

ePLAN[®] 5

News
04/ 2003

Version 5.50



Copyright © 2003 by EPLAN Software & Service GmbH & Co. KG

EPLAN Software & Service GmbH & Co. KG assumes no liability either for technical or printing errors nor for deficiencies in this technical information and cannot be held liable for damages that may result directly or indirectly from delivery, performance, and use of this material.

This technical information includes information on the basis of a proprietary system that is protected by copyright laws. All rights reserved. These information or part thereof may not be copied or reproduced by any other means without the previous permission of EPLAN Software & Service.

The software described in this technical information is subject to a license agreement. The software may only be used and copied within the scope of this agreement.

EPLAN[®] is a registered trademark of EPLAN Software & Service GmbH & Co. KG.

MS-DOS[®] / Windows NT[®] / Windows 2000[®] / Windows XP[®] / Microsoft Windows[®] are registered trademarks of the Microsoft Corporation.

Microsoft[®] Excel[®] and Microsoft[®] Access[®] are registered trademarks of the Microsoft Corporation.

RITTAL[®] is a registered trademark of the Rittal Werk Rudolf Loh GmbH & Co. KG.

Clip Project[®] is a registered trademark of the Phoenix Contact GmbH & Co.

INTERBUS[®] is a registered trademark of the Phoenix Contact GmbH & Co.

ÖLFLEX[®], ÖLFLEX-SERVO[®] and ÖLFLEX CLASSIC[®] are registered trademarks of the U.I. LAPP GmbH.

AutoCAD[®] and Volo[™]View Express are registered trademarks of Autodesk, Inc.

SIMATIC HW Konfig[®] is a registered trademark of the Siemens AG.

SCAN[®] is a registered trademark of McAfee Associates.

PKZIP/PKUNZIP[®] are registered trademarks of PKWARE, Inc.

ObjectStore[®] is a registered trademark of eXelon Corporation.

D-ISAM[®] is a registered trademark of Informix Software Inc.

Hardlock E-Y-E[®] is a registered trademark of Aladdin Knowledge Systems, Ltd Int.

InstallShield[®] is a registered trademark of InstallShield, Inc.

PMS[®] is a registered trademark of PMS Compelec GmbH.

MicroStation[®] is a registered trademark of Bentley Systems.

All other product names, trade names and company names mentioned are trademarks or registered trademarks of their respective owners.

Note: The hardware requirements indicated by the operating system manufacturers must be considered. Hardware combinations that cause malfunctions according to the manufacturer's specification may also have an effect on the operation of EPLAN. Therefore an accurate basic installation of hardware, operating system, and hardware drivers (e.g. graphics board driver etc.) is essential for a smooth operation of EPLAN. For this reason, the main conditions and areas of application as specified by the hardware manufacturers also apply to the operation of EPLAN.

Table of Contents

1	Preface	7
2	Notes on Reading	9
3	New Terms in EPLAN 5.50	11
4	Redesign of the Graphics Editing Module.....	13
4.1	Working Window	15
4.2	Page Overview	17
4.3	Message Window.....	18
4.4	New Menu Design	19
5	New and Changed Functions of the Page Overview	25
5.1	New Icons on the Page Toolbar.....	26
5.2	Creating a New Page.....	27
5.3	Opening Pages	29
5.4	Saving a Page.....	29
5.5	Closing Pages and Exiting Graphics Editing	30
5.6	Copying Pages.....	30
5.6.1	Copying Pages within a Project.....	31
5.6.2	Copying Pages Into / From Another Project	32
5.7	Renaming and Moving Pages	34
5.7.1	Renaming/Moving Pages by Making an Entry in Tree View	34
5.7.2	Renaming/Moving Pages via the Menu Option.....	35
5.8	Positioning New Identifiers in the Higher-level Assignment and Location Designation Database	36
5.9	Editing and Displaying Page Properties.....	37
5.10	Reading in a Backup Page.....	40
5.11	Deleting and Emptying Pages.....	41
5.12	Browsing through the Pages in the Working Window.....	42
5.13	Page Preview	42
5.14	Filtering Pages	44
5.15	Updating a Filter.....	47
5.16	Printing Pages.....	47
5.17	Exporting Pages	48
6	Block Editing	49
6.1	Selecting/Deselecting Schematic Objects.....	49
6.2	Calling up and Using the Block Editing Function.....	57
6.3	Understanding Object-related Differences for Editing Blocks.....	59

7	Including External Documents	67
7.1	Creating Pages of the External Document Type	67
7.2	Opening External Documents	69
7.3	Editing Pages of the External Document Type	69
8	Inserting Hyperlinks on Project Pages	71
9	Designing Schematics in Single-pole Display	73
9.1	Setting the Page Type	73
9.2	Using Symbol Files	74
9.3	Special Points when Inserting Symbols in Single-pole Display	76
9.4	Special Points when Using Macros in Single-pole Display	77
9.5	Special Points when Entering Parts Data in Single-pole Display	78
9.6	What to Consider when Generating Interruption Point Cross-references.....	79
9.7	What to Consider when Inserting Wire Designations on Pages in Single-pole Display	79
9.8	Search Function.....	79
10	Online Data Transfer when Copying Schematics Pages	81
10.1	Transferring Interruption Point Cross-references Online	81
10.2	Transferring Contact Cross-references Online	81
10.3	Transferring Online PLC Cross-references	82
10.4	Transferring Device Designations Online	82
11	Moving Symbol Texts	87
11.1	Moving Symbol Texts via Menu Command	89
11.2	Moving Symbol Texts by Specifying Defined Coordinates	90
11.3	Displaying the Relocation of Symbol Text	91
11.4	Undoing the Symbol Text Move	93
12	Moving Strip Designations and Connection Numbers	95
12.1	Moving Strip Designations/Connection Numbers by Specifying Defined Coordinates	96
12.2	Displaying the Move of the Strip Designations and the Connection Numbers	97
12.3	Undoing the Move of the Strip Designation and the Connection Number	98
12.4	Examples	99
13	Evaluating and Editing Messages	103
13.1	Marking Messages as Done / Undone	103
13.2	Selecting the Display according to Message Types	104
13.3	Displaying the Device Causing the Message in the Schematic	104
13.4	Displaying the Second Position of a Device in the Schematic	105
13.5	Sorting Messages	105

14	Drawing Polylines	107
15	Adjusting the Function Text for Distributed Components	109
16	Redesigned Help System	113
17	Creating the Table of Contents.....	115
17.1	New Location in the Menu	115
17.2	Changed Parameters for the Table of Contents	115
18	Designing Mounting Panels.....	119
18.1	Functions of the Panel Layout.....	119
18.2	Creating Component Macros (First Step)	120
18.2.1	Creating a Combined Graphics Object	120
18.2.2	Creating a Combined Macro	122
18.3	Entering Component Macros into the Parts Master Data (Second Step)	122
18.4	Assigning Part Numbers to a Device (Third Step)	123
18.5	Generating the Equipment File (Fourth Step)	124
18.6	Inserting the Mounting Panel (Fifth Step)	124
18.7	Equipping the Mounting Panel (Sixth Step)	125
18.8	Configuring the Placeholder Texts (Seventh Step)	130
18.9	Creating Legends (Last Step)	131
18.9.1	Creating a Window Legend	132
18.9.2	Outputting a Page Legend	133
18.10	Updating Mounting Panel Data	134
18.11	Editing Special Text Coordinates in the Legend Form.....	135
19	Information Management	137
19.1	General.....	137
19.2	Expansion of the User Data.....	137
19.3	Cases of Conflict.....	138
19.4	Structure of the Message.....	139
19.5	Informing Other Users by E-mail.....	139
19.6	Confirming a Message.....	140
19.7	Updating the Status of Processing	140
20	Using Long File Names	141
21	Outputting Wiring Lists.....	143
21.1	New Menu Item in the Cross-reference Module.....	143
21.2	Outputting Wiring Lists as Graphics Pages.....	144
22	Expansion of the Foreign-language Functions.....	147
22.1	Excluding Texts from the Translation.....	147
22.2	Generating Line Feeds in a Text-oriented Way during the Translation.....	149
22.3	Having an Influence on how the Texts are Taken over during the Translation.....	150

23	Standard Labeling	151
23.1	Devices	151
23.2	Wirings.....	152
23.3	Cables.....	153
23.4	Conductors.....	154
23.5	Terminals	155
24	Changing the Drive during Project Selection.....	157
25	New and Modified Parameters.....	161
25.1	Project-specific Parameters	161
25.2	Personal Parameters.....	164
25.3	Workstation-specific Parameters	165
26	Further Extensions, Modifications and Corrections	167
26.1	Graphics Editing	167
26.1.1	Editing Symbols in a Symbol Project.....	167
26.1.2	Search function.....	167
26.2	Cross-reference Module	168
26.3	Terminal and Interconnect Diagram Module	170
26.4	Translation Module.....	170
26.5	Data Backup.....	170
26.6	Terminal Line-up Diagram.....	171
26.7	Conversion of HPGL files	171
26.8	Message Management	172
26.9	Extension of the Form-controlled Output for the Printing of Interconnect Diagrams.....	172
26.10	Outputting Lists	173
26.11	New Viewer for HTML/DWF Documents.....	173
26.12	Program Corrections.....	174
27	New Add-on Programs	175
27.1	SIMATIC HW Config	175
27.2	EPLAN Project Reference.....	176
28	New and Modified Master Data	179
28.1	Designing in A1 Format.....	180
28.2	New Symbol Files	180
28.2.1	Single-pole Display	180
28.2.2	DIN EN61346-2	180
28.3	Extended DIC_WUPD.SYM Symbol File.....	180
28.4	Expansion of the Standard Labeling	181
28.5	Extended Cable Types.....	181

1 Preface

Dear User,

With EPLAN Version 5.50 we are pleased to present a new system with which you can fulfill your requirements faster and better. To achieve this we have implemented new developments in almost all the fields. The new version thus sets new yardsticks both in the field of handling as well as functionality and integration compatibility. The redesign of the user interface opens up new possibilities in the fields of engineering. The multiple-window feature means that you can access any number of pages And that you can navigate comfortably and rapidly through the entire project by parallel use of the page structure and further message windows.

An enormous saving in time is possible when setting up new projects – You can access the complete structure of the higher-level assignments and location designations, which you can either use completely or adapt certain segments to your requirement – all of which in a single step. Whereby the device designations and all the cross-references are updated completely on-line. Generation runs are a matter of the past. A saving in time which makes itself paid.

Block functions mean that you only have to enter the same properties once for many individual elements. This also saves time and prevents errors during the input.

Future-oriented technologies, such as the use of Open DWG, allow the direct import of design drawings in the form of DXF and DWG.

The openness of the system is illustrated once again by the multitude of possibilities of integrating all the documents belonging to a project into EPLAN. With the Version 5.50 EPLAN becomes the "switching point" of the complete project and plant design system. This ensures that you have control over all the data – even beyond the electrotechnical documentation – all the time.

The pioneering new technology of single-pole display provides not only an optimization of data and space requirements but also new fields of application, for example installation technology as well as high-voltage and medium-voltage technology, and furthermore allows enormously flexible engineering. It is possible to carry out parallel development and production even when all the detailed information is not yet available. This ensures that plant overviews and evaluations, such as bills of materials and overview diagrams, are already available early on. Steps which could only be carried out sequentially in the past can now be carried out in parallel – thus saving time and costs.

Mounting panel management has been revised completely, thus leading to more comfort, in particular through free design and positioning of the legends by the use of individual forms, and in particular to the flexibility and information which you as a user expect from a system adapted to your requirements. Sections which have been completed can be included in the production at an early stage since the legends contain all the required bill-of-materials information. Here again the optimization potential through parallel engineering is enormous.

We are sure that the present document will be of great help to you. In order to optimize your use even further we also provide a concise and, at the same time, extensive range of training.

Ensure that you maintain your advantage over your competitors by availing yourself of our qualified courses to get to know and use to the full extent the wide range of new features.

Best regards

Martin Witte

EPLAN Product Manager

2 Notes on Reading

Before reading the EPLAN News, please note the following:

Bold text is used to indicate information that is **particularly important** and that you must note. Furthermore, it marks important dialog elements. Menus, menu options, and menu paths are shown in **bold** in the text.



Texts preceded by this graphics are warnings. Read this information before continuing!



Texts introduced by this graphics and shown within these framing lines signify additional information, for example, interdependencies or supplementary information.

Several consecutive notes are indicated by the following line:

Notes:



Examples are introduced by this graphics symbol and are printed in italics.

Code examples, directory names, direct entries etc. are printed in a non-proportional font.

3 New Terms in EPLAN 5.50

The continuous improvement process in EPLAN not only incorporates the implementation of new functions and function expansions, but also the design of an easy-to-use and convenient user interface. From time to time, this requires to modify the EPLAN terminology, i.e. the terms used in EPLAN.

EPLAN version 5.50 will introduce two new terms: First, the terms "potential arrow" / "potential cross-reference" were changed, and, second, the term "polyline" in the free graphics menu was altered.

We have changed these terms as follows:

<i>Previous term</i>	<i>New term</i>
Potential arrow	Interruption point
Potential cross-reference	Interruption point cross-reference
Polyline	Set of lines

Therefore please note that you no longer insert a potential arrow but an **interruption point** into the schematic. Furthermore, you do not generate a potential cross-reference, but an **interruption cross-reference**. In previous EPLAN versions, the German term "Linienzug" had been translated as "Polyline". Since in EPLAN version 5.50 the German term "Polylinie" was added to the Free graphics menu, which had to be translated as "Polyline", we had to change the translation for "Linienzug" to "**Set of lines**".

4 Redesign of the Graphics Editing Module

We have redesigned the graphics editing module in EPLAN in order to make it more user-friendly and efficient. In addition to the modified page selection and management as well as to the redesigned menus, you will notice the new three-window structure of the graphics editing module.

By default, EPLAN now displays the page overview and the message window, which we also altered, next to the main window of graphics editing. At first, the actual working window is gray, since no page has been opened yet. When you open a page (see Chapter 5.3 on Page 29), a working window is displayed for this page (see Fig. 1).

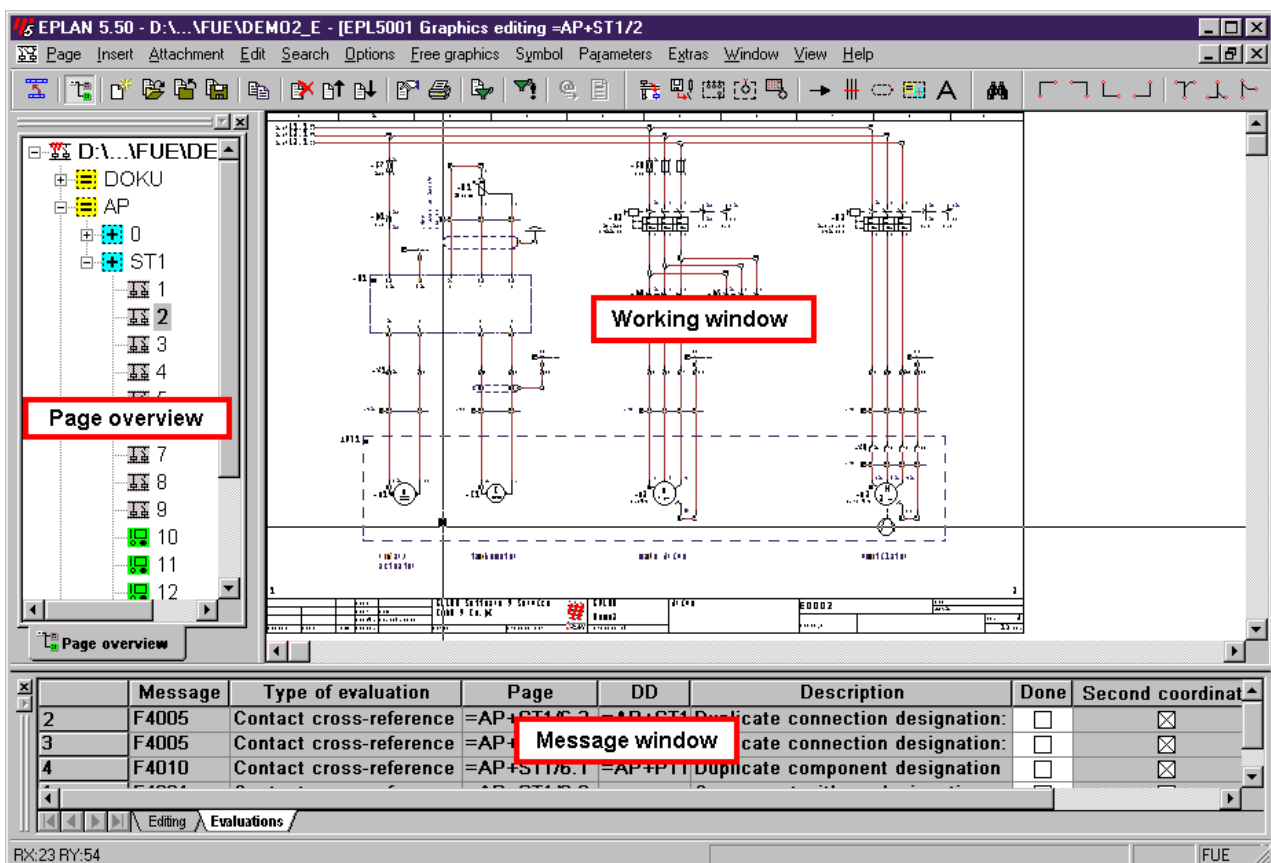


Fig. 1: New, three-window structure in graphics editing

The page overview window as well as the message window are separate "dockable" windows. This means that these windows can be freely positioned inside and outside the EPLAN main window, just like EPLAN toolbars. Within the main window, you can place the dockable windows at the window margins.

If you wish to use the whole screen of the main window for page editing, just close the two dockable windows. To do so, simply click the  button of the respective window, or select **Messages** from the **View** menu or **Page overview** from the **Page** menu. To redisplay the windows on your screen, select the corresponding menu options again.

An icon in front of the respective menu option indicates whether the corresponding dockable window is shown on your screen (see Fig. 2).



Fig. 2: Activated **Messages** or **Page overview** menu option

When exiting the graphics editing module, EPLAN saves the last screen setting. This way, you can immediately start working in the desired graphics editing setting when starting graphics editing the next time.

Switching between the individual windows (page overview, working, and message window) is done by a mouse click, by selecting **Next window** from the **View** menu, or by pressing  . If only one window or no window is displayed, this menu item or shortcut has no effect at all.

The following sections provide additional details of the various windows of graphics editing.

4.1 Working Window

As a new feature in EPLAN, you can now also open several working windows at the same time (see Fig. 3). The working windows are default windows offering the familiar Windows® functions such as Move, Minimize etc. In Chapter 5.3 on Page 29 you will learn how to open several working windows.

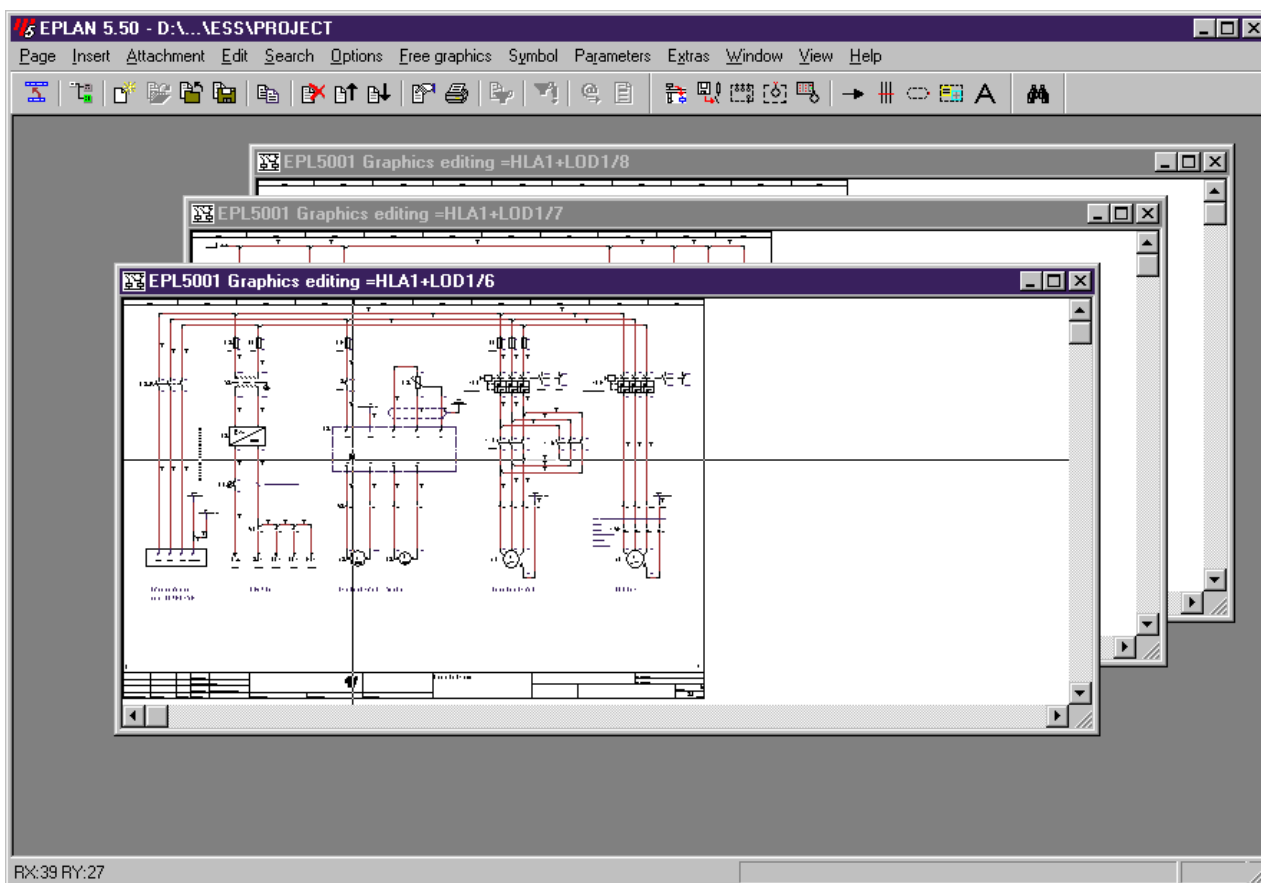


Fig. 3: Graphics editing with several working windows

Though it is possible to open several working windows at the same time, pages can be edited in **only one window** at a time. The window containing the page in which the current cursor movements and functions are carried out is called the "active window". You can recognize the active working window by its colored title bar.

If you have modified a page and want to make a change on the second page, EPLAN will first ask you whether you want to save the modifications made on the previous page. This is the case when you open the second page in the same working window as well as when you switch to another working window to edit the second page there. In EPLAN 5.50, you can recognize a modified page by the asterisk shown in the title bar of the working window (see Fig. 4).



Fig. 4: Title bar of a working window for a modified page

If you want to switch from a working window to another open working window you can simply press **Ctrl** + **Alt** + **W**. It is now also possible to select the desired window via the **Window** menu (see Fig. 5). All open working windows are listed in this menu. The active, topmost window is marked by a preceding checkmark (✓). Here again, a modified page is marked with an asterisk (see the highlighted third menu entry in the figure below). If more than 9 working windows or pages are open, you can select the **More Windows...** option to call up a dialog box in which all open pages are available.

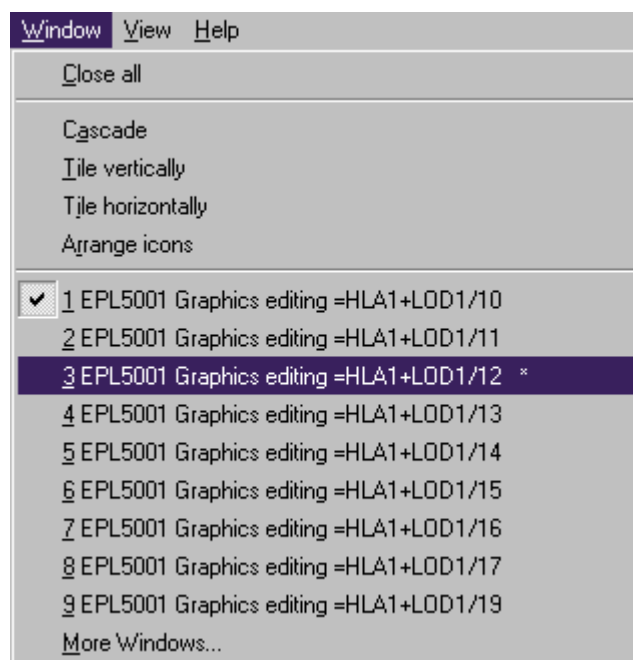


Fig. 5: **Window** menu showing a list of the open pages

To close all open working windows at one go, select the new **Close all** option from the **Window** menu.

Unlike previous EPLAN versions, EPLAN 5.50 now also allows projects without pages. If, for example, you call up the graphics editing module of a newly created project, the previous prompt whether you want to copy or create pages no longer appears.

4.2 Page Overview

In EPLAN 5.50, the window for selecting and managing project pages is no longer called Page selection, but **Page overview**. This page overview is no longer a page dialog box that is divided into two sections (tree view on the left and page properties on the right-hand side of the dialog), but a separate dockable window, that only displays the tree view (see Fig. 6).

"Pinnable" dialog boxes for displaying and editing the page properties (see Chapter 5.9 on Page 37) and a "pinnable" page preview window (see Chapter 5.13 on Page 42) are now available to get quick information on the pages selected.

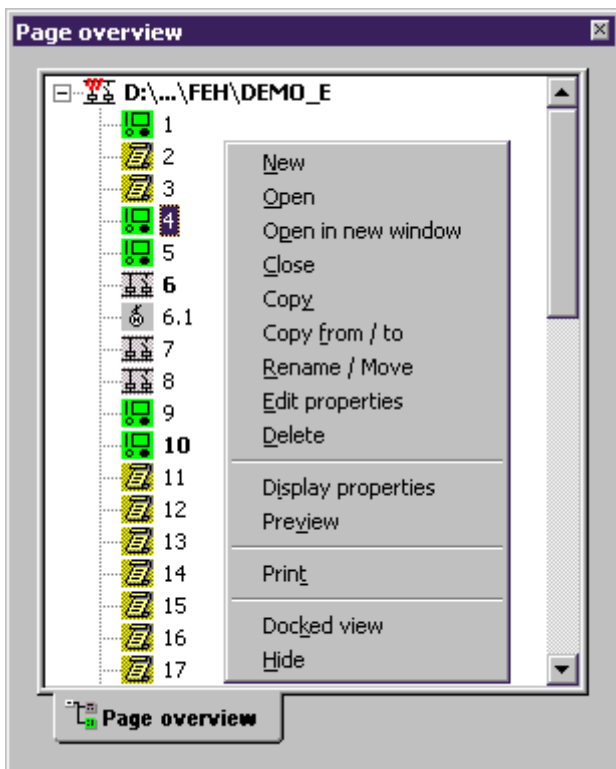


Fig. 6: Page overview as dockable window with pop-up menu

It is now also possible to call up a pop-up menu in the page overview. A simple click with the right mouse button will provide you with the most important functions of the page overview. In addition to the above-mentioned possibilities of exiting the page overview (see Chapter 4 on Page 13), you can close the window by selecting the **Hide** menu option from the pop-up menu.

You can also configure EPLAN to exit the page overview after every action. The corresponding new Close page overview after action parameter can be found in the working parameters (menu path in parameter management: **Personal > Working parameters -> Working parameters 1 tab**, see Fig. 7).

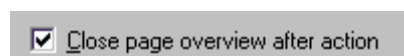


Fig. 7: Working parameters for the page overview

4.3 Message Window

The "old" message window and the dialog for the evaluation messages have been combined into one dockable message window in EPLAN Version 5.50. This message window is structured as a table (see Fig. 8), which is familiar to you e.g. from the Number device dialog box. As already mentioned in Chapter 4 on Page 7, the new message window can be activated/deactivated via the **View > Messages** menu path.

	Message	Type of evaluation	Page	DD	Description	Done	Second coordinate
1	W4014	Contact cross-reference run	=HLA1+LOD1/8.0	=HLA1+	Unnumbered DD	☐	☐
2	W4014	Contact cross-reference run	=HLA1+LOD1/8.0	=HLA1+	Unnumbered DD	☐	☐
3	W4014	Contact cross-reference run	=HLA1+LOD1/8.0	=HLA1+	Unnumbered DD	☐	☐
4	W4014	Contact cross-reference run	=HLA1+LOD1/8.1	=HLA1+	Unnumbered DD	☐	☐
5	W4014	Contact cross-reference run	=HLA1+LOD1/8.1	=HLA1+	Unnumbered DD	☐	☐
6	Filter	Errors	n	=HLA1+LOD1/6.7	=HLA1+ Contact with no coil	☐	☐
7	Jump to DD	Warnings	n	=HLA1+LOD1/6.7	=HLA1+ Contact with no coil	☐	☐
8	Jump to second coordinate	Notes	n	=HLA1+LOD1/6.7	=HLA1+ Contact with no coil	☐	☐
9	Set to 'done' / reset	Done	n	=HLA2+LOD2/6.3	=HLA2+ Contact with no coil	☐	☐
10	W4025	Contact cross-reference run	=HLA2+LOD2/6.6	=HLA2+	Contact with no coil	☐	☐
11	W4025	Contact cross-reference run	=HLA2+LOD2/6.6	=HLA2+	Contact with no coil	☐	☐
12	W4025	Contact cross-reference run	=HLA2+LOD2/6.6	=HLA2+	Contact with no coil	☐	☐

Fig. 8: Redesigned message window in graphics editing

The **evaluation messages** which were previously called up via the menu path **Edit > Messages** are now displayed on the Evaluations sheet. Messages that, during editing, were displayed in the "old" message window (menu path: **Options > Display message window**) in previous program versions, now appear on the Editing sheet.

For detailed information on how to evaluate and process messages, see Chapter 13 on Page 103.

4.4 New Menu Design

Due to various modifications we had to redesign many menus of graphics editing. This section will give you a small overview of the most important modifications.

The **Page** menu has been restructured because of the new page overview (see Chapter 4.2 on Page 17).

The following menu items are new: **Page overview**, **Open in new window**, **Copy from / to**, **Empty**, **Display properties**, **Preview**, **Filter**, **Update filter**, and **Return to main menu**. The former **Properties** option is now the **Edit properties** option.

As announced in the "News 5.40", the HPGL output and thus the **HPGL plot** menu item were removed. In addition, the menu options have been regrouped (the basic items are shown first, afterwards the options for page editing, etc.) and some icons have been changed (see Chapter 5.1 on Page 26). For more information on the new and modified menu items, please refer to Chapter 5, starting on Page 25.

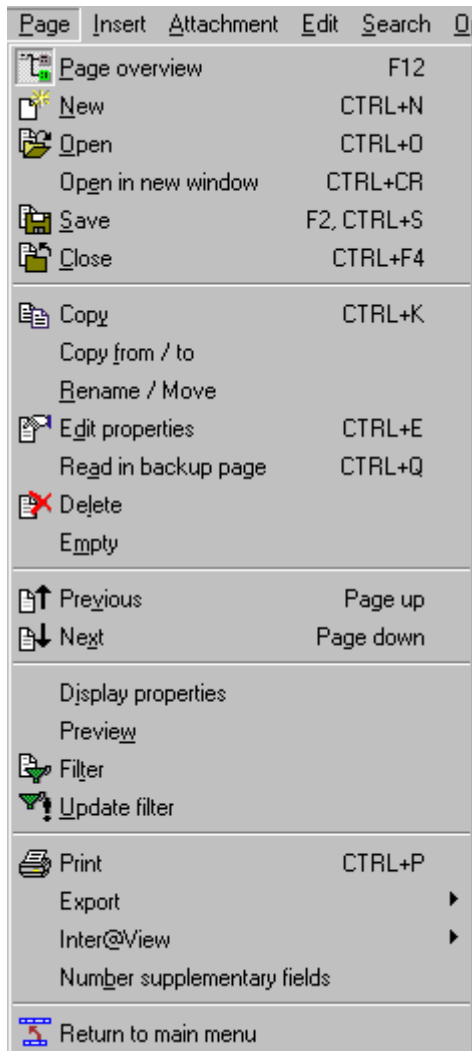


Fig. 9: **Page** menu with new and modified menu options

In EPLAN 5.50, the previous **Draw** menu is called **Insert** (see Fig. 10). Consequently, we have renamed the **Insert symbols** and **Insert macro** options to **Symbols** and **Macros**. Due to the changes made in the EPLAN terminology, the previous **Potential arrows** menu item is now called **Interruption points**. The **Hyperlink** menu item was newly inserted, allowing you to create links to other documents on your project pages (see Chapter 8 on Page 71).

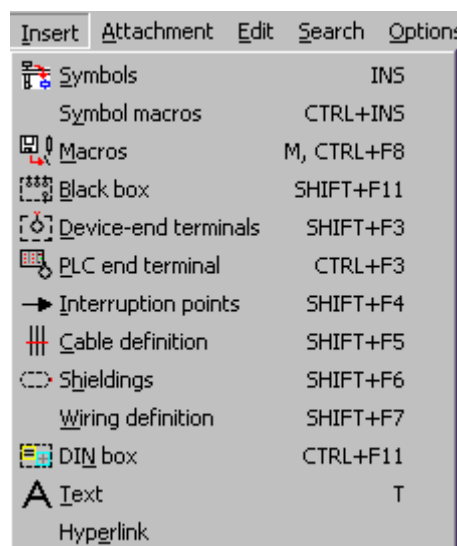


Fig. 10: Renamed **Insert** menu

Please note that we have added further editing functions to the graphics editing of Version 5.50. For this reason, the **Edit** menu now contains several new menu options (see Fig. 11 on Page 21).

If you wish to edit devices and other schematic symbols in blocks, you can now use the **Block editing** option (see Chapter 6 on Page 49). **Open hyperlink** allows you to jump to a connected document stored previously by you via **Insert > Hyperlink** on a project page (see Chapter 8 on Page 71).

Furthermore, the **Change size** option was added to the **Edit** menu.

Using another new menu item **Adjust function text** you can adapt function text at distributed components. For detailed information, please refer to Chapter 15 on Page 109.

The previous **Move DD** option was renamed **Move symbol texts** in Version 5.50. We also changed the sequence of this function. In addition, you can also move strip designations (terminal and connector strip designations) and connection numbers (terminal and connector connection numbers) now. For this purpose, the **Edit** menu now offers the new **Move strip designation** and **Move connection number** options. For detailed information on these shift functions, please see the Chapters 11 and 12 starting on Page 87.

The **Window** menu option was also renamed and is called **Select** in EPLAN 5.50.

It is no longer possible to open the message dialog for the evaluation messages via the **Messages** option. Instead, the new submenu contains several functions for editing the new message window (see Chapter 13 starting on Page 103).

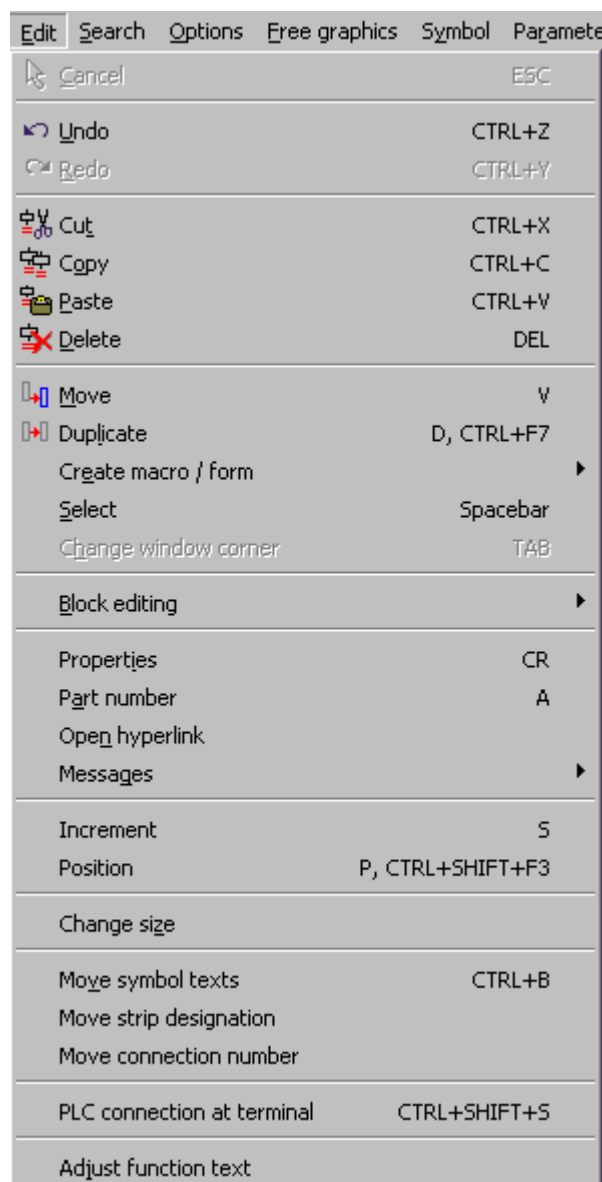
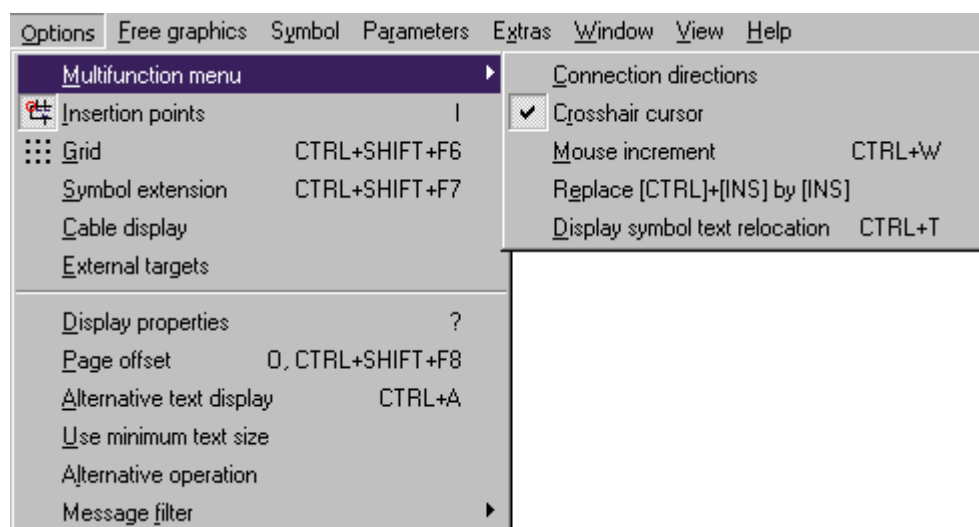


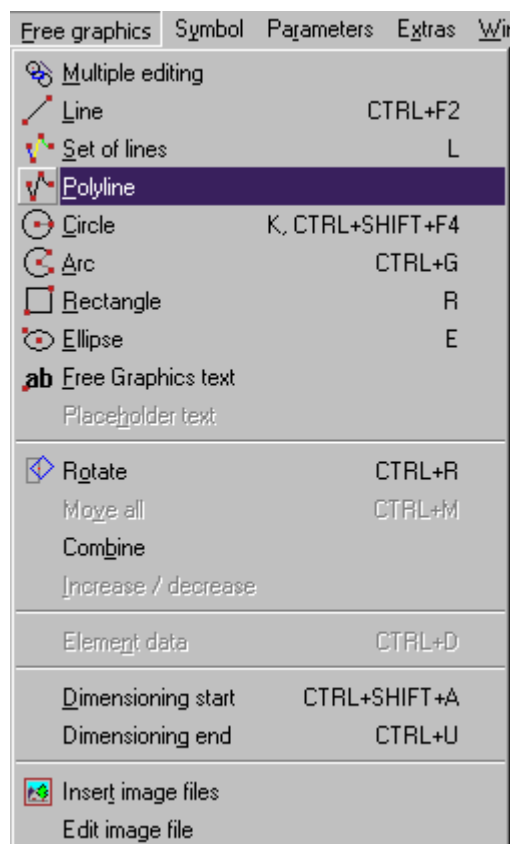
Fig. 11: **Edit** menu with new and renamed menu options

We have also redesigned the **Options** menu including the **Multifunction menu** (see Fig. 12 on Page 22). The **Point and/or target wiring**, **Only target wiring**, **Save cursor position**, and **Display message window** options were removed.

The **Cable display** and **External targets** options are no longer included in the **Multifunction menu**, but were moved to the **Options** menu. The **Page offset** and **Alternative text display** options were moved and regrouped within the **Options** menu. The **Display symbol text relocation** (see Chapter 11.3 on Page 91) and **Message filter** functions are new. The latter allows you to define which message types are to be displayed in the message window (see Chapter 13.2 on Page 104).

Fig. 12: Modified **Options** menu

The **Free graphics** menu was enhanced to include the **Polyline** option (see Fig. 13). To help you make a clear distinction between the **Set of lines** and **Polyline** functions, we have changed the icon for the Set of lines (). Further information can be found in Chapter 14 on Page 107. The **Placeholder text** menu item is also new on this menu. This function is used when creating the component macros for the panel layout (see Chapter 18.2 on Page 120).

Fig. 13: New menu option **Polyline** in the **Free graphics** menu

When editing symbols in a symbol project you can now also import graphics macros. To this purpose the **Symbol** menu now includes the new menu item **Import macro** (see Fig. 14). Further information about this topic can be found in Chapter 26.1 on Page 167.

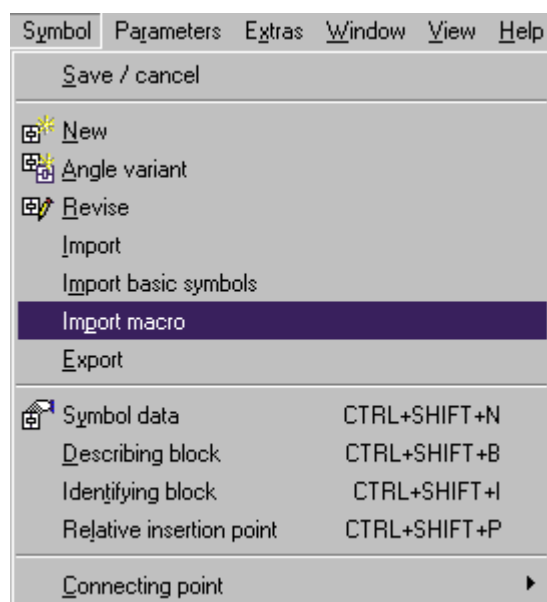


Fig. 14: New menu option **Import macro** in the **Symbol** menu

The **Extras** option is completely new in the graphics editing menu (see Fig. 15). Using the various functions of the **Table of contents** menu item you can now create, update, and delete a table of contents from within graphics editing (see Chapter 17 starting on Page 115). The **Mounting panels** menu item includes all functions available for the panel layout.

The **External programs** option which you can use to launch "non-EPLAN" programs was contained in the **Page** menu in previous EPLAN versions.

The **BUS data export** menu item is also new. This is used to export EPLAN PLC BUS data. At present this contains only the option **SIMATIC HW Config** with which you can transfer BUS data to the SIEMENS configuration software of the same name.



Please note that this function is part of an add-on module. The **SIMATIC HW Config** menu option cannot be carried out until you have acquired a license for this module (refer to Chapter 27 on Page 175).

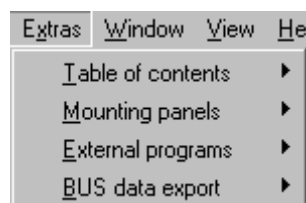


Fig. 15: New **Extras** menu

The new functions of the **Window** menu were already explained in Chapter 4.1 (refer to the description starting on Page 15).

The **View** menu also contains some new functions. The new **Screen section** menu item allows you to shift the current screen section to the top, bottom, left, or right. Below the regrouped **Toolbars** option, the **View** menu contains the new **Messages**, **Next window**, **Update**, and **Display of page properties** menu options (see Fig. 16).

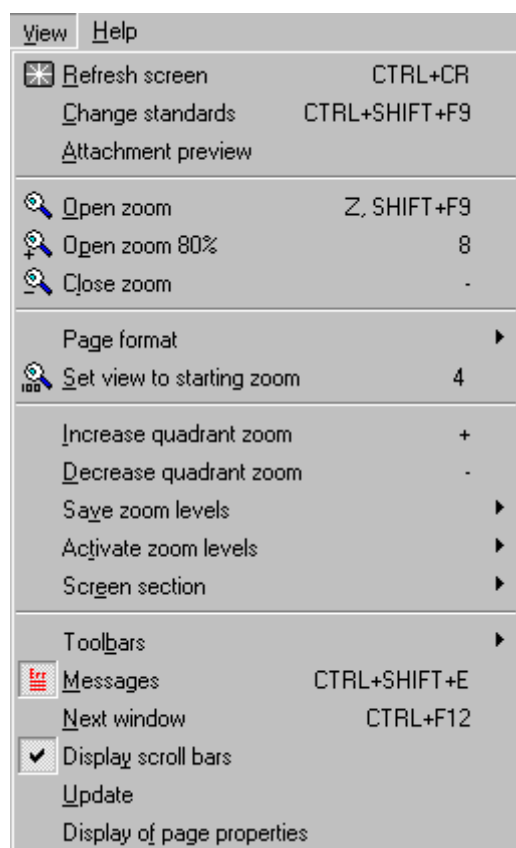


Fig. 16: Modified **View** menu

As already mentioned in Chapter 4.3 on Page 18, the new message window in graphics editing can be activated/deactivated with the **Messages** option. The new **Next window** menu item allows you to switch between the different page types in graphics editing – the page overview, the working window, and the message window. The **Update** menu item is used to keep the processing status of a page up to date (also refer to Chapter 19 starting on Page 137). Using the new **Display of page properties** option you can call up a dialog box which is used to specify the display of the page properties in the **Page** menu (**Display properties** menu item).

In addition, the **DIN format** and **Display scroll bars** options were repositioned in the menu. **Page format** is now in a group with the **Set view to starting zoom** option, and the **Display scroll bars** option was shifted to the last group of the menu.

5 New and Changed Functions of the Page Overview

The page overview in graphics editing was completely redesigned for EPLAN Version 5.50. This means that the sequence of many menu options was changed and that some functions were added to the corresponding **Page** menu.

Many functions can only be carried out when the page overview or a working window is active. If a working window is active, the function will only be carried out for the page opened there. This is the case for the **New**, **Copy**, **Rename / Move**, **Edit properties**, **Delete**, **Print**, functions and for all export functions. You can "activate a window" by highlighting e.g. one or several pages in the page overview or by left-clicking e.g. on a working window.



1. You highlight the Pages 12, 13, 14, 15 in the Page overview window, and select the **Delete** option from the **Page** menu afterwards. The corresponding pages are deleted after a prompt for confirmation.
2. You are in the working window of Page 7. If you select the **Delete** option, Page 7 is deleted after a prompt for confirmation (regardless of whether you have highlighted several pages in the page overview beforehand).

The **Open**, **Open in new window**, **Filter**, and **Update filter** functions only refer to the page overview. In other words, in these cases it is not relevant whether a working window is active or not. When the page overview is not open, the respective menu option are not available.

The **Save**, **Read in backup page**, **Empty**, **Previous**, and **Next** functions, however, only apply to the active working window. This means that these functions are carried out in the active working window, even if your cursor is located in the page overview. If no working window is open, the respective menu options cannot be activated (grayed out display).

The functions mentioned from Chapter 5.2 onwards are described for default settings. This means, the page overview is active and is not closed after each action.

5.1 New Icons on the Page Toolbar

In view of the modified and new functions of the page overview we also redesigned several icons of the corresponding toolbar.

New symbol



Function

Return to the main menu.

Show and hide page overview.

New page.

Open page.

Close page.

Save page.

Copy pages.

Delete pages.

Previous page.

Next page.

Edit page properties.

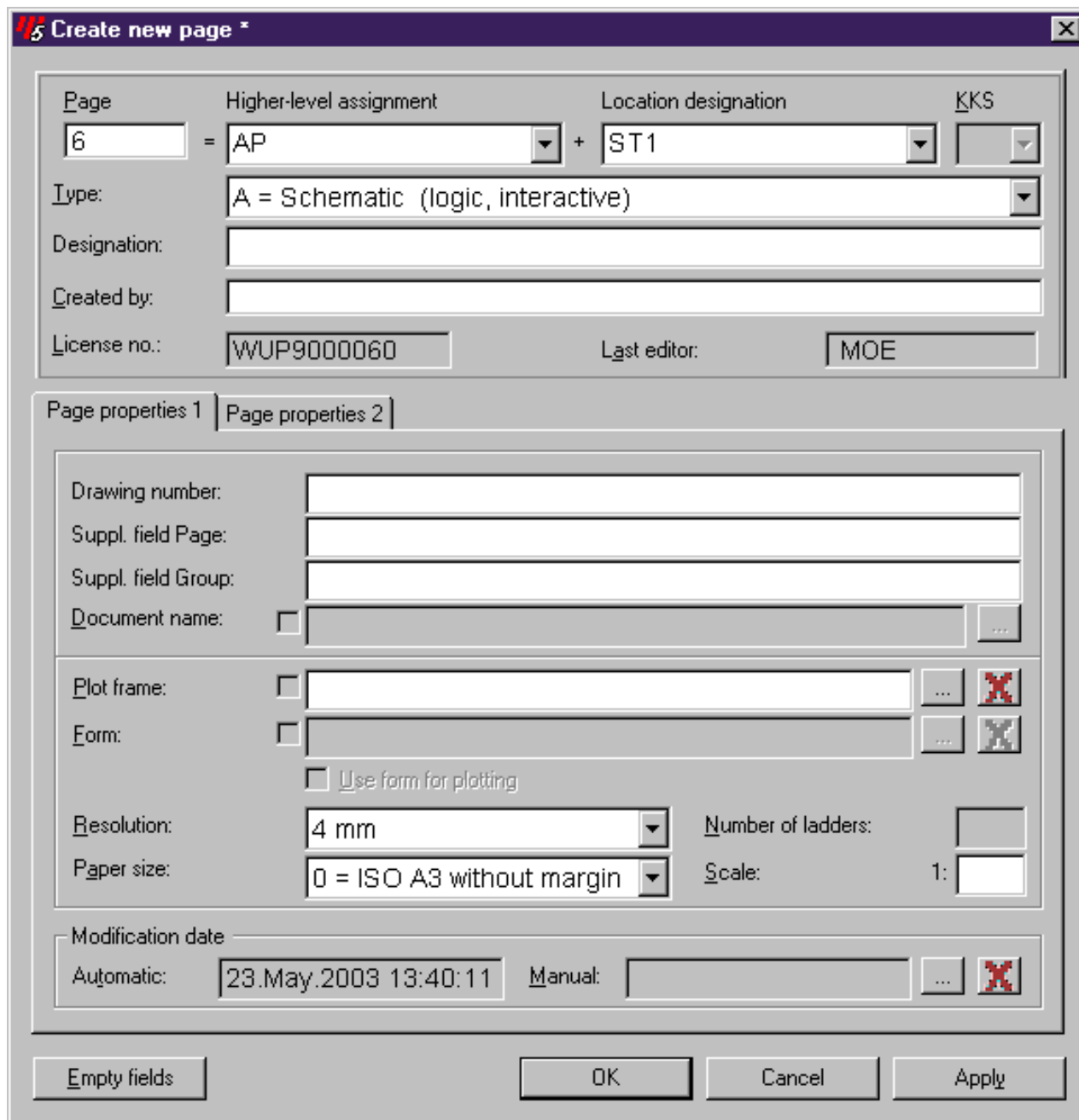
Print pages.

Filter pages.

Update filter.

5.2 Creating a New Page

When creating a **new page**, proceed in the same way as in previous EPLAN versions – e.g. by selecting the corresponding **New** option. Alternatively, you can also click the redesigned  icon, or press **Ctrl+N**. Contrary to the previous two-part dialog box, the redesigned Create new page dialog box only contains the fields for entering the page properties (see Fig. 17).



The dialog box "Create new page" is shown with the following fields and options:

- Page:** 6
- Higher-level assignment:** AP
- Location designation:** ST1
- KKS:** (empty)
- Type:** A = Schematic (logic, interactive)
- Designation:** (empty)
- Created by:** (empty)
- License no.:** WUP9000060
- Last editor:** MOE
- Page properties 1:**
 - Drawing number:** (empty)
 - Suppl. field Page:** (empty)
 - Suppl. field Group:** (empty)
 - Document name:** ☐ (empty) ...
 - Plot frame:** ☐ (empty) ... 
 - Form:** ☐ (empty) ... 
 - ☐ Use form for plotting
 - Resolution:** 4 mm
 - Number of ladders:** (empty)
 - Paper size:** 0 = ISO A3 without margin
 - Scale:** 1: (empty)
- Page properties 2:**
 - Modification date:**
 - Automatic:** 23.May.2003 13:40:11
 - Manual:** (empty) ... 
- Buttons:** Empty fields, OK, Cancel, Apply

Fig. 17: Create new page dialog

The actual process of creating a new page including the selection of the page type and the input of page properties has not changed much.

The **Document name** input field is new. It is activated when a page of the **External document page** type is created. Using this page type you can integrate external documents into your EPLAN project. Further information can be found in Chapter 7 on Page 67.

Slight optical modifications took place for the Plot frame, Form, Resolution, Number of ladders, and Scale fields. Considering the fact that you can also use long file names for plot frames and forms now (cf. Chapter 20 on Page 140), we extended the corresponding fields and regrouped some of the other fields. The Plot field is now called "Use form for plotting".

The **Paper size** dropdown list is also new on the Page properties 1 tab. Apart from the paper size "ISO A3 without margin" that has been always been available in EPLAN you can now also select the following formats:

- ISO A1
- ISO A2
- ISO A3
- ISO A4 Landscape
- ISO A4 Portrait
- ANSI-B
- ANSI-C
- ANSI-D
- Legal

Project pages can also be printed out in the respective paper sizes. In the EPLAN master data, a sample plot frame is available for each ISO format.

As non-EPLAN documents are now integrated through a new page type, the **[Ext. File]** button is no longer required in the Page properties dialog box. We also took out the **Copy with graphics** check box, since it was misplaced here. Furthermore, the **[Filter]** button is no longer included in the Page properties dialog box. Page selection can now also be carried out via the **Filter** option (also refer to Chapter 5.14 on Page 44).

When you close the dialog box by clicking **[OK]**, EPLAN opens the new page that was most recently created. The working window of the page is activated, and can immediately start to edit the page.

If, when creating new DIN project pages, you have entered a new higher-level assignment and/or location designation, which did not exist in the project before, EPLAN will display another dialog box once the Create new page dialog has been confirmed by clicking **[OK]** or **[Apply]**. It will help you to position the new identifiers in the higher-level assignment and location designation database. In Chapter 5.8 on Page 36 you will learn how to position identifiers.

5.3 Opening Pages

To open a page, you have to highlight it in the page overview first. Afterwards, select the **Open** option from the **Page** menu or from the pop-up menu. As an alternative, you can also double-click with the left mouse button on the selected page or press . Another possibility to open a page is to click on the redesigned  button or to press  .

If a working window is already open, EPLAN will use the page to be opened for this working window. In case of several pages being selected in the page overview, the lowest page selected is opened.

As from EPLAN Version 5.50, you can also **open several project pages** for editing. For this purpose, the new **Open in new window** option is available in the **Page** menu. Before carrying out this function you must select the respective page in the page overview again. It is also possible to select several pages here. A corresponding working window is called up for each selected page.

5.4 Saving a Page

Just like in the previous versions, you can save a page by selecting the corresponding **Save** menu option, by pressing , or the shortcut  , or by clicking the modified  icon on the Page toolbar. When this function is executed, the page in the active working window is saved. It is not possible to save several pages at a time.

Due to the fact that you can open more than one page in EPLAN now, the prompt whether you want to save a modified page is no longer displayed when another page is opened, but when you edit another page.

5.5 Closing Pages and Exiting Graphics Editing

In previous EPLAN versions, using the **Close** option not only meant closing the current page but also exiting graphics editing. As it is now possible that several pages or no page at all are open in graphics editing, we had to change this function and to add some new functions to the page overview.

Selecting the **Close** menu option will close the **active working window** opened in EPLAN 5.50. Alternatively, you can also click the redesigned icon  or press **Ctrl** **F4**.

To **exit the graphics editing module**, select the new **Return to main menu** function. To do so, select the respective menu option, or click the new  button on the Page toolbar. If no working window is open, you can also press **Ctrl** **F4** to exit graphics editing.



Please note: The **Close** menu option was replaced by the **Return to main menu** option throughout EPLAN!

5.6 Copying Pages

EPLAN 5.50 offers **two** different copy functions in the page overview:

- **Copy**
Use this option known from previous EPLAN versions to copy pages within the current project.
- **Copy from / to**
Use this new menu option to copy pages from another project into the current project and vice versa.

5.6.1 Copying Pages within a Project

In the page overview, first select the page(s) to be copied. Next, select the **Copy** option from the **Page** menu or the pop-up menu, or click on the  button on the toolbar. EPLAN then opens the Copy pages dialog box (see Fig. 18).

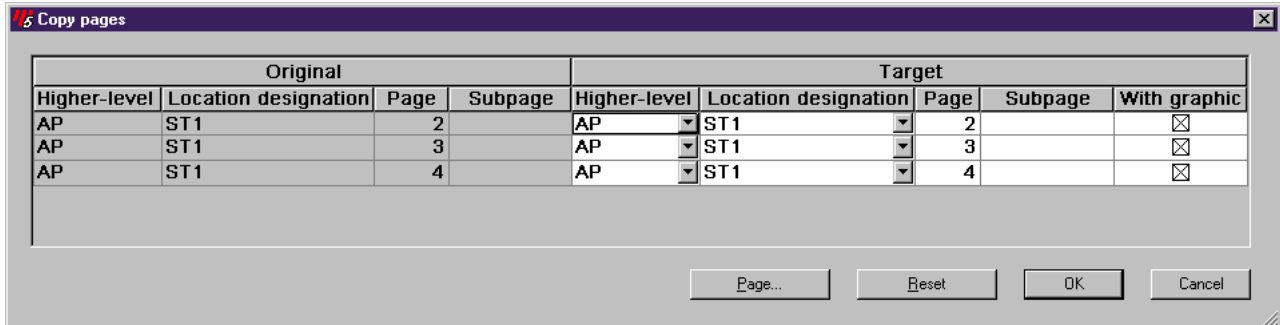


Fig. 18: Copy pages dialog box for copying pages within a DIN project

This dialog box allows you to specify page numbers and page designations (for DIN projects) for the pages to be copied. On the left-hand side of the dialog, you will find the designations of the selected pages (original pages), on the right-hand side the designations of the target pages are displayed. By default, the entries on both sides are the same.

However, you can edit the entries on the right-hand side. To do this, click into a field, and enter the desired page number or the desired higher-level assignment or location designation there. You can also make one entry for several pages. To do so, highlight either a whole column in the Target table or several fields in one column, and make the desired entry.



You want to specify a new location designation for all pages to be copied. First highlight the Location designation column.

Target			
Level	Location designation	Page	
▼	LOD3	▼	2
▼	LOD2	▼	3
▼	LOD2	▼	4

Then right-click into the upper field. Enter the new designation afterwards. It will immediately be applied to all pages.

Target			
Level	Location designation	Page	
▼	LOD4	▼	2
▼	LOD4	▼	3
▼	LOD4	▼	4

In the With graphics column you can specify for every page if it is to be copied with the graphics. By default, the check boxes are activated here.

Using an additional function which is activated with the **[Page...]** button, you can assign the page numbers automatically. The Assign page numbers dialog box will then appear (see Fig. 19).

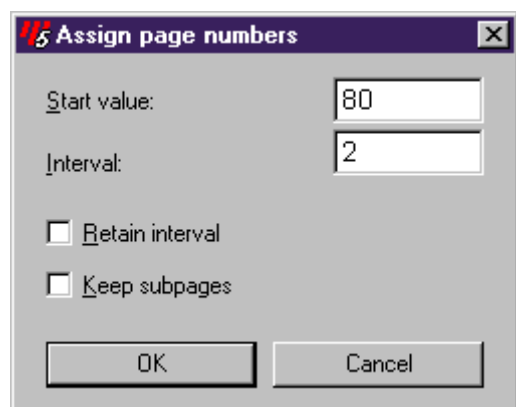


Fig. 19: Dialog box for the automatic assignment of page numbers

First, specify the page number of the first page in the Start value field of this dialog box. Then enter the new interval between the pages to be copied. Using the Retain interval check box you can specify whether the originally determined interval is to be kept. If you activate this check box, the Interval field cannot be edited any more. Furthermore, if you want to retain subpages while copying, you must activate the Keep subpages check box. When you confirm with **[OK]** you will be returned to the Copy pages dialog box.

Clicking **[Reset]** in the Copy pages dialog box will restore all original entries in the changed lines on the right side. The pages are copied after you have confirmed by clicking **[OK]**. If page designations already exist in the project, a message appears asking you if you want the existing pages to be overwritten or retained.

In case you have entered new higher-level assignments and/or location designations in the Copy pages dialog box, the Position identifiers dialog box is opened afterwards. Before you can copy the pages, you must "sort" the new identifiers into the higher-level assignment/location designation database by means of this dialog box. For detailed information on positioning identifiers, please refer to Chapter 5.8 on Page 36.

5.6.2 Copying Pages Into / From Another Project

For the "external" copying of pages, select the **Copy from / to** option from the **Page** menu. A dialog box divided into two sections will then be opened (see Fig. 20 on Page 33). As for appearance and functionality, this dialog box is similar to the Copy dialog box of previous EPLAN versions. On the left-hand side you will find the pages of the current project. Using the **[...]** button on the right-hand side you can select a target or source project. If no project has been selected yet, the right-hand side is empty.

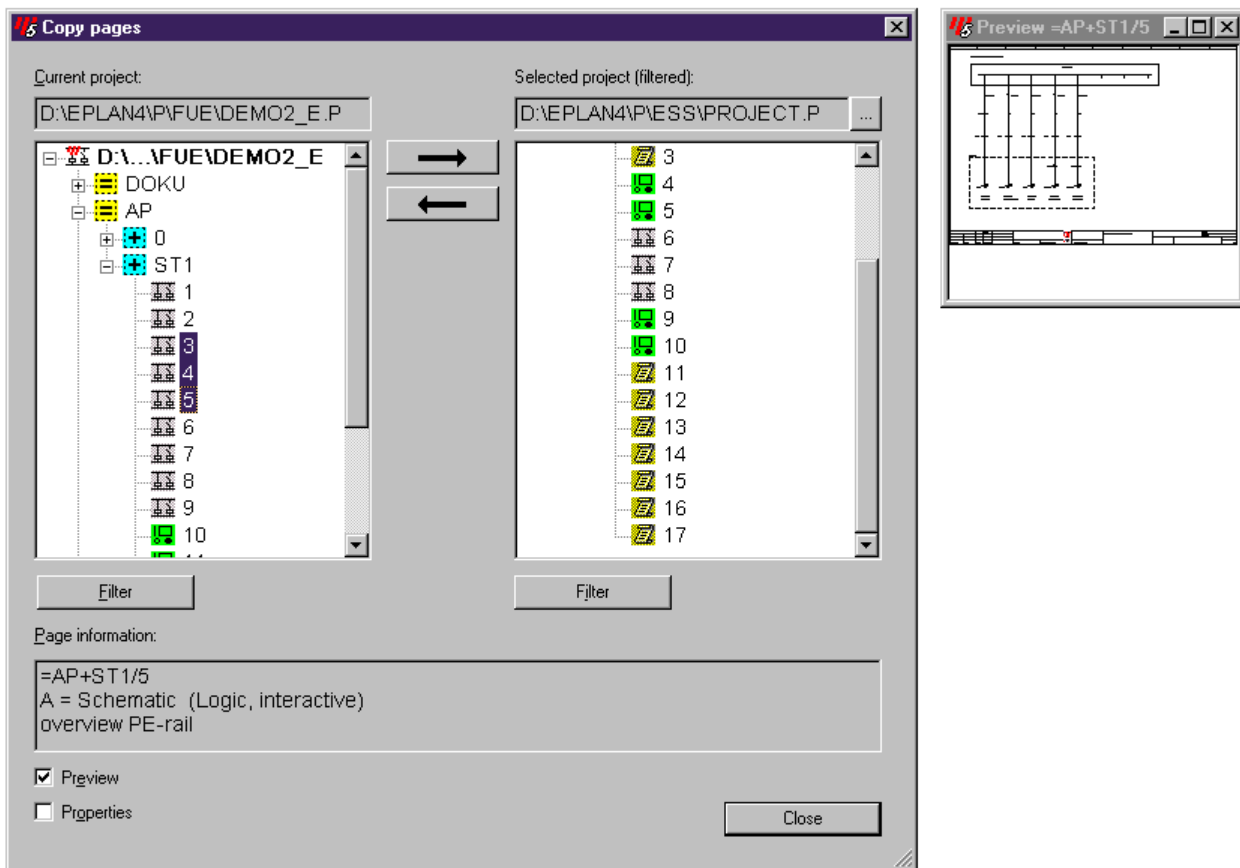


Fig. 20: Copy pages dialog box for copying pages from a project into another project

Like in the "old" Copy pages dialog box, the new dialog box provides information on the highlighted page in the Page information field. However, the **Preview** and **Properties** check boxes are new. If you activate the Preview check box, an additional window appears. This shows a preview of the selected page. This preview allows you to get a quick overview of the pages of the currently open project (cf. Chapter 5.13 on Page 42). In case you need additional information on a page to be copied, activate the Properties check box. The Page properties dialog box giving the data of the selected page will then be opened.

You can use a page filter for the current project as well as selected project. To do this, click on the respective **[Filter]** button. The page filter allows you to restrict the display of project pages to the pages needed for the copy procedure.

If you have defined a filter for a page of the copy dialog box, this will be displayed by EPLAN in the dialog box (e.g. Selected project (filtered)). The filters for the copying of pages may differ from the filter for the page overview (see also Chapter 5.14 on Page 44).

Using the arrow keys you can copy the selected pages into the "current project" or the "selected project". When doing this, the dialog box for determining the target page numbers or the target page designations (in the case of a DIN project) appears (see Fig. 18 on Page 31).

If you specify a higher-level assignment or location designation that does not already exist in the target project, EPLAN opens a dialog box in which it can be positioned (see Fig. 23 on Page 36). Further information about this topic can be found in Chapter 5.8 on Page 36.

Once the copy procedure is completed, click **[Close]**. The Copy pages dialog box will then be closed, and you will be returned to graphics editing.

5.7 Renaming and Moving Pages

There are two different ways to rename or move pages in EPLAN 5.50.

- Renaming/moving by making an entry in tree view and
- Renaming/moving via the corresponding menu option.

5.7.1 Renaming/Moving Pages by Making an Entry in Tree View

It is now also possible to rename or move a page within the tree structure of the page overview, just like in the Windows® Explorer (see Fig. 21).

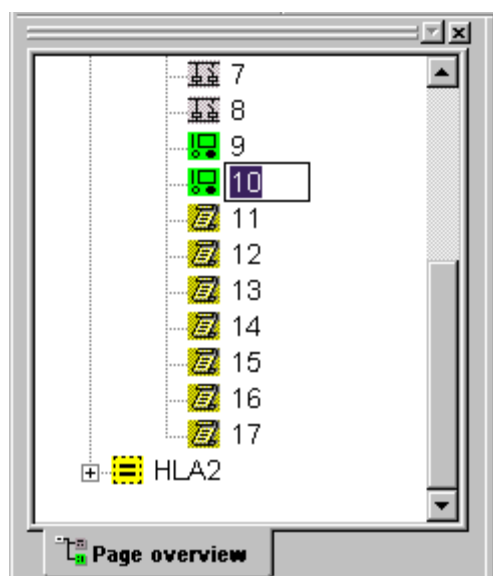


Fig. 21: Directly renaming or moving a page in the Page overview

In the page overview, first select the page to be shifted or renamed. Then click on the page number, or press **[F2]**. A small box appears around the page number, into which you can enter the desired number.



This way, you can only change the **page number of a page**. However, it is **not** possible to change the higher-level assignment or location designation of one or several pages.

5.7.2 Renaming/Moving Pages via the Menu Option

Once you have highlighted one or several pages in the page overview and selected the **Rename / Move** option from the **Page** menu or the pop-up menu, the Rename pages dialog box is called up (see Fig. 22). As for appearance and functionality, this dialog is similar to the dialog EPLAN calls up during the copy procedure for you to determine the page designation (cf. Fig. 18 on Page 31).

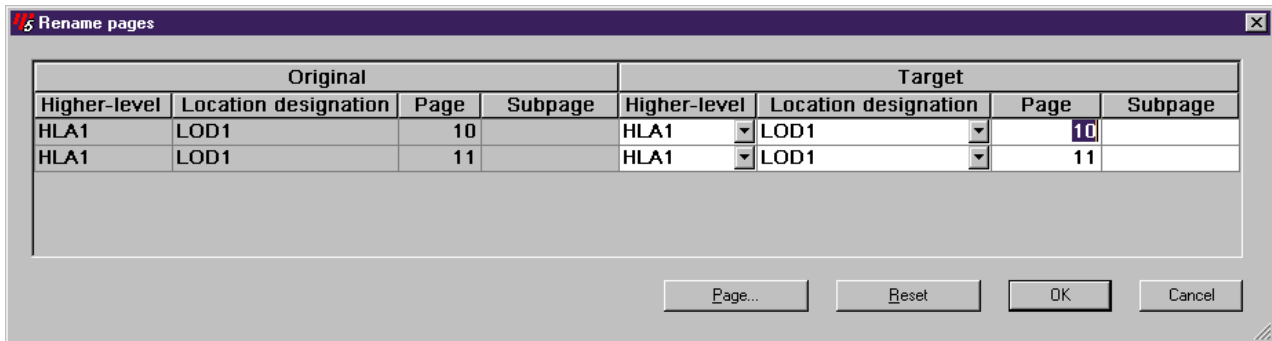


Fig. 22: Rename pages dialog box for renaming/moving pages

If you enter new identifiers here and confirm by clicking **[OK]**, the dialog box for positioning identifiers appears – as already described for the copy procedure (cf. Fig. 23 on Page 36). The following chapter will provide you with detailed information on this subject.

If the page renamed by you already exists in the project, an error message appears and the procedure is canceled. In this case, you must make another entry in the Rename pages dialog box.

5.8 Positioning New Identifiers in the Higher-level Assignment and Location Designation Database

Various functions in the page overview (e.g. the copying of schematics pages) allow you to create new higher-level assignments and/or location designations. Before applying the new identifiers to the higher-level assignment and location designation database, the dialog box for **positioning the identifiers** appears (see Fig. 23).

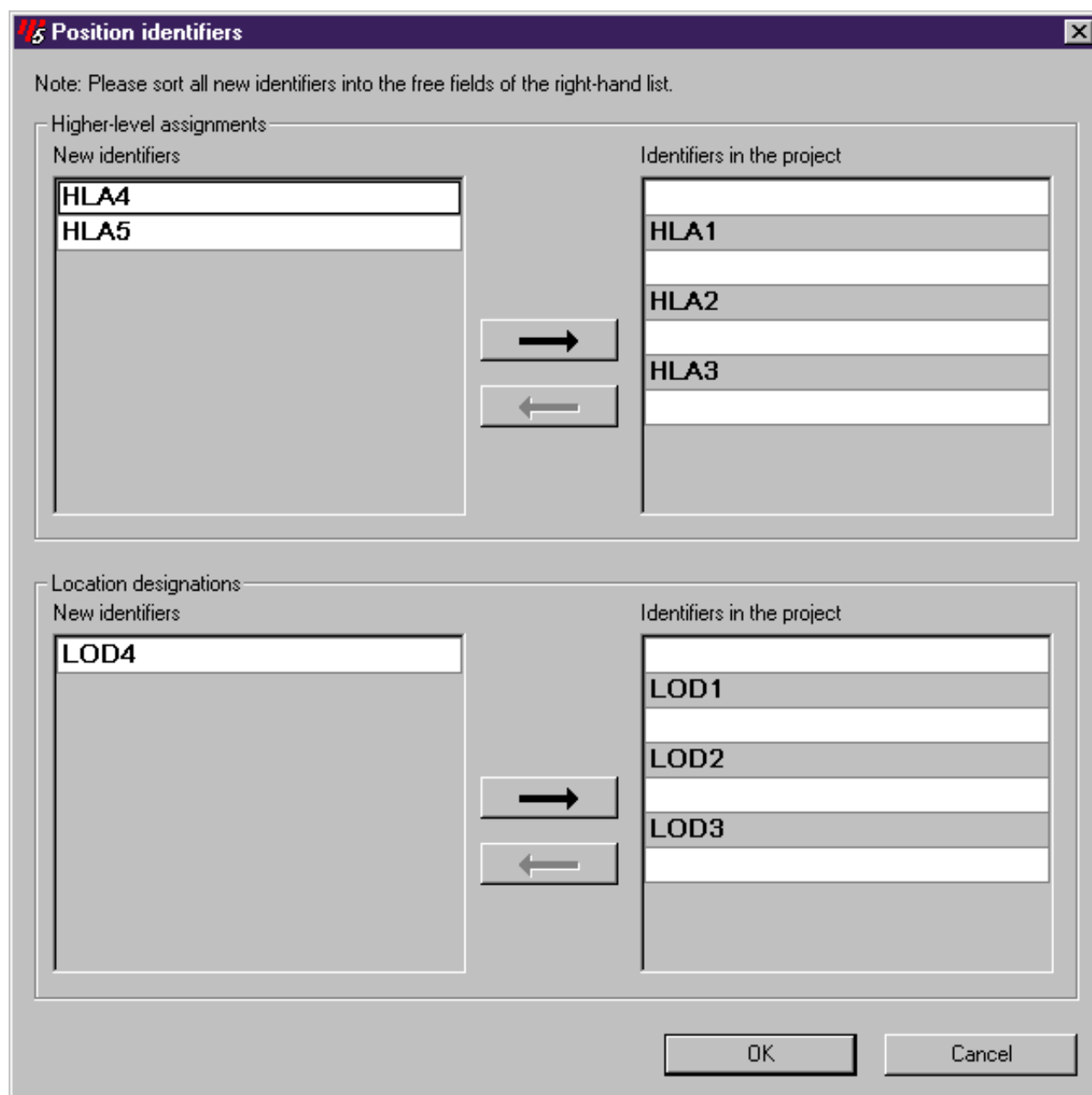


Fig. 23: New dialog box 'Position identifiers'

This dialog box helps you to sort new higher-level assignments and location designations into the higher-level assignment and location designation database. If only higher-level assignments (or only location designations) are to be positioned, only the corresponding group box is displayed in the dialog box.

On the left-hand side, you will find all new identifiers, the right-hand side shows all identifiers existing in the project. An empty line in the identifier list on the right-hand side indicates where you can position an identifier. To position it, first highlight an empty line on the right-hand side. On the left-hand side, highlight the identifier to be positioned, and click the  button afterwards. The identifier will then be positioned correspondingly. As an alternative, you can copy an identifier from the left-hand side into the Clipboard by pressing  and then paste it at the appropriate place with the  key combination.

You can also undo this action. To do so, highlight the corresponding identifier on the right-hand side, and click the  button. The identifier will be transferred to the list on the left side.

If you do not want to sort any identifiers into the database, or not all of them, first click the **[OK]** command button and then confirm the subsequent query by using **[Yes]**. All the identifiers which have not been sorted in are then attached at the end of the respective database table.

5.9 Editing and Displaying Page Properties

In EPLAN 5.50, the page properties can be modified via the **Edit properties** function. To do this, select the page whose properties you want to modify in the page overview. You can also select several pages. Afterwards, select the respective menu option from the **Page** menu or from the pop-up menu. Alternatively, you can also click the redesigned  icon, or press . The page property dialog box then appears (cf. Fig. 17 on Page 27).

You can edit the properties here just like in previous EPLAN versions. When doing this, please keep in mind that grayed fields such as Page number or Higher-level assignment and Location designation cannot be edited.

In case you have selected several pages, the fields that have identical properties are filled correspondingly. Fields with differing properties are marked by the entry "<<...>>" (also see the following example). Changes made in an input field are applied to all selected pages.



The following figure shows a section of the page properties for two pages of the Free graphics type. The two pages have differing page numbers and designations. As a result, the corresponding fields contain the entry "<<...>>". However, only the Designation field can be edited, since the Page field is grayed out.

For example, the Type and the Last editor fields have the same entries and thus identical page properties here. Therefore, the entries are displayed in these fields. Both pages don't have an entry for the Created by field.

In the Edit page properties dialog box, you will find a small button showing a pin icon in the title bar (see Fig. 24). When you click on this icon, the properties dialog box is fixed on the screen, "pinned" so to speak. This button is also available in other dialog boxes, such as the Page preview and the Display page properties dialog box described below.



Fig. 24: Examples for a title bar with pin icon; a) unpinned and b) pinned dialog box

For an **unpinned** dialog box, you can select another page in the page overview. However, once you start to edit a page, the unpinned dialog box is closed. EPLAN thus closes the unpinned Edit page properties dialog box if e.g. you open a page or click on a working window.

A **pinned** dialog box, however, remains on top of the screen when you edit a page. If you click on the pin again (i.e. "remove" the pin), the dialog box is closed again when you start to edit the next page.

Both the unpinned and the pinned dialog boxes are closed when you click on the respective buttons for closing the dialog box ([OK], [Cancel], [Close], or).

As a new feature, EPLAN now offers a possibility of a pure display of the page properties. For this purpose, the **Display properties** option is available in the **Page** menu or the pop-up menu. Once you have highlighted a page in the page overview and selected this option, EPLAN calls up the Display page properties dialog box (cf. Fig. 25 on Page 39).

This dialog box is for your information only, i.e. the page properties are shown in a table and cannot be edited. In the left column you will find the page properties, and in the right column the respective value is given. If you have selected several pages at the same time, the properties of the most recently selected page are displayed in the dialog box.

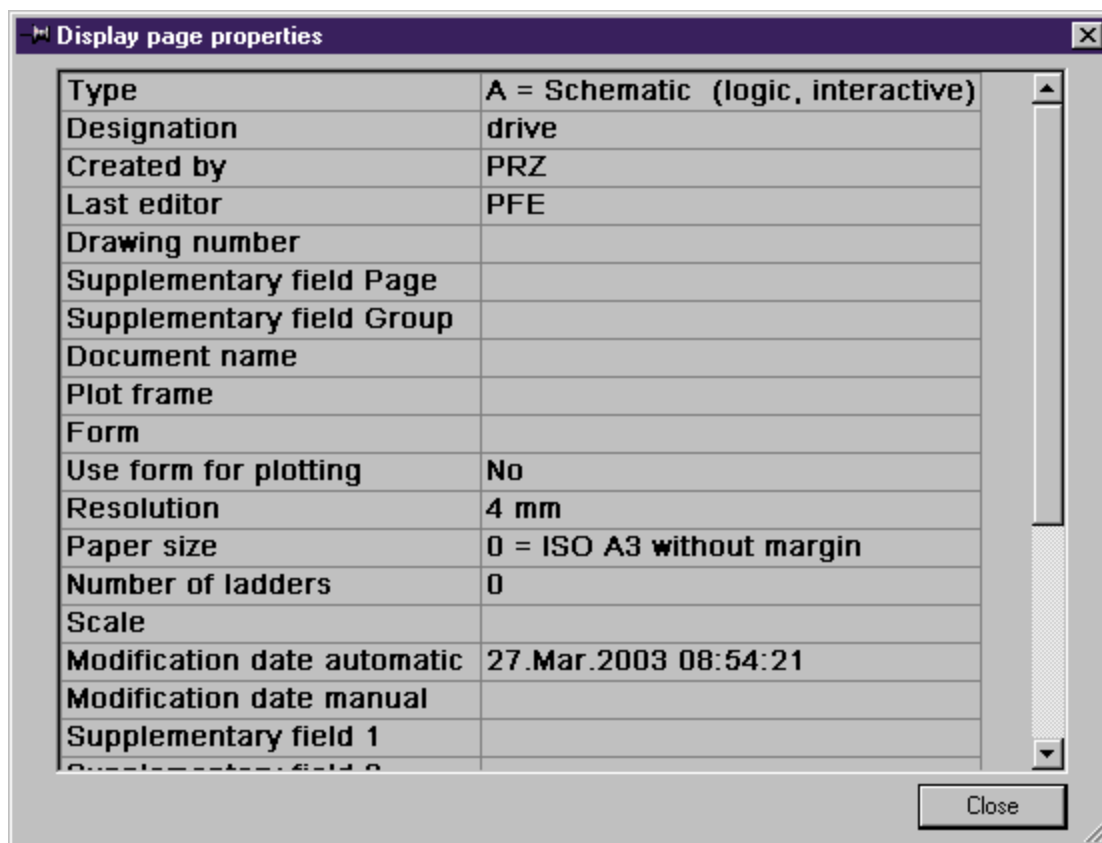


Fig. 25: New dialog box "Display page properties"

Furthermore, you can change the size of the dialog box and define which page properties are to be displayed in this dialog box. Call up the corresponding configuration dialog box in the graphics editing module by following the menu path **View > Display of page properties** (see Fig. 26 on Page 40).

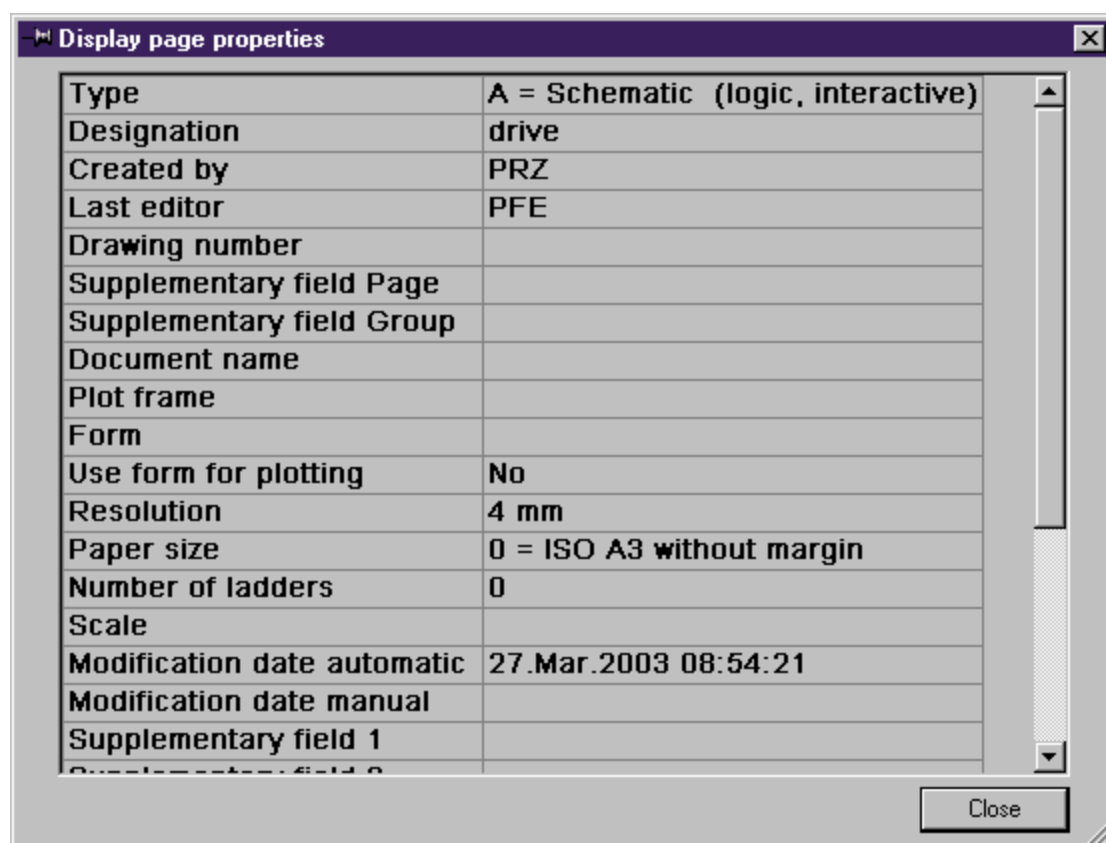


Fig. 26: New dialog box "Set display of page properties"

Deactivate the check boxes of the page properties you no longer want to be displayed. Furthermore, you can change the order of the displayed page properties. To do so, highlight the row of a page properties you want to position elsewhere in the Display page properties dialog box. Afterwards, click one of the two arrow buttons at the right-hand margin of the configuration dialog box to place the property at the desired position.

If you now confirm with **[OK]**, EPLAN will only show the page properties that are of interest to you, sorted in the desired order, when the Display page properties dialog box is called up the next time.

5.10 Reading in a Backup Page

Compared to previous EPLAN versions the actual insertion of a backup page has not been changed.

Due to the changes made for the page overview, the respective **Read in backup page** option is only active if a working window is open. If several working windows are open, the backup page is loaded in the active window.

5.11 Deleting and Emptying Pages

EPLAN 5.50 now offers a new function "Empty pages" in addition to the familiar Delete pages function.

Deleting pages is done in almost the same way as in previous EPLAN versions. First, select the page(s), that you want to remove from your project, in the tree view of the page overview. Afterwards select the **Delete** option from the **Page** menu or the pop-up menu.

Alternatively, you can also click the redesigned  icon on the Page toolbar.

EPLAN will then ask you if you wish to delete the respective page(s). Only when you have confirmed the prompt by clicking **[Yes]**, will the pages be deleted. If you delete a page that is opened in a working window, the working window will be closed. If no other working window is open, EPLAN will then display the previous page in a working window. If there is no previous page in the project, the next page is opened.



Your project consists of the pages "ANL1/1" to "ANL1/45". If you remove the open page "ANL1/45", and no other working window is open, the page "ANL1/44" is opened. However, if you remove the open page "ANL1/1", EPLAN opens the subsequent page "ANL1/2" afterwards.

Using the **Empty** function you can remove the contents of a page. The "actual" page including the page properties defined by you is retained. You can call up the corresponding option only for one open page. Selecting several pages is not possible in this case. However, you can undo the Empty command via the menu path **Edit > Undo**.

5.12 Browsing through the Pages in the Working Window

As you already know from previous EPLAN versions, you can use the **Previous** and **Next** menu options to browse through the project pages in a working window. Alternatively, you can also use the  and  keys or the  and  icons on the toolbar.

Browsing through project pages is only possible if a working window is open. If this is not the case, the corresponding menu options and icons cannot be activated.



If your cursor is located in the page overview, you can use the  and  keys to browse through the pages of the page overview. That is, the cursor will jump to the page which is right at the top or right at the bottom of the tree view.

In case you have opened several working windows, the pages of the active window are "browsed through". If, while browsing, you open a page that is already open in another working window, a corresponding message will be displayed (see Fig. 27).

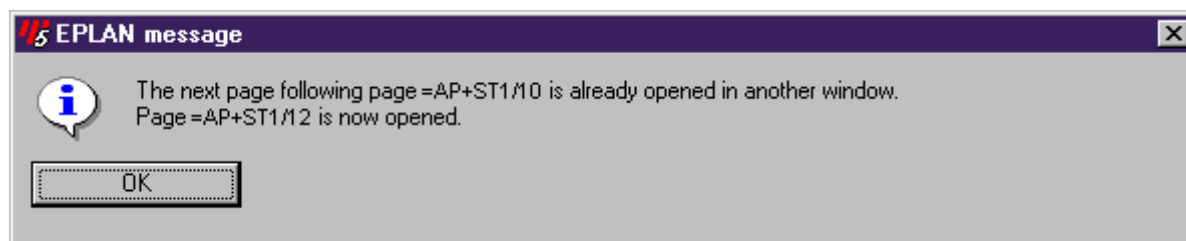


Fig. 27: Message that may appear while browsing through pages

When you confirm by clicking **[OK]**, EPLAN skips the already open page and shows the next page in the working window. Pages of the **External document** page type are not considered (see Chapter 7 on Page 67).

5.13 Page Preview

In order to support you when selecting pages in the page overview, we have implemented a page preview function for EPLAN Version 5.50, which is displayed in a separate window on the screen (see Fig. 28 on Page 43).

Call up the page preview window by selecting the **Preview** option from the **Page** menu or from the pop-up menu. If you have already selected a page in the page overview, this page will be displayed in the preview window. The number or designation of the displayed page is shown in the title bar of the page preview window. If no page has been selected in the overview, the preview window remains empty. In case you have selected several pages, the most recently selected page is displayed in the preview.

Use the page preview to browse through your project pages. This way, you will get a quick overview of the pages of a project.

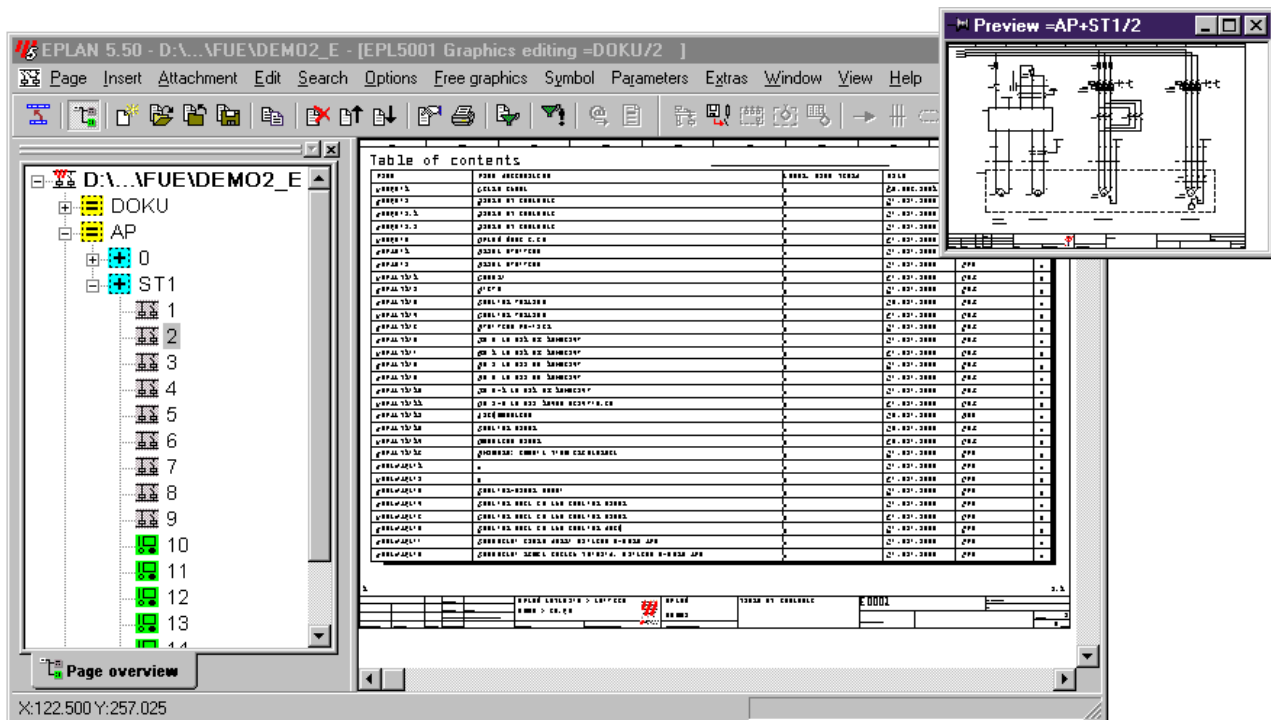


Fig. 28: Graphics editing with open page preview window

The preview window can be scaled and disposes of the Windows® standard buttons "Minimize", "Maximize", and "Close". As mentioned already, the preview window can also be fixed to the screen by means of the "pin button".

The pinned window stays in the foreground of your screen until you either minimize the window (i.e. as an icon on the task bar) or close it. Page editing is not affected by the pinned preview window. Although the pinned page preview is always in the foreground, it is possible to edit the project pages in the page overview or in a working window.

To close the Page preview window, simply select the **Preview** menu item again, or click on the  button. The unpinned page preview is also closed when you start editing a page (e.g. open a page).

5.14 Filtering Pages

Setting up a page filter is also possible for the new EPLAN page overview. This filter allows you to "hide" pages in the page overview that you don't want to edit.

In EPLAN 5.50, the new **Filter** menu option in the **Page** menu is used to call up the selection dialog box by means of which you determine which pages EPLAN displays in the page overview. Alternatively, you can also click the new  icon on the Page toolbar. In previous EPLAN versions, this is done by clicking the **[Filter]** button in the Page properties dialog box. This button was removed for Version 5.50 (cf. Fig. 17 on Page 27). We have also modified the actual Filter dialog box (see Fig. 29), the subsequent dialog box for defining filter criteria, and the way you can use those criteria.

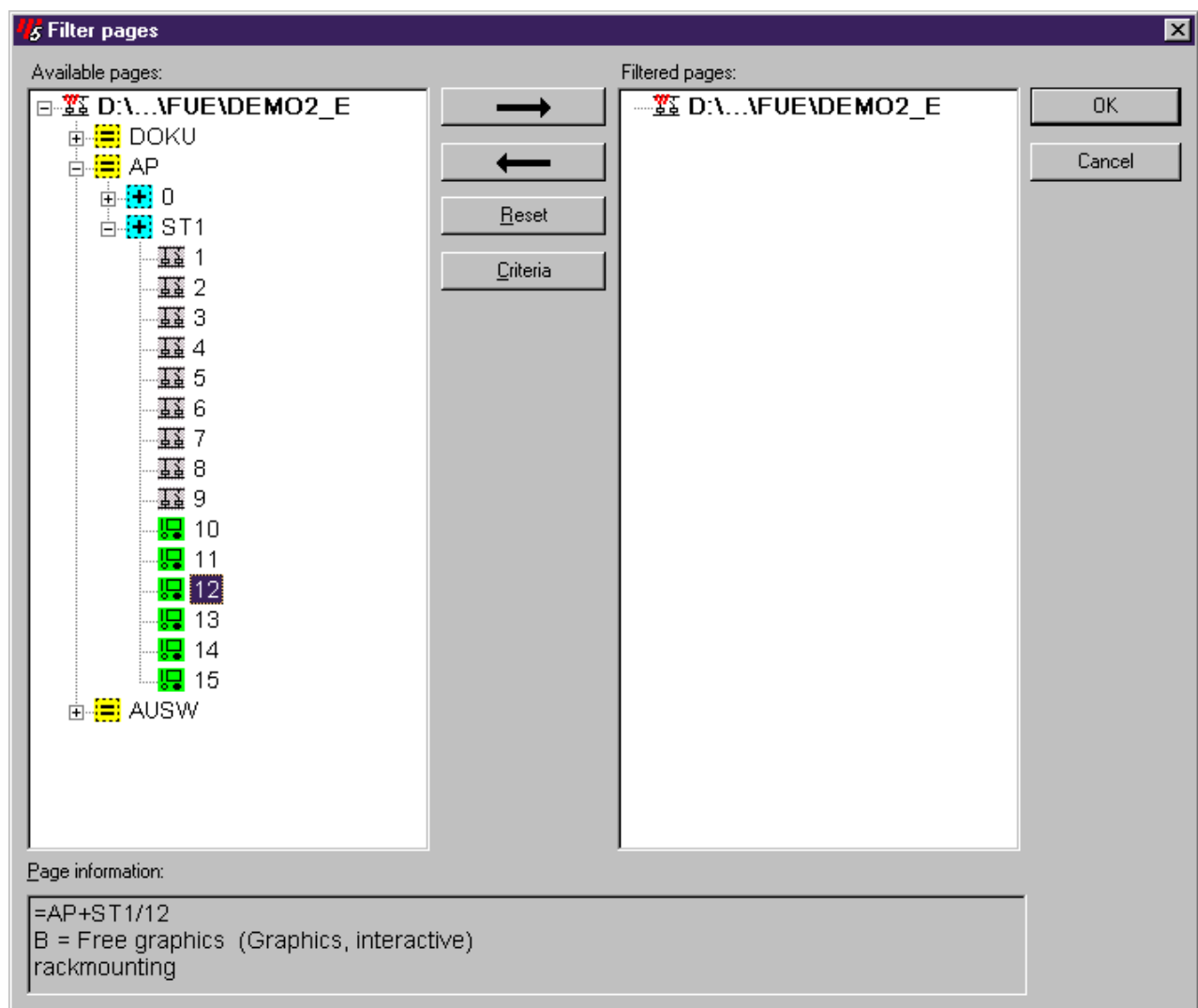


Fig. 29: Dialog box for setting up a page filter

First of all, the terminology was standardized. We no longer talk of "selecting pages" but of "filtering pages". Correspondingly, you will find all "available pages" in the project on the left side of the dialog box. And just like in previous EPLAN versions, you will then transfer the pages you want to have displayed in the page overview to the right side of the dialog box ("Filtered pages" field).

The button used to undo this selection is now called **[Reset]**, and the renamed **[Criteria]** button is used to open the subsequent dialog box for specifying the criteria.

For EPLAN Version 5.50, new and extended functionalities were included in the Filter criteria dialog box (see Fig. 30). We have added several **new filter criteria**. It is now possible to filter the pages of the page overview by using a drawing number. Furthermore, you can use a filter for **entries in the search database**. To do this, you must activate the corresponding check boxes in the dialog box. However, before you can use the search database as a basis for a page selection you must search for a corresponding search term in your project pages.



First, search for all fuses in your project whose device designation contains the identifier "F". Afterwards the search database contains 5 entries that are all located either on project Page "6" or "7". If you now use the search database as a filter criterion, only the Pages "6" and "7" are displayed in the page overview.

In addition, you can now also use the wildcards "*" and "?" for the "KKS", "higher-level assignment" and "location designation" fields. When specifying DIN or KKS identifiers as filter criteria, you can use the newly inserted [...] button to select the DIN or KKS identifiers existing in the project.

In addition, you can now also select the new option "Include higher-level assignment/location designation at devices" when filtering for DIN identifiers. When you activate the corresponding check box, the pages containing devices with the specified DIN identifier are selected.

Fig. 30: Extended dialog box for specifying the filter criteria

As mentioned already, the way in which you can use filter criteria has also been changed. In previous EPLAN versions, the defined criteria - such as e.g. the respective activated check box - were stored until the dialog box was called up the next time. If you then specified new criteria for the page selection (e.g. another page type), the previously defined criteria were removed and the previous page filter "overwritten".

The new EPLAN version now provides the possibility to use several **filters with differing criteria consecutively**. If you now filter according to a specific criterion and call up the Filter criteria dialog box again later, the corresponding check box is deactivated again. You can now specify further criteria which gives you the possibility to add further pages to a page filter. Only when you click the **[Reset]** button in the Filter pages dialog box, will the filter be deleted.



First, select the Schematic page type as filter criterion. The page overview for this project will then only display schematic pages. Call up the filter criteria again, and specify the Free graphics page type as a further criterion. Afterwards, the page overview will also display the free graphics pages in addition to the schematic pages.

Once you have set up a page filter for a project, this is displayed in the tab caption of the page overview (see Fig. 31).

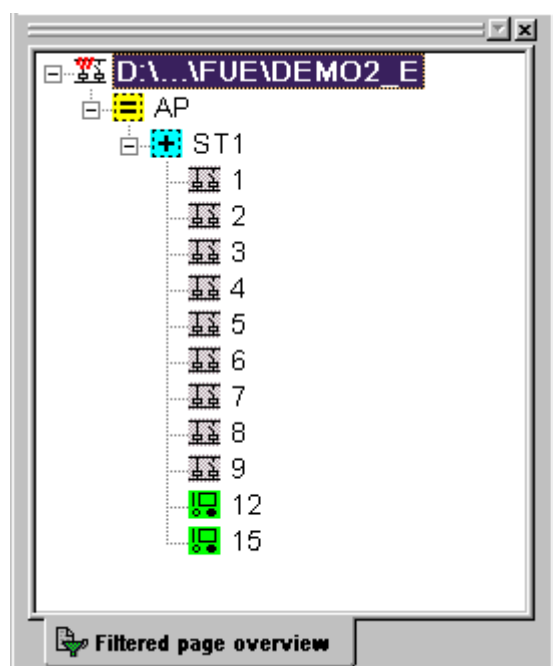


Fig. 31: Page overview for a filtered project

5.15 Updating a Filter

Under certain conditions it may be necessary to update a defined filter for the page overview, to have the filtered pages correctly displayed in the page overview. For this purpose, select the **Update filter** option from the **Page** menu, or click the  button in the Page toolbar.



Example 1:

*You have set up a filter with the criterion "Page type = Schematic" for the pages of a project. As a result, the page overview only displays schematic pages. Afterwards you create a new page of the Free graphics type. This graphics page also appears in the page overview. However, after having carried out the **Update filter** option, the page overview only displays the schematic pages that correspond to the defined filter.*

Example 2:

First, you have searched your project pages for a specific search term (e.g. the device designation "M"), and then you have used the entries in the current search database as filter criteria. The page overview displays the corresponding pages on which devices with the DD "M*" are located. If you delete entries in the search database later on or extend it by a new search, you must **update the filter** afterwards. Only then will the page overview correspond to the defined filter criterion.*



Please note that the **Update filter** function only considers the filter criteria defined via the **[Criteria]** button. Pages that you manually shifted to the Filtered pages field on the right side of the Filter pages dialog box, are reset by the **Update filter** option.

5.16 Printing Pages

As from EPLAN Version 5.50, the printing of pages can also be started via the pop-up menu (cf. Fig. 6 on Page 17).

First, highlight the page(s) to be printed in the page overview. If no pages are selected in the page overview, the page of the active working window is printed.

Afterwards, select the **Print** option from the **Page** menu or from the pop-up menu. As an alternative, you can also click on the  icon on the Page toolbar or press  . The familiar Print dialog box then appears (see Fig. 32 on Page 48). Please note, that the Print hidden lines and texts check box is new here. If you activate this check box, EPLAN will also print all objects marked as invisible in the schematic.

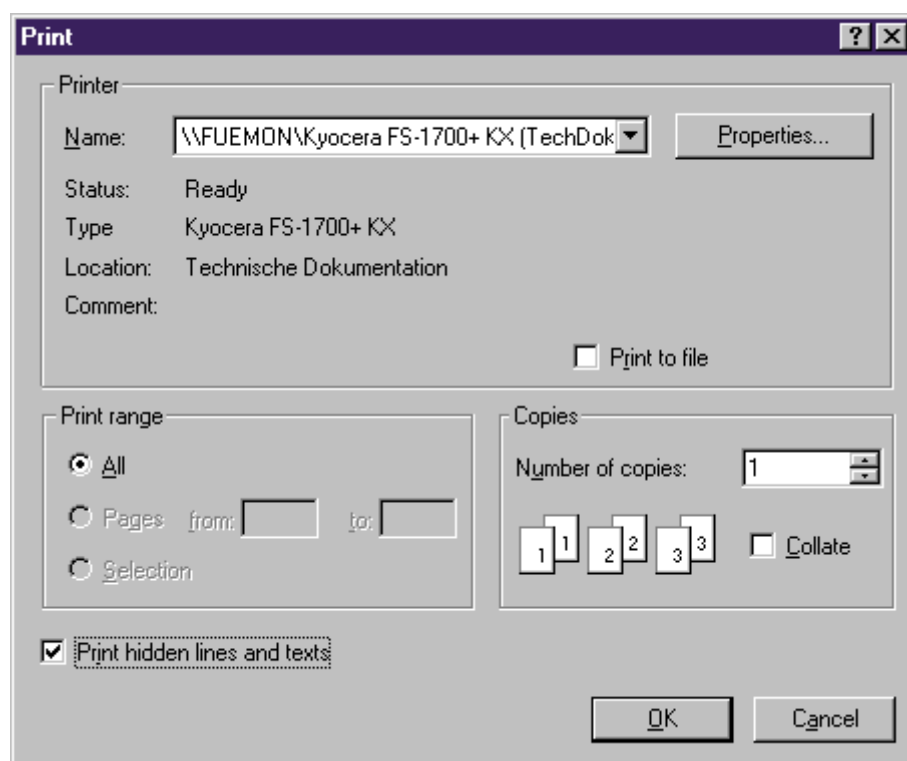


Fig. 32: Modified Print dialog box

After you have confirmed your settings by clicking **[OK]**, the selected pages will be printed on the specified printer.

5.17 Exporting Pages

Apart from a changed sequence as regards the page overview, the export functions have not been considerably changed. After one or several pages have been selected in the page overview, the highlighted pages are output (you may have to specify a target directory first). If no pages are selected in the page overview, the page of the active working window is exported.

6 Block Editing

Up to now, it was only possible to edit or change the properties of individual symbols in EPLAN. This was very time-consuming and sometimes resulted in incorrect entries. EPLAN 5.50 now allows you to edit symbols in blocks, i.e. you can quickly and conveniently modify the properties of several symbols on the current schematic page at one go. Block editing includes the following schematic objects:

- Symbols from a symbol file
- Black, DIN, and PLC boxes
- Device- and potential-end terminals
- PLC end terminals
- Terminals / connectors (from symbol file)
- Cables
- Wiring definitions
- Interruption points
- Texts
- Objects of free graphics.

The special feature of a block function is that you can edit the common properties of similar as well as of heterogeneous schematic objects.

In the following sections you will learn how to select / deselect schematic objects, how to call up the block editing function, and which object-related differences exist during block editing.

6.1 Selecting/Deselecting Schematic Objects

Before using the block editing function, you must highlight the schematic objects whose common properties you want to modify. For this purpose, EPLAN offers various highlighting methods. You can also undo the selection (deselect) of schematic objects. In detail, the following sections will inform you on how to:

- Highlight schematic sections with the mouse
- Highlight schematic sections via a menu option
- Highlight various individual objects with the mouse
- Highlight various individual objects via the keyboard
- Highlight various individual objects via a menu option
- Highlight specific schematic objects

- Undo the selection of a schematic section
- Undo the selection of individual objects.

Highlight schematic sections with the mouse

Proceed as follows in order to highlight a schematic section with the mouse:

Place the cursor **next** to the insertion point of the first object in the schematic. Press the left mouse button, and keep it down for a moment.

The cursor will then change its shape as follows:



When the mouse button is released, the cursor changes again and is displayed as follows:



Now draw a box around all schematic objects (or their insertion points) within the section defined by you. If you click briefly again with the left mouse button the box disappears, and the selected schematic objects are shown in a different color. You are now working in the highlight mode. The color change of the schematic objects depends on the background color setting defined by you in the personal parameters. When working in the highlight mode, you can undo selections of individual schematic objects (see sections "Undoing the selection of a schematic section" and "Undoing the selection of individual objects").

Highlight schematic sections via a menu option

As an alternative to using the mouse, EPLAN also offers a menu option that allows you to highlight schematic objects within a specific section for block editing.

To do so, move the mouse over the **Block editing** option of the **Edit** menu in graphics editing. From the pulldown menu that is opened, choose the **Select** function. Then left-click in the schematic, and draw a box around the schematic objects to be edited, which will then change their color as mentioned above.

Highlighting various individual objects with the mouse

You can also use the mouse to highlight specific schematic objects that are distributed all over the schematic page. Proceed in the same way as if you were highlighting a section with the mouse, until the cursor is displayed as a box that can be opened. Then open the box around the insertion point of the first schematic object to be selected, and press the left mouse button briefly. The box disappears, and the schematic object is displayed in another color. Then place the cursor **next** to the insertion point of the next schematic object, draw a new box around it, and press the left mouse button again.

Continue with the same procedure for all schematic objects to be highlighted.

Highlighting various individual objects via the keyboard

You can also use a key combination to highlight individual schematic objects.

To do this, press the  key and keep it pressed. Then use the arrow keys to move the cursor **on** the insertion point of the first schematic object to be selected, and press the spacebar (). Afterwards, the schematic object is displayed in another color. Move the cursor to the insertion point of the next schematic object to be selected, and press the spacebar again.

Continue with the same procedure for further schematic objects.

Highlighting various individual objects via a menu option

You can also follow a menu path to select individual schematic objects.

Choose the **Select** option from the **Block editing** menu, and place the cursor next to the first schematic object. Then press the left-hand mouse button briefly. As soon as the cursor changes its shape to a box, you can drag it around the insertion point of a schematic object and press the left mouse button again.

Proceed in the same way for further schematic objects.

Highlighting specific schematic objects

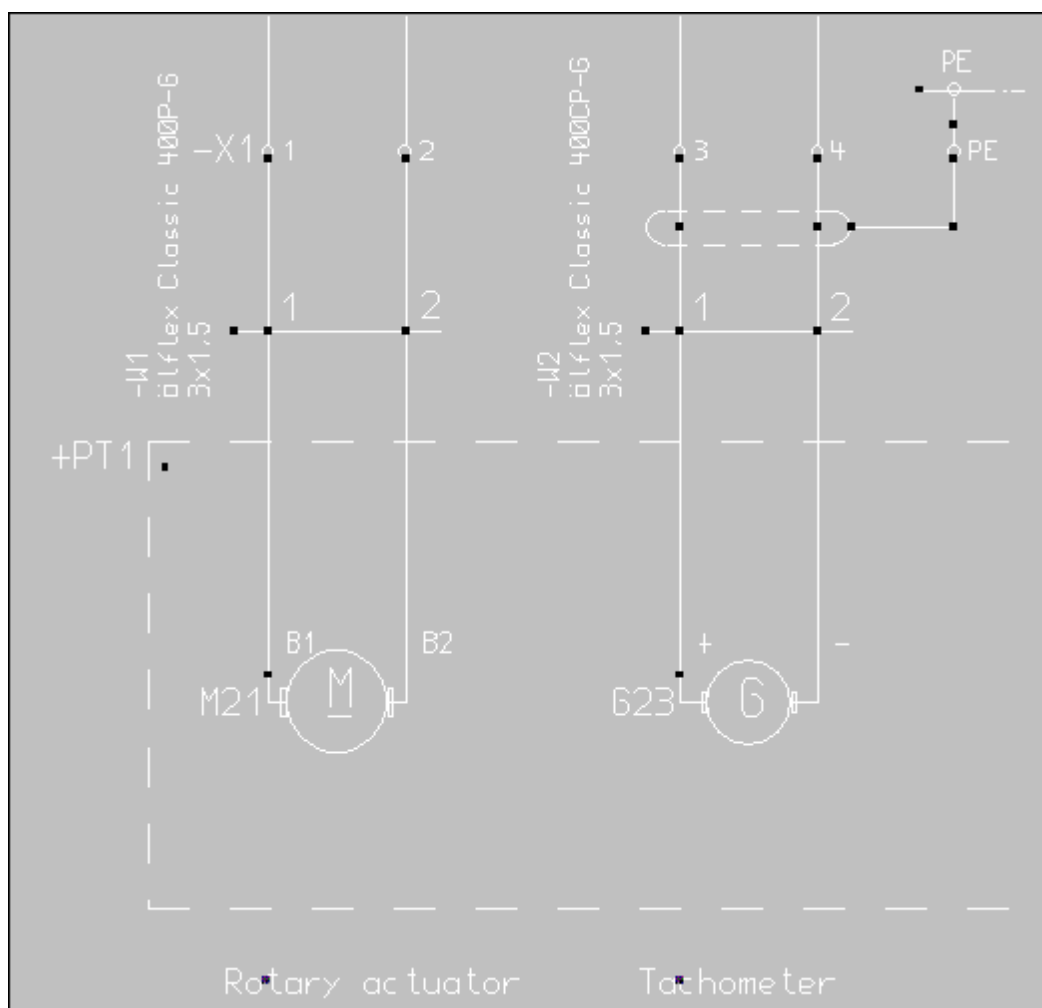
This highly efficient and quick highlighting method allows you to select only specific schematic objects. We advise to use it if you want to assign unique symbol properties to a specific group of schematic objects from any section of the schematic, for example only symbols from a symbol file (except for terminals and connector pins), or only symbol file-independent symbols (e.g. black boxes, device-end terminals), or other schematic objects (texts, objects of free graphics). The following example shows how to use this method.



To help you distinguish highlighted symbols more easily in the following sample schematic, we have selected a "gray" background color and the symbol color "white" for all schematic objects.



The schematic section below contains various schematic objects.

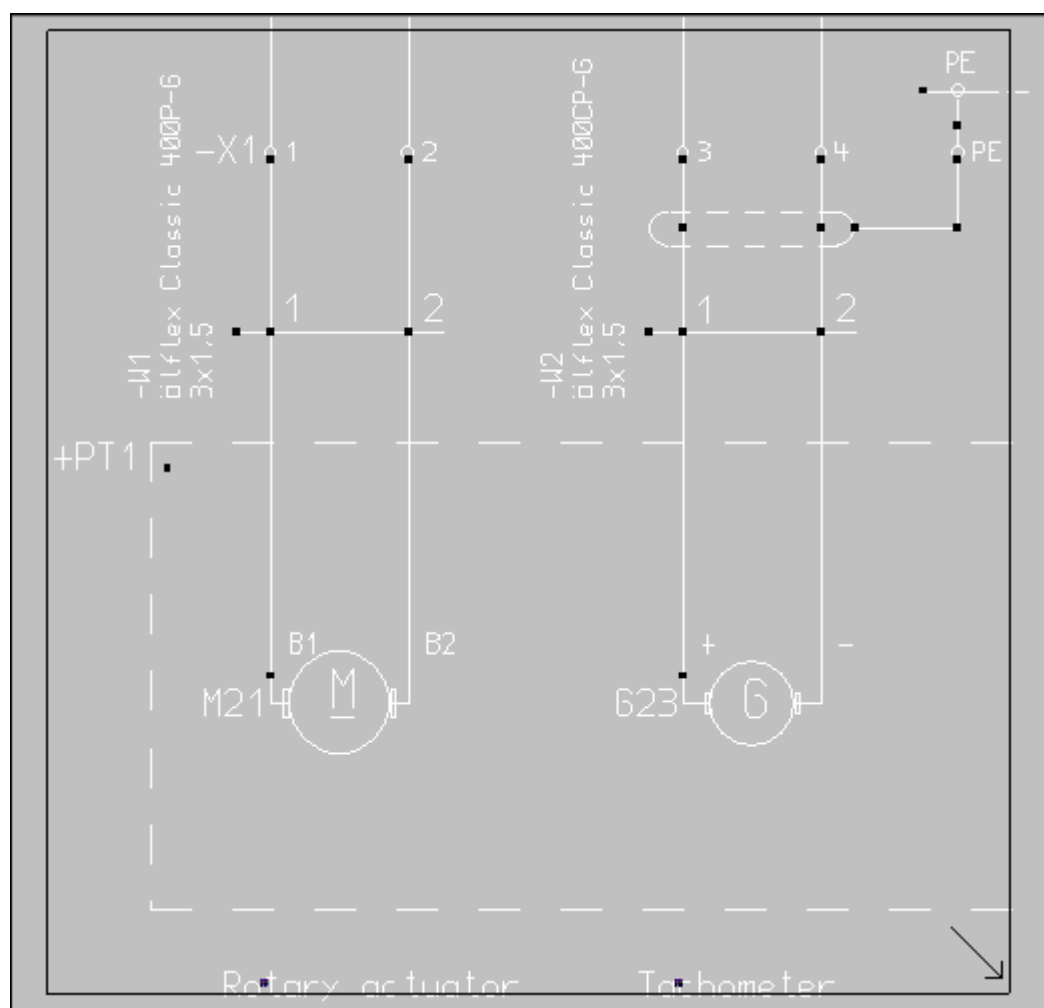


First, you want to highlight only the symbols that are taken from a symbol file (except for terminals). Afterwards you will highlight the texts only.

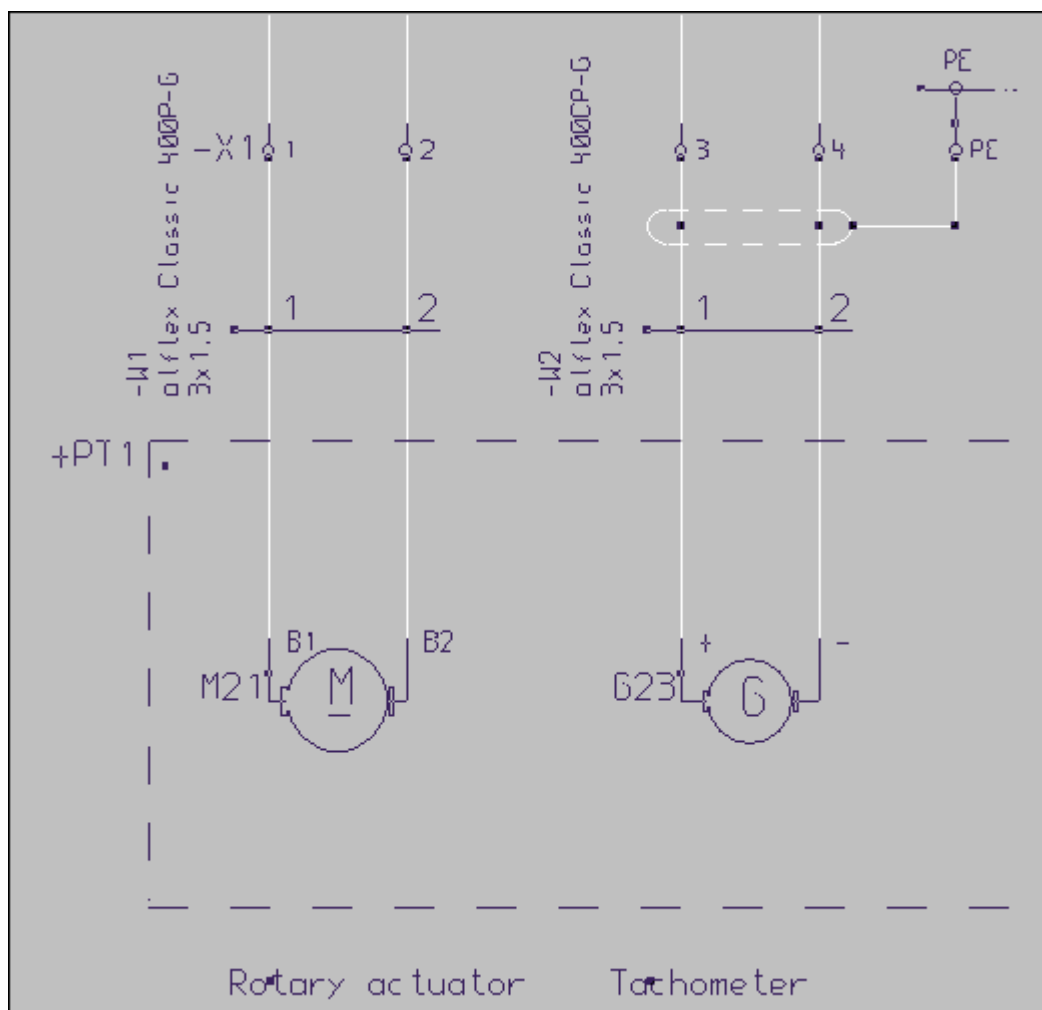
Selecting symbols from a symbol file

Follow the menu path **Edit > Select** in graphics editing. You will then get an angle-shaped cursor, which you will place in the upper left corner of the schematic.

Now press the left-hand mouse button briefly. Using the cursor, which is now shaped like a frame, you can define the extent of the selection. As shown in the figure below, you can draw a box around all objects in the schematic section.

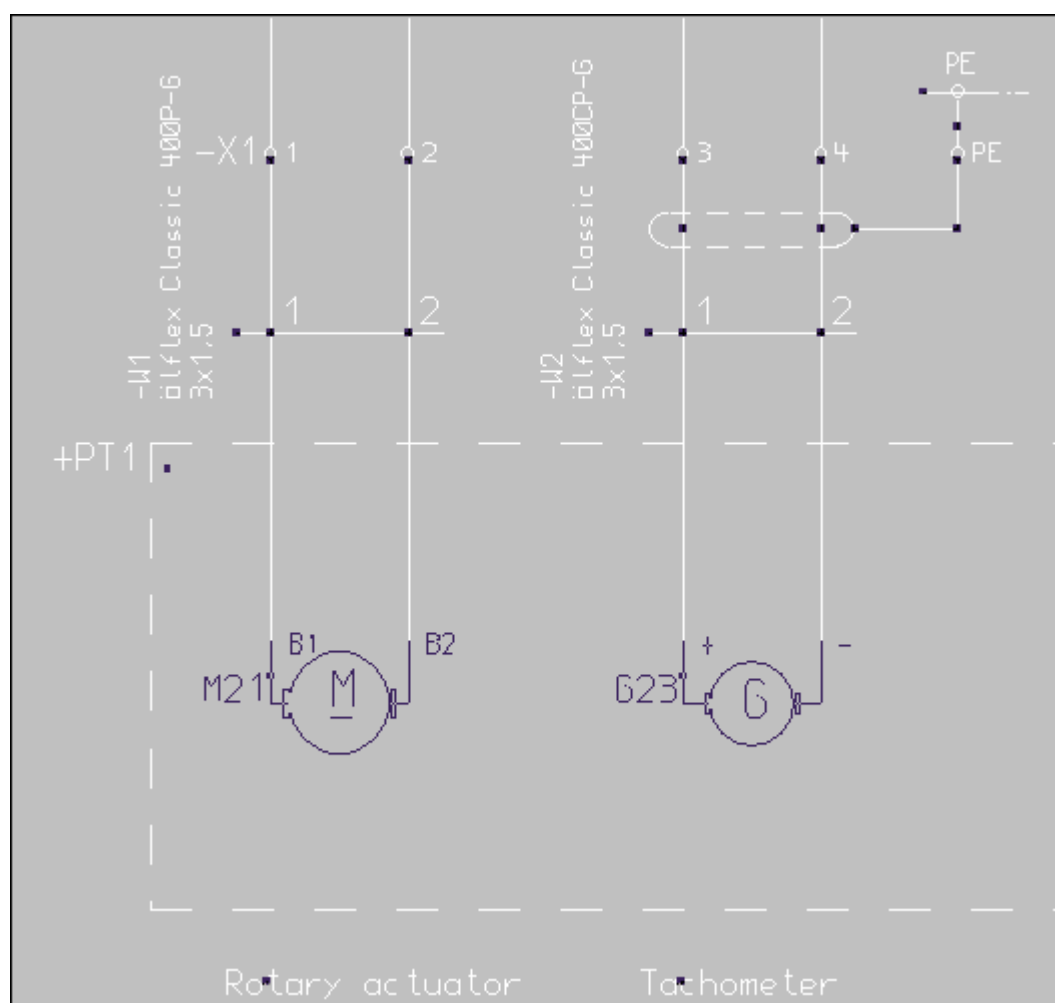


Then press the left mouse button again briefly so that the box disappears. EPLAN highlights all schematic objects that were included in the box before in another color (here: black) (see figure on the next page).



To select only the symbols taken from a symbol file from this highlighted section (terminals are not included), press  on the motor M21 or on the generator G23.

All schematics objects except for M21 and G32 will then lose their (black) marking and will be displayed in their original color (white) again. This is illustrated by the following figure: M21 and G23 are now ready to be edited in a block.

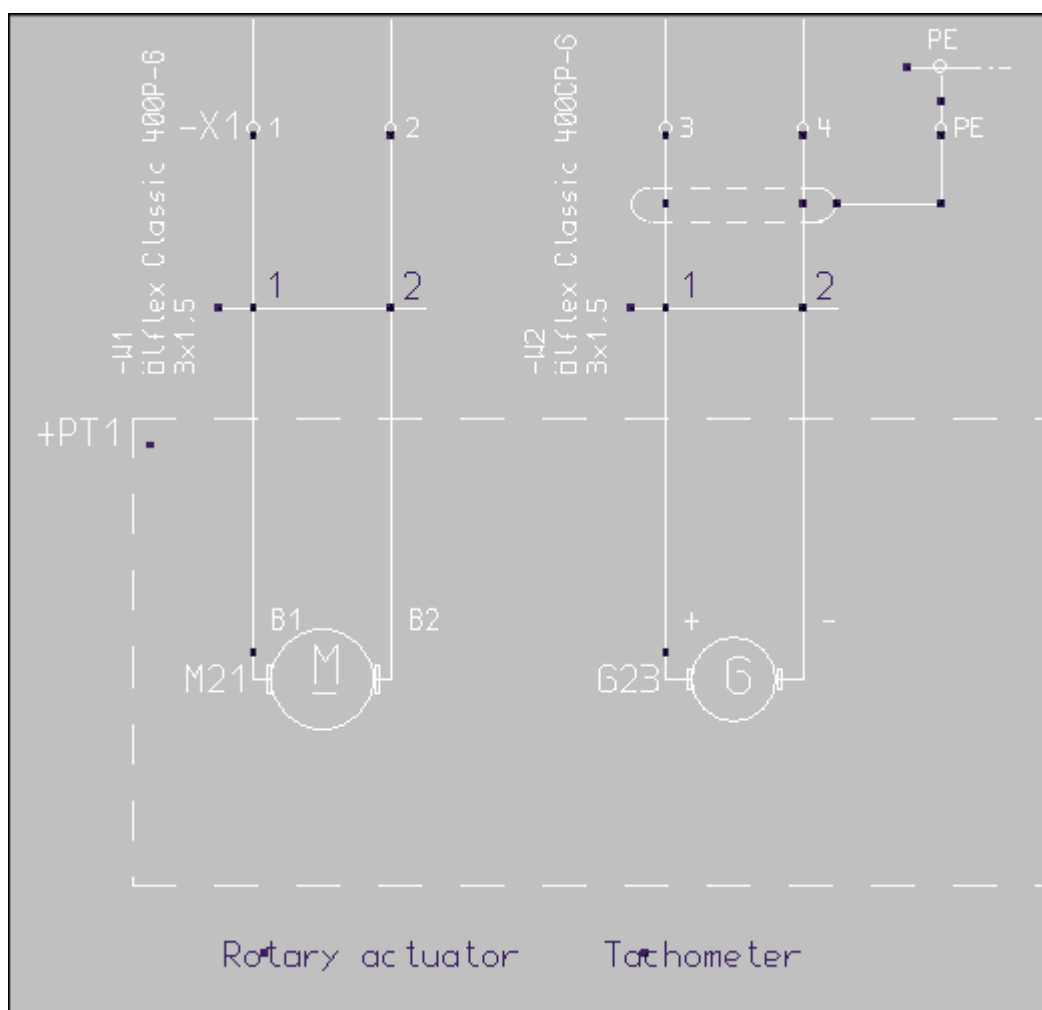


Selecting texts

To select texts with this highlighting method, first press **Esc**. M21 and G23 are then deselected again (displayed in white).

Select the entire schematic section again, so that all schematic objects lying within the box are highlighted in a specific color (here: black).

Then place the cursor on the insertion point of a text object, e.g. on the text with the designation "Rotary switch" and press the **Enter** key. All schematic objects except for the texts are deselected (see next figure). As the conductors of the cables –W1 and –W2 were designated by means of text objects of the Cable conductor designation text type, EPLAN has included them in the selection.



Undoing the selection of a highlighted schematic section

To undo the selections made within any schematic section, simply press the **[Esc]** key. All schematics objects will then be shown in their original colors again.

Undoing the selection of individual objects

In case you have highlighted too many schematic objects, EPLAN allows you to deselect individual schematic objects again. Press the **[Ctrl]** key and keep it pressed. Then click briefly **on** the insertion point of the highlighted schematic object you want to deselect. As an alternative to the left mouse button, you can also press the spacebar (**[]**) **on** the insertion point. Afterwards EPLAN removes the selection from the schematics object and displays it in its original color.

6.2 Calling up and Using the Block Editing Function

After EPLAN has highlighted the desired schematic objects you can call up the block editing function. Select **Edit > Block editing** in the Graphics editing menu. As EPLAN differentiates between various property types for schematic objects, the **Block editing** menu item offers the following options in a fly-out menu:

- **Devices**
- **Text display**
- **Line data**
- **Parts.**

Depending on the highlighted schematics objects, this EPLAN submenu already indicates which type of elements you can edit in a block. Only active options (**not grayed out**) can be carried out. For detailed information on this subject, refer to the section "Understanding Object-related Differences for Editing Blocks" on Page 59.

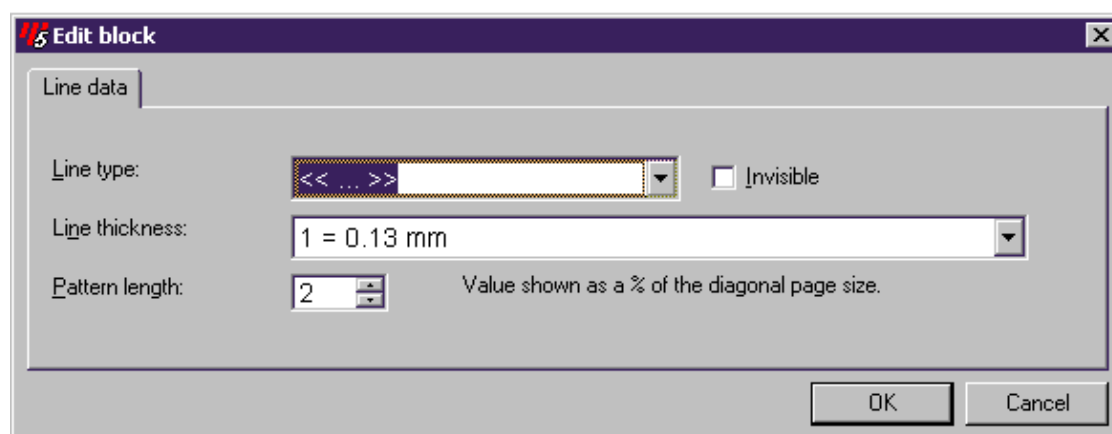
Once you have selected an executable option, the **Edit block** dialog box appears, in which the tab of the selected property type is active. In this tab, you will only find the properties that the selected schematics objects have in common. In some cases, EPLAN only displays one common property. If you have highlighted schematic objects with differing values, the entry "<<...>>" is shown in the corresponding input fields (cf. the example below).

To standardize the symbol properties of all highlighted schematic objects, simply change the values indicated in the fields or dropdown lists and confirm by clicking **[OK]**.



You have highlighted two schematic objects, a black box and a cable-definition line whose line data you want to edit at one go.

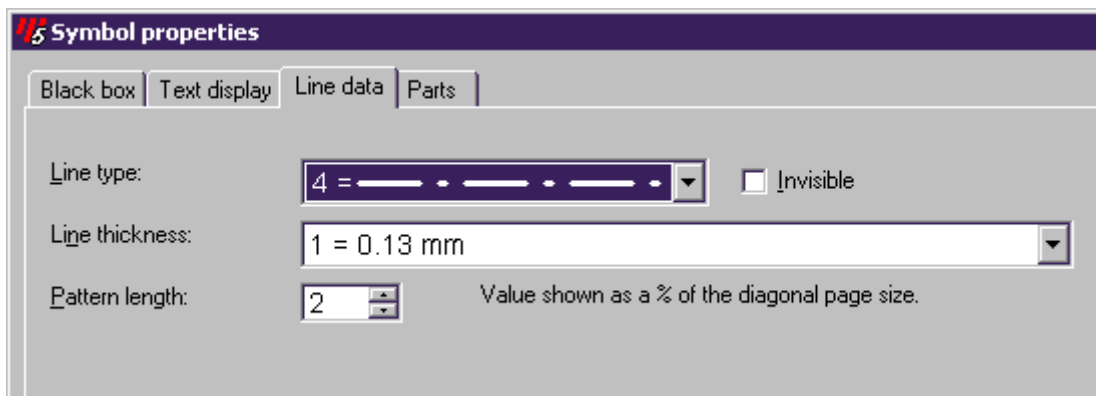
In the Edit block dialog box, which appears after the selection of the Block editing function, you will find the "<<...>>" entry in the Line type field in the Line data tab.



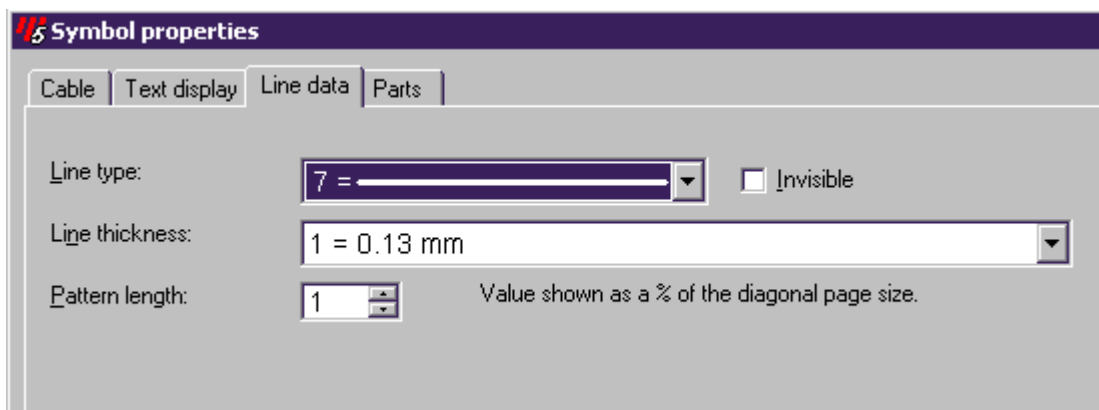
The values of the common symbol properties "Line thickness" and "Pattern length" are identical, whereas the "<<...>>" entry in the Line type dropdown list indicates that the two schematic objects have different values for this symbol property.

To check these differing entries, exit the block editing function, and call up the symbol properties of schematic objects individually.

The Line data tab of the **black box** displays the entry "4" (dash-dot-line) in the Line type field (see figure below).



However, the value selected in the same field in the Line data tab of the **Cable definition** is "7" (solid line) (see next figure).



6.3 Understanding Object-related Differences for Editing Blocks

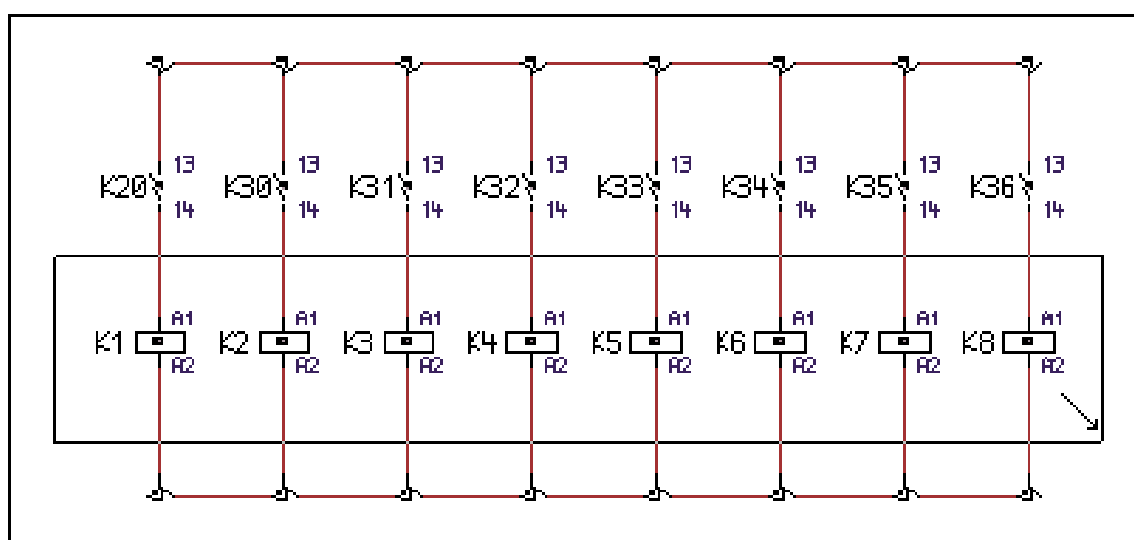
When editing a block, it is not only possible to edit the properties of similar schematic objects, but also of heterogeneous ones, e.g. a combination of cable-definition lines, black boxes, and objects of the free graphics.

Editing similar schematic objects

Similar schematic objects are objects whose properties are completely identical. When, for example, highlighting terminals that are taken from one and the same symbol file or all contactor coils of the current project page, you are selecting similar schematic objects.



You want to modify the device designation and the connection designation of all contactor coils inserted on the current schematic page. First, highlight the contactor coils as shown in the schematic below.



The following options can then be activated in the submenu of the **Block editing** function:

- **Devices** (symbol texts and symbol data)
- **Text display** (symbol texts)
- **Parts**.

Once you call up the **Devices** option, EPLAN opens the Edit block dialog box which contains tabs for the options that were available in the submenu (cf. figure below). The Devices tab is in the foreground of this dialog box and can be edited immediately. If you want to edit the text display or the part data first, simply click on the respective tab.

Edit block

Device | Text display | Parts

Device designation: << ... >>

Connection designations: A1 A2

Techn. characteristics:

Supplementary field:

Function text:

Engraving data:

Mounting location:

Symbol number: 20

Symbol text: K [K] DIC_WUPE(1) Relay coil 07-15-01

Angle variant: 0 = 0°

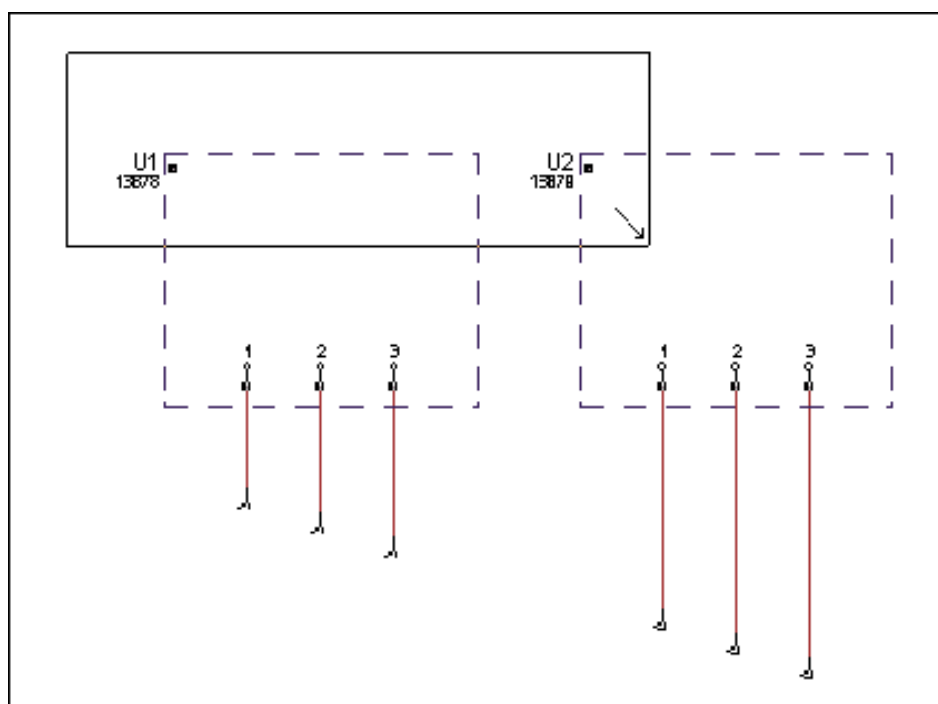
Symbol type: 50 = Main coil

Contactor selection Reserve contacts

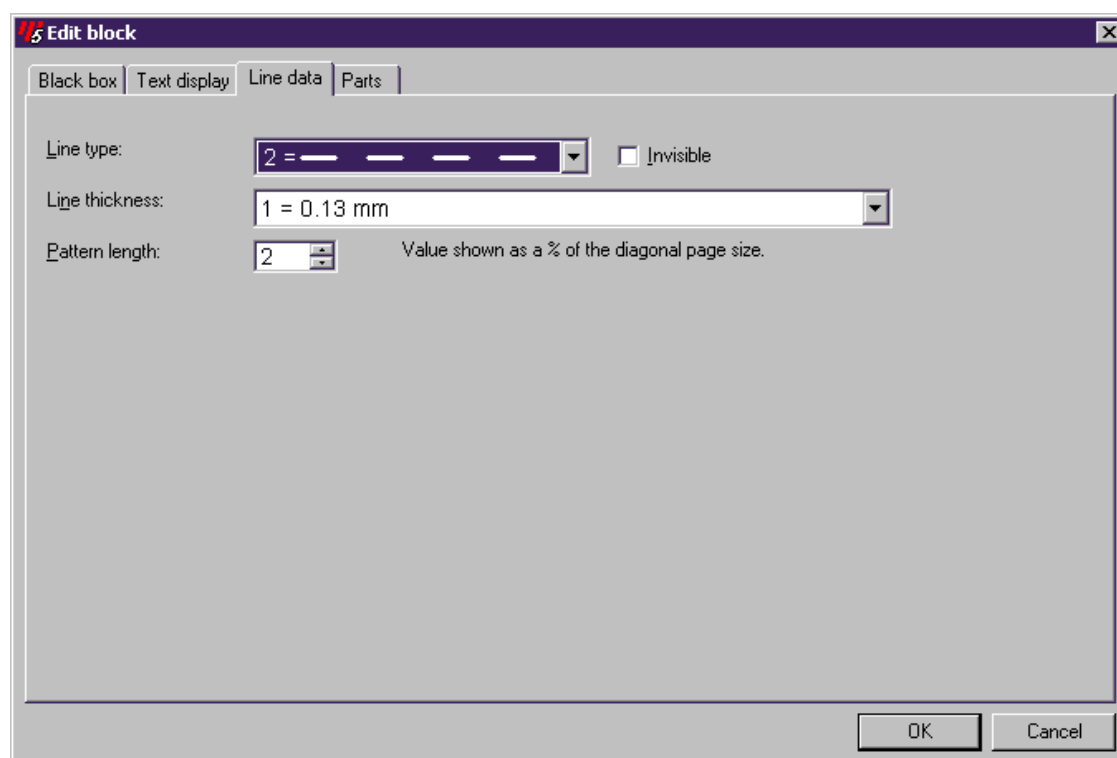
OK Cancel

As contactor coils don't possess line data properties, the **Line data** option and its corresponding tab are not activated and displayed respectively.

This is no longer the case if you include, for example, two black boxes into your selection for block editing as shown in the following figure.



Both schematics objects possessing line data properties, these data can be modified together in the Line data tab of the Edit block dialog box (see the following figure).



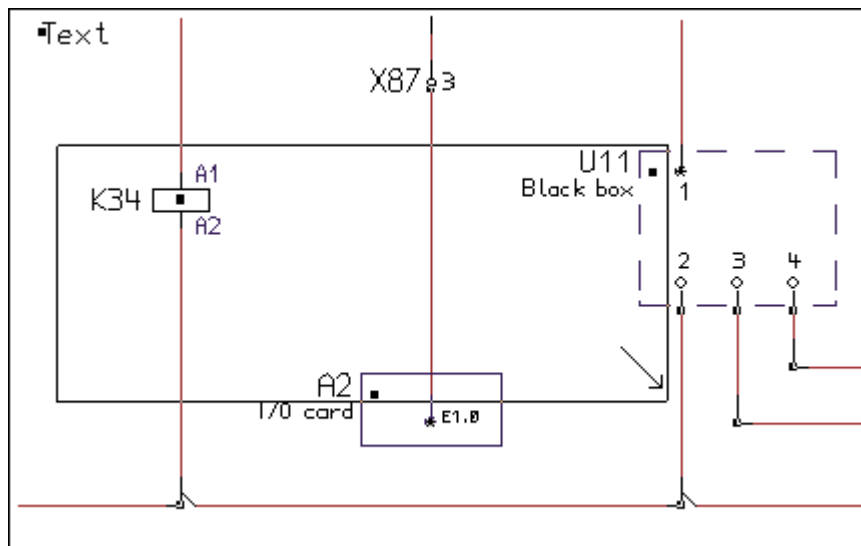
If you have highlighted several **text objects** on the schematic page whose symbol properties you want to modify at one go, you can only use the **Devices** menu option to do so. When in block editing mode, EPLAN treats the text object like an inserted device.

Editing heterogeneous schematic objects

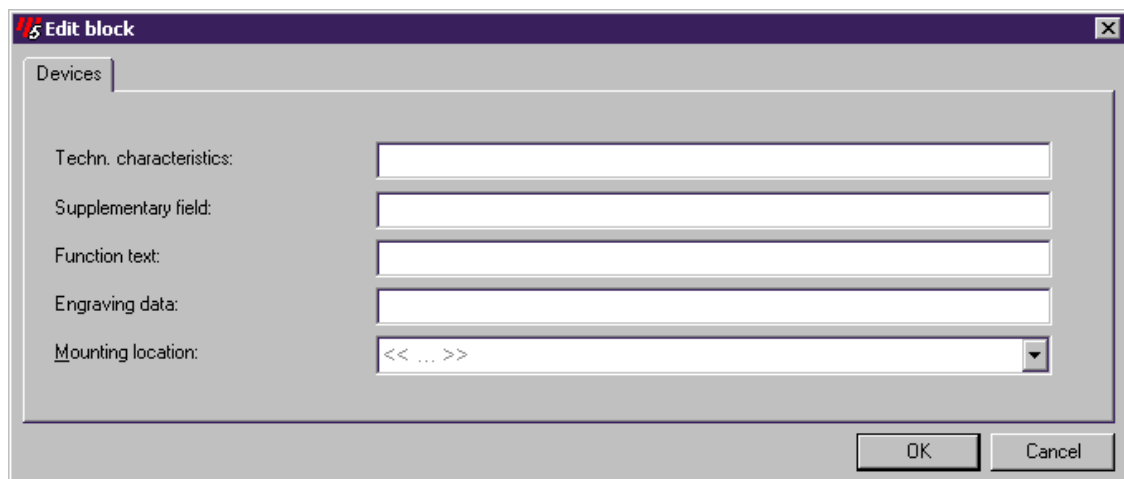
Heterogeneous schematic objects comprise objects that have both common properties and differing ones.

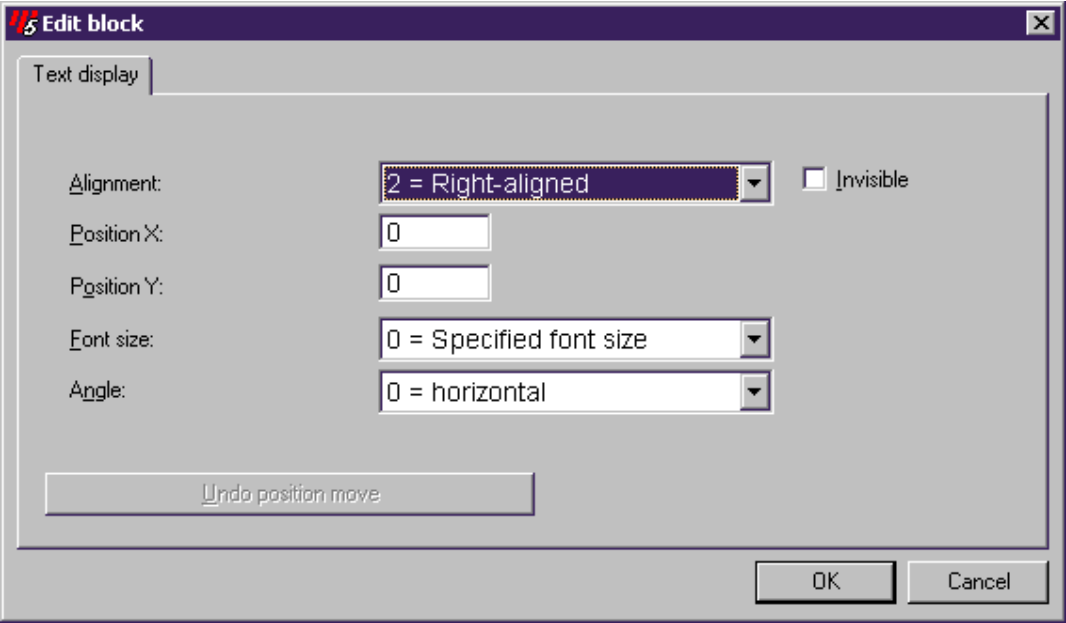


Highlight a contactor coil (K34), a black box (U11), and an I/O board (A2), as shown in the schematic section below.



You can now change the device data (see the following figure), the text display, and the parts data (see figures on the following page) of these objects. The device data only contain the symbol texts. The symbol data (symbol number, symbol type, and angle variant) are not available since they are not properties of all these schematic objects.





Edit block

Text display

Alignment: 2 = Right-aligned ☐ Invisible

Position X: 0

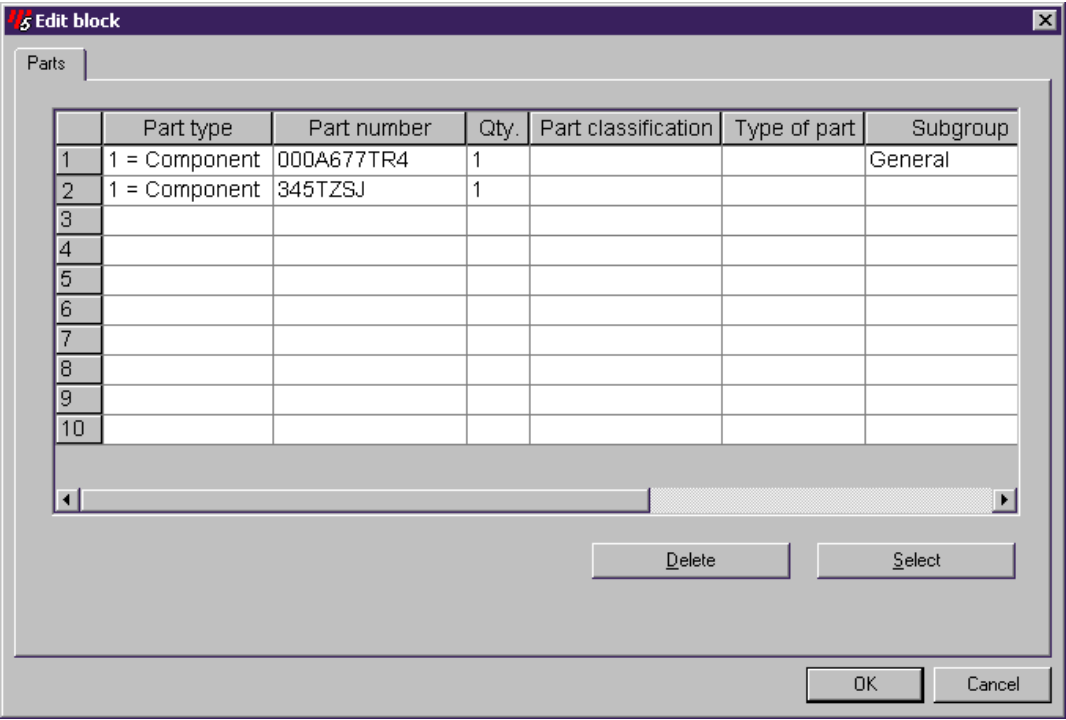
Position Y: 0

Font size: 0 = Specified font size

Angle: 0 = horizontal

Undo position move

OK Cancel



Edit block

Parts

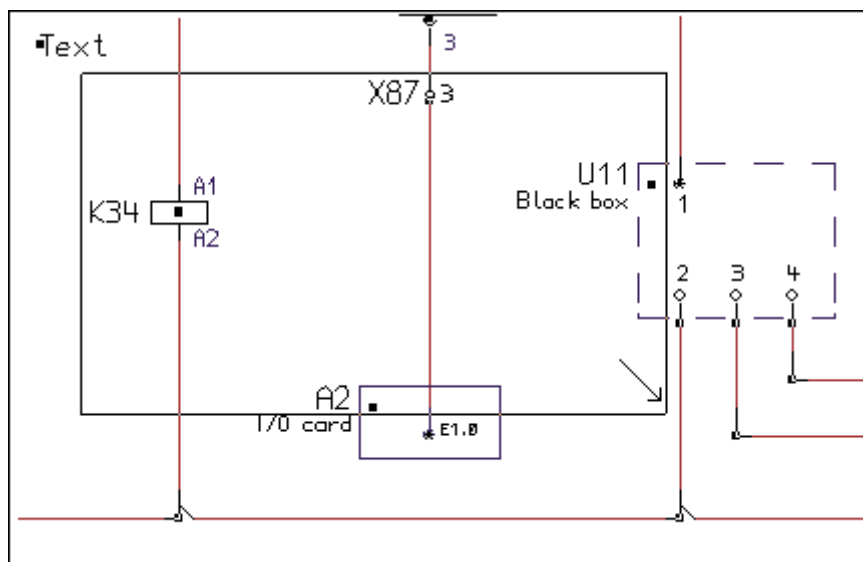
	Part type	Part number	Qty.	Part classification	Type of part	Subgroup
1	1 = Component	000A677TR4	1			General
2	1 = Component	345TZSJ	1			
3						
4						
5						
6						
7						
8						
9						
10						

Delete Select

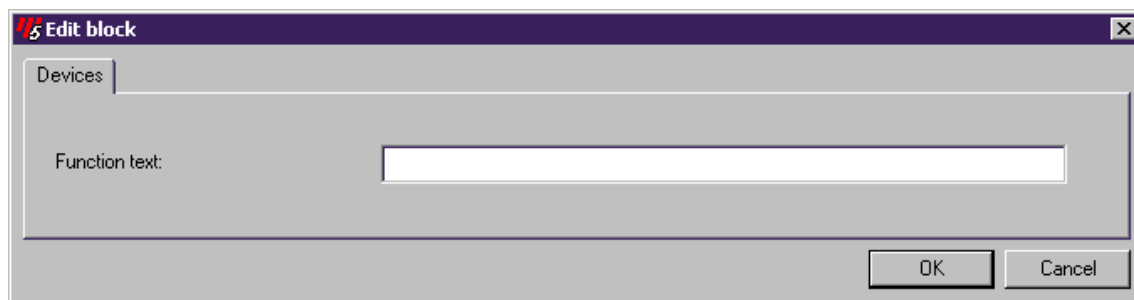
OK Cancel

The line data cannot be edited either, because only the black box and the I/O board have these properties.

If you now change your selection and, for example, include a terminal (X87:3) taken from a symbol file, as shown in the schematic section below, the number of fields offered on the Devices tab is further reduced.

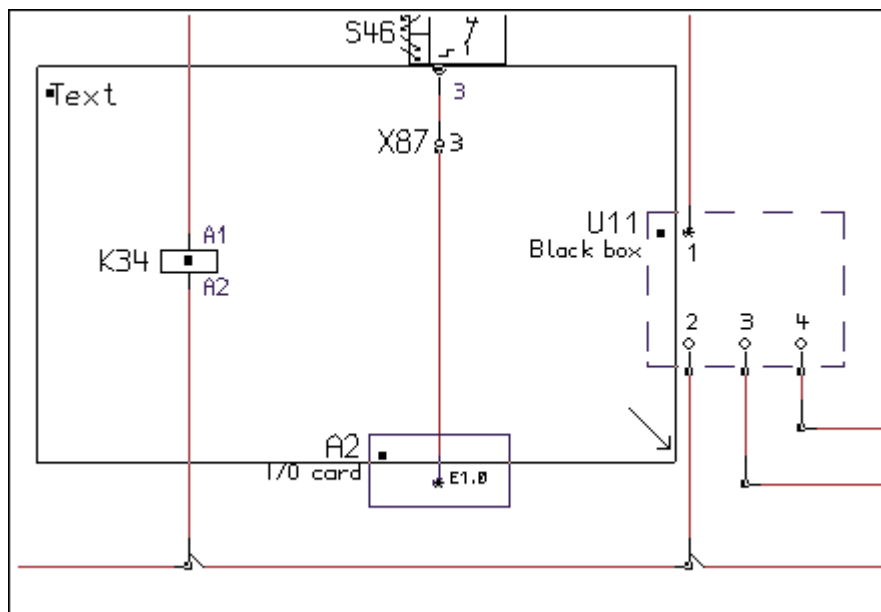


It is then only possible to edit the function text for all schematic objects at the same time.

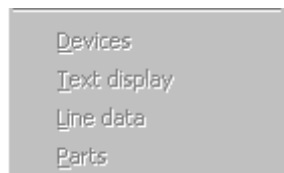


The Text display tab is not available to be edited for this selection because the terminal (X87:3) does not have this property type.

If you now include a text object (text) into your selection (cf. the following schematic section), you can no longer change any device data.



*In EPLAN, this is made clear by the grayed out **Devices** menu option. The text display, the line and parts data cannot be edited, either.*



7 Including External Documents

EPLAN 5.50 now offers a new page type allowing you to include non-EPLAN documents in your project. For this reason, the External files function is not needed for EPLAN Version 5.50. This means that the integration of external documents has been considerably facilitated, since, for example, external documents are no longer called up via a control file.

One project page can always include only **one** external document. However, it is possible to create several pages with external documents in a project.

7.1 Creating Pages of the External Document Type

You can select the corresponding page type "X = External document" when creating a new page (see Chapter 5.2 on Page 27). The Document name input field in the page properties dialog box will then be activated (see Fig. 33). Specify the non-EPLAN document here you want to include in your project.

The screenshot shows the 'Seiteneigenschaften' (Page Properties) dialog box. The 'Typ:' dropdown menu is set to 'X = Fremddokument'. Below it, the 'Benennung:' field is empty. The 'Ersteller:' field contains 'MOE'. The 'Lizenznr.:' field contains 'WUP9000060'. The 'Letzter Bearbeiter:' field contains 'TBL'. The 'Seiteneigenschaften 1' tab is selected. In the 'Seiteneigenschaften 1' section, the 'Zeichnungsnummer:' field is empty. The 'Zusatzfeld Seite:' field is empty. The 'Zusatzfeld Gruppe:' field is empty. The 'Dokumentname:' field is active and contains the path 'D:\TMP\Fremddokument.doc'. A checkbox next to 'Dokumentname:' is checked. A small icon button is visible to the right of the 'Dokumentname:' field.

Fig. 33: Section of the page properties for an external document

If this document is **not** located in your **current project directory** (e.g. \EPLAN4\P\

If the external document is located **in your current project directory**, you only need to specify the file name and extension.

A checkmark in the grayed ☒ check box indicates whether the specified document exists.

Notes:

- When entering the document name for the external document, you cannot specify relative paths such as \\Server\Directory\File.end.
- If the external document should be considered for the data backup, it must be located in the current project directory. This is, for example, important if you want to send an "entire" project (i.e. including the external document) to customers.

Once the page properties dialog box has been closed with **[OK]** or the properties of the new page have been confirmed by clicking **[Apply]**, the new page will be displayed in the page overview.

In the page overview, the icon that has been specified in the operating system for the corresponding file type is used for the external document. For example, the familiar Word[®] icon  is assigned to a Microsoft[®] Word[®] document. A document with an unknown file type receives the EPLAN icon  (see Fig. 34).

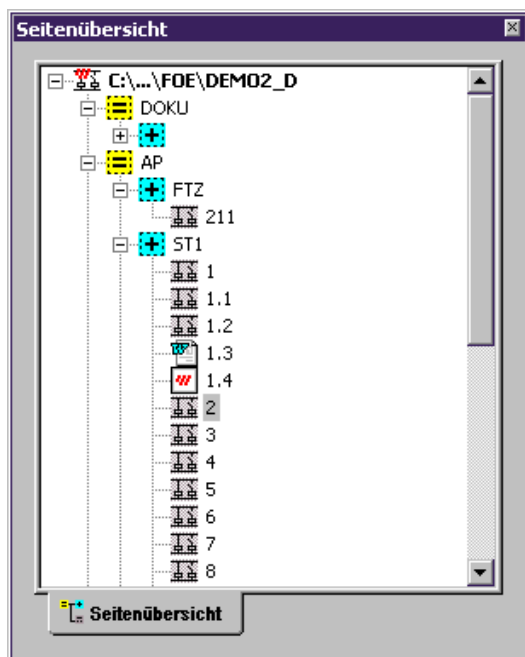


Fig. 34: A sample project including two external documents

7.2 Opening External Documents

An external document is opened in the same way as any "normal" EPLAN page, e.g. by selecting the **Open** menu option (see Chapter 5.3 on Page 29). In contrast to a project page, no EPLAN working window is launched afterwards but the program linked to the file type (e.g. Microsoft® Word).

You can edit and modify the document in this "external" program and save these modifications there. It is also possible to keep this program in the background or on the task bar while editing a "normal" project page in EPLAN. If you want to close the external document, exit the respective program.

7.3 Editing Pages of the External Document Type

As external documents cannot be displayed and edited in an EPLAN working window, not all options of the **Page** menu can be selected for this page type. The following functions are not supported for an external document:

- Open page in new window
- Save page
- Close page
- Read in backup page
- Empty page

and

- Next/Previous page.

Furthermore, external document pages are not displayed in the page preview.

However, renaming, moving, and deleting an external document and displaying and editing its page properties is done in the same way as for "normal pages".

When copying a project containing pages of the External document type, the external document is only copied to the project directory of the target project if the external document "belongs" to the source project. In other words, only the file name and extension were entered in the Document name input field which means that the external document is located in the project directory of the source project.

If you select the **Print** menu option for an external document, the familiar Print dialog box is called up first (see Fig. 32 on Page 48). Once you confirm with **[OK]**, EPLAN opens the corresponding program briefly and closes it again. The external document will be printed afterwards.

Notes:

- Pages of the External document type can **only** be exported in EXF as well as in HTML/DWF format. During the EXF export, EPLAN only outputs the page information and the document name. When exported to the HTML/DWF format, external document pages are output as HTML/DWF pages with a hyperlink to the corresponding included external document.
- An EPLAN project page **cannot** be subsequently converted into a page of the External document type by modifying the page properties!

8 Inserting Hyperlinks on Project Pages

External documents cannot only be included in your project as foreign documents (see Chapter 7 on Page 67). A further new function now allows you to insert **links to external documents**, so-called "hyperlinks", on all project pages. These external documents may be local documents on your computer, documents on a network, or documents on a network. This way, you can for example store additional information about a component on a schematic page without risking to impair the clear structure of this page.

To **insert a hyperlink**, select the **Hyperlink** option from the **Insert** menu. The corresponding Properties dialog box will then be opened (see Fig. 35).

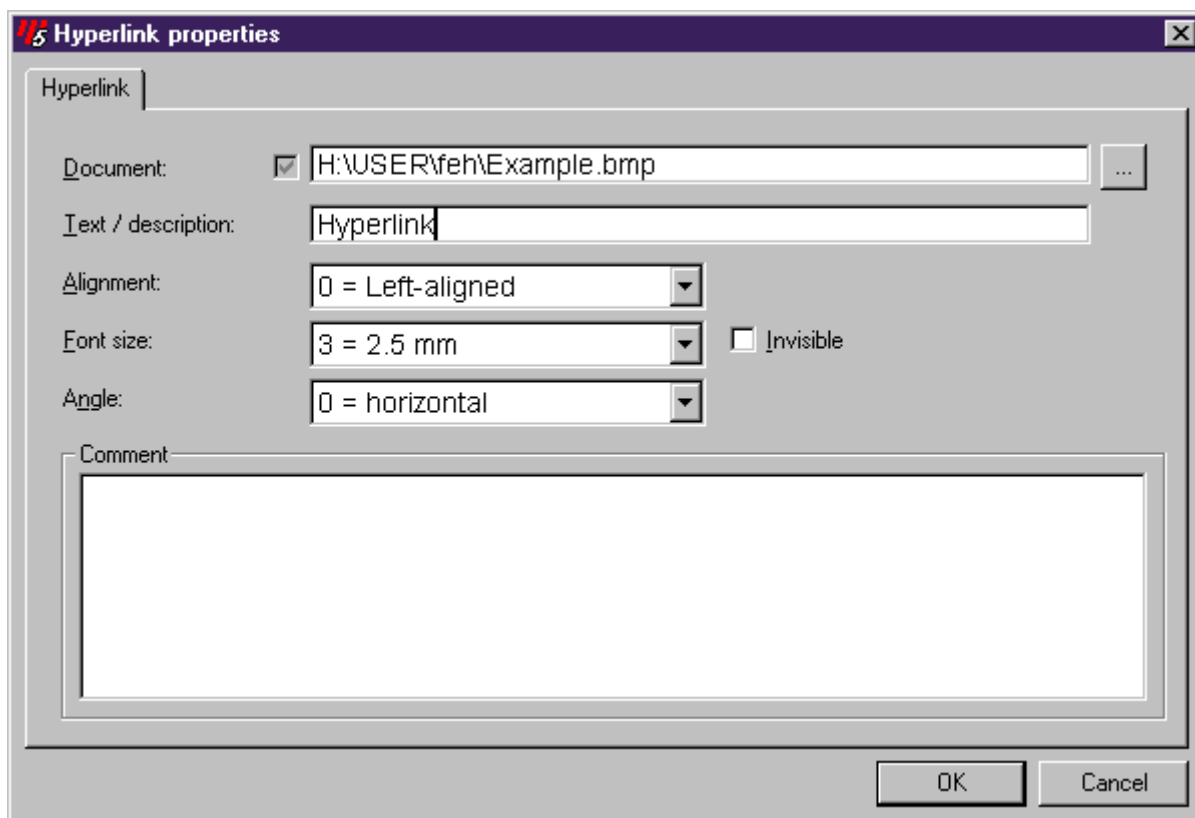


Fig. 35: Properties dialog box for hyperlinks

First, enter the document to be linked into the Document field. Similar to the foreign documents, the exact entries depend on the fact whether the document to be linked is located **in the current project directory** or **not**.



If the document to be linked should be considered for the data backup, it must be located in the current project directory. This is, for example, important if you want to send an "entire" project (i.e. including the linked document) to customers. If the document is not located in the current project directory, only the inserted hyperlink text on the page will be saved during the data backup. The linked document won't be considered.

Enter the text, by which the hyperlink will be stored on the project page, into the Text / Designation field. The display of this hyperlink text can be specified by the familiar Alignment, Size, Angle fields and by the Invisible check box, just like when inserting a "normal" text.

You can enter any text into the Remark field. For example, this text can describe the linked document in more detail (author, changes to the document, etc.). The remark will not be displayed on the project page.

After having confirmed your entries by clicking **[OK]**, you can place the hyperlink text with the insertion point on the project page. This way, you can also insert several hyperlinks on a page.

To **open** a **hyperlink**, move the cursor to the insertion point of a hyperlink. Select the **Open** option from the pop-up menu. The linked document is then started by the corresponding program.

As an alternative, you can also select the **Open hyperlink** option from the **Edit** menu. If you call up the function this way, the cursor symbol first changes. Draw a box around the insertion point of the desired hyperlink text. The linked document is then also called up by the corresponding program.

9 Designing Schematics in Single-pole Display

In addition to creating schematics in multipole display, some industries prefer the single-pole display of schematics. Schematics in single-pole display show a simplified structure and operation of the plant.

Up to now, it was only possible to create multipole schematics in EPLAN. As from Version 5.50, you can also document your projects in the single-pole display. For this reason, you can now also create overview diagrams in the form of single-line diagrams according to DIN.

And that is not all, EPLAN even enables you to manage the multipole display along with the corresponding single-pole display within the same project (not the same page). It doesn't make a difference which display was created first.

When designing wiring diagrams in single-pole display, you have to keep some special features in mind, which are described in the following sections. You will be provided with detailed information on the topics:

- Setting the Page Type
- Using Symbol Files
- Special Points when Inserting Symbols in Single-pole Display
- Special Points when Using Macros in Single-pole Display
- Special Points when Entering Parts Data in Single-pole Display
- What to Consider when Generating Interruption Point Cross-references
- What to Consider when Inserting Wire Designations on Pages in Single-pole Display
- Search Function.

9.1 Setting the Page Type

To create a schematic page in single-pole display, a specific page type must be set in the page properties dialog box.

To do so, point the mouse on the **Page** option of the graphics editing menu. In the subsequent pull-down menu select the option **New**. The Create new page dialog box appears. In the Type field, select the option "**W = Single-pole display**".

When inserting symbols into this schematic page you can only take symbols from a "single-pole" symbol file.

9.2 Using Symbol Files

To insert symbols in single-pole display on a page of type "W" in the schematic, a separate symbol file must be available, which must be entered in the parameter program first. For this purpose, follow the menu path **Utilities > Parameters > Project > Symbol file** in the EPLAN main menu. The **Symbol file selection** dialog box comes up. You will immediately notice that we have extended and modified this dialog box compared to previous EPLAN versions.

Instead of 8 symbol files, it is now possible to enter or select twice the number of symbol files into the corresponding fields. Multipole symbol files are entered into the **Multipole** column on the left, whereas the fields in the **Single-pole** column on the right-hand side are intended to be used for entering or selecting single-pole symbol files.

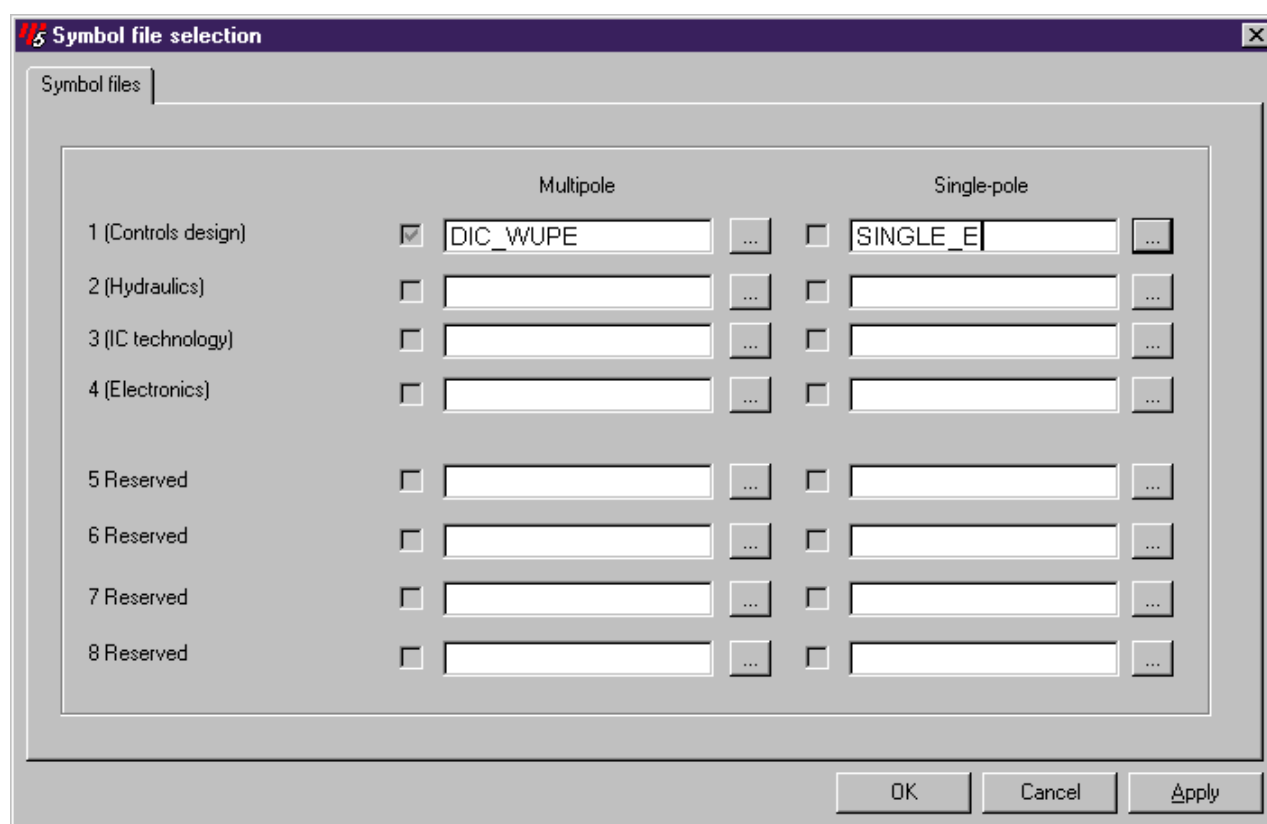


Fig. 36: Dialog box for the symbol file selection

Notes:

- Our supply of master data includes the single-pole symbol file **SINGLE_E.SYM**, which is available on the EPLAN CD. Its contents corresponds to the multipole symbol file **DIC_WUP**.
- If you try to insert a symbol on a page of the Single-pole display type and no single-pole symbol file was entered, EPLAN displays a corresponding message.
- Note that there will be no check for the type of the symbol file entered. This means that if you have unintentionally entered a "single-pole" symbol file as a "multipole" symbol file or vice versa in the parameter program, this will result in an incorrect display. You will also receive incorrect data if you start an evaluation run with an erroneous symbol file setting. For this reason, always take care in assigning the symbol files to the **right** category!

9.3 Special Points when Inserting Symbols in Single-pole Display

When symbols from a "single-pole" symbol file are inserted into a project page of type "W = Single-pole display", the connection data-relevant input fields are not available in the respective symbol properties dialog boxes. Which fields are no longer required on the single-pole schematic pages depends on the symbol that is inserted in the schematic.

The display of the Symbol properties dialog box is changed for the following symbols:

Symbols from a symbol file (without terminals and connectors)

The Connection designations input field **is no longer available** in the Device tab of the Symbol properties dialog box.

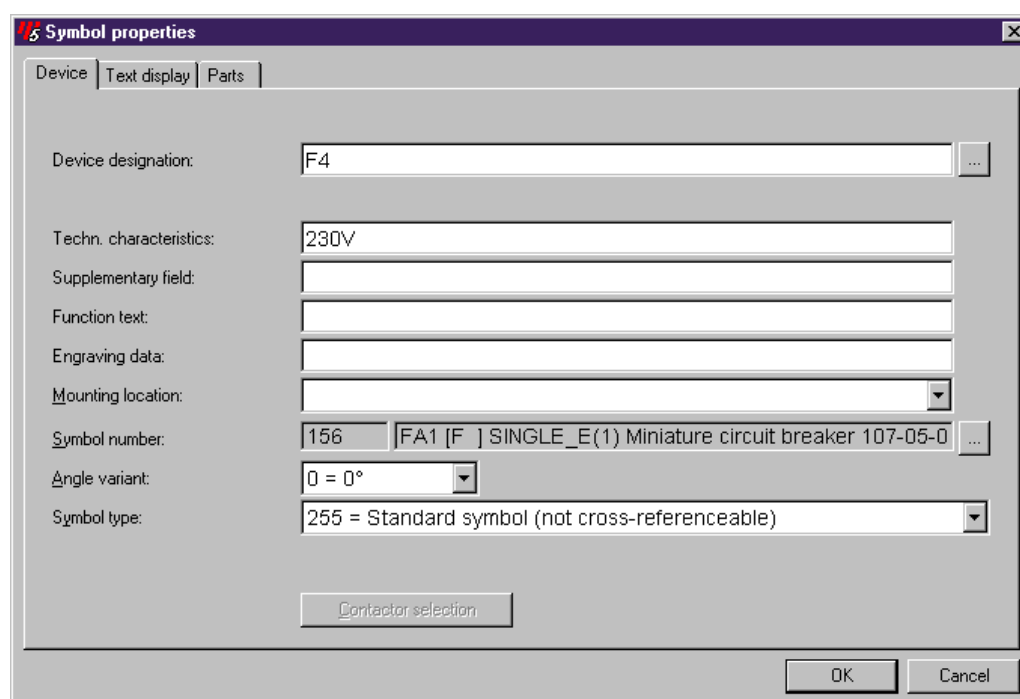


Fig. 37: Symbol properties dialog box for symbols taken from a single-pole symbol file

Terminals / connectors from a symbol file

In the Symbol properties dialog box for terminals or connectors, the following fields are **not displayed** in the Terminal / connector pin tab:

- Connection of the external output position in form
- Connection for cable generation
- PLC designation
- Function text
- Terminal strip search direction
- Sorting identifier

- Check identifier
- Jumper bars.

Symbol properties

Terminal strip / connector strip | Text display | Parts

Strip designation:

Specify strip definition:

☒ None ☐ Terminals ☐ Connectors

Definition text:

Symbol number: Angle variant:

Symbol type:

Terminal-strip definition

Minimum number of numerical terminals/strip:

Form:

Include in

☒ Print output

☒ Graphics output

☒ Output of the terminal connection diagram

☒ Output of terminal bill of materials / of terminal line-up diagram

Connector strip definition

Connector type name:

Form:

Include in

☒ Print output

☒ Graphics output

☒ Output of the terminal connection diagram

☒ Output of terminal bill of materials / of terminal line-up diagram

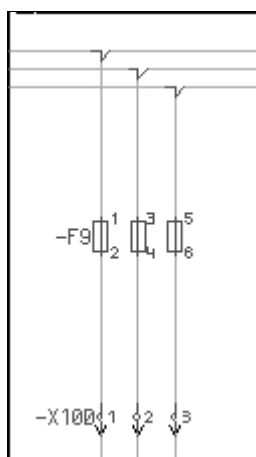
Fig. 38: Symbol properties dialog box for terminals/connectors from a single-pole symbol file

9.4 Special Points when Using Macros in Single-pole Display

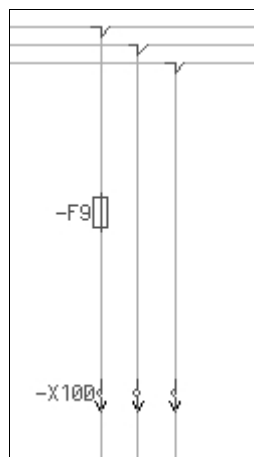
When macros drawn in multipole display are inserted into a page of the Single-pole display type, EPLAN removes the conductor and connection designations. This is also the case if macros are pasted from the EPLAN Clipboard.

If you create a macro on a page of the Single-pole display type and insert it into a schematic page in multipole display, EPLAN displays the single-pole symbols as multipole symbols. This behavior also applies to the EPLAN Clipboard function.

The other way round, i.e. if you insert a schematic macro drawn in multipole display into a page of the Single-pole display type, all multipole symbols contained in the macro are shown as single-pole symbols (see example below). This behavior also applies to the EPLAN Clipboard function.



Multipole macro...



... after the insertion into the "single-pole" page

9.5 Special Points when Entering Parts Data in Single-pole Display

Similar to the "multipole" page types, you can assign a part number to each device that you insert on a "single-pole" schematic page. If you want to know whether you have assigned the same part number to a device in the multipole display as well as in the single-pole display, you can have this checked during the generation of the bill of materials by activating a menu item. When EPLAN discovers that the same parts data were specified in both display variants for a device, a message will be output in the message window.

9.6 What to Consider when Generating Interruption Point Cross-references

You can insert, edit, and cross-reference interruption points on pages in single-pole display, just like in schematics in multipole display. However, please note: interruption points located on a "single-pole" schematic page can only be cross-referenced to interruption points of the same page type, thus "W = Single-pole display"!

9.7 What to Consider when Inserting Wire Designations on Pages in Single-pole Display

It is not possible to insert wire designations on pages of the Single-pole display type.

9.8 Search Function

A search for schematic objects by means of search terms can be also be carried out on single-pole logic pages.

10 Online Data Transfer when Copying Schematics Pages

EPLAN 5.50 now offers new convenient online functions, which make additional generation runs after the copying of schematics pages unnecessary and thus help you to save time.

The following section provides information about these on-line functions when copying schematic pages. EPLAN automatically updates cross-reference databases as well when renaming and deleting schematic pages.

10.1 Transferring Interruption Point Cross-references Online

If you copy schematics pages with interruption points (was: "potential arrows"), the interruption point designations are entered online into the interruption-point cross-reference database. As – because of the existing original page – identical interruption point designations already exist in the database, EPLAN creates a cross-reference. This cross-reference automatically appears at the interruption points on the copied schematics page. Therefore you do not need to start a new interruption-point cross-reference run.

10.2 Transferring Contact Cross-references Online

If you copy schematics pages containing devices that can be cross-referenced, the device designations are entered online into the contact cross-reference database. Here, too, EPLAN creates a cross-reference since identical device designations already exist in the database because of the existing original page. This cross-reference automatically appears at the respective device on the copied schematics page. Starting a new contact cross-reference run is therefore no longer necessary.

10.3 Transferring Online PLC Cross-references

If you copy pages of the type "A" or "Q" (logic pages) or of type "T" (overview pages) containing PLC terminals, EPLAN enters the new PLC data into the PLC cross-reference database. EPLAN creates a cross-reference since PLC data already exist in the database because of the original page. This cross-reference is now displayed at the respective PLC terminals on the logic or overview page without you having to start a new PLC cross-reference run.

10.4 Transferring Device Designations Online

Up to now, it was only possible for the insertion of macros or the copying of devices to determine – automatically or via selections – the behavior of the copied devices during the insertion. The selection is influenced by the setting of the project-specific graphics parameter 'Transfer of DDs from macros / during copy actions' (Graphics 4 tab).

As from EPLAN Version 5.50, this function, which is defined in a selection dialog box, is also available for the transfer of device designations when copying schematics pages.

This selection is made in the Copy pages dialog box (menu path **Project > Graphics editing, output > Page > Copy from / to**) after the target page number was entered and confirmed with **[OK]**. EPLAN then opens the Use of the device designation dialog box (see Fig. 39). The following options are available here:

- **Complete**
EPLAN inserts the identifiers without any changes.
- **Until identifier**
The counter behind the identifier is replaced with a "?".
- **Numbering by means of online DD**
The numerical counter of the DD is increased.

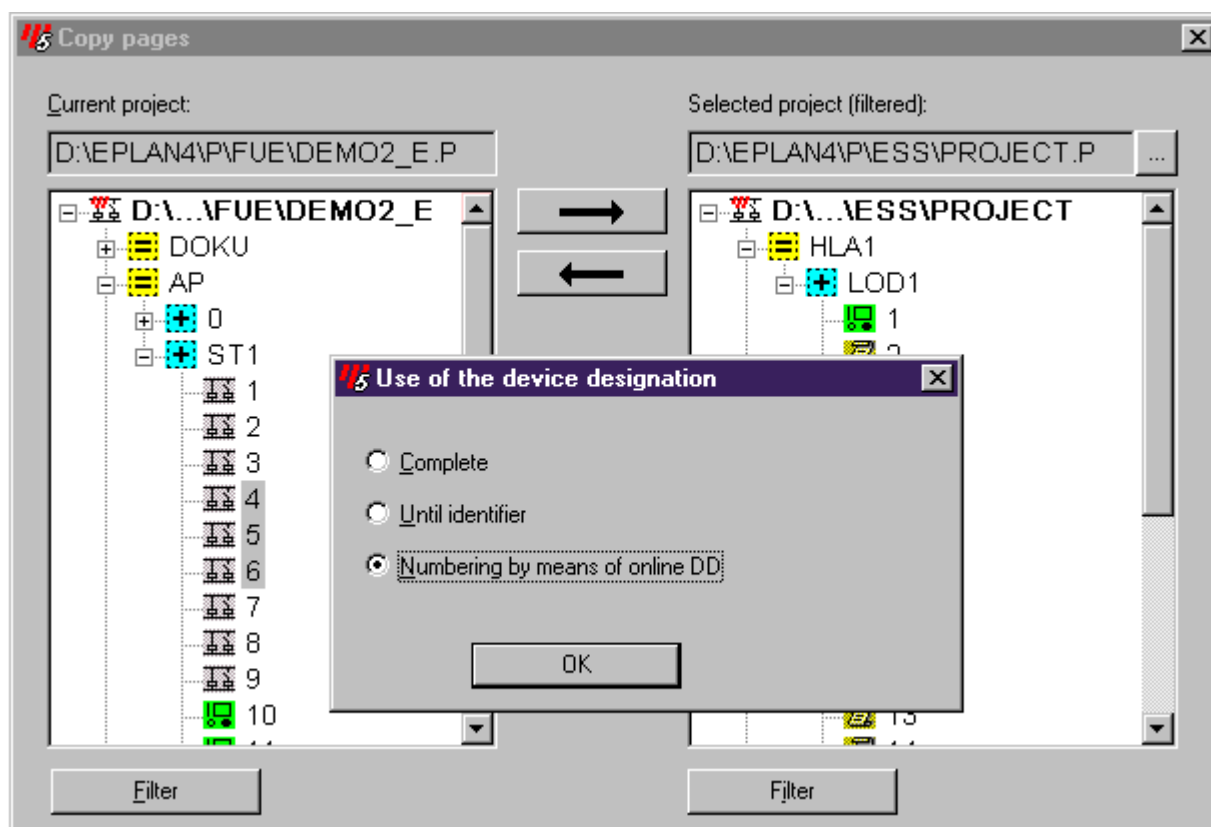


Fig. 39: Transfer of device designations when copying pages

Similarly to the graphics parameter, which controls the transfer of device designations when devices are copied and macros are inserted, the display of the selection dialog box during the copying of pages depends on the setting of the project-specific graphics parameter "Transfer of DDs when copying pages", available in EPLAN as from Version 5.50. This parameter can also be found on the Graphics 4 tab and offers the same options as the above-mentioned parameter (see Fig. 40).

The following applies again: EPLAN only displays the selection window in the Copy pages dialog box if the 2 = Additional prompt option was selected. The default setting is 0 = Complete.

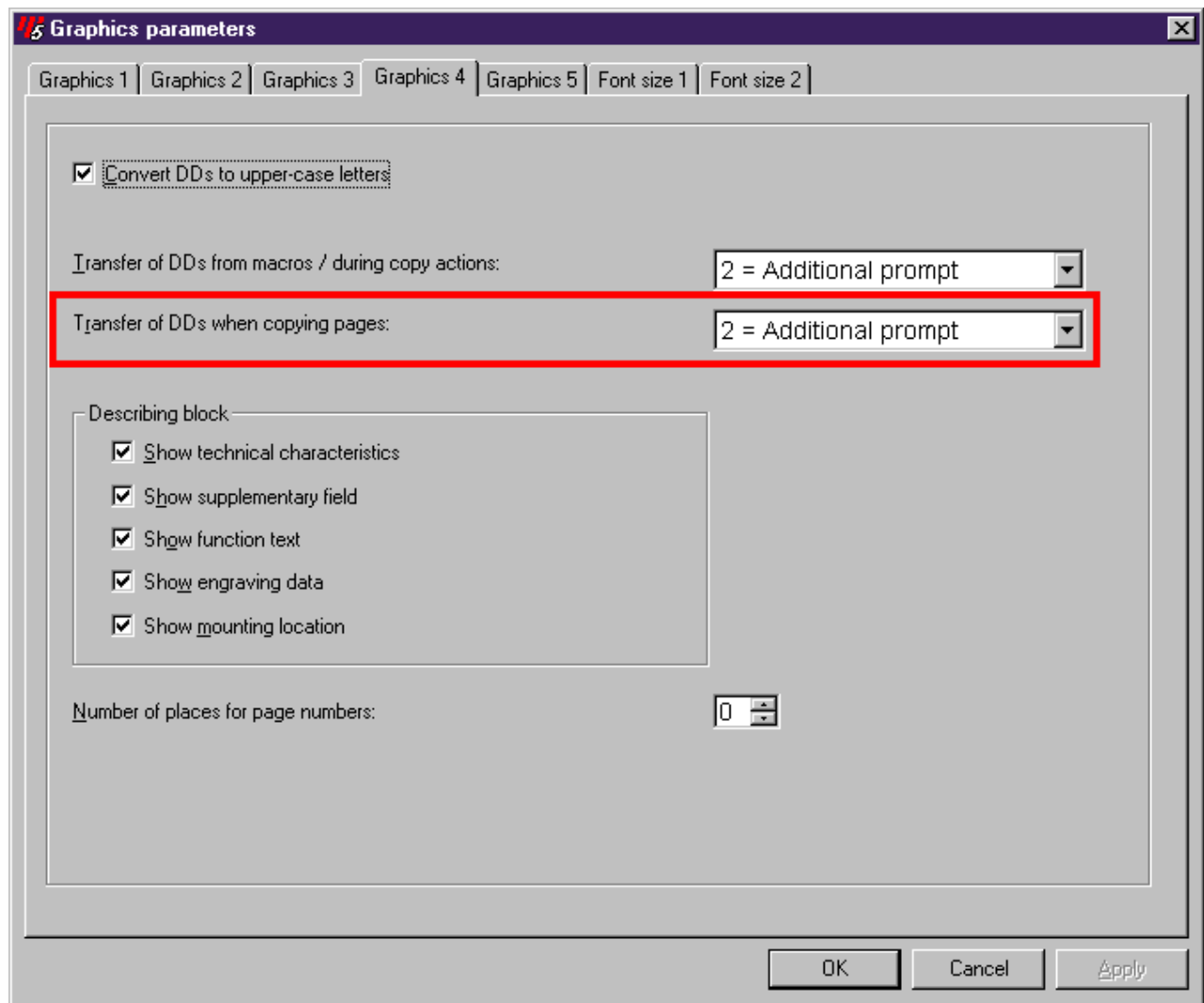


Fig. 40: Parameter for the transfer of device designations when copying pages

If you select the Numbering by means of online DD option from the Transfer of device designation selection dialog box and confirm by clicking **[OK]**, EPLAN copies the page and enters it at the specified position in the target project. During this process, all numerical counters of the device designations are increased by "1".

Notes:

- All devices on a page with the same identifier will receive the same identifier on the target page. Cross-references are also retained page-specifically, even if the numerical counter is changed.



Example 1:

The contactor coil K1 and the contact K1 are located on one specific schematic page. Copy this page to any position within the project. Both counters are increased by "1". After the copying process both devices have the same designation: "K2". The cross-reference is retained.

Example 2:

The contactor coil K1 and the contact K1 are located on two different schematic pages. Highlight and copy both pages together to any position in the project. Both counters are increased by "1". After the copying process both devices have the same designation: "K2". The cross-reference is retained.

- If you copy pages within the current project, EPLAN applies the designations of cables and interruption points without any changes.
- If you copy pages from the current project into another project, EPLAN also applies all device designations in addition to the interruption point and cable designations.
- Combined components (such as coils and contacts) can only be copied in **one** copying process, since otherwise the cross-references would be incorrect.



The contactor coil K1 is located on Page 1 and the associated contact K1 is positioned on Page 2. First copy Page 1. Afterwards, the contactor coil has the device designation "K2". Then copy Page 2 where the contact K1 is positioned. After the copying process has been completed, the contact has the device designation "K3". The original cross-reference therefore no longer exists.

If the pages to be copied have higher-level assignments or location designations which are not yet contained in the higher-level assignment/location designation database, EPLAN first displays the dialog box for positioning the identifiers before the Use of the device designation selection dialog box (also refer to Chapter 5.8 on Page 36).

11 Moving Symbol Texts

You will certainly know the Move DD function from previous EPLAN versions. We have improved this function and, in addition, renamed it with good reason.

In previous EPLAN versions, it was only possible to use this move function via the pop-up menu which is opened by right-clicking on the insertion point of a symbol. As from Version 5.50, you will find this function in the menu of the graphics editing module by selecting **Edit > Move symbol texts** (see Fig. 41).

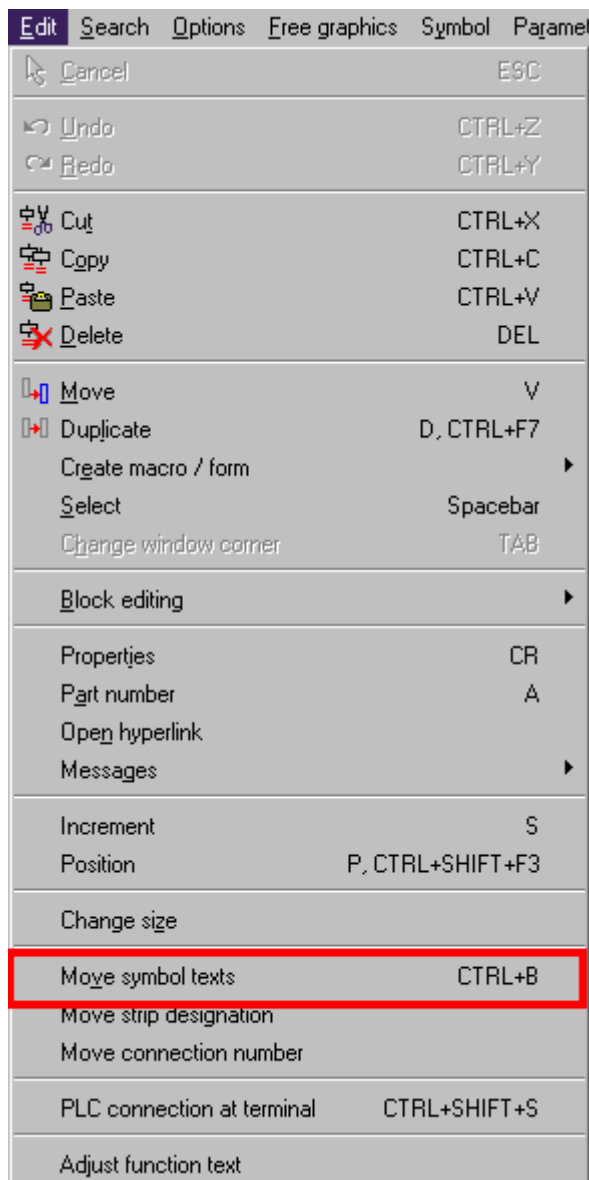


Fig. 41: **Move symbol texts** function in the **Edit** menu

We have renamed the function as you cannot only freely move device designations on the schematics page with the schematics object itself remaining at the insertion point, but also all texts of schematics objects which you have specified in the symbol properties dialog.

Please note that EPLAN also moves generated **cross-references** at symbols, if existing.

The following overview shows which texts of schematics objects are included in the move operation.

Symbols	Movable text (block)	
Symbols from a symbol file (without terminals and connectors)	Device designations Technical characteristics: Supplementary field Function text Engraving data Mounting location Contact cross-reference (if DD is cross-referenceable)	
Black, DIN, and PLC boxes	Technical characteristics Supplementary field Function text Engraving data Mounting location	
Interruption points	Designation Interruption point cross-reference	
PLC end terminals	Q page (PLC pages)	T page (PLC overview)
	Address PLC cross-reference Symbolic address Function text	Address PLC cross-reference Symbolic address Function text
Cable definition lines	Designation Cable type Number of conductors Cross-section Length Voltage Remark	
Wiring definition lines	Cross-section Length Color Remark	

Symbol texts can be moved via a menu option or by entering the coordinates directly. The following two sections provide detailed information on this subject.

11.1 Moving Symbol Texts via Menu Command

Select **Edit > Move symbol texts** in the Graphics editing menu.

The cursor then changes its shape: 

Now press the left-hand mouse button briefly. The cursor is transformed again and is now shaped like a frame:



Drag the frame around the insertion point of the corresponding symbol, and press the left mouse button again.

The cursor is displayed again as originally set, anchored to the symbol text (block).

Using the mouse or the arrow keys, you can now move the text or text block away from the symbol and shift it to any position within the schematic.

Press the left mouse button again to place it at the desired position, or hit the  key if you prefer to use the keyboard.

If you select the **Move symbol texts** option from the pop-up menu, which is called up by right-clicking on the insertion point of the schematic object, you can immediately move the text (block) to the desired position.

Symbol texts can be moved to the following project page types:

- A = Schematic page (logic, interactive)
- Q = PLC page (logic, automatic) (logical connection of PLC end terminal)
- T = PLC overview (logic, interactive) (logical connection of PLC end terminal).

11.2 Moving Symbol Texts by Specifying Defined Coordinates

Another possibility to move symbol texts is provided in the Symbol properties dialog box. Here, you can directly enter the coordinates for the symbol text position.

To do this, click on the insertion point of the desired symbol. In the Symbol properties dialog box that appears switch to the **Text display** tab (see Fig. 42). Overwrite the old values in the **Position X** and **Position Y** fields with the new data, and confirm your entries with **[OK]**.

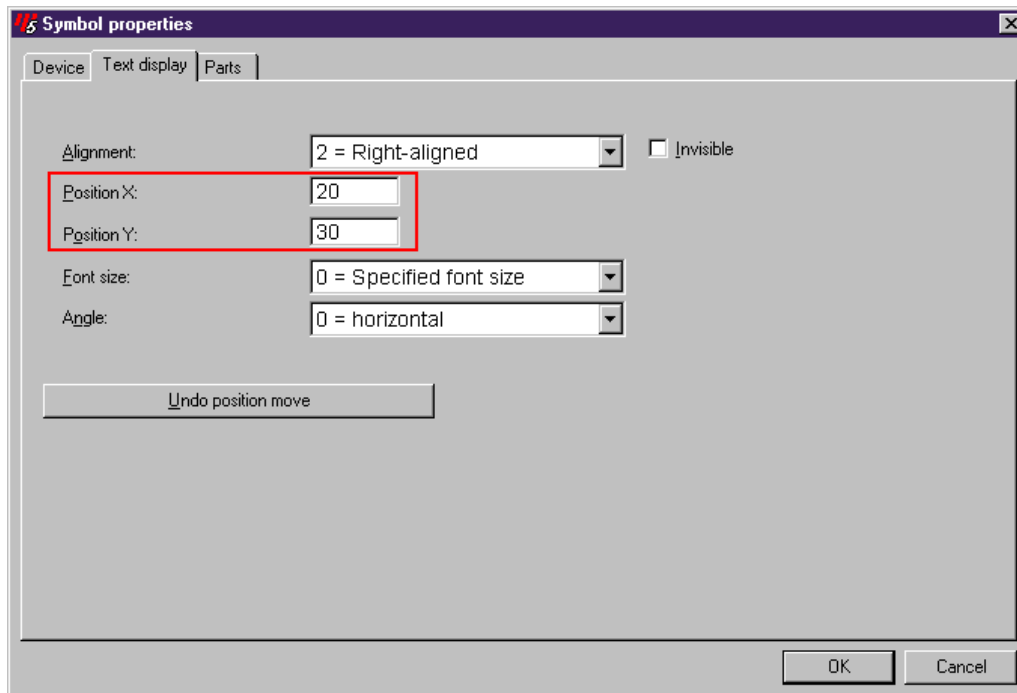


Fig. 42: Coordinate fields on the Text display tab

If you want to use this method to edit the coordinates of symbol texts at cross-referenceable PLC end terminals located on pages of type "T" (PLC overview), activate the **Text display 1** tab of the symbol properties dialog box (see Fig. 43 on Page 91).

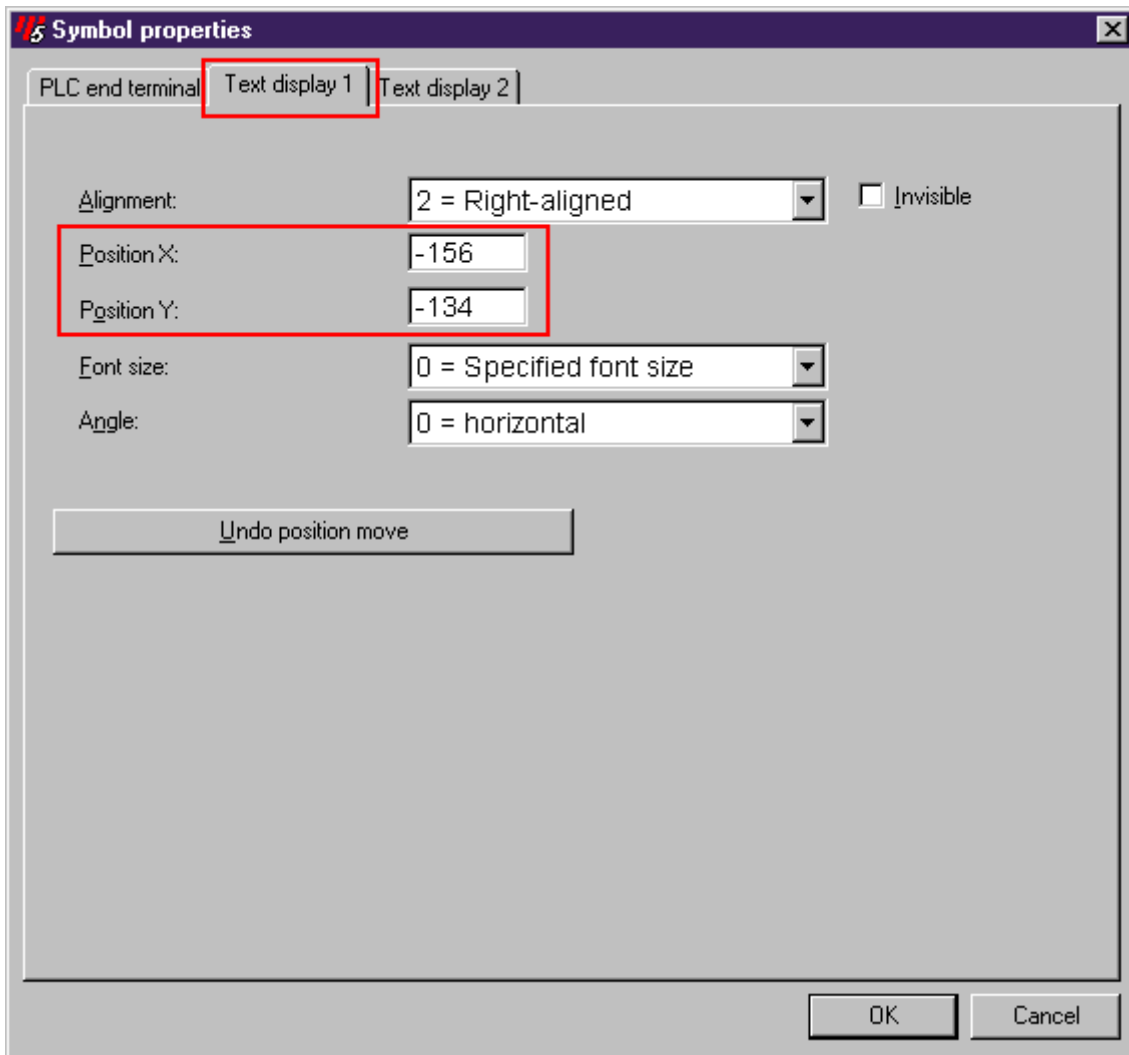


Fig. 43: Coordinate fields of the Text display 1 tab for PLC end terminals

11.3 Displaying the Relocation of Symbol Text

In previous EPLAN versions, the DD relocation was displayed by selecting **Edit > Move DD** from the menu of the graphics editing module. EPLAN then indicated the relocation in the schematic by means of a light green line.

This function has also been revised, given a new place in the menu, and renamed according to the **Move symbol text** function.

If you now wish to display the relocation of the symbol texts in the schematic, follow the menu path **Options > Multifunction menu > Display symbol text relocation** (see Fig. 44). EPLAN will then display a light green line between the insertion point of the graphics object and the new symbol text position, see schematic section in the following example). Selecting this function again will de-activate the display. A checkmark in front of the **Display symbol text relocation** option indicates whether the display is activated or not.

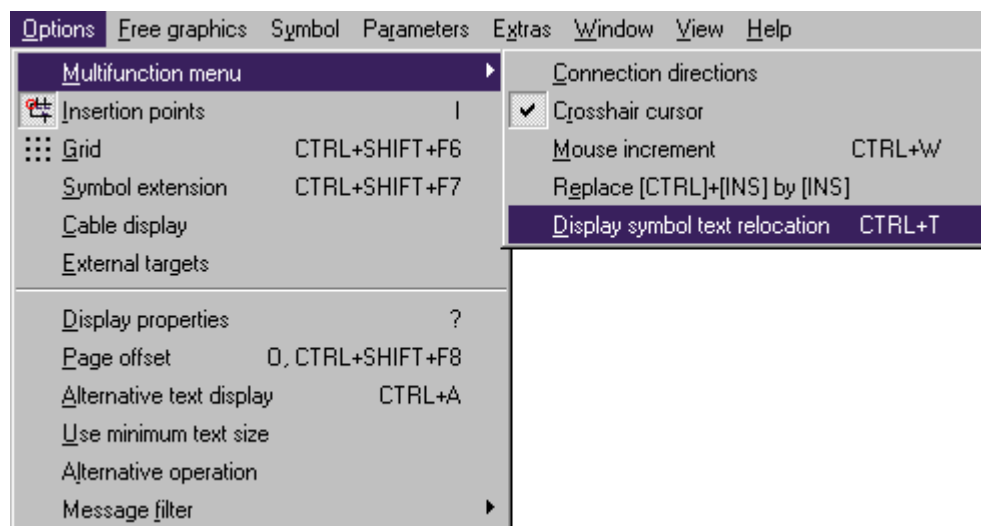
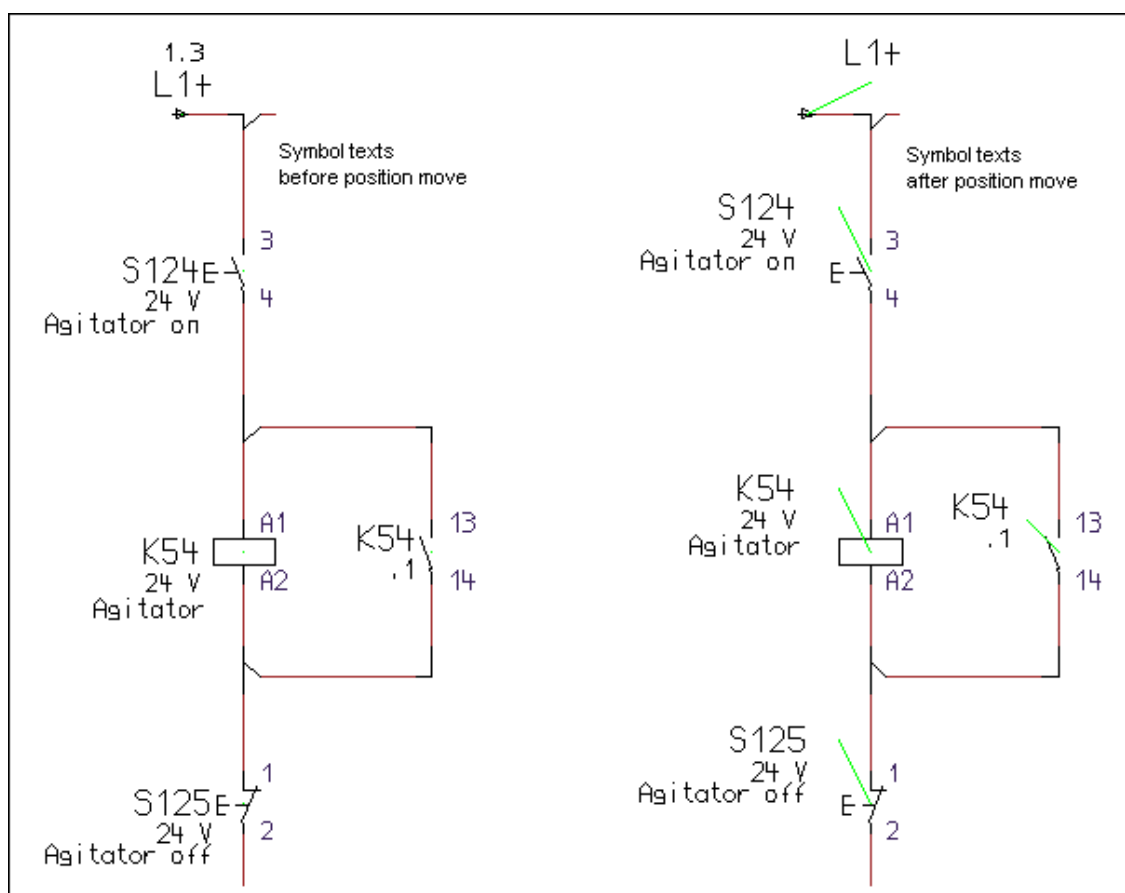


Fig. 44: **Display symbol text relocation** menu option in the multifunction menu



11.4 Undoing the Symbol Text Move

A relocated symbol text (block) can be reset to its original coordinates (X=0/Y=0) in EPLAN (relative to the position of the insertion point).

To do this, move the cursor to the insertion point of the schematic object whose symbol text(s) you want to move back to its original position, and press the left mouse button. In the Symbol properties dialog box that comes up, select the **Text display** tab, and hit the **[Undo position move]** button (see Fig. 45).

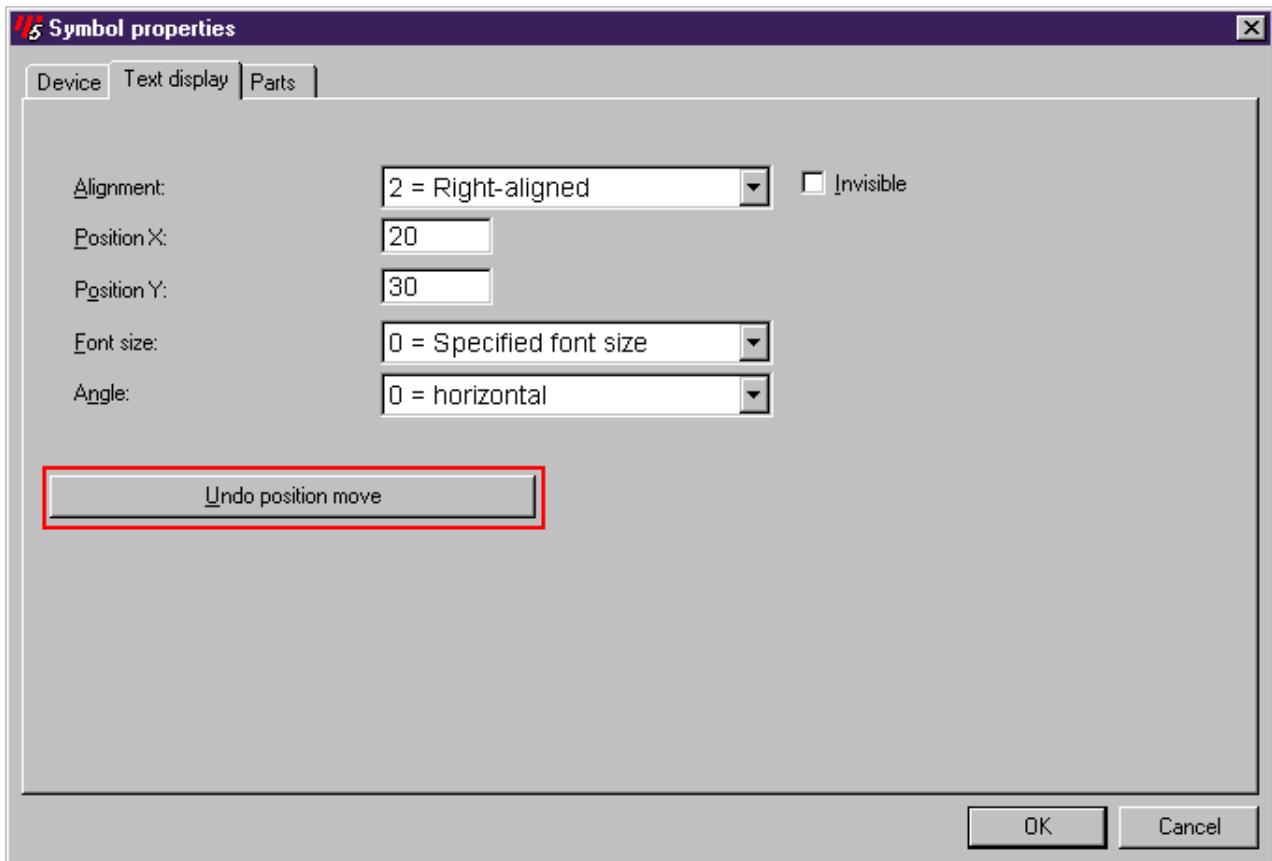


Fig. 45: Text display tab with the **[Undo position move]** button for symbol texts

In case of cross-referenceable **PLC end terminals** on pages of type "T" (PLC overview) this button is available on the **Text display 1** tab.

12 Moving Strip Designations and Connection Numbers

As is known, the device designation of a terminal or a connector pin consists of two differing elements – first, of the terminal or connector strip designation (e.g. "X1") and, second, (separated by ":") of the terminal connection or connector connection number.

Terminal connection or connector connection numbers can contain purely numeric or alphanumeric information (e.g. "1", "2A", or "B12").

The terminal strip or connector strip designation as well as the terminal connection or connector connection number can be moved to any position within the schematic independently of one another. These functions are to your advantage if you must insert terminals or connector pins very close to each other in the schematic, with overlappings occurring quite often.

Strip designations and connection numbers can be moved to the following project page types:

- A = Schematic page (logic, interactive)
- Q = PLC page (logic, automatic)
- T = PLC overview (logic, interactive) (designation of the physical connection of PLC end terminals).

If you want to move the terminal or connector strip designation and/or the terminal connection or connector connection number, select the **Move strip designation** or **Move connection number** functions, which were newly implemented in EPLAN 5.50, from the **Edit** menu of the graphics editing module (see Fig. 41 on Page 87).

After having selected the respective option, press the left mouse button briefly, and drag the box-shaped cursor around the insertion point of the corresponding terminal / connector pin. Press the left-hand mouse button shortly again afterwards. The strip designation or connection number can now be moved in the schematic. Place the strip designation or connection number, which is tied to your cursor, at the desired position in the schematic via mouse click or by pressing the  key.

Naturally, you can also move the strip designation or connection number via the pop-up menu. It is called up by right-clicking on the insertion point of the terminal/pin connector.

12.1 Moving Strip Designations/Connection Numbers by Specifying Defined Coordinates

Similarly to moving symbol texts, you can also move the terminal strip or connector strip designation and/or the terminal connection or connector connection number by defining coordinates in the Symbol properties dialog box. To do this, click on the insertion point of the desired terminal or connector pin. In the Symbol properties dialog box that appears switch to the **Text display** tab (see Fig. 46). To move the strip designation, overwrite the values of **Position X** and **Position Y** fields in the **Strip designation** group box with the new data, and confirm your entries with **[OK]**. The coordinate fields of the **Connection number** group box are modified correspondingly if you wish to re-position the connection number.

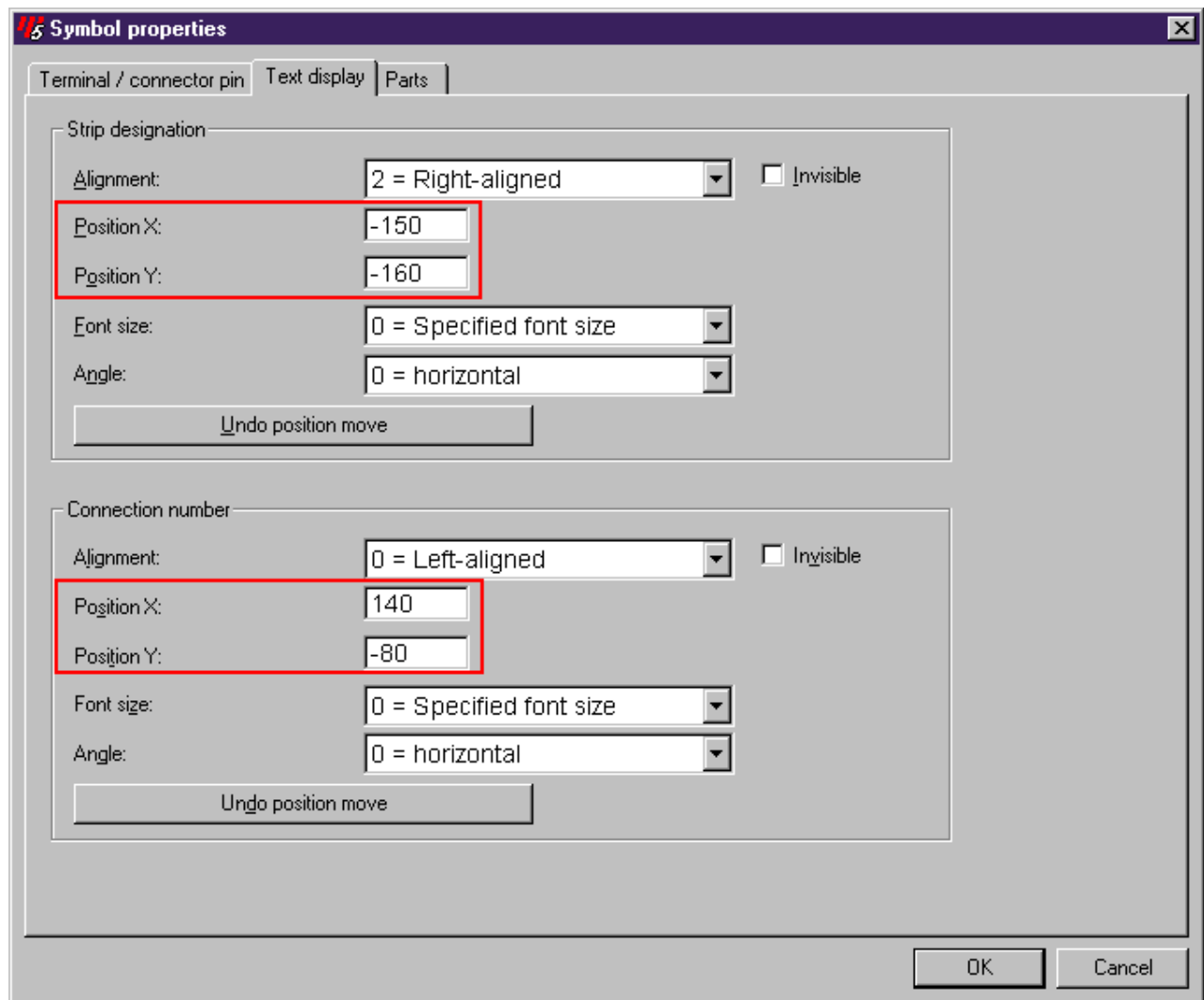


Fig. 46: Coordinate fields of the Text display tab for terminals and connector pins

If you want to use this method to edit the coordinates of the strip designation or connection number at cross-referenceable PLC end terminals located on project pages of type "T" (PLC overview), activate the **Text display 2** tab of the symbol properties dialog box (see Fig. 47 on Page 97).

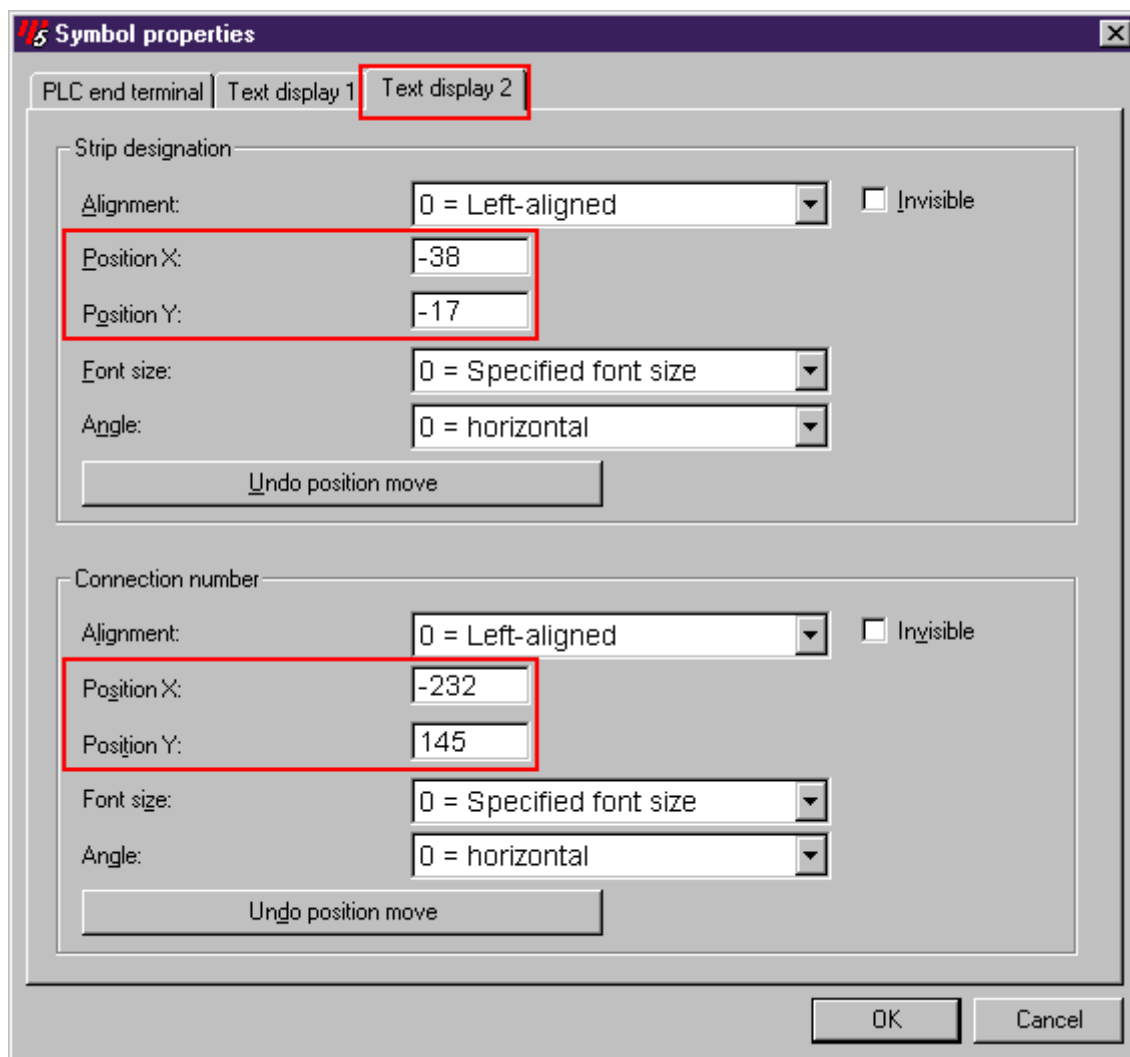


Fig. 47: Coordinate fields of the Text display 2 tab for PLC end terminals on PLC overview pages

12.2 Displaying the Move of the Strip Designations and the Connection Numbers

It is also possible to display the move of the terminal strip/connector strip designations and of the terminal connection/connector connection numbers in the schematic. To do this, select the **Display symbol text relocation** function from the Multifunction menu of the graphics editing module. EPLAN will then indicate the relocation by means of a light green connection line which is displayed between the moved strip designation or connection number and the corresponding terminal or connector pin (cf. the following section "Examples" starting on Page 98).

12.3 Undoing the Move of the Strip Designation and the Connection Number

You can undo the move of the terminal strip/connector strip designation and/or of the terminal connection/connector connection number, just as it is possible for the symbol text move.

For this purpose, call up the Symbol properties dialog box of the corresponding terminal, connector pin, or PLC end terminal. Select the **Text display** tab (see Fig. 48). If the strip designations and connections are located at cross-referenceable PLC end terminals on pages of type "T" (PLC overview), select the **Text display 2** tab.

In this case, you must click the [**Undo position move**] button in the **Strip designation** group box in order to reset the strip designation to the original coordinates (X=0/Y=0). To reset the connection number, activate the corresponding button in the **Connection number** group box.

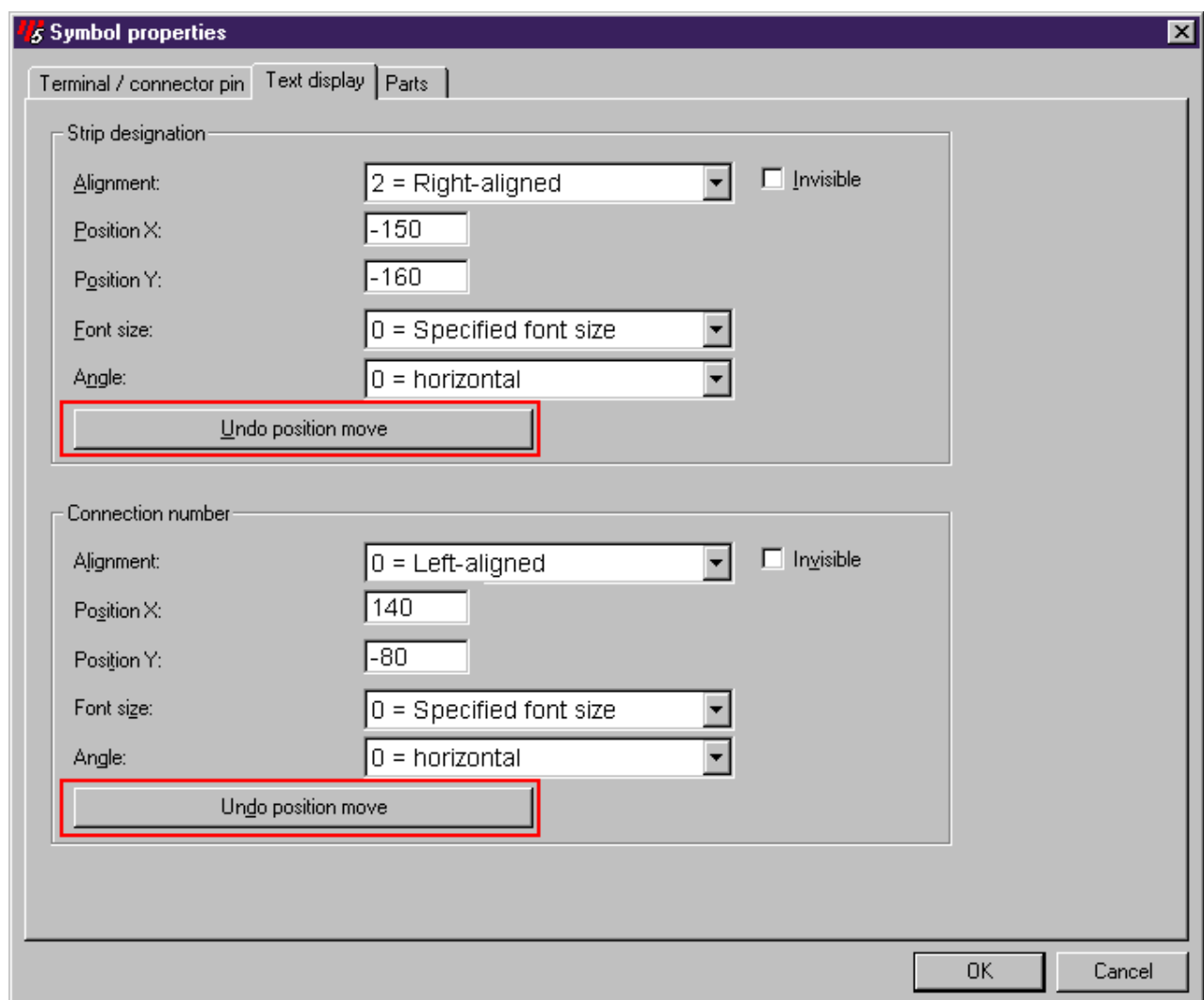


Fig. 48: Text display tab with the [**Undo position move**] buttons for the strip designation and the connection number

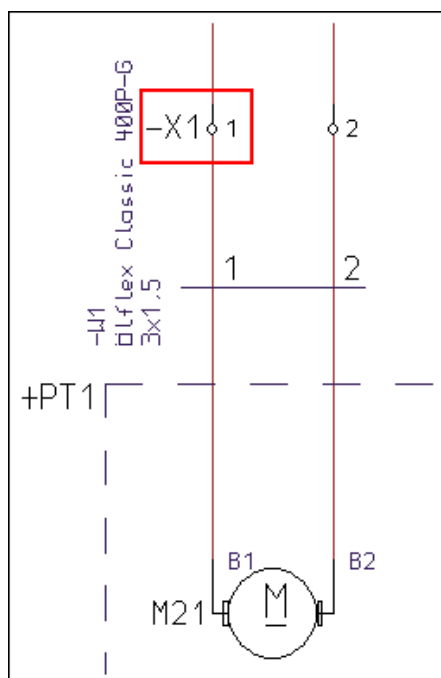
12.4 Examples



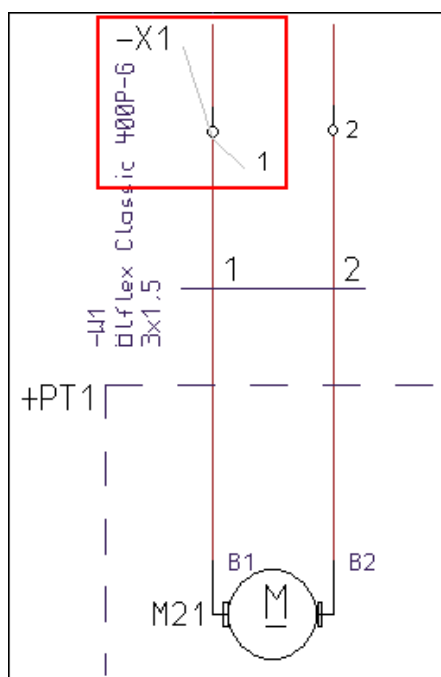
Example 1:

The strip designation and the terminal connection number of the terminal "-X1:1" are moved by defined values on a project page of type "A".

Positions before the relocation



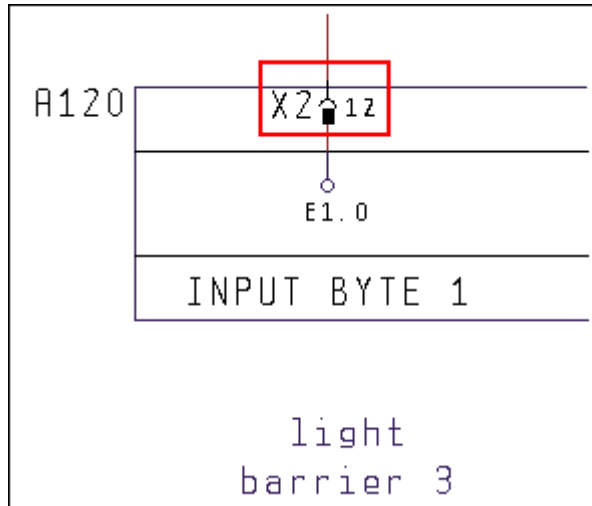
Positions after the relocation



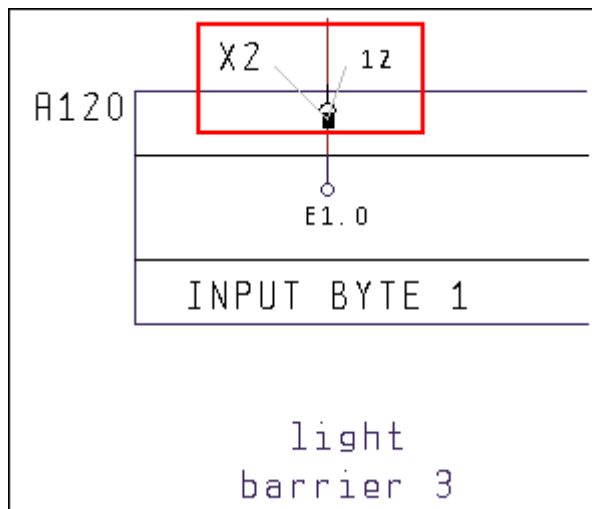
Example 2:

The strip designation and the connection number of the plug-in connection "X2:12" (physical connection) at the PLC end terminal "I 1.0" is moved by defined values on a project page of the "Q" type (PLC page).

Positions before the relocation



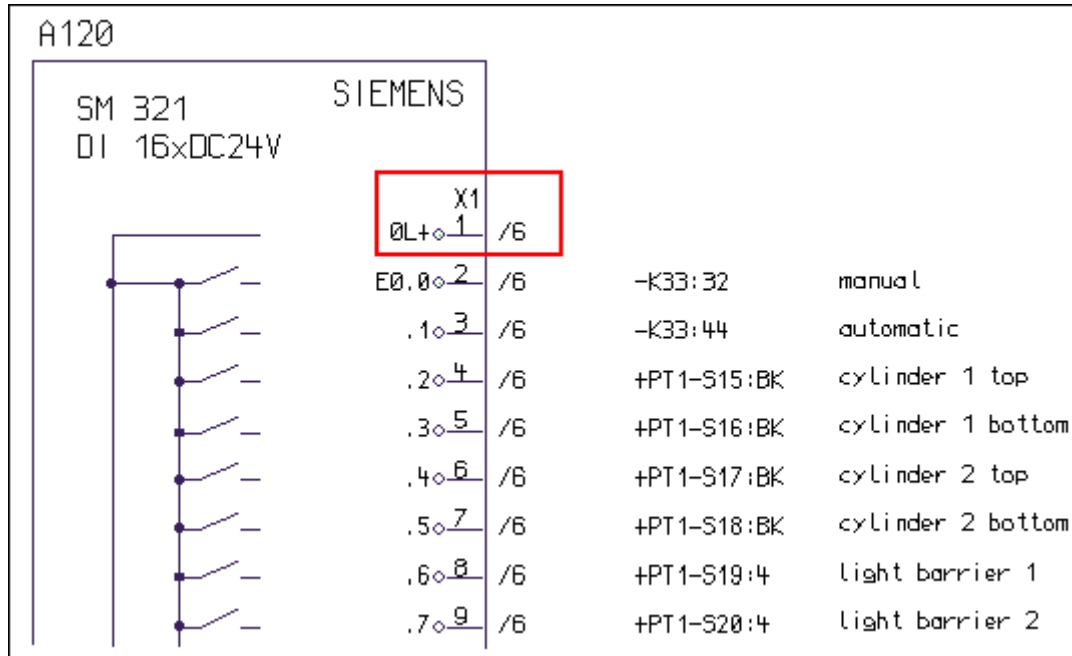
Positions after the relocation



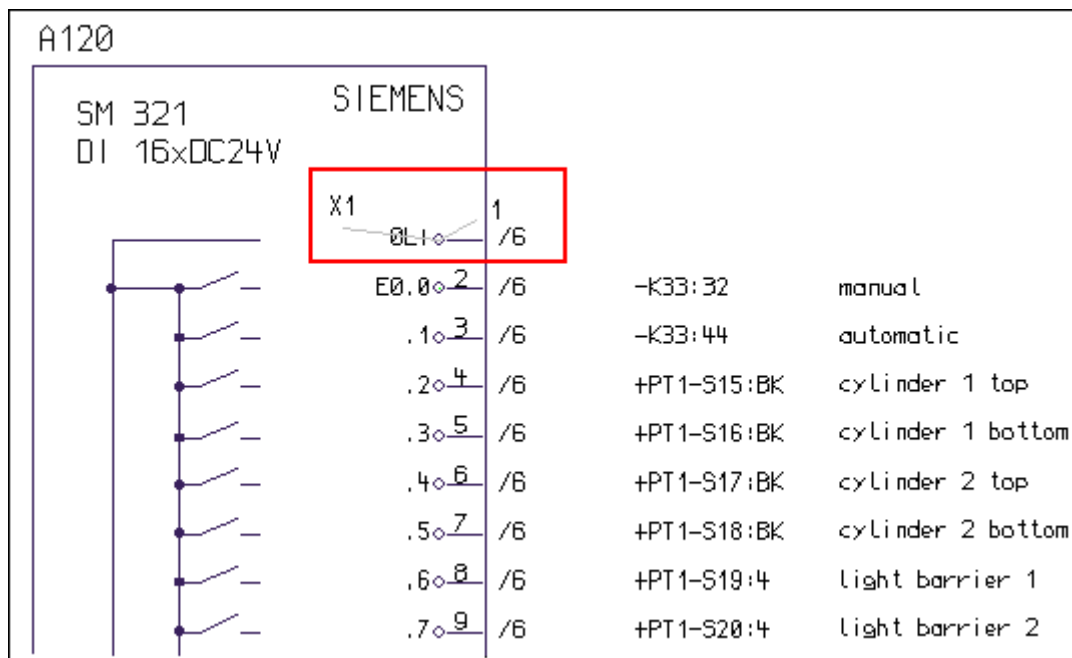
Example 3:

The strip designation and the connection number of the plug-in connection "X1:6" at the feeder terminal "0L+" is moved by defined values on a project page of the "T" type (PLC overview).

Positions before the relocation



Positions after the relocation



13 Evaluating and Editing Messages

As already mentioned in Chapter 4.3 on Page 18, EPLAN now allows the online display of processing and evaluation messages in a separate, dockable window of the graphics editing module. In contrast to Version 5.40, in which you could use buttons and check boxes to evaluate and edit messages, messages are handled via menus only in Version 5.50.

13.1 Marking Messages as Done/Undone

The function to mark messages as done or undone familiar to you from EPLAN 5.40 is also available in EPLAN Version 5.50. Use the mouse or the keyboard to highlight the desired messages in the table, and follow the menu path **Edit > Messages > Set to 'done'/reset** in the graphics editing module. As an alternative, you can also select **Set to 'done'/reset** from the pop-up menu, which appears if you right-click in the message window. EPLAN will activate or deactivate the respective check boxes afterwards. You can, of course, also activate/deactivate the respective check boxes individually with the mouse in the Done column of the message window.

Notes:

- Messages marked as "Done" are not automatically deleted from the project-specific message management, but will be shown again in the message table after the next evaluation run if the problem hasn't been solved.
- When the graphics editing module has been started, the Done messages option is activated by default.

13.2 Selecting the Display according to Message Types

Follow the menu path **Options > Message filter** in the graphics editing module or, as an alternative, right-click in the message window, and select the **Filter** option from the pop-up menu to specify whether or not you want EPLAN to display errors, warnings, notes, and/or done messages. When you have activated a message type, EPLAN will put a tick in front of the respective entry in the pop-up menu. Message types without a checkmark are not activated.

Supplementary Information on Displaying/Suppressing Messages Marked as Done

If you entirely suppress the display of the message types Error, Warning, and Note (no checkmark in front of the menu option), EPLAN won't show any messages even if the Done messages setting is activated.

EPLAN saves the display status of the **Done messages** menu item when exiting the graphics editing module.

13.3 Displaying the Device Causing the Message in the Schematic

The device or object causing the message can be displayed in the schematic, e.g. to eliminate the error (just like in EPLAN Version 5.40). To do this, click on the line number in front of the desired message in the message window. The line with the message is then highlighted. Double-click on the line, or press . The cursor will immediately "jump" to the corresponding object in the project, that caused the message. You can also use the menu, for example to locate a schematic object causing an error within your project pages. When having selected the respective line in the graphics editing module, follow the menu path **Edit > Messages > Jump to DD**, or open the pop-up menu in the message window, and select the **Jump to DD** option.

13.4 Displaying the Second Position of a Device in the Schematic

The 2nd coordinate function, which allows you to jump to the second position of a device in the schematic (if existing), is also included in the menu of Version 5.50. Select **Edit > Messages > Jump to second coordinate** in the graphics editing module to have the source of the error, the warning, or the note displayed in the project. Of course, you can also call up this command via the pop-up menu.



You can only activate the **Jump to second coordinate** option if a second position of the device exists which caused the error, otherwise this menu item is grayed out.

13.5 Sorting Messages

Just like in Microsoft® Excel® or Microsoft® Access®, you can sort the entries in the individual columns of the message table alphanumerically in ascending or descending order. To do this, click on the desired column header. All entries of the column will then be sorted in descending order. If you relick on the column header, the entries will be sorted in ascending order again.

14 Drawing Polylines

We have added a new graphics object to the Free graphics menu, the **polyline**, which can now be placed on your project pages. The procedure is similar to that of drawing a set of lines.



Please note that the former **Polyline** in the **Free Graphics** menu has been renamed **Set of lines** in Version 5.50!

In contrast to the **Set of lines** consisting of several lines, a polyline consists of successional line sections. Whereas you can assign individual element data to each line of a set of lines, the element data of a polyline apply to all line sections.

You can also modify the display of a polyline in the element data. Corresponding options allow you to change the polyline to a closed or a closed and filled polyline (see Fig. 49). In other graphics programs, a closed polyline is also called "polygon".

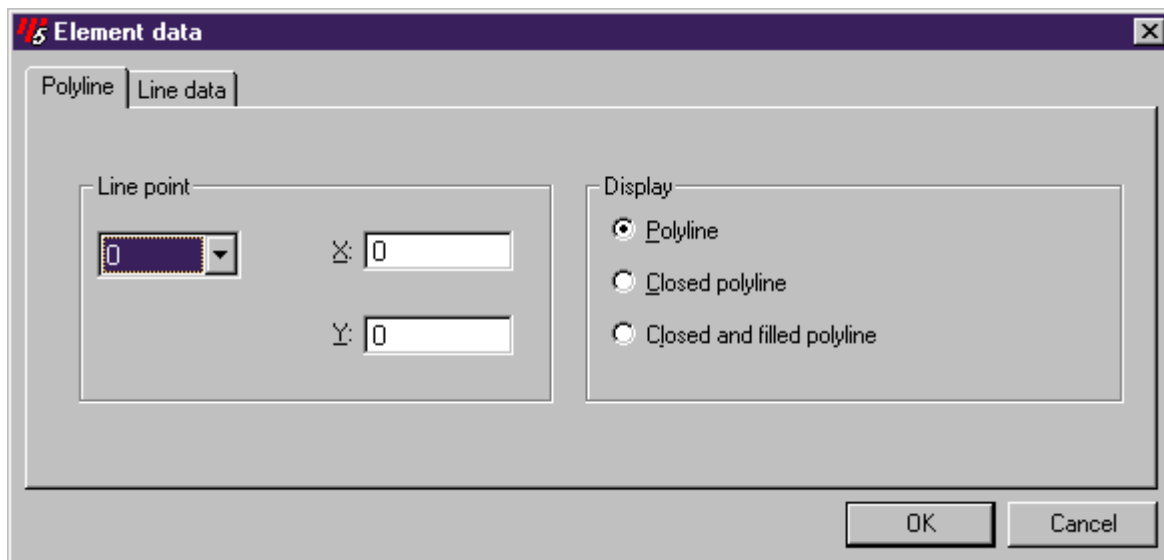


Fig. 49: Element-specific data of a polyline in the Element data dialog box

Furthermore, the line data of the graphics elements were extended for EPLAN Version 5.50 (see Fig. 50 on Page 108). You can now also assign **variable line thicknesses** to the lines of the graphics elements. If you select the option "7 = variable" in the corresponding dropdown list, the input field below is activated. The desired line thickness can then be entered (in mm) into this field.

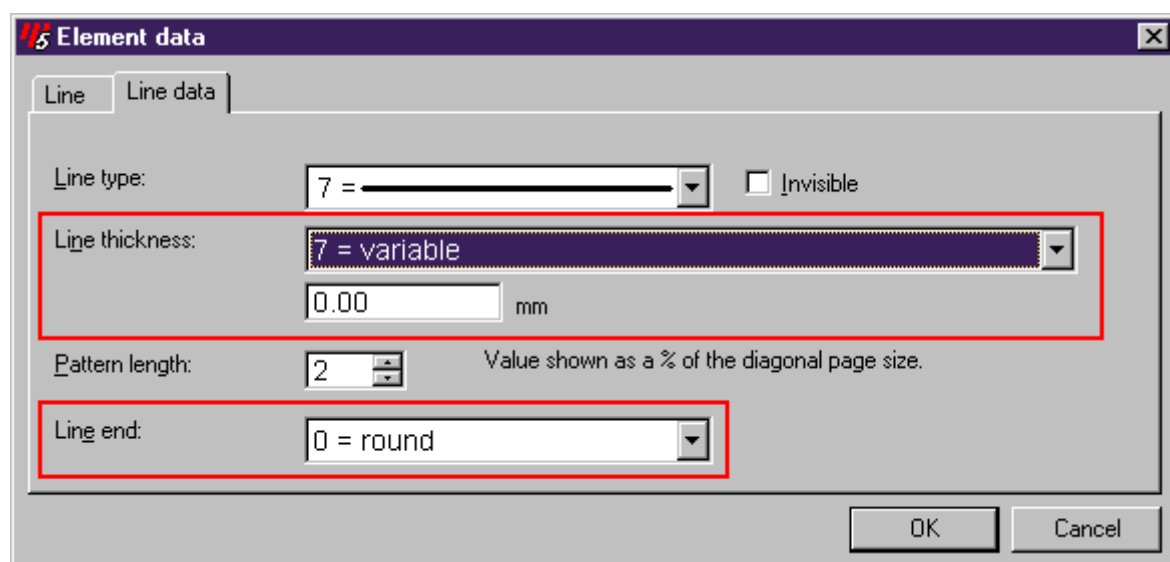
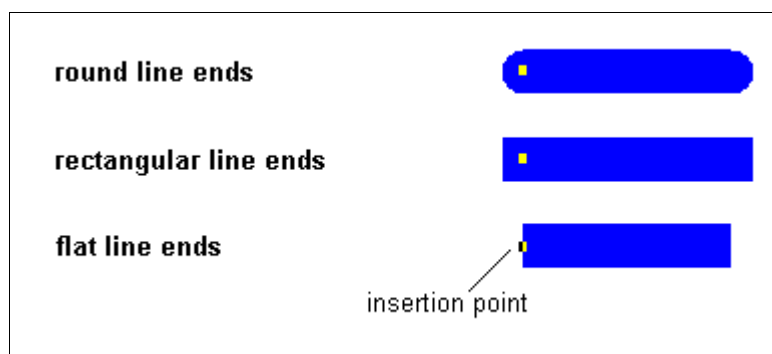


Fig. 50: Line data for graphics elements with new input possibilities



The Line type dropdown list is not available for **polylines**, since they can only be drawn with a solid line. For this reason, the pattern length field is not offered for polylines, either.

In addition, you have the possibility to select one of three **line ends** ("round", "rectangular", or "flat") for the lines. This is illustrated by the following example: The default end is "0 = round". Line ends are important for the display and printout of lines, e.g. if you have selected large line thicknesses.



15 Adjusting the Function Text for Distributed Components

A device often consists of various individual components, that are inserted in EPLAN as separate symbols with their own function texts at different positions in the schematic. An example for this is a contactor, which generally consists of a contactor coil and a contact block, but which in addition can also be equipped with an auxiliary block or an auxiliary switch or with additional components.

As from EPLAN Version 5.50, you can adjust the function texts of the components of a device that are displayed at various locations in the schematic.



Please note that the components can only be adjusted if contact cross-references are assigned to them, i.e. you have carried out a contact cross-reference run. Furthermore, please keep in mind that the adjustment is only carried out for the function text you have entered for the respective component in the symbol properties dialog box. Path function texts will **not** be considered!

To adjust function texts automatically, you must first highlight a component of a device that can be adjusted in the schematic. It doesn't make a difference whether you highlight the main element or one of the auxiliary elements.

Select the menu path **Edit > Adjust function text** in graphics editing.

The cursor will then change its shape as follows:



Once you press the left mouse button briefly the cursor changes its shape to a frame:



Drag this frame around the insertion point of the selected component.

If you briefly press the left mouse button again, the frame disappears, and the **Function text transfer** dialog box is opened (see Fig. 51).

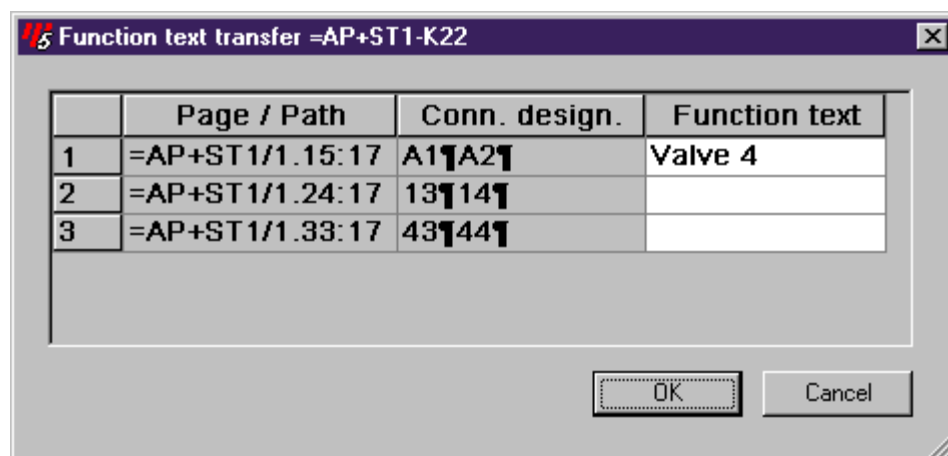


Fig. 51: Function text transfer dialog box

This dialog box also comes up if you right-click on the insertion point of a component to which cross-references are assigned and select the **Adjust function text** option from the pop-up menu.

A table is shown here giving all associated components which are cross-referenced and have the same device designation.

Information about page, path and position of the component as well as about the connection designation and the function text is provided in this table. Only the Function text column can be edited.

You now have the possibility to modify the function text at the desired components by clicking into the respective cell and overwriting the text contained therein.

In addition you can adjust the function text at all components of a device at one go. To do this, place the mouse pointer on the **Function text** column header and press the left mouse button. EPLAN then highlights all cells of this column (see Fig. 52). Now type the common function text into the topmost, non-inverted cell. When the text is entered, it is simultaneously written into all highlighted cells of the column. Click **[OK]** to confirm your entry and exit the dialog box. EPLAN will then write the adjusted function text to all components of the corresponding device in the schematic.

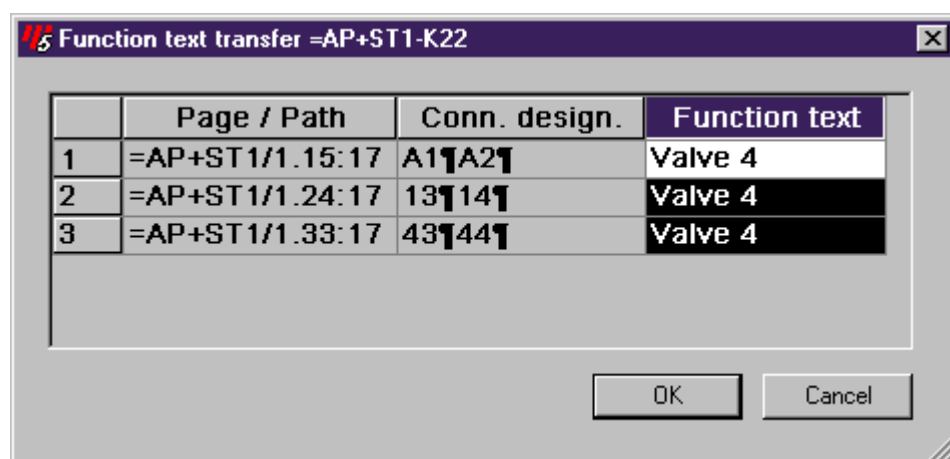


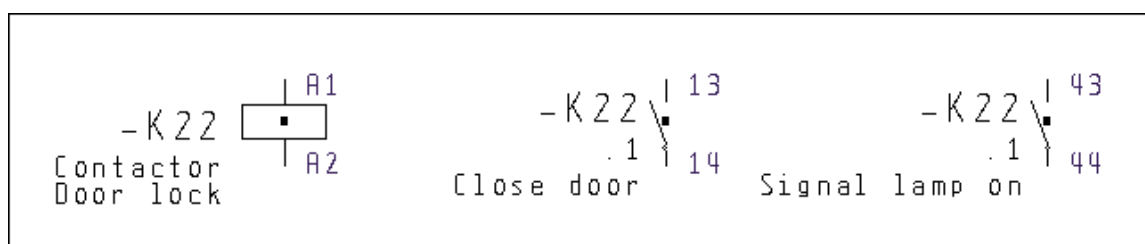
Fig. 52: Function text transfer dialog box with highlighted function text column



The function text can only be adjusted if the corresponding pages are not locked by other users. If pages are locked, you will be informed by the information management function of the program (see the section "Information Management" on Page 137).

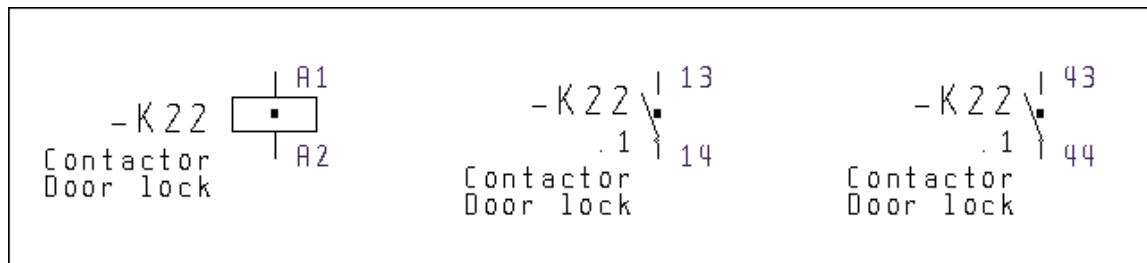


The following simplified schematic section shows the distributed components of a contactor with the device designation "K22", consisting of a contactor coil and two NO contacts. The function texts vary for all components.



The function text of the contactor coil is now to be transferred to the contacts with the connection designations 13 / 14 and 43 / 44.

Once the function text is adjusted, the schematic section displays identical function texts at all components of the device.



16 Redesigned Help System

In addition to the numerous new functions and changes of EPLAN Version 5.50, we have also redesigned the online help system. This applies to the following points:

- Improved orientation thanks to a new help window design
- Reader-friendly layout of the help texts.

The new **help window design** is characterized by the navigator area on the left side of the help window which is always displayed even when a topic is called up.

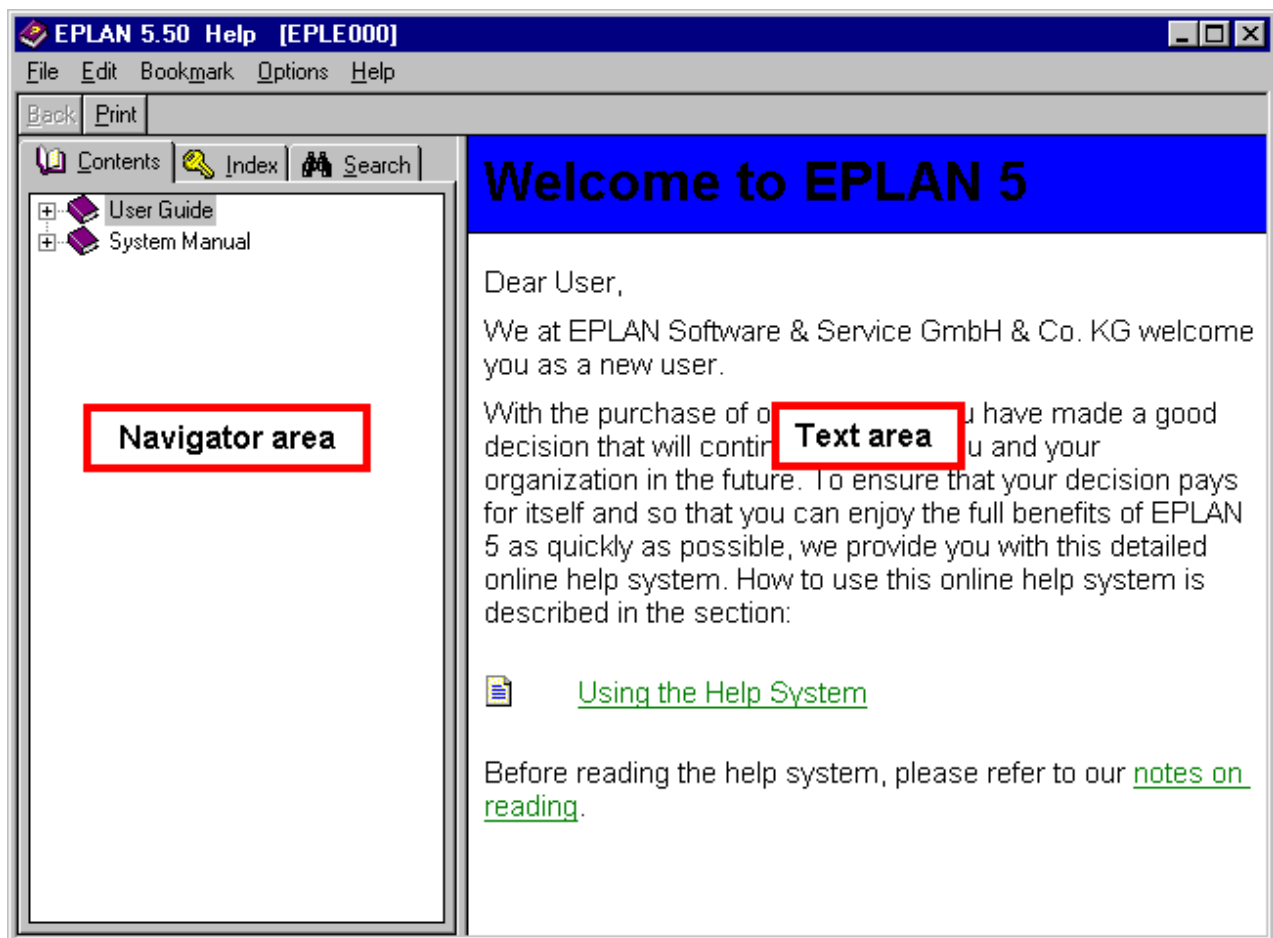


Fig. 53: Redesigned help window in EPLAN 5.50

This area contains the familiar Contents, Index, and Find tabs, offering you quick access to any desired information of the help system.

Since the navigator window is permanently visible, we are offering you an improved orientation in our online help system. Whether you browse through the pages of a main chapter or a "book" (e.g. "Using EPLAN") via the [**<<**] / [**>>**] buttons or use a link to jump to the next help topic, the respective highlighted topic in the Contents tab indicates which subchapter is currently open.

The **layout of the help texts** has been improved, too. For example, a reader-friendly font makes it easier for you to read the texts. The removal of the margin resulted in a reduced space requirement of the help topics.

Last but not least we have taken care of your health and now present the background of the help window in a light gray color. This way, the contrast to the black text has been reduced, being easier on the eyes.

17 Creating the Table of Contents

17.1 New Location in the Menu

Up to now, you have created the table of contents under the **Generate** menu item of the EPLAN main menu. This was, however, inconvenient, because you had to exit the graphics editing module. This is no longer necessary. As from EPLAN Version 5.50, you can generate the table of contents from within **graphics editing**. Call up this function by selecting **Extras > Table of contents > Create**.

17.2 Changed Parameters for the Table of Contents

Further changes were made as for the parameters for the table of contents.

The parameter dialog box that was called up via **Utilities > Parameters > Project > Table of contents** in previous EPLAN versions was removed. The parameters were shifted to other parameter dialog boxes and some of them are no longer available at all. One parameter has been added. The following sections will give you a detailed description of the changes.

Shifted parameters

The form ("form name") used to output the table of contents as graphics page is now set in the Graphics output parameters dialog box (menu path **Utilities > Parameters > Project > Graphics output -> Graphics forms tab** "; refer to Fig. 54 on Page 116).

The Create subpages and Character for subpages parameters are now also to be found in the Graphics output parameters dialog box. To edit the parameters, select the above-mentioned menu path and switch to the **Graphics output** tab (Table of contents group box; refer to Fig. 55 on Page 118).

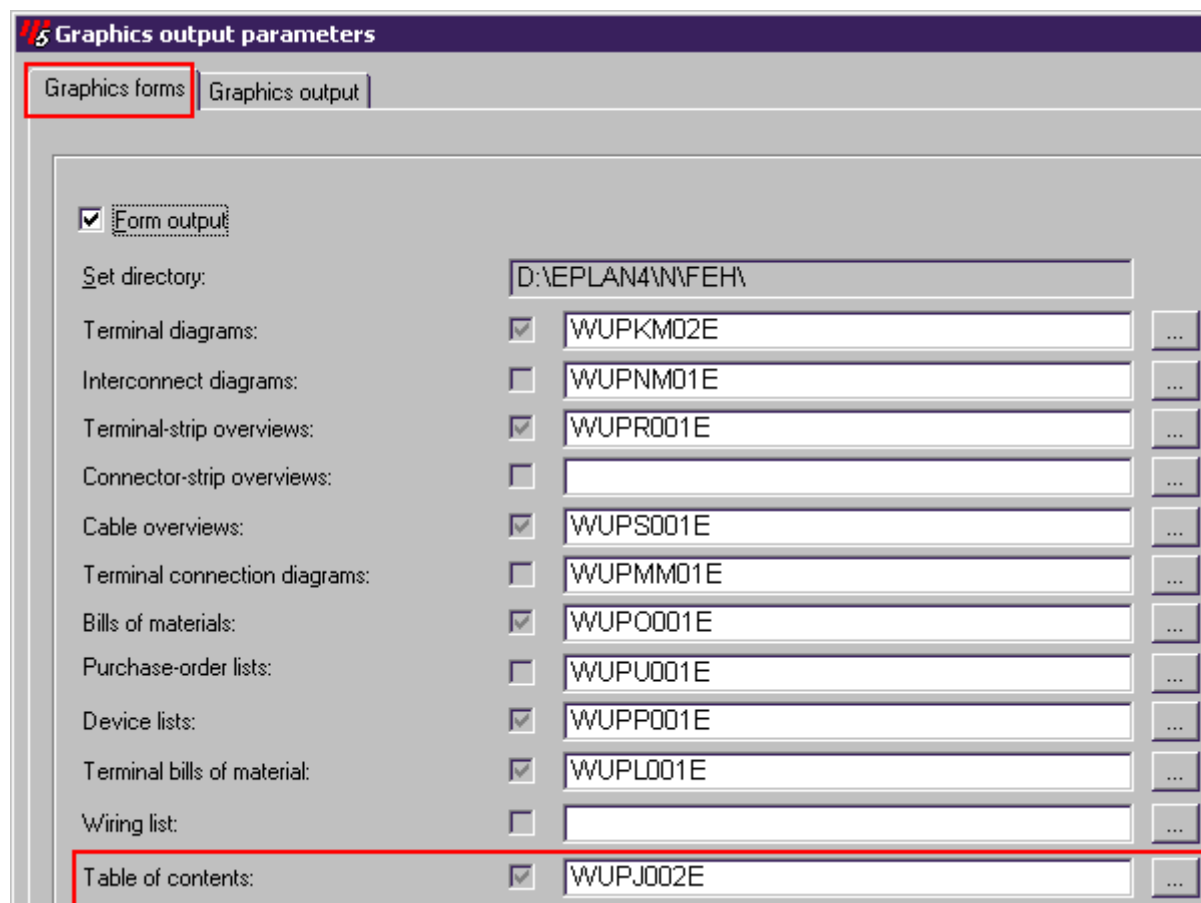


Fig. 54: New Table of contents parameter in the Graphics forms tab

Parameters that are no longer required

Pages can only be created automatically and no longer interactively. For this reason, the Pagination parameter was removed.

Furthermore, the Spare page identifier, Forms transfer identifier and Position identifier with DIN parameters are no longer available.

The same applies to the following parameters:

- Lines per page
- Line spacing in mm
- Number of columns
- Column spacing in mm
- Orientation.

These data for the page layout are now only taken from the form. To this purpose the corresponding special texts have to be entered in the form.

New parameters

The following parameters were implemented in the Table of contents group box of the Graphics output tab (see Fig. 55 on Page 118):

- **Number of blank pages:**

Use this parameter to specify how many pages will be left blank between the most recently used page and the table of contents.

- **Round up to page:**

This parameter, which operates together with the Number of blank pages parameter, is used to specify the page number from which you want the table of contents to start.



Both parameters are set as follows:

"Number of blank pages" = "4"

"Round up to page" = "10"

Case 1:

Let us assume the last page number is "47" --> The table of contents will then start on Page "61".

Explanation:

The last page number is "47". You have to add 4 blank pages, which results in "51" as last page number. This result is rounded up the next full "10" so that the last page is "60". Therefore, "61" is the first page number of the table of contents.

Case 2:

Let us assume the last page number is "46" --> The table of contents will then start on Page "51".

Explanation:

The last page number is "46". You have to add 4 blank pages, which results in "50" as last page number. This result is rounded up the next full "10" so that the last page number is "50". Therefore, "51" is the first page of the table of contents.

- **Output method:**

Use this parameter to adjust the output of the table of contents to your project type. It is, for example, possible to output a separate table of contents for each higher-level assignment of your DIN project.

The dropdown list offers the following options:

- **Entire output:** This option enables you to output the entire table of contents at a defined position.
- **Higher-level assignment:** This option allows you to output the table of contents separately for each existing higher-level assignment.
- **Location designation:** This option allows you to output the table of contents separately for each existing location designation.
- **KKS:** Using this option, you can output the table of contents separately for each document and documentation type.

- **Entire output and by higher-level ass.:** This option enables you to output the entire table of contents at a defined position and, in addition, by higher-level assignments.
- **Entire output and by location des.:** Using this option you can output the entire table of contents at a defined position and, in addition, by location designations.
- **Entire output and KKS:** This option enables you to output the entire table of contents at a defined position and, in addition, KKS-oriented.
- **Manual:** Using this option you can highlight the pages to be output in the page overview of graphics editing and manually define the page designation, the starting page, and the output positions.

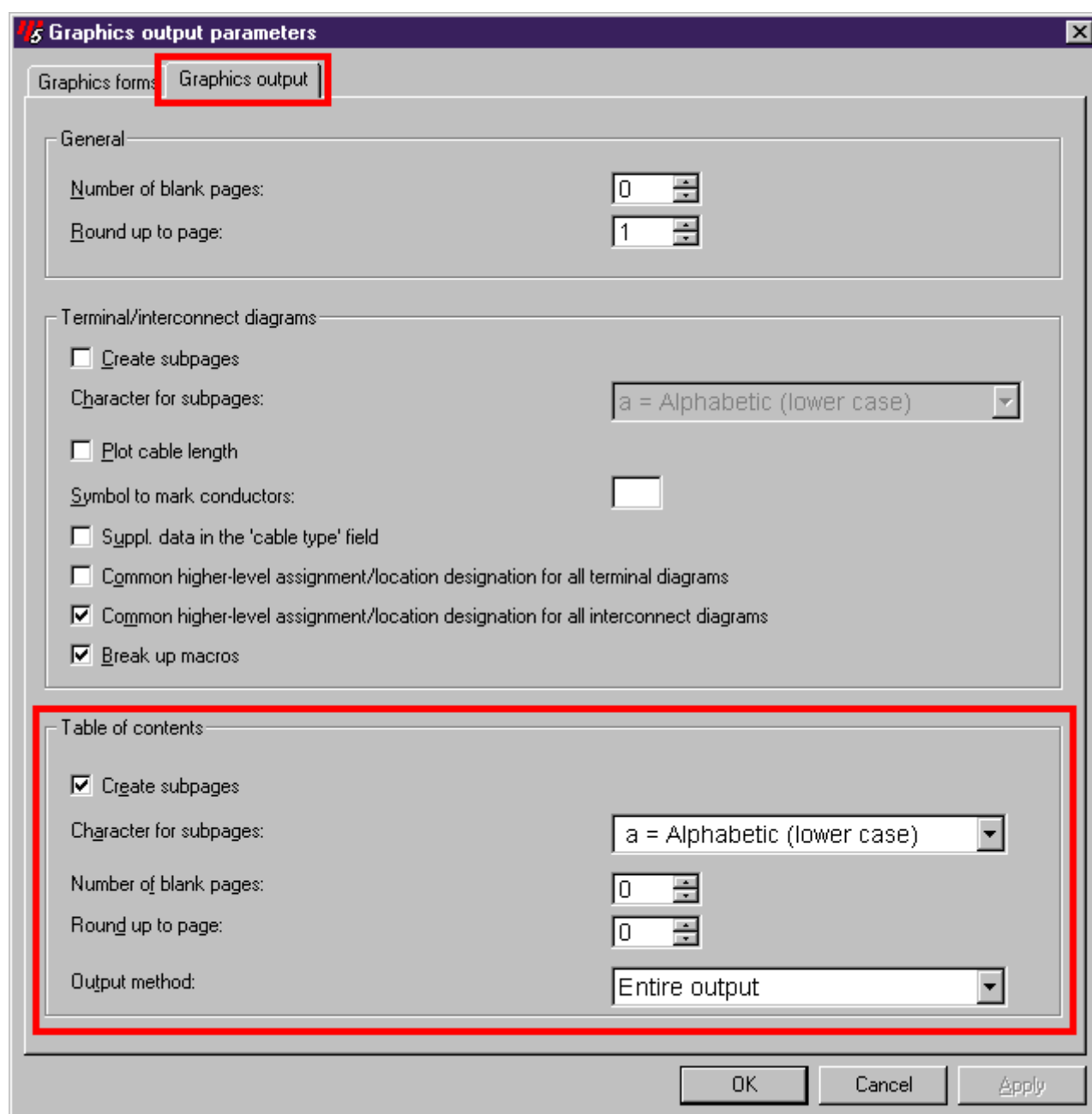


Fig. 55: Parameters for the graphics output of the Table of contents in the Graphics output tab

18 Designing Mounting Panels

The EPLAN panel layout has been revised completely. The following sections will provide you with information on the new functions as well as with an introduction on how to design mounting panels.

18.1 Functions of the Panel Layout

All functions relevant to the panel layout are now available under the menu path **Extras > Mounting panels** in the graphics editing module. Opening the **Mounting panel** submenu will provide you with the following options:

- The **Insert** option enables you to draw a mounting panel just like a black box. Note that mounting panels can only be designed on pages of the "C = Control panel layout" page type.
- If you select the **Equip** option, a dialog box for equipping the mounting panel with components is opened. This dialog box allows you to specify selection criteria and thus influence the table overview of the components to be placed. Furthermore you can coordinate the equipment itself by choosing application options.
- The **Update** menu option must be used when you have changed device designations in the schematic or removed components from the mounting panel.
- The **Insert window legend** option enables you to create a window legend that can be placed directly next to the mounting panel.
- If you want to output the equipment legend as graphics pages in the project, select the **Create page legend** option.
- The **Coordinates** option enables you to change the special text coordinates in the legend forms.

18.2 Creating Component Macros (First Step)

To create components on mounting panels, you must use graphics objects or so-called "**Combined macros**". The latter are always used when you want to place components **to scale** on the mounting panel. Now let us have a closer look at the working method that uses this macro type.

Combined macros can be created on pages of the type "**B = Free graphics**" as well as on pages of the type "**C = Control panel layout**". These macros have the ***MZF** file extension.

Enter the **scale** that EPLAN saves in the macro into the properties dialog box of the page (Page properties 1 tab; see Fig. 56).

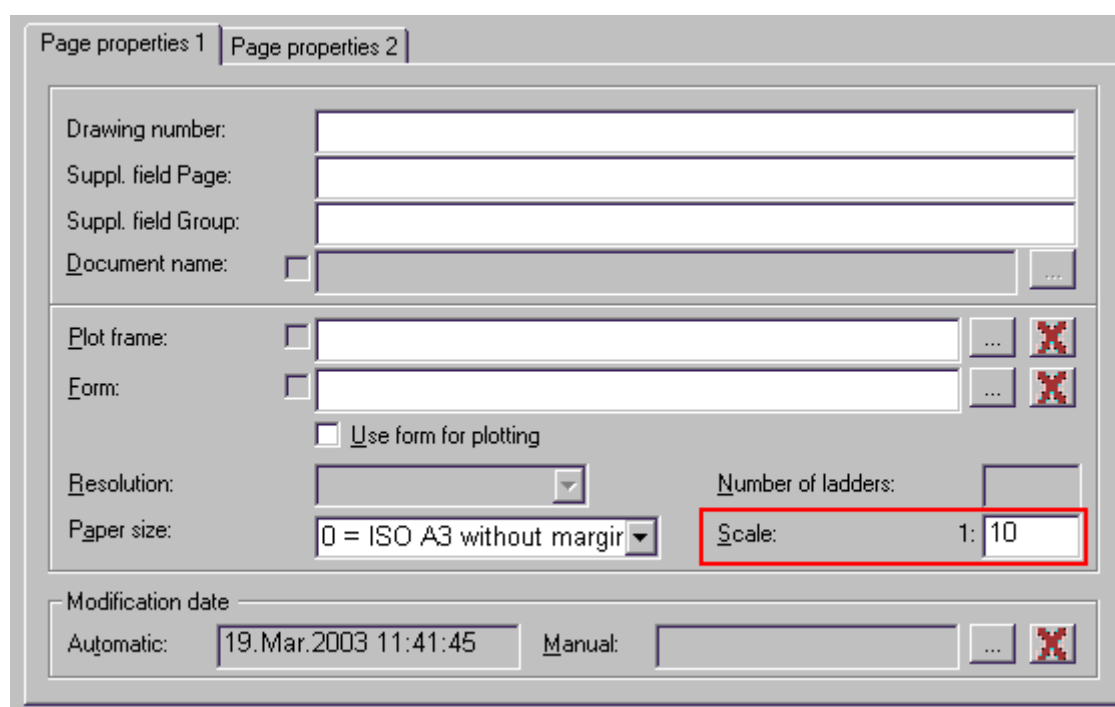


Fig. 56: Setting the scale for combined macros

Combined macros, also known as component macros, consist of one or several scale graphics objects and a **placeholder text**. In the following sections you will learn how to draw a combined graphics object and how to create a combined macro by using this object.

18.2.1 Creating a Combined Graphics Object

The quickest way to draw a combined graphics object is to use the **Multiple editing** function in EPLAN. This function is either activated via the Free graphics menu or by clicking the  button on the Free graphics toolbar.

If, for example, the mounting panel is to be equipped with a contactor, select the **Rectangle** option for this graphics object from the Free graphics menu. Once you have defined the outlines and placed the rectangle on the graphics page by clicking **[OK]**, it changes its color. (The change of color depends on the specified color parameters.)

Then select **Free graphics > Placeholder text** to insert the text object, the placeholder, at the component for the text to be generated at a later point. When doing this, the cursor changes its shape to an "X" to which three question marks are attached (refer to Fig. 57).

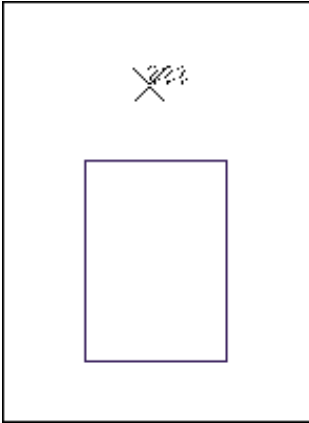


Fig. 57: Inserting a placeholder

Move the cursor with the three interrogation points to the desired position (by preference within the rectangle). EPLAN then branches to the **Element data** dialog box. The Placeholder text tab enables you to influence the text type, the font, and the graphics data of the text object.

The following text types are available:

- Device designation
- Part number
- Sequential number (position number of the component in the legend)
- Type designation.



Although the text object "Free graphics text" can be used to designate the component, it does not offer the versatility of the placeholder text, since only the device designation can be output here! In this context we would like to mention that you can update combined macros which you created in an earlier EPLAN version by using the menu tree **Utilities > Automatic revision > Revise macros** in EPLAN 5.50.

Then click on **[OK]**. EPLAN then closes the Element data dialog box and places the interrogation points in the graphics object.

Now, you have created a new combined graphics object disposing of one common insertion point (see Fig. 58).

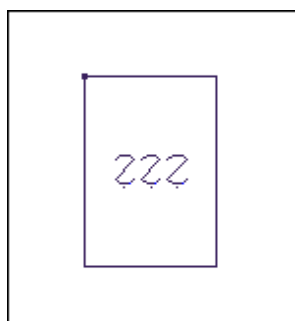


Fig. 58: Combined graphics object consisting of a rectangle and a placeholder text

18.2.2 Creating a Combined Macro

In the next step you will create a combined macro from the new combined graphics object. To do this, select **Edit > Create macro/form** in the graphics editing module, and choose the **Combined macro** submenu item. In the **Define combined macro** dialog box that appears, enter a sensible name (e.g. `CONTACTOR1`) for the macro to be created, and, if necessary, a commentary text. It will be displayed when the macro is called up later. When you click **[OK]**, EPLAN will use the combined graphics object as combined macro (component macro).

18.3 Entering Component Macros into the Parts Master Data (Second Step)

To create a connection between the commercial and the graphics data, you must enter the name of the component macro into the parts master data. Furthermore, you specify the mounting site there.

For this purpose, follow the menu path **Part > Management** in the main menu, and double-click on the **Components** entry in the tree view of the EPLAN parts. EPLAN will then display the individual product groups. In the next step, double-click on the desired product group (e.g. "Relays / contactors"). EPLAN then displays the manufacturer level. Another double-click on a manufacturer icon will open the part level. Double-click on the desired part. The window for the input of parts data appears (see Fig. 59 on Page 123).

Activate the **Supplementary data** tab, and enter the name of the component macro you have created beforehand (e.g. CONTACTOR1) into the **Graphics macro** field. You can alternatively use the [...] command button to open the Macro selection window, and select the desired macro there.

The screenshot shows a software dialog box titled "Components - MOE.010042". It has four tabs: "Gen. data", "Prices", "Supplementary data" (which is active), and "Technical data". In the "Supplementary data" tab, there are several input fields. The "Graphics macro" field is highlighted with a red rectangular box and contains the text "CONTACTOR1". To the right of this field is a small button with three dots "...". Other fields include "Group number", "Identifier" (containing "K"), "Part classification", "Type of part", "Transfer identifier" (a dropdown menu showing "Only DD"), "Technical characteristics", "Connection designations" (containing "A1", "A2", "13", "14", "21", "22", "31", "32", "4" with a button "..."), "Symbol file", and "Symbol number" (containing "0" with a button "...").

Fig. 59: The connection of the component macro is made in the Suppl. data tab of the parts master data

Afterwards switch to the **Technical data** tab. Select the position where the component macro is to be placed (e.g. mounting panel) using the dropdown list of the **Mounting site** field.

18.4 Assigning Part Numbers to a Device (Third Step)

To establish a relation among the respective device in the schematic, the part management, and the component macro, you must assign a part number to all devices in the schematic that you want to place on the mounting panel. Since we assume that you are familiar with entering part data in the symbol properties dialog box, we refrain from describing this procedure here.

18.5 Generating the Equipment File (Fourth Step)

You can only equip mounting panels with component macros if you have generated a so-called **equipment file** beforehand. This file saves all device designations to which a part number has been assigned. To generate the equipment file, follow the menu path **Generate > Panel layout > Generate** from the EPLAN main menu. The file will be created immediately when the function is called up.

18.6 Inserting the Mounting Panel (Fifth Step)

After having generated the equipment file, you can insert the mounting panel. Please note that you must have created and opened a project page of **type "C = Panel layout"** beforehand. Mounting panels can be designed on this page type only. If you want to carry out true-to-scale design, enter the same scale in the page properties as has been selected for the component macros.

In contrast to previous EPLAN versions, a mounting panel is no longer drawn by means of a black box but inserted as a separate "mounting panel" graphics object (see Fig. 60 on Page 125).

Selecting **Extras > Mounting panels > Insert** will activate the insertion mode. First use the cursor to define the starting point of the mounting panel. Press the left mouse button briefly, and draw a frame in the desired size, just like a black box is drawn. The end point is determined by pressing the left mouse button again. The Symbol properties dialog box appears in which you can specify the symbol texts, the text display, as well as the line and parts data.



Do not forget to enter a device designation for the mounting panel. EPLAN requires this specification for the creation of the legend data.

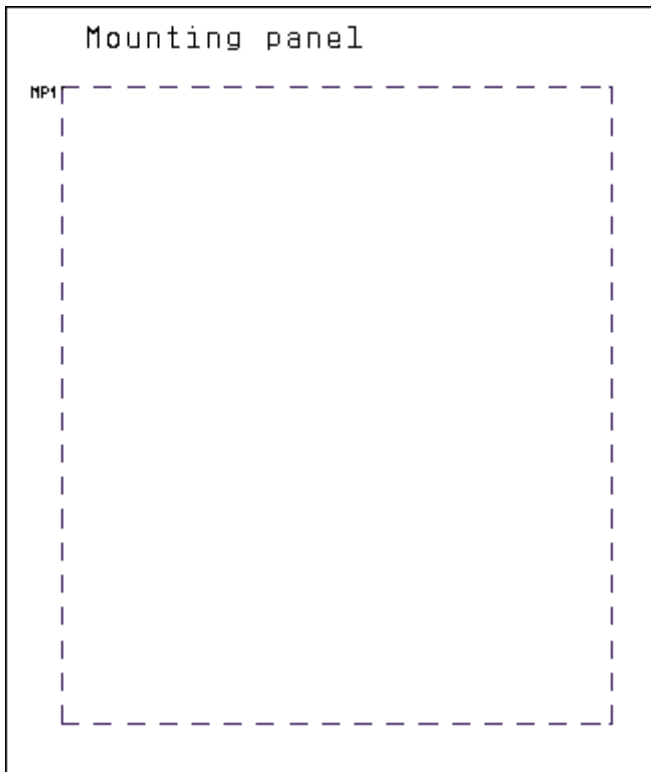


Fig. 60: Inserted mounting panel

18.7 Equipping the Mounting Panel (Sixth Step)

Next, equip the mounting panel with the component macros. To do this, follow the menu path **Extras > Mounting panels > Equip** in graphics editing to call up the Components for panel layout dialog box. This equipment dialog box contains a table with an overview of all devices used in the schematic (see Fig. 61 on Page 126). A large number of data columns is displayed here. EPLAN fills most of the columns with the corresponding information from the system-specific parts management.

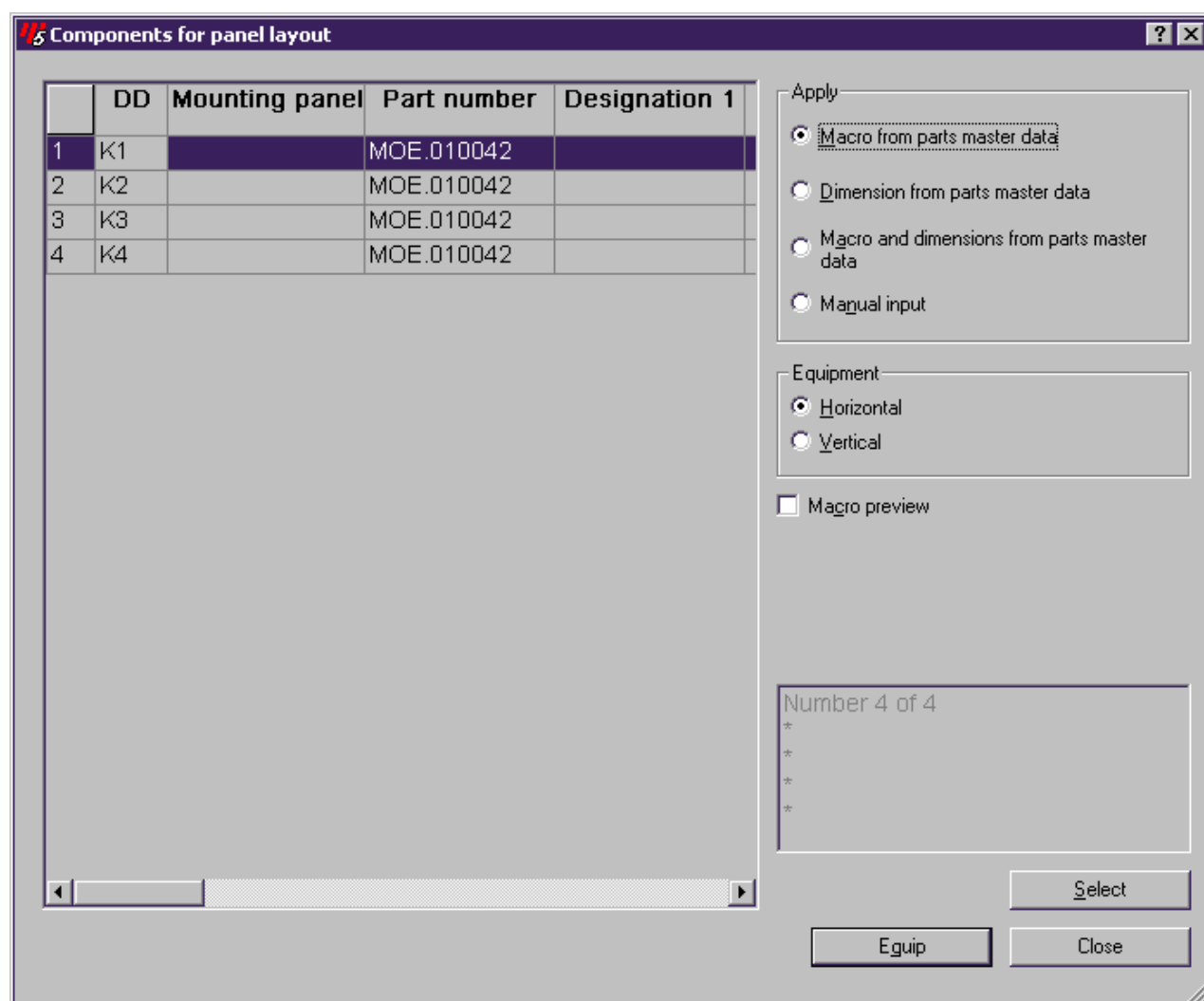


Fig. 61: Equipment dialog box

Using the **[Select]** button of the equipment dialog box, you can adapt the component overview to suit your needs. In the **Configure and select** dialog box that comes up you can configure the display of the equipment data and have an influence on the component overview according to specific criteria (see Fig. 62 on Page 127).

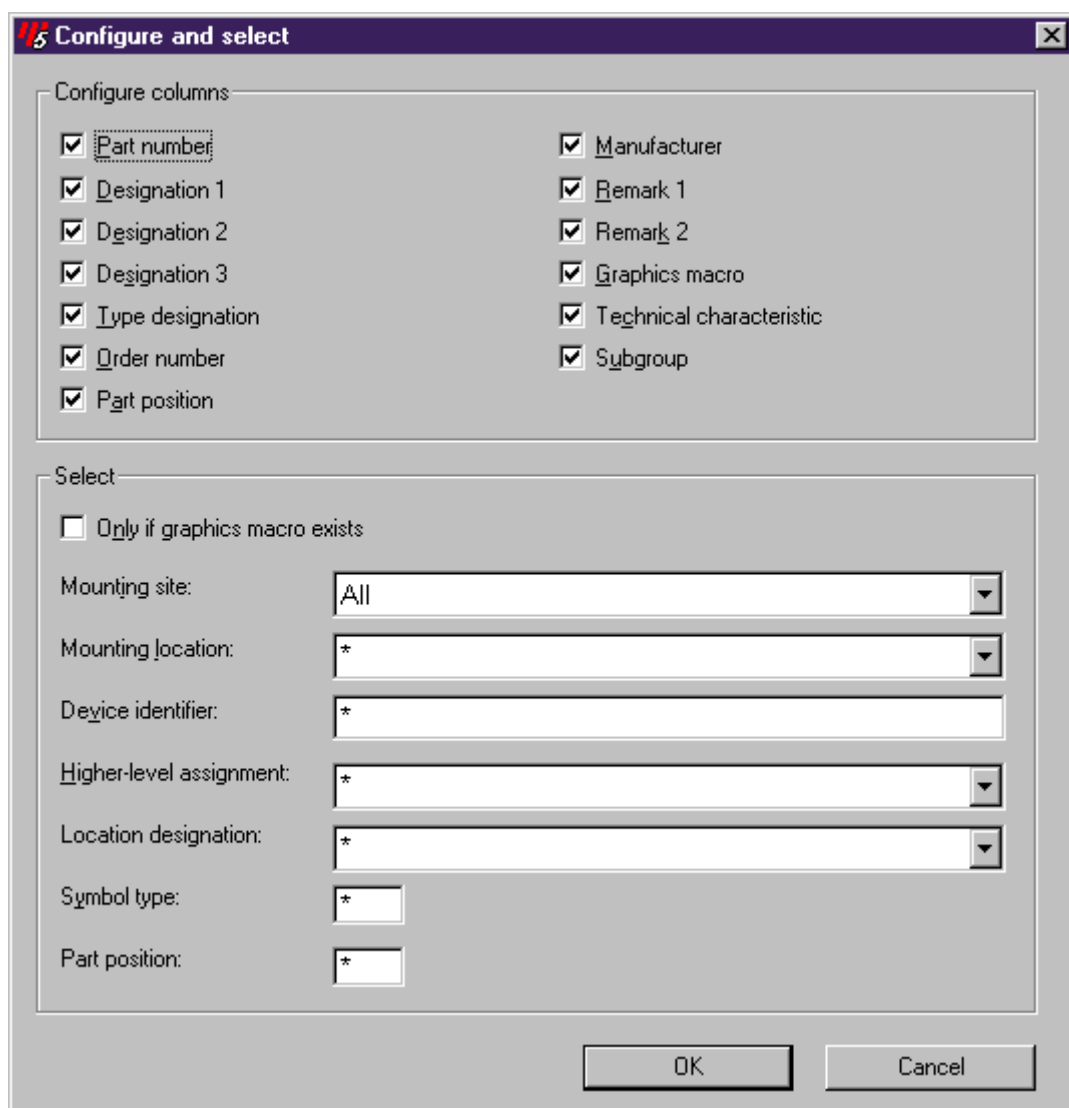


Fig. 62: Configure and select dialog box

Configuring table columns

In the Configure columns group box you can define which data columns are to be displayed in the overview. Activate or deactivate the check boxes of the data you wish to be shown/hidden in the Components for panel layout dialog box.

Selecting components

In the Selection group box you can specify which components are to be displayed in the overview. The following selection criteria are available (with the possibility to define a combination of several criteria):

- **Only if graphics macro exists:**
EPLAN only displays components for which a component macro was entered in the parts master data.

- **Mounting site:**

Here you can select the mounting site as the selection criterion. The dropdown list offers the following options:

- Mounting panel
- Pivoting frame
- Door
- Side section
- Top cover
- Not defined
- All.

- **Mounting location:**

In this field, the mounting location specified in the symbol properties of devices is chosen as the selection criterion.

- **Identifier:**

Enter the identifier here according to which you want to select the components. You can also use logic functions for the selection.



For example, you only want to have the components containing a "K" or "F" in their device designation displayed in the overview. In this case, enter "K||F" as the selection criterion into the field. To enter the "|" character, press  .

- **Higher-level assignment / Location designation:**

Select the higher-level assignment or location designation you want to use for the component selection. You can open the two dropdown lists only when editing a DIN project.

- **Symbol type:**

Enter the symbol type here according to which you want to select the components.

Defining the application

The radio buttons in the Apply group box are used to specify how the component macro is displayed on the mounting panel and which data are used for it. You can choose between the following options:

- **Macro from parts master data:** If this option is selected, EPLAN will use the component macro entered in the parts master data (combined macro).
- **Dimension from parts master data:** In order to display the component, EPLAN does not use a macro but automatically draws a rectangular graphics object. The measurements (height and width) are taken from the Technical data tab of the parts master data. If all mounting panel parameters are activated when this option is used, up to four placeholder texts will be displayed at the component macro after the object has been placed (see Fig. 65 on Page 131).
- **Macro and dimensions from parts master data:** This option is a combination of the first and second option.

- **Manual input:** Here, too, EPLAN does not use a macro but draws a rectangular graphics object to display the component. In this case, the measurements are taken from the subsequent **Graphics object** dialog box. Enter the extension of the object in the X and Y direction and the distance in the X direction. Even if you have entered a component macro in the parts master data, the manual input takes precedence in this case. If all mounting panel parameters are activated when this option is used (see the section "Configuring the Placeholder Texts (Seventh Step)" on Page 130), up to four placeholder texts will be displayed at the component macro after the object has been placed (see Fig. 65 on Page 131).

Additional settings

- **Equipment:**
In this group box you can determine how the component macros are to be placed on the mounting panel:
 - **Horizontal:** The component macros are placed from left to right on the mounting panel.
 - **Vertical:** The component macros are placed from bottom to top on the mounting panel.
- **Macro preview:**
Activate this check box if you want to have a preview window displayed. Once you highlight a component in the overview, it will be displayed there.

Equipping a Mounting Panel

When you have specified the selection criteria and selected the transfer identifier, you start the actual equipment procedure. To do this, highlight one or several components in the overview, and hit the **[Equip]** button. The corresponding graphics objects or component macros can then be booked consecutively from the overview and placed on the mounting panel with a mouse click (see Fig. 63 on Page 130). If a graphics object or component macro exceeds the edges of a mounting panel, a message is displayed.

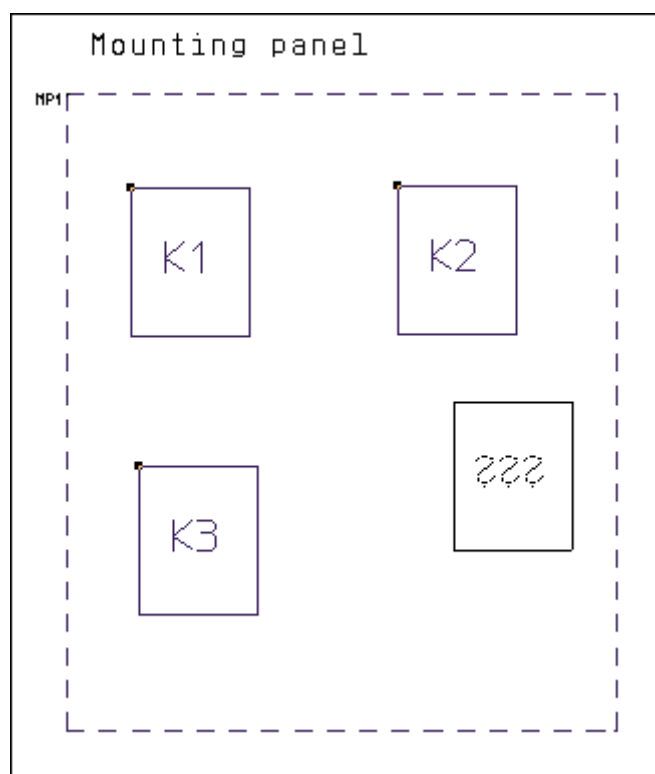


Fig. 63: Mounting panel equipped with components

18.8 Configuring the Placeholder Texts (Seventh Step)

When the graphics objects or component macros have been placed, you can decide which placeholder texts are to be displayed on the component or component macro.

For this purpose, follow the menu path **Parameters > Project > Mounting panel** in the graphics editing menu. This will open the dialog box for the mounting panel parameters (see Fig. 64).

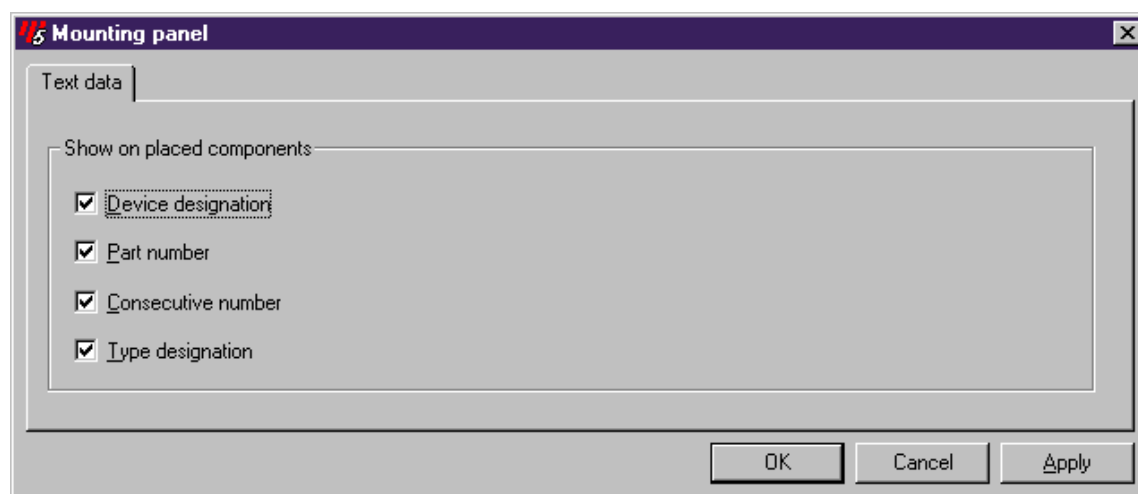


Fig. 64: Mounting panel parameters

In the Show on placed components group box, activate the check boxes before the respective text types for the placeholder texts to be displayed, and confirm your parameter selection by clicking **[OK]**. The corresponding placeholders will then be displayed at the component (see Fig. 65) Please note:



Using the options in the Apply group box you can specify whether a component macro or a rectangular graphics object is to be inserted on the mounting panel.

When selecting the **Macro from parts master data** or **Macro and dimensions from parts master data** options, you can output only one placeholder text, namely the one whose text type you selected in the Element data dialog box when creating the **combined macro**. The text is only shown at the component if you have defined the same text type in the parameter dialog box!

However, if you select one of the **Dimension from parts master data** or **Manual input** options, which create a rectangular graphics object as component on the mounting panel, you can have up to four placeholder texts displayed if you activate the corresponding check boxes for the text types in the parameter dialog box.



Fig. 65: Component with the four possible placeholder texts

18.9 Creating Legends (Last Step)

EPLAN now offers two legend types, which you can create separately or together in the project.

- Window legend
- Page legend.

Forms of the *.skc type are required to display the two legends. Here, you can define special texts whose number and position can be individually set. Furthermore, you have the possibility to add supplementary texts, e.g. table headers, and lines to the form.

18.9.1 Creating a Window Legend

The window legend is handled like a symbol. It is therefore possible to place it freely on a schematic page or directly next to the mounting panel.

As window legends dispose of symbol properties, they also have a device designation.



To output a window legend, the device designation of the mounting panel must match the device designation of the window legend!

To create the window legend, select **Extras > Mounting panel > Insert window legend** in the graphics editing module. After having selected the desired form from the subsequent Legend forms dialog box, you can position the window legend next to the mounting panel by means of the mouse (see Fig. 66).



Please make sure to select a window legend form for the creation of a window legend. Otherwise the data output will be incorrect!

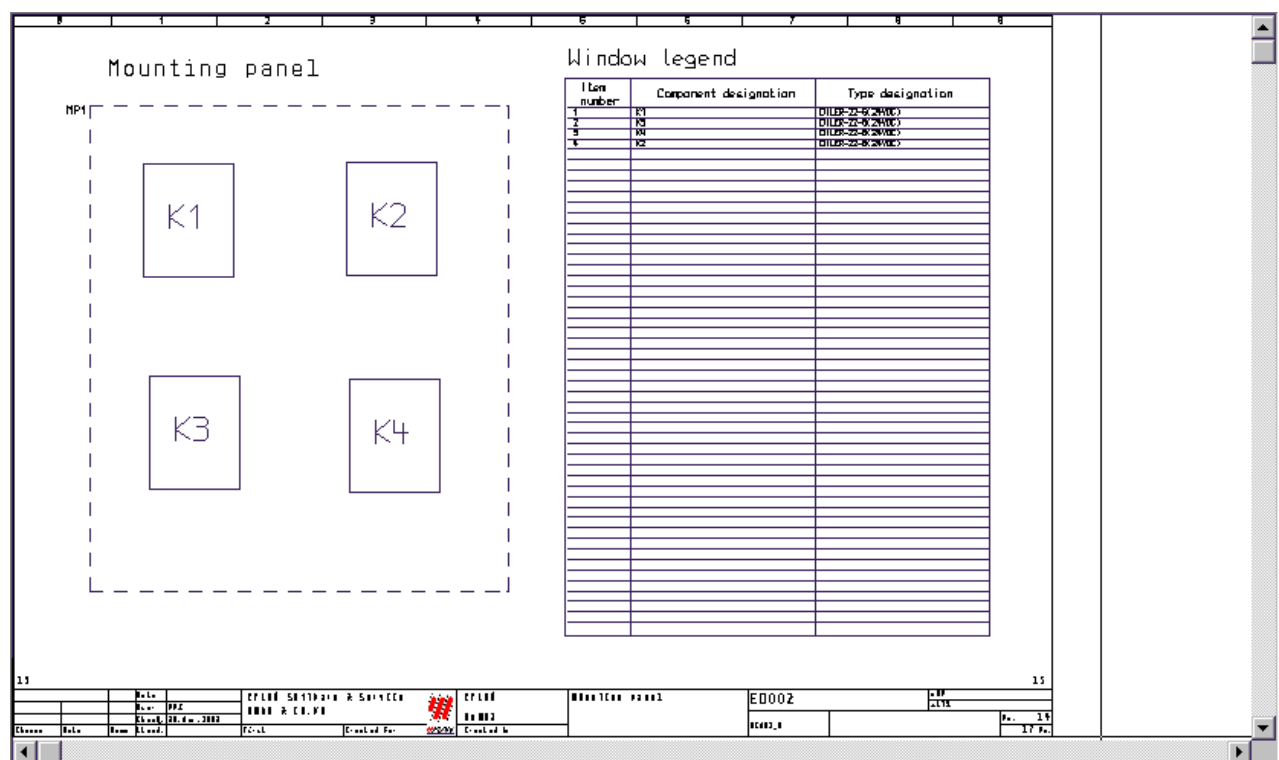


Fig. 66: Inserted window legend next to the mounting panel

18.9.2 Outputting a Page Legend

Unlike a window legend, a page legend is output as supplementary graphics pages in the project – just like terminal or interconnect diagrams. The data for the legend creation and the output are set in the **Page legend** group box of the mounting panel symbol properties (see Fig. 67).

The screenshot shows the 'Symbol properties' dialog box with the 'Page legend' section highlighted. The 'Start page' field is set to 4. The 'Running direction' dropdown is set to '0 = Horizontal'. The 'Form' field is set to 'Essc002e'. The 'Character for subpages' dropdown is set to 'a = Alphabetic (lower case)'. There are also checkboxes for 'Create subpages' and 'Higher-level assignment', 'Location designation', and 'KKS'.

Fig. 67: The fields for specifying page legend data in the symbol properties dialog box

In the **Start page** field you can determine on which page the output of the legend is to be started.

If you are working with a DIN or KKS project, you can also specify the higher-level assignment and/or location designation or the KKS identifier.

The **Running direction** dropdown list allows you to select the evaluation method for the macros concerning the legend creation. The **Horizontal** setting means that EPLAN will conduct the evaluation from left to right and from top to bottom. If you select **Vertical**, the evaluation is carried out from top to bottom and from left to right.



Note that the evaluation depends on the position of the insertion points on the mounting panel!

Use **Form** to select the corresponding page legend or window legend form.



Remember to select a page legend form when outputting a page legend. Otherwise the data output will be incorrect!

If you want to output **subpages**, activate the corresponding check box. In the **Character for subpages** dropdown list you can determine whether a lower-case letter, an upper-case letter, or a number is to be used for the subpage designation.

A page legend is created by following the menu path **Extras > Mounting panels > Create page legend**. The following figure shows an extract of a page legend.

Page legend			
Item number	Component designation	Type designation	Part number
1	K1	DILER-22-6(24VDC)	MDE. 010042
2	K2	DILER-22-6(24VDC)	MDE. 010042
3	K3	DILER-22-6(24VDC)	MDE. 010042
4	K4	DILER-22-6(24VDC)	MDE. 010042

Fig. 68: Extract of a page legend

18.10 Updating Mounting Panel Data

If you have modified the device designation of a component located on a mounting panel in the schematic, you must update the mounting panel data afterwards. To do so, select **Extras > Mounting panel > Update** in the graphics editing module. EPLAN then replaces the old device designation with the new one at the component placed on the mounting panel as well as in the legend. Also use the update function after having deleted a component from the mounting panel!

If you modify device designations outside the graphics editing module, e.g. by using the numbering module for devices, you must regenerate the equipment file first (to do so, follow the menu path **Generate > Panel layout > Generate** in the main menu). Then select the **Update** function in the Mounting panel menu to keep the legend up-to-date.

18.11 Editing Special Text Coordinates in the Legend Form

Just as it is possible for the output of the terminal diagram, the interconnect diagram and other generable graphics pages, you can change the special text coordinates in the selected form for the output of legends.

Open the coordinates dialog box by selecting **Extras > Mounting panel > Coordinates** in graphics editing.

First, the selection dialog box for legend forms appears. Select the desired form there, and confirm by clicking **[OK]**. Afterwards the **Coordinates of mounting panel legend** dialog box is opened.

If you want to change the position of a specific special text in the form, overwrite the value in the "X" or "Y" field or in both fields. Use the **[Move]** button to change **all** special text coordinates at one go. A dialog box comes up in which you enter the value that is to be used for all X and/or Y coordinate changes, and confirm your entry with **[OK]**.

Clicking **[Preview]** will open a preview window, which displays the changed special text positions in the form.

If you click **[Select]** you will either be returned to the form selection dialog box, or a dialog box is opened in which you can edit the font size, the text angle and/or the text alignment of the special texts. Which dialog box is opened depends on the field in which the cursor is located when the function is called up. The following table provides information in this regard:

Cursor in the field		Call of dialog box
"Plot forms"	[Select]	"Form selection"
"Style"		"Font size, text angle, text alignment"
Other fields		No call up possible

In the following section, you will find a table containing the new special texts for the legend creation.

New special texts

The following new special texts are available for the legend forms:

Designation	Special text number
Sequential number of legend sequence	1300
Device designation Mounting panel (incl. DIN identifier)	1301
Device designation Mounting panel	1302
Referenced DD (incl. DIN identifier)	1303
Referenced DD	1304

Designation	Special text number
Connection designation (from parts database)	1305
Connection cross-section	1306
Part number	1307
Trigger current	1308
Remark 1	1309
Remark 2	1310
Designation 1	1311
Designation 2	1312
Designation 3	1313
Width	1314
Mounting clearance - width	1315
Mounting clearance - height:	1316
Technical characteristic (from parts database)	1317
Part classification	1318
Weight	1319
Graphics macro	1320
Group number	1321
Holding power	1322
Manufacturer short name:	1323
Height	1324
Power	1325
Required space	1326
Voltage	1327
Current	1328
Type of part	1329
Depth	1330
Type designation	1331
Power dissipation	1332

19 Information Management

19.1 General

EPLAN allows several users at various workstations to edit one project at the same time. This may lead to conflicts when several users try to access the same record. In this connection, we have developed a so-called "information management" function which, should a conflict occur, will inform you which other user(s) is (are) currently also editing the project.

Please note the following conditions or default settings in order to use information management correctly and to the full extent in a multi-user system.

- The network parameter must be activated on all workstations (menu path **Utilities > Parameters > Workstation > Network**).
- In addition, an e-mail client must be installed and set up on all workstations.
- The user data in the working parameters must be completely specified (see following chapter).

19.2 Expansion of the User Data

The **Phone (number)** and **E-mail (address)** fields have been added to the user properties on the Working parameters 1 tab of the Parameters for language and workstation dialog box (menu path **Utilities > Parameters > Personal > Working parameters**) (see Fig. 69). In the case of a conflict, these data are output in a message together with the short name, the user name, and the workstation number (see Fig. 70 on Page 139).

This information enables you to ask the user by phone or e-mail to release the used data or to cancel the process.

The information must be completely entered and saved by clicking **[OK]** or **[Apply]**.

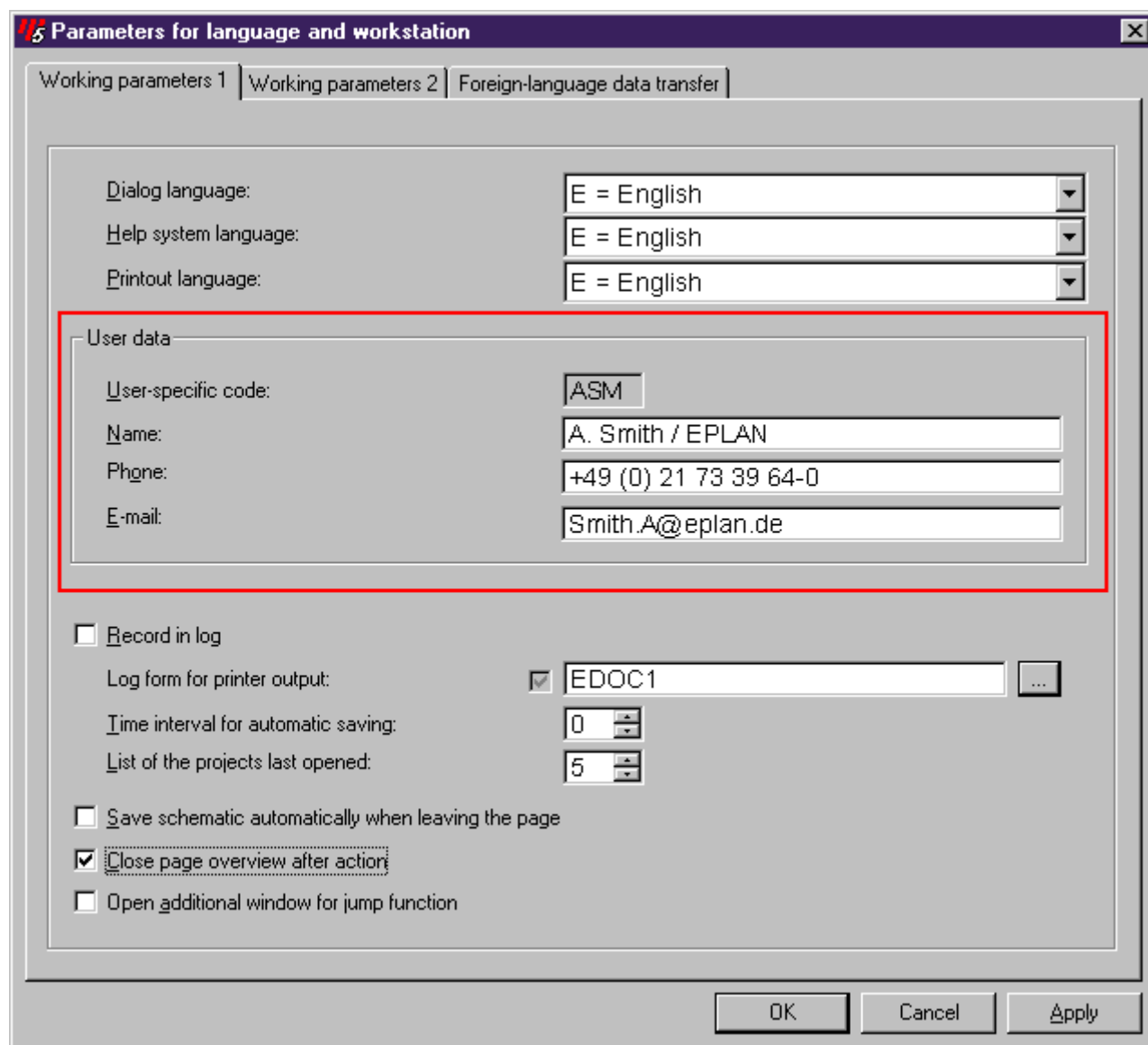


Fig. 69: User data for the information management

19.3 Cases of Conflict

If several users are editing the same project, there are various possibilities of conflicts. Conflicts may occur if

- You wish to open a page which is currently being edited by another user.
- You wish to edit a record in the parts management which is currently being accessed by another user.
- You want to call the higher-level assignment/location designation editor while another user has opened pages during graphics editing.

- You try to start an evaluation run while another user or several users has/have opened pages during graphics editing.
- You wish to start an evaluation run while another user is carrying out the same evaluation or an evaluation of the entire project.
- You want to call the graphics editing module while another user is carrying out evaluation runs.

19.4 Structure of the Message

Messages output in EPLAN in the case of a conflict (see Fig. 70) are structured as follows:

The header of the message tells you why the desired action cannot be carried out at the moment. Below you will find a table containing all users who are currently editing the project. It shows all important data of the users, such as the name, identifier, workstation number, phone number, and e-mail address. If other users get a conflict message they will also find your user data in this table.

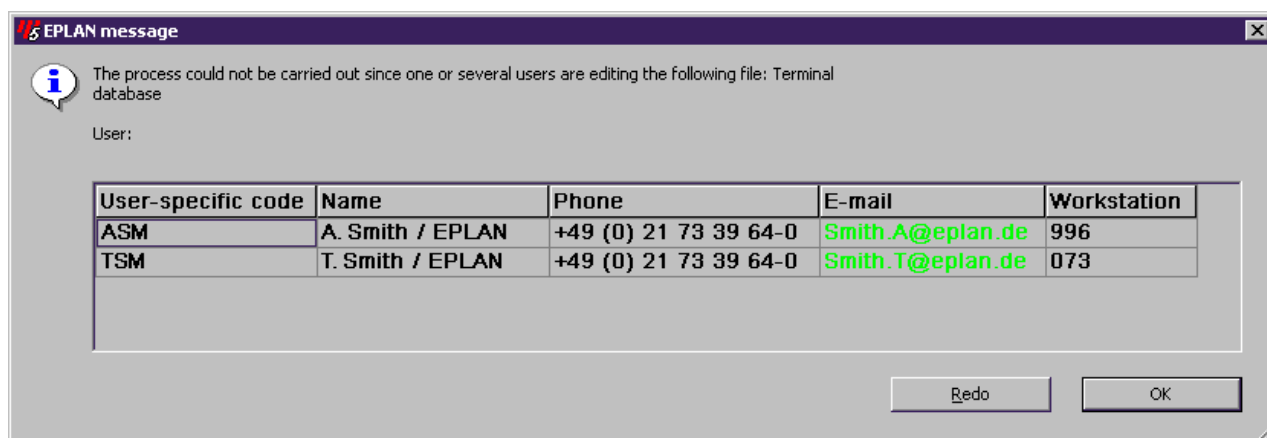


Fig. 70: Example of a message in a case of conflict

19.5 Informing Other Users by E-mail

Informing an Individual User

If you want to inform an individual user by e-mail, click on the e-mail address in the message window. As an alternative, you can also select **Send e-mail** from the pop-up menu, or highlight any field in the corresponding message line, and press the space bar.

The e-mail program installed on your computer will then be launched and you can enter your message. The field for the e-mail address is automatically completed by EPLAN.

Sending an E-mail to All Users

If several users are listed in the message, you can send an e-mail to all users at the same time. This may be the case if you want to start an evaluation run while several other users have already opened various pages in graphics editing.

To do so, highlight the E-mail column in the user table. Call up the pop-up menu for the highlighted column, and select the **Send e-mail** option. The e-mail program installed on your computer will then be started. As you will notice, EPLAN has already transferred all selected user addresses. Therefore you only have to enter the desired text and send your mail to the respective users.

19.6 Confirming a Message

If you click **[OK]**, EPLAN will close the message window. Depending on the case of conflict, the further procedure varies. If you e.g. try to open a page, which is currently edited by another user, EPLAN allows you to do so, however only in the read-only mode.

The **[Retry]** button allows you to re-access the data or to start a specific process (e.g. graphics editing or an evaluation run).

19.7 Updating the Status of Processing

If, for example, you have opened a schematic page, which is edited by another user, in the read-only mode, and this page is released for editing shortly afterwards, you won't receive a message.

In order to be informed of the current status of processing, follow the menu path **View > Update** in the graphics editing module. We recommend to always use this function before accessing a page or before starting an evaluation run in a multi-user system with EPLAN.

20 Using Long File Names

Up to now, file names in EPLAN had to conform to the "8.3 convention". This standard format consists of a file name of up to eight characters, followed by a period and a three-character file extension. As from Version 5.50 it is possible to use long file names for the files (apart from some exceptions). This allows you to assign a self-explanatory name to the file.

In this context, the maximum number of characters must be distinguished according to the following two cases:

- A maximum of **55 characters** when a file name is assigned to a "referenced" file.
Referenced files are files you specify in parameters, page properties, or in the parts management module, and which EPLAN saves internally in specific databases. This applies to all **forms** and **plot frames** as well as **parameter files** and **graphics macro files** specified in the parts management.
- A maximum of **255 characters** when a file name is assigned to an "unreferenced" file.
"Unreferenced files" are files that are not stored in the parameters, page properties, or parts management. This applies to all files used for the **EPLAN output functions** (import, export, printing) as well as all **macro files of the graphics editing module**.



In both cases, the length of the file extension is still limited to three characters.

If it is possible to enter the file path when entering a file name, the maximum file path length is 255 characters, as is usual in Windows® operating systems. In this case you must consider the directory structure used and the length of the directory names for the possible file name length. In EPLAN, it is for example allowed to indicate the file path when a project is exported to the EXF format.



Please note that long file names may pose a problem when data are exchanged with a user working with a previous EPLAN version (prior to EPLAN 5.50). For example, forms may not be found since their file names are cut as a result of the limited file name length.

Similar problems may occur if EPLAN is run on a network which supports file names only according to the 8.3 convention (e.g. Novell). In this case we recommend you to continue to assign file names in accordance with the 8.3 convention.

Notes:

- File names of **projects**, **symbol files**, **contactor definition files**, **contactor lists**, **connector type files**, **cable type files**, and **language files** are still limited to eight characters.
- The use of blanks, periods, or special characters when entering file names is not allowed. The period may only be used as separator for the file extension.

21 Outputting Wiring Lists

EPLAN 5.50 now enables you to output wiring lists as graphics pages. In the following sections, you will be familiarized with the new menu for the graphics output, and you will learn how to proceed when outputting the wiring list.

21.1 New Menu Item in the Cross-reference Module

We have added the **Graphics output** item to the menu of the cross-reference module to enable you to output wiring lists as graphics pages (cf. Fig. 71).

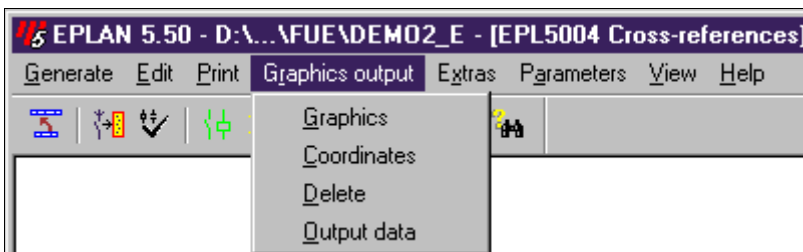


Fig. 71: Extended menu in the cross-reference module

The **Graphics output** pulldown menu offers the following options:

Graphics:

Select this option to output the wiring list as graphics pages. Detailed information on this topic can be found on Page 144.

Coordinates:

This option is used to specify the position of special texts in the selected plot form.

Delete:

Selecting this option will delete the wiring list, which is output as graphics pages, in the project.

Output data:

As already mentioned under **Graphics**, you can use this option to call up or change the output data for the graphics pages.

21.2 Outputting Wiring Lists as Graphics Pages

Before outputting the wiring list, you must select the desired form in the graphics output parameters (menu path **Utilities > Parameters > Project > Graphics output -> Graphics forms tab**). If you click on the [...] button next to the Wiring lists field, a dialog box appears in which forms of the *.skv type are available.

Then follow the menu path **Generate > Cross-references / wiring list** to launch the cross-reference module, and select **Graphics output > Graphics** there. At first, the Output data dialog box appears – unless you have not specified the output data yet. Here, you can specify the **project name**, name of the **editor**, and the **date** before outputting the graphics pages. These data will later be displayed in the corresponding fields of the graphics pages.



If you have already defined the output data via the corresponding option in the **Graphics output** menu, this dialog will not come up.

EPLAN then opens the Output wiring list dialog box, in which you can specify specific sorting criteria for the output (see Fig. 72 on Page 145). The following fields are available:

Using the **Presort by mounting location** check box you can output a wiring list which is presorted by the mounting locations. The check box has to be activated to this purpose.

Within the mounting location, it is possible to sort the wirings by the **internal potential number**, the **cross-sections**, the **designations**, or the **location designations**. These options can be found in the Sorting group box.

If the Presort by mounting location check box remains deactivated, EPLAN outputs the wiring list without considering the mounting locations and uses the criterion selected in the Sorting group box.

The first and last value resulting from the sorting criterion which you previously selected in the Sorting group box are assigned to the **From** and **To** fields.

If you choose the cross-section as sorting criterion, the values 0.5 and 2.5 are, for example, assigned to these fields. You can restrict the scope of the list by assigning different values.

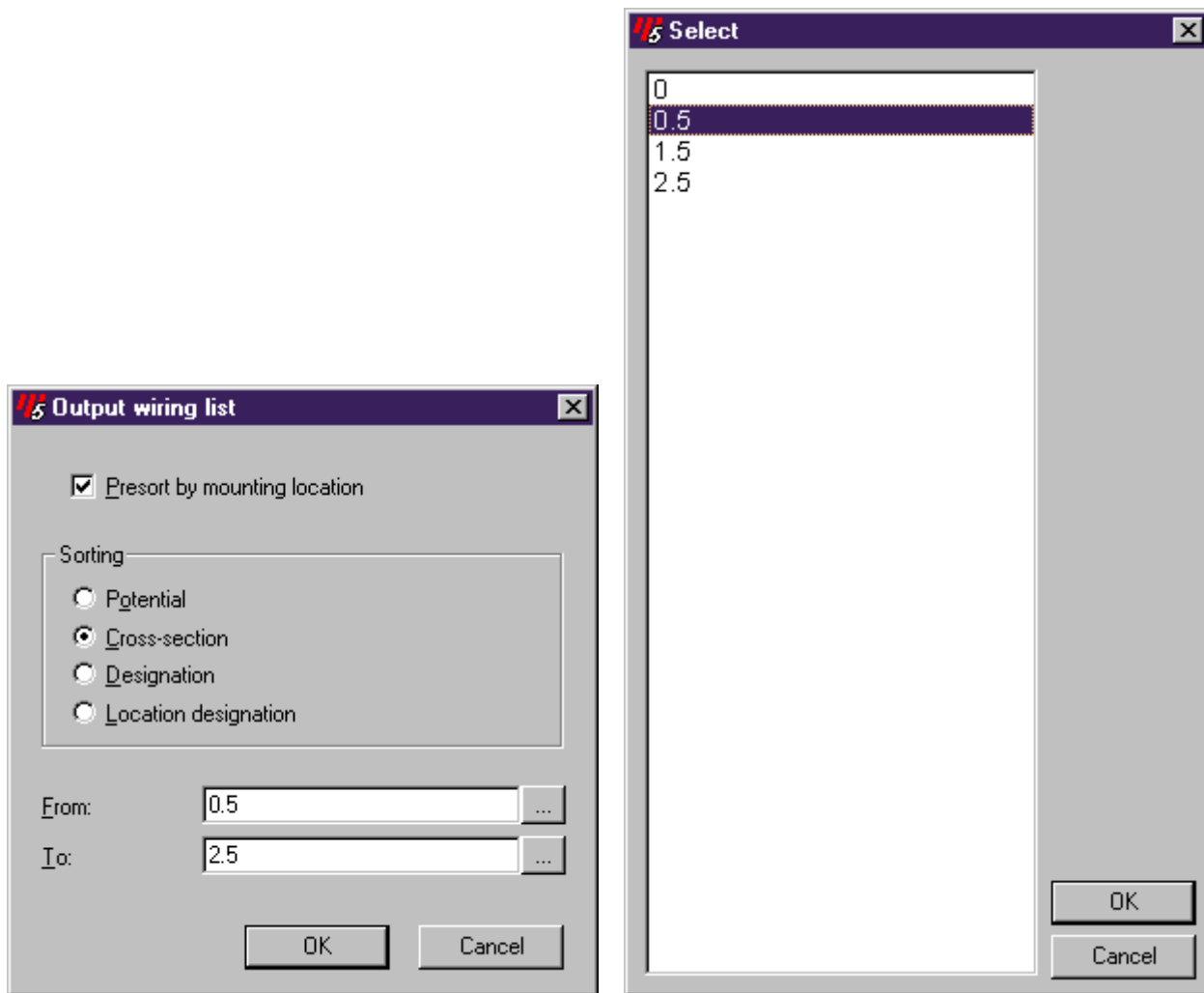


Fig. 72: Dialog box for defining sorting criteria

Clicking on one of the [...] buttons will open a list from which you can select the values corresponding to the chosen sorting criterion. The values are entered as predefined data in the From or To input fields.

Click **[OK]** to confirm the entered sorting criteria, and exit the Output wiring list dialog box.

If the current project is a DIN project, EPLAN will open a selection dialog box, in which you can define the output location of the graphics pages (see Fig. 73 on Page 146). Select the desired higher-level assignment and location designation and confirm by clicking **[OK]**.



In projects with serial numbering, this dialog box is not displayed. In this case, EPLAN immediately opens the dialog box shown in Fig. 74 on Page 146.

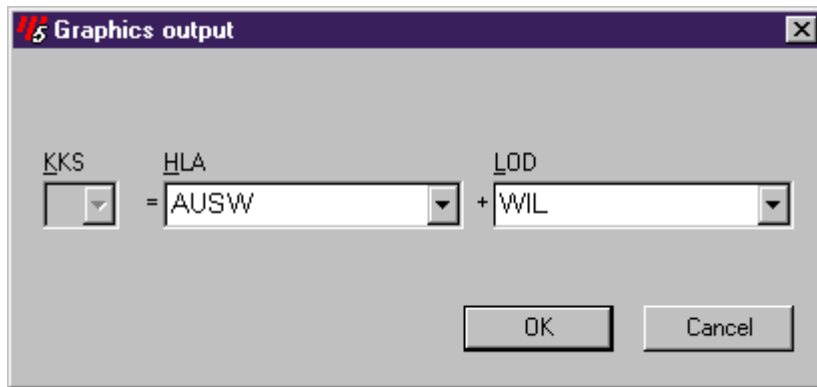


Fig. 73: Dialog box for defining the output location

If you have not output a wiring list before, EPLAN will open a dialog box asking you for the page number from which the graphics output of the wiring lists is to start (see Fig. 74). EPLAN will suggest the next free page. After you have specified the page number and confirmed with **[OK]**, EPLAN creates the graphics pages of the wiring list.

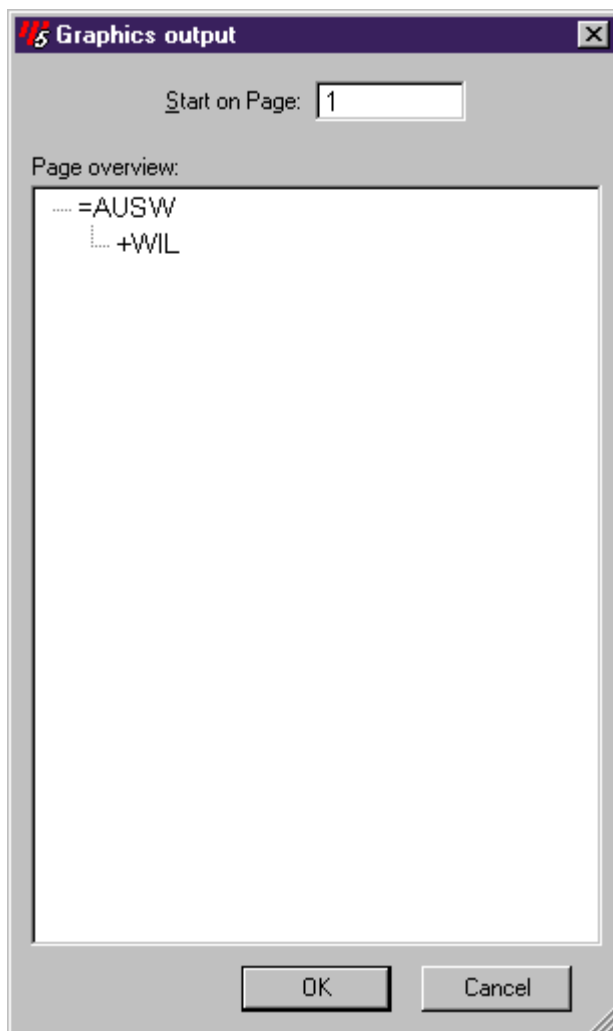


Fig. 74: Dialog box in which you specify the starting page for the output

22 Expansion of the Foreign-language Functions

Three new options have been added to the foreign-language functions. You can now:

- Highlight text parts, that are not to be translated, in translatable input fields
- Generate line feeds in a text-oriented way during the translation
- Have an influence on how the texts are taken over during the translation.

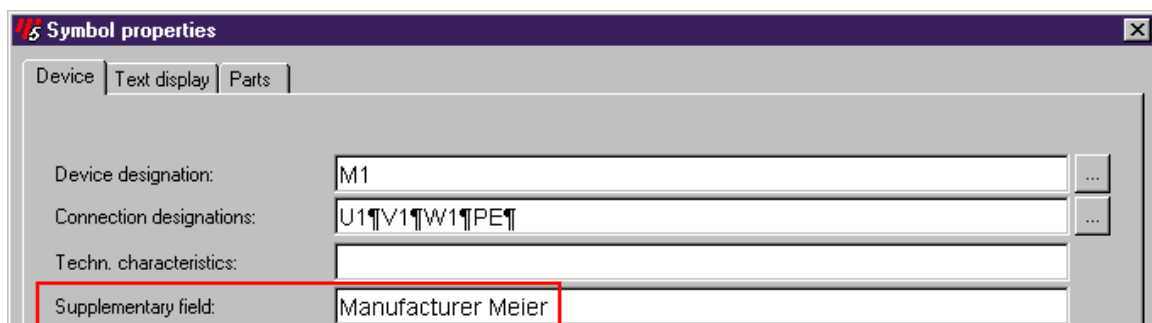
22.1 Excluding Texts from the Translation

It is now possible to exclude texts from the translation in translatable input fields. For this purpose, EPLAN offers a new function which is called up via a pop-up menu.

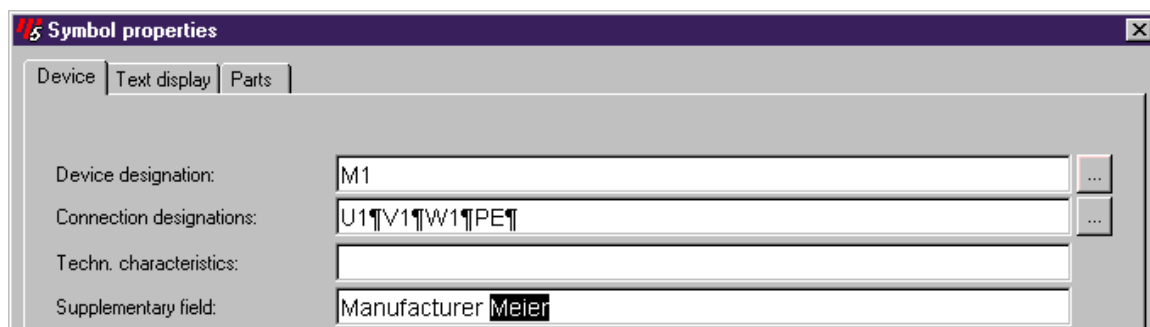
We would like to explain the application of the new foreign-language function by means of the following example:



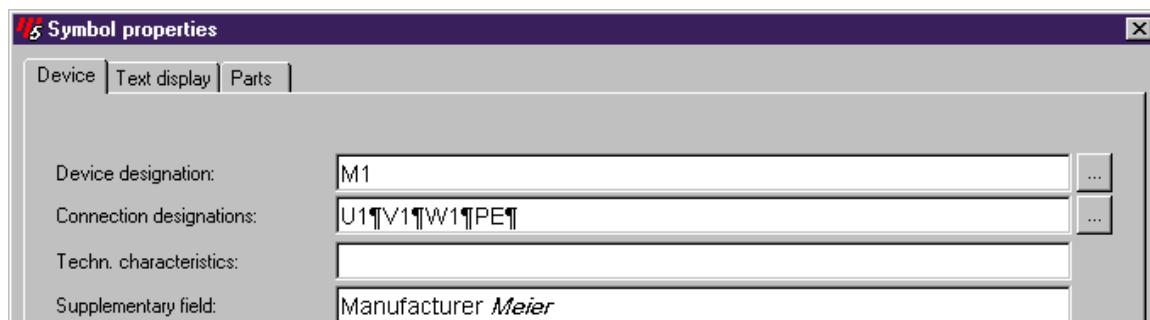
You have drawn a motor symbol into the schematic and specified the manufacturer in the supplementary field of the Symbol properties dialog box. You now want to output the text "Manufacturer Meier" (see the following figure) not only in the English but also in the German language in the schematic. The text part "Manufacturer" is contained in the dictionary and is to be translated, in contrast to the text part "Meier", which is not included in the dictionary.



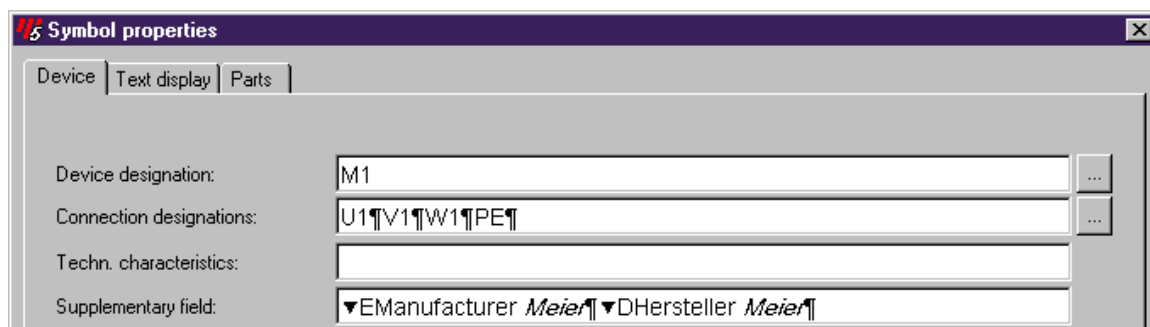
First highlight the word "Meier" as shown in the following figure.



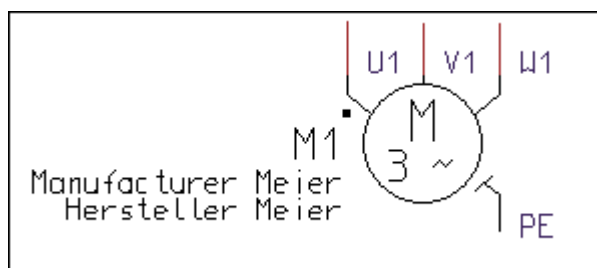
To exclude the text part "Meier" from the translation, right-click on the supplementary field. Select the **Highlight non-translatable text option** from the pop-up menu that appears. EPLAN will then display the text part "Meier" in italic letters (cf. the following figure).



Right-click on the supplementary field again, and select the Translate option from the pop-up menu. Without opening the dictionary EPLAN now translates the entry into the German language, with only the text part "Manufacturer" being considered. This is illustrated by the following figure:



In the schematic, EPLAN displays the texts as follows:



22.2 Generating Line Feeds in a Text-oriented Way during the Translation

In previous EPLAN versions, line feeds for formatting purposes were automatically generated in the target text. This **separator-oriented** placing of line feeds during the translation process was not changed for EPLAN Version 5.50. However, in addition, it is now possible to generate line feeds in a **text-oriented** way, i.e. irrespective of the length and the number of words in the target text.

In order to enable you to decide which kind of line feed EPLAN is to create during the translation, we added two options to the parameters for foreign-language translation (see Fig. 75).

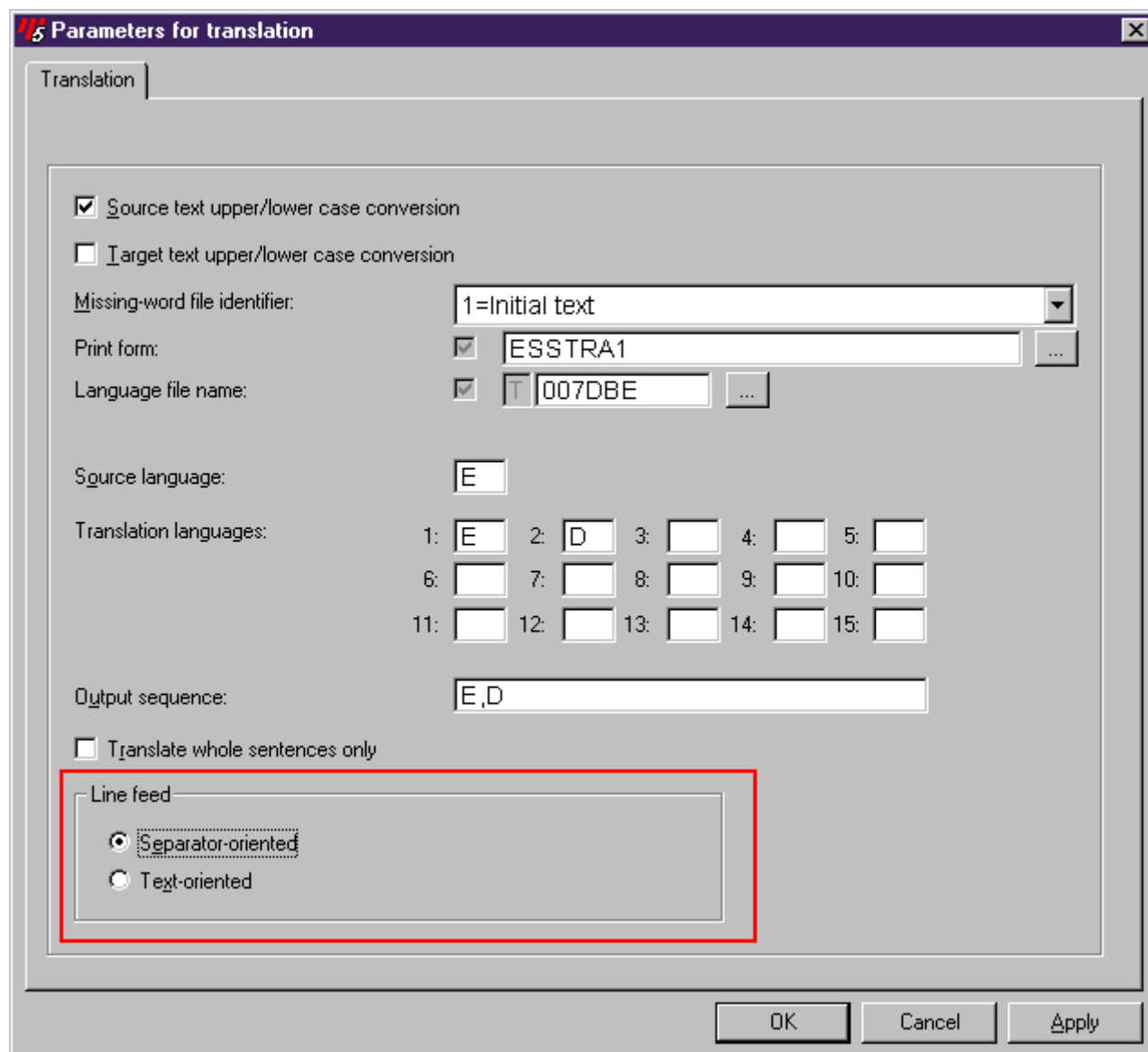


Fig. 75: Extended dialog box for foreign-language translation parameters

If you choose the **Separator-oriented** option, EPLAN will place the line feeds based on the separator, as usual.

If, however, you go for the Text-oriented option, the placing of the line feeds depends on their position in the original text.



EPLAN only places line feeds correctly for the Text-oriented option if you have the text translated word by word. A word-by-word translation is only made if the text to be translated is **not completely** contained in the dictionary and the Translate whole sentences only parameter is **not activated**.



The following table illustrates how the original texts are translated according to the Separator-oriented and according to the Text-oriented option.

Original:	manual-actuation button for lifting cylinder	starting circuit-breaker ventilator drive on
Separator-oriented:	manual-actuation button for lifting cylinder Handtaster für Hubzylinder	starting circuit-breaker ventilator drive on Motorschutzschalter Lüfter Antrieb ein
Text-oriented:	manual-actuation button for lifting cylinder Handtaster für Hubzylinder	starting circuit-breaker ventilator drive on Motorschutzschalter Lüfter Antrieb ein

22.3 Having an Influence on how the Texts are Taken over during the Translation

The Words found dialog box containing the suggestion list from the dictionary now offers a check box, which you can use to have an influence on how the texts are taken over during the translation.

If the **Lower-/upper-case conversion** is activated, EPLAN translates the original text according to the settings made in the Source text upper/lower case conversion and Target text upper/lower case conversion check boxes, which are available in the Parameters for translation dialog box of the parameter management.

However, EPLAN accepts the chosen word from the suggestion list as original text and translates it as it was entered in the dictionary of the respective language if the check box remains deactivated.

23 Standard Labeling

We have added several format elements to the standard labeling module.

These extensions apply to:

- Devices
- Wirings
- Cables
- Conductors
- Terminals.

23.1 Devices

When working with devices you now have the possibility to output the entire device designation to DIN. The format element required for this can be found under **"=HLA+LOD-designation"** in the list of available format elements (see Fig. 76). Furthermore, you can also output the information of the supplementary field of the symbol properties dialog box. In this case, you have to select **Supplementary field** from the list of available format elements.

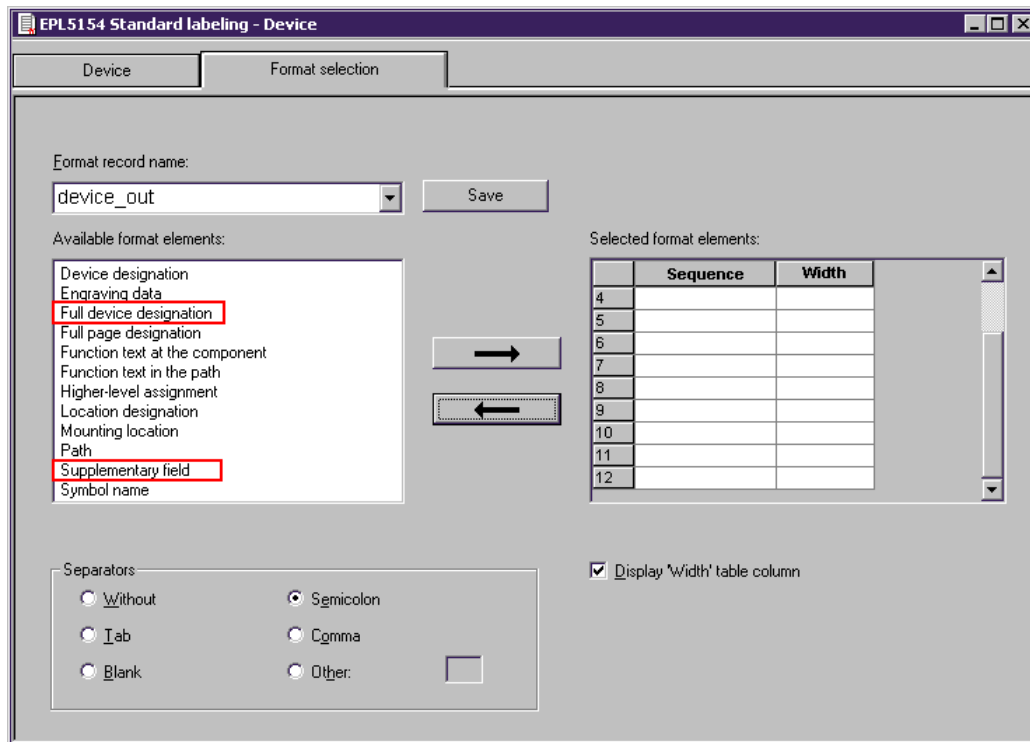


Fig. 76: Standard labeling for devices with new format elements

23.2 Wirings

Further output data were added to the wiring standard labeling (refer to Fig. 77). This means that you can now output the designations of both wiring lists in addition to the connection designation. To do so, select the entries "**Designation with connection designation (Target 1)**" or "**Designation with connection designation (Target 2)**" from the list of the format elements available. If you want this output to contain the higher-level assignment and the location designation as well, select the format elements "**Full designation with connection designation (Target 1)**" and/or "**Full designation with connection designation (Target 2)**". In addition, you can use **Remark** to write the contents of the remark field from the symbol properties dialog into the target file.

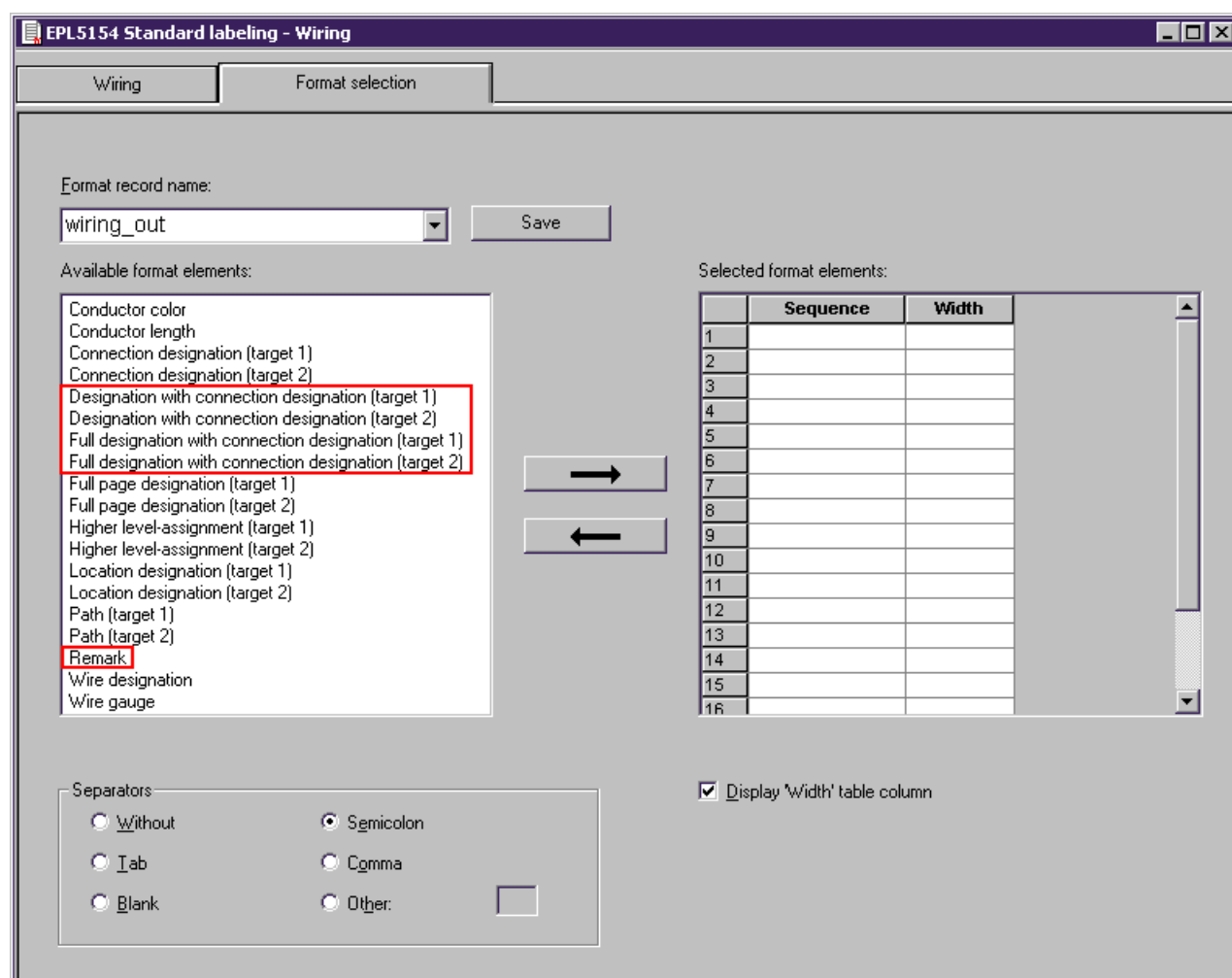


Fig. 77: Standard labeling for cables with new format elements

23.3 Cables

The **Voltage** and **Remark** format elements from the symbol properties dialog were added to the format element list for cables (see Fig. 78) Furthermore, the cable designation can now be output without the higher-level assignment and location designation. This entry is contained under "**Cable designation**" in the list of available format elements.

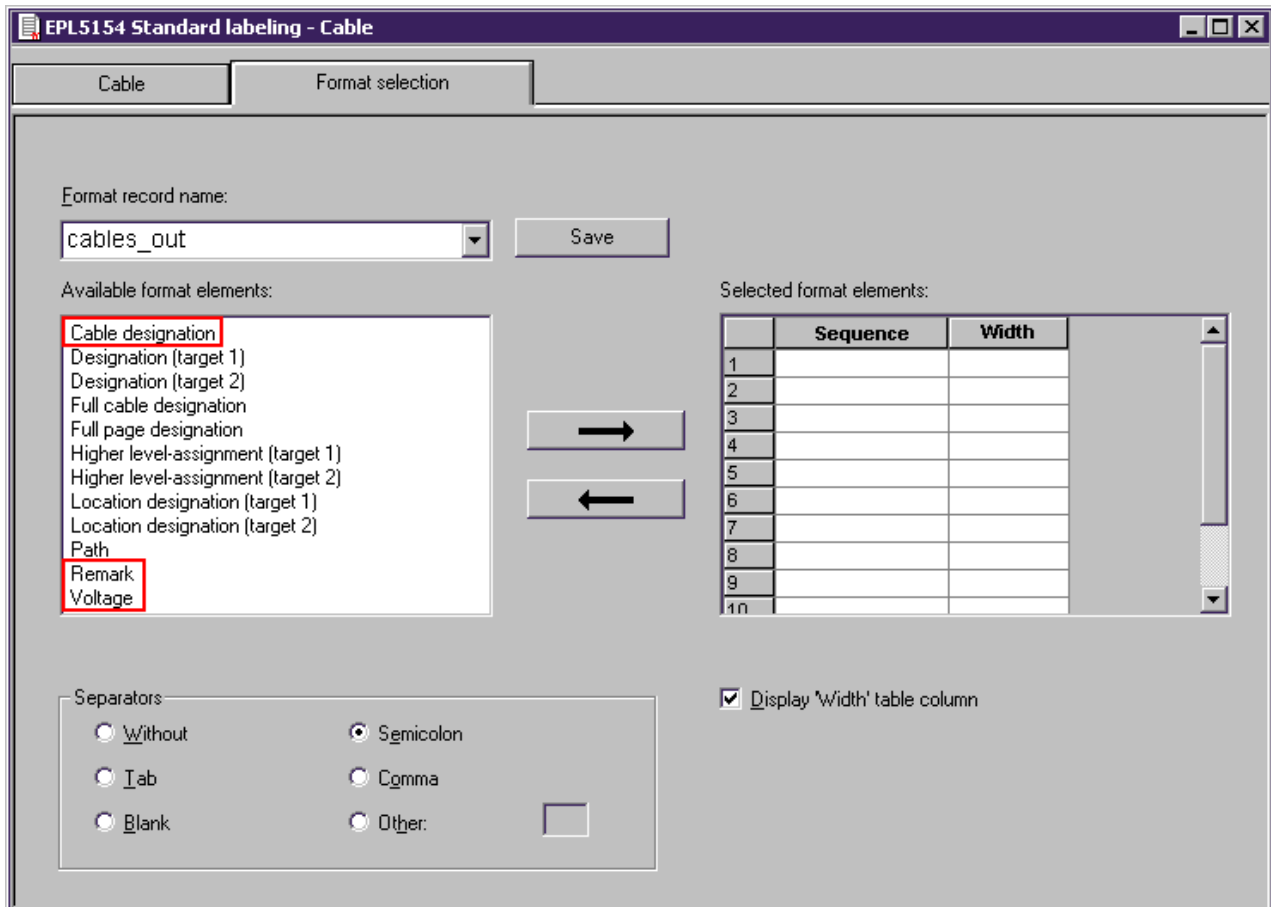


Fig. 78: Standard labeling for cables with new format elements

23.4 Conductors

The dialog box for conductors has also been supplemented by additional output data (see Fig. 79). You can now output the designation of the two conductor targets including the connection designation. To do so, select the entries "**Designation with connection designation (Target 1)**" or "**Designation with connection designation (Target 2)**" from the list of the format elements available. Similarly to the wiring you can also output the full designation of the conductor targets in accordance with DIN including the connection designation. To do so, select "**Full designation with connection designation (Target 1)**" and/or "**Full designation with connection designation (Target 2)**". Furthermore, the cable designation can now be output without the higher-level assignment and location designation. This entry is contained under "**Cable designation**" in the list of available format elements.

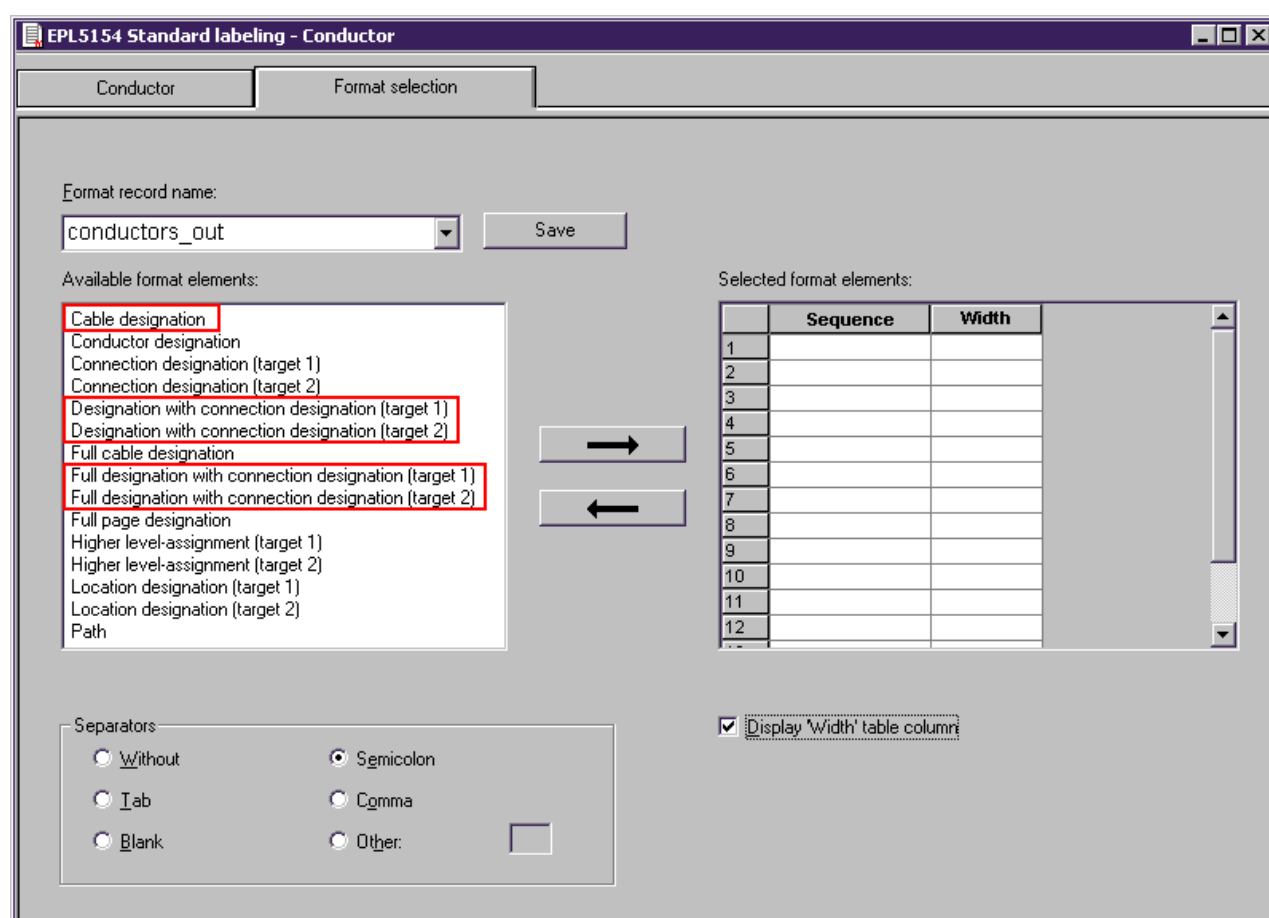


Fig. 79: Standard labeling for conductors with new format elements

23.5 Terminals

The dialog box for terminals now allows to output the entire strip designation according to DIN for the labeling (see Fig. 80). To this purpose we have added the entry **"Full strip designation"** to the list of available format elements. In addition, you can now output the **symbol type of the terminal**.

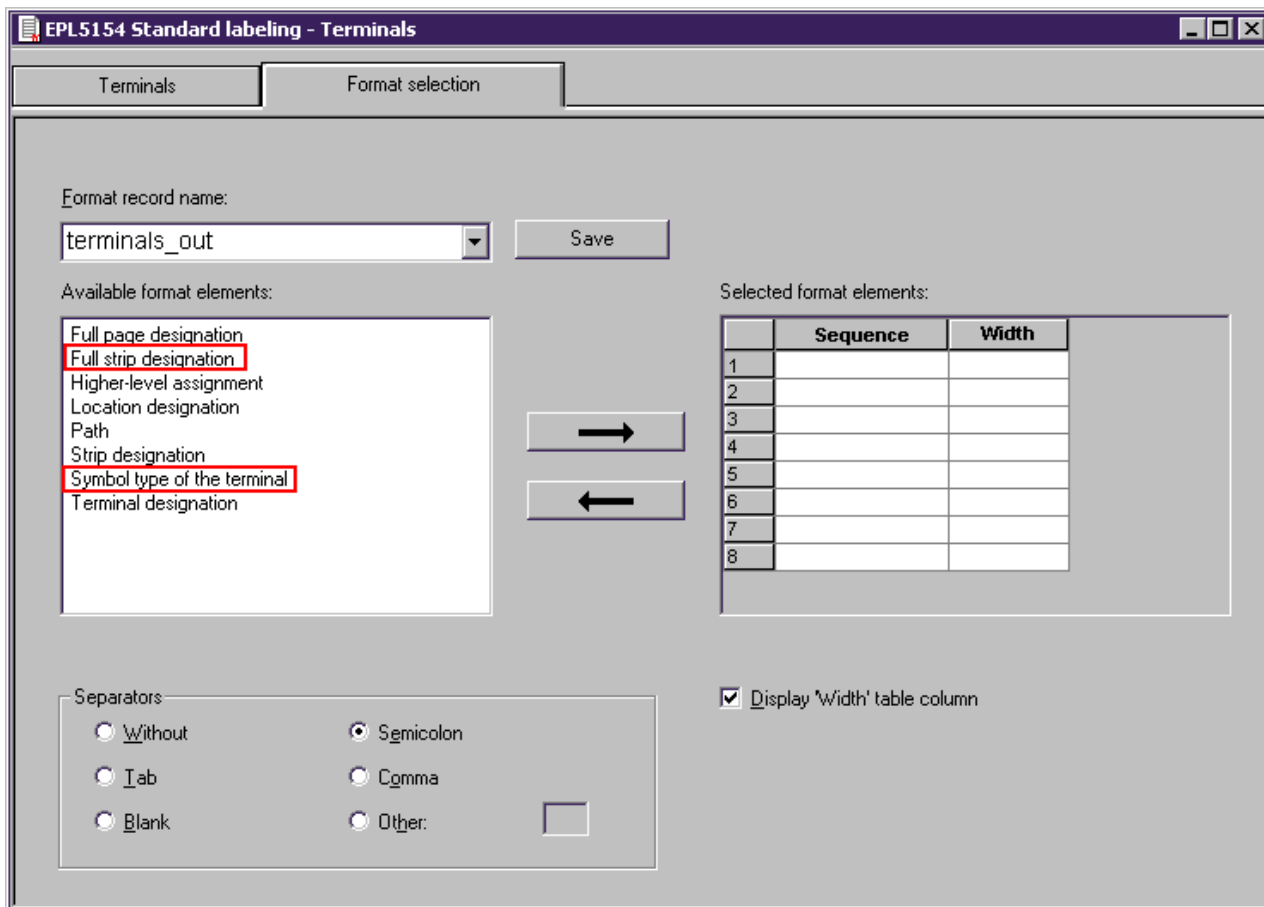


Fig. 80: Standard labeling for terminals with new format elements

24 Changing the Drive during Project Selection

When projects are exchanged, directories and drive characters often do not match due to the differing drive mappings. A manual change of settings is then required.

In order to minimize the work required for changing the directory or drive, we have optimized the process of changing the drive.

What has been improved?

First, we have redesigned the user interface of the Directory selection dialog box (see Fig. 81 on Page 158).

1. The fields for setting the parameter-specific (formerly called "Current") and project-specific directories are no longer arranged next to one another but one upon the other.
2. All fields for setting directories now contain a check box that cannot be manually changed. If the box is checkmarked, this means that the directory path specified exists. If, however, the box remains empty after a change has been made, the selected path does not exist.
3. In the group box for the project-specific directory settings, two new buttons – **[Current]** and **[Project]** – have been integrated which can be used to influence the selection of the drive.

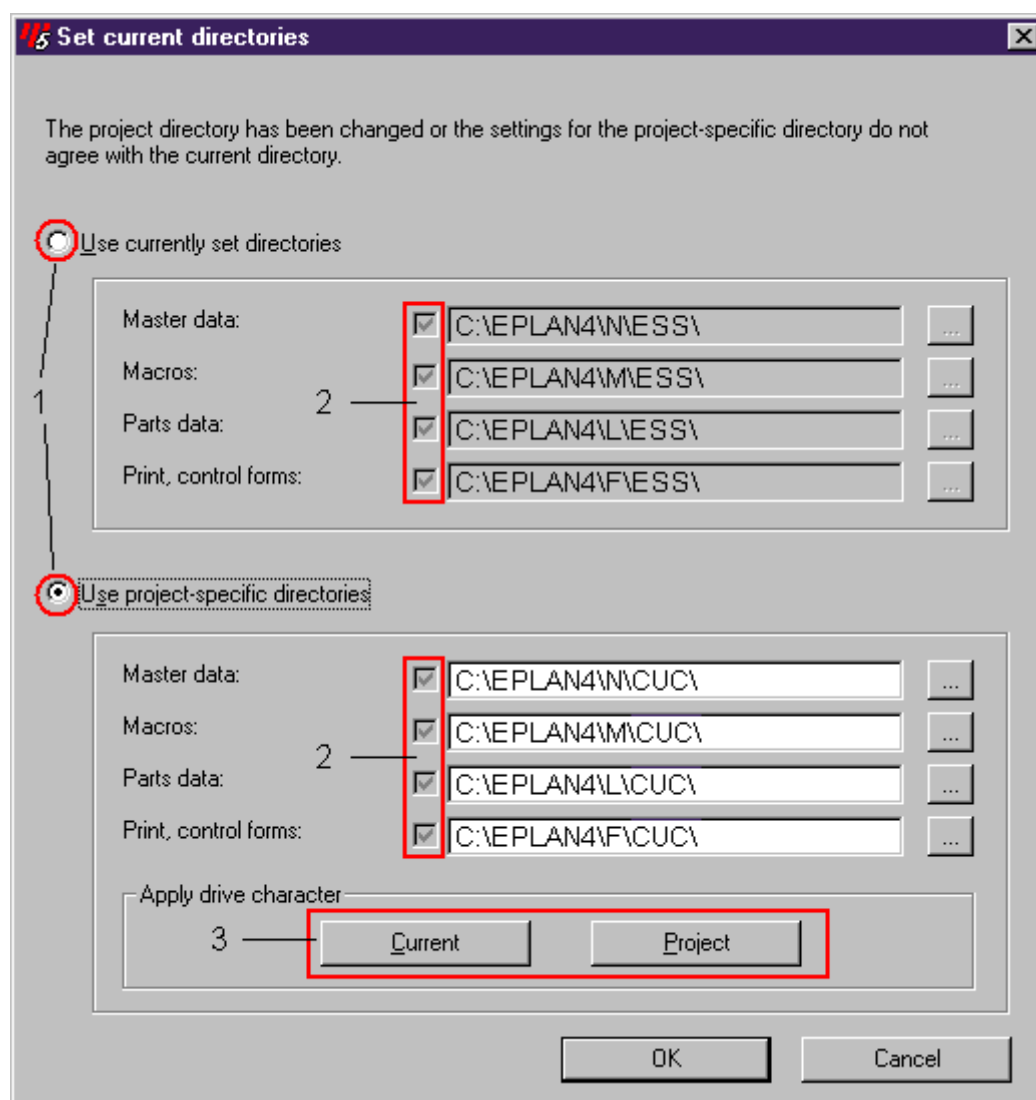


Fig. 81: Modified Directory selection dialog box

To what extent do the buttons described under 3. affect the selection of the drive?

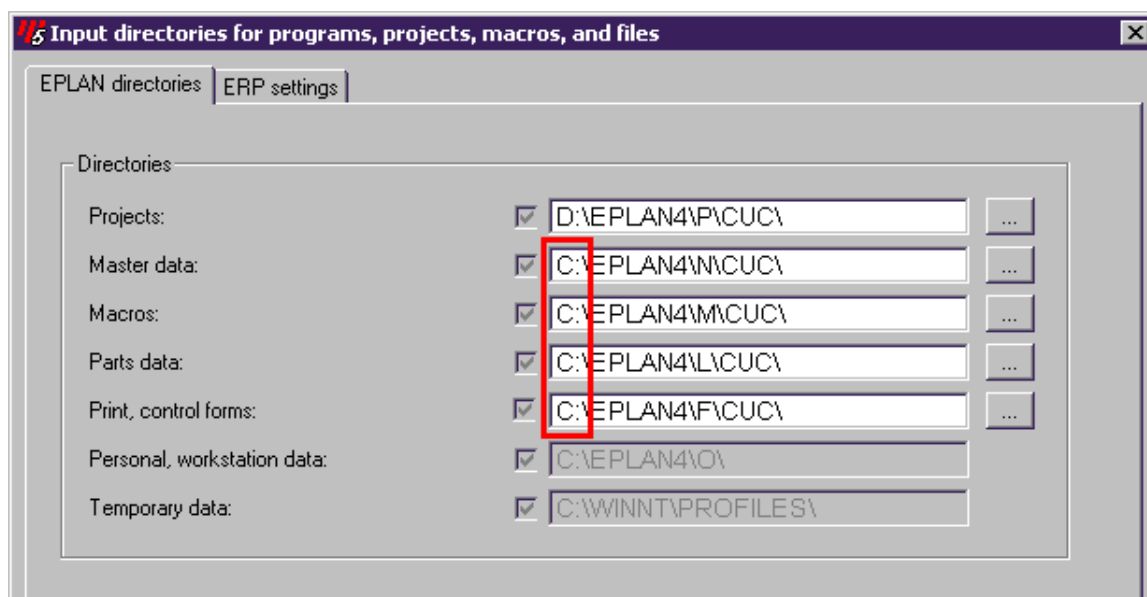
First, please notice that they are only active if the Use project-specific directories option has been selected, i.e. if the respective directory fields are not grayed out.

If you click on **[Current]**, you will use all drive letters specified for the EPLAN directories in the personal parameters (menu path **Utilities > Parameters > Personal > Directories**).

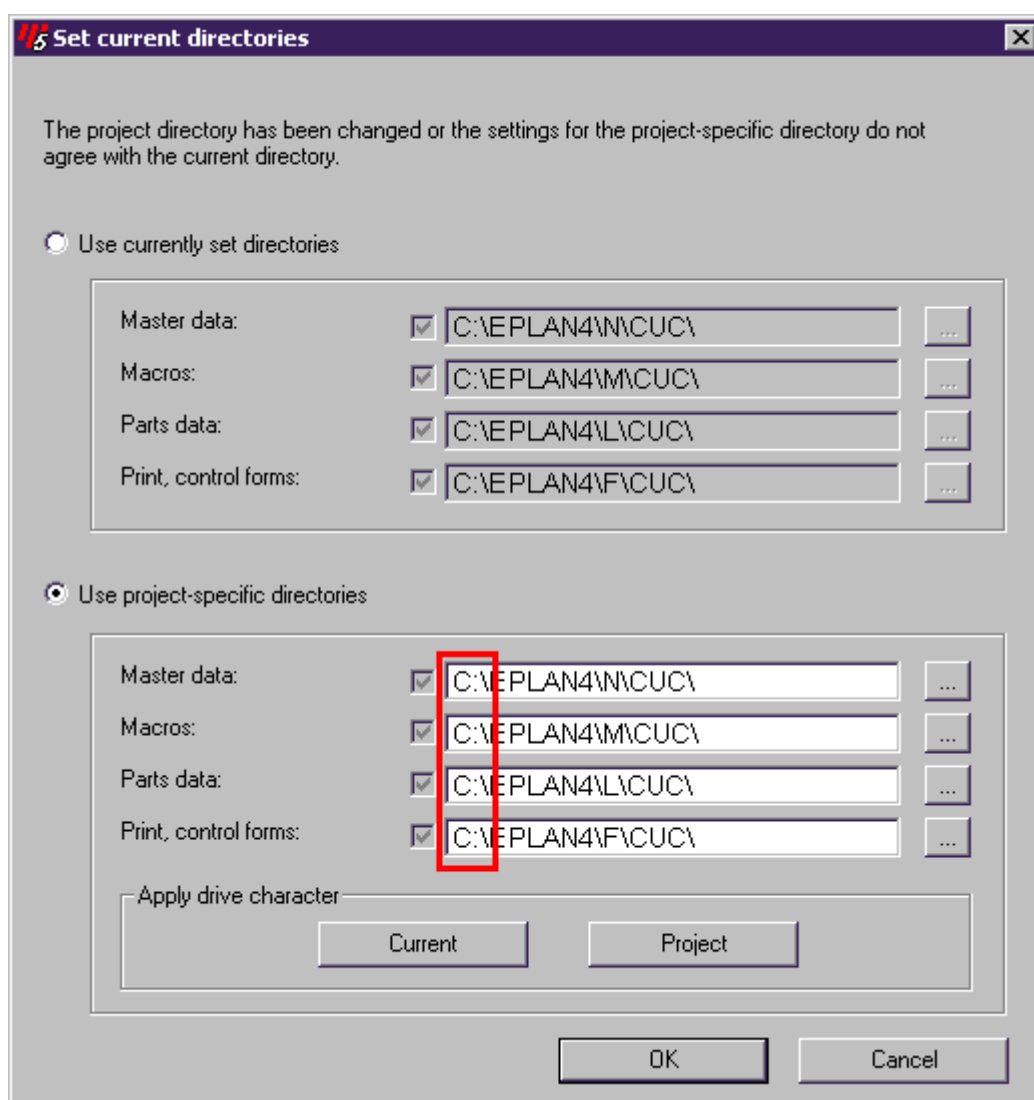
If you click on **[Project]**, you will use the drive letter of the currently selected project for all directories.



*You have selected a project which is located on the **D** drive. The EPLAN directories selected for the personal parameters indicate "C" as drive letter (see figure below).*

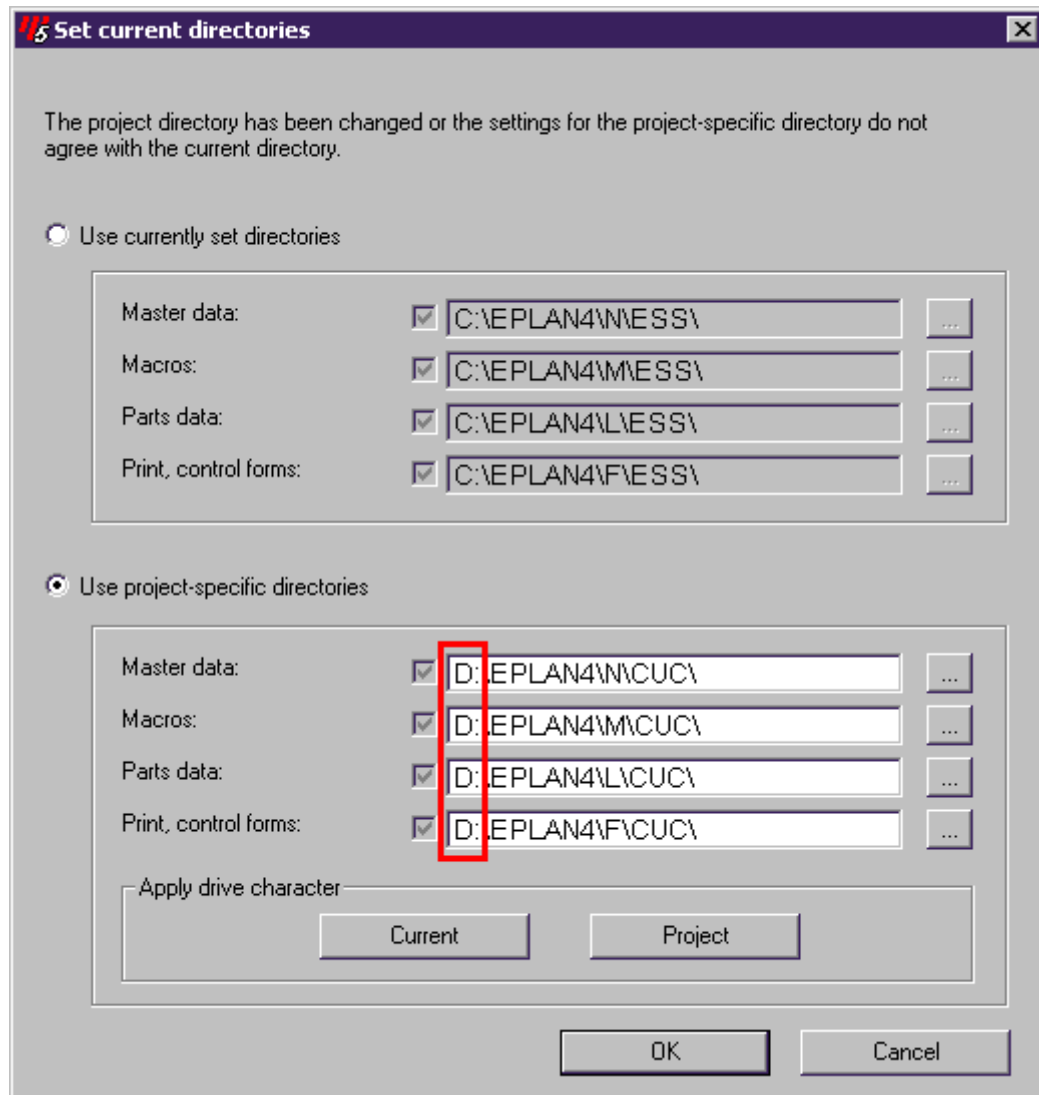


In the EPLAN main menu, select the menu path **Project > Select, create, copy projects** and click the **[Directories]** button. EPLAN will then open the Directory selection dialog box shown in the next figure.



The *Use project-specific directories* option is defaulted. Since "C" has been entered as drive letter for the EPLAN directories in the personal parameters, this drive letter is also shown in the directory fields from *Macros to Forms*.

If you now click the **[Project]** button, EPLAN will use the drive letter of the currently set project for all directories – in this case, this would be "D". This is illustrated by the following figure:



If you click the **[Current]** button afterwards, EPLAN will take the drive letter ("C") of the EPLAN directories from the personal parameters.

25 New and Modified Parameters

This chapter gives a list of all parameters from the parameter management which have been altered, added, or removed compared to EPLAN Version 5.40.

25.1 Project-specific Parameters

Graphics parameters (menu path: Project > Graphics)

Parameters	Modification
Graphics 1 tab:	
Line size	New parameter
Graphics 4 tab (was renamed; previously called: Graphics 5 tab):	
Transfer of DDs when copying pages	New parameter
Font size 1 tab (was renamed; previously called: Graphics 4 tab):	
PLC addresses	New font size parameter
Cable definition texts	New font size parameter
Wiring definition texts	New font size parameter
Graphics 5 tab (was renamed; previously called: Graphics 6 tab)	
Font size 2 tab	
Texts	New font size parameter
Function texts	New font size parameter
Terminal-/connector-strip designations	New font size parameter
Wire designations	New font size parameter

Graphics parameters (menu path: Project > Graphics)

Parameters	Modification
Font size 2 tab	
Cable conductor designations	New font size parameter
Hyperlink texts:	New font size parameter

Graphics output parameters (menu path: Project > Graphics output)

Parameters	Modification
Graphics forms tab	
Wiring lists	New parameter
Table of contents	New parameter
Graphics output tab	
In this tab, new scopes of application were defined with the following parameters:	
Scope of application "General"	
Number of blank pages	
Round up to page	
Scope of application "Terminal/interconnect diagrams"	
Create subpages	
Character for subpages	
Plot cable length	
Symbol to mark conductors	
Suppl. data in the "cable type" field	
Common higher-level assignment/location designation for all terminal diagrams	
Common higher-level assignment/location designation for all interconnect diagrams	New parameter
Break up macro	

Parameters	Modification
Scope of application "Table of contents"	
Create subpages	Shifted from the Table of contents parameter dialog box
Character for subpages	Shifted from the Table of contents parameter dialog box
Number of blank pages	
Round up to page	
Output method	
The following parameters were canceled due to the definition of new scopes of application:	
Page generation	
Spare page identifier	
Forms transfer identifier	
Position identifier with DIN	

Selection of the symbol files (menu path: Project > Symbol file):

Parameters	Modification
1 (Controls design) (single-pole)	New parameter for the single-pole display
2 (Hydraulics) (single-pole)	New parameter for the single-pole display
3 (IC technology) (single-pole)	New parameter for the single-pole display
4 (Electronics) (single-pole)	New parameter for the single-pole display
5 Reserved (single-pole)	New parameter for the single-pole display
6 Reserved (single-pole)	New parameter for the single-pole display
7 Reserved (single-pole)	New parameter for the single-pole display
8 Reserved (single-pole)	New parameter for the single-pole display

Parameters for foreign-language translations (menu path: Project > Foreign languages)

Parameters	Modification
Separator-oriented	New parameter for the line feed
Text-oriented	New parameter for the line feed

Parameters for the message management (menu path: Project > Message management / DD syntax check)

Parameters	Modification
Message management tab:	
Done messages	New parameter for the output of messages

25.2 Personal Parameters

Directory settings (menu path: Personal > Directories)

Parameters	Modification
EPLAN directories tab:	
Master data	Parameter was renamed
Parts data	Parameter was renamed
Print, control forms	Parameter was renamed
Personal, workstation data	Parameter was renamed
Temporary data	Parameter was renamed

Working parameters (menu path: Personal > Working parameters)

Parameters	Modification
Working parameters 1 tab:	
Show window number	Parameter was removed
User data (phone)	New parameter
User data (e-mail)	New parameter
Prompt for confirmation when quitting dialog boxes with ESC	Parameter was removed
Display page dialog box when starting the graphics	Parameter was removed

Working parameters (menu path: Personal > Working parameters):

Parameters	Modification
Working parameters 1 tab:	
Close the page overview after action	New parameter
Open additional window for jump function	New parameter

Parameters for the external programs (menu path: Personal > External programs)

Parameters	Modification
Display standard menu	Parameter was removed.
Display personal menu	Parameter was removed.
Display general menu	Parameter was removed.

Connection designations (menu path: Personal > Connection designations):

In EPLAN 5.50, the former tab of the working parameters is an independent dialog box.

Personal color parameters for the graphics screen (menu path: Personal > Colors)

Parameters	Modification
Hyperlinks	New parameter for the display of hyperlinks.
Variable line thickness	New parameter for the display of free graphics on graphics pages.

25.3 Workstation-specific Parameters

DXF / DWG parameter record (menu path: Workstation > DXF/DWG > Parameter record):

The parameter settings for the DXF import and export can now be saved as a parameter record.

Network parameters (menu path: Workstation > Network):

Parameters	Modification
Network operating system	Parameter was removed.
Workstation address	Parameter was removed.

Data backup parameters (menu path: Workstation > I/O parameters):

Parameters	Modification
I/O backup identifier	Parameter was removed.
Compress data	New parameter.
Use old backup format	New parameter.

26 Further Extensions, Modifications and Corrections

26.1 Graphics Editing

26.1.1 Editing Symbols in a Symbol Project

In a symbol project you can now also **directly import and export** symbols. Activating the symbol editor – for example by selecting the **Revise** menu item in the **Symbol** menu – is no longer required.

During the symbol export the familiar dialog box for the symbol selection is called up. The symbol to be exported is specified in this dialog box. If you export symbols with **angle variants**, you can use a subsequent dialog box to specify which of the symbol angle variants are to be exported.

You can now also **import macros** in the course of symbol editing in EPLAN. This way, you have the possibility to use a previously created graphics as symbol graphics for a newly created or revised symbol. It is possible to import window macros `*.mff` as well as combined macros `*.mzf` as graphics macros. The new option **Import macro** is now available in the **Symbol** menu for the macro import.

26.1.2 Search function

During the search for part numbers, the DD and the function text of the component are now also displayed in the Select search term dialog box.

26.2 Cross-reference Module

- The contact cross-reference, the interruption point cross-reference, and the wiring list are now started via one common menu item. To do this, select **Generate > Generate** in the cross-reference module. The generation runs are now initiated by activating check boxes in the subsequent Generate dialog box. Evaluations that have already been carried out are indicated in a separate field by a checkmark. In addition, you can now update the EPLAN text database and carry out a check of function texts of cross-referenced components.
- The **Extras** (in previous versions called: **Options**) menu item now allows you to launch the backup management from within the cross-reference and terminal/interconnect diagram modules.
- The symbol number is now also considered at coils managed by the contactor selection.
- Due to the new possibility to design schematics in single-pole display and the possibility to set various sheet formats, the contact and interruption point cross-reference function was revised.

This has the following effects:

If, prior to a generation run, EPLAN 5.50 detects a project that was created with a previous EPLAN version, the contactor data are written into the schematic. A new generation run reads the contact arrangement only from the schematic. If all contact can be sorted into the schematic, the contactor receives the status of being selected. This is indicated by a green contactor when the cross-references are displayed. If additional contacts are available, the contactor is displayed in red. Following the **Generate > Delete** menu path in the cross-reference module you can delete the contact arrangements in the schematic.

- In the case of interruption points designated according to DIN, you can now also rename the higher-level assignments and location designations in the dialog box for editing the interruption point cross-references.
- The following F codes and special texts are available for outputting the wiring list *.vdr and the wire bill of materials *.vst:

Designation of the data field	F code	Special text
Conductor number	F20	1400
Target 1	F21	1401
Target 2	F22	1402
Cross-section	F23	1403
Conductor color	F24	1404
Conductor length	F25	1405
Page/Path 1	F26	1406

Designation of the data field	F code	Special text
Page/Path 2	F27	1407
Page of the wiring definition	F28	1408
Text of the wiring definition	F29	1409
HLA/LOD in the page header	F50	
Mounting location 1	F51	1410
Mounting location 2	F52	1411
Function text 1	F59	1412
Function text 2	F60	1413
Number (wire bill of materials)	F91	
Conductor number list (wire bill of materials)	F92	
Cross-section dimension	F93	1414
Dimension length	F95	1415
Subsequent connection in next line	F98	1416
Internal potential number	F99	1417
Form alignment		1480
Number of lines		1481
Line spacing		1482
Character for external terminals	S1	1483

- The /SORT execution option used when printing the wiring list (#43) was extended by the identifier y for the piping list (#45). The parameter with the corresponding switches is indicated as /SORTxy, with the possibility to add several y in any order:

x=0/4:	Wire	(/4.../7: Presorting Mounting location
x=1/5:	Cross-section	
x=2/6:	Designation	
x=3/7:	Location designation	
y=E:	Electrical engineering	
y=H:	Hydraulics	
y=P:	Pneumatics	
y=S:	Lubrication	
y=K:	Cooling	

26.3 Terminal and Interconnect Diagram Module

- The dialog boxes for the processing of terminal and interconnect diagrams can now be resized.
- The dialog box for processing terminal and interconnect diagrams now offers new Delete buttons (X) as for the selection of output forms in the Strip or Cable data tabs. They are used to remove the selected form entries.
- Using key combinations you can now switch tabs in the Terminal diagram coordinates dialog box. The following combinations are available:



26.4 Translation Module

- If a sentence with line feeds is translated word by word, the last line feed is inserted if the sentence end behind only contains special characters.
- The additional fields and the Engraving data field in the symbol properties dialog box of devices can now be translated online.

26.5 Data Backup

- As from EPLAN Version 5.50, only the internal compression program is used for compressing projects.
- When reloading projects, an additional dialog box is called up once the project name has been specified. The Specify directories dialog box only appears for projects for which the "project-specific files" were included in the backup. Here, you can specify the directories not only manually, but also via a selection dialog box.
- When backing up projects with associated files, the `SPEZIAL.SSY` files is backed up as well, without being included in the contents file. The project can then be directly called up from the storage medium. The `SPEZIAL.SSY` file is not copied when being reloaded from associated files.

26.6 Terminal Line-up Diagram

In the graphics output dialog box of the terminal and interconnect diagram module (menu path **Output > Graphics**), the terminal line-up diagram is now output via a check box. The terminal line-up diagram contains the graphics display of all terminals located on a strip.

For this purpose, we have created new special texts:

Number	Text
713	Macro for mounting rail
714	Macro for terminal
715	Macro for separating panel
783	Combine multi-level terminals

The macros are used to display the associated graphics. If you set the special text 783 to "1", the multi-level terminals are drawn next to each other. In addition, the special text 706 (Jumper spacing) must be set to the width of the terminal graphic 714.

Always start with Level 1, followed by Level 2 etc... You may enter part numbers for Level 1 only, otherwise they are output on top of each other.

If the `KSARTIK` parts file does not exist in the parts directory or is empty, the `EPLC01?` parts file is used for the output of the type designations.

Furthermore, you can select part numbers for strip designations and terminals if the `KSARTIK` parts file exists.

26.7 Conversion of HPGL files

Using the menu path **Plot file > Scaling** of the Data exchange module "Conversion of HPGL files" (HPGL → EPLAN), you can now convert HPGL files to the page sizes A2 and A1.

26.8 Message Management

- New messages were added to the message management. These messages refer to the following generation runs:

Cable generation:

- During cable generation, the new warning messages W5213 "Higher-level assignment does not exist" and W5214 "Location designation does not exist" may be output.
- In addition, there is the new error message F5324 "Identical cable/target DD for terminals/non-terminals".

Wiring list:

The new warning message W4413 "Duplicated wire designation" may occur.

- Furthermore, the message database now saves the version number during every generation run. It can be displayed and printed with the execution data. For this purpose, however, the F29 field must be enlarged in the form.

26.9 Extension of the Form-controlled Output for the Printing of Interconnect Diagrams

When printing the cable types, you now have the possibility to output the conductors of cable type "3" in detail. For this purpose, the variable .Z1 must be set to "1" and the data records "..DA5" with "\F35" and "..DA6" must be added. For the record DA6, there are the following new F codes available:

- F41 Position
- F42 Color
- F43 Type
- F44 Index
- F45 Cross-section.

26.10 Outputting Lists

- The data for outputting bills-of materials purchase-order lists and device lists can now also be read out directly from the schematic in EPLAN. To this purpose the "Schematic" option in the output dialog boxes has to be activated. It is furthermore possible to create the lists by means of a so-called "Transfer file". This is a file which can be created and edited in EPLAN e.g. via the "Generate bills of materials" module.
- Before outputting the list it is possible to check logic pages with single-pole display and logic pages with multipole display for duplicate parts assignment by using the "Check" function.

26.11 New Viewer for HTML/DWF Documents

Instead of the "Whip!" program which you used in the past to view the HTML/DWF documents generated from EPLAN (**Export > HTML/DWF** menu path in the graphics editing function), we are now supplying the "VoloView Express" program of Autodesk®.

VoloView Express is started automatically as a plug-in for the Internet browser when viewing the pages generated under **Export > HTML/DWF**. The program is supported in an English version (vve201_setup.exe) and a German version (vve2setup_deu.exe).

Brief Description of the Installation of VoloView Express

Call up the file vve201_setup.exe or vve2setup_deu.exe from the directory ..\Service\Autodesk\VoloView Express on the installation CD and follow the instructions of the installation wizard. Here you can specify the directory in which VoloView Express is to be installed. After the installation this directory contains the file voloview.exe which is used to start the program as well as an English-language help file VoloView.chm.

26.12 Program Corrections

We have made the following program corrections for EPLAN 5.50:

- During the conductor selection of numbered cables, only the PE conductor is offered in addition to the number of conductors, provided that there is the same number of conductors with and without PE.
- The generation of jumpers for adjacent terminals has been changed. It is now carried out as shown in the following table:

Jumper between	Type of jumper
"normal" terminal / "normal" terminal	Jumper bar
"normal" terminal / PE terminal	Wire jumper
PE terminal / PE terminal	No jumper, target suppressed

- When the wiring list is printed, the F57 field no longer contains blanks.
- The message 7002 (Error in DRAW command) is no longer output for draw commands with a length exceeding 32k.
- The error-free transfer of wiring definitions via interruption points is now guaranteed.
- For the Evaluation method 3 parameter, a corresponding text now exists in the language DLL.
- A period is placed between page and path during the automatic assignment of device designations with page and path indication if the entry of the corresponding separator parameter contains any value.
- The selection of the symbol type for devices was corrected for the range of 151 to 154.

27 New Add-on Programs

In addition to the new EPLAN Version 5.50 we also offer two new add-on programs which are subject to licensing and with which the performance of EPLAN can be enhanced even further.



Please note that the functions mentioned here are part of the add-on modules described below. Only when the license for the respective module has been purchased can you carry out these functions.

27.1 SIMATIC HW Config

A new add-on module can be used to output the PLC-BUS data of your EPLAN project. The two new tabs "Configuration" and "Supplementary fields" are available for PLC boxes on PLC overview pages in order to configure the BUS data. Exporting itself is carried out in graphics editing via the menu path **Extras > BUS data export**. At present this contains only the option **SIMATIC HW Config** with which you can transfer BUS data to the SIEMENS configuration software of the same name.

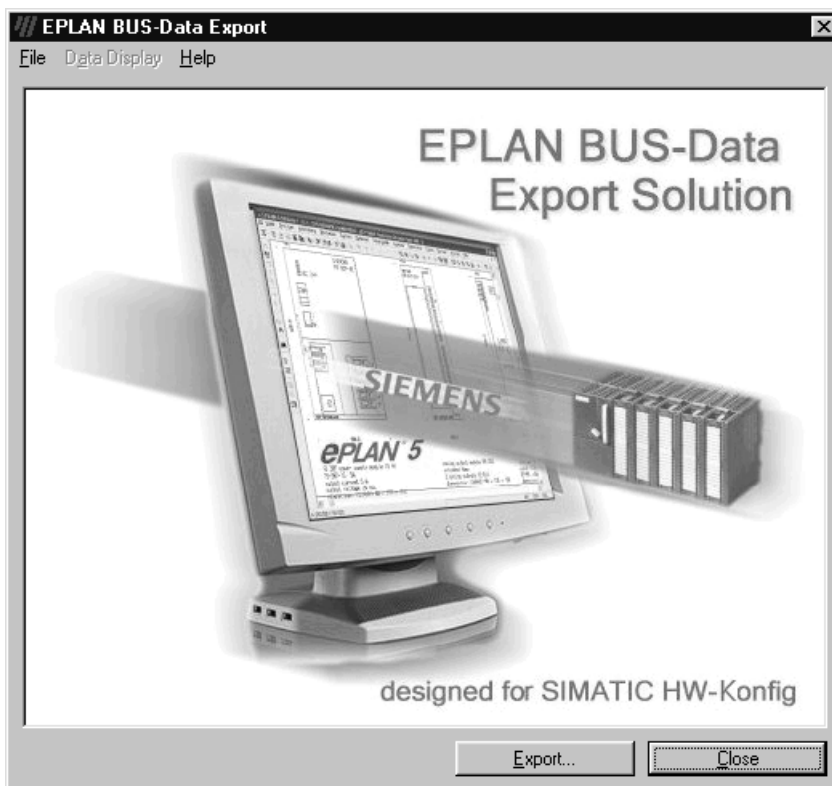


Fig. 82: EPLAN bus data export for SIMATIC HW Config

27.2 EPLAN Project Reference

EPLAN provides a further add-on module which makes functions for automatically checking your projects available. The new functions have been integrated in EPLAN at central program points:

- In the "Data backup" utility
- and in the "Project selection" function.

Add-on functions in the data backup function

Two of the new functions are included in the data backup utility in the **Extras** menu (see Fig. 83). You can use the **Parameter comparison** menu item to compare the parameter settings of your current project with those of another project. EPLAN writes the differences found into an "error list" which is stored in the current project directory.

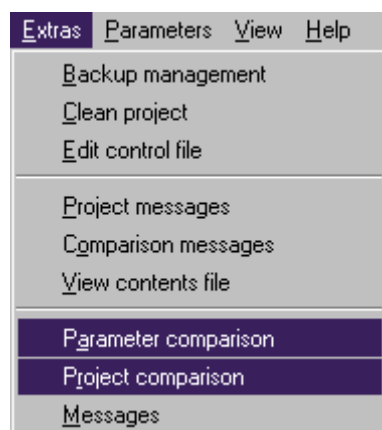


Fig. 83: Options of the "Project reference" add-on in the **Extras** menu

The **Project comparison** menu item is used to compare the project properties of different projects. The comparison of all the project properties is carried out on the basis of a standard project specified beforehand.

In addition you will be informed in the message management function if a parameter or project comparison detected differences between the projects. The corresponding dialog box is also accessed in the data backup function in the **Extras** menu via the **Messages** option.

Add-on functions in the project selection function

A further component of the project reference is the new **"Status" tab** contained in the project selection (see Fig. 84 on Page 177). This tab displays all the generation runs carried out for the project as well as further information. The "Generation date", "EPLAN Version", "Build No." of the version and the number of errors, warnings, and information which occurred at the respective run are listed for each run. This information is read directly from the message database.

General [1]	General [2]	Customer	End user	Editing	Status
Generation runs	Generated on	Version	Build	Messages	
Generate bill of materials	13.Dec.2001 12:01:11	5.40.3		0F/0W/0H	
Contact cross-reference r	23.May.2003 13:40:17	5.50.2	39900	111F/11W/	
Interruption point cross-re	23.May.2003 13:40:17	5.50.2	39900	2F/0W/0H	
Wiring and wire list	30.May.2003 15:21:05	5.50.2	40200	0F/0W/103	
PLC cross-references	27.Mar.2003 08:54:24	5.50.0	35600	0F/0W/0H	
Terminal generation	27.Mar.2003 08:54:24	5.50.0	35600	29F/1W/0H	
Generate cables	27.Mar.2003 08:54:24	5.50.0	35600	0F/0W/0H	
Graphic terminal overview	27.Mar.2003 08:54:25	5.50.0	35600	0F/0W/0H	
Graphic cable overview	27.Mar.2003 08:54:26	5.50.0	35600	0F/0W/0H	
Graphic terminals	27.Mar.2003 08:54:25	5.50.0	35600	0F/0W/0H	
Graphic cable	27.Mar.2003 08:54:26	5.50.0	35600	0F/0W/0H	
Graphic terminal connecti	27.Mar.2003 08:54:26	5.50.0	35600	0F/0W/0H	
Graphic bill of materials	13.Dec.2001 12:01:14	5.40.3		0F/0W/0H	
Graphics Wiring list	30.May.2003 15:53:25	5.50.2	40200	0F/0W/0H	
Project comparison	02.Jun.2003 11:40:38	5.50.2	40300	0F/0W/0H	

Fig. 84: New "Status" tab in the "Project selection" dialog box

In addition you can also use a function belonging to the add-on to have the user displayed who is currently working on the project currently selected in the project management function. To do so, select the **Display user** option in the pop-up menu of the "Project selection" dialog box (see Fig. 85).

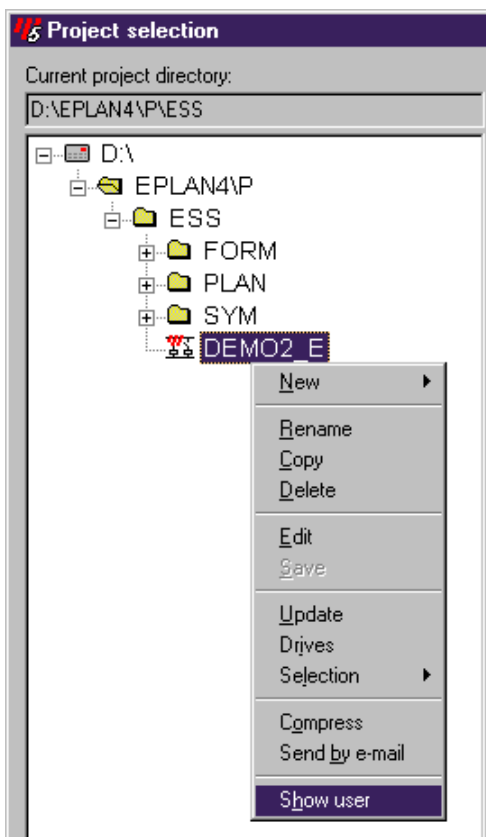


Fig. 85: New "Display user" option in the pop-up menu of the project selection function

The dialog box called up subsequently contains detailed information on the person editing the project. As in the information management module (see Chapter 19 from Page 137 on), you have to carry out corresponding entries in the user data of the working parameters in order to have the "Name", "Phone" and "E-mail" address displayed. Also similarly to the information management module you can also contact the other user by e-mail by using the **Send e-mail** menu item in the pop-up menu of the dialog box.

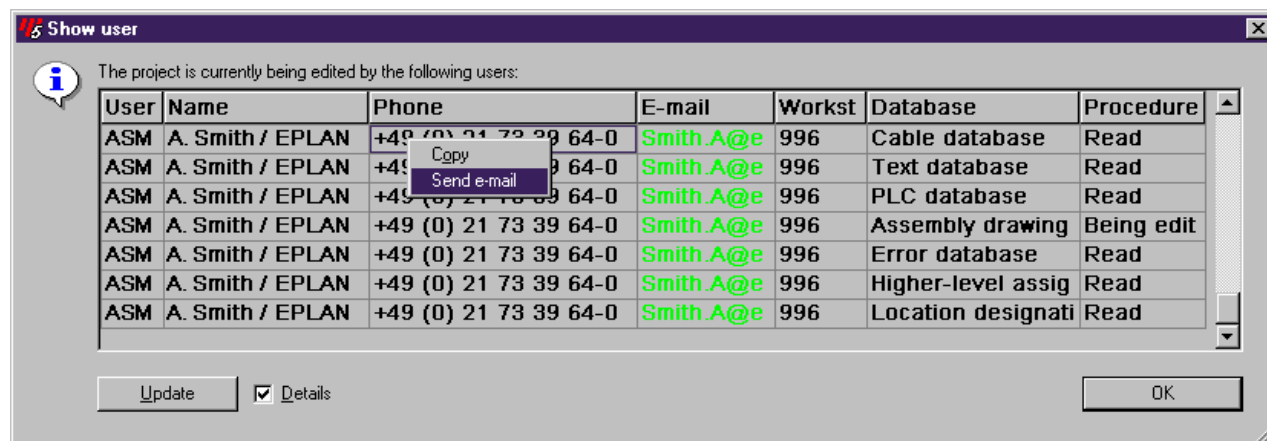


Fig. 86: "Display user" dialog box of the "project reference" add-on

In contrast to the information management module this is a "real" dialog box with additional information and not just a message which is displayed at a corresponding action. The additional information displayed in the "Display user" dialog box includes the "database" the respective user is accessing and the "actions" currently being carried out.

28 New and Modified Master Data

The following master data were either added to the scope of delivery of EPLAN Version 5.50 or were modified:

Name	Path specification	Description
PFADPOS.DAT	\EPLAN4\N	Data definition file; was extended by a path subdivision in A1 format
SINGLE_E.SYM	\EPLAN4\N\<Customer Code>	Symbol file for single-pole display
SINGLE_E.P	\EPLAN4\P\<Customer Code>\SYM	Symbol project; contains the symbols of the symbol file for single-pole display
DIC_WUPE.SYM	\EPLAN4\N\<Customer Code>	Revised symbol file; new symbols added
DIC_WUPE.P	\EPLAN4\P\<Customer Code>\SYM	Revised symbol project, contains the changes made in the symbol file
DIC_ESSD.SYM	\EPLAN4\N\<Customer Code>	Symbol file with identifiers; based on DIN EN61346-2
DIC_ESSD.P	\EPLAN4\P\<Customer Code>\SYM	Symbol project; contains the symbols of the DIC_ESSD symbol file
KABEL.KLB	\EPLAN4\N\<Customer Code>	Revised cable type file
N5154BMK.FRM	\EPLAN4\F\<Customer Code>\EPLAN\E	Transfer form for labeling components (devices)
N5154VDR.FRM	\EPLAN4\F\<Customer Code>\EPLAN\E	Transfer form for labeling components (wirings)
N5154KBL.FRM	\EPLAN4\F\<Customer Code>\EPLAN\E	Transfer form for labeling components (cables)
N5154ADR.FRM	\EPLAN4\F\<Customer Code>\EPLAN\E	Transfer form for labeling components (conductors)
N5154KLM.FRM	\EPLAN4\F\<Customer Code>\EPLAN\E	Transfer form for labeling components (terminals)
STDADR.KAB	\EPLAN4\F\<Customer Code>\EPLAN\E	Revised print form for outputting the added identifiers
STDKAB.KAB	\EPLAN4\F\<Customer Code>\EPLAN\E	Revised print form for outputting the added identifiers
STANDARD.VDR	\EPLAN4\F\<Customer Code>\EPLAN\E	Revised print form for outputting the

Name	Path specification	Description
		added identifiers

28.1 Designing in A1 Format

For a display in A1 format, the data definition of path positions was extended to include this drawing size. In this context, the `PFADPOS.DAT` data definition file included in the EPLAN standard scope of delivery was adapted to this.

In the EPLAN master data, a sample plot frame is additionally available for each ISO format.

28.2 New Symbol Files

28.2.1 Single-pole Display

This EPLAN version also offers a symbol file allowing you to design schematics in single-pole display. The `SINGLE_E.SYM` symbol file contains the same symbols as the `DIC_WUPE.SYM`.

28.2.2 DIN EN61346-2

To comply with DIN EN 61346-2, we have created the new `DIC_ESSD.SYM` symbol file, which assigns the identifiers to the symbols in accordance with the categorizations.

28.3 Extended DIC_WUPD.SYM Symbol File

New symbols were added to the standard `DIC_WUPD.SYM` symbol file, and the associated `DIC_WUPD.P` symbol project has been revised completely.

28.4 Expansion of the Standard Labeling

The following transfer forms are available for the new component labeling:

- Devices N5154bmk.frm
- Wiring N5154vdr.frm
- Cables N5154kbl.frm
- Conductor N5154adr.frm
- Terminals N5154klm.frm.

The following print forms were adjusted correspondingly, to enable you to print the new format elements:

- Stdadr.kab
- Stdkab.kab
- Standard.vdr.

28.5 Extended Cable Types

In addition, current cable types were added to the cable type file. These include, among others, the most common cable types of the ÖLFLEX-SERVO® and ÖLFLEX-CLASSIC® product range.



EPLAN Software & Service GmbH & Co. KG
An der alten Ziegelei 2
40789 Monheim am Rhein Germany
Phone: +49 (0) 21 73 / 39 64-0
Fax: +49 (0) 21 73 / 39 64-25
mailto:Info@eplan.de · <http://www.eplan.de>