com/manuals.
DWG/DXF Map settings	For setting up parameters for DWG/DXF import/export. Described in the **PCSCHEMATIC Tools** manual.
Create variable list in (Sub)drawing	Described in "Sub drawings" on page 299.
Project generator	Described in the **Project generator** manual.
Mouse Chasing System	For reducing repetitive strain injuries — see "Preventing Repetitive Strain Injuries" on page 883.
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	See "Scripts" on page 865.

HOW TO INSTALL AND UNINSTALL

To install or uninstall modules choose **File => Modules**, and you enter the **Modules** dialog box:
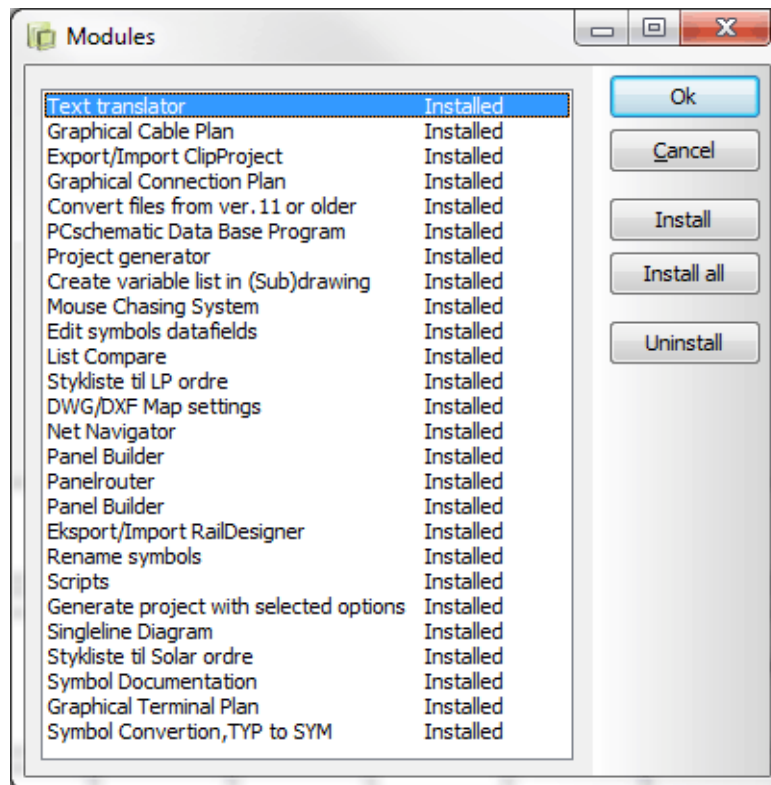
Here you get a survey of which modules can be installed, as well as which modules are installed in your case.

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.
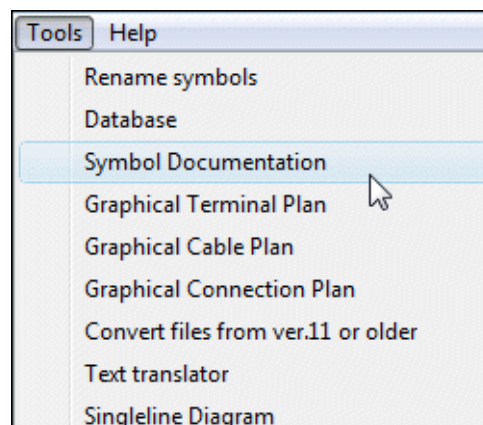
SCRIPTS

You can create your own scripts from the build-in interface for COM/OLE Automation.

The documentation for how to create your own scripts this can be downloaded at www.pcschematic.com under **Download => Modules, drivers etc**. At www.pcschematic.com/articles you can read a little more about the options.

NO UNDO ON SCRIPTS

PLEASE NOTE THAT IT IS *NOT POSSIBLE TO UNDO* THE EXECUTION OF A SCRIPT. SO, AS A PRECAUTION, PLEASE SAVE THE PROJECT, AND SAVE IT AGAIN UNDER A NEW NAME, BEFORE EXECUTING THE SCRIPT.

When you choose **Tools => Scripts**, you enter the **Scripting** dialog box. (You may need to install *Scripts* under **Files => Modules** first). In this dialog box you find a list of scripts, which you can choose to execute.

In the dialog box **Scripting** you can execute, add and edit scripts:

Short description of the selected script

If you select *Include in Tools menu*, you can subsequently execute the selected script directly from the **Tools** menu.

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 PCSCHEMATIC A/S about **Automation**.

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/ License**.

In the lower-left corner of the **System** tab you can then choose:

- **22.XX update**: Here you automatically receive information when updates exist for the version you are using.

- **Newer version**: Here you automatically receive information when a new version is released.

- **Information**: Here you automatically get new information from PCSCHEMATIC A/ S — see "Information from PCSCHE- MATIC A/S via the internet" on page 867. 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**: Here you can manually activate a search for new updates and new versions.
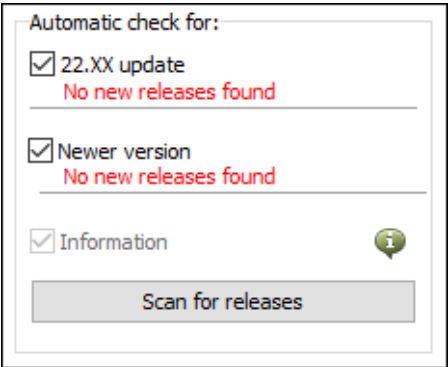
When new updates exist, this is displayed below the relevant checkbox. A button will also be displayed to the right of the checkbox. When you click this button, you enter PCSCHEMATIC's website, from which you can download the update.

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/License** on the **System** tab, 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 PCSCHEMATIC's website, from which you can download the update.

INFORMATION FROM PCSCHEMATIC 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/License** on the **System** tab, by selecting *Information* in the lower-right corner of the dialog box, as displayed in "Updating via the internet" on page 866.

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.
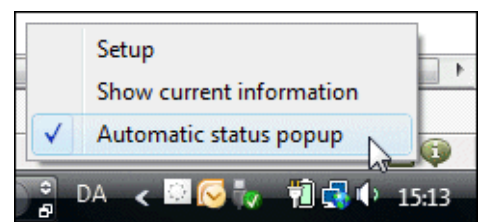
ADAPTING THE SYSTEM TO YOUR NEEDS

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. 868)   — ADDITIONAL OPTIONS .... (P. 868)

DESCRIBED IN OTHER CHAPTERS

You will find a summary of some of these options below:

OPTION	PAGE
DOCKING THE PROGRAM WINDOWS	23
INSTALLING EXTRA MODULES	861
DEFINE YOUR OWN SHORT-CUT KEYS	870
CHANGE THE SYSTEM COLORS	874
OTHER PROGRAM SETTINGS	907

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**.
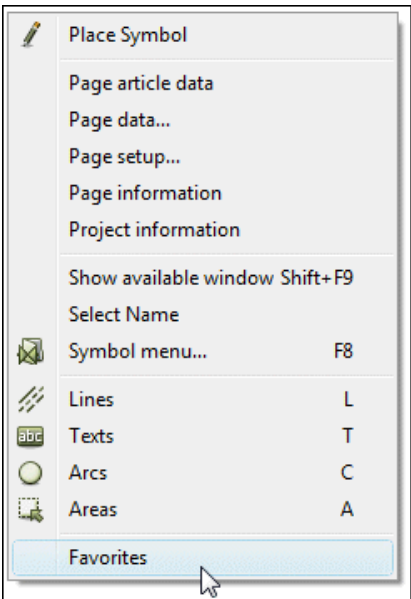
This gives you the option to specify some favorite functions:
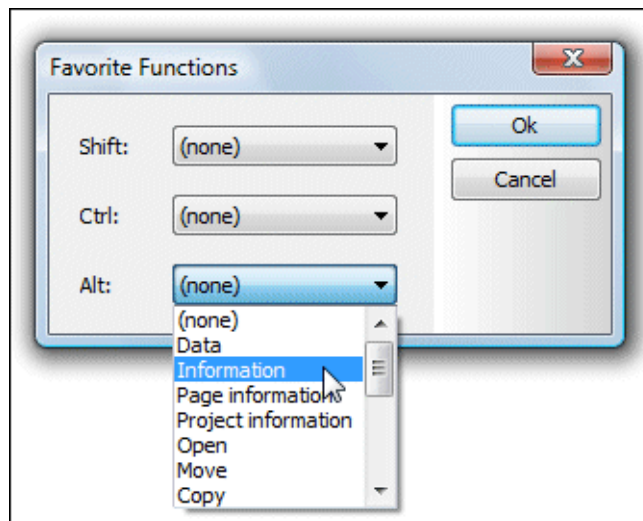
These functions are activated when you press down <Shift>, <Ctrl> or <Alt> and *right-click* on an object.

Specify character set

Under **Settings => System/License** on the **System** tab, 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 872.

FUNCTION	SHORT-CUT KEY	FUNCTION	SHORT-CUT KEY
AREAS	\<A\>	*PANELBUILDER*	\<SHIFT+P\>
ARTICLE	\<V\>	*PANELROUTER*	\<SHIFT+R\>
AUTO COMMAND CHANGE	\<SHIFT+Z\>	*PRECEDING*	\<F6\>
CHANGE STATE	\<SHIFT + SPACEBAR\>	*PREVIOUS PAGE*	\<PAGEUP\>
CIRCLES / ARCS	\<C\>	*PREVIOUS SELECTED PAGE*	\<F10\>
COMPONENT GROUPING	\<CTRL+F7\>	*PRINT*	\<CTRL+P\>
COMPONENT WIZARD	\<W\>	*RECTANGLE*	\<CTRL+F\>
COORDINATES	\<CTRL+ I\>	*REDRAW*	\<CTRL+G\>
COPY	\<CTRL+C\>	*REPLACE*	\<CTRL+H\>
DATABASE	\<D\>	*REPLACE SYMBOL*	\<F4\>
DELETE	\<DEL\>	*ROTATE 90°*	\<SPACEBAR\>
DRAW ON/OFF	\<INSERT\>	*ROUTER, ACTIVATE*	\<R\>
ELEVATION ON/OFF	\<F11\>	*SAVE PROJECT*	\<CTRL+S\>
FIND	\<CTRL+F\>	*SELECT ALL*	\<CTRL+A\>
FIND ARTICLE	\<V\>	*SELECT COMPONENT*	\<SHIFT+C\>
FIND NEXT	\<F3\>	*SLANTED LINES*	\<CTRL+K\>
FIND TYPE	\<B\>	*SNAP, FINE*	\<SHIFT\>
GO TO: AVAILABLE SYMBOLS	\<F9\>	*STRAIGHT LINES*	\<CTRL+R\>
GO TO: COMPONENTS	\<SHIFT+F9\>	*SURVEY WINDOW ON/OFF*	\<F12\>
GO TO: SYMBOLS FOR COMPONENT	\<ALT+F9\>	*SYMBOL MENU*	\<F8\>
INSERT POTENTIAL	\<P\>	*SYMBOLS*	\<S\>
KEYBOARD ENTRY	\<K\>	*TEXTS*	\<T\>
LINES	\<L\>	*TYPE*	\<B\>

THE SHORT-CUT KEYS

FUNCTION	SHORT-CUT KEY	FUNCTION	SHORT-CUT KEY
MOUNTING ASSISTANT	<SHIFT+M>	UNDO	<CTRL+Z>
NET NAVIGATOR	<N>	VIEW ARTICLE DATA	<CTRL+F2>
NEXT	<F5>	VIEW CONDUCTING	<SHIFT+F2>
NEXT PAGE	<PAGEDOWN>	WRITE TEXT	<K>
NEW DOCUMENT	<CTRL+N>	ZOOM	<Z>
OBJECT LISTER	<F7>	ZOOM IN	<CTRL+HOME>
ON-LINE MANUAL	<F1>	ZOOM, MOVE / PAN	<CTRL>+ARROW KEYS
OPEN	<CTRL+O>	ZOOM OUT	<CTRL+END>
PAGE MENU	<SHIFT+P>	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 => Shortcuts**.

You then see the following dialog box:

Choose Menu (1)　　　**Choose Command (2)**

Customize Keyboard

Categories:	Commands:		
File	Commands:	Shortcut	
Edit	New...	Ctrl+N	
View	Open...	Ctrl+O	
Insert	Close		
Functions	Save	Ctrl+S	
Lists	Save As...		
Settings	Save All		
Router	Open Unit...		
Tools	Save Unit as...		
All commands	DWG...		
	DXF...		
	PDF...	E	
	Print Page		
	Print..	Ctrl+P	
	Printer Setup...		
	Send to...		
	Modules...		
	Exit		

Close
Apply
Remove
Show All
Export
Import

Press new shortcut key:

Current key:

Ctrl+N

Reset all...

Type in New short-cut key (3)　　　**The Current short-cut key**

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 870.

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 both create custom colors and set the system colors as you like.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO SET CUSTOM AND SYSTEM COLORS.
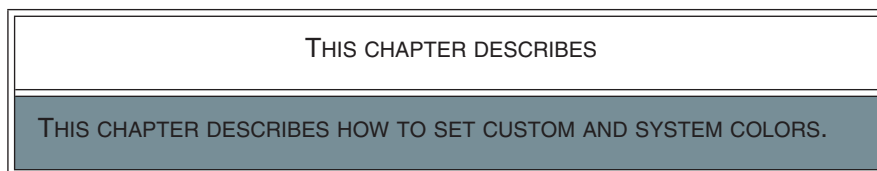
To set the colors in **Automation**, click **Settings => Basic settings**.

In the dialog box **Basic settings** you can choose whether to edit **System** colors of **Custom** colors.

System colors

If you click on **System** you enter the following dialog box:
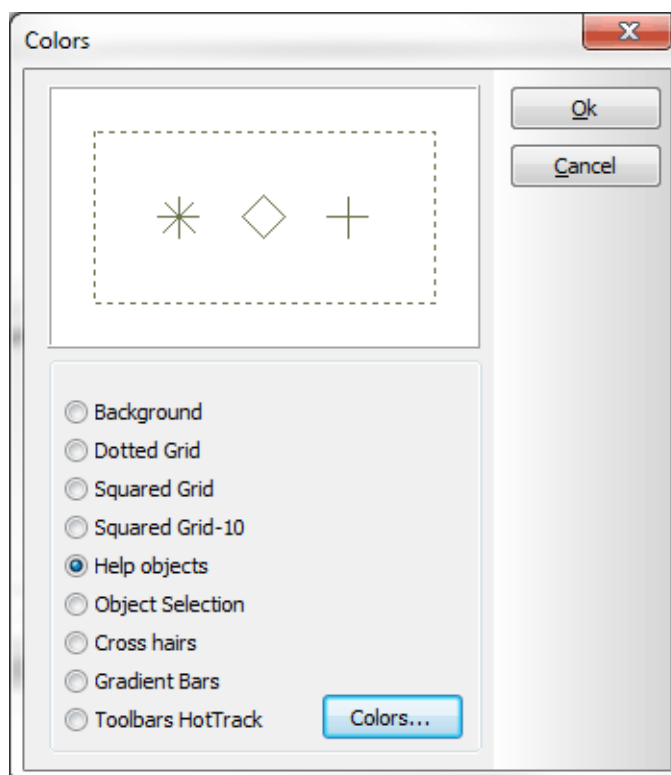
Here you specify which elements you want to change the color settings for.

When you right-click the **Show reference points** button in the left toolbar and choose **Color**, you also enter the dialog box.

You have 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.
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 *COMPLIMENTARY* 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 918), 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.

In the dialog box **Color** you either click on a *Basic color* or a *Custom color* and click **OK**.

CUSTOM COLORS

When you in **Settings => Basic settings** click on **Custom** at the bottom of the dialog box, you get the option of defining your own colors, which subsequently will be easily accessible anywhere you need to select colors in the program.

Create a new color

This is how to create a new color:
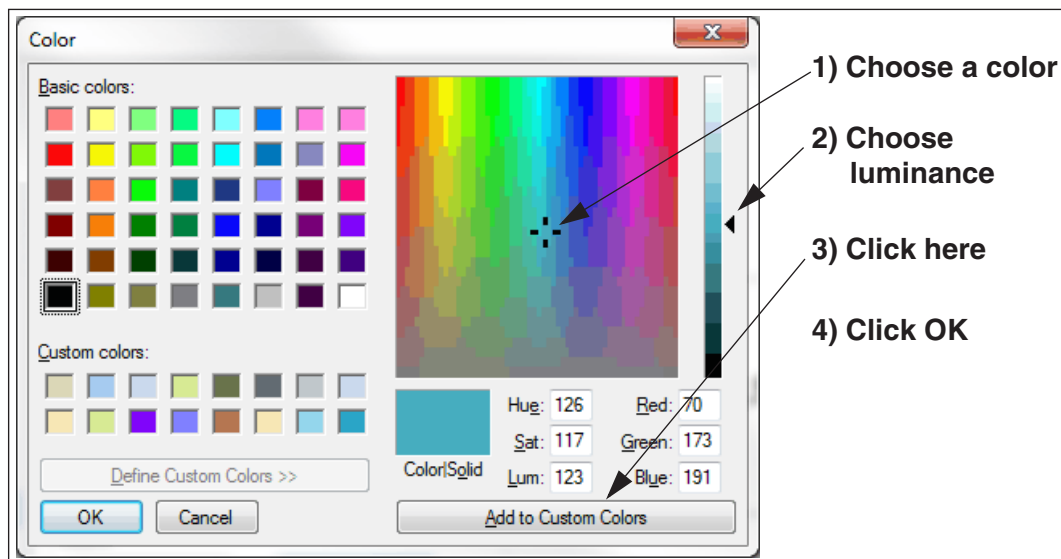
The created color is displayed under **Custom colors**, when you select new colors using the color field in the toolbar for lines. — See "Line color: F" on page 77.

It is possible to create 16 custom colors.

TANKING LICENSES

THIS CHAPTER DESCRIBES

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 878.

2) By transferring a license file to a PC which is not on the net — see "Transferring licenses via license tanking" on page 879.

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 PCSCHEMATIC 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/License**, click on the **License** tab, and the license key settings are displayed on the lower-left corner of the tab:
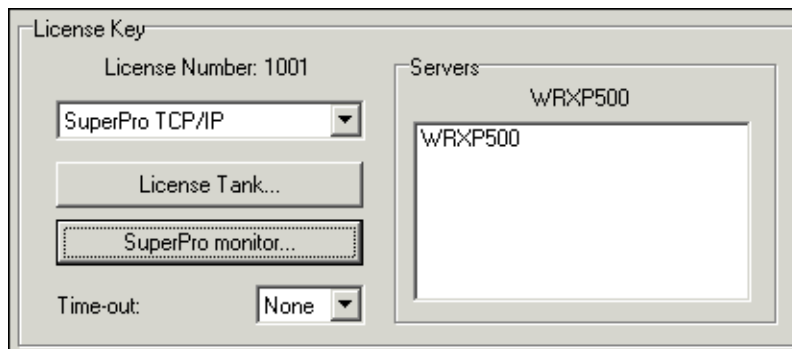
2) Click on **License Tank**:
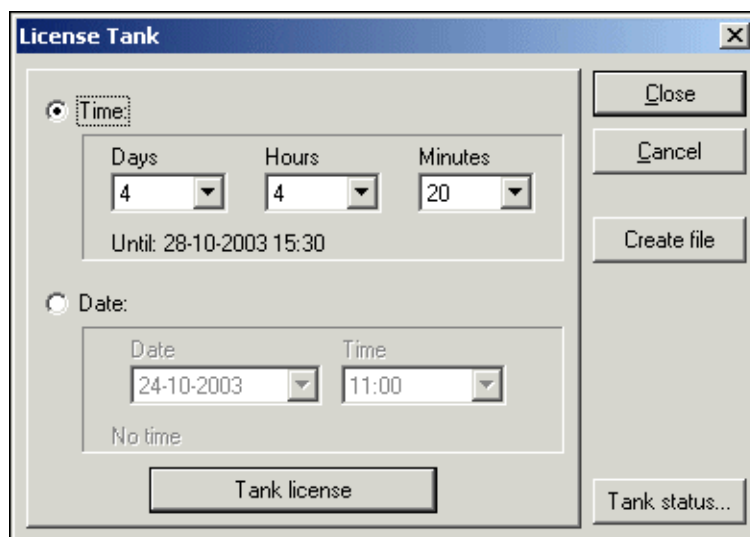
3) In the dialog box **License Tank**: Select a *Time* or a *Date* and click on **Tank license**.

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 herby 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 => License => 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 879.

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 880.

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 879.

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/License => License**: The license key settings are displayed in the lower-left corner — as displayed in "Transferring licenses by License tanking" on page 878.

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 **Pcselcad** 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 **Pcselcad**.

2) Start **Automation**. You now get the following message:
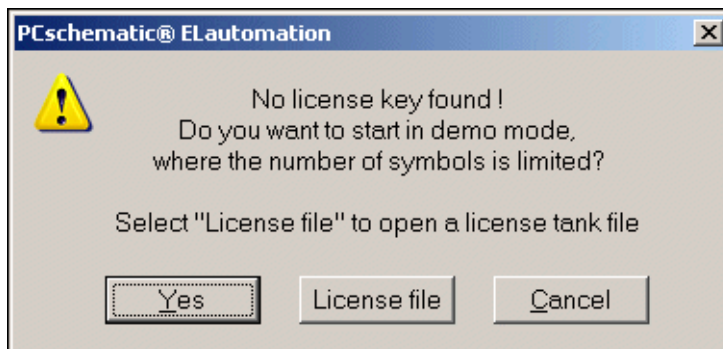
3) Click on **License file**.

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 => License => License Tank**, you can see for how much longer you have the license:
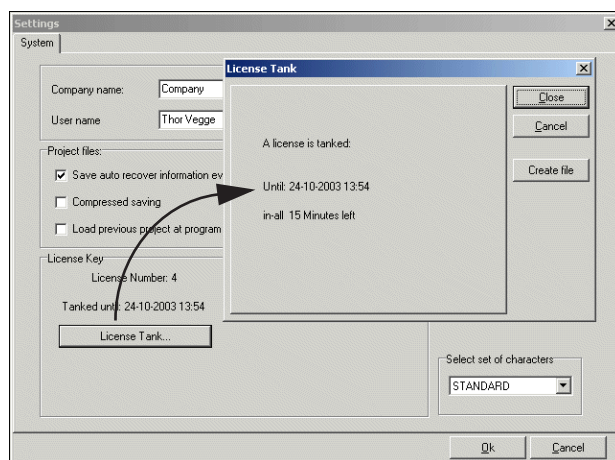
When the tanked license expires

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 **Pcselcad** folder, or on your own PC.
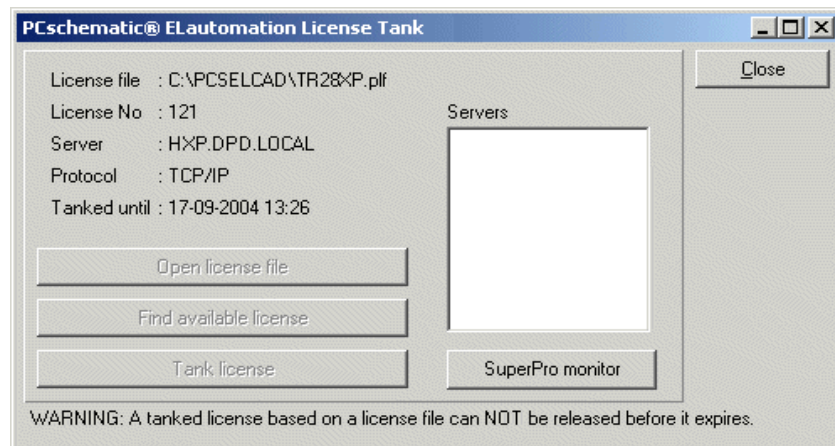
2) In the program **Automation License Tank**: Click on **Open license file**.

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.
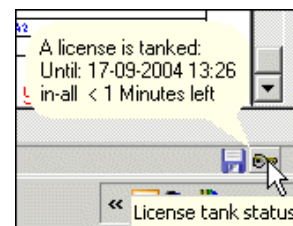
SUPERPRO MONITOR

If you choose **Settings => System/License => License => 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.

Set a check mark here to look for license keys

Double-click on a license key for further information
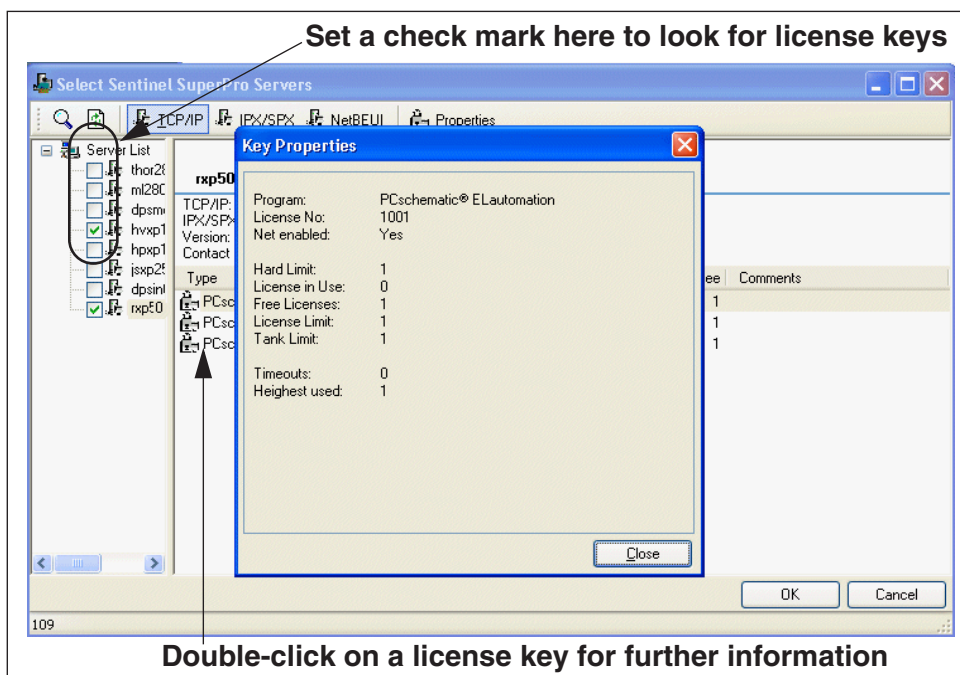
When you select a PC in the left-hand side of the dialog box, its license keys are displayed in the right-hand side of the dialog box.

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:
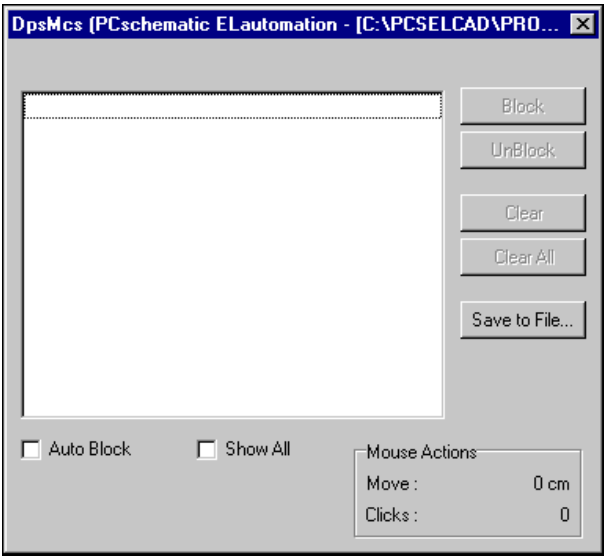
When you click anywhere in the **Automation** program, the dialog box disappears. Click the **Mouse Chasing System** button at the bottom of the screen when you want to see the dialog box again.

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.

Blocks the selected function

Remove blocking of selected function

Clear chosen function

Clear alle functions

Save the information in a file

Statistical information
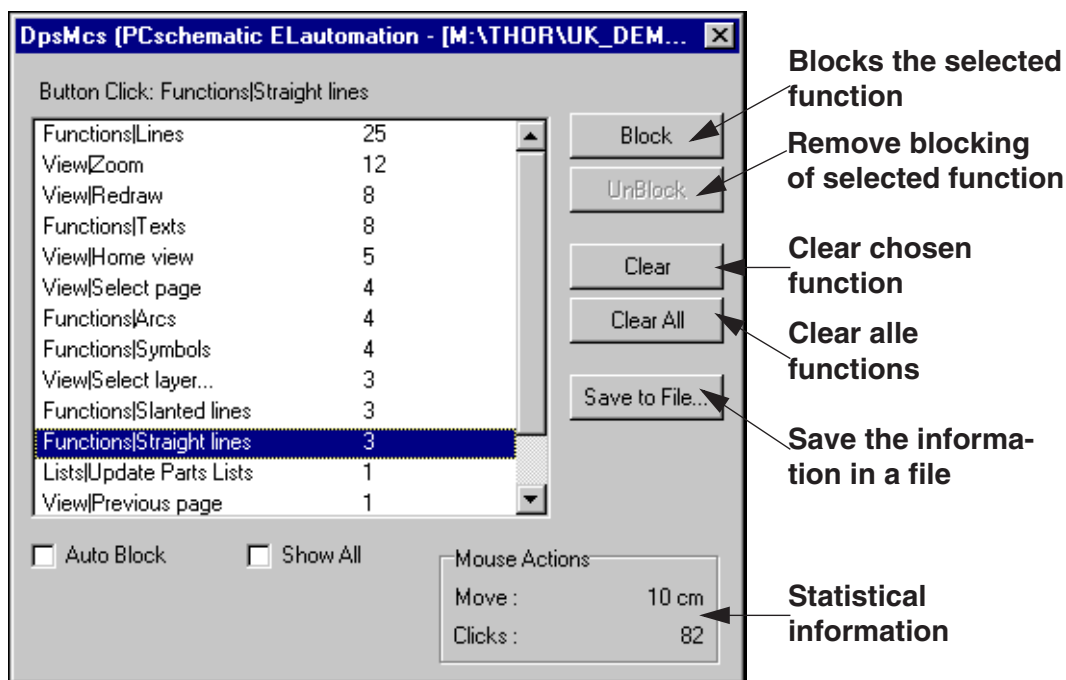
When you click on a function in the dialog box, and then click on **Block**, the function is blocked when you *click* on it in **Automation**.

Clicking on a blocked function in Automation

When you click on a blocked function in **Automation** you get the following message:
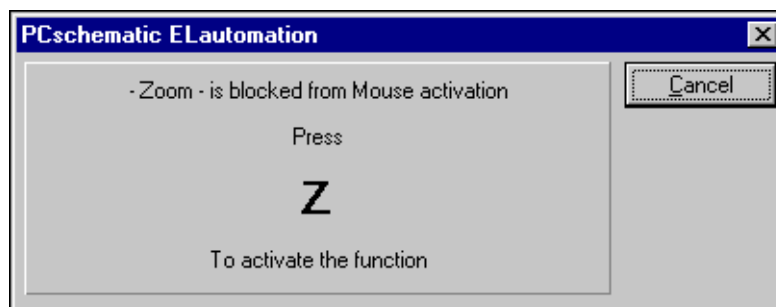
This way the program informs you about the short-cut key to the function. When you press this key, the dialog box disappears automatically, and the function is activated.

If no short-cut key is defined, you can create a short-cut key as described in "User defined short-cut keys" on page 872.

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.
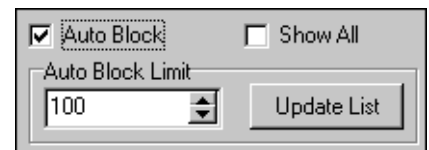
Learning the short-cut keys by need

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 click-
ing 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 auto-
matically 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:

PC | SCHEMATIC

PASSWORDS ON PROJECTS

In **Automation** you can create a password for your project files, in order to prevent others from opening or saving them.

THIS CHAPTER DESCRIBES
THIS CHAPTER DESCRIBES HOW TO USE PASSWORDS IN **Automation**.

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

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

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 893.

It is also possible to print the edited pages — see "Printing during revision control" on page 893.

STARTING A REVISION

1) Choose **Settings => Project data => Revision**

Click here
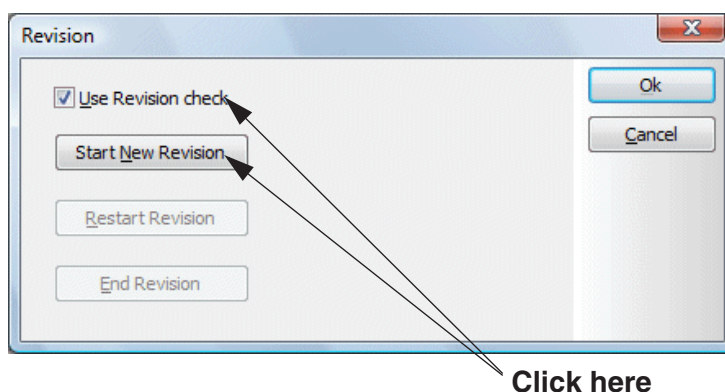
2) Select *Use Revision check*

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 892 — 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 558.
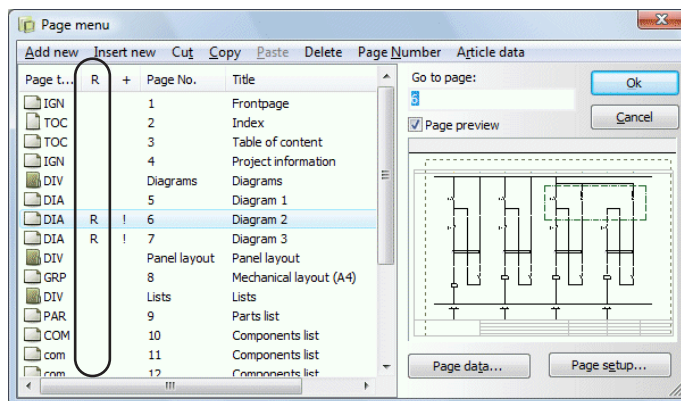
The Object Lister during revision control

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**.
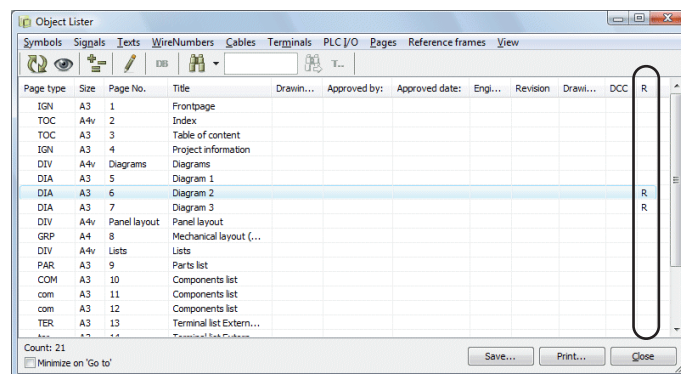
If the **R** column is not displayed, choose **View => Pages => Revised**. (Please note that the contents of the column **Revision** comes from the user defined Page data datafield of the same name). — Read more about pages in the **Object Lister** in "Editing Page data in the Object Lister" on page 360.

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**
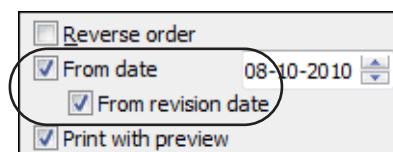
The edited pages are now printed. — Read more about the dialog box **Print** in "Printing" on page 371.

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 895.

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 893.

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 889.

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 894 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 the 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 769, particularly "Adding New datafields to Project data and Page data" on page 772.

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**:
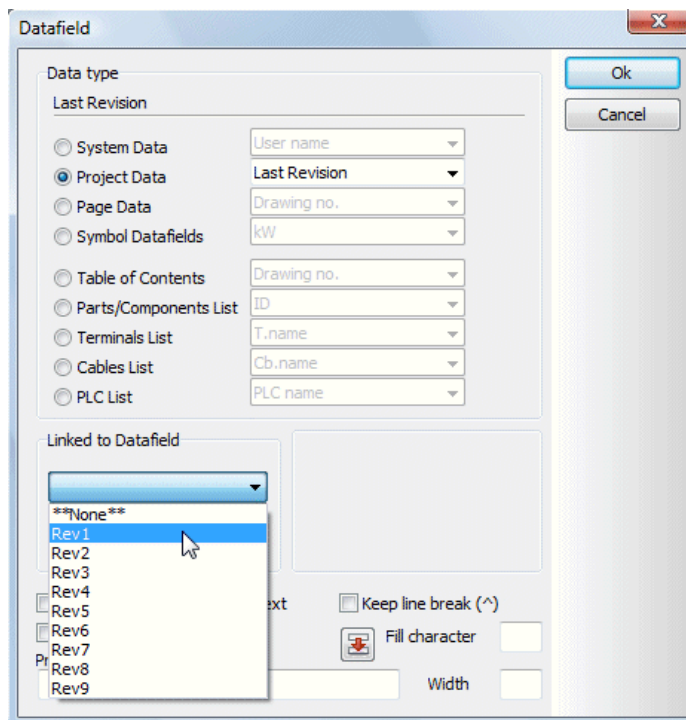
3) Fill out the datafields with revision information as the revisions are performed:

Rev1	1/7-2008
Rev2	12/8-2009
Rev3	17/9-2010
Rev4	8/10-2010
Rev5	
Rev6	

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 PCSCHEMATIC FILE TYPES

THIS CHAPTER DESCRIBES
— REGISTRATION OF PCSCHEMATIC FILE TYPES .... (P. 897)
— REGISTRATION OF .PRO FILES .... (P. 898)
— REGISTRATION OF .SYM FILES .... (P. 898)

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 1373.

To register the program file type in Windows, do this:

1) Choose **Settings => System/License => System** and click on **File type registration**:

Click here
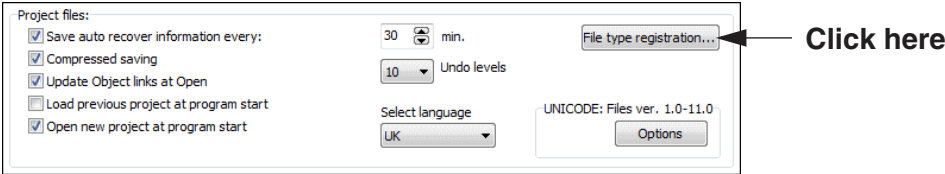
2) You then enter the dialog box **File type registration**:
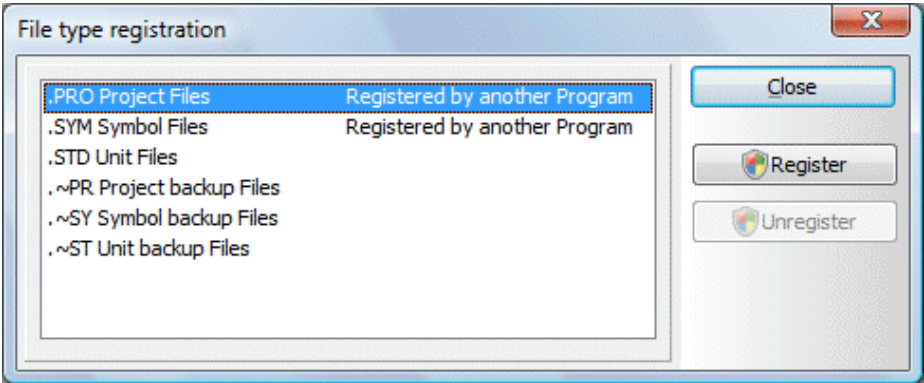
3) To register **.pro** files, first click on *.PRO Project Files* and then on **Register**. The file type is now registered. (**.sym** files are registered correspondingly).

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 PCSCHEMATIC 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 1373.

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 => PCSCHEMATIC 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 897.

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 PCSCHEMATIC icon is displayed in front of **Automation** symbols in the Windows Explorer.

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.

<table>
<tr><td align="center">THIS CHAPTER DESCRIBES</td></tr>
<tr><td>

</td></tr>
</table>

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 756.

5. Export lists

You can open **Automation** and a specific project in the program from other software systems / programs, and generate a list from the project. The command opens **Automation** and subsequently closes it again without changing the contents of the project in the process.

Write for instance (must be written in a single line):

C:\PCSELCAD\PCSELCAD.EXE C:\PCSELCAD\PROJECT\Project.pro /LISTEXPORT /FormatFile=c:\Test\PartList.pff /OutputPath=c:\test\output

The command contains the following sections:

- The path to the exe file for **Automation** (C:\PCSELCAD\PCSELCAD.EXE)
- The path to the project (C:\PCSELCAD\PROJECT\Project.pro)
- The parameter /LISTEXPORT
- The path to the format file (/FormatFile=c:\Test\PartList.pff)
- The path to the folder where the generated list is saved (/Output-Path=c:\test\output)

It is not required that the list is already generated in the project. It is exclusively the applied format file that specifies which type of list that is generated in the process. When executing the command the most recent data from the project pages are included in the generated list. (If a not updated similar list exists in the project, the contents in the generated list will therefore differ from the not updated list in the project). The project is not subsequently saved, so the project itself will in no way be changed by executing the command.

The format file also specifies whether this causes an Excel file or a comma separated file to be generated.

PC|SCHEMATIC

— Read more about format files in "Format files" on page 730, and get an overview over the possible format files in "The List files and their Extensions" on page 739.

The name of the generated list file will be the name of the project file followed by a dash, which is followed by a letter code for the list type.

This could for instance be *Projectname-PRT.csv*, where the list type is a parts list.

The following letter codes are applied:

LIST TYPE	LETTER CODE
PARTS LIST	PRT
COMPONENTS LIST	CMP
TERMINALS LIST	TRM
CABLES LIST	CBL
PLC LIST	PLC
CONNECT POINTS LIST	NET
CONNECTIONS LIST	CNP

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 901 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:\Pcselcad\Pcselcad.exe C:\PATH-TO-CMDFILE\SAVEA-SPDF.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:\Pcselcad\Project*.pro

All .pro-files in the folder **C:\Pcselcad\Project** are saved as PDF files in the same folder.

FILESAVEAS .PDF C:\Pcselcad\Project*.pro C:\Pcselcad\Project\PDFSAVE

All .pro-filer in the folder **C:\Pcselcad\Project** are saved in the folder **C:\Pcselcad\Project\PDFSAVE** as PDF files.

The folder **C:\Pcselcad\Project\PDFSAVE** must be created before executing the command.

FILESAVEAS .PDF @C:\Pcselcad\Project\ProList.txt

All project files in the listfile **C:\Pcselcad\Project\Prolist.txt** are saved as PDF files in the same folder as each individual PRO file.

FILESAVEAS .PDF @C:\Pcselcad\Project\ProList.txt C:\Pcselcad\Project\PDF-SAVE

All project files in the listfile **C:\Pcselcad\Project\Prolist.txt** are saved in the folder **C:\Pcselcad\Project\PDFSAVE** as PDF-files.

The folder **C:\Pcselcad\Project\PDFSAVE** must be created before executing the command.

Command line files for creating PDF files via quick-settings

If you have created a set of quick-settings for export of PDF files — see "Quick-settings for command line files" on page 378 — the quick-settings can also be applied in command line files:

FILESAVEAS .PDF PDF_exportsettings_1.pdfset C:\Pcselcad\Project\MyProject.pro

The project **C:\Pcselcad\Project\MyProject.pro** is generated as a PDF file — based on the quick-settings specified in **PDF_exportsettings_1.pdfset** — and is saved in the same folder as the project.

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:\Pcselcad**), 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:\Pcselcad**), 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

PCSCHEMATIC Automation

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

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/License => System** and activate *Compressed saving*:

Click here ➝
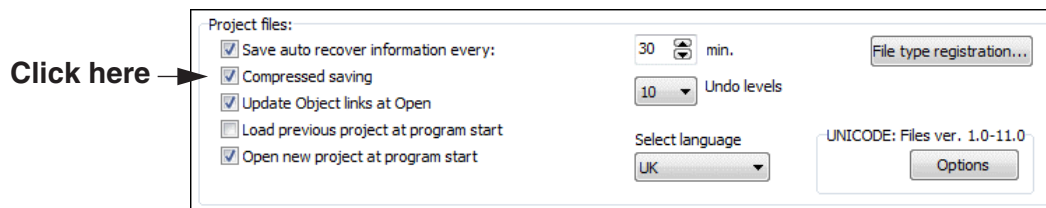
The projects still gets the extension ***.pro**, though they are zipped.

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:

PC | SCHEMATIC

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 907.

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 items, 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 **(P. 909)**	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 **(P. 912)**	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.
PAGE SETUP **(P. 915)**	SPECIFICATION OF PAPER SIZE, PAGE TYPE, PAGE SCALING, AS WELL AS GRID AND SNAP FOR THE PAGE YOU ARE WORKING ON.
PAGE REMARKS **(P. 917)**	HERE YOU CAN TYPE IN REMARKS FOR THE ACTIVE PROJECT PAGE.
POINTER/SCREEN **(P. 918)**	SETTINGS DEALING WITH WHAT IS DISPLAYED ON THE SCREEN, AND HOW IT IS DISPLAYED.

TAB	CONTENTS
DIRECTORIES (P. 923)	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).
TOOLS (P. 926)	HERE YOU SPECIFY WHICH OTHER PROGRAMS CAN BE STARTED FROM **Automation**.
DATABASE (P. 928)	CHOOSE DATABASE FILE AND SETTINGS. ACTIVATING AND DEACTIVATING THE DATABASE.
SYSTEM (P. 930)	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.
LICENSE (P. 933)	LICENSE KEY SETTINGS.
TEXT/SYMBOL DEFAULTS (P. 934)	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.
BASIC SETTINGS (P. 938)	SPECIFICATION OF BASIC SETTINGS FOR DIFFERENT TYPES OF LINES AND SPECIFICATION OF TEXT PROPERTIES FOR TEXTS IN REFERENCE FRAMES, CLOUDS AND MEASUREMENT TEXTS. YOU CAN ALSO DEFINE SYSTEM COLORS AND CUSTOM COLORS.
SPECIAL SETTINGS (P. 940)	SETTINGS WHICH ONLY APPLY FOR THE ACTIVE PROJECT.
SHORTCUTS (P. 943)	SPECIFICATION OF USER-DEFINED SHORTCUTS.

PROJECT DATA

On the tab **Settings => Project data** you handle information and 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.

Project title

Here you type in a title for the project.

List of project titles

It is possible to create a list of project titles, consisting of predefined values for the datafield.

To apply/create such a list of project titles, click the **Values** button to the right of the **Project title** field, and then administrate the list as described in "The Values dialog box" on page 124.

Remarks in projects

In the **Remarks** field at the top of the dialog box you can type in remarks for the 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.

Passwords

For further details about passwords on projects, please refer to "Passwords on Projects" on page 889.

Logo

— See "Inserting logos in projects" on page 595.

Project data

In *Project data* you type in information regarding the entire project, which you can read more about in "Project data and Page data" on page 128.

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 118.

Buttons for handling Project and Page data datafields

On the tab **Settings => Project data** as well as on the tab **Settings => Page data** you will find the following buttons for handling datafields. (The buttons are described for the **Project data** tab, but have similar functions on the **Page data** tab).

Save as default

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 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 project data from file** button, you can import project data from an Excel file or a comma separated file.

Add, Edit and Remove datafields

Using the buttons **Add datafield**, **Edit datafield** and **Remove datafield** you can delete, add or edit field names in the menu, as described in "Adding New datafields to Project data and Page data" on page 772.

Values

When you click on a datafield and then click the **Values** button, you can create / maintain a list of predefined values for the datafield. Read more about lists of values in "The Values dialog box" on page 124.

Remove unused datafields

When you click the **Remove unused data fields** button, all unused datafields are removed.

Move datafields up and down in the dialog box

When you click on a datafield and then click the **Move datafield up** or the **Move datafield down** button, you can move the datafield up or down in the list of datafields in the dialog box. You can also click on a datafield in the dialog box, keep the mouse button pressed down, and *drag* the datafield to another position.

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 440.

Revision

— See "Revision control" on page 892.

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 "On reference designations" on page 385 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 102 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

On the tab **Settings => Page data** you specify various kinds of settings regarding the project page you are working on right now.

Clicking the **Page data** button also brings you directly into this dialog box.

Page title

Here you type in the title of the page.

List of page titles

It is possible to create a list of possible page titles, consisting of predefined values for the datafield.

To apply/create such a list of page titles, click the **Values** button to the right of the *Project title* field, and then administrate the list as described in "The Values dialog box" on page 124.

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.

Mark as last used page

When you select *Mark as last used page*, this setting has effect on the datafields *Next used page*, *Used pages* and *Last used page*.

REMEMBER TO UPDATE TABLES OF CONTENTS

WHEN YOU SELECT *MARK AS LAST USED PAGE*, YOU SHOULD SUBSEQUENTLY REMEMBER TO UPDATE THE TABLES OF CONTENTS IN THE PROJECT, SO THAT THEY ONLY CONTAIN PAGES FROM THE FIRST PROJECT PAGE TO THE "LAST" PROJECT PAGE.

— See "Last used page active" on page 373.

Page data

Under **Page data** you type in information, which is attached to the page you are working on in the project. — Read more about this in "Project data and Page data" on page 128.

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 118.

Buttons for handling Page data datafields

See how to use the buttons under *Page data* for handling page data datafields in "Buttons for handling Project and Page data datafields" on page 910.

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 "On reference designations" on page 385 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 911.

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 600.
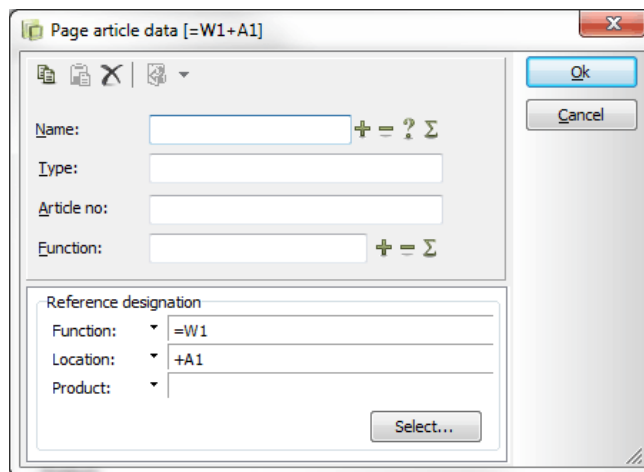
References

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 445.

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.

PAGE SETUP

On the tab **Settings => 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 572.

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 575.

Page type and Page function

See how to select the **Page type** in "Page type" on page 574.

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 560 for further details.

Please note that a **Page function** can be set to **Normal**, as described in "Change page function & Transfer page function to other pages" on page 563.

Grid and Snap

Grid and **Snap** is also specified here — read more about this in "Screen / Image Functions" on page 42.

You can also select **Follow page scale** in order to specify that snap and grid follows the page scale — see "Setting the scale of the page" on page 547.

"OTHER SYMBOLS"

IN **Automation** VERSION 19 AND EARLIER VERSIONS, YOU WERE HERE ABLE TO SPECIFY FROM WHICH DATAFIELD IN THE DATABASE, THAT THE SYMBOLS ON THE PAGE WERE FETCHED. THIS IS NOW DONE IN THE **COMPONENT MENU**. — SEE "THE PAGE DATABASE FIELD FOR SYMBOLS" ON PAGE 266.

Scale factor and Reading direction

— Read more about **Scale factor** in "Setting the scale of the page" on page 547.

— **Reading direction** is described in "Specify reading direction" on page 144.

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 841.

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 575.

PAGE REMARKS
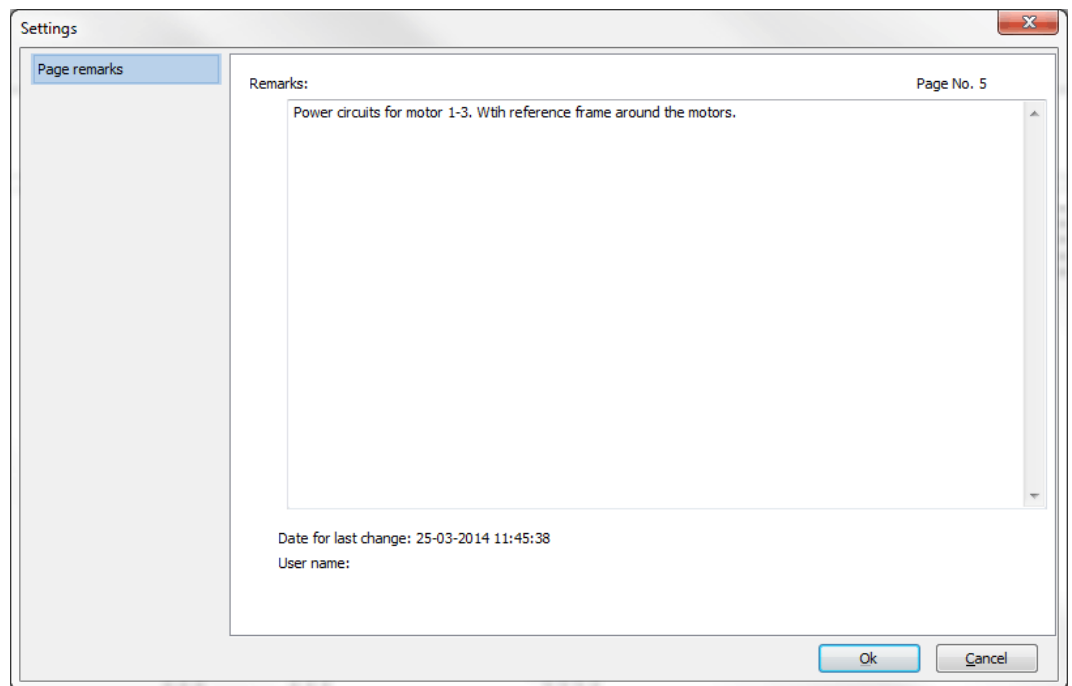
On the tab **Settings => Page remarks**, you can type in remarks about the selected page in the project.

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 843.

POINTER/SCREEN

The tab **Settings => Pointer/Screen** consists of a number of check boxes, in which you specify how you want the screen and pointer settings.
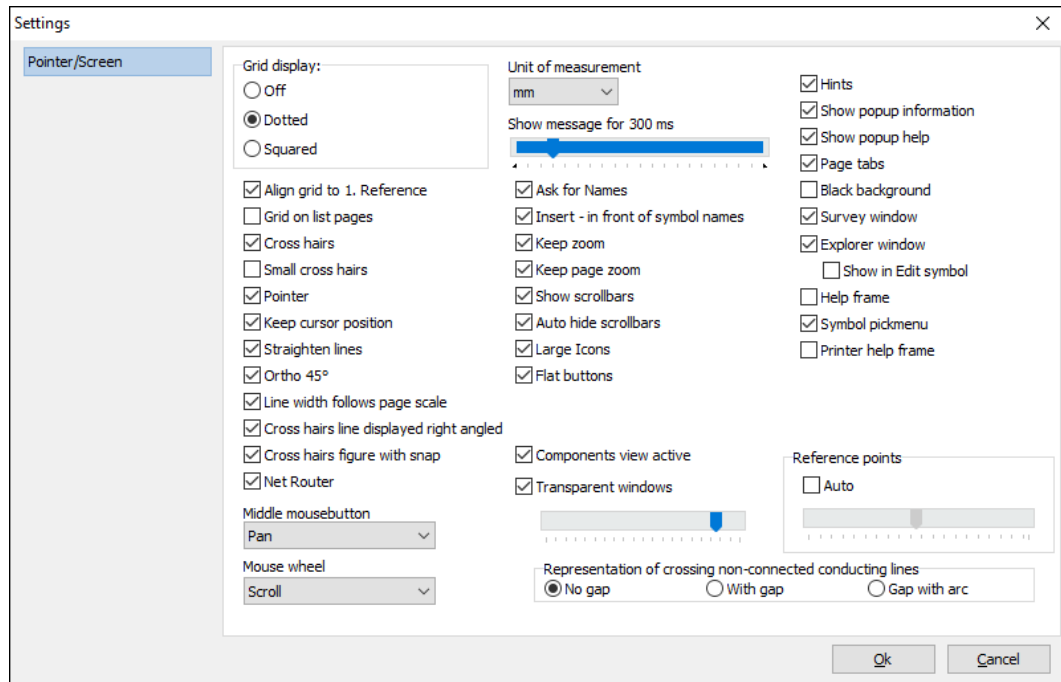
Grid display: Here you specify the appearance of the grid on the screen. Please refer to "Grid" on page 48 for further details.

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 980.

Pointer: Specifies whether a mouse pointer is displayed on the screen. Is automatically selected if *Cross hairs* is deselected.

Keep cursor position: When you open a dialog box while working on a diagram page, you have to move the cross hairs in order to make choices in the dialog box. Therefore, you can use *Keep cursor position* to control where to position the cursor when the dialog box is closed:

- When *Keep cursor position* is *not* activated: The cursor remains where you last clicked in the dialog box.

- When *Keep cursor position* *is* activated: The cursor jumps back to the exact same position in the diagram, as it had before opening the dialog box.

Straighten lines: Here you specify whether the connecting lines should be straightened when you move a symbol. Is enabled by default.

PC|SCHEMATIC

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.

Line width follows page scale: When ***Line width follows page scale*** is activated, lines on pages with page scale are drawn with a line width that reflects the selected page scale.

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 715.

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***, ***Zoom: Wheel Forward*** or ***Zoom: Wheel Backward***. 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 scroll-bars 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: This option only exist in Windows 7 and previous versions of Windows. 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 874.
— Please note that dialog boxes aways have *gradient look*.

Gradient Pickmenu: This option only exist in Windows 7 and previous versions of Windows. Is only displayed when you have selected ***Gradient toolbars***. Select ***Gradient Pickmenu*** if you also want *gradient look* in pickmenus.

Transparent windows: Some windows can remain on screen while you are work-ing on the drawing area in the project. These windows can be set to become trans-parent 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 win-dows*** 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**.

Component menu active: Here you can specify whether the **Component menu** is active. Please note that this change will first occur when the program is restarted.

Please note that choosing to deactivate this window will make it impossible to use the short-cut [F9] for searching for components with available symbols.

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.

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 345 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.

Symbol pickmenu: Here you turn the pickmenu on and off. Read more about the symbol pick menu in "Fetching symbols from the Pickmenu" on page 155.

Reference points: Reference points are created to make it easier to work with the diagrams on the screen (and to make it easier to attach information to objects intelligently). However, when you have zoomed out on a page, the reference points can take too much attention, and for instance make it difficult to read connection names. When you select *Auto*, the program only makes reference points visible when you zoom in on a page. You can adjust when to show the reference points by dragging the slider.

You can also specify whether or not to display reference points on screen by clicking the **Reference points** button in the left toolbar, or by choosing **View => Reference points**. Read more about reference points in "Dictionary" on page 980. The reference points are not visible when you print out the project.

Representation of crossing non-connected conducting lines: Here you specify how to represent non-connected conducting lines, when they cross on diagram pages. You have the following options:

- ***No gap***: The lines cross.

- ***With gap***: A gap is opened on one of the lines.

- ***Gap with arc***: A gap is opened on one of the lines and an arc is drawn on the other line.

— Also see "Priority on lines" on page 86.

DIRECTORIES
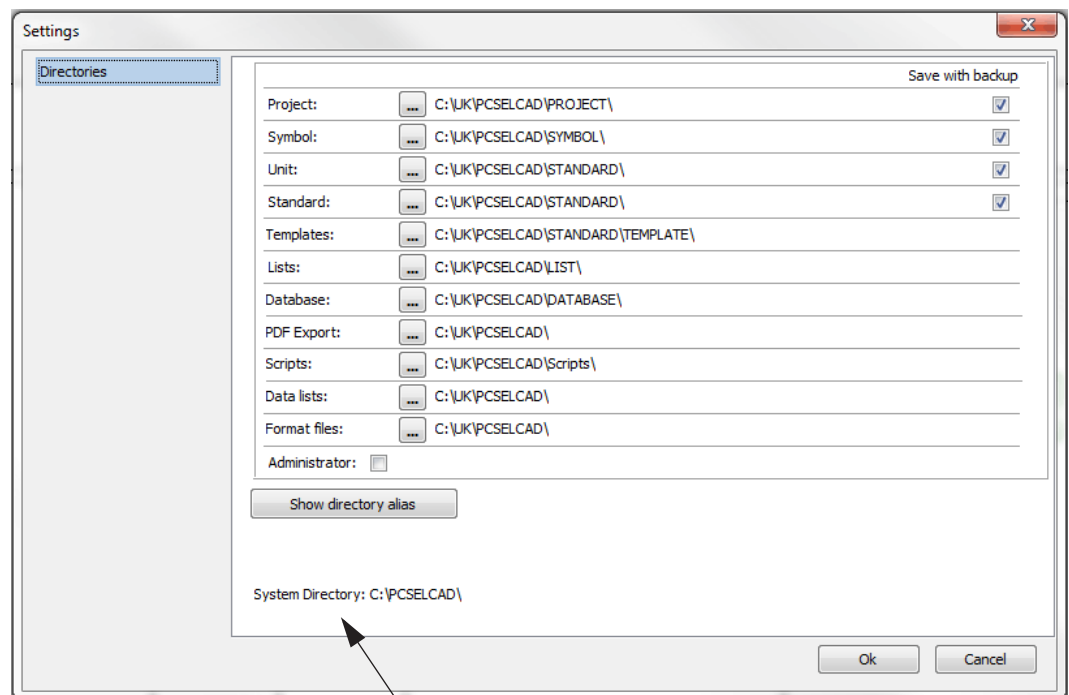
**Here the System catalog is displayed,
as well as the Workstation catalog**

Directories

On the tab **Settings => Directories** 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.
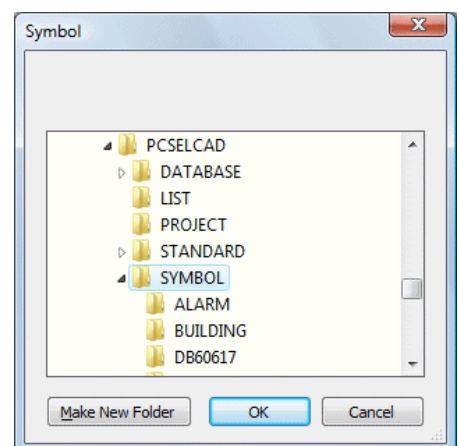
Save with backup

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**.

Alias for directories

When you on the tab **Settings => Directories** click on **Show directory alias**, you enter the following dialog box:
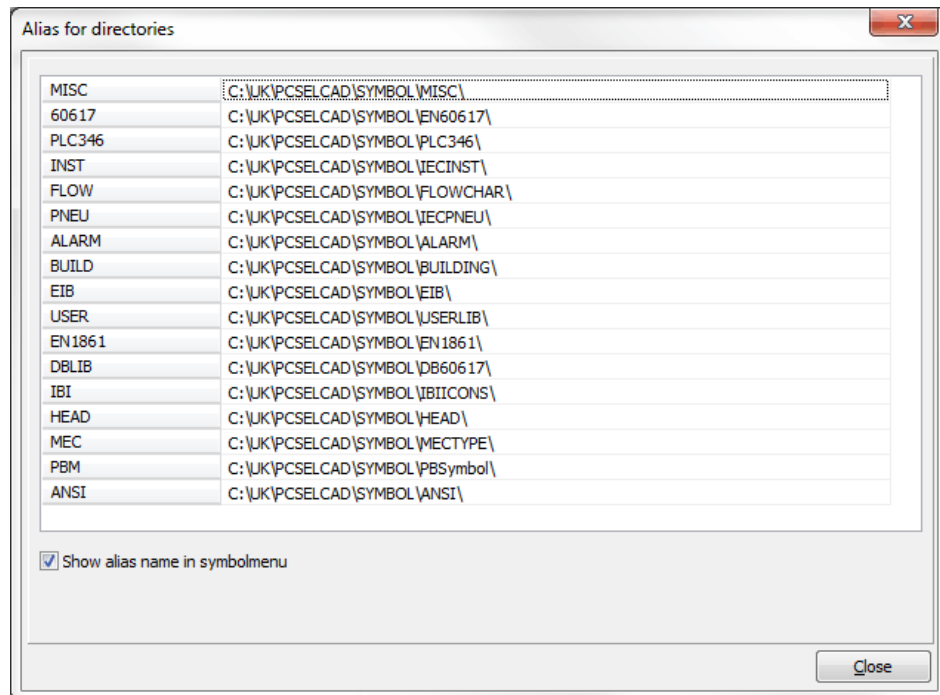
Here you get the option to assign an *alias* to each symbol folder/directory.

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:\SYM-BOLS\MISC**, you only have to correct the path here. The program will then fetch symbols from the new symbol library.

You will normally not need to type in the alias **MISC** anywhere in **Automation**; it is merely a name you have given to the program, so that it internally can distinguish between the various aliases. But 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 to search in first

When you fetch a symbol from the database, and the symbol is not already in the project, the program will search the symbol folders in the order you have specified under **Alias for directories**, starting from the top. The program then finds the first symbol with the given symbol name. The order of the directories is therefore 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.

Replacing all symbols - based on Alias

If you apply **Functions => Special functions => Replace all symbols** all symbols in the projects are loaded into the project again — based on the order in which they appear under **Alias for directories**.

— See "Replacing all symbols in a project" on page 224.

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.
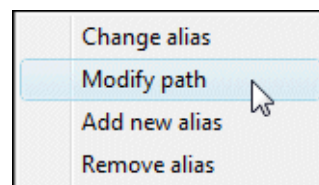
If you want to change an alias name, you then choose **Change alias**, and if you want to remove an alias, you choose **Remove alias**.

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.

TOOLS
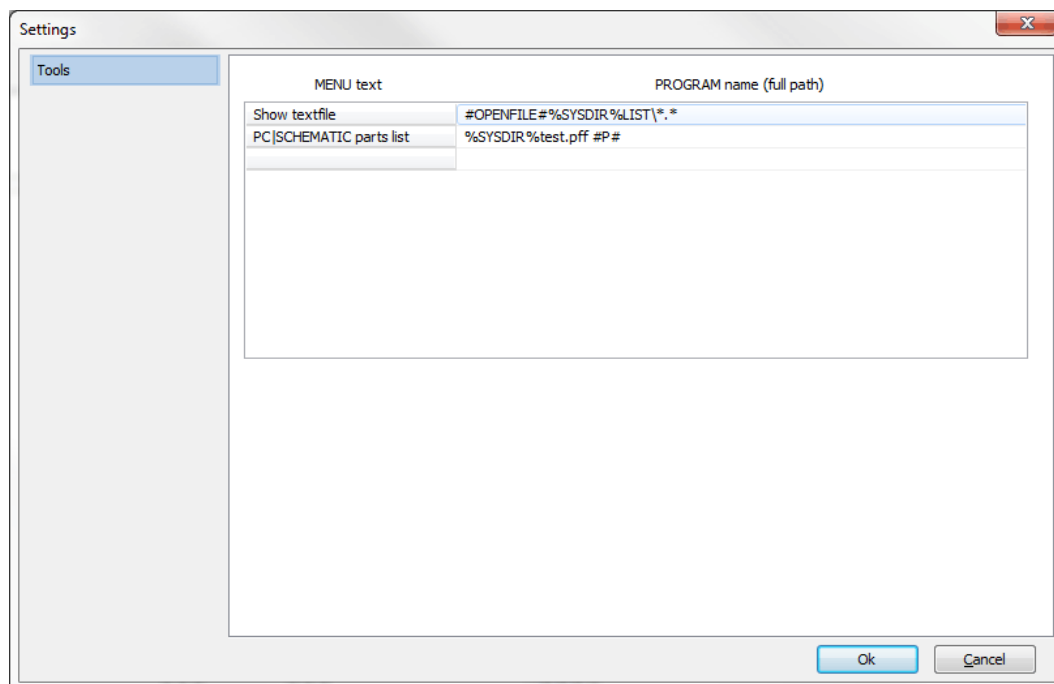
In the program tool bar you can choose a menu called **Tools**. From this menu you can for instance change back and forth between **Automation** and other programs, or create list files based on predefined settings.

Starting other programs from the Tools menu

On the tab **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.

%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.

%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.

Creating list files from the Tools menu

If there is a specific list format that you frequently use, you can choose to make it available directly from the **Tools** menu. See how to do this in "Placing fixed list formats in the Tools menu" on page 731.

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 *NP* on it, in order to be able to change to Notepad quickly, you just add **#B(NP)#** after the program name for Notepad.

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 **NP** in the example above: **#B(**bitmap file**)#**. The size of the bitmap file should be 20x20 pixels.

Removing or editing menu items in the Tools menu

If you want to remove or edit a menu item, just right-click it in the **Settings => Tools** tab and choose **Remove menu item** or **Edit menu item** respectively.

MENU text	PROGRAM name (full path)
Components list file	%SYSDIR%CompList_uk.cff #P#
Parts list file	%SYSDIR%PartsList_uk.pff #P# #SLIENT"
Insert menu item	
Remove menu item	
Edit menu item	

DATABASE
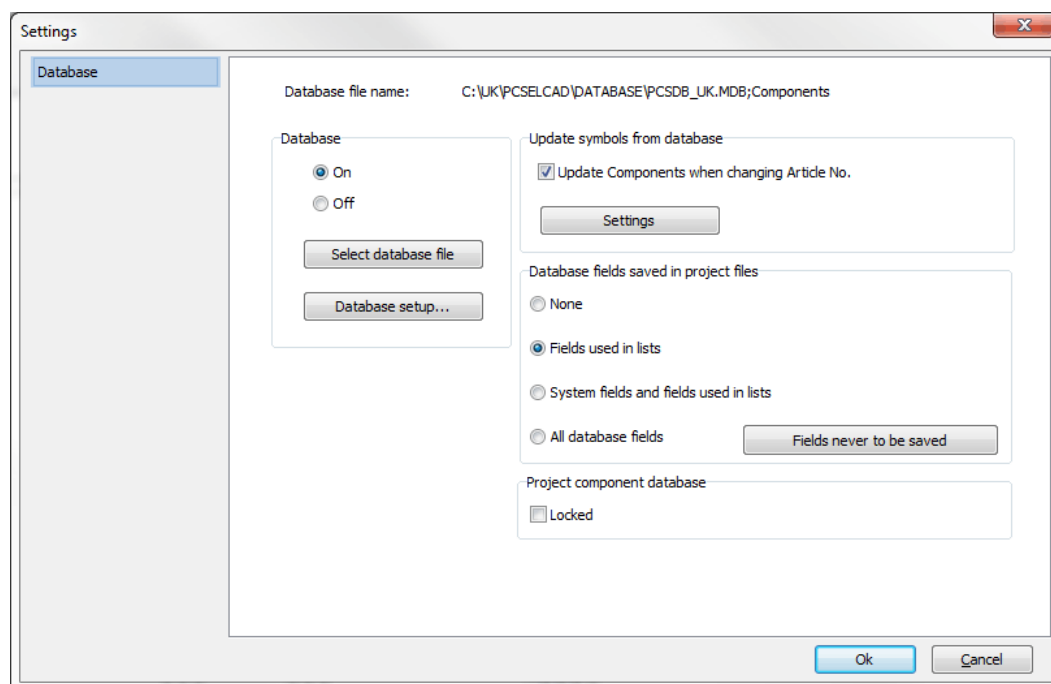
On the tab **Settings => Database** you specify whether to work in **Automation** with- or without a connected database, by clicking on *On* or *Off* respectively.

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 699.

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 682.

You can read more about the database program in the **PCSCHEMATIC Database** manual.

Database fields saved in project files

Settings for project component database. — See "Project component database" on page 654.

Locking the project component database

— See "Locking the project component database for the active project" on page 657.

![PC|SCHEMATIC logo]

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 Component 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.
UPDATE CABLE WIRES	UPDATES ALL CABLE WIRES, SO THAT THEY GET THE NAMES SPECIFIED IN THE DATABASE.
CHECK FOR CORRECT SYMBOLS	IT IS CHECKED WHETHER THE SYMBOLS FOR THE COMPONENT HAVE THE CORRECT SYMBOL FILE NAMES — 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

On the tab **Settings => System/License => System** you can 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 "Data-fields" on page 118.

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 **Pcselcad** using the same project name, but with the extension **.recoveryPRO**.

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 904.

PC | SCHEMATIC

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.

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, American English, German, French, Spanish, Brazilian Portuguese, Czech, Polish, Hungarian, Russian, Chinese, Norwegian, Swedish and Danish.

If these options are not available in your installation, you can have them sent to you by contacting PCSCHEMATIC A/S.

When changing language the program must be restarted.

USING STATES IN DRAWING HEADERS FOR CHANGING LANGUAGE

YOU CAN USE STATES IN DRAWING HEADERS FOR CHANGING LANGUAGE FOR TEXTS IN DRAWING HEADERS.

WHEN YOU USE THIS OPTION, THE LANGUAGE YOU CHOOSE HERE DETERMINES WHICH LANGUAGE IS SELECTED IN THE DRAWING HEADERS YOU SUBSEQUENTLY ASSIGN TO PAGES IN THE PROJECT.

— READ MORE ABOUT THIS IN "DRAWING HEADERS WITH STATES" ON PAGE 829.

— Also see "Symbol settings: Title" on page 789 to see how to specify titles for symbols in different languages.

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 **22.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 866.

You can also check for new updates by choosing **Help => Check for updates** from the menu line at the top of the screen.

Automatic check for:
☑ 22.XX update
No new releases found
☑ Newer version
No new releases found
☑ Information
Scan for releases
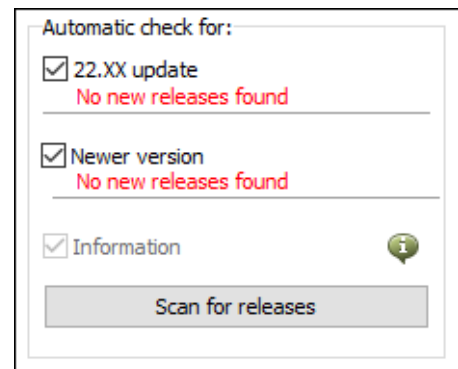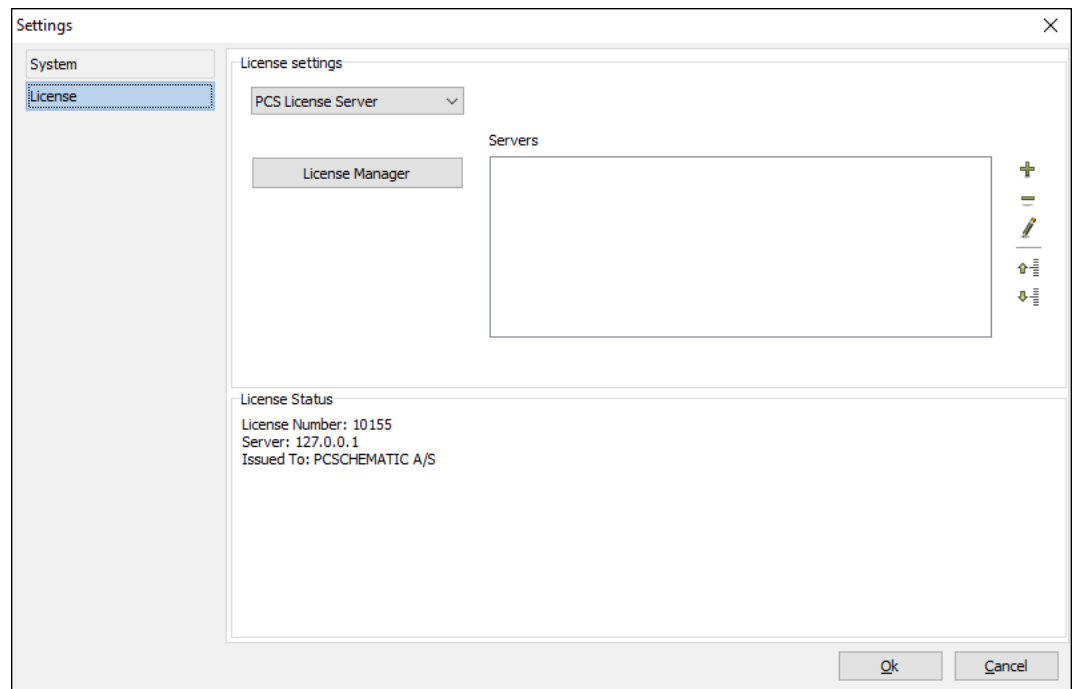
LICENSE

On the tab **Settings => System/License => License** you have the following options:

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 => License => License key**.

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.

Software license key

Up to version 15, it was only possible to purchase **Automation** along with a physical license key. Since version 17 was released you normally purchase **Automation** with a software license key. How to install this license system is described in the *PCS License Server - Install guide*.

TEXT/SYMBOL DEFAULTS

The tab **Settings => Text/Symbol defaults** 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 141.

— 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 448.

— See also "Default text properties for free texts" on page 114, "Appearance of references" on page 438, "Changing the reference crosses" on page 447 and "Specifying default text settings for Edit symbol mode" on page 796.

Transferring text defaults between projects
If you wish to transfer all text and naming settings between projects, click **All**:

Here you get the following options:

Transfer all text settings to the project: Click here to transfer all the text settings that you have specified under **Text defaults** to all objects in the project. — This means that all settings specified under **Free**, **Symbol**, **Connection**, **Reference** and **Line** are hereby transferred to all objects in the project.

Load all default text settings: Click here to transfer all *default* text settings to the project. (See below).

Save all text settings as default: Click here to save all text settings in the active project as default text settings.

Use project default text settings when placing objects: When you select this option, all settings that you have specified here under **Text defaults**, will be applied when placing objects in the project.
Therefore, if a symbol is designed with specific text properties, these will be overruled by the specified text defaults, when the symbol is placed in the project.

Settings for different customers

If you have created different project templates, which contains the text and naming standards for different customers, you can use the possibilities above to easily transfer these settings from a template to the active project.

Symbol defaults

The program contains lists specifying which symbols can be used as for instance drawing headers or signal symbols, and other lists specifying the names of the user defined datafields for symbols and lines. Under **Symbol defaults** you can specify which symbols are represented in these lists, and also specify the names of the various types of datafields for symbols and lines.

Adding a symbol to a list requires that the symbol is already created — read more about creating symbols in "Creating Symbols" on page 775.

Inserting symbols in lists / Create datafields

In order to place a symbol/datafield in a list, you first specify — on the left-hand side — which type of list you wish to place the symbol/datafield in. Then click the **Add** button.

For symbols: You now enter the **Symbol menu** where you click on the symbol you want to add to the list, and click **OK**.

For datafields: You now enter the dialog box **Enter text** where you type in the name of the datafield that you wish to create, and click **OK**.

— Read more about adding a symbol (here a drawing header) to a list in "Adding drawing headers to the List Of Drawing Headers" on page 828.

CALCULATED LINE DATAFIELDS

It is also possible to create calculated *line* datafields, where you for instance can combine different line datafields in a single datafield. — See "Calculated line data-fields (Formula editor)" on page 83.

Removing symbols or datafields from lists

You can remove symbols and datafields from the lists by clicking on the down-arrow in the ***Drawing headers*** field, then click on the symbol or datafield you wish to remove, click on the **Remove** button, and answer **Yes** to remove it 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 828.

— Read more about drawing headers in "Creating Drawing Headers" on page 820.

— Read more about signal symbols in "Creating Signal Symbols" on page 816.

— Read more about join signal symbols in "Creating Join symbols" on page 819.

— Read more about symbol datafields and line datafields in "Creating symbol datafields" on page 811.

— Read more about reference symbols in "Changing the reference crosses" on page 447.

List of signal names

There also exists a list of signal names. When you place a signal symbol on a pro-ject 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 234, and about adding signal names to the list in "Frequently used signal symbol Names" on page 819.

Cable direction and Default cable quantity

At the bottom of the tab you can specify the default cable direction for horizontal and vertical cables.

You can also specify whether the ***Default cable quantity*** is *0* or *1*:

- When you choose *0*: The ***Quantity*** field is used for typing in the lengths of the individual cables, which can then be accumulated in the project lists.

- When you choose *1*: The *Quantity* field is used for counting the number of cables (cable symbols) in the project.

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 97.

BASIC SETTINGS
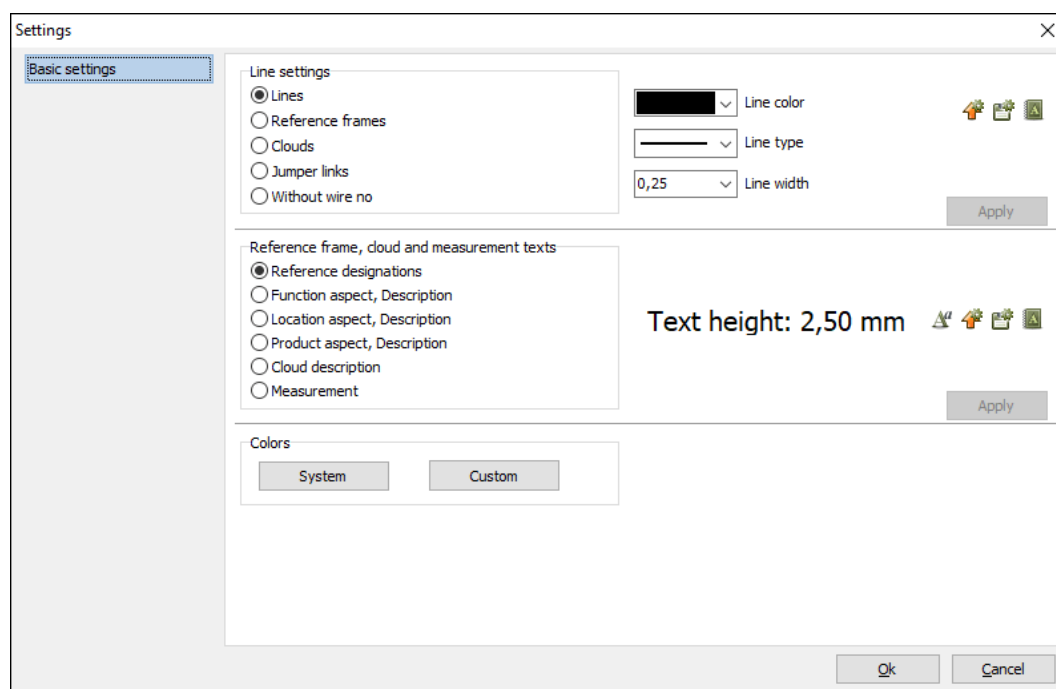
On the tab **Settings => Basic settings** you have the following options:

Line settings

Here you can specify the basic settings for different types of lines in the project.

To change how a line type from now on will be drawn in the project, do this:

1) Click on the desired line type - for instance **Lines**

2) Specify **Line color**, **Line type** and **Line width**

3) Click **Apply**

You also have these options:

When you click the **Transfer standard settings as project settings** button, the saved standard settings are transferred for all of the line types as project settings.

When you click the **Transfer project settings as default settings** button, the project settings for all of the line types are transferred as standard settings for future projects.

When you click the **Transfer line setting to project** button, the settings for the selected line type in the dialog box, are transferred to all lines of the this type in the project.

Reference frame, cloud and measurement texts

Here you can specify the basic settings for how texts appears for reference frames, clouds and measurement objects in the project. To edit how these texts appear in the project from now on, do this:

1) Click on the desired text type - for instance **Reference designations**

2) Click the **Text properties** button

3) In the dialog box **Text properties**: Specify the appearance of the selected text type and click **OK**.

4) In the **Basic settings** tab: Click **Apply**.

Beyond this, you have the same options as described above.

System colors and custom colors

At the bottom of the tab, you can change **System** colors and **Custom** colors. Read more about this in "Colors" on page 874.

SPECIAL SETTINGS

On the tab **Settings => Special settings** you can set specific settings, which only apply for the active project.

> **CLICK APPLY TO ACTIVATE THE CHANGES**
> WHEN YOU CHANGE SETTINGS IN **SETTINGS => SPECIAL SETTINGS**, YOU MUST SUBSEQUENTLY CLICK THE **APPLY** BUTTON TO ACTIVATE THE CHANGES.

Default setting in the dialog box: Rename symbols

Here you can specify the default setting for all dialog boxes for renaming symbols. If for instance you work with pneumatics, you can choose to set it to *Do NOT rename symbols*.

Default setting in the dialog box: Component data

If you select *Show full Reference designation* here, the setting *Show full Reference designation* will by default be selected on the **Ref.design.** tab in the dialog box **Component data** when you place new symbols in the project.

Do not show dots on: Conducting lines or Connection points

When you under **Do not show dots on** select *Conducting lines*, dots will not be shown on conducting lines in the diagrams. — Please note that all connections must be indicated by either dots or bend according to the EN61082 standard.

When you under **Do not show dots on** select *Connection points*, dots will not be shown on connection points in the diagrams.

Signal name from PLC or component

When you draw a line from a connection point on a PLC symbol, and terminate the line in a signal symbol (*without* connecting the line to a connection point on another symbol), you can here specify that the program by default suggests to name this signal symbol from either the **PLC address/Function text** or the **Connected component/Connection point (-K1:14)**.

This means that the signal symbol is either named from the *function text* on the connection point on the connected symbol, or from the symbol and the connection point. — See "Signal names assigned from Function text or Connection name" on page 237.

Printer and PDF dialog

If you choose **Show list update reminder before printing and before exporting PDF files**, you will be asked whether to update project lists before printing and before exporting the project as a PDF file.

Settings for the page type Ground plan / Mechanical

If you select **Show lines in symbols as thin lines**, symbols on **Ground plan/ Mechanical** pages are drawn using thin lines, when displayed on the screen. This is meant to increase the on-screen readability, but if this is not the case on your screen, just disable this setting.

Update lists when saving a project

Here you choose whether lists are *not* updated when projects are saved, if they are *always* updated when projects are saved, or whether you are *asked* first.

Update from component database when saving a project

Here you choose whether the project components are *not* updated from the attached component database when projects are saved, if they are *always* updated from the attached component database when projects are saved, or whether you are *asked* first.

Fixed line break on texts

When you place datafields in projects, you can specify a maximal **Width** for the datafields in the dialog box **Datafield**. If you select **Wrap text**, the text will continue on the line below, when the text is longer than the specified width. If you select **Keep line break (^)**, these line breaks are also transferred to lists. — See "Datafields" on page 118.

Under **Fixed line break in texts** you can specify whether line breaks in the project are made exactly at the specified number of characters in the field **Width**. If it for instance says *10* in the **Width** field, the line break will be made after exactly 10 characters, regardless of whether the line break is made in the middle of a word.

In lists the line break will always be made exactly at the specified width, because longer texts risk overwriting the contents of the next column in the list.

If you want the program to make exact line breaks on diagram pages also, select **On all pages**.

If you want the program only to make exact line breaks in lists, select **Lists only**.

ONLY EFFECT FOR SMALL TEXT WIDTHS

THE SETTING WILL TYPICALLY ONLY MAKE A DIFFERENCE FOR VERY SMALL LINE WIDTHS. IF FOR INSTANCE *WIDTH* IS SET TO 25, THE PROGRAM WILL PLACE AS MANY WORDS AS POSSIBLE ON EACH LINE, AS LONG AS THE TOTAL NUMBER OF CHARACTERS ON THE LINES IS BELOW 25 CHARACTERS — NO MATTER WHICH OF THE SETTINGS YOU HAVE CHOSEN.

This setting also has effect on free texts, if you have specified a line width and selected *Wrap text*. — Read more about free texts in "General purpose / Free texts" on page 111.

SHORT-CUTS
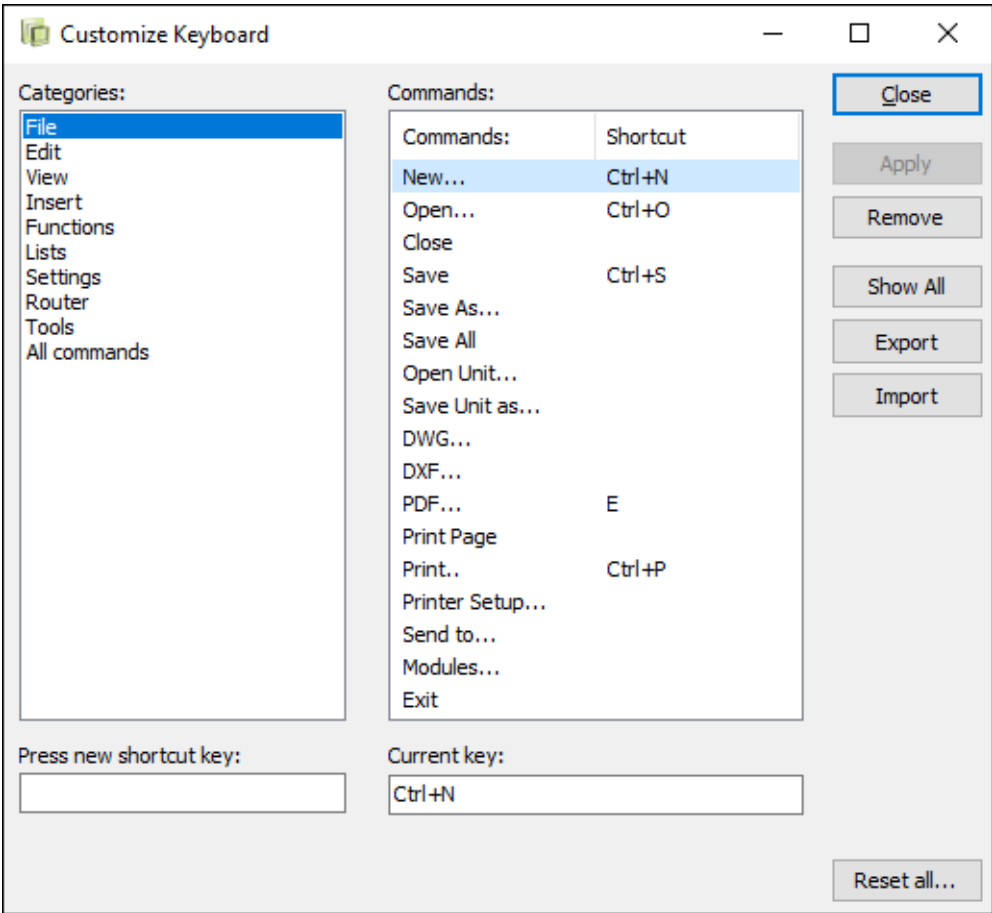
On the tab **Settings => Shortcuts** you can define your own keyboard shortcuts, or edit the already created shortcuts. See how to do this in "User defined short-cut keys" on page 872.

You can see the predefined shortcuts in "Short-cut Keys" on page 870.

DATAFIELDS IN AUTOMATION

In the following it is displayed which datafields can be inserted on project pages and drawing headers / lists via 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.

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

DATAFIELD	DESCRIPTION
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*.
FILE DATE/TIME	FILE DATE/TIME FOR THE LAST TIME THAT THE PROJECT WAS SAVED. CHOOSE EITHER *DATE AND TIME*, *DATE*, *TIME* OR *YEAR*.
REFERENCE DESIGNATION	REFERENCE DESIGNATION DEFINED IN THE PROJECT: CHOOSE BETWEEN *ALL*, *FUNCTIONAL ASPECT* AND *LOCATIONAL ASPECT*, AND SPECIFY WHETHER THE INSERT THE *DESIGNATION*, *DESCRIPTION* OR *DESCRIPTION 2*.
LOGO1 OR LOGO2	THE SELECTED LOGO IS INSERTED. — SEE "INSERTING LOGOS IN PROJECTS" ON PAGE 595.
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 895.
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.

DATAFIELD	DESCRIPTION
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* (THE REFERENCE DESIGNATIONS ARE ONLY DISPLAYED IF THEY DIFFER FROM THE REFERENCE DESIGNATIONS ON THE PREVIOUS PAGE), *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT 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*, *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT 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), *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT 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), *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
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*, *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT 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, LOCATION ASPECT* AND/OR *PRODUCT 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 895.
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*.

DATAFIELD	DESCRIPTION
LAST PRINTED	TIME AND DATE FOR WHEN THE PAGE WAS PRINTED LAST. HERE YOU CAN CHOOSE BETWEEN *DATE AND TIME*, *DATE*, *TIME* AND *YEAR*.
ACTIVE LAYER	SPECIFIES THE NAME OF THE ACTIVE LAYER. IF IT HAS BEEN CHOSEN TO SHOW A LAYER GROUP, THE NAME OF THE GROUP IS DISPLAYED HERE.
GROUP OF LAYERS	SPECIFIES WHICH LAYERS A GROUP CONTAINS.
PAGE TAB COLOR	INSERTS A THREE CHARACTER LONG FIELD, WHICH HAS THE SAME COLOR AS THE PAGE TAB. IT IS NOT POSSIBLE TO INSERT A PRE-TEXT FOR THIS DATAFIELD. — READ MORE ABOUT COLOR ON PAGE TABS IN "COLORS ON PAGE TABS" ON PAGE 571.

SYMBOL DATAFIELDS

Contains user-designed symbol datafields. — See "Symbol datafields" on page 122.

Symbol datafields can also be created with a list of predefined values. — See "Symbol datafields with predefined values" on page 123.

When using symbol datafields of this type in the definition of a symbol, you can also fetch data from any datafield in the attached database. — See "Extra symbol Datafields in symbols" on page 790, particularly "Inserting symbol datafields from database in symbol definition" on page 791.

Symbol datafields for terminal, cable and PLC lists

When you use symbol datafields in terminal, cable and PLC lists, you can specify whether to fetch the values for the fields from the actual symbol or from the component (same text for all symbols for the component in the list).

Edit symbol: Description and reference designation as extra Symbol datafield

When you are designing a symbol — in the Edit symbol mode — you have the following extra datafields at your disposal:

DATAFIELD	DESCRIPTION
=+-DESIGN.	THE REFERENCE DESIGNATION FOR THE COMPONENT: CHOOSE EITHER *DESIGNATION*, *DESCRIPTION* OR *DESCRIPTION 2* — AND … CHOOSE EITHER *FULL REFERENCE DESGINATION*, *FUNCTION ASPECT*, *LOCATION ASPECT* OR *PRODUCT ASPECT*.
DESCRIPTION	INSERT THE CONTENTS OF THE DATAFIELD, WHICH IS MAPPED TO THE *DESCRIPTION* FIELD IN THE DIALOG BOX *SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA* — SEE "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 684 AND ITS SUBSECTION "DESCRIPTION" ON PAGE 685.
QUANTITY	THE CONTENTS OF THE *QUANTITY* FIELD ON THE *GENERAL* TAB IN THE DIALOG BOX *COMPONENT DATA*. — SEE "PREDEFINED SYMBOL DATAFIELDS" ON PAGE 126.

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 **FULL REFERENCE DESIGNATION**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT** — AND SPECIFY WHETHER TO PLACE THE **DESIGNATION**, THE **DESCRIPTION** OR **DESCRIPTION 2**.
CREATION DATE/TIME	CREATION DATE/TIME FOR THE PAGE. CHOOSE BETWEEN **DATE AND TIME**, **DATE**, **TIME** AND **YEAR**.
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 895.
REMARKS	REMARKS FROM THE REMARKS FIELD FOR THE PAGE. — SEE "PAGE REMARKS" ON PAGE 917.
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** (THE REFERENCE DESIGNATIONS ARE ONLY DISPLAYED IF THEY DIFFER FROM THE REFERENCE DESIGNATIONS ON THE PREVIOUS PAGE), **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT 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** (THE REFERENCE DESIGNATIONS ARE ONLY DISPLAYED IF THEY DIFFER FROM THE REFERENCE DESIGNATIONS ON THE PREVIOUS PAGE), **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**. — SEE "LAST USED PAGE ACTIVE" ON PAGE 373.
STATE. PRIMARY HEADER	THE SELECTED STATE OF THE PRIMARY DRAWING HEADER. — SEE "TWO DRAWING HEADERS ON THE SAME PAGE" ON PAGE 605.
STATE. SECONDARY HEADER	THE SELECTED STATE OF THE SECONDARY DRAWING HEADER. — SEE "TWO DRAWING HEADERS ON THE SAME PAGE" ON PAGE 605.
PAGE TAB COLOR	THE COLOR OF THE PAGE TAB.

DATAFIELD	DESCRIPTION
PAGE PAPER FORMAT	FOR INSTANCE A3.
PAGE PAPER NORM	FOR INSTANCE ISO.
PAGE PAPER SIZE	FOR INSTANCE 420MM X 297MM.

PARTS/COMPONENTS LIST DATAFIELDS

The list also contains datafields from the attached database. If you use a database from PCSCHEMATIC, the datafield names from the database are written in capitals. This makes it easy to distinguish these datafields from the other datafields. If you create your own database, it is recommended that you also write these names in capitals.

Fields from the PCSCHEMATIC database are not displayed below. You can see these in "Datafields in the component database" on page 969.

DATAFIELD	DESCRIPTION
NAME	THE NAME OF THE COMPONENT. YOU CAN HERE ALSO CHOOSE WITH QUANTITY, FUNCTION ASPECT, LOCATION ASPECT AND/OR PRODUCT 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 299.
FUNCTION	THE FUNCTION OF THE COMPONENT
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE.
OBJECT GROUP NAME	IF THE COMPONENT IS PART OF AN OBJECT GROUP, THIS DATA-FIELD DSIPLAYS THE NAME OF THE OBJECT GROUP. — READ MORE ABOUT OBJECT GROUPS IN "OBJECT GROUPS" ON PAGE 293.
THUMBNAILS FIELD	THE PICTURE OF THE COMPONENT FROM THE THUMBNAILS / PICTURE FIELD IN 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.

PC | SCHEMATIC

DATAFIELD	DESCRIPTION
QUANTITY	THE QUANTITY OF THE COMPONENT. CHOOSE BETWEEN **QUANTITY** ("NUMBER OF UNITS"), **QUANTITY COMPL. PACKAGES** (ROUNDED UP TO THE NEAREST INTEGER), OR **QUANTITY IN PACKAGES** (SPECIFIED WITH DECIMALS). WHEN YOU SELECT **WITH QUANTITY** (ONLY POSSIBLE FOR **QUANTITY COMPL. PACKAGES** OR **QUANTITY IN PACKAGES**), BOTH QUANTITY AND PACKAGE INFORMATION IS DISPLAYED. PLEASE NOTE, THAT THE INFORMATION **QUANTIY 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 684). YOU CAN ALSO SELECT/DESELECT **+(NO OF CABLES)**. THIS GIVES YOU THE NUMBER OF CABLES IN BRACKETS AFTER THE QUANTITY IN PARTS LISTS.
TOTAL QUANTITY	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). WHEN YOU SELECT **WITH QUANTITY** (ONLY POSSIBLE FOR **QUANTITY COMPL. PACKAGES** OR **QUANTITY IN PACKAGES**), BOTH QUANTITY AND PACKAGE INFORMATION IS DISPLAYED. PLEASE NOTE, THAT THE INFORMATION **QUANTIY 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 684). YOU CAN ALSO SELECT/DESELECT **+(NO OF CABLES)**. THIS GIVES YOU THE NUMBER OF CABLES IN BRACKETS AFTER THE QUANTITY IN PARTS LISTS.
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	DISPLAYS A LIST OF ALL COMPONENTS LINKING TO THE UNIT DRAWING IN THE PROJECT. HERE YOU HAVE THE OPTION OF SELECTING/DESELECTING THE **FUNCTION ASPECT**, THE **LOCATION ASPECT** AND/OR THE **PRODUCT ASPECT**. — READ MORE ABOUT THIS IN "INSERTING LINKS TO COMPONENTS IN THE UNIT DRAWING" ON PAGE 848.
UNIT NAME LIST	UNIT NAME LIST. HERE YOU CAN SELECT/DESELECT **FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT 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 **CURRENTPATH, PAGE, FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT ASPECT.**

DATAFIELD	DESCRIPTION
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 717 FOR FURTHER INFORMATION.
MECHANICAL PAGE	THE PAGE NUMBER FOR THE GROUND PLAN PAGE, WHERE THE MECHANICAL SYMBOL IS PLACED. YOU CAN HERE CHOOSE WHETHER TO ADD THE *LOCATION ASPECT*, THE *FUNCTION ASPECT* AND/OR THE *PRODUCT ASPECT* ALONG WITH THE PAGE NUMBER.
=+DESIGN.	THE REFERENCE DESIGNATION FOR THE COMPONENT: CHOOSE EITHER *DESIGNATION*, *DESCRIPTION* OR *DESCRIPTION 2* — AND … CHOOSE EITHER *FULL REFERENCE DESGINATION*, *FUNCTION ASPECT*, *LOCATION ASPECT* OR *PRODUCT ASPECT*.
SYMBOL TYPE	THE 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 BAR-CODE FOR THE COMPONENT. YOU CAN HERE CHOOSE THE BETWEEN STANDARD SIZES *NORMAL 12x31 MM*, *SMALL 10x25 MM* AND *LARGE 20x62 MM*.
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

The list also contains datafields from the attached database. If you use a database from PCSCHEMATIC, the datafield names from the database are written in capitals. This makes it easy to distinguish these datafields from the other datafields. If you create your own database, it is recommended that you also write these names in capitals.

Fields from the PCSCHEMATIC database are not displayed below. You can see these in "Datafields in the component database" on page 969.

DATAFIELD	DESCRIPTION
T.NAME	THE NAME OF THE TERMINAL. YOU CAN HERE SELECT *WITH SUB NAME*, *WITH CONNECTION NAME*, *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
T.TYPE	THE TYPE OF THE TERMINAL.
T.ARTICLE	THE ARTICLE NUMBER OF THE TERMINAL.
T.FUNCTION	THE FUNCTION OF THE TERMINAL.
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE. CHOOSE EITHER *TERMINAL*, *EXTERNAL* (THE DESCRIPTION FROM THE INTERNALLY CONNECTED COMPONENT) OR *INTERNAL* (THE DESCRIPTION FROM THE EXTERNALLY CONNECTED COMPONENT).

DATAFIELD	DESCRIPTION
T.=+DESIGN	THE REFERENCE DESIGNATION FOR THE TERMINAL: CHOOSE EITHER **DESIGNATION**, **DESCRIPTION** OR **DESCRIPTION 2** — AND … CHOOSE EITHER **FULL REFERENCE DESGINATION**, **FUNCTION ASPECT**, **LOCATION ASPECT** OR **PRODUCT ASPECT**.
C.NAME	THE CONNECTION NAME OF THE TERMINAL. CHOOSE EITHER **OUTPUT SIDE**, **INPUT SIDE**, **INPUT SIDE/OUTPUT SIDE** OR **OUTPUT SIDE/INPUT SIDE**.
C.FUNCTION	THE CONNECTION FUNCTION OF THE TERMINAL. CHOOSE EITHER **OUTPUT SIDE**, **INPUT SIDE**, **INPUT SIDE/OUTPUT SIDE** OR **OUTPUT SIDE/INPUT SIDE**.
C.LABEL	THE CONNECTION LABEL OF THE TERMINAL. CHOOSE EITHER **OUTPUT SIDE**, **INPUT SIDE**, **INPUT SIDE/OUTPUT SIDE** OR **OUTPUT SIDE/INPUT SIDE**.
C.DESCR.	THE CONNECTION DESCRIPTION OF THE TERMINAL. CHOOSE EITHER **OUTPUT SIDE**, **INPUT SIDE**, **INPUT SIDE/OUTPUT SIDE** OR **OUTPUT SIDE/INPUT SIDE**.
C.REF	THE CONNECTION CURRENT PATH OF THE TERMINAL. YOU CAN HERE SELECT **FUNCTIONAL ASPECT**, **LOCATIONAL ASPECT**, **PAGE** AND **CURRENT PATH**.
IE.NAME	INTERNAL/EXTERNAL COMPONENT NAME. THIS CAN EITHER BE **EXTERNAL** OR **INTERNAL**. YOU CAN HERE SELECT **WITH SUB NAME**, **WITH CONNECTION NAME**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT 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**, **LOCATION ASPECT** OR **PRODUCT 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**, **LOCATION ASPECT** OR **PRODUCT 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**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.

DATAFIELD	DESCRIPTION
IE.SIGNAL	INTERNAL/EXTERNAL SIGNAL NAME. THIS CAN EITHER BE **EXTERNAL** OR **INTERNAL**. YOU CAN ALSO CHOOSE **FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
IE.WIRENO	INTERNAL/EXTERNAL WIRE NUMBER. THIS CAN EITHER BE **EXTERNAL** OR **INTERNAL**. YOU CAN ALSO CHOOSE **FUNCTION ASPECT, LOCATION ASPECT** OR **PRODUCT ASPECT**.
IE.CB.NAME	INTERNAL/EXTERNAL CABLE NAME. THIS CAN EITHER BE **EXTERNAL** OR **INTERNAL**. YOU CAN HERE SELECT **WITH SUB NAME, WITH CONNECTION NAME, FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT 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 **FULL REFERENCE DESIGNATION, FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
IE.CB.=+DESCRIPTION	DESCRIPTION FOR THE INTERNAL/EXTERNAL CABLE FUNCTION/LOCATION. THIS CAN EITHER BE **EXTERNAL** OR **INTERNAL**. CHOOSE BETWEEN **FULL REFERENCE DESIGNATION, FUNCTION ASPECT, LOCATION ASPECT** OR **PRODUCT ASPECT**.
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 **CURRENT PATH, PAGE, FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
JUMPER LINK	— SEE "JUMPER LINKS IN LISTS AND TERMINAL PLANS" ON PAGE 94.
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

The list also contains datafields from the attached database. If you use a database from PCSCHEMATIC, the datafield names from the database are written in capitals. This makes it easy to distinguish these datafields from the other datafields. If you create your own database, it is recommended that you also write these names in capitals.

Fields from the PCSCHEMATIC database are not displayed below. You can see these in "Datafields in the component database" on page 969.

DATAFIELD	DESCRIPTION
CB.NAME	CABLE NAME. THIS CAN EITHER BE *WITH SUB NAME*, *WITH CONNECTION NAME*, *FUNCTION ASPECT*, *LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
CB.TYPE	CABLE TYPE
CB.ARTICLE	CABLE ARTICLE NUMBER
CB.FUNCTION	CABLE FUNCTION
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE. CHOOSE EITHER *CABLE*, *FROM* (THE DESCRIPTION FROM THE COMPONENT IN THE "FROM" END) OR *TO* (THE DESCRIPTION FROM THE COMPONENT IN THE "TO" END).
CB.=+DESIGN.	THE REFERENCE DESIGNATION FOR THE CABLE: CHOOSE EITHER *DESIGNATION*, *DESCRIPTION* OR *DESCRIPTION 2* — AND ... CHOOSE EITHER *FULL REFERENCE DESGINATION*, *FUNCTION ASPECT*, *LOCATION ASPECT* OR *PRODUCT ASPECT*.
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 *CURRENT PATH*, *PAGE, FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
FT.NAME	FROM/TO COMPONENT NAME. THIS CAN EITHER BE *WITH SUB NAME*, *WITH CONNECTION NAME, FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
FT.TYPE	FROM/TO COMPONENT TYPE
FT.ARTICLE	FROM/TO COMPONENT ARTICLE NUMBER
FT.FUNCTION	FROM/TO COMPONENT FUNCTION
FT.=+DESIGN.	FROM/TO COMPONENT REFERENCE DESIGNATION. THIS CAN EITHER BE *FULL REFERENCE DESIGNATION, FUNCTION ASPECT, LOCATION ASPECT* OR *PRODUCT ASPECT*.
FT.=+DESCRIPTION	DESCRIPTION FOR THE FROM/TO COMPONENT REFERENCE DESIGNATION. THIS CAN EITHER BE *FULL REFERENCE DESIGNATION, FUNCTION ASPECT, LOCATION ASPECT* OR *PRODUCT 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

DATAFIELD	DESCRIPTION
FT.C.REF.	FROM/TO COMPONENT CONNECTION CURRENT PATH. HERE CAN BE REFERRED TO **CURRENT PATH**, **PAGE**, **FUNCTION ASPECT, LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
CB.W.SIGNAL	CABLE WIRE SIGNAL NAME. YOU CAN ALSO SELECT **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
CB.W.WIRENO	CABLE WIRE WIRE NUMBER. YOU CAN ALSO SELECT **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
CABLE WIRES	CHOOSE BETWEEN **WIRE COUNT**, **USED WIRES COUNT**, **FREE WIRES COUNT** OR **FREE WIRE NAMES**.
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. — SEE "CABLE DIRECTION AND DEFAULT CABLE QUANTITY" ON PAGE 936 AND "THE QUANTITY FIELD IN THE DIALOG BOX COMPONENT DATA" ON PAGE 179.

PLC LIST DATAFIELDS

The list also contains datafields from the attached database. If you use a database from PCSCHEMATIC, the datafield names from the database are written in capitals. This makes it easy to distinguish these datafields from the other datafields. If you create your own database, it is recommended that you also write these names in capitals.

Fields from the PCSCHEMATIC database are not displayed below. You can see these in "Datafields in the component database" on page 969.

The PLC connected component

For all datafields from the database it is possible to specify whether to display data for the **PLC** or the **Connected component**. If **Connected component** is chosen, data will be fetched from the component that the PLC either activates data from or receives data from.

When you have selected a symbol on a project page and enter the dialog box **Component data**, you can specify that the component represented by the symbols should be understood as the connected component by selecting **The PLC connected component**. — See "The PLC connected component" on page 180.

PC | SCHEMATIC

DATAFIELD	DESCRIPTION
PLC NAME	PLC NAME. THIS CAN EITHER BE **WITH SUB NAME**, **WITH CONNECTION NAME**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
PLC TYPE	PLC TYPE
PLC ARTICLE	PLC ARTICLE
PLC FUNCTION	PLC FUNCTION
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE. CHOOSE EITHER **PLC** OR **CONNECTED COMPONENT**.
PLC =+DESIGN.	PLC REFERENCE DESIGNATIONS. THIS CAN EITHER BE **FULL REFERENCE DESIGNATION**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT 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
I/O STATUS	SPECIFIES WHETHER THE I/O STATUS IS **INPUT** OR **OUTPUT**.
I/O STATUS TYPE	I/O STATUS TYPE — SPECIFIES WHETHER IT FOR INSTANCE IS AN ANALOGUE OR A DIGITAL I/O.
C.REF.	CONNECTION CURRENT PATH OF PLC. HERE CAN BE REFERRED TO **CURRENT PATH, PAGE**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
CC.NAME	CONNECTED COMPONENT NAME. THIS CAN EITHER BE **WITH SUB NAME**, **WITH CONNECTION NAME**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**
CC.TYPE	CONNECTED COMPONENT TYPE
CC.ARTICLE	CONNECTED COMPONENT ARTICLE NUMBER
CC.FUNCTION	CONNECTED COMPONENT FUNCTION
CC.=+DESIGN.	CONNECTED COMPONENT FUNCTIONAL AND LOCATIONAL ASPECT. THIS CAN EITHER BE **FULL REFERENCE DESIGNATION**, **FUNCTION ASPECT**, **LOCATION ASPECT** OR **PRODUCT 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 **CURRENT PATH, PAGE**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
C.SIGNAL	CONNECTED SIGNAL NAME. YOU CAN ALSO CHOOSE **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
C.WIRENO.	CONNECTED WIRE NUMBER. YOU CAN ALSO CHOOSE **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
SIGNALPATH	PLC SIGNAL PATH THROUGH TERMINALS AND CABLES. THIS CAN EITHER BE **WITH SUB NAME**, **WITH CONNECTION NAME**, **FUNCTION ASPECT**, **LOCATION ASPECT** AND/OR **PRODUCT ASPECT**.
LINE NUMBER	LINE/SUBJECT NUMBER FOR THE LIST POINT. CHOOSE BETWEEN **ON ALL LINES** AND **ON USED LINES**.

DATAFIELD	DESCRIPTION
LINE-NAME	NAME OF THE CONNECTED LINE. CHOOSE BETWEEN *AT PLC CONNECTION* AND *AT CONNECTED SYMBOL*. — SEE "ARTICLE DATA ON LINES" ON PAGE 80.
LINE-TYPE	TYPE OF THE CONNECTED LINE. CHOOSE BETWEEN *AT PLC CONNECTION* AND *AT CONNECTED SYMBOL*. — SEE "ARTICLE DATA ON LINES" ON PAGE 80.
LINE-ARTICLE	ARTICLE NUMBER OF THE CONNECTED LINE. CHOOSE BETWEEN *AT PLC CONNECTION* AND *AT CONNECTED SYMBOL*. — SEE "ARTICLE DATA ON LINES" ON PAGE 80.
LINE-FUNCTION	FUNCTION OF THE CONNECTED LINE. CHOOSE BETWEEN *AT PLC CONNECTION* AND *AT CONNECTED SYMBOL*. — SEE "ARTICLE DATA ON LINES" ON PAGE 80.

CONNECTIONS LIST DATAFIELDS

The list also contains datafields from the attached database. If you use a database from PCSCHEMATIC, the datafield names from the database are written in capitals. This makes it easy to distinguish these datafields from the other datafields. If you create your own database, it is recommended that you also write these names in capitals.

Fields from the PCSCHEMATIC database are not displayed below. You can see these in "Datafields in the component database" on page 969.

If you have created your own line datafields, these also appears in the list of connections list datafields. — See "Creating symbol datafields" on page 811 and "Line datafields" on page 83.

DATAFIELD	DESCRIPTION
SIGNAL	THE SIGNAL NAME FOR THE CONNECTION. YOU CAN CHOOSE TO INCLUDE *LOCATION ASPECT*, *FUNCTION ASPECT* AND/OR *PRODUCT NAME*.
NAME	THE NAME OF THE COMPONENT. YOU CAN CHOOSE *FROM* (THE COMPONENT IN THE "FROM" END), *TO* (THE COMPONENT IN THE "TO" END), *FROM/TO* OR *TO/FROM*. YOU CAN ALSO CHOOSE TO INCLUDE *WITH SUB NAME*, *WITH CONNECTION NAME*, *FUNCTION ASPECT*, *LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
TYPE	THE TYPE DESCRIPTION FOR THE COMPONENT. YOU CAN CHOOSE *FROM* OR *TO*.
ARTICLE NO.	THE ARTICLE NUMBER FOR THE COMPONENT. YOU CAN CHOOSE *FROM* OR *TO*.
FUNCTION	THE FUNCTION OF THE COMPONENT. YOU CAN CHOOSE *FROM* OR *TO*.
DESCRIPTION	THE DESCRIPTION OF THE COMPONENT FROM THE DATABASE. CHOOSE EITHER *FROM* (THE DESCRIPTION FROM THE COMPONENT IN THE "FROM" END), *TO* (THE DESCRIPTION FROM THE COMPONENT IN THE "TO" END), *LINE ARTICLE DATA* OR *WIRE NUMBER*.
=+DESIGN.	THE REFERENCE DESIGNATION FOR THE COMPONENT: CHOOSE *FROM* (THE COMPONENT IN THE "FROM" END) OR *TO* (THE COMPONENT IN THE "TO" END) — AND ... CHOOSE EITHER *FULL REFERENCE DESIGNATION*, *FUNCTION ASPECT*, *LOCATION ASPECT* OR *PRODUCT ASPECT*.
=+DESCRIPTION	THE DESCRIPTION OF THE REFERENCE DESIGNATION FOR THE COMPONENT: CHOOSE *FROM* (THE COMPONENT IN THE "FROM" END) OR *TO* (THE COMPONENT IN THE "TO" END) — AND ... CHOOSE EITHER *FULL REFERENCE DESIGNATION*, *FUNCTION ASPECT*, *LOCATION ASPECT* OR *PRODUCT ASPECT*.
C.NAME	THE COMPONENT CONNECTION NAME
C.FUNCTION	THE COMPONENT CONNECTION TYPE
C.LABEL	THE COMPONENT CONNECTION LABEL
C.DESCR.	THE COMPONENT CONNECTION DESCRIPTION
C.REF.	THE COMPONENT CONNECTION CURRENT PATH. HERE CAN BE REFERRED TO *CURRENT PATH*, *PAGE, FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
WIRENO	THE WIRE NUMBER FOR THE CONNECTION. THIS CAN EITHER BE WITH *FUNCTION ASPECT, LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.

DATAFIELD	DESCRIPTION
WIRENO TYPE	THE TYPE OF THE WIRE NUMBER FOR THE CONNECTION.
WIRENO ARTICLE	THE ARTICLE NUMBER OF THE WIRE NUMBER FOR THE CONNECTION.
WIRENO FUNCTION	THE FUNCTION OF THE WIRE NUMBER FOR THE CONNECTION.
WIRENO QUANTITY	THE QUANTITY OF THE WIRE NUMBER FOR THE CONNECTION.
LINE NUMBER	THE LINE OR ITEM NUMBER OF THE LIST ITEM. CHOOSE EITHER *ON ALL LINES* OR *ON USED LINES*.
LINE-NAME	THE NAME OF THE CONNECTED LINE.
LINE-TYPE	THE TYPE OF THE CONNECTED LINE.
LINE-ARTICLE	THE ARTICLE NUMBER OF THE CONNECTED LINE.
LINE-FUNCTION	THE FUNCTION OF THE CONNECTED LINE.
LINE LENGTH	WIRE LENGTH OF THE CONNECTION (LINE QUANTITY).
JUMPER LINK	— SEE "JUMPER LINKS IN LISTS AND TERMINAL PLANS" ON PAGE 94.
CB.NAME	THE CABLE NAME. YOU CAN ALSO CHOOSE TO INCLUDE *WITH SUB NAME*, *WITH CONNECTION NAME*, *FUNCTION ASPECT*, *LOCATION ASPECT* AND/OR *PRODUCT ASPECT*.
CB.W.NAME	THE CABLE WIRE NAME.

SYNTAX FOR DATABASE FIELDS IN THE DATABASE

In this appendix you can see the syntax for what you can specify in the most important database fields in the component database.

These database fields are described in this chapter:

DATABASE FIELD	APPENDIX
PCSTYPE	SEE "THE DATABASE FIELD PCSTYPE IN THE COMPONENT DATABASE" ON PAGE 961.
PINDATA	SEE "THE DATABASE FIELD PINDATA IN THE COMPONENT DATABASE" ON PAGE 963 AND "THE DATABASE FIELD PINDATA: COUNTING" ON PAGE 965.
MECTYPE	SEE "THE DATABASE FIELD MECTYPE" ON PAGE 967.

Beyond these database fields, these database fields are described here:

- The database field ACCESSORY: See "Accessories in the database" on page 676

- The database field REFID: See "Specifying Ref. Letter for the symbol in the Database" on page 679

The interpretation of the fields in the database (mapping)

How **Automation** interprets the information in the different fields in the database, is specified in the dialog box **Settings => Database => Database setup => Component data**.

As a default setting it is here for instance specified that the database field PCSTYPE contains the names of the electrical symbols, and that the database field MECTYPE contains the names of the mechanical symbols — and so on. When this appendix refers to the database fields PCSTYPE, MECTYPE etc. this is in the context that the settings ("mapping") in this dialog box has not been changed from the standard settings.

Read more about this in "Transferring article data to symbols and lines" on page 684.

General description of the database fields in the database

In "Electrical and Mechanical symbols for components" on page 663 you will find a general description of the database fields in the database.

THE DATABASE FIELD PCSTYPE IN THE COMPONENT DATABASE

Read more about the database field PCSTYPE in "Electrical symbols" on page 663.

PREREQUISITE: "DIAGRAM SYMBOL" IS SET TO PCSTYPE

IN ORDER FOR THE PROGRAM TO UNDERSTAND THAT THE PCSTYPE FIELD IN THE DATABASE CONTAINS NAMES OF ONE OR MORE ELECTRICAL SYMBOLS FOR THE COMPONENT, THE FIELD **DIAGRAM SYMBOL** MUST BE SET TO *PCSTYPE* IN THE DIALOG BOX **SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA**. THIS IS ALSO THE DEFAULT SETTING WHEN INSTALLING **Automation**.

READ MORE ABOUT THIS IN "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 684.

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**=P**#8

Options:

N	Normal
R	Relay
O	Open
C	Close
S	Switch
T	Terminal
P	PLC
X	PLC Reference

M   MasterReference (Obsolete - is converted to: *With reference* activated for the symbol, with status *Master reference*)

W   WithReference (Obsolete - is converted to: *With reference* activated for the symbol, with status *With reference*)

T,R Terminal/Relay (Obsolete - is converted to **R**)

T,O Terminal/Open (Obsolete - is converted to **O**)

T,C Terminal/Close (Obsolete - is converted to **C**)

T,S Terminal/Switch (Obsolete - is converted to **S**)

T,P Terminal/PLC Obsolete - is converted to **P**)

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 IN THE COMPONENT DATABASE

Read more about pindata in "Connection names/Pindata" on page 669 and "Addressing PLC symbols via the database" on page 672.

PREREQUISITE: "PIN NAMES" IS SET TO PCSTYPE

IN ORDER FOR THE PROGRAM TO UNDERSTAND THAT THE PCSTYPE FIELD IN THE DATABASE CONTAINS NAMES OF ONE OR MORE ELECTRICAL SYMBOLS FOR THE COMPONENT, THE FIELD **PIN NAMES** MUST BE SET TO *PINDATA* IN THE DIALOG BOX **SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA**. THIS IS ALSO THE DEFAULT SETTING WHEN INSTALLING **Automation**.

READ MORE ABOUT THIS IN "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 684.

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, I/O Status type = ***None***

OPA Output PLC connection, I/O Status type = ***Analog***

OPD Output PLC connection, I/O Status type = **_Digital_**

IP Input PLC connection, I/O Status type = **_None_**

IPA Input PLC connection, I/O Status type = **_Analog_**

IPD Input PLC connection, I/O Status type = **_Digital_**

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

Read more about counting functions in the PINDATA field in "Addressing PLC symbols via the database" on page 672.

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.++(**D**0)];}#8	=>	{++(1)[10.++(0)];}#8	Decimal
{++(1)[10.++(**H**0)];}#8	=>	Hexadecimal	
{++(1)[10.++(**O**0)];}#8	=>	Octal	
{++(1)[10.++(**B**0)];}#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,**O0**;2,**O2**;3,**O3**;……..

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 to 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;

THE DATABASE FIELD MECTYPE

Read more about the database field MECTYPE in "Mechanical symbols in the database" on page 674.

PREREQUISITE: "MECHANICAL SYMBOL" IS SET TO MECTYPE

IN ORDER FOR THE PROGRAM TO UNDERSTAND THAT THE MECTYPE FIELD IN THE DATABASE CONTAINS NAMES OF ONE OR MORE ELECTRICAL SYMBOLS FOR THE COMPONENT, THE FIELD **MECHANICAL SYMBOL** MUST BE SET TO *MECTYPE* IN THE DIALOG BOX **SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA**. THIS IS ALSO THE DEFAULT SETTING WHEN INSTALLING **Automation**.

READ MORE ABOUT THIS IN "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 684.

One symbol:

PCSMV3

Multiple symbols: ;

Specification of two different symbols:

PCSMV3;PCSMV5

Multiple symbols of the same kind: #n

Specification of 8 PCSMV3 symbols:

PCSMV3**#8**

PCSMV5;PCSMV3**#8** (1 PCSMV5 symbol and 8 PCSMV3 symbols)

Or-function with multiple drawing options: [A|B|C]

The symbol PCSMV3 and the symbol PCSMV4 or PCSMV5:

PCSMV3;**[**PCSMV4|PCSMV5**]**

States: §n

The symbol PCSMV3 displayed in state 1:

PCSMV3**§1**

Dimensions on wire tryay: #xXXmmyYYmm

A wire tray with the width 30mm and the height 40mm is specified like this:

#x30mmy40mm

These values are also used for calculating capacity in wire trays in **PCSCHEMATIC Panelrouter**.

Diamater on wires and cables: #R

A diameter of 1mm including insulation is specified like this:

#R0,5mm   ("R" is for "radius")

Distance between connections in jumper links: #X

A distance of 5mm between connections in jumper links is specified like this:

#X5mm

This value is also used as a control function for jumper links in **PCSCHEMATIC Panelrouter**.

FILE=

If the **MECTYPE** field is to short, you can type in *FILE=xxxxxx.pcs*, where *xxxxxx.pcs* is the name of the file with the **MECTYPE** contents, specified according to the syntax described above. (You have the same option for **PCSTYPE** and **PINDATA**).

The file must be placed in the database folder, which you have selected under **Settings => Directories**.

Automatic generation of symbols

The syntax for automatic generation of symbols via the MECTYPE field is described in "Syntax for automatic generation of symbols" on page 229.

DATAFIELDS IN THE COMPONENT DATABASE

In this appendix you get an overview over the created datafields in the component database, which are installed with the **Automation** software.

INTERPRETATION OF THE CONTENTS OF THE DATAFIELDS IN AUTOMATION
IN THE DIALOG BOX **SETTINGS => DATABASE => DATABASE SETUP => COMPONENT DATA** YOU CAN SPECIFY HOW **Automation** INTERPRETS THE CONTENTS IN THE FIELDS IN THE ATTACHED COMPONTENT DATABASE (HOW THE "MAPPING" IS BETWEEN THE COMPONENT DATABASE AND **Automation**).
READ MORE ABOUT THIS IN "TRANSFERRING ARTICLE DATA TO SYMBOLS AND LINES" ON PAGE 684.

Datafield: EANNUMBER

Contains the EANNUMBER for the component.

In standard installations of **Automation** the applied *Article no* is set to be fetched from this field in the database. The contents of this filed will therefore appear as the article number for components in projects.

— See "Transferring article data to symbols and lines" on page 684.

Datafield: STOCKNO

Contains the stock number for the component. The stock number can be used as an alternative article number.

— See "Alternative article number" on page 684.

Datafield: ORDERNO

Contains the order number for the component specified by the component manufacturer.

Datafield: TYPE

Contains the component type.

In standard installations of **Automation** the applied *Type* is set to be fetched from this field in the database. The contents of this field will therefore appear as the type for components in projects.

— See "Transferring article data to symbols and lines" on page 684.

Datafield: DESCRIPT

Contains the description for the component in Danish.

Datafield: UKDESCRIPT

Contains the description for the component in English.

In UK standard installations of **Automation** the applied *Description* is set to be fetched from this filed in the database. The contents of this field will therefore appear as the description for components in projects.

— See "Transferring article data to symbols and lines" on page 684.

Datafield: DEDESCRIPT

Contains the description for the component in German.

Datafield: REMARKS

General remarks and/or additional information about the component from the component manufacturer.

Datafield: CATALOGUE

Here you can specify a link to a data sheet for the component.

— Please also see "URL links" on page 696, where you can read about how to apply a so-called *alias* for easier specification of the path to data sheets.

Datafield: SOURCE

Specifies the component vendor / source of the component.

— See "Searching for components in the Database menu" on page 646 and "Selecting a Fabricate / Vendor" on page 646 to see how select components by fabricate/vendor.

— See "Sorting in parts and components lists" on page 614 to see how to sort lists by source.

Datafield: MANUFACTUR

Specifies the component manufacturer.

— See "Searching for components in the Database menu" on page 646 and "Selecting a Fabricate / Vendor" on page 646 to see how select components by fabricate/vendor.

Datafield: UNITPRPACK

Specifies "the number of units in each package".

— See "Correct amount of jumper links in parts lists" on page 94.

Datafield: PRICE

Contains the list price for the component.

Datafield: DISCOUNT

Contains a discount factor — if any — for the component. The discount factor can by included in the calculation for some datafields in parts/component lists.

— See "Parts/components list datafields" on page 949.

Datafield: NETPRICE

Specifies the net price for the component.

Datafield: TAX

Is not used normally.

Datafield: WEIGHT

Specifies the weight of the component.

Datafield: PCSTYPE

Specifies which electrical symbol(s) that represent the component in the diagram drawings.

— Read more about this in "Electrical symbols" on page 663.

Datafield: PINDATA

Contains the pindata for the component.

— Read more about this in "Connection names/Pindata" on page 669.

Datafield: MECTYPE

Specifies which mechanical symbol(s) that represent the component on the mechanical drawings.

— Read more about this in "Mechanical symbols in the database" on page 674.

Datafield: HEIGHT

Specifies the average height for the connection terminals for the component. The height is entered in mm, without specification of the unit of measurement. The datafield is used in conjunction with the **PCSCHEMATIC Panelrouter**.

Datafield: SINGLELINESYMBOL

Specifies single line symbols — if any — for the component.

— See "Single line diagrams" on page 473.

Datafield: INSTTYPE

Used for installation symbols on ground plan / mechanical pages.

Datafield: OPT_ACCESSORY

Here you can specify which article number(s) — if any — that can appear as optional accessories in the dialog box **Component data**.

— See "Optional accessories - Mechanical" on page 689.

Datafield: OPT_ADDON

Specifies the article number(s) for optional addon ("electrical accessories") — if any — for the component.

Optional addon appears in **Show available** on both diagram and arrangement drawings.

— See "Optional accessories - Electrical" on page 689.

Datafield: ACCESSORY

Here you can specify the article number(s) for fixed accessories — if any — for the component.

Fixed accessories appear alone on parts and components lists. In lists fixed accessories appear under the component it is accessories for.

— See "The Accessory tab" on page 184 and "Fixed accessories" on page 689.

Datafield: PICTURE

Can contain a link to a picture. The picture is for instance displayed in the dialog box **Component data**.

Datafield: TABLECODE

Specifies which component group the component belongs to. The component groups are displayed in the **Database menu**.

— See "About the Database menu" on page 645, "Choosing a folder in the article menu" on page 646 and "Choosing a database Menu" on page 693.

Datafield: REFID

Here you can specify the reference id for the component.

— See "Specifying Ref. Letter for the symbol in the Database" on page 679.

Datafield: SUBDRAWING

This datafield makes it possible to include subdrawings as components in the database. When you have created a subdrawing in **Automation**, you specify the path (alias + file name) to the subdrawing file in the SUBDRAWING field. When you then place a symbol in the pickmenu, and from this symbol in the pickmenu link to the article number for the subdrawing, you can subsequently insert the subdrawing in projects directly from the pickmenu.

— See "Placing symbols in the pickmenu" on page 203.

Datafield: RATING

Can contain specification of component ratings. The information can be displayed next to the electrical and mechanical symbols for the component, if these are created with an inserted Datafield of the type **Symbol Datafields => DB[RATING]**.

Datafield: APPROVED

Here you can specify whether a component is approved.

— See "Approved and Obsolete components in the database" on page 691.

Datafield: OBSOLETE

Here you can specify whether a component is obsolete.

— See "Approved and Obsolete components in the database" on page 691.

Datafield: CWCODE

This datafield is not meant for users. The datafield is used by the **Component Wizard** for creating components in the PCSCHEMATIC component database, and specifies the component type.

— Read more about the **Component Wizard** this in the **Component Wizard** manual, which can be downloaded at www.pcschematic.com/manuals.

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 designations

You can insert individual variables in all aspects in reference designations. To insert a variable in an aspect designation, enter the dialog box **Reference desig-nations** and create an aspect designation, in which you insert a variable — for instance **+23/@3** in a position aspect.

When you then want to insert an aspect designation with an attached variable on either a symbol, a reference frame or a page, just do this by selecting the relevant aspect designation from either the **Component data** dialog box, the **Reference frame data** dialog box or the **Page data** dialog box.

VARIABLES IN REFERENCE DESIGNATIONS - IN PREVIOUS VERSIONS

IN VERSION 18 AND PREVIOUS VERSIONS OF **Automation** YOU HAD TO HANDLE VARIABLES IN REFERENCE DESIGNATIONS DIFFERENTLY.

IN ORDER TO HANDLE PROJECTS CREATED IN PREVIOUS VERSIONS IT IS STILL POSSIBLE TO USE THE OLD PROCEDURE, BUT WE STRONGLY RECOM-MEND TO USE THE NEW PROCEDURE DESCRIBED ABOVE.

IN PREVIOUS VERSIONS OF **Automation** YOU HAD TO DO THE FOLLOWING TO USE VARIABLES FOR CONTROLLING REFERENCE DESIGNATIONS ON REF-ERENCE FRAMES:

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)

IN PREVIOUS VERSIONS OF **Automation** YOU HAD TO DO THE FOLLOWING TO USE VARIABLES FOR CONTROLLING REFERENCE DESIGNATIONS ON PAGES:

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 var-iable (e.g. /@7) in the *Name* field

---

STRUCTURE FOR PROJECT DEFINITION FILES

A project definition file consists of three parts, of which the first two are optional.

Part 1: Specifying aliases / applied folders (Optional)

The settings in this section, corresponds to the settings which can be set from the dialog box **Options for project generator**, in the tab **Generate/update** project.
— See "The Generate/update project tab" on page 23.

FOLDERS	
Name for Alias 1 E.g.: **FOLDER1**	Path to the *first* folder to examine for drawing templates, sub drawings and projects. E.g.: **C:\PCschematic\MySubDrawings**
Name for Alias 2	Path to the *second* folder to examine for drawing templates, sub drawings and projects.
Name for Alias 3 (Relative to other alias)	E.g.: **%FOLDER1%\SubDrawings** — Can both be relative to aliases defined in the **Project generator** dialog box, and to aliases defined on a previous line here in the project definition file.
...	...
GENERATOR OPTIONS	
UpdateLists=yes	Options: 'yes' / 'no'
UpdateFromDB=yes	Options: 'yes' / 'no'
AutoNumSymbolNames=yes	Options: 'yes' / 'no'
PLCSync=FromPLCref	Options: 'none' / 'FromPLCref' / 'ToPLCref'
MakeInvisibleUnrefSignals=yes	Options: 'yes' / 'no'
MakeInvisibleUnrefCables=yes	Options: 'yes' / 'no'

Part 2: Specifying Project data and/or Reference designations
Please note that this is optional.

PCSCHEMATIC DATA	
PROJECTDATA	
Datafield from Project data	Value for datafield
Datafield from Project data	Value for datafield
...	...
REFERENCE DESIGNATIONS	
FUNCTION	
Function aspect	Description
Function aspect	Description
...	...
LOCATION	
Location aspect	Description
Location aspect	Description
...	...
PRODUCT	
Product aspect	Description
Product aspect	Description
...	...

Part 3: Specifying Drawing templates, Sub drawings and Projects

PCSCHEMATIC PAGES				
TEMPLATE	PAGES			
Multipage project	Page no. / Empty			
Drawing template / Single-page project	Page no. / Empty	@1	@2	...
Sub drawing	~	@1	@2	...
Project / Page	#REMAIN			

When Updating projects

NEWPAGES>>				
Multipage project	Page no. / Empty			
Drawing template / Single-page project	Page no. / Empty	@1	@2	...
Sub drawing	~	@1	@2	...
Project / Page	#REMAIN			
<<NEWPAGES				

When Merging projects
MERGE

Project 1

Project 2

Project 3

...

Project n

DICTIONARY
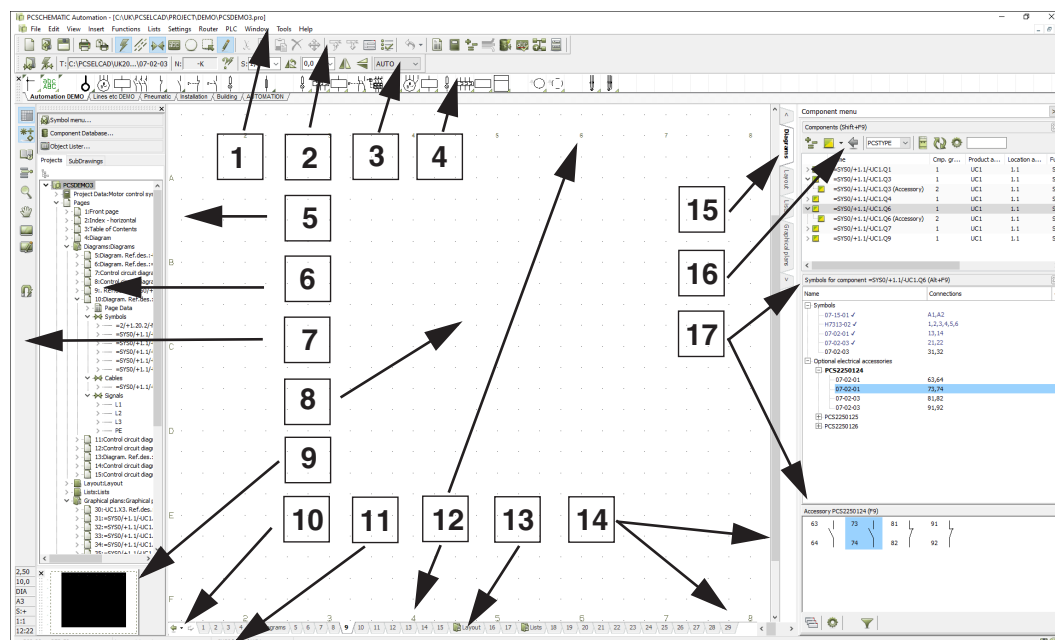
Activate next (in lists): The first time the program meets a check mark in an *Activate next* check box, you tell the program to collect information about the first component in the list. Subsequent datafields, such as *Description* and *Quantity*, will then be regarded as the description and quantity 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 831 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 136. 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 980.

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 779 for more information.

Cross hairs: When you are working on a project drawing, the location of the mouse pointer i 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 299.

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 820.

Read more about how to create lists in "Creating Lists" on page 831.

Read more about how to create unit drawings in "Creating Unit Drawings" on page 846.

And, finally, you can read about how to insert an already created drawing header in "Insert drawing headers/formulas" on page 599.

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 546.

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 980. Described in further detail in "The Explorer window: The Projects tab" on page 345.

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.

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 111.

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 48.

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 980.

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.

PCSCHEMATIC Automation

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 980.

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 136.

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 79 and "Signal symbols" on page 234.

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:**Pcselcad**\Project\project.pro.

Pickmenu: A place you can place symbols for easy access while working. Point number **4** on the figure on page 980. Described in further detail in "Fetching symbols from the Pickmenu" on page 155.

PC|SCHEMATIC

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 980.

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 775 and in "Specifying text properties / Appearance of text" on page 136.

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 816.

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 980.

Sub drawing: See "Sub drawings" on page 299.

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 980. Read more in "The Survey window" on page 46.

Tanking: See "Tanking licenses" on page 877.

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 980.

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

Automation

Automation

PC | SCHEMATIC

Automation

Automation