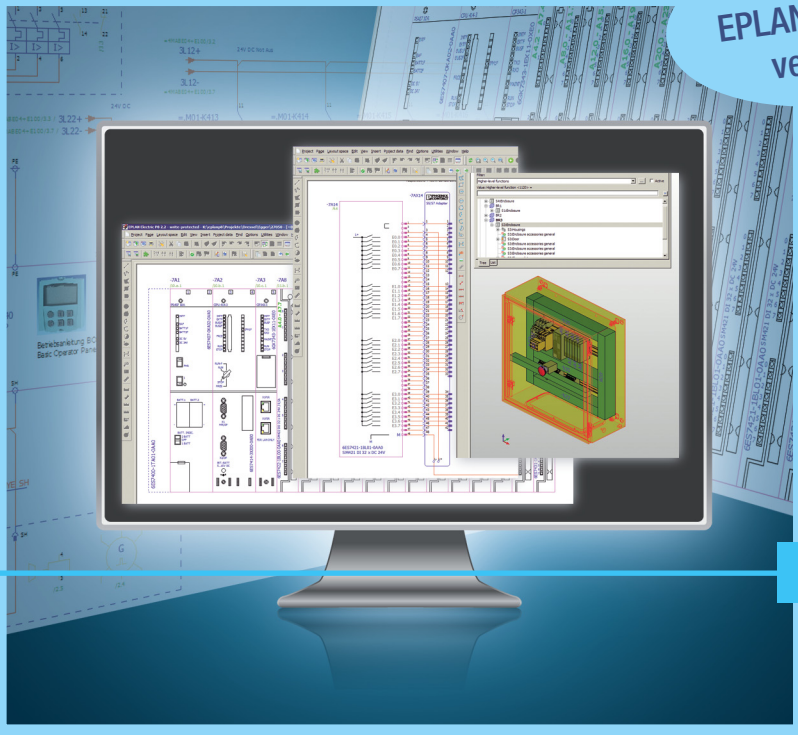
Bernd Gischel

EPLAN Electric P8
Reference Handbook

Based on EPLAN Electric P8 version 2.2



3rd edition, revised

HANSER

Bernd Gischel

EPLAN Electric P8 Reference Handbook

Bernd Gischel

EPLAN Electric P8 Reference Handbook

3rd edition, revised

HANSER

The Author: Dipl.-Ing. Bernd Gischel, Lünen, Germany
Translated by Lindsey Translations, Deisenhofen, Germany

Distributed in North and South America by Hanser Publications
6915 Valley Avenue, Cincinnati, Ohio 45244-3029, USA
Fax: (513) 527-8801
Phone: (513) 527-8977
www.hanserpublications.com

Distributed in all other countries by Carl Hanser Verlag
Postfach 86 04 20, 81631 Munich, Germany
Fax: +49 (89) 98 48 09
www.hanser-fachbuch.de

The use of general descriptive names, trademarks, etc., in this publication, even if the former are not especially identified, is not to be taken as a sign that such names, as understood by the Trade Marks and Merchandise Marks Act, may accordingly be used freely by anyone.

While the advice and information in this book are believed to be true and accurate at the date of going to press, neither the authors nor the editors nor the publisher can accept any legal responsibility for any errors or omissions that may be made. The publisher makes no warranty, express or implied, with respect to the material contained herein.

Cataloging-in-Publication Data is on file with the Library of Congress.

Bibliografische Information der deutschen Bibliothek:
Die Deutsche Bibliothek verzeichnet diese Publikation in der Deutschen Nationalbibliografie; detaillierte bibliografische Daten sind im Internet über <http://dnb.d-nb.de> abrufbar.

All rights reserved. No part of this book may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying or by any information storage and retrieval system, without permission in writing from the publisher.

© Carl Hanser Verlag, Munich 2013
Production Management: Andrea Stolz
Coverconcept: Marc Müller-Bremer, www.rebranding.de, Munich
Coverdesign: Stephan Rönigk
Typeset, printed and bound by Kösel, Krugzell
Printed in Germany
ISBN 978-1-56990-476-3
E-Book ISBN 978-1-56990-477-0

Contents

Foreword XIII

1 Installing EPLAN Electric P8 1

1.1 Hardware 1
1.2 Installation 2
1.3 Note for users of previous versions 10
1.3.1 Parallel operation with previous versions 10

2 The basics of the system 11

2.1 Five principles for working with EPLAN Electric P8 11
2.2 Directory structure, storage locations 13
2.3 Settings – General 15
2.3.1 Settings – Project 16
2.3.2 Settings – User 28
2.3.3 Settings – Station 35
2.3.4 Settings – Company 35
2.4 EPLAN and multiple starts? 37
2.5 Properties 37
2.5.1 Project properties 38
2.5.2 Page properties 40
2.5.3 Symbol properties (components) 41
2.5.4 Form properties 41
2.5.5 Plot frame properties 42
2.6 Buttons and popup menus 42
2.6.1 Device dialog buttons 42
2.6.2 Buttons in dialogs (configuring) 43
2.6.3 Buttons in dialogs such as filter schemes 44
2.6.4 Small blue triangular button 44
2.6.5 Property arrangements (components) 45

2.6.6 Format properties ... 46
2.6.7 Buttons (small black triangles) ... 47
2.6.8 Dialogs for schemes ... 48
2.7 Master data ... 48
2.8 Operation ... 49
2.8.1 Using the keyboard ... 50
2.8.2 Using the mouse ... 51
2.9 The user interface – more useful information ... 51
2.9.1 Using workspaces ... 51
2.9.2 Dialog display ... 52

3 Projects ... 57
3.1 Project types ... 58
3.1.1 Project types in EPLAN ... 59
3.1.2 Project templates and basic projects ... 60
3.2 Creating a new project ... 62
3.2.1 New project (from a basic project) ... 63
3.2.2 New project (with a project creation wizard) ... 65

4 The graphical editor (GED) ... 75
4.1 Page navigator ... 75
4.1.1 Page types ... 77
4.1.2 The popup menu in the page navigator ... 78
4.1.3 Page navigator filter ... 94
4.2 General functions ... 96
4.2.1 The title bar ... 97
4.2.2 The status bar ... 97
4.3 Coordinate systems ... 98
4.3.1 Graphical coordinate system ... 98
4.3.2 Logical coordinate system ... 99
4.4 Grid ... 99
4.5 Increments, coordinate input ... 100
4.5.1 Increment ... 100
4.5.2 Coordinate input ... 101
4.5.3 Relative coordinate input ... 102
4.5.4 Move base point ... 103
4.6 Graphical editing functions ... 104
4.6.1 Graphical objects – lines, circles, rectangles ... 104
4.6.2 Trim, chamfer, stretch and more 106
4.6.3 Group and ungroup ... 111

4.6.4 Copy, move, delete, ... 113
4.6.5 Dimensioning 113
4.7 Texts 116
4.7.1 Normal (free) texts 117
4.7.2 Path function texts 118
4.7.3 Special texts 119
4.7.4 The Properties – Text dialog 121
4.8 Components (symbols) 126
4.8.1 Insert components (symbols) 126
4.8.2 Properties (components) dialog – [device] tab 127
4.8.3 The Display tab 129
4.8.4 The Symbol/function data tab 131
4.8.5 The Parts tab 134
4.9 Cross-references 135
4.9.1 Contact image on component 135
4.9.2 Contact image in path 138
4.9.3 Special feature: Pair cross-reference 139
4.9.4 Device selection settings 142
4.10 Macros 144
4.10.1 Types of macros 144
4.10.2 Macros with value sets 149

5 Navigators 157
5.1 Overview of the most important navigators 158
5.1.1 Additional navigators and modules 162
5.2 Device navigator 164
5.2.1 Device navigator/Assign main function 166
5.2.2 Device navigator/Cross-reference DTs 168
5.2.3 Device navigator/Synchronize function texts 169
5.2.4 Device navigator/Interconnect devices 170
5.2.5 Device navigator/Number (offline) 177
5.2.6 Device navigator/New ... 181
5.2.7 Device navigator/New device ... 182
5.2.8 Device navigator/Place/[Representation type] 183
5.3 Terminal strip navigator 185
5.3.1 Terminal strip navigator/Edit ... 185
5.3.2 Terminal strip navigator/Correct 188
5.3.3 Terminal strip navigator/Number terminals 189
5.3.4 Terminal strip navigator/New ... 193
5.3.5 Terminal strip navigator/New functions ... 193
5.3.6 Terminal strip navigator/New terminals (devices) ... 195
5.3.7 Terminal strip navigator/Place/[Representation type] 197

5.4 Plug navigator (Socket navigator) 197
5.4.1 Plug navigator/Edit 197
5.4.2 Plug navigator/Correct 198
5.4.3 Plug navigator/Number pins 199
5.4.4 Plug navigator/New 199
5.4.5 Plug navigator/New functions 200
5.4.6 Plug navigator/Generate plug definition 200
5.4.7 Plug navigator/Generate pin 201
5.4.8 Plug navigator/Place/[Representation type] 201
5.5 Cable navigator 202
5.5.1 Cable navigator/Edit 203
5.5.2 Cable navigator/Number 204
5.5.3 Cable navigator/Generate cables automatically 207
5.5.4 Cable navigator/Automatic cable selection 209
5.5.5 Cable navigator/Assign conductors 211
5.5.6 Cable navigator/New 213
5.5.7 Cable navigator/Place/[Representation type] 213
5.5.8 Cable navigator/Number cable DT 215
5.5.9 Cable navigator/Number DT 215
5.6 PLC navigator 217
5.6.1 PLC/Write back connection point designations 219
5.6.2 PLC/Set data types 221
5.6.3 PLC/Addresses/assignment lists 221
5.6.4 PLC navigator/Address 223
5.6.5 PLC navigator/New 225
5.6.6 PLC navigator/New functions 225
5.6.7 PLC navigator/New device 227
5.6.8 PLC navigator/View 227
5.7 Part navigator/(Devices/Parts) 228
5.7.1 Popup menu functions 230
5.8 Assign function (navigators) 235
5.9 Navigators/Quick input filter 237

6 Reports 239

6.1 What are reports? 240
6.2 Report types 241
6.3 Types of graphical reports 241
6.3.1 Report types (forms) 241
6.3.2 Special connection diagrams 256
6.4 Settings (output options) 258
6.4.1 The Display/output project setting 258
6.4.2 The Parts project setting 260
6.4.3 The Output to pages project setting 261

6.5 Generate reports 267
6.5.1 The Reports dialog 268
6.5.2 Generate reports without templates 270
6.5.3 Popup menus in the Reports tab 279
6.5.4 Generate reports with templates 280
6.6 Other functions 286
6.6.1 Generate project reports 287
6.6.2 Update 287
6.6.3 Generate report project 287
6.6.4 Settings for automatic updates 287
6.7 Labeling 288
6.7.1 Settings 289
6.8 Edit properties externally 295
6.8.1 Export data 297
6.8.2 Importing 303

7 Management tasks in EPLAN 307
7.1 Structure identifier management 307
7.1.1 Tabs in structure identifier management 311
7.1.2 Graphical buttons 313
7.1.3 Sort menu 315
7.1.4 Extras menu 316
7.2 Message management 318
7.2.1 The visual appearance of message management 319
7.2.2 Project checks 320
7.2.3 Message classes and message categories 320
7.2.4 Filters in message management 326
7.2.5 Various message edition options 328
7.3 Layer management 330
7.3.1 Standard layers 330
7.3.2 Export and import of layers 331
7.3.3 Create and delete your own layers 332
7.3.4 Uses of layers 333
7.4 Parts management 336
7.4.1 Structure of parts management 338
7.4.2 Tabs in parts management 339
7.5 Revision control 346
7.5.1 General 347
7.5.2 Generate new revision 347
7.5.3 Execute changes 349
7.5.4 Complete page(s) 352
7.5.5 Generate reports 353
7.5.6 Complete a project 354

8 Export, import, print 357
8.1 Export and import of DXF/DWG files 358
8.1.1 Exporting DXF and DWG files 358
8.1.2 Import of DXF and DWG files 362
8.2 Image files 365
8.2.1 Exporting image files 366
8.2.2 Insert image files (import) 368
8.3 Print 371
8.3.1 The Print dialog and its options 372
8.3.2 Important export/print setting 374
8.4 Export and import of projects 375
8.4.1 Exporting projects 375
8.4.2 Importing projects 375
8.5 Print attached documents 376
8.6 Import PDF comments 377
8.6.1 Importing commented PDF documents 377
8.6.2 Deleting PDF comments 380
8.7 Generate PDF documents 382
8.7.1 Export of PDF files 383

9 Data backup 391
9.1 Zipping and unzipping of projects 392
9.1.1 Zip projects 392
9.1.2 Unzip projects 393
9.2 Backing up and restoring projects 393
9.2.1 Back up projects 394
9.2.2 Restore projects 400
9.3 Other important settings 402
9.3.1 Default settings for project backup (global user setting) 402
9.3.2 Compress project (remove unnecessary data) 403
9.3.3 Automated processing of a project 405
9.4 Backing up and restoring master data 408
9.4.1 Backing up master data 409
9.4.2 Restoring master data 411
9.5 Send project by e-mail directly 413

10 Master data editors 415
10.1 Preparatory measures 418
10.2 Overview of forms 418
10.2.1 First option – manual overview 418
10.2.2 Second option – automatic overview 420
10.3 Forms 422
10.3.1 Create new form (from copy) 425
10.3.2 Edit existing form 431
10.3.3 Create new form 432
10.4 Plot frames 433
10.4.1 Create new plot frame (from copy) 434
10.4.2 Edit existing plot frame 436
10.4.3 Create new plot frame 437

11 Old EPLAN data (EPLAN 5 . . .) 439
11.1 Import options 440

12 Extensions 443
12.1 EPLAN Pro Panel 443
12.1.1 What can EPLAN Pro Panel do? 443
12.1.2 General 444
12.1.3 Toolbars in EPLAN Pro Panel 446
12.1.4 Menus in EPLAN Pro Panel 458
12.1.5 Navigators in EPLAN Pro Panel 472
12.1.6 Settings for EPLAN Pro Panel 477
12.1.7 A practical example with EPLAN Pro Panel 479
12.2 EPLAN Data Portal 493
12.2.1 What are the advantages of the EPLAN Data Portal? 493
12.2.2 Before the first start 493
12.2.3 How the EPLAN Data Portal works 496
12.3 Project options 500
12.3.1 What are project options? 501
12.3.2 Terminology in the Project options module 501
12.3.3 Creating options and sections 502
12.3.4 Generate options overview report 508

13 FAQs 509
13.1 General 509
13.2 Parts 536
13.3 Terminals, plugs 541
13.4 Cables 545
13.5 PLC 548
13.6 Properties, layers 549
13.6.1 Master data 553
13.7 Data exchange 560
13.8 Reports 561

14 Creating a schematic project – step by step 565
14.1 Create a project 565
14.2 Generate pages 569
14.3 Create a schematic 572
14.3.1 Insert symbols 573
14.3.2 Insert cables 578
14.4 Generate reports 583
14.4.1 Generating other report pages 586

Index 587

GENERAL ICONS IN THE NAVIGATORS

	Multiple projects are open
	Devices without DT, or devices
	Project structure level (view independent of standards)
	Project structure level (preceding sign view)
	Project structure level (user-defined structure)
	Prefix / identifier; prefix

UNPLACED OBJECTS IN THE NAVIGATORS

	Main function, »multi-line« representation type
	Main function, »single-line« representation type
	Auxiliary function, »multi-line« representation type
	Auxiliary function, »single-line« representation type
	Overlaid main function
	Overlaid function
	Function relevant to safety
	Protected main function
	Function template
	Black box, PLC box (main function)
	Overlaid black box, PLC box (main function)
	Black box, PLC box (auxiliary function)
	Shield (main function)
	Shield (auxiliary function)
	Cable definition (main function)
	Cable definition (auxiliary function)
	Plug definition (main function)
	Overlaid plug definition (main function)
	Plug definition (auxiliary function)
	Terminal strip definition (main function)
	Overlaid terminal strip definition (main function)
	Terminal strip definition (auxiliary function)
	Channel (PLC)
	Channel (PLC) with connection point relevant to safety
	Part
	Isolated wire termination

© Hanser 2013: Bernd Gischel, EPLAN Electric P8 Reference Handbook, 3rd Edition;
ISBN: 978-1-56990-476-3

PLACED OBJECTS IN THE NAVIGATORS

	Main function, »multi-line« representation type
	Main function, »single-line« representation type
	Auxiliary function, »multi-line« representation type
	Auxiliary function, »single-line« representation type
	Auxiliary function, »pair cross-reference« representation type
	Auxiliary function, »overview« representation type
	Main function, »overview« representation type
	Overlaid main function
	Overlaid function
	Function relevant to safety
	Protected main function
	Black box, PLC box (main function)
	Overlaid black box, PLC box (main function)
	Black box, PLC box (auxiliary function)
	Shield (main function)
	Shield (auxiliary function)
	Cable definition (main function)
	Cable definition (auxiliary function)
	Plug definition (main function)
	Overlaid plug definition (main function)
	Plug definition (auxiliary function)
	Terminal strip definition (main function)
	Overlaid terminal strip definition (main function)
	Terminal strip definition (auxiliary function)
	Channel (PLC)
	Channel (PLC) with connection point relevant to safety
	Net
	Net definition point
	Parts placement, »panel layout« or »detailed panel layout« representation type
	Isolated wire termination
	Function contains inconsistent or incomplete data. After checking the project data, incorrect functions are identified by exclamation marks in the navigators.
	Layout space EPLAN Pro Panel X axis - Red; Y axis - Green; Z axis - Blue

Foreword

Dear Users,

EPLAN Electric P8 is CAE software that is constantly being further developed. It offers innumerable project editing options and provides new innovations with each new version.

The current version 2.2 is the result of continual development of previous versions. This version once again incorporates a wide range of user requirements and requests that have arisen during the practical use of EPLAN.

This fourth edition of this book has been revised and expanded based on Version 2.2 to demonstrate the wide range of functions in EPLAN Electric P8. The book is meant to make it easier to start using the software and to smoothly guide you around initial hiccups when working with EPLAN Electric P8. Numerous practical examples show you what is possible with Version 2.2.

Of course, like its predecessors, this edition cannot and will not describe all of the software's functions or provide examples for every conceivable function. EPLAN Electric P8 becomes increasingly comprehensive with every new version, and it offers a variety of functions that cannot be completely covered in a single book. A book that describes all the functions would have thousands of pages and be impractical for the reader.

In Version 2.2, there are also many ways to reach the same goal. I will present and discuss some solutions. Others you will discover yourself and ask yourself why no one has ever tried it this or that way before.

This book will recommend solution approaches and demonstrate solutions that will help simplify your everyday work. It will help you make necessary decisions.

The book is addressed to everyone who uses EPLAN Electric P8 for electrical engineering designs – both daily and sporadic EPLAN Electric P8 users as well as engineers, electrical engineers, pupils and students.

I would like to express my thanks to Julia Stepp and her team at the Carl Hanser Verlag for the opportunity to write and publish this book. I would also like to sincerely thank my family, especially my wife Susanne. They have always been, and continue to be, very patient with me.

I would also like to thank all of the readers who have made this book a success. All feedback, whether criticism or praise, has always been a strong motivator for me to revise this book.

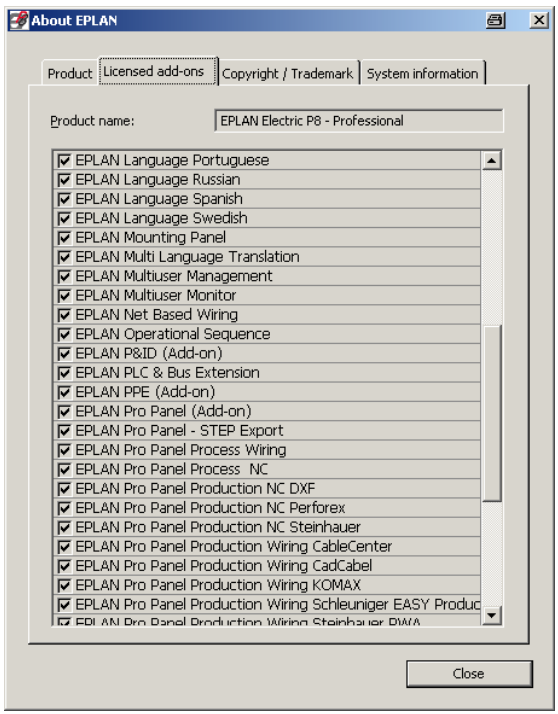
And finally, I would again like to thank EPLAN Software & Service GmbH & Co. KG for their consistent and very friendly support and collaboration in compiling some of the information for this edition of the EPLAN Electric P8 Reference Handbook.

Important notes

All of the examples and explanations assume local installation and local operation of EPLAN. Furthermore, the book assumes that the user has all of the user rights in EPLAN and is logged in as the local administrator.

It is possible that, depending on the user's license and module package, certain functionality or a certain function described in the book will not be available or executable in the way in which it is explained and illustrated. Therefore, you should always check to see which licensed add-ons you have (via **HELP/ABOUT/LICENSED ADD-ONS TAB**).

For this book, EPLAN Electric P8 Professional Edition 2.2.5 Build 6338 was used with the licensed add-ons shown in the figure.



Help/Info/Licensed add-ons tab (excerpt)

NOTE FOR USERS OF PREVIOUS VERSIONS: Certain parts of the functions described here may exist in EPLAN Electric P8 Versions 1.7.x, 1.8.x, 1.9.x, 2.0.x and 2.1.x, but their use, settings and range of functionality may differ from Version 2.2.

The examples used in the book are available as an EPLAN Electric P8 project at *www.eplan-efficient-engineering.com/handbook*. The example project can also be downloaded in the EPLAN Electric P8 Education Version format.

Some of the settings used in this book, such as those for filters or schemes, differ from the standard EPLAN installation. All of this additional data is available in the sample data. In addition, some custom, non-standard shortcut keys were also used.

The following text boxes are used to visually highlight notes, tips, etc.

NOTE: This box contains important notes that should be observed when using EPLAN Electric P8.

TIP: This box contains helpful tips for everyday working with EPLAN Electric P8.

TO DO: This box gives you practical exercises to let you try out what you have learned.

This box provides additional information and tips.

Whenever this symbol appears in the book's margin, you will find questions and answers to problems that occurred during actual use of EPLAN Electric P8.

1 Installing EPLAN Electric P8

Since installation requires few steps and can only be performed by the system administrator, this chapter provides only a basic description of this process. EPLAN is usually already installed on the workstation.

Installation of EPLAN generally requires administrator rights. The system administrator also designates at least one EPLAN administrator who will later manage the EPLAN users (also known as rights management). If rights management is not used, then EPLAN can be started by all users without requiring passwords, etc.

User management (an add-on that must be purchased separately and is not always included with every license) is not described in this book. Brief general information is provided as necessary at the appropriate points.

1.1 Hardware

For Version 2.x, EPLAN recommends a CPU comparable to a 3-GHz Intel Pentium D or a 2.4-GHz Intel Core 2 Duo with at least 4 GB of RAM.

EPLAN has no special requirements for the graphics card or other hardware components. A standard computer as used for Office applications, for example, is sufficient. Even for the graphic card, the more available memory, the smoother EPLAN runs. Certain add-ons, such as EPLAN Pro Panel and its extensions, have other hardware requirements that affect the graphics card and its drivers.

I feel that a single-screen solution can no longer be recommended for EPLAN due to the many additional modular dialogs that can be displayed, such as the various navigators. A two-screen solution is clearly preferable, and a three-screen solution with each screen having a resolution of at least 1680 × 1050 pixels is ideal.

Of course, EPLAN still functions with just one screen. But this screen should have a resolution of at least 1280 × 1024 pixels.

1.2 Installation

As far as installation is concerned, EPLAN is a normal Windows program. Apart from a few entries, the new EPLAN Setup Manager (as of Version 2.1.x) performs most of the work. There are only a few entries in the Windows registry, which is commendable and not always the case today.

Installation is usually started using the installation CD. The *Setup.exe* file in the root directory of the CD is run to begin installation. Installation after downloading the installation package from the EPLAN homepage works the same way (the downloaded ZIP file unpacks the installation data into the same directories that would be on the installation DVD).

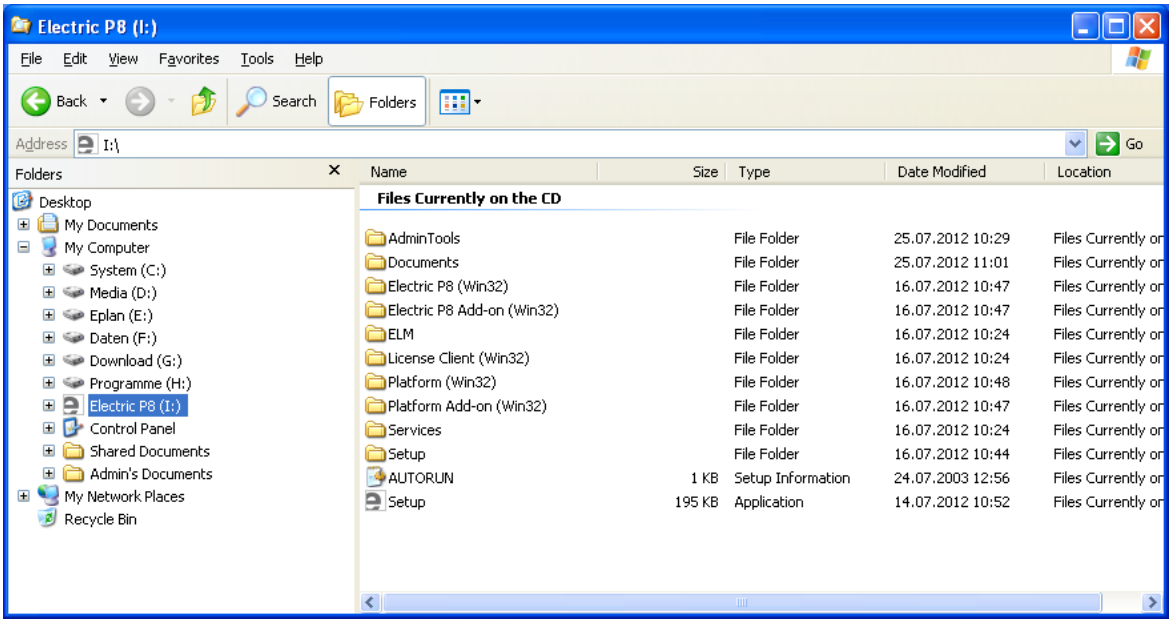


Fig. 1.1 Installation directory

NOTE: Running EPLAN Electric P8 requires a specific version of .NET Framework. If .NET Framework is not installed, or is not installed in the required version, it must be installed before you can proceed with the installation of EPLAN.

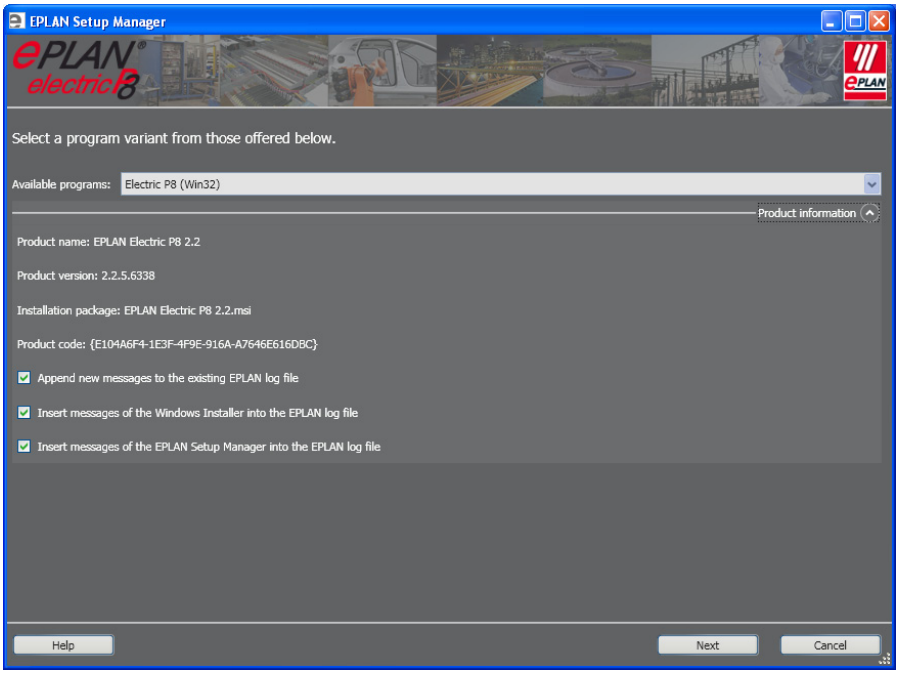


Fig. 1.2
The EPLAN Setup Manager's basic settings

When the **NEXT** button is clicked, the license agreement dialog is displayed. This must be accepted in order to use the EPLAN Setup Manager.

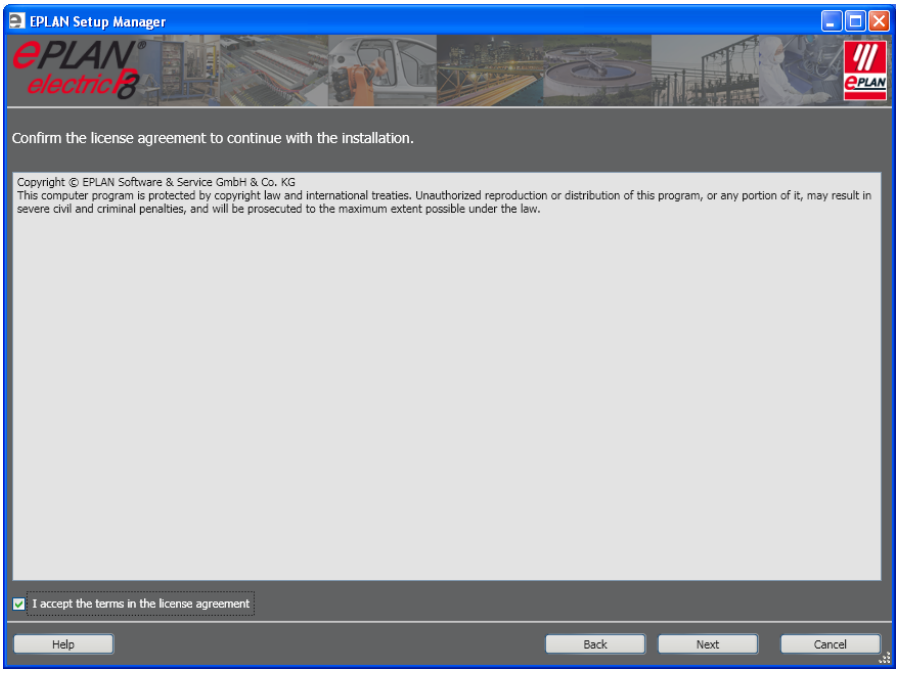


Fig. 1.3
Licensing agreement dialog

After the dialogs showing the EPLAN Setup Manager's basic settings and the available programs, the first installation dialog appears.

When the **NEXT** button is clicked, the **Target directories, settings** dialog is displayed. This is where you set the *program directory*, the *system master data directory*, the *company code*, and the directories for *user, workstation and company settings*. You must also define the *units of measurement* for the system and the directory for the *EPLAN original master data*. This ensures that your own master data is always synchronized with the original EPLAN master data.

EPLAN always suggests default directories for the installation.

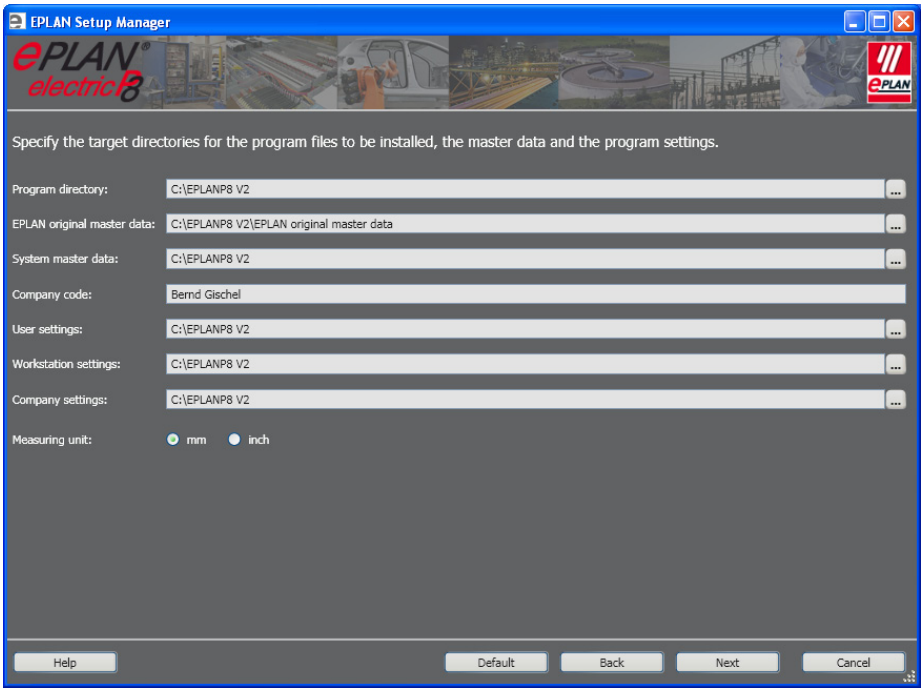


Fig. 1.4 Definition of target directories

These suggested directories can be kept or changed at this point. I always change these directories (and of course the company code) to my own target directories.

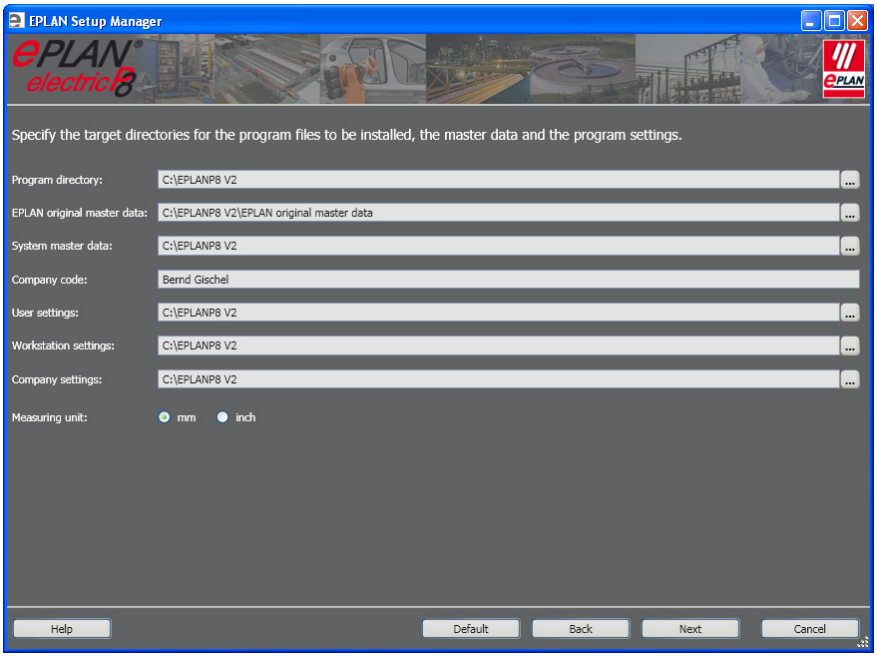


Fig. 1.5
Changed target directories and company code

When you have checked or changed all directories, you exit the dialog by clicking **NEXT**. EPLAN continues with the installation and asks what program components, master data and languages should be installed.

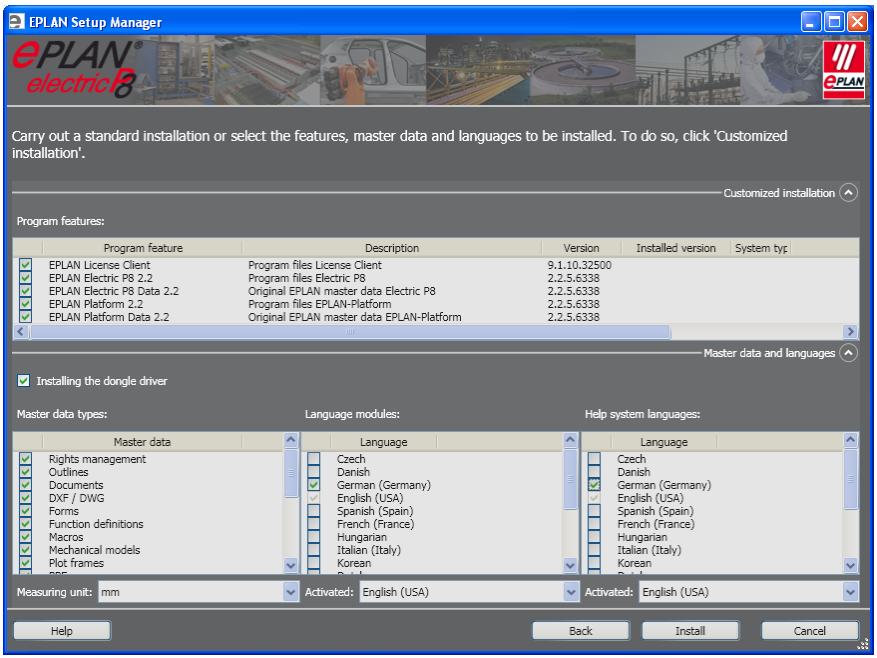


Fig. 1.6
Scope of components to be installed

After you set all the required information and click **INSTALL**, the Windows Installer prepares the required components and the actual installation begins.

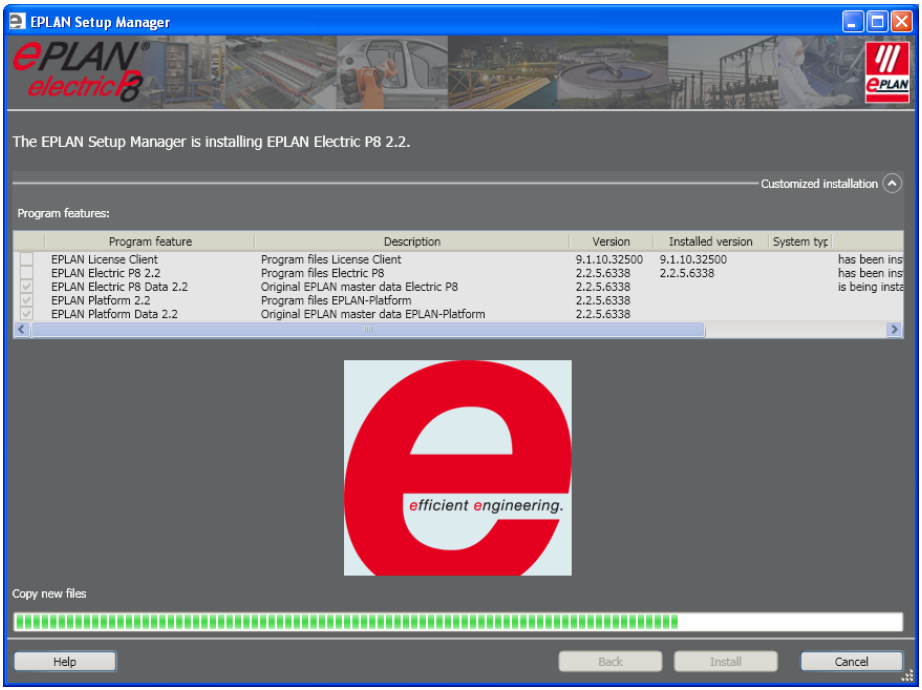


Fig. 1.7 EPLAN being installed

NOTE: EPLAN does not replace your system master data. If you would like to work with EPLAN's new system master data at a later date, then you must synchronize this data.

By design, EPLAN does not overwrite user-related master data because the user may have modified the original system master data and saved this under the original name assigned by EPLAN. During installation, EPLAN does not recognize whether this data has been changed and would therefore simply replace it. Usually the user does not want this to happen.

Once installation is complete, EPLAN displays the completion dialog. Here, you have to click **FINISH**. Installation of EPLAN Electric P8 is now complete.

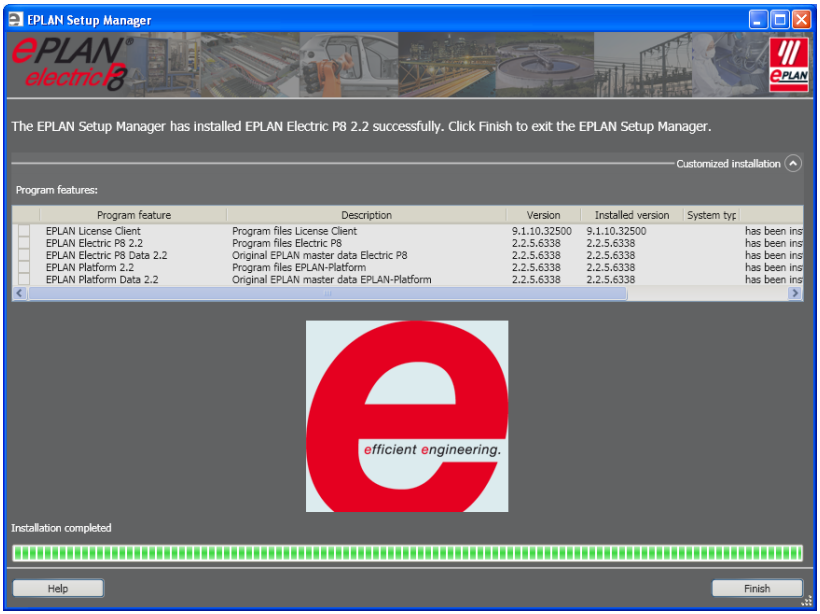


Fig. 1.8
Finishing the installation

EPLAN Electric P8 can now be started from the Start menu or the desktop icon.

If a license has not yet been installed, a one-time dialog requesting the validation code (license number) for the installed dongle (hardware protection) is displayed before the program starts.

Fig. 1.9
Activating EPLAN

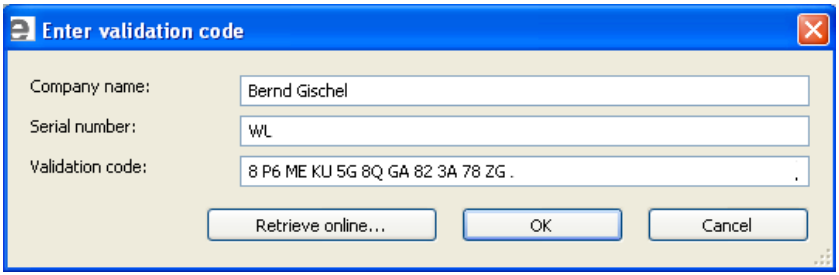


Fig. 1.10 Entering the validation code or request the code online

EPLAN starts with the **Select scope of menu** dialog. Here you can choose between the options *Beginner* (only the basic menus allowing graphical drawing of a project and/or working with macros), *Advanced* (more extensive display options, such as minimum text size or empty text boxes, can be displayed and used), or *Expert* (all menus and functions available), and then confirm your selection by clicking **OK**. The *Beginners*, *Advanced*, and *Expert* options are hard-coded into EPLAN and cannot be changed or extended.

NOTE: The **Select scope of menu** dialog is only displayed when EPLAN is used without rights management.

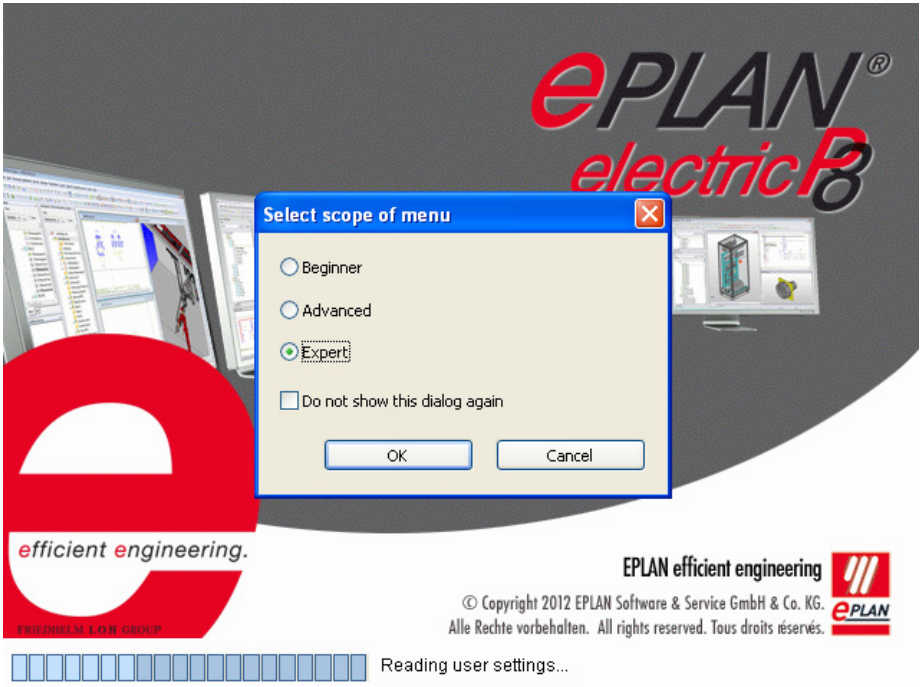


Fig. 1.11 Starting up EPLAN for the first time

If a previous version is being used, then there is a **one-time** option to import the settings (user, workstation and company) from this version in the subsequent dialog. If you click the **CANCEL** button, none of the previous version's settings will be imported.

EPLAN now opens the *Default* workspace (a standard view) and displays the **Tip of the day**.

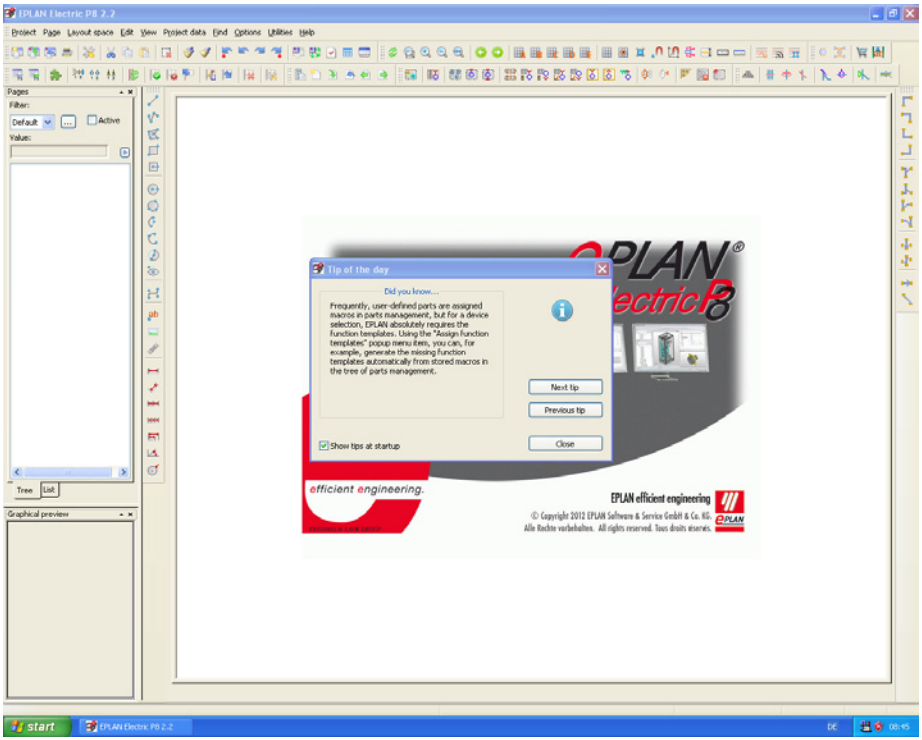


Fig. 1.12 Starting up EPLAN for the first time

1.3 Note for users of previous versions

You should use the export function to export the schemes, filters, etc. that you have created in a previous version so that you will be able to import them later into Version 2.2 as necessary.

1.3.1 Parallel operation with previous versions

Because EPLAN has again in Version 2.2 made some changes to the databases, which are no longer compatible with the previous versions, we recommend that you install Version 2.2 and its master data in a separate directory.

This especially applies to changes in the parts database. If you open the parts database with Version 2.2 and reformat it for Version 2.2, you will no longer be able to write to the file when you open it with previous versions. However you will still be able to read it with the previous versions.

2 The basics of the system

This chapter provides a brief explanation of some important EPLAN principles, functions, and working methods, and uses a number of examples to illustrate selected facts and system settings.

Important points are the directory structure, settings, multiple starts, notes on project and page properties, notes on particular dialog properties, handling of schemes, forms, plot frames, symbol libraries and an overview of my personal shortcut keys.

2.1 Five principles for working with EPLAN Electric P8

1st principle: Errors during project editing are allowed in EPLAN Electric P8.

In general, mistakes are allowed when working with EPLAN Electric P8. This basic principle of allowing mistakes is illustrated in the following example. A contactor may have two auxiliary contacts in the schematic, with both of them initially having the same connection point designation. Things that are not physically possible are initially "allowed" by EPLAN while working on a project. The user is not slowed down by "irritating" errors or messages during editing. This type of error naturally appears in message management, but only as a message entry. This entry initially has no further consequences. When project editing has progressed far enough, or is finished, EPLAN can perform certain project checks. Erroneous entries such as those described above will be listed in the project, if they do not already exist in message management.

Of course this error must be fixed in order to have a correct practical reference. However, this is not **compulsory**. EPLAN allows the user to decide whether a project is error free or not and which priority a message (error, warning, or note) should have. It is also pos-

sible to prevent such errors. With the *Prevent errors* check option, the above approach would not be possible. But this is a user-settable setting.

2nd principle: In EPLAN Electric P8, what is selected is what is edited.

The following example clarifies this principle. If I select three texts on a page and start the translation function, then exactly these three texts will be translated. If I select this page in the page navigator, then the translation function will translate all texts on the entire page based on their settings.

3rd principle: EPLAN Electric P8 stores data and any references online.

EPLAN is an online system. All references and device data are constantly (i. e. online) updated. The only limitations (if you want to call them this) relate to connection data. For performance reasons, these are only updated on request via a few specific actions. EPLAN performs the rest completely independently.

A typical example of this is the editing of a page followed by a page change. Here you need to manually start any required updating of the connections. There is of course a setting that allows EPLAN to do this type of connection updating automatically. However, this can negatively affect project performance.

In my opinion, constant (online) updating of connections is not really necessary because relevant actions such as graphical project reports or automated procedures such as device numbering automatically update the connections before the actual action is performed.

4th principle: EPLAN Electric P8 can be operated using a graphical approach.

This means that the devices (symbols) can first be placed in the schematic and subsequently be assigned the parts, including the associated function definitions. This is not compulsory, and you have a completely free hand when editing a project.

5th principle: EPLAN Electric P8 can also be operated using an object-oriented approach.

This means that external motor lists or other component lists can be read into the system as device lists, and the project can be started from this end.

NOTE: Principles 4 and 5 can be combined with each other. There are absolutely no limitations when working with EPLAN Electric P8.

2.2 Directory structure, storage locations

EPLAN can use any desired directory structure. EPLAN allows the user a free choice here. Data, such as project or master data, can therefore be easily integrated into an existing company data storage structure.

EPLAN recommends installing/running the program files locally, and only storing the data on the network. I agree with this recommendation. Any further structuring or organization of the remaining master data is the responsibility of the user.

By default, EPLAN creates the program directory with the following sub-directories:

- **BIN:** contains the program modules and, in additional subdirectories, the language files (e. g. en-US for U.S. English).
- **CFG:** contains configuration files for the users, company, workstation and projects.
- **P_ID:** is only of interest to PPE users.

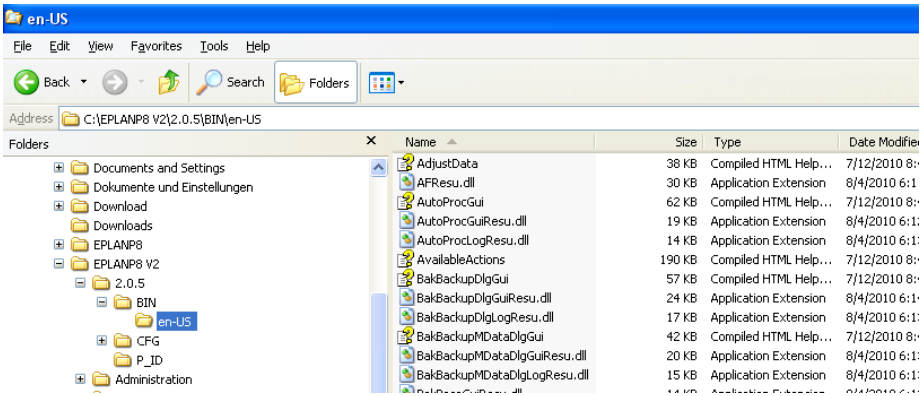


Fig. 2.1 Program directory with subdirectory

Similarly, during installation EPLAN also creates certain default directories for particular system data and other data.

For example, EPLAN creates a main directory called *Parts* with the subdirectory *Company code*. EPLAN gets this code from the information entered during installation – in this case *Bernd Gischel*.

The following *directories* are initially used as default directories and filled with relevant data during installation:

- *Part:* contains the parts databases (Microsoft Access **.mdb* format) and the configuration files for importing and exporting parts
- *Images:* contains all images

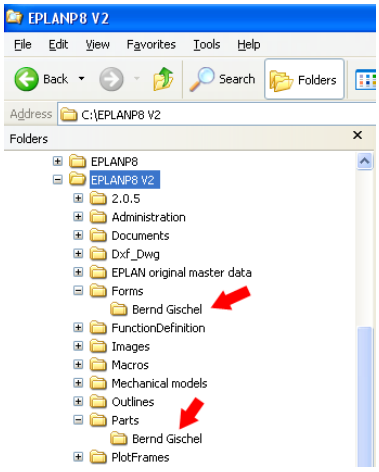


Fig. 2.2 Default customer directories

- *Documents:* contains documents such as *PDF* documents or Excel tables
- *Dxf_Dwg:* contains CAD drawings in *DXF* or *DWG* format
- *Forms:* contains all forms (system master data)
- *Function definition:* contains the files for the function definitions
- *Macros:* contains all macros, such as window macros (**.ema*) and page macros (**.emp*)
- *Mechanical models:* contains mechanical data
- *Plot frames:* contains all plot frames (system master data)
- *PPE:* contains files for the EPLAN PPE module
- *Projects:* default directory for projects
- *Revisions:* contains the revision database
- *Schemes:* contains preconfigured or user-created schemes, for example personal filter or sorting settings.
- *Scripts:* contains the corresponding **.cs* or **.vb* script files
- *Symbols:* contains all symbol libraries (system master data)
- *Symbol macros:* contains all symbol macros (**.ems*)
- *Translation:* contains the dictionaries (translation databases in Microsoft Access format, **.mdb*)
- *Management:* standardly contains the project database into which all projects are imported from project management and other management databases such as the rights management database
- *Templates:* contains templates, basic projects and exchange files for exporting project data (labels)
- *Xml:* contains *XML* files

These are the standard EPLAN defaults. EPLAN users can, of course, change these directory structures to suit their needs. For practical reasons, this should be done after the initial program start and before starting project management for the first time.

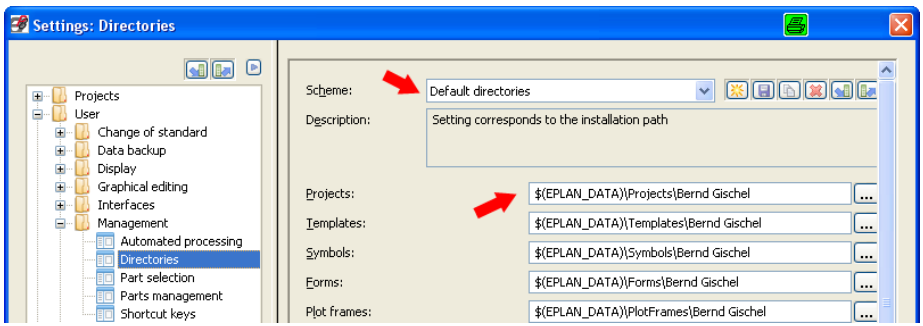


Fig. 2.3 Directory settings

User/Management/Directories

These directories can be viewed and modified under **OPTIONS/SETTINGS/USER/MANAGEMENT/DIRECTORIES**. A useful feature is that separate schemes can be created and configured for different directory settings here.

TIP: It is also possible to start EPLAN Electric P8 with an appropriate command line parameter to automatically set a particular directory structure. Example: “E:\EPLANP8 V2\Platform\2.2.5\BIN\Eplan.exe”/Variant: “Electric P8”/PathsScheme:OwnSchemeName (without prefixes and suffixes).

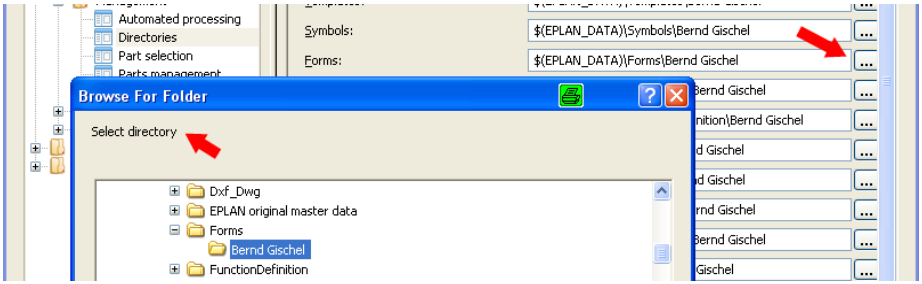


Fig. 2.4 Selecting other directories

Clicking the ... button causes EPLAN to display the **Browse For Folder** dialog. You can select a different directory here, or use the **CREATE FOLDER** button to create a folder at the desired storage location.

NOTE: These changed settings will not be used until EPLAN is restarted.

2.3 Settings – General

To put it simply, a project contains all the relevant data that it uses and requires. This means that a project is independent of the general system master data because EPLAN stores all required and used data within the project automatically upon first-time use.

The advantage of this method of “storage” should not be underestimated because even after several years this type of project can still be opened and edited with the original master data such as plot frames, symbols, etc. This is also true, for example, when the graphics of a company plot frame changes (e. g. because the company logo has been revised).

If the user wants, this behavior can be configured in EPLAN via settings (parameters). There are four basic setting areas as described below.

The settings are accessed via the **OPTIONS/SETTINGS** menu item. Storing this variety of project-related data naturally increases the data volume of the project.

An empty project has an average size of approx. 4 MB to 5 MB.

The following sections provides a brief overview of the settings. Many of these settings are explained in more detail in other chapters where they are actually used and can be more clearly explained.

2.3.1 Settings – Project

All project properties (and only these) are defined under **SETTINGS/PROJECTS [PROJECT NAME]**. The settings in the *Projects* node are available only if at least one project is open in the **page navigator**.

In the project settings you can, for example, define whether the project master data should be automatically synchronized with the system master data when the project is opened. You should always carefully consider the use of this setting, and many other project settings (i. e. "automation" when a project is opened), because changing these settings can have wide-reaching effects on the project to be opened. EPLAN has no "undo" button at this point, and if no data backups exist the old data is irretrievably lost.

Some important project properties, for example, are located under **OPTIONS/SETTINGS/PROJECTS/PROJECT NAME/MANAGEMENT/PAGES** and under the entry for the *Default plot frame*. This entry for the global plot frame applies to all pages, even any report pages, if no other plot frame has been assigned via the page properties. Symbol libraries are located under **OPTIONS/SETTINGS/PROJECTS/PROJECT NAME/MANAGEMENT/SYMBOL LIBRARIES**. After the settings are selected and entered, they are also stored with the current state in the project.

There are numerous settings – too many to include in the scope of this book. For this reason, only specific, important project properties are listed and explained.

2.3.1.1 Setting Projects [project name]/Reports

This area affects the output of graphical reports (see chapter 6, "Reports", for further explanations).

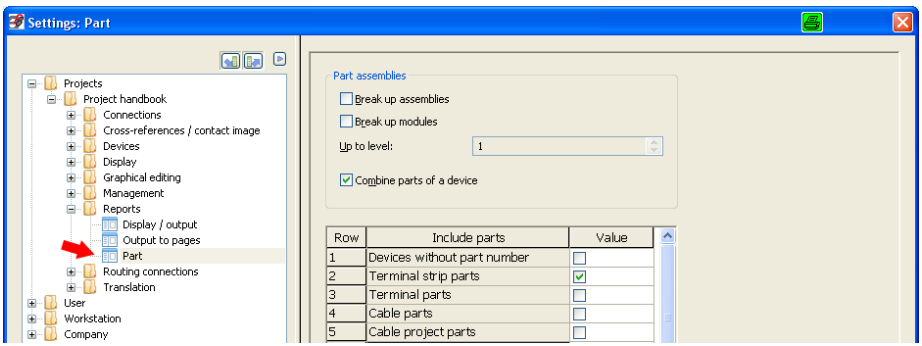


Fig. 2.5
Settings for reports

- **Display/Output:** Settings that affect the output or display of data in reports (forms)
- **Part:** Settings that affect the output of parts data and their behavior in reports and their display in certain navigators
- **Output to pages:** Project-wide settings for reports (forms) for the different report types in a project

2.3.1.2 Setting Projects [project name]/Devices

The device area is where you define different settings such as numbering schemes or syntax checks for devices.

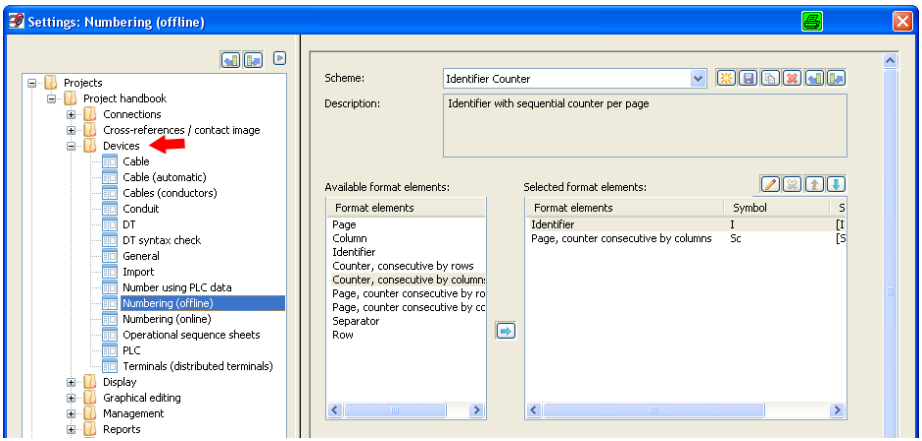


Fig. 2.6 Settings for devices

- **Sequence sheets:** Contains defaults for the symbols that are placed in sequence sheets (such as simple sequences (sequence chains) for a graphical listing of plant or machine processes according to the GRAFCET standard).
- **General:** Contains general settings, including the parameter for *conversion to uppercase* for devices. If the parameter is changed, uppercase letters are not automatically converted to lowercase letters or vice versa. It is only possible to work with either uppercase letters (parameter activated) or with uppercase and lowercase (parameter not activated). Generally, EPLAN initially always uses uppercase letters for identifiers. This setting can also be used to define the devices that automatically receive a prefix on insertion.
- **DT:** This is where settings for prefixes and conversion to uppercase are defined.
- **DT syntax check:** These settings define the *special characters* that can be used for the structure identifiers and for devices.
- **Cables:** Default settings for inserting cable definition lines (cables), shields, and their connection definition points (i. e. which connection definition point graphical symbol is the first choice).
- **Cables (conductors):** These settings allow for default settings; such as defining the symbols that EPLAN should use when inserting a connection definition.

- **Cables (automatic):** Contains the settings for automatic cable generation, cable selection, and cable numbering. All settings are defined via filter schemes. You can use predefined schemes or define your own schemes.
- **Terminals (distributed terminals):** Contains defaults for working with distributed terminals, such as: What function definition should be used for distributed terminals or at what point should terminals automatically become distributed terminals when placed (number of connection points)?
- **Numbering with PLC data:** Default values of a scheme for numbering connected devices with PLC data. PLC numbering can be used to number the following devices (among others): Terminals, pins, and general devices.
- **Numbering (offline):** Settings for a scheme for numbering devices offline (subsequently). Offline numbering is used to subsequently give a schematic a different DT layout. For example, the devices were first numbered with the default [identifier counter]. However, the devices must now be numbered according to the scheme [page identifier column]. The default for the scheme is set here. However, this does not have to be used. The scheme for offline numbering can be changed at any later time.
- **Numbering (online):** Contains the defaults for a numbering scheme for online device assignment. Such schemes are applied immediately when you create a schematic or insert symbols, macros or copy operations. This setting also defines the identifier set used for the project or how identifiers are to be handled when inserting symbols or macros.
- **Conduit:** Contains the settings for which connection definition point symbol should be placed when creating the EPLAN Fluid add-on's conduit definition lines (such as a tube or pipe connection).
- **PLC:** The defaults for outputting lists such as the PLC assignment lists and for the output format.

2.3.1.3 Setting Projects [project name]/Display

This setting is used to change the display of project structures in the navigators as well as in the page navigator. The *Display* node also contains the display or format of numbers, time and date.

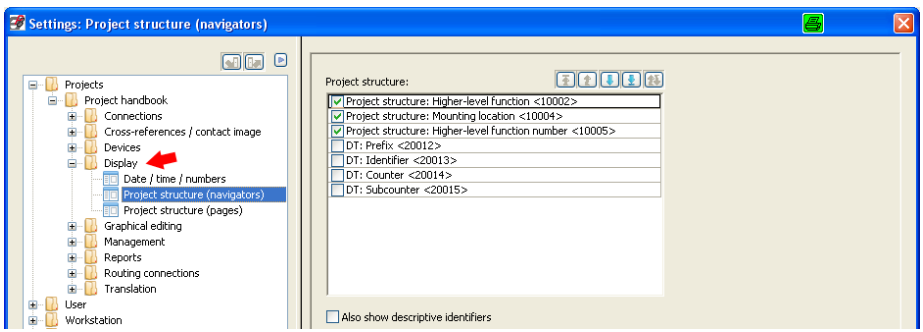


Fig. 2.7 Setting the project structure and format display

- **Date/Time/Numbers:** Settings for selecting the display format of the date, time and numbers in accordance with the operating system or a project-specific format (for example to set the date or time format to match typical foreign displays).
- **Project structure (navigators):** These settings allow you to change the display sequence of structure identifiers and/or of the DT in the navigators (except for the page navigator). However, this relates only to the display in the various navigators. Devices are not changed.
- **Project structure (pages):** These settings are used to adjust the display sequence, especially in the page navigator.

NOTE: This setting changes only the representation or display of the project structure or devices in the navigators. The page structure and the device tag structure are not changed.

2.3.1.4 Setting Projects [project name]/Graphical editing

This section covers the general settings for graphical editing (selected).

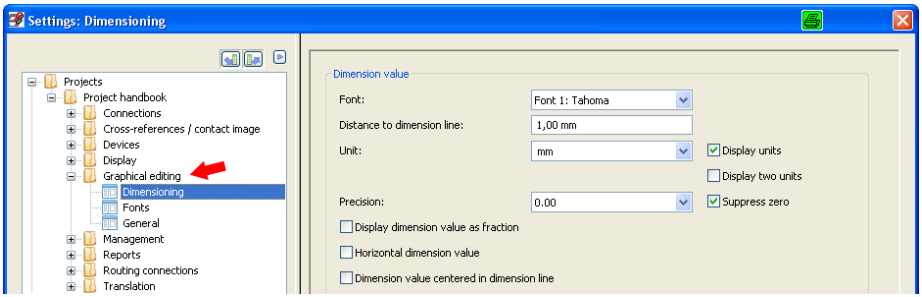


Fig. 2.8 Settings for graphical editing

- **General:** Settings such as the default values for representing connection junctions are set here, i. e. whether the display should be target-oriented, a point, or shown as drawn. However, the setting depends on the **SETTINGS/USER/GRAPHICAL EDITING/CONNECTION SYMBOLS** user setting and especially the *Connection symbols with prompt* parameter.
- **With target specification:** This default means, for example, that a T-node is drawn as target wiring. This T-node is also first placed and can then subsequently be changed to point wiring or a different target. This parameter has the effect that the representation remains as target wiring, regardless of how the T-node is internally set.
- **Point wiring:** This default allows T-nodes (junctions) to be entered with the visual display of point wiring. Internally, this type of junction can also be set for target wiring.
- **As drawn:** This means that a T-node, for example, is generally placed as target wiring. It is not possible to change this while placing (inserting) in the schematic. After the T-node has been placed, the **T-node [direction]** dialog can be called up by double-clicking and the targets can be changed. It is possible to change the display setting of this

T-node from target wiring to point wiring. This representation is then also shown in the schematic. This setting thus allows a mixed representation of target and point wiring.

With these parameters, it is generally possible to switch between the point wiring and target wiring representations. This means that if all T-nodes (junctions) were drawn with the point representation, the target wiring parameter allows the representation to be changed to target wiring.

Insertion of T-nodes (junctions) is controlled at a higher level by the **SETTINGS/USER/GRAPHICAL EDITING/CONNECTION SYMBOLS** parameter using the *Connection symbols with prompt* parameter.

If this query parameter is not switched on, the T-node (junction) is first placed and can be changed later. If the dialog is switched on, it is displayed before the T-node (junction) is placed in order to set the target tracking or point/target wiring.

Aside from the settings for the connection symbols, there are other additional general settings contained in this node.

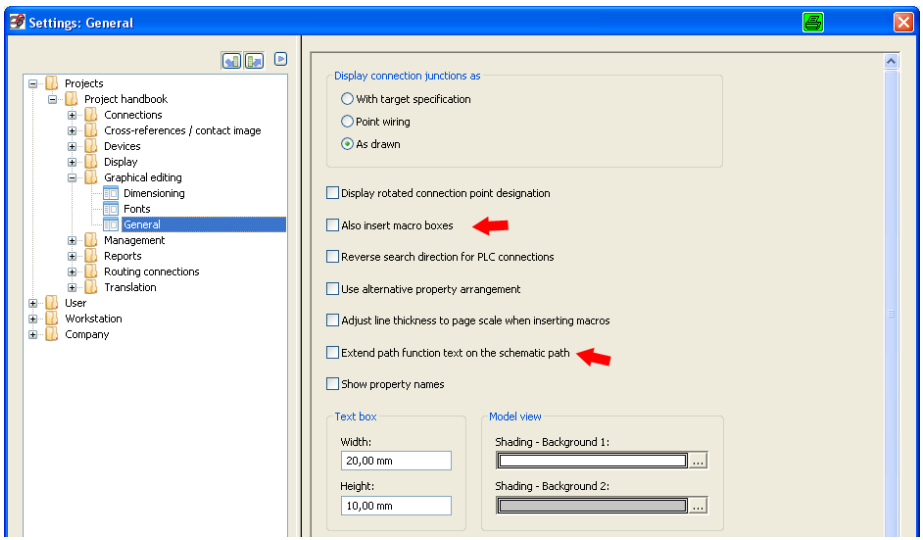


Fig. 2.9 Additional general settings

- **Display rotated connection point designation:** When this setting is activated, the connection point designations are displayed rotated by 90 degrees. This setting also applies to plug and terminal designations.
- **Also insert macro boxes:** If this setting is activated, any macro boxes present on the macro will also be inserted.
- **Extend path function text on the schematic path:** If this setting is activated, the insertion point of a path function text does not have to be directly in the path of the device in order to be applied. It is sufficient if the path function text is in the path (of the plot frame).

- **Dimensioning:** Format setting for entering dimensions, such as font or dimension line termination. Changing this setting has no effect on existing dimensions, and only affects dimensions that are subsequently inserted. The number of decimal places is defined in the USER > DISPLAY > DISPLAYED UNITS OF MEASURE setting.
- **Fonts:** Defines the fonts that should be used in the project (up to ten fonts). These settings have priority over the font settings in the company settings (*Company/Graphical editing/Fonts*).

2.3.1.5 Setting Projects [project name]/Cross-references/contact image

These settings offer many parameters defining the representation of references for interruption points, motor overload switches, contactors, general areas (for example the representation of references between different page and report types and the symbols to be used) and the display itself (the options for the overview cross-references between the types multi-line, single-line and the overviews are set here).

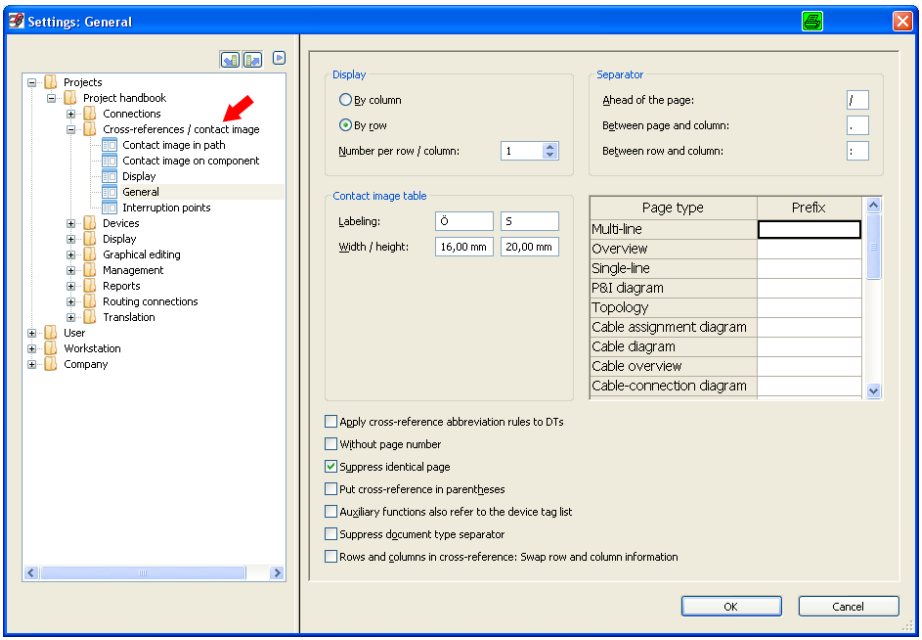


Fig. 2.10 Cross-references/contact image settings

The device settings always have priority.

If no changes are made to the device settings, for example to the cross-reference representation, the default settings are taken from these parameters (depending on the symbol properties).

- **Interruption points:** Default settings for displaying cross-references such as by column or row, with or without page names, and other settings

- **General:** Settings for displaying general cross-references, how and which separators are placed between pages and columns and if the cross-reference is enclosed in brackets
- **Display:** Setting defining between which representation types (multi-line, single-line overview, etc.) device cross-references should be displayed
- **Contact image on component:** Default setting for displaying cross-references at components, such as motor overload switches, circuit breakers, etc.
- **Contact image in path:** Default setting for displaying, for example, cross-references of the arrangement (contact image) of contactor coils

2.3.1.6 Setting Projects [project name]/Connections

The default values for connection properties are defined in this node.

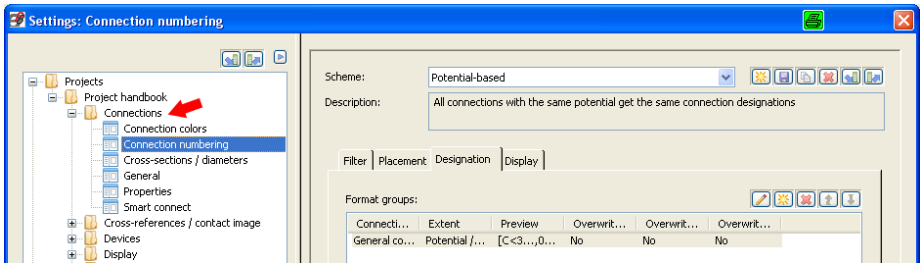


Fig. 2.11 Connection settings

- **General:** Setting for determining the source and target
- **Properties:** Default settings for certain connection properties such as the unit (mm^2), etc.
- **Cross-sections/diameters:** Default project setting for cross-sections and diameter values
- **Smart connect:** Defaults for settings such as using a special interruption point symbol and for interruption point designations for interruption points that EPLAN automatically generates when using smart connect
- **Connection colors:** Editable, extendable table of color codes, color names and colors to be used in a layout space
- **Connection numbering:** Defines the scheme used for numbering connections. You can use predefined schemes or your own scheme.

2.3.1.7 Setting Projects [project name]/Routing connection

This is where you enter default values for connections, their routing and associated terminations such as double-ended sleeves. The data/settings are needed during export.

This is also where you will find the settings for importing (text file) connections and their information, such as source and target DT, conductor color and cross-section.

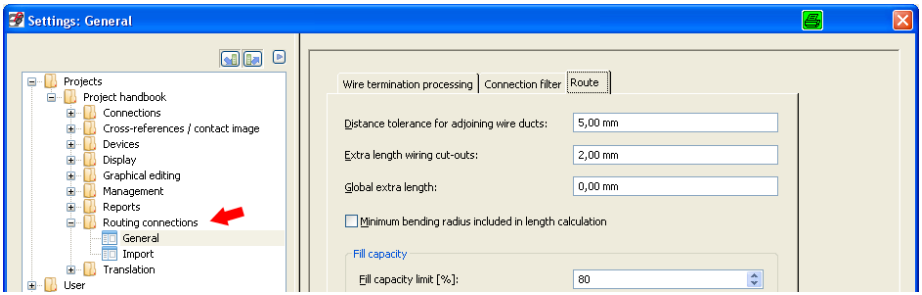


Fig. 2.12 Settings for routing connections

2.3.1.8 Setting Projects [project name]/Management

This includes default settings defining how EPLAN should handle, for instance, master data or particular modules in the project.

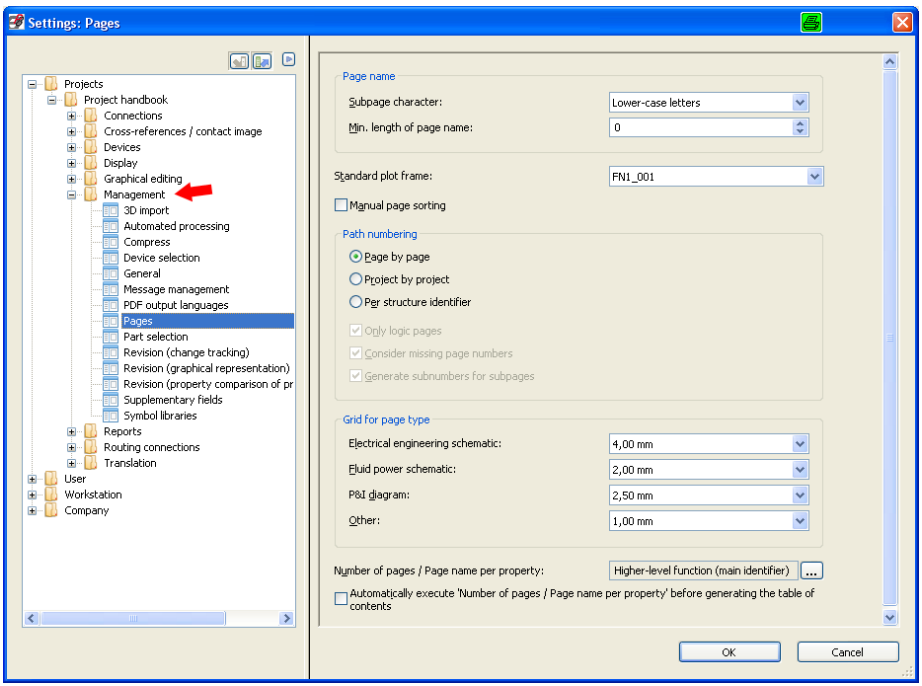


Fig. 2.13 Settings for managing the project

- **3D import:** Default setting of a function definition when importing 3D data
- **General:** This setting contains three very important parameters. The *Synchronize project master data when opening* parameter means that when the project is opened, the project master data, e. g. a terminal diagram form or a symbol library, is synchronized with the system master data and possibly updated. The *Synchronize plot frames* setting

synchronizes plot frames immediately after the project start. The last setting, *Synchronize stored parts when opening*, means that parts are automatically synchronized with the master data.

NOTE: Forms are only synchronized when, in the **PROJECTS [PROJECT NAME]/REPORTS/OUTPUT TO PAGE** setting, the *Synchronize* option for the corresponding forms has also been set.

Before EPLAN synchronizes the master data, it displays a security message that must be confirmed by clicking **YES**. Only then is the project master data updated, and only when it is older than the system master data.

- **Part selection:** Preselection of parts (filter schemes, for example, can be used to display only particular types of contactor in the parts selection), and also which parts database EPLAN should use
- **Automated processing:** A definable active script. This script is used as the default setting for the *Automated processing* function.
- **Device selection:** In addition to part selection, EPLAN also has more effective device selection. Whereas the part selection is independent of the function definitions used in the schematic, device selection occurs based only on the function definitions belonging to the devices. A preselection of how EPLAN should handle the device selection is defined here: For example, are existing function data to be used and if so, which ones, etc.?
- **Compress:** Default scheme used for compressing the project
- **Message management:** An important setting that defines the extent and type of project checks for the current project. Project checks are not essential and do not necessarily have to be performed. However, they do uncover possible editing or data entry errors during project editing and list the problem areas in message management. This setting defines the project checks and the extent of each check for this project. The number of project check messages can also be limited here. This scheme is stored in the project the first time it is used, but is not automatically synchronized with a newer version of this project check. This is important to remember when it is used.
- **PDF output languages:** Default setting for outputting languages in the PDF document to be generated
- **Revisions (property comparison of projects):** Settings for placing revision markers at connections
- **Revision (graphical representation):** Setting for the revision control of projects: What color or line thickness should be assigned to revision markers for changed, deleted or added objects or changes in the reports?
- **Revisions (change tracking):** Settings for revision control with change tracking, for example: What watermark is to be displayed on an incomplete page? Should a project be stored as a PDF upon completion?
- **Pages:** Defines the handling of pages. What combinations of letters and page names are allowed for subpages? How should paths be numbered: by page, by project or by structure identifier? The most important parameter in these settings is the definition or

(subsequent) changing of the global plot frame for the project. A separate plot frame can be defined for every page in the page properties independently of the global plot frame setting. The page settings always have priority over the global project settings.

- **Symbol libraries:** Defines the symbol libraries used in this project. New entries (addition) of symbol libraries are automatically stored in the project after the setting is saved.
- **Supplementary fields:** EPLAN provides the user with 100 freely definable supplementary fields for each of the project, page, and function areas. The *Part reference* tab has five freely definable supplementary fields. Freely definable means that users of the *User supplementary field 1* property can provide their own name for the project. This name, and not the previous name, is then displayed in the relevant areas.

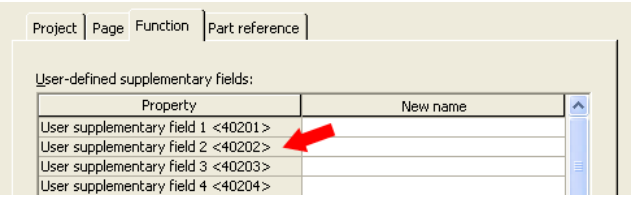


Fig. 2.14 User-defined supplementary fields tabs

In the **MANAGEMENT/SUPPLEMENTARY FIELDS** setting, the default is that user-defined supplementary fields do not have names of their own. The properties with the standard names defined by EPLAN would thus be displayed on a function.

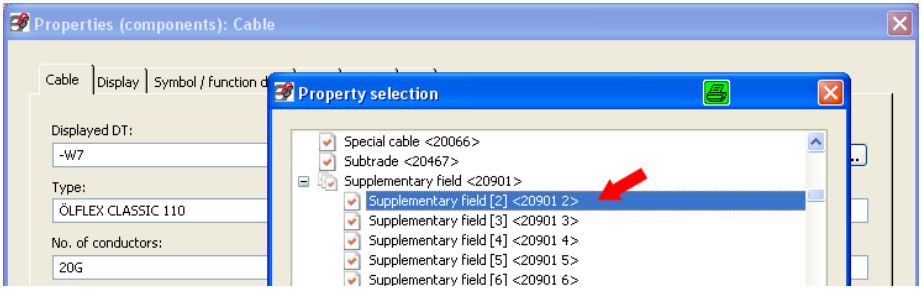


Fig. 2.15 User-defined supplementary fields on a function (here: cables)

You can assign separate names to these user-defined supplementary fields so that they are more easily recognizable.

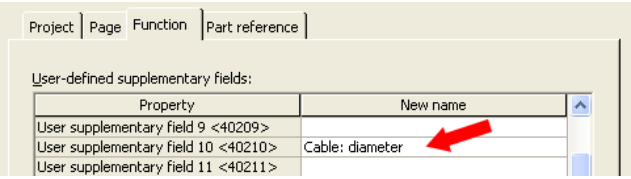


Fig. 2.16 Assignment of a separate name for the user supplementary field 10 <40210>

Using a function (here: a cable), this *User supplementary field 10 <40210>* can then be added with its new name.

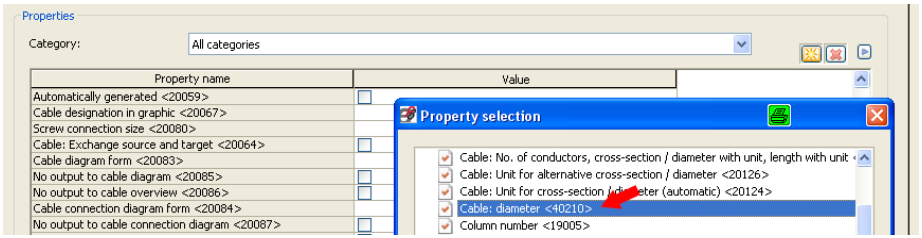


Fig. 2.17
Property <40210> with its "new" name

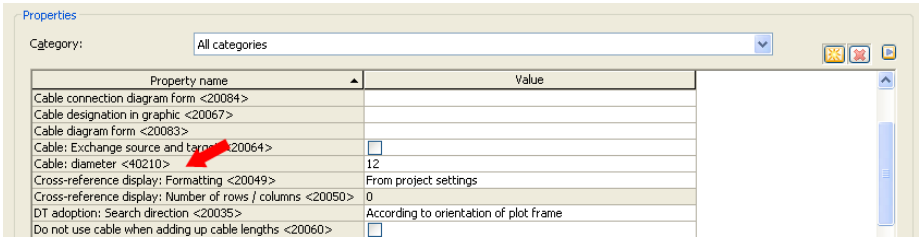


Fig. 2.18
The user-defined supplementary field integrated into the properties

The advantage of assigning separate names for user-defined supplementary fields is self-evident: clear defaults for entries with very specific properties that may have to be evaluated correctly later on. This helps avoid incorrect entries and tedious extra work to fix values that have been assigned to the wrong properties.

2.3.1.9 Setting Projects [project name]/Translation

This setting defines the databases to be used for translation or the scope of translation that EPLAN should use.

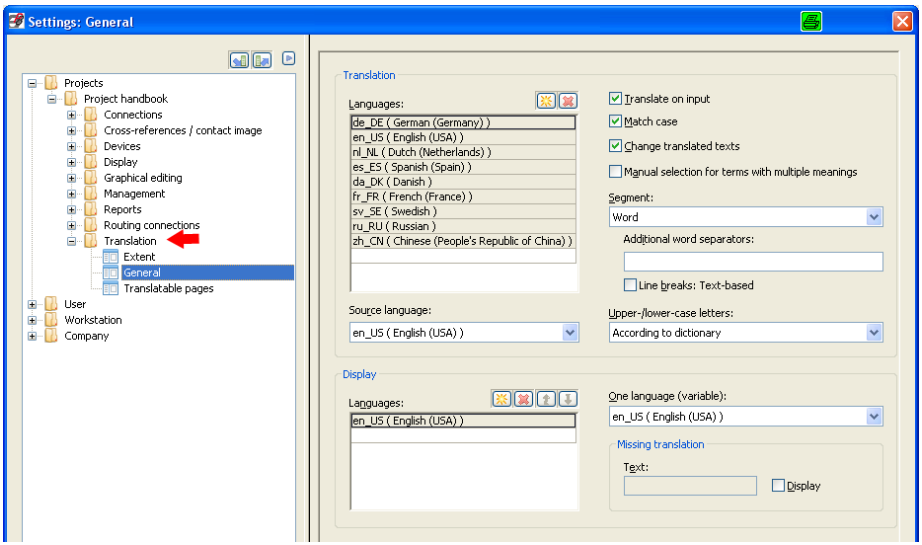


Fig. 2.19
Translation settings

- **General:** This is where you enter project language settings, settings defining the handling of texts when translating during data entry. and settings for displaying translation

languages (displaying translations in the project). You also have to define a source language here.

- The definition of a **source language** is crucial, because this is the **first** language EPLAN displays in the various dialog boxes. This helps to prevent operating errors, e. g. entering English in the wrong "language line" by mistake.

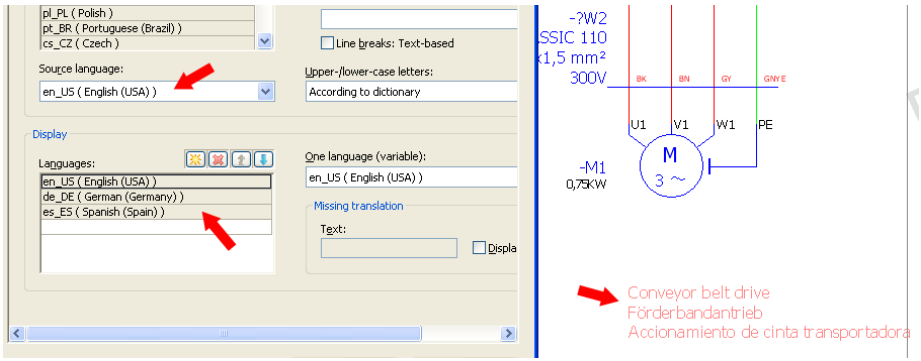


Fig. 2.20
Setting the translation sequence to "English, German, Spanish"

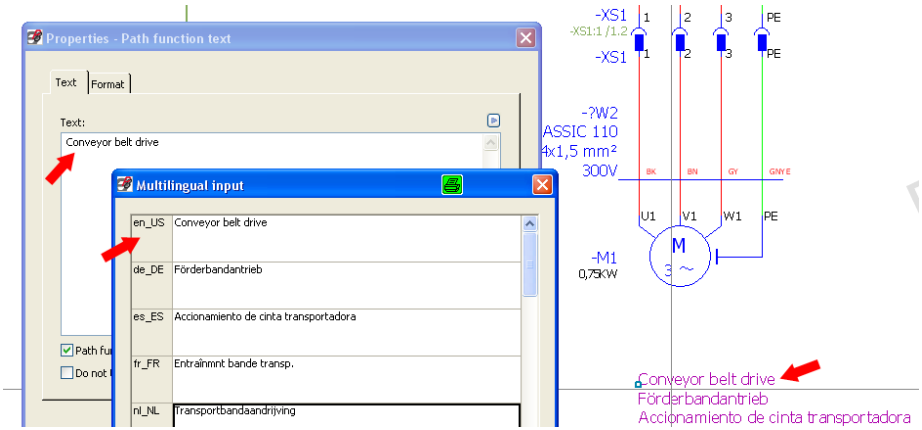


Fig. 2.21
Display in graphical editing

If the sequence of the *displayed languages* is changed in the settings, the *source language* will still be shown first in the dialogs.

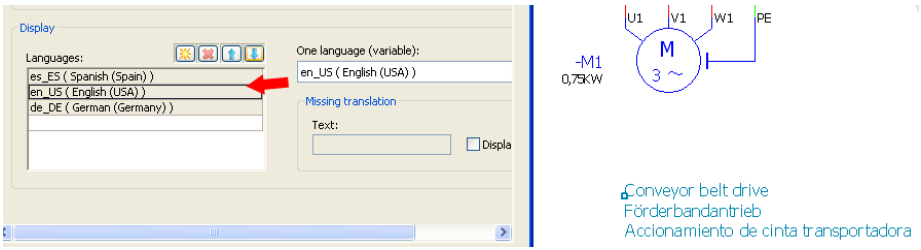


Fig. 2.22
Changing the translation sequence to "Spanish, English, German"

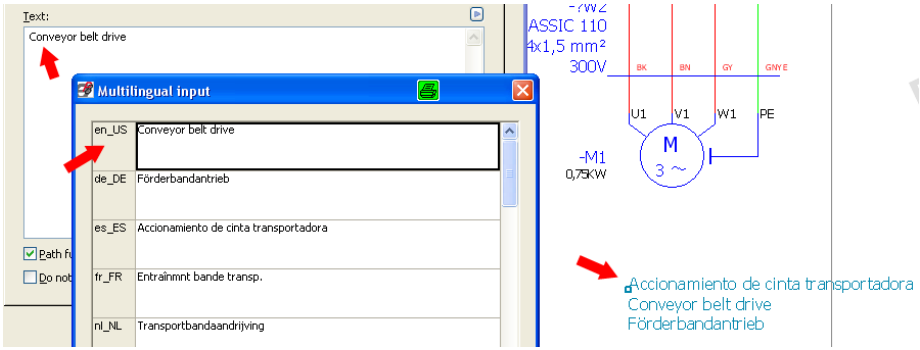


Fig. 2.23 Display in graphical editing

The sequence of the displayed languages has been changed correctly, but the sequence of the languages in the dialogs remains as is. This way, translations can no longer be entered by mistake.

- **Extent:** This item allows the user to define the range of the properties to be translated for particular texts in forms, projects or components. Properties set to active are translated, and when the check box is deselected, they are not translated. This allows specific properties to be excluded from translation. The properties of the areas are defined by EPLAN and cannot be extended.
- **Translatable pages:** This setting is similar to the **Extent** setting. However, it generally relates to a page type. If page types are excluded from a translation, then they are not translated during an automatic translation run.

2.3.2 Settings – User

User settings relate to personal settings such as the user interface structure, which dialogs are opened on which screen, the directory structure for certain personal data, and the different representations of settings. All these settings depend on the login name entered at the Windows login and are stored in the directory *\\Basic\...\USER...CFG* in a workstation-specific manner (by computer). If you place these settings on a network drive, for example, then you can call up your own EPLAN configuration from any computer in the network.

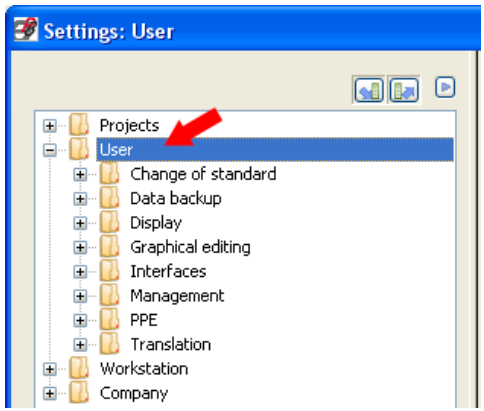


Fig. 2.24
User settings

2.3.2.1 Setting User – Display

This user settings area contains the personal user interface settings or default values for a user-related workspace.

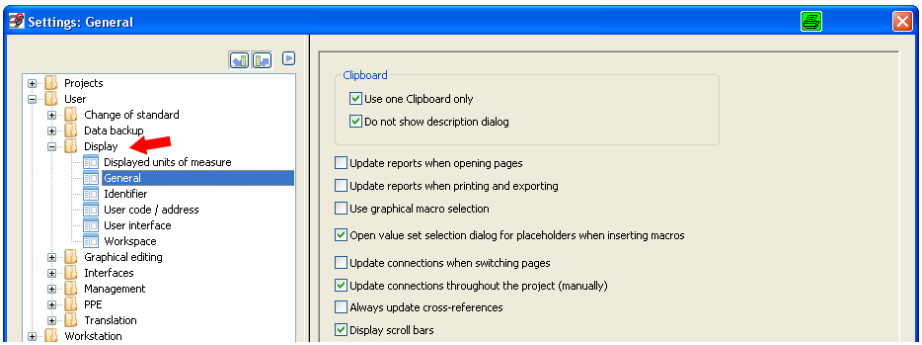


Fig. 2.25 General display settings

A number of important settings are explained below.

- **General:** Settings defining whether multiple clipboards or associated description dialogs should be used.

This settings area contains other important settings:

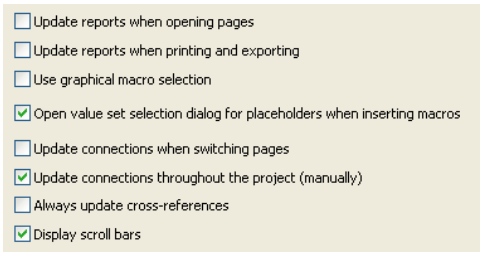


Fig. 2.26 Additional general settings

- **Update reports when opening pages:** If this parameter is active, then reports that are already generated in the project and which exist as graphical output are automatically updated by EPLAN when a page of this type is opened.

  This may be intentional, but it may also be the case that these reports should not be graphically changed by EPLAN, e.g. for revision reasons or for tracking changes. This parameter should then be switched off by deselecting the option check box.

  This parameter should also be switched off for performance reasons if project editing becomes very slow (EPLAN must constantly update the data in the background and then update the report as soon as it is opened – this requires a fast computer). This also applies to the **Update connections when switching pages** setting. The **Update connections throughout the project (manually)** setting has the following effect: If this setting is active, the connections will be updated throughout the entire project. If the setting is inactive, only the selected connections (one or several pages or selected devices, etc.) will be updated.
- **Update reports when printing/exporting settings:** This way, the reports are always up-to-date when the project is printed or exported.
- **Use graphical macro selection:** This setting can only be used in combination with the *Company/Display/Graphical macro selection* node settings.

The nodes are also found in the following settings:

- **Displayed units of measure:** Here you define the units of length and weight as well as the number of displayed decimal places.
- **Workspace:** This is where you define the preferred user workspace. The workspace can be changed at any time during project editing. Workspaces will be handled in more detail later in section 2.9.1.
- **User code/Address:** This is where you enter the user's code, name, phone and e-mail address. Some of these entries are used for the page or project properties, and others are used for system messages, for example when a conflict with another user occurs during project editing. These entries are also useful for forms, plot frames, etc.
- **Identifiers:** Default settings defining how EPLAN should sort structure identifiers such as higher-level function or installation site that are reassigned during project editing

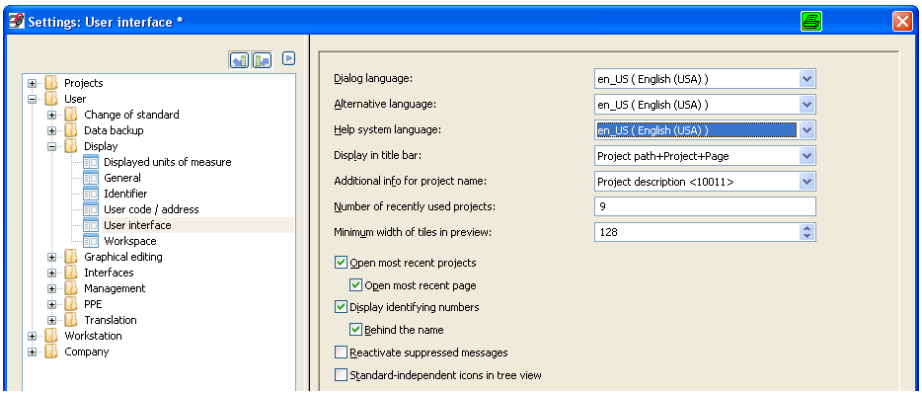


Fig. 2.27 Extended interface settings

- **User interface:** General settings for the dialog language, and language settings for the help files. Further user interface settings are possible here: Should the last projects be reopened when the program starts, or should the property number be displayed with the properties in EPLAN (= my recommendation)? The *Reactivate suppressed messages* setting is interesting. If this is set, then dialogs whose displays have been deactivated are once more displayed. Apart from this, there is also the *Display in title bar* setting. Here you can define what information is to appear in the title bar: only the current page, only the project name and the current page, or the project name including project path and the currently open page.

2.3.2.2 Setting User – Data backup

These user settings are for defining default values for backing up master data.

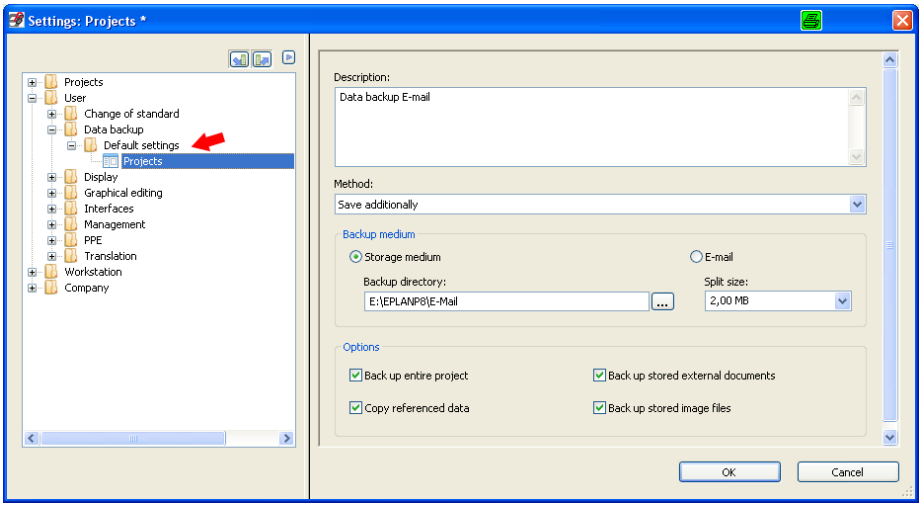


Fig. 2.28
Data backup settings

- **Default settings/Projects:** Default settings for project backup details such as the e-mail message split size or the backup drive.

2.3.2.3 Setting User – Graphical editing

These settings define further user interface settings such as colors or settings for placing connectors (T-nodes, etc.).

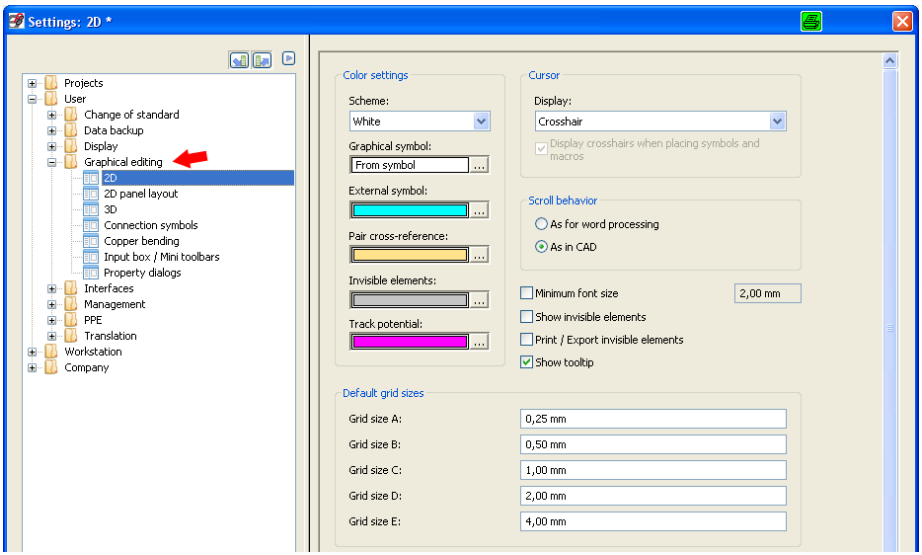


Fig. 2.29
Settings for graphical editing

- **2D:** Display of color settings, background color, grid sizes, cursor settings, and scrolling behavior

The following settings, especially the grid sizes, have proven useful in practice. I have found the 4; 2; 1; 0.5, and 0.25 levels to be very practical.

- **2D panel layout:** Settings for part placements, their mounting options, and settings for applying dimensions from the parts master data or manual input
- **3D:** Settings for the 3D area, such as grids, color settings and how terminal strips are to be handled during placement
- **Property dialogs:** Settings defining whether certain property dialogs, for example **Shield properties**, should be automatically opened after an object is placed
- **Connectors:** Important settings for defining whether or not the connection dialog should be displayed before a connection junction (T-node) is placed and whether or not changes to connection designations should be transferred to subsequent connections

2.3.2.4 Setting User – Change of standard

This is an important area for projects impacted by a change in a standard. These entries remain empty until a change of standard has occurred.

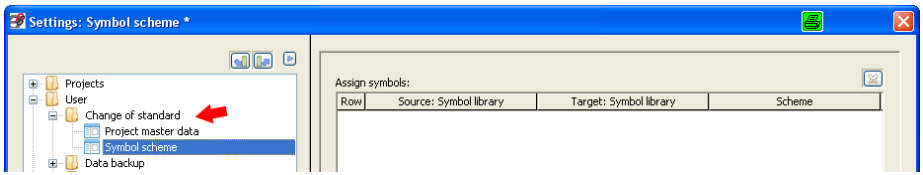


Fig. 2.30 Change of standard

- **Project master data:** Contains project master data assignments. This area is only filled with data when a change of standard has occurred.
- **Symbol scheme:** Contains symbol assignments. Similar to project master data, if no change of standard has been performed then this area is empty.

2.3.2.5 Setting User – Interfaces

EPLAN offers a number of different interfaces for importing and exporting data. The interface settings are defined in this area.

These settings are adopted as default values if no other settings are used during the direct output.

Important interface settings

- **Labeling:** This is where you define default settings for the most frequently used schemes, in order to transfer data to the labeling system.
- **Image file export:** Default settings for outputting image files in BMP and TIFF format.
- **Export and import of DXF/DWG:** Default settings for importing and exporting DXF/DWG files.
- **EPLAN 5 data import:** Settings for importing EPLAN 5.x data.

- **EPLAN 21 data import:** Settings for importing EPLAN 21 projects such as symbol library assignments, directory settings, languages, and general import parameters.
- **External editing [type]:** Defines the schemes most frequently used for external editing. These settings affect a number of different areas, such as functions, pages, and connections.
- **PDF export:** Settings for internal PDF output of pages or projects.

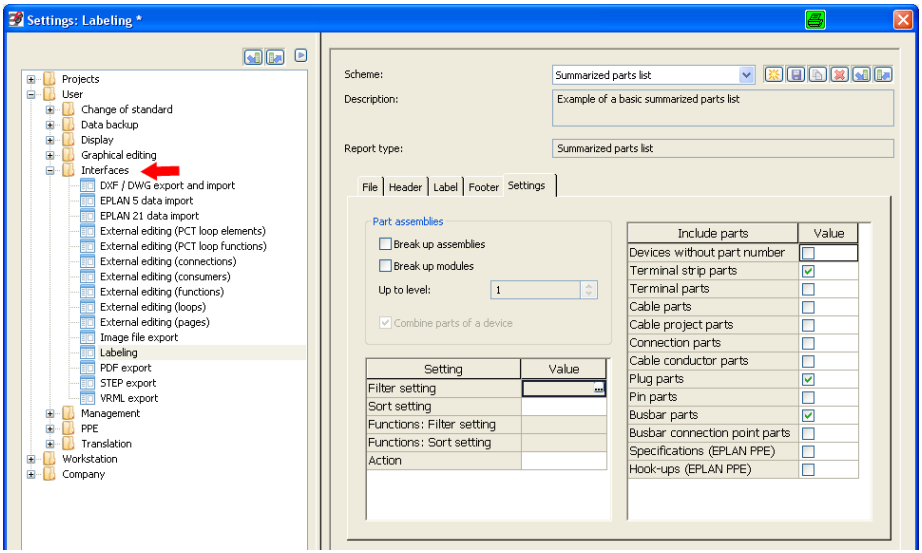


Fig. 2.31 Various interface settings

2.3.2.6 Setting User – Management

In this area, EPLAN allows user-specific settings for management tasks.

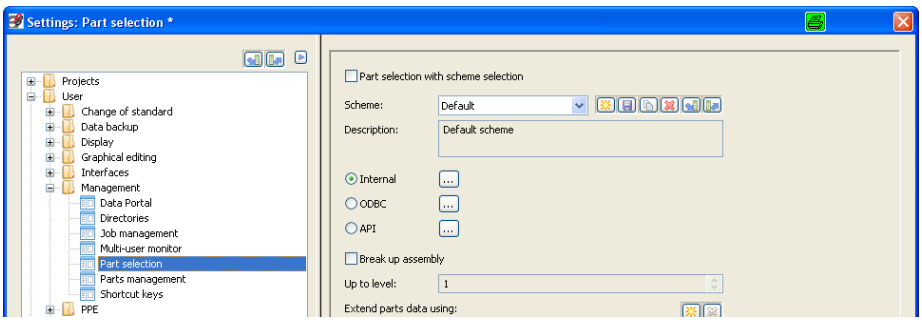


Fig. 2.32 Management settings

- **Part selection:** Defines part selection, where the parts are selected and the interface to be used (API, internal, ODBC or COM interface). This can also be an external PPC system.

- **Parts management:** Here is where you define the parts database and various settings affecting the parts and how they are displayed. For example, you can define whether EPLAN should import parts from the internal parts management database or an SQL database.
- **Data portal:** Settings for the user, the portal and the connection settings to the portal.
- **Shortcut key:** User-specific settings for defining shortcut keys for EPLAN commands. The shortcut keys can be individually assigned or deleted here, or the keyboard assignments can be restored to the default settings.
- **Directories:** A complete overview of the installed directories for the various master data and databases in EPLAN. You can also use your own schemes to change all the directories in a single step.

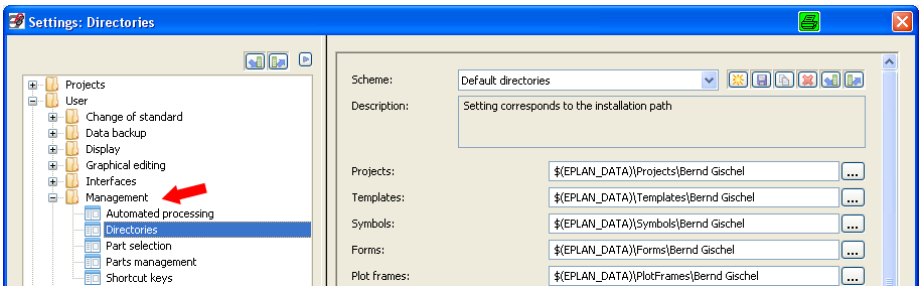


Fig. 2.33 Default directories

2.3.2.7 Setting User – Translation

These are general settings for defining how languages and translations are to be handled.

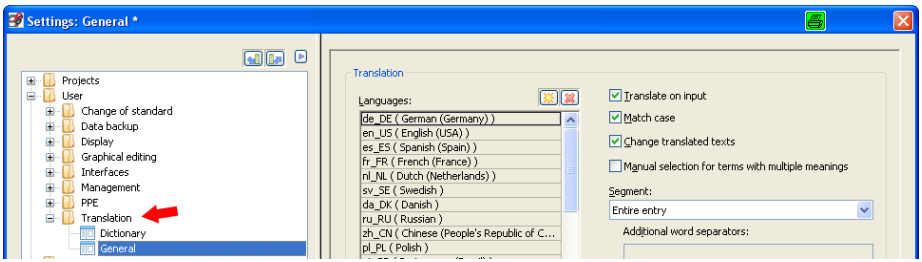


Fig. 2.34 Translations, foreign languages

- **General:** Definition of how project-independent texts are to be translated and which source language is to be used. The source language is the language for which translations will be later generated.
- **Dictionary:** Defines the dictionary the user would like to work with. The foreign languages available in the database are displayed simultaneously. The parameter also allows default settings for importing new translations and defines the behavior when texts are entered (AutoComplete or AutoCorrect).

2.3.3 Settings – Station

These are general settings for the current workstation.

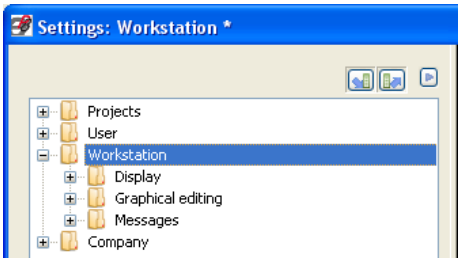


Fig. 2.35
Station settings

- **Display/General:** This allows you to set operational optimization if you are using a terminal server.
- **Graphical editing/Print – PDF output size:** Here you can set various global options related to printing, such as print margins or print size, and enter settings that affect the size (scaling) of a PDF output.
- **Messages/System:** This is where you define the path to, and the maximum size of, the system messages file.

2.3.4 Settings – Company

The company settings contain a number of parameters that, for example, can make comparisons of projects from external suppliers easier.

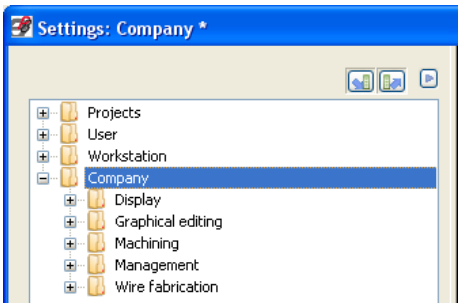


Fig. 2.36
Company settings

- **Display/Graphical macro selection:** This setting allows you to more easily find and select macros and their directories using your own texts (descriptions) and/or images.

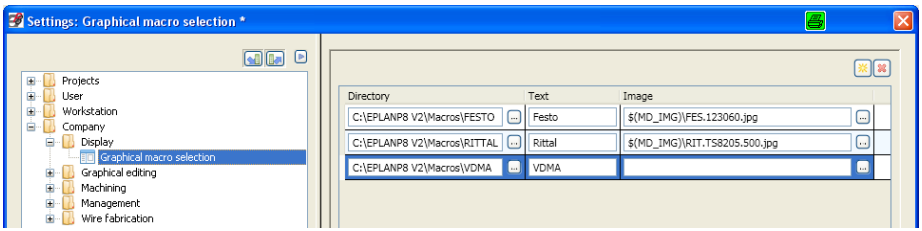


Fig. 2.37 Defining the settings for graphical macro selection

These default settings can only be used in the *graphical editor* if the *Use graphical macro selection* option has be selected in the *User/Display/General* settings.

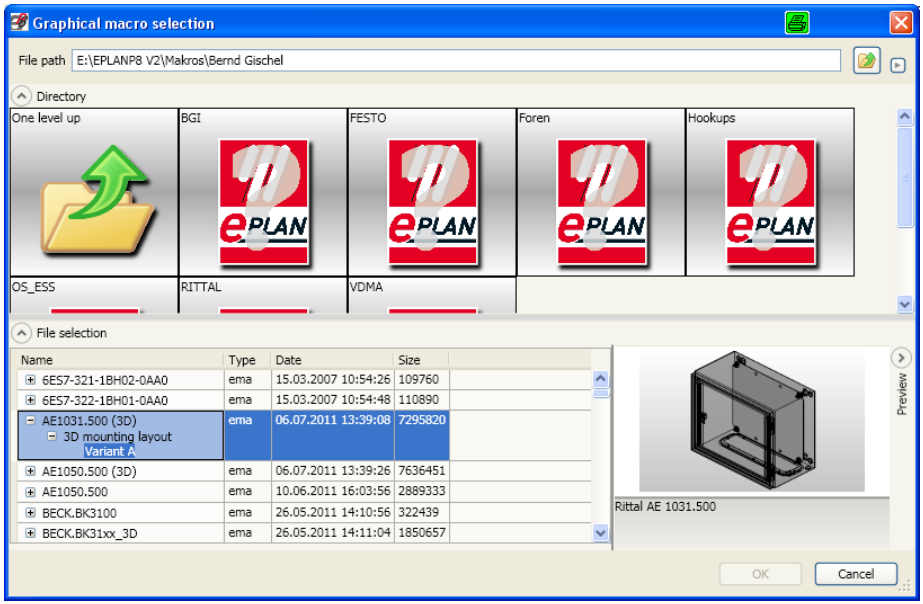


Fig. 2.38 Graphical macro selection dialog (started from the graphical editor)

- **Graphical editing/Fonts:** Defaults for up to ten company fonts. These ten different fonts allow you to later switch from one font to another in the entire project without any problems, provided the affected object uses one of the global fonts. However, priority is always given to the project-specific fonts (setting under *Projects/[project name]/Graphical editing/Fonts*).
- **Management/Property comparison** of projects): Settings determining the properties to be used for project comparison. You can create your own schemes for comparisons.
- **Management/Project management database:** This is where you can set the storage location of the company-specific project database.
- **Management/Add-ons:** Allows you to automatically control EPLAN Electric P8 when it is started using the manual defaults defined in an *Install.xml* file, whose location is defined here.
- **Management/Change tracking (numbering of the revision index):** Settings (controllable via your own schemes) for indexing revisions.

2.4 EPLAN and multiple starts?

EPLAN can be started as often as desired. You may ask why, since you can work on several projects at the same time in EPLAN. In general, you do not need to start EPLAN multiple times, but it is possible.

One of the practical benefits of EPLAN multiple starts is that you can have parts management and the dictionary open at the same time, allowing you to enter missing entries without having to interrupt your work on open projects.

When you are finished editing, for example in parts management, you of course should and must synchronize this data with the project and decide where these parts are to be stored. This can be done automatically on each subsequent start. Here too, you must carefully consider whether automation at this point is actually useful. EPLAN lets the user decide.

2.5 Properties

The different types of properties in EPLAN

In addition to the actual graphics of symbols, forms, or plot frames, EPLAN also outputs logical information. This logical information must be specified for the symbols, forms, or plot frames so that it is visible. This is done by assigning properties.

Every property has a property name and an associated property number (known as an ID, and possibly also an index) that is only valid for this specific property. You can use the property number to gain an *approximate* idea of the area that the property belongs to (project, page, etc.). The property value assigned to each property is unique to the property.

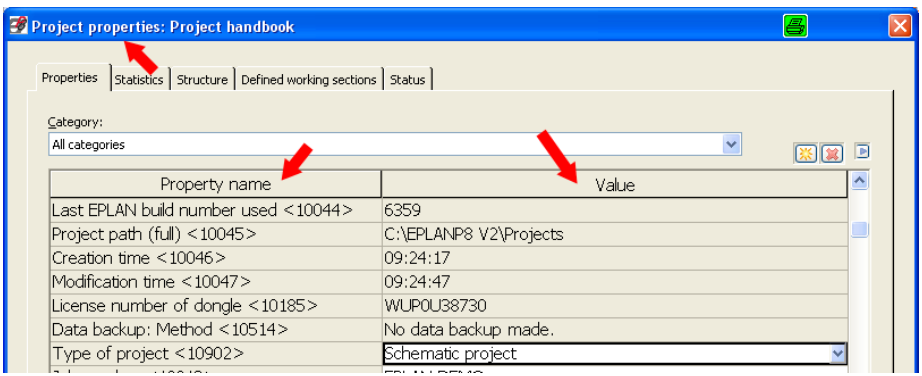


Fig. 2.39 Project properties

The *Project name <10000>* property contains the *<Project name>* value (the name of the project).

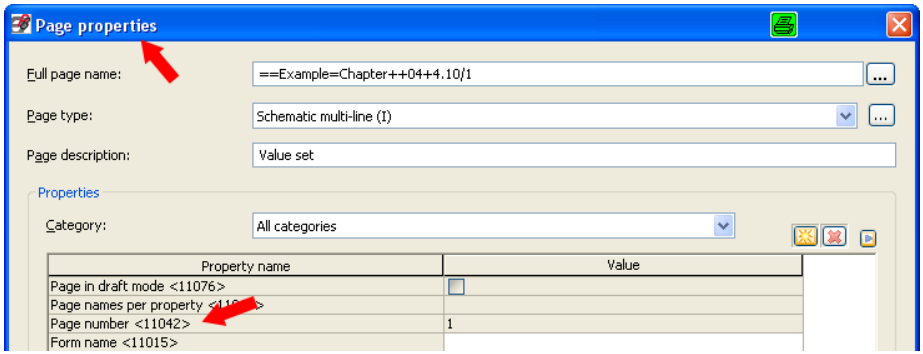


Fig. 2.40
Page properties

The *Page number <11042>* property contains the *<n>* value (n = the actual page number).

EPLAN generally distinguishes between several different types of properties. There are *Project properties* (relevant project information such as the project name; 10000), *Page properties* (properties of a page in the project itself, e. g. the current page number; 11042), *Symbol properties* (property of a symbol such as the displayed device tag; 20010), *Form properties* (properties for constructing a form and reporting data such as terminal and pin designations; 20030), *Plot frame properties* (e. g. the search direction for transferring the device tag, whether the device tag should be automatically transferred from the first device to the left; 12103), and numerous other function properties.

2.5.1 Project properties

Project properties can also be described as global properties. These properties can be used everywhere as special text (project property), in graphical editing for example via the INSERT/SPECIAL TEXT/PROJECT PROPERTIES menu item. The **Special text – Project properties** dialog will open.

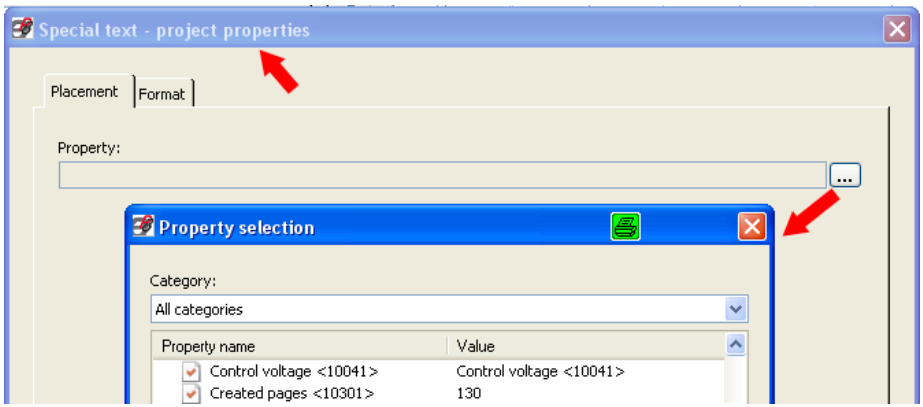


Fig. 2.41
Selecting and inserting a project property

Clicking the ... button opens the **Property selection** dialog. Here you can select the desired project property and accept it by clicking **OK**. The project property is loaded into the Property field of the **Special text – Project properties** dialog.

Fig. 2.42 Transferred project property

The special text can then be formatted (font, width, italics, etc.) on the **Format** tab. When you have finished entering data, you confirm by clicking **OK**. The project property now hangs on the cursor and can be placed.

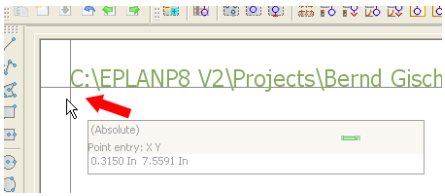


Fig. 2.43 Placing project properties

Project properties can be added to the project via the graphical button in the project properties. Use the **PROJECT/PROPERTIES** menu to open the project properties. Now click the button to open the **Property selection** dialog.

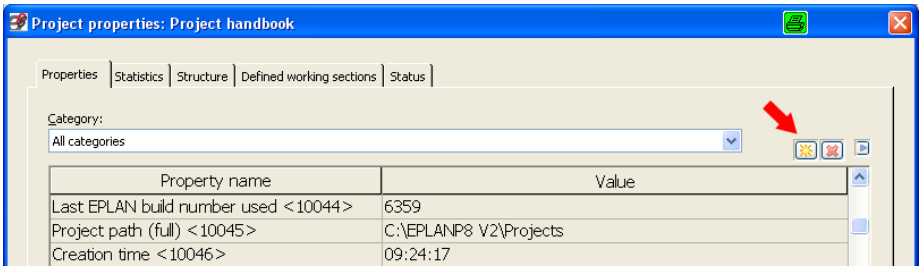


Fig. 2.44 Add project property

In the **Property selection** dialog, the usual Windows functions can be used to select one or more properties and these are then loaded into the project properties of the project by clicking the **OK** button.

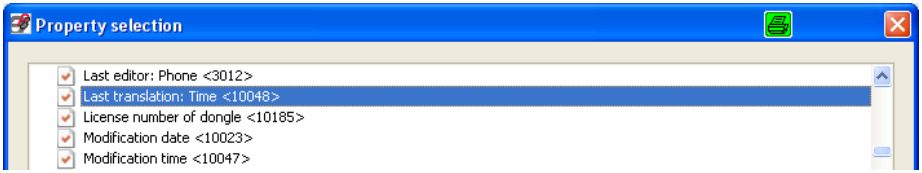


Fig. 2.45 Property selection

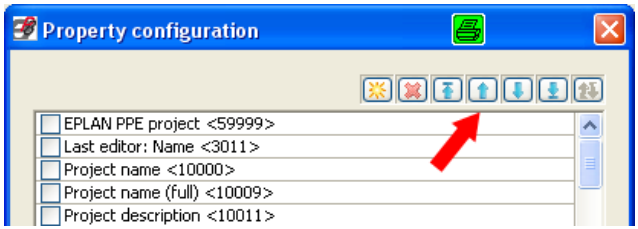


Fig. 2.46 Property configuration

You can use the right-click popup menu or the graphical button to sort the display of the project properties. Select the **CONFIGURE** command in the popup menu. The **Property arrangement** dialog opens. The graphical buttons can be used to arrange the properties as you wish.

2.5.2 Page properties

EPLAN **page properties** are those properties belonging to a page. Page properties can also be used everywhere. The **INSERT/SPECIAL TEXTS/PAGE PROPERTIES** menu item in graphical editing opens the **Special text – Page properties** dialog. In the *Property* field, you can use the ... button to open the **Property selection** dialog.

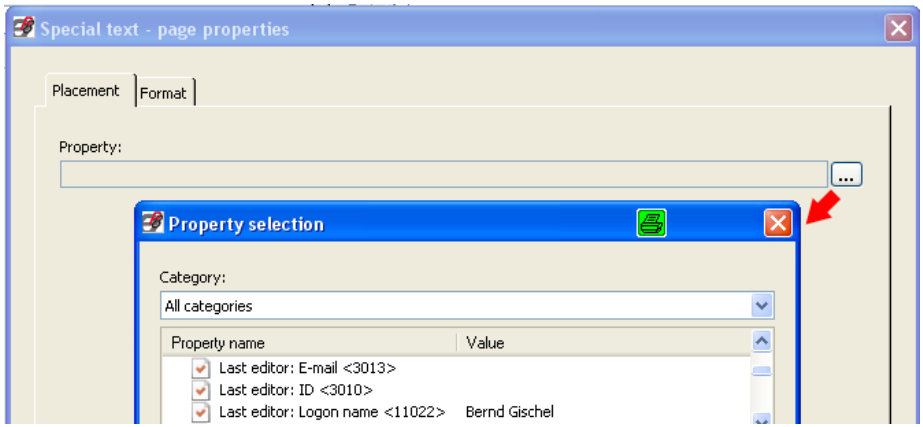


Fig. 2.47
Add page properties

Then you select the desired property. When you click **OK**, the selected property is loaded into the *Property* field of the **Special text – Page property** dialog.

After formatting the text (if necessary, on the *Format* tab), you close the **Special text – Page properties** by clicking **OK**. The special text now hangs on the cursor and can be placed as desired.

Fig. 2.48
Placing a page property

2.5.3 Symbol properties (components)

Symbol properties are properties that are assigned to special symbols (components). Some symbol properties, such as connection point designations or device tags, can be accessed directly.

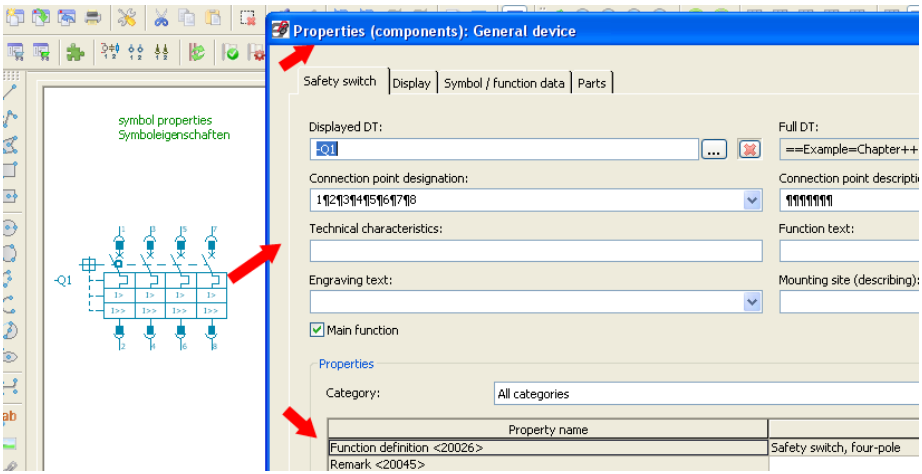


Fig. 2.49 Properties of a symbol (component)

New properties are added in the same way as other properties (project properties or page properties). You click the graphical button on the *Property (device type)* tab. EPLAN opens the **Property selection** dialog. Here you can select the desired property or properties to be used in the symbol properties and confirm by pressing **OK**. The property has now been added.

It is also possible to adjust the sequence in which properties are displayed in a symbol. To do this, you either right click and use the popup menu or click the button. Select the CONFIGURE entry in the menu that opens up. EPLAN opens the **Property arrangement** dialog. Here you can now use the familiar graphical buttons to move or sort the properties.

2.5.4 Form properties

In EPLAN, **Form properties** are the properties of a form. Forms (reports) are distinguished by report type. Some properties are shared by all types of forms, but there are also properties that apply only to specific report types. Form properties can only be edited in the **form editor**.

2.5.5 Plot frame properties

Plot frame properties are properties assigned to the structure and reports of normal page types, such as the schematic multi-line *page type*. An example of this is the *Path areas* property and its size. Plot frame properties can only be edited in the **plot frame editor**.

2.6 Buttons and popup menus

EPLAN has many dialogs. This is not unusual in such an extensive program. At this point, I would like to draw special attention to a number of dialog elements that occur repeatedly in many dialogs. These dialogs are similar in many different places in EPLAN. It is therefore important to know what they mean and how they can be used.

Graphical buttons play an important role because they provide easy access to most of the functions and procedures in EPLAN.

2.6.1 Device dialog buttons

Apart from a few small details, all EPLAN device dialogs are very similar. This means that they always have the same structure, regardless of whether it is a device dialog for a motor overload switch or a transformer or any other component.

EPLAN uses different buttons in different tabs to simplify operation. Here is a list of the most commonly used buttons and what they mean.

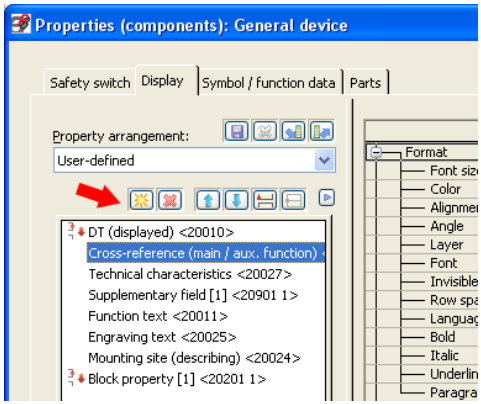


Fig. 2.50
Various buttons

	Add new property to property arrangement
	Remove property from property arrangement
	Move property up one place
	Move property down one place
	Undock property (release)
	Dock property (link to upper area)
	Save
	Delete
	Import
	Export

2.6.2 Buttons in dialogs (configuring)

You can use the **CONFIGURE** menu item to better arrange or re-sort properties in dialogs to suit your personal needs.

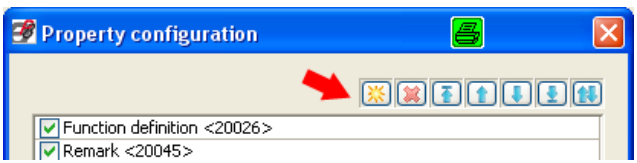


Fig. 2.51 Buttons in the property configuration dialog

These can be page properties and also settings in other dialogs such as the **UTILITIES/GENERATE REPORTS/SETTINGS/OUTPUT TO PAGES** dialog. The number of buttons can vary here. In principle, the meaning/function of the buttons remains the same.

	Move the selected entries to the start of the list
	Move the selected entries to the end of the list
	Swap two selected entries (swap positions)

In contrast to the graphical buttons in the device dialog, multiple entries can be selected here. The functions provided by the other graphical buttons have already been described.

2.6.3 Buttons in dialogs such as filter schemes

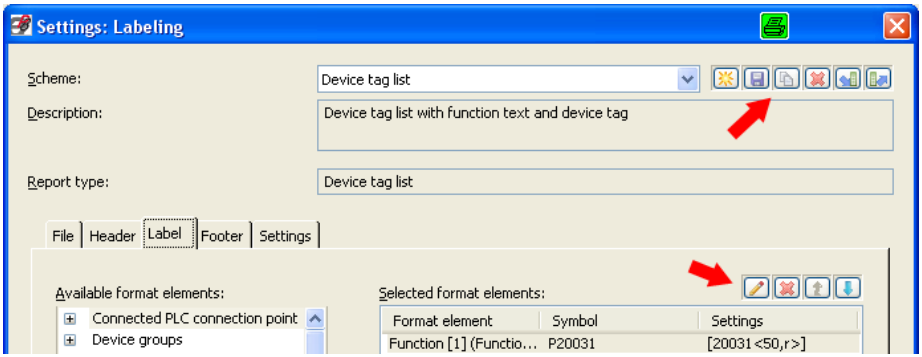


Fig. 2.52 Buttons in various settings dialogs

In addition to the already familiar buttons, dialogs such as **Filter, Sorting** and other settings dialogs have additional graphical buttons.

Additional buttons in the **Filter** or **Sorting** dialogs:

Edit entry

Copy entry

2.6.4 Small blue triangular button

As soon as the button appears in any dialog, you can click the right mouse button to call up a popup menu. Clicking the button itself displays the same popup menu.

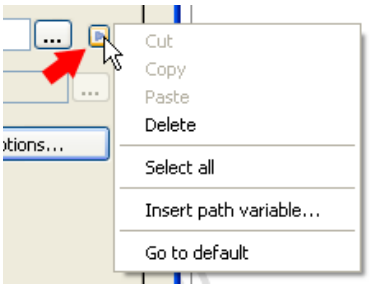


Fig. 2.53
Blue button

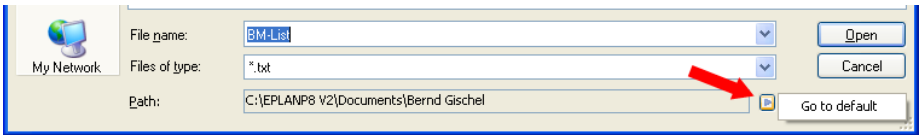


Fig. 2.54 Other function options

Depending on the dialog, the popup menu "behind" the blue button can have different functions. It is very useful in dialogs that open files or directories. Here you can quickly set the default directories, without having to click all the way through to the default directory.

2.6.5 Property arrangements (components)

EPLAN always offers a *Default setting* for property arrangements. The default setting is defined in the symbol structure and cannot be changed here. To change these default settings, you have to edit the symbols. I recommend that you do not make any changes to the symbol libraries that came with EPLAN since changes may make them incompatible with newer versions of EPLAN symbol libraries, making synchronization impossible.

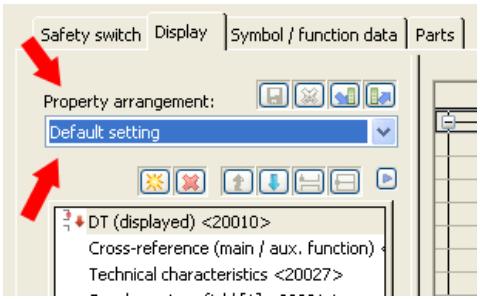


Fig. 2.55
Default setting

You can also change these property arrangements. Once they have been changed, they initially appear in the property arrangement with the designation *User-defined*.

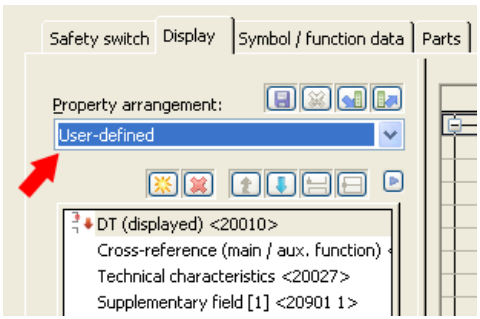


Fig. 2.56
User-defined arrangement

To undo this user-defined property arrangement, simply select the *Default setting* from the *Property arrangement* selection list. This returns all manually changed settings on a symbol back to the default values defined for the symbol.

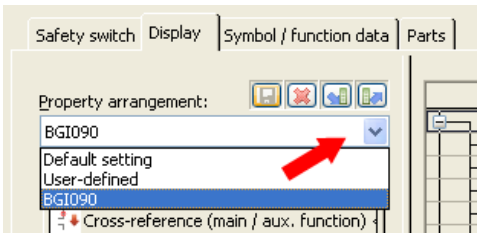


Fig. 2.57
Undo

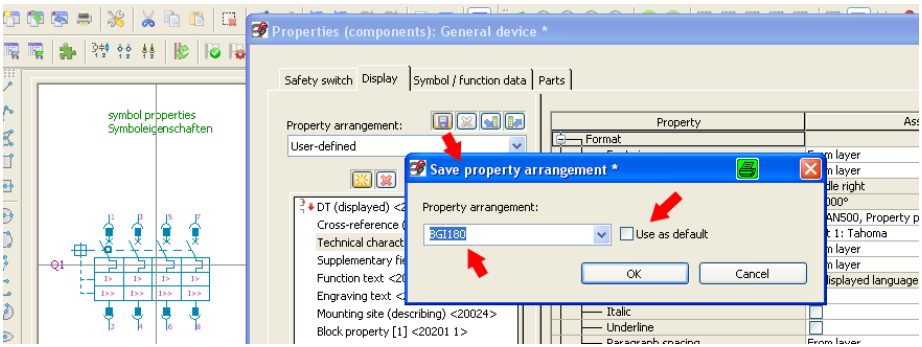


Fig. 2.58
Saving your own property arrangement

If a property has been changed, then EPLAN no longer regards it as a default value and the display in the *Property arrangement* field changes to *User-defined*. This user-defined arrangement can also be saved via the button and then called up or set at similar symbols. You can assign a descriptive name to the property arrangement. After you click **OK**, this property arrangement is saved, entered into the selection field and is then available to all symbol variants.

Property arrangements can only be transferred to the same "types" of devices, i. e. the property arrangement of a safety switch cannot be transferred to the property arrangement of a terminal.

The *Use as default* option allows EPLAN to immediately (the next time you insert the same symbol) and automatically use your own property arrangement.

2.6.6 Format properties

EPLAN makes it easy to use different formats, e. g. for texts. Though the dialogs may be somewhat different, the basic principle always remains very similar.

A normal text dialog is used as an example here. To insert a text, you use the **INSERT/GRAPHIC/TEXT** menu to open the **Properties** text dialog.

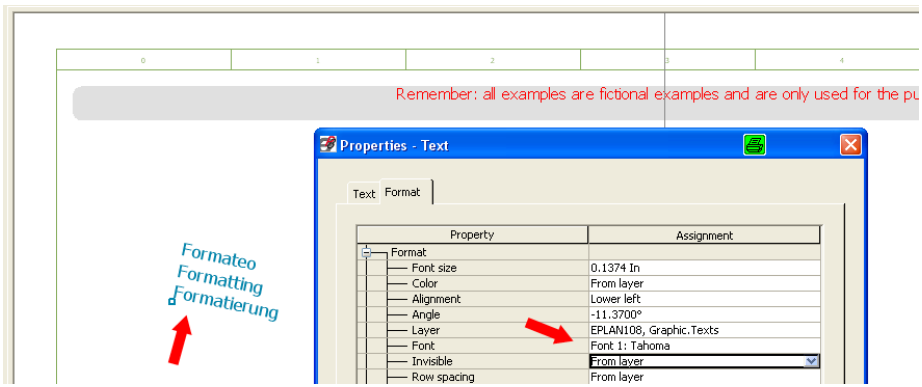
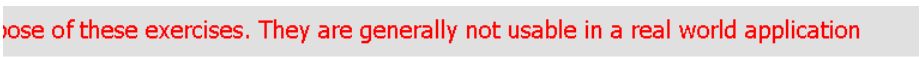


Fig. 2.59
Formatting options for texts (example)

You can now enter the desired text into the **Enter text** field on the *Text* tab in the opened dialog. To format the text according to your personal wishes, you need to switch to the **Format** tab. You can freely format the text you just entered. You can edit all selection fields or select default values.



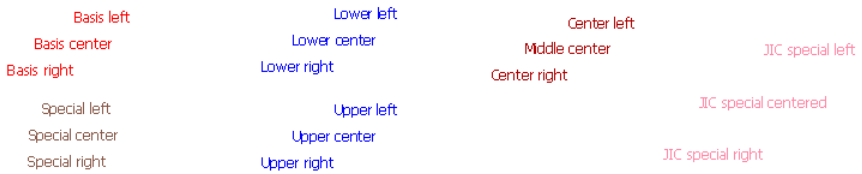


Fig. 2.60 Examples of different formats

This example shows how different the arrangement and appearance of a text can be if it is formatted using the default setting *Alignment* (the small black dots are the anchor points of the text).

2.6.7 Buttons (small black triangles)

In a number of dialogs, the buttons have a small black triangle. Additional menus are "hidden" behind these buttons.

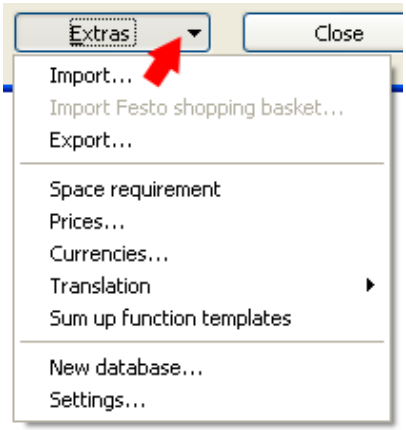


Fig. 2.61 Parts management

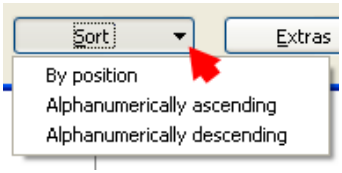


Fig. 2.62 Structure identifier management

2.6.8 Dialogs for schemes

Dialogs for schemes allow you to export or import created schemes.

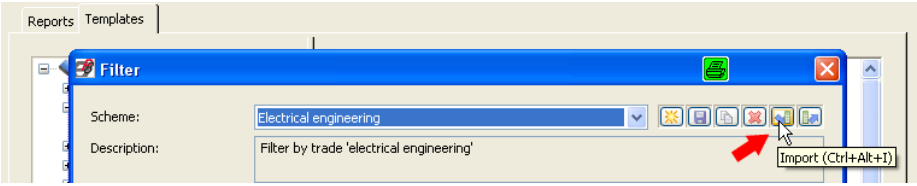


Fig. 2.63 Import and export options for schemes

TIP: User-created schemes should be regularly exported, preferably as soon as they have been created. These schemes are then always available for new versions, other projects or other workstations (depending on the installation).

In many areas, EPLAN offers a special identifier for the scheme type. This way, during an import, only those scheme types that can actually be imported into the current scheme are displayed.

CBnu..xml*	Scheme prefix for cable numbering
LB..xml*	Scheme prefix for labeling
WS..xml*	Scheme prefix for workspaces

2.7 Master data

EPLAN distinguishes between two types of master data: **system master data** and **project master data**.

System master data is stored in the directories with the associated user directory that were set during installation. In addition to the user-specific system master data, the original EPLAN system master data is also installed in the *EPLAN original master data directory* (depending on the directory selected during the installation).

It should be noted that on installation, only the original EPLAN system master data is overwritten or updated in the \EPLAN directories. In case of a new installation or installation of an update, the user-specific system master data is **not** overwritten or updated. Generally, though, it is possible to synchronize the user's own system master data with the original EPLAN system master data. To do so, in the **UTILITIES/MASTER DATA** menu,

you call up the **SYNCHRONIZE SYSTEM MASTER DATA** menu item. In the following dialog, you can then update your own system master data accordingly.

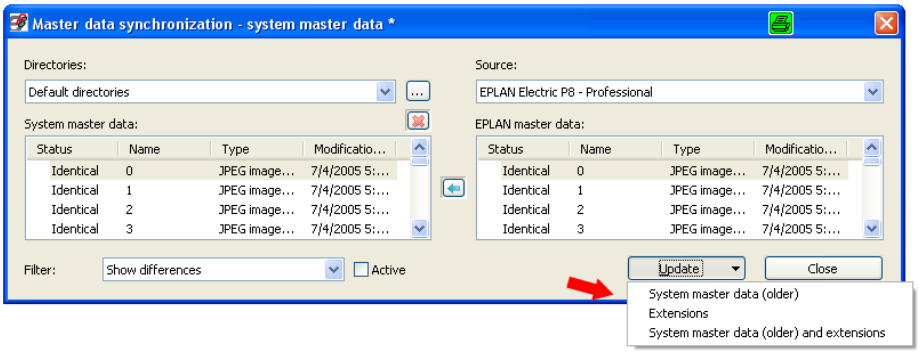


Fig. 2.64 Master data synchronization

If, for example, you have modified the supplied table of contents form *F06_001.f06*, EPLAN would not replace this form in the user-specific master data directories *\\Forms\Company code* with an updated, possibly improved, version. It will only overwrite this form in the *\\Forms\EPLAN* directory.

System master data contains data such as symbol libraries with the associated symbols, function definitions, forms and plot frames.

Project master data is the other type of master data. Project master data, after initial use, is moved from the system master data and stored in the project; after it is stored it is independent of the system master data.

2.8 Operation

EPLAN can be conveniently operated using the mouse. However, to increase your own working speed, EPLAN is very flexible in allowing functions accessed with the mouse to also be accessed via definable keyboard shortcuts. Normally functions can be assigned to a keyboard shortcut if they are also accessible in the main menus, such as Page, Project etc.

To translate text in a Text dialog, you can select the appropriate command with the mouse via **UTILITIES/TRANSLATION**. This function can be assigned a keyboard shortcut because it is a normal menu command. One possibility is the key combination **CTRL + T**.

2.8.1 Using the keyboard

EPLAN allows the user a great deal of freedom in assigning keyboard shortcuts to functions and commands.

NOTE: This is one limitation here: Certain shortcuts are often only permitted in combination with the **CTRL** and/or **ALT** keys.

Selected shortcuts

Of course, important standard Windows shortcuts such as **CTRL+ C (COPY)** or **CTRL + V** (paste) should **not** be reassigned if at all possible. It is possible, but not recommended.

Here are a number of recommended keyboard shortcuts that have proven useful in practice.

Copy page from/to	**CTRL + SHIFT + P**
Edit/Copy format	**ALT + F**
Edit/Assign format	**ALT + A**
Utilities/Reports/Generate	**CTRL + ALT + G**
Find/Synchronize selection	**CTRL + SHIFT + Y**
Project data/Device navigator	**STRG + SHIFT + D**
Project data/Update connections	**F11**
Utilities/Parts/Management	**ALT + V**
Utilities/Translation/Dictionary	**ALT + T**

To assign your own shortcut to functions, in the **OPTIONS/SETTINGS** menu, select **USER/MANAGEMENT/SHORTCUT KEYS**.

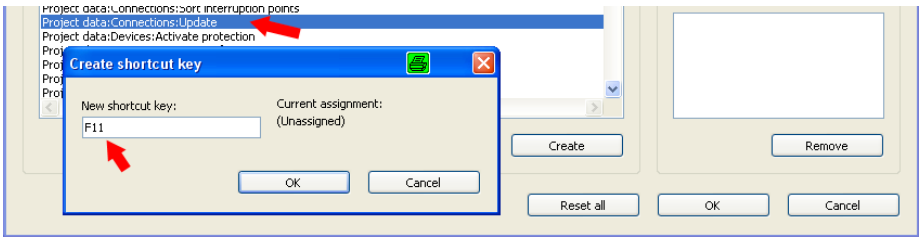


Fig. 2.65 Assigning a shortcut key to a function

In this dialog, you select the desired action and assign it a shortcut key using the **CREATE** button. The shortcut key is applied and fully "operational" as soon as you click **OK**.

Fig. 2.66 Assigned shortcuts in the Connections menu

2.8.2 Using the mouse

EPLAN can be completely operated using the mouse. Admittedly, many menu items are easier to reach with the mouse than with cryptic key combinations that one usually cannot remember.

2.9 The user interface – more useful information

EPLAN provides a great deal of support to the user when editing projects, but also for general operation of the program.

A very nice and practical feature is the ability to freely configure the EPLAN user interface. The so-called workspaces are defined depending on the logged-in EPLAN user. This makes it possible to always call up your own user interface configuration when you start up EPLAN on the same computer.

2.9.1 Using workspaces

Put simply, a **workspace** is the dialog layout and the toolbars you want to have when performing certain operations in EPLAN. EPLAN allows you to create separate workspaces for particular areas that contain exactly these desired toolbars, views or dialogs.

The Workspace function is accessed from the graphical editing menu via VIEW/WORKSPACE. When you select this function, EPLAN opens the **Workspace** dialog and you can use the selection field to select one.

2.9.1.1 Creating and selecting a workspace

A workspace is customized by placing desired dialogs or toolbars to suit your personal preferences. To permanently save a new workspace, you open the **Workspace** dialog via the VIEW/WORKSPACE menu.

A new workspace is created via the button. EPLAN then opens the **New scheme** dialog. You should enter a name in the empty *Name* field and a sensible description for the workspace in the *Description* field.

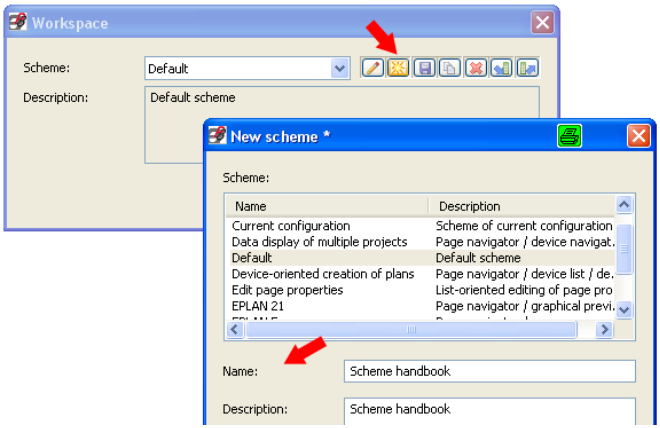


Fig. 2.67
Creating a workspace

When you click **OK**, the workspace is saved and can then be set as the current workspace.

2.9.2 Dialog display

EPLAN provides a lot of data and additional information. EPLAN has a number of different dialog representations in the navigators to ensure information is clearly displayed.

2.9.2.1 Tree view

The **tree view** is one type of representation. Similar to Windows Explorer, all information is displayed in a tree with small symbols.

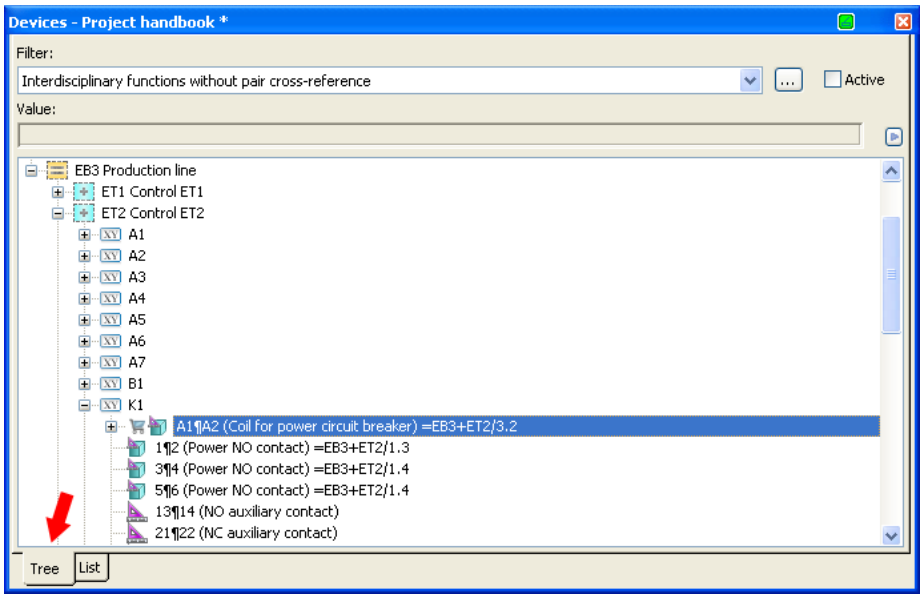


Fig. 2.68
Tree view

This representation type provides a clear overview, making it relatively easy to find objects. In this representation, EPLAN always shows all devices (depending on the function definition). No changes to the form and content of the tree view are possible. EPLAN provides no options for this here.

The display sequence of the devices can be configured under **OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/DISPLAY/PROJECT STRUCTURE (NAVIGATORS)** and/or **PROJECT STRUCTURE (PAGES)**. This is where you can define how devices should be shown in the tree view, for example, by identifier or by the page prefix.

2.9.2.2 List view

The **list view** is another representation type. This representation type offers many more customization options to suit your working habits. As with the tree view, EPLAN shows all devices here as well (again depending on the navigator that was selected). The cable navigator shows only cables, the terminal navigator shows only devices with the terminal function type, etc.

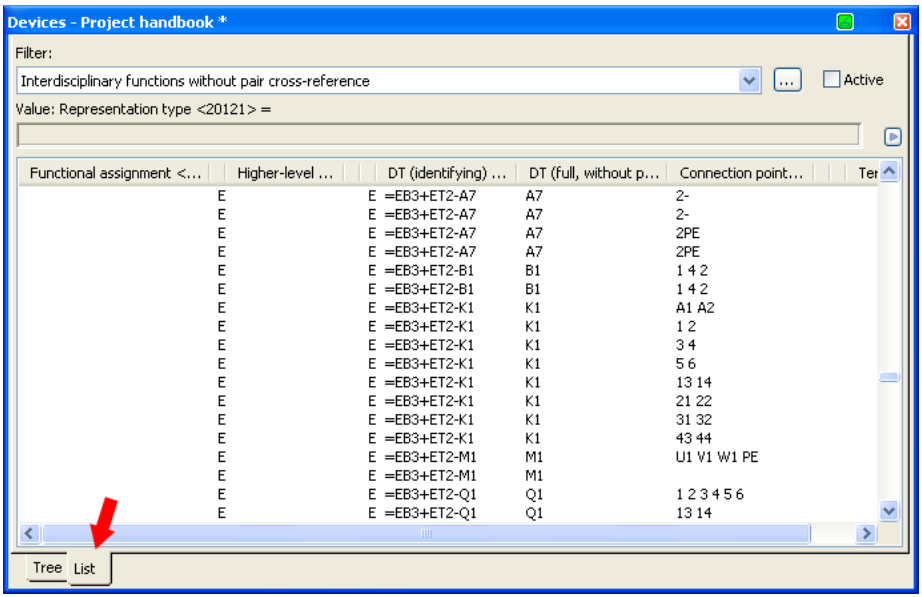


Fig. 2.69 List view

Unlike the tree view, it is possible to add further information to the list view. To add more information, you right click to call up the popup menu or press the ▶ button and then select **CONFIGURE COLUMNS**. EPLAN opens the **Column configuration** dialog. You can now select or deselect columns and change their order. Click the **OK** button to save the settings. You now have a personalized list view.

2.9.2.3 Combined tree and list view (part)

In addition to the previous representations, the *Part master data* navigator also allows a combined display of list and tree view, the **combination** view.

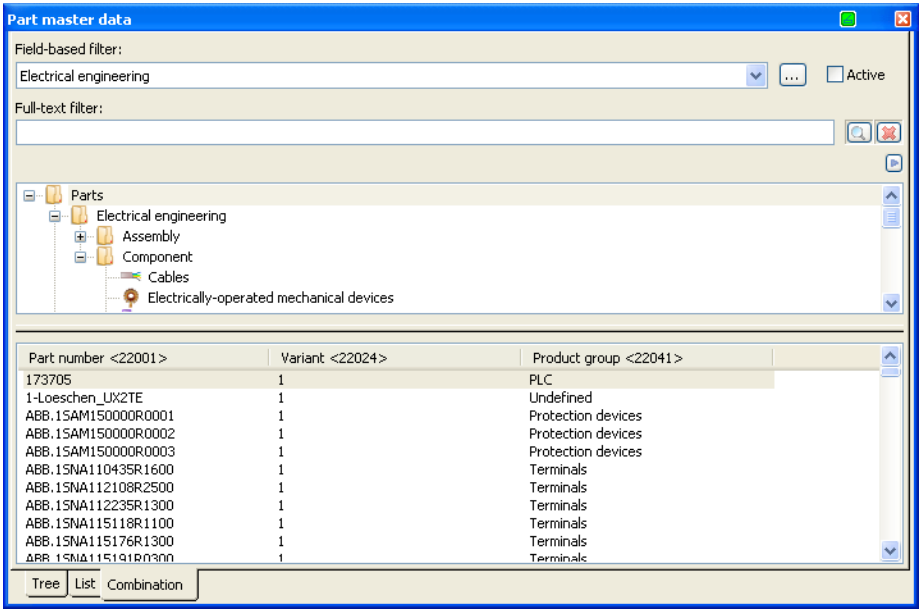


Fig. 2.70 Combined display

The tree view is displayed in the upper area. Here it is possible to preselect, for instance by selecting the *Cable* node. In the lower area, the preselected parts are shown in a clear list with additional specific information.

2.9.2.4 Edit in table

Edit in table is another method of conveniently displaying and editing properties. Edit in table is usually accessed from the tree view by right clicking to access the popup menu or via the button.

Unlike the previous representations (tree or list view), only the devices and their functions that were previously selected in the navigator are displayed.

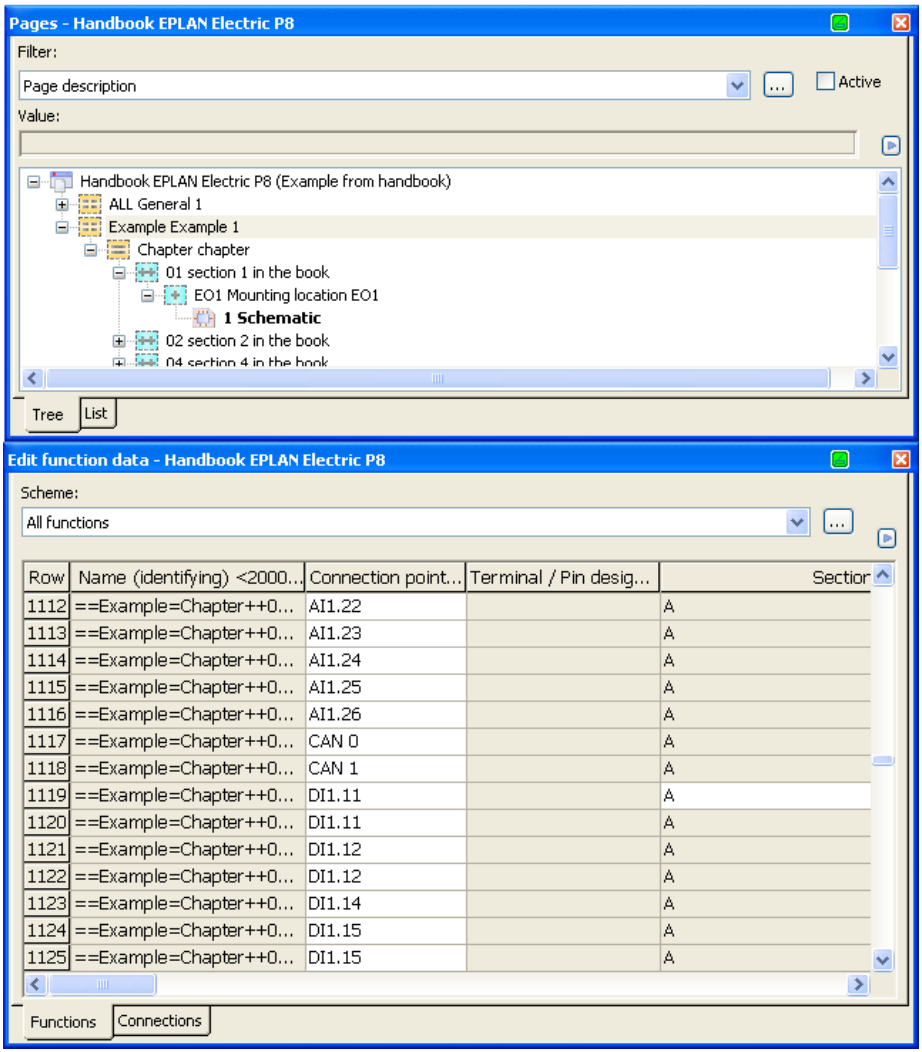


Fig. 2.71 Editing function data in table

The displayed devices can now be edited in a table in the **Edit function data** dialog. All editing functions, such as Copy, Paste and Edit, can be used when editing tables.

3 Projects

A project must first be created before you can edit schematics in EPLAN.

NOTE: In EPLAN, the storage location of projects and the names of the individual projects can be freely chosen. The only limitations are the naming conventions of the Windows system being used.

When the page and device structures are defined, a project has all the properties it needs, such as the *device structure* (the structure identifier, e. g. higher-level function or installation site, that is to be used) and also all the master data, such as forms, parts data, function definitions, symbol libraries, etc., that is necessary for editing projects.

Page structure of a project

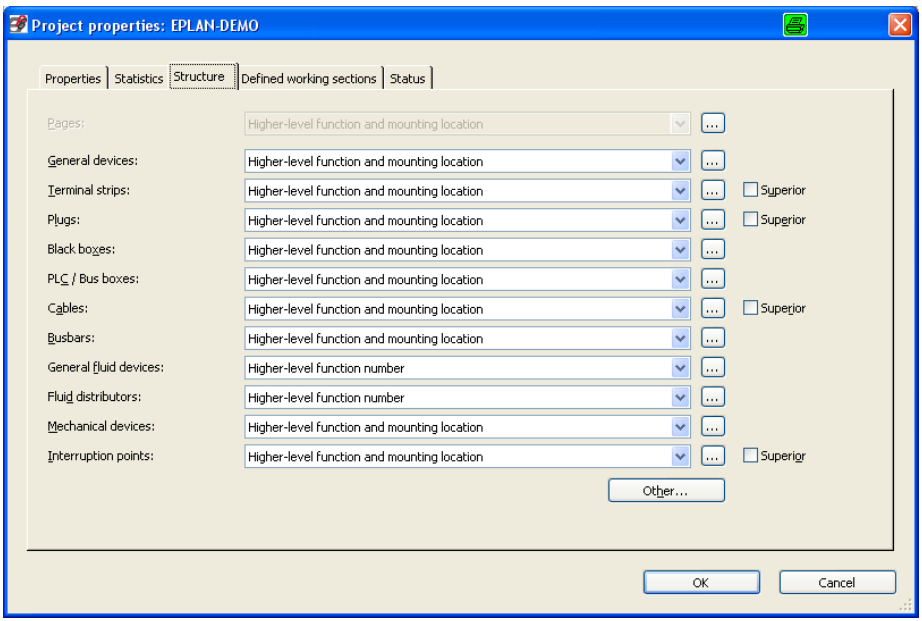


Fig. 3.1 Project properties – Structure tab

This data is all **completely** stored in the project. Data added later, such as forms, is also stored in the project.

This ensures that this project can later be edited with exactly the same data used when the project was created, or which was generated at the beginning of project editing, or was later stored in the project.

3.1 Project types

EPLAN basically distinguishes between two **project types**: a *schematic project* (this is the usual project in practice) and a *macro project*.

A *macro project* is used for creating and automatically generating window macros, and for managing macros. Logical functions such as cross-references or connection information are not supported in a macro project nor are they displayed.

If necessary, a *schematic project* can easily be modified by changing the *<10902 Project type>* property from *schematic project* to *macro project*.

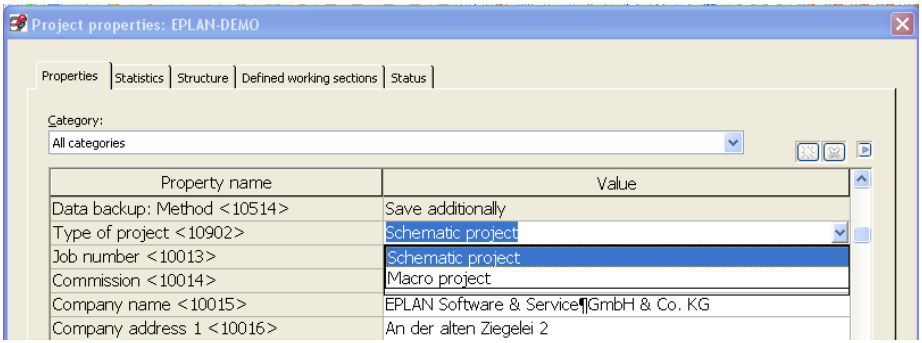


Fig. 3.2 Project properties – Properties tab

Other project types such as form project or symbol project do not exist. Master data (e.g. forms or symbols) is always temporarily edited directly in the project (with existing system master data), stored after the form is closed, and then possibly automatically (depending on the settings) synchronized with the current project (depending on the settings and usage).

And of course the system master data can also be manually synchronized with the project master data. It is also possible to synchronize in the other direction, to synchronize the project master data with the system master data. If EPLAN finds inconsistencies in the master data at this point, then a message is generated and EPLAN cancels the synchronization because incompatible master data cannot be used to overwrite existing master data. This is the case, for example, when connection points are added to existing, placed

symbols. The logical data of this symbol no longer "fits" already placed or existing symbols and therefore this data cannot be synchronized.

An EPLAN project consists of a directory **Projectname.edb** and an operation **Projectname.elk** (the *.elk file extension indicates a normal editable schematic project).

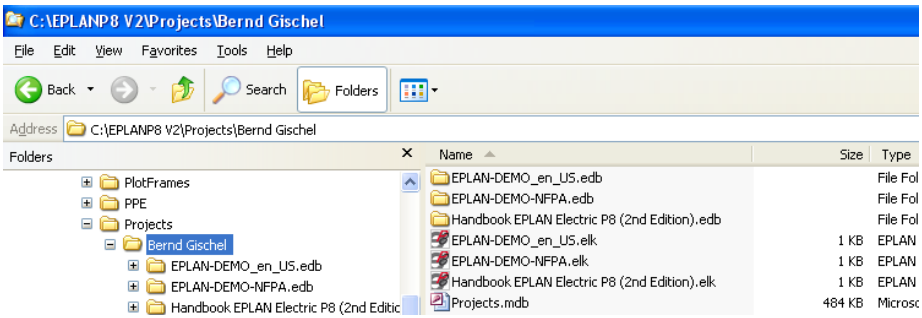


Fig. 3.3 Project directory with projects

You can also start the project from the file manager (e. g. Explorer) by double clicking the *Projectname.elk* link. Another possibility is to drag *Projectname.elk* into the page navigator while holding down the left mouse button. EPLAN then opens the project.

3.1.1 Project types in EPLAN

Normal schematic projects in EPLAN are subdivided into different **project types**.

EPLAN manages different project types as schematic projects.

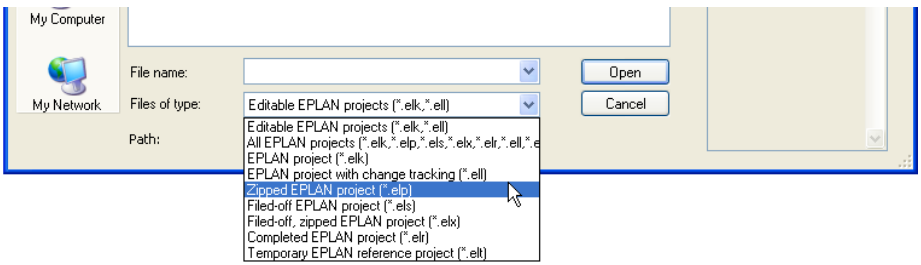


Fig. 3.4 Open project dialog with various file types

Project types define the project by its functional meaning, for example, as a normal project or a revision project. EPLAN distinguishes between the project types described in the following section.

The most important project for the user is the “normal” EPLAN project (*.elk indicates a schematic project). EPLAN has the following project types:

- ***.elk: a normal editable project (normal EPLAN schematic project)**
- ***.ell: a project with change tracking**
- ***.elp: a zipped project**
- ***.els: a filed-off project**
- ***.elx: a filed-off and zipped project**
- ***.elr: a completed project**
- ***.elt: a temporary EPLAN reference project** (comparison)

Central project editing occurs in a normal project. All other types of projects (e.g. a completed project, an archived project, a basic project, or a project template) are derived from the normal EPLAN project (schematic project).

3.1.2 Project templates and basic projects

EPLAN allows the user to quickly and precisely create new projects based on existing *basic projects* and *project templates*.

A project template contains preconfigured values. Later, when using the template for a new project, the project and page structure can be changed **one time**.

Opening a projekt template

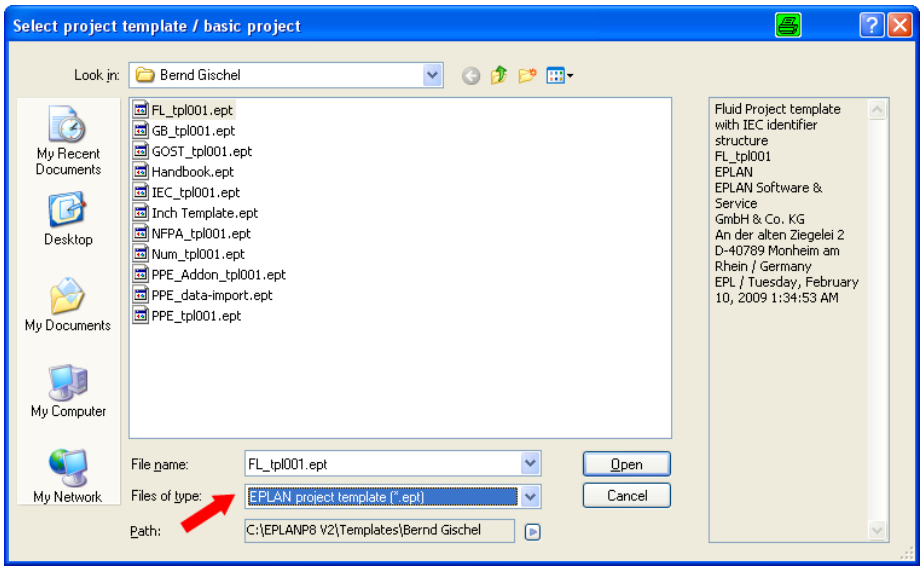


Fig. 3.5 *.ept project template

In contrast to a basic project, a new project created from a project template (via **PROJECT – NEW (WIZARD)**) usually has no pages because the page structure still has to be defined. But project templates can also contain pages.

A *project template* has the ***.ept** file extension (and ***.epb** – old project templates from previous program versions and should not be used) and cannot be directly opened or modified. You can, however, easily overwrite an existing project template that has incorrect or changed properties with a new project template.

Project templates can be generated from existing projects in the graphical editor via the **PROJECT/ORGANIZE/CREATE PROJECT TEMPLATE** menu. In the page navigator, you first use the mouse to select the project that is later to become a project template and then open the **PROJECT/ORGANIZE/CREATE PROJECT TEMPLATE** menu.

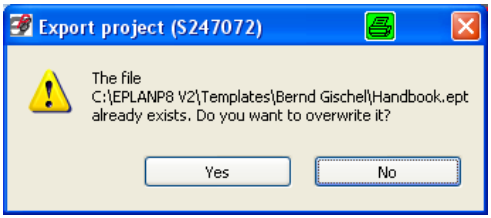


Fig. 3.6 Overwriting a project template

In the **Create project template** dialog that is displayed, you need to define the storage location and the name of the new project template. It is a good idea to create a *project templates* directory below the root directory, possibly with separate folders for each customer.

Basic projects are, for example, projects prefilled with appropriate customer values such as a predefined page structure, sample pages, graphical report templates, various master data, and much more.

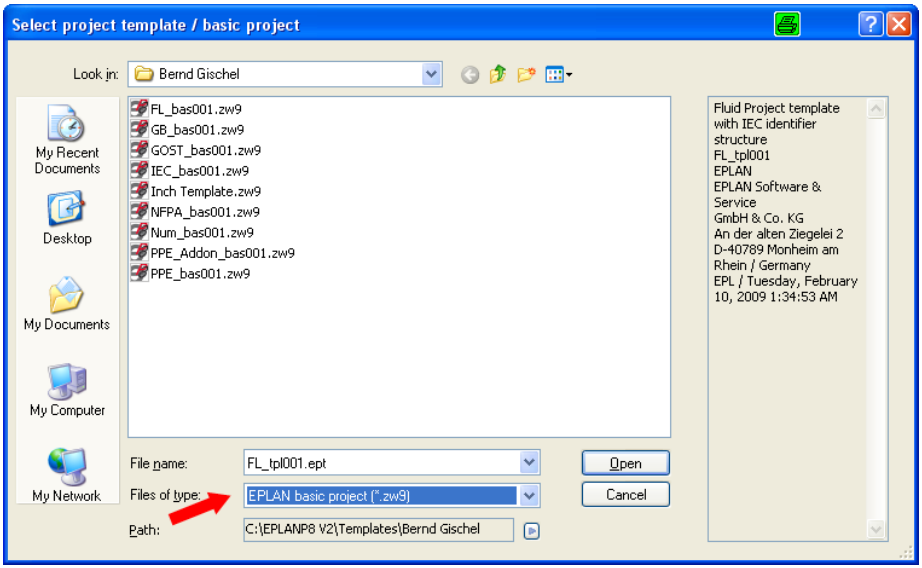


Fig. 3.7 Opening a basic project

In a project created from a basic project, the project and page structures are fixed and can no longer be modified.

A basic project has the ***.zw9** file extension and, like project templates, cannot be directly opened. Basic projects, like project templates, can be generated from existing projects in the graphical editor via the **PROJECT/ORGANIZE/CREATE BASIC PROJECT** menu.

EPLAN then executes a number of functions to create the basic project. Afterward, the **Create basic project** dialog opens and this is where you define the directory and the project name of the basic project to be created.

Here too it is a good idea to create a **basic projects** folder below the root directory, possibly with separate customer folders.

Once basic projects or project templates have been created, they cannot be later changed (i. e. directly edited). However, they can be overwritten with new or modified data.

3.2 Creating a new project

To quickly get started with a project, new projects can be generated in various ways in EPLAN. There are two ways of doing this directly from *project editing*.

A new project based on an existing basic project or project template can be created "at the push of a button" via the **PROJECT/NEW** menu.

EPLAN comes with several basic projects and project templates. The project template *IEC_tpl001.ept*, for example, contains the IEC identifier structure, and the supplied project template *Num_tpl001.ept* contains a sequential numbering structure.

Selecting **PROJECT/NEW (WIZARD)** generates a new project in several steps, each of which requires you to enter certain data (EPLAN provides detailed information at the bottom of each dialog).

Basic projects (*.zw9) and project templates (*.ept. and *.epb) can be used to create a new project.

The third way to create a new project is via the optional project management. A new project based on an existing basic project or project template can be created here just as fast as via the **PROJECT** menu.

Of course it is also possible to simply copy a project and give it a different name.

3.2.1 New project (from a basic project)

You can either select the menu item **PROJECT/NEW** or use your own keyboard shortcut to create a new project.

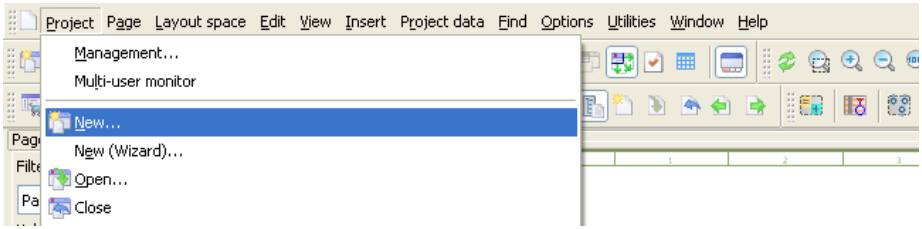


Fig. 3.8 Project/New menu

The **Create project** dialog is displayed. In this dialog, you must/can define the following settings:

- Project name (you need not worry about file extensions; these will be automatically added by EPLAN)
- Storage location
- Creation date
- Creator

After defining the above entries, you need to select a template. This can be a project template or a basic project.

You simply click the More button. The **Select project template/basic project** dialog opens. In this example, the basic project **Handbook.zw9** will be used to create the new project. You can select the relevant project type in the *File type* field.

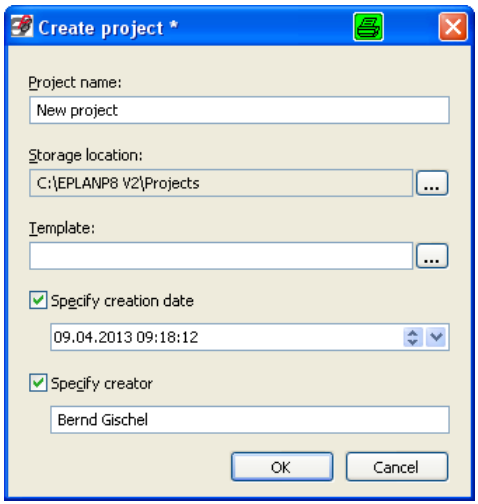


Fig. 3.9
Create project dialog

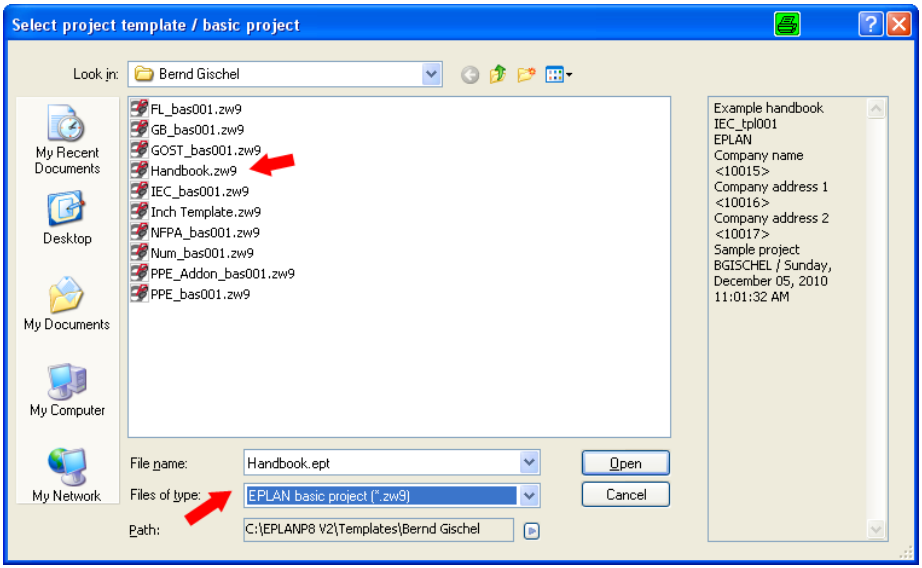


Fig. 3.10 Select project template/basic project dialog

Creating a new project based on a basic project

In the **Select project template/basic project** dialog, you now press the **OPEN** button and select and use an existing basic project as your template.

EPLAN then returns to the **Create project** dialog and imports the selected basic project (or the template project) into the *Template* field.

Now, you only have to confirm this dialog by clicking **OK** and EPLAN will generate the new project in the specified directory.

The new project is created from the selected basic project. This may take a while depending on the hardware and the storage location (server, local).

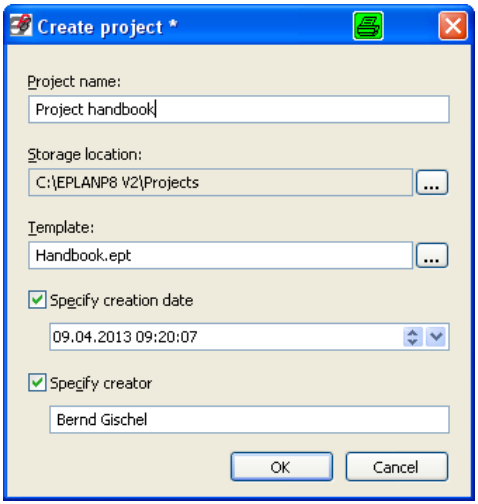


Fig. 3.11
Create project dialog, with the template

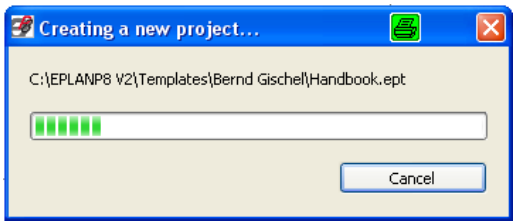


Fig. 3.12
The new project is being generated

Once EPLAN has successfully generated the data for the new project, the **Project properties** dialog is displayed. You do not necessarily need to edit these at the moment. This can be done later during project editing.

EPLAN opens the **Project properties** dialog with the *Properties* tab. The project properties can now be adjusted or completely changed on the *Properties*, *Structure* etc. tabs.

There is one limitation: The *structure of the pages*, located in the *Structure* tab, can no longer be changed (grayed out). It is fixed because the page structure was defined in the basic project.

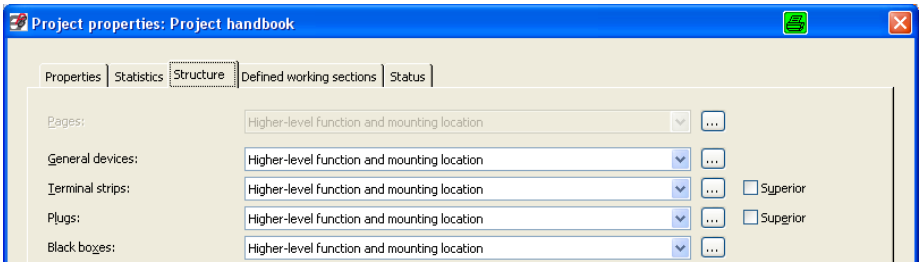


Fig. 3.13 Project properties – Structure tab

When you click **OK**, the project properties are saved and the project is immediately opened in the page navigator (it can also be opened using the **F12** key or you can display the overview with the **PAGE/NAVIGATOR** menu item). The project can now be edited.

3.2.2 New project (with a project creation wizard)

You can either select the menu item **PROJECT/NEW (WIZARD)** or use your own keyboard shortcut to create a new project.

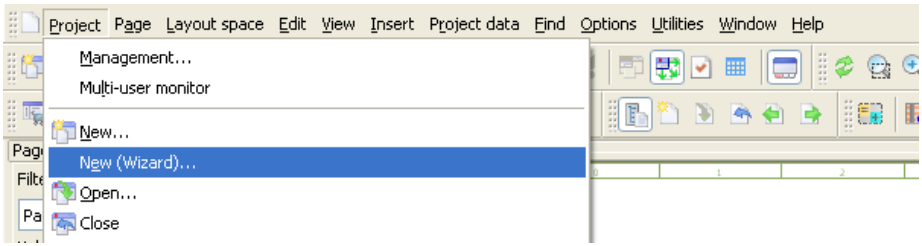


Fig. 3.14
Project/New (Wizard) menu

The wizard opens the **Create project** dialog. This dialog contains several tabs (*Project*, *Structure* or *Numbering* and *Properties*). You must enter data into at least the *Project* tab (the PPE tab is only visible when you have a license for this).

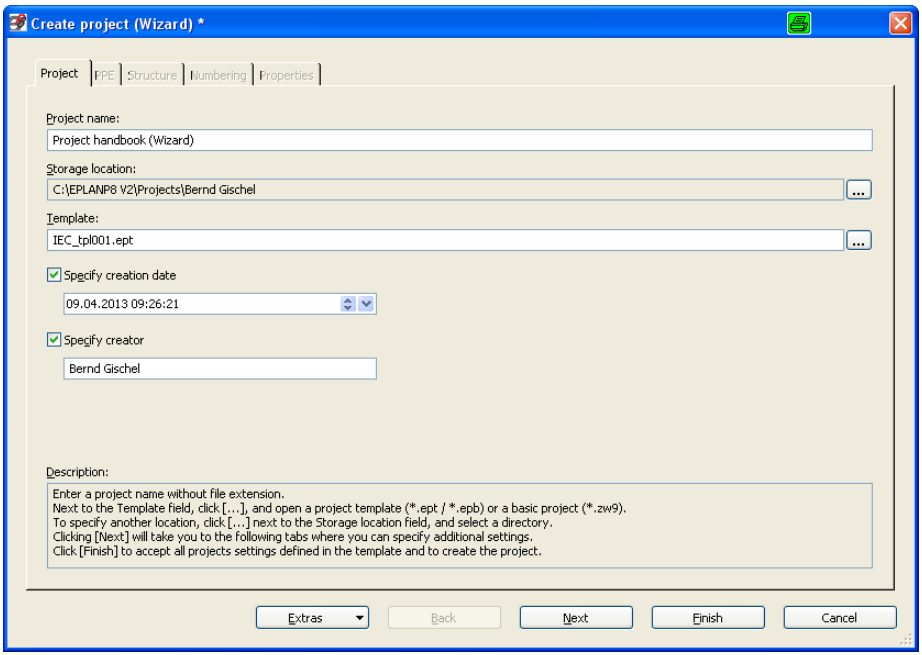


Fig. 3.15 Creating a project using the wizard

Creating a new project with the wizard

The lower area of the **Create project** dialog contains five buttons (**EXTRAS**, **BACK**, **NEXT**, **FINISH**, and **CANCEL**) whose meaning is described below.

The **EXTRAS** button allows you to view a summary of the project in the browser. However, this only functions after the first *Project* tab has been successfully completed.

The **BACK** button returns you to the previous tab. Here too, this only functions when all necessary information has been entered in the *Project* tab and EPLAN has created the project, or if you can move to the next tab by pressing **NEXT**.

Once all necessary entries have been made on the current tab, the **NEXT** button becomes enabled and you can switch to the next tab.

The **FINISH** button has a special function. It allows the project creation to be finished without making any further manual entries. A condition is that a new *Project name*, a *Template* (template project or basic project) and *Storage location* have been defined in the new project's *Project* tab.

The **CANCEL** button is used to stop and exit the **New project** wizard at any time.

Back to the tabs: The *Project* tab is the main core of the project wizard. The information required on this tab is mandatory for creating a new project.

TIP: All the other tabs can be edited, but this is not compulsory when creating a project via the wizard. These tabs can be edited or changed later (**Exceptions:** the *Structure* tab and the *Pages* selection field cannot be changed at a later point in time).

The *Project* tab has the following mandatory fields:

- **Project name** input field: The name of the new project is entered here.

NOTE: The project name is not checked to see if it already exists until a template (project template or basic project) has been selected and the **NEXT** button has been clicked. The action can then still be cancelled.

EPLAN always suggests the project name *New name,* possibly also with a consecutive number. This can, of course, be changed to any other name.

- **Template** selection field: You select a template using the ... selection button. The usual range of templates and basic projects are available for selection.
- **Storage location** selection field: Specifies the storage location of the new project. Any storage location can be selected using the ... button.

EPLAN requires no information other than these three entries. When you click the **FINISH** button, the wizard can immediately create the new project. In this case (clicking the **FINISH** button), EPLAN would just use the settings on the other tabs for the new project, generate the project, and close the project wizard.

You can, but do not have to, fill in the information under *Specify creation date* and *Specify creator*.

NOTE: These fields cannot be changed later on. It is therefore very important to make sure you enter the correct data.

When you click **NEXT**, EPLAN generates the basic elements of the new project based on the selected template. The **Import project** dialog is displayed. The process can be cancelled here if desired. Depending on the amount of data, the import may take a while.

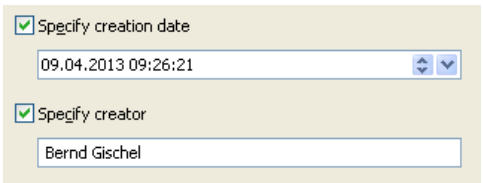


Fig. 3.16
Information on creation date and creator

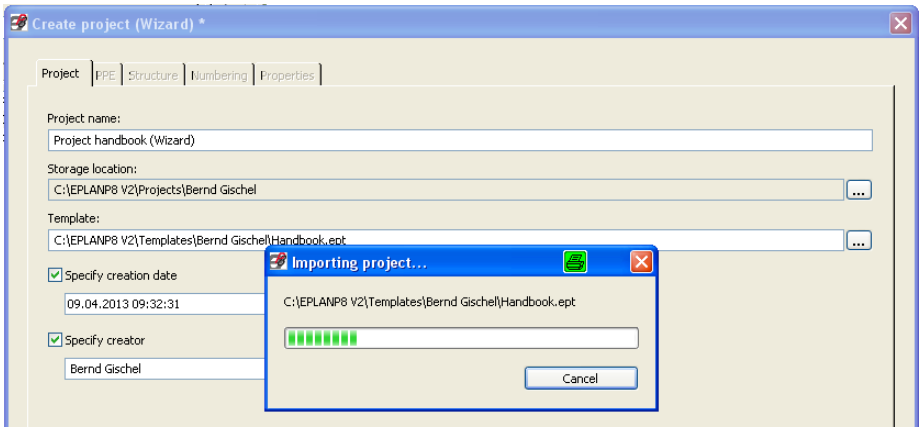


Fig. 3.17
The wizard is creating the new project

EPLAN then remains at the *Project* (or *PPE*) tab. You can still select, for instance, a different template project or storage location by clicking the **BACK** button.

On the *Structure* tab, you define the subsequent page structure and the structure of the individual device groups for the project.

NOTE: Once you have saved the structure set for the pages, it cannot be changed. This applies only to the structure settings of pages. All other structure settings, such as general devices, can be changed later.

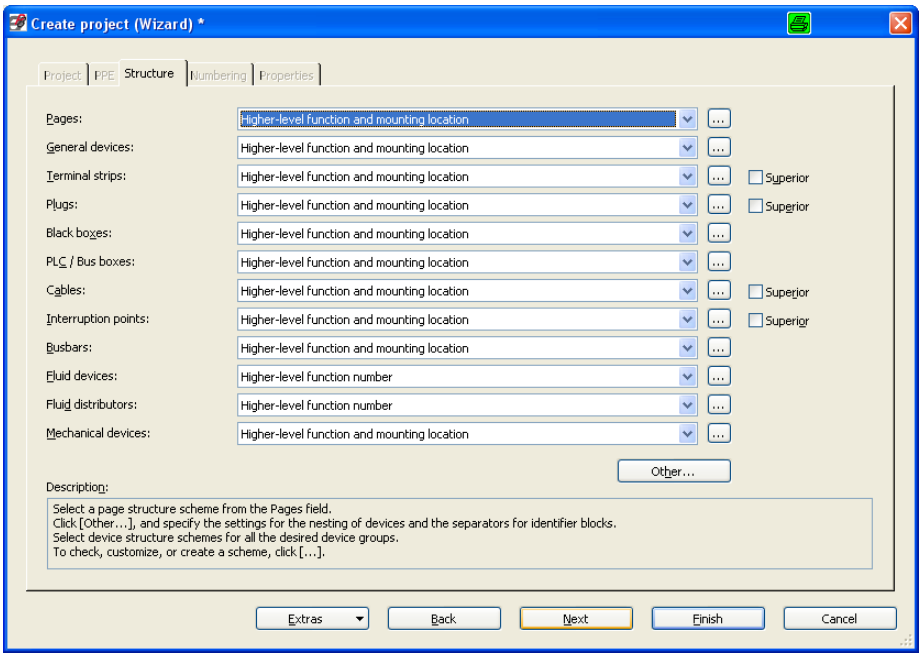


Fig. 3.18
Defining the project structure and its components

You can use the ... button at the end of each device group to select a different finished structure (scheme) via the **SCHEME** selection field, or use the button to create a new structure (new scheme).

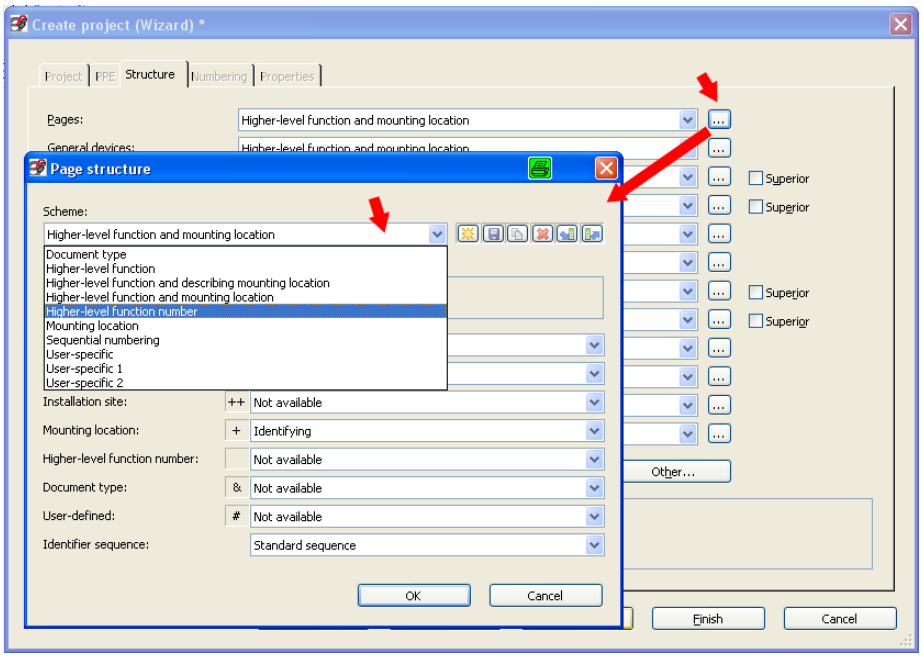


Fig. 3.19 Selecting a different structure scheme

The **Superior** option allows you to define reports for terminal strips, plugs, cables, and interruption points independently of the structure scheme that is set. Here EPLAN separately evaluates the presence or absence of the "–" character preceding a device tag.

Example:

The *Higher-level function and mounting location* scheme was set for interruption points. The *Superior* option was also set. If an interruption point in the schematic is designated with –**L12**, EPLAN assigns the device tag the higher-level function and mounting location in the schematic (if present) based on the defined *Higher-level function and mounting location* scheme.

Input: *–L12*

Result: *= Higher-level function + mounting location L12*

The *Higher-level function and mounting location* scheme for interruption points also has the *Superior* option set. If an interruption point in the schematic is designated with **L12** (without the preceding sign "–") in the schematic, the *Superior* option means that EPLAN does **not** assign the device tag the *higher-level function and mounting location* in the schematic (if present).

Input: *L12*

Result: *L12*

The OTHER... button in the **Structure** tab opens the **Extended project structures** dialog, where you can define the *separators* and the nesting of devices for the new project.

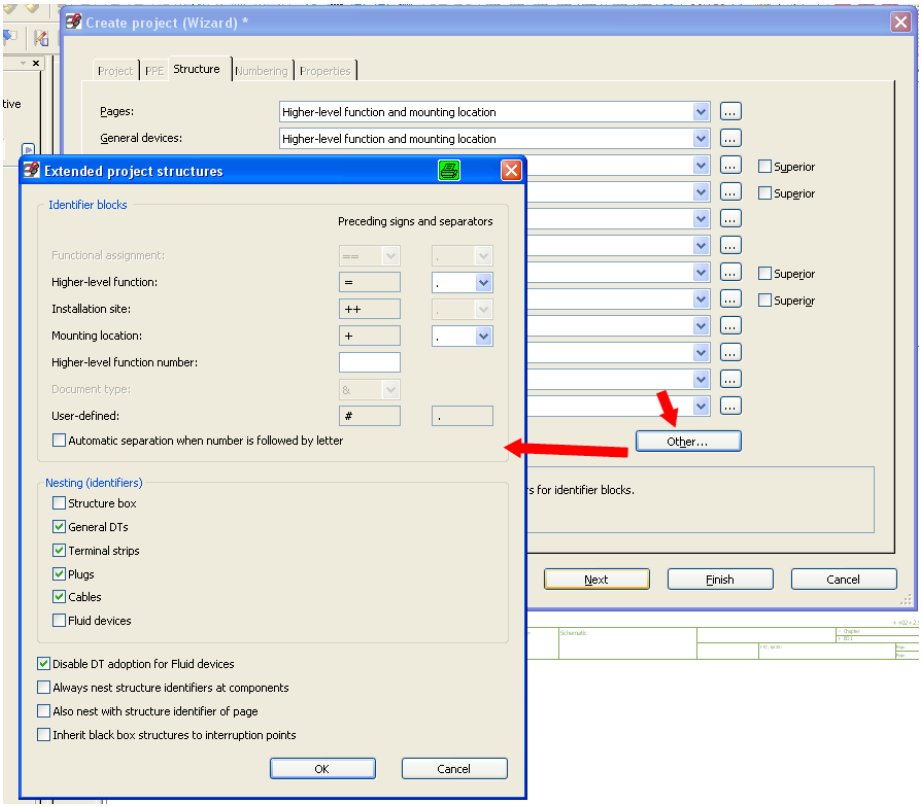


Fig. 3.20 Settings for other project structures

Extended project structures

Here you can check existing separators and enter new separators. The possible changes, however, depend on the page scheme set. Certain preconditions exist here for the *nesting of devices*.

NOTE: Apart from the page structure, which is the most important property, all other settings can be changed at a later date.

It is a good idea to select at least the *Higher-level function and mounting location* scheme. When such a scheme is set, EPLAN does not necessarily expect a higher-level function or mounting location (this is also true for other structure identifiers). Therefore, you can also use this scheme to create schematics with consecutively numbered pages.

If you later discover that one of the two identifiers is to be used after all, the pages can still easily be changed to the desired higher-level function or mounting location structure.

After setting and saving all the desired settings, click **NEXT**. The **Synchronize master data** dialog opens.

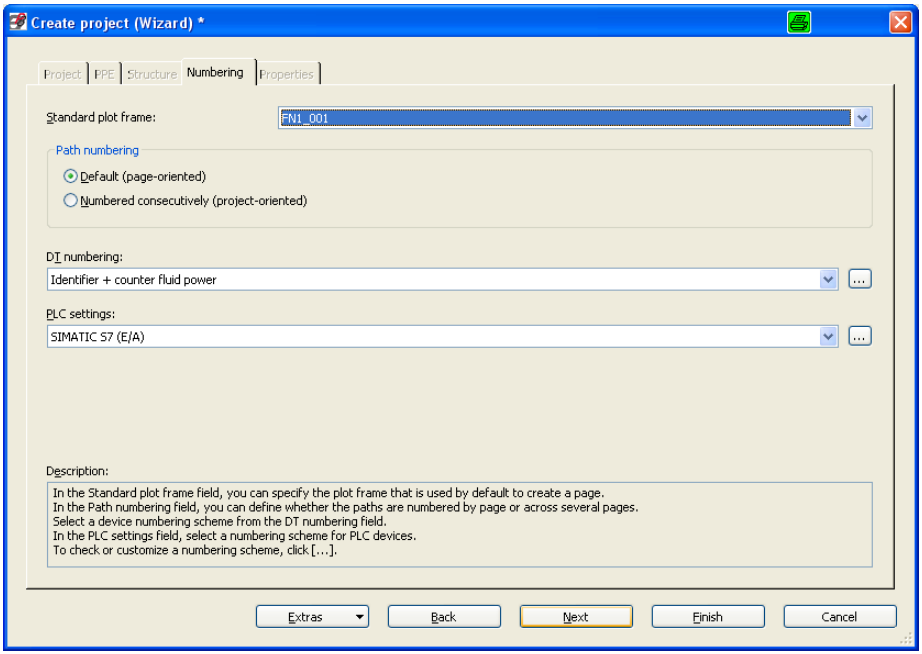


Fig. 3.21 Options and defaults for numbering

Here you can enter general project settings, such as a *Standard plot frame* (default setting for each new page) or the *Path numbering* (page-oriented or path-oriented).

The standard plot frame can be selected from the system master data pool and is then stored by EPLAN in the project automatically.

The path numbering can be globally set as page-based here (default value *Page-oriented*: every page then begins, for example, with path 1 and ends with path 10), or the path numbering is to function across all pages.

Across pages (default value *Project-oriented*) means that the paths are numbered across all pages. For example, path 1 starts on page 1 and path 30 ends on page 3.

NOTE: All these settings can be modified as desired later on in the project settings.

Defining numbering schemes

On this tab, you can also use ▾ in the drop-down field to specify a predefined *DT numbering* scheme for how new devices are to be numbered online when they are inserted.

You can use the ... button to select a scheme from the list, or you can create a new scheme via the **NEW** button, enter the desired values, and use this new scheme for the current project.

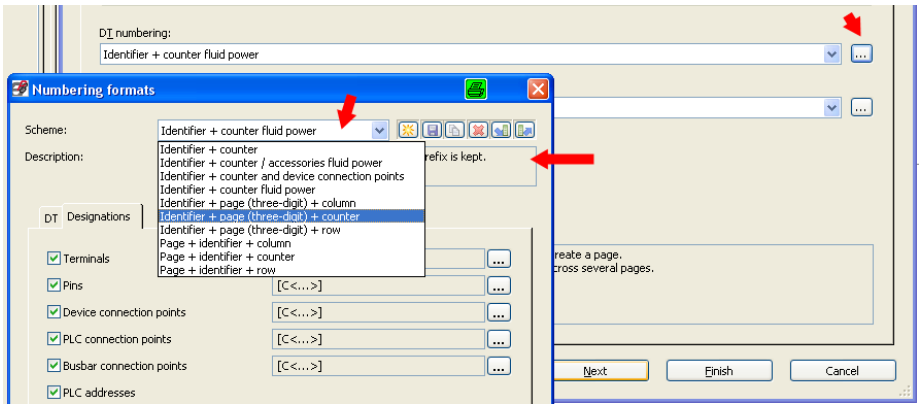


Fig. 3.22 Settings for DT numbering

The same applies to the setting in the *PLC numbering* selection field on the *Numbering* tab. Here too you can select existing schemes from the selection list. You can also use the ... button to select a scheme (from the selection list). They can be applied or changed at this point.

You use the **NEW** button to create new schemes and later assign them to the project.

TIP: All schemes created in EPLAN can be exported or imported. This makes it easy to exchange these settings between different EPLAN workstations or users. You should use a unified directory structure to make exported schemes easier to find.

All fields on the *Numbering* tab have now been entered. Clicking the **NEXT** button takes you to the *Properties* tab, which contains general, non-essential descriptive information about the project.

For example, to fill the plot frame with information, such as customer name or the name of the person responsible for the project, from the project properties, this information must be entered in the project properties.

This is not essential at this point because this is only descriptive information about the project that can be entered at a later date.

When you click the **FINISH** button, the project is generated and can be opened in the page navigator (the page overview). You can access the page navigator via the **F12** key or the **PAGE/NAVIGATOR** menu.

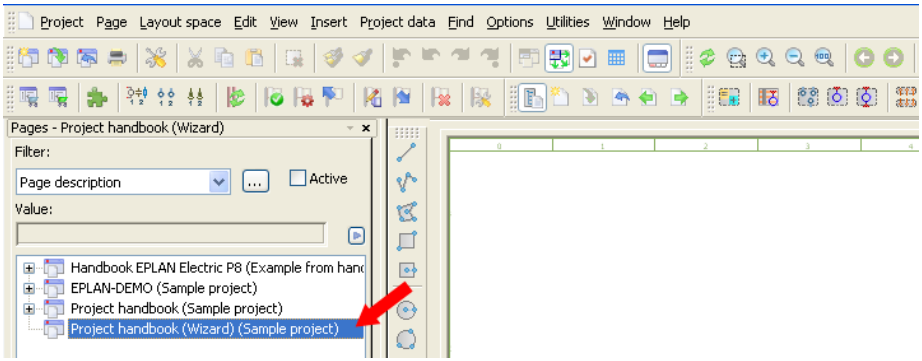


Fig. 3.23 The newly created project in the page navigator

You can now proceed with editing the project.

4 The graphical editor (GED)

In EPLAN Electric P8 you mainly work in the graphical editor, also known as the GED. The graphical editor contains all the functions you need to edit a project. It is, as it were, the main hub of EPLAN Electric P8.

4.1 Page navigator

In addition to the actual data such as parts data, a project also contains function templates consisting of individual pages, e.g. schematics, construction drawings or terminal diagrams that can be divided into particular structures such as higher-level functions or mounting locations and installation sites.

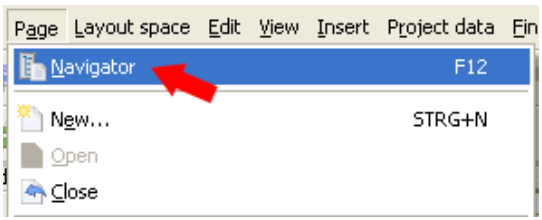


Fig. 4.1
Opening the page navigator

To provide a certain level of clarity and make it easy to edit the page properties, EPLAN has a **page navigator**. You open the **page navigator** via the PAGE/NAVIGATOR menu. As is usual in EPLAN, the **page navigator** offers a choice between tree structure or list view.

Page navigator Tree/list

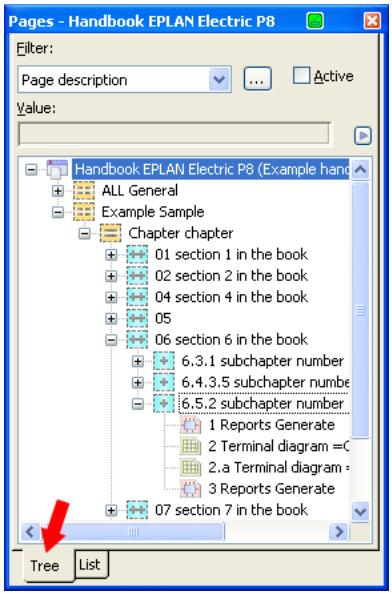


Fig. 4.2 Tree view

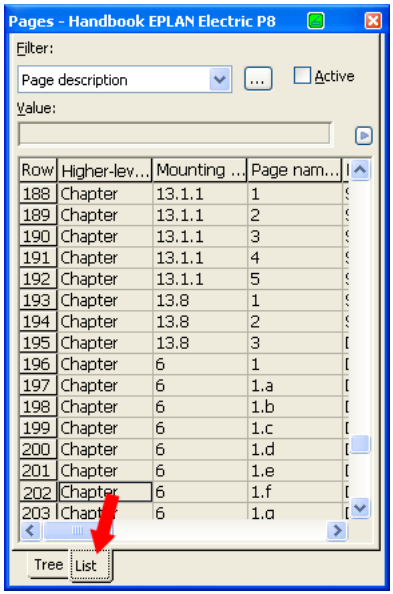


Fig. 4.3 List view

The tree view is ideal for providing a visual overview of all pages in a project. The small symbols preceding the names graphically differentiate identifiers and pages. You can define the display sequence via the parameter in the **OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/DISPLAY/PROJECT STRUCTURE (PAGES)** menu.

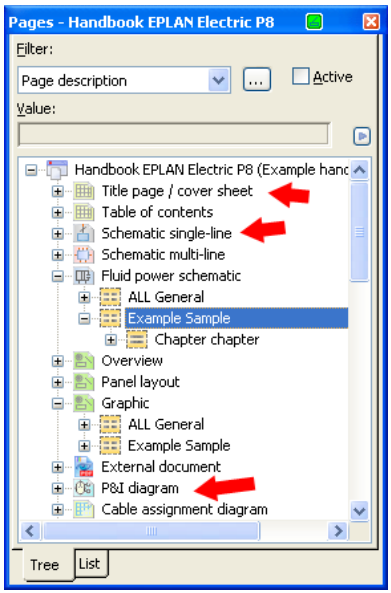


Fig. 4.4 Structured according to page type

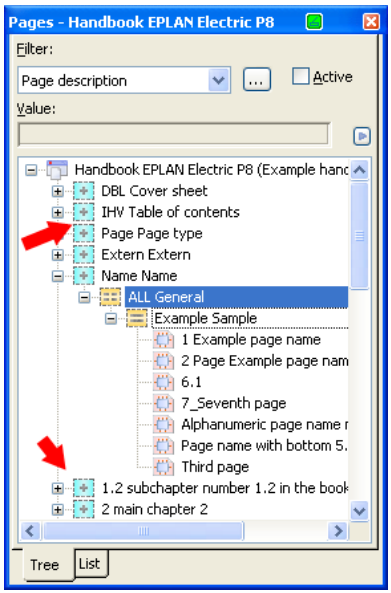


Fig. 4.5 Structured according to mounting location

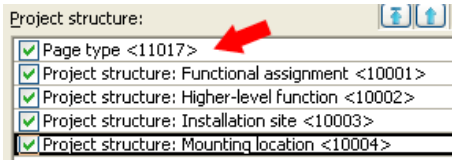


Fig. 4.6 Project setting by page type

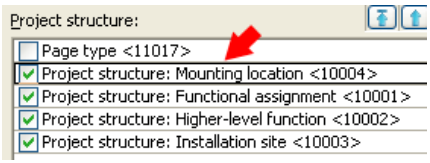


Fig. 4.7 Project setting by mounting location

This only affects the display in the **page navigator**. The sorting of the identifiers is defined in *structure identifier management*, i. e. whether (e. g.) higher-level function = A1 is sorted before or after higher-level function = A2.

4.1.1 Page types

EPLAN offers a range of different page types for specific purposes. In EPLAN, small symbols make these page types easier to be recognized at a glance.

EPLAN distinguishes between logical and graphical pages.

Examples of logical pages are, for example, *Schematic multi-line*, *Schematic single-line* and *P&I diagram*. EPLAN examines these page types for logical information and evaluates them accordingly (cross-references, etc.).

The *Graphical* page type or the *model view*, on the other hand, are purely graphical (non-logical) pages that initially do not contain logical information.

EPLAN also makes a distinction between pages that can be edited (interactive pages) and pages that are generated (automatic pages). The *Panel layout* page type is an example of an interactive page and the *Terminal diagram* page type is an example of a generated (automatic) page.

In the page navigator, all page types in EPLAN are identified by small symbols, as described above.

List of page types (incomplete)

Fig. 4.8 Page types

The exception is that there is no page type in the *layout space*. The layout space is the basis of the 3D representation of enclosures or other components used for the panel layout; here it only provides a view of the 3D data and its further processing.

4.1.2 The popup menu in the page navigator

Page editing (but not graphical page editing), is done via the **PAGE** menu or via the popup menu in the page navigator.

You access the *popup menu* functions via the right mouse button or the button in the **page navigator**.

Page menu and page navigator

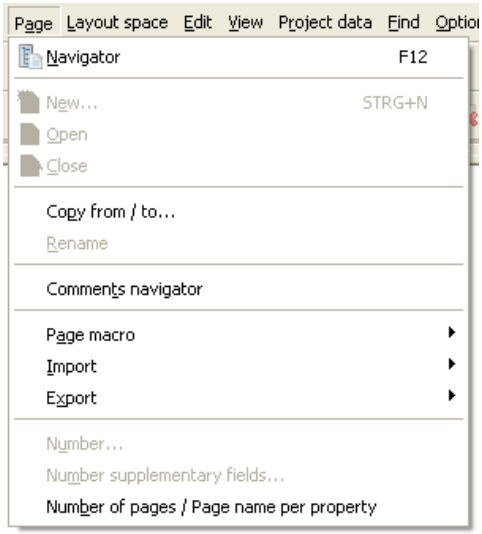


Fig. 4.9 Page menu

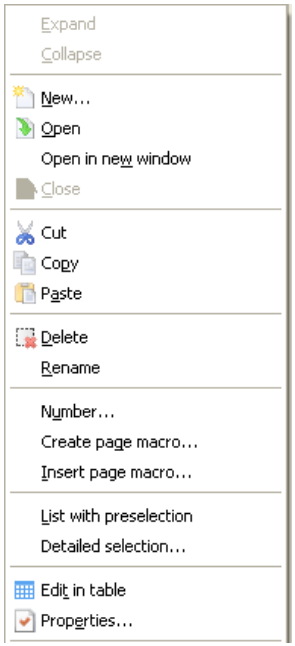


Fig. 4.10 Popup menu

4.1.2.1 Creating new pages

Usually the pages of a project are created from scratch (leaving aside copying for the time being). This can be done using the **CTRL + N** shortcut key or using the **PAGE/NEW** menu. The procedure is the same when using the popup menu, except that in this case you select the **NEW** menu entry. The **NEW PAGE** dialog then opens.

New page dialog

There are now two ways to integrate the new page into an existing page structure. You can define the full page name in the *Full page name* field. You can either manually change the existing entry or enter a completely new value with a prefix for the structure identifier. Alternatively, you can use the ... button to call up the **FULL PAGE NAME** dialog.

EPLAN opens the **FULL PAGE NAME** dialog, whereby every identifier has its own input field and own ... selection button, which can be used to branch to the selection dialog for the selected identifier.

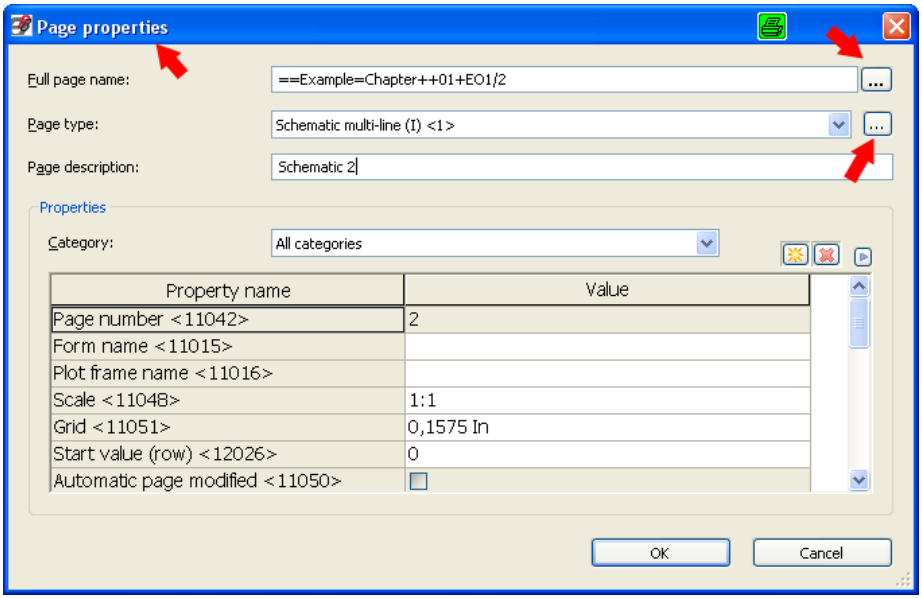


Fig. 4.11 New page dialog

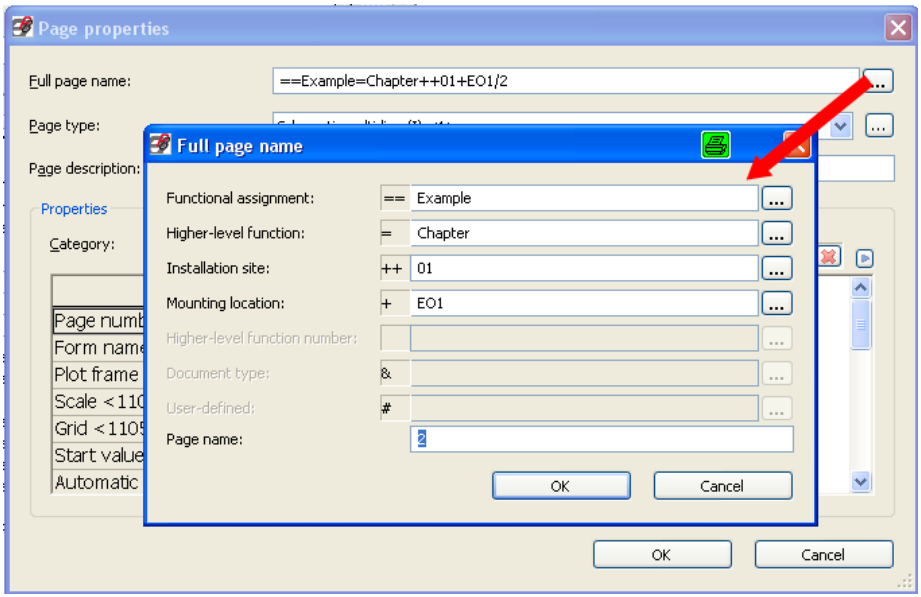


Fig. 4.12 Full page name dialog

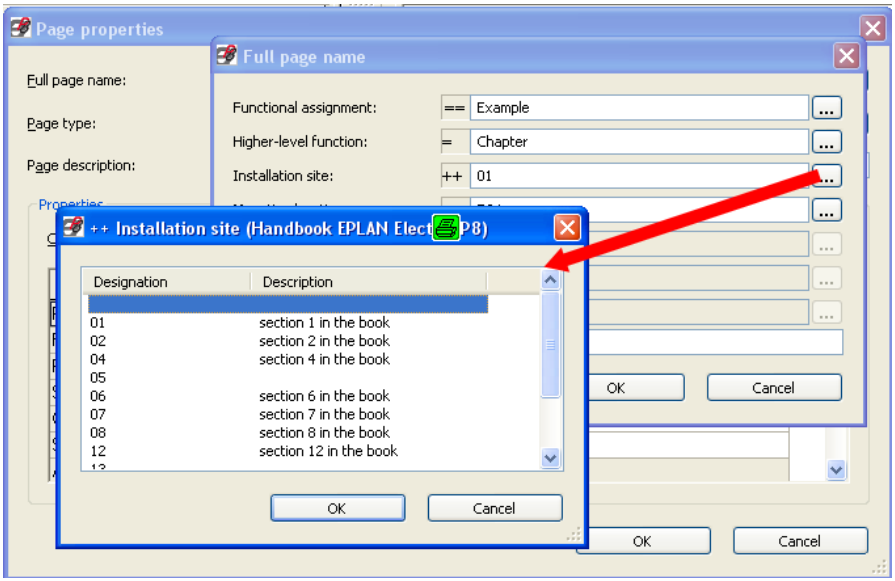


Fig. 4.13
Identifier dialog

Depending on the selected page structure, you can now click to select a *higher-level function*, *mounting location*, etc. Clicking **OK** applies the selected identifiers.

NOTE: If an existing identifier cannot be applied, then the new identifier can also be directly entered into the [Identifier type] input field in the **FULL PAGE NAME** dialog.

EPLAN saves new identifiers in the order defined in the settings. This setting is under **OPTIONS/SETTINGS/USER/DISPLAY/IDENTIFIER**. There are no other query dialogs.

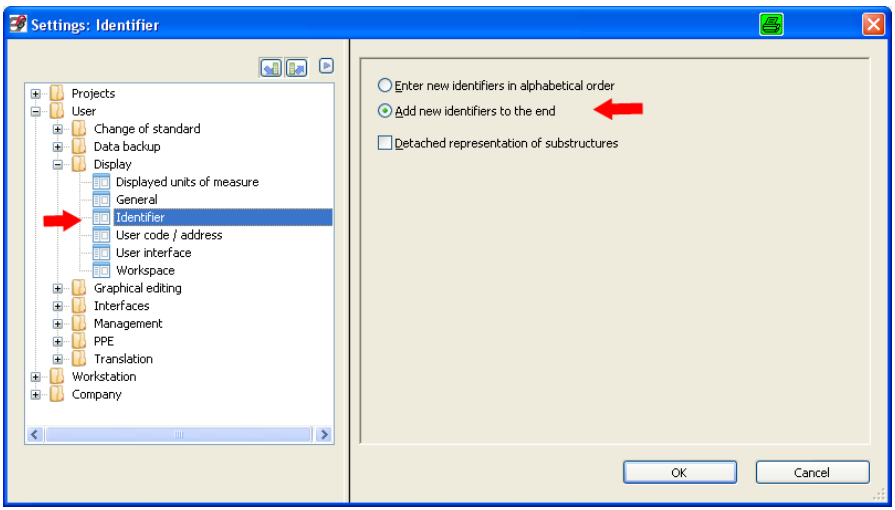


Fig. 4.14
Connection settings

Sorting can be performed manually or semi-automatically later in structure identifier management.

If all settings and entries have been entered in the **NEW PAGE** dialog and any missing structure identifiers have been created, the dialog can be closed. EPLAN will create the new page and then show it in the page navigator in the chosen structure.

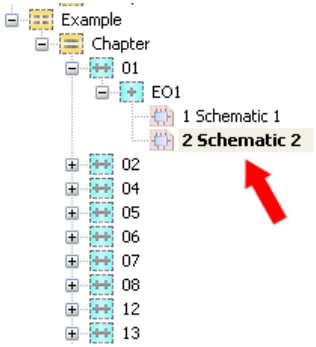


Fig. 4.15
Newly created page

4.1.2.2 Open pages

As the name of the function indicates, the **PAGE/OPEN** menu item opens a page. This is nothing new. The **OPEN IN NEW WINDOW** menu item in the page navigator popup menu is more interesting and useful. This menu item allows you to open several pages or even open the same page **several times**.

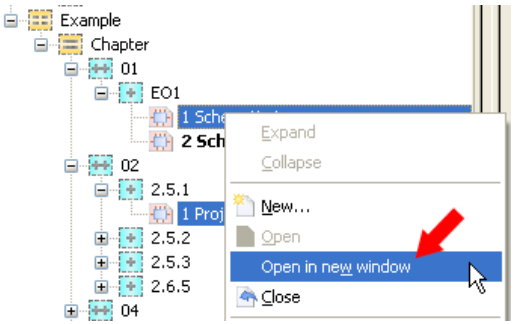
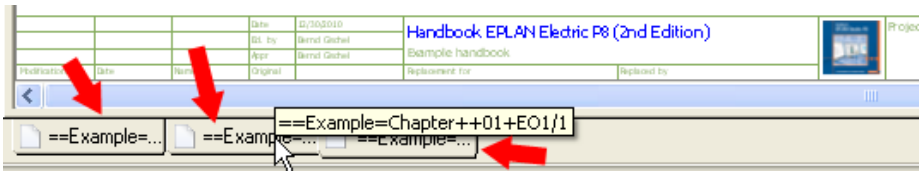


Fig. 4.16
Opening in new window

The procedure is simple. In the page navigator, you select the page or pages (it is possible to open several pages at once), right click to open the popup menu and select **OPEN IN NEW WINDOW**. You can also use the **CTRL + ENTER** keyboard shortcut. EPLAN then opens all selected pages.



Open in new window function

Fig. 4.17 Several open pages

If the **WORKBOOK** menu item in the **VIEW** menu is activated, then all opened pages are displayed in Excel style (tabbed page list) at the bottom of the workspace. You can then use the mouse to switch back and forth between these pages – or the **CTRL + TAB** keyboard shortcut to sequence through all the pages. With three pages, the sequence starts at page 1 followed by page 2, page 3, back to page 1, etc.

If several pages are open, you can use the **WINDOW** menu to go directly to the desired page.

To close open pages, simply select the pages in the page navigator and in the **PAGE** menu, select **CLOSE**. EPLAN then closes all selected pages.

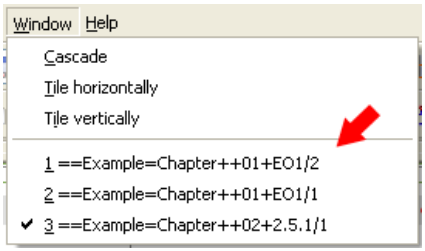


Fig. 4.18
Window menu

4.1.2.3 Copy pages

When you create new pages, they are created without any page content (devices, graphics, etc.). To create a new page that uses the content of an existing page, you need to copy one or more pages. There are several ways to do this in EPLAN. You can use the standard Windows functions like **CTRL + C** (copy page) and **CTRL + V** (paste page), or the **PAGE** menu (or the page navigator popup menu).

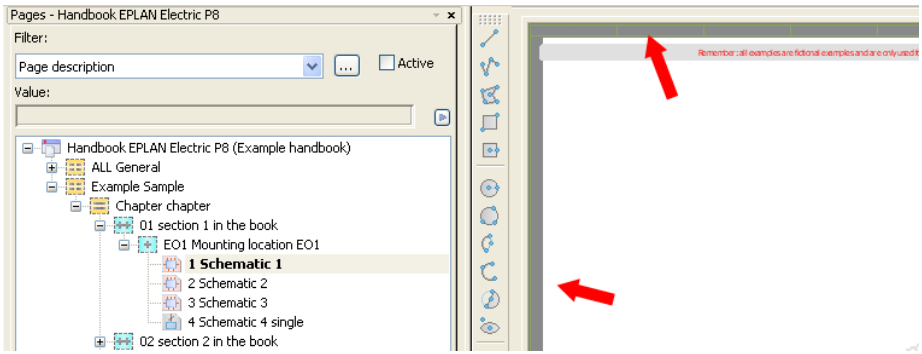


Fig. 4.19
Using CTRL + M to select a page

Use **CTRL + M** to select the page in the graphical editor. EPLAN then draws a thick gray border around the page (to show it is selected).

When you press **CTRL + C** (standard Windows function for copying), EPLAN copies the page to the clipboard. Pressing **CTRL + V** (paste) pastes the copied pages to a different location. EPLAN then opens the **ADAPT STRUCTURE** dialog.

Copying pages within a project and across projects

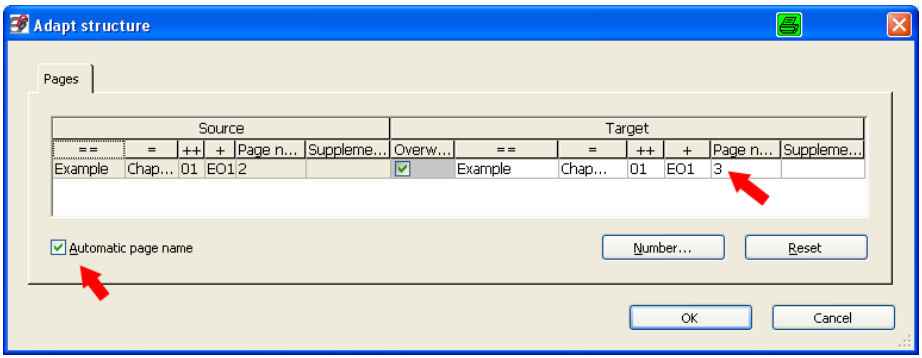


Fig. 4.20
Adapt structure dialog

Relevant entries, such as the assignment of identifiers or a new page name, can now be edited in this dialog. However, EPLAN can also determine the new page name automatically (*Automatic page name* setting). EPLAN always suggests the highest free page name available in this structure. Once all entries are correct, you click **OK** to apply the page.

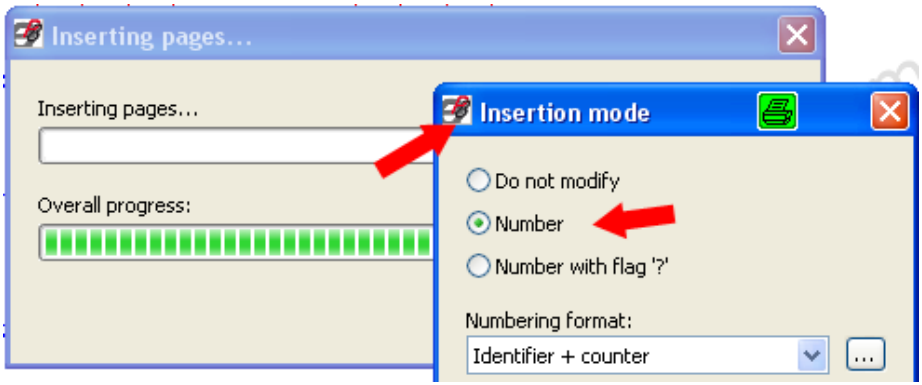


Fig. 4.21 Selecting the insertion mode

Depending on the settings, the **INSERTION MODE** dialog opens to allow you to number existing devices on the copied page. After selecting the desired setting, you exit the dialog by clicking **OK**. EPLAN then copies the (source) page, generates a new (target) page, and sorts it into the page structure.

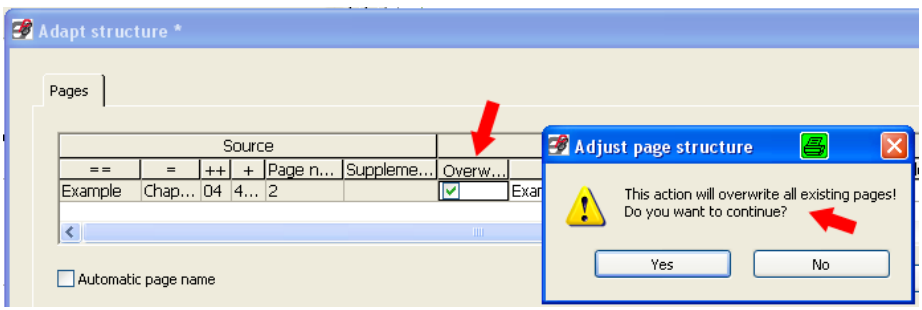


Fig. 4.22 Overwrite option

At this point, the **ADAPT STRUCTURE** dialog also lets you overwrite existing pages(using the **Target – Overwrite** column). Before actually overwriting pages EPLAN displays a prompt, which must be confirmed by clicking **YES** (copy action continues) or **NO** (copy action is canceled and EPLAN opens the page navigator once more).

Again the following applies when copying pages: Without asking for confirmation, EPLAN automatically sorts new identifiers into the existing structure (alphabetically or at the end, depending on the setting). They can be resorted later in structure identifier management.

You can, of course, copy more than one page. In the page navigator, you can select multiple pages and use **CTRL + C** to copy them to the clipboard.

Copying pages in the page navigator

When you press **CTRL + V** (paste), EPLAN again opens the already-familiar **ADAPT STRUCTURE** dialog. After you make all desired entries and confirm any subsequent dialogs, these pages are inserted into the EPLAN project in the selected structure.

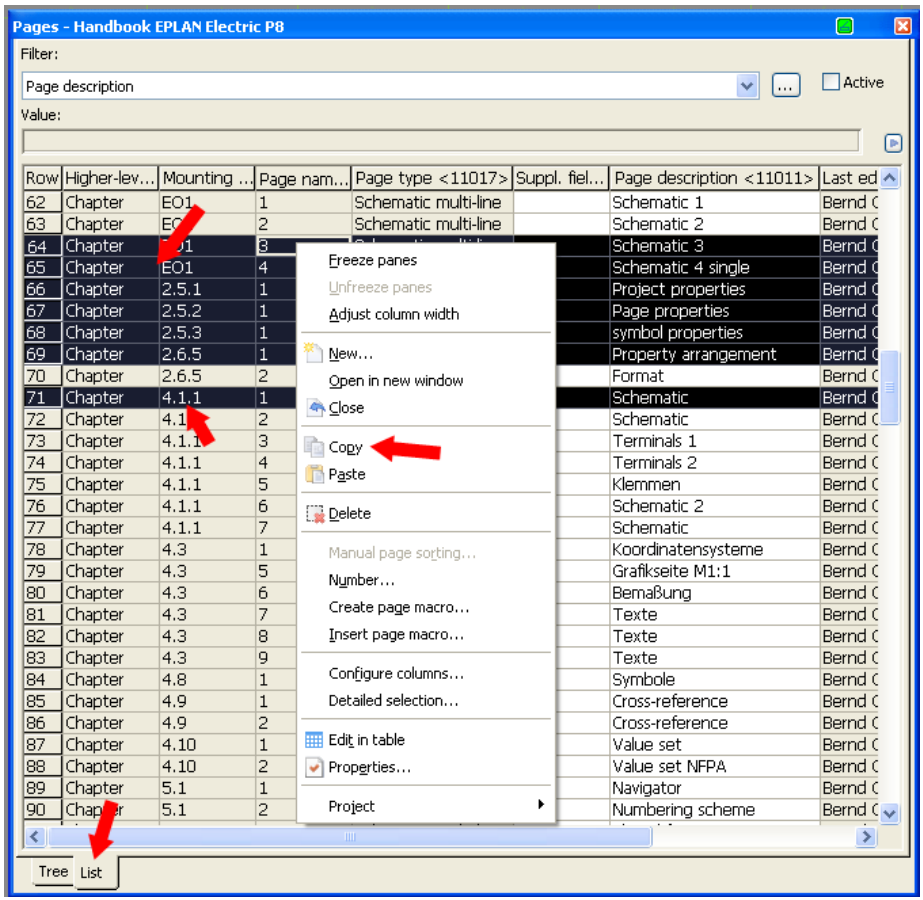


Fig. 4.23 Copying in list view

Page navigator List view

Instead of **CTRL + C** and **CTRL + V**, you can also use the **COPY** and **PASTE COMMANDS IN THE page navigator POPUP MENU** to copy pages. These two commands always relate to the selected pages. You can select and copy/paste pages from either the tree view or the list view.

TIP: CTRL + M (SELECT PAGE) is an important keyboard shortcut to remember. All page-related commands, i. e. copying a page, deleting a page or editing page properties, are implemented by pressing **CTRL + M** and then the subsequent action command.

In addition to copying pages within a project, you can also copy the pages of a project to the clipboard, switch to another project (which should be open in the page navigator) and there, paste the pages from the clipboard.

4.1.2.4 Copy pages from/to

The counterpart to simple page copying (within a project and to other projects) is the **COPY FROM/TO...** function. Since you can open and edit several projects at once in the EPLAN page navigator, it is very useful to have a function that lets you conveniently copy pages from one project to another.

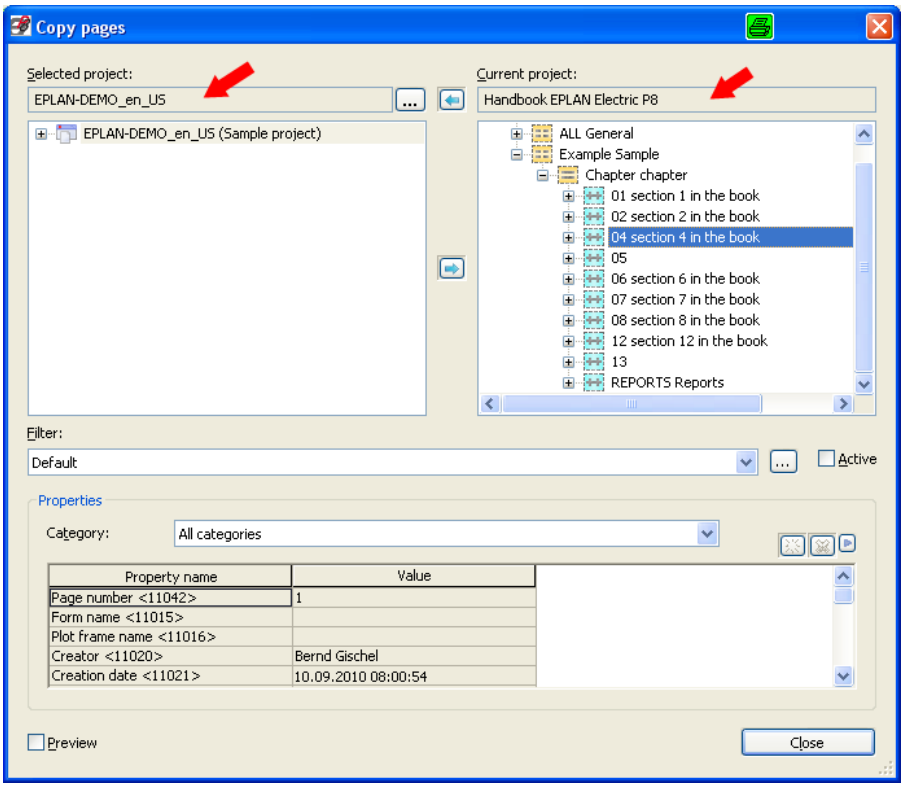


Fig. 4.24 Copy pages dialog

Copy pages from/to function

The **COPY PAGE/PAGES FROM/TO** menu lets you copy pages from different projects (even from projects that are not open in the page navigator) to the current project. When you select **COPY PAGES FROM/TO** from the menu, EPLAN opens the **COPY PAGES** dialog. As already mentioned, a project does not need to be open in the page navigator in order to copy pages from it.

The lower area of the **COPY PAGES** dialog shows additional information about the selected page. A preview for the selected pages can also be activated. The right field in the **COPY PAGES** dialog is fixed and always indicates the *current project*. It cannot be changed. The

selected project in the left field is variable. You can use the ... button to select and open any other project.

When you click ..., EPLAN opens the **PROJECT SELECTION** dialog. Initially, only projects currently open in the page navigator are listed.

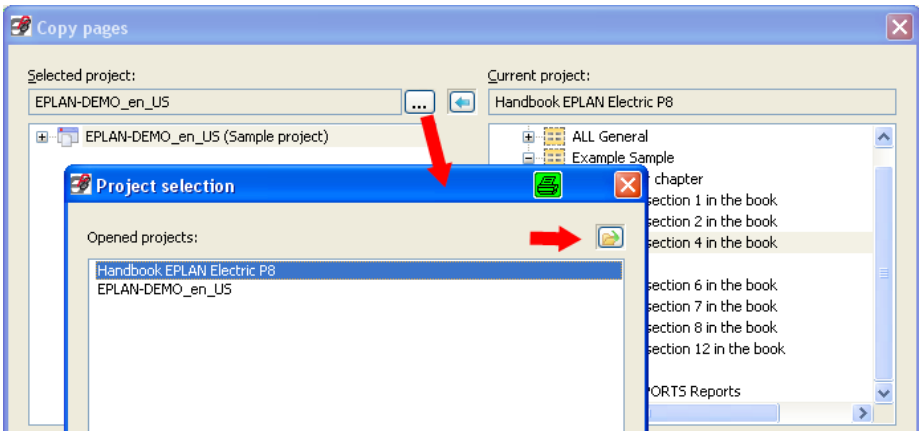


Fig. 4.25 Project selection dialog

When you click (Open), EPLAN opens the **OPEN PROJECT** dialog. In this dialog, you can select and open one or more projects to be used in the **PROJECT SELECTION** dialog. First you select the project(s), then click **OPEN** to add the selected project(s) to the **PROJECT SELECTION** dialog.

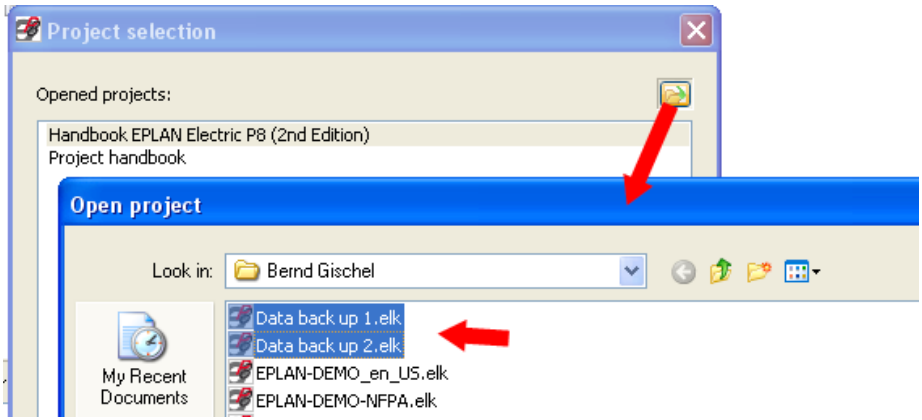


Fig. 4.26 Open project dialog

When you have finished, the selected projects are listed in the **PROJECT SELECTION** dialog ready for page copying and are opened simultaneously in the page navigator.

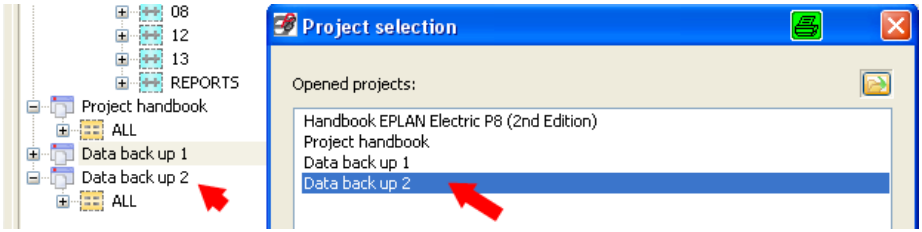


Fig. 4.27 Opened projects

You can now select the desired project in the **PROJECT SELECTION** dialog and add them to the **COPY PAGES** dialog by clicking the **OK** button.

NOTE: Only one project at a time can be added to the selection for the **COPY PAGES** dialog. However, EPLAN remembers the projects that were added to the **PROJECT SELECTION** dialog so that later copying can take place without having to re-select a project.

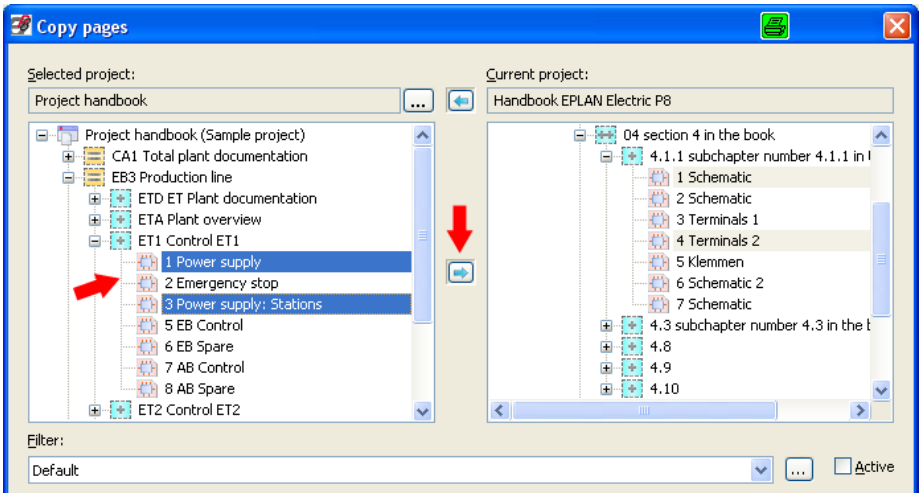


Fig. 4.28 Copying pages from a project

In the **COPY PAGES** dialog, on the left you *select* the pages you want to copy from the external project and use the button in the middle to copy (move) them to the current project. EPLAN then opens the **ADAPT STRUCTURE** dialog.

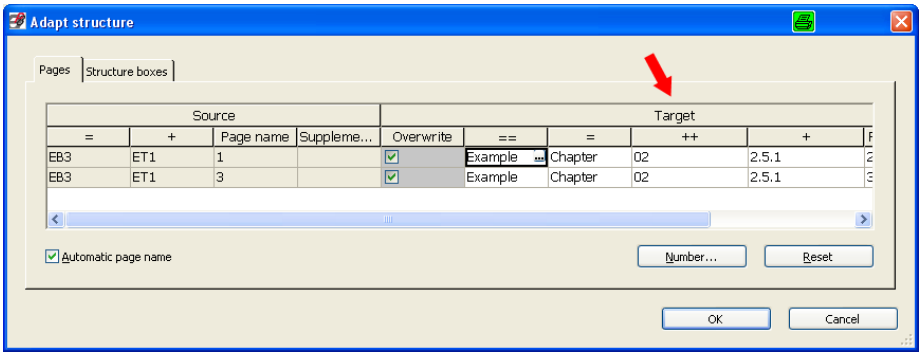


Fig. 4.29 Inserting pages into the page structure

Adapt structure

In the **ADAPT STRUCTURE** dialog, you can adjust the page structure as desired.

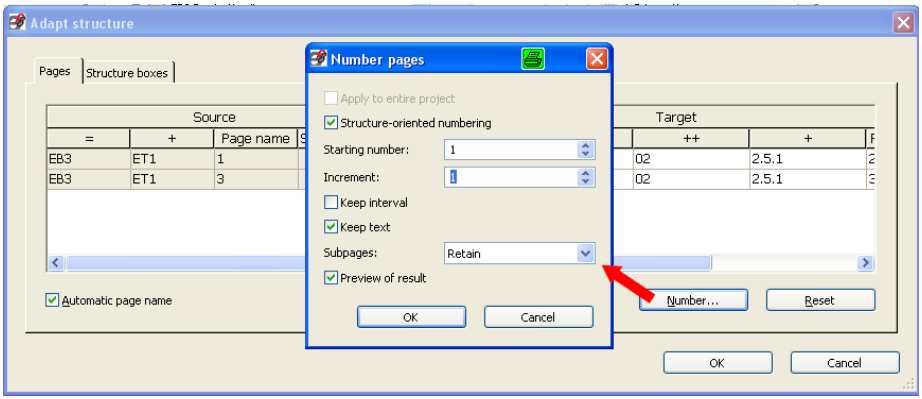


Fig. 4.30 Number pages

In the **NUMBER PAGES** dialog, it is possible to re-number the new pages according to specific settings. When you need to copy many subpages and convert them to main pages, for example.

After you click the **OK** button in the **ADAPT STRUCTURE** dialog, EPLAN copies the pages into the current project. Any follow-up dialogs, such as the **INSERTION MODE** dialog, must be confirmed as required.

This completes the copying of pages from external projects. EPLAN closes all dialogs and again focuses on the **COPY PAGES** dialog. You can now copy pages from other projects. If this is not required, you can exit the **COPY PAGES** dialog with the **CLOSE** button. EPLAN closes the dialog and all temporarily opened projects and returns to graphical editing of the currently opened project.

4.1.2.5 Rename pages

Pages in EPLAN are given a unique page name (page number). This page name **can** include alphanumeric characters. A page could have the name = *ANL2+ORT2/14_Overview*, for example, whereby the word "Overview" is part of the page name and not the page description.

The page name (number) is renamed using the **RENAME PAGE** menu item. In the **page navigator**, you can also simply select the page and press the **F2** key.

EPLAN then lets you change the page name (number) without directly opening the page properties of this page. Pressing **ENTER** saves the changed page name (number).

Renaming the page name (number)

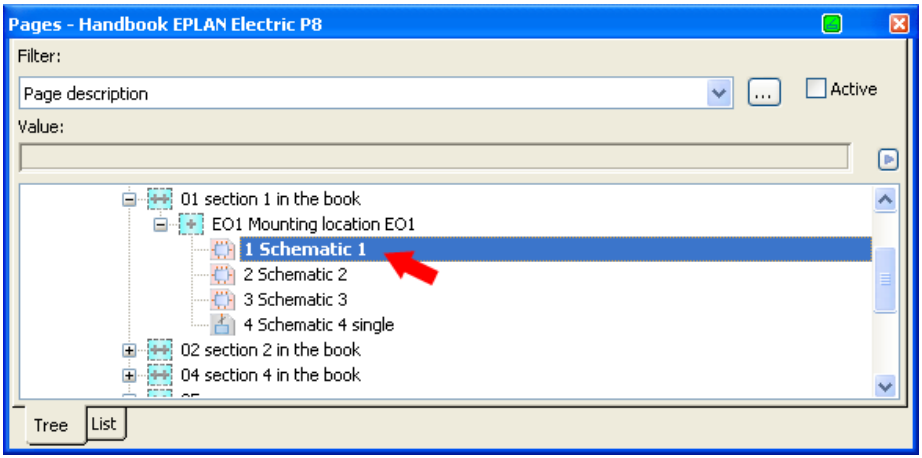


Fig. 4.31 Selecting a page

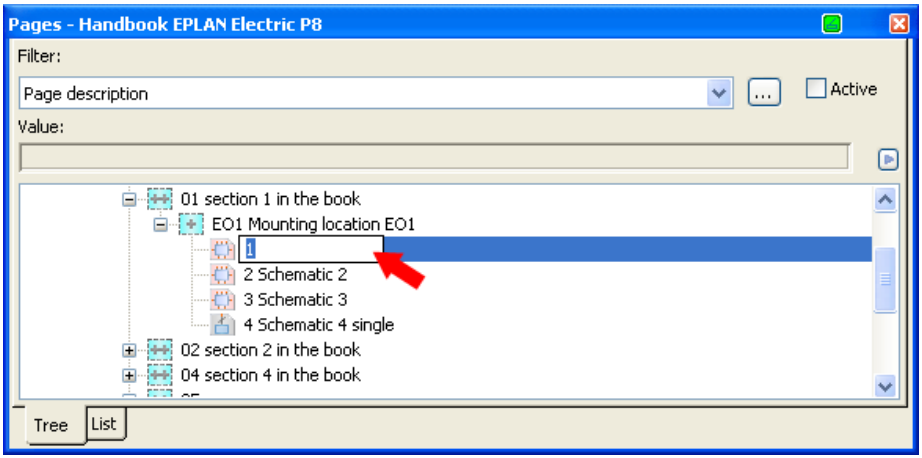


Fig. 4.32 Direct editing via F2

4.1.2.6 Delete pages

But you don't just create or copy pages. Sometimes you need to delete them. In the popup menu of the page navigator, EPLAN provides the **DELETE** function for this purpose.

You are already familiar with the procedure. The pages to be deleted are selected in the page navigator and then deleted using the Delete function in the page navigator popup menu or by pressing the **DEL** key.

Before the actual deletion, EPLAN displays a **DELETE PAGES** dialog with a warning message asking if the page(s) should actually be deleted.

NOTE: I recommend that you read the messages carefully before **blindly** clicking **YES** or **YES TO ALL**.

NOTE: The following is very important and should be noted when deleting pages. It makes a difference how pages are selected in thepage navigator.

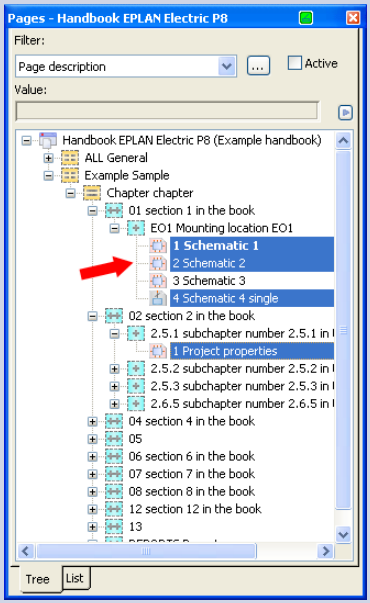


Fig. 4.33 Selection 1: 4 pages

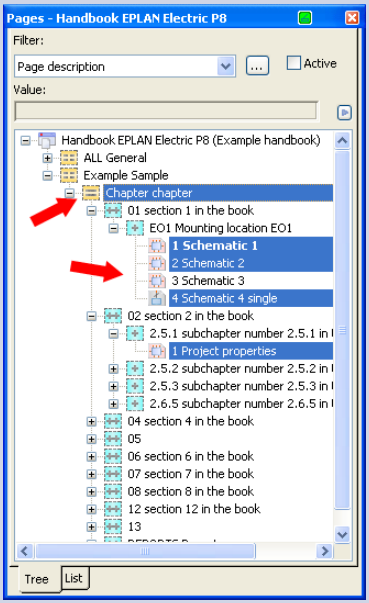


Fig. 4.34 Selection 2: Entire higher-level function (chapter)

In Fig. 4.33 four pages are selected for deletion. Only these four pages will be deleted. The warning message in the **DELETE PAGES** dialog is very clear: "Do you want to delete Page … out of 4 selected pages?"

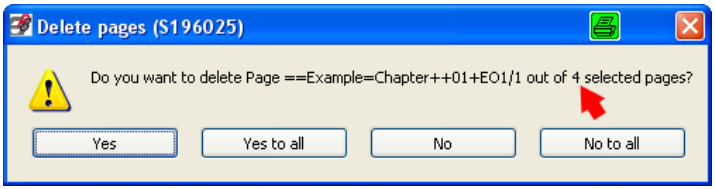


Fig. 4.35 Warning message: Delete 4 pages?

In Fig. 4.34, however, the selection has been extended by mistake to the entire higher-level function *=Chapter*, because the higher-level function was selected in the **page navigator**.

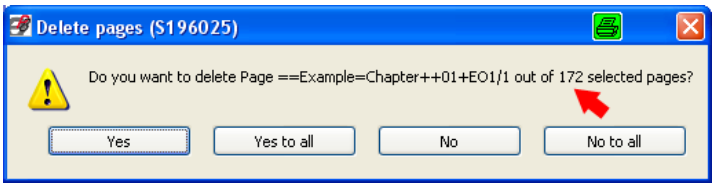


Fig. 4.36 Warning message: Delete 172 pages? (instead of the desired 4)

If you try to delete these selected pages, EPLAN displays the familiar **DELETE PAGES** dialog with an appropriate warning message, but with the minor difference that the *number of pages to be deleted* is no longer correct.

If you unthinkingly confirm this dialog with **YES**, then EPLAN will delete the entire higher-level function *=Chapter* from the project. Unintentional deletion of pages is usually noticed immediately, since the page navigator is updated after a deletion and the missing pages, higher-level functions, or locations are (or should be) immediately obvious. The usual UNDO command **CTRL + Z**, or the **UNDO** or **UNDO LIST** functions in the **EDIT** menu can then be used to restore the pages *immediately after deletion*.

NOTE: If the project was closed in the interim, the deleted pages are irretrievably lost!

NOTE: If the project name is selected, then EPLAN disables the **DELETE** function. This prevents accidental deletion of entire projects from the page navigator popup menu.

4.1.2.7 Close pages

This function is self-explanatory. It closes open pages. The **CLOSE** function is accessible in the **PAGE** menu and also via the page navigator popup menu. It is possible to select several open pages in the page navigator and close them all at once. The **CLOSE** function only closes the project page(s) and not the project, not even if it is selected.

4.1.2.8 Edit page properties

To edit the page entries, the so-called *Page properties*, you need to select the relevant page in the page navigator and use the popup menu to call up the **PROPERTIES** function.

Of course, this can also be done without the page navigator, directly on the opened page using the **CTRL + M** and **CTRL + D** shortcut keys. **CTRL + M** selects the page and then **CTRL + D** calls the **EDIT PROPERTIES** function. EPLAN then opens the **PAGE PROPERTIES** dialog.

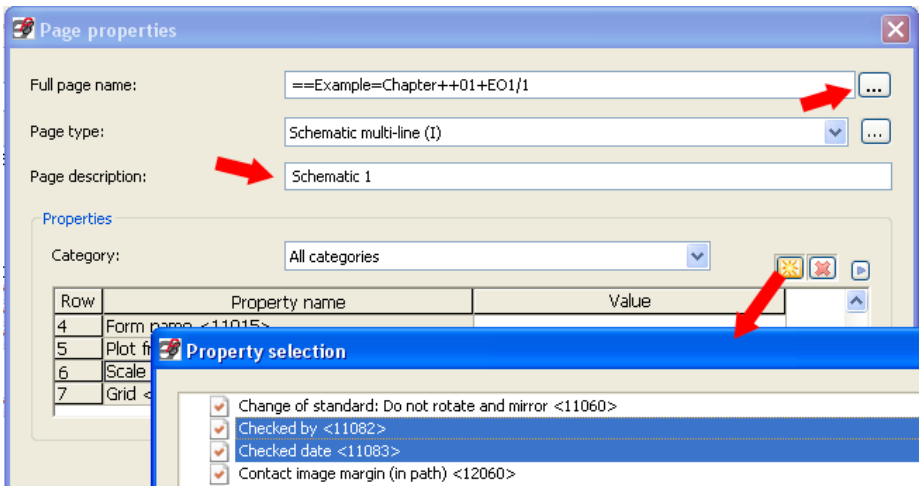


Fig. 4.37 Edit page properties dialog

Edit page properties

You can now edit all the page properties in the **PAGE PROPERTIES** dialog. The *Full page name* can be changed (for example, you could change the page number from 2 to 3, which will move the page) or modify the *Page description*.

The *Properties* area displays additional properties, such as page scale (*Scale <11048>*) or the currently set grid (*Grid <11051>*). These properties can all be edited when they are not grayed out.

New properties are added via the button. When this button is clicked, EPLAN opens the **PROPERTY SELECTION** dialog. Here you can select the desired properties (multiple selections are possible) and apply them in the **Page properties** display by clicking the **OK** button.

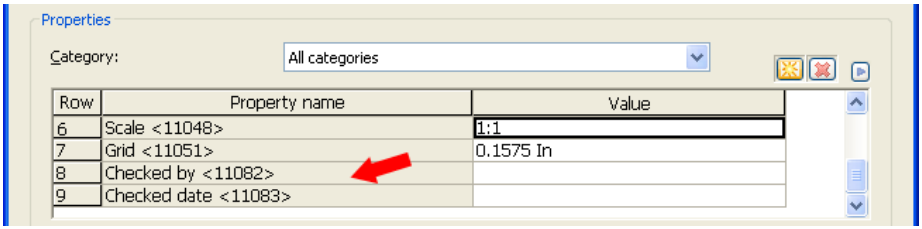
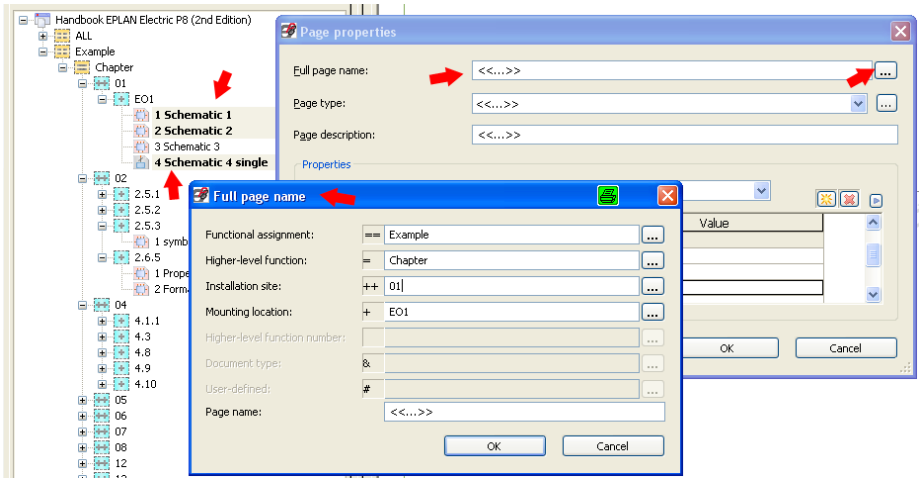


Fig. 4.38 Newly added properties

In the **page navigator**, you can edit more than one page at once, which is not the case for the **CTRL + M** and subsequent **CTRL + D** keyboard shortcuts.

If several pages are selected in the page navigator and the popup menu is then used to call the **PROPERTIES** function, then the properties of all selected pages can be edited in a single step.



Edit the properties of multiple pages

Fig. 4.39 Editing the properties of multiple pages

All properties that are the same in the selected pages are displayed in "plain text". Examples would be the scale or grid of the pages. If the page properties are different from each other, then they are displayed visually using the <<…>> string in the corresponding input fields.

NOTE: Care should be taken when changing fields containing the <<…>> string because when an entry in a field is changed, all other entries that are not the same on other pages are overwritten with the new (changed) value.

When all entries are complete, you can exit the **PAGE PROPERTIES** dialog by clicking **OK**. EPLAN saves all entries and then closes the dialog.

4.1.2.9 Edit page properties directly

In addition to starting the **PAGE PROPERTIES** dialog via the page navigator popup menu or the **CTRL + M** and **CTRL + D** shortcut keys in order to adjust or modify properties such as the page description, etc., it is now possible to simply double-click an existing object (page description, customer name) in the plot frame to have EPLAN open the page properties.

Fig. 4.40 Double-click any object in the plot frame to open page properties

But it has to be an area that contains content. Double-clicking an empty area will *not* open the page properties.

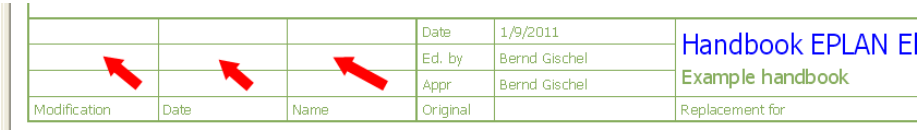


Fig. 4.41 Empty areas do not work

4.1.3 Page navigator filter

In addition to the functions previously described, with EPLAN Electric P8 2.2 it is now possible to directly activate a filter for previously created schemes and use the quick entry functions in the page navigator and in other navigators.

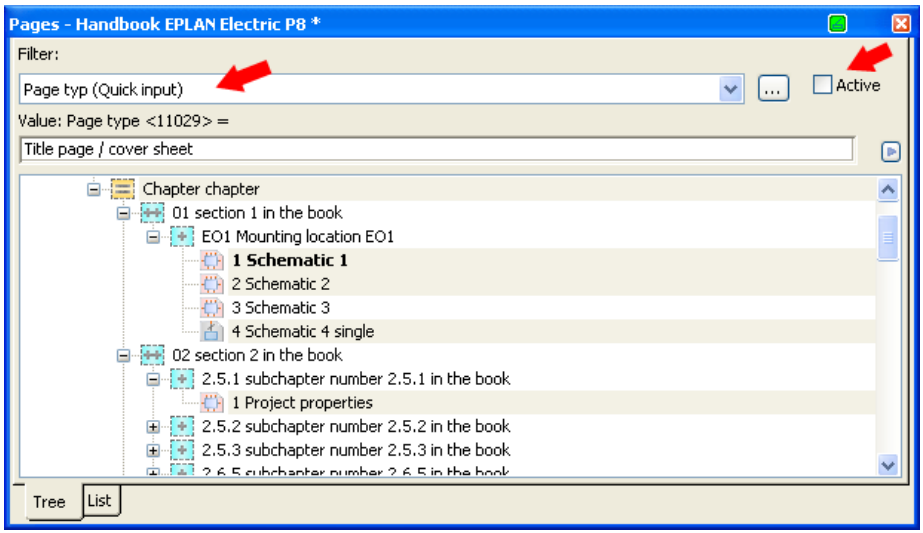


Fig. 4.42 Filter options

This lets you quickly and easily filter, for example, the page display by a specific **page type** (“quick entry”). To do so, you first create a filter scheme and then activate the **quick entry** column.

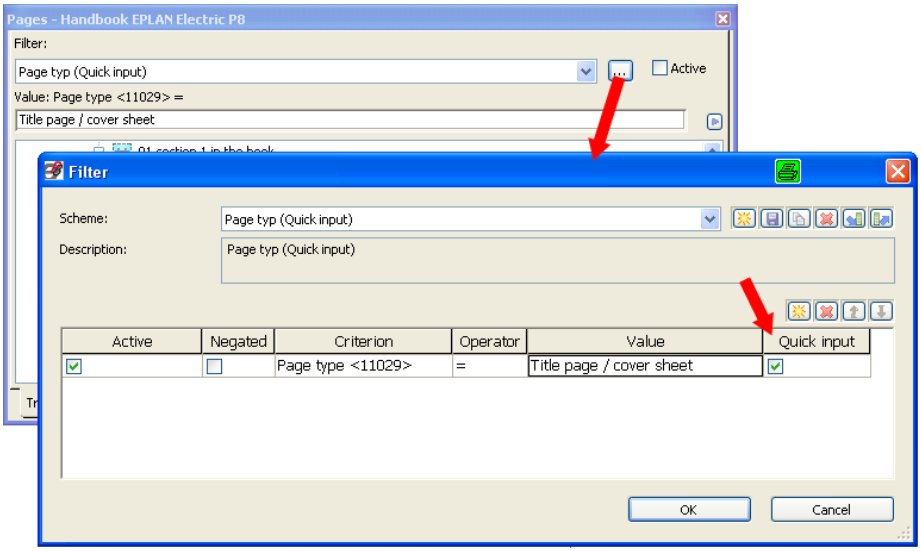


Fig. 4.43 Filtering by quick entry

When the filter has been activated, you can use the **Value selection: [criterion]** to execute a quick entry (**page type** in this example).

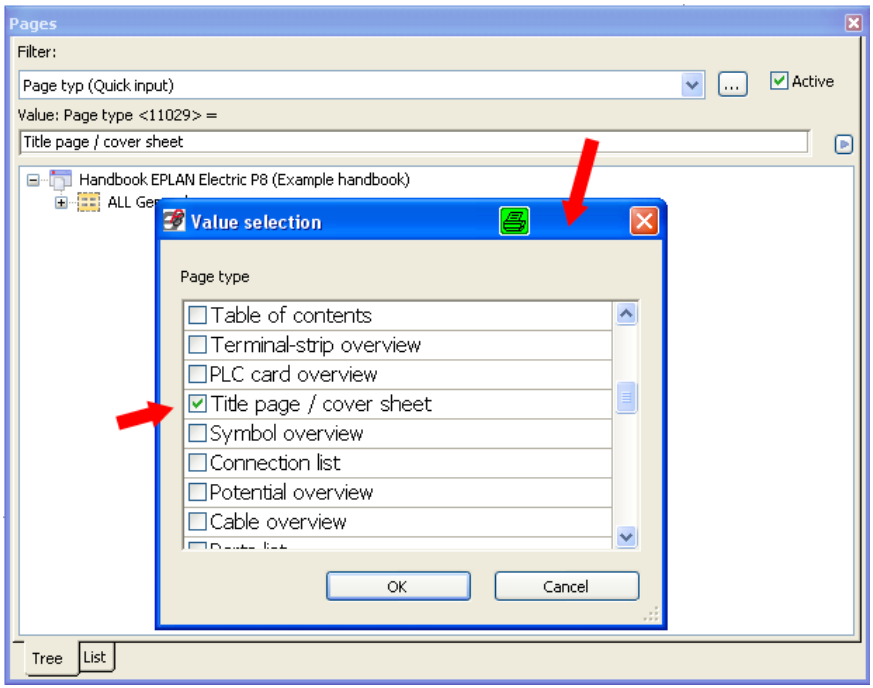


Fig. 4.44 Active page type filter

The main advantage of being able to combine filters with quick entry is that you do not have to create an extra filter scheme for each possibility. You only have to set one, like here in the **Page type**. You simply activate the required value, here the to-be-filtered **page type**.

NOTE: The filter function from previous versions (prior to the Versions 2.0.x) is now located in the popup menu of the **page navigator** and is called **DETAILED SELECTION**. This can still be used to filter search results or to find and filter specific structure identifiers.

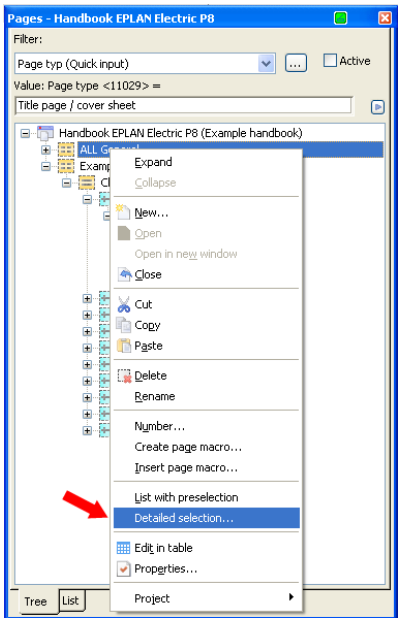


Fig. 4.45 "Previous" detailed selection

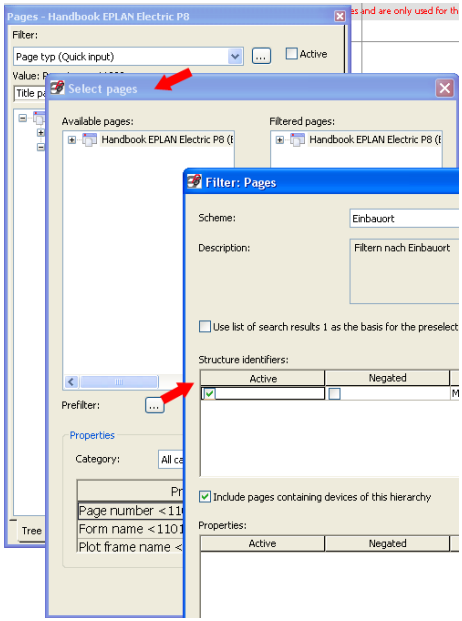


Fig. 4.46 "Previous" filter options

4.2 General functions

The graphical editor can be divided into several areas. The main area is the working area for editing the schematic (the graphical editor). A number of dialogs can be associated with the main area. These range from the page navigator to user-defined toolbars.

The graphical editor is not restricted to a single representation. The familiar idea of **workspaces** can be used to rapidly switch the entire EPLAN user interface from one representation to another, including all the associated dialogs, toolbars, etc.

4.2.1 The title bar

Aside from the actual working area, there is the *Title bar*, which displays only little, but crucial, information. The most important function of the title bar is the display of the currently open project name.

This display can be customized by the user. To do so, go to (**OPTIONS/SETTINGS/USER/DISPLAY/USER INTERFACE** menu) and modify the *Display in title bar* item.

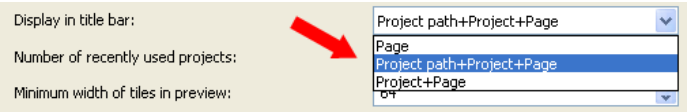


Fig. 4.47 Customizable display of the title bar

The following entries are possible here: **Project+Page; Project path+Project+Page** or only **Page**. In Fig. 4.48 the **Project path+Project+Page** setting has been selected.

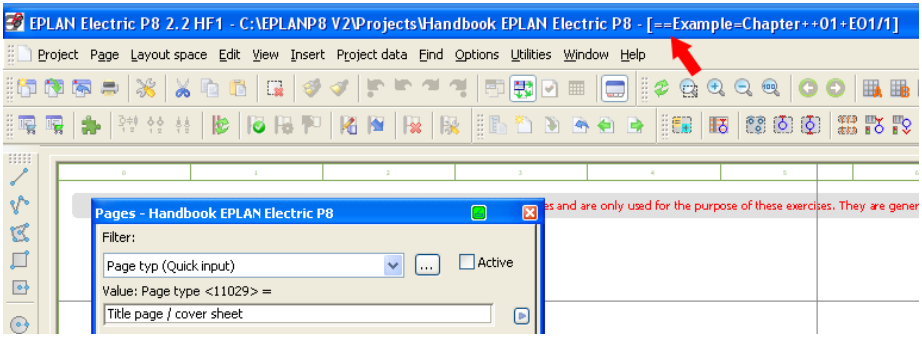


Fig. 4.48 Sample setting

You cannot change or add to the content of the title bar.

4.2.2 The status bar

The status bar in the graphical editor constantly shows various pieces of current information.

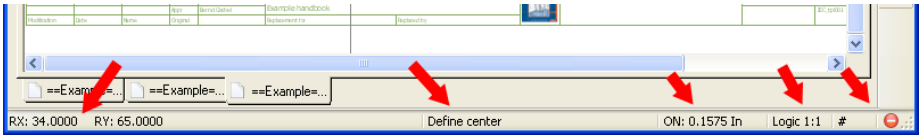


Fig. 4.49 Status bar

For example, the RX and RY coordinates of the current cursor position, information on the current grid (here 4.00 mm), and whether or not the grid is switched on (here ON), are all displayed here. The information logic 1:1 shows the page type and the scale of this page. The example shows an open logical page at a scale of 1:1.

In the status bar, EPLAN also displays supporting information for the function currently in use (in this case, a circle is to be drawn). This does not apply to all functions, but looking at the status bar can be a great help for some EPLAN actions.

The last characters in the status bar have the following meaning:

#	The hash means that the project contains connections that have not been updated.[1)]
*	The asterisk means that the current page contains connections that have not been updated.[1)]
	If a new error occurs after the last time the system messages were opened, this icon will be displayed again.[2)]
+	This character is displayed when the “advanced” support-log mode has been enabled (using the “ELogFileConfigToolu.exe” tool supplied by EPLAN).

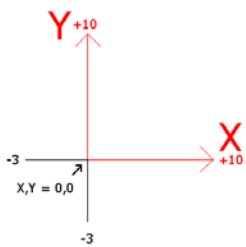
1) Since it is possible to assign menu commands to keyboard shortcuts in EPLAN, I would recommend assigning the **PROJECT DATA/CONNECTIONS/UPDATE** menu command to the **F11** key. This is easy to remember and the command then only requires a single keystroke.

2) The icon can be double-clicked with the mouse. This opens the **SYNCHRONIZE MASTER DATA** dialog. The current system messages can then be read here. When the **SYSTEM MESSAGES** dialog is closed, the icon disappears until the next system message occurs. The **UTILITIES/SYSTEM MESSAGES** menu path can be used to call up the **SYSTEM MESSAGES** dialog at any time. All system messages are visible until EPLAN is exited. The current system messages are only deleted from the list when EPLAN is exited.

4.3 Coordinate systems

The EPLAN coordinate system has the following structure. There are different points of view: the graphical system and the logical systems for the areas of electrical engineering, fluid power and process engineering.

4.3.1 Graphical coordinate system



The graphical system has its origin (X,Y = 0,0) at the lower left of the page. This means that when the page scale changes, the page is enlarged upwards and to the right.

Fig. 4.50
Graphical

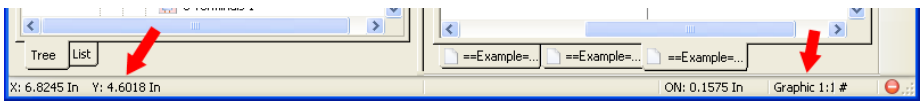


Fig. 4.51
Display in the status bar

4.3.2 Logical coordinate system

The logical system has its origin (X,Y = 0,0) at the upper left. These positions are always measured from the original element and its origin (usually the so-called insertion point) in different directions.

In this case these are negative values, since the property is moved downwards and to the left based on the coordinate input X/Y in the property dialog.

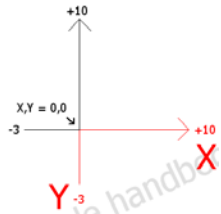


Fig. 4.52 Logical

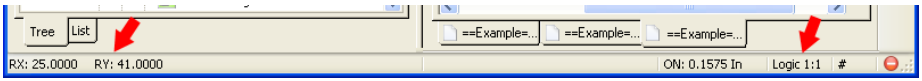


Fig. 4.53 Display in the status bar

4.4 Grid

In EPLAN can, a fixed **grid** can be used to assist the user when placing (e.g. symbols) or when drawing (e.g. panel structures). The grid is visually displayed with small dots. The display of the grid can be enabled or disabled via the **VIEW/GRID** menu or the **CTRL + SHIFT + F6** shortcut key.

Five different grids can be entered in the **OPTIONS/SETTINGS/USER/GRAPHICAL EDITING/2D** settings. I recommend the following values for the grid settings: grid A = 0.25 mm; grid B = 0.5 mm; grid C = 1 mm; grid D = 2 mm and grid E = 4 mm.

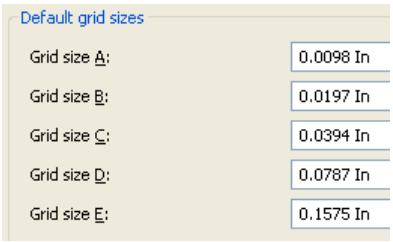


Fig. 4.54 Grid settings

These grid settings can then be selected in the **EDIT/OTHER/GRID** [A to E] menu or via the **VIEW** toolbar.

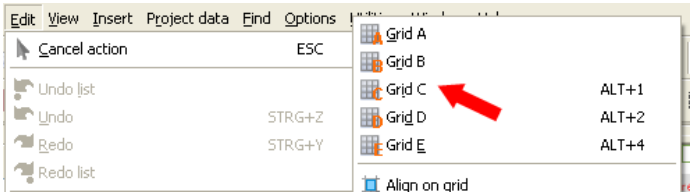


Fig. 4.55 Other menu item, with grid settings

The **VIEW** toolbar has other buttons that will be briefly explained here. They can all be assigned to a keyboard shortcut as well.

Fig. 4.56 Toolbar view

	This button turns the grid on or off.
	This function enables or disables Snap to grid. When Snap to grid is enabled, elements can only be placed on the activated grid. Intermediate positions or free placement are not possible. When the grid snap is switched off, then all elements can be freely placed (this also applies to logic elements such as symbols).
	This Snap to grid function allows you to align different elements (outside the grid) to the currently set grid.[1)]
	During drawing, the object snap will let you, for example, automatically link a line to insertion points (logical objects) or element points (graphical objects) (let them "capture" it).
	The design mode allows you to align graphical elements to specific points, i. e. to place them on specific coordinates. If the design mode is enabled, first you select the action, Move for example, and then you select the object and define its starting and end points.

[1)] The procedure is as follows: Click the button on the **View** toolbar. Then use the mouse to pull a window around the misaligned elements. The window can be pulled in any direction. When finished (all elements are contained in the window), release the mouse button and EPLAN will align all elements that are inside the window to the grid. It also works the other way around. First select the object(s) you want to align and then click the button. EPLAN will then align the selected objects to the set grid.

4.5 Increments, coordinate input

In addition to the grid settings, EPLAN also allows the input of increments and coordinates. These two features relate to the cursor position.

4.5.1 Increment

The increment is set via the **S** key or in the **OPTIONS/INCREMENT** menu. EPLAN opens the **SELECT INCREMENT** dialog.

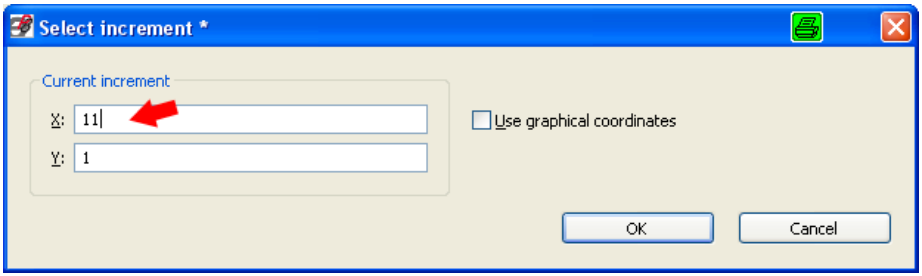


Fig. 4.57
Integer increment setting

The increments for the cursor can be entered into the **Current increment X and Y** fields of this dialog. If the **USE GRAPHICAL COORDINATES** option is deactivated, then it is only possible to enter integer values, such as 1, 3 or 17.

Increment input

If the **USE GRAPHICAL COORDINATES** option is activated, then it is also possible to enter real numbers such as 1.75 or 3.88 because the graphical coordinates are then evaluated.

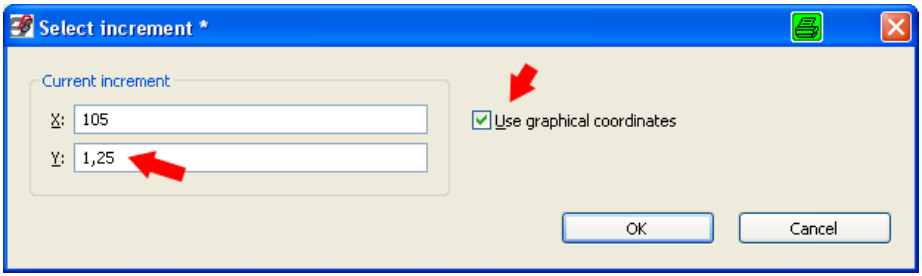


Fig. 4.58
Real-number increment setting

EPLAN also displays the information in the status bar, regardless of whether in the **SELECT INCREMENT** dialog the **USE GRAPHICAL COORDINATES** option has been activated or deactivated.

4.5.2 Coordinate input

Coordinates can also be directly entered into EPLAN to allow jumping to a particular point by clicking **OK**.

Coordinate input

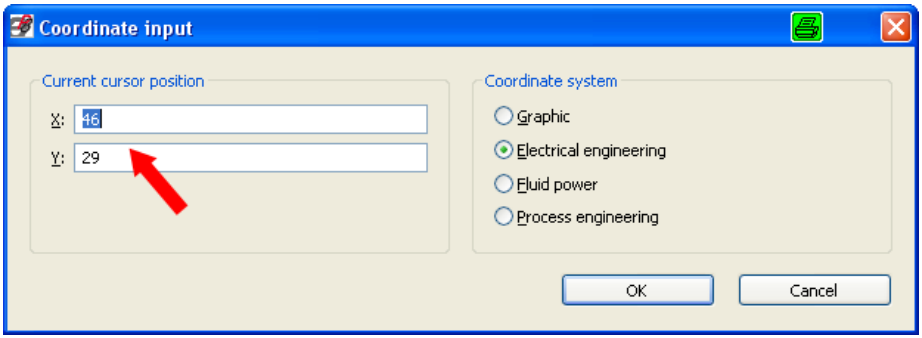


Fig. 4.59
Coordinate input

The **COORDINATE INPUT** dialog is started with the **P** key or via the **OPTIONS/COORDINATES** menu. The X and Y positions can be directly entered in this dialog. As soon as the dialog is closed using the **OK** button, EPLAN jumps to the specified X/Y coordinates.

NOTE: The mouse should not be used here because it would probably shift the position where you want to jump.

During coordinate input, it is also possible to switch between the different coordinate systems (graphic, electrical engineering, fluid power and process engineering).

4.5.3 Relative coordinate input

EPLAN allow both **static** coordinate input (described in the previous chapters) and **relative** coordinate input. The **RELATIVE COORDINATE INPUT** function is accessed via the **OPTIONS** menu.

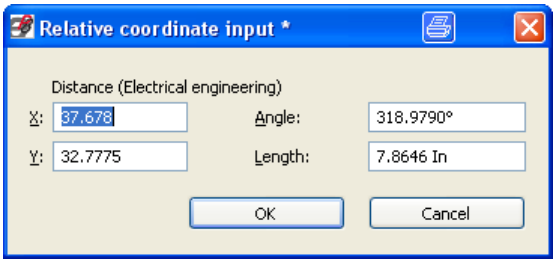


Fig. 4.60 Relative coordinate input

This function can be used to place the cursor (on which a line for placement might "hang") in a new position. This means that you could start drawing a new line and then call the relative coordinate input. In the simplest case, you could enter the length of the line, and EPLAN would use this length, place the cursor in the position and terminate the drawing of the line. In contrast to the static coordinate input, the input here is always relative to the current cursor position.

NOTE: If a grid is enabled and the jump to the cursor end point is not on the grid, EPLAN will still jump to the next grid point. You should take this into consideration when drawing.

4.5.4 Move base point

The **MOVE BASE POINT** function, accessible via the **OPTIONS** menu or the **O** key, sets a new base point (X,Y = 0), which can be manually set to be different from the **standard base point** (depending on the coordinate system, e. g. graphic, electrical engineering).

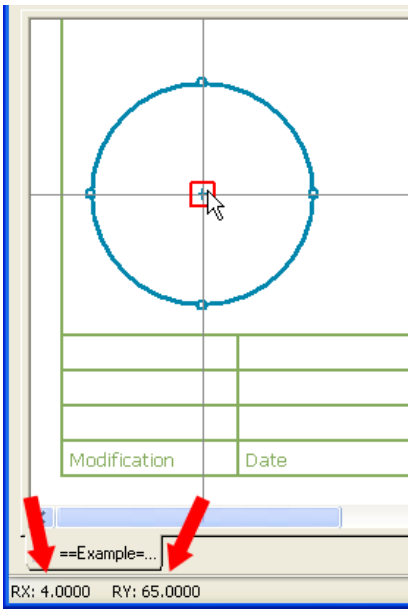


Fig. 4.61 Standard base point

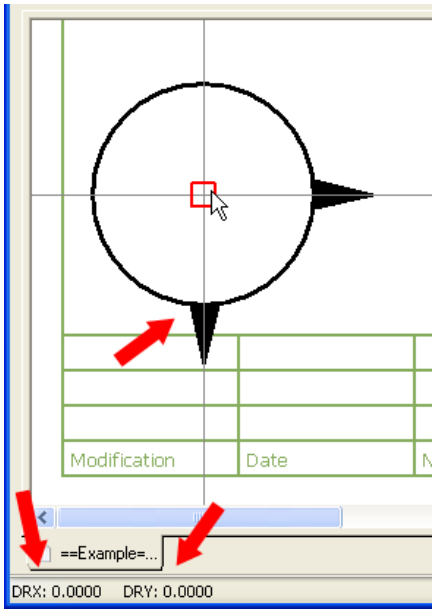


Fig. 4.62 New base point

In the example (center of the circle), the base point is X = 256.00 mm and Y = 244.00 mm. In order to redefine the base point, for example, to calculate the horizontal values from the center of the circle, you place the cursor on the **center**, press the **O** key and then press **ENTER**.

EPLAN now generates a new current base point for this page with X,Y = 0. All positions are immediately calculated using this new base point and displayed in the status bar. To differentiate from the usual values displayed (X = n; Y = n), EPLAN displays the values of the base point shift with a starting D. In the status bar, the X and Y values are displayed as DX = n and/or DY = n.

You can reset the base point shift using the **O** key or you can simply change pages.

4.6 Graphical editing functions

In addition to schematics, EPLAN allows you to create a type of CAD drawing (CAD = Computer Aided Design). EPLAN also offers true to scale drawings with many dimensioning possibilities and many additional well-known CAD functions, such as stretching, trimming (modifying the length) and grouping of elements.

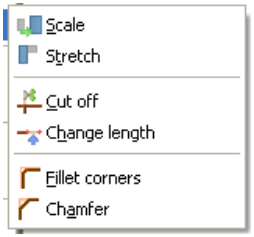


Fig. 4.63 Edit/Graphic menu

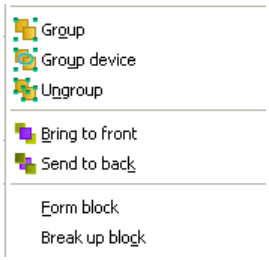


Fig. 4.64 Edit/Other menu

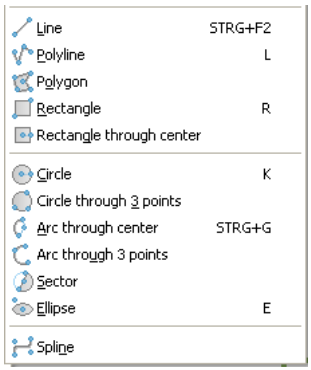


Fig. 4.65 Insert/Graphic menu

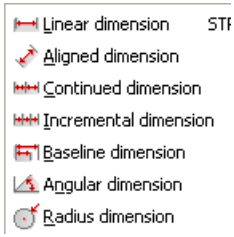


Fig. 4.66 Insert/Dimensioning menu

These graphical functions can be accessed via the **INSERT/GRAPHIC [FUNCTION TYPE]** menu. Special functions such as stretching, grouping and converting a component to a graphic are located in the **EDIT/GRAPHIC** menu and others such as Group, Bring to front, etc. are located in the **EDIT/OTHER** [function type] menu.

The dimensioning functions are located in the **INSERT/DIMENSIONING** menu.

EPLAN remains a pure CAE system (CAE = Computer Aided Engineering), but the ability to work with CAD functions makes certain pure drawing tasks much easier.

4.6.1 Graphical objects – lines, circles, rectangles

With the *Circle*, *Rectangle*, *Line* or *Polyline* functions and many other options, EPLAN enables you to create graphical drawings. This will not be explained in detail here because the calls and the functions are very simple and easy to understand.

TIP: When using the graphical functions, it is a good idea to switch on the *Grid* or to enable the **SNAP TO GRID** function.

You should also "play" around with the page's set grid. If you require 1 millimeter increments, then you should set the grid for this page to 1 (1 millimeter), either by using the buttons of the grid settings or directly in the page properties (i.e. via the **Grid <11051>** page property).

To draw a rectangle, for example, you use the **INSERT/GRAPHIC/RECTANGLE** menu or the **R** key call up a rectangle and place the starting point on the drawing. Now there are three options.

Option 1: You draw rectangle using the mouse or the keyboard by counting grid increments. You can also adjust the height and width later in the rectangle's properties.

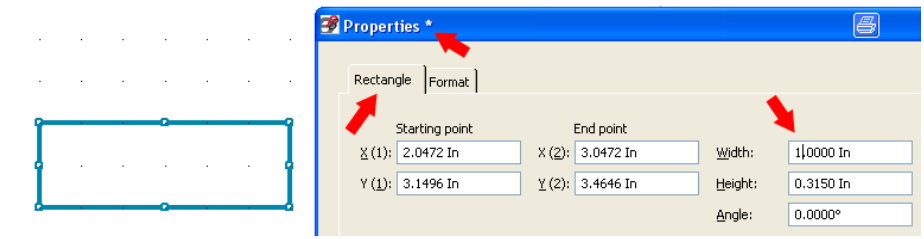


Fig. 4.67 Drawing a simple rectangle with manual width and height

Option 2: You start drawing the rectangle and define its size using the relative coordinate input.

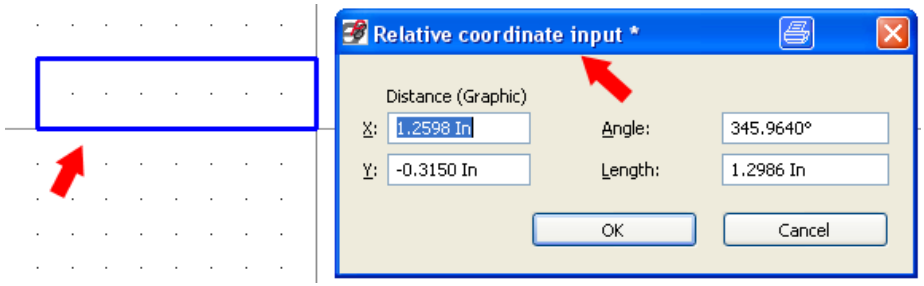


Fig. 4.68 Drawing a rectangle on the basis of relative coordinate input

Option 3: You use the **OPTIONS** menu to activate the **INPUT BOX** (always "hangs" on the cursor). Here, by directly entering values such as width and height, you can let EPLAN draw the rectangle.

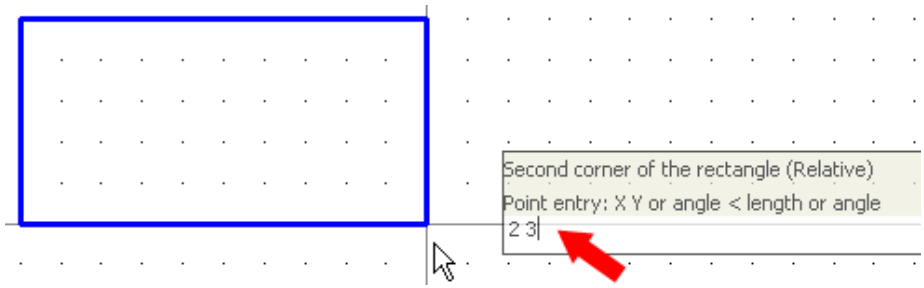


Fig. 4.69 Drawing a rectangle by direct input in the input box

4.6.2 Trim, chamfer, stretch and more ...

In addition to the pure graphical functions, such as drawing a rectangle or circle, EPLAN also lets you edit these graphical objects later. Such editing could include the following: changing the length of graphical elements, cutting out parts of graphical elements, rounding corners and much more. A few examples of the edit function in the **EDIT/GRAPHIC** menu will briefly illustrate the options.

NOTE: All these examples were carried out with the input box (in the **OPTIONS MENU**) activated.

4.6.2.1 Rotate

The **ROTATE** function allows you to rotate a selected object by any angle. It is called via the **EDIT/GRAPHIC/ROTATE** menu. Now, EPLAN waits for you to select the object to be rotated. Then, you have to define the center of rotation (around which the object is to be rotated). Now you enter the rotation angle directly in the input box and confirm with **ENTER**. EPLAN has rotated the object.

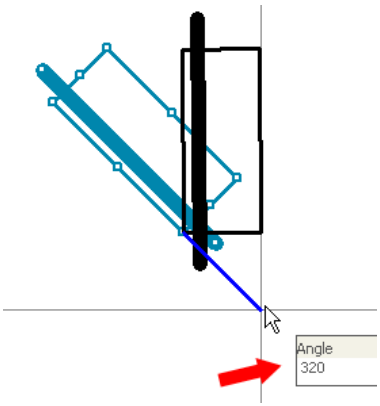


Fig. 4.70
Rotate

NOTE: Rotating is not limited to graphical objects. You can also rotate pure symbols (components). This eliminates the need to change the component variant in the properties (components) on the *Symbol/function data* tab. But you can only use the rotation angles that already exist as a symbol variant.

4.6.2.2 Mirror

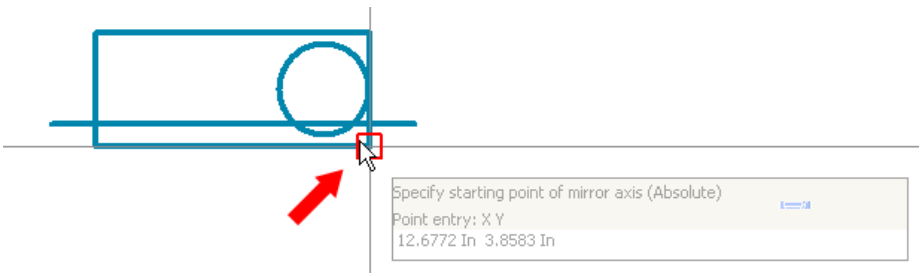


Fig. 4.71
Defining the starting point of the mirror axis

Another function is the **MIRROR** function. This function is called via the **EDIT/MIRROR** menu. Then you select the object to be mirrored. Now EPLAN waits for you to input and/or define the *Starting point of the mirror axis*.

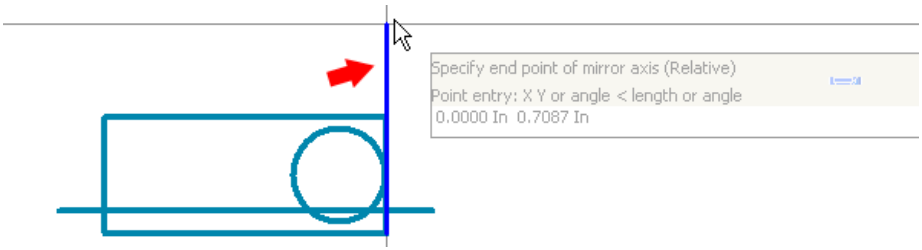


Fig. 4.72 Defining the end point of the mirror axis

After you have set the starting point of the mirror axis, you have to define the *End point of the mirror axis*. When you click the end point, EPLAN will mirror the object along the mirror axis defined.

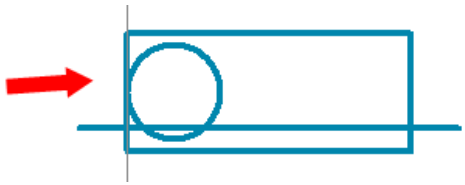


Fig. 4.73
Mirrored object

NOTE: Mirroring is not limited to graphical objects. You can also mirror pure symbols (components). This again eliminates the need to change the component variant in the properties (components) on the *Symbol/function data* tab. But you can only use the mirroring angles that already exist as a symbol variant.

4.6.2.3 Scale

Aside from rotating and mirroring, objects can also be enlarged or reduced. In EPLAN, these actions are summarized under the term **Scale**. The Scale function is called via the **EDIT/GRAPHIC/SCALE** menu.

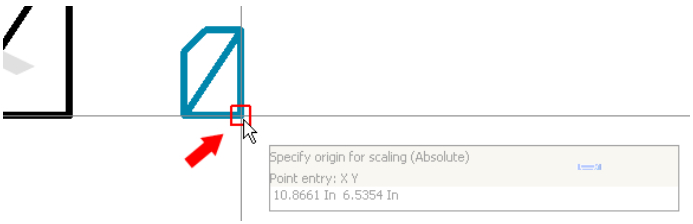


Fig. 4.74
Defining the origin for scaling

After you select the *Origin* you can click it. EPLAN opens the **SET SCALE** dialog.

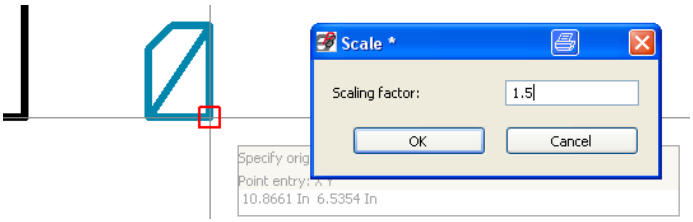


Fig. 4.75 Setting the scaling factor

When you confirm this dialog with **OK**, EPLAN scales (here: reduces) the selected object.

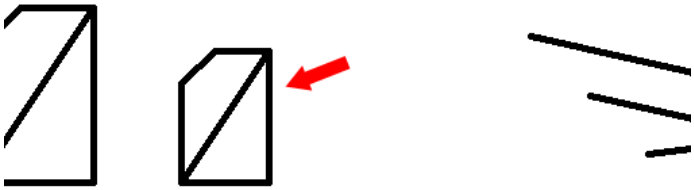


Fig. 4.76 Scaled object

When using the **Scaling factor** note that EPLAN expects a factor less than 1 to reduce and a factor greater than 1 to enlarge. You must enter a decimal point in the scaling factor for it to work.

4.6.2.4 Stretch

The **STRETCH** function allows you to lengthen and shorten objects. It is called via the **EDIT/GRAPHIC/STRETCH** menu. Then, you select the right-hand section of the rectangle (in this example) with a window. EPLAN places round markers in the corners and waits for you to define the starting point of the stretch.

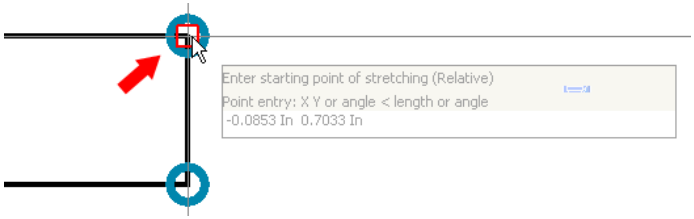


Fig. 4.77 Defining the starting point of the stretch

Once you have defined the **Starting point**, you use the mouse or keyboard to set the end point.

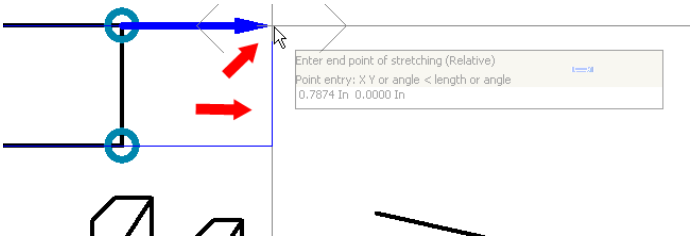


Fig. 4.78 Defining the end point of the stretch

After you have defined the **End point**, confirm it with **OK**, and EPLAN will execute the stretch.

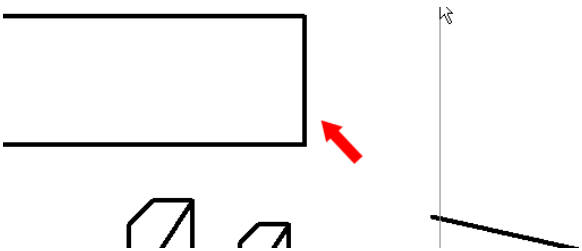


Fig. 4.79 Stretched object

Of course, you can change the length of more than just simple rectangles.

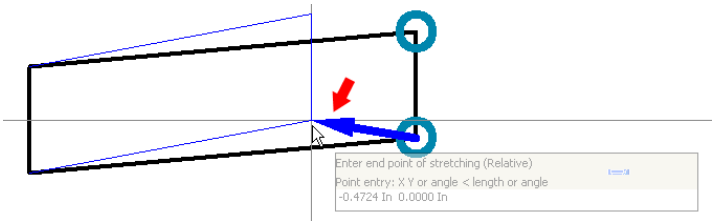


Fig. 4.80 Lengthening or shortening a different object

4.6.2.5 Cut off

The **CUT OFF** function, also called **Trim** in the CAD world, is used to "cut" a so-called section out of elements. It is called via the **EDIT/GRAPHIC/CUT OFF** menu. After you have selected the function, EPLAN waits for you to select the section to be removed.

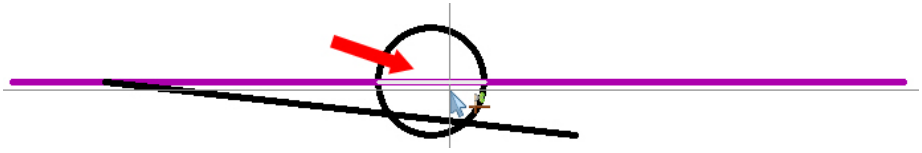


Fig. 4.81 Selecting the element to be removed

If, as shown in Fig. 4.81, you click the small section within the circle, EPLAN will cut it out.

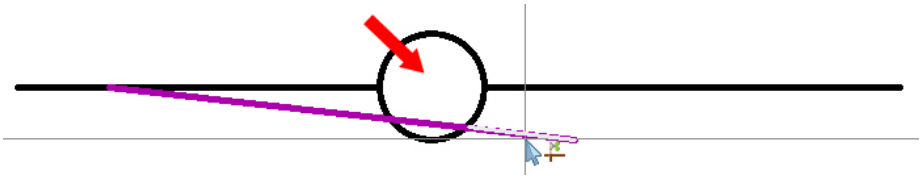


Fig. 4.82 Cut-off section

A typical example of this function would be the cutting off of sections of a populated horizontal rail or a mounting rail in a mounting panel layout.

NOTE: Using cutoff for grouped objects is not quite the same. If you want to cut sections off grouped objects, you have to keep the **SHIFT** key pressed and then cut the relevant section.

4.6.2.6 Change length

The **CHANGE LENGTH** menu item, as the name indicates, changes the lenght of objects. It is called via the **EDIT/GRAPHIC/CHANGE LENGTH** menu. Once this function has been called, you click the relevant object to change its length via the input box or by using the keyboard and/or mouse.

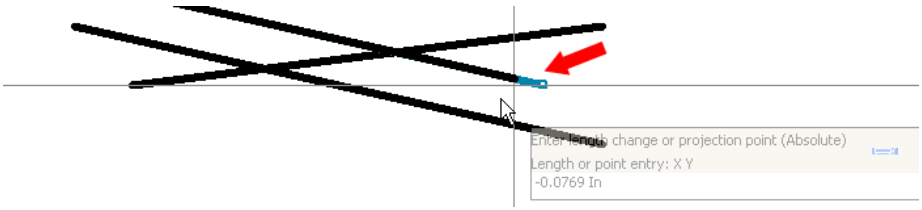


Fig. 4.83 Changing the length of a line

4.6.2.7 Fillet corners

The **FILLET CORNERS** function rounds off corners. It is called via the **EDIT/GRAPHIC/FILLET CORNERS** menu. After launching the function, you can click the desired corner and define the *fillet radius* in the input box.

Fig. 4.84 Filleting corners

When you press the **ENTER** key, EPLAN fillets the corner according to the entered radius.

Fig. 4.85 Filleting the next corner

If you then click the next corner, it will be filleted automatically using the same radius.

4.6.2.8 Chamfer

The last function is **CHAMFER**. Chamfer does not fillet corners, it gives them a sloping edge. Similar to all other functions, it is called up via the **EDIT/GRAPHIC/CHAMFER** menu.

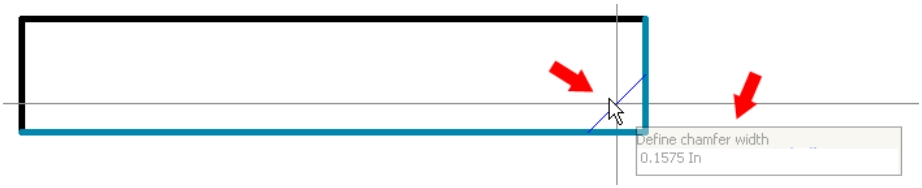


Fig. 4.86 Chamfering a corner

After starting the function, selecting the object and clicking the first corner, you enter your chamfer parameters directly in the input box. EPLAN uses these parameters after you press **ENTER** or left-click the object. The same chamfer setting can be applied immediately to the next corner.

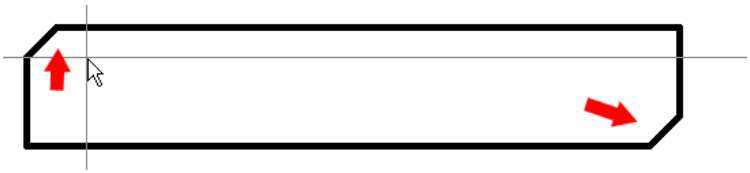


Fig. 4.87 Several chamfered corners

4.6.3 Group and ungroup

When you have several graphical elements, it can be difficult to move or, for example, to scale (enlarge, reduce) all these individual graphical elements. Every element must be manually adjusted to its new size via the **PROPERTIES** dialog, or when multiple elements are selected, the elements that are not to be scaled must be manually deselected.

With the **GROUP** and **UNGROUP** functions in the **EDIT/OTHER** menu, EPLAN allows particular elements that belong together to be grouped and then be subsequently regarded as a single element. You use the mouse to select all elements that belong together and then call up the **GROUP** command from the menu (or alternatively use the **G** key).

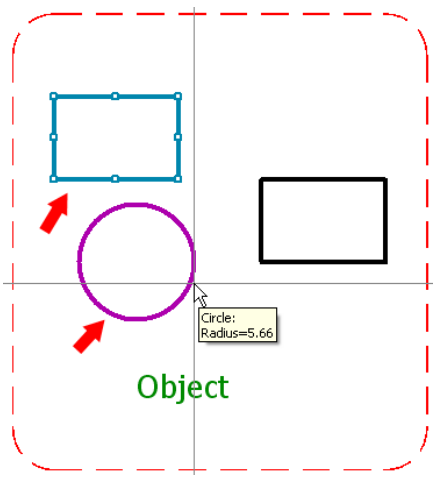


Fig. 4.88 Individual objects

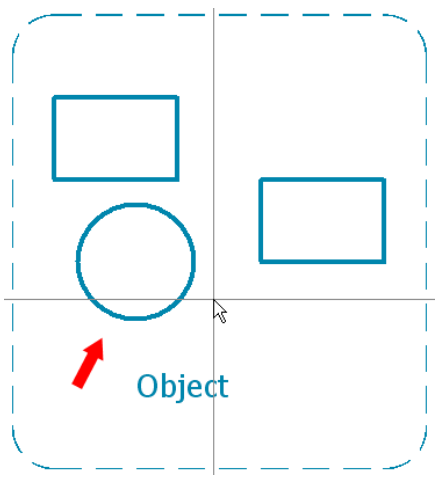


Fig. 4.89 Grouped objects

The grouping can, of course, be removed again. You simply select one of the grouped elements (EPLAN automatically selects all the elements in the group). In the **EDIT/OTHER** menu, you then select **UNGROUP**. EPLAN removes the grouping and all elements are once more individually accessible.

Individual elements within a grouping can also be edited independently. Press and hold the **SHIFT** key, and then double-click the desired object within the group. The object's property dialog opens, and the properties can be edited – without have to ungroup the objects. When all entries are complete, EPLAN applies the changes to this object. The remaining group objects are not affected.

4.6.4 Copy, move, delete, ...

In addition to inserting and drawing new elements, you can also copy and delete elements.

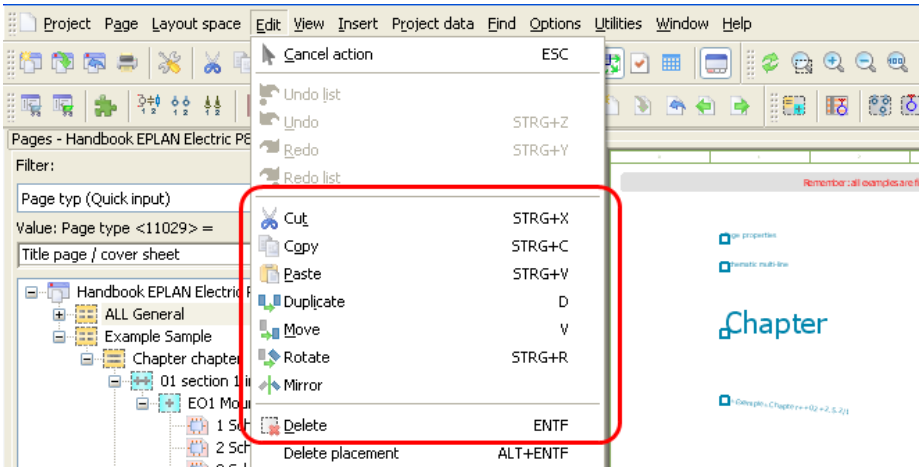


Fig. 4.90 Editing functions

These functions are available in the **EDIT** menu. In this menu, you can call up the functions **COPY**, **MOVE**, etc. You can also use the keyboard to call up these functions: **CTRL + C**, for example, for **COPY**, or **CTRL + X** for the **CUT** function. To use the functions on elements, you have to first select an element (left mouse click) and then select the function.

4.6.5 Dimensioning

In addition to actually drawing, you can dimension the drawn elements. EPLAN offers accurately scaled dimensioning, depending on the page scale that is set.

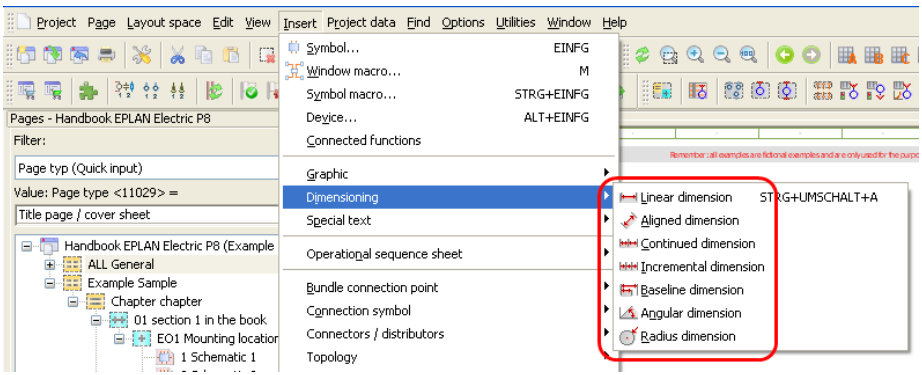


Fig. 4.91 Dimensioning functions

EPLAN comes with a number of dimensioning functions. Dimensioning types include, for example, continued dimension and radius dimension. The procedure for dimensioning elements is just as simple as drawing the elements themselves. The dimensioning types can be accessed in the **INSERT/DIMENSIONING [DIMENSIONING TYPE]** menu.

The various dimensioning methods will be illustrated with an example.

TIP: It is recommended that you enable the **OBJECT SNAP** function (accessible via the **OPTIONS/OBJECT SNAP** menu) when dimensioning. This allows EPLAN to start precisely at the ends (or the midpoints) of elements, without you having to tediously try to reach the ends of a line, for example. This usually cannot be done precisely.

In this example, a rectangle will be dimensioned. Use the **INSERT/DIMENSIONING** menu to call up the **LINEAR DIMENSION** function.

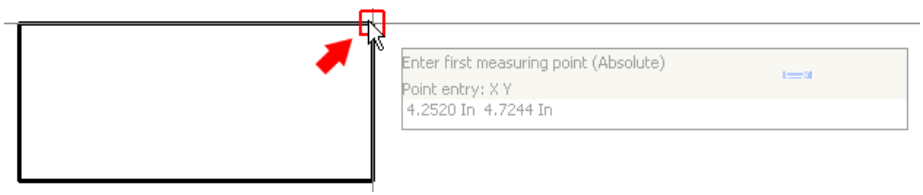


Fig. 4.92 Dimensioning starting point

Use the cursor to select the first **Dimension point** (corner of the rectangle) and then click it. EPLAN sets this point as the start of the dimensioning.

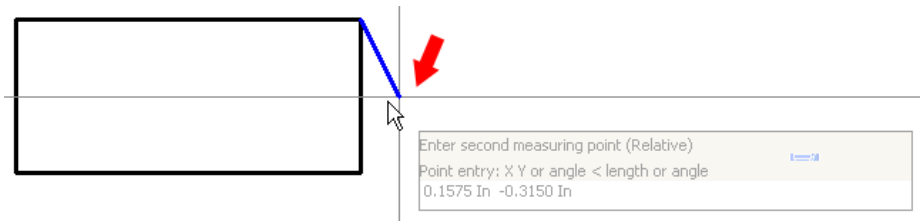


Fig. 4.93 Defining the second dimension point

Now look for the second dimension point. The first dimension "hangs" on the cursor and is placed using the mouse or with a click.

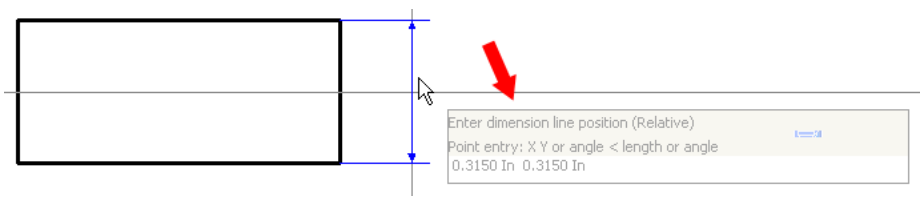


Fig. 4.94 Defining the position of the dimension line

Once the position of the dimension line has been set, EPLAN will dimension the object. For the dimensioning itself, there are a number of properties that can be called by double-clicking the dimension itself.

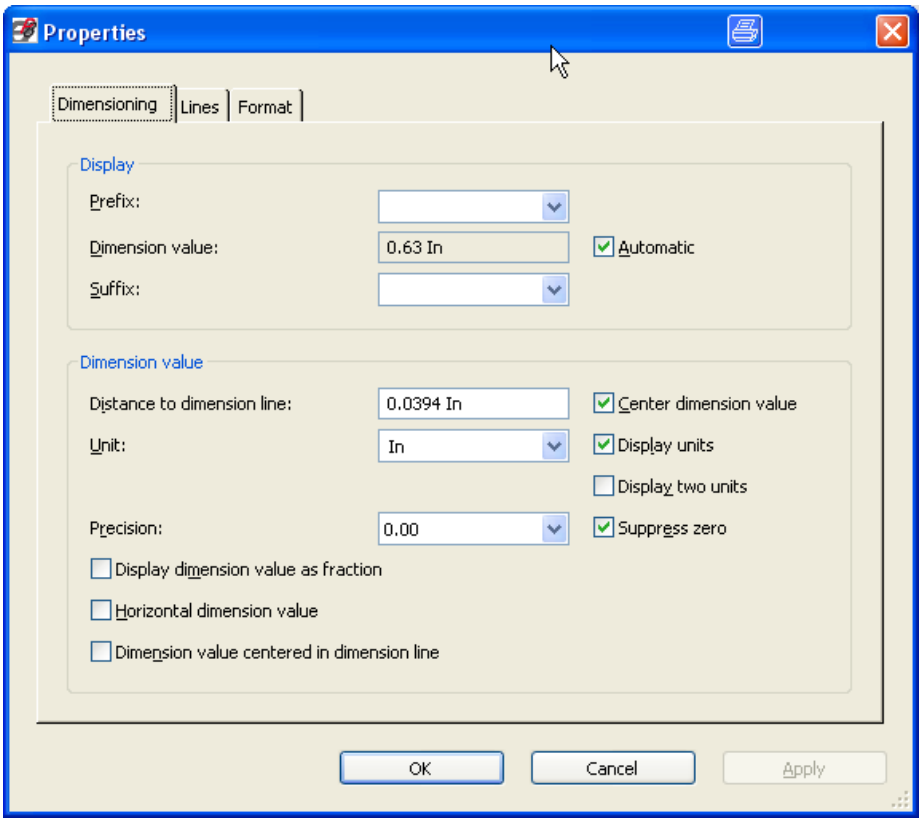


Fig. 4.95
Setting the current dimension

There are also additional tabs for *lines* and the *format* of the dimension labeling.

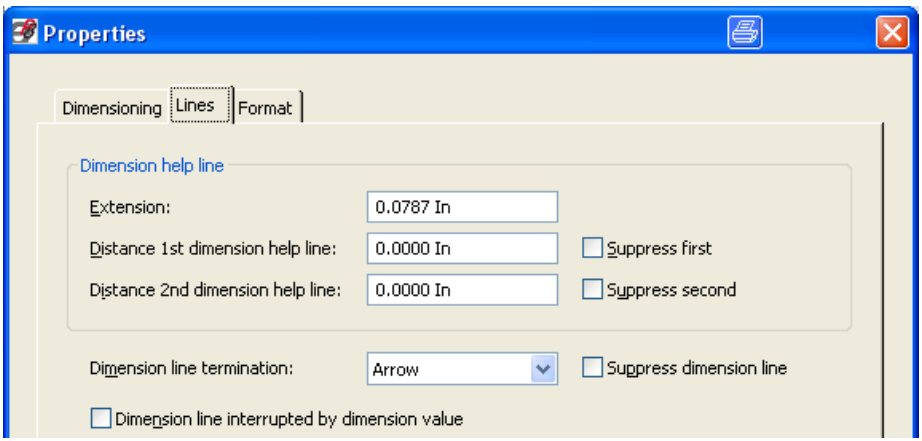


Fig. 4.96
Setting the dimension lines

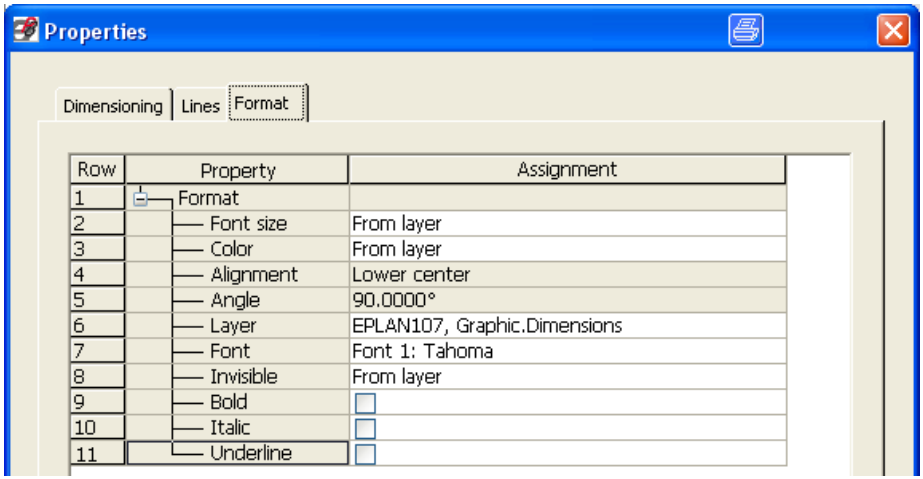


Fig. 4.97 Setting the format (dimension values, etc.)

The *Format* tab is where you define the settings for dimension values, etc. This does not affect the dimension lines. To control the dimension lines directly, you have to modify the **EPLAN107** layer in layer management.

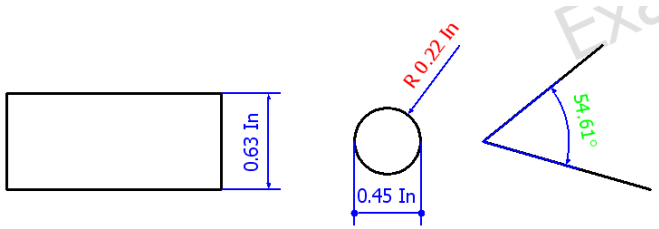


Fig. 4.98 Examples of different dimensions

4.7 Texts

In addition to graphical or electrical engineering functions such as symbols or devices, EPLAN also allows you to enter simple free texts.

Fig. 4.99 Free graphical text

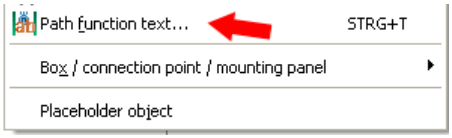


Fig. 4.100 Function text

This is not a new feature but rather a necessity to allow various schematic functions to be clarified with text entries, and to provide a better explanation of the schematic and its functions, or to fill reports with additional information (or to have EPLAN fill them).

Plain text is always used in EPLAN. No numbering or coding system is used, so that the user can always see what the text is and does not need to look up a number in a text file to know which text entry the number refers to.

4.7.1 Normal (free) texts

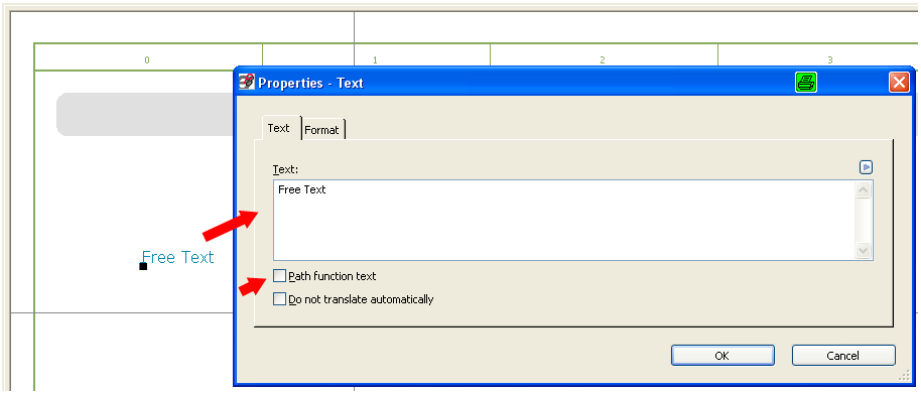


Fig. 4.101 Free text

EPLAN allows the entry of normal (free) texts. Normal texts are texts that have no further functionality (except their visual display).

Translation of this type of text is possible without problems. Normal text should usually only be used to place text that does not need to be used in any other way, for example to provide extra information in a black box. Normal texts are inserted via the **INSERT/GRAPHIC/TEXT** menu or by pressing the **T** key.

It is only possible to later change normal (free) text to function text by activating the **PATH FUNCTION TEXT** selection box.

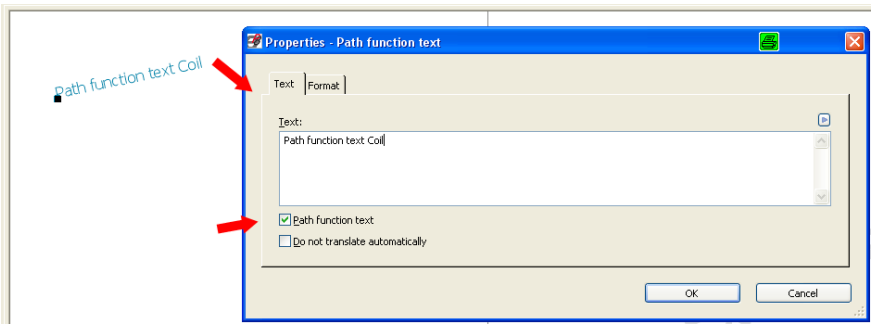


Fig. 4.102 Path function text selection box

4.7.2 Path function texts

In addition to normal (free) texts, EPLAN also has path function texts. These are equipped with “intelligence”. Path function texts should generally be used to automatically fill specific devices with function text, for example based on the page path.

Automatic path function text

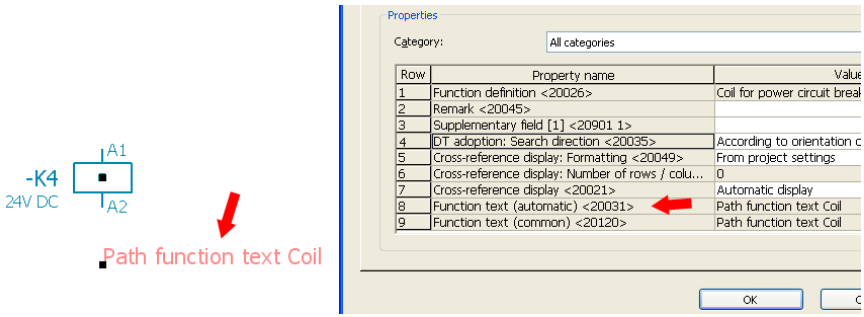


Fig. 4.103 Path function text

EPLAN uses the path function text for embedding in different reports (depending on the structure of the form). It is therefore advantageous to always work with path function texts, since subsequent conversion from normal texts to path function texts is possible, but requires unnecessary extra work.

Path function text is inserted via the **INSERT/PATH FUNCTION TEXT** menu. If path function texts are generally to be used, the keyboard shortcut for free text should be redefined to path function text. In this case, the keyboard shortcut for normal (free) text should be set to a different keyboard shortcut.

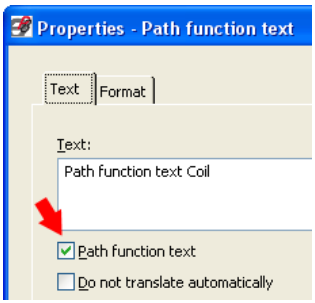


Fig. 4.104
Path function text

TIP: Normal (free) texts can be converted to path function texts by simply activating the *Path function text* setting in the **PROPERTIES – TEXT** dialog. After you exit the **PROPERTIES – TEXT** dialog, the layer is also automatically set to the standard layer for path function texts: **LAYER110.Graphic. Path function texts**.

To generate a report for a path function text for a device, it normally has to be directly (in the path) below or above the devices (the insertion points of the path function text and device are then "in sequence").

But this might not be desired, for reasons of space or because the path function text is to apply equally to several devices. To avoid having to align the path function text several times in the path of the individual devices, you can set the **GRAPHICAL EDITING/GENERAL** project setting and, in this case, *Extend path function text on the schematic path*.

This way, it will not matter whether the insertion point is directly below the device or not. It is sufficient if the path function text and the device are located in the same path (the column).

NOTE: But extending the reporting of text to the path can also result in the text being assigned to objects in the path where this is not wanted. Therefore, you need to be careful when using the *Extend path function text on the schematic path*.

4.7.3 Special texts

Special texts are another type of text used in EPLAN. Special texts can be *page property texts* or *project property texts*. They are generally used in forms (project properties) or in the plot frame (page properties) and less so as text on project pages.

Page property or project property texts are inserted via the **INSERT/SPECIAL TEXT/PROJECT PROPERTY** menu or **PAGE PROPERTY**. This way you can place text from the project properties or the page properties on pages.

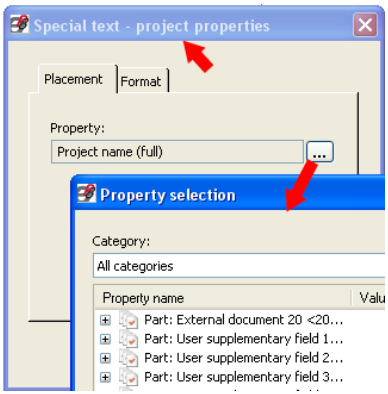


Fig. 4.105
Special texts

The following example clarifies this application. In the project properties, 400V AC for the main current voltage has been entered in **Supplementary field [10] <10901 10>**.

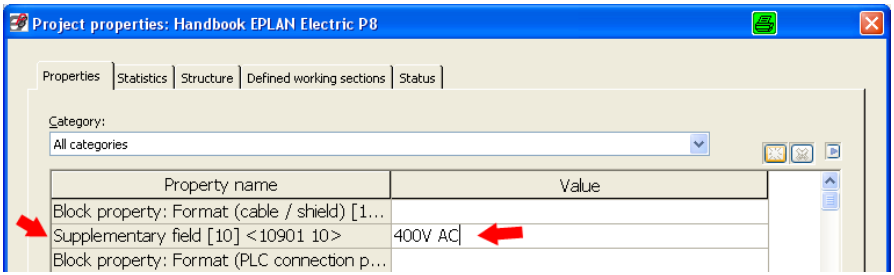


Fig. 4.106
Supplementary field [10] <10901 10> project property

This project property can now be used in the schematic, for example to show the main current voltage. The project property is called and placed via **SPECIAL TEXT/SPECIAL TEXT/PROJECT PROPERTY**.

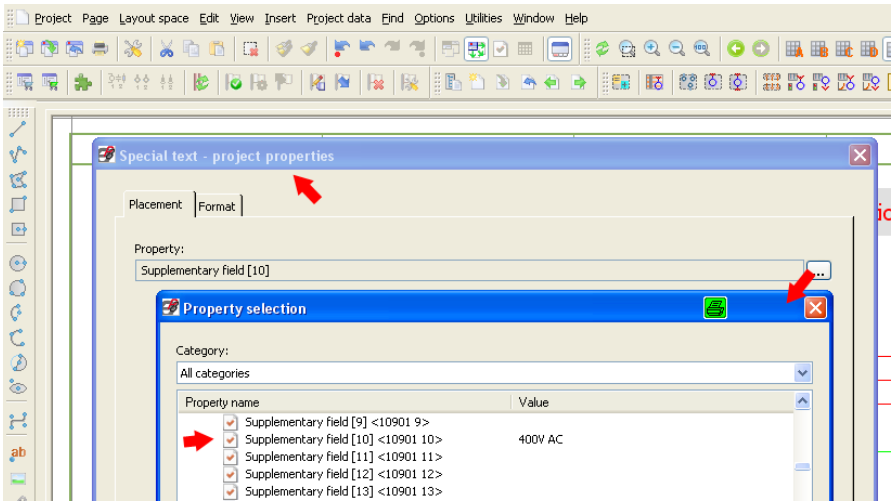


Fig. 4.107
Selecting project properties

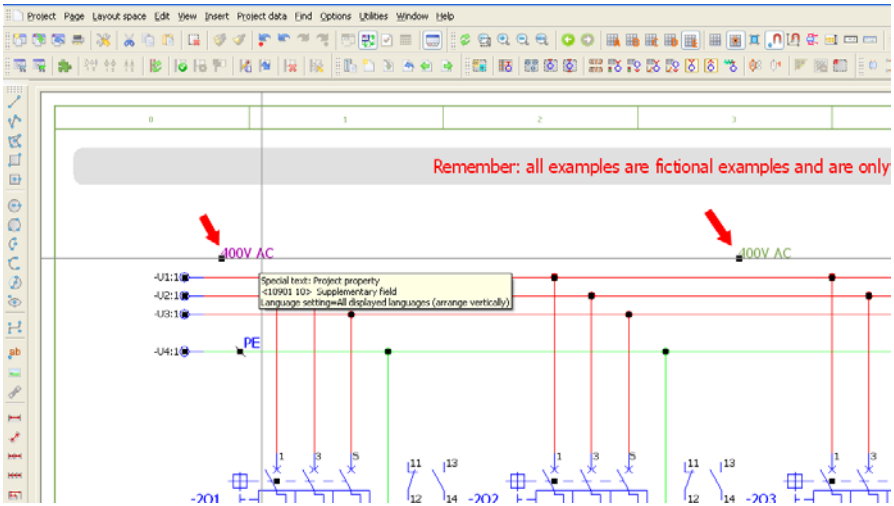


Fig. 4.108
Placed project property

If the voltage later changes or additional information is needed, the input in the project properties can simply be changed, and everywhere in the project where this project property is displayed will be automatically updated. This can save a lot of time in manually changing this information.

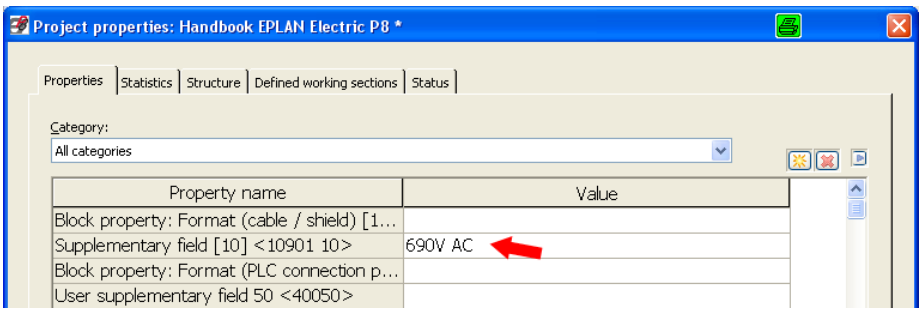


Fig. 4.109 Changed input in the project property

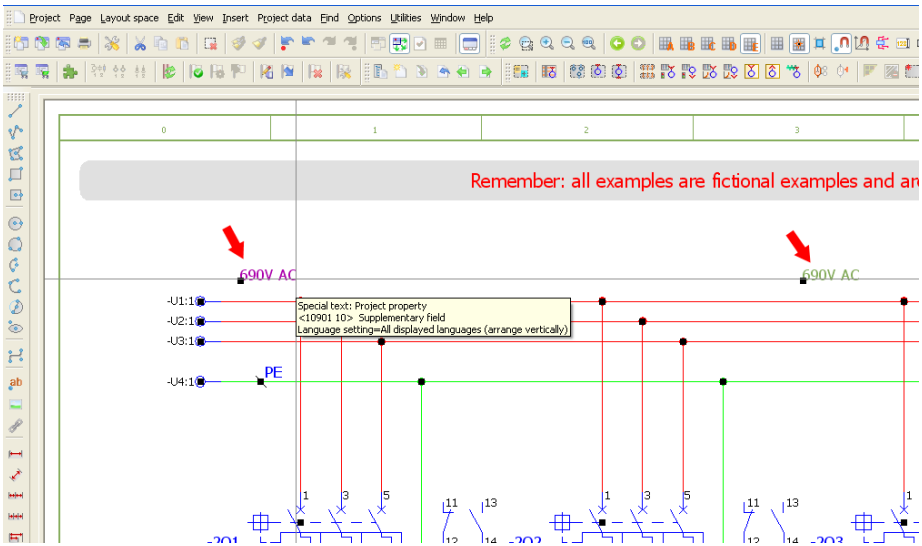


Fig. 4.110 Changed display in the project

4.7.4 The Properties – Text dialog

To insert and format texts according to requirements, you have to call up the text's **PROPERTIES – TEXT** dialog. You select text and press **ENTER** key or a left click to open the dialog. The dialogs for normal text and path function text are the same. Only the window title bar of the text input field has a different text type name.

4.7.4.1 Text tab

This *Text* tab contains the *Input field* for the text. You enter text in the *Text* field. You can translate the text at this point, or remove the translation.

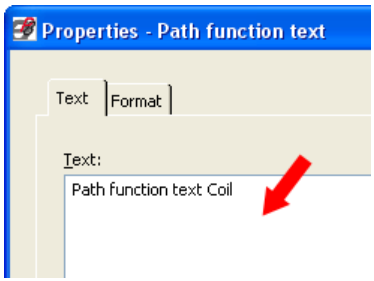


Fig. 4.111
Text input field

You call up the popup menu using the right mouse button or the button. In addition to the usual Windows text functions such as **COPY**, **PASTE**, **DELETE**, and **CUT**, the popup menu also contains other functions such as **SPECIAL CHARACTERS** or **TRANSLATE**, **REMOVE TRANSLATION**, **REMOVE MARKING** and **MULTILINGUAL INPUT**.

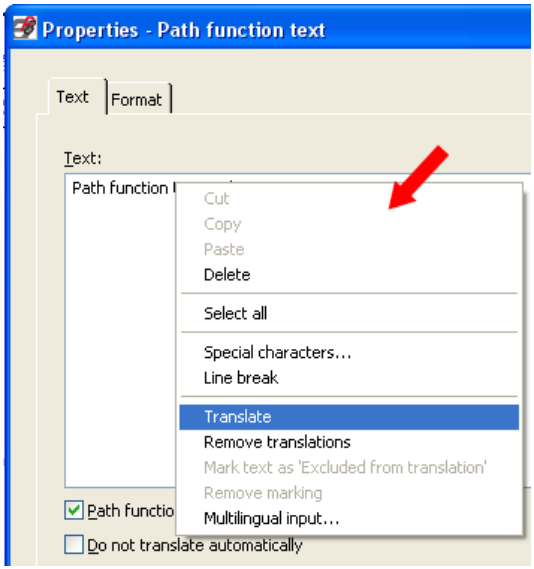


Fig. 4.112
Popup menu

4.7.4.2 Format tab

The *Format* tab contains all functions needed to format the text (*Font size* or *Font*) or, for example, to assign it a different layer. The formatting options and settings can be divided roughly into the three nodes; *Format*, *Border* and *Value/unit*.

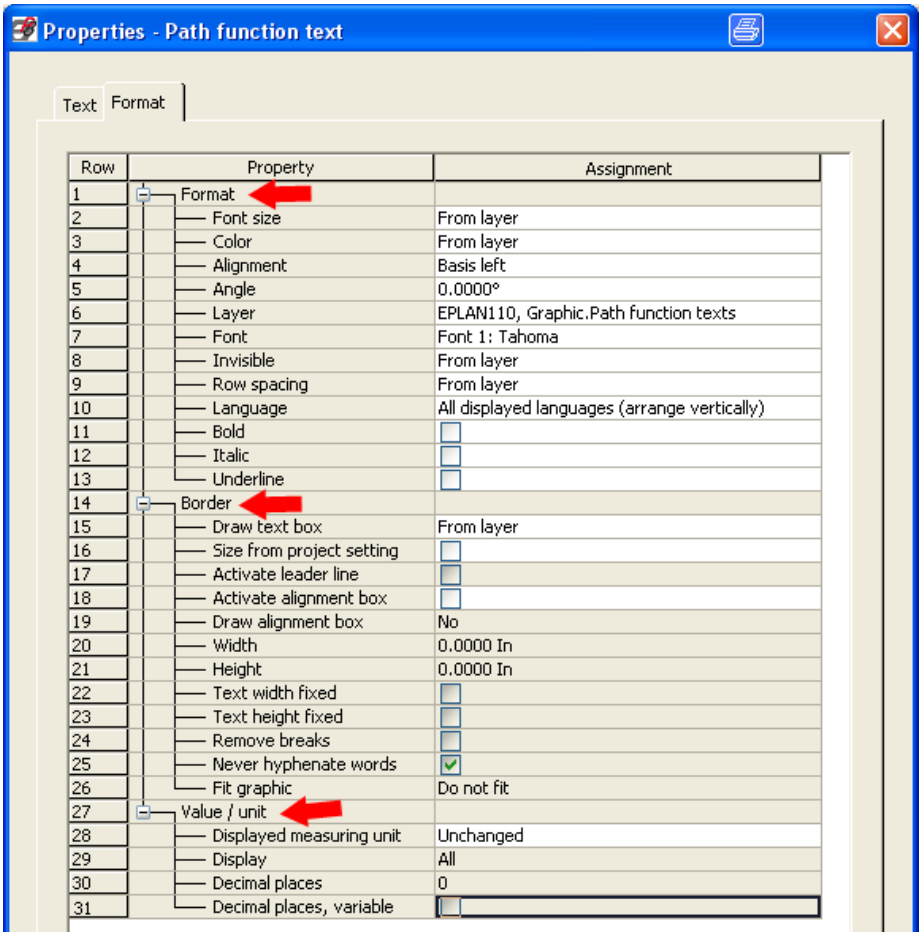


Fig. 4.113 Formatting options for texts

Most fields are selection fields, meaning that when a field like the **Alignment** field is opened, all selection options are displayed.

A number of fields are also free input fields, such as the field for the **Angle** property. Here, you can select default values, such as the layer setting default for the angle, or enter your own values.

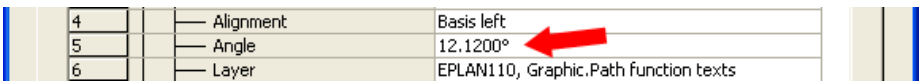


Fig. 4.114 Free input fields

The next section contains a few explanations regarding selected properties of the *Format* tab.

4.7.4.2.1 Format tab – Format node – Language

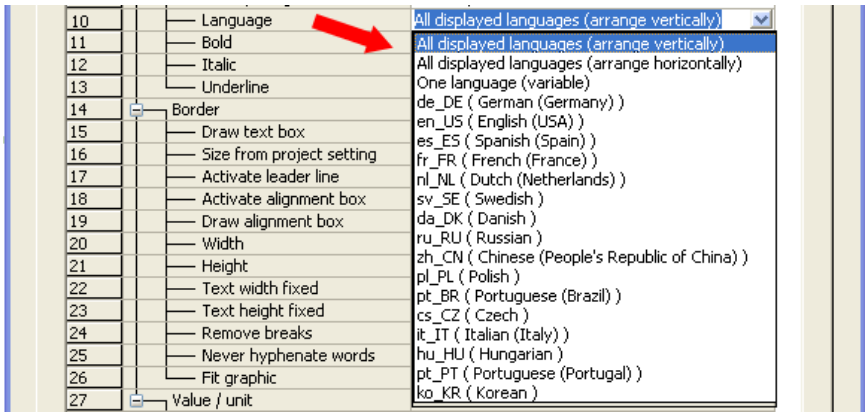


Fig. 4.115 Language selection list

The *Language* node contains the functions for displaying translated texts in the project. The **ONE LANGUAGE (VARIABLE)** selection entry depends on the settings in **UTILITIES/TRANSLATION/SETTINGS**. These settings (**SETTINGS/TRANSLATION [PROJECT NAME]** dialog) are also where the general project languages and the display languages in the project are defined.

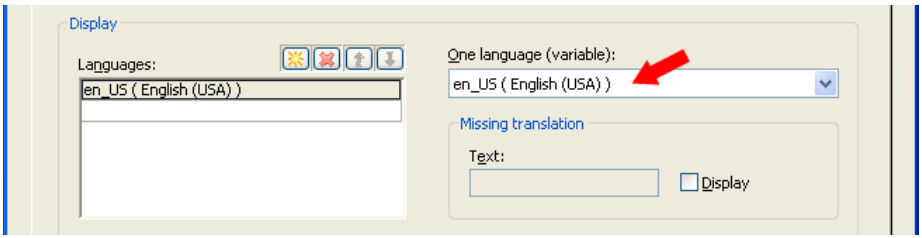


Fig. 4.116 Some translation settings

From the existing *project languages*, you can select the *displayed languages*. Depending on the existing project languages, you can then select in the *One language (variable)* selection field the desired *Displayed language*.

4.7.4.2.2 Format tab – Border node – Activate alignment box

In addition to the *Format* and *Language* nodes, there is also a *Border* node on the *Format* tab. This node contains properties, for example, to always limit the *Width* or *Height* of a text. Other options include removing breaks from texts or defining that words (in the case of automatic breaks due to the width/height limit) must not be split.

These functions are very useful, especially for translations. This prevents translations from overlapping each other since the width of the text expansion can be limited.

Fig. 4.117 shows a typical example. In this case, it does not matter whether it is pure text or device text. The procedure is always the same.

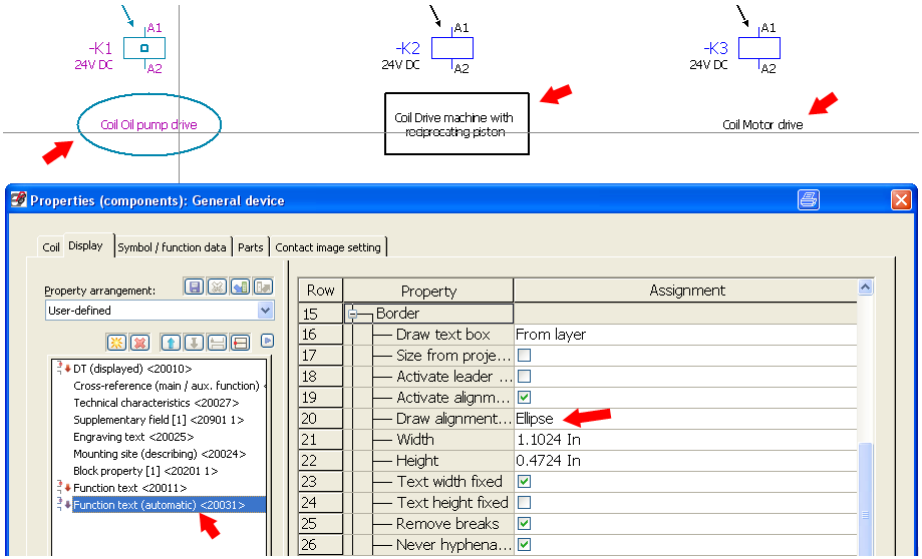


Fig. 4.117 Examples of an activated alignment box

TIP: To better read the often large number of properties and their values in dialogs, you can change the height of lines. You click the table and, while keeping the **CTRL** key pressed, turn the **SCROLL WHEEL** of the mouse. Depending on the direction, the lines will increase or decrease in height, improving legibility.

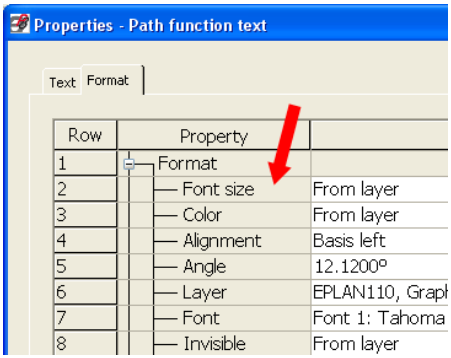


Fig. 4.118 Large row height

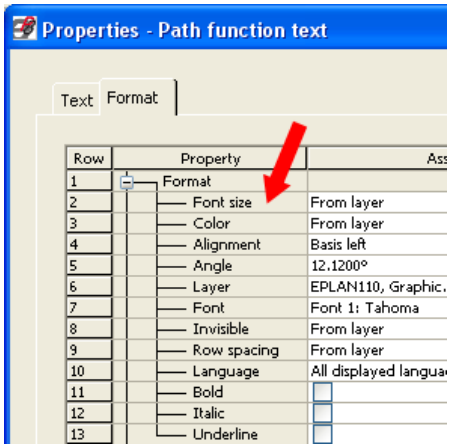


Fig. 4.119 Small row height

EPLAN remembers the settings, but handles all dialogs differently. This means that, if you wanted to, you could set different line heights everywhere.

4.8 Components (symbols)

Components, also called symbols, are among the most important elements in EPLAN and, naturally, of any schematic. Logical connections and reports can be generated based on components and their entered properties or assigned function definitions.

4.8.1 Insert components (symbols)

Symbols can be inserted in EPLAN in different ways. The usual way is via the **INSERT/SYMBOL** menu or the **INS** key. EPLAN then opens the **SYMBOL SELECTION** dialog. In this dialog, you can select the appropriate symbol. You can either use direct entry (*List view*), in which case you must have a certain amount of knowledge of identifiers, and enter the symbol names directly into the **Direct entry** field, or you can select the symbol selected from the list using the mouse.

Symbol selection(list) dialog

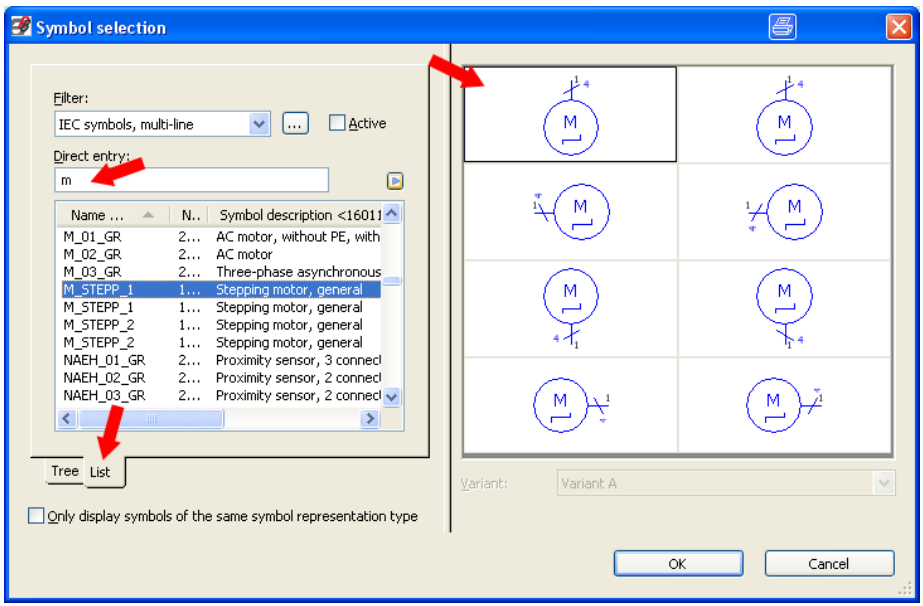


Fig. 4.120 Symbol selection via list view

Next to the symbol list is the *Symbol preview*. EPLAN (generally) provides symbols with eight variants. The normal four variants are Variant A = 0°, variant B = 90°, variant C = 180° and variant D = 270°. The remaining four variants E to H are 180° mirrored variants of the first four variants.

As with many other dialogs, it is possible to set a filter in this dialog via the ... selection button. You can use the filters to display only *Multi-line symbols* or only a special *Symbol library*.

For example, the *Only symbols of the same symbol representation type* parameter can be used to display only multi-line symbols from all displayed symbol libraries.

In addition to the *List view*, the **SYMBOL SELECTION** dialog also lets you select symbols from a *tree view*. In contrast to the list view, symbols here are listed according to a structure. Depending on the application, a symbol can be found more quickly when you search, for example, for the *motor overload switch* symbol using plain text.

Symbol selection (tree) dialog

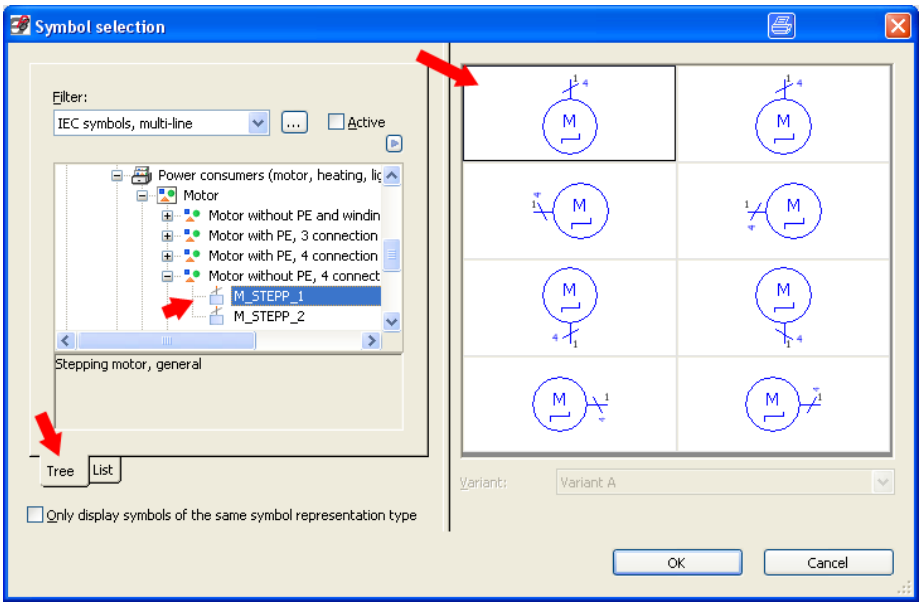


Fig. 4.121 Symbol selection via tree view

Once you have selected a component (symbol), it can be applied to the schematic with **ENTER** or the **OK** button.

4.8.2 Properties (components) dialog – [device] tab

The symbol that has been applied "hangs" on the cursor and can be placed in the schematic.

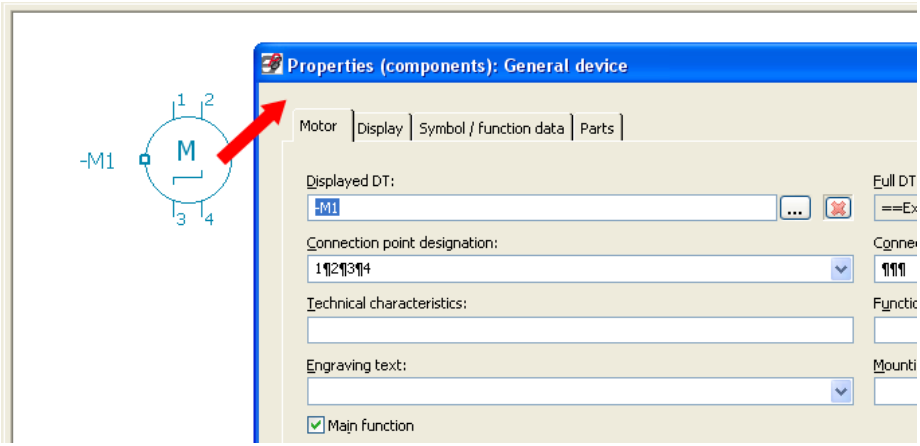


Fig. 4.122 Properties (components) dialog

When you place the symbol, EPLAN opens the **PROPERTIES (COMPONENTS) [DEVICE TYPE]** dialog. The dialog consists of the different tabs *[Device type]*, *Display*, *Symbol/Function data* and *Parts* (in case of a main function).

Here is where you find fields for the **Displayed DT** (can be edited), the **Full DT** field (can be edited indirectly via the ... selection button and the follow-up **Full DT** dialog where the DT is split into its elements, which can be edited individually), the **Connection point designation** (selectable from the selection list), **Connection point description** (selectable from the selection list), **Technical characteristics**, **Function text** (of the symbol!) fields, the **Engraving text field** (can be edited and selected from a selection list if there are already other entries in the project for this) and the **Mounting site (describing)** field, which can also be selected from a selection list.

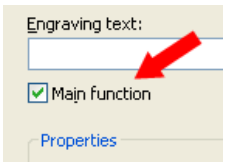


Fig. 4.123
Main function

Aside from these items, there is also an important check box: *Main function*. This box lets you decide whether this symbol is to be a main function (can carry a part), or an auxiliary function (does not contain parts data).

The lower half of the **PROPERTIES (COMPONENTS) [DEVICE TYPE]** dialog consists of the **Property name** and **Value** columns (of the properties). The button can be used to add new properties.

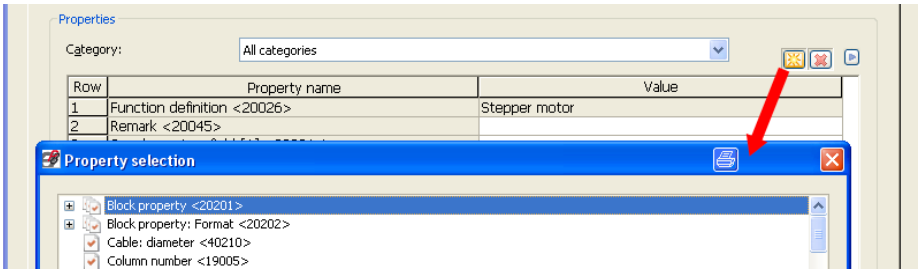


Fig. 4.124
Add new properties

When you click the button, EPLAN opens the **PROPERTY SELECTION** dialog. Here, you can select a new property and then click **OK** to apply it in the **PROPERTIES (COMPONENTS)** dialog. All properties that are not gray in the **Value** column of this dialog can be changed.

4.8.3 The Display tab

The *Display* tab allows you to add new symbol properties to the existing *for display* default symbol properties at the symbol, or delete existing properties.

The *Display* tab also contains all elements for changing the display of the properties. On this tab, you can change the format, font size or font for each property, including the connection points of devices. In this case, you would open the *Connection points* tab in the property arrangement.

The toolbar can be used to insert new properties, move the sequence of properties, or dock and undock them.

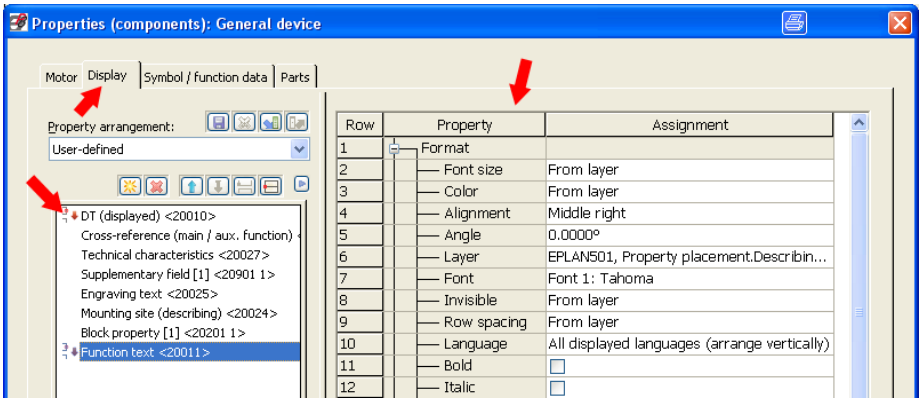


Fig. 4.125 Display tab and its options

This section explains *docking and undocking of properties* in more detail. *Docked* properties are those that do not have a symbol in front of the property name. These properties are assigned to the next higher property that has a symbol.

In Fig. 4.126, all properties framed in red would make up a "block". If you move one of the properties out of this "block", all the other properties in this "block" will be moved as well.

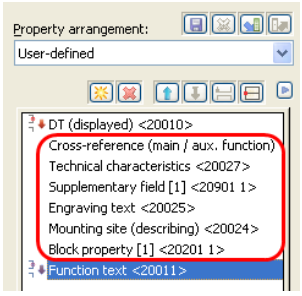


Fig. 4.126 Related properties

If you undock a property, it will be removed from the "block" and is now free. Now it is independent of the other properties and can be placed freely.

To undock a property, you select it and use the button to undock it. Then, using the direction buttons, you move it to the end of the property arrangement. If you do not do this, the undocked property and all other properties that follow it in the display will form another "block".

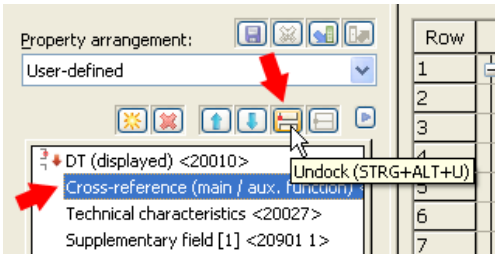


Fig. 4.127
Undocking a property

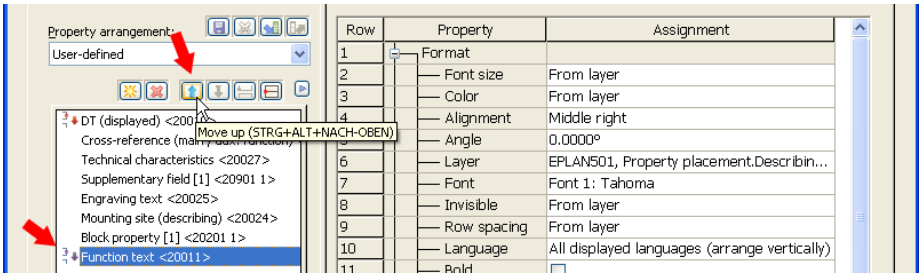


Fig. 4.128 Undocked and moved property

The *Display* tab also contains all the functions you need to format or position text. From placing a border around text to setting a different font size for a specific property or defining/adjusting a position (X/Y).

The procedure is the same as with texts and will not be discussed further here. The control of the language display will also not be discussed again.

4.8.3.1 Activate leader line property

There are some differences from formatting text when it comes to certain properties. One example is the *Activate leader line* setting.

The *Activate leader line* setting allows you to activate a leader line for each property displayed on a device. This way, for example, you can undock or move properties, while the leader line ensures a type of "connection" with the actual device.

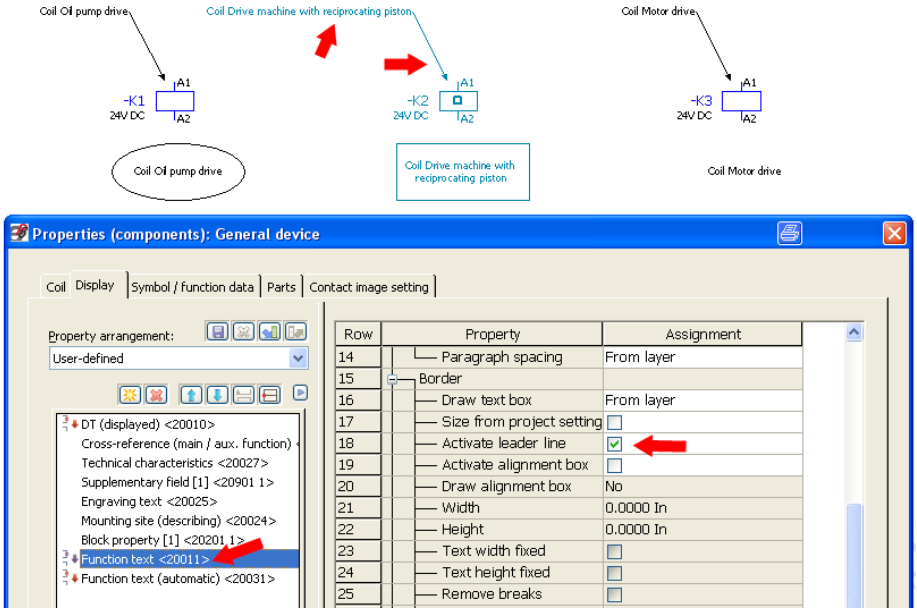


Fig. 4.129 Example of activated leader line

4.8.4 The Symbol/function data tab

The *Symbol/Function data* tab contains important information for assigning **logic** to the graphical symbol.

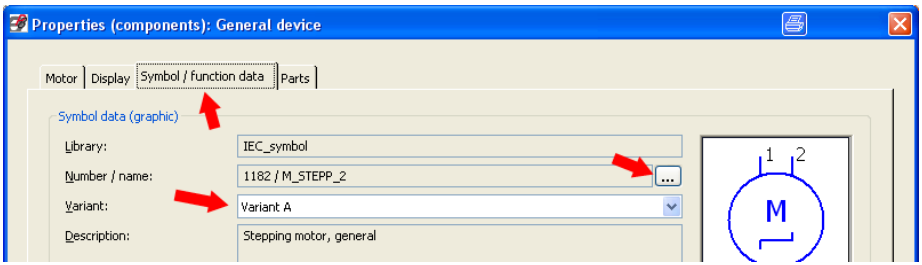


Fig. 4.130 Graphical information for the symbol

The upper area contains the graphical symbol data. In the **Number/Name** field, the ... button can be used to exchange the existing symbol for a different one. A different angle variant of the symbol can be selected in the **Variant** field. You simply open the selection list and select the desired variant. All other fields (**Library**, etc.) are derived from the graphical symbol data and cannot be edited.

The lower area of this tab contains the logical symbol information, such as which *Function definition* is attached to the symbol, the *Representation type*, or whether it is a **MAIN FUNCTION**, etc.

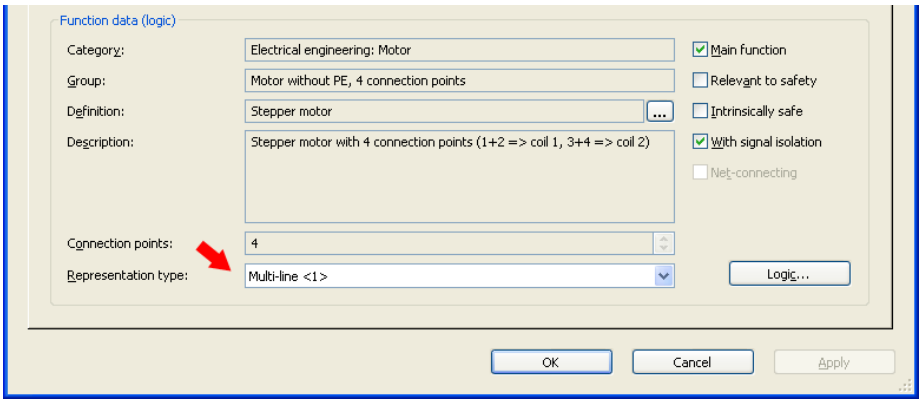


Fig. 4.131 Logical symbol functions

In the **Definition** field, the ... button can be used to assign a different function definition to the symbol if necessary. All other fields are then derived automatically from the applied function definition.

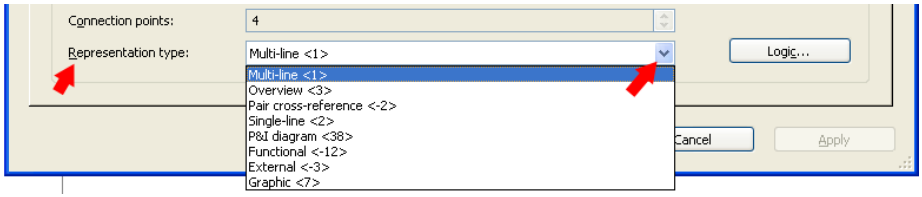


Fig. 4.132 Representation type selection

The **Representation type** here can be modified from a selection list. This may be necessary, for example, if you wish to create or represent an overview on a multi-line schematic page.

Aside from the previous graphical and functional information, there are additional settings options under the **LOGIC** button. In a targeted manner, you can change or adjust logical information such as **Connection point type** or the **Number of targets**, **Potential type** and much more for the individual connection point of a device.

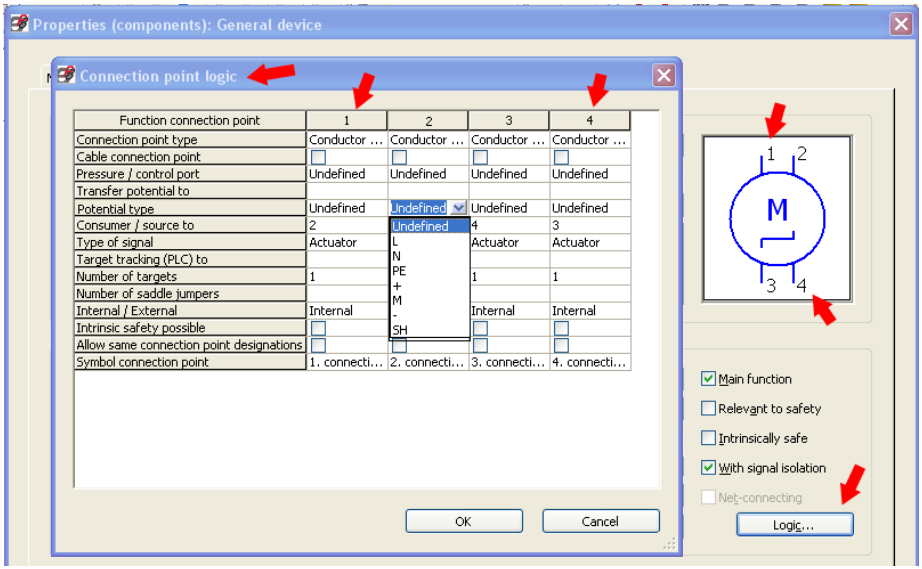


Fig. 4.133 Logical functions of individual connection points

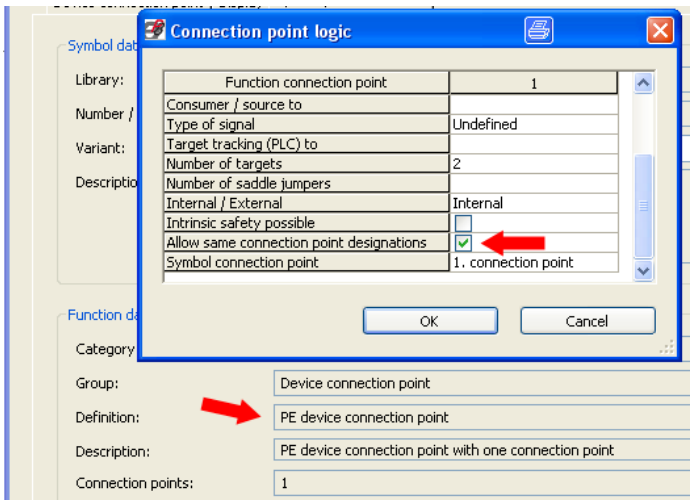


Fig. 4.134 Allow same designation

A typical example of changing the *Connection point logic* is the enabling of the *Allow same connection point designation* setting when using device connection points (here PE). This makes it possible, like with terminals, to use the same designation several times (PE rail), and the check run will not find any duplicate connection point designations.

4.8.5 The Parts tab

On the *Parts* tab, you can assign one or more parts to a symbol. Click the **Part number** field and then use the ... that appears to switch to parts management.

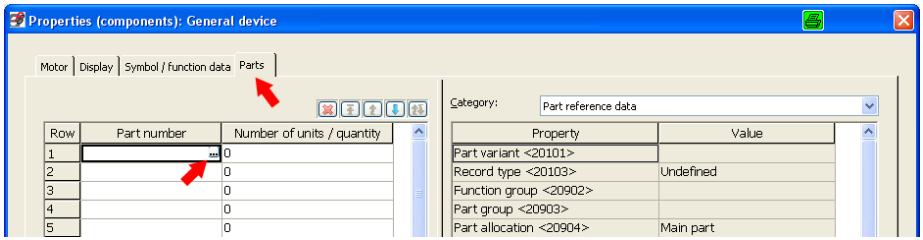


Fig. 4.135 Simple part selection

In **parts management**, you select the necessary part and click **OK** to apply it to the symbol. Parts management automatically closes after the part is applied.

In addition to simple part selection, this dialog also has a **DEVICE SELECTION** button. In contrast to simple part selection, this will offer only devices for selection that fit the existing functions of the symbol in the project. This ensures that EPLAN selects, for example, a coil for a coil rather than a motor overload switch. **DEVICE SELECTION** is the preferred method.

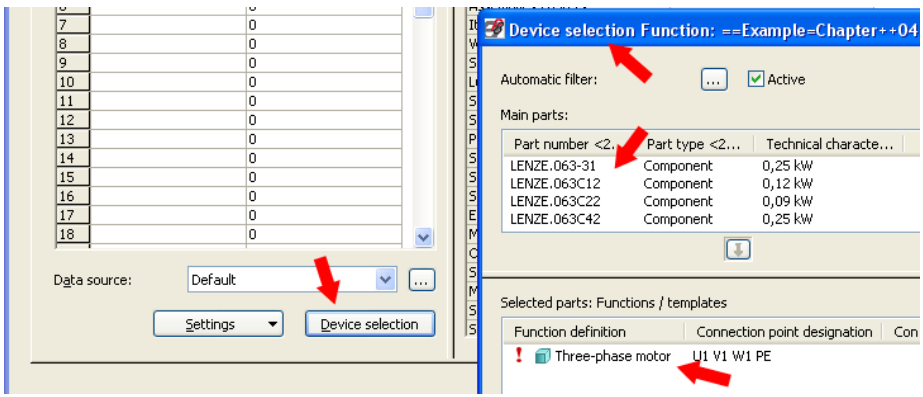


Fig. 4.136 Device selection

4.9 Cross-references

Cross-references are one of the most important elements in EPLAN. When automatically generated, they create visual and logical connections, e.g. between *main and auxiliary functions*.

NOTE: Inserting and filling out of the corresponding symbols for the cross-reference display examples will not be explained at this point. More details are available in the following sections.

4.9.1 Contact image on component

For example, a motor overload switch or a pushbutton switch can have auxiliary contacts. The auxiliary contacts physically belong to the **motor overload switch** or **pushbutton switch** symbol and, for the sake of clarity, should also be displayed at the switch.

In the following, a motor overload switch is inserted into the schematic. The motor overload switch initially has no auxiliary contact at the symbol. As such, this is not a problem, because EPLAN creates a report on the auxiliary contacts used in the schematic on the basis of the settings on the *Display* tab in the *Contact image* selection field (the selection here is set to **ON COMPONENT**). This way, the contacts used and their cross-references are displayed automatically.

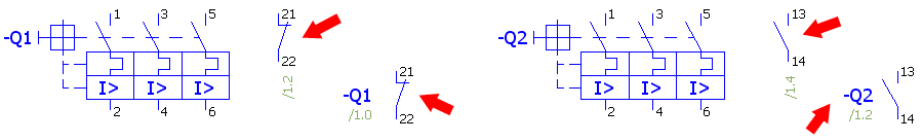


Fig. 4.137 Automatic cross-referencing

This does not matter provided that both or all auxiliary contacts that have such a motor overload switch are also used in the schematic.

Not all auxiliary contacts are always used, and EPLAN would display the motor overload switch only with the auxiliary contacts used (as graphical representation at the motor overload switch). The other auxiliary contacts physically exist but are “suppressed” with this setting. To display those contacts anyway, a device with the appropriate *Function definition* must be assigned to the **Motor overload switch** symbol.

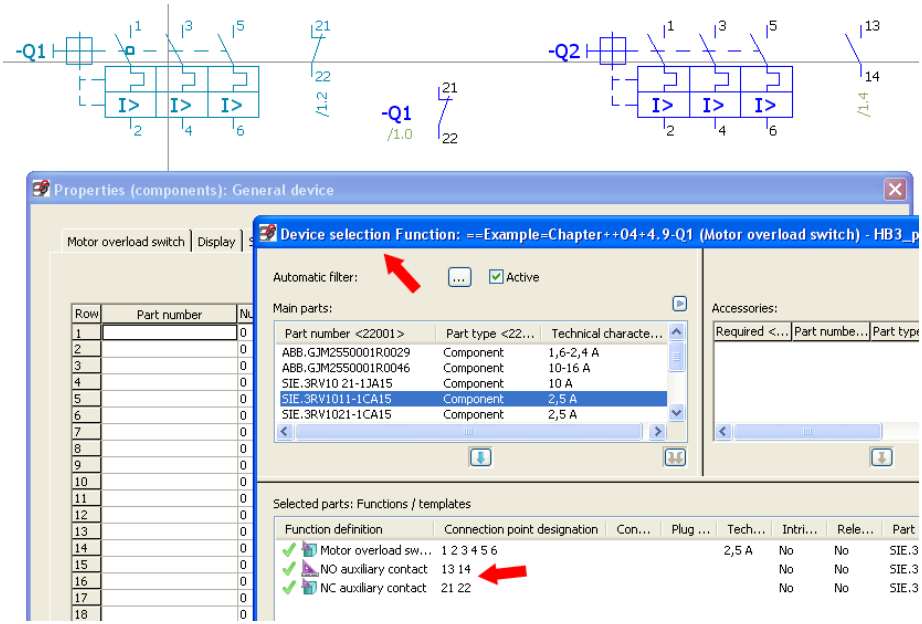


Fig. 4.138 New device selection

This is best done using the Device selection. You open the properties of the symbol, switch to the *Parts* tab and click the **DEVICE SELECTION** button.

TIP: The **SETTINGS** button provides access to the settings defining how EPLAN should behave during device selection: whether existing function data is used or not; whether any other criteria of devices or functions are be taken into account, etc. All these have to be taken into consideration during device selection.

This is important to know because in certain situations this can result in devices not being displayed in the device selection. In situations such as “incorrect” or inappropriate settings in the device selection, or when there is no part with the appropriate functions in parts management.

Back to device selection: EPLAN recognizes that the motor overload switch consists of the motor overload switch itself and a placed auxiliary contact (considered a function). Now all parts that fit the selected motor overload switch (including its used functions), i. e. the parts with their function definitions, are offered for device selection.

In the upper area of the **DEVICE SELECTION FUNCTION** dialog, you can now select the right part and click to apply it. EPLAN “moves” the part across the existing functions of the motor overload switch and adds the functions not yet placed to the function template.

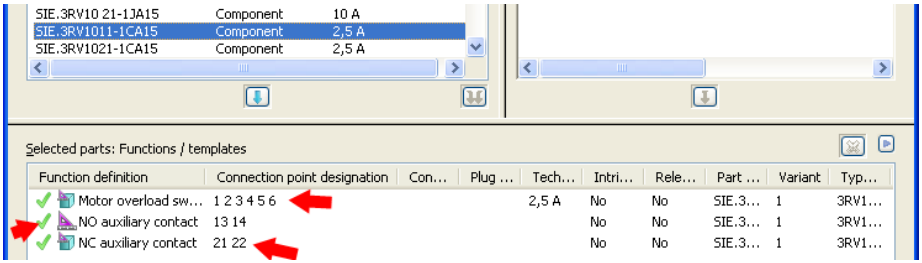


Fig. 4.139 Selected part with its function template

The **DEVICE SELECTION FUNCTION** dialog can now be closed with the **OK** button. EPLAN transfers the part into the *Parts* tab. If you now leave the **SYMBOL PROPERTIES** dialog by clicking the **OK** button, EPLAN will use the stored function definitions to graphically recreate the entire contact image for the auxiliary contacts. It makes no difference whether the auxiliary contacts have been used in the schematic or not.

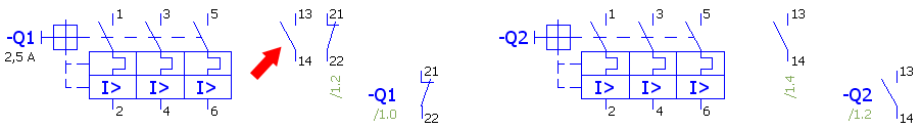


Fig. 4.140 Complete contact image

But this representation can be achieved only if the display of the contact image has been set accordingly in the symbol properties of the *Display* tab.

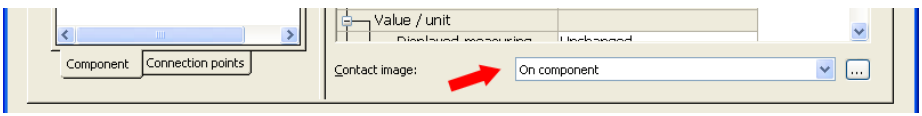


Fig. 4.141 Contact image setting

This display of the placed contacts is set by EPLAN automatically at first. But if you wish to, or must, deviate from this automatic setting, you can modify the item manually. You simply click the **MORE** button. In the **CONTACT IMAGE POSITION** dialog that follows you can manually edit the position for the Y and X values.

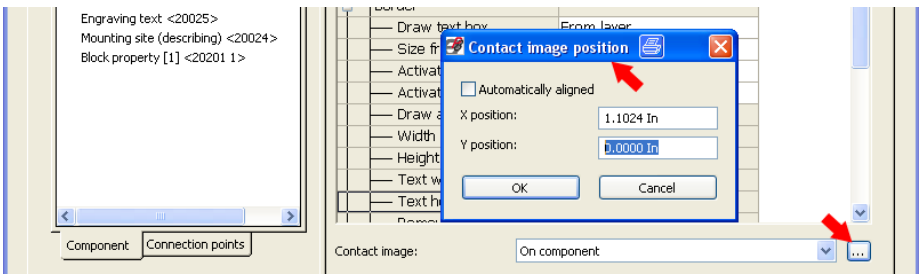


Fig. 4.142 Manual editing of the contact image position

4.9.2 Contact image in path

In contrast to a contact image at the component, which is usually displayed to the right of the symbol, for (e. g.) contactors the contact image is generated by EPLAN below the contactor coil in the column (path).

Similar to the previous example, at first only the contacts used are automatically displayed in the contact image.

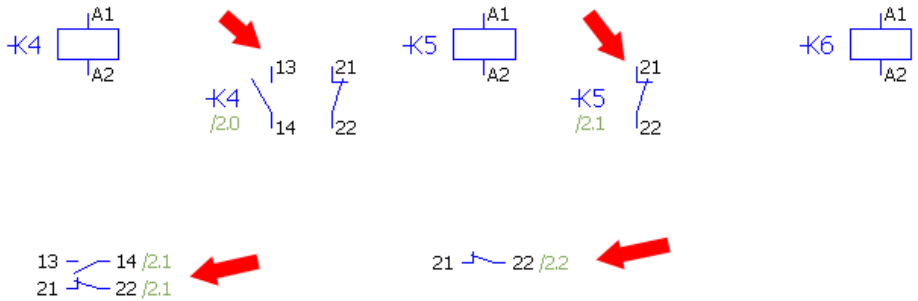


Fig. 4.143 Automatically generated contact image of the functions in use

To obtain a complete *contact image*, **DEVICE SELECTION** is performed in the same way as was done in the example of the motor overload switch. The procedure is identical to that already described in the motor overload switch example. After device selection, the full contact image is displayed. If this does not occur, then the *Contact image* entry in the **DISPLAY** tab of the *Symbol properties* dialog must be checked. It must be set to the **In path** entry.

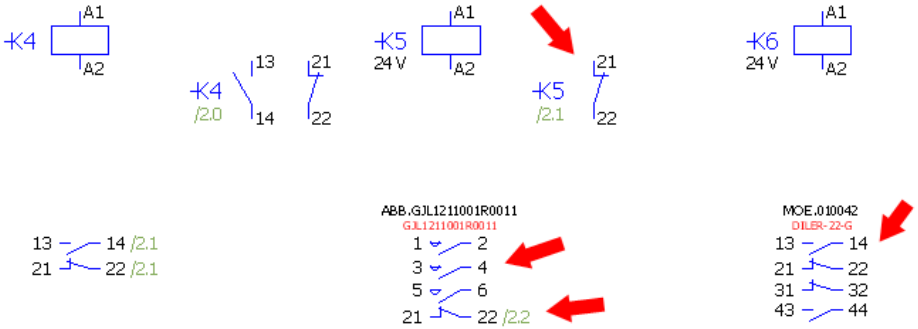


Fig. 4.144 Complete contact image following device selection

NOTE: EPLAN also lets you move contact images. Every contact image can be individually moved (it makes no difference whether the contact image type is *On component* or *In path*).

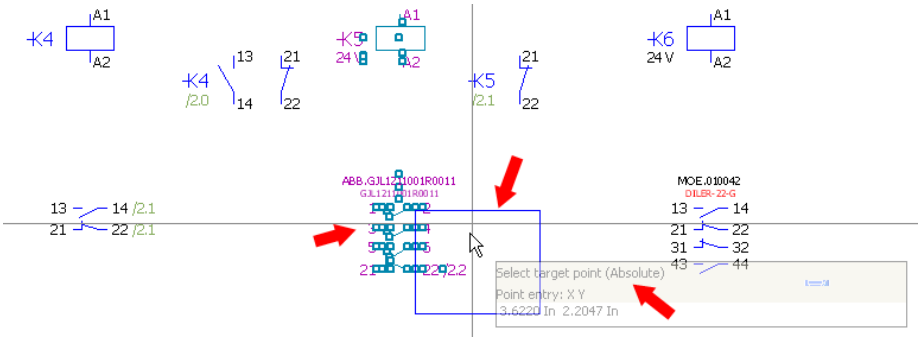


Fig. 4.145 Moving the contact image manually

You first select the symbol, and then press **CTRL + B** or use the **EDIT/TEXTS** menu to call the **MOVE PROPERTY TEXTS** function. EPLAN then displays a small blue rectangle for "grabbing" with the mouse. You then click the contact image. It will then "hang" as a blue rectangle on the cursor and can be placed as desired (while keeping left mouse button pressed). You left click to place it in the desired position.

At this point, it should be mentioned that you can define the positions of the contact images for the motor overload switch and/or contactor coils in the respective plot frame (or only for this page, in which case, in the Page properties), for example, as **Contact image margin (on component) <12059>** and **Contact image margin (in path) <12060>** plot frame properties.

3	Contact image margin (in path) <12060>	0.0000 In
4	Contact image margin (on component) <12059>	0.0000 In
5	Contact image offset <12061>	0.0000 In

Fig. 4.146 Properties of contact image offsets

Regardless of these default values, all contact images can, of course, be individually and manually set with the desired values and placed at the respective devices in the schematic.

4.9.3 Special feature: Pair cross-reference

Aside from the previous examples, the functionalities of EPLAN have been used on the basis of the device selection and its functions. But situations can arise where there are no parts with correct function templates. It is, after all, only because of the function templates that EPLAN knows, for example, that an illuminated pushbutton consists of several functions.

Using an illuminated pushbutton as an example, we will now see how contact images can be referenced to the interconnected functions even without parts. This is done using the **Pair cross-reference** representation type (in the **PROPERTIES (COMPONENTS)** dialog on the *Symbol/Function data* tab in the **Representation type** selection list).

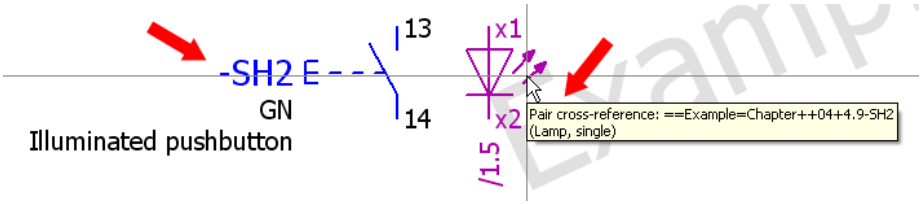


Fig. 4.147 Insert and designate symbols

A pushbutton is inserted. It is assigned a DT and is designated as a main function. In addition to the pushbutton, a lamp is also inserted. It is **not assigned a DT**, nor is it **designated as a main function**. The representation type of the lamp is set to **Pair cross-reference**.

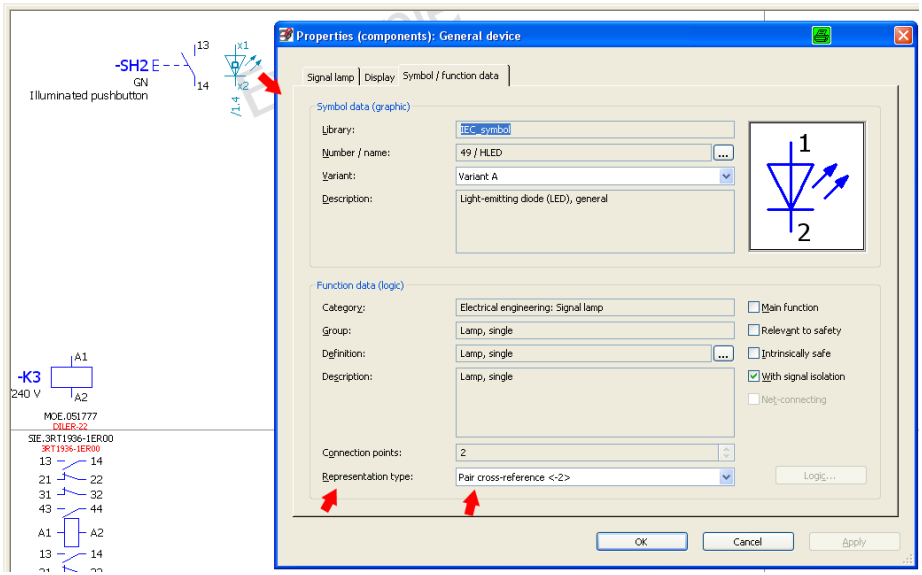


Fig. 4.148 Setting the pair cross-reference representation type

Now, at the auxiliary function (of the lamp), the **Cross-reference (main/aux. function) <20300>** property is set to the optimal position of the display of the cross-reference. The following settings are made: the **angle to 90°**, the **alignment to center right**, the **position** of the **X coordinate to 0.00 mm** and of the **Y coordinate to −6.00 mm**.

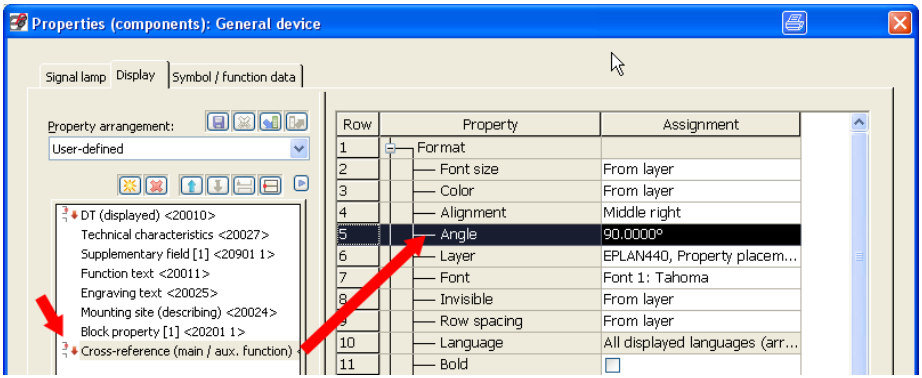


Fig. 4.149 Setting appropriate values for the cross-reference

If all values have been set accordingly, the cross-reference will be rotated subsequently by 90° directly below the lamp.

Now the lamp can be placed, either from the **device navigator** via the **PLACE/MULTI-LINE** popup menu or via the symbol selection.

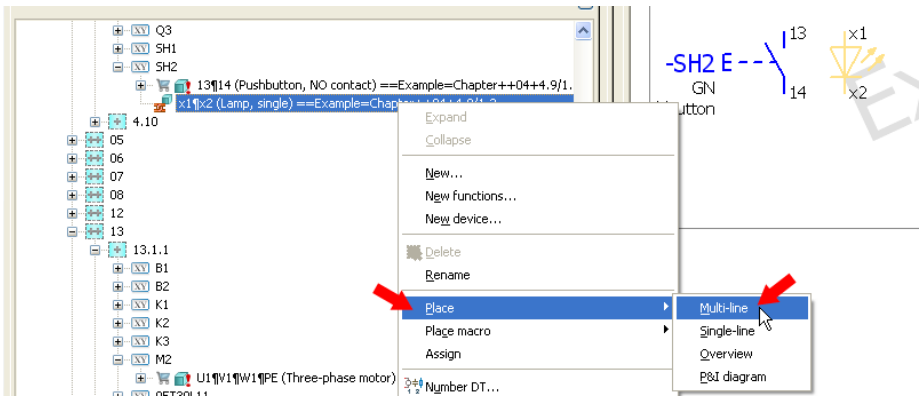


Fig. 4.150 Place multi-line function from the device navigator

After this function has been placed, EPLAN immediately displays the cross-reference on both pages: on the complete device illuminated pushbutton and on the device of the lamp itself, which is displayed as distributed.

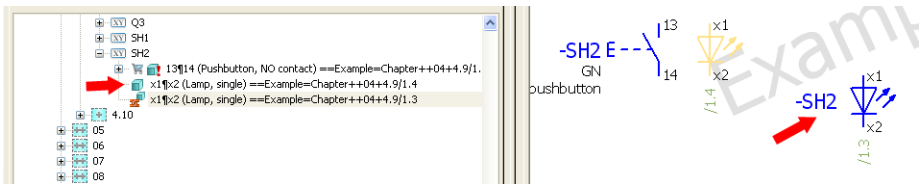


Fig. 4.151 Cross-reference with pair cross-reference representation type

Finally, the default property arrangement setting on the display of the distributed symbol is set. This puts the cross-reference in the “right” position.

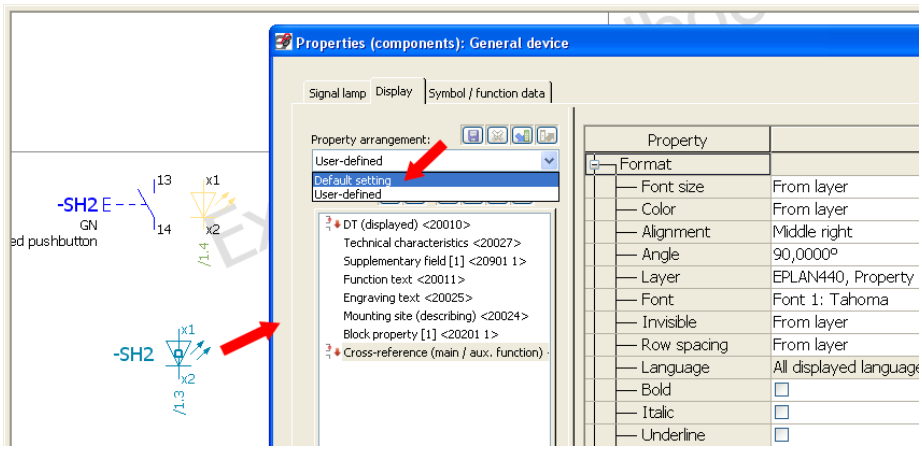


Fig. 4.152 Default property arrangement setting

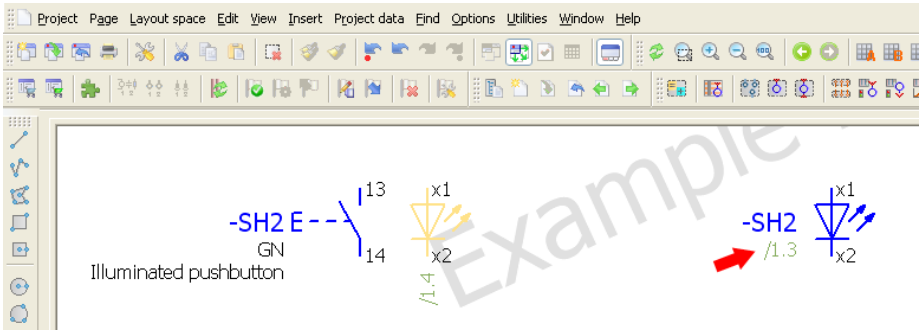


Fig. 4.153 Result

NOTE: The placing of functions from the device navigator is always preferable to manual placing (insert symbol, fill out DT, etc.), because EPLAN implements all key settings automatically, which means that everything fits all the time.

4.9.4 Device selection settings

There are a number of settings for the device selection. Depending on how they are set or adjusted, they can influence **DEVICE SELECTION** and thus the selection of parts.

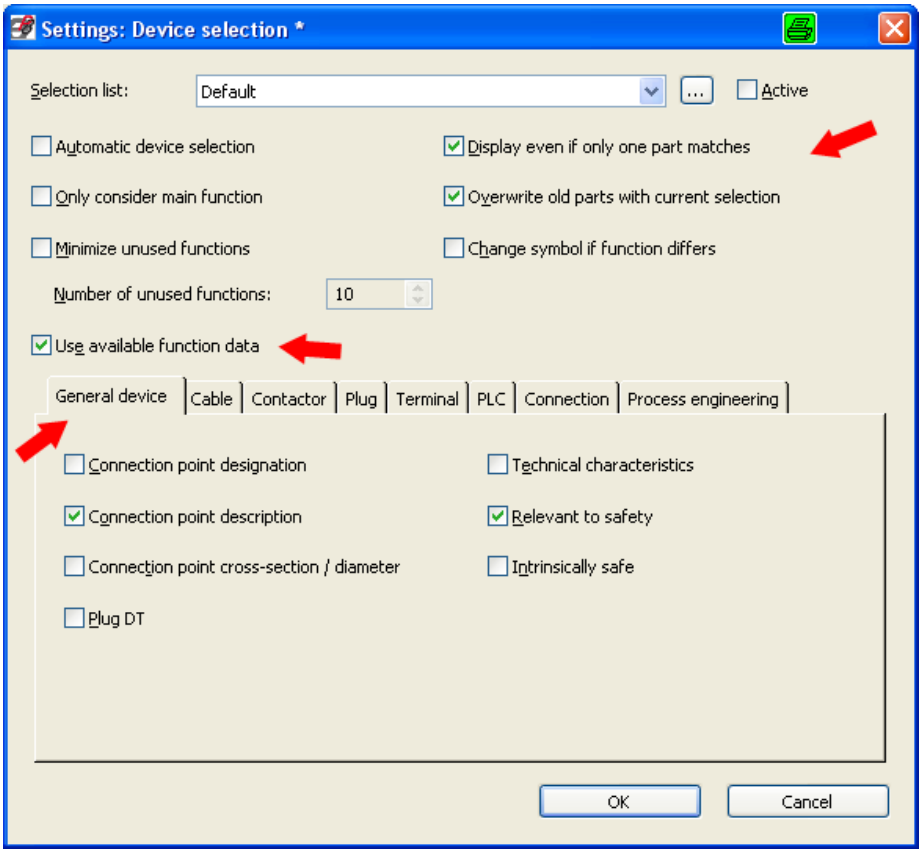


Fig. 4.154 Device selection settings

It helps if the *Use available function data* setting and the desired properties have been enabled. This way, only parts will be offered for selection that match the identifying function data. Identifying function data is the data that is located on the tabs, e. g. *General devices*, *Cable*, *Contactor*, etc.

If the *Use available function data* setting is disabled, you can select devices freely. This means that the function data available in the project (at the function) does not have to match the function data in the parts selection.

4.10 Macros

In addition to designing using devices (symbols), EPLAN also lets you work with partial circuits, so-called macros.

4.10.1 Types of macros

EPLAN has different types of macros. These can be window, page and symbol macros. Special macros are macros with value sets. These can be window and page macros. When using or creating macros (exception: macros with value sets), it generally makes no difference whether it is a window macro on a multi-line page or a macro on a graphical page.

4.10.1.1 Window macros

Window macros are (can be) the smallest partial circuits in EPLAN. Window macros can include single or multiple devices and objects within an area, or several items within a page.

To create a window macro, you first use the mouse to select the associated devices on the respective page. These may be individual devices or unrelated parts circuits on a project page. These are "gathered" by selecting the first device, and then selecting the other devices that are to belong to the window macro while holding down the **CTRL** key.

You then press **CTRL + F5** or use the **EDIT/CREATE WINDOW MACRO** menu item to start the function. EPLAN opens the **SAVE AS** dialog. The first thing you should do is enter a descriptive file name for the window macro. The macro directory is usually the company-specific macro directory, but this can be changed if desired.

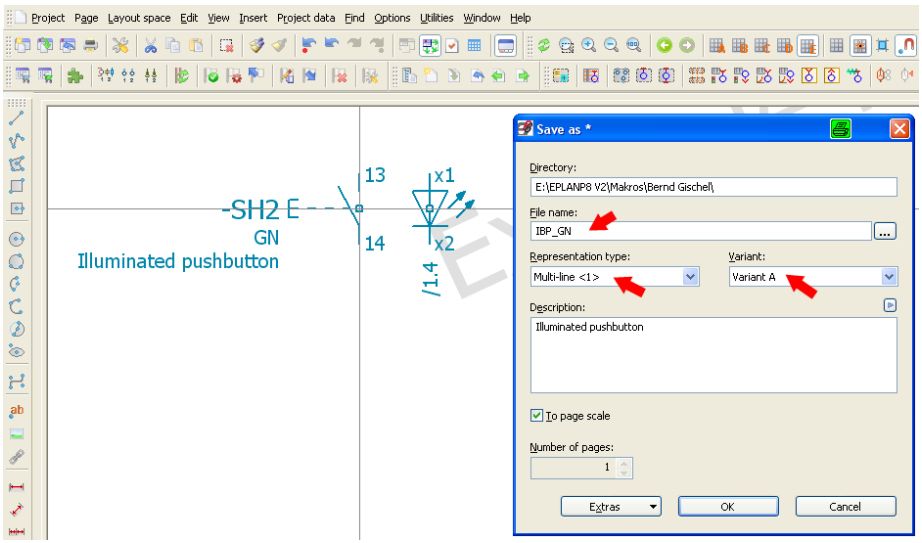


Fig. 4.155
Saving the macro

Window macros are inserted into a page with the **M** key or via the **INSERT/WINDOW MACRO** menu item. The **SELECT MACRO** dialog opens. Here you can select the desired *variant* or *representation type* from the default directory (or a different macro directory by selecting a different directory in the *Search in:* selection field). When you click **OPEN**, the macro will hang on the cursor and can be placed anywhere on the page.

To summarize, window macros are only possible on *one* (the same) page but with different variations, such as the gathering of objects that do not all have to be within a single window. A window macro can also be the contents of the entire page (all objects on the page). Window macros have the file extension **.ema*.

4.10.1.1.1 Define handle setting

The **SAVE AS** dialog and the **EXTRAS** button contain other settings related to the macro.

The **MOVE HANDLE** menu item lets you save a (separately created) handle (base point) with the macro. A handle means that when the macro is inserted, it "hangs" on the cursor crosshairs using this handle as the base point. When you click the **MOVE HANDLE** menu item EPLAN temporarily closes the **SAVE AS** dialog, and you can define the handle by clicking with the left mouse button.

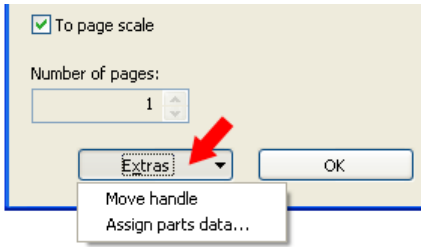


Fig. 4.156 Extras settings

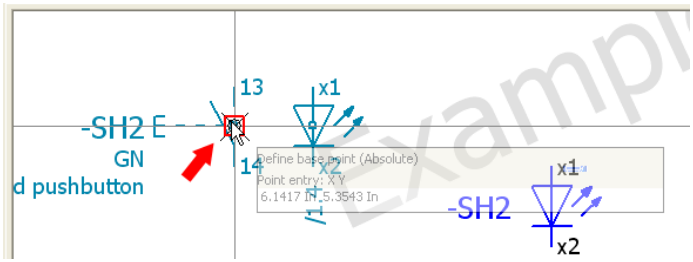


Fig. 4.157 Defining a new handle

The cursor changes when doing this, as can be seen in Fig. 4.157. It is a good idea to enable the **OBJECT SNAP** setting here. Once the handle has been defined, EPLAN returns to the **SAVE AS** dialog, and the macro can be saved.

4.10.1.1.2 Define representation type setting

The **Representation type** selection allows you to define the representation type for the macro.

The individual representation types are usually defined via the devices at the symbol (**SYMBOL/FUNCTION DATA** tab). However, EPLAN allows a different representation type to be defined for each individual macro. This way, you can save many representations under one macro name.

It is possible to have a macro for the **multi-line** representation type, one for the **part placement** and one for the **single-line display**. When selecting a macro, EPLAN chooses

the right representation type based on the page type defined. Of course, this must be present in the macro.

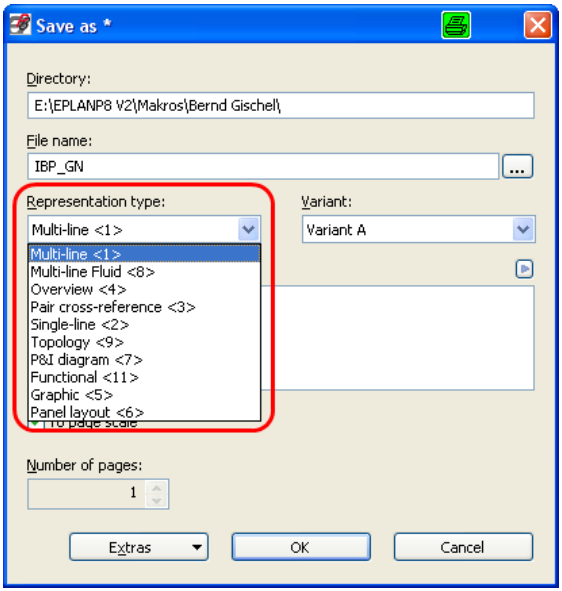


Fig. 4.158
Representation types

4.10.1.1.3 Define variant setting

The **Variant** selection box provides another way of working more effectively.

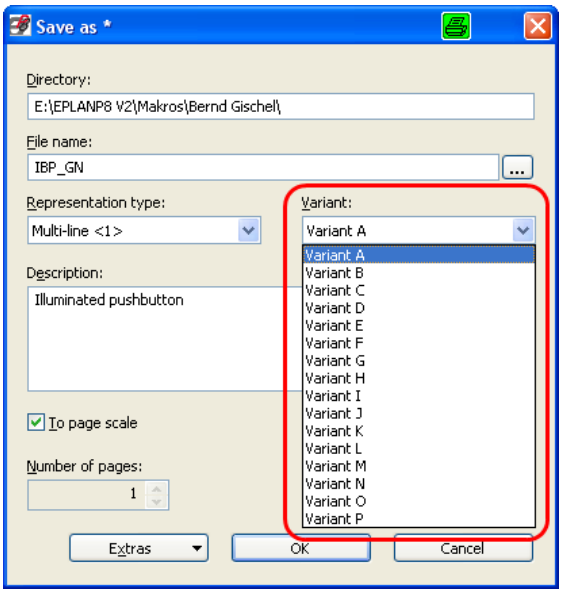


Fig. 4.159 Variants

This box allows different variants (so far, sixteen variants A to P) to be stored within a macro. This just means that the same macro name (file name of the macro) can have different content. Depending on the variant used and in connection with the representation types, you can have up to 160 possible variants (macro variants) within one macro. For example, you could save different macro variants of a PLC card in one macro.

4.10.1.1.4 To page scale setting

In addition to all the other settings for a window macro, there is also the **To page scale** option. If this option is activated, then EPLAN reduces or enlarges the macro to suit the page scale when it is inserted into a page.

4.10.1.2 Page macros

Page macros are the counterpart to window macros. Page macros can comprise one or several pages within an EPLAN project. Page macros are created using the **CTRL + F10** keyboard shortcut (at least one page must be open and the user must be on this page), or using the **CREATE PAGE MACRO** popup menu item on selected pages in the **page navigator**.

The familiar **SAVE AS** dialog, as described in the 4.10.1.1 "Window macro" section, is displayed. The *file name* and *description* can be entered here. **No** handles, **no** representation types, and **no** variants are possible with page macros. However, page macros store all the information relating to a page or pages (page properties). This includes, for example, the structure identifiers or the form pages stored in the page properties.

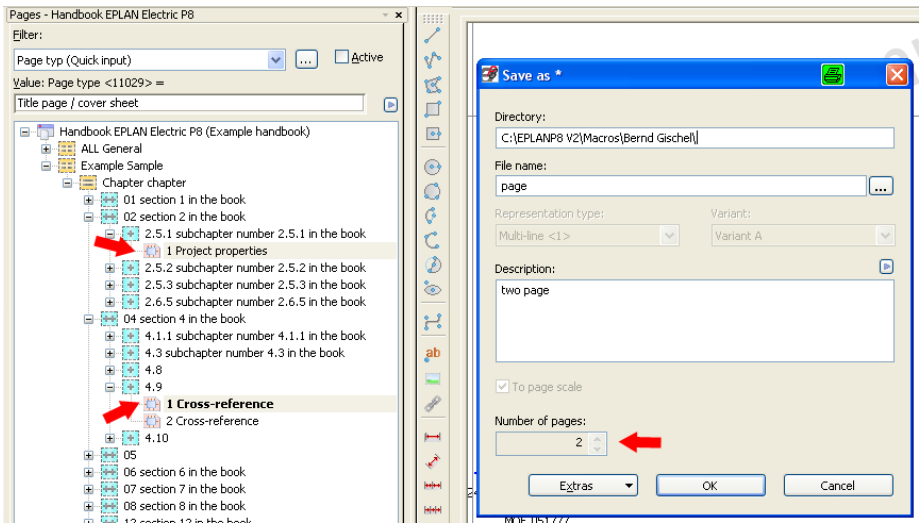


Fig. 4.160 Creating page macros

In addition to the individual page macro (directly on the page), in the **page navigator** it is possible to write multiple pages (even pages that are not related) into a page macro. You open the **page navigator**, select the desired pages and use the right-click popup menu to call the **CREATE PAGE MACRO** command.

You can use custom keyboard shortcuts for inserting page macros, e. g. **CTRL + ALT + F10**, or call up the popup menu in the **page navigator** and select the **INSERT PAGE MACRO** command. EPLAN then opens the **SELECT MACRO** dialog. Here you select the desired macro and click **OPEN** to apply it.

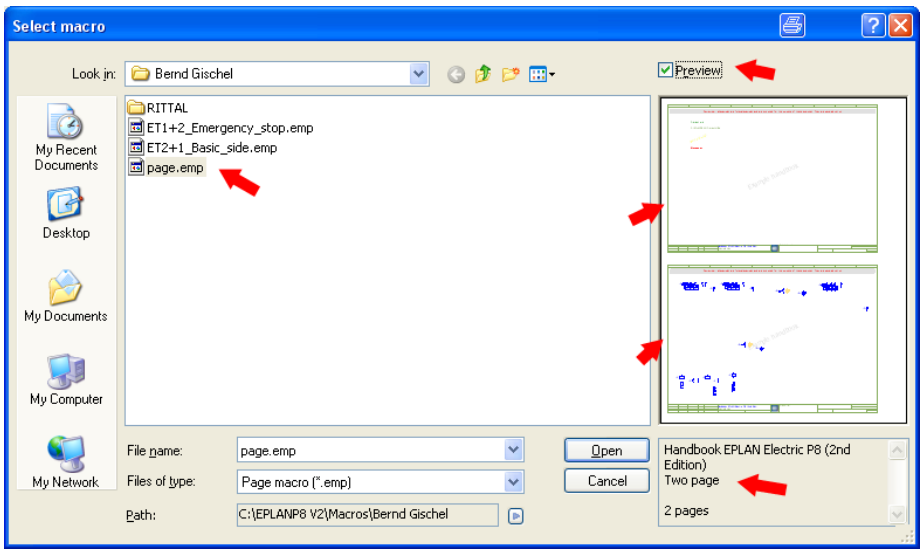


Fig. 4.161 Inserting a page macro

If the preview is switched on in this dialog, then all pages are displayed in a small preview. In addition to the preview display (which has a fixed size), the **GRAPHICAL PREVIEW** window can also be permanently displayed (**VIEW** menu). The advantage of this window is that the size can be changed, and you can see the details better.

After applying the macro, EPLAN opens the **ADAPT STRUCTURE** dialog. Here, the page(s) can be stored in the project and sorted into the existing page structure. At this point, it is, of course, possible to adjust the structure of the pages as desired.

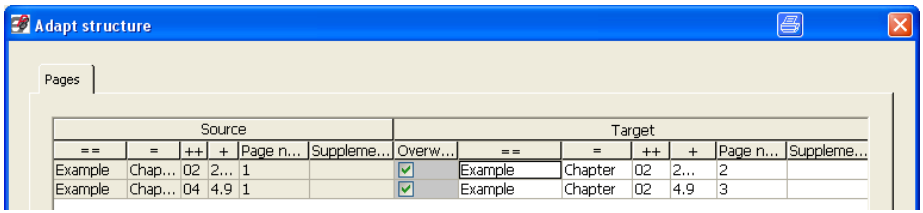


Fig. 4.162 Adapt structure

To summarize, page macros are macros containing one or more pages and their page information (page properties). Page macros have the file extension **.emp*.

4.10.1.3 Symbol macros

Symbol macros are an additional type of macro. Generally, symbol macros are completely identical to window macros in terms of handling and creation. Further explanation will therefore not be provided. Symbol macros have the file extension **.ems* and are inserted via the **INSERT/SYMBOL MACRO** menu.

4.10.2 Macros with value sets

Macros with value sets are a special function type in EPLAN Electric P8. These are usually window macros equipped with additional functionality that can make project planning much easier. In addition to the basic macro (partial circuit) and its properties, such as technical characteristics, part numbers, etc. of the various objects, this type of macro also contains additional sets of properties. These additional properties are called **Value sets**. Value sets are activated via a particular symbol – the placeholder object. This symbol is used to switch between value sets.

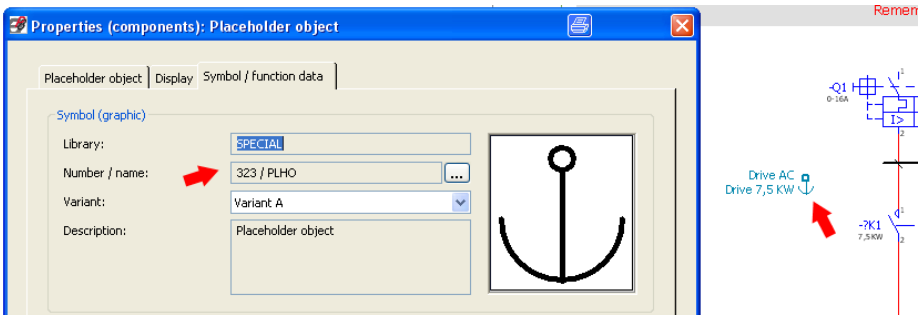


Fig. 4.163 Placeholder object symbol

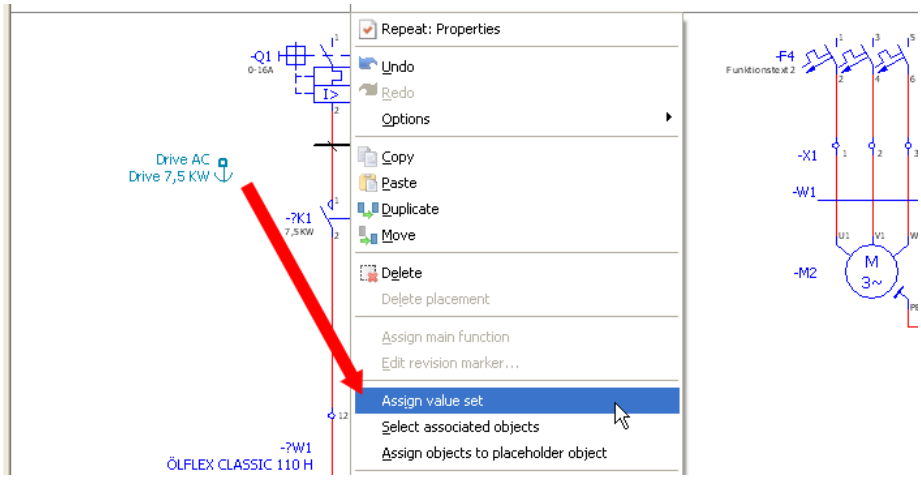


Fig. 4.164 Assigning a value set

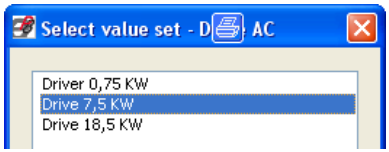


Fig. 4.165
Selecting a value set

For example, all devices of the window macro could be equipped with “Siemens parts”. A second value set could then contain “Moeller parts data” for all devices. The value set can then be used to switch between Siemens and Moeller parts data for the same macro with a mouse click.

Once these macros and the properties they contain have been developed and tested, they provide a source of error-free data.

4.10.2.1 Placeholder object

The most important object in a macro with value sets (subsequently called a value set macro here) is the *placeholder object*. A placeholder object can be inserted via the **INSERT/PLACEHOLDER OBJECT** menu, making it easy to later switch between value sets. The symbol for a placeholder is an anchor. Since the placeholder object is a symbol, it has settings and options similar to other symbols.

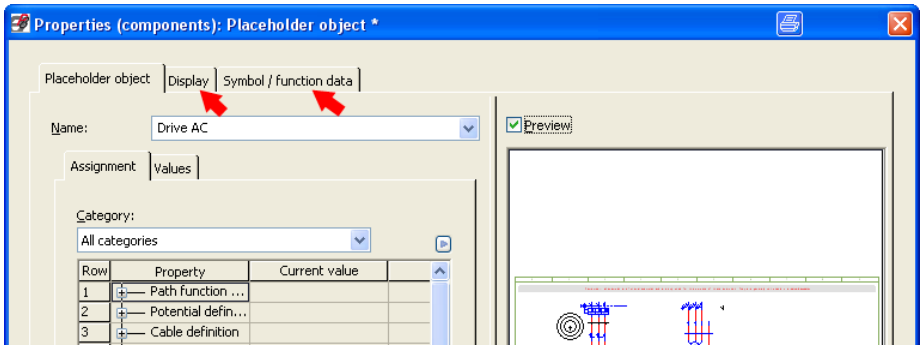


Fig. 4.166 Placeholder object symbol properties

4.10.2.2 Value set

A *value set* is a collection of variables of selected objects stored in a window macro. Value sets are managed in a type of table “behind the placeholder object” (*Placeholder object* tab) and, in addition to all the device properties (filled or empty), they contain additional information such as the actual values or the variables for the values.

4.10.2.3 Variables of a value set

Variables bring a value set to life. Without these variables, there would be no table for the actual values used for switching the value set. Every property of a device can be provided with any desired variable name. Variable names are always surrounded by the “less than **(<)**” and “greater than **(>)**” characters on the keyboard. For example: **<Variable name>**.

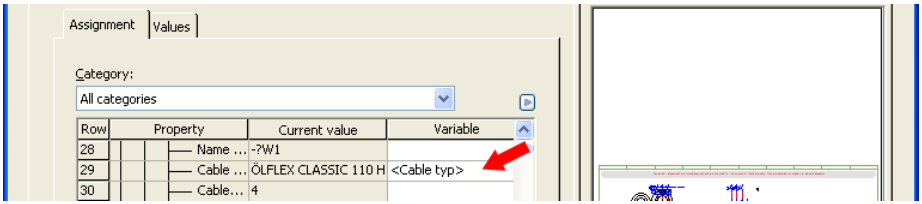


Fig. 4.167 Example of variables

Apart from one exception, all characters can be used. Square brackets are the exception. These have a special purpose within a variable. They are used to define a line break.

For example, a variable **<Variable name [12]>** is split at the twelfth position (line break) and the words are simply split at this twelfth position. If the additional option 1 is inserted into the square brackets: **<Variable name [12,1]>**, then a split also occurs at the twelfth position (i.e. a maximum of twelve characters) but an attempt is made to recognize the ends of words. In other words, the words are not "torn apart" but remain intact, and line breaks are only inserted between the words.

NOTE: The square brackets **must** be entered directly after the variable name. Spaces are allowed, but then the defined line break option does not take effect.

4.10.2.4 Create a macro with a value set

Only a few steps are needed to create a value set macro. First, you create the partial circuit with all the required or desired devices and their associated functions such as part numbers, technical characteristics, or function texts. Then you insert the **placeholder object** via the **INSERT/PLACEHOLDER OBJECT** menu. There are several ways of assigning objects to a placeholder object in EPLAN.

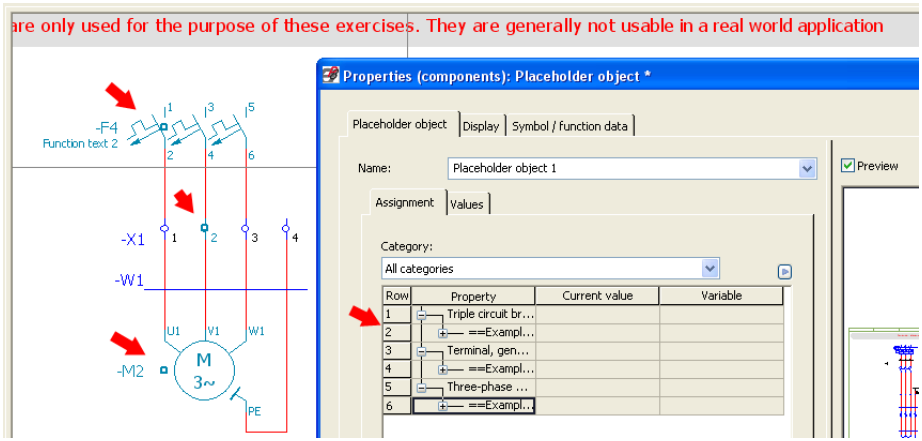


Fig. 4.168 Assigning individual objects to the placeholder object (option 1)

Option 1: You first select all objects and then use the **INSERT/PLACEHOLDER OBJECT** menu to insert the placeholder object. After you place the placeholder object, EPLAN opens the **PROPERTIES (COMPONENTS) PLACEHOLDER OBJECT** dialog, and the data can be edited. This approach is recommended, because it allows you to accurately select the objects that are to be applied to the placeholder object.

Option 2: You first insert the placeholder object in the page via the **INSERT/PLACEHOLDER OBJECT** menu. But before EPLAN places it, all objects must be selected with a window (pulled open with the mouse).

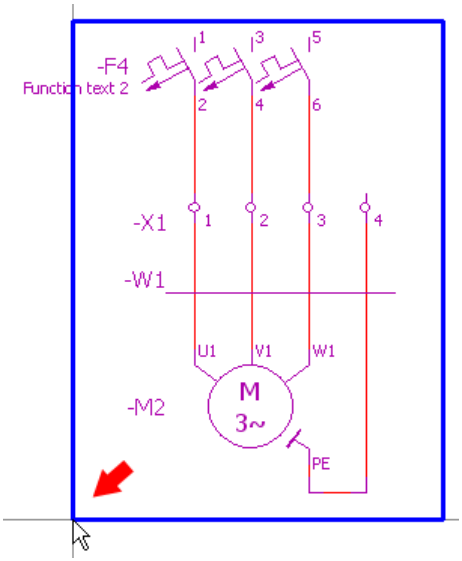


Fig. 4.169
Selecting objects with a window (option 2)

NOTE: Disadvantage of option 2: remote objects lying outside a window cannot be included in the value set.

Once you have chosen one of the methods, you can place the placeholder object. It is recommended that you place the placeholder object close to the macro. Simply use the left mouse button to move the placeholder object to the desired position and click to place it.

EPLAN immediately opens the **PROPERTIES (COMPONENTS) PLACEHOLDER OBJECT** dialog. Here you should enter a descriptive name for the placeholder object into the **Name** field.

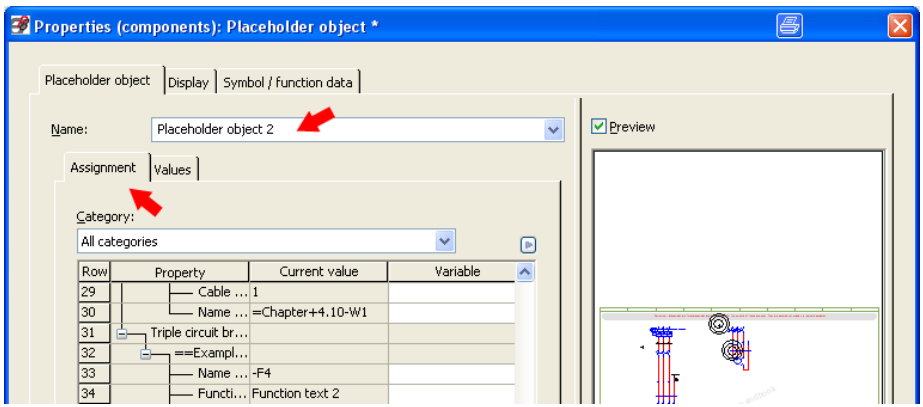


Fig. 4.170 Edit placeholder object

On the *Placeholder object* tab, you can now open the *Values* tab and define the variables that are to be later switched when the value set is selected. Right click in the free field and select the **NAME NEW VARIABLE** function.

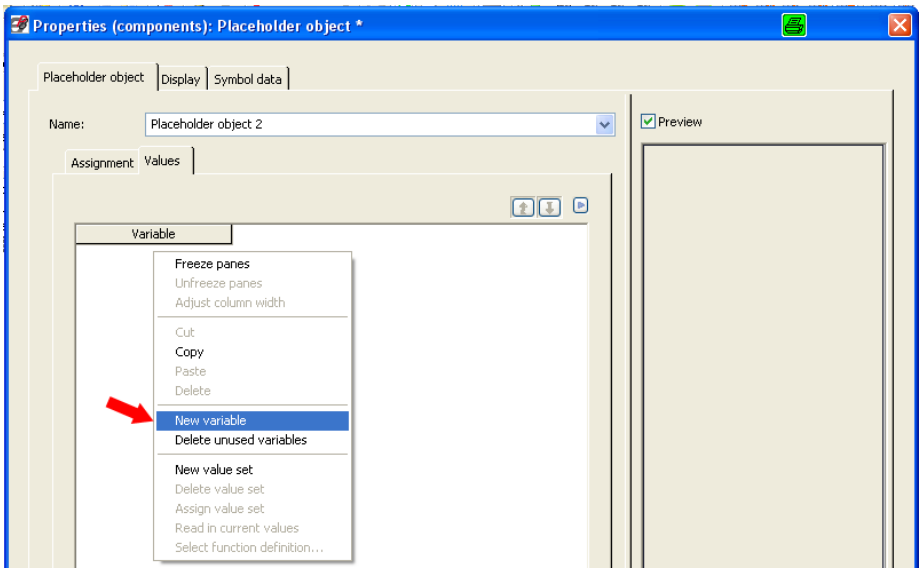


Fig. 4.171 Naming a new variable

EPLAN opens the **NAME NEW VARIABLE** dialog. The *Variable name* is now defined. When you click **OK**, the variable is applied in the *Values* tab.

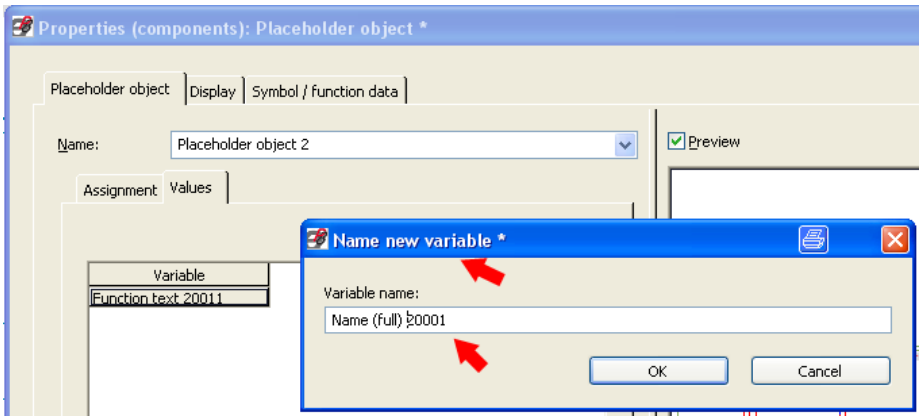


Fig. 4.172 Naming a new variable

If the variable name contains characters that are not permitted, EPLAN will display a message to this effect and will not apply the variable.

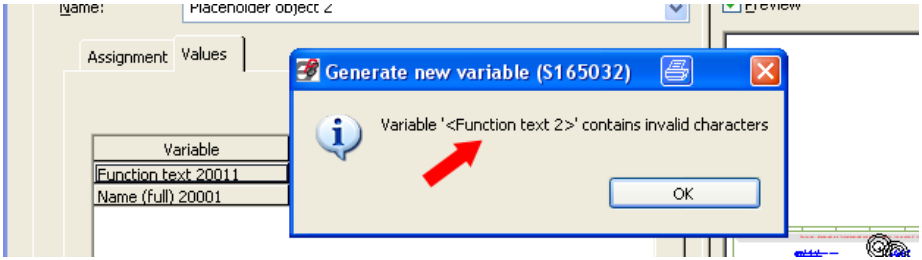


Fig. 4.173 Message indicating invalid characters

The value sets, which will be available for selection later on, must now be defined so that data is entered into the variables for the different value sets.

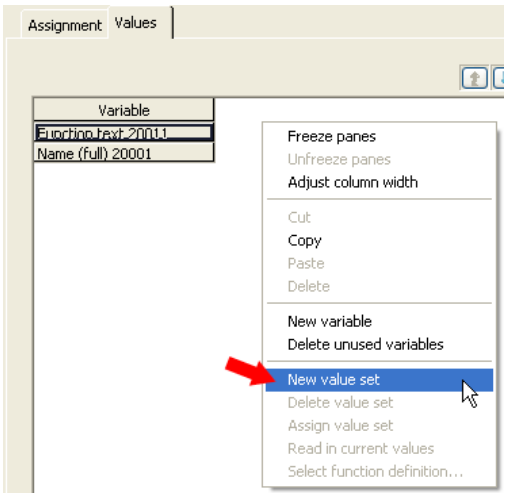


Fig. 4.174
Creating a new value set

Click the right mouse button again, and select the **NEW VALUE SET** function. EPLAN inserts a new column next to the **Variable** column. Enter the desired designation of the value set into the header of the column. Proceed in the same way for the next new value set. You will now have several value sets at your disposal for the subsequent selection of "switchable" values.

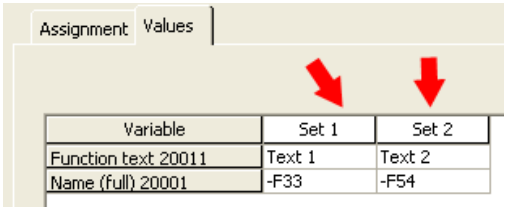


Fig. 4.175
Filling value sets with data

Data (the actual values) for the variables can now be entered below the value set name.

To assign the variables (and their values) to the device properties, you switch to the *Assignment* tab, and select the property that is later to be switched.

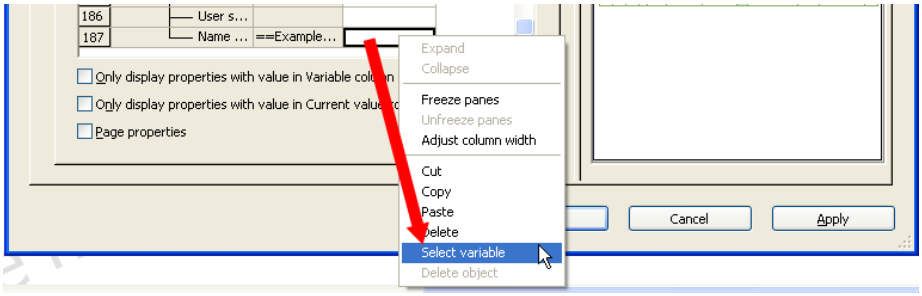


Fig. 4.176 Selecting a variable

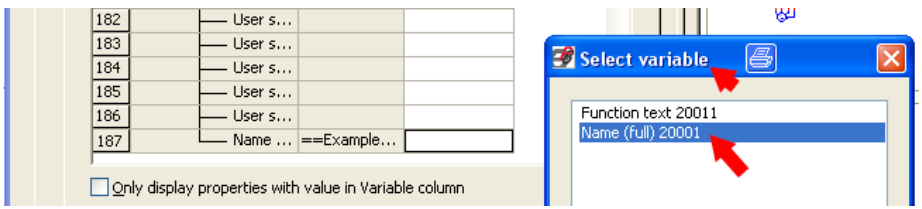


Fig. 4.177 Applying a variable

Now you can select the variable from the **SELECT VARIABLE** dialog and apply it. EPLAN applies the variable and enters it correctly in the **Variable** column.

TIP: This approach is to be recommended, since variables can be simply applied from a dialog, and EPLAN enters these with the correct syntax. This excludes the possibility of incorrect entries, which is not always the case with manual entry.

Once the variable has been applied, EPLAN establishes the assignment between the variable value and the value set (name). The dialog can now be saved and closed with **OK**. To now switch between the individual value sets, you select the placeholder object and right click to select the **ASSIGN VALUE SET** command from the popup menu. EPLAN opens the **SELECT VALUE SET** dialog. Here you select the desired entry and apply it by clicking **OK**. EPLAN now switches over to the variables/values inserted into this value set. In the example, that could be the function text.

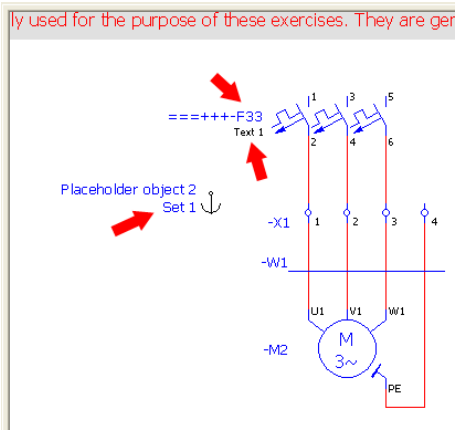


Fig. 4.178 Example value set 1

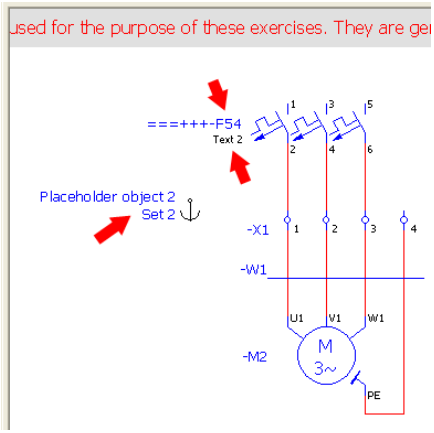


Fig. 4.179 Example value set 2

In conclusion, it can be said that this type of macro can be used in many different ways in projects. The example shown here was deliberately simple, merely pointing out the option of macros with value sets. The number of potential ideas is unlimited and possible sources of errors, e. g. incorrect part data or wrong cable cross-sections when changing motor power, are eliminated once and for all.

5 Navigators

The graphical editor with its many aids, such as the many navigators, is the central focus of project editing. Whatever approach you use, whether graphically-oriented, object-oriented, or a mixture of both, you always remain in the graphical editor. Only the way in which you use it can differ.

What is a navigator? A navigator provides a particular "viewpoint" for looking at project data. As the name indicates, the cable navigator only 'looks' at devices with the 'cable' function. The PLC navigator only looks at devices with the 'PLC' function.

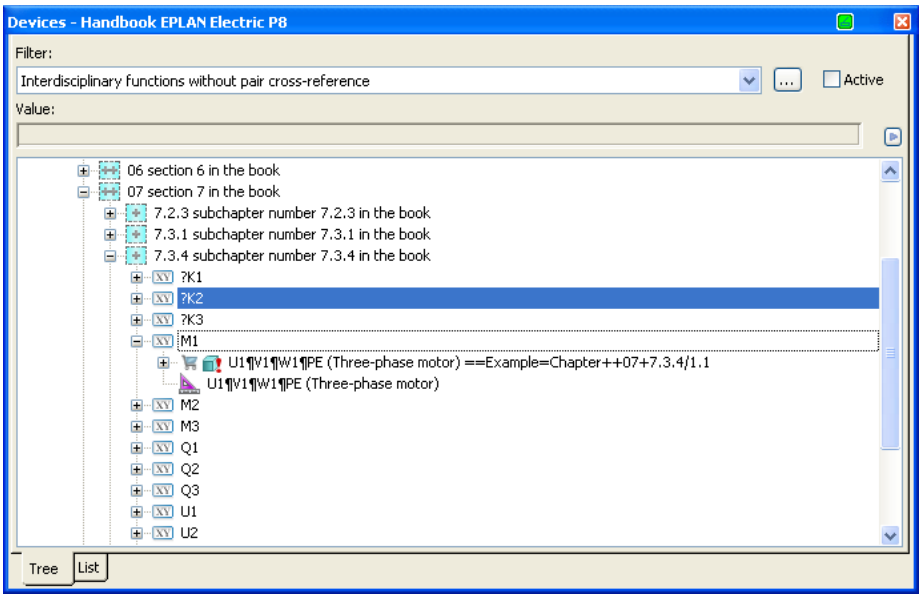


Fig. 5.1 A navigator and its view of devices

This chapter covers navigators, their basic functions and the additional functions in the **PROJECT DATA** menu. Numerous explanatory examples are used to demonstrate the correct use of devices, their settings and their properties.

To work quickly and efficiently with EPLAN, you must make use of the various navigators because they allow certain project actions to be performed much faster and more effectively – the key phrase here is **bulk editing**.

This chapter will not deal with all navigator functions, nor will it deal with all of the navigators in EPLAN Electric P8. The essential purpose of this chapter is rather to explain functions that are often used in daily work.

5.1 Overview of the most important navigators

When possible, it is a good idea to keep some navigators open all the time, for example the **device navigator** (this is the most comprehensive navigator as it displays all of the project data).

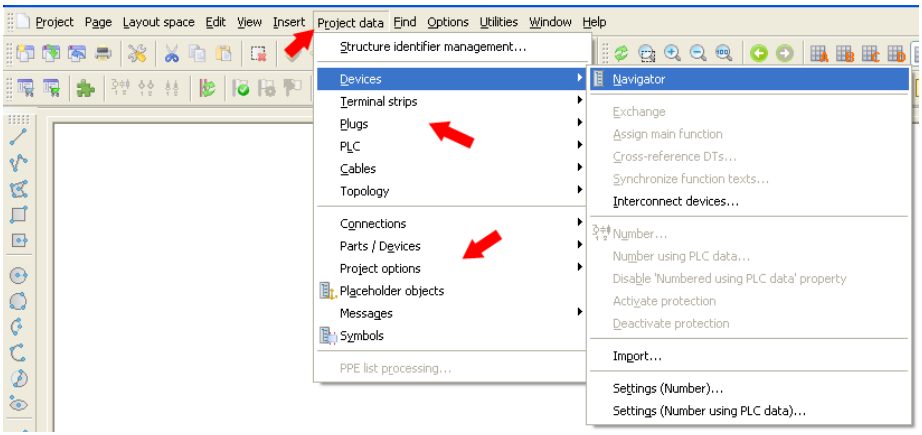


Fig. 5.2 Various navigators

But the device navigator is not the only navigator. EPLAN has a separate navigator for each of the different **device types** in the project data. Each of these navigators has specific tasks and features relating to the selected project data.

In addition to this, every **PROJECT DATA/[DEVICE TYPE]** menu item contains extra functions with a functional scope similar to the navigator menus. Some of the EPLAN navigators are listed here.

The **Device navigator** shows all *project data* and can, for example, directly edit a device's properties. But it cannot, for example, number terminals; this can only be performed by the special terminal strip navigator.

The **Terminal strip navigator** only shows project data for *terminals*. It has terminal editing functions such as renumbering and creating terminal strips.

The **Plug navigator** only shows project data for *Plugs/Sockets*. It has functions for editing plugs and sockets.

The **PLC navigator** only contains project data relating to *PLC* functions such as, for example, creating new PLC cards. It provides an overview of other PLC box functions (these can also be frequency converters with an integrated EA layer). The PLC navigator provides different views (representations) of the project data.

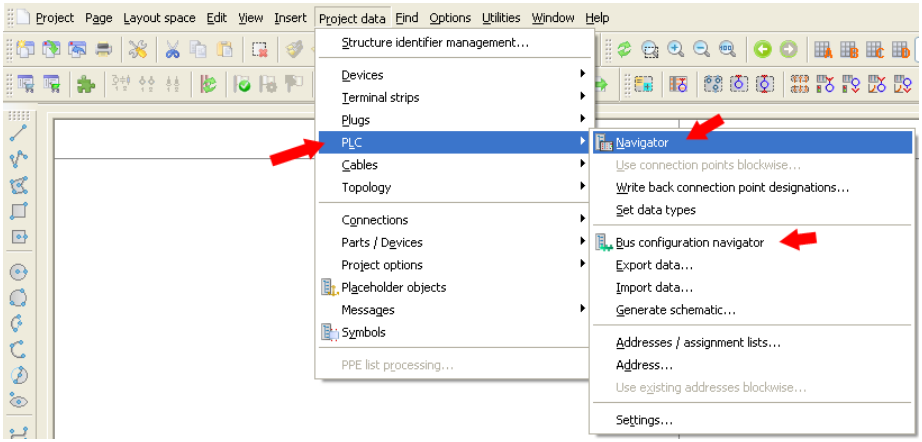


Fig. 5.3 Another navigator: Bus configuration navigator

The PLC navigator also contains the **Bus configuration navigator** (an additional navigator for PLC objects). *Master/slave bus configurations* and *rack/module* assignments are managed here.

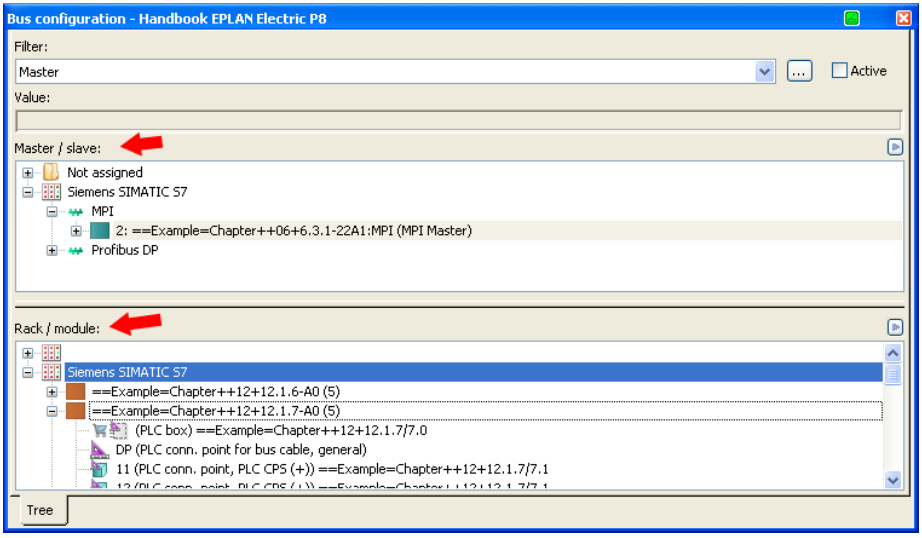


Fig. 5.4 Bus configuration navigator

The **Cable navigator** contains all project data for *cables* and their properties. With the cable navigator, you can, for example, place or collect *conductors*.

The **Topology navigator** contains information pertinent to the *Routing path networks* (2D method), such as routes and routing points.

The **Connection navigator** contains all project data related to the *connections* in a project. You can use this navigator, for example, to use existing connections.

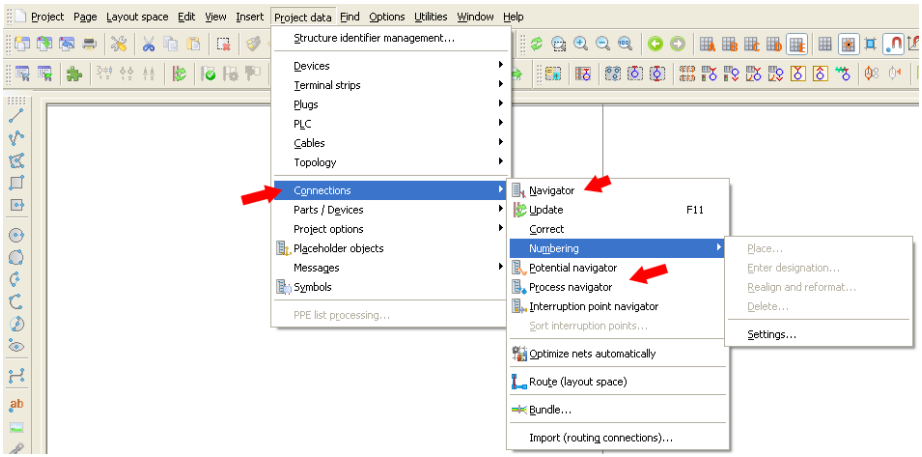


Fig. 5.5 Additional navigators for connections project data

The connections project data also includes the **Potential navigator**, the **Process navigator** and the **Interruption point navigator**. These navigators manage potentials, signals, networks, interruption points and the processes defined in the project.

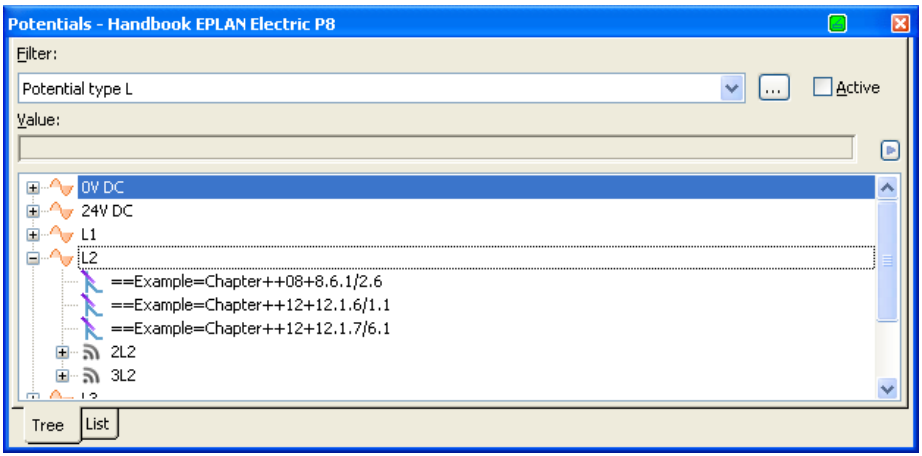


Fig. 5.6 Potential navigator

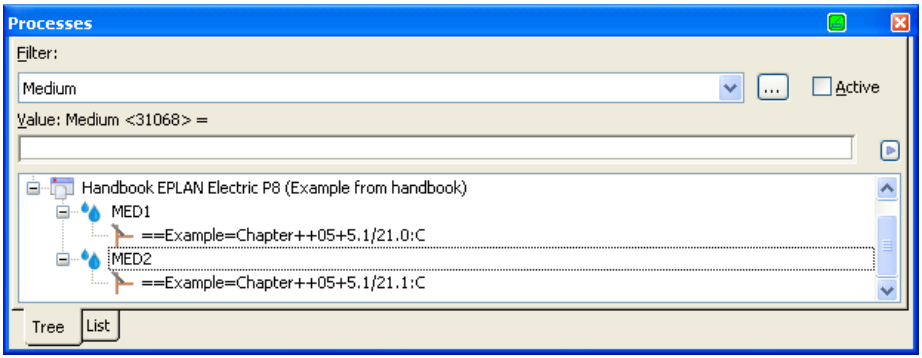


Fig. 5.7
Process navigator

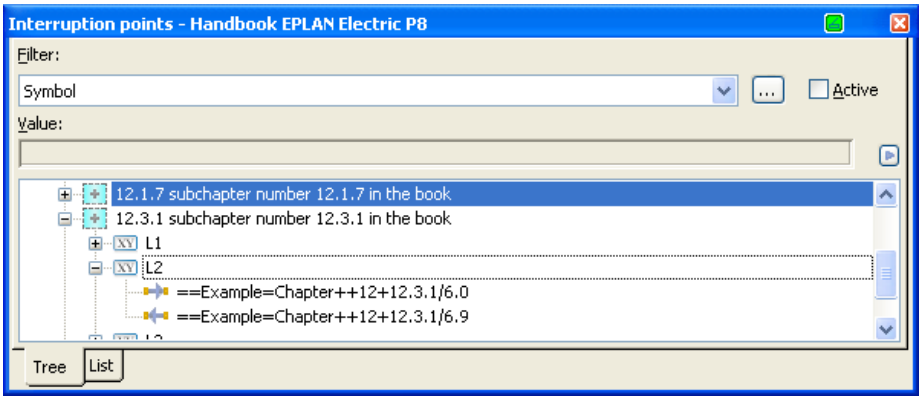


Fig. 5.8
Interruption point navigator

There are additional navigators in the **PROJECT DATA/PARTS/DEVICES** menu. The **bill of materials navigator** contains all functions for centrally editing parts, such as swapping parts or selecting a contactor. Unlike the other navigators, the bill of materials navigator also provides special views of the project data in a tree view.

There are also two more navigators in the **PARTS/DEVICES** menu.

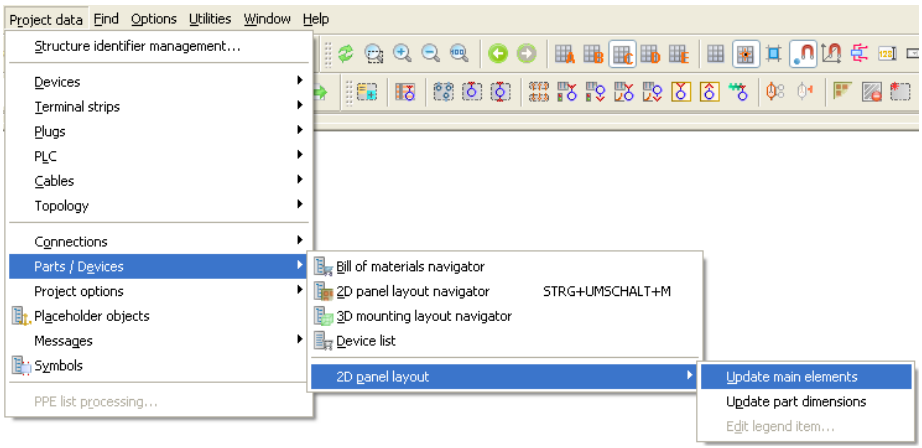


Fig. 5.9
Navigators in the parts/devices menu

The **2D panel layout navigator** allows, for example, the creation of a 2D structure diagram of the devices used in the project, based on technical data (at least width and height) from the parts master data or manually entered data.

The **3D mounting layout navigator** allows a 3D diagram of the devices used in the project, based on technical data (at least width, height and depth) from the parts master data.

The **Project options navigator** allows creation of schematics using various options. The options can be easily switched on and off, by page, across pages, or in page sections.

The **Placeholder objects navigator** conveniently manages macros with value sets from a single location, from changing global value sets editing selected objects.

5.1.1 Additional navigators and modules

The following entries in the **PROJECT DATA/SYMBOLS** menu are not navigators in the sense of those described in section 5.1. This entry allows symbol selection to stay open all the time like a navigator.

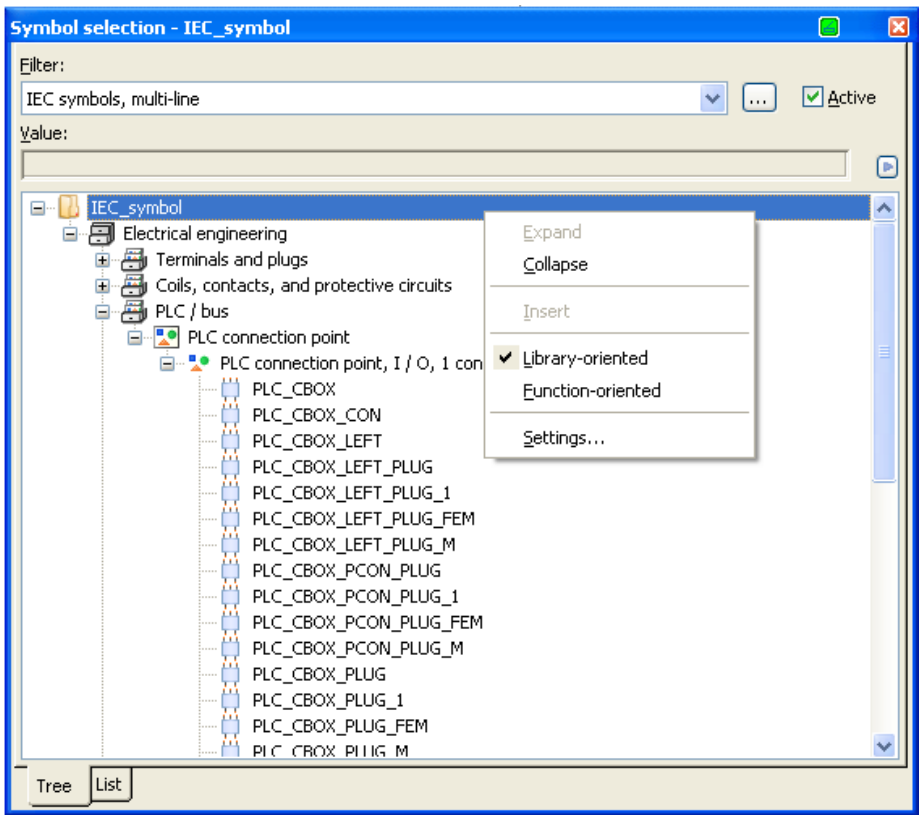


Fig. 5.10 Constantly open symbol selection

The following navigators are found at other locations: the **Part master data navigator** and the EPLAN **Data portal navigator**. The part master data navigator is started in the **UTILITIES/PARTS/PART MASTER DATA NAVIGATOR** menu, and the EPLAN data portal navigator is started in the **UTILITIES/DATA PORTAL NAVIGATOR** menu.

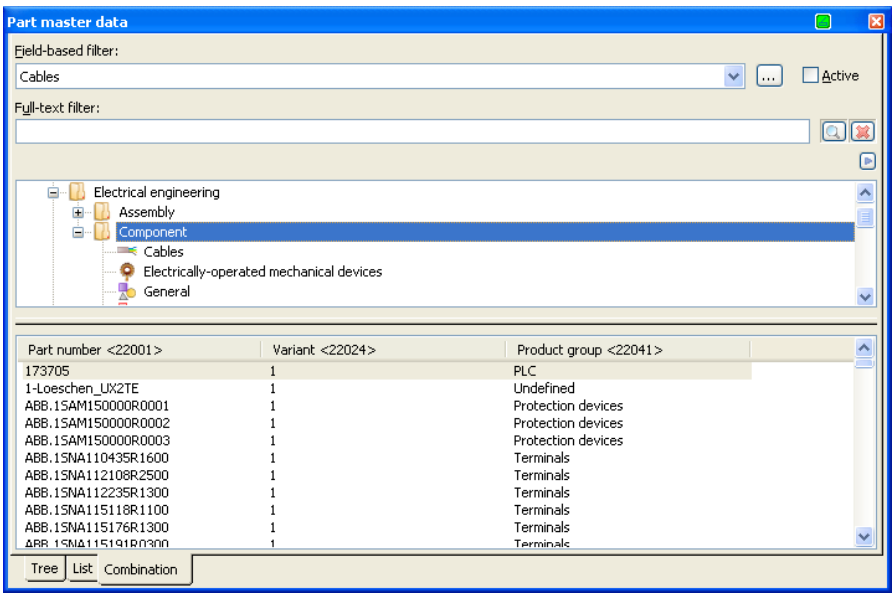


Fig. 5.11
Part master data navigator

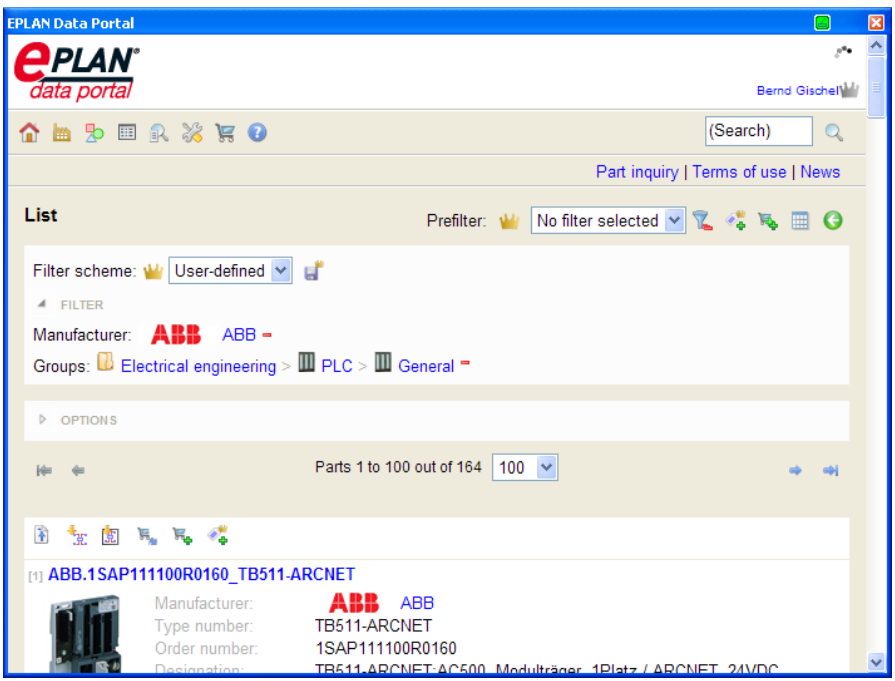


Fig. 5.12
EPLAN Data Portal navigator

These navigators are independent dialogs that can remain open in the same way as 'true' navigators. For example, the **Parts master data navigator** always allows access to parts management without having to open parts management via **UTILITIES/PARTS/ADMINISTRATION**.

5.2 Device navigator

All information and project data come together in the **device navigator**. The device navigator can be accessed via the **PROJECT DATA/DEVICES/NAVIGATOR** menu item, and can then be positioned anywhere on the desktop. The device navigator can do everything that can be manually done to a device during device editing, i. e. changing device tags, changing the display format, editing a device's properties, and much more. But the major advantage of the device navigator is that it does everything from a central point while maintaining a view of the entire project.

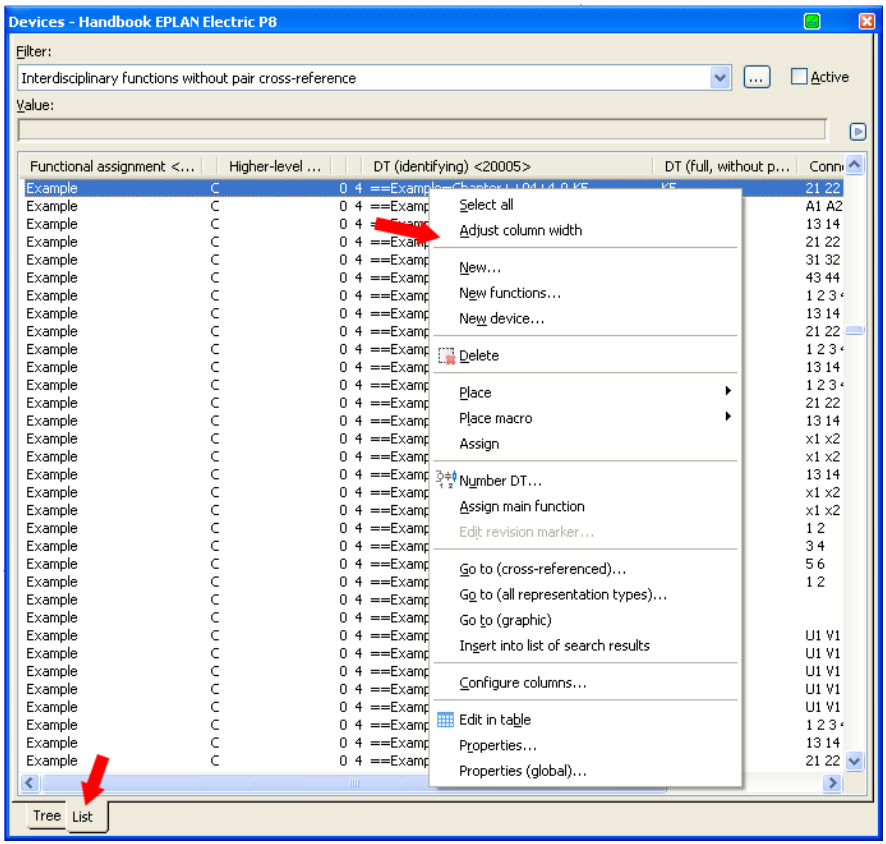


Fig. 5.13
List view in the device navigator

The *filter* and *sorting features* of the device navigator can be used to swap device navigator function definitions or connection point designations simply and conveniently. The device navigator can do more than just display and edit the properties of a device. You can also use it to edit or number several devices at once via the block functions (bulk editing).

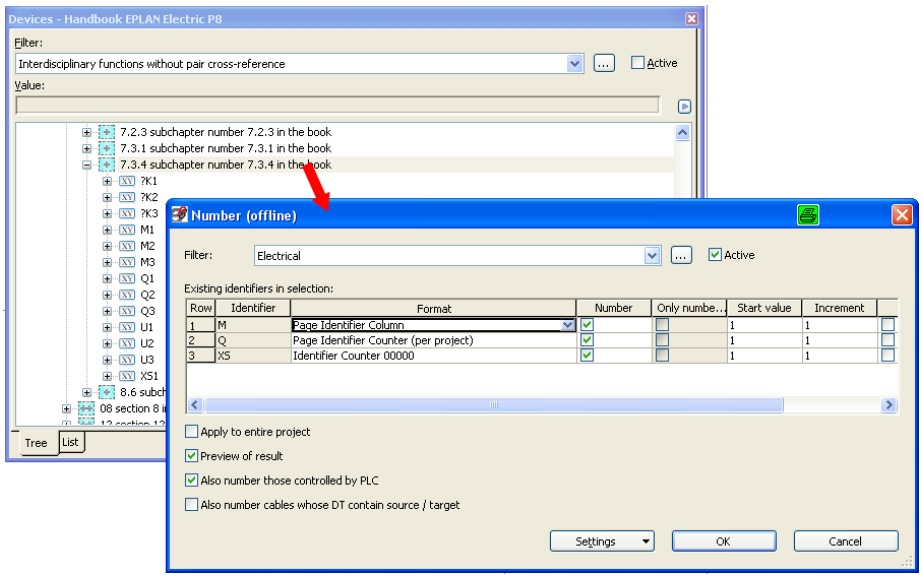


Fig. 5.14 Bulk editing with the device navigator

This section not only deals with the device navigator and its functional scope, but also with a series of other functions in the **PROJECT DATA/DEVICES** menu. For reasons of space, this chapter does not provide an overview of all of the menu items and their functions.

For the device navigator, it is a good idea to create your own keyboard shortcut for opening and closing the navigator, since it is used most often and must be opened and closed constantly (unless it remains open all the time anyway). My recommendation is **CTRL + SHIFT + D** for the device navigator and to always keep it and the page navigator open.

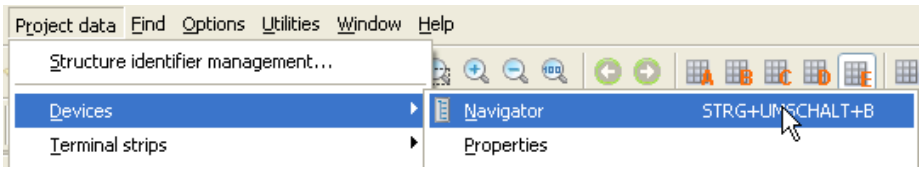


Fig. 5.15 Keyboard shortcut for the device navigator

5.2.1 Device navigator/Assign main function

EPLAN distinguishes between a **main function**, for example a coil, and an **auxiliary function**, such as the contacts belonging to a coil.

The *main function* setting on the first tab in the **PROPERTIES (COMPONENTS)** dialog, or on the *Symbol/function data* tab, turns a device into a main function.

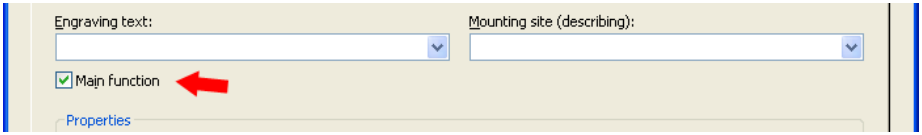


Fig. 5.16 Main function setting in the Properties (components) dialog

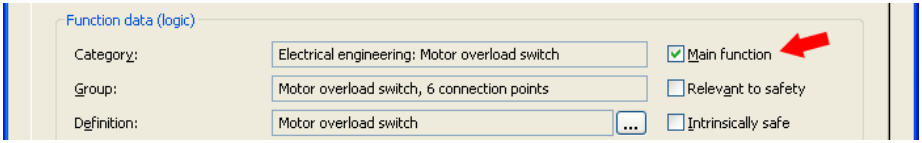


Fig. 5.17 Main function setting on the Symbol/function data tab

This is very important because, for example, only devices that have a main function can have parts stored on them or have devices selected for them. If devices do not have a main function characteristic then, for example, they cannot be numbered.

Therefore, every independent item should initially be a main function. However, if other items with the same device tag exist in the project, then these are forced to become auxiliary functions.

NOTE: Only one main function per device is allowed at all times. Double main functions are not allowed and are (usually) detected with a check run and reported (if the corresponding check run message is used).

In the **PROPERTIES (COMPONENTS)** dialog, both ways of activating a main function are always set when one of them is set. Regardless of where you set the check box for the main function, the other input check box is also automatically activated or deactivated.

The advantage of the **ASSIGN MAIN FUNCTION** menu function here is that the main function need not be manually set for every device where this is necessary. It is entirely possible that copy operations or other similar actions can cause a device to have no main function.

NOTE: Usually, EPLAN automatically checks the main function checkbox after inserting a new symbol.

In the example, all of the devices on the schematic page were selected in order to renumber them. As an alternative, of course, these devices can also be selected in the device navigator.

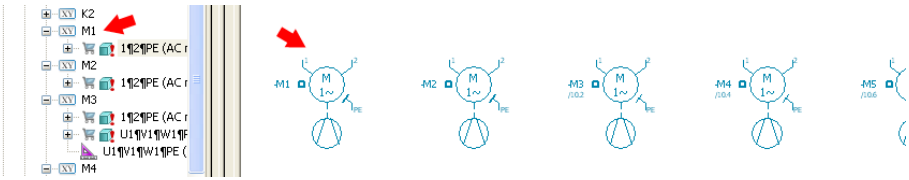


Fig. 5.18
Selected devices

After they were selected, the **NUMBER** function in the **PROJECT DATA/DEVICES/NUMBER** menu was executed. EPLAN, however, will not want to number all of the devices. Some of the devices will not be numbered because they do not have the **main function** characteristic.

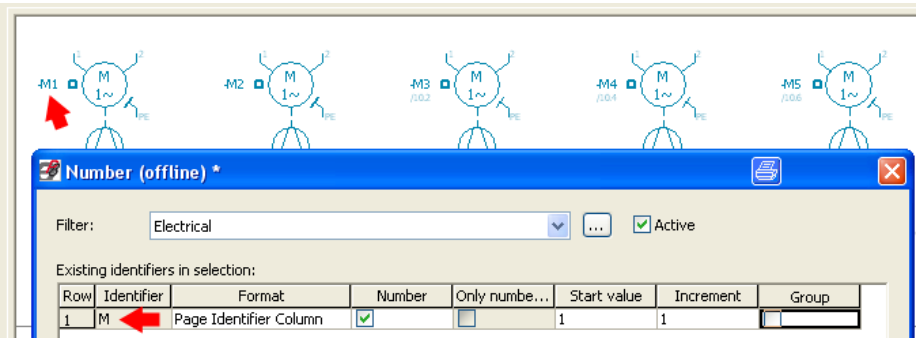


Fig. 5.19
Definitions before numbering

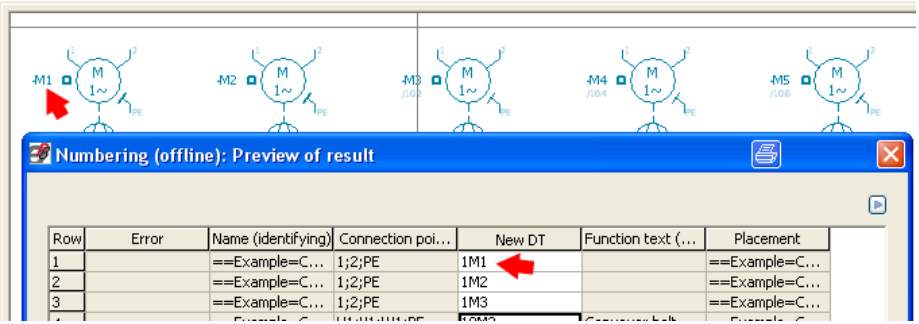


Fig. 5.20
Preview of result before final numbering

TIP: I recommend wherever possible to display the preview of result in the follow-up dialogs. Even if this may take a moment longer, this preview is advantageous because you can cancel immediately if necessary without something "happening" to the project.

The results of the numbering are shown in Fig. 5.21. All except a few devices were numbered. All devices without a main function were not numbered.

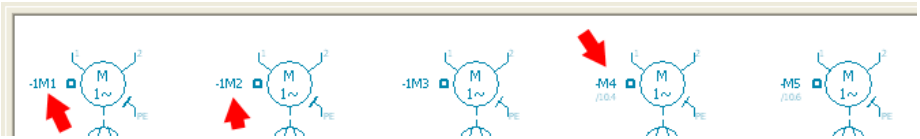


Fig. 5.21 Result of numbering

Assign main function

You can now use **ASSIGN MAIN FUNCTION** to transfer the characteristic to devices without the main function. You first select the devices in the schematic, one after the other, and then select the **ASSIGN MAIN FUNCTION** item in the **PROJECT DATA/DEVICES** menu.

After performing this action (i. e. by clicking the menu item), all devices now have the main function. If you now start the **NUMBER** function again for comparison purposes, the preview dialog will show all devices waiting to receive the new device tag.

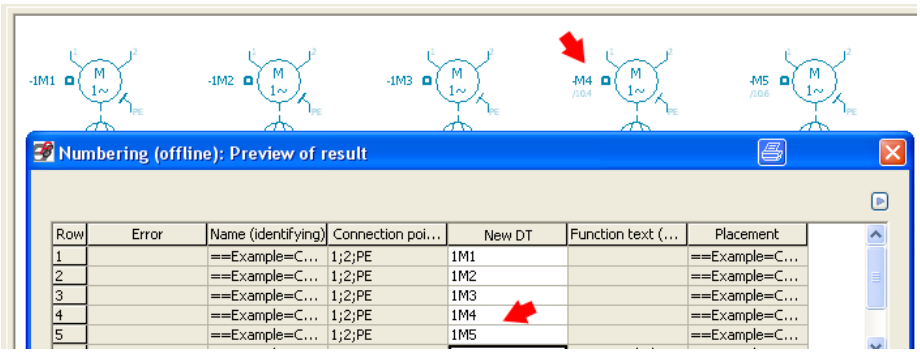


Fig. 5.22 All devices are now numbered.

ASSIGN MAIN FUNCTION can, of course, be assigned to a keyboard shortcut.

5.2.2 Device navigator/Cross-reference DTs

This function cross-references the device tags of the selected devices. To put it more precisely, the device tag is set to the same value for all the selected devices. The important phrase here is "selected devices". This does not include the main or auxiliary functions belonging to the devices, rather only the *actually* selected devices. You should take this into consideration when using this function.

All the devices in Fig. 5.23 are to be given the device tag **–K2**. First you select all the devices (at least two must always be selected, otherwise the **CROSS-REFERENCE DTS** function cannot be executed) and then select the **CROSS-REFERENCE DTS** function from the **PROJECT DATA/DEVICES** menu.

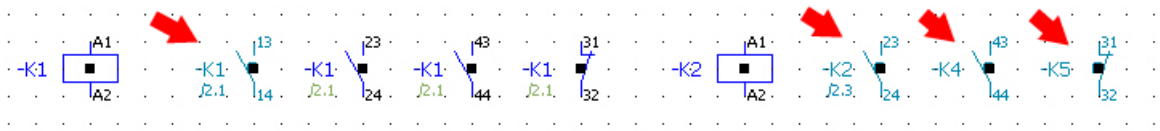


Fig. 5.23 Selecting desired devices

The **CROSS-REFERENCE DTS** dialog is then displayed. Here you click **only** the device tag that is to be transferred to all the selected devices. In the example, you click the row –K2. This transfers device tag –K2 to the other device tags.

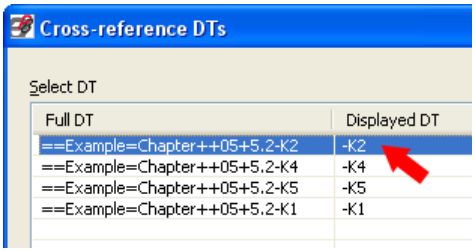


Fig. 5.24 Cross-reference DTs dialog

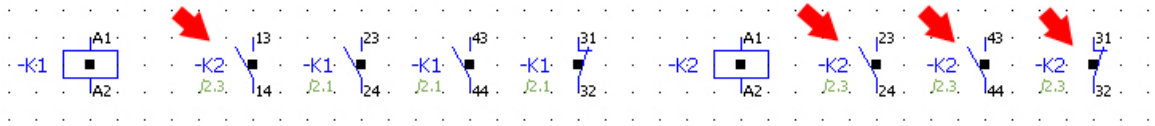


Fig. 5.25 Cross-referenced devices

All other devices retain their original device tags because they were not selected.

5.2.3 Device navigator/Synchronize function texts

This function allows you to synchronize the function texts of the selected devices, including all auxiliary functions and the main function. It makes no difference whether the main function or one of the auxiliary functions is selected.

It must always be the same device, though. Different devices cannot be edited with this function.

All devices should receive the same function text. First you select an element. This can be the main function or any auxiliary function in the schematic.

Then you execute the **SYNCHRONIZE FUNCTION TEXTS** function via the **PROJECT DATA/DEVICES** menu item.

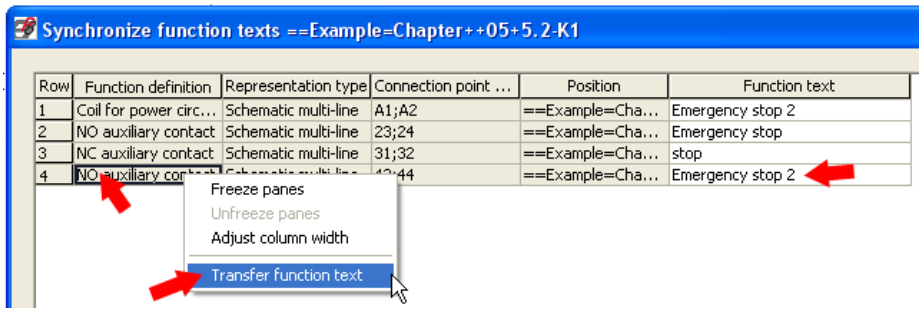


Fig. 5.26 A selected element and the Synchronize function texts dialog

EPLAN opens the **SYNCHRONIZE FUNCTION TEXTS** dialog. In this dialog, only the row containing the function text to be transferred to all other functions is selected. In the example, the function text of row 2 will be transferred to all other elements. Right-click to open the popup menu and select the **TRANSFER FUNCTION TEXT** function. EPLAN now transfers the function text to all other entries in the dialog.

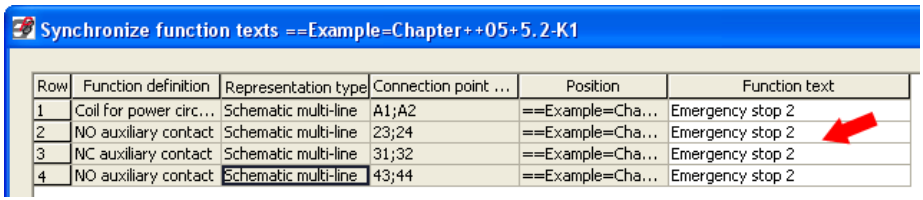


Fig. 5.27 Transferred function text in the dialog

After the transfer, you click the **OK** button. EPLAN exits the dialog and transfers the function text to all elements and writes this text to the **function text <20011>**, **function text (automatic) <20031>**, and **function text (common) <20120>** symbol properties.

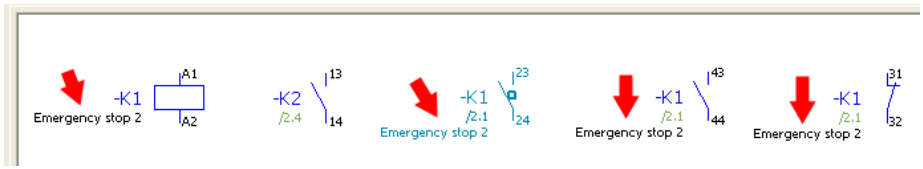


Fig. 5.28 Transferred function text in the schematic

5.2.4 Device navigator/Interconnect devices

The EPLAN **INTERCONNECT DEVICES** function allows interconnection of devices. In this case, interconnection means the generation of connections between the selected interconnected devices. These can be placed devices , i. e. those that are already graphically in the schematic, or unplaced devices, i. e. those that are only in the device navigator. A mixture of both types of placement is also possible, of course, and they can be any devices.

In the example, two terminal strips (with several unplaced terminals and isolated cable ends) are interconnected with an existing cable and its reserve conductors without being placed graphically on a page.

Icon for a placed terminal in the device navigator

Icon for an unplaced terminal in the device navigator

The **INTERCONNECT DEVICES** function is started via the menu item **PROJECT DATA/DEVICES**. EPLAN then opens the **INTERCONNECT DEVICES** dialog.

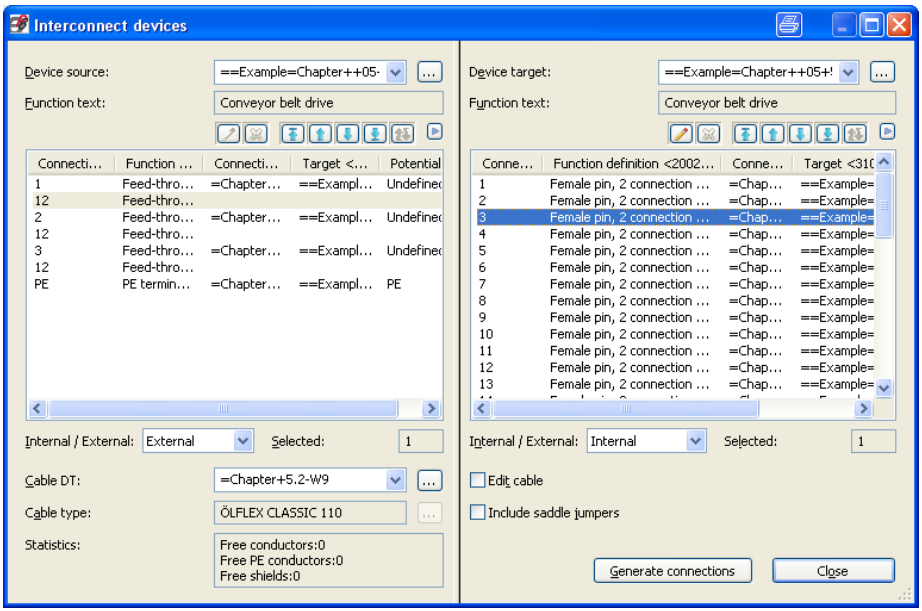


Fig. 5.29 Interconnect devices dialog

The **INTERCONNECT DEVICES** dialog can be filled with various data after it is called up. If these entries are not required, then you can simply delete them. You delete entries by selecting them and then clicking the delete button.

If this is not possible, then you can also delete the entries in the **device source**, **device target**, and **cable DT** fields. Click the field, select the desired entry, and then press the **DEL** key. All entries are deleted, and new devices can be selected and interconnected.

New devices are selected in the **device source** and **device target** fields using the ... button. When you click this button, EPLAN opens the **SELECT DEVICE** dialog. In this dialog, you select the desired devices for the source and target (one of each) and click **OK** to apply them.

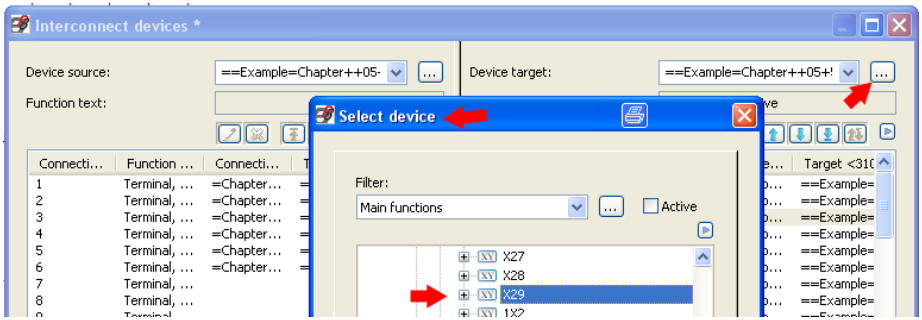


Fig. 5.30 Selecting the device source and target

EPLAN then loads the selected device into the **device source** and **device target** fields. An additional cable is to serve as a connection between the two terminal strips to be interconnected. You select an existing cable in the **cable DT** field for this purpose.

TIP: It is also easy to create a new, non-existent cable at this point. You just click the field and enter the cable DT.

To use an existing **cable DT**, you simply click the ... button in the SELECT CABLE dialog to select it and then click **OK**.

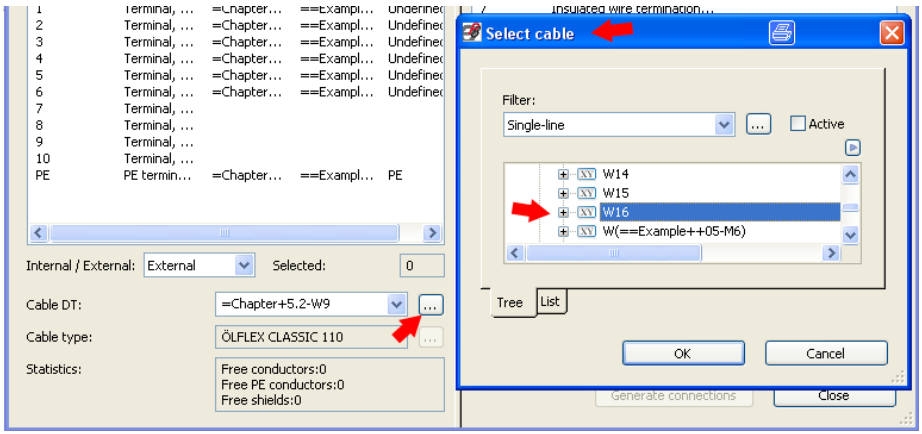


Fig. 5.31 Selecting cables for interconnection

Once the preparatory measures have been completed, both terminal strips, including the intended functions (unplaced terminals and unplaced isolated wire terminations), can now be interconnected. Select the **terminals** and **isolated wire terminations** in the fields (click and then hold down the CTRL button and the mouse to select the desired terminals and isolated wire terminations) and then click the GENERATE CONNECTIONS button.

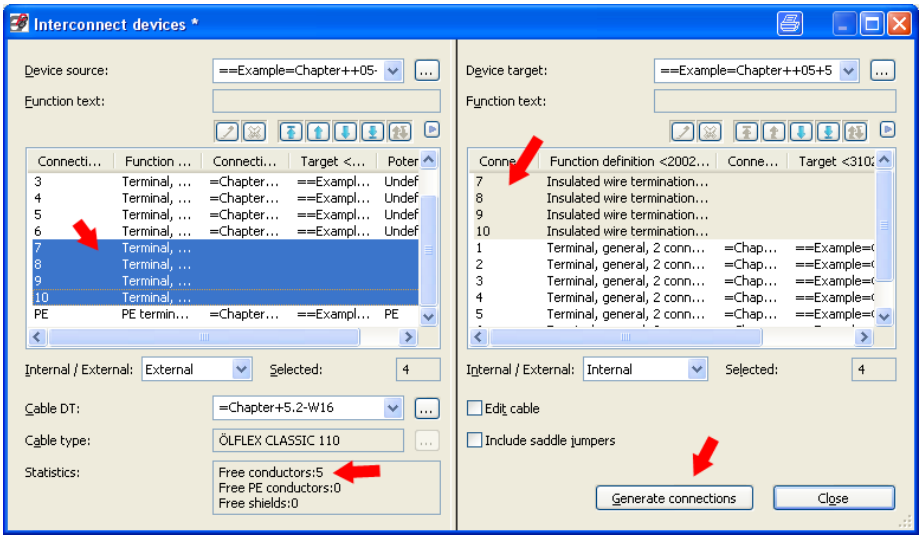


Fig. 5.32 Generating connections

EPLAN now generates the unplaced connections between the terminals and the isolated wire terminations and assigns the cable and cable conductor.

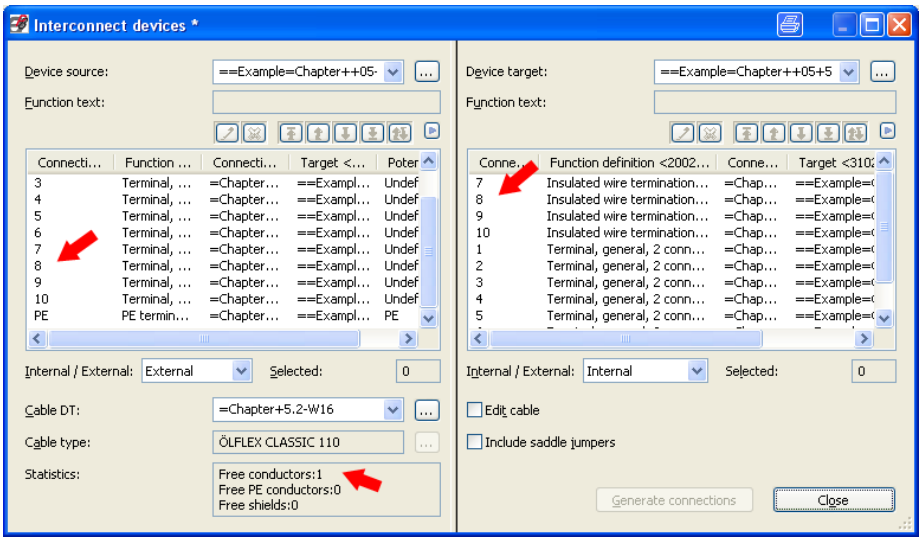


Fig. 5.33 Connections were generated.

You can also still edit these connections directly in the **INTERCONNECT DEVICES** dialog. Simply double-click the connection. EPLAN opens the **PROPERTIES (COMPONENTS)** dialog.

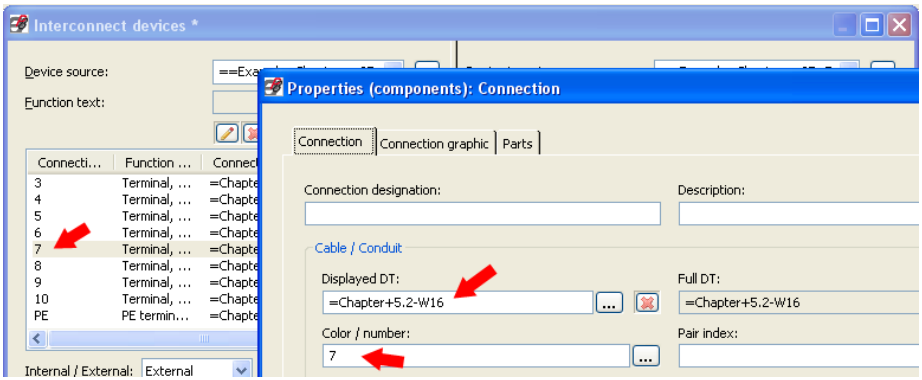


Fig. 5.34 Direct editing of individual connections

This basically completes the creation of connections. If you had created a report before interconnecting the devices (in this case, a cable diagram for the affected cable), EPLAN would only have listed the devices used, including the cable conductors used, in the report.

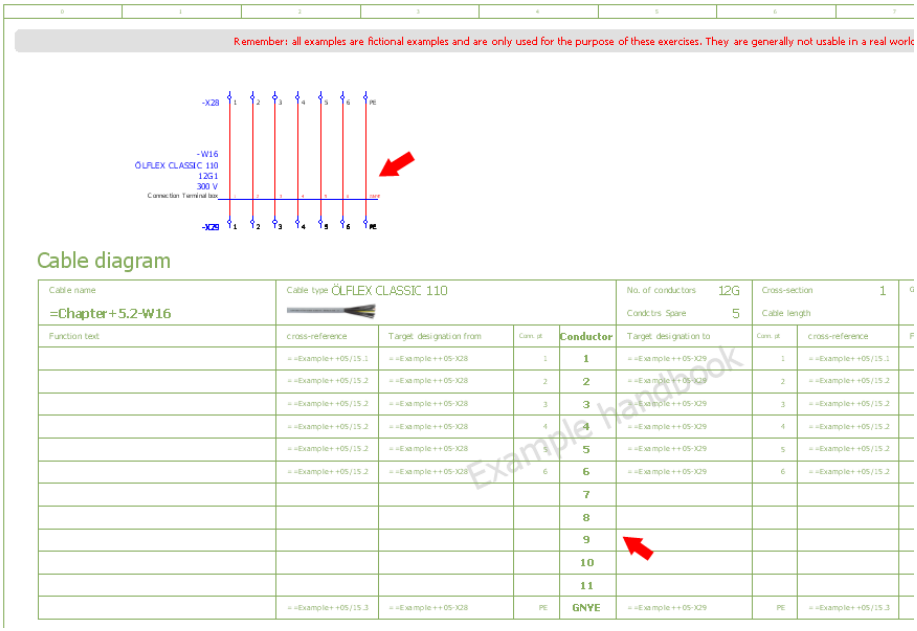
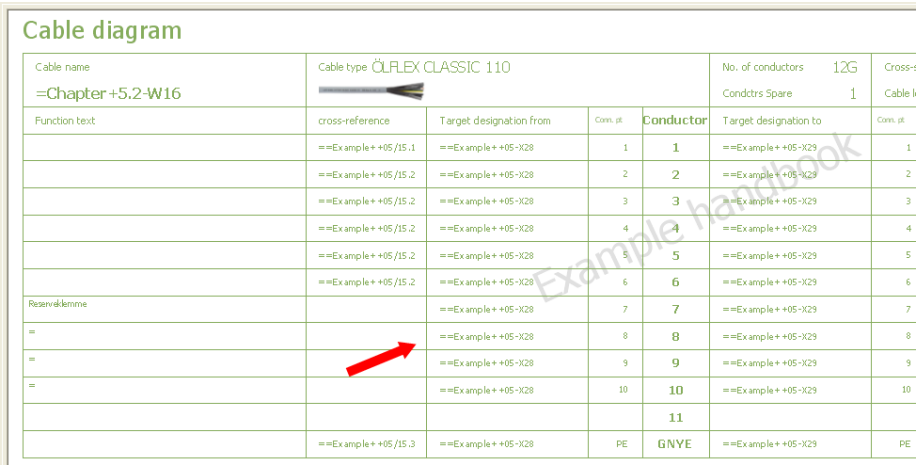


Fig. 5.35 Report before interconnecting devices

After updating the **cable diagram** report, EPLAN adds the interconnected devices to the report. The devices don't receive a cross-reference (page/path) because they are not placed anywhere.



Cable diagram

Cable name =Chapter+5.2-W16	Cable type ÖLFLEX CLASSIC 110				No. of conductors 12G Condctrs Spare 1	Cross-s Cable l
Function text	cross-reference	Target designation from	Conn. pt	Conductor	Target designation to	Conn. pt
	==Example+ +05 /15.1	==Example+ +05-X28	1	1	==Example+ +05-X29	1
	==Example+ +05 /15.2	==Example+ +05-X28	2	2	==Example+ +05-X29	2
	==Example+ +05 /15.2	==Example+ +05-X28	3	3	==Example+ +05-X29	3
	==Example+ +05 /15.2	==Example+ +05-X28	4	4	==Example+ +05-X29	4
	==Example+ +05 /15.2	==Example+ +05-X28	5	5	==Example+ +05-X29	5
	==Example+ +05 /15.2	==Example+ +05-X28	6	6	==Example+ +05-X29	6
Reserveklemme		==Example+ +05-X28	7	7	==Example+ +05-X29	7
=		==Example+ +05-X28	8	8	==Example+ +05-X29	8
=		==Example+ +05-X28	9	9	==Example+ +05-X29	9
=		==Example+ +05-X28	10	10	==Example+ +05-X29	10
				11		
	==Example+ +05 /15.3	==Example+ +05-X28	PE	GNYE	==Example+ +05-X29	PE

Example handbook

Fig. 5.36 Updated report

The devices themselves are displayed with a red mark on their symbol to visually identify them. This lets you know that this is an unplaced connection.

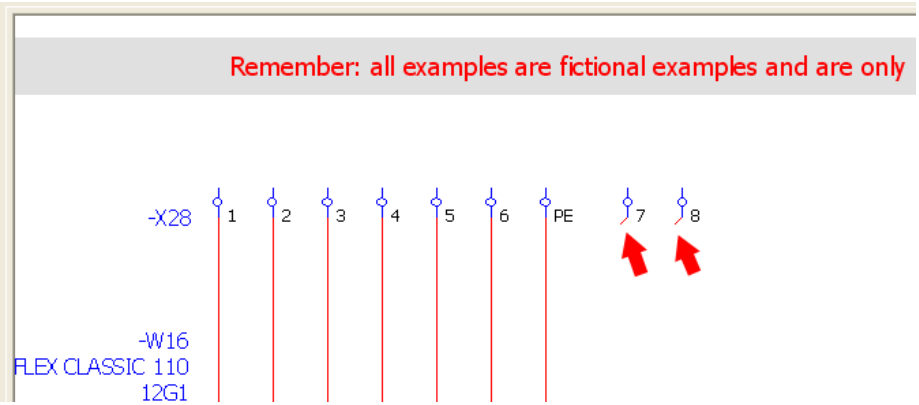


Fig. 5.37 Visual identification of unplaced connections

5.2.4.1 Place interconnected unplaced devices

Of course, it is also possible to place these interconnected devices, including connection information (cable DT and their conductors), without having to redraw everything. In the device navigator, you multi-line place either the **device source** or the **device target** on a schematic page.

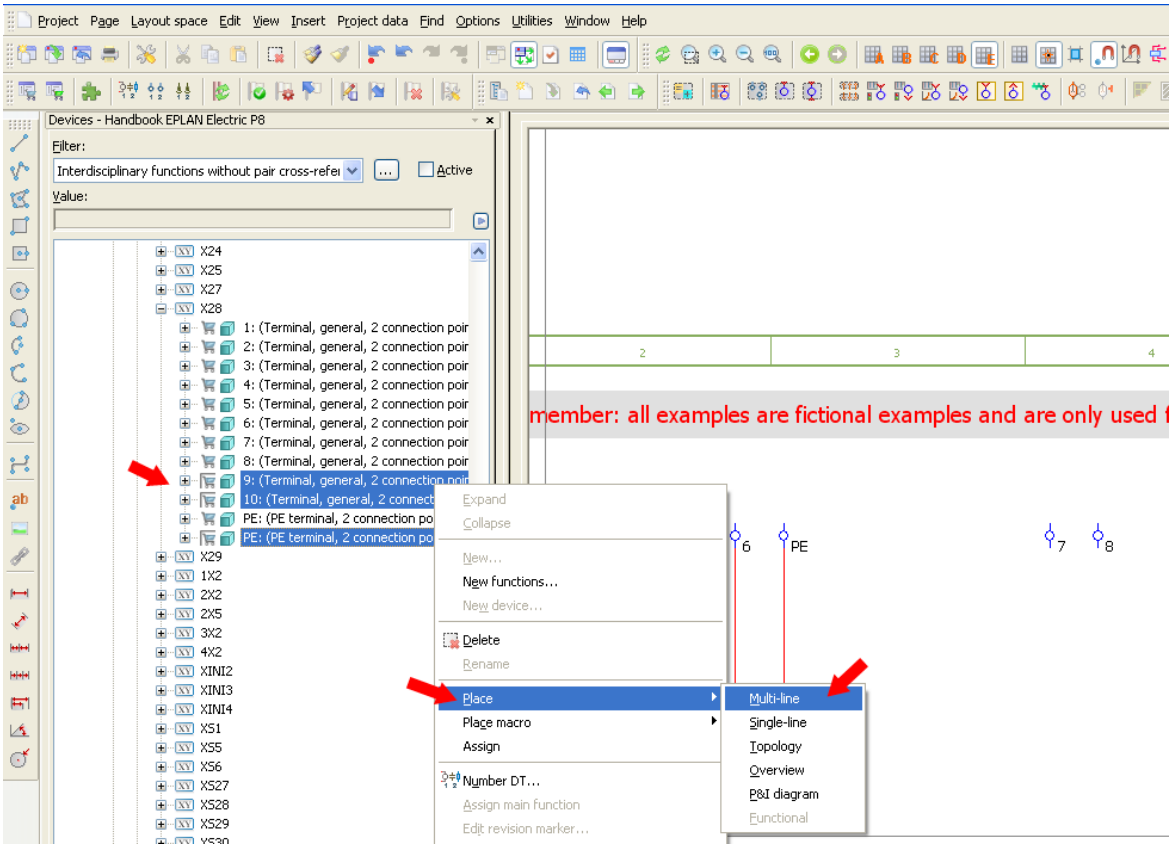


Fig. 5.38 Placing multi-line functions from the device navigator

After the devices are placed, the **CONNECTED FUNCTIONS** menu item can be called up via the **INSERT** menu.

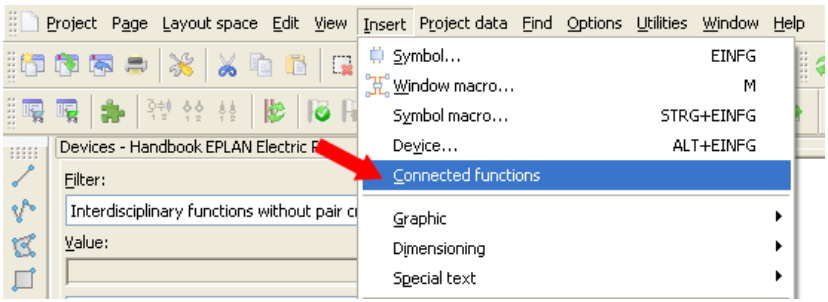


Fig. 5.39 Inserting connected functions

EPLAN will ask for which devices the connected functions should be displayed. These are selected with a window.

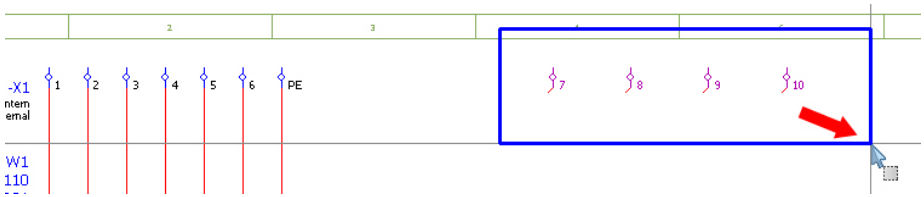


Fig. 5.40 Selecting the relevant functions

When you define (click) the window's second corner, EPLAN immediately displays the corresponding function on the schematic page.

If you again update the associated report, then the data of the previously unplaced functions will be updated in the cable diagram with a cross-reference specification.

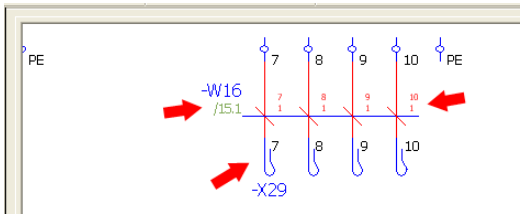


Fig. 5.41 Placed functions

Cable diagram

Cable name =Chapter+5.2-W16	Cable type ÖLFLEX CLASSIC 110				No. of conductors 12G Condctrs Spare 1	Cross-sect Cable leng
Function text	cross-reference	Target designation from	Conn. pt	Conductor	Target designation to	Conn. pt
	==Example++05/15.1	==Example++05-X28	1	1	==Example++05-X29	1
	==Example++05/15.2	==Example++05-X28	2	2	==Example++05-X29	2
	==Example++05/15.2	==Example++05-X28	3	3	==Example++05-X29	3
	==Example++05/15.2	==Example++05-X28	4	4	==Example++05-X29	4
	==Example++05/15.2	==Example++05-X28	5	5	==Example++05-X29	5
	==Example++05/15.2	==Example++05-X28	6	6	==Example++05-X29	6
Reserveklemme	==Example++05/15.4	==Example++05-X28	7	7	==Example++05-X29	7
=	==Example++05/15.4	==Example++05-X28	8	8	==Example++05-X29	8
=	==Example++05/15.4	==Example++05-X28	9	9	==Example++05-X29	9
=	==Example++05/15.4	==Example++05-X28	10	10	==Example++05-X29	10
				11		
	==Example++05/15.3	==Example++05-X28	PE	GNYE	==Example++05-X29	PE

Example handbook

Fig. 5.42 Updated report

5.2.5 Device navigator/Number (offline)

EPLAN can number devices according to a definable numbering pattern as soon as they are inserted (according to the personal setting). If the schematic and its devices need to be renumbered in a different format later, a fast and effective un-numbering method is important.

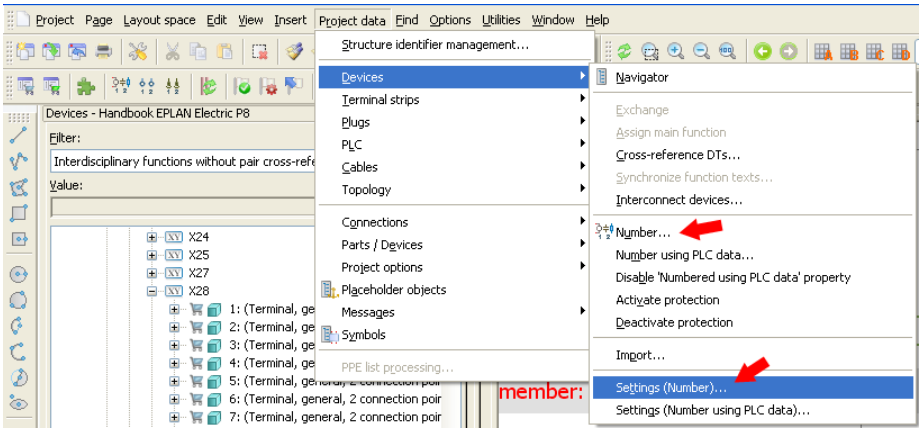


Fig. 5.43 Numbering devices

EPLAN offers the **NUMBER DEVICES** function for this. In contrast to numbering when inserting symbols (**online numbering**), subsequent renumbering is **offline numbering**. Certain filters and schemes can be used for numbering, and predefined schemes can also be applied, exported or imported.

One important thing should be noted about offline numbering. Auxiliary functions cannot be numbered. If these items are selected and numbering is then attempted, EPLAN displays an appropriate message.

How does the numbering function work? Very simple. You select the devices to be numbered and then start the **NUMBER** function from the **PROJECT DATA/DEVICES** menu.

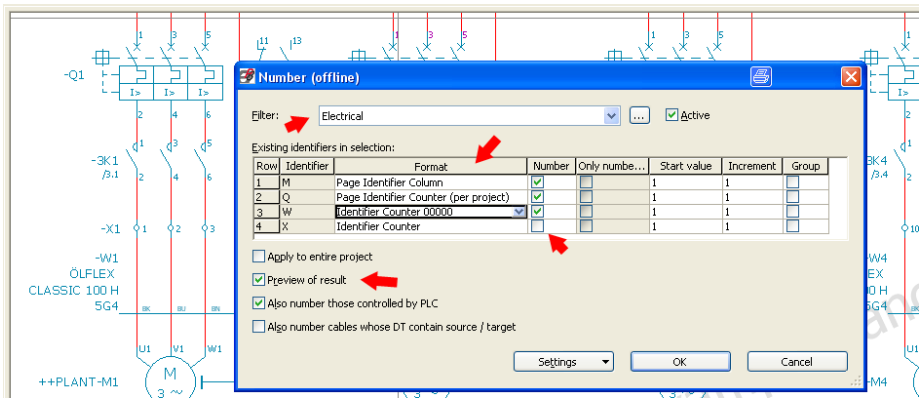


Fig. 5.44 Settings in the Number (offline) dialog

Number (offline)

EPLAN opens the **NUMBER (OFFLINE)** dialog and lists the found **identifiers** in a table. In this dialog, you can set your own **numbering format**, start value and increment for each identifier entry. Entire identifiers can be excluded here from numbering so that you are not forced to select only very specific devices when selecting devices.

NOTE: It is important to always use the *Preview of result* setting. This enables you to check one more time that everything is numbered as desired before EPLAN writes the new numbering back to the project.

A different numbering format can simply be selected in the *format* selection list, or the **EXTRAS** button in the lower area of the dialog can be used to create a new numbering scheme.

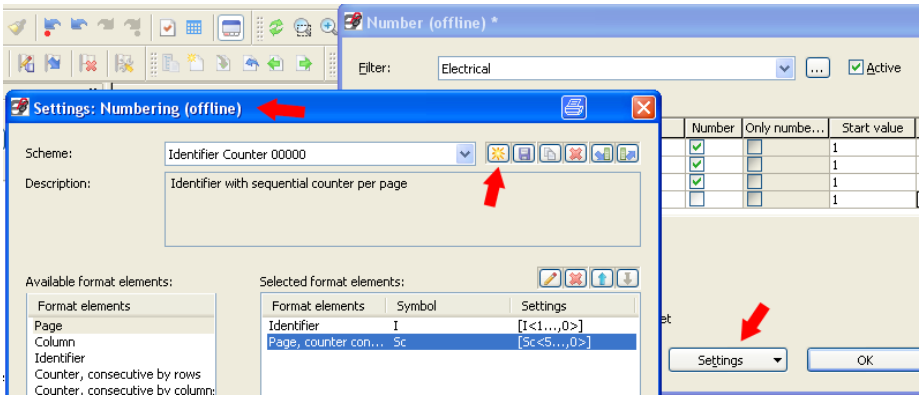
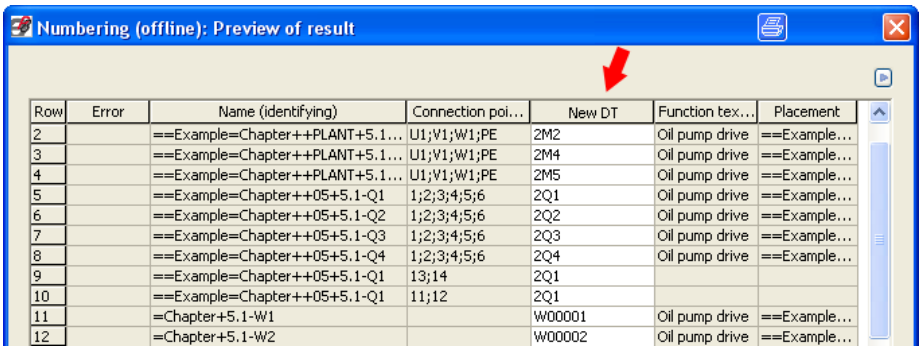


Fig. 5.45 Settings: Numbering (offline) dialog

After you have selected the numbering format, you click **OK**. EPLAN numbers all devices according to the specified format and displays this numbering in a *preview of result*.



Row	Error	Name (identifying)	Connection poi...	New DT	Function tex...	Placement
2		==Example=Chapter++PLANT+5.1...	U1;V1;W1;PE	2M2	Oil pump drive	==Example...
3		==Example=Chapter++PLANT+5.1...	U1;V1;W1;PE	2M4	Oil pump drive	==Example...
4		==Example=Chapter++PLANT+5.1...	U1;V1;W1;PE	2M5	Oil pump drive	==Example...
5		==Example=Chapter++05+5.1-Q1	1;2;3;4;5;6	2Q1	Oil pump drive	==Example...
6		==Example=Chapter++05+5.1-Q2	1;2;3;4;5;6	2Q2	Oil pump drive	==Example...
7		==Example=Chapter++05+5.1-Q3	1;2;3;4;5;6	2Q3	Oil pump drive	==Example...
8		==Example=Chapter++05+5.1-Q4	1;2;3;4;5;6	2Q4	Oil pump drive	==Example...
9		==Example=Chapter++05+5.1-Q1	13;14	2Q1		
10		==Example=Chapter++05+5.1-Q1	11;12	2Q1		
11		=Chapter+5.1-W1		W00001	Oil pump drive	==Example...
12		=Chapter+5.1-W2		W00002	Oil pump drive	==Example...

Fig. 5.46 Preview of result dialog

If the **PREVIEW OF RESULT** (if used) is confirmed, then EPLAN writes the modified device tags back to the schematic.

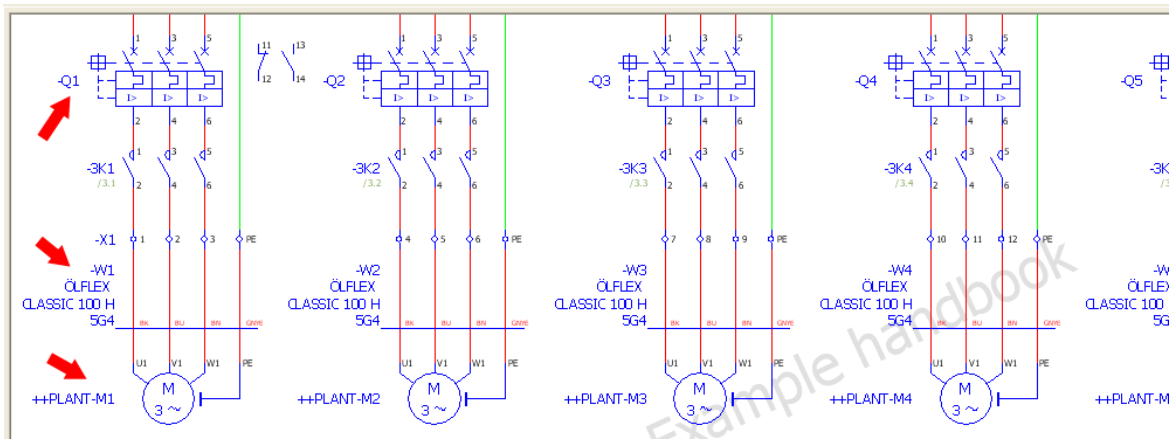


Fig. 5.47 Before numbering

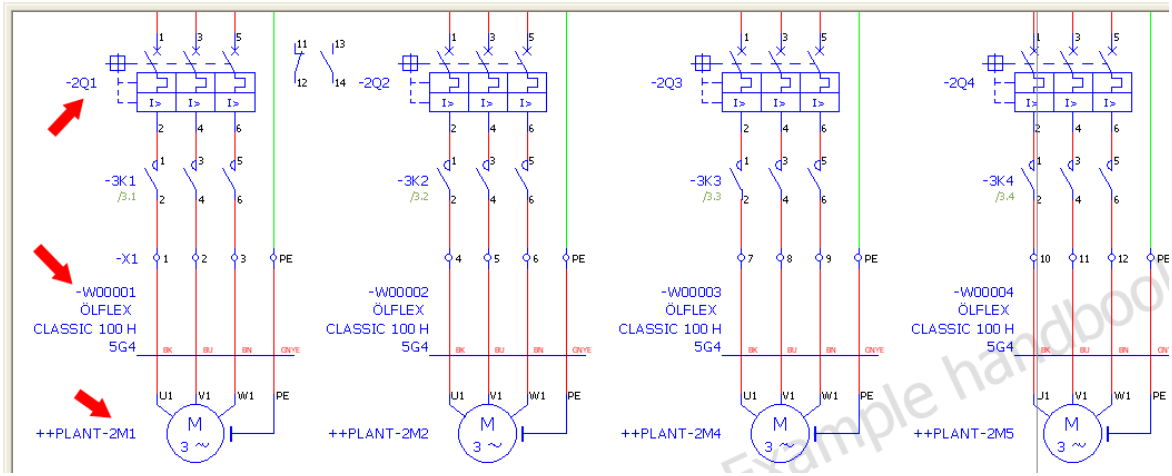


Fig. 5.48 After numbering

In this example, it is obvious that EPLAN only numbers devices that are also a main function. In this example that would be everything except the coil contacts. These were not included in the numbering dialog.

This is why it is important to know that although you can number or re-number devices very quickly by selecting them on a page, these devices must be main functions. If you want to avoid this, you can perform numbering via the device navigator. Select the devices, right click to open the popup menu and select the **NUMBER DT** menu item.

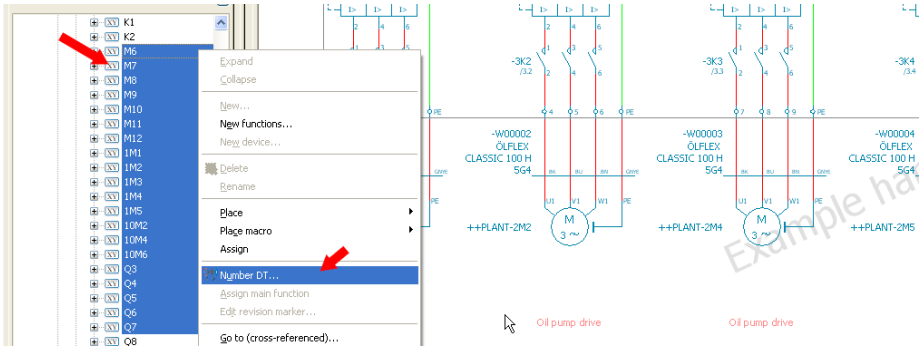


Fig. 5.49 Number DT in the popup menu

Then EPLAN restarts the **NUMBER (OFFLINE)** dialog, and numbering can proceed from here as described above. This time, however, all functions are included because the contacts *and* the coil have been selected (in the device navigator).

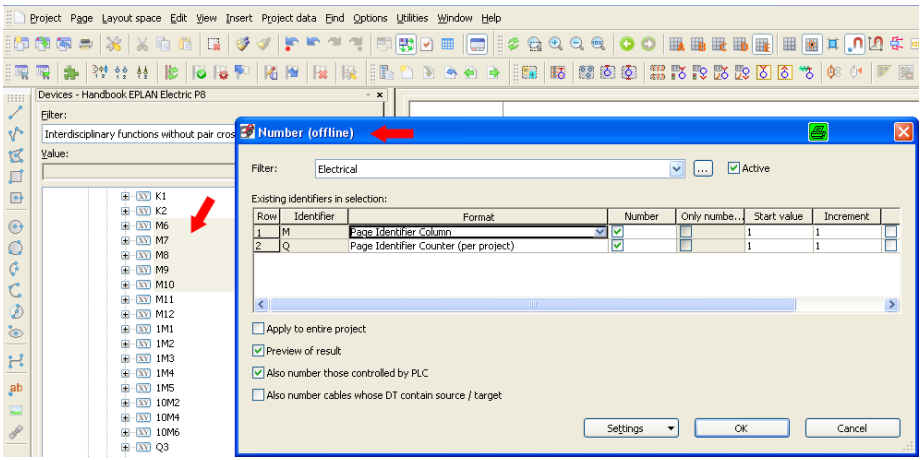


Fig. 5.50 Number (offline) dialog in the device navigator

5.2.6 Device navigator/New ...

The **NEW...** function inserts a new function definition, for example an additional auxiliary NC contact for a motor overload switch.

First you select the motor overload switch in the device navigator. Then you open the popup menu in the device navigator and select the **NEW...** function. EPLAN opens the **FUNCTION DEFINITIONS** dialog.

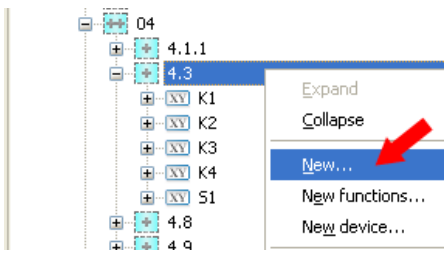


Fig. 5.51 New... menu item

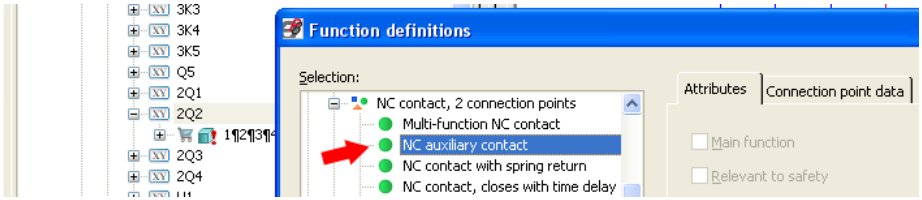


Fig. 5.52 Function definitions dialog

In this dialog, you select the *NC auxiliary contact* function definition and click **OK** to apply it. In the subsequent **PROPERTIES (COMPONENTS)** dialog, EPLAN asks for the *device tag* and other information.

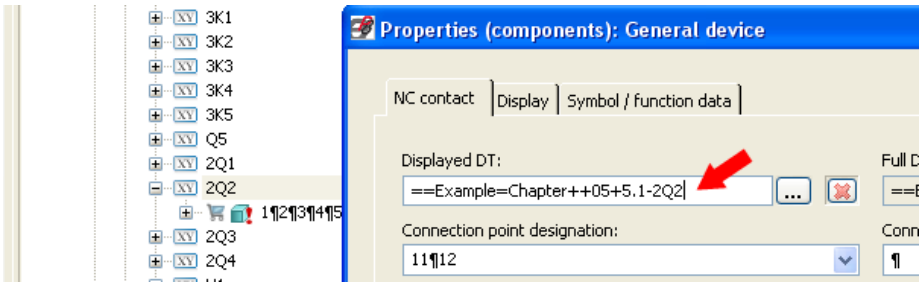


Fig. 5.53 Properties (components) dialog

When you click **OK** to exit the dialog, the *NC auxiliary contact* is assigned to the selected motor overload switch in the device navigator.

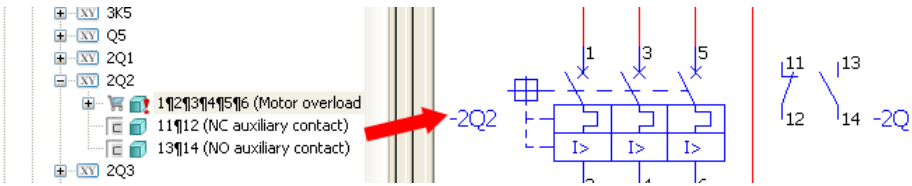


Fig. 5.54 New function on motor overload switch

5.2.7 Device navigator/New device ...

To insert a new device in the device navigator you select the **NEW DEVICE ...** function in the popup menu of the device navigator.

When the **NEW DEVICE ...** function is selected, EPLAN starts *parts management*. You select the relevant device and click **OK** to apply it.

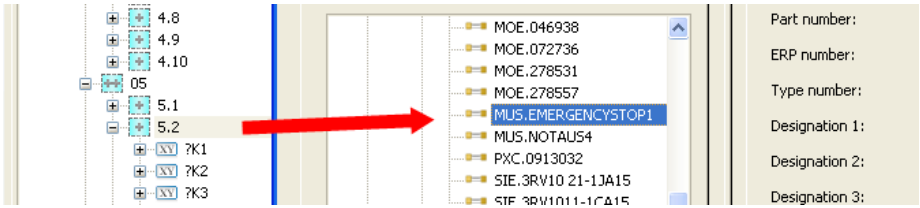


Fig. 5.55 Select device

EPLAN sorts the device according to the *identifier* into the device navigator. It can still be edited. Open the popup menu and select the **PROPERTIES** function, which then opens the **PROPERTIES (COMPONENTS)** dialog.

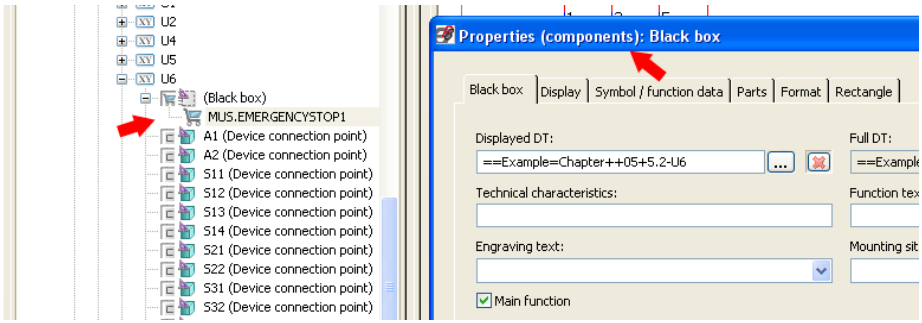


Fig. 5.56 Subsequent device editing

Only new devices with at least one of the following properties (macro, function template, symbol file or symbol number) can be assigned in the device navigator. If this is not the case, then EPLAN displays an appropriate message.

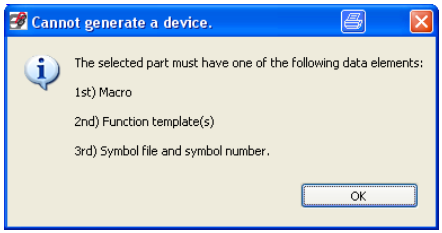


Fig. 5.57
Prompt if a device cannot be generated

5.2.8 Device navigator/Place/[Representation type]

In the last example in section 5.2.7, a device was inserted into the navigator. It was inserted in the navigator as an unplaced device. To put this device in the schematic (i. e. to place it), you select the **PLACE** function and **MULTI-LINE** in the popup menu of the **device navigator**.

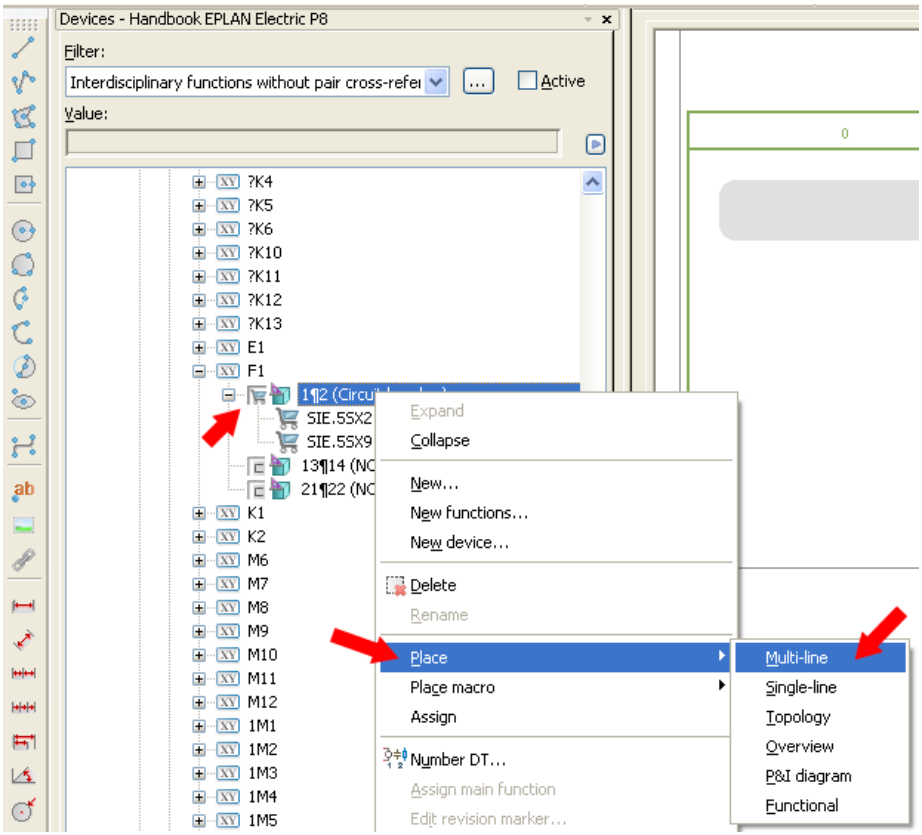


Fig. 5.58 Placing a device

The device then "hangs" on the cursor and can be placed on the schematic page. It is also possible to drag more than one device onto the page. Multiple devices can be marked in the device navigator and placed on the page. EPLAN then places an item every time the mouse is clicked.

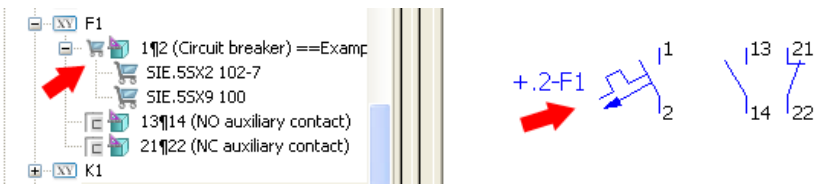


Fig. 5.59 Placed device

The major advantage of using the device navigator for placement is that EPLAN updates and enters all important information and data. This means that you won't have any incorrect DTs or missing technical characteristics on the device because EPLAN enters the correct (existing) information from the navigator.

5.3 Terminal strip navigator

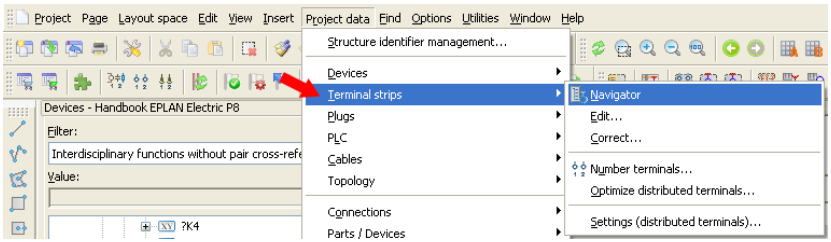


Fig. 5.60
Terminal strip menu

There is not much to say about terminal strips. The **terminal strip navigator** helps the user perform many repetitive actions, for example numbering terminals.

Besides the actual terminal strip navigator, the **PROJECT DATA/TERMINAL STRIPS** menu contains three additional functions. These can be executed directly on the selected elements on a page or on selected pages in the **page navigator**. The **terminal strip navigator** also has a popup menu, accessible by right-clicking the mouse or clicking the button.

5.3.1 Terminal strip navigator/Edit ...

The **EDIT** function in the **PROJECT DATA/TERMINAL STRIPS** menu opens the **EDIT TERMINAL STRIPS** dialog. This dialog contains different functions from the popup menu for editing terminals. Examples of these functions are *Number terminals*, *Move*, *Add strip accessories*, and delete or *Ungroup multi-level terminals*.

Like all EPLAN menus, the popup menu in the terminal editor can be called up via the right mouse button or the button.

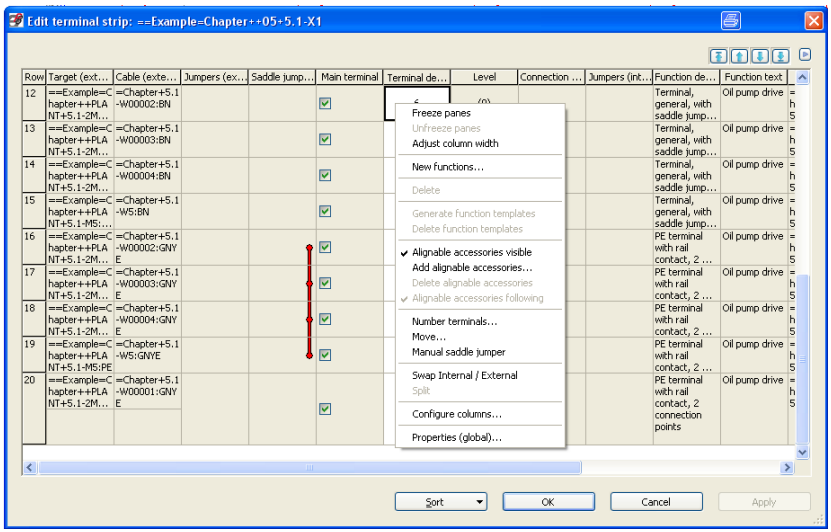


Fig. 5.61
Editing terminal strips dialog

Additional functions for editing terminals are located in the lower area of the dialog after clicking the **SORT** button.

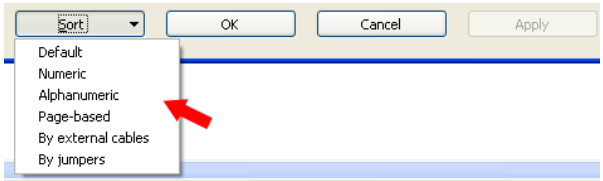


Fig. 5.62
Sort button and its functions

The **SORT** button opens another menu containing functions for sorting terminals. The **DEFAULT** function, for example, restores the existing sorting of the terminals back to the default. The **NUMERIC** function sorts the terminals by number, and the **ALPHANUMERIC** function sorts terminals by the alphanumeric components in the terminal designation.

The other buttons, **OK** and **CANCEL**, speak for themselves. The **APPLY** button applies the changes in the **EDIT TERMINAL STRIPS** dialog, but does not close the dialog.

To edit a terminal strip using the terminal editor, you first select the terminal strip in the schematic or in the terminal strip navigator (you can also just select one terminal in the terminal strip to be edited) and then select the **EDIT** function in the **PROJECT DATA/ TERMINAL STRIPS** menu.

TIP: To change the row height in the **EDIT TERMINAL STRIPS** dialog, position the cursor on a row and then pull the row while holding down the left mouse button to make it smaller or larger

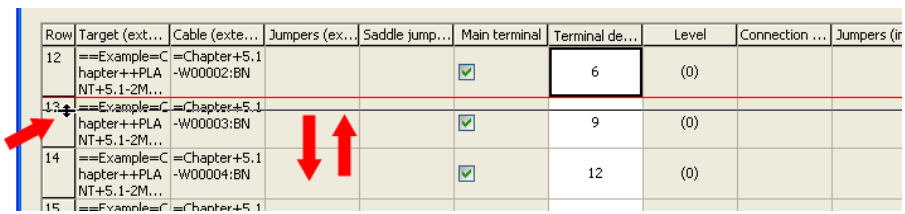


Fig. 5.63 Changing the row height

EPLAN opens the **EDIT TERMINAL STRIPS** dialog. The terminals are sorted here according to the standard. Click the **SORT** button and select the **STANDARD** item. EPLAN immediately sorts the terminal strip according to the standard.

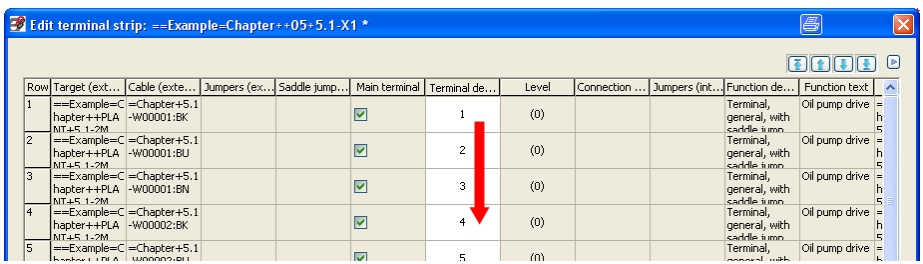


Fig. 5.64 Resorted terminal strip

For example, to change the sequence of terminals 1, 2 and 3 (from 1-2-3 to 3-2-1), you select each of the terminals and use the buttons or the associated keyboard shortcuts to move them to the desired position.

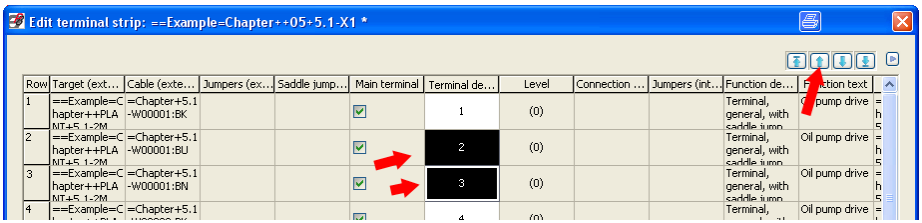


Fig. 5.65 Manually moving a terminal

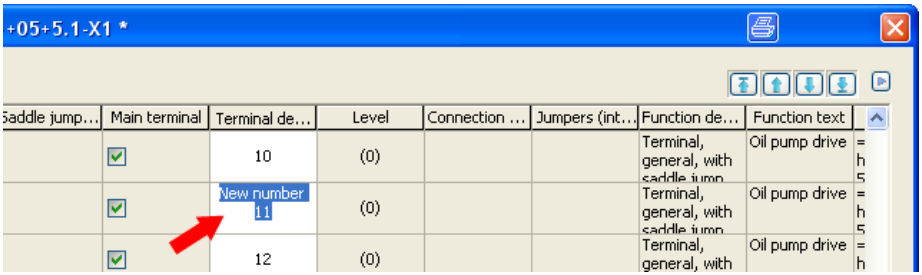


Fig. 5.66 Renaming a terminal designation

To give one or more terminals another terminal designation, you need only edit the **terminal designation** field directly. EPLAN writes the changed terminal designation to the corresponding terminals immediately after you save the changes and exit the dialog.

Terminals can also be numbered in the dialog. Simply select the desired terminals with the mouse, open the popup menu and select the **NUMBER TERMINALS** menu item. EPLAN then starts the **NUMBER TERMINALS** dialog where the settings are made. EPLAN can then number the terminals.

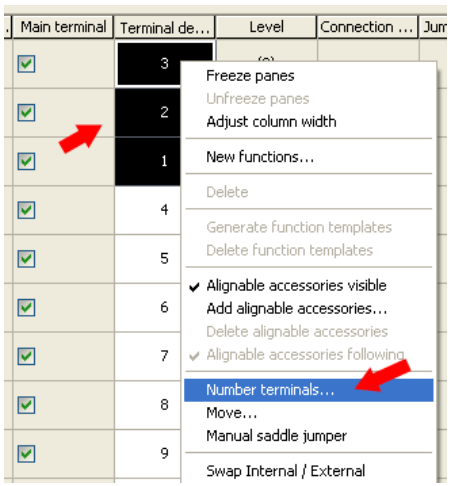


Fig. 5.67
Number terminals

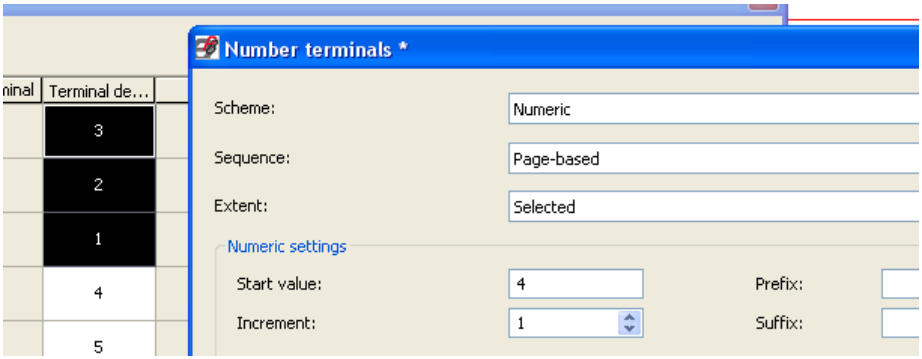


Fig. 5.68 Number terminals dialog

5.3.2 Terminal strip navigator/Correct

Another important function in the **PROJECT DATA/TERMINAL STRIPS** menu is the **CORRECT** function. At least one terminal or terminal strip must be selected before this function can be called. When the **CORRECT** function is called, EPLAN opens the **CORRECT TERMINAL STRIPS** dialog.

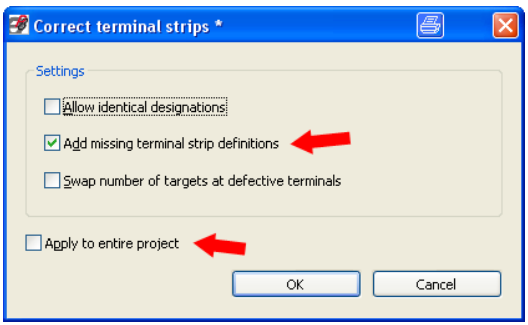


Fig. 5.69 Correct terminal strips dialog

The dialog offers four settings. The *Allow identical designations* setting means that terminals with the same terminal number are not logged as errors and also sets the **Allow multiple entries<20811>** property for all terminals with the same terminal number.

The *Swap number of targets at defective terminals* setting means that EPLAN corrects terminals with a different number of targets.

You can also use the *Add missing terminal strip definitions* setting so that EPLAN automatically generates all missing terminal strip definitions and adds them to the unplaced terminal strip functions in the terminal navigator.

What is a terminal strip definition, and why should you use them? Only through generated terminal strip definitions is it possible to, for example, exclude terminal strips from various reports (**properties <20851, 20853 and 20857> Do not assign an output** ...), assign a descriptive terminal strip definition text (**Function text** field) to terminal strips, or graphically output terminal strips using a special form field (**Terminal diagram and/or Connection-point diagram form**).

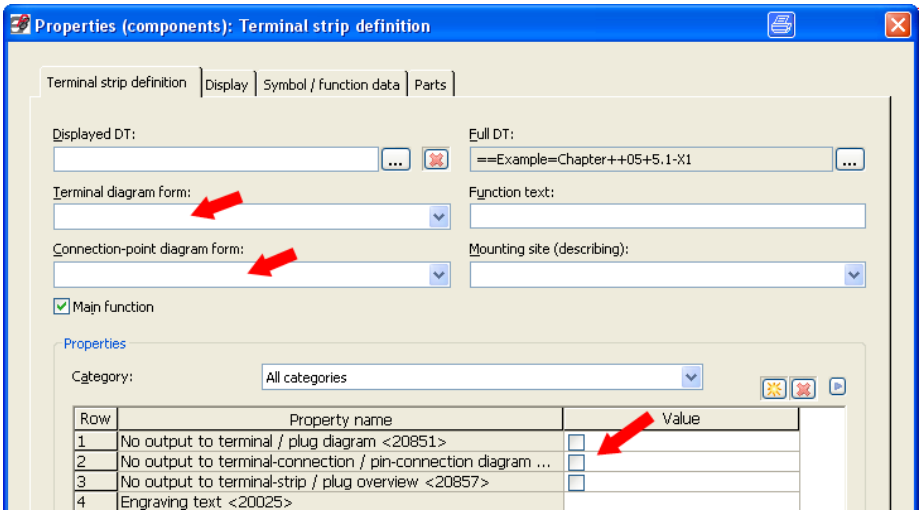


Fig. 5.70 Terminal strip definition and its properties

This information can only be entered when a terminal strip definition has been created. This information cannot be entered or stored in the properties of a terminal.

If the *Add missing terminal strip definitions* check box is set, and the *Apply to entire project* setting is set, and the dialog is confirmed by clicking the **OK** button, then EPLAN generates a terminal strip definition for all terminal strips that do not already have one.

5.3.3 Terminal strip navigator/Number terminals

As a result of various project actions in EPLAN, such as copying or copying of pages from other projects, terminals can end up with the wrong or completely incorrect designations. To avoid having to manually renumber all these terminals, the **PROJECT DATA/TERMINAL STRIPS** menu offers the **NUMBER TERMINALS** function.

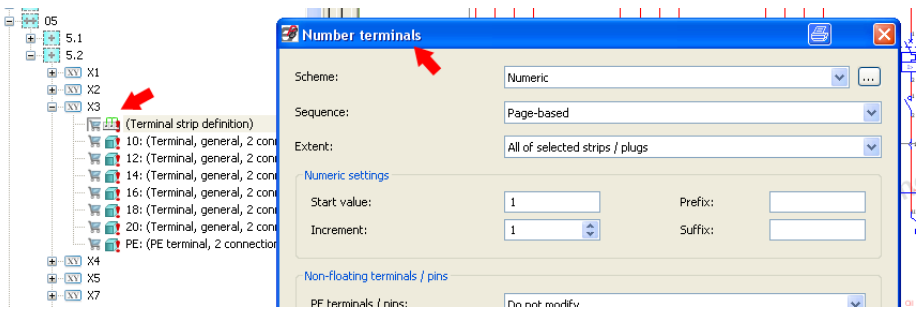


Fig. 5.71 Number terminals dialog

You can number terminals using more than a simple scheme such as numbering integer values from 1 to 10. EPLAN also allows you to define your own schemes, e. g. for initiator terminals.

The terminal strip to be renumbered is selected in the schematic (or in the terminal strip navigator). You only have to select one (any) terminal belonging to this terminal strip.

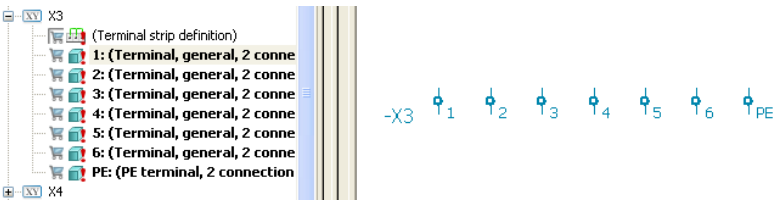


Fig. 5.72 Selecting terminals

You then start the **NUMBER TERMINALS** dialog via the **NUMBER TERMINALS** item in the **PROJECT DATA/TERMINAL STRIPS** menu.

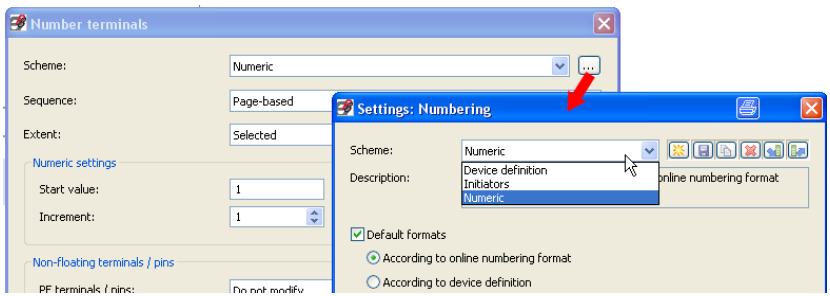


Fig. 5.73 Number terminals dialog with selection of numbering scheme

In the **NUMBER TERMINALS** dialog, the desired scheme is now selected in the **Scheme** selection field; in the example, this is the *Numeric* scheme. The scheme can only be changed before numbering by using the … button or by generating a new scheme in the **SETTINGS: NUMBERING** dialog.

Before you start the actual numbering, you should check and adjust the other settings, such as the numeric settings, the range, and the handling of PE terminals.

When you click **OK** in the **NUMBER TERMINALS** dialog is clicked, EPLAN numbers the terminal strips as specified.

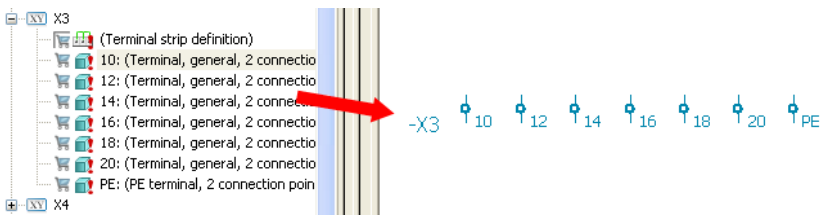


Fig. 5.74 Numbered terminals

This simple example is only meant to illustrate the process of using the **NUMBER TERMINAL STRIPS** function. More complex numbering tasks can be accomplished by making use of the many possible settings in terminal schemes.

5.3.3.1 Number terminals settings

In the **NUMBER TERMINALS** dialog, in addition to the basic numbering settings, there are other options, which are explained below.

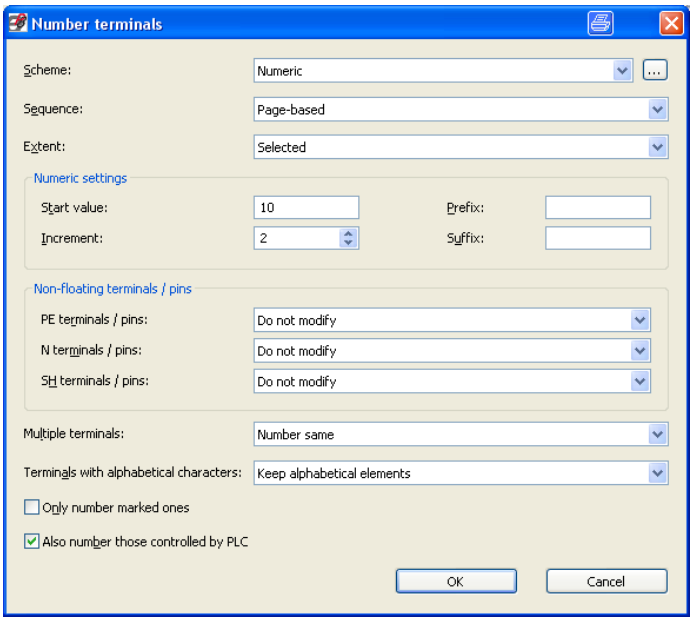


Fig. 5.75 Number terminals dialog

Scheme: The desired scheme can be set here from a selection list. You can add as many schemes as you want to this selection list.

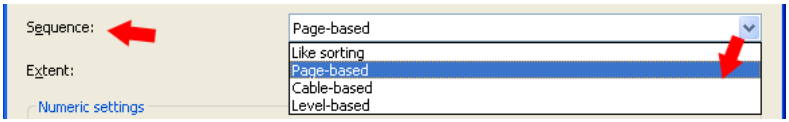


Fig. 5.76 Sequence setting

Sequence: The numbering sequence is set in this selection list. Should the sequence be oriented to pages, cables or levels, or should the existing sorting method be applied?

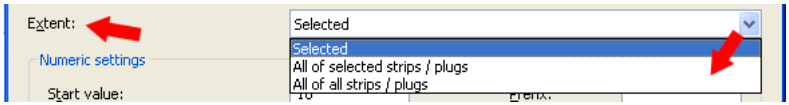


Fig. 5.77 Extent setting

Extent: This setting determines the range of the terminals to be numbered. What should be numbered – only selected terminals, the entire terminal strip, or all terminals in the project?

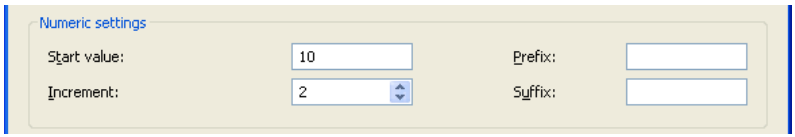


Fig. 5.78 Numeric settings

Numeric settings: This is where the start value, increment, prefix (a character preceding the terminal designation) and suffix (a character after the terminal designation) are set.

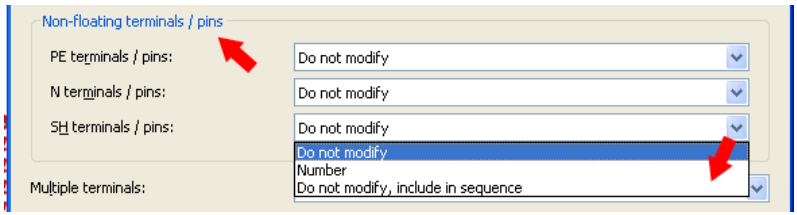


Fig. 5.79 Settings for certain terminal types

Non-floating terminals/pins: This is where settings for PE, N and SH terminals are set. Should they stay in the set sequence and not be changed, numbered or modified? The potential type of the terminals is crucial here!

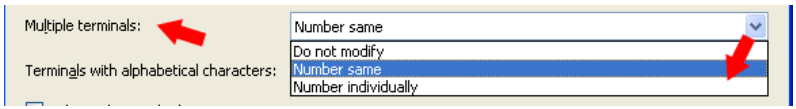


Fig. 5.80 Settings for multiple terminals

Multiple terminals: How should multiple terminals be handled during numbering operations? Should they be left unchanged, numbered all the same, or numbered individually?

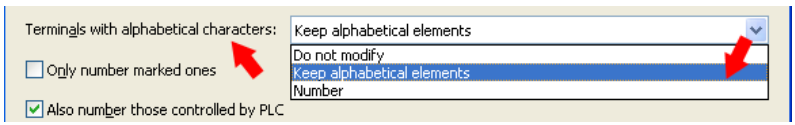


Fig. 5.81 Settings for alphabetically designated terminals

Alphabetically designated: This setting means that terminal designations that contain alphabetical components are taken into account. This setting determines whether these will be left unchanged and the components retained, or whether these will be completely renumbered. In the latter case, the alphabetical components would be lost.

In addition to these settings, there are two more options.

Only number marked ones: Only those terminals with the ? placeholder are numbered. The last setting, *Also number those controlled by PLC*, only numbers those terminals that are controlled by PLC.

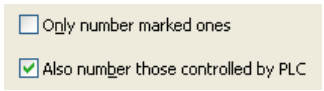


Fig. 5.82
Further options

5.3.4 Terminal strip navigator/New ...

The **NEW ...** function is basically the same as in the **device navigator**. In contrast to the device navigator, which can insert any function definition, only terminal or terminal strip function definitions can be inserted in the **terminal strip navigator**.

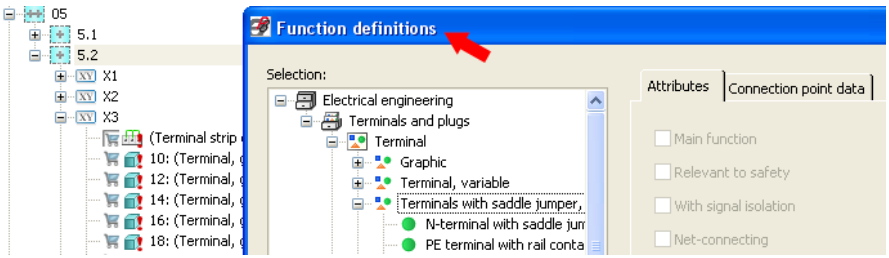


Fig. 5.83
Inserting new terminals

5.3.5 Terminal strip navigator/New functions ...

The **NEW FUNCTIONS ...** function in the **terminal navigator** allows fast generation of complete terminal strips using specific (own) numbering schemes.

The **NEW FUNCTIONS ...** function is called up in the **terminal strip navigator** popup menu. EPLAN opens the **GENERATE FUNCTIONS** dialog, which is initially empty, or a terminal strip designation is applied. The full DT is already displayed, depending on the current focus in the terminal strip navigator.

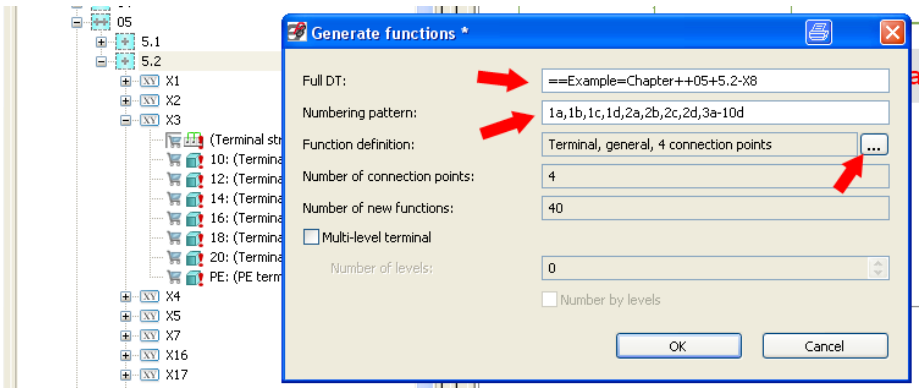


Fig. 5.84 Generate functions dialog

Here you should select/enter the full DT (the terminal strip designation), the numbering pattern and the setting defining whether or not these are multi-level terminals.

Generate functions

It is also possible to select a different function definition instead of the default terminal. Just click the ... button. EPLAN switches to the **FUNCTION DEFINITIONS** dialog where you can select a different terminal and apply it by clicking the **OK** button.

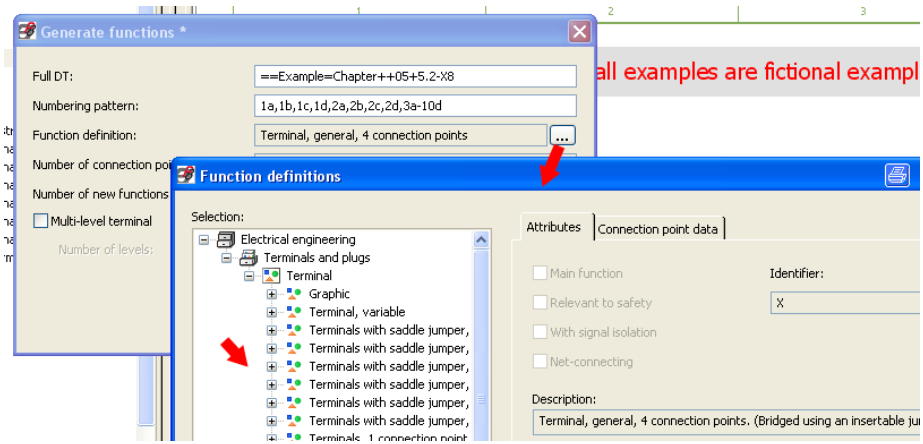


Fig. 5.85 Function definitions dialog

However, this is not a precondition. All function definitions can be easily changed later using block editing in the terminal strip navigator.

We return once more to generation of the functions: The functions can only be generated when all entered data is correct (EPLAN will enable the **OK** button). If an error exists in the numbering pattern, e. g. a missing comma, then the **OK** button is not enabled. You must check for this. When the **OK** button is clicked, EPLAN generates a terminal strip according to the specified options.

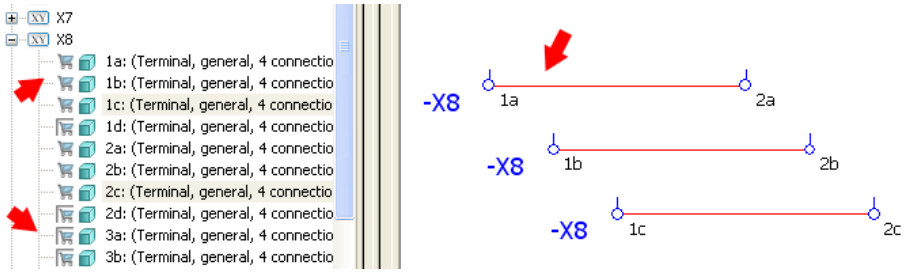


Fig. 5.86 New terminal strip with placed and unplaced terminals

NOTE: The newly created terminals do not have any parts data. This data must be assigned later using device selection.

5.3.6 Terminal strip navigator/New terminals (devices) ...

In addition to the easy generation of new terminal strips and new terminals, it is also possible in the terminal strip navigator popup menu to generate new terminals as devices on the basis of existing parts and their function templates.

Start the **NEW TERMINALS (DEVICES) ...** menu entry from the popup menu.

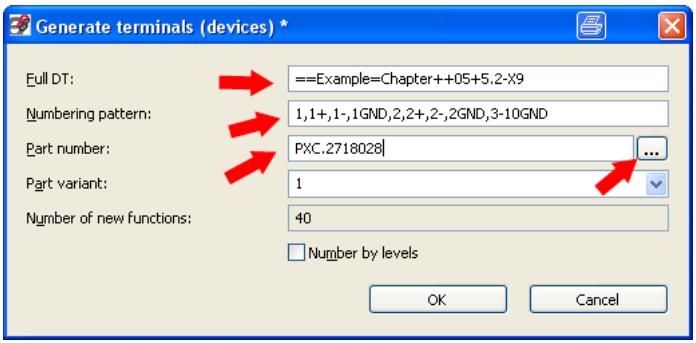


Fig. 5.87
Generate terminals (devices) dialog

EPLAN opens the **GENERATE TERMINALS (DEVICES)** dialog. The entries are similar to the previous section, You need to select a DT, a numbering pattern, and, most importantly, a part (the device) from parts management.

When all entries are correct, you can start device generation by clicking the **OK** button. EPLAN saves the functions in the navigator as unplaced functions.

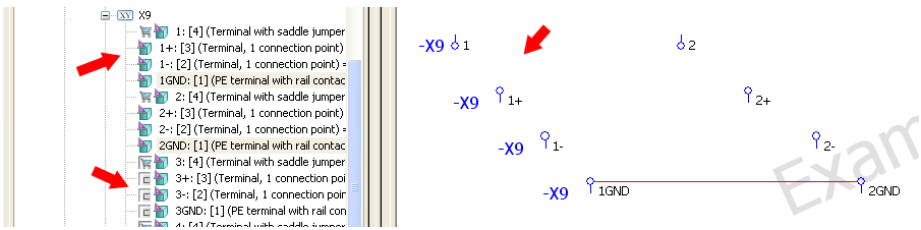


Fig. 5.88
Newly created terminals (devices)

In contrast to the last section, these new terminals are real devices. They already have parts data.

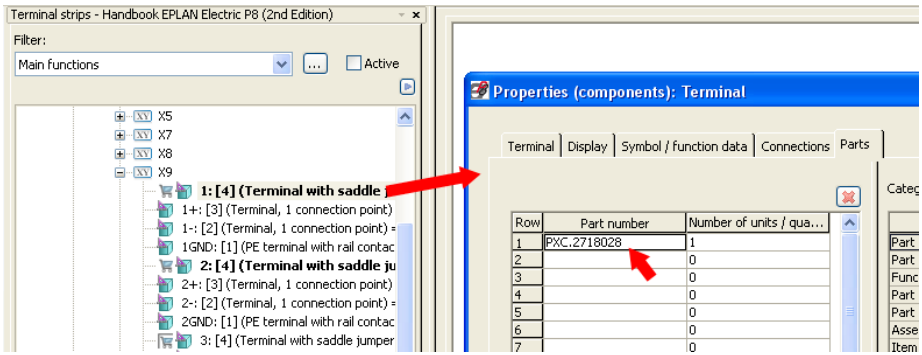


Fig. 5.89
Parts data for a terminal

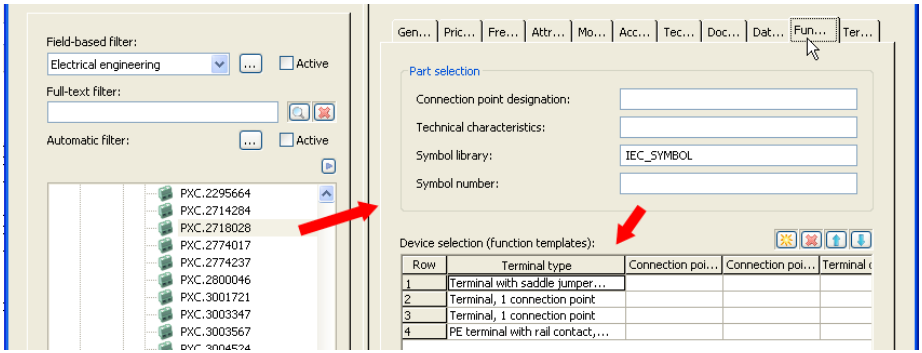


Fig. 5.90 Function template for a terminal part

5.3.6.1 Main terminals/Auxiliary terminals

It is important to know that, in addition to the generation of terminals or of terminals as devices, EPLAN has introduced new terminal functions and a new term in Version 2.0 – the **Main terminal**.

What is a main terminal? It is comparable to a main function; a terminal with multiple function definitions can only be identified once as a main terminal. All of this terminal's other functions cannot be designated as main functions. They are **Auxiliary functions**.

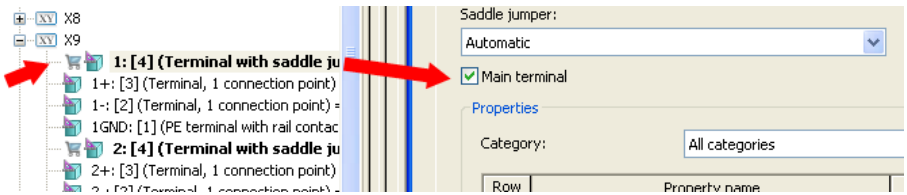


Fig. 5.91 Main terminal

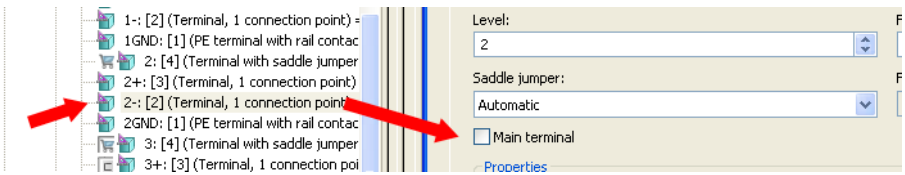


Fig. 5.92 Auxiliary terminal

Only main terminals can have *parts data*. These new main and auxiliary terminals are now handled like all other devices and can be used like other devices in EPLAN. You therefore have the option of using or placing some of this terminal's functions, e. g. as a grounding contact or a coil. Main and auxiliary terminals have the same relationship to each other. Device selection can be performed for these main terminals. The terminal strip definition, however, remains a terminal strip's **main function**.

5.3.7 Terminal strip navigator/Place/[Representation type]

If, for example, unplaced terminals are needed in the schematic, you can use the **PLACE** function in the terminal strip navigator to put them there.

You use the mouse or the keyboard (the mouse is easier) to select the desired terminals in the terminal strip navigator. You then select the **PLACE/MULTI-LINE** function (for a multi-line representation) from the right-click popup menu in the terminal strip navigator.

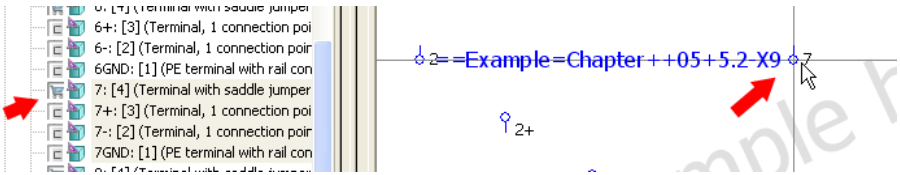


Fig. 5.93 Multi-line placement of terminals

EPLAN now "hangs" all the selected terminals on the cursor, and you can place them one at a time by clicking the left mouse button.

You can also hold down the left mouse button and (e.g.) drag the mouse to the right over the page. EPLAN then places multiple terminals in a single operation until you release the mouse button. Counter targets must be present, so that EPLAN can generate an automatic connection.

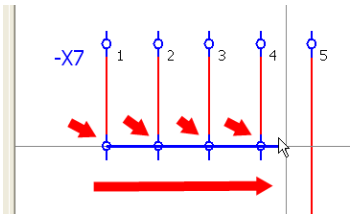


Fig. 5.94
Multiple placement

5.4 Plug navigator (Socket navigator)

In addition to terminals, there are also plugs and sockets. In principle, the **plug navigator** offers the same functions as those used for editing terminals and terminal strips. This section will therefore only cover the plug/socket functions that are different from the functions provided in the terminal strip navigator.

Plug is a generic term in EPLAN. Plugs are distinguished by male and female pins. Corresponding function definitions exist for these, and the various menu items are also labeled as such.

5.4.1 Plug navigator/Edit

In addition to the actual **plug navigator**, the **plug** project data also has a submenu with additional special functions. The **EDIT PLUG** function works in a way very similar to the **EDIT TERMINAL** function. In this case, you edit only male pins or female pins.

After you call up the menu, EPLAN displays the **EDIT PLUG** dialog. The functions available in the popup menu, such as *Number pins* or *Edit properties*, can be used here.

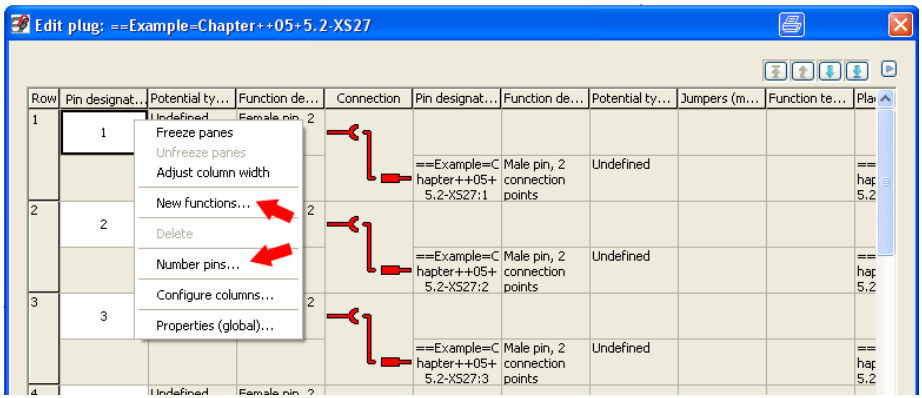


Fig. 5.95 Edit plug dialog

5.4.2 Plug navigator/Correct

Among other features, the **CORRECT** function in the **PROJECT DATA/PLUGS** menu creates correct plug definitions. Whereas terminals only have one definition, the terminal strip definition, plugs in EPLAN have three possible definitions.

There are *plug definitions* for female pins, male pins, and for a combination of male and female pins.

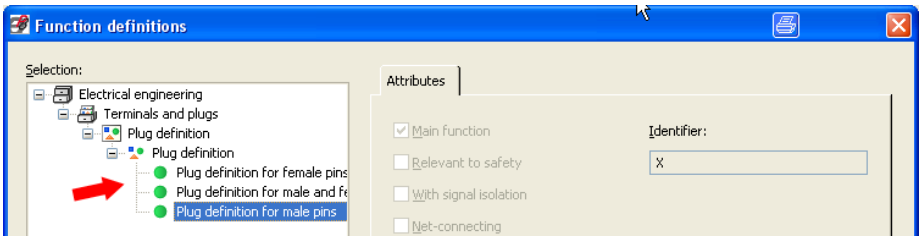


Fig. 5.96 Different types of plug definitions

It is very important that correct *plug definitions* that match the corresponding plug, socket, or plug strip are generated. If for example, a socket strip with an entered report form (different from the default plug diagram form so that it can later be manually placed on a schematic page) is assigned the wrong plug definition, then EPLAN will not generate a manually placed report for this socket strip.

NOTE: Since this is an important point, I would recommend that you always automatically generate plug definitions. EPLAN then automatically assigns the different functions to the correct plug definition since it determines the required information, such as the function definitions, from the project.

5.4.3 Plug navigator/Number pins

This function is identical to the **NUMBER TERMINALS FUNCTION** – with one exception. In the **NUMBER PINS** dialog, the **MULTIPLE TERMINALS** option is not possible for plugs.

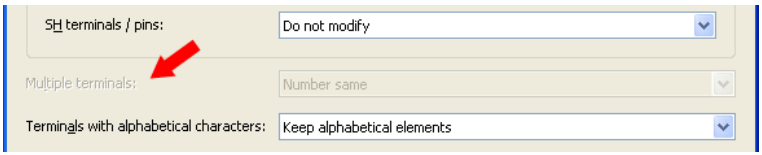


Fig. 5.97 Number plugs dialog

5.4.4 Plug navigator/New ...

The **NEW ...** menu item is in the **plug navigator** popup menu (accessible by right-clicking or clicking the button). The procedure here is identical to the **NEW ...** function in the **terminal navigator**. A new function definition can be selected here as well. In contrast to terminals, this is only possible for plugs and sockets.

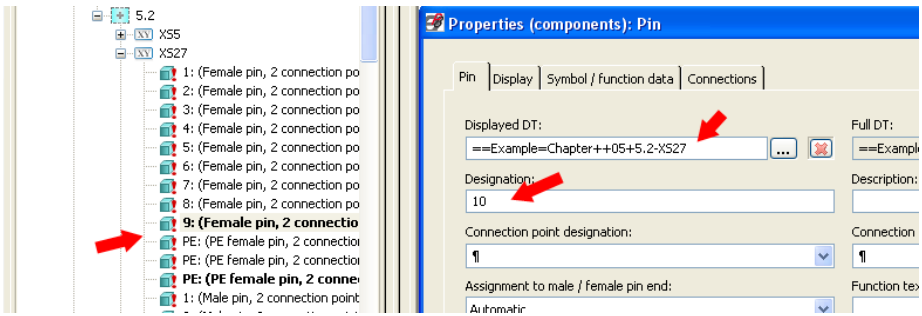


Fig. 5.98 Creating a new plug pin

5.4.5 Plug navigator/New functions ...

New plug strips are created using the popup menu item in the **plug navigator** in exactly the same way as in the **terminal strip navigator**. Whereas the terminal strip navigator creates new terminal strips, in the plug navigator you create new plugs, sockets, and plug/socket strips – depending on the function definition settings.

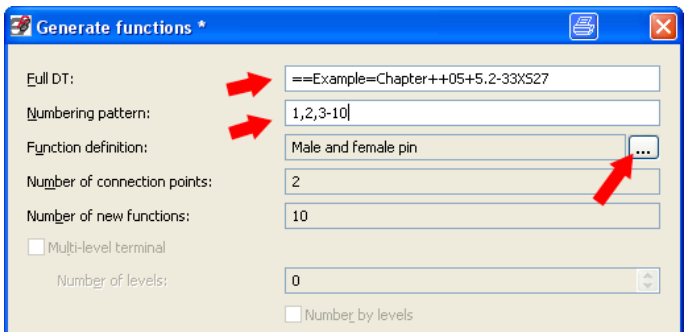


Fig. 5.99 Example: Generating new functions

In contrast to generating new terminal functions, here it is not possible to create multi-level terminals. The remaining procedures are the same as in the terminal strip navigator.

5.4.6 Plug navigator/Generate plug definition

As in the **terminal strip navigator**, you select new plugs in the **plug navigator** via the popup menu's **GENERATE PLUG DEFINITION**. Then you select **ONLY MALE PINS** or **ONLY FEMALE PINS** in the **MALE AND FEMALE PINS** field.

If, for example, *Only male pins* is selected, EPLAN displays the **PROPERTIES (COMPONENTS)/PLUG DEFINITION** dialog and enters the correct function definition. The remaining entries, such as device tag or form, can be entered and updated accordingly.

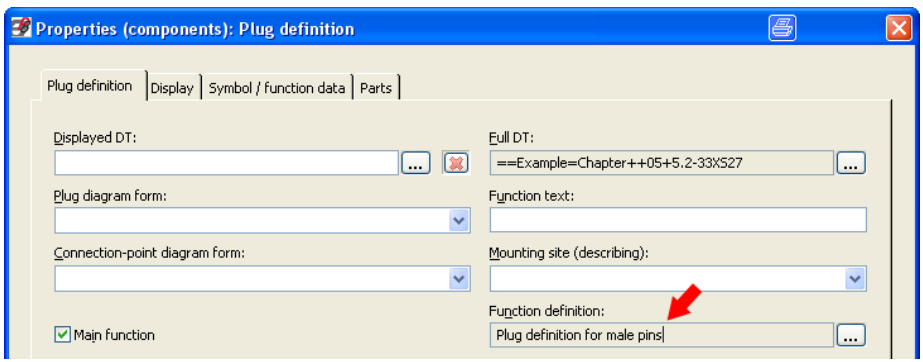


Fig. 5.100 New definition for male pins

5.4.7 Plug navigator/Generate pin

The **GENERATE PIN** function generates a new pin for the plug, socket or plug strip selected in the plug navigator.

You select the **GENERATE PIN** function from the popup menu in the plug navigator and then select, as in this example, the **MALE PIN** entry. EPLAN immediately opens the **PROPERTIES (COMPONENTS)/PIN** dialog. The device tag of the plug is automatically applied.

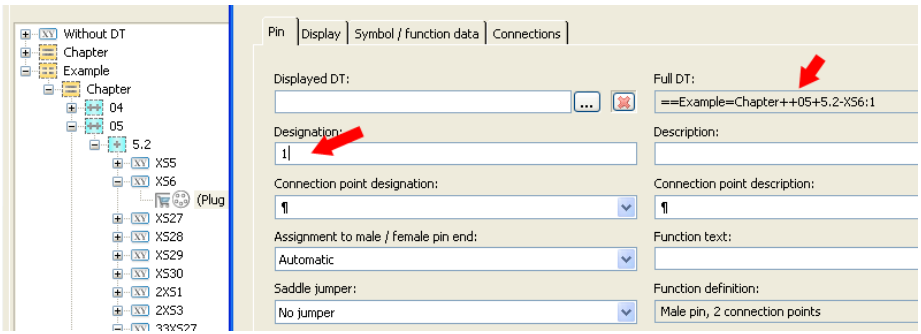


Fig. 5.101 Creating a male pin

You can use the ... button in the **Displayed DT** field to open **HELP** in the **DT SELECTION** dialog. All the plug's pins are listed here. A male pin can be designated and then assigned to the plug as an unplaced pin by clicking **OK**.

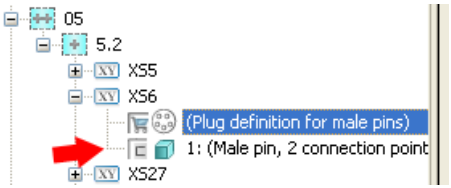


Fig. 5.102
Unplaced male pin

5.4.8 Plug navigator/Place/[Representation type]

In the same way as in the terminal strip navigator, to place unplaced plugs, sockets or combined plug/sockets in the schematic, you select the **PLACE** function in the popup menu and then select the representation type.

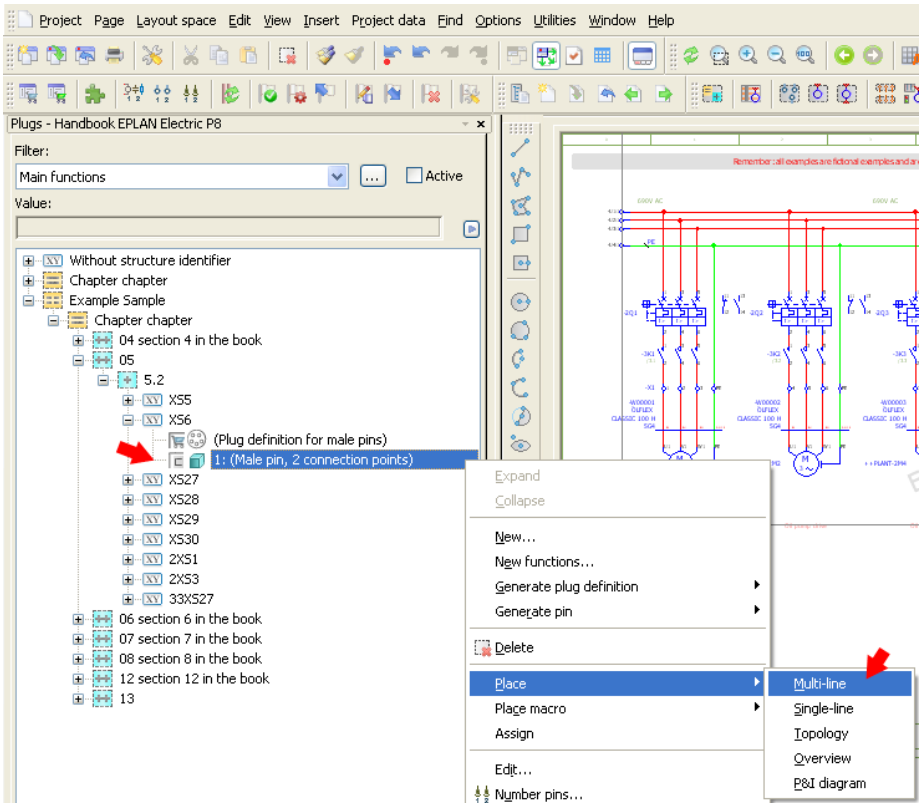


Fig. 5.103 Placing an unplaced male pin

This function is again identical to how the terminal strip navigator works.

5.5 Cable navigator

Cables and their conductors connect the individual devices. The **cable navigator** provides an overview of all used and unplaced cables in the project.

NOTE: To edit cables with all the functions provided by the cable navigator, such as automatic cable selection, the logical connection data must always be complete and up to date.

In most cases, EPLAN automatically performs **UPDATE CONNECTIONS** before such actions. But in the event that the connection data is not up to date, it is possible that subsequent actions will not work as desired or expected. This is why you should check the *status of connections* (up to date or not) in the status bar (symbol # or *), and update the connections yourself as required.

As I have mentioned before in this book, my personal recommendation is to assign the **UPDATE CONNECTIONS** function to the **F11** key. Then updating connections is only one keystroke away.

As with all the other navigators, the cable navigator also has a popup menu with additional functions for cable editing. As usual in EPLAN, the popup menu is called up via the right mouse button or the button.

5.5.1 Cable navigator/Edit

The **PROJECT DATA/CABLES** menu has an **EDIT** menu item. This function allows you to manually edit the assignments of the individual connections (which cable conductor – source/target) without requiring you to manually change the individual conductors of the cable in the schematic.

After various copy and delete actions, the assignment of conductors/terminals for a cable no longer reflects the desired assignments. To conveniently change the assignment, in the schematic you first select the cable to be changed (click the cable symbol) and then open the **EDIT** function in the **PROJECT DATA/CABLES** menu.

EPLAN opens the **EDIT CABLE** dialog. The left field lists the cable with its properties (stored **function templates** – conductor/wire). The cable's determined connections are shown in the right field.

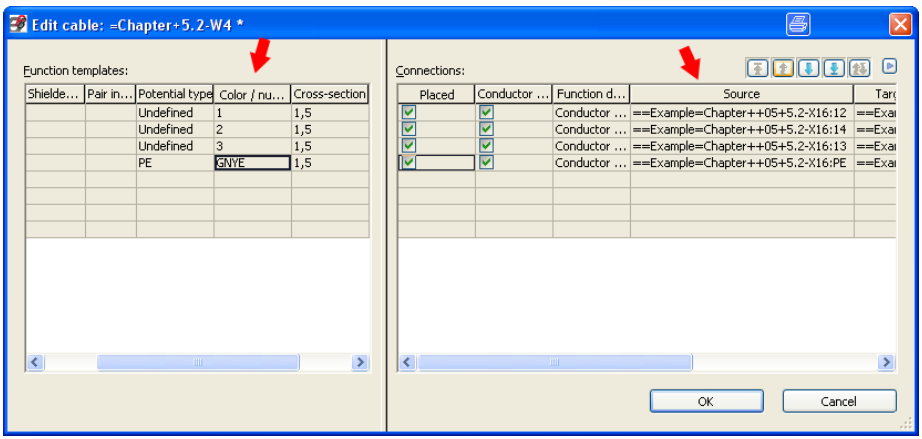


Fig. 5.104 Assignment of conductor to source and target DTs

The corresponding conductors can now be clicked in the **Connections** field and can then be moved using, for example, the buttons (move up or down) and assigned to the correct (desired) connections. You can also exchange connection places. Select the two connections that are to be exchanged and click the button. EPLAN then exchanges the assignments in the table.

The other "wrong" assignments can be corrected in the same way. After you finish changing the assignment, you exit the **EDIT CABLE** dialog. EPLAN then automatically updates the conductor – source/target assignments in the schematic, as can be seen Fig. 5.105 and Fig. 5.106.

Fig. 5.105 Conductor assignment before editing

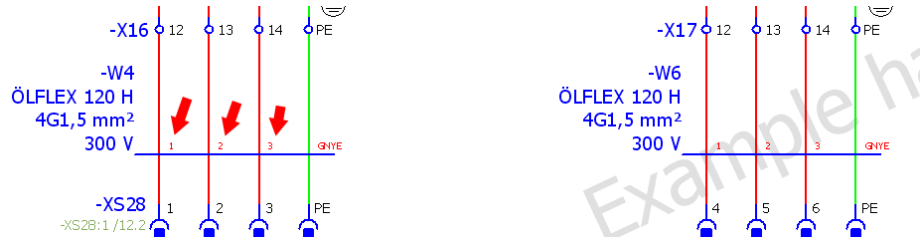


Fig. 5.106 Conductor assignment after exiting the Edit cable dialog

5.5.2 Cable navigator/Number

Schematics can be composed of many different macros or other project pages. The **cable DTs** can be very different. The **NUMBER** function in the **PROJECT DATA/CABLES** menu allows these to be consistently numbered across the entire project.

In the schematic or in the **cable navigator**, select the cables to be numbered.

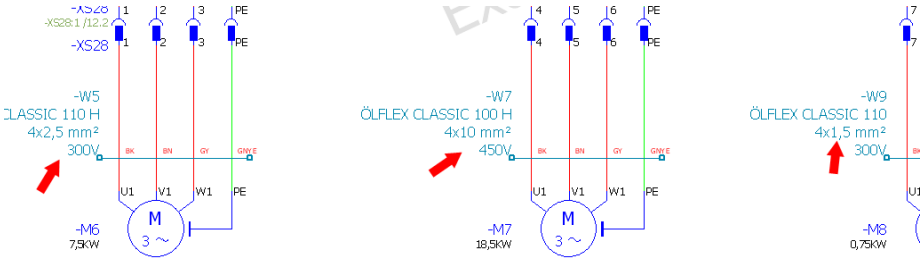


Fig. 5.107
Selected cables

Then call up the **NUMBER** function in the **PROJECT DATA/CABLES** menu. EPLAN opens the **NUMBER CABLES** dialog. The numbering options can be set in the *Settings* selection field.

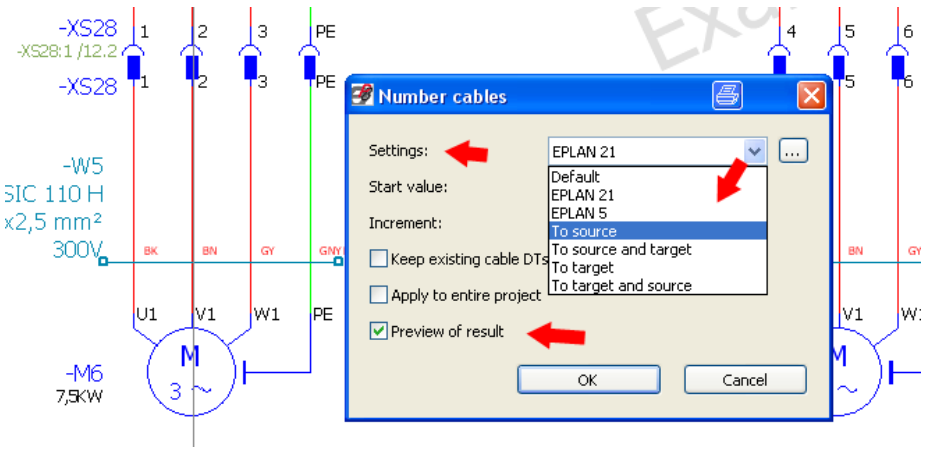


Fig. 5.108 Number cables dialog

Use the ... button to switch to the **SETTINGS/CABLE NUMBERING** dialog and create your own scheme for cable numbering, or modify and save an existing scheme.

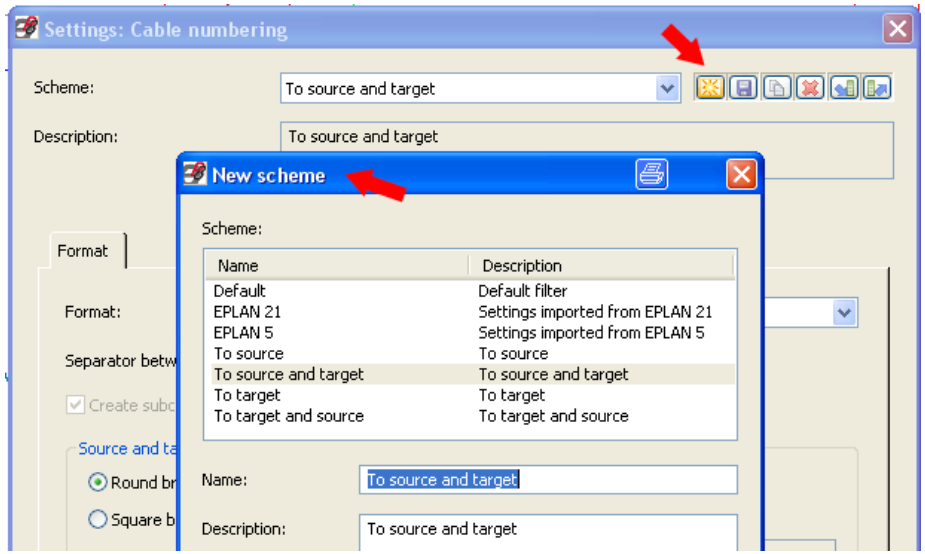


Fig. 5.109 Settings: Cable numbering dialog

In the example, the existing *Scheme: according to target* was selected in the **EDIT CABLE** dialog for numbering the selected cables. Here too it is a good idea to activate the *Preview of result* parameter. This allows the new cable DTs to be checked before they are written

back to the schematic, and if necessary, the numbering can be stopped via the **CANCEL** button.

After you click **OK**, EPLAN renumbers the **cable DTs** according to the selected scheme and displays the result in the **NUMBER CABLES: PREVIEW OF RESULT** dialog.

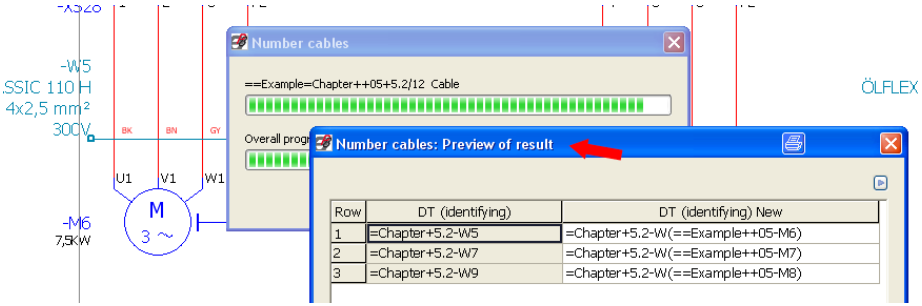


Fig. 5.110 Preview of result dialog

If the result is as desired, you can exit the **NUMBER CABLES: PREVIEW OF RESULT** dialog by clicking **OK**. EPLAN finishes the numbering process by writing back the new cable device tags.

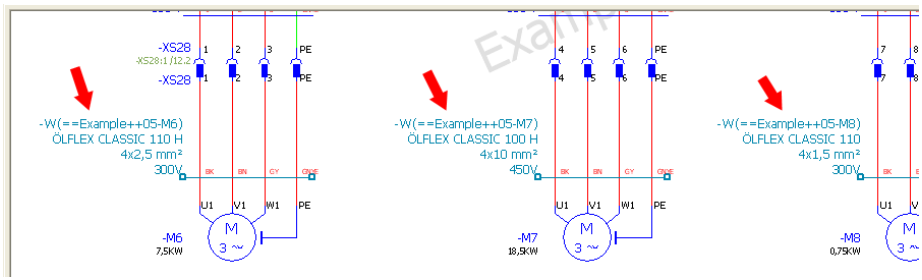


Fig. 5.111 Changed cable device tags

The default settings for the schemes for the different cable editing functions, such as the **Numbering function**, are defined in the project settings. The settings can be found under **OPTIONS/SETTINGS/PROJECT [PROJECT NAME]/DEVICES/CABLE (AUTOMATIC)**.

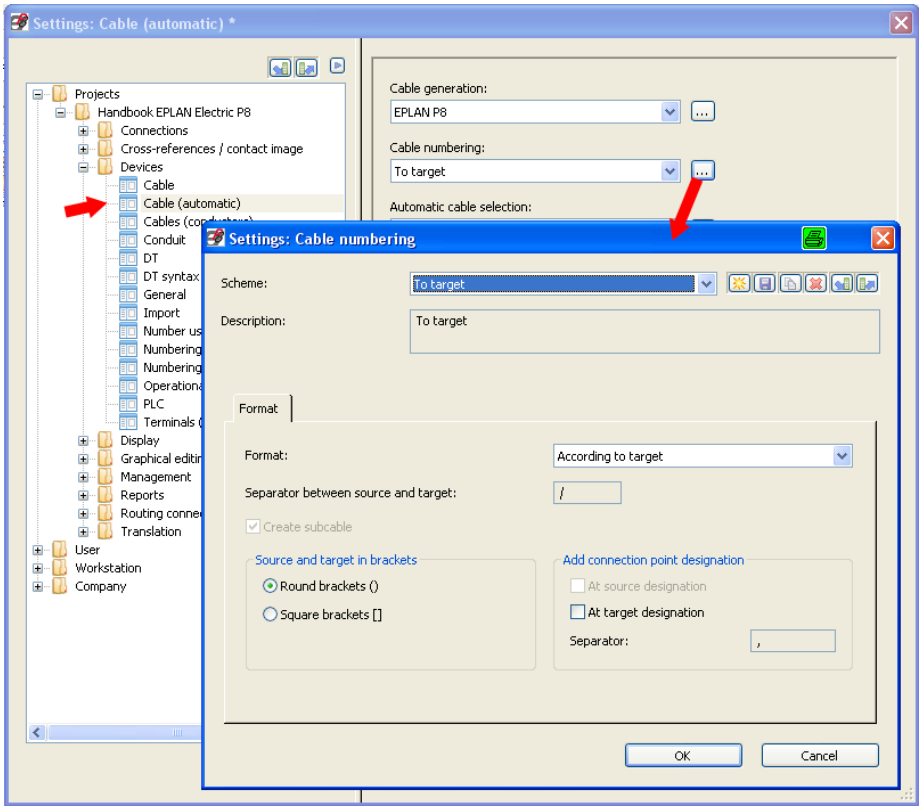


Fig. 5.112 Default settings for cable editing functions

5.5.3 Cable navigator/Generate cables automatically

EPLAN can automatically generate cables and their associated functions such as cable type etc. and thus complete schematics for you.

Preconditions: The cable for which automatic generation is intended already has a cable definition line entered at the connection points in the schematic. The cable device tag is adopted exactly from the online numbering. The sequence of the cable device tags plays a subordinate role here. A subordinate role means that the format of the cable device tag can still be defined retroactively in the settings for automatic cable generation.

Before calling up the GENERATE CABLES AUTOMATICALLY menu item, you first have to select the corresponding schematic pages in the **page navigator** or the individual cables in the schematic that are to be automatically generated.

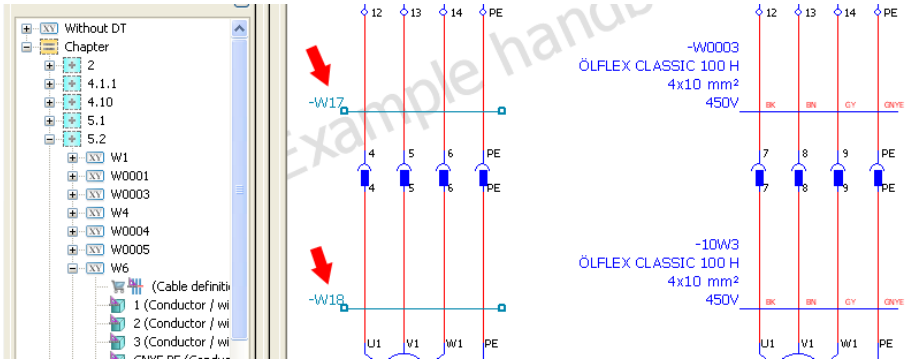


Fig. 5.113 Selecting existing cables

The **GENERATE CABLES AUTOMATICALLY** function in the **PROJECT DATA/CABLES** menu is then called up.

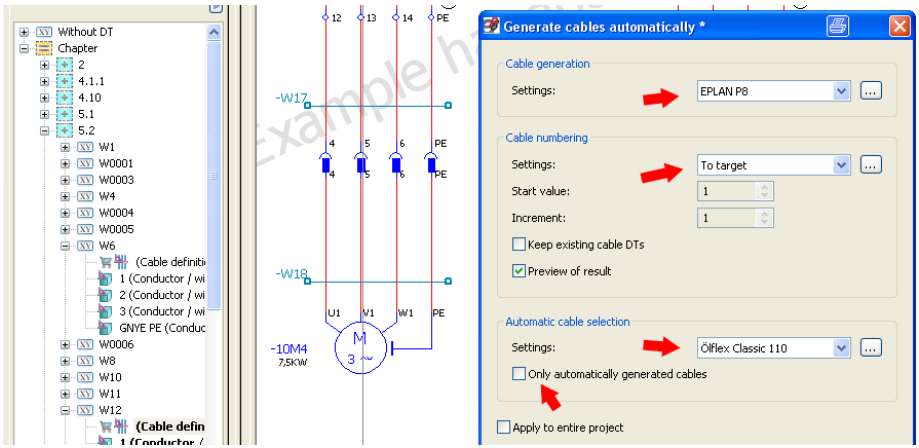


Fig. 5.114 Place cables automatically dialog, with its settings

EPLAN opens this dialog and applies the project settings for automatic cable generation. You can still change all the settings for this particular automatic generation of cables. Use the ... button to open the next settings dialog. In this dialog, you can create new schemes or modify and save existing schemes.

It is also a good idea to activate the *Preview of result* setting when using this function. This allows you to cancel the operation before actually generating the cables.

NOTE: The *Automatic cable selection* and *Only automatically generated cables* settings must be deselected if they do not exist in the cable main function. Otherwise, EPLAN will search for the **Automatically generated <20059>** cable property and, if this isn't activated, will not automatically update the cable data, such as cable type, etc.

Once you have correctly entered all the settings in the **NUMBER CABLES AUTOMATICALLY** dialog, you click **OK**. EPLAN then generates the cables and displays the future result in the **PREVIEW OF RESULT** dialog.

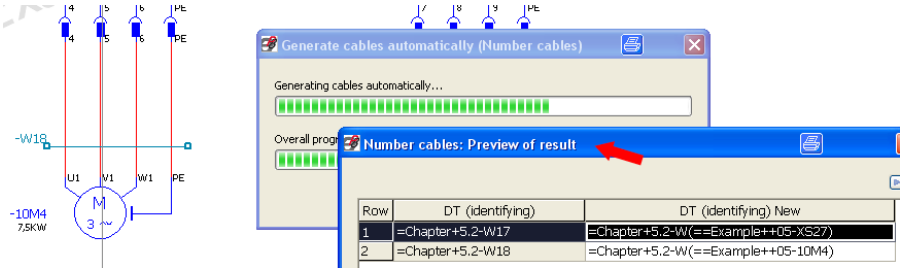


Fig. 5.115 Preview of result

As in other Preview of result dialogs, you can use the popup menu or the button to call the **USE ORIGINAL DT** function. You select the device tag and restore it back to the old DT. It is also possible to manually change the identifier directly in the **DT (identifying) New** field of the preview dialog.

Once all settings are correct, you can leave the **PREVIEW OF RESULT** dialog by clicking **OK**. EPLAN writes the modified cable data back to the selected cable definition lines in the project.

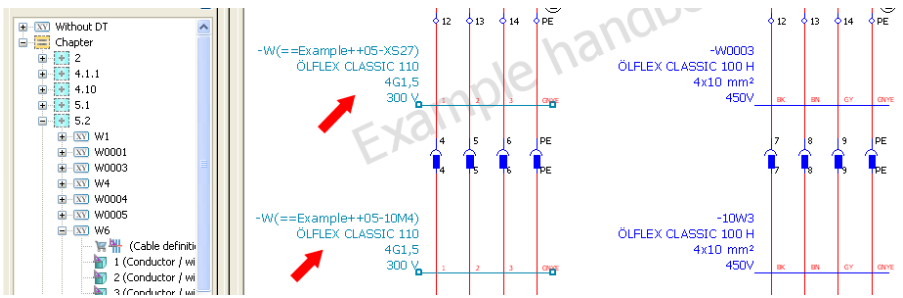


Fig. 5.116 Automatically generated cable data

5.5.4 Cable navigator/Automatic cable selection

The **AUTOMATIC CABLE SELECTION** dialog allows you to add a specified cable type to existing cables in the schematic.

To automatically assign cables, in the schematic you again select the cables to be modified and then call up the **AUTOMATIC CABLE SELECTION** function in the **PROJECT DATA/CABLES** menu. EPLAN opens the **AUTOMATIC CABLE SELECTION** dialog.

The dialog offers three settings:

- A choice of which cable should be used (an existing scheme can be used or you can create a new one by clicking the ... button and then selecting it)

- Which cables should be used the automatic cable selection
- Whether or not the automatic cable selection should be applied to the entire project

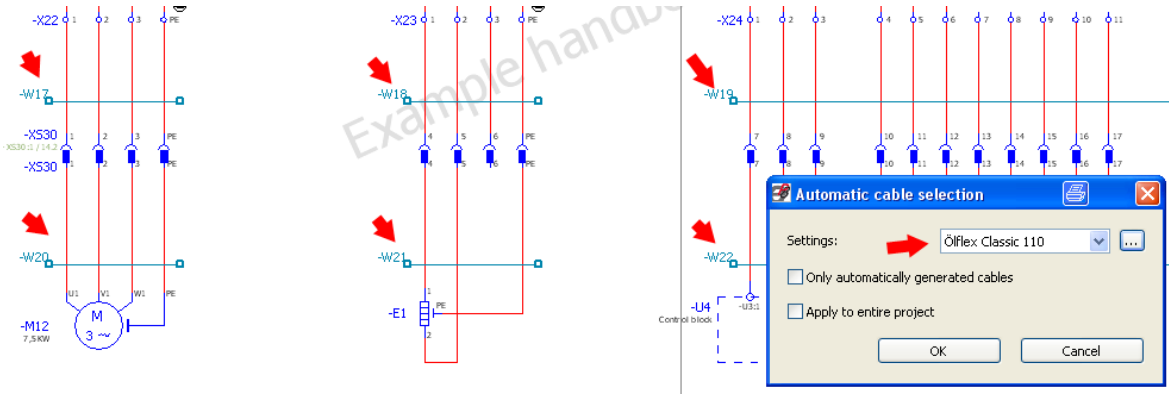


Fig. 5.117 Automatic cable selection dialog

The *Scheme for cable types* (settings) can contain multiple cable types. EPLAN automatically determines the required number of conductors and applies this cable to the selected cable DT in the schematic.

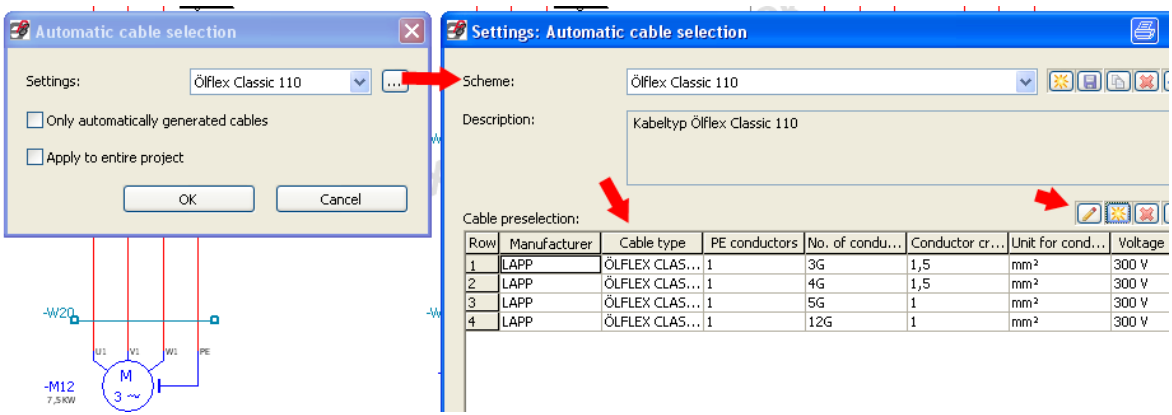


Fig. 5.118 Automatic cable (types) selection settings dialog

Once all settings in the **AUTOMATIC CABLE SELECTION** dialog have been correctly set, you click **OK** to start automatic cable selection. EPLAN compares the existing cable and connection data with the entered default cable types and assigns the new cable type(s) to the connections. After this, the cable data is also transferred to the cable definition lines and/or updated, depending on the settings.

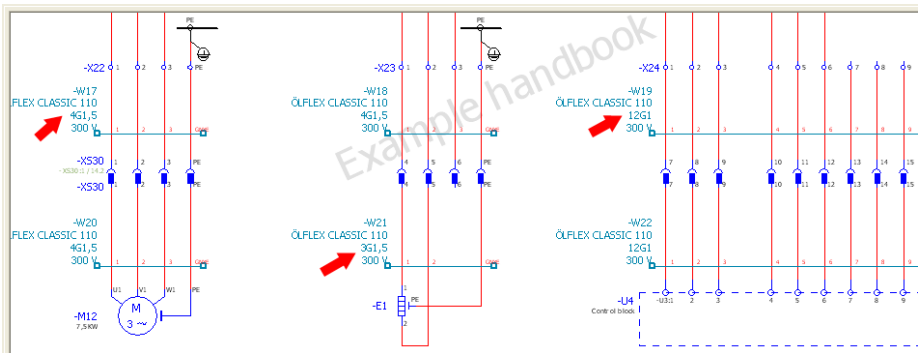


Fig. 5.119
Automatically generated cable data

5.5.5 Cable navigator/Assign conductors

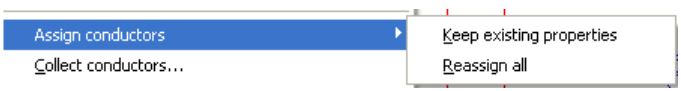


Fig. 5.120
Assign conductors menu item in the Cables menu

In the **PROJECT DATA/CABLES/ASSIGN CONDUCTORS** menu item, there are two more menu items, the importance and effects of which will be explained in the following sections.

5.5.5.1 Keep existing properties

The **KEEP EXISTING PROPERTIES** menu item enables the new assignment of conductors without affecting conductor assignments that already exist. Accordingly, they keep the cable and conductor assignments as previously assigned. This would be necessary, for example, if cable conductors have already been generated and their assignments should not be changed later. New conductor assignments should be made using this menu item.

All conductors are normally designated during the *Device selection* for the cable. If components, like new cable connections, are to be added from the reserve conductors, the previous conductor assignments should not change. Nevertheless, it is preferable for you to be able to automatically assign the rest of the new conductors to the cable and its connections. This is done via the **KEEP EXISTING PROPERTIES** menu item.

You select the main function of the cable to be edited (to be updated).

Fig. 5.121
Cables are selected by main function.

Now you call up the **KEEP EXISTING PROPERTIES** function via the **PROJECT DATA/CABLES/ASSIGN CONDUCTORS** menu. Based on the function definitions for the conductor designations stored in the cable part, EPLAN writes to the previously undefined connection definition points (conductor designations), whereby existing conductor designations are not overwritten.

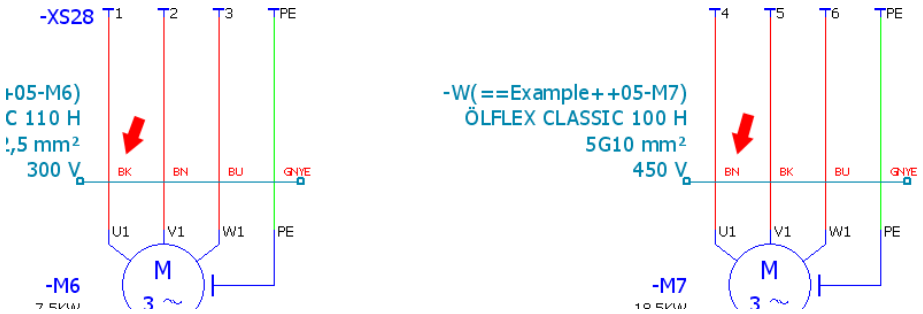


Fig. 5.122 Previously undesignated conductors were given designations

5.5.5.2 Reassign all

The counterpart to the "Keep existing properties" menu item described in section 5.5.5.1 is the **REASSIGN ALL** menu item This function overwrites (rewrites) previously designated conductors with the (conductor) designations from the cable's function definition. The previous conductor designations are replaced completely.

The procedure is as follows. Select the cables for which the conductor designations should be completely reassigned.

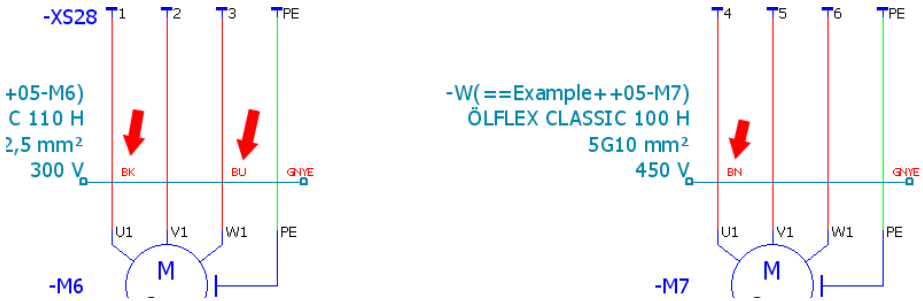


Fig. 5.123 Selecting cables

Next select the **REASSIGN ALL** menu item from the **PROJECT DATA/CABLES/ASSIGN CONDUCTORS** menu. EPLAN then reassigns the conductors for all of the selected cables without further prompts.

Fig. 5.124 All conductors were reassigned.

5.5.6 Cable navigator/New ...

The **Cable navigator** offers functions similar to those in the other navigators. The **NEW ...** function in the cable navigator popup menu creates a new **CABLE DEFINITION** function.

EPLAN opens the **PROPERTIES (COMPONENTS)/ CABLE** dialog. Here you make the usual entries, such as a device selection, and save the dialog. EPLAN then adds this cable definition to the cable navigator as an unplaced function.

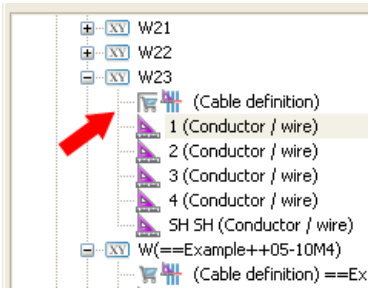


Fig. 5.125 New cable

5.5.7 Cable navigator/Place/[Representation type]

Devices, in this case cables, can of course also be inserted into schematics via the **PLACE** function in the **cable navigator**.

NOTE: If the **CABLE DEFINITION** function is selected in the navigator, only the cable definition lines will be placed.

In the **cable navigator**, you select the cables and conductors that are to be placed. Alternatively, you can also directly select the **cable DT**. In this case, all functions belonging to this cable DT will be offered for placement.

Now you open the popup menu and start the **PLACE** function, here **MULTI-LINE**.

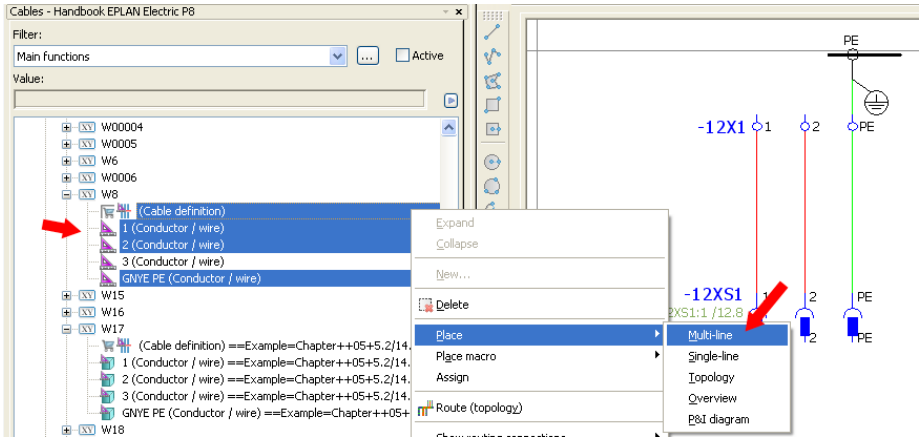


Fig. 5.126 Placing cables

EPLAN then “hangs” the cable on the cursor and you can place it in the schematic. You define the starting point of the cable by left clicking at a position to the left of the first connection. The cable is then pulled to the right over the connections and then placed by left clicking to the right of the last connection. After the cable is placed, EPLAN fills out the cable properties.

The cable has been placed and you can now place the conductors that are still “hanging” on the cable.

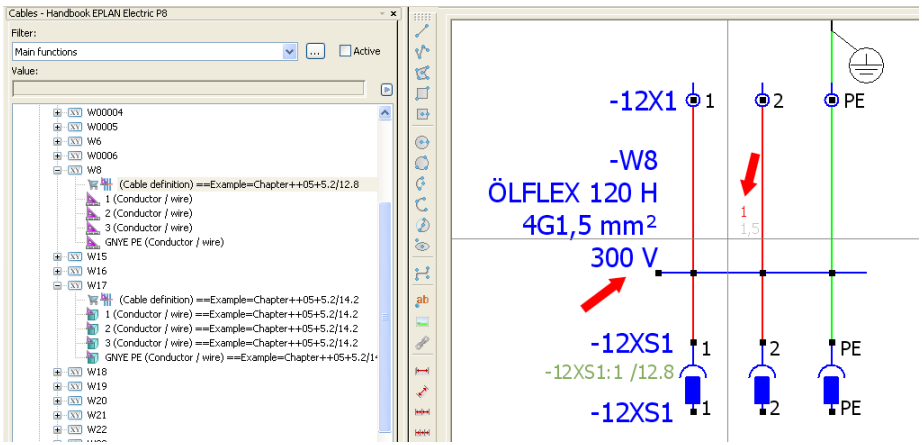


Fig. 5.127 Placing cables

You can place the conductors directly on the connection. Since EPLAN has already placed a connection definition point on this connection, you are asked whether the existing connection definition should be replaced. You can confirm with **YES** and EPLAN will replace the data.

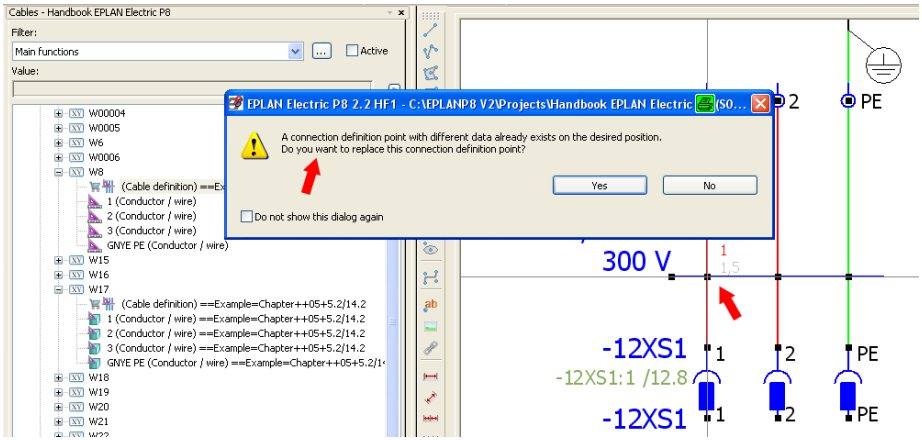


Fig. 5.128 Replace connection definition point dialog

This completes multi-line placement of a cable, including the conductors.

5.5.8 Cable navigator/Number cable DT

The function and procedure used by the **NUMBER CABLE DT** function in the **cable navigator** is identical to the function in the **PROJECT DATA/NUMBER CABLES** menu. Here too, you select the cables to be numbered in the cable navigator and then call the **NUMBER CABLE DT** function.

The familiar dialogs and settings are then displayed. This will not be explained further because it was described in the previous section.

5.5.9 Cable navigator/Number DT

A cable in EPLAN is also a normal device, albeit with special properties. EPLAN not only allows *cable numbering* with, for example, source/target in the **cable device tags**, it also allows normal *numbering formats* such as [Page Identifier Column].

You select the cables to be numbered in the **cable navigator** or in the schematic, open the popup menu and start the **NUMBER DT** function.

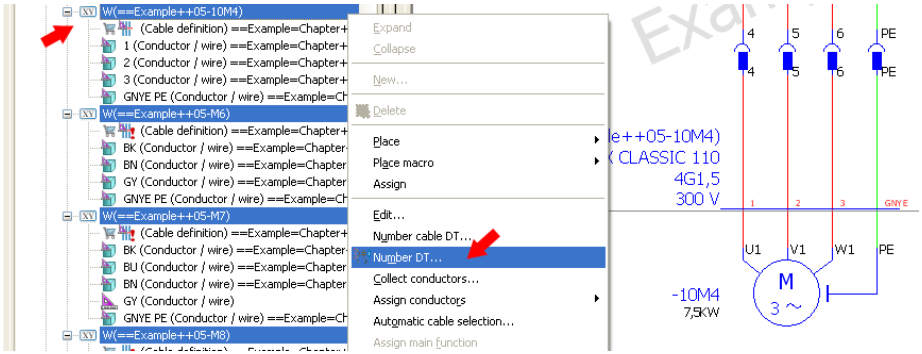


Fig. 5.129 Selected cables in the cable navigator

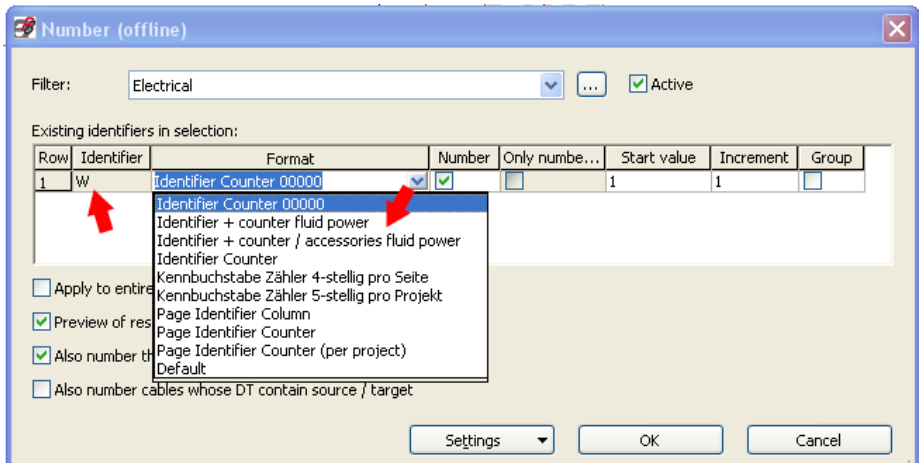


Fig. 5.130 Number (offline) dialog

EPLAN opens the familiar **NUMBER (OFFLINE)** dialog. This is where the desired *numbering scheme* and the other possibly required settings are defined. When you click **OK**, EPLAN numbers the cables like normal devices.

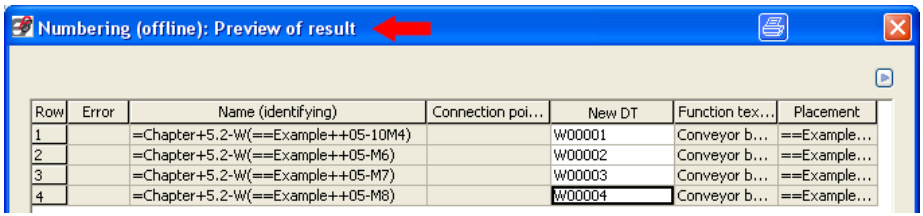


Fig. 5.131 Preview

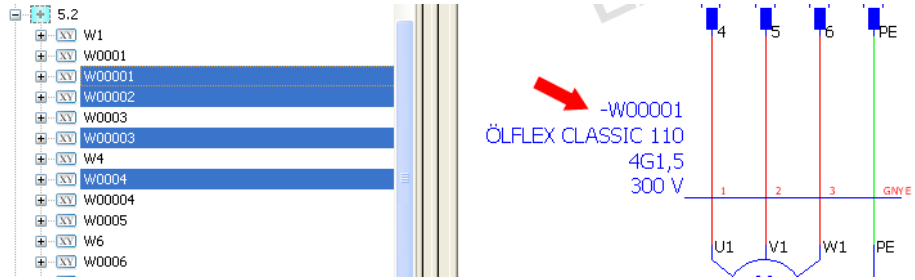


Fig. 5.132 Final result, numbered cables

5.6 PLC navigator

The **PLC navigator** allows the data of PLC cards and other PLC components used in the project to be managed conveniently from a central place. The PLC navigator shows the individual functions of the PLC cards, such as *addresses*, *power supply connections*, and *channels*.

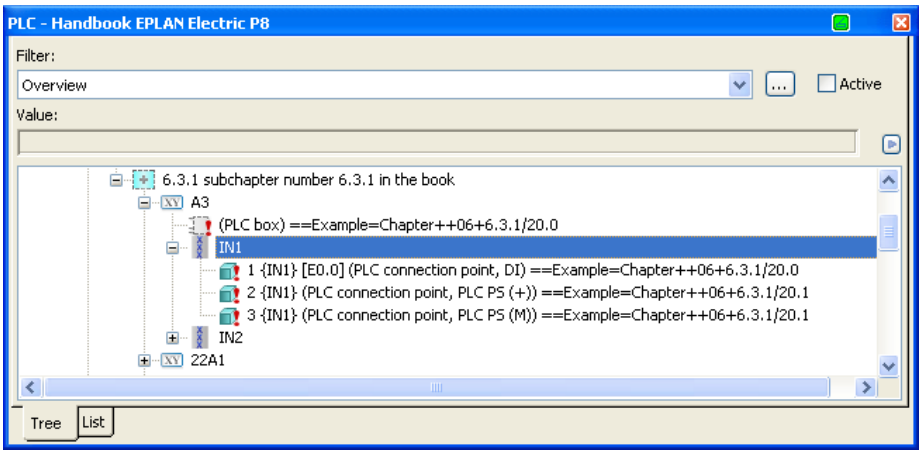


Fig. 5.133 The PLC navigator

In addition to the PLC navigator, the **PROJECT DATA/PLC** menu contains other functions for editing PLC data, such as **SET DATA TYPES** or **ADDRESS CARDS**.

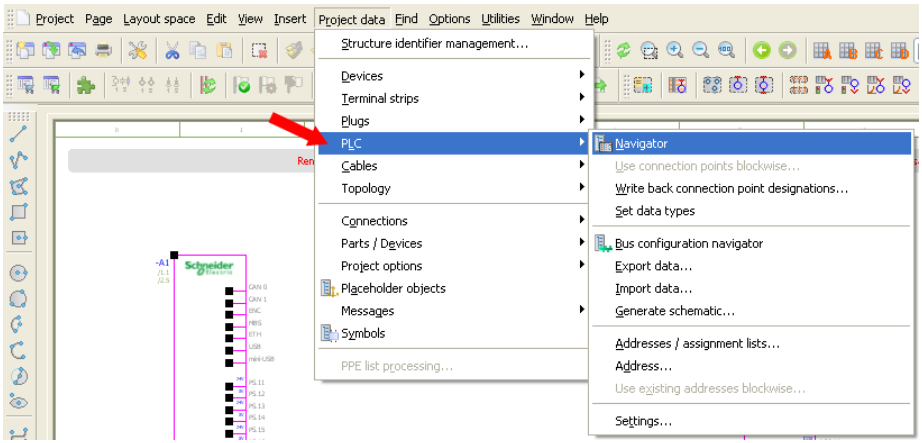


Fig. 5.134 Project data/PLC menu

NOTE: A very important fact is that EPLAN uses the **connection point designation** as the **identifying property** of a PLC function. This means that, for example, the designation E1.0 or A23.7 is not the identifying property, but rather the connection point designation (PIN number or terminal number) of the function of the PLC item.

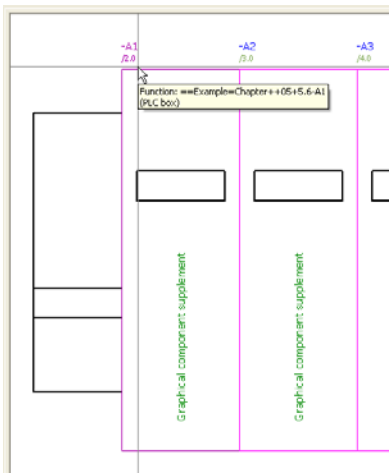


Fig. 5.135 Graphical overview (rack layout)

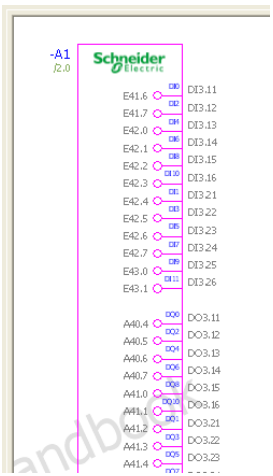


Fig. 5.136 Logical overview (references, etc.)

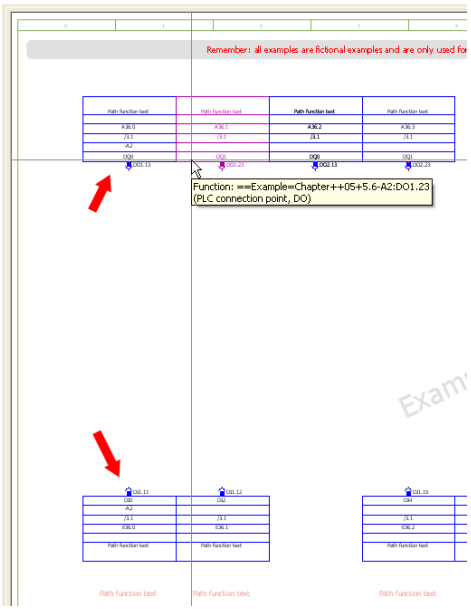


Fig. 5.137 Distributed view (schematic)

Representations of PLC components

The representation of PLC items can vary. One option is “dividing by three”. This means that there is a *graphical PLC overview* (rack layout), a *logical PLC overview* (card with cross-references), and the *distributed representation* in the schematic (inputs and outputs used, power supply connections, etc.). This representation is not essential, but it makes it easier to deal with PLC data in a project.

The various ways of editing are described in sections 5.6.1 to 5.6.8, whereby it is assumed that the PLC structure is the same as that described previously, i. e. there is at least one finished logical PLC overview with correct entries (address assignments and connection point designations) and one or more inputs or outputs represented in a distributed manner in the schematic (multi-line display).

5.6.1 PLC/Write back connection point designations

The **WRITE BACK CONNECTION POINT DESIGNATIONS** function in the **PROJECT DATA PLC** menu writes the connection point designations from the PLC overview back to the PLC terminals that are represented as distributed in the schematic.

To use the function, you first draw PLC terminals in the schematic or insert them from a macro. The PLC addresses are known and are already correctly entered into the schematic during project editing. The distributed representation of the PLC terminals is finished.

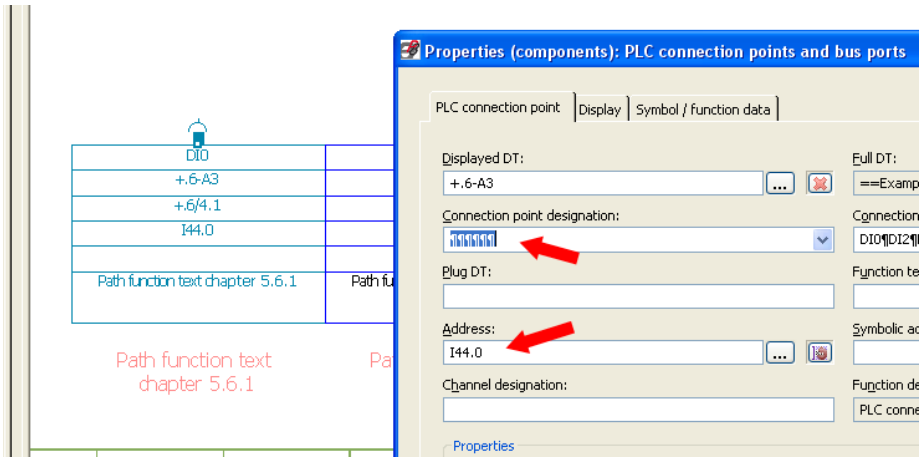


Fig. 5.138 Added distributed PLC data in schematic

When editing is complete, the view is changed to the logical overview page (PLC). Here you select the corresponding card(s) and address ranges. Then you select the **WRITE BACK CONNECTION POINT DESIGNATIONS** item in the **PROJECT DATA/PLC** menu.

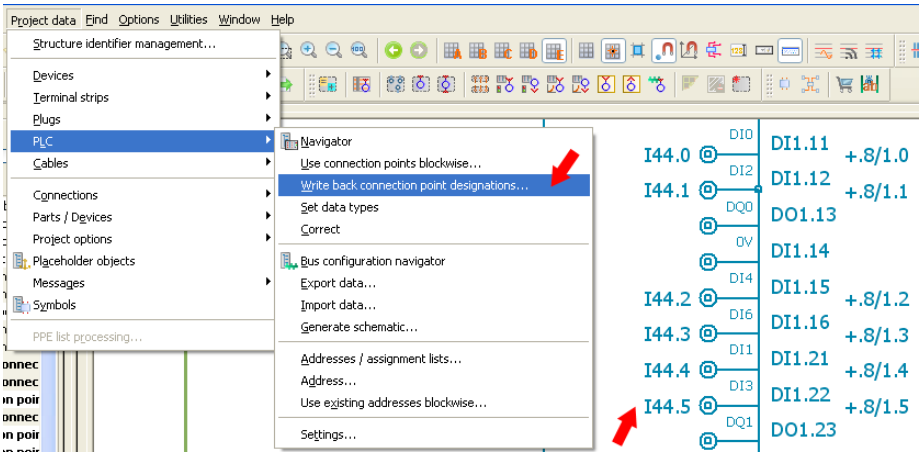


Fig. 5.139 Run the Write back connection point designations function

EPLAN opens the **WRITE BACK CONNECTION POINT DESIGNATIONS** dialog. You confirm this selection by clicking **OK**. The connection point designations from the logical PLC overview (of the selected PLC terminals) in the schematic are now written back to the PLC data that is in distributed view.

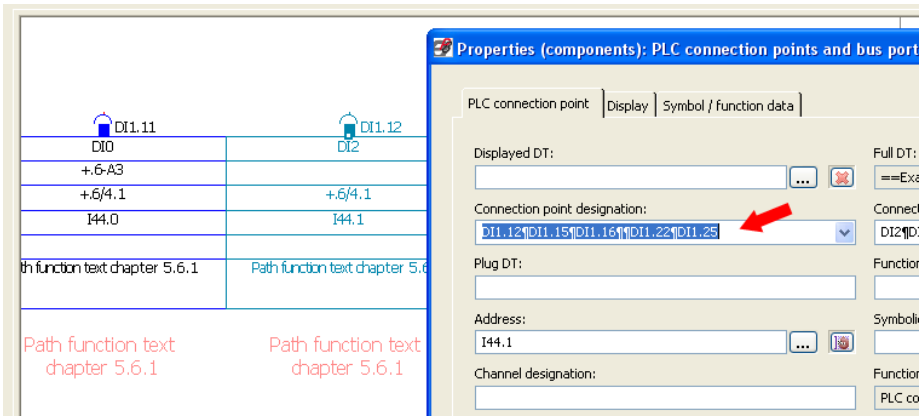


Fig. 5.140 Written back connection point designations

The result can be seen in Fig. 5.140. All connection point designations are correctly transferred to the PLC terminals. But this only works if the addresses themselves have been entered correctly. Otherwise, EPLAN cannot recognize the relationship between the PLC terminal in the logical PLC overview and the PLC terminal in the distributed view.

5.6.2 PLC/Set data types

The **SET DATA TYPES** function is used to automatically update and write the corresponding data types to the PLC terminals if they are not already there. EPLAN derives the data types from the functions. You select the PLC terminals in the *logical PLC overview* or the *PLC terminals in the schematic* and call the **SET DATA TYPES** function in the **PROJECT DATA/PLC** menu. EPLAN writes the data type into the properties of the PLC terminal.

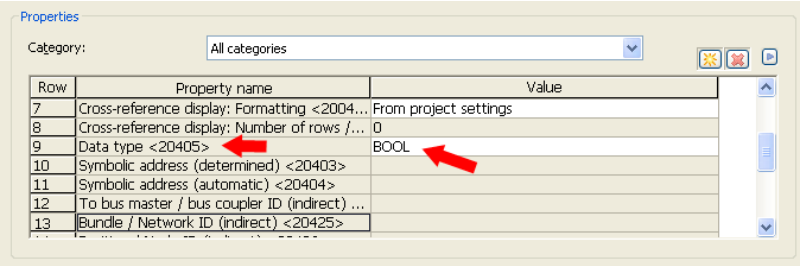


Fig. 5.141 Set data type

5.6.3 PLC/Addresses/assignment lists

In the **PROJECT DATA/PLC** menu, the **ADDRESSES/ASSIGNMENT LISTS** function opens the dialog of the same name (a table of all addresses used in the diagram).

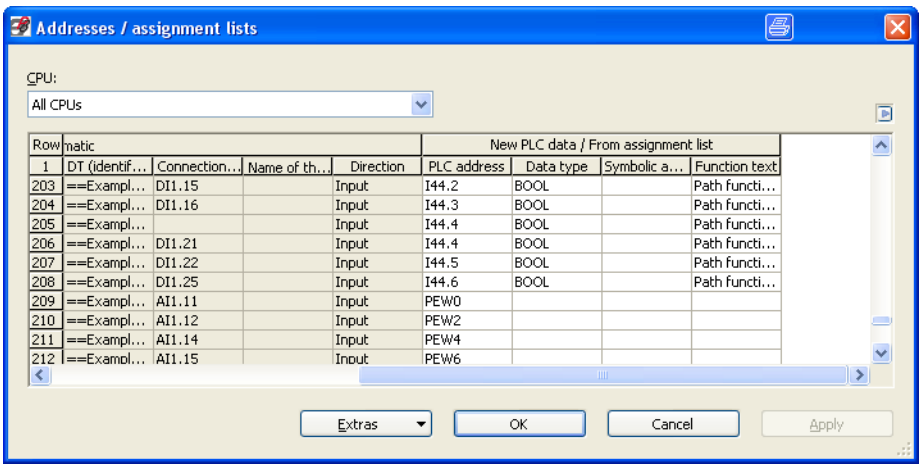


Fig. 5.142 Addresses/assignment lists dialog

In this dialog, the *PLC address*, *Data type*, *Symbolic address*, and the defined *Function text* properties can also be manually changed. EPLAN then automatically writes back the modified data to the PLC terminals and the modified function text to the manually placed path function text in the PLC terminal path.

At this point, if you remain at the function texts, it is also possible to edit the function texts so that they can be written back to the schematic. Since this is done at a central place, it avoids tedious paging through the schematic in order to modify the placed path function text at the corresponding PLC terminals.

In addition, an *assignment list* can be exported from this dialog (for the downstream PLC software) or imported (from the PLC software). Click the **EXTRAS** button in the lower area of the dialog. EPLAN opens another menu.

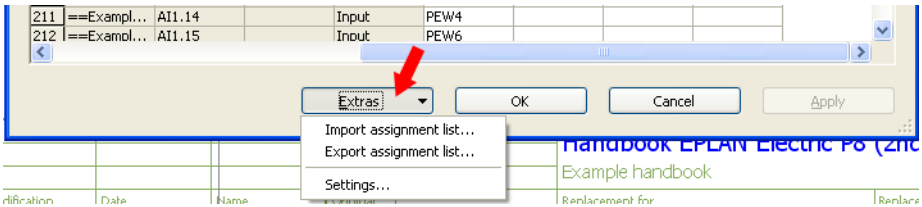


Fig. 5.143 The Extras button and its functions

This allows *assignment lists* to be *exported* or *imported*. Different settings may be required, depending on the PLC and its software.

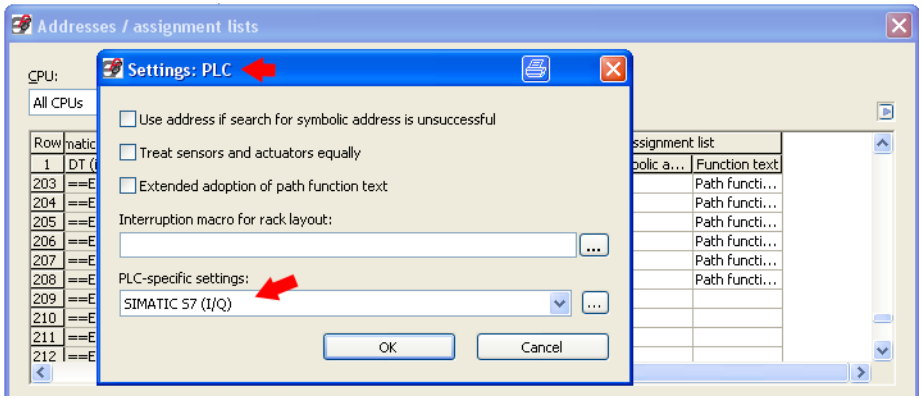


Fig. 5.144 PLC settings

EPLAN already contains predefined schemes for a range of PLC software for export and import, so that all you need to do is select the correct PLC software in the settings under *PLC-specific settings*.

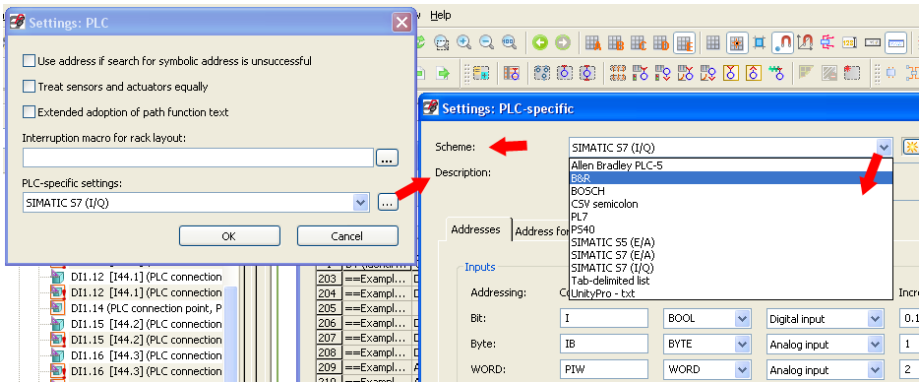


Fig. 5.145 Different PLC schemes

If all of the settings were properly selected, EPLAN can now generate the assignment list. Use the **EXTRAS** button in the **ADDRESSES/ASSIGNMENT LISTS** dialog to select **EXPORT ASSIGNMENT LISTS**. In the **EXPORT ASSIGNMENT LISTS** dialog, you can check the settings once more and adjust the settings if necessary. Select the correct option based on the available CPUs (there could be multiple controls in a project). You then simply confirm this dialog with **OK**. EPLAN generates the assignment list in the desired PLC software format and saves it in the specified directory.

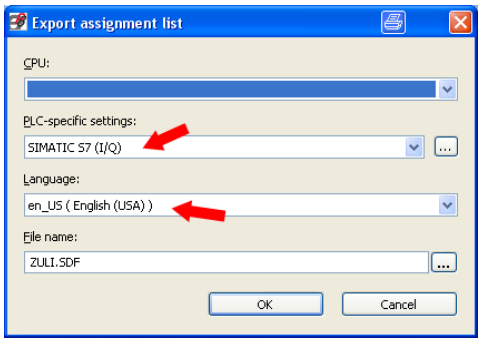


Fig. 5.146 Export assignment lists dialog

5.6.4 PLC navigator/Address

EPLAN allows simple addressing or readdressing of PLC terminals via the **ADDRESS** function in the **PROJECT DATA/PLC** menu. The functions in the logical PLC overview and the functions in the schematic for the PLC terminals in distributed view will be re-addressed or addressed for the first time.

NOTE: In order for all PLC functions to be addressed, it is necessary to select the PLC card in the PLC navigator beforehand. This ensures that all of the functions, those of the logical PLC overview and the PLC functions represented in distributed view, are included and addressed.

In the PLC navigator, you select the card to be addressed. Then you call up the **ADDRESS** function in the **PROJECT DATA/PLC** menu.

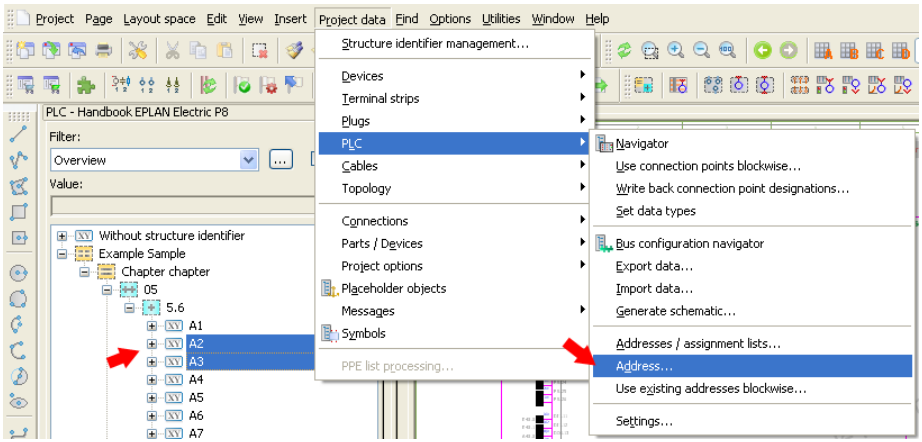


Fig. 5.147 Selecting the card to be addressed and calling up addressing

EPLAN now opens the **READDRESS PLC CONNECTION POINTS** dialog.

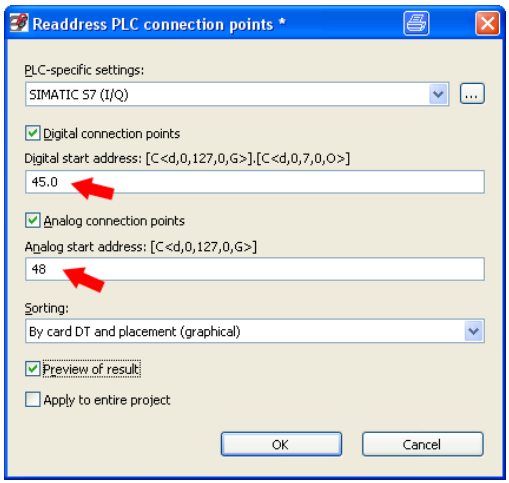


Fig. 5.148 Defining start addresses

In the **Digital start address** and **Analog start address** fields, the new address is entered *without* a prefix. EPLAN obtains the prefix, meaning whether an entry begins with E or with I, according to the selected scheme, which is entered in the *PLC-specific settings* selection field. It is possible, however, to enter a prefix in the start address entry field, an "IX" for example. EPLAN would then use this during addressing.

I recommend leaving the *Preview of result* parameter switched on. When you click **OK**, the **ADDRESS PLC CONNECTION POINTS/PREVIEW OF RESULT** dialog is displayed so that you can check the results and use the **CANCEL** button to stop the process if necessary.

If the result is correct, you click **OK** to confirm the **ADDRESS CONNECTION POINTS/PREVIEW OF RESULT** dialog and EPLAN readdresses the PLC functions of the selected cards in the entire project.

Address PLC connection points

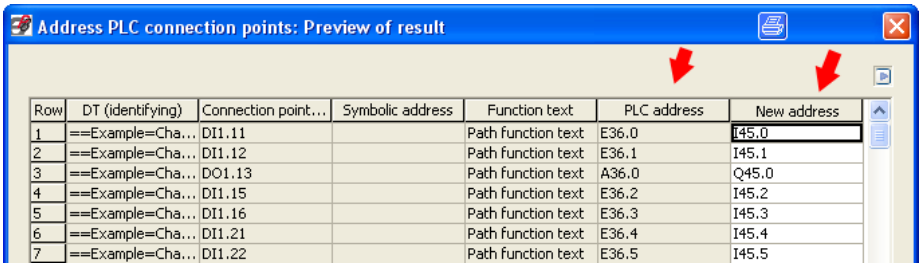


Fig. 5.149 Checking the new addressing

5.6.5 PLC navigator/New ...

As in the other navigators, new devices with a function definition can also be created in the **PLC navigator**. The **NEW ...** item in the popup menu is used for this.

EPLAN opens the **FUNCTION DEFINITIONS** dialog. In the PLC navigator you can only select PLC function definitions. After applying the function definition by clicking the **OK** button, EPLAN opens the **PROPERTIES (COMPONENTS) PLC BOX** dialog. Here you must make the desired entries and save the properties. You can also immediately perform a *device selection* for the new PLC card.

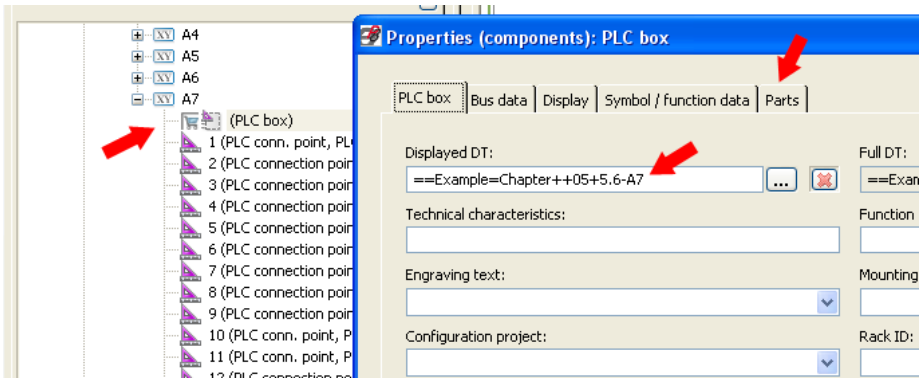


Fig. 5.150 New unplaced PLC card in the navigator

The new device is stored in the **PLC navigator**, with a function definition of **PLC BOX** and the selected part's function definitions as an unplaced PLC box.

5.6.6 PLC navigator/New functions ...

To add additional functions to devices in the PLC navigator, you use the **NEW FUNCTIONS ...** function. This is started in the usual way via the popup menu in the **PLC navigator**.

EPLAN opens the **GENERATE FUNCTIONS** dialog. In the **Function definition** field, you can select a PLC connection point, DI (for example) via the ... button and the subsequent **FUNCTION DEFINITIONS** dialog. For a card with 20 connection points, the numbering pattern is set to 1 – 20.

Generating new PLC functions

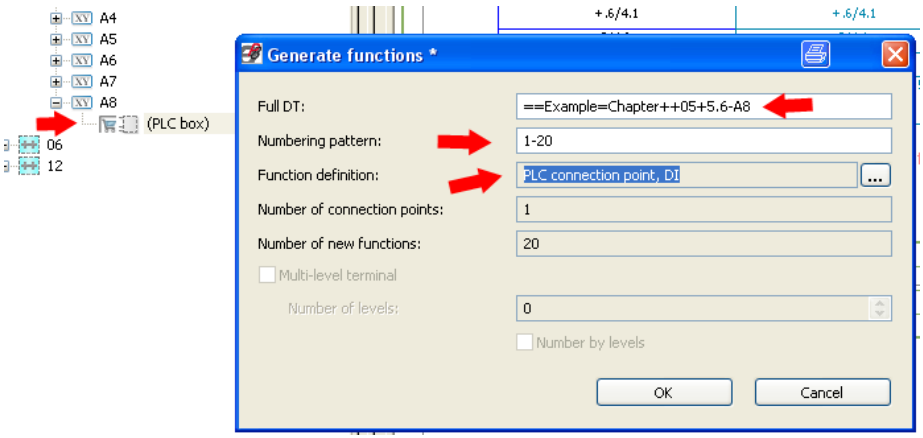


Fig. 5.151 Generating new PLC functions

These functions are generated when **OK** is clicked. This is also the approach for generating supply connection points for a card. This method creates a complete PLC card in the PLC navigator, with correct *PLC addresses* and their *connection point designations* ready to be placed.

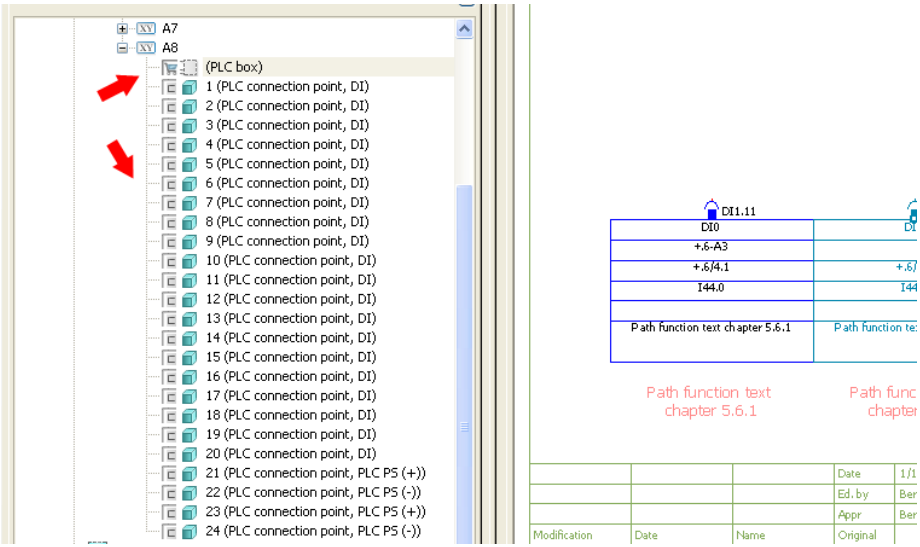


Fig. 5.152 New PLC functions

5.6.7 PLC navigator/New device ...

In contrast to creating individual new functions, the **NEW DEVICE ...** item in the popup menu creates a completely new device that already has default function definitions; in the PLC navigator this is, of course, a PLC function, such as an input card.

After you select **NEW DEVICE ...** in the popup menu of the **PLC navigator**, EPLAN opens the **PARTS MANAGEMENT** dialog. Only PLC parts can be selected. The selected device is now applied in the PLC navigator by clicking **OK**.

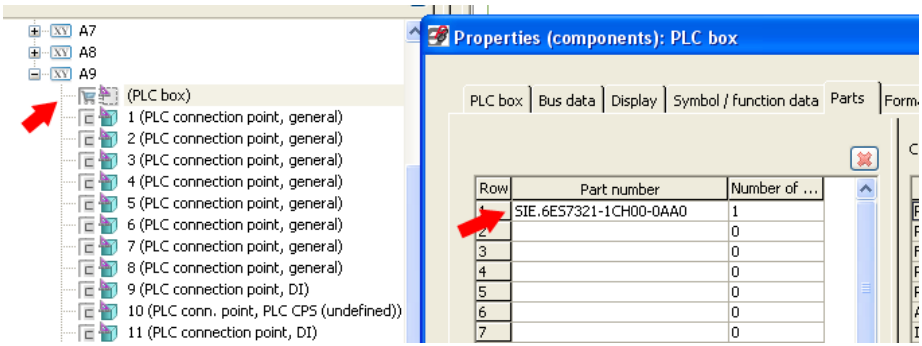


Fig. 5.153 New unplaced device in the navigator

EPLAN automatically assigns the next available device tag to the new device in the PLC navigator. The device is then ready to be used in the schematic or ready for its functions to be placed.

5.6.8 PLC navigator/View

The **PLC navigator** in EPLAN provides different *views* of the PLC data: the **DT-oriented** view, the **address-oriented** view, and the **channel-oriented** view.

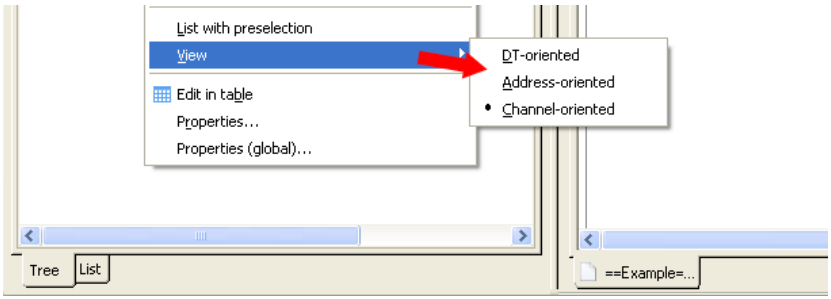


Fig. 5.154 View selection in popup menu

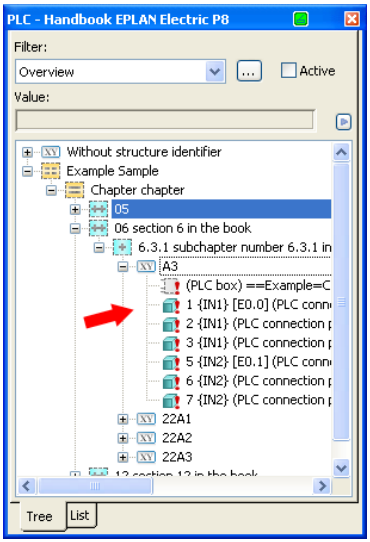


Fig. 5.155 DT-oriented view

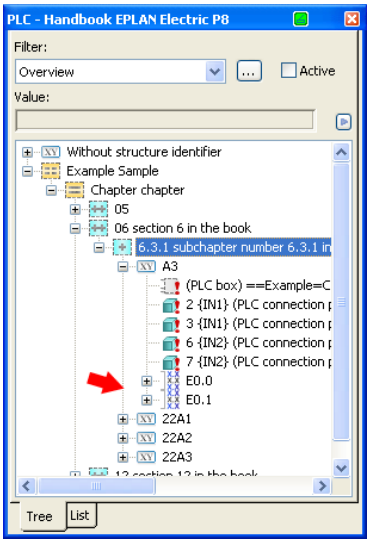


Fig. 5.156 Address-oriented view

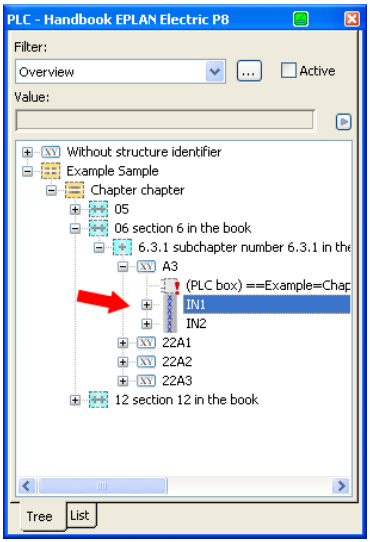


Fig. 5.157 Channel-oriented view

In the DT-oriented view, the PLC data is listed according to its device tag. In the address-oriented view, the PLC navigator organizes the PLC data by address. And lastly, the channel-oriented view shows the channel groups and all of their PLC functions. The channel-oriented view has the advantage of allowing you to select and place all of the functions belonging to a channel “at once”. This ensures that you do not forget any of the functions in the channel group.

5.7 Part navigator/(Devices/Parts)

The **bill of materials navigator** has a view of all devices with or without the part numbers that exist in a project. The bill of materials navigator is started via the **PROJECT DATA/PARTS/DEVICES** menu.

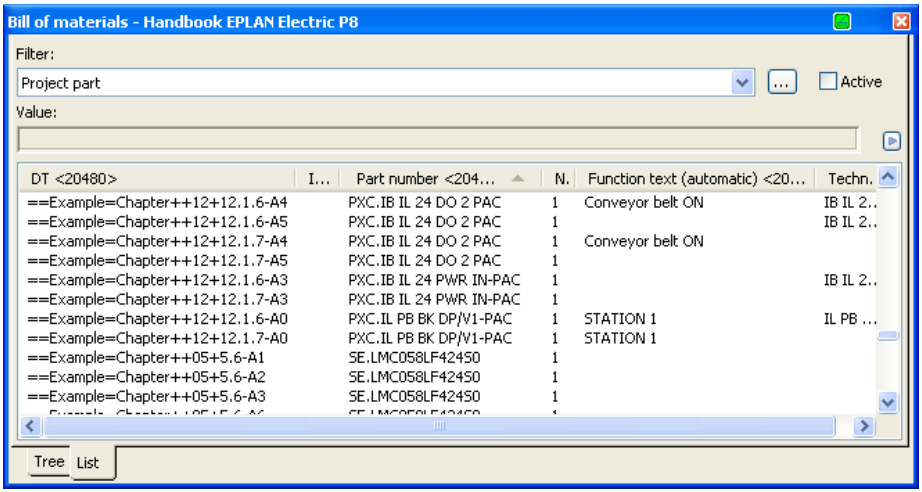


Fig. 5.158 The bill of materials navigator in list view

Like the other navigators, the bill of materials navigator offers fundamentally different viewpoints of the project and its data.

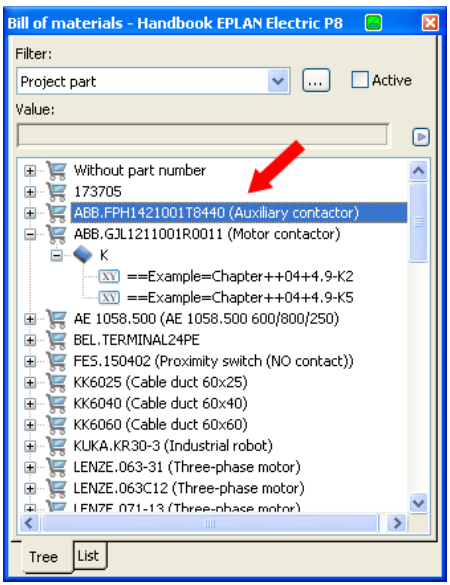


Fig. 5.159 Part number view (leading)

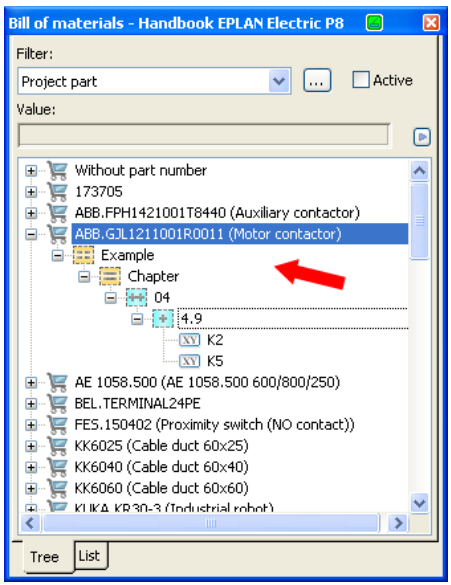


Fig. 5.160 Part number/Structure identifier view

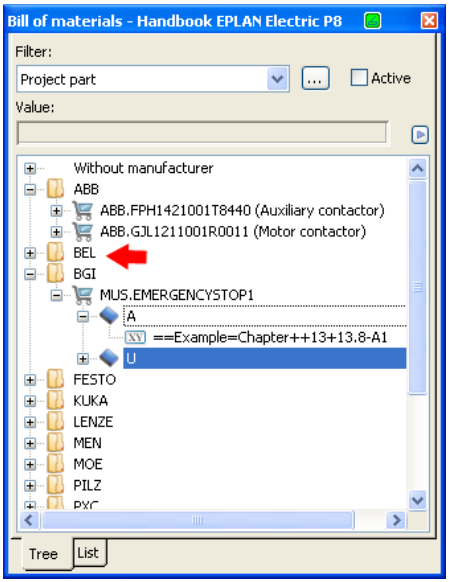


Fig. 5.161 View by manufacturer

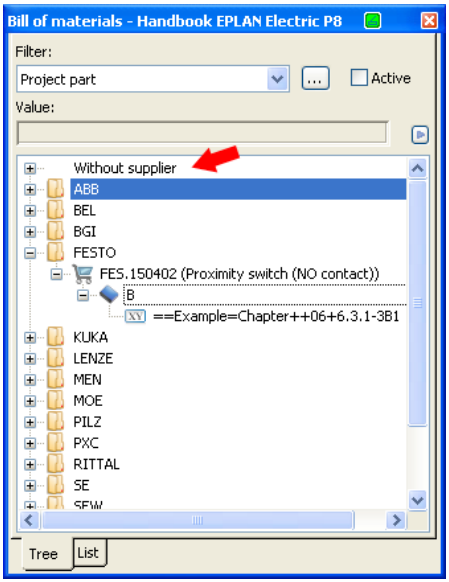


Fig. 5.162 View by supplier

The bill of materials navigator provides many ways to quickly and easily edit parts data.

5.7.1 Popup menu functions

The **bill of materials navigator** offers different functions in its popup menu to make editing the project parts easier.

5.7.1.1 Add project part

If you select the **ADD PROJECT PART** menu item, EPLAN opens *parts management*, where you can add a part to the project as a *project part*.

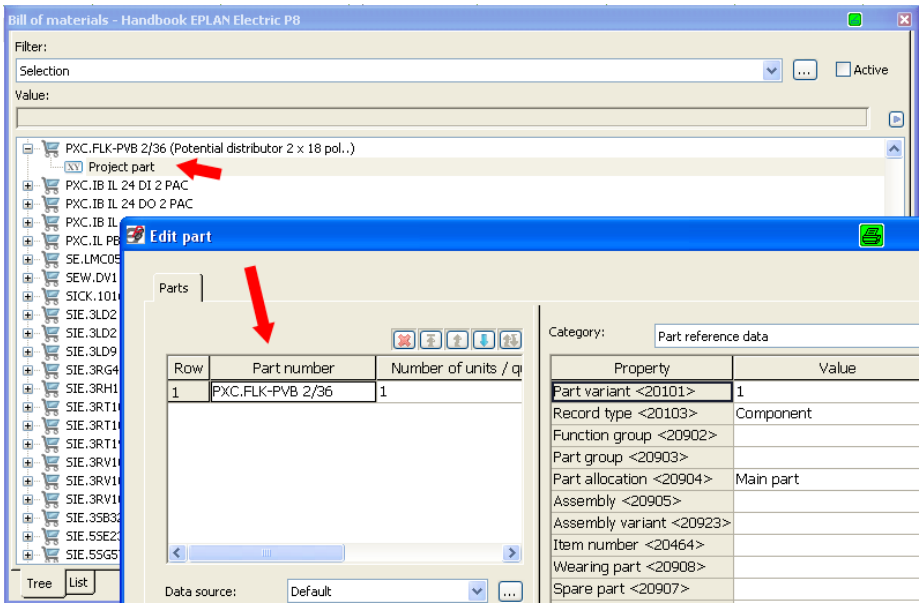


Fig. 5.163 Added project part

Project parts initially do not have device tags. These are normally parts that are shipped as spare or loose parts. Project parts can also be filtered or evaluated.

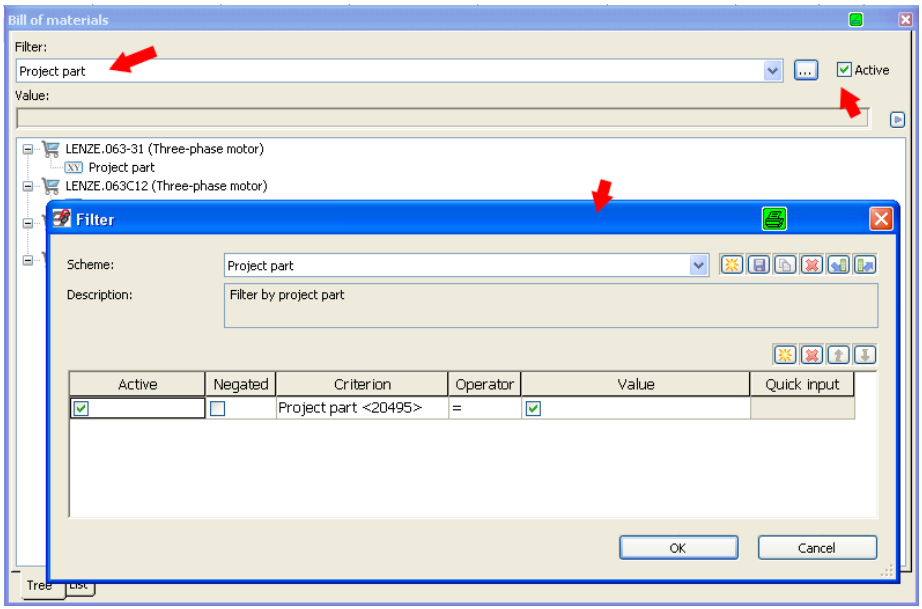


Fig. 5.164 Filtering by project part

5.7.1.2 Add part

You can easily add additional parts to an existing device with the **ADD PART** menu item. Select the device and call the **ADD PART** menu item.

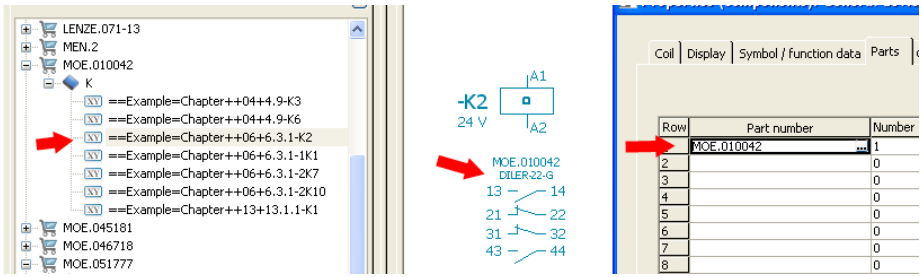


Fig. 5.165 Before running the Add part function

EPLAN starts *parts management* after running the **ADD PART** function. The desired part is chosen, selected and applied to the selected device by clicking **OK**.

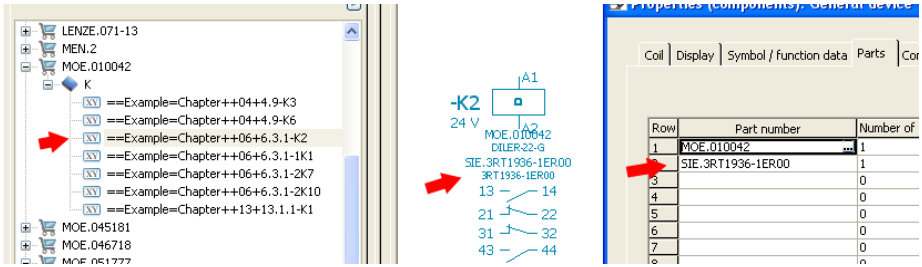


Fig. 5.166 A part was added.

This function of course also works with multiple selections. For example, you could add surge arresters for several contactors “at once” if you “forgot” them.

5.7.1.3 Place via ...

The PLACE VIA ... menu item contains additional submenu items.

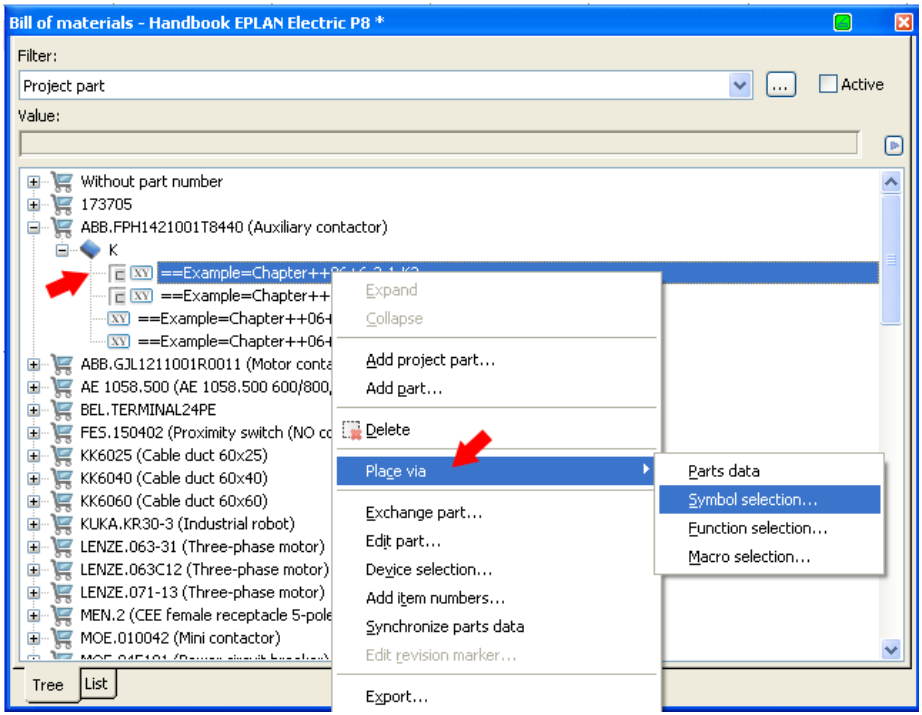


Fig. 5.167 Placement options

The submenu item PLACE VIA PARTS DATA means that EPLAN places the *symbols* and *macros* stored with the part when the part is placed on the schematic.

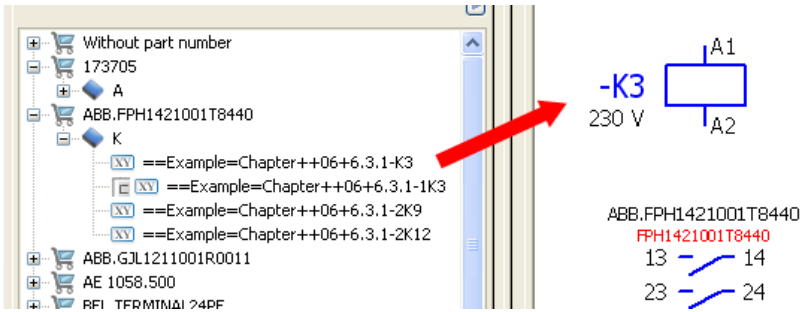


Fig. 5.168 Placing via parts data

The submenu item PLACE WITH SYMBOL DATA allows EPLAN to open symbol selection before placement. Here you select the appropriate or desired symbol. Then you can place this symbol on the schematic page and EPLAN will enter the parts data for this symbol.

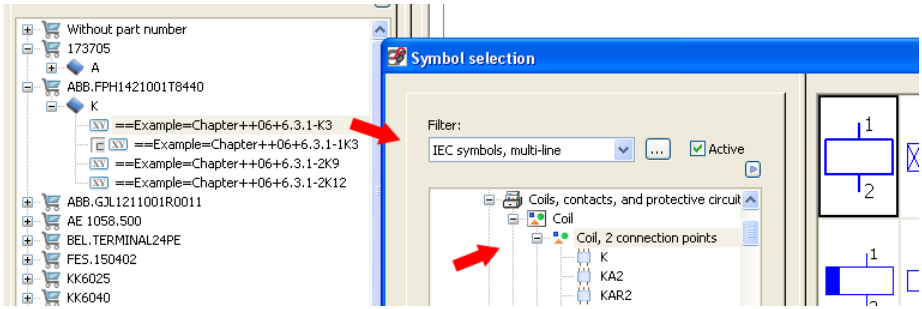


Fig. 5.169
Place via symbol data

The **FUNCTION SELECTION** submenu item starts before placement. In the **FUNCTION DEFINITIONS** dialog, you select and apply the desired function definition before placing the part. Then the part can be placed. EPLAN then assigns the previously selected *function definition* to the part.

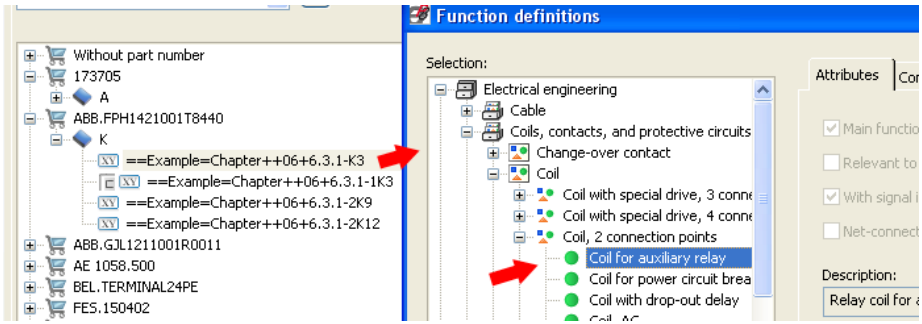


Fig. 5.170
Placing via function data

The last menu item allows part placement via **MACRO SELECTION** beforehand. When you select this menu item, EPLAN opens the **SELECT MACRO** dialog. Here, you can select the macro and then click the **OPEN** button to apply and place it.

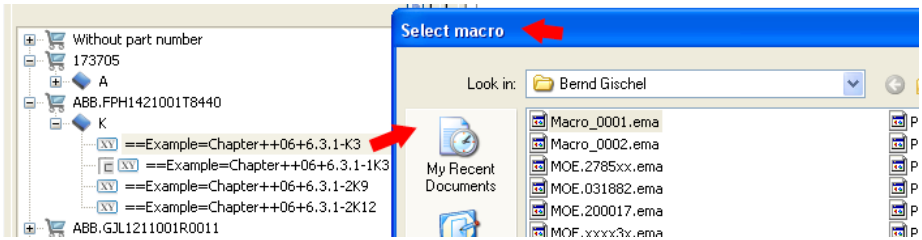


Fig. 5.171
Macro selection before placement

5.7.1.4 Exchange part

This menu item offers the option to exchange a part quickly and easily. Select the part in the **bill of materials navigator** and open the **EXCHANGE PART** menu item. EPLAN then opens **parts management**, where you select the *part to be exchanged* is and then click **OK** to apply it. EPLAN instantly exchanges this part with the selected one.

5.7.1.5 Device selection

This menu point is an interesting enhancement. For example, you can use this to perform *contactor selection* retroactively for multiple selected parts data (contactor coils). Select the part and run the **DEVICE SELECTION** menu item.

EPLAN now allows you to perform device selection for every device, one by one. If device selection has not been performed for a device, then it suffices to press the **ESC** key. EPLAN then asks whether the entire action should be cancelled, or whether the selected device should be skipped.

Fig. 5.172
Cancel option in device selection

5.7.1.6 Synchronize parts data

The **SYNCHRONIZE PARTS DATA** menu item offers the option of synchronizing the master data of specific, selected parts with parts management. This operation compares the parts data with parts management and overwrites the saved parts data in the project if necessary.

5.8 Assign function (navigators)

A number of navigators have the **ASSIGN** function in their popup menu.

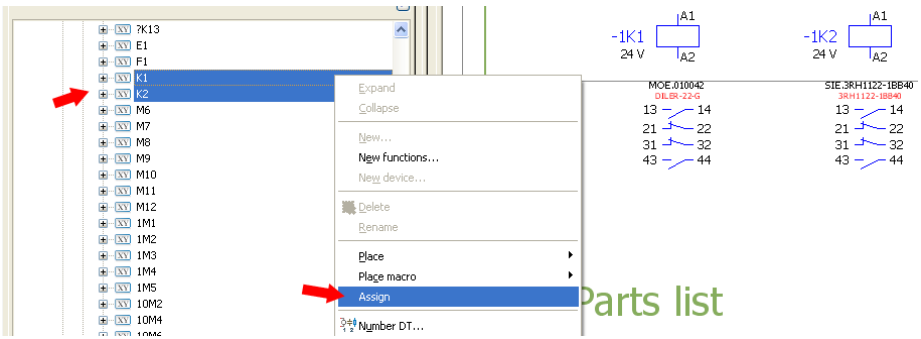


Fig. 5.173
The Assign menu item in the device navigator

The **ASSIGN** menu item allows the *properties of objects* selected in the navigator, e. g. a *device tag* or parts of a device such as a PLC connection point, to be assigned the properties of an existing object (e. g. a power contact).

It is also possible to select several objects in the navigator, execute the **ASSIGN** function, and then assign these properties to other objects by clicking them in the schematic.

You select a few devices in the **device navigator** and then execute the **ASSIGN** function. EPLAN now visibly "hangs" all of the properties for this device, such as for example the *device tag* or the *function definition*, on the cursor (the remaining unassigned device tags remain visible in the foreground).

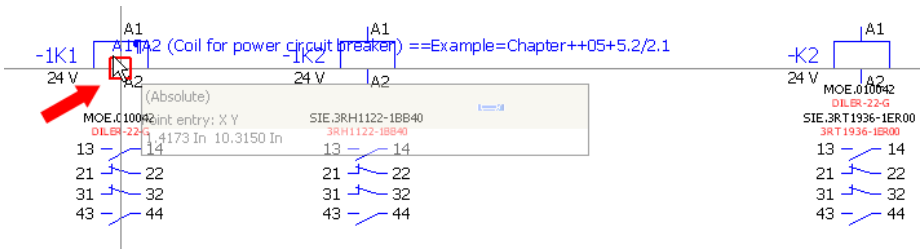


Fig. 5.174 The assigned properties "hang" on the cursor.

You then click the individual power contacts in the schematic one after another (always exactly those that are to receive a *device tag*).

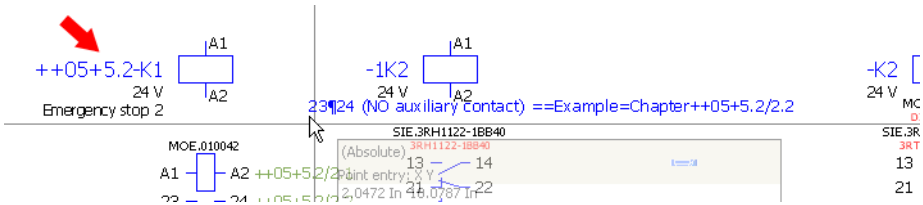


Fig. 5.175 Assigned device tags

EPLAN transfers one DT after another to the power contacts without requiring this to be manually entered at each contact. This is a very useful function that saves time and ensures the error-free transfer of information. It is available in the following navigators (in addition to the device navigator):

- Terminal strip navigator
- Plug navigator
- PLC navigator
- Cable navigator
- Topology navigator
- Connection navigator
- Interruption point navigator

5.9 Navigators/Quick input filter

To simplify work, most navigators can also filter by a scheme according to different values. This is similar to quick input.

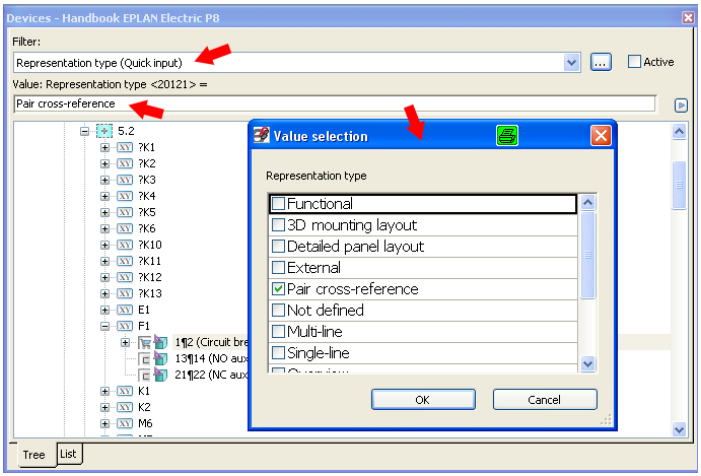


Fig. 5.176 Quick entry filter

This type of filter, i.e. the ability to use quick input, eliminates the need to create individual schemes for each value.

In the **Device navigator** example shown in Fig. 5.176, this would be filtering by representation type. Previously, you had to create a scheme for each of the representation types that you wanted to use for filtering. This results in a large number of possible, selectable schemes, which over time would mean that selecting a scheme would be confusing and cumbersome.

The ability to use the **QUICK ENTRY** function for properties reduces the number of different schemes that have to be created, so that is again clear and easy to select schemes.

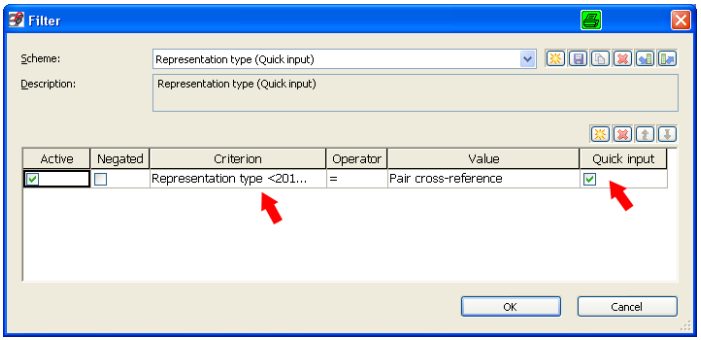


Fig. 5.177 Quick entry

Not all criteria are suitable for the quick entry option when filtering.

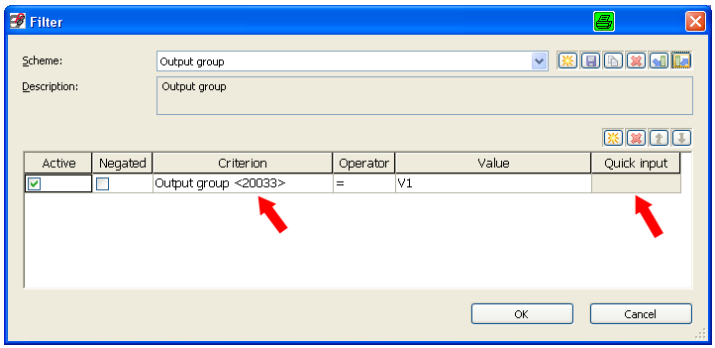


Fig. 5.178 Quick entry is not possible here.

Quick entry can be used in the following navigators:

- Page navigator
- Device navigator
- Terminal strip navigator
- Plug navigator
- PLC navigator
- Cable navigator
- Topology navigator
- Connection navigator
- Potential navigator
- Process navigator
- Interruption point navigator
- Bill of materials navigator
- 2D panel layout navigator
- 3D mounting layout navigator
- Device list navigator
- Project option navigator
- Placeholder object navigator
- Symbol navigator (symbol selection)
- Comment navigator

6 Reports

This chapter deals with reports: Which ones are necessary, what settings are needed for which reports, how do I make the most effective use of templates and reports, and what are the advantages for the user? Some of the possibilities offered by external editing are also touched upon using simple, short examples.

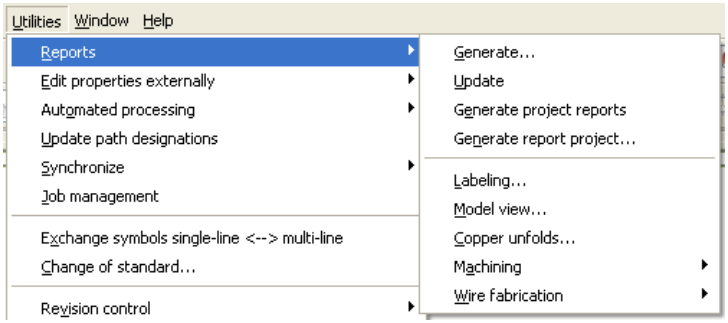


Fig. 6.1 Reports menu

The **UTILITIES/REPORTS** menu contains the functions for generating graphical reports such as terminal diagrams, parts lists and connection lists. The **UTILITIES/REPORTS/LABELING** menu item allows the user to export various reports such as cable overviews or summarized parts lists in (e. g.) Excel format.

EPLAN thus allows users to generate internal outputs (graphical outputs for project documentation) and to transfer evaluated data into an external format (Excel, text format, etc.), for passing on the generated data from a schematic, for example.

Generation of graphical reports is only one option offered by EPLAN. Previously, you could externally edit the generated data via the *Labeling* function, but there was no way of synchronizing this with the EPLAN project, so an additional functional scope was added to EPLAN allowing properties to be exported, externally edited, and then re-imported back into EPLAN.

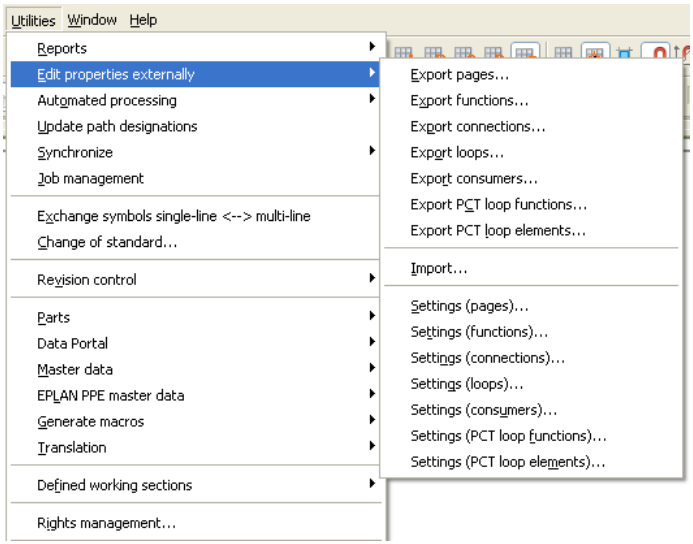


Fig. 6.2 The Edit properties externally menu

The **UTILITIES/EDIT PROPERTIES EXTERNALLY** menu item contains numerous options allowing you to more conveniently edit a wide range of properties, in Excel for example, and then import the externally edited data to the EPLAN project. The data in the project is automatically changed without you having to make these changes in EPLAN.

6.1 What are reports?

Reports are selected project data that can output, for example, a *terminal diagram* in graphical form on newly generated project pages (internal report) or that can write data into a *text file* (external report).

EPLAN also offers another special type of report. You can, for instance, directly place a motor plug as a report onto the page containing the motor. These types of reports are *manually placed*, and the versatile form design provides the user with a rich variety of custom uses.

Since EPLAN keeps all the data online and automatically updates the connection data before generating a report, the user does not need to update manually.

6.2 Report types

As mentioned previously, EPLAN distinguishes between normal report pages, embedded reports and frozen reports. Normal report pages are automatically newly created in the project based on their settings, whereas embedded reports are manually placed directly on the page on which the information (e.g. pin assignments, etc.) is to be displayed.

Frozen reports are automatically generated once and then “frozen” so that these reports can no longer be changed via an update, neither automatically nor manually. Frozen reports (the report pages) must be deleted and generated again if they need to be updated.

6.3 Types of graphical reports

All graphical reports are based on a particular report type. Every form has a specific area of application for form properties. However, not all form properties can be used in every type of form. A number of specific form properties can, however, be used in a range of different forms.

This section lists all possible report types with a brief description of each one.

6.3.1 Report types (forms)

EPLAN manages different **report types (forms)** for the reports. Broadly speaking, report types are distinguished by their file extension **.fnn*, where *nn* is a number.

- F**01** Parts list
- F**03** Device tag list
- F**13** Terminal diagram
- F**22** Plug diagram

NOTE: Forms should **always** be edited using the internal EPLAN form editor because, when editing and saving forms, EPLAN also saves internal information in the form to allow later master data synchronization to be correctly performed.

6.3.1.1 Parts list *.f01

A parts list contains a list of all the parts but not the totals. This form corresponds to a normal single parts list.

Parts list

designation	part number	Quantity	Device Designation
Mini contactor	MOE.010042	1	==Example=Chapter++06+6.3.2
Auxiliary contactor	SIE.3RH1122-1BB40	1	==Example=Chapter++06+6.3.2
Auxiliary contactor	ABB.FPH1421001T8440	0	==Example=Chapter++06+6.3.2

Fig. 6.3
Sample parts list
(file extension *.f01)

One special feature of this form should be noted. Parts with a quantity of “0” are also initially output by default. Unplaced parts are also listed in the standard forms. If this data should **not** be output, then you can use a filter to limit the contents of the output (see also section 6.3.1.2, “Summarized parts list*.f02”).

6.3.1.2 Summarized parts list *.f02

The summarized parts list is the output of all parts, but unlike the parts list, all of the same parts are output here as a total. This corresponds to a part quantity list.

Summarized parts list

supplier	Order number	Quantity	designation	Type number
MOE				
	010042	2	Mini contactor	DILER-22-G
SIEMEN				
	3RH1122-1BB40	0	Auxiliary contactor	3RH1122-1BB40
ABB				
	FPH1421001T8440	2	Auxiliary contactor	FPH1421001T8440

ample handbook

Fig. 6.4
Sample summarized parts list
(file extension *.f02)

The same applies for this form: Parts with a quantity of “0” are output by default. In the standard forms supplied with EPLAN, unplaced parts are also listed by default. If this data should **not** be output in the summarized parts list, then you again have the option of using a filter to limit the data that is included in the output.

6.3.1.3 Device tag list *.f03

The device tag list allows you to output all the devices used in a project, along with their information.

Device tag list

device tag part number Type number	function text Article designation	X-Ref	symbol
-3B1 FES.150402 SIEN-M12B-PS-K-L	Proximity switch Proximity switch (NO contact)	/3.3	
-3H1 TEL.XB5 AV61	Indicator light	/3.0	

Fig. 6.5
Sample device tag list
(file extension *.f03)

Whether or not a graphical representation of the corresponding component is to be displayed depends on the form used and/or the **<13050 Component graphic>** property. By default, unplaced devices are also output in forms, or you use the corresponding filter functions to output only the project data that is required or desired.

6.3.1.4 Forms documentation *.f04

Forms documentation provides an overview of which forms are stored and used in a project.

Fig. 6.6
Sample forms documentation (file extension *.f04)

This form has no additional filter possibilities, i. e. all forms stored in the project are output.

6.3.1.5 Device connection diagram *.f05

A device connection diagram allows the representation of a device with all its device connections and the internal/external devices connected to these.

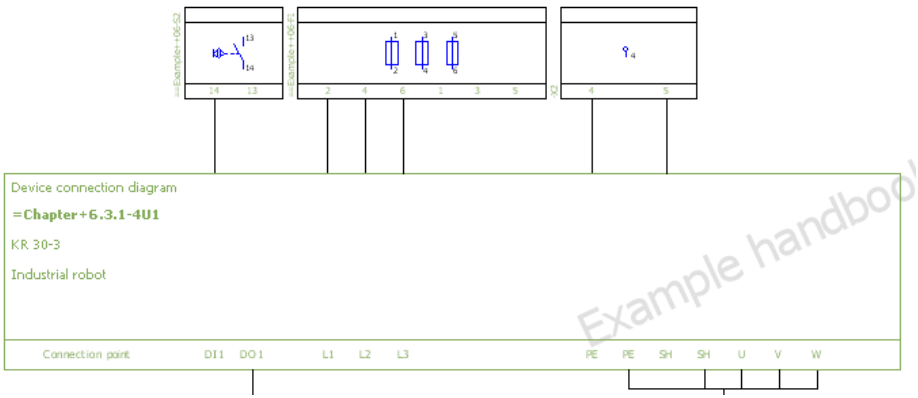


Fig. 6.7
Sample device connection diagram (file extension *.f05)

With this form and others, EPLAN offers the option of displaying the connected components over several connection levels on the internal and external pages.

In this form, internal connection points are not device-internal circuit diagrams, etc.

6.3.1.6 Table of contents *.f06

The table of contents is one of the typical uses for forms in EPLAN. The table of contents, also called a drawing directory, generates a list of the complete project documentation (the project pages).

Table of contents

Page	Page description
==ALL=DOC+DBL/1	Titel- / Deckblatt
==ALL=DOC+IHV/1	Table of contents
==ALL=DOC+IHV/1.a	Table of contents
==ALL=Example+Page/1	Parts list (Report)

Fig. 6.8 Sample table of contents (file extension *.f06)

As with other forms, the table of contents has a number of functions, such as filters, sortings, etc., available for controlling the graphical output to suit your needs.

6.3.1.7 Cable connection diagram *.f07

The cable connection diagram generates a form for a single cable (device) with all internal and external targets that are connected to the corresponding cable conductors.

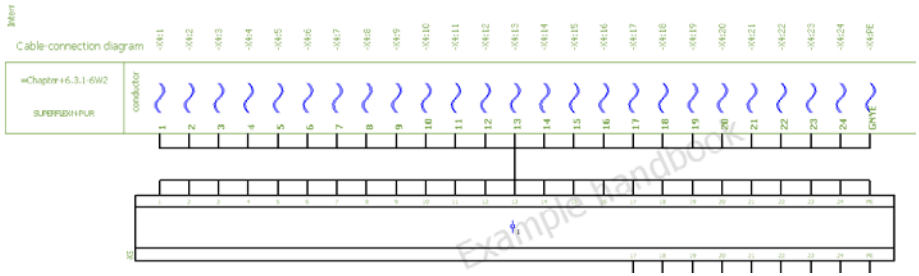


Fig. 6.9 Sample cable connection diagram (file extension *.f07)

The cable connection diagram belongs to the series of forms that can display the targets (external/internal) over several levels. Once again, predefined or user-defined filters can be used to control the output of the cable connection diagram and thus display only the required project cables.

6.3.1.8 Cable assignment diagram *.f08

The cable assignment diagram is the only form that cannot be generally entered for all cables in the *Output to pages* setting.

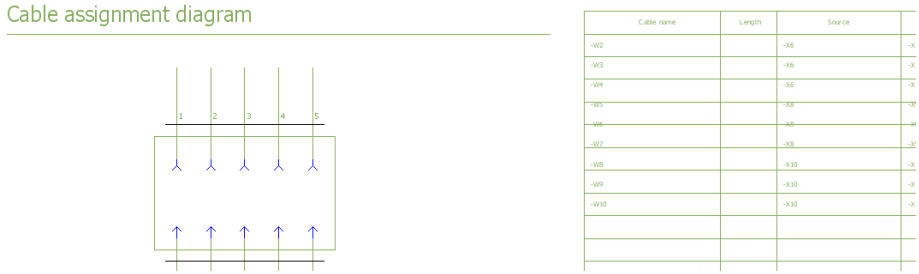


Fig. 6.10 Sample cable assignment diagram (file extension *.f08)

This is a special form that can be used, for example, to display a detailed representation of the internal structure of a cable together with a listing of the corresponding cables in the project (including various other information such as the cable device tag). Everything is displayed together on a single report page.

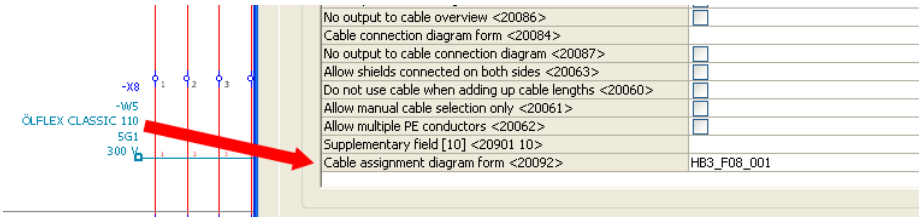


Fig. 6.11 Cable assignment diagram form entry on a cable

Since different cable types are usually used in a project, it makes no sense to make a global entry for a cable assignment diagram or a global selection for all cables in the settings.

The cable assignment form can be selected directly from the selection list in the cable properties (the property value would be **<20092 Cable assignment diagram form>**) or saved with the part in parts management (*Cable data* tab).

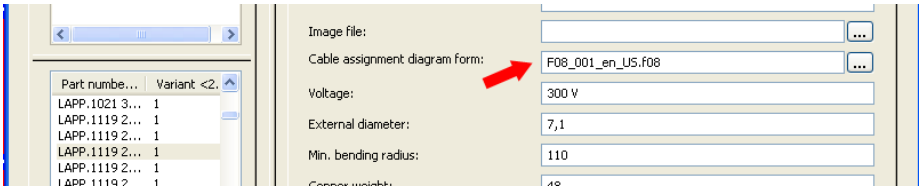


Fig. 6.12 Cable assignment diagram form entry in parts management

6.3.1.9 Cable diagram *.f09

The cable diagram report is the representation of a single cable with all its conductors and their information.

Cable diagram

Cable name	=Chapter+6.3.1-9W11		cable type		OLFLEX CLASSIC 110
function text	supply 24V DC		no. of conductors		12G
function text	Page / column	Target designation from	Connection point	conductor	Target designation to
supply 24V DC	= =Example++06/9.0	= =Example++06-X12	1	1	= =Example++06-X13
signal	= =Example++06/9.1	= =Example++06-X12	2	2	= =Example++06-X13

Example handbook

Fig. 6.13 Sample cable diagram (file extension *.f09)

Depending on the structure of the form used, you can display information such as the internal and external targets of the connected devices. You can use filter settings to control the output of the cable diagram, etc. Without a filter setting, all cables used in the project, for example, would be output.

6.3.1.10 Cable overview *.f10

By default, a cable overview contains a listing of all cables used in the project with all the desired information such as cable type, number of conductors, etc.

Cable overview

cable type	Cable description	device tag	Condctrs	cross-se
ÖLFLEX CLASSIC 100 H	Cable halogen free, 5G2,5	-W1	5G	
ÖLFLEX CLASSIC 110	Control line, 5G1.0	-W2	5G	
	Control line, 5G1.0	-W3	5G	
	Control line, 5G1.0	-W4	5G	

Fig. 6.14 Sample cable overview (file extension *.f10)

You can again control the output by setting filters or use sorting to, e. g. display the cables by higher-level function or mounting location.

6.3.1.11 Terminal connection diagram *.f11

The terminal connection diagram belongs to the category of reports that can display several levels of connected internal and external targets at the corresponding terminal strip.

Terminal-connection diagram

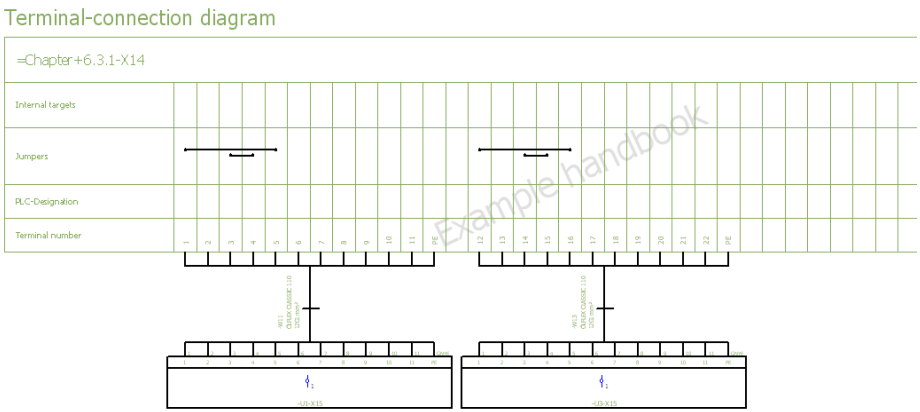


Fig. 6.15 Sample terminal connection diagram (file extension *.f11)

The number of levels to be displayed can be defined by the form structure and/or in the form properties. Once again, this form has extensive filtering and sorting functions.

6.3.1.12 Terminal line-up diagram *.f12

The terminal line-up diagram generates a graphical report for each terminal strip with the associated terminal (strip) parts.

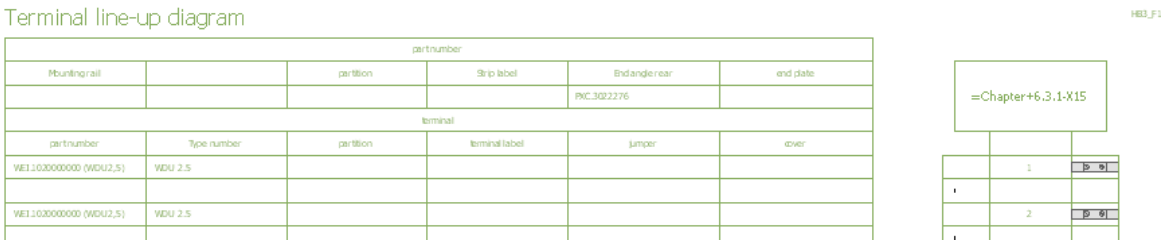


Fig. 6.16 Sample terminal line-up diagram (file extension *.f12)

The terminal line-up diagram can be graphically structured, for example, in a similar way to a terminal diagram, or you can use graphics to represent the parts and insert these visually into the form as properties. You can also use the filter and sorting functions to selectively control the output in this report.

6.3.1.13 Terminal diagram *.f13

You can use a terminal diagram to output a single terminal strip with connected internal and external targets, for example, including the cables used.

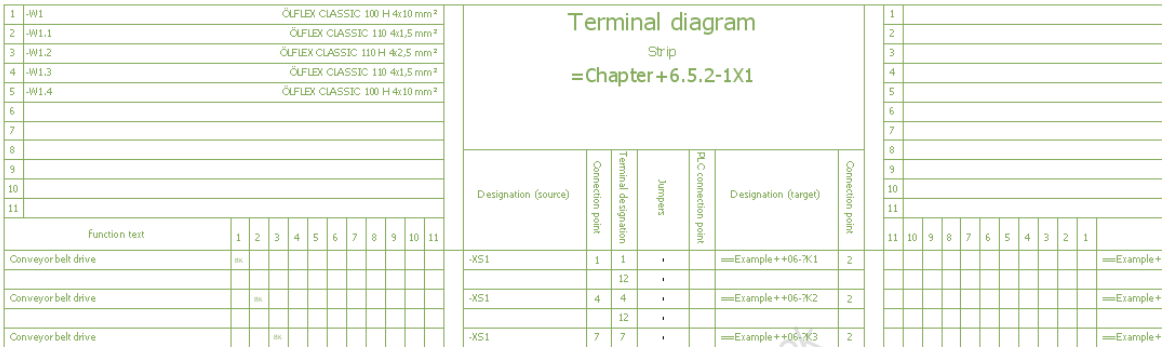


Fig. 6.17 Sample terminal diagram (file extension *.f13)

Terminal diagrams can be controlled via the numerous filter and sorting options. The structuring of the terminal diagrams is also very versatile.

6.3.1.14 Terminal strip overview *.f14

You can use the terminal strip overview form to obtain an overview of all terminal strips used in the project.

Terminal-strip overview

terminal strip	function text	Terminals			
		first	last	Total PE	Total N
		12	PE	4	0
-U1-X15	terminal 400V AC	1	PE	1	0
-U2-X15		1	PE	1	0
-U3-X15		1	PE	1	0
-X2	Power 690V AC	1	5	0	0

Fig. 6.18 Sample terminal strip overview (file extension *.f14)

In order to control the output of the desired terminal strips while editing the project, you can either enter the **<20857 No output to terminal strip/plug overview>** property into the terminal strip definition or limit the output later by using a filter. As with many other forms, actions such as sorting using predefined or user-defined filters are also possible here.

6.3.1.15 Plot frame documentation *.f15

The plot frame documentation form is used to obtain an overview of all plot frames stored in the project.

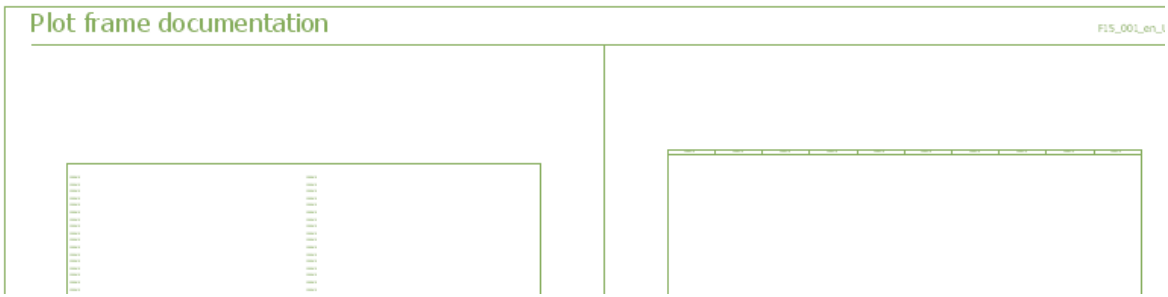


Fig. 6.19 Sample plot frame documentation (file extension *.f15)

There are no filtering or sorting options available for plot frame documentation forms. This always produces a total output.

6.3.1.16 Potential overview *.f16

The potential overview report lists all potentials and signals used in the project, as well as further information, depending on the settings and structure of the form.

Potential overview

HB3_F16_001

Name of potential	Potential value	Frequency	Potential type	Placement
0V DC			-	/17.3
24V DC	24V DC		+	/17.2
L1	400	50		++08+8.6.1/2.6
L1				++07+7.3.4/1.1

Fig. 6.20
Sample potential overview
(file extension *.f16)

You can use the filter and sorting settings to control the output of the graphical report.

6.3.1.17 Revision overview *.f17

You use the revision overview form to obtain a graphical overview of the existing revisions and their corresponding entries such as the revision creator or the revision index.

Revision overview

Project name Revision control (change tracking)
Date 1/8/2011
revision Revision 3

Revision name Index	Revision comment (page) Revision description (page)	Page name Page description
Revision 3	Index 5 / - End of project	=A1+O1/3 Table of contents : =A1+O1/1 - =REPORT+REVUEB/1
Revision 3	Index 5 / - End of project	=A1+O1/3 Table of contents : =A1+O1/1 - =REPORT+REVUEB/1
Revision 3	Index 5 / - End of project	=A2+O2/1
Revision 3	Index 5 / - End of project	=A2+O2/2

Fig. 6.21
Sample revision overview
(file extension *.f17)

The form can evaluate the respective page properties, e. g. the property **<11073 1 Reason for revision change>** or even associated project properties such as the property **<10155 1 Revision name>**.

There are no additional filter or sorting possibilities for the revision overview, apart from the normal settings for graphical output.

6.3.1.18 Enclosure legend *.f18

In a panel layout, you can include a legend of the items (parts) placed, for example, on the mounting panel. The enclosure legend form in EPLAN allows this task to be automatically completed using the project data.

Enclosure legend

==Example=Chapter++06+6.3.1-M1

item number	device tag	Type number
1	3H1	XB5 AV61
2	1K1	DILER-22-G
3	3Q1	3RV10 21-1JA15
4	3B1	SIEN-M12B-PS-K-L

Fig. 6.22
Sample enclosure legend
(file extension *.f18)

An enclosure legend can be graphically generated as a type of window legend (embedded as a manual placement in a page) or as completely new report pages. Other filter and sorting options are available for graphical output using this form.

6.3.1.19 PLC diagram *.f19

In addition to the familiar (manual) logical PLC (card) overview, EPLAN can also use the existing PLC project data to automatically generate these in a similar graphical form, including additional information.

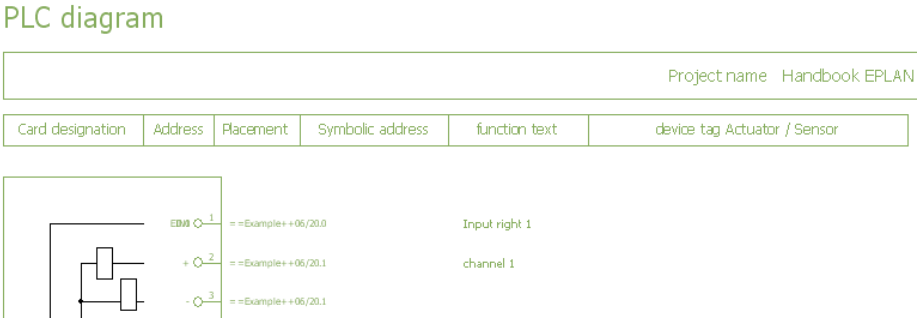


Fig. 6.23 Sample PLC diagram (file extension *.f19)

Data from the PLC modules, and other data such as rack or module number, path function texts and automatically determined symbolic addresses are all taken into account.

You can again use numerous familiar filter and sorting options with PLC diagram forms.

6.3.1.20 PLC card overview *.f20

The PLC card overview provides a clear overview list of all PLC components used, such as power supply, CPU or input cards.

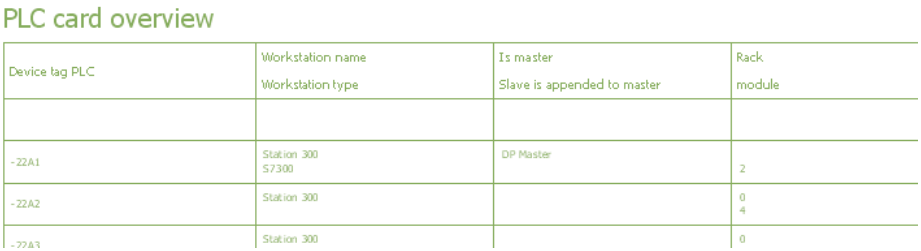


Fig. 6.24 Sample PLC card overview (file extension *.f20)

You can, of course, display each PLC component on its own graphical report page of type PLC card overview. This depends on the structure of the PLC card overview form. Once more, you can also use filters and sorting to affect the graphical output in PLC card overview forms.

6.3.1.21 Pin connection diagram *.f21

Pin connection diagrams belong to the category of graphical reports that can manage several levels of internal and external targets and can output these graphically (depending on the form and settings used).

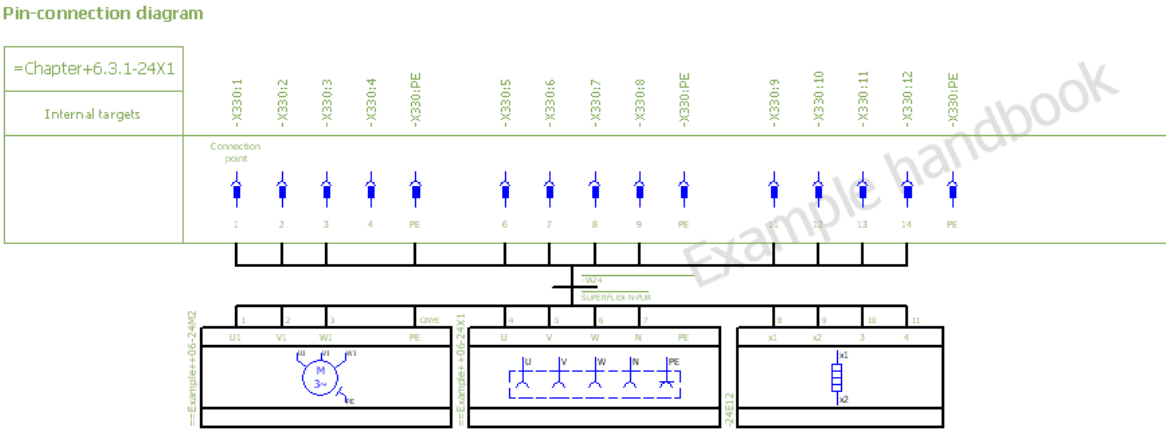


Fig. 6.25 Sample pin connection diagram (file extension *.f21)

A graphical output usually also includes a pin connection diagram. These can also be set to dynamic in the settings so that several pin connection diagrams can be displayed on a single page (combined report). You can adjust the graphical output via filters and sorting settings.

6.3.1.22 Plug diagram *.f22

The plug diagram is a listing of all the internal and external targets connected to a plug with the maximum evaluation of one level. This is the normal evaluation of the standard forms.

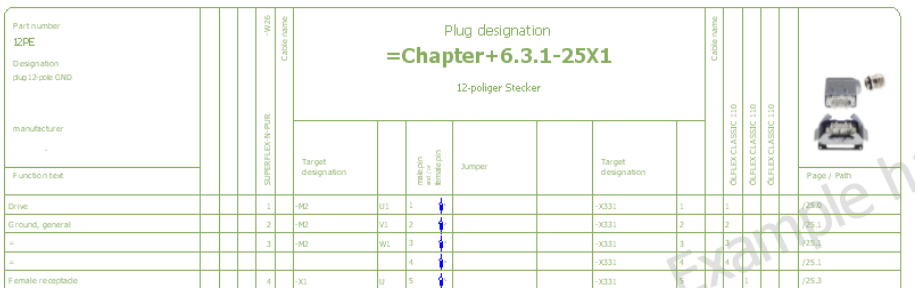


Fig. 6.26 Sample plug diagram (file extension *.f22)

The plug diagram can also be set to dynamic in the form properties so that several plugs are graphically output combined (consecutively) on a single page.

You can also use the familiar filter and sorting options to, for example, limit the output for the plug diagram.

6.3.1.23 Plug overview *.f23

The plug overview provides a listing of all plugs and/or sockets used in the project, depending on the structure of the plug overview form.

Plug overview

Plug designation	function text	pin			Page of plug connection diagram
		Normal	N	PE	
+2-XS1		3	0	1	
-24X1	Hybrid cable	12	0	3	
-25X1		12	0	1	

Fig. 6.27 Sample plug overview (file extension *.f23)

Like almost all forms, you can sort and filter the plug overview in many ways.

6.3.1.24 Structure identifier overview *.f24

EPLAN can manage the various structure identifiers such as functional assignment (==), higher-level function (=), or mounting location (+). You can assign additional descriptions to every structure identifier in EPLAN. You use the structure identifier overview to obtain an overview of all the structure identifiers assigned (or also unused) in the project.

Structure identifier overview HB3_F24_001

Full designation	Structure description
Funktionale Zuordnung	
==ALL	General
==Example	Sample
=DOC	Info
=Example	Sample
=Chapter	chapter

Mounting location	
+7.3.4	subchapter number 7.3.4 in the book
+8	subchapter number 8 in the book
+8.6	subchapter number 8.6 in the book
+8.6.1	subchapter number 8.6.1 in the book

Fig. 6.28 Sample structure identifier overview (file extension *.f24)

No filters are possible for the output, but a range of sorting options are available.

6.3.1.25 Symbol overview *.f25

A symbol overview is used to display the symbol libraries used and/or saved in a project. Depending on the form property settings you can, for example, only output the symbols used in the project (Form property **“Output only used symbols <13114>”**).

Symbol overview SPECIAL_en_US F25_001_en_US

0 DC Black box Black box	8 BP 1 interruption point	16 CABDCP2 Cable definition, two-sided Cable definition
1 PLCC PLC box PLC box	9 BPOL 1 interruption point (without line)	17 DCP Device connection point Device connection point

Fig. 6.29 Sample symbol overview (file extension *.f25)

Simple filtering and sorting options are available for the symbol overview.

6.3.1.26 Title page/cover sheet *.f26

The title page/cover sheet is usually the page with which a project begins. EPLAN allows the user to automatically generate these pages.

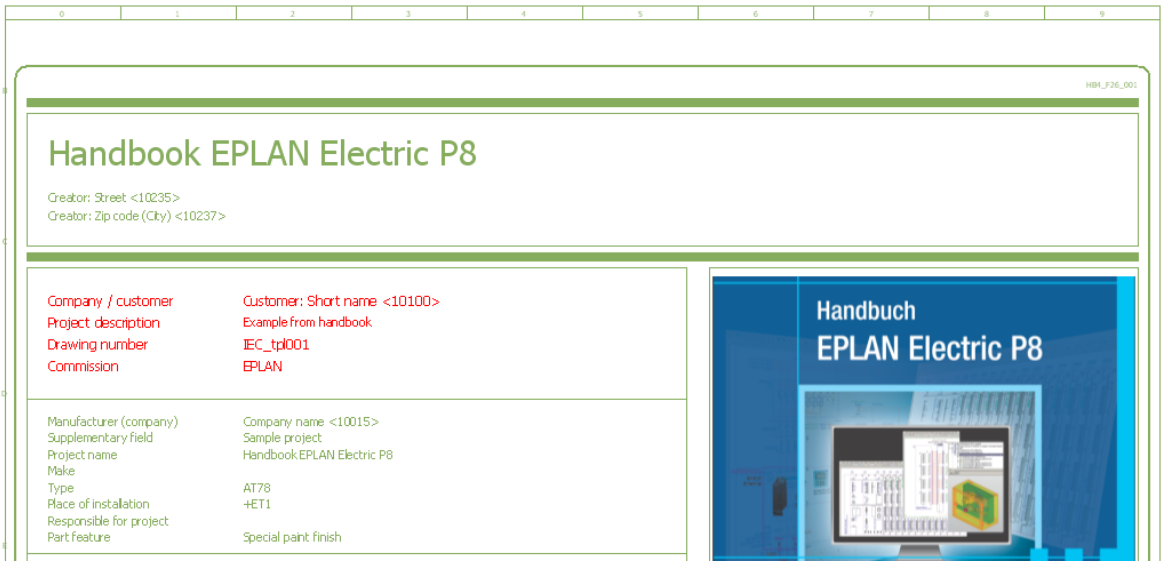


Fig. 6.30 Sample title page/cover sheet (file extension *.f26)

Various project or page properties can be assigned to the form. For example, you can output a title page/cover sheet for the entire project documentation and a cover sheet for each individual higher-level function. Additional filters cannot be used or evaluated.

6.3.1.27 Connection list *.f27

The connection list form generates a report containing all the project connection data.

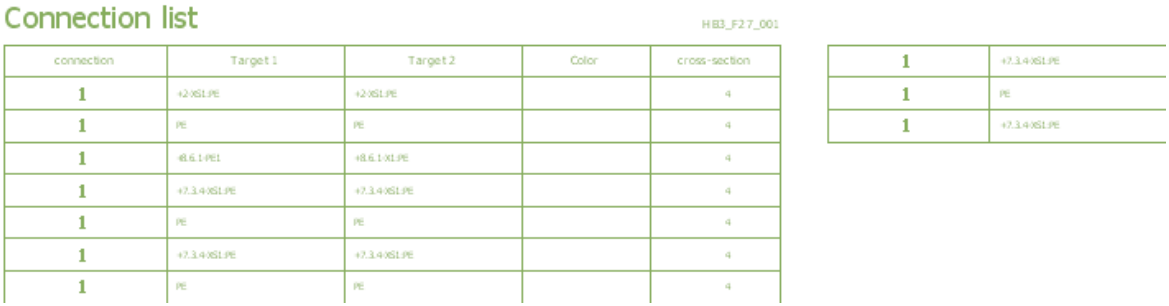


Fig. 6.31 Sample connection list (file extension *.f27)

The output can also be adjusted via existing or user-defined filters and sorting settings.

6.3.1.28 Graphic *.f28

The graphic form is not a “real” form in the sense of a report. I mention it here for the sake of completeness. Graphical forms have the file extension *.f28 and can be entered into the page properties as a “placeholder form graphics”.

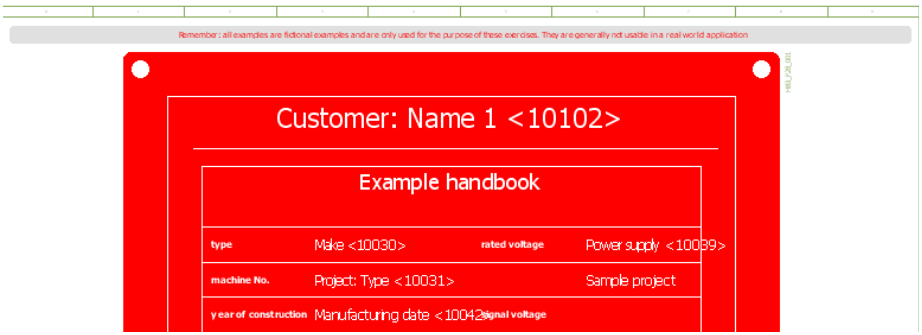


Fig. 6.32 Sample graphical form (file extension *.f28)

For example, you can generate a report for specific pages and fill it with specific information on page and/or project properties.

Since there are no report templates for a graphical “report”, the form is entered directly in the page properties.

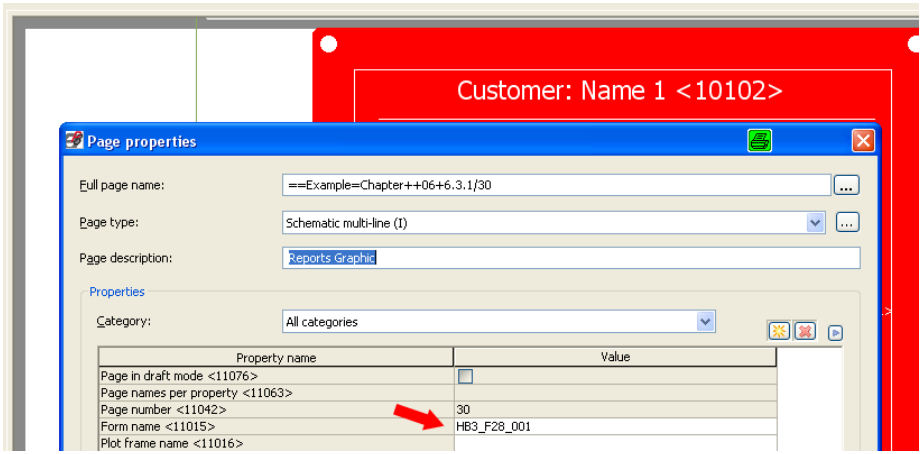
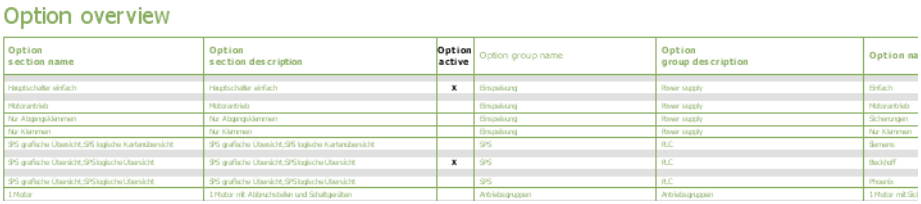


Fig. 6.33 Entering a form in the page properties

6.3.1.29 Project options overview *.f29

The project options overview report contains information on possible project options.

This can be information about the project options themselves, the project option segments or information about placeholder objects with the appropriate value sets. With this form, you can use a range of filter and sorting options for the graphical output.


Option overview

Option section name	Option section description	Option active	Option group name	Option group description	Option na
Hauptschalter einfach	Hauptschalter einfach	x	Einspeisung	Power supply	Einfach
Motorantrieb	Motorantrieb		Einspeisung	Power supply	Motorantrieb
Nur Abgangsklemmen	Nur Abgangsklemmen		Einspeisung	Power supply	Sicherungen
Nur Klemmen	Nur Klemmen		Einspeisung	Power supply	Nur Klemmen
SPS grafische Übersicht,SPS logische Kartenübersicht	SPS grafische Übersicht,SPS logische Kartenübersicht		SPS	PLC	Siemens
SPS grafische Übersicht,SPS logische Übersicht	SPS grafische Übersicht,SPS logische Übersicht	x	SPS	PLC	Beckhoff
SPS grafische Übersicht,SPS logische Übersicht	SPS grafische Übersicht,SPS logische Übersicht		SPS	PLC	Phoenix
1 Motor	1 Motor mit Abzweigklemmen und Schaltgeräten		Antriebsgruppen	Antriebsgruppen	1 Motor mit Sic

Fig. 6.34 Sample project options overview (file extension *.f29)

6.3.1.30 Placeholder object overview *.f30

The placeholder object overview report contains information about the placeholder objects used and their different value sets.

Placeholder object overview

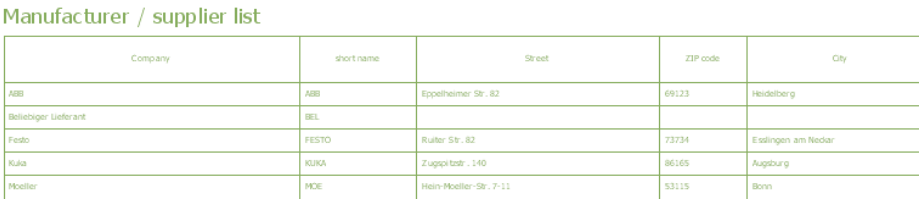
Placeholder object: Last value set selected	Placeholder object name	Placement	Placeholder object: Last value set selected
		+SYMB	

Fig. 6.35 Sample placeholder object overview (file extension *.f30)

You can control the output via filters and sorting options.

6.3.1.31 Manufacturer/supplier list *.f31

The manufacturer/supplier list report enables the specific listing of the manufacturers and/or suppliers used.


Manufacturer / supplier list

Company	short name	Street	ZIP code	City
ABB	ABB	Eppelheimer Str. 82	69123	Heidelberg
Beliebiger Lieferant	BEL			
Festo	FESTO	Ruiter Str. 82	73734	Esslingen am Neckar
Kuka	KUKA	Zugspitzstr. 140	86165	Augsburg
Moeller	MOE	Hein-Moeller-Str. 7-11	53115	Bonn

Fig. 6.36 Sample manufacturer/supplier list (file extension *.f31)

EPLAN's usual filter and sorting options are available here as well.

The following reports are special reports for other add-ons such as EPLAN Pro Panel, PPE or even the new Topology add-on (FieldSys). I list them here for the sake of completeness.

6.3.1.32 Mounting list *.f32

The mounting list report lets you list the structure of a mounting layout in order to map specific part placements in the EPLAN Pro Panel layout space.

6.3.1.33 PCT loop list *.f33

The PCT loop list report lets you clearly list the data of a loop or consumer. The listed data includes functions, loop data and even the medium data of a loop or consumer.

6.3.1.34 Topology: Routing path list *.f34

The Topology: Routing path list allows you to generate, for example, an overview of the routing path and their routing points and the associated data.

6.3.1.35 Topology: Routing path diagram *.f35

The Topology: Routing path diagram report generates a list of the individual paths and associated data such as cables and connections.

6.3.1.36 Topology: Routed cables/connections *.f36

The Topology: Routed cables/connections outputs all routed cables and connections in the current project.

6.3.1.37 Process overview *.f37

The Process overview report outputs the media, states and properties such as pressure, flow, etc. that are defined in a process-engineering process.

6.3.2 Special connection diagrams

In addition to the previous report types, EPLAN has special connection diagrams. These diagrams depict connection overviews between devices, independent of size and placement on the mounting panel.

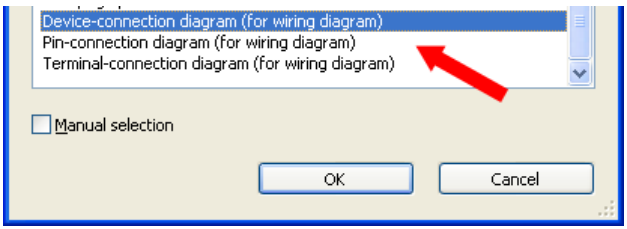


Fig. 6.37 Special connection diagrams

EPLAN offers connection diagrams for the following reports: device connection points, terminal connection points, and pin connection points. A prerequisite for generating connection diagrams is that the devices must be placed on a mounting panel. Fig. 6.38 and Fig. 6.39 show examples of such connection diagrams.

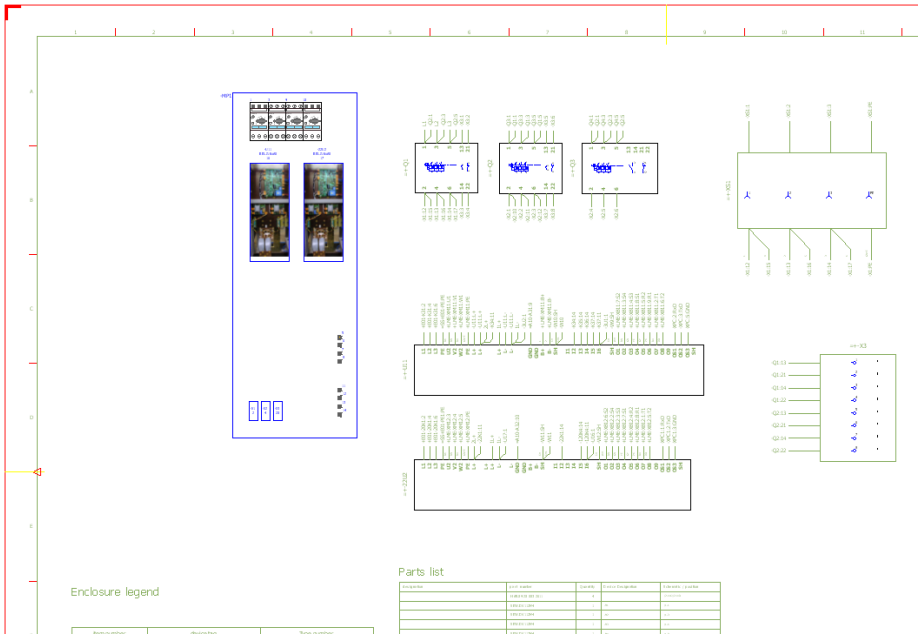


Fig. 6.38 Connection diagrams (here device connection points)

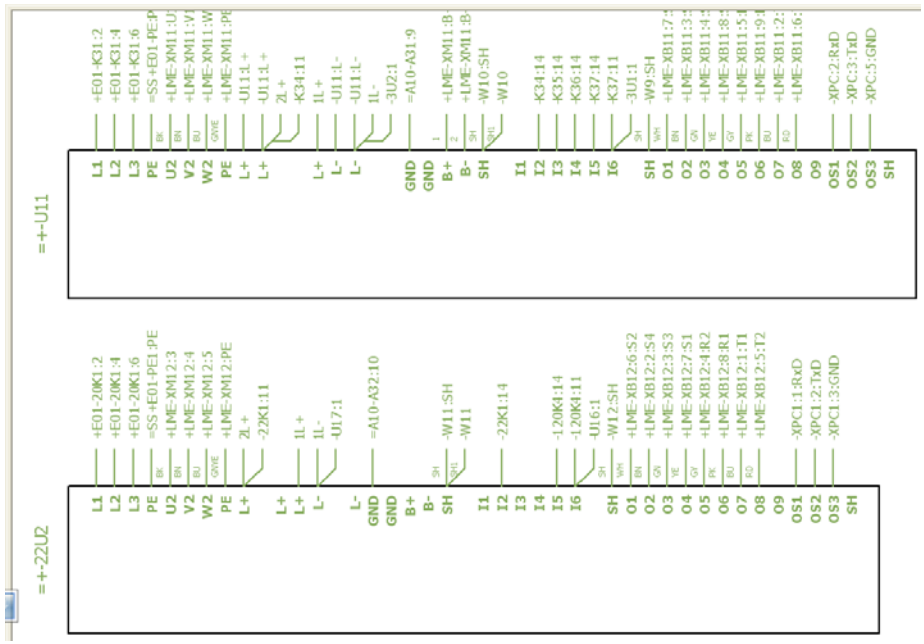


Fig. 6.39 Connection diagram showing detached representation of a device's connection points

6.4 Settings (output options)

Before generating reports for a project, you will typically need to adjust certain settings to suit your requirements. These settings relate only to the output of data such as the form settings, or how EPLAN is to handle path function texts in the forms.

NOTE: You do not need to perform report runs or the like, because EPLAN keeps all data online, and this is therefore always up to date. The system automatically updates the connections before EPLAN generates a report.

6.4.1 The Display/output project setting

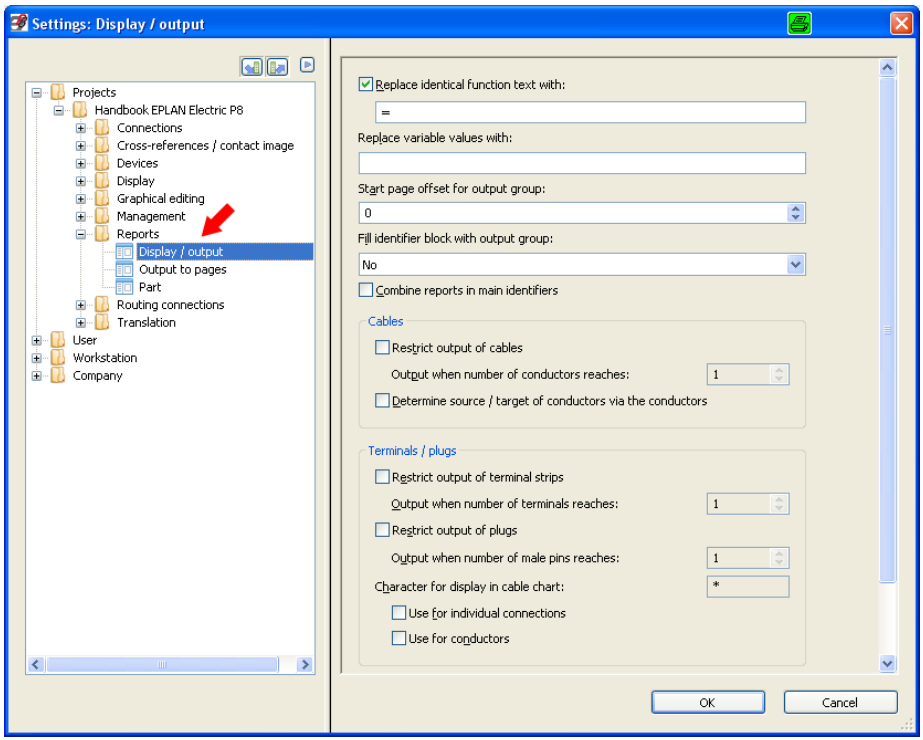


Fig. 6.40 Settings for the display and output of reports

These settings are used to define basic information for graphical evaluation of the forms. You can influence the way in which device tags are displayed in reports or from what number of data records particular reports should be output. You will find the *Display/*

output settings in the **OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/REPORTS/DISPLAY/OUTPUT** menu item.

Replace identical function text with: Often, for example for a row of terminals, function text is repeated in the report form. If this type of graphical output is not wanted, then here you can define one or more characters to represent the repetition of the function text. The equals sign "=" is often used for this.

Replace variable values with: This setting is only valid for summarized parts lists. It takes effect when placeholder objects with different value sets are used in a project. In the *summarized parts list* report, EPLAN can then replace the current placeholder text data with the data from this field (e. g. a reference to an extra page in the project). A precondition for this is that the **<13108> Replace variable values with text** property is activated in the form.

Start page offset for output group and *Identifier block for output groups:* These settings are for when you use the **Output module** component property. Here you can add a report variant when outputting reports. For these settings to take effect, certain preconditions are necessary. One of these is that there is an entry in the **Output group <20033>** property for the component in the **SYMBOL PROPERTY** dialog. If this property is not filled in, EPLAN cannot output a report by output group.

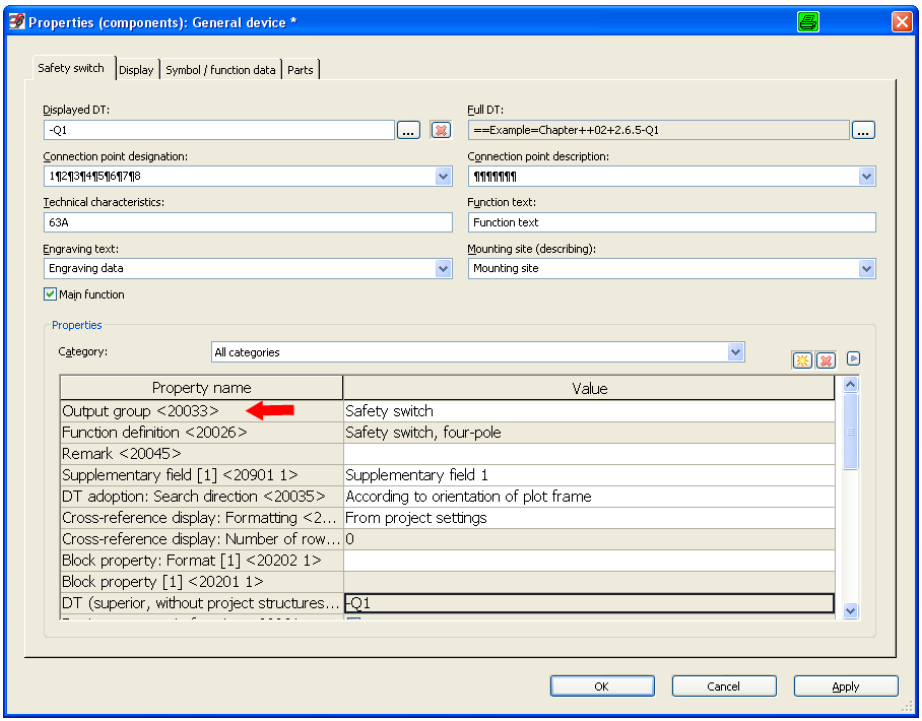


Fig. 6.41 Output group <20033> component property

Combine reports in main identifiers: When this setting is activated, EPLAN lists the graphical reports for the project data one after the other in the form. If the identifier changes, then it is immediately placed after the last different identifier in the graphical report without EPLAN generating a page break in the graphical output. When the setting is not active, EPLAN generates a page break in the graphical report when a main identifier changes.

Other settings for *Terminals/cables/plugs:* This allows specific project data to be output only when it possesses a minimum number of data records. If, for example, a limit of five conductors is set for cables, then all cables with less than five conductors are not included in the graphical report. The default values are set to *not active* and they should usually remain so.

Character for display in cable chart: If the correct display of the cable conductors (or individual connections) in the terminal diagram and their **conductor color** property values is not important, then instead of the correct color designation, you can enter a value here to be entered instead of the conductor color in the graphical output.

For example, this could be an "X". Multiple characters can be entered, but in practice one character is better since it must also fit in the available space on the forms.

6.4.2 The Parts project setting

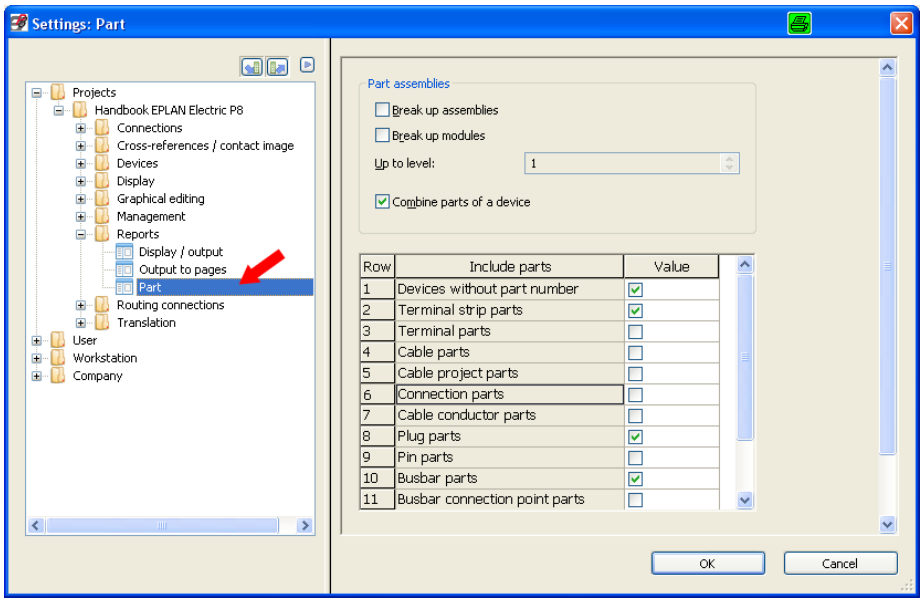


Fig. 6.42 Settings for the output of parts

This dialog contains settings that define how EPLAN is to handle parts when outputting graphical reports. You will find the settings for viewing parts in the **OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/REPORTS/PARTS** menu.

These settings are mainly intended for evaluating forms related to **parts** project data. These would be parts lists or summarized parts lists, for example. These settings also apply to displays in navigators, such as the bill of materials navigator.

Part assemblies/Break up assemblies & modules: You can use the level setting here to define the level to which EPLAN should break up assemblies (and assemblies within assemblies) and modules when generating graphical reports.

Combine parts of a device: This setting combines, for example, the terminals of a terminal strip in a parts list so that not every terminal part is listed individually in the parts list.

Include parts: You can influence the graphical output for each type of part here by activating the individual options such as terminal parts or cable parts.

6.4.3 The Output to pages project setting

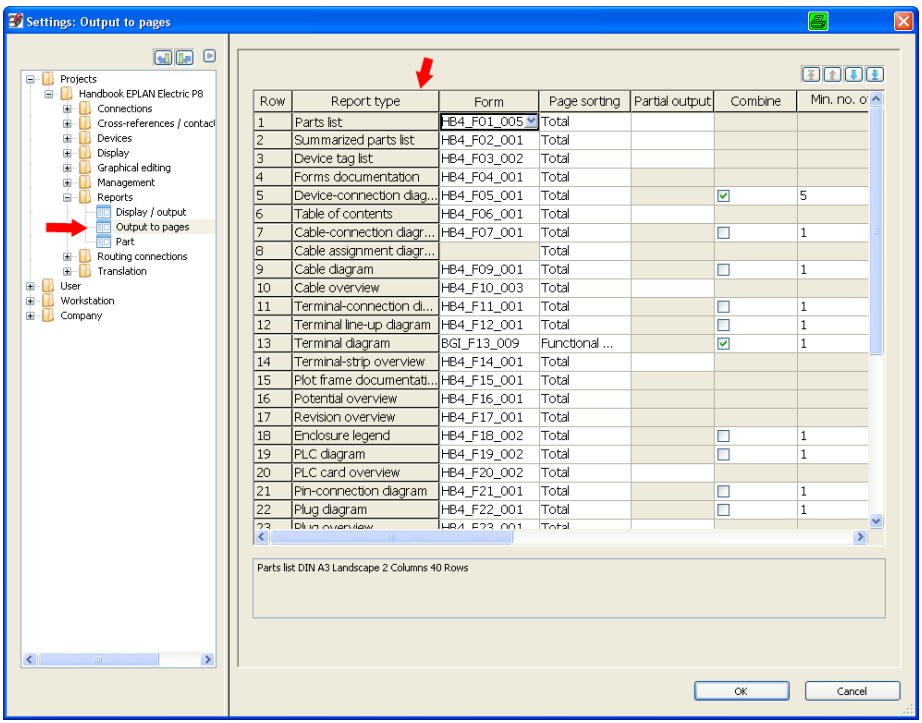


Fig. 6.43 Settings for the output to pages

Forms are a fundamental element in outputting reports. They are always set on a project-specific basis via **OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/REPORTS/OUTPUT TO PAGES** or adopted from the templates or basic projects as default settings for the project. These settings define a number of global default conditions for graphical output of the project data.

6.4.3.1 The Report type column

The **Report type** column cannot be changed in this table. EPLAN provides different default report types. This does not mean that all these reports must be generated for a given project. The user can decide whether only one terminal diagram or the full palette of all possible reports should be output for the schematic.

Row	Report type
1	Parts list
2	Summarized parts list
3	Device tag list
4	Forms documentation
5	Device-connection diagram
6	Table of contents
7	Cable-connection diagram
8	Cable assignment diagram

Fig. 6.44
Report types

6.4.3.2 The Form column

The desired form can be set in the **Form** column. Click the **FORM** row and then click the **BROWSE** button in the selection list that appears. The **SELECT FORM** dialog opens. Now you can select another form from the directory and press the **OPEN** button to load it into the **Form** column.

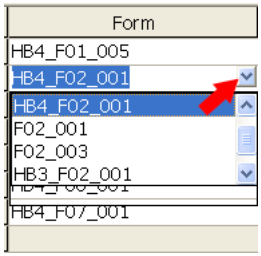


Fig. 6.45
Forms

TIP: The form is automatically stored in the project if it does not already exist in the project master data. Initially it makes no difference from which directory the new form was selected.

6.4.3.3 Page sorting column

The **Page sorting** column plays a decisive role in defining how the graphical output pages are later sorted into the existing page structure of the project. Selecting *Total* from the *Sort by/into* selection list causes all reports to be summarized under the selected total identifier. If you want, for example, to output all terminal diagrams by mounting location, then you should select the *Mounting location* entry.

NOTE: Not all types of page sorting are available for all report types. It is not possible, for example, to output forms documentation by higher-level function and mounting location as this does not make sense.

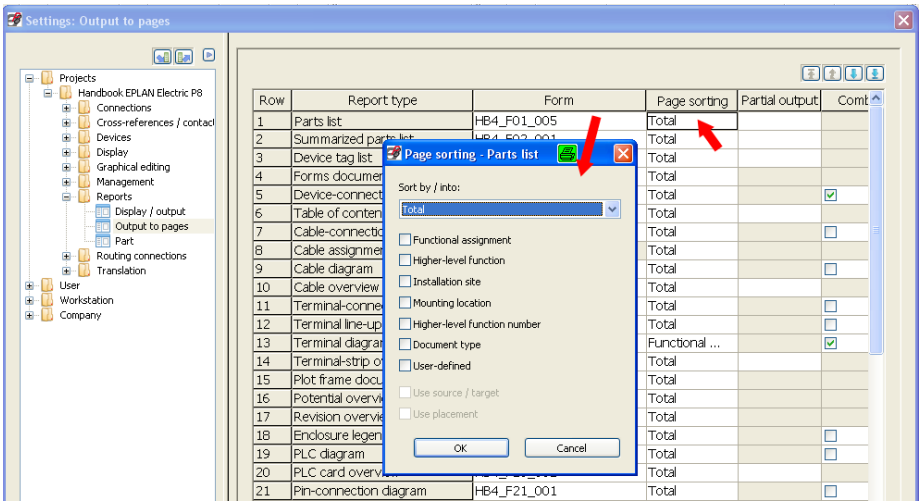


Fig. 6.46 Page sorting

6.4.3.4 Partial output column

Partial output is an interesting setting. For reports where this is possible, you can automatically generate a partial cover sheet for each higher-level function in addition to a main form such as a total cover sheet. For this to work properly, the page sorting must also be properly set, to *Total + Higher-level function*, for example. Using this setting, EPLAN then generates a (total) cover sheet and a (partial) cover sheet for each individual higher-level function. We explicitly point out here that the structure and graphical content of these forms can be different.

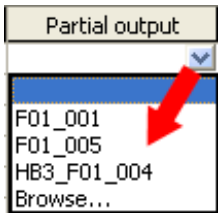


Fig. 6.47 Partial output

6.4.3.5 The Combine column

The **Combine** function is an ideal way, for example, to place several terminal strips that only have a few terminals on *one* graphical output page (limited only by the available space on the output page; when there is no more space EPLAN generates a new page as before).

The examples in Fig. 6.49 and Fig. 6.50 make this clearer. In Fig. 6.49, the **Combine** option was *not* selected. EPLAN generates one page of the terminal diagram report type for each *terminal strip*.

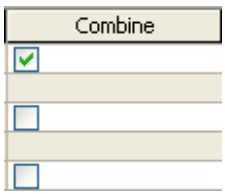


Fig. 6.48 Combining

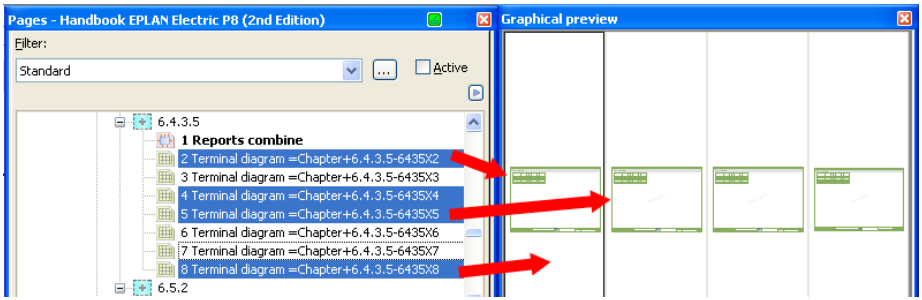


Fig. 6.49
Combine option not activated

In Fig. 6.50, the **Combine** option was *activated*. EPLAN now generates one or more report pages of the *terminal diagram* type for several terminal strips, limited only by the size of the plot frame and the set maximum number of data rows that EPLAN should display on a report page.

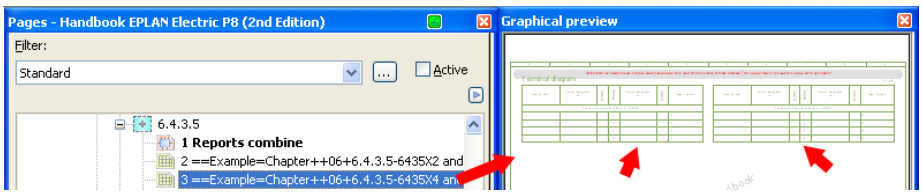


Fig. 6.50 Combine option activated

NOTE: The **Combine** option can only be used for dynamic forms. The *Combine* setting **cannot** be used for static forms.

6.4.3.6 The Minimum number of report rows column

The *Min. no. of rows on report page* setting specifies a particular number of data sets to be output before EPLAN generates a page break.

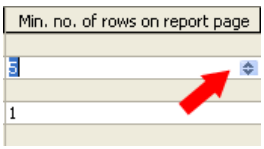


Fig. 6.51
Minimum number of rows

This setting only makes sense when used together with the **Combine** option. When the **Combine** option is not used, then changing the minimum number of rows has no effect on the graphical output.

The usual value is **1**. This places no limitations on the graphical output pages, and the project data is output consecutively. If the *minimum number* is now changed to **10**, then at least ten terminals of a terminal strip must be displayed on a single graphical output page before EPLAN can force a new page break.

6.4.3.7 The Subpage column

This setting defines whether or not EPLAN should generate subpages for the graphical output pages. Report pages in EPLAN do not have to be generated using consecutive integers; sometimes the actual page number for the report should not change. This means that *subpages* must be created for the graphical output pages.

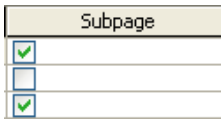


Fig. 6.52 Subpages

6.4.3.8 The Character column (for subpages)

This property only makes sense when used together with the **Subpages** property. The *Character* setting defines the format of the subpage. There are multiple selections available, but all subpages are generally separated from the main page with a point.

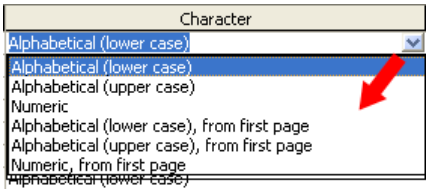


Fig. 6.53 Characters

- **Alphabetical (lower case)** → Report pages begin with **20; 20.a; 20.b; 20.c**
- **Alphabetical (upper case)** → Report pages begin with **15; 15.A; 15.B; 15.C**
- **Numeric** → Report pages begin with **301; 301.1; 301.2; 301.3**
- **Alphabetical (lower case), from first page** → Report pages begin with **5.a; 5.b; 5.c**
- **Alphabetical (upper case), from first page** → Report pages begin with **7.A; 7.B; 7.C**
- **Numeric, from first page** → Report pages begin with **43.1; 43.2; 43.3**

6.4.3.9 The Blank pages column

EPLAN can automatically maintain a certain spacing between the report pages and the actual schematic pages, the so-called *blank pages*. This setting defines the size of this spacing.

The last schematic page is 2. The blank pages setting is set to 5. 2 + 5 = 7; the report pages then begin with page number 8.

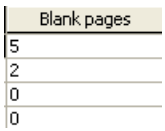


Fig. 6.54 Blank pages

6.4.3.10 The Round column

The *Round* setting relates to the setting for blank pages.

In the example, we round to 10. If the last schematic page 12 has a spacing of 5 (setting for blank pages results in 17), then the setting *Round* = *10* will cause the first report page to have a page number of 20.

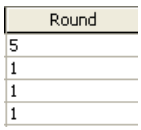


Fig. 6.55 Rounding

6.4.3.11 The Hide structure identifier if identical column

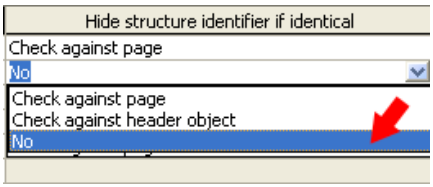


Fig. 6.56
Hide structure identifier if identical

This setting checks to see if structure identifiers should be included in the graphical output or suppressed when certain conditions are satisfied. This allows "abbreviation" of device tags (DTs) in the reports, meaning a check for corresponding identical entries.

There are three possible settings:

- *Check against page:* This causes the structure identifier of the DT to be checked against the structure identifier of the report page. If EPLAN finds a match here then the DT is shortened by this matching value in the report.
- *Check against header object:* This setting is intended for function-related reports such as terminal or cable diagrams. In this case, EPLAN checks the output DT against the header object of the report page (e.g. a terminal strip definition or a cable definition) and abbreviates the output DT by this structure identifier if it is the same.
- *No:* This setting causes the complete structure identifier to always be output.

6.4.3.12 The Synchronize column

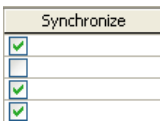


Fig. 6.57
Synchronize

Synchronize is an important setting for keeping project master data up to date, i.e. synchronized with the system master data. The Synchronize setting in EPLAN means that forms having synchronizing *active* (check box set) are automatically synchronized with the system master data when the project is opened.

NOTE: A precondition for synchronizing is that the **Synchronize project master data when opening** setting is set in the OPTIONS/SETTINGS/PROJECTS [PROJECT NAME]/MANAGEMENT/GENERAL menu.

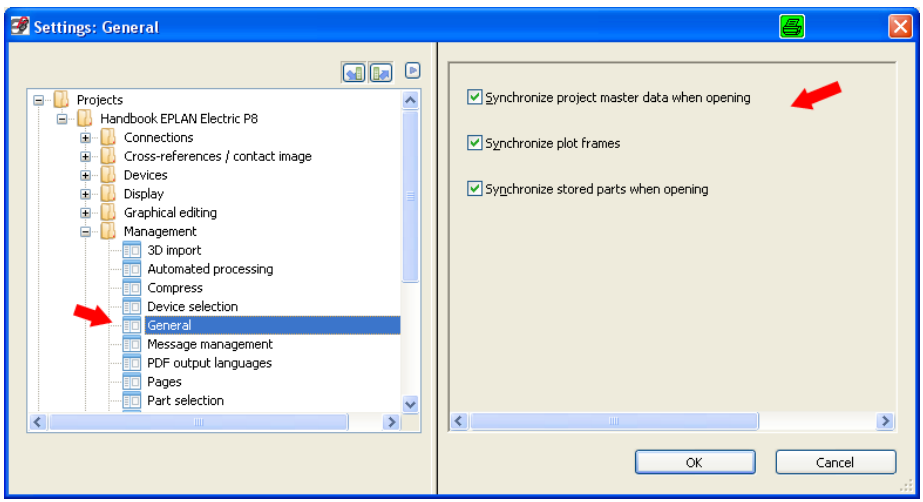


Fig. 6.58
Allow automatic synchronization of master data

When the **Synchronize** check box is *not* set in the column, then it will **not** be synchronized despite the active project setting. This must be carefully considered for a project. You should not synchronize if the project master data should retain its original edited status. In this case, this setting should not be activated.

6.4.3.13 The Next form column

Next forms are forms that display additional information to the information in the main report (e. g. parts data).

For example, a cable assignment diagram can be followed by a summarized parts list. This is only possible if the main form (in this case the cable assignment diagram) has been created with a form insertion point. Only then does EPLAN "know" that it must generate a corresponding next form containing the subsequent data (parts data).

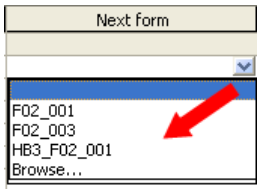


Fig. 6.59
Next form

NOTE: At present, only the summarized parts list is available as a next form.

6.5 Generate reports

This section deals with the generation of reports and the most important aspects of the various ways for doing this. EPLAN can generate reports with or without templates – without templates when a quick report is needed or no templates are available, or with templates when these are available or have been created by the user or imported from other projects. However, you do not have to use templates to generate reports in EPLAN. They enable project editing when no reporting structure has been defined and reports are output as desired.

6.5.1 The Reports dialog

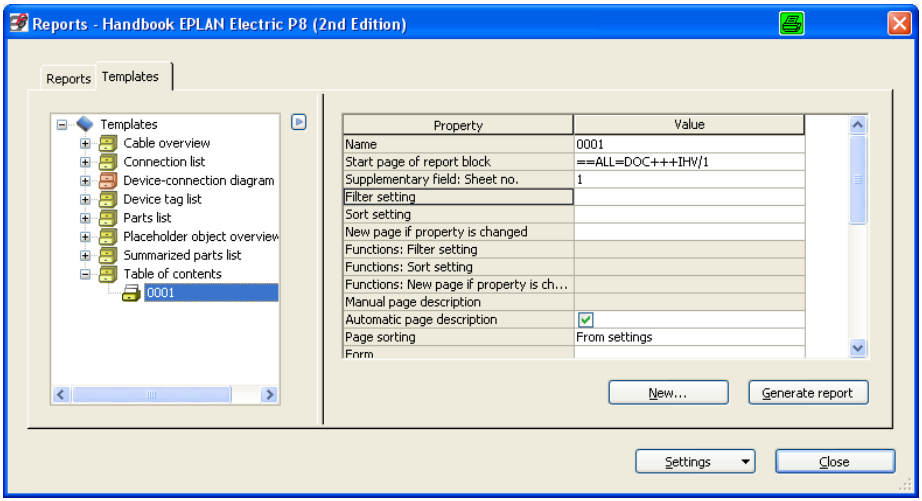


Fig. 6.60 The Reports dialog

To generate graphical reports while editing a project, you use the EPLAN **UTILITIES/REPORTS/GENERATE** menu item to open the **REPORTS** dialog.

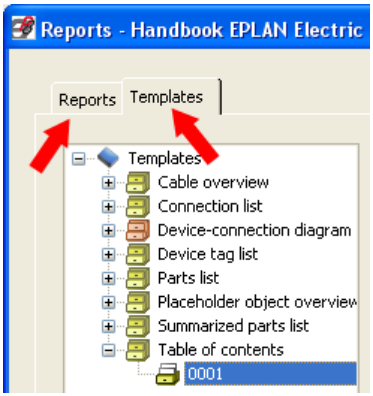


Fig. 6.61
Left area with tabs

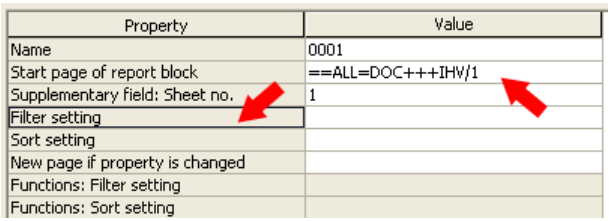


Fig. 6.62 Right area with additional information

The dialog is divided primarily into the *Reports* and *Templates* tabs, and the right part, which contains the report data, e.g. from which page the report should be generated, which filter and sorting settings should be used, and much more.

The *Reports* tab is usually empty before reports are generated from the project data for the first time, i.e. there are no entries, and the *Templates* tab is also usually empty.

NOTE: If the project was copied with the *Copy with reports* option, then the old reports will, of course, be visible in the *Reports* tab. Templates may also already be present.

After graphical reports, such as a parts list, have been generated from the project data for the first time, the *Reports* tab will contain an overview of all reports in the project. Normal graphical outputs (each graphical output on a new page) are then shown in the *Pages* folder and are visually distinguished by a small graphical symbol.

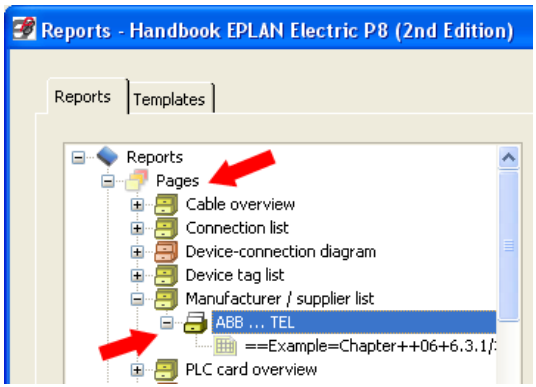


Fig. 6.63
Pages tree

The graphical symbols have the following meanings:

	Node includes general report pages
	Node includes function-related reports, such as terminal diagrams
	Node includes all pages that belong to a report block (function-related reports)
	Node includes report overviews, such as a cable overview
	Node includes all pages that belong to a report block (report overviews)
	Node includes placed report pages
	Node includes reports embedded in pages

6.5.2 Generate reports without templates

In general, you can directly generate reports in EPLAN. You do not need to create any templates for this. To generate a report, you simply begin by clicking the **NEW ...** button.

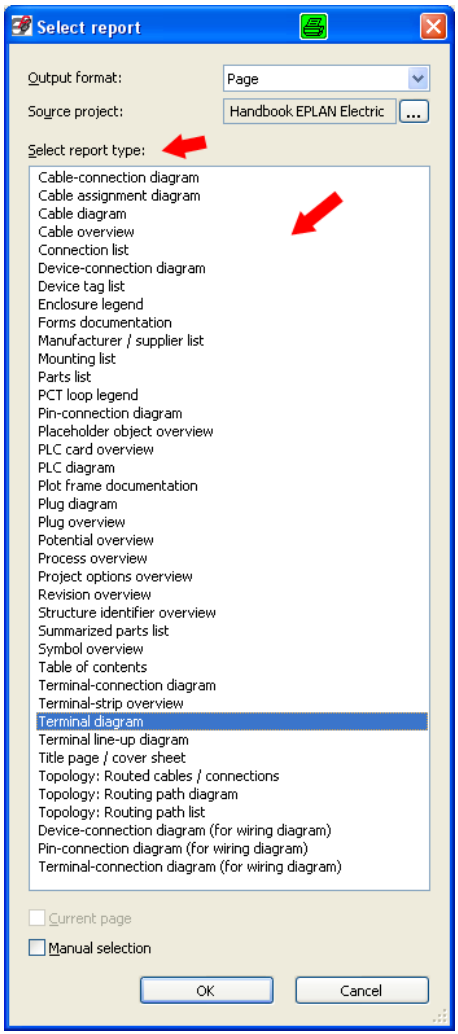


Fig. 6.64 Selecting a report

6.5.2.1 Output of project data for graphical output on new report pages

EPLAN opens the **SELECT REPORT** dialog. Now you select the desired report (report type) in the window. *Only one* type of report can be selected each time this dialog is called up. You can only generate several reports at once by using suitable templates. After selecting the report, e. g. a terminal diagram, you can now set the remaining options in the **SELECT REPORT** dialog.

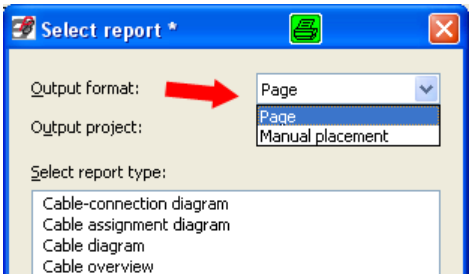


Fig. 6.65
Output format

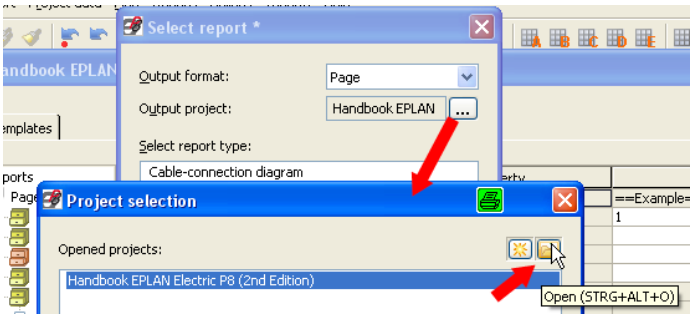


Fig. 6.66
Selecting a source project

There are two available *output formats:*

- Output in pages (to be generated by EPLAN)
- Output as manual placement (in an existing page)

The *source project* is usually the current project, the source of data for reports

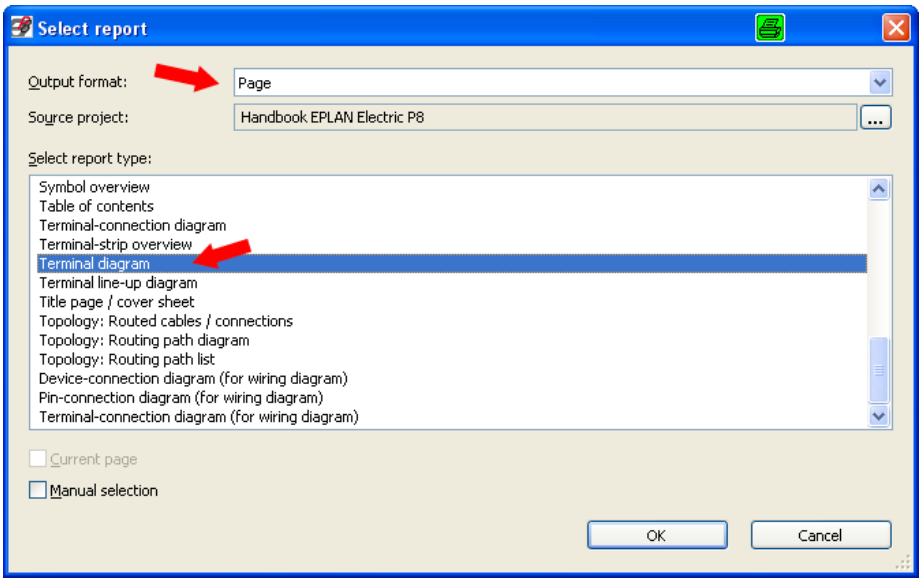


Fig. 6.67 Selecting a report

When you click the **OK** button (we will ignore the **Manual selection** and **Current selection** check boxes at this point, because all of the terminal strips are to be output), EPLAN allows you to define the sorting into the project via particular predefined filter and/or sorting schemes, or your own filter and sorting schemes (the possibilities here may be limited by the particular report type).

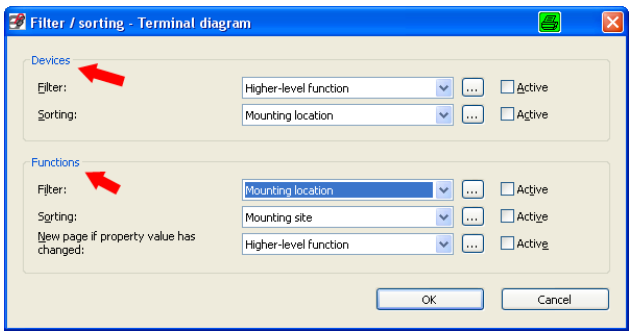


Fig. 6.68
Filter and sorting options for a report

If EPLAN is to use a filter, then the *Active* check box must be set. Only then is a filter active. If the option is grayed out, then no filters can be used with this area of graphical output. If the check boxes are not set, then the set filters have no effect.

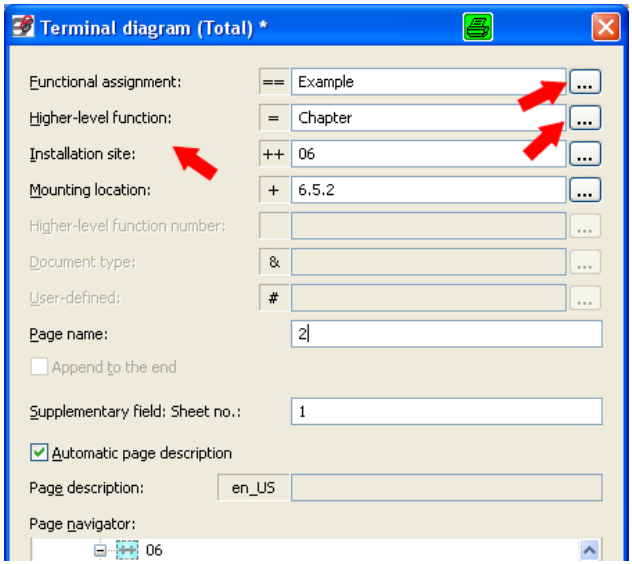


Fig. 6.69
Selecting the output

EPLAN next displays the **[REPORT TYPE] (TOTAL)** dialog. Here you can define default values for sorting, such as the selection of *structure identifiers* or specific entries in the page properties, e. g. **automatic page description**. EPLAN then automatically creates the page properties from the form properties, whereby something must, of course, be assigned to the **<13019 Format for automatic page description>** form property. The remaining dialog fields contain the usual entries and will not be further explained here. When you click **OK**, EPLAN generates the reports.

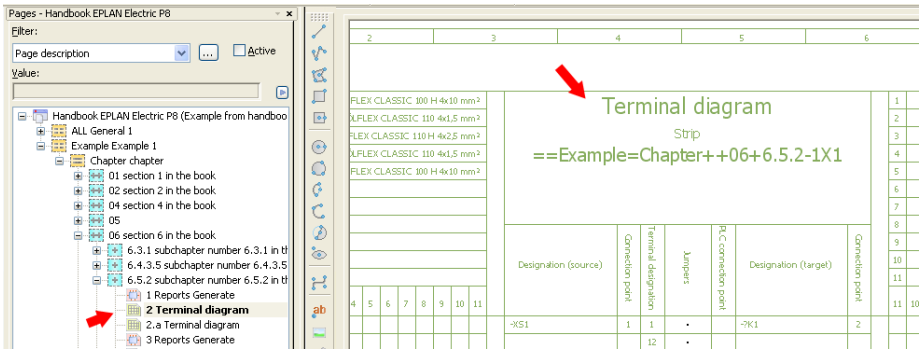


Fig. 6.70
Generated output of terminal diagram report

After you leave the dialog by clicking **OK**, EPLAN generates the graphical report pages and sorts them into the project according to your settings. For example, the *terminal diagram* report is generated as graphical output and inserted into the project as a report page.

6.5.2.2 Manual selection of project data on the current page for graphical output

When you select *Current page* in the **REPORT** dialog, EPLAN lets you generate a report based on the data on the currently open page, e.g. the terminals that are placed on this page.

TIP: Before you use the **Current page** function to generate a report, the schematic page where the report with the current page's data is to be placed must be open and active. It is possible to scroll between pages once the report has been generated, but you will "lose" the previously generated report.

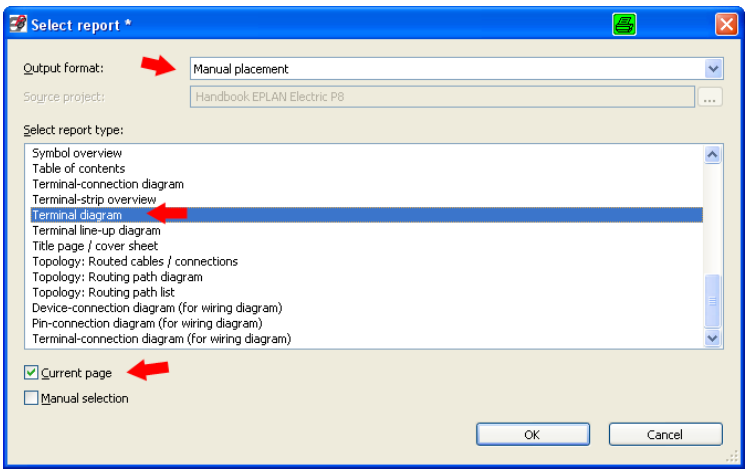


Fig. 6.71
Output options for data on the current page

The form entered in the settings will be used for the graphical output. However, a form entered in the default settings takes precedence. In the example, the entered form is the terminal strip definition form.

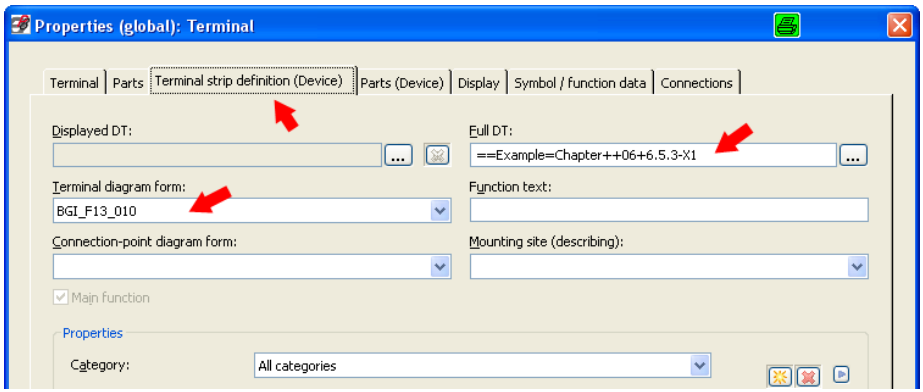


Fig. 6.72 Setting the default form

When you click **OK**, EPLAN asks about possible sorting and filter criteria, generates the report for the current data, and hangs the graphical output on the cursor. This can now be placed on the current page.

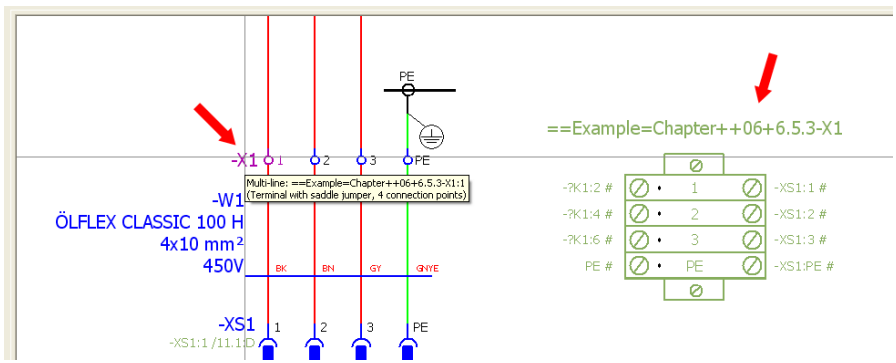


Fig. 6.73 Manually placing the report

This gives you a simple way to output the data (as a report) on the page the data is on, without having to tediously search through a manual selection list for the exact data.

6.5.2.3 Manual selection of specific project data for graphical output with manual placement

When reports are generated as described above, then either all project data or only the data on the current page is output. This means that if the project contains ten terminal strips, then all of these ten terminal strips are output to the project as graphical report pages (depending on the settings in the terminal strip definition, e. g. *<20851 No output to terminal/plug diagram>*).

In the following, data is selected via manual selection, but not on new report pages as in section 6.5.2.2; instead, the data is manually placed on existing schematic pages.

TIP: Before you use the **Manual placement** function, the schematic page where the report will finally be placed must be open and active. You can no longer scroll between the pages once the report has been generated.

Manual placement works in principle with all form sizes (in this case the graphical extent of the form), but the form should be suitable for the respective schematic page – optically and in terms of space and the data to be output.

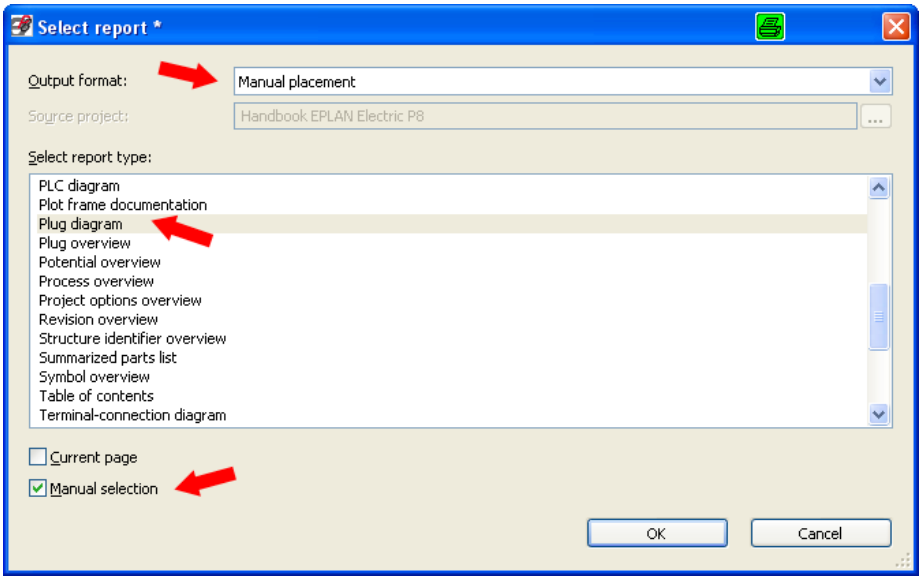


Fig. 6.74 Selecting a report

The **Manual selection** option allows you to define which particular project data is to be output, e. g. specific plug strips. Select the corresponding report and activate the **Manual selection** option. The data should be placed on an active schematic page. You do this by setting the output format to **Manual placement**. Then you can confirm the dialog by clicking **OK**.

The **MANUAL SELECTION** dialog is then displayed. The left side of the dialog shows all data that is *available* for the selected report. You use the buttons with the blue arrows to add the desired data to the *Selected* field at the right of the dialog. The button inserts data from left to right. The lower button (which is only available when there is data in the right field) removes the selected data from the selection.

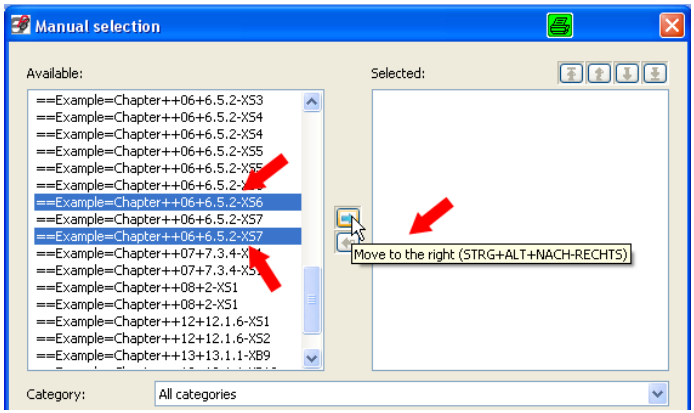


Fig. 6.75 Manual selection of project data

You select the data via **CTRL** (or **SHIFT**) + left mouse button, or you can use the usual Windows functions such as **CTRL + A** (select all). To select data, the cursor must be located either in the left (*Available*) field or the right (*Selected*) field of the dialog.

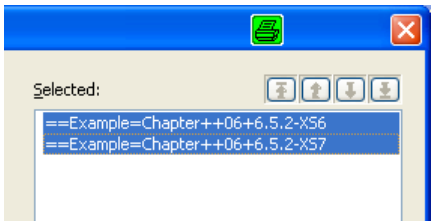


Fig. 6.76 Selected data

Once all desired data has been selected, it can also be sorted as necessary with the **SORT** button. EPLAN sorts the data with this function based on the sorting of the existing structure identifiers.

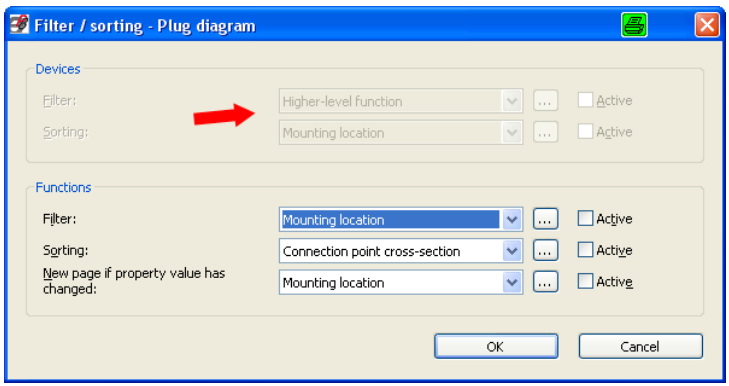


Fig. 6.77 Filter and sorting options for manually selected functions

When you click **OK**, EPLAN displays the familiar **FILTER/SORTING [REPORT TYPE]** dialog. Filters and sorting cannot be selected for devices because these were already manually selected. Filters and sorting can, however, be used for the functions themselves (data such as plug pin or terminal).

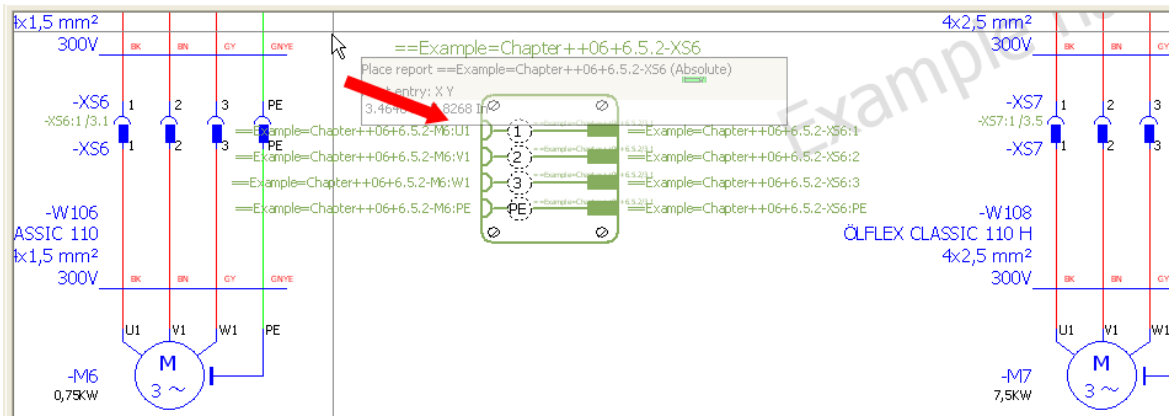


Fig. 6.78 Manually placing the report

When you click **OK**, EPLAN closes all dialogs. The reports hang on the cursor and can now be placed on the active schematic page.

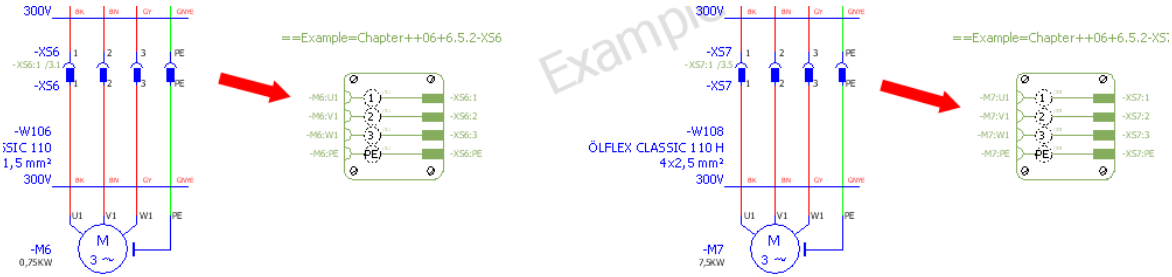


Fig. 6.79 Manually placed reports

NOTE: The **Manual selection** option cannot be used with all report types.

6.5.2.4 Graphical output of data using the Current page/Manual selection combination

The Current page and Manual selection options can also be combined.

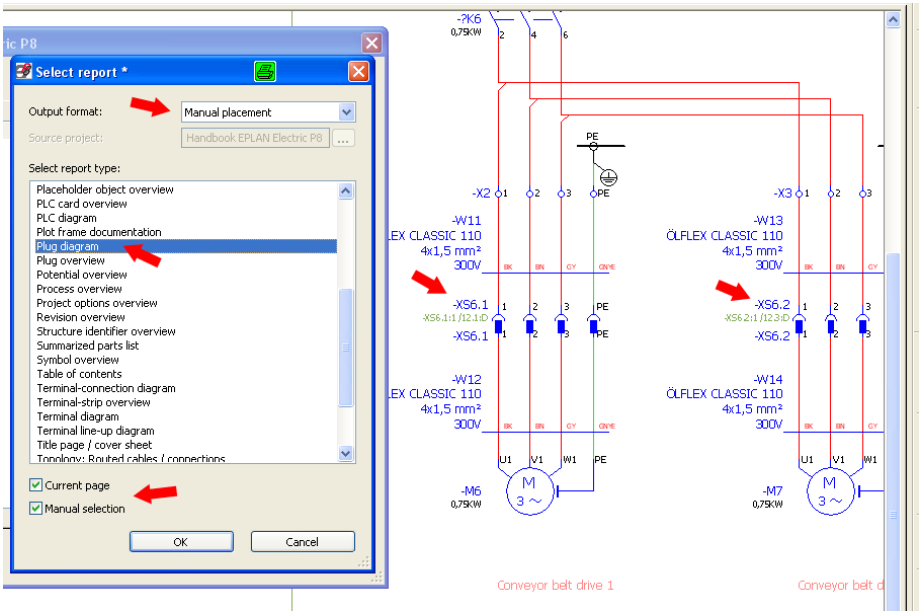


Fig. 6.80 Combining both options for the graphical output of the data

All data on the current page will be used for graphical output, but you now can manually intervene and, for example, remove a terminal strip from the graphical output.

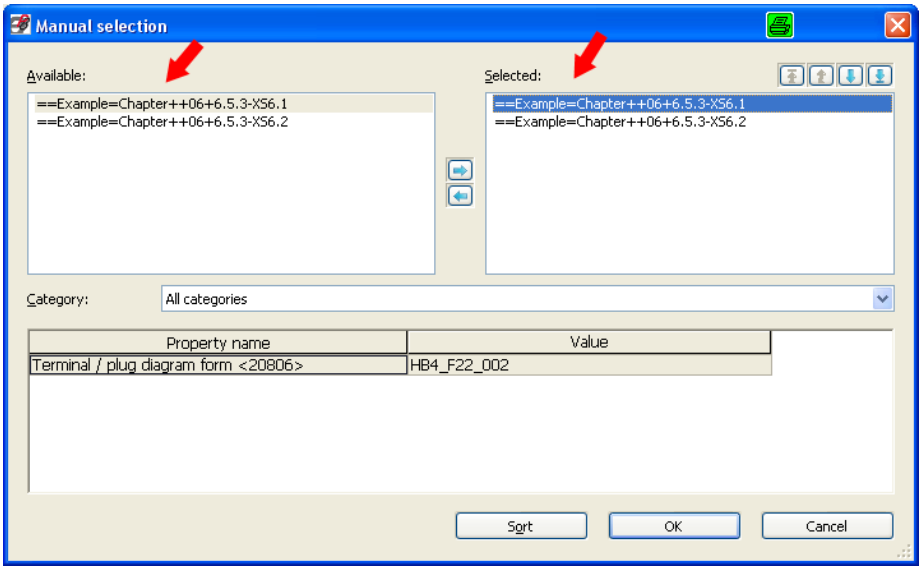


Fig. 6.81
Selecting data

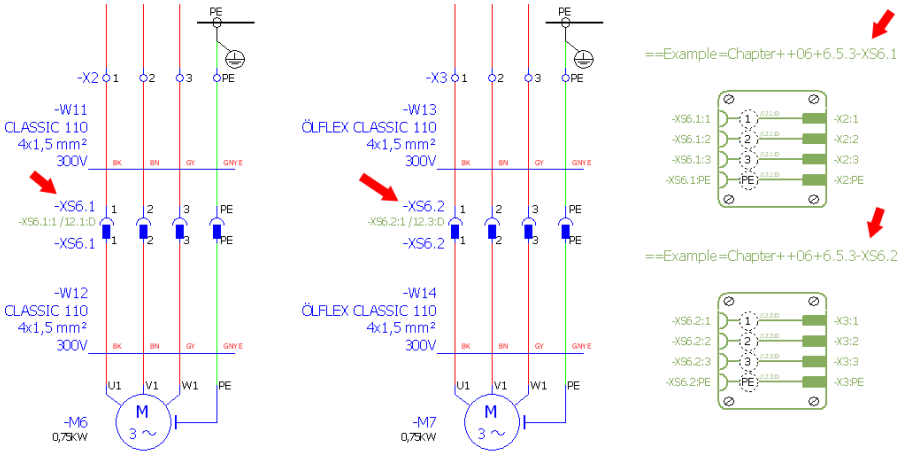


Fig. 6.82 Placed data

6.5.3 Popup menus in the Reports tab

There are a number of other functions in addition to the previously described functions, such as **Creating reports** and **Manually placing reports**. Reports should, of course, be able to be deleted, or quickly updated from within the dialog (e.g. after changing a setting).

These and other functions are located in the right-click popup menu (click a field in the *Reports* tab with the left mouse button and then click the right mouse button or open the popup menu using the button). In addition to the usual **EXPAND** and **COLLAPSE** functions for the report tree, the menu contains the following functions.

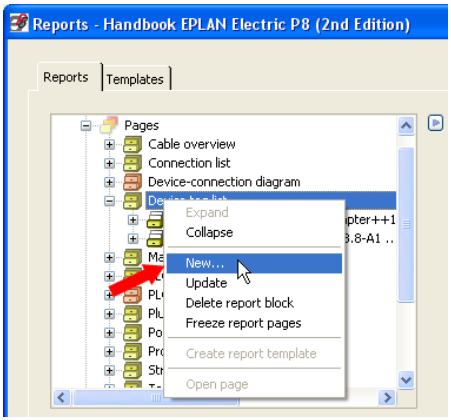


Fig. 6.83 Popup menu

Menu entries in the Reports tab

- **NEW:** Creates a new report and has the same function as the **NEW ...** button in the **REPORTS** dialog.
- **UPDATE:** This function updates one or more reports. This is useful when, for example, you have changed the report sorting and wish to quickly check where the graphical output pages were placed.
- **DELETE REPORT BLOCK:** Removes the reports (i.e. the graphical output pages) from the project. Here too, you can delete one or more reports at the same time.
- **FREEZE REPORT BLOCK:** "Freezes" the selected reports and removes them from the list of reports in the dialog. EPLAN displays a safety warning before executing this function. This allows you to cancel the action before it is executed.

- **CREATE REPORT TEMPLATE:** If a report is suitable for creating new reports, then you can click here to create a template from it. The template can then be found in the *Templates* tab.
- **OPEN PAGES:** This is a useful function that allows the user to directly navigate to the selected report in the project.

6.5.4 Generate reports with templates

What are *templates*? You could describe this term in another way. These are special *defaults* that define how, where, and to what extent EPLAN is to generate the graphical output of various project data. The procedure for creating a template is the same as that for creating regular reports, except that no reports are generated when the template is finished.

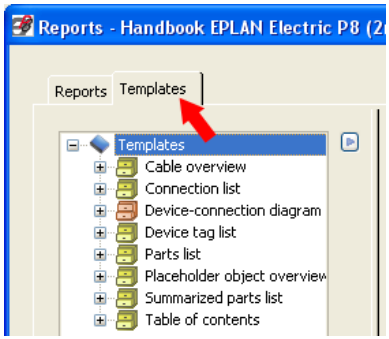


Fig. 6.84
The Templates tab

NOTE: The following restrictions apply to the use of templates to generate reports: The *output format* is fixed in the *page* selection and you cannot define a *manual selection*.

And how are templates created? The "how" involves a number of different settings for a template.

To define the report type, you open the **REPORTS** dialog via the **UTILITIES/REPORT/GENERATE** menu and switch to the *Templates* tab. You then click the **NEW ...** button to display the **SELECT REPORT** dialog. You now select the desired *report type* here. Confirm your selection with **OK**.

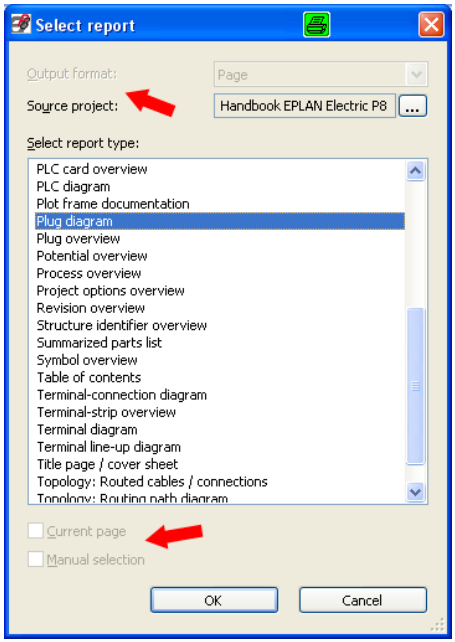


Fig. 6.85
Selecting a report type

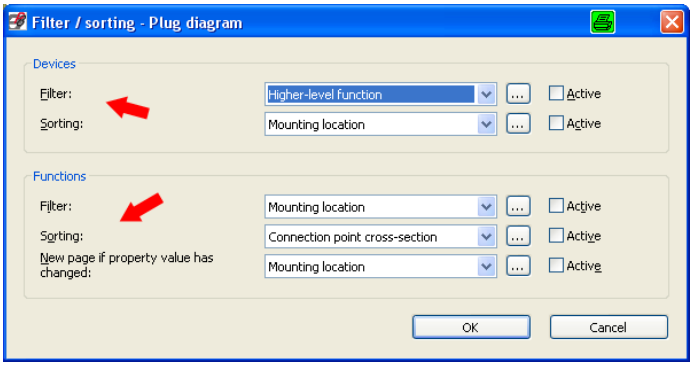


Fig. 6.86 The Filter/sorting dialog

As usual in EPLAN, you can use predefined filter and sorting options, or set up your own, in the **FILTER/SORTING [REPORT TYPE]** dialog. However, you must always make sure that the check boxes are correctly set, otherwise the filter and sorting options will not be active.

NOTE: Depending on the report type, the **FILTER/SORTING** dialog may look different, or a few of the options might not be displayed at all.

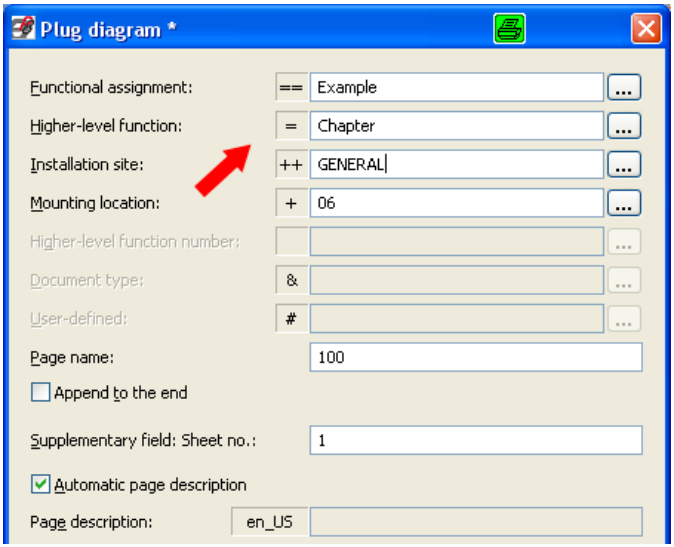


Fig. 6.87 Setting the structure of report pages

After you have selected any data filters and sorting, the *Selection of structure identifier* dialog for the output of report pages opens. The structure identifiers can be directly entered into the data entry fields or, if they already exist, they can be selected and applied from a table via the ... selection button.

NOTE: EPLAN initially automatically sorts new identifiers into the project's identifier structure. If these need to be rearranged, then this must be done later in structure identifier management.

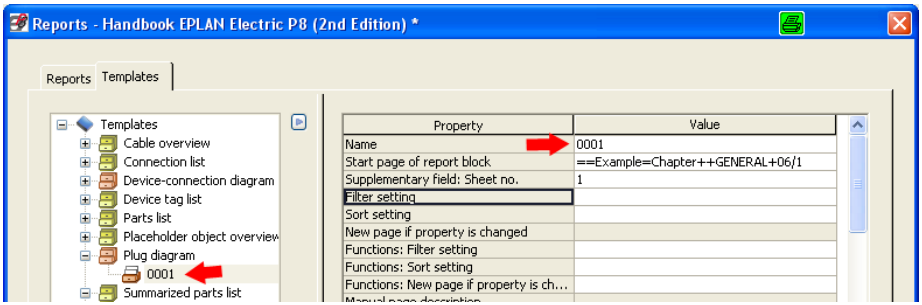


Fig. 6.88 Template addition complete

EPLAN now adds the template and its settings to the *Templates* tab. In order to keep templates functionally separate, EPLAN automatically assigns a certain structure to them when they are created (according to report type). The template is given the standard name 0001.

NOTE: If filters were selected previously, EPLAN assigns the filter's name to the template.

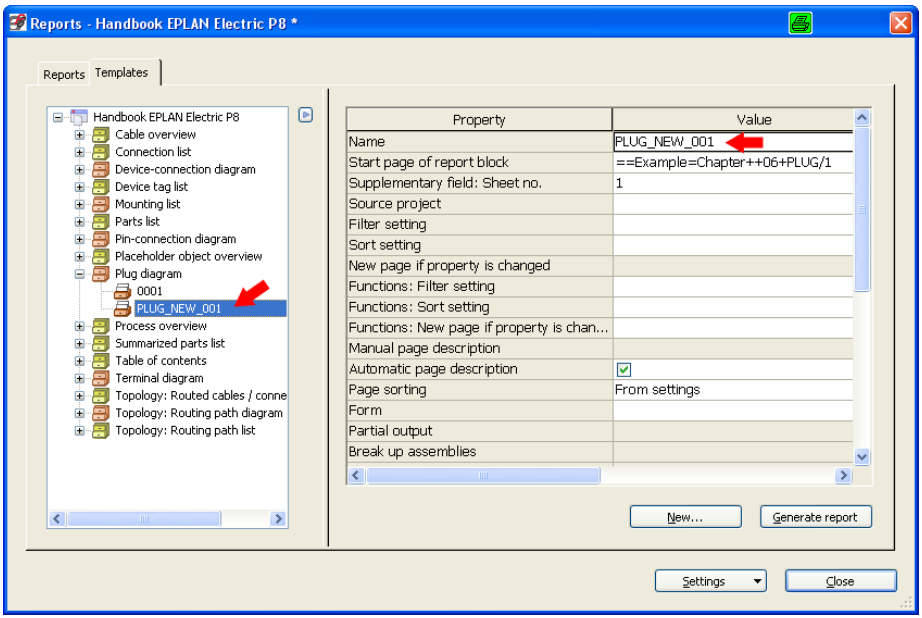


Fig. 6.89 Template name changed

Because there can be several templates generating report types, it is possible to manually change the names of templates later. You click in the **PROPERTY – NAME: VALUE** line and simply enter another name.

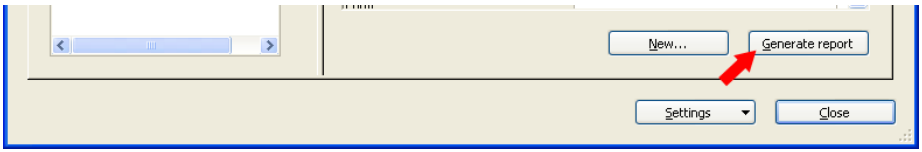


Fig. 6.90 Generate this template report

This completes the generation of a template. If you now select this template and click the **GENERATE REPORT** button, EPLAN will generate the report pages and sort them in the predefined structure of the project.

This and other information can be seen at the *right side* of the *Templates* tab. The table shows additional properties and their associated values. In contrast to the *Reports* tab (which also has this table), here you can change the corresponding values of basic properties.

Other properties of the Templates tab

Property	Value
Name	Complete
Start page of report block	==Example=Chapter++GENERAL+06/1
Supplementary field: Sheet no.	1
Filter setting	
Sort setting	
New page if property is changed	
Functions: Filter setting	
Functions: Sort setting	
Functions: New page if property is changed	

Fig. 6.91 Basic properties for structure and output

Name: Contains the name of the template. It can be manually changed to suit your personal requirements.

Start page of the report block: This is the start page of the graphical output and can be later changed. To change the name, click the field and then click the ... selection button that appears. The familiar dialog for **ENTERING THE STRUCTURE IDENTIFIER** is then displayed.

Supplementary field: Sheet no.: Additional information that appears in the page properties of the report pages.

Filter setting: This field is filled with the selected filter settings and can also be changed when you click it. You can then use the ... selection button to can assign a new filter scheme.

Sort setting: Can be subsequently changed in exactly the same way as the **Filter setting** property.

New page if property is changed: Allows you to enter a default value that causes an automatic page break when the value of a particular property change; the entered value usually cannot be changed

Functions: Filter/Sort setting and *New page if property is changed*: These filters operate directly on the device functions, e. g. on a terminal.

Manual page description	
Automatic page description	☑
Page sorting	From settings
Form	
Partial output	

Fig. 6.92 Properties that affect the report page

Manual page description: Can also be subsequently changed.

Automatic page description: When this property is activated, the **Manual page description** property can no longer be edited.

Page sorting: This field is usually empty because the basic value is defined in the default settings (**OUTPUT TO PAGES SETTING**). However, this value can still be changed.

Form: Here you can enter a different form to that specified in the default settings (**OUTPUT TO PAGES SETTING**). This form then has priority over the global settings.

Partial output: Can be changed, but only for certain report types that support partial output, such as the table of contents.

Break up assemblies	From settings
Break up modules	From settings
Devices without part number	From settings
Terminal strip parts	From settings
Terminal parts	From settings
Cable parts	From settings Yes No
Cable conductor parts	
Cable project parts	
Connection parts	From settings
Plug parts	From settings
Pin parts	From settings
Busbar parts	From settings
Busbar connection point parts	From settings

Fig. 6.93 Properties that affect parts output

Break up assemblies and *Break up modules*; *Device without part number*; other additional settings for *terminals*, *cables*, *project* and *connection parts:* These settings can be changed later but they only affect some of the *part* report types.

Project filter	FACTORY_PRINT_OK
Template active	☑

Fig. 6.94 Special report settings

Project filter: Setting a project filter enables you to generate reports that are linked to a condition. You can generate conditional reports independently of certain project properties.

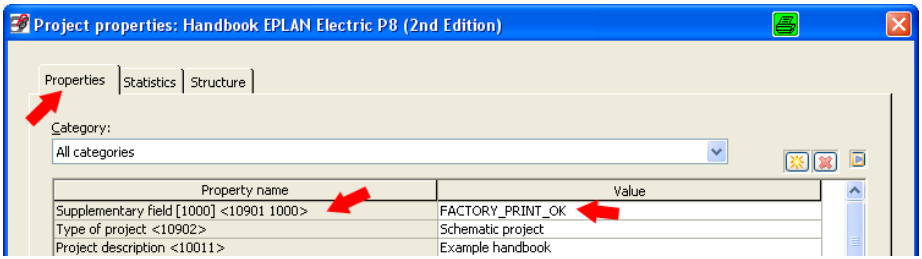


Fig. 6.95 Selecting a project property and defining its value

This means that if a defined project property's value matches the filter value, a report will be generated.

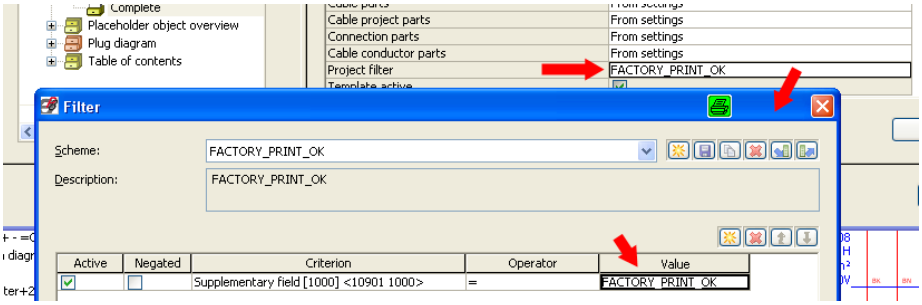


Fig. 6.96 Selecting a filter for the desired project property and its value

Template active: This setting cannot be edited; it is automatically set or not set by EPLAN. The check box depends on whether the project filter was fulfilled or not. When the condition is fulfilled, the check box is set; when the condition is not fulfilled, EPLAN removes the check. This means that no report will be generated.

6.5.4.1 Export and import of templates

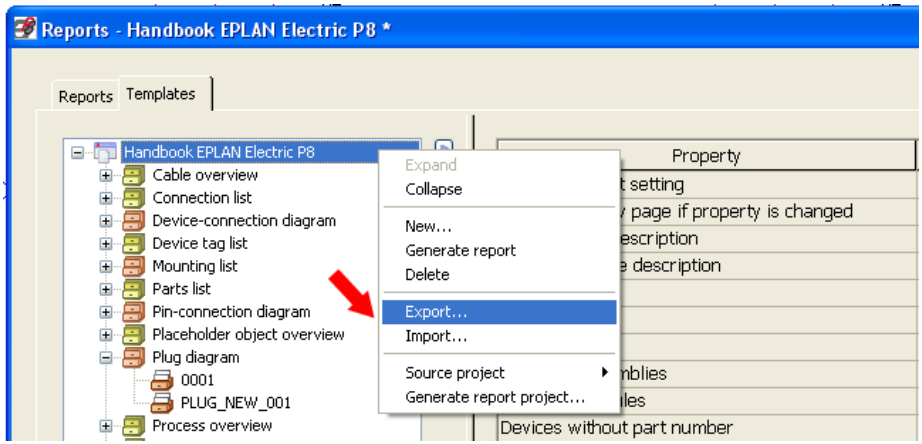


Fig. 6.97 Export/import of templates

Once you have created useful templates for graphical reports in a project, EPLAN can also use these for other projects. EPLAN provides *export and import functions* for this. However, this applies only to templates. Existing reports cannot be exported or imported.

To export or import templates, you open the **REPORTS** dialog via the **UTILITIES/REPORTS/GENERATE** menu. If not immediately displayed, then switch to the *Templates* tab. You access the *Export* and *Import* functions via the popup menu (accessible via the button or the right mouse button).

EPLAN allows the user to decide which templates to export and import. If the tree is selected at the highest level, for example, then all templates below this are exported together. You can also export individual templates. Expand the template tree, select the desired template only, and run the **EXPORT** function. EPLAN then displays the **EXPORT REPORT TEMPLATES** dialog. Select a file name for the template and then save it by clicking the **SAVE** button. The template is then exported.

Importing existing templates functions in exactly the reverse direction. You open the **REPORTS** dialog and start the *Import* function (right mouse button or popup menu). EPLAN opens the **IMPORT REPORT TEMPLATES** dialog. Here you select the desired template and open it via the **OPEN** button. EPLAN now inserts the new template into the project.

6.6 Other functions

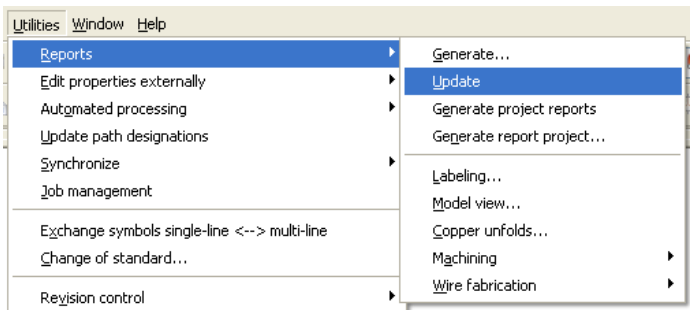


Fig. 6.98 Generate project (templates) reports

You will normally not just constantly create new reports and develop pretty templates for reports. Reports with particular report types are usually not completely finished until the end of project editing and then only updated when project data changes, such as terminal designations or a different cable type. EPLAN offers two more functions that allow this to be done quickly.

6.6.1 Generate project reports

The *Generate project reports* function is accessed via the **UTILITIES/REPORTS** menu. If you select this function, all of the reports (that exist as report templates) are evaluated completely. For large projects, this can take a while.

TIP: Once the menu item has been selected, EPLAN begins project evaluation immediately (without any prompt dialogs). You can, however, use the **CANCEL** button in the **GENERATE PROJECT REPORTS** dialog to immediately exit the function. The reports then remain in their original state or are returned to their original state.

6.6.2 Update

An additional function that EPLAN offers is the **UPDATE** function for *existing* reports. This function is also available in the **UTILITIES/REPORTS** menu. In contrast to the *Generate project reports* function, the *Update* function always only affects the currently open report or a report selected in the page navigator (a page, but can also be a manually placed report) in the project. If you run the *Update* function, the corresponding report is updated. If the Update function is used on a schematic page, EPLAN displays a message stating that you are not on a report page or that no report page is selected in the page overview.

6.6.3 Generate report project

This menu item is used to define a report project. Using the open project (source project), EPLAN generates an identical project (report project), including the header data, but without pages. This makes it possible to later output the data reports in the source project to the external project report.

6.6.4 Settings for automatic updates

If this is not enough for you, then you can also enter a user setting that defines whether or not report pages should be automatically updated when opened, printed, or exported. This is not a project setting but rather a user-specific setting. It is located in the **OPTIONS/SETTINGS/USER/DISPLAY/GENERAL** menu. The two relevant settings are *Update reports when opening pages* and *Update reports when printing and exporting*.

However, you should note that this type of constant updating slows down project editing, e. g. when scrolling through project pages. Opening report pages also takes longer when this user setting is active because the reports are always brought up-to-date. You should

therefore carefully consider whether or not the report pages should be constantly kept up-to-date. In my opinion, this is not absolutely necessary, but EPLAN offers this setting. It is up to the user to decide.

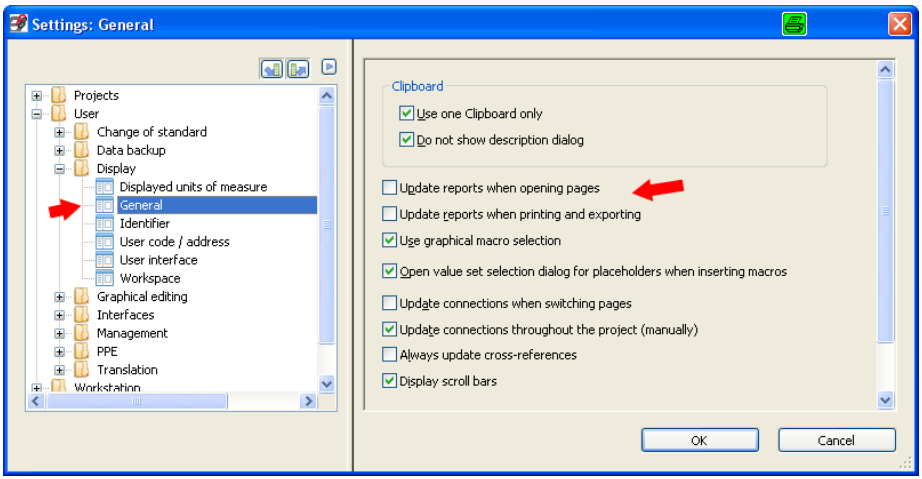


Fig. 6.99 Settings for automatically updating reports

6.7 Labeling

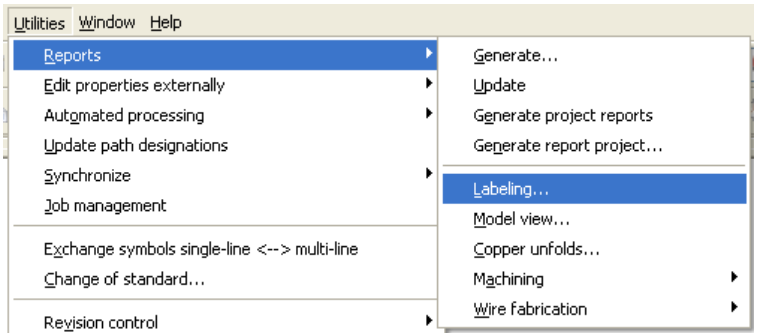


Fig. 6.100 Labeling

The second way to generate a report from project data is via the function under the **UTILITIES/REPORTS/LABELING** menu item. This function does not generate graphical report pages but rather writes the project data to external files. These external files can then be passed on and subsequently edited, for example to print terminal diagrams in Excel format or labels.

6.7.1 Settings

This section deals with the basic settings for outputting labeling. When the **LABELING** function is started, EPLAN displays the **OUTPUT LABELING** dialog.

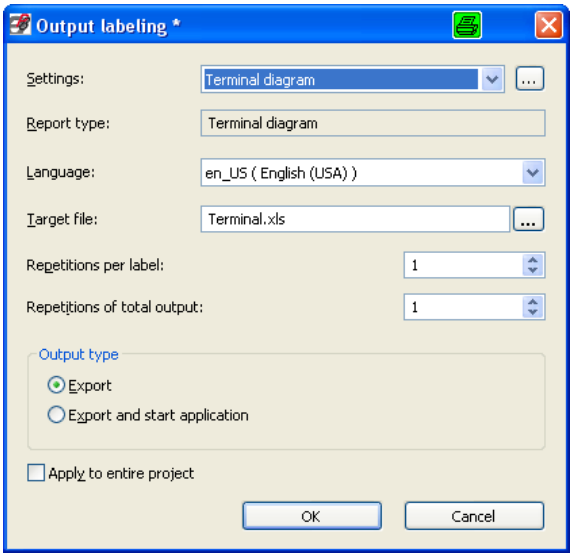


Fig. 6.101 Output labeling

This dialog offers the following settings options:

Settings: Here you can select what data is to be output. You can filter and/or sort the output.

Report type: This field merely displays which report type is used as the basis of the set labeling scheme.

Language: This setting controls the language output of the labeling file. Only the languages set in the project are available here. EPLAN offers two ways of selecting the language: A single language or all project languages at once can be output.

Target file: This is where you enter the name of the output file (according to the selected scheme) for the labeling data to be generated.

Value for repetitions: All values greater than 1 increase the output of the labeling data by the value specified here. The default value here is 1 (output the data once).

Output type: The generated file is exported, or you can cause the exported file to be subsequently opened with a specified editor so that you can check or edit the contents.

TIP: The editor to be used is defined by an entry in the user settings. This entry is located under **OPTIONS/SETTINGS/USER/DISPLAY/IDENTIFIER**. You select the **OPTIONS** button in the schemes.

Apply to entire project: Regardless of what data is selected, this setting always causes the entire project's data to be written.

6.7.1.1 Preparatory settings

The most important area in the **OUTPUT LABELING** dialog is the *settings:* What data should be output here and under which relationships? This is defined via predefined schemes or your own schemes.

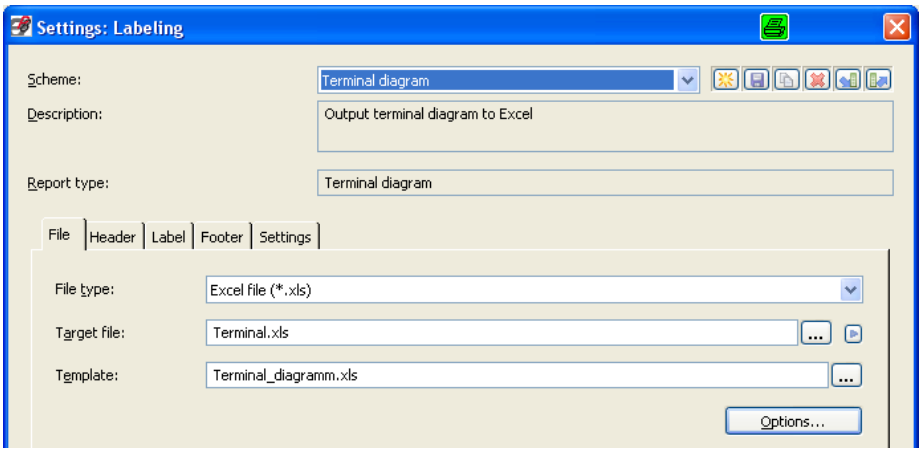


Fig. 6.102 The Labeling dialog with the scheme and the settings

To add a setting, you click the ... selection button next to the *Settings* field. The **SETTINGS/LABELING** dialog is displayed. The dialog is basically divided into two areas: the area with information on the scheme itself (name of the scheme, description, and report type), and the lower area with the tabs for various data and settings such as file, header, label, and footer, and other output settings such as filter, sorting, etc.

The file type to be created is important when creating a labeling file. Other settings, such as which program to use for opening the generated file, depend on this value.

EPLAN offers several different file types here.

File types for outputting labeling

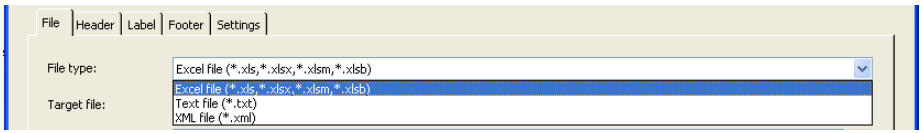


Fig. 6.103 Selecting the file type

*The Excel file *.xl*:* Files with this format, as the name indicates, are opened with Excel. Whereas a text file is usually output without formatting, the Excel format can be used, for example, to output cable diagrams formatted in an Excel template. This produces report files that look similar to those generated in the graphical report pages. You can use the **OPTIONS** button to also output column designations.

*The text file *.txt:* A normal text file is created with this file type. It can be edited with a conventional text editor. The **OPTIONS** button allows further settings to be defined for a text file. Examples are the ANSI output character set or separators for multi-language output.

*XML file *xml:* The last possible data type generates an XML file that can be opened and edited with an appropriate XML editor. No other options are available for this file type.

When outputting a labeling file, the correct file format should be chosen before generating the output. The **.txt* file format is usually the right one for outputting project data such as cable device tags (e. g. for labeling cable shields). In practice, you should always check what data format can be edited by the downstream systems.

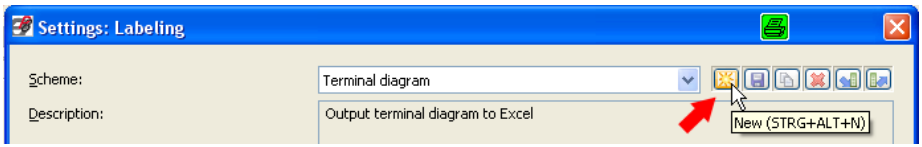


Fig. 6.104 Toolbar for editing the selected scheme

Next to the *Scheme* selection field is the toolbar with editing functions for the scheme. You use these buttons to create, edit, and delete schemes, etc.

6.7.1.2 Labeling as text output

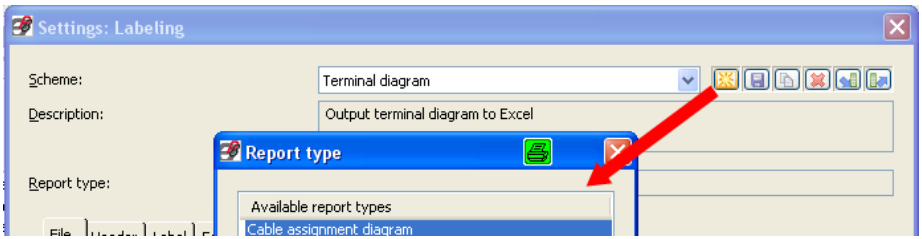


Fig. 6.105 Select report type

You create a new scheme via the button. EPLAN first asks for the *report type*. You should enter a suitably descriptive name and description in the subsequent dialog. One advantage of this is that every user has an immediate understanding of the purpose of the scheme. Another advantage is that this scheme name can also be used to export the scheme, eliminating duplicate work later on.

Once the header of the scheme has been defined, you can fill the scheme with the required format elements. Format elements in EPLAN represent the *available properties* of the project, pages, and devices. Format elements are thus nothing more than *properties*. The file type is set to **.txt*. You can now use the *Header*, *Label* and *Footer* tabs to assign *format elements* to the scheme.

The tabs are all structured in a similar way. To the left is the field with the *available format elements* (the actual properties) and the right area contains the overview of currently

selected format elements. You use the button to “copy” selected elements from the left field to the right field.

Depending on the property selected, you can open an additional dialog to make a more detailed selection of one of the properties.

You can delete elements from the right field using the button on the toolbar. You must, of course, first select the element you wish to delete.

NOTE: Clicking the button immediately removes the element. There are no other confirmation prompts.

To assign a format element to the scheme, you switch to the *Label* tab. EPLAN offers a lot of possible project data (properties) that can be used for labeling.

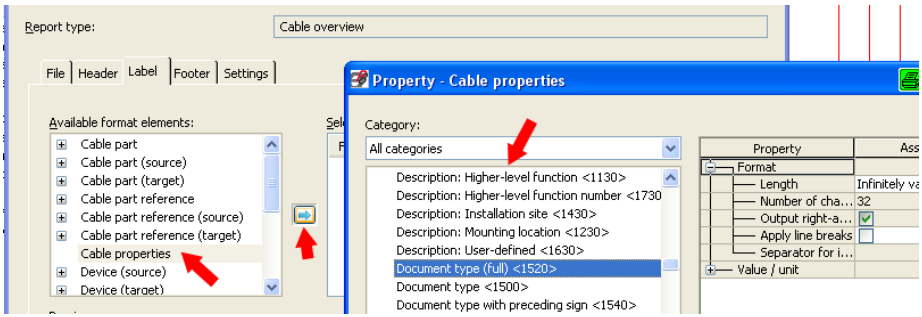


Fig. 6.106 Selecting desired properties

The cable device tag should be output in a cable overview. The cable device tag belongs to the cable properties group, since the DT is a property of a symbol (terminal, cable, etc.). You select the format element in the left field with the left mouse button. You now use the button to move the *Cable property* format element from the left field (*Available format elements*) to the right field (*Selected format elements*).

EPLAN then opens the **PROPERTY – CABLE PROPERTIES** dialog. All available cable properties are listed here.

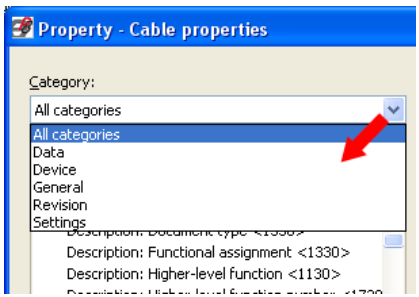


Fig. 6.107 Categories

You can limit this property overview by selecting a more suitable value in the **Category** field, *devices* for example. This hides all properties that do not belong to this category. This greatly simplifies the overview.

To allow the cable device tag to be output later in the labeling file, the **<20006 DT (full)>** property must be selected from the list and applied by clicking **OK**.

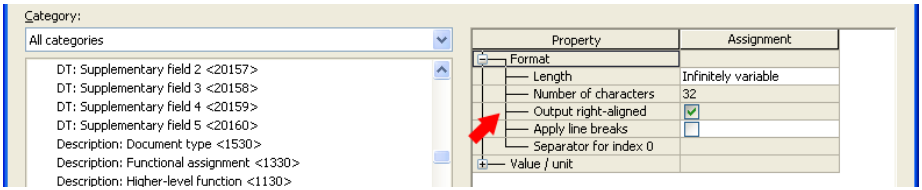


Fig. 6.108 Possible assignments for selected properties

The additional options in the right side of the dialog should be set as needed. This requires no further explanation. After applying the property, EPLAN closes the dialog. Property **<20006>** is now located in the right field with the selected format elements and with the correct settings (unlimited variable length). These entries can be changed at any later time. You use the toolbar above the field to do this. This applies to all available tabs in the Labeling dialog.

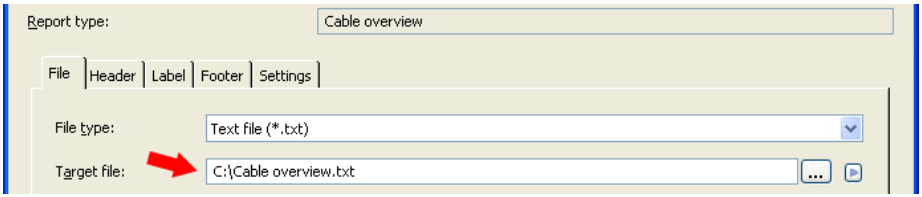


Fig. 6.109 Select target file

Now only the name of the target file is missing. This completes all entries and you can save the scheme by clicking . Click **OK** to exit the settings dialog. EPLAN returns to the **LABELING** dialog.

The labeling file can now be generated. You only need to set the desired output type.

If the file is only going to be exported, then the *Export* setting is sufficient. If the file is to be subsequently edited, you should set the *Output type* setting to *Export and start application*.

The result of the example is shown in Fig. 6.110 (opened in a text editor):

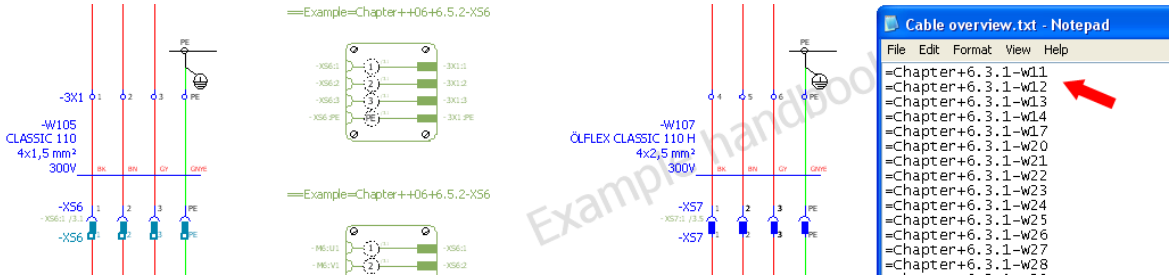


Fig. 6.110 Generated text file with information

6.7.1.3 Labeling as Excel file

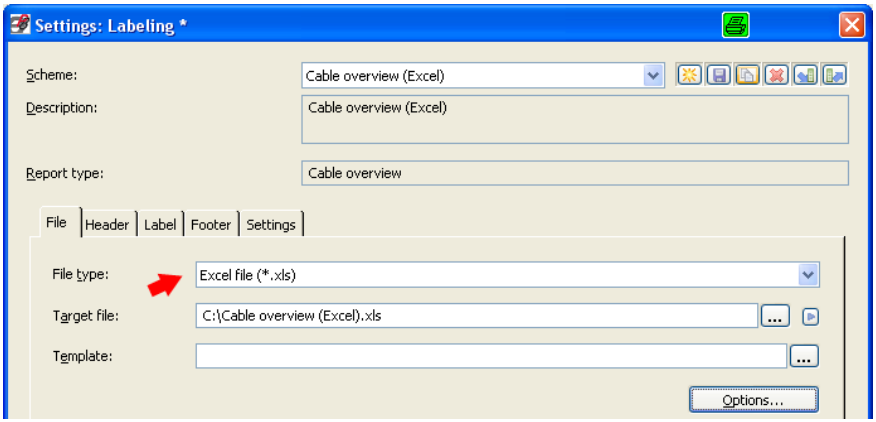


Fig. 6.111 Scheme for output to Excel

In addition to text output, EPLAN also allows output in the Excel format **.xls*. This allows you to replicate the graphical output in Excel if necessary. The preparatory procedure is similar to that for outputting a text file. This means that you again generate a scheme containing assigned format elements.

However, the following points must be noted when generating Excel files. For an Excel output, a template **can** be stored in the scheme. This template is created in Excel with variables before the data is output and then later assigned to the scheme. EPLAN later writes the project data (as specified in the scheme) into these variables.

The following example is explained **without** an Excel template, since an Excel template is not absolutely necessary.

NOTE: You *can*, but you don't *have* to use an Excel template.

You move the data from the pool of available properties to the list of *selected format elements* in the usual way.

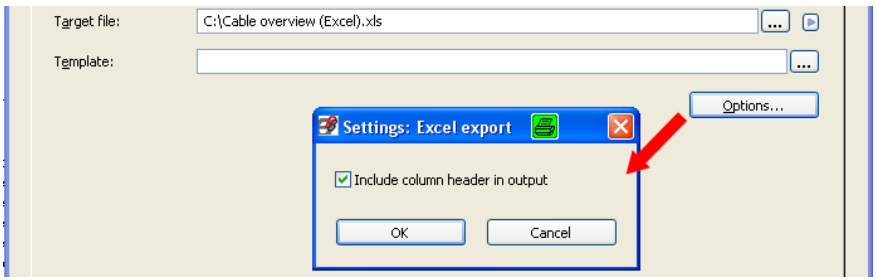


Fig. 6.112 The Options button

The **OPTIONS** button on the *File* tab is important when outputting to Excel. If the *Include column header in output* setting is activated, the column headings in Excel are given the names of the corresponding format elements (here the full DT).

Once these two things have been done and the data has been correctly stored in the scheme and the scheme has been saved, EPLAN can start the output of project data in the Excel format. As you already know, this is done via the **UTILITIES/REPORTS/LABELING** menu item. The **OUTPUT LABELING** dialog then opens. Here you select the desired scheme and set the corresponding output type. You now confirm the output by clicking **OK**. EPLAN generates the data and saves it in the Excel format. When you open the Excel file you can see, as in our example, the transferred cable overview with all the data that was previously assigned to the scheme.

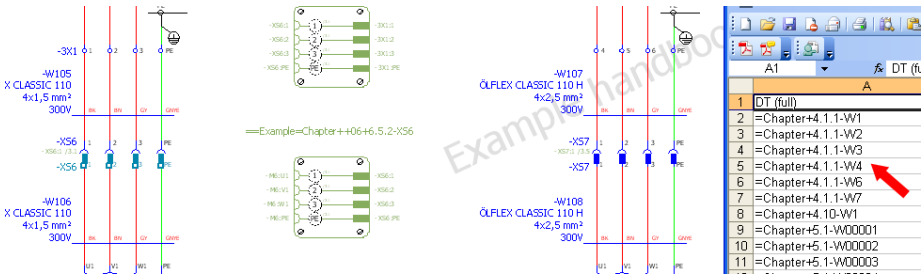


Fig. 6.113 Generated Excel file with the data from EPLAN

6.8 Edit properties externally

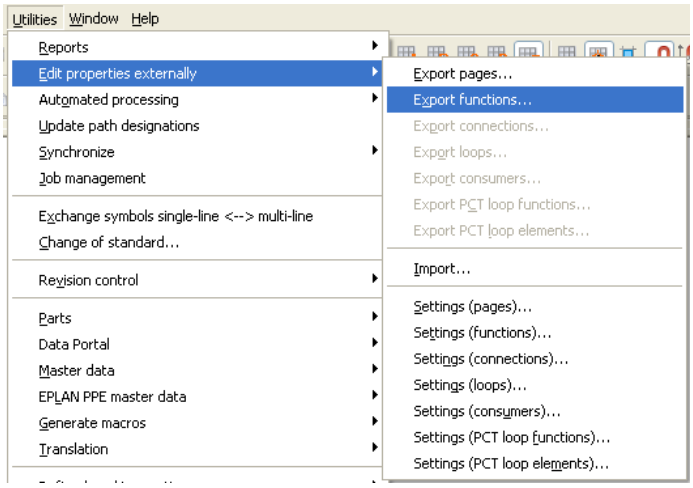


Fig. 6.114 The Edit properties externally menu

In addition to generating reports in the form of graphical output pages or files, EPLAN provides another powerful tool with the *Edit properties externally* function.

NOTE: One important thing should always be noted: Whatever you wish to externally edit must always first be highlighted (selected) in EPLAN.

If you wish to edit page properties then you must select the desired pages or select the entire project (project name) in the page navigator. This also selects all pages. If you wish to edit only particular terminals (functions) on a page then you must select precisely these terminals.

These functions can be described very simply. They allow project data to be transferred to Excel, edited there, and then transferred back to EPLAN.

NOTE: At the moment, Excel is the only program available for external editing.

The data, which is easy to rename, exchange or enhance in Excel, is then synchronized with the existing project data. This avoids subsequent editing in EPLAN; this is a very powerful function.

However, we should explicitly mention here that there is no UNDO function after transferring the data back. Once the data has been synchronized with the schematic, there is no way back. You should therefore always be very careful when using this powerful function. If you are not sure, then you should work with a copy of the current project. Here you can check the results and adjust the settings for the external properties as necessary.

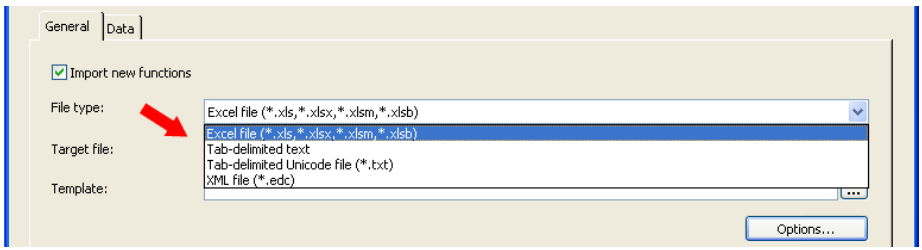


Fig. 6.115 File types for external editing

Using Excel is, of course, not the only way of doing this. EPLAN offers a number of other file types for output that are listed below:

- Excel file formats (**.xl**)
- Tab-delimited Unicode file (**.txt*)
- Tab-delimited text
- XML file (**.edc*)

However, the following examples will only deal with output to Excel. Exporting of page data will be used to illustrate the creation of a scheme, the export to Excel, and the import into EPLAN.

The other possibilities of exporting functions or connections are performed in an identical manner. The only differences are usually just the data and/or the property values.

6.8.1 Export data

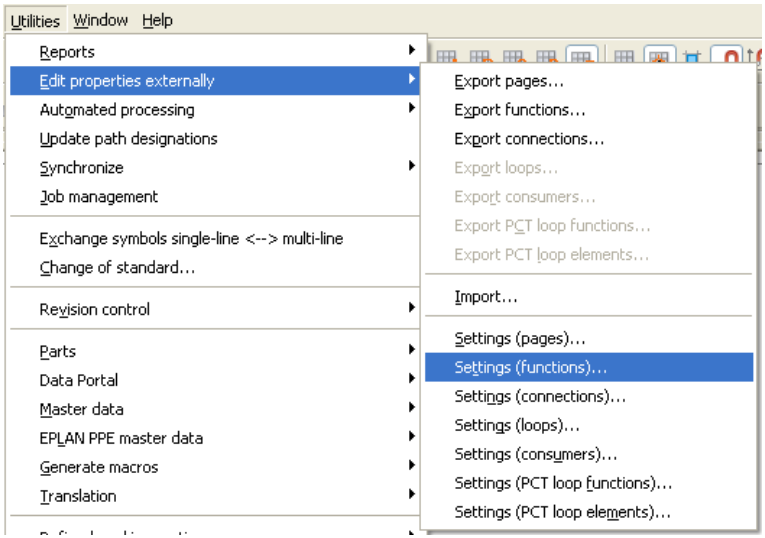


Fig. 6.116 Settings for external editing

A number of preconditions must be met before EPLAN can transfer data to Excel. These are determined in the settings under **EDIT PROPERTIES EXTERNALLY/SETTINGS** (pages, functions, connections).

EPLAN offers several ways of transferring project data to Excel. This relates to the following project data (extract):

Pages: For example, all data related to pages, such as page names, page supplementary fields and other properties assigned to a page (such as a form)

Functions: Functions are device tags, cable comments, connection point designations, and everything relating to devices, symbols, or the associated parts. The range of possible data is, of course, much larger, but this is sufficient as an example.

Connections: All data related to connections

NOTE: Basically, most properties are suitable for exporting. However, you should note that not all properties can be externally edited.

This is because EPLAN needs certain important information in order to write back the modified data at the correct position in the schematic. If this data basis is denied to EPLAN, then it has no idea where the data should be placed and may end up generating incorrect data.

Some properties are locked for editing to prevent this problem. You can still change these externally, but EPLAN simply ignores these changes when reading back the data tables.

6.8.1.1 Settings – sample pages

EPLAN allows a large amount of data for pages and their descriptions. Most of this data can be conveniently edited in the page navigator (page overview). This is even easier when you create an export to Excel for the pages and their properties. You can then use many Excel functions that do not exist, or are very difficult to implement, in EPLAN. To use this feature, you must first create a scheme in EPLAN with the associated properties. An Excel template is also useful, but not absolutely necessary.

Without an Excel template, for example, you get the result shown in Fig. 6.117.

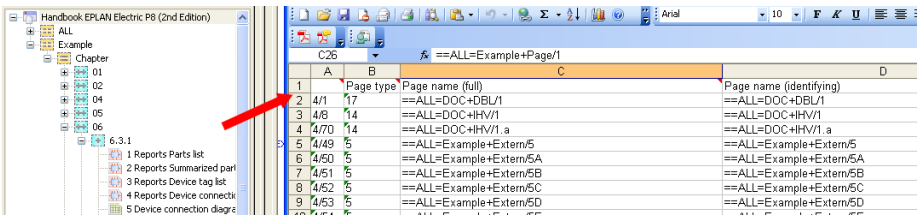


Fig. 6.117 Exported data without Excel template

The problem with this representation is that all cells look the same here and can also be edited. However, as already mentioned, not all edited values are re-imported into EPLAN. To make it clear that this is not caused by errors in the program but rather errors in the application, it is a good idea to use a formatted Excel template. A formatted Excel template provides an overview of which properties have changed and which properties are not able to be written back to EPLAN for programming reasons.

An example of a formatted Excel template is shown in Fig. 6.118:

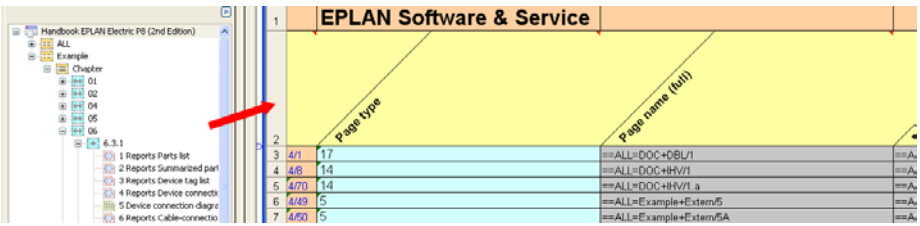


Fig. 6.118 Exported data with Excel template

The easiest way to create a template is to modify one of the templates provided in the EPLAN master data. You use the usual Excel editing functions for this.

The following points are important for the template: There is a *header* and a *data area*. The header area is marked with the **#H#** identifier and the data area with the **###** identifier. The number of properties transferred by EPLAN is irrelevant for these templates. The rest of the template can be structured in any desired manner.

To allow EPLAN and Excel to select the different areas (which properties are editable or not), the Excel template has a second *Format* worksheet.

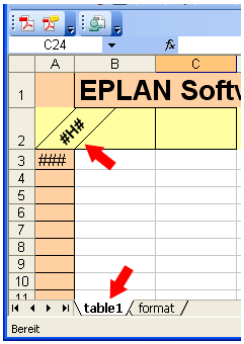


Fig. 6.119
Worksheet table 1

This worksheet has several entries. These entries should not be deleted, nor should the text be modified. However, the format of the cells (background color, font, text size, etc.) can be changed to suit your needs. Changing the cell format affects the later visual appearance of the worksheet.

For the sake of completeness, the meanings of these entries are listed here.

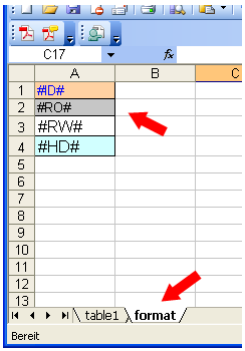


Fig. 6.120
The Format worksheet

- **#H#**: Header area (headers)
- **#HD#**: Automatically added header data
- **#F#**: Footer
- **###**: Data area
- **#ID#**: Identifier for a property
- **#RO#**: Property has the read-only format and cannot be changed; EPLAN ignores these values when reading back the data.
- **#RW#**: Property has the read-write format and the values can be changed as desired. However, illogical values are not adopted by EPLAN.

The Excel template is now complete. You now save this in any desired directory. The ideal location would be the EPLAN **templates** directory. We still don't have the EPLAN scheme allowing the properties to be transferred to Excel.

To create this scheme, you use the **UTILITIES/EDIT PROPERTIES EXTERNALLY/SETTINGS (PAGES)** menu to open the **SETTINGS/EXTERNAL EDITING** dialog.

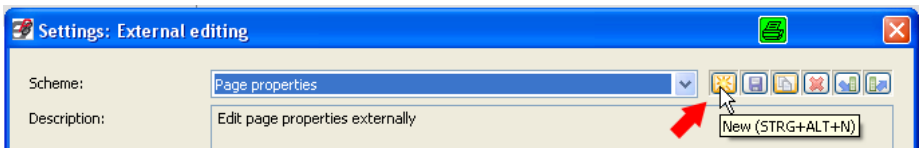


Fig. 6.121 Creating a new scheme

Settings for transferring data to Excel

Click the button to create a new scheme. The **NEW SCHEME** dialog is displayed. Here you enter a descriptive name and a description for the scheme and apply these settings by clicking **OK**. You can now fill the new scheme with the desired values.

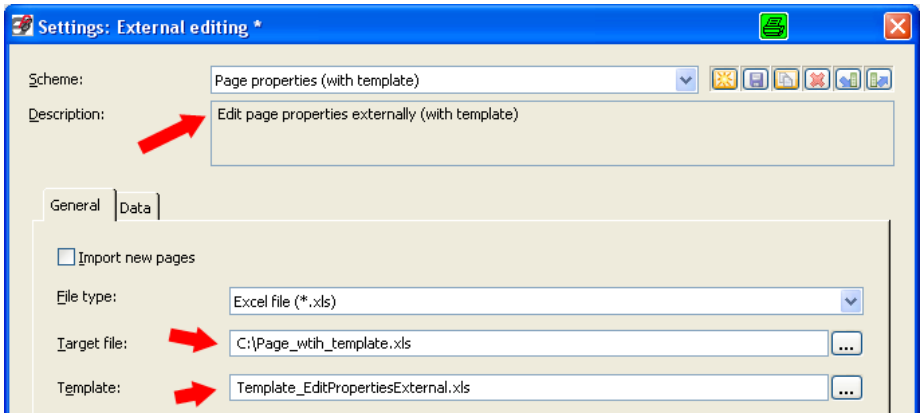


Fig. 6.122 Settings in the General tab

In the *File type* selection field, you select the Excel file (**.xls*) value. You can enter any value into the *Target file* field. The *Template* field is important. Here, you should select the Excel template that you just created for the scheme. This completes the entries in the *General* tab.

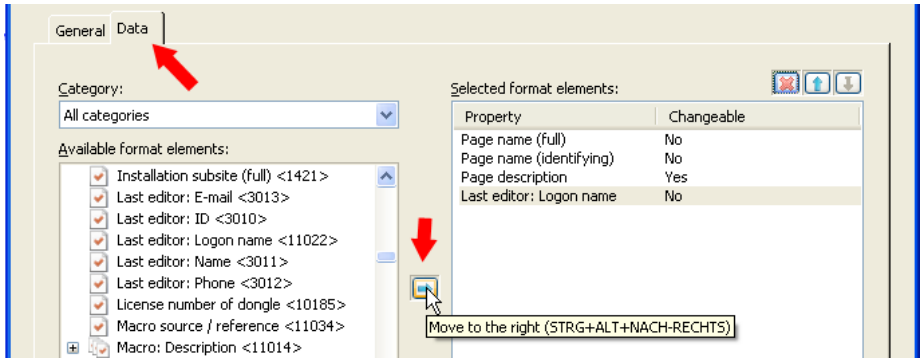


Fig. 6.123 Settings in the Data tab

To transfer the properties to Excel, you must now select the desired properties in the *Data* tab. This is done in a similar way to selecting properties for labeling data. As you can see in Fig. 6.123 for this example, a number of properties have been added to the *Selected format elements* field.

This completes all of the required steps. The scheme is now saved, and the dialog can be confirmed by clicking **OK**.

6.8.1.2 Export data – sample pages

To be able to export page properties of a project with this scheme so that you can externally edit them, you call up the **page navigator** and select the desired project with the left mouse button or individually select the desired pages in the page navigator.

NOTE: If no pages are selected, then the **EDIT PROPERTIES EXTERNALLY/ EXPORT PAGES** function cannot be executed.

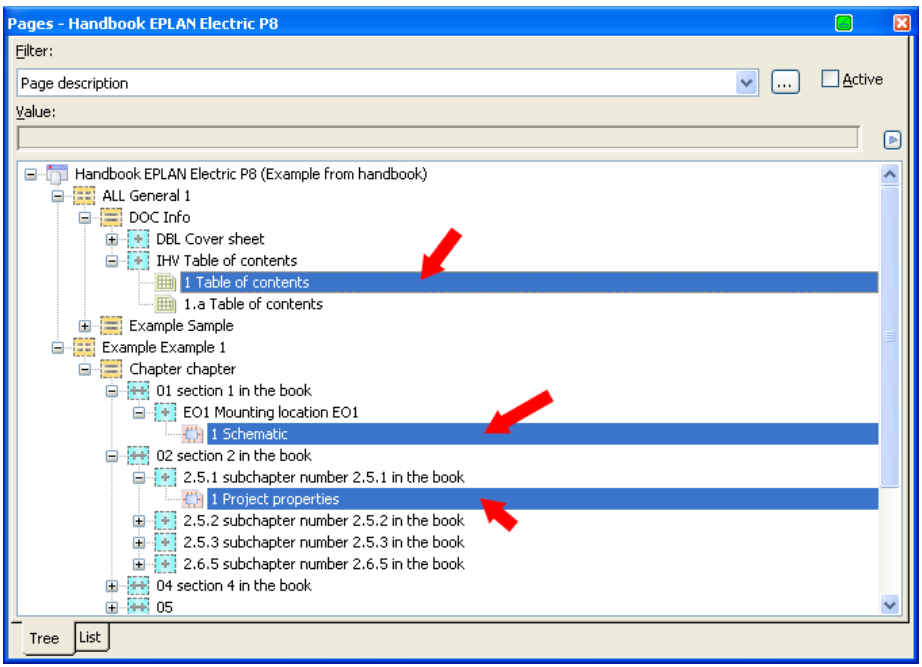


Fig. 6.124
Selected pages

You then use the **UTILITIES/EDIT PROPERTIES EXTERNALLY/EXPORT PAGES** menu item to open the **EDIT PAGES EXTERNALLY** dialog. You must select and apply the required *scheme* in the **Scheme** selection field. Select the desired option in the **Language** selection field and then select either a single language or all of the languages set in the project.

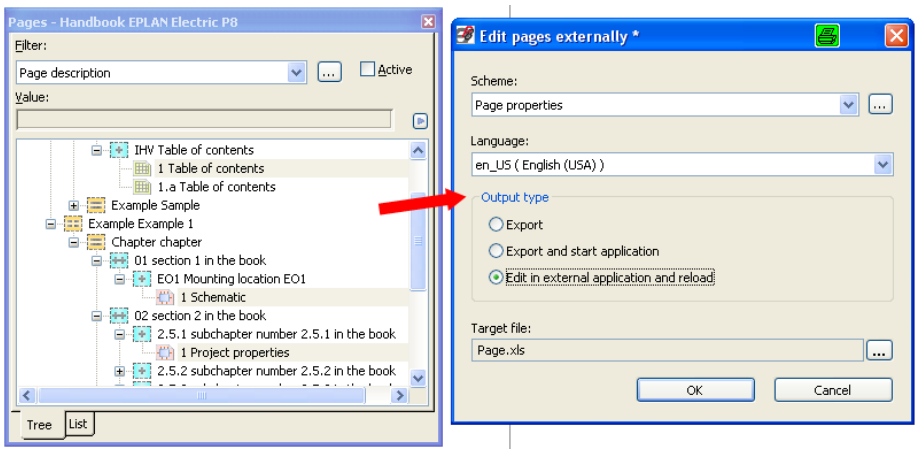


Fig. 6.125
Export settings

The **Output type** options are the most important point for outputting the data. The *Export* output type generates the data and creates the output file, but does not open it. The *Export and start application* option generates the output file, and then opens this in the specified application (in this case Excel), where you can then edit and save the output file. This does not update any data in EPLAN. The saved output file can later be read back into the EPLAN project via **UTILITIES/EDIT PROPERTIES EXTERNALLY/IMPORT**. This option is useful when the data is to be further edited, e. g. by a customer.

The last option, *Edit in external application and reload*, includes a transfer of the data from EPLAN to Excel. It can now be edited there. After editing, the file can be saved and Excel can be exited. This is followed by a confirmation prompt asking if the data really should be imported. If you confirm the dialog by clicking **YES**, then EPLAN synchronizes the data from the Excel file with the project data.

To export the data, edit it in Excel, and then re-synchronize it with the EPLAN project, you should select the last output type. The target file (where the data should be written to) is specified in the scheme.

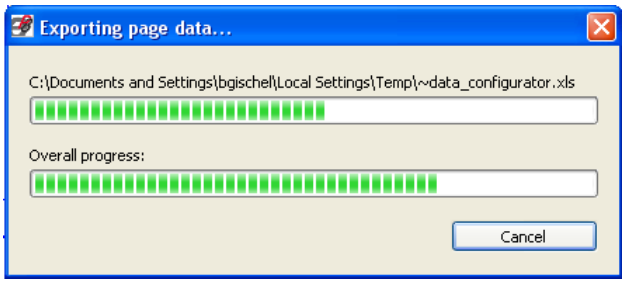


Fig. 6.126 EPLAN is exporting the data

Clicking the **OK** button generates the output file and then starts Excel. The loaded data can now be edited or changed.

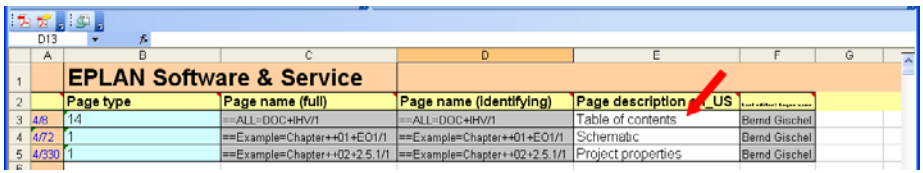


Fig. 6.127 Editing data in Excel

After making all your changes, you save the Excel file (click the button once or press **CTRL + S**) and can now close Excel.

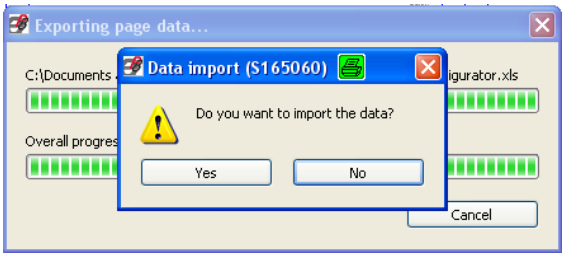


Fig. 6.128 Data import dialog

This is followed by a confirmation prompt asking if you really want to import the changed data. When you confirm this dialog by clicking **YES**, then all modified data is written back to the project and is now available there.

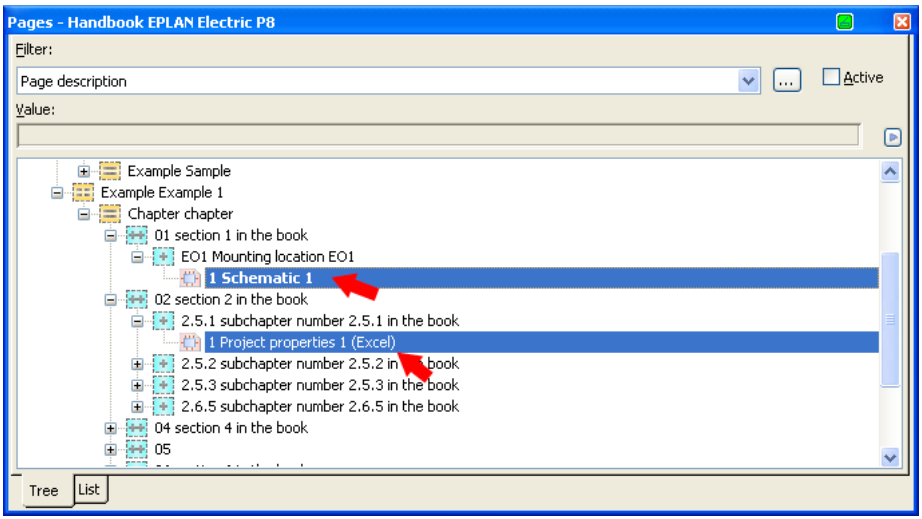


Fig. 6.129 Changed page description

NOTE: It is not possible to undo this action (importing changed data).

6.8.2 Importing

Up to now, we have exported data, edited the data externally, and then written it back to the EPLAN project. Immediately writing back the data may sometimes not be desired.

In this case, in the **EDIT PAGES EXTERNALLY** dialog, you should set the output type to *Export* or *Export and start application*. This then only transfers the data to Excel and either

immediately stores it in the target file, or writes it to the application where it can then be edited further and subsequently saved.

You can also exit project editing in the aforementioned cases. The exported data can now be edited as desired, and then read back into the project at a later time.

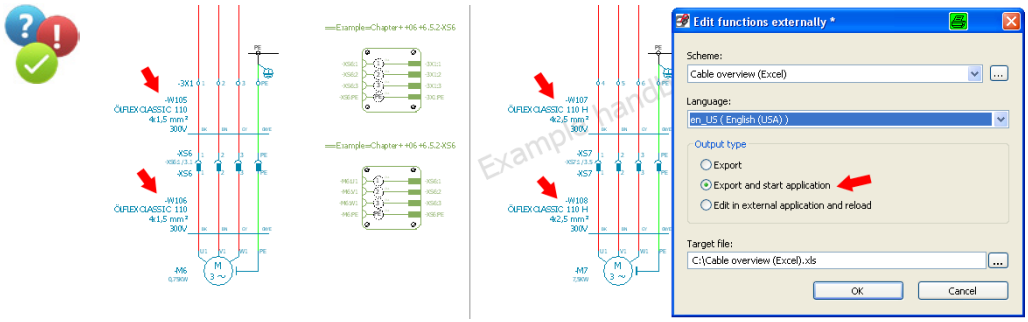
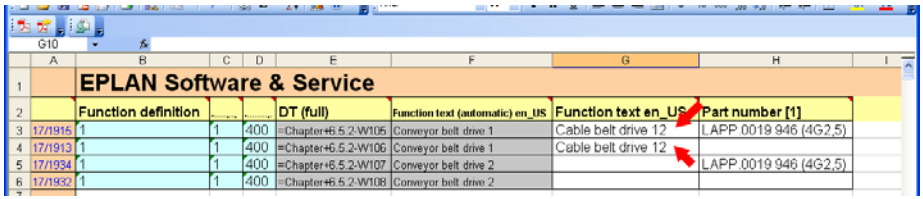


Fig. 6.130 Data sets selected for export (here cables)

This will be explained with an example: In this example, cable function texts will be entered and changed. You either select all cables in the cable navigator (by selecting the project) or select only the required cables on a page. Using a scheme that contains the properties you want to change, you export the functions and save them to an Excel file. This file can now be edited by an employee who might not have EPLAN installed.



	A	B	C	D	E	F	G	H
1	EPLAN Software & Service							
2		Function definition			DT (full)	Function text (automatic) en_US	Function text en_US	Part number [1]
3	17/1915	1	1	400	=Chapter+6.5.2-W105	Conveyor belt drive 1	Cable belt drive 12	LAPP 0019 946 (4G2,5)
4	17/1913	1	1	400	=Chapter+6.5.2-W106	Conveyor belt drive 1	Cable belt drive 12	
5	17/1934	1	1	400	=Chapter+6.5.2-W107	Conveyor belt drive 2		LAPP 0019 946 (4G2,5)
6	17/1932	1	1	400	=Chapter+6.5.2-W108	Conveyor belt drive 2		

Fig. 6.131 Editing an external file

The cable function texts can now be entered or modified in the Excel file, and the saved Excel file can now be read back (imported) into EPLAN.

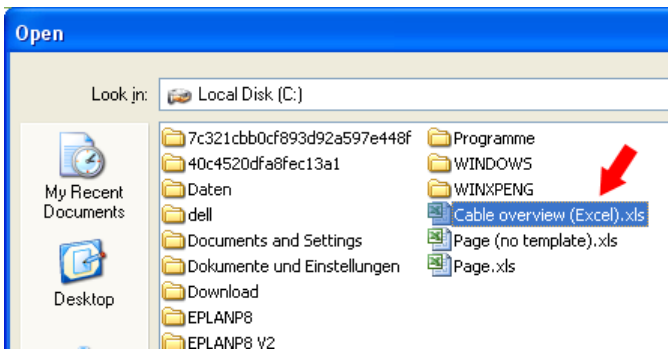


Fig. 6.132 Selecting the saved file

Open the **UTILITIES/EDIT PROPERTIES EXTERNALLY/IMPORT** menu item and select the Excel file to be imported. The **OPEN** dialog is displayed. Here, you use the normal Windows functions to select and open the file using the **OPEN** button. EPLAN now imports the file **immediately and without any other prompts** and writes the modified data back to the project. This completes the import.

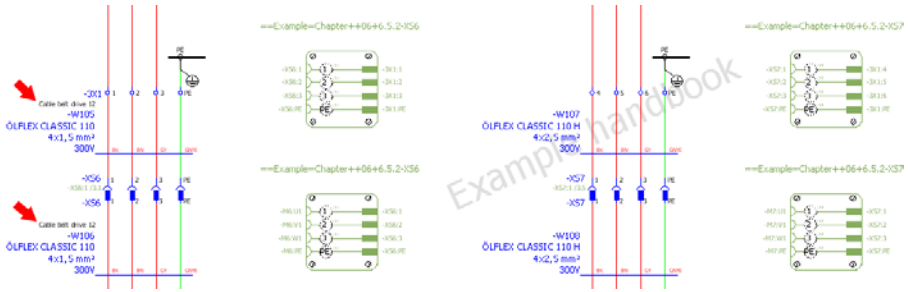


Fig. 6.133 Externally modified data after the import

NOTE: No confirmation prompt is displayed at this point. EPLAN imports this data into the project immediately after the file is opened. There is no **UNDO** function at this point.

7 Management tasks in EPLAN

A lot of different types of data must be managed in EPLAN. EPLAN has functions that can handle these various, and sometimes different, tasks. This chapter will explain some of these functions.

In EPLAN, you not only need to manage all types of structure identifiers, but also project messages that occur either online while editing projects, or offline as a result of project checks. You need to organize and manage parts data and the various layers of a project as well. Thus the collective term “management”.

7.1 Structure identifier management

As soon as a project has a page structure other than **Sequential numbering**, the different structure identifiers in the project will eventually be used somewhere.

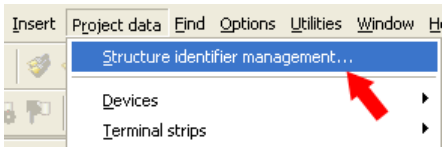


Fig. 7.1
Structure identifier management

Every possible identifier block in a project is assigned a specific identifier that can consist of a string of letters and/or numbers. These identifiers are known as structure identifiers.

Structure identifier management is called up via **PROJECT DATA/STRUCTURE IDENTIFIER MANAGEMENT** menu item. EPLAN opens the **IDENTIFIERS – [PROJECT NAME]** dialog.

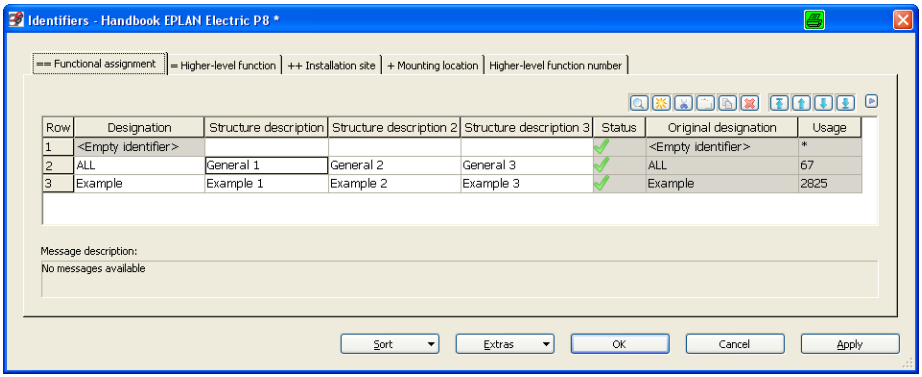


Fig. 7.2 The control center of structure identifier management

The possible structure identifiers such as **higher-level function**, **installation site** or **mounting location** are managed centrally in EPLAN. The sequence of these structure identifiers and the sequence of the pages in a project are closely related. The page structure is initially sorted in the page navigator in the same way as the structure identifiers.

Several mounting locations are entered in structure identifier management (not all mounting locations need to have been used previously in the page structure).

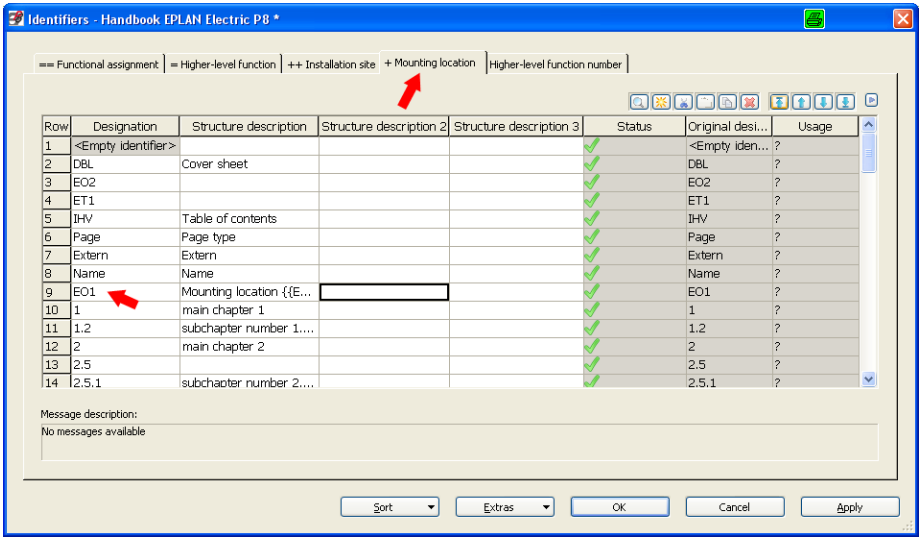


Fig. 7.3 Sequence of the mounting location

The page structure in the page navigator is now sorted according to the defaults in structure identifier management (Fig. 7.4).

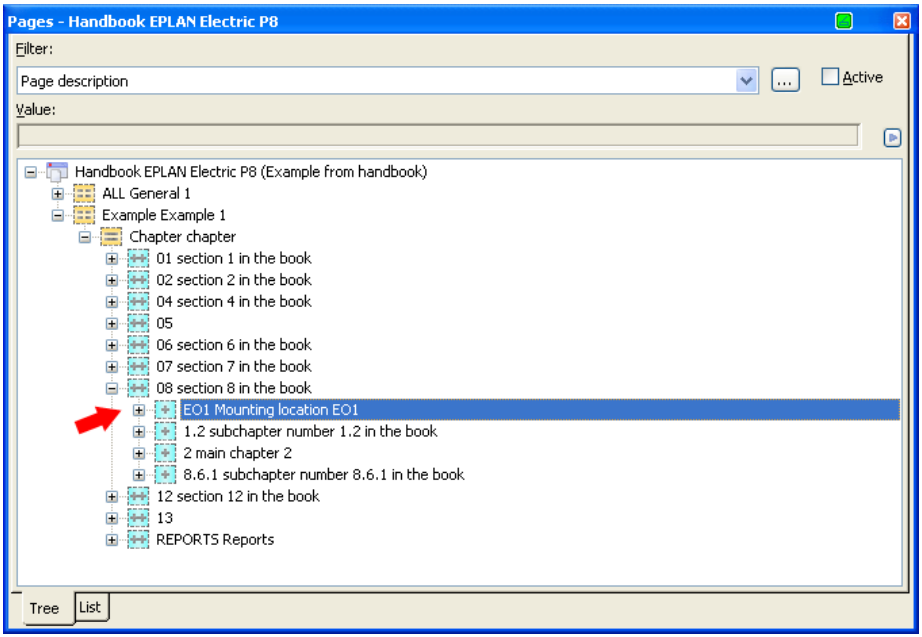


Fig. 7.4 Sequence in the page navigator

For example, if you want to sort mounting location ++EO1 after mounting location ++8.6.1 in the page structure, you change this sequence in structure identifier management.

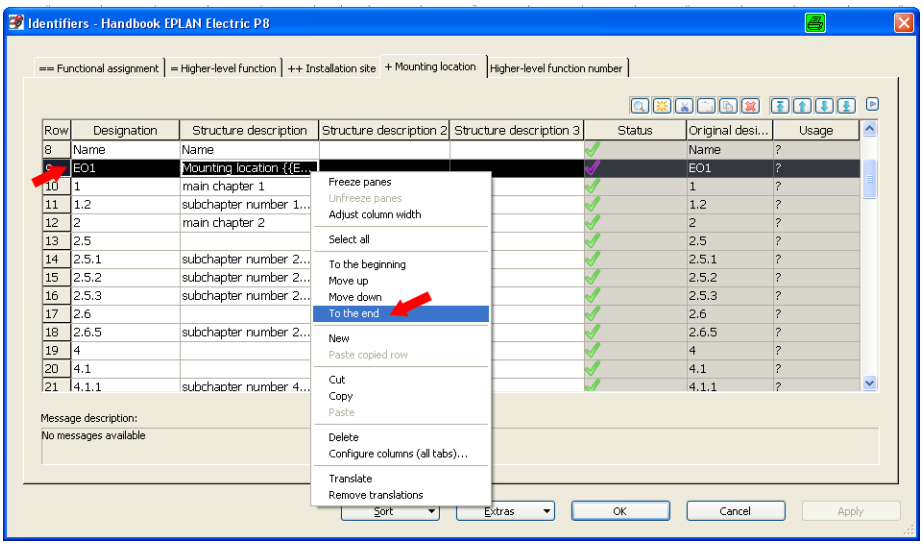


Fig. 7.5 Changing the sequence of structure identifiers

This is done, for example, by selecting the row and then calling the popup menu via the right mouse button. Here, you will see functions such as **TO THE BEGINNING** (will be moved there) or **DOWN** (will be moved down by a row) and many others.

Once sorting has taken place and after you click the **OK** button at the bottom of the dialog, EPLAN will ask whether the changes should be applied.

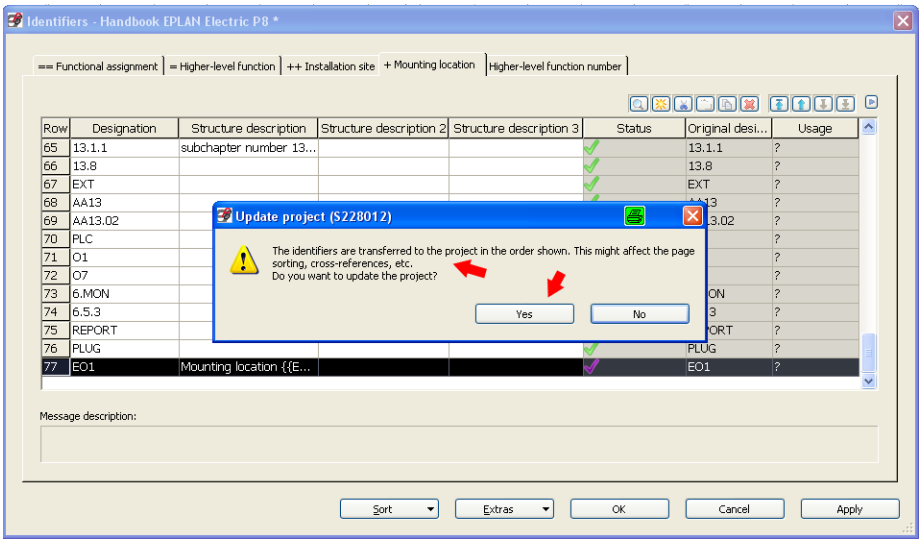


Fig. 7.6
Apply project data changes dialog

When you confirm, the changes are applied and the page structure is adjusted in the page navigator.

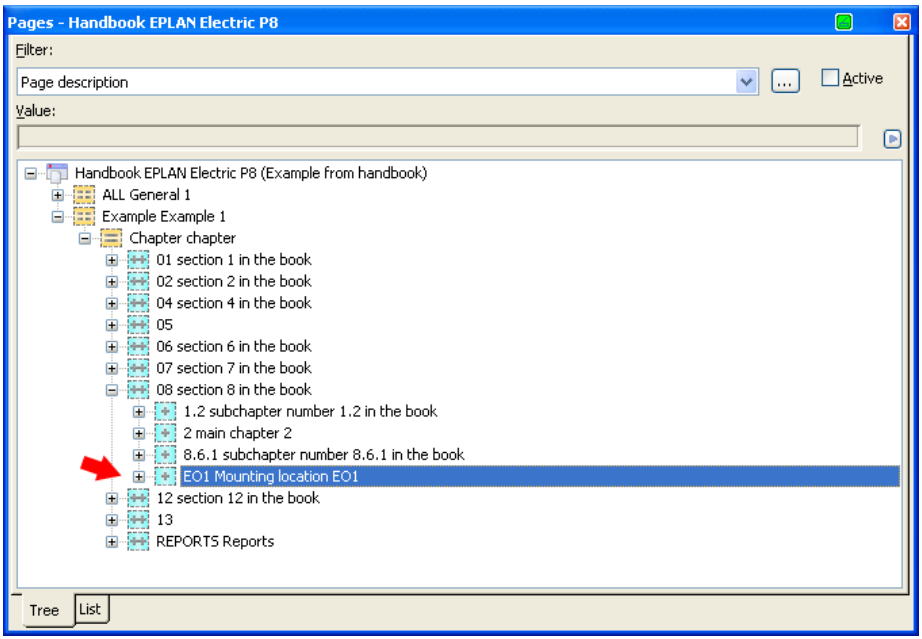


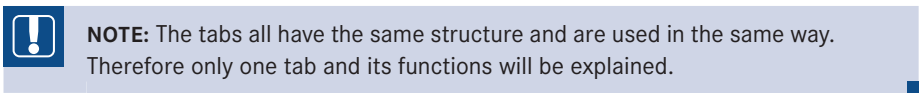
Fig. 7.7
Changed sequence

This is only one of many functions in structure identifier management. Among other features, it is also possible to copy structure identifiers from an Excel table into structure identifier management. This eliminates the time-consuming process of manually entering the data into EPLAN.

A detailed description can be entered for every structure identifier. As a result, reports or graphical generation of structure identifier overviews can be filled with this information and the data need not be entered manually. The graphical output and the information contained therein depend, of course, on the structure of the forms used.

7.1.1 Tabs in structure identifier management

The available tabs correspond to the page structure that was selected at the time the project was created.



NOTE: The tabs all have the same structure and are used in the same way. Therefore only one tab and its functions will be explained.

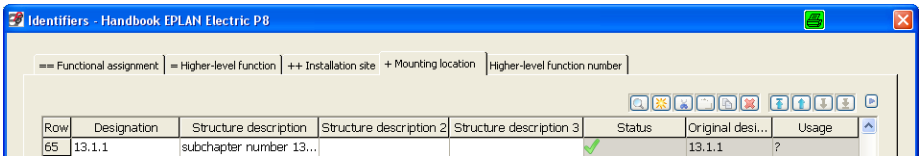


Fig. 7.8 Tabs in structure identifier management

EPLAN can manage up to seven structure identifiers in one project, the so-called identifier blocks. The following structure identifiers are possible:

Possible structure identifiers in EPLAN

- **Functional assignment**: Superior to the **higher-level function** identifier block. The following prefixes are possible for the functional assignment in EPLAN: "==", "#" or "≠"
- **Higher-level function**: The **higher-level function** identifier block is used, for example, for the identifiers of system parts. A possible prefix in EPLAN is "=".
- **Installation site**: The **installation site** identifier describes the physical site of devices or system parts. EPLAN uses "++" as the identifier.
- **Mounting location**: The mounting location describes the place where devices are mounted/installed. For example, pushbuttons would have a mounting location of "control panel". EPLAN allows "+" as a prefix for this.
- **Higher-level function number**: The higher-level function number is another structuring of devices that is used, for example for fluid devices. The prefix can usually be freely chosen, but should be compatible with the other identifier blocks.
- **Document type**: The document type is an identifier block for the so-called KKS (power station identifier system). EPLAN allows a prefix of "&". A space character can also be used for this.

- **User-defined**: This identifier block can be freely chosen. Any character can be used as the prefix. However, the prefix used should, of course, not conflict with the existing identifiers.

Any number of identifiers can be entered into an **Identifier block**. New identifiers are (or can be, depending on the setting) sorted into structure identifier management, either at the end or alphabetically (depending on the setting).

Which type of sorting is used during editing is defined in a user-specific setting. This setting is accessible via **OPTIONS/SETTINGS/USER/DISPLAY/IDENTIFIER**.

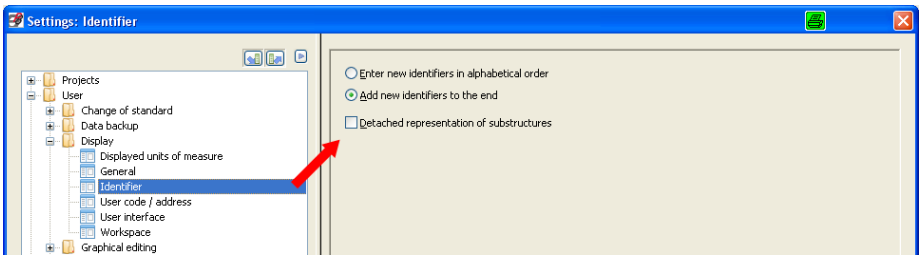


Fig. 7.9 Settings for Place identifiers

In order to assign **Descriptions** to these automatically sorted identifiers, structure identifier management must be started and the descriptions entered later.

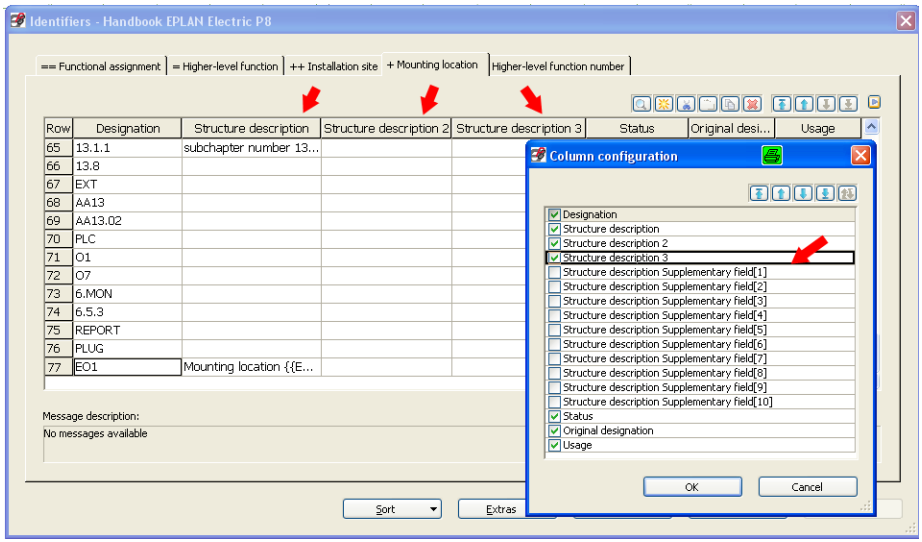


Fig. 7.10 Entering structure designations

EPLAN allows you to manage several structure designations per identifier. These can be structure designations 1 to 3 or the structure designation supplementary fields 1 to 10. These are activated by right clicking to call up the popup menu and then selecting **CON-**

FIGURE COLUMNS (ALL TABS). EPLAN then displays the CONFIGURE COLUMNS dialog. In this dialog, you can select or deselect the appropriate properties for display.

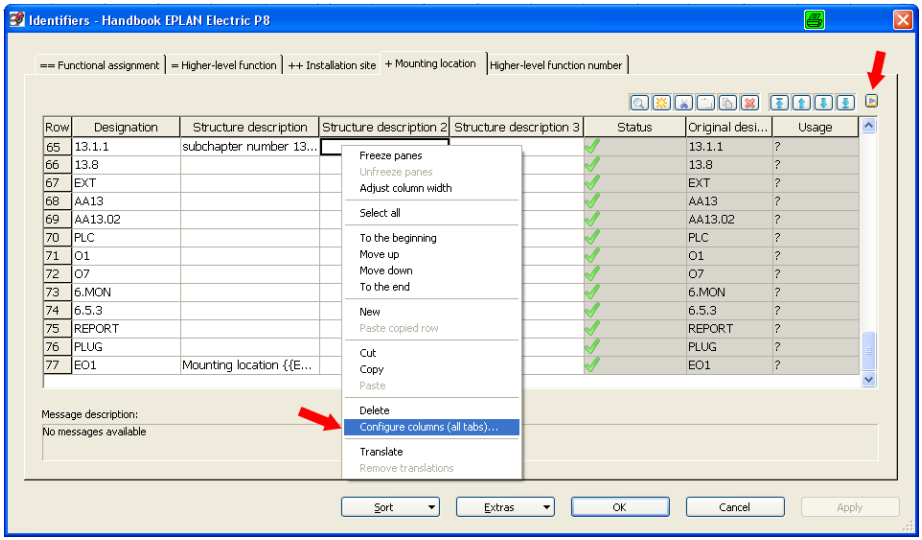


Fig. 7.11 Configure identifier/column dialog

7.1.2 Graphical buttons

When working on a project, it is not always possible to ensure that only desired identifiers are entered into the identifier structure. Various types of copy and macro actions usually create unwanted entries in structure identifier management.

For this situation, EPLAN allows convenient subsequent editing of the structure identifiers via functions such as Move or Delete in the graphical toolbar . These (and more) functions are also available via the right-click popup menu. The graphical buttons have the following functions:

	Opens the **REPLACE** dialog. Allows the user to perform extensive find and replace actions. This usually replaces the manual editing of identifiers.
	Inserts a new row before the currently selected row
	Cuts the selected cell(s) and allows this (these) selected cell(s) to be inserted at a different place in the **IDENTIFIERS** dialog
	The **INSERT** function inserts copied (or cut) cells at the specified location
	Copies selected rows to the clipboard
	Deletes selected rows
	Moves the selected row to the beginning

	Moves the selected row one position upwards
	Moves the selected row one position downwards
	Moves the selected row to the end

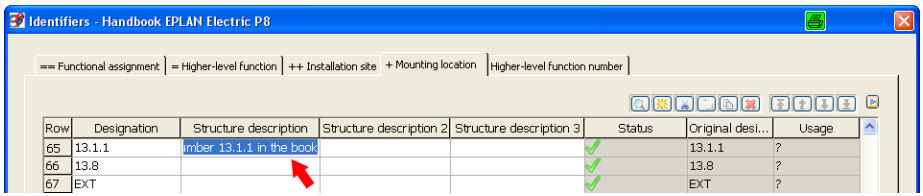


Fig. 7.12 Direct editing using F2

To edit an identifier, you click the corresponding field and can then edit the entry. Or you can press the **F2** key to edit entries. Either option opens up different right-click popup menus. But in either popup menu you can use similar functions, such as **COPY**, **REMOVE** or **TRANSLATE**.

Single selection

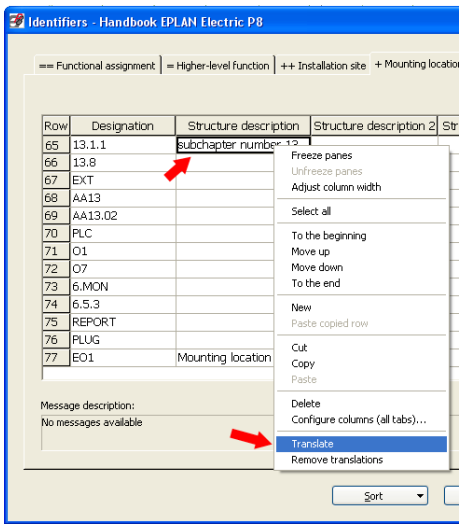


Fig. 7.13 Selected row

Direct selection

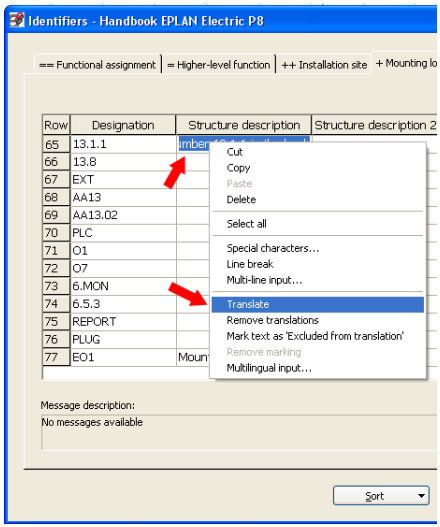


Fig. 7.14 Row selected via F2

TIP: The right-click popup menus can also be accessed via the button.

To translate column entries, you first select the entries you want to translate and then use the **CTRL + L** shortcut key or the right-click popup menu to translate them. If EPLAN finds multiple matching entries in the dictionary, it displays the **FOUND WORDS** dialog. You can then select the most suitable entry and apply it by clicking **OK**.

7.1.3 Sort menu

The **SORT** menu provides a few automated sorting options for identifiers. Regardless of the sorting used, it is initially a temporary sorting and will not be applied until the **IDENTIFIERS** dialog is closed by clicking **OK** or **APPLY**.

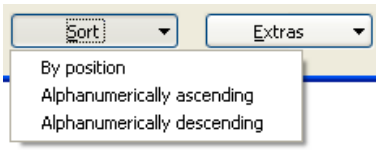


Fig. 7.15
Sort button

Using the **SORT** menu and the **BY POSITION** option, EPLAN sorts the identifiers according to the internal item numbers. The **ALPHANUMERICALLY ASCENDING** option sorts the identifiers according to their character strings (numbers take precedence over letters). The **ALPHANUMERICALLY DESCENDING** option sorts the identifiers in descending order.

The identifiers do not generally need to be selected in order to arrange them using the **SORT** option. The cursor must only be in a field. Note that when the identifiers are automatically sorted, the sorting of the page structure follows that of the identifiers.

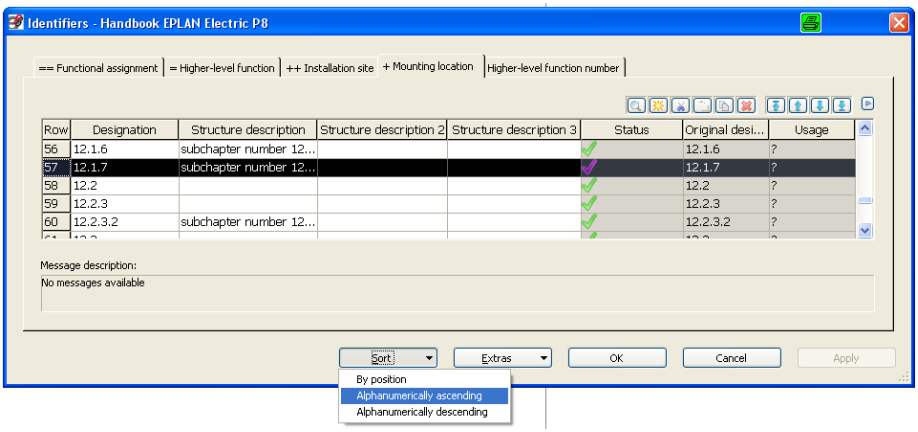


Fig. 7.16 Cursor position

NOTE: A new sorting always applies to all identifier types collectively. It is not possible to sort individual identifiers automatically.

7.1.4 Extras menu

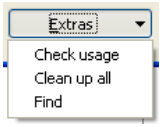


Fig. 7.17
Extras button

The **EXTRAS** button displays an additional menu. This menu contains options allowing unnecessary entries to be cleaned from the database and contains a **FIND** function for searching the database.

The **CHECK USAGE** option works as follows: When this function is executed, EPLAN checks all identifiers to see if they are used and exist in the project or if they are only entries in the identifier database. These can be entries, for example, that were created from previous copy operations and which can now be deleted. EPLAN designates such identifiers that are no longer used with a **No** in the **Usage** column. These identifiers can be safely removed from the database using the graphical **[DELETE]** button.

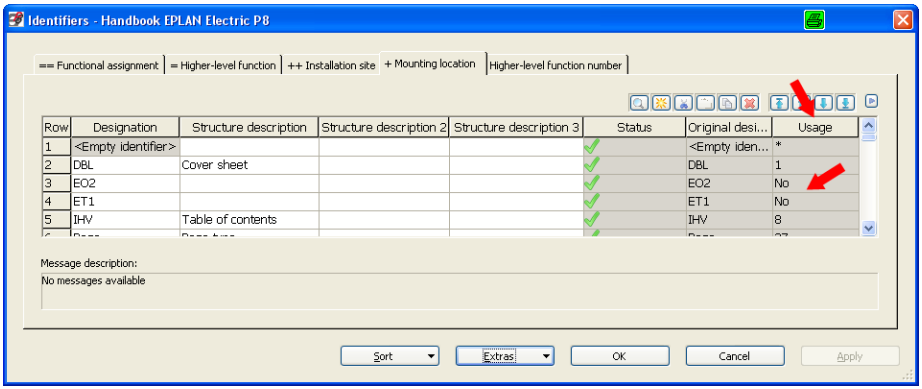


Fig. 7.18 Check usage

The **CLEAN UP ALL** option automatically removes all entries from the database that are no longer required, without further prompting.

NOTE: Once this function is started, no additional prompts are displayed. The entries are immediately checked for usage, and all unnecessary identifiers are immediately removed.

Since the system does not "know", for example, when identifiers for higher-level functions or mounting locations have been inserted for later use, these identifiers are also removed. You should therefore be careful when using this function. When you click the **OK** button, these additional (unused) entries are permanently removed from the database.

The **FIND** option displays the **REPLACE** dialog. You can use this to locate entries in the database and replace them with other strings. Find and replace allow for a simple way of replacing terms (or partial terms) with other terms. Here too you should work carefully because no UNDO function is available. Replaced really means replaced!

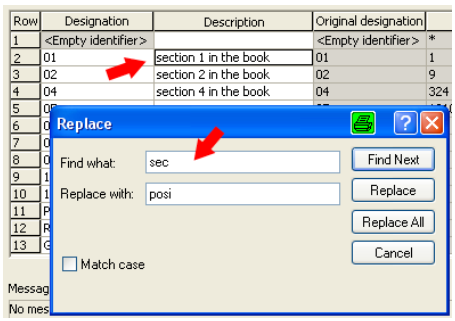


Fig. 7.19 Before replacement

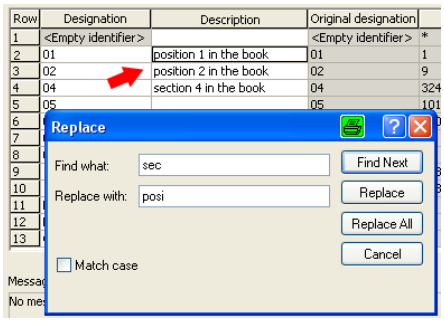


Fig. 7.20 After replacement

NOTE: An unwanted replace action can only be undone by exiting the **IDENTIFIERS** dialog with the **CANCEL** button.

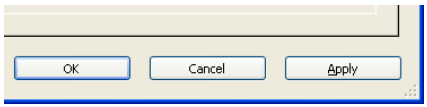


Fig. 7.21 More buttons

In addition to the previously described **SORT** and **EXTRAS** buttons, the dialog also has the **OK**, **CANCEL** and **APPLY** buttons.

You use the **OK** button to exit the **IDENTIFIERS** dialog and, depending on the actions you have performed, you may need to confirm your changes in a subsequent dialog.

Since identifier editing has no UNDO function, the **CANCEL** button is very important, because it allows you to discard all changes made.

The **APPLY** button applies (saves) the changes made in structure identifier management without closing the **IDENTIFIERS** dialog. When performing comprehensive actions such as renaming or deleting of unnecessary identifiers, you should first save the contents of the **IDENTIFIERS** dialog via the **APPLY** button before performing the replacement action. If something goes wrong with the replacement action, you can click **CANCEL** to exit the Identifiers dialog. The status will then be exactly the same as before the replacement action started.

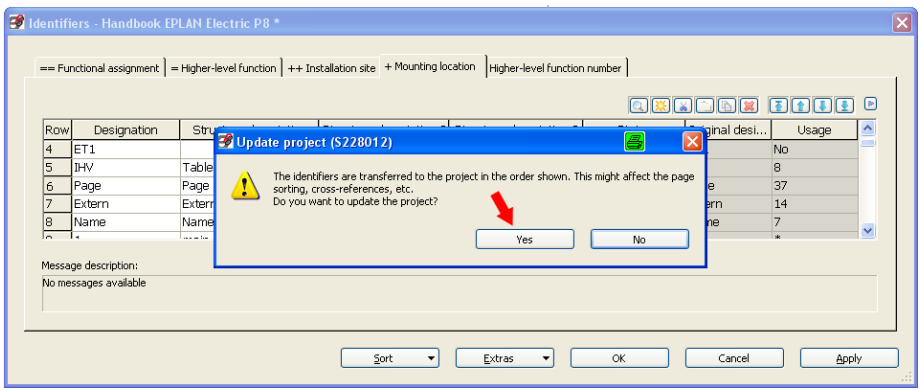


Fig. 7.22 Prompt after you click the **APPLY** button

NOTE: A kind of “interim save” (**APPLY** button) is necessary when editing the identifier structure so you don’t lose all changes.

7.2 Message management

Why does message management exist? EPLAN is an online system, and message management is a perfect tool for immediately checking (i. e. online) whether the action you have just performed was correct (depending on the project check settings).

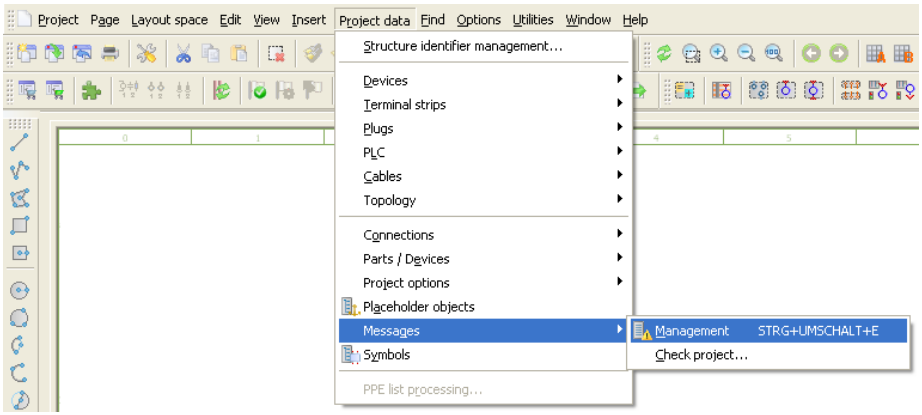


Fig. 7.23 Message management

Message management is started as an extra dialog via the **PROJECT DATA/MESSAGES/MANAGEMENT** menu, or via a shortcut key, such as **CTRL + SHIFT + E**.

The **MESSAGE MANAGEMENT** dialog can be placed anywhere on the desktop and can be displayed/hidden via a (user-defined) keyboard shortcut such as **CTRL + SHIFT + E**. You do not need to keep this dialog open when editing projects. However, if you wish to receive immediate notification of faulty data entries (depending on the types of checks set), then you should keep message management open.

TIP: The greater the number of messages in message management, the slower project and/or graphical editing may be. Therefore, you should hide the dialog unless you need it.

When project editing is almost finished and the project is to be checked for faulty entries, the dialog can be displayed for extra information, since all **Errors, Warnings**, and **Notes** generated by the corresponding **Check run** are listed here.

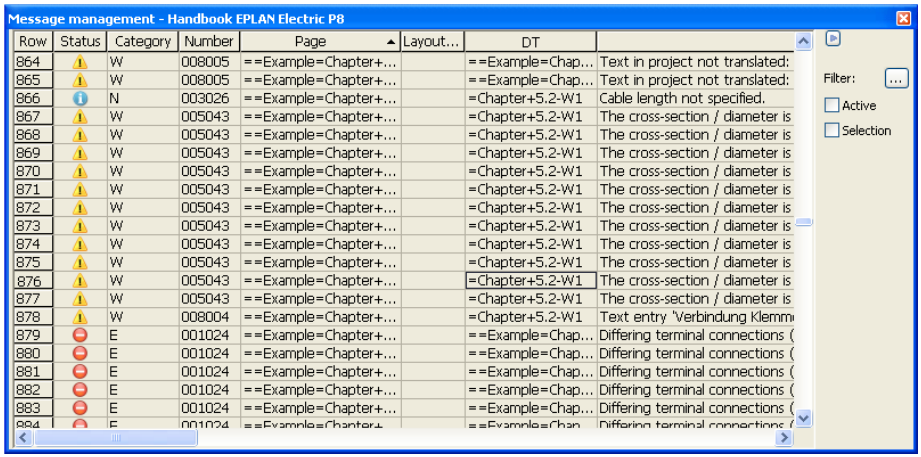


Fig. 7.24 Message management

When you start editing a project in EPLAN, various actions, such as inserting entire pages from other projects or inserting existing macros into the project, can result in unwanted data in the various databases. For example, unwanted device combinations may be created or new structure identifiers may be added to the project. You cannot really avoid the creation of such data that does not really belong to the project without being constantly "interrupted" in your actual project editing. To make sure that you do not forget to change this unwanted data, EPLAN provides an online monitoring feature, or you can use manual check runs to check the project using a specified scheme (offline) and generate messages in the message database.

7.2.1 The visual appearance of message management

The dialog supports the user with the following features: Display of the message type, the message priority, and other information such as a brief message description or the jump point to the faulty position in the project. Small colored icons make it easy to visually distinguish between the different types of messages.

	Error	For example: "Duplicate connection point description"
	Warning	For example: "Connection point description missing"
	Note	For example: "A placement lies outside the range"

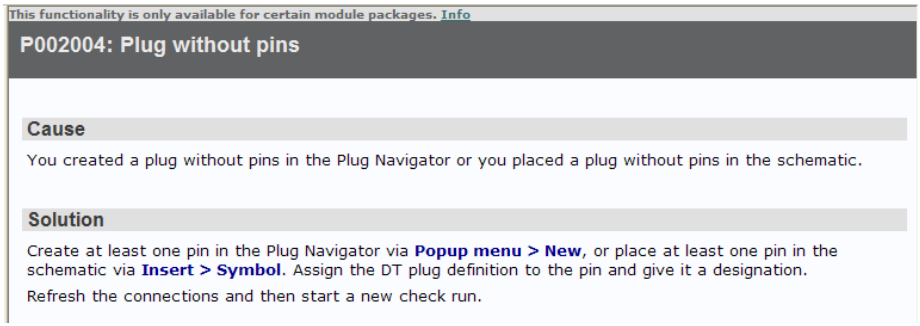


Fig. 7.25 Note on message in online help

If you position the cursor in a message row and then press the F1 key, EPLAN displays the online help for the corresponding message number, lists the possible cause(s), and suggests one or more solutions for the selected message.

7.2.2 Project checks

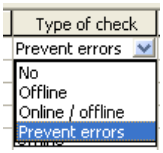


Fig. 7.26
Types of check

EPLAN generally distinguishes between **offline checks**, **online/offline checks** and the **Prevent errors** check type, and also allows you to "remove" checks entirely from the check run, i.e. to switch off the check with the *No* check type. You should not use this option very much since you run the risk of certain true project editing errors not being discovered (e.g. duplicate DTs) until the actual plant is built, which will present a much bigger problem.

The user decides what type of check routines to use. The **online check routine** is the first choice, but is also the most computing-intensive because all the data and resulting messages must be continually checked and updated. This, of course, has a negative effect on the performance during project editing. The **offline check routine** is the "second choice", but I prefer to use this one. After all, who manages to constantly correct all the online messages (errors, warnings or notes) while editing a project.

Check runs are started via the **PROJECT DATA/MESSAGES/CHECK PROJECT** menu or directly via the **CHECK PROJECT** function in the opened message management via the right-click popup menu.

7.2.3 Message classes and message categories

EPLAN allows the user to define the categories that messages belong to. This has the great advantage of allowing company-specific check runs to be developed that exactly match the way in which projects are developed within the company.

Not all messages generated or supplied by EPLAN are, for example, true error messages. The message *<<017003 Incorrect identifier>>* is one example. It can be an error message, a warning, or a note, depending on the project editing sequence. You can create your own schemes for check runs in message management so that these messages are adjusted to exactly suit your project editing process.

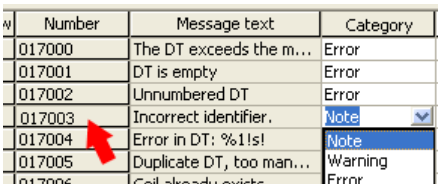


Fig. 7.27 Check run message 017003

Use the **PROJECT DATA/MESSAGES/CHECK PROJECT** menu to open the **CHECK PROJECT** dialog. In this dialog, you can select a scheme in the *Settings* field for the subsequent check run.

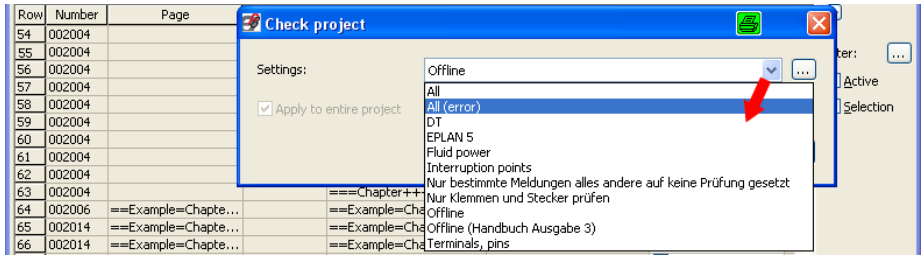
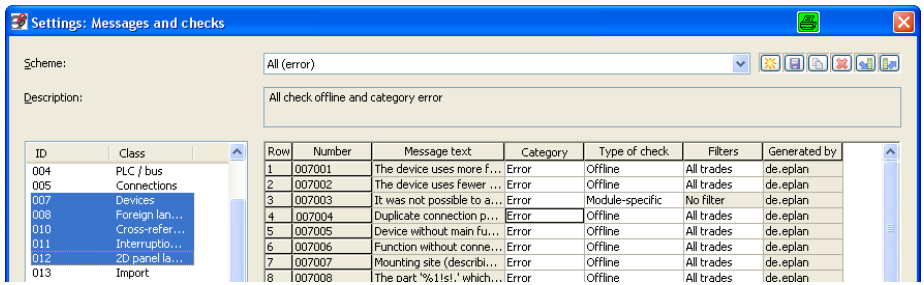


Fig. 7.28 Check project dialog

To add your own scheme to this selection, you click the ... selection button. The **SETTINGS/MESSAGES AND CHECKS** dialog is displayed. To create a new scheme, you click the **[NEW]** button in the upper toolbar.



Settings for messages and check runs

Fig. 7.29 Dialog for creating test schemes

EPLAN opens the **NEW SCHEME** dialog. Here you assign a suitably descriptive **Name** and **Description**. When you click the **OK** button, the scheme is saved and automatically entered into the **Scheme** selection field of the **SETTINGS/MESSAGES AND CHECKS** dialog.

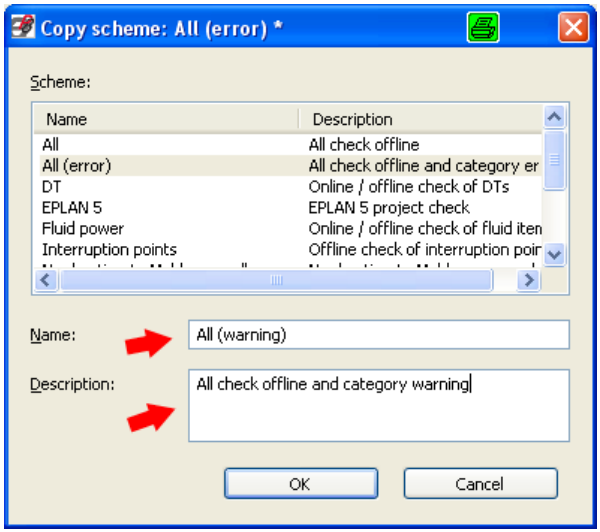


Fig. 7.30
Creating a new scheme

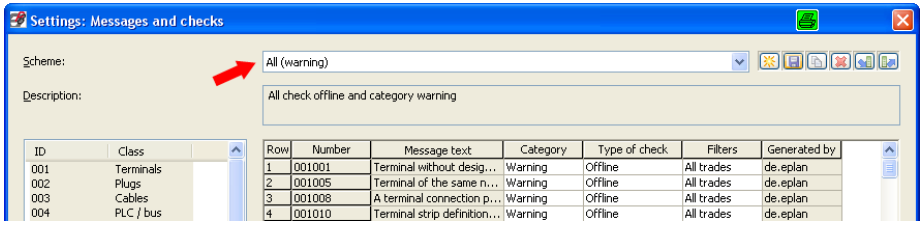


Fig. 7.31 Example of a new scheme

The individual messages can now be set and edited according to the desired specifications.

Every message belongs to a message class. There are currently 26 message classes. They range from message class 001 Terminals to message class 999 External. The message classes cannot be extended or changed; they are permanently defined by EPLAN.

This means that every message begins with a specific three digit number. This defines the message class. In our example, the message *<017003 Incorrect identifier>* is assigned the DT message class. The rest of the message number is a three-digit number.

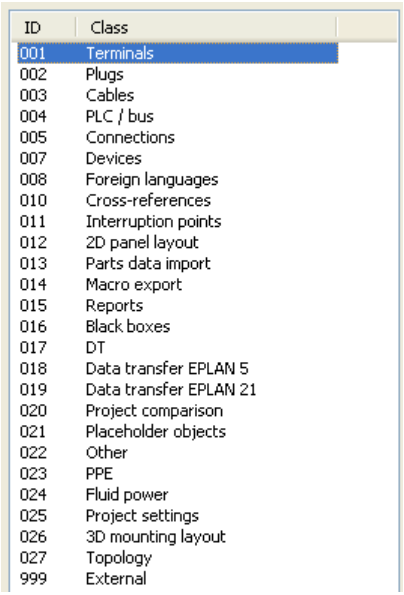


Fig. 7.32
Message classes

Every message number has a corresponding message text. As with the message number, this message text is defined by EPLAN and cannot be changed. However, in order to return to their own scheme, users can set the category and type of check for this message.

Message text
A pin connection point has too many targets: %1!s!. Maximum num...
Pin without targets.

Fig. 7.33
Message text

EPLAN divides messages into three categories. There are **Notes**, **Warnings**, and **Errors**. The user must now decide on one of these three message categories. Once the category for the message has been defined, a check type must be decided. There are several check types in EPLAN.

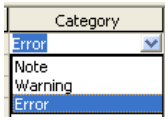


Fig. 7.34
Category

The **No** type of check does not carry out any checks for this message either automatically or manually. This will be clarified with a short example.

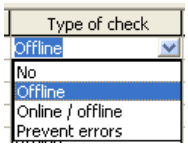


Fig. 7.35
Types of check

Number	Message text	Category	Type of check
001001	Terminal without designation.	Error	No

Fig. 7.36
Type of check set to "No"

A terminal will be positioned in the schematic. This terminal will not given a designation, and the check for the message *Terminal without designation* will be set to **No**. If a check run is now carried out, there will not be any check run message *Terminal without designation*.

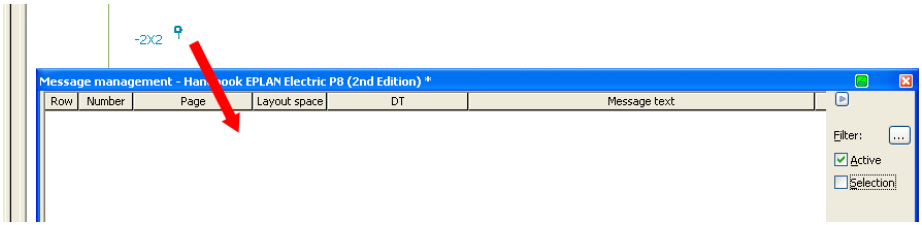


Fig. 7.37
Message management after completed check run

The check type **Offline** only generates messages when a check run is manually started.

Row	Number	Message text	Category	Type of check
1	001001	Terminal without designation.	Error	Offline

Fig. 7.38
Type of check set to "Offline"

Now, a terminal will again be positioned in the schematic. This terminal will also not be given a designation, and the check for the message *Terminal without designation* will be set to **Offline**. EPLAN does not yet check this terminal during placement. Only when the check run has been started manually (offline) does EPLAN generate the check run message *Terminal without designation*.

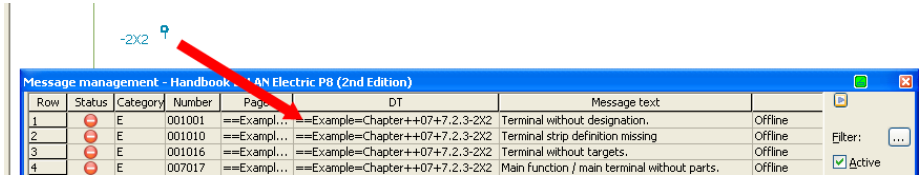


Fig. 7.39
Type of check set to "Offline"

The check type **Online/offline** generates messages online and immediately enters them into the opened message management. Completed messages are not removed from the message database until the next check run.

Fig. 7.40
Type of check set to "Online/Offline"

Row	Number	Message text	Category	Type of check
1	001001	Terminal without designation.	Error	Online / offline

Another terminal is positioned in the schematic. This terminal is not given a designation, and the check for the message *Terminal without designation* has been set to **Online/offline**. In contrast to the previous check runs, EPLAN checks whether a designation has been entered already during the placement of the terminal. Since in our example no designation was assigned, EPLAN generates immediately (online) the corresponding check run message *Terminal without designation*.

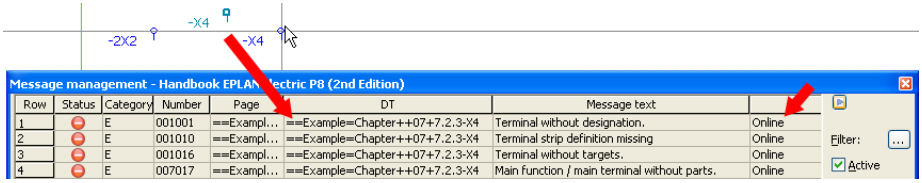


Fig. 7.41 Type of check set to "Online/offline", EPLAN reports the error immediately.

The **Prevent errors** type of check means that EPLAN checks the input during an action, for example, when inserting devices, and, if this results in errors, shows a message immediately and then "prevents" the action.

Fig. 7.42
Type of check set to "Prevent errors"

Row	Number	Message text	Category	Type of check
1	001001	Terminal without designation.	Error	Prevent errors

Again, a terminal is inserted and placed without a designation. During the placement, EPLAN checks whether this action violates any check and immediately reports the error in the **PREVENT ERRORS** dialog.

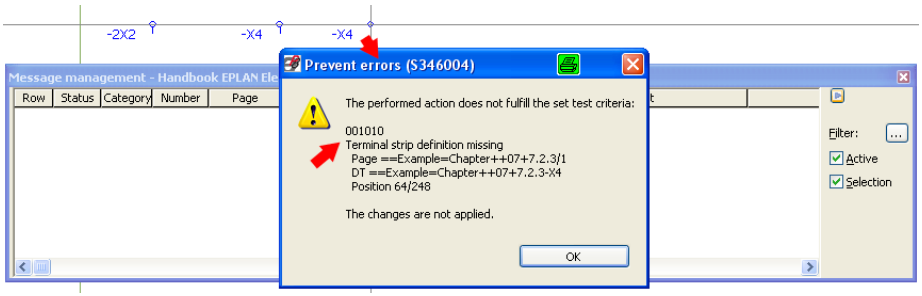


Fig. 7.43
EPLAN prevents the error on the basis of the type of check defined.

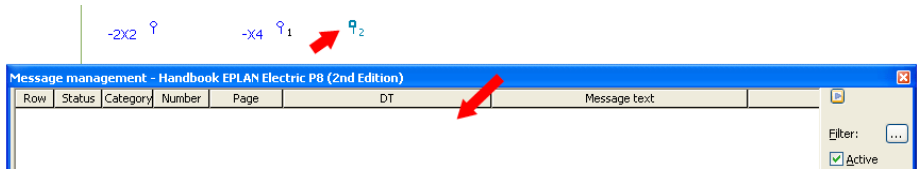


Fig. 7.44
Terminal can be repositioned.

EPLAN terminates this action, and the terminal can be repositioned and relabeled. If the default requirements are met (terminal designation exists), the terminal can be placed.

NOTE: The **Prevent errors** type of check, of course, is very strict and should (in my personal estimation) not be used during ongoing project editing. But when, for example, schematics are revised, this is a very good opportunity to immediately prevent errors while revising schematics instead correcting them later during a check run.

The **Module-specific check** type of check also generates messages during certain automated actions, but cannot be influenced any further.

Row	Number	Message text	Categ...	Type of check	Filters	Generate...
1	003001	The automatic cable selection is being canceled, because n...	Error	Module-specific	No filter	de.eplan
2	003002	It is not possible to make an automatic cable selection for t...	Error	Module-specific	No filter	de.eplan
3	003003	Mismatch between connection DT and cable DT.	Error	No	All tra...	de.eplan

Fig. 7.45 Module-specific check

If you do not want that, set this type of check to **No**.

7.2.3.1 Filters for all check run messages

Besides the previous settings for types of check or categories, it is possible to define a separate filter for each check run message.

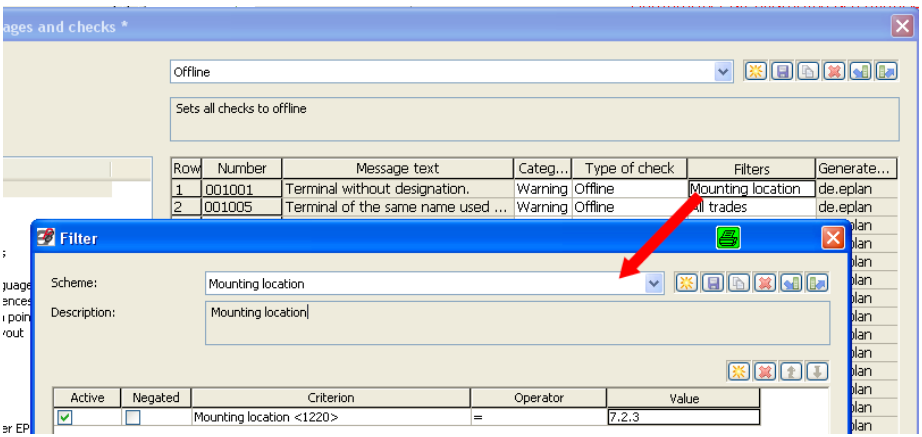


Fig. 7.46 Filter options for each check run message

These filters allow you to limit certain checks to specific areas or properties. In other words, checks can be targeted quite specifically.

The *Terminal without designation* check is set to the **Offline** type of check, but is to be checked only in a specific mounting location. This will exclude from the check all other faulty terminals that do not match the defined filter criterion.

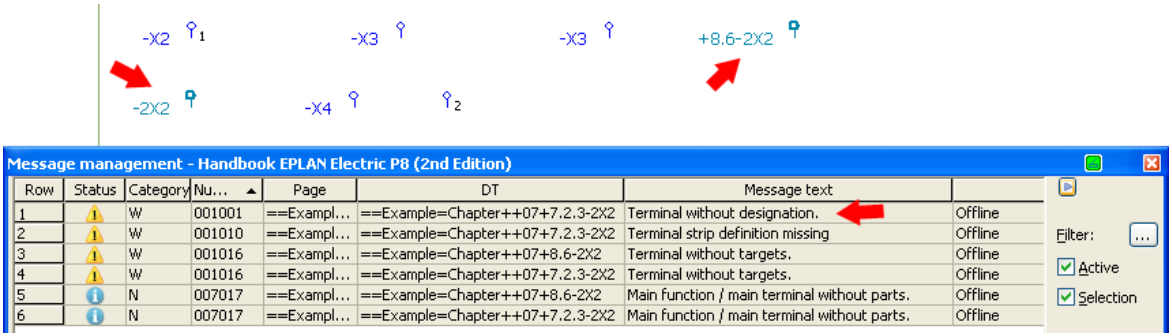


Fig. 7.47 Only checks defined by filter will be carried out.

In the example, the terminal without a designation, whose mounting location was different than the one defined in the filter, was ignored by the check run. The terminal whose mounting location matches the filter, however, was captured by the check run, and EPLAN generated a corresponding check run message.

TIP: This feature (placing filters on individual check run messages) should be used very sparingly. The check run could be slowed down considerably as a result, because EPLAN would always have to evaluate the check run message as well as the filters.

7.2.4 Filters in message management

As described in the section 7.2.3, EPLAN messages are categorized into specific message classes. All message classes and areas (electrical engineering, fluid, or e. g. errors) can be switched on and off via filters.

7.2.4.1 Filter active setting

Select the **ACTIVE** check box and use click the ... button to open up the **FILTER: MESSAGES** dialog.

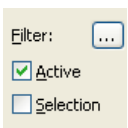


Fig. 7.48
Filter active

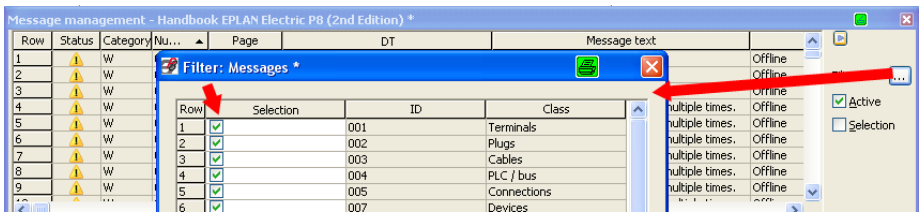


Fig. 7.49
Filter settings for message class selection

In this dialog, by deselecting the **message classes** (terminals, plugs, cables, etc.) and also the **areas** (electrical engineering, process engineering, etc.), it is possible to filter different data. When you are finished making entries, you close the dialog by clicking the **OK** button. To activate the filter for the message management display, you must set the check mark in the **Active** check box below the filter (see Fig. 7.51).

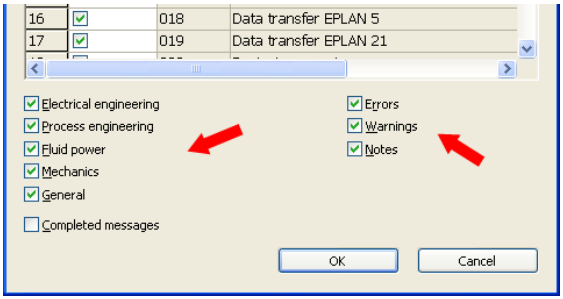


Fig. 7.50 Filter settings for the areas

A typical example of a filtered display would be, for example, removal of the completed messages from message management until the next check run.

7.2.4.2 Filter selection setting

If the **SELECT** check box is set, then only messages from selected objects are displayed in message management. All other messages from other objects, etc. are hidden. This avoids tedious searching for messages relating to particular objects.

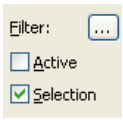


Fig. 7.51 Filter selection

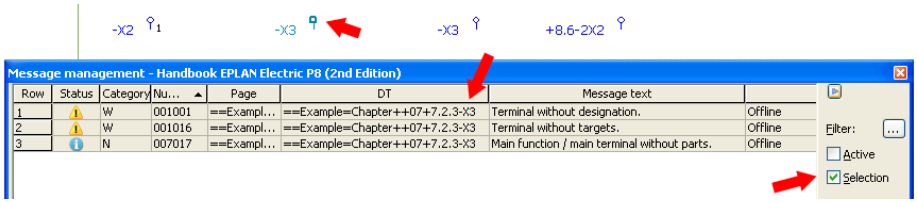


Fig. 7.52 Selection has been activated.

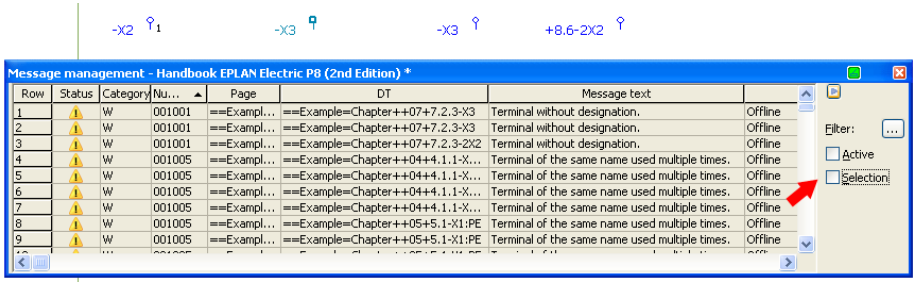


Fig. 7.53 Selection has not been activated.

If the **SELECTION** check box is not active, message management displays all messages. A selection of objects does not play any role here.

7.2.5 Various message edition options

This section does not deal with the correction of generated error, warning, or note messages. I refer you instead to the online help, which explains problems and related solutions for check run messages on a much wider scale.

Instead, this section focuses more on the ways in which messages can be localized in the schematic, and what tools EPLAN provides for this. The general functions for displaying, for example, Freeze panes or Adjust column width as well as copy functions will not be explained any further here.

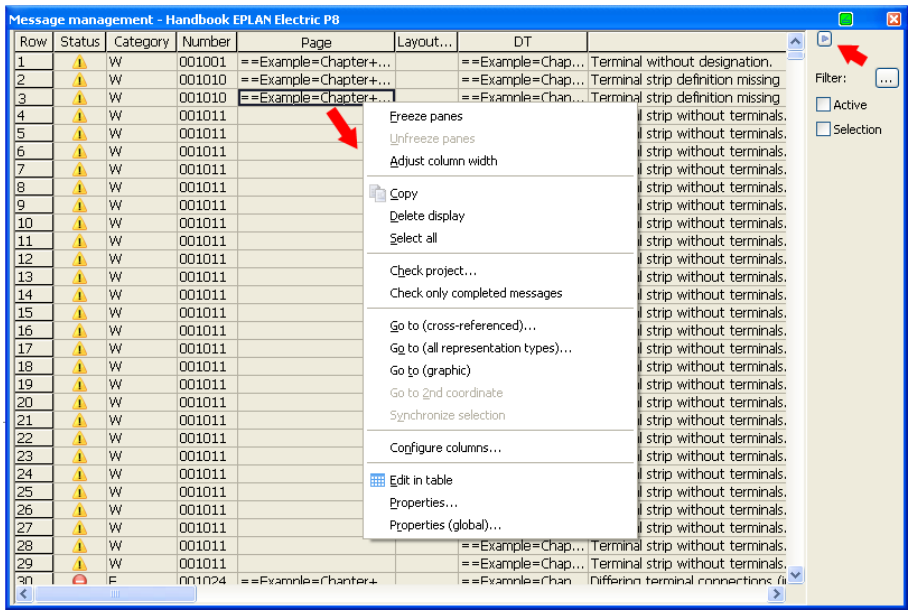


Fig. 7.54 Popup menu

The right-click popup menu provides several options for easily finding errors. The key ones are described below.

- **DELETE DISPLAY**: This option deletes all messages from the message management display. The messages are still present, but are no longer displayed. They re-appear after the next check run.
- **CHECK PROJECT**: This function carries out the set check run; depending on the selection, this is done for the entire project or only for a single page.
- **GO TO (CROSS-REFERENCED)**: This function causes EPLAN to enter all cross-referenced objects in the *Go to* list and then open the **FIND** dialog with the *Go to* tab.
- **GO TO (ALL REPRESENTATION TYPES)**: This function determines all representation types (schematic, report pages, etc.) of the function and also enters them in the *Go to* list. From here, you can navigate directly to the desired representation.
- **GO TO (GRAPHIC)**: This function allows you to navigate to the object in question directly from message management.

- **GO TO 2ND COORDINATE:** If, for example, there are two duplicate terminals, you can use this function to jump directly to the other terminal.
- **SYNCHRONIZE SELECTION:** This familiar function lets you jump to and select exactly this device in other open navigators, such as the device navigator. This function eliminates time-consuming searching for a device in the navigators.
- **CONFIGURE COLUMNS:** This function allows the number of columns displayed in message management to be increased or decreased.

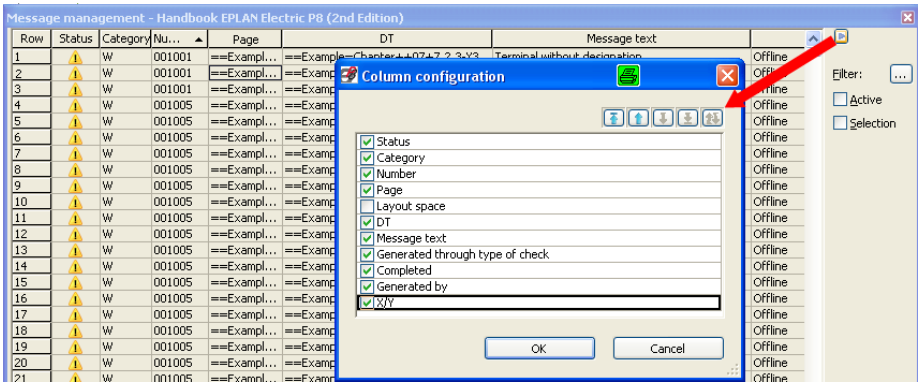


Fig. 7.55 Configure columns

When this function is executed, EPLAN displays the **COLUMN CONFIGURATION** dialog. Here you can remove or add columns, or change the sequence of the columns.

TIP: To obtain a preview of the incorrect objects, the **GRAPHICAL PREVIEW** should be enabled in the **VIEW** menu. Ideally, this preview should be used with a multi-screen system.

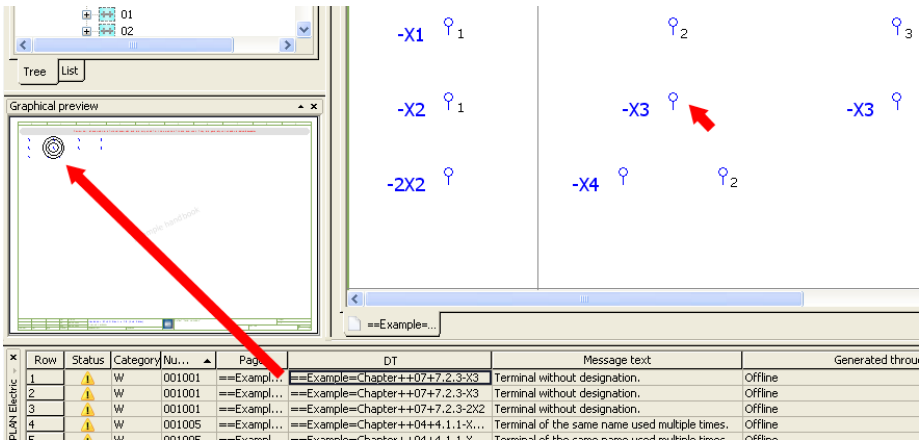


Fig. 7.56 Using the graphical preview to find errors

7.3 Layer management

What are layers? Layers are an essential element originating from mechanical engineering (CAD). Information of the same type (e. g. dimensions) is placed on the same layer. For example, the form, color, font size or other formats of this layer can later be changed at a central place, easily and without errors.

This is a great advantage, because otherwise every property that is not allocated to a layer must be manually edited. Layer management makes such editing actions a "piece of cake". EPLAN has taken this well-known idea from mechanical engineering and extended it into the CAE area.

You start layer management via the **OPTIONS/LAYER MANAGEMENT** menu. The **LAYER MANAGEMENT** dialog is displayed. Layer management itself is a project-specific setting, and you can recognize this from the window title bar that displays the currently active project name. The dialog has windows on the left and right sides. The left window contains a tree with the superior layer designations (nodes and any subnodes and then the layer designation EPLANxyz), and the right window contains the associated subentries (the actual layer information).

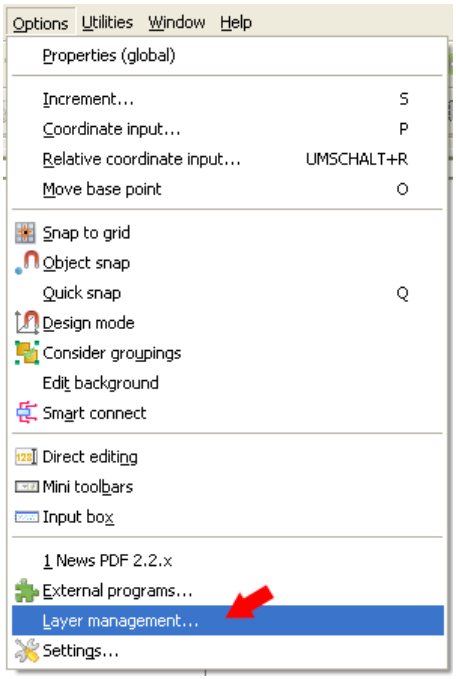


Fig. 7.57
Layer management

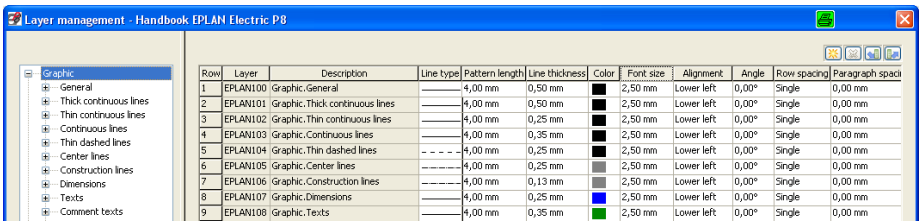


Fig. 7.58
Layer management dialog

7.3.1 Standard layers

EPLAN comes with a number of standard layers. They begin with the name EPLAN, followed by a three-digit number.

NOTE: Original EPLAN layers cannot be removed (deleted).

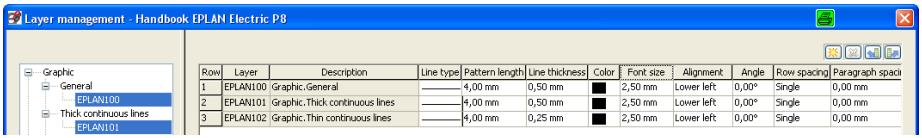


Fig. 7.59 Standard EPLAN layer

7.3.2 Export and import of layers

All layers can be modified and/or adapted to your requirements, including the original EPLAN layers. However, you should first export all layers before making any changes to the standard layers. You then have a functioning backup copy of the previous layer settings.

Layers are exported as follows: Before starting the export, you must first select all desired layers using the usual Windows functions. The upper right of the dialog contains the graphical buttons. You use the button to start the export. EPLAN opens the **LAYER EXPORT** dialog. The file name contains a default value set by EPLAN, which is the name of the project. When you confirm the dialog by clicking **SAVE**, the layers are exported and saved.

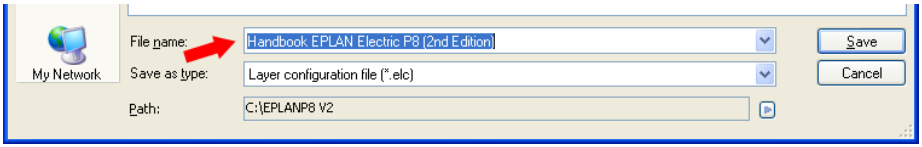


Fig. 7.60 Layer export

Layer import is done in a similar way. Clicking the button causes EPLAN to open the **LAYER IMPORT** dialog. Here you select the desired layer configuration file and import it into the opened EPLAN project by clicking the **OPEN** button.

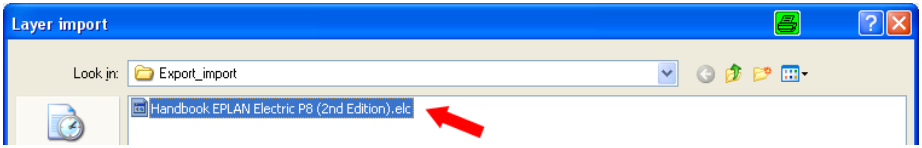


Fig. 7.61 Layer import

NOTE: All layers not already present in the project are imported and created as new layers. If layers of the same name already exist in the project, then the data in the import file always has priority. This means that layers in the project might be updated.

7.3.3 Create and delete your own layers

In addition to the standard layers provided with EPLAN, you can, of course, create your own layers. You should be somewhat systematic in doing so, otherwise you will end up with “proliferated chaos”. Layers can only be (directly) created in existing nodes such as graphics or property placements. New (superior) nodes cannot be directly created.

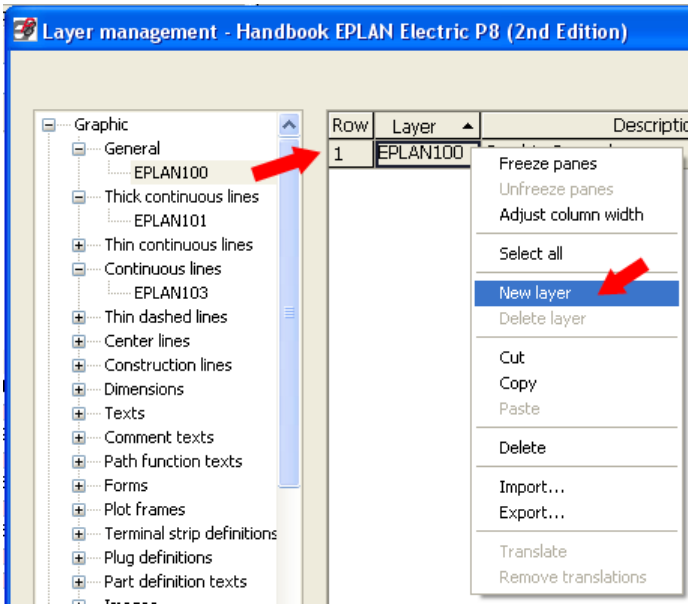


Fig. 7.62 Creating a new layer

You create new layers from the **LAYER MANAGEMENT** dialog by right-clicking and selecting the **NEW LAYER** popup menu item (in the right area of the layer management dialog), or by using the button. Use the mouse to select the node in which the desired layer should be created. Now you call up the **NEW LAYER** function. EPLAN generates a new layer with the name NEW_LAYER_1.

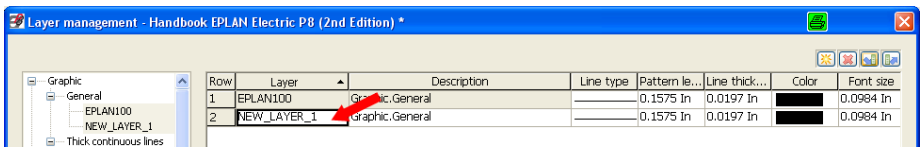


Fig. 7.63 New user-created layer

This layer can now be edited. The key columns here are **Layer** and **Description**.

The **Layer** column bears the name of the layer. When you create your own layers, you should give them sensible names. One possibility is to name the layers after the company and to include a range of numbers, e. g. COMPANY1000.

The **Description** column describes the layer itself. This description can be viewed later at the time of assigning layers to individual objects in the project. You should therefore enter a descriptive text here that allows a function to be assigned to the layer based on the description.

The layer will be later sorted into the left-hand area of layer management on the basis of the description.

We will now change the layer from NEW_LAYER_1 to HB3-0100. The layer description will be changed to **Handbook.Graphic.Spezial**. Immediately after you save the layer, EPLAN will sort it as follows.

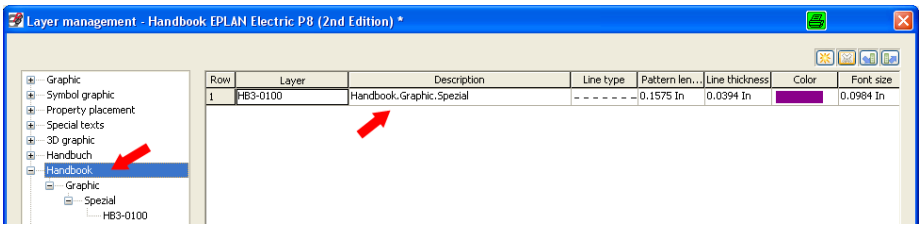


Fig. 7.64 Newly sorted layer

Properties of layers

Properties such as **Line type**, **Pattern length**, **Line thickness**, etc. are self-explanatory and are stored or selected in accordance with the layer requirements. It is also possible to define default values for angles and their orientation or whether this level should be in the background.

Row	Layer	Description	Line type	Pattern length	Line thickness	Color	Font size	Row spacing	Paragraph spacing	Text box	Visible	Print	Locked
1	EPLAN100	Graphic.General	———	0.1575 In	0.0197 In		0.0984 In	Single	0.0000 In	No	☑	☑	☐
2	EPLAN101	Graphic.Thick continuo...	———	0.1575 In	0.0197 In		0.0984 In	Single	0.0000 In	No	☑	☑	☐
3	EPLAN102	Graphic.Thin continuou...	———	0.1575 In	0.0098 In		0.0984 In	Single	0.0000 In	No	☑	☑	☐

Fig. 7.65 Other layer settings options

7.3.4 Uses of layers

What can you use all these layers for? You can use your own layers, for example, to provide extra project information for the workshop. This could be cabling notes or deadlines. The possible uses are limitless.

As described in section 7.3.3, a new layer (HB3-0101) was created with the description **Handbook.Graphic.Workshop.Information**. This layer is for displaying further information for workshop production in the schematic. When the workshop has finished using this information, it should then be easy to hide this information for the entire project.

Row	Layer	Description	Line type	Pattern length	Line thickness	Color
1	HB3-0101	Handbook.Graphic.Workshop.Information	———	0.1575 In	0.0138 In	

Fig. 7.66 Workshop layer

A schematic page with corresponding entries is then opened in the project. Now, special information for the workshop is placed on the page as text using the EPLAN default settings for texts.

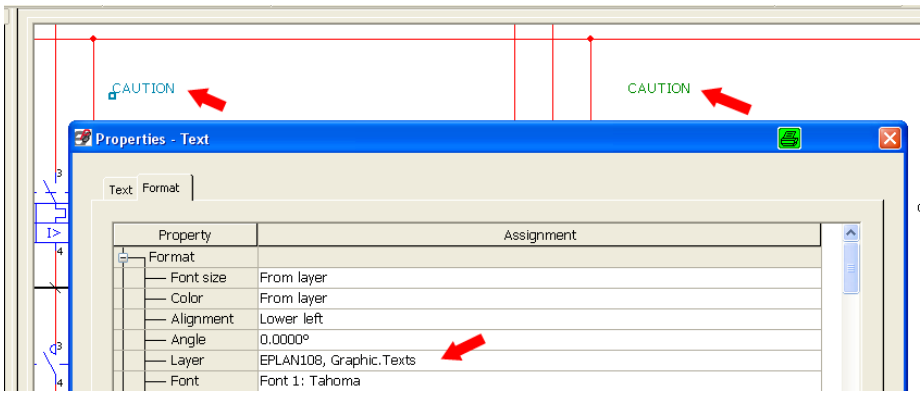


Fig. 7.67 Text with EPLAN default setting

Then, these texts are selected and placed on the newly created layer **Handbook.Graphic. Workshop.Information**.

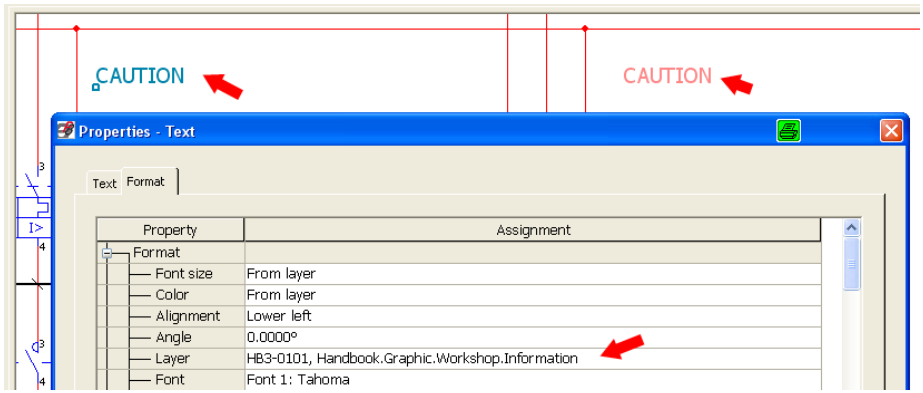


Fig. 7.68 Texts placed on the new layer

EPLAN handles the settings that pertain to this layer and formats the texts accordingly. Via layer management, this workshop information can now be hidden, for example, prior to printing.

Row	Layer	Description	Color	Font size	Row spacing	Paragraph spacing	Text box	Visible	Print	Locked
1	HB3-0100	Handbook.Graphic.Spezial		0.0984 In	Single	0.0000 In	No	☑	☑	☐
2	HB3-0101	Handbook.Graphic.Workshop.Information		0.1378 In	Single	0.0000 In	No	☑	☐	☐

Fig. 7.69 Excluding a layer from printing

This is only a small example of what is possible with layer management. The user has practically unlimited possibilities here. The biggest advantage of layers is that it is easy to globally change the format, representation, etc. of the respective element on this layer with just a few mouse clicks.

7.3.4.1 Remove unnecessary layers from the project

To remove layers that have been rendered superfluous, for example, layers that were stored in the project and subsequently changed as a result of copy actions or the insertion of DXF/DWG files, you can use the compress the project.

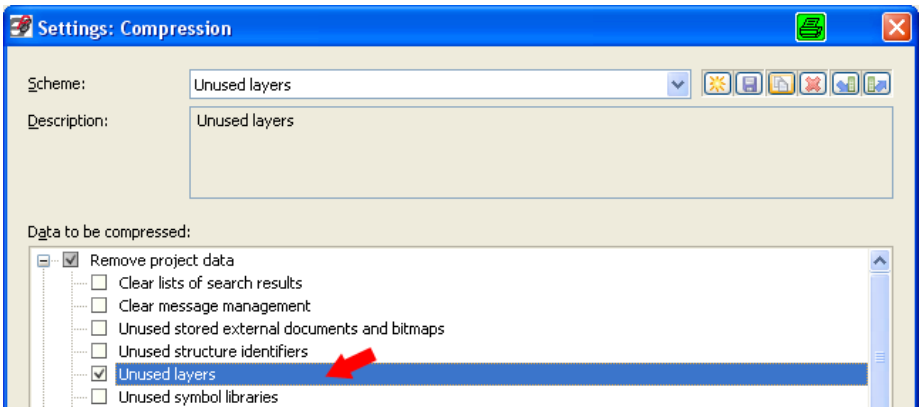


Fig. 7.70 Compression scheme to remove unnecessary layers

After the scheme is launched, EPLAN removes the unnecessary layers and then displays a summary of these removed layers.

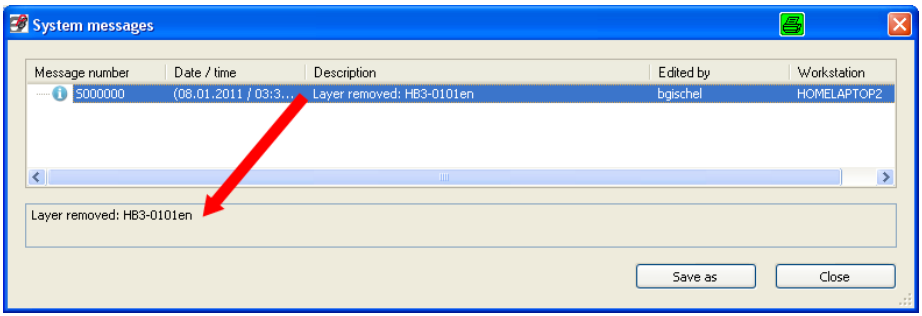


Fig. 7.71 System message

7.4 Parts management

All devices in EPLAN, with all their technical and commercial data such as technical characteristics, dimensions (width, height, depth) or prices, are managed in **Parts management**. However, not only device-specific data is managed in parts management – the corresponding function definitions, symbols and also symbol macros can be stored for every device.

Parts management is called via the **UTILITIES > PART > MANAGEMENT** menu.

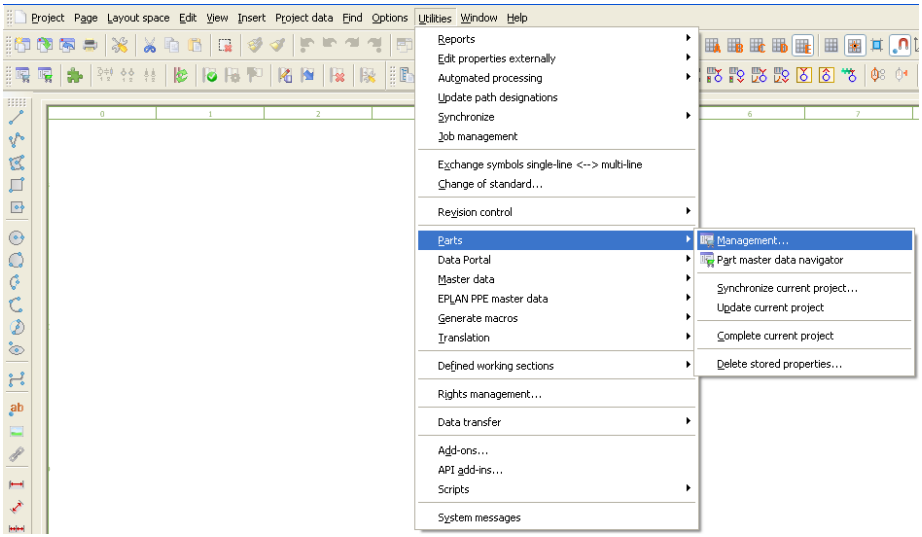


Fig. 7.72 Calling up parts management

The actual parts database used is defined via **OPTIONS/SETTINGS/USER/MANAGEMENT/PARTS MANAGEMENT**. You can also run parts management from an SQL server database instead of an Access database.

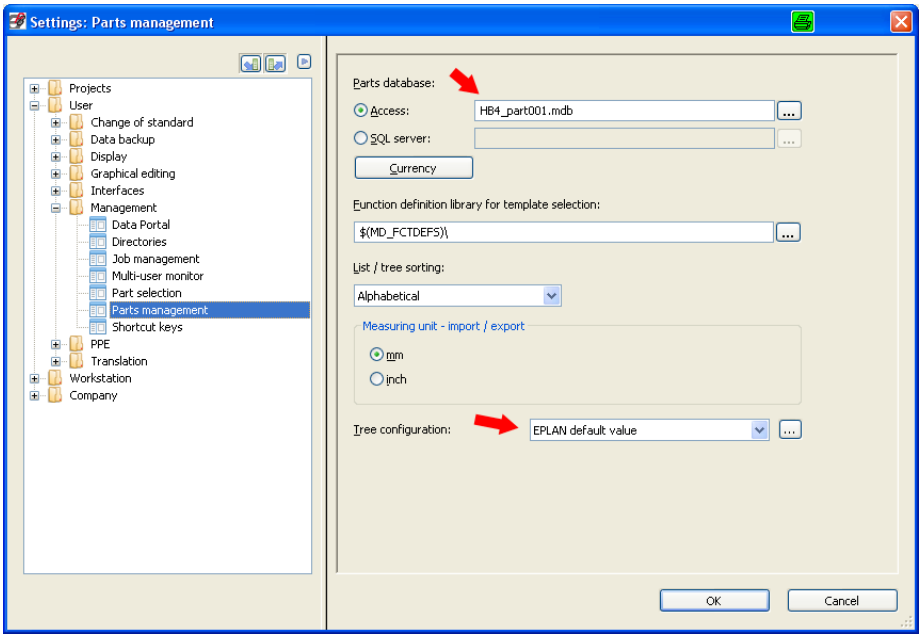


Fig. 7.73 Parts management user settings

In these **User-specific settings**, you can also set the parts management **View** (tree configuration). There are various options to adjust the tree configuration view to your own specifications by using your own schemes.

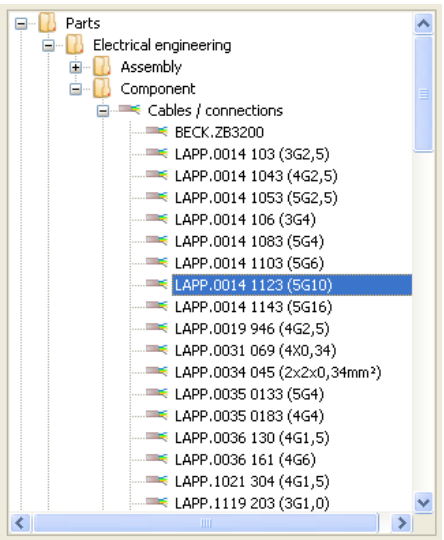


Fig. 7.74 Standard EPLAN view

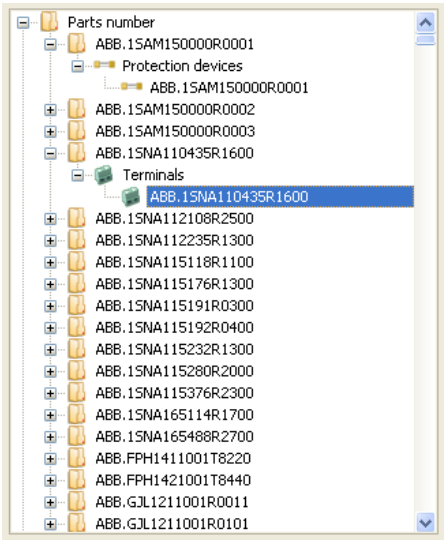


Fig. 7.75 View by part

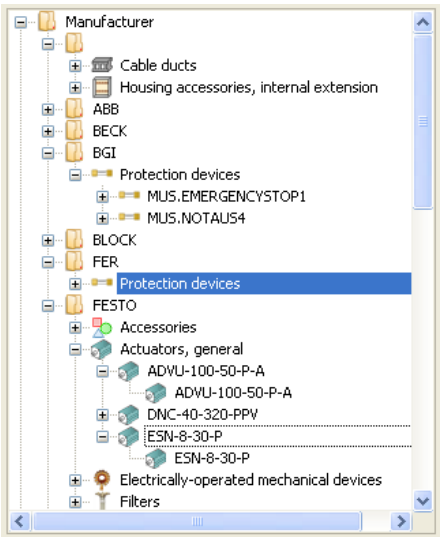


Fig. 7.76 View by manufacturer

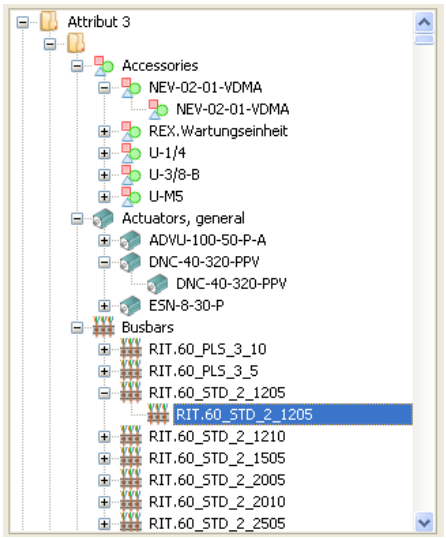


Fig. 7.77 View by attribute 3

7.4.1 Structure of parts management

It would take an entire book to describe parts management in detail. In this section, I cover the most important information that is of interest and useful in everyday practice.

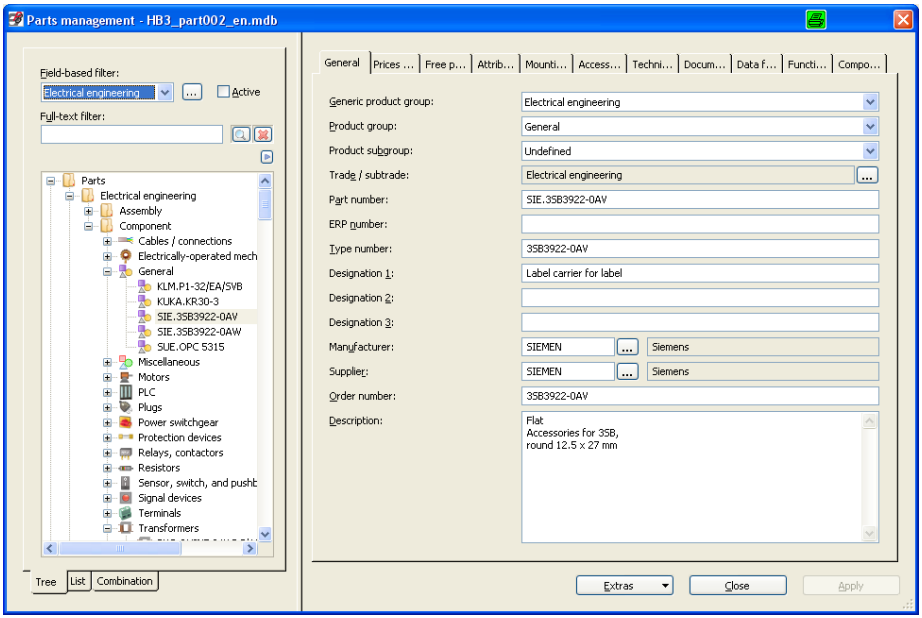


Fig. 7.78
Overview of the parts management dialog

The **PARTS MANAGEMENT** dialog basically consists of an area on the left containing an overview of all parts, customers and manufacturers/suppliers – and an area on the right containing detailed information. Both parts are separated by "splitters" and, therefore, scalable to (almost) any size. The area on the right always relates to one (or more) selected parts in the area on the left – the parts or customers and/or manufacturers/suppliers. The area on the right, i. e. the parts data, is further divided into several tabs containing the various data relating to the part.

7.4.2 Tabs in parts management

Each part has several tabs. Various technical and commercial data for the part are defined in these tabs. The existing tabs of the part Siemens SIE.3LD2 504-0TK53 will serve as our example here.

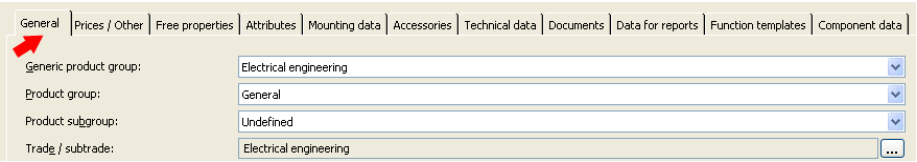


Fig. 7.79 The different tabs of a part

NOTE: The tabs shown here *may* differ from part to part. For example, there are differences for terminals, plugs, PLC, etc. You need to take this into consideration when editing.

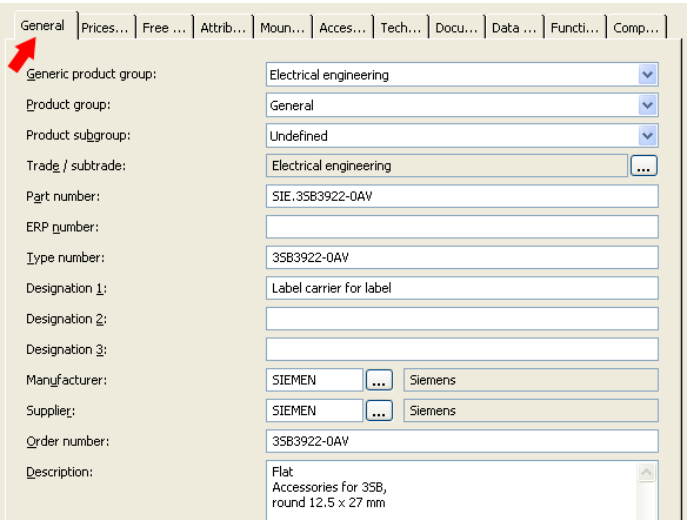


Fig. 7.80 General tab

The *General* tab contains general information on the part, such as the part number, ERP number, **product group** (e. g. a contactor or the **trade**). You will also find information here such as the **part number**, **ERP number**, **order number**, the device **designations** and the **manufacturer**.

NOTE: A part is identified by its part number and the ERP number. This means that they have to be unambiguous and can only be used once in parts management.

Most of these fields are pure input fields; where this is impossible, for example for the generic product group, there are fixed selection lists. If a record for the manufacturer and/or supplier has been created in parts management, the **Manufacturer** and **Supplier** fields can be selected from the **MANUFACTURER** and/or **SUPPLIER DIALOG** that opens when you click ... EPLAN then applies this entry and automatically fills in the full name of the manufacturer or supplier.

NOTE: The user cannot add entries to these selection lists (generic product group, product group and product subgroup). Only EPLAN can do that.

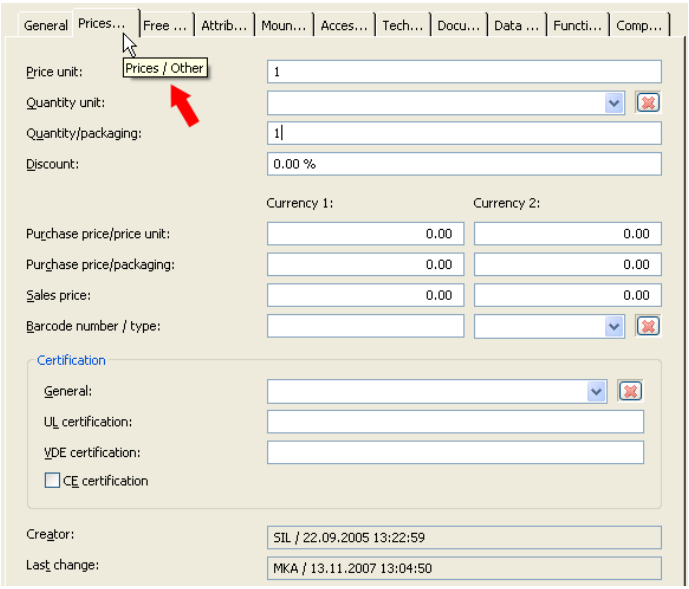


Fig. 7.81 Prices/Other tab

The *Prices/Other* tab contains commercial and organizational entries like **Quantity unit**, **Package sizes**, **Prices** and even linked documents for **Certification**.

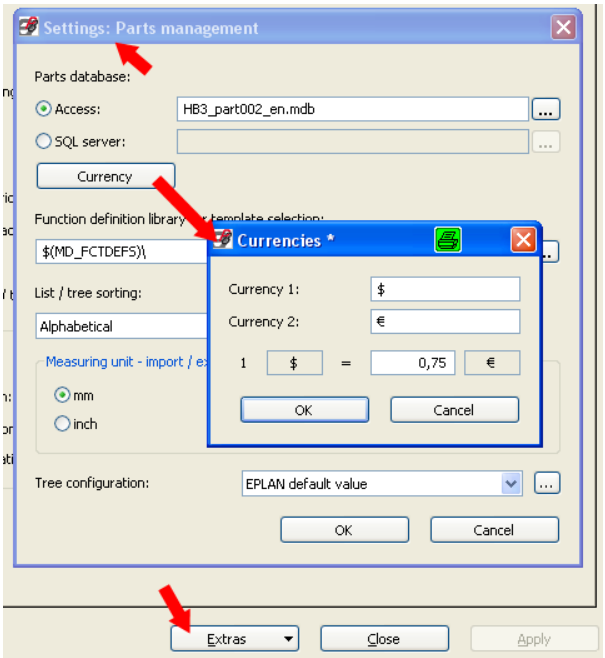


Fig. 7.82 Currency definition

To define the currency, you first click the extras button in parts management, then select the **SETTINGS** menu item, and then click the currency button in the **SETTINGS/PARTS MANAGEMENT** dialog.

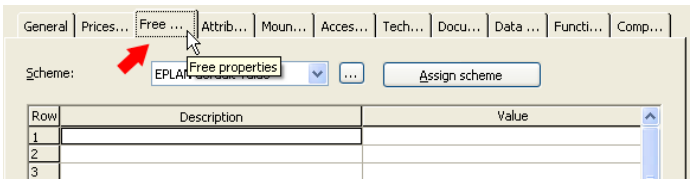


Fig. 7.83 Free properties tab

The *Free properties* tab covers 100 properties that are freely available for any desired purpose. Each of these free properties consists of a **Property**, a **Value** for the property, and a **Unit**.

All fields can be freely used, for instance, for information that is not available as a standard field for the part. Block properties can be used with the free properties to display necessary or additional information at a specific symbol in the schematic (with stored parts).

You can create your own schemes with free assignment of the free properties and assign parts to these schemes without having to manually enter the contents of the free properties for every part.

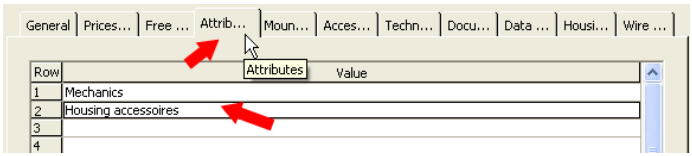


Fig. 7.84 Attributes tab

The *Attributes* tab allows you to store additional information (in the **Value** column) that is not available as standard in parts management. For example, these entries can be used for filtering or as a sorting characteristic for the tree configuration. The entries themselves are limited to 200 characters.

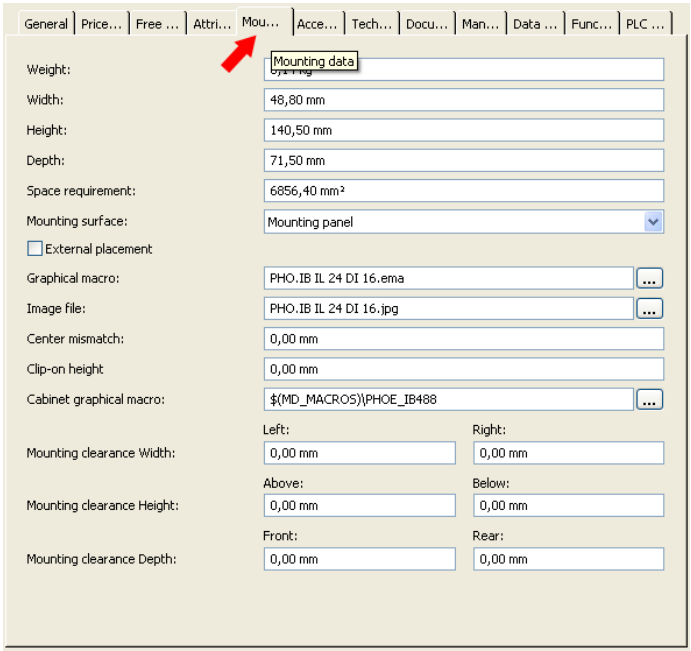


Fig. 7.85 Mounting data tab

The *Mounting data* tab contains all the information that is necessary to use the part, for example, for a mounting panel layout. It is recommended to meticulously maintain this information, such as **Width**, **Height** or **Depth** (especially important for the 3D mounting layout (EPLAN Pro Panel)) for the part and to update any missing data. A stored image file can also be useful, because this information is evaluated at different locations in EPLAN. This greatly simplifies work steps such as equipping mounting panels or generating data reports like parts lists, device tag lists, etc. because the necessary data can be processed automatically from parts management.

NOTE: The **graphic macro** field on this tab is intended for the Pro Panel (3D macro). For 2D panel layout, the 2D macro should be stored in the **Macro** field on the *Technical data* tab.

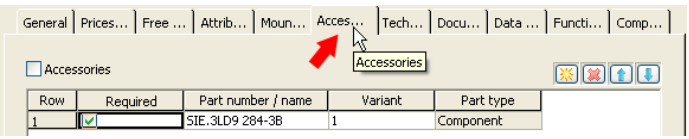


Fig. 7.86 Accessories tab

The *Accessories* tab allows you to assign specific accessories to "main" parts. This accessory is also created as a normal part. You turn it into an accessory part by activating the **Accessory** check box. Then it can be selected and assigned under the "main" parts on the *Accessory* tab.

Selected accessories can be marked as **required** or, if the identifier is missing, as optional accessories.

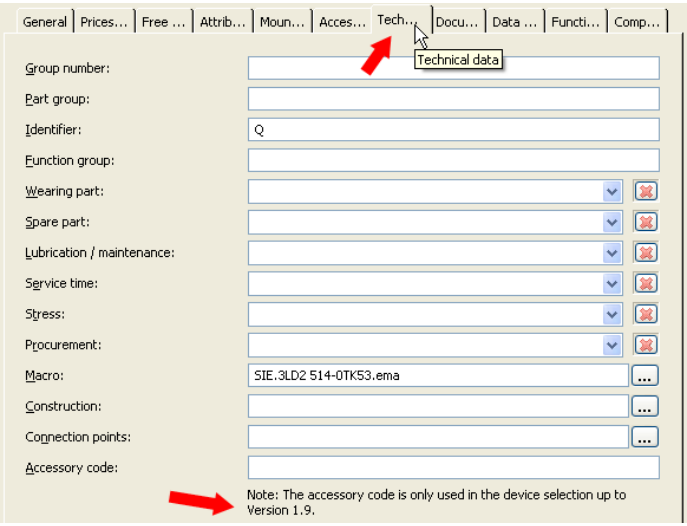


Fig. 7.87 Technical data tab

The *Technical data* tab contains part data such as **service time** and whether the part is a **wearing part**. Most fields are pure input fields, such as the **Group number** or **Function group** fields.

NOTE: The **Identifier** field is not used to automatically assign identifiers during the placement of a device. Instead, you can use this field, for example, to set a filter for the identifier Q in parts management.

The **Wearing part**, **Spare part**, **Lubrication/maintenance**, **Service time**, **Stress** and **Procurement** fields are also input fields. However, each entry is recognized by EPLAN and written to a selection list (internal) so that, over time, this selection list contains many entries that have already been used (and can be selected again).

The remaining fields of this tab, such as **Macro**, **Construction**, **Connection points**, can be selected via the ... selection button. The **Macro** field can contain the corresponding window macro for the project. Using the familiar macro functions with the different representation types (e. g. part placement, overview or other representation types), this macro can be set up in such a way that it can be put to multiple uses during project editing.

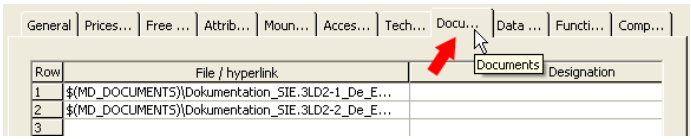


Fig. 7.88 Documents tab

The *Documents* tab can hold up to 20 different documents. These can be user instructions, manuals, etc. Different language versions of the documents can also be stored for the part.

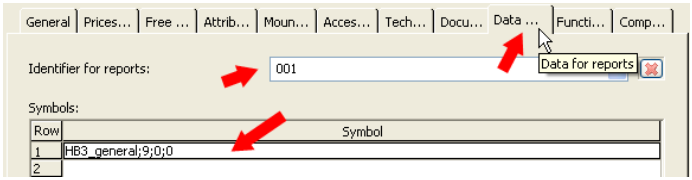


Fig. 7.89 Data for reports tab

It makes sense to use the *Data for reports* tab in combination with the **device tag list** report type and conditional forms (so-called subforms for dynamic forms). This way, certain graphical and more complex requirements regarding the output of parts in a device tag list can be displayed in a specific manner.

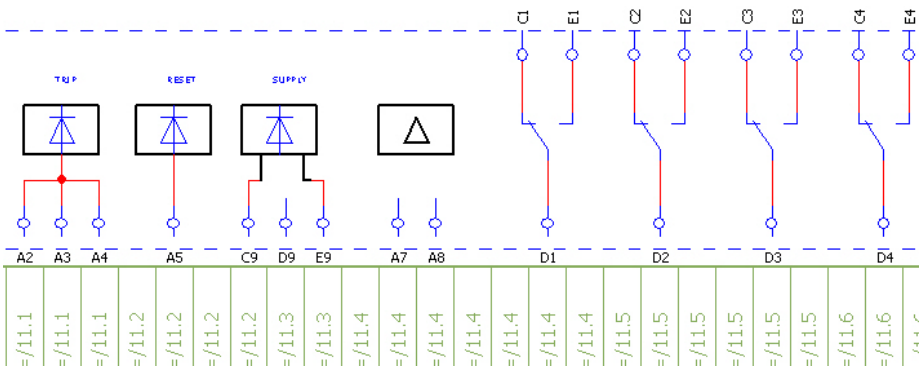


Fig. 7.90 Example view of a complex graphical report of a part

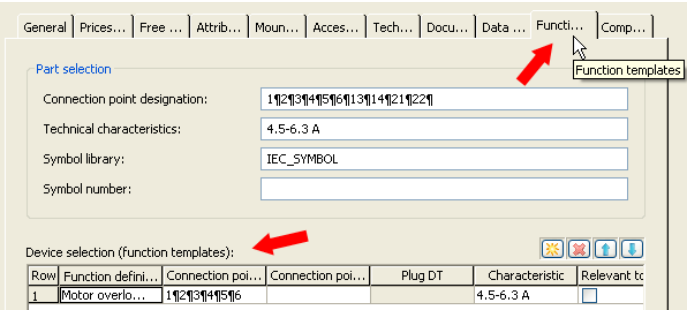


Fig. 7.91 Function templates tab

The *Function templates* tab contains key information about the actual part and for when selecting devices for the part. The *Function templates* for selecting devices for a part are defined in this tab. I. e. this is where you define whether this part is a lamp or an auxiliary contact.

EPLAN needs function templates in order to compare the devices already used in the project with the devices in parts management when performing device selection (not to be confused with parts selection). This allows you to select devices that match the items in the schematic.

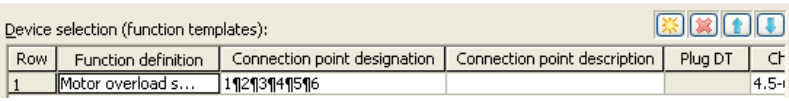


Fig. 7.92 The first columns of the function template

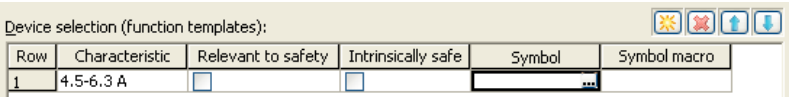


Fig. 7.93 The last columns of the function template

Entries such as the **Function definition**, the **Connection point designation**, the **Connection point description** or the **Characteristic** allow comparison criteria to be defined. For example, when selecting a device, a pushbutton that was defined in parts management with a pushbutton function definition and connection point designations 7 and 8 can only be assigned to an item in the schematic that has exactly these properties, assuming that the device selection settings have been set for this.

For each function definition, you can define, in addition to the previous values, specific **symbols** or **symbol macros** that EPLAN is then to use for graphical project editing should the standard symbol not fit. You can also define for each function definition whether the function is *relevant to safety* or *intrinsically safe*.

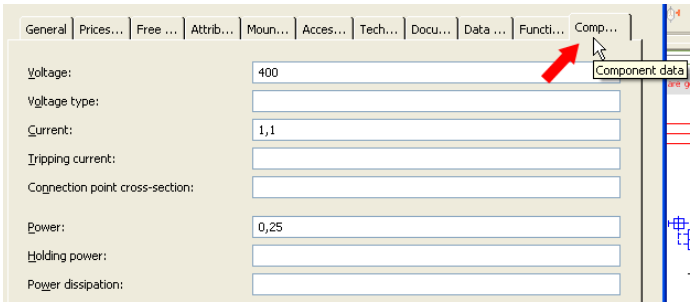


Fig. 7.94 Component data tab

The last tab, *Component data*, contains further technical data for the part, which can also be evaluated – for use in reports, calculations or simply as an additional representation in the schematic on the device.

7.5 Revision control

In addition to actual project creation, EPLAN Electric P8 provides a management tool that allows you to record, evaluate or document revisions (e. g. changes to the site, end customers, etc.) to this project at the end of project editing.

In EPLAN Electric P8, the task of recording, managing and documenting revisions is handled by revision control. EPLAN distinguishes between two fundamental procedures for executing revision control: Revision based on a **project comparison** and revision based on **change tracking**.

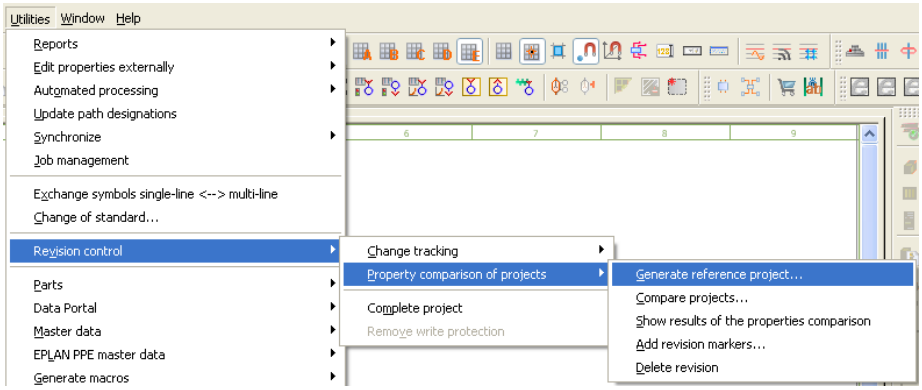


Fig. 7.95 Revision control with property comparison

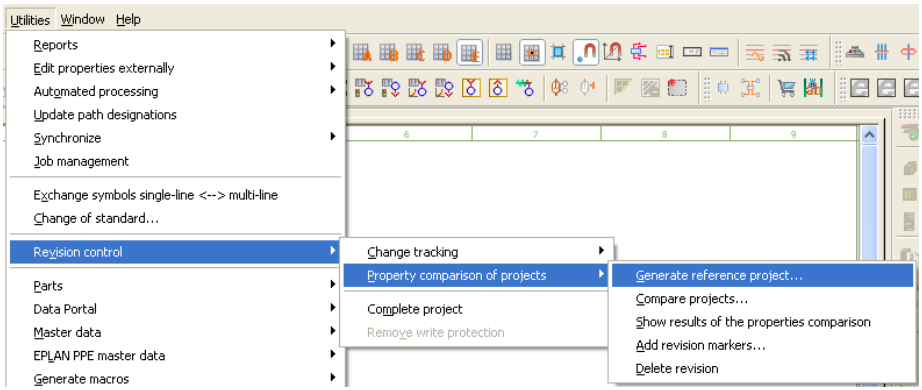


Fig. 7.96
Revision control with change tracking

7.5.1 General

A revision in EPLAN Electric P8 is essentially defined as follows: A revision is an edited project in which the various changes are marked and then tracked via revision. A revised project is stored with the file extension **.ell*.

The following will explain revision control using change tracking as an example.

7.5.2 Generate new revision

Before a revision, and thus change tracking of modified data, can be executed, you must first activate it (the revision) for the project. In the **UTILITIES/REVISION CONTROL/CHANGE TRACKING** menu, start the **GENERATE REVISION** menu item.

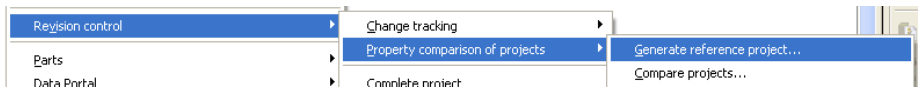


Fig. 7.97 Generate revision

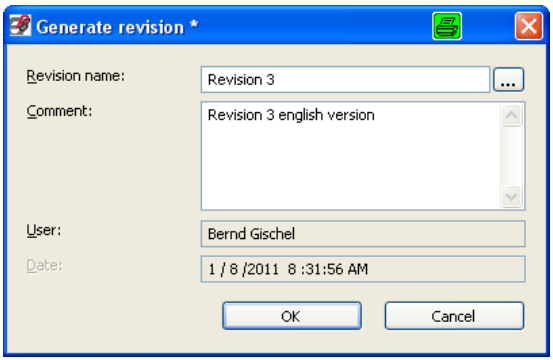


Fig. 7.98
Entering revision data

EPLAN opens the **GENERATE REVISION** dialog. Here you must enter a *Revision name*. The *Comment* on the revision is optional. The *User* and *Date* cannot be changed and are set by EPLAN automatically. This data always applies to a complete revision. It is a kind of global "header" for the revision data. This data in the **GENERATE REVISION** dialog is part of the project properties.

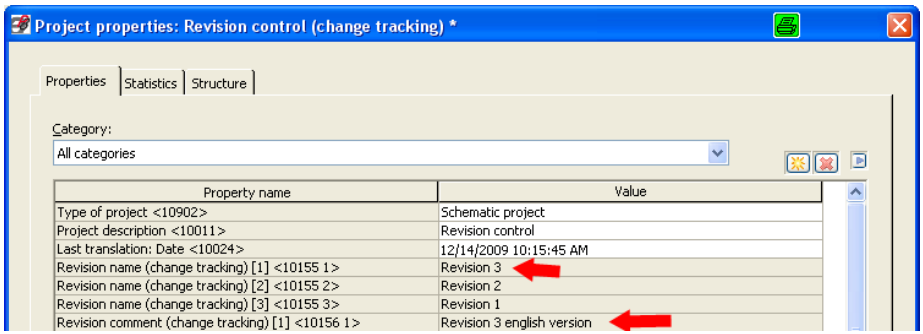


Fig. 7.99 Project property names of revision entries

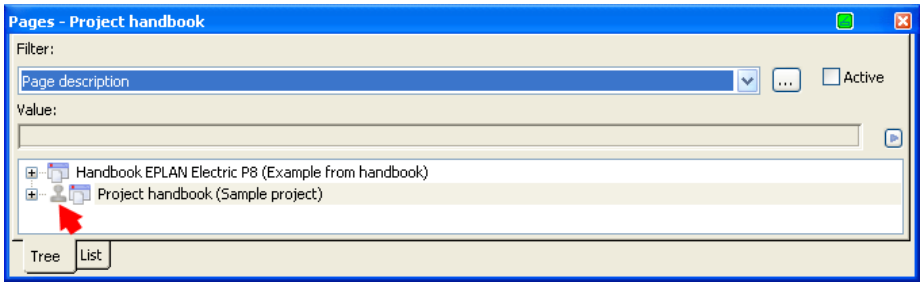


Fig. 7.100 Modified icon

This global revision data can, for example, be integrated into the plot frame. If all data has been entered, you can close the **GENERATE REVISION** dialog by clicking **OK**. EPLAN now generates the revision and, for the purposes of the visual check, changes the project icon in the page navigator.

7.5.3 Execute changes

After the revision has been created, the changes can be entered in the schematics.

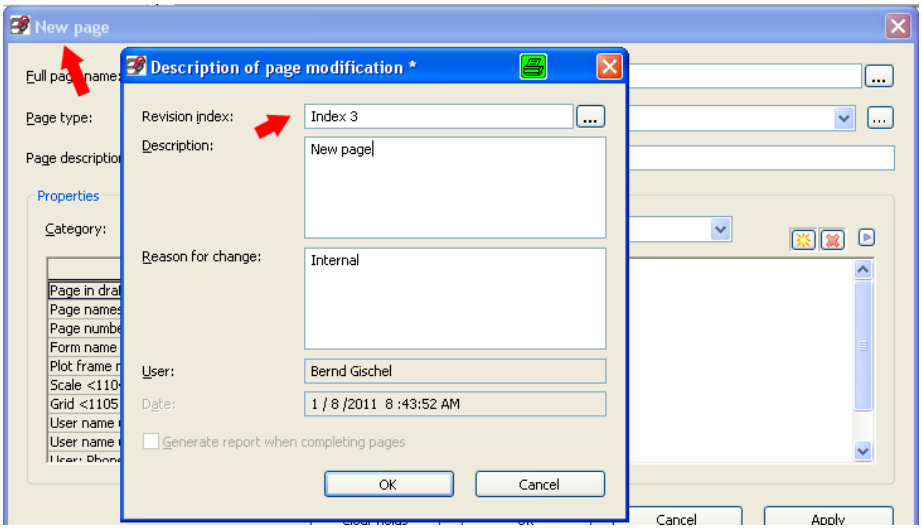


Fig. 7.101 Index query following a change

Following changes, EPLAN queries a revision index. EPLAN opens the **DESCRIPTION OF PAGE MODIFICATION** dialog. Here the *Revision index* is defined as well as a *Description*, which can also be optional. Again, the user and date of the modification are defaulted by EPLAN from the system.

For the revision index, you should choose one system to avoid chaos and to prevent the numbering of the revision index from becoming mixed up upon completion of the project. It would be a good idea to integrate the revision index, and possibly its description, into the plot frame. You would immediately be able to see, on a printout for example, that a page has been modified.

6				
Index 3	1/8/2011	Bernd Gischel	Date	1/8/2011
			Ed. by	Bernd Gischel
			Appr	
Modification	Date	Name	Original	

Fig. 7.102 Adding the revision index to the plot frame

The properties of the revision index and their additional data such as the description are pure page properties.

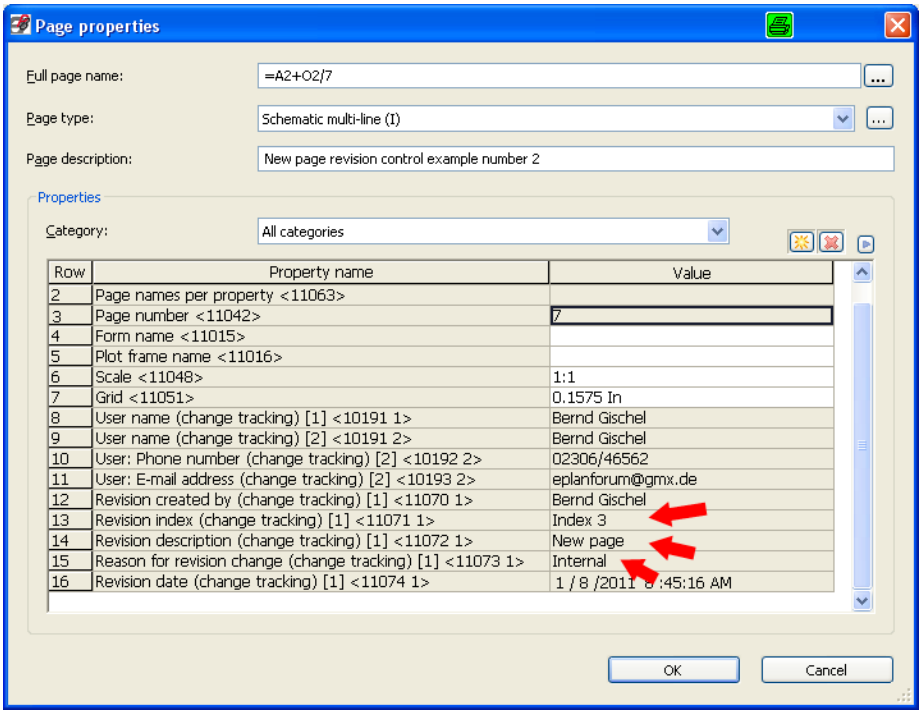


Fig. 7.103 The revision index in the page properties

Generally, you can now make any desired modifications to the project. Changes on a page are marked with the **Watermark** (the text of the watermark can be modified in the settings) across the plot frame, and in the title bar of the window you will also see the word "**Draft**". This serves as a visual indication that this page and the modifications have not been completed yet.

EPLAN allows you to place revision markers for every modification. You can also use visually different revision markers for different types of changes.

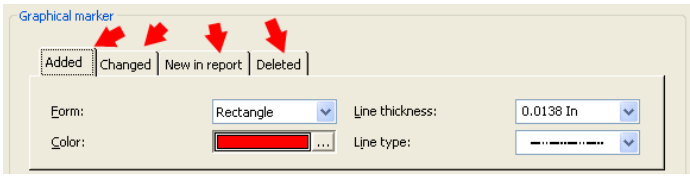


Fig. 7.104 Graphical markers for various modifications

Here, EPLAN differentiates between *Added*, *Changed*, *New in report* and *Deleted*. Depending on your requirements, you can adjust these graphical markers for each project differently.

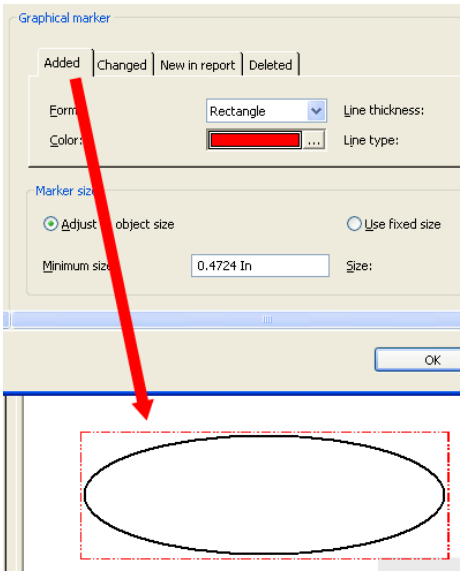


Fig. 7.105 Graphical marker for added objects

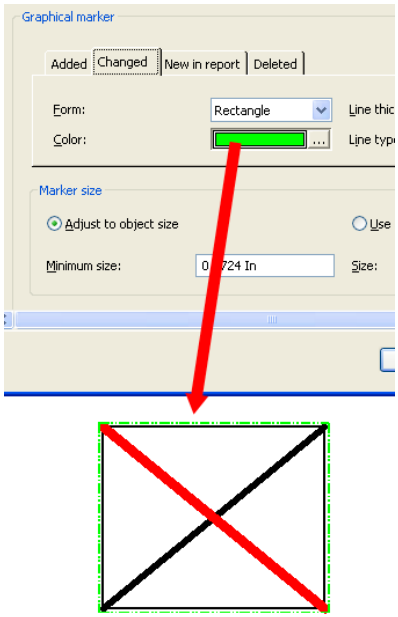


Fig. 7.106 Graphical marker for changed objects

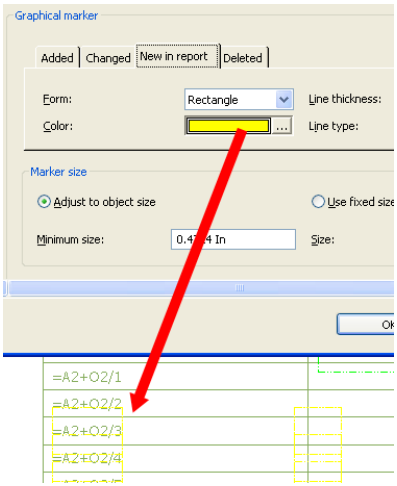


Fig. 7.107 Graphical marker for “New in report”

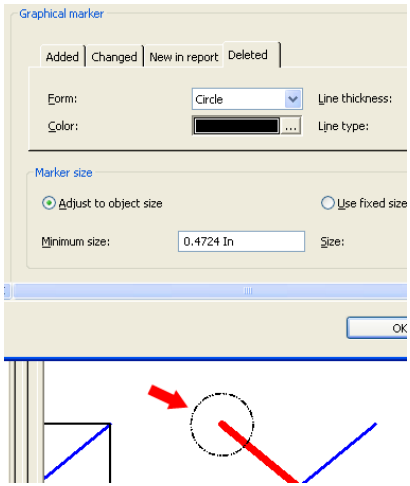


Fig. 7.108 Graphical marker for deleted objects

7.5.4 Complete page(s)

After the modifications have been incorporated into the project, you need to complete the page or pages. Go to the **UTILITIES/REVISION CONTROL/CHANGE TRACKING** menu and start the **COMPLETE PAGES** menu item.

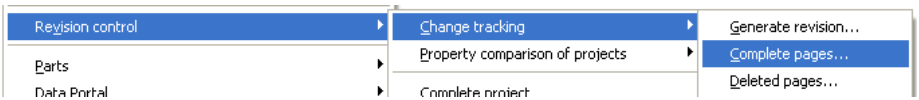


Fig. 7.109
Complete pages

EPLAN opens the **DESCRIPTION OF PAGE MODIFICATION** dialog. Here you must now enter the revision index. It is also possible to switch to the **SELECT REVISION INDEX** dialog via the More button. From there, select, apply or modify an existing revision index.

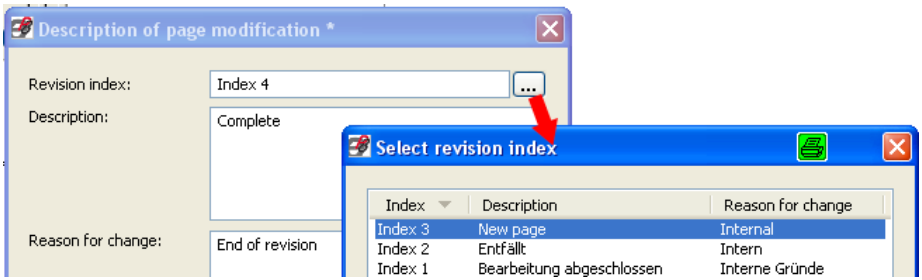


Fig. 7.110 Defining the description of page modification

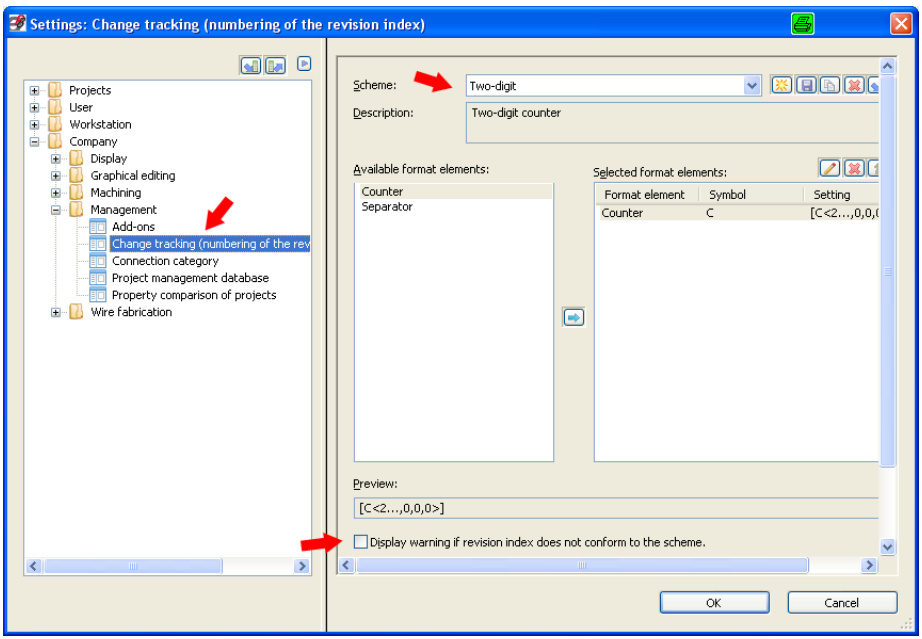


Fig. 7.111 Defining the numbering of the revision index

If a numbering scheme has been defined for the revision index in the settings (**OPTIONS/UTILITIES/COMPANY/MANAGEMENT/CHANGE TRACKING (NUMBERING SCHEME FOR REVISION INDEX)**), a revision index will be recommended automatically based on the selected scheme.

NOTE: As long as the project itself has not been completed, any completed page can be modified again. In that case, you merely have to complete the page again.

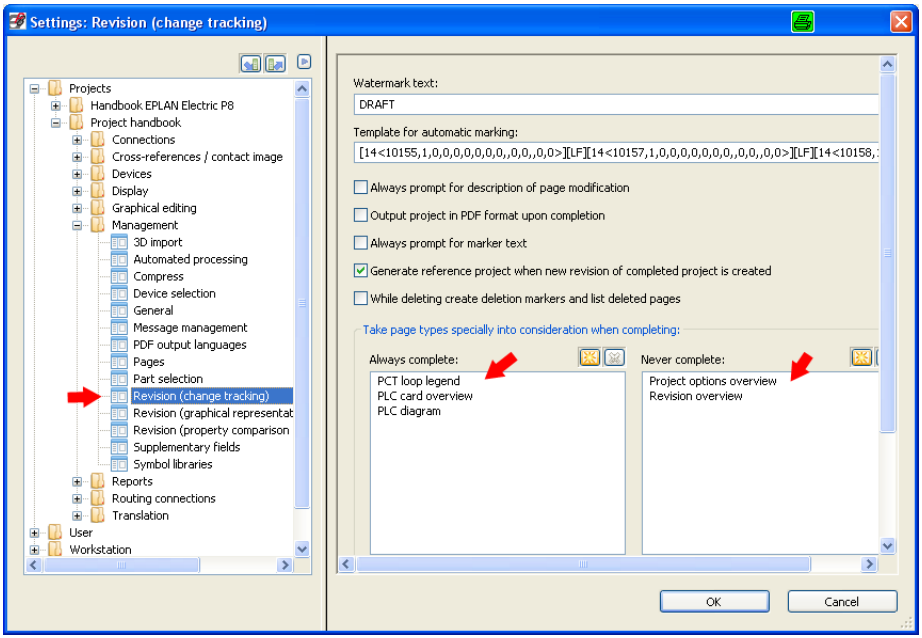


Fig. 7.112 Special handling of page types

The **Always complete pages** option means that these pages are always assigned a revision index even if no modifications have been made. But EPLAN can assign an automatic revision index only if a scheme for this has been defined in the settings.

Never complete pages has the opposite effect. These pages are ignored for revisions even if modifications have been made.

7.5.5 Generate reports

Once all modifications have been made, you can generate and/or update the reports. Special revision overviews will give you an overview of the changes made to the project.

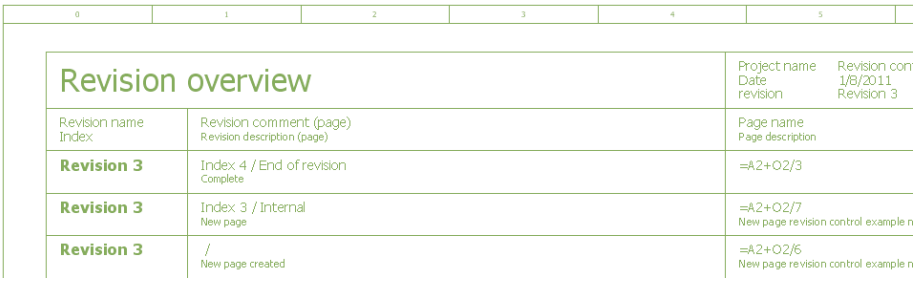


Fig. 7.113 Revision overview report

A special menu item allows you to list even the pages that were deleted from the project. You can call up this menu item via **UTILITIES/REVISION CONTROL/CHANGE TRACKING/DELETED PAGES**. When this function is started, EPLAN displays the **DELETED PAGES** dialog. This contains information about when and how pages have been deleted (e.g. when reports were updated), plus the data you entered, such as a reason for the change.

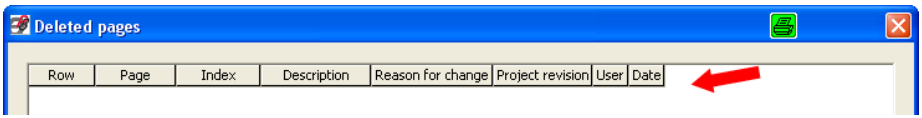


Fig. 7.114 Deleted pages dialog

7.5.6 Complete a project

Fig. 7.115 Complete project

If all modifications have been finished, the reports generated and updated, and the pages completed, you are ready to complete the project itself. Go to the **UTILITIES/REVISION CONTROL** menu and start the **COMPLETE PROJECT** menu item. EPLAN opens the **COMPLETE PROJECT** dialog. Here, you enter the already familiar data like revision index or the reason for the change. If you select the **GENERATE REPORT WHEN COMPLETING PROJECT** option, the following happens: EPLAN updates the project, evaluates all reports and completes each page, i.e. it removes the entry “Draft” and/or the watermark. All revision indices that EPLAN assigns for this automatically now show the most recently defined revision index.

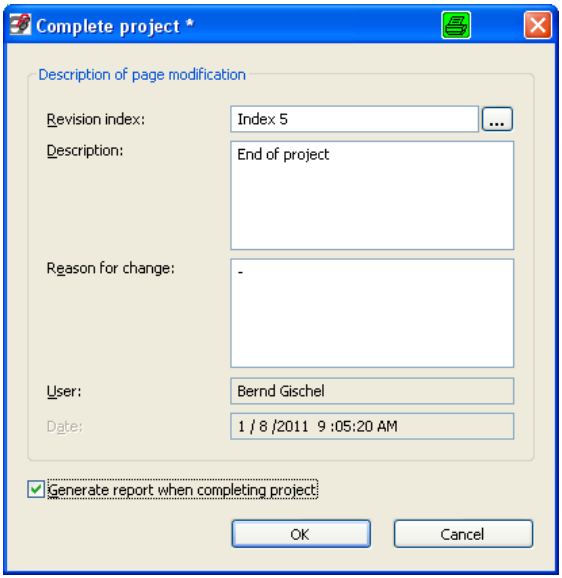


Fig. 7.116 Complete project

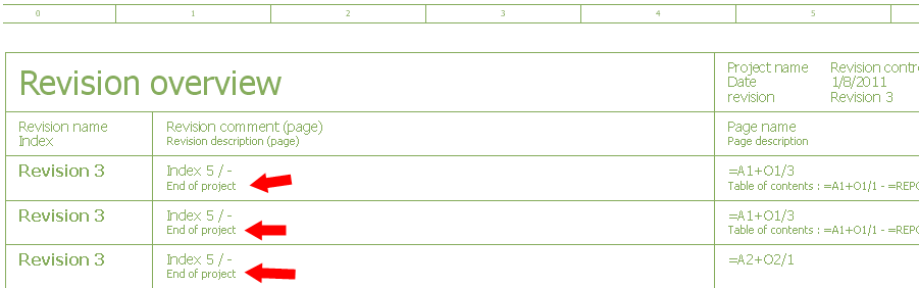


Fig. 7.117 Automatically completed project

A completed project is represented in the page navigator by a corresponding icon.

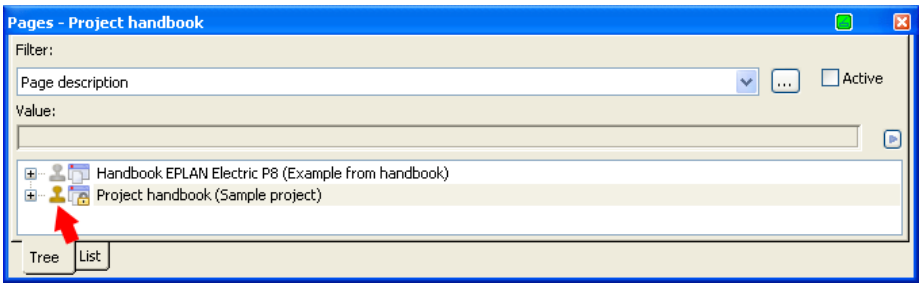


Fig. 7.118 Icon for a completed project

8 Export, import, print

EPLAN offers a range of export and import functions for the entire project or also individual pages. Export and import are relatively simple processes that are only carried out for the selected pages (or all pages) of a project.

An export can be launched via the **PAGE/EXPORT [EXPORT TYPE]** menu; an import via the **PAGE/IMPORT [IMPORT TYPE]** menu.

Export menu

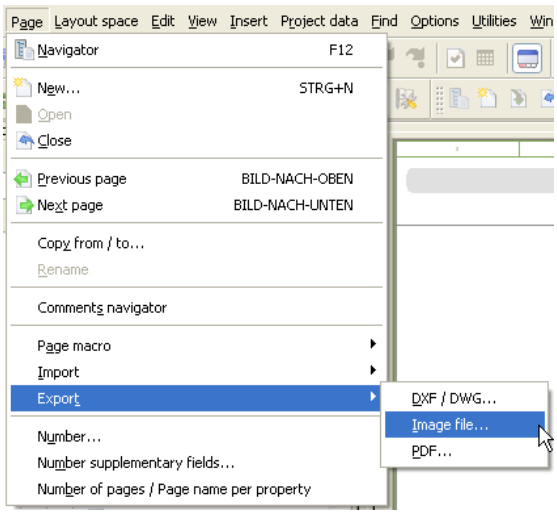


Fig. 8.1 Menu of miscellaneous export options

Import menu

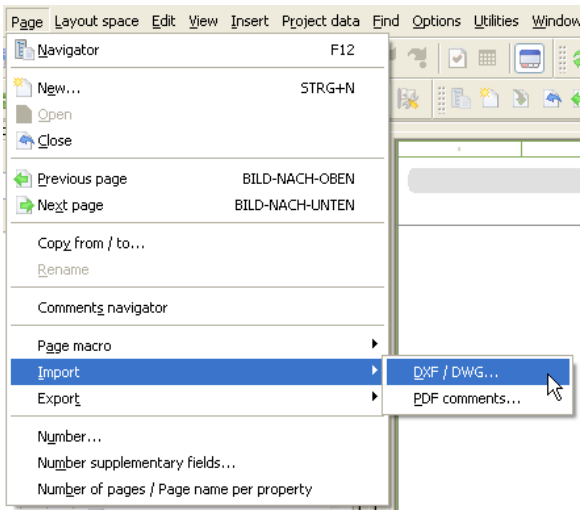


Fig. 8.2 Menu of miscellaneous import options

NOTE: Exported pages in an image format or DXF/DWG format do not contain any (electrical engineering) logic after the export.

This is why this export function is not suitable for exchanging projects. To exchange projects, you should always do a proper data backup (the *Back up* or *Restore* project options).

NOTE: Pages can only be exported from *one* project at a time. Pages selected from different projects in the page navigator cannot be exported together.

The export options are then grayed out and cannot be selected.

8.1 Export and import of DXF/DWG files

Files in the DXF or DWG format have a neutral exchange format that EPLAN can use. However, after export or import, these files contain only purely graphical elements such as circles, lines, etc.

The DXF and/or DWG formats are widely used and popular in various CAD/CAE programs, for example, to represent machine layouts or similar elements. Therefore, they are ideal for exchanging non-program-specific drawings, etc., even if they are purely graphical.

8.1.1 Exporting DXF and DWG files

To export pages in DXF or DWG format, you first select the desired pages in the **page navigator** and then use the mouse to start the function from the **PAGE/EXPORT/DXF/DWG** menu (the **ALT + S** keyboard shortcut does not work when the page navigator is open). EPLAN opens the DXF/DWG export dialog and loads the **Source** field with the selected pages (this field cannot be edited by the user).

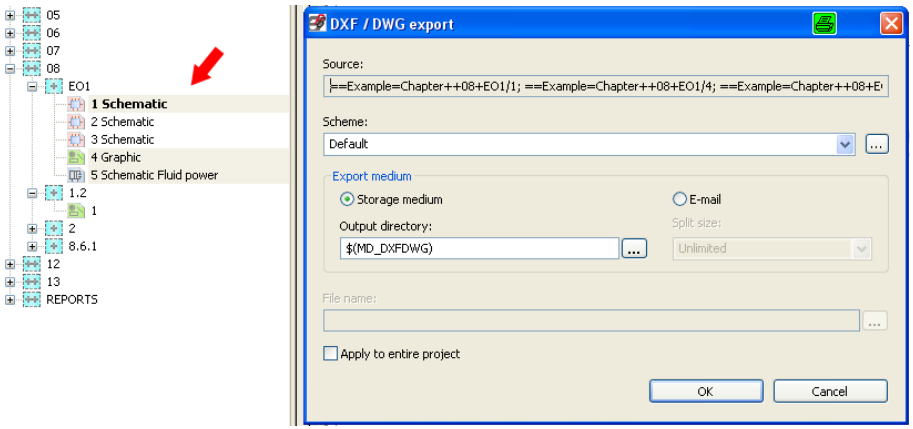


Fig. 8.3 Export pages to DXF/DWG format dialog

The **Export** scheme is more interesting and can be modified by the user. Existing schemes can be selected in the selection field here and then applied.

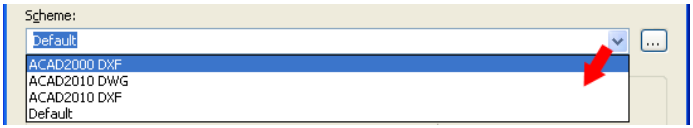


Fig. 8.4 Schemes can be adjusted in any manner.

The ... button opens the **Settings: DXF/DWG export and import** dialog, where the current scheme can be edited.

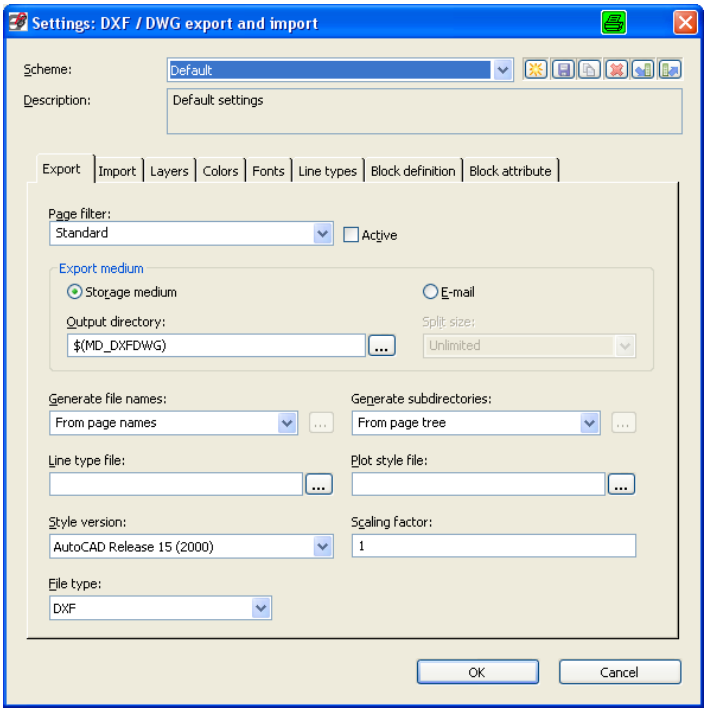


Fig. 8.5 Settings for exporting DXF/DWG files

These settings are not project-specific and are regarded as user settings. You can find them under **OPTIONS/SETTINGS/USER/INTERFACES/DXF/DWG EXPORT AND IMPORT**.

DXF/DWG export and import settings

The **Settings: DXF/DWG export and import** dialog offers the user many options/parameters for influencing or controlling the export and import of DXF/DWG files.

A number of parameters are important for exporting pages in the DXF/DWG format. However, at this point you should create a new scheme via the **NEW** button and not edit the currently set scheme. An exception to this is when the scheme settings are not optimal. You can then change these and save the scheme under the same name.

Once a name and optional description for the scheme have been defined, you close the dialog by clicking **OK**. EPLAN enters the new scheme name into the **Scheme** selection field. The desired export settings can now be adjusted.

You can set the **Output directory** to a special directory for this specific project, instead of the usual global output directory for DXF/DWG files.

You should adjust the **Style version** to suit the target system. EPLAN provides a selection field with different entries and a certain number of different target systems.

The **File type** can be set to either DXF or DWG. This setting is also made via a selection field and must be set to suit your requirements.

The **Scaling factor** controls enlargement or reduction of the page being output. The usual value is 1, i.e. the page is written to the DXF/DWG format on a 1:1 basis without enlargement or reduction. This entry should be left at the default value.

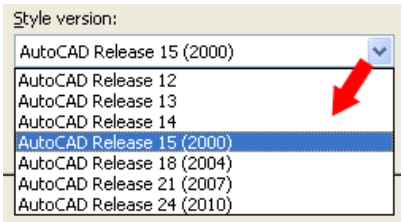


Fig. 8.6
Selection of formats

The *Generate file names* option allows the user to define the name of the generated DXF/DWG files. You can allow EPLAN to automatically generate the names using *consecutive numbers*, or you can use your own scheme to generate the file names individually from various page and/or project properties (*From properties* option).

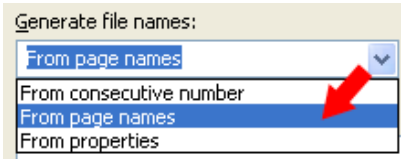


Fig. 8.7
File name options

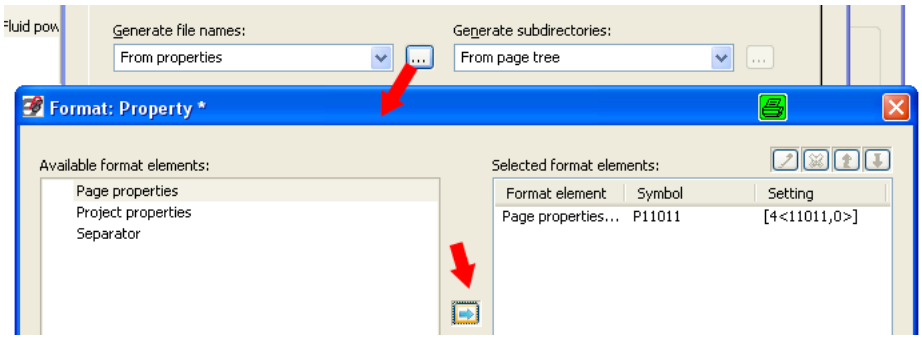


Fig. 8.8 Generating file names from properties (format elements)

Once all options or parameters have been set, you save the scheme via the graphical **[SAVE]** button. It is then always available and can also be used for other projects, as described previously.

It is also possible to output the DXF/DWG files to be generated in defined and fixed **subdirectories**. This would be practical, for example, when you want to structure the output in such a way that a subdirectory is generated for each higher-level function, with all pages belonging to the higher-level function being output there.

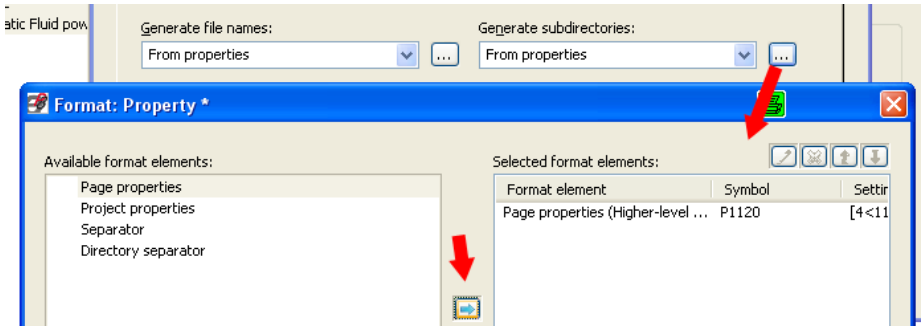


Fig. 8.9 Option of output in subdirectories

The two remaining settings, Line type file (*.lin) and Plot style file (*.ctb), are provided by the target system and must exist as files so that they can be included in reports as well.

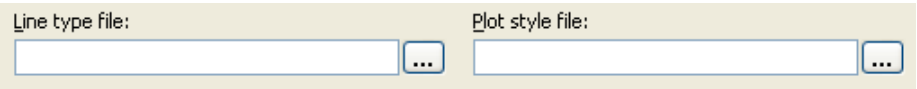


Fig. 8.10 Options to let the target system define types/styles

You exit the **Settings: DXF/DWG export and import** dialog by clicking the **OK** button. The **CANCEL** button discards all changes you may have made to the scheme settings. EPLAN applies the newly defined scheme in the **DXF/DWG export** dialog and adjusts the target (directory) for the output, if it was changed in the scheme. When you click the **OK** button, EPLAN generates a DXF or DWG file for each selected page using the defined output parameters.

NOTE: If this should result in duplicate file names, EPLAN will display a message and then cancel the export without any further messages.

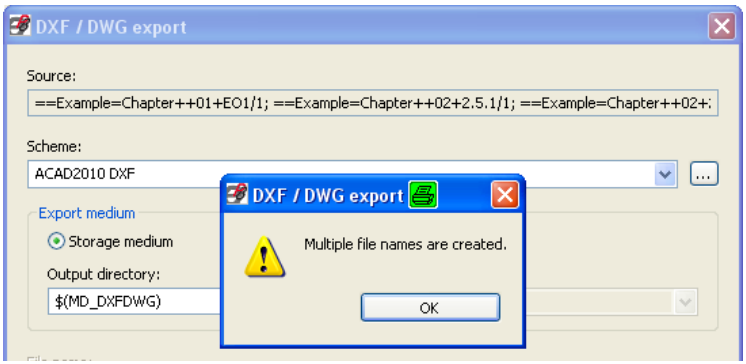


Fig. 8.11 Error message when generating the output

If the output goes through without any problems, the output will be generated. EPLAN then closes all dialogs, and again focuses on graphical editing.

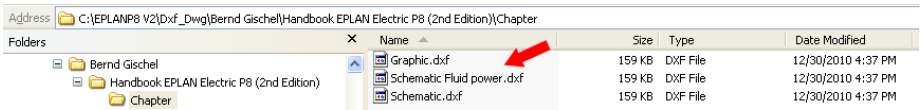


Fig. 8.12 Generated DXF files in the stored directory structure

8.1.2 Import of DXF and DWG files

In addition to exporting (generating) DXF/DWG files, EPLAN can import CAD drawings in DXF and DWG formats and create project pages from them.

Fig. 8.13 Import DXF/DWG files

The **DXF/DWG file selection** dialog is opened via the **PAGE/IMPORT DXF/DWG** menu. By default, EPLAN opens the DXF/DWG directory defined at installation for selecting the files. You can, of course, select any other directory.

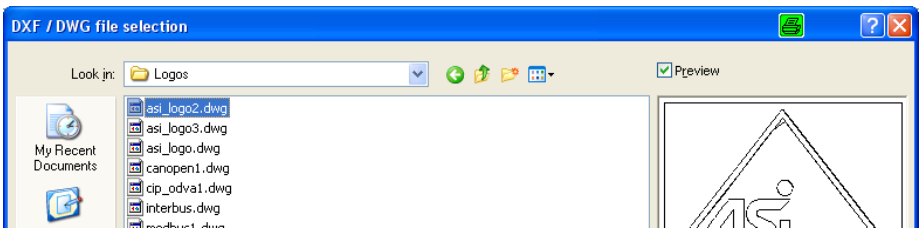


Fig. 8.14 Dialog for importing DXF or DWG files

The desired file type for the import (DXF or DWG) is set in the lower area of the dialog.

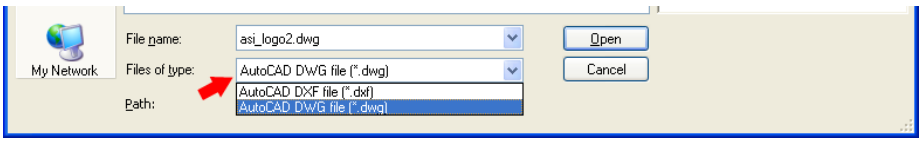


Fig. 8.15 Selection of the file type

You can switch on the **PREVIEW** option to make initial selection of the file easier. EPLAN then displays a small preview of the relevant file. If you select multiple files, then no preview is shown.

You can now select one or more files in the **DXF/DWG file selection** dialog and apply this selection by clicking the **OK** button. EPLAN opens the **DXF/DWG import** dialog.

The scheme set in the lower area is the same as that used to export DXF/DWG files (because this scheme also contains the import setting).

To adjust the import scheme, or create a new one, you click the **[...]** button to open the now-familiar **Settings: DXF/DWG export and import** dialog.

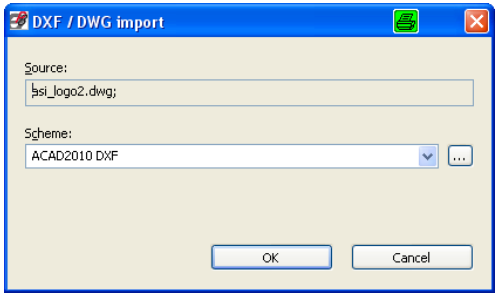


Fig. 8.16 Import dialog

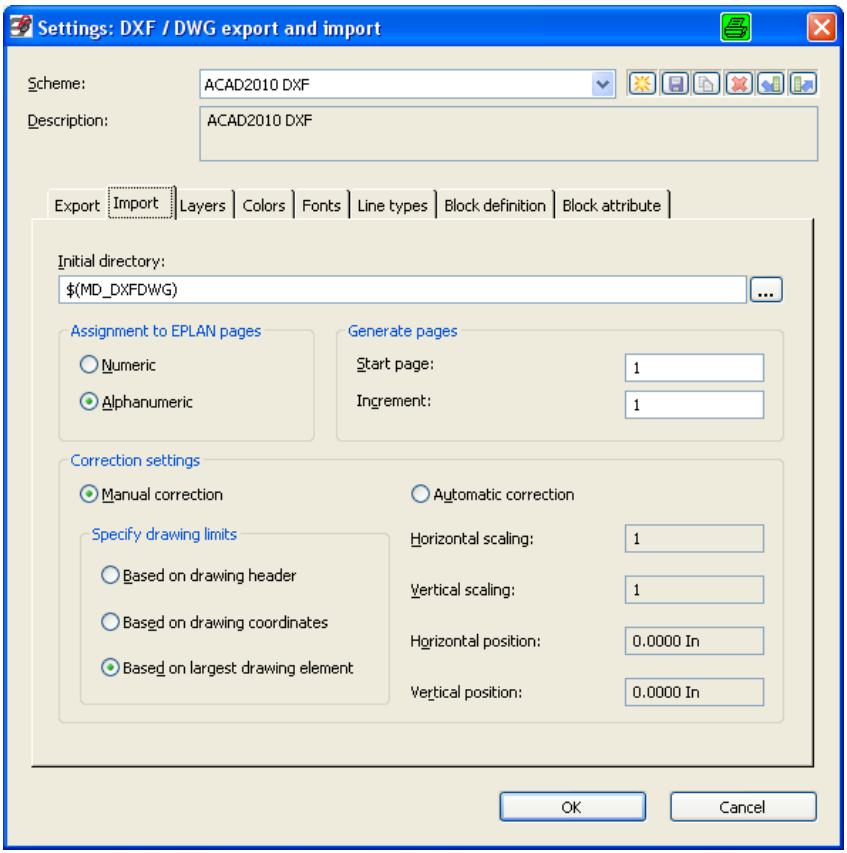


Fig. 8.17 Settings for importing DXF/DWG files

On the **Import** tab you can now define parameters and options for importing the DXF/DWG files before starting the actual import.

TIP: It is a good idea to initially accept the settings as they are defined in the EPLAN default scheme. You can specify the **Generate pages** setting in advance, if you wish. However, this is queried again in a later dialog, with more options for placing the DXF/DWG files at specific locations in the project.

If you have made no changes, you can click **CANCEL** to exit the dialog. EPLAN returns to the **DXF/DWG import** dialog. When you click **OK** here, EPLAN starts the import.

This is followed by the **Assign pages** dialog, which asks for important information defining how the DXF/DWG files are to be sorted into the existing page structure of the project.

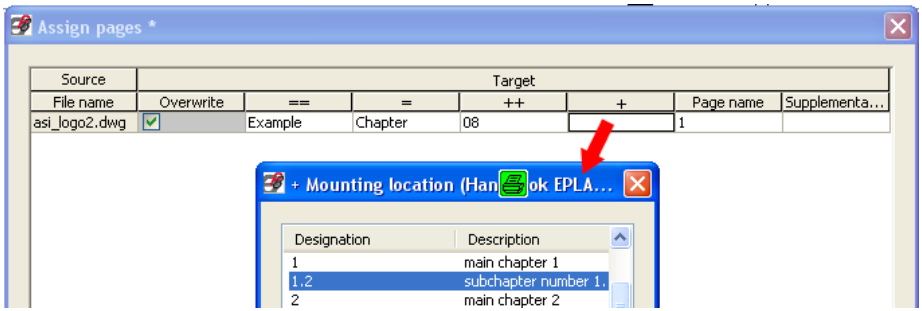


Fig. 8.18 Assign pages prior to the actual generation of the project pages

You enter the desired target designations into the dialog. You can use the **NUMBER** button to renumber the target pages before importing them if necessary.

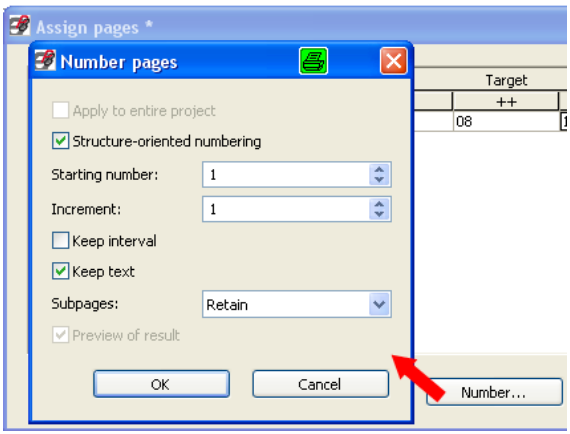


Fig. 8.19 Numbering pages automatically

When you click **OK** in the **Assign pages** dialog, the pages are imported (possibly accompanied by an **Import formatting** prompt, which can be confirmed by clicking **OK**) and sorted into the page structure of the project as specified.

This completes the import of the DXF/DWG files. The DXF/DWG files are read in and displayed on pages of the graphic type. They can now be edited in the usual manner with the normal graphical editing functions.

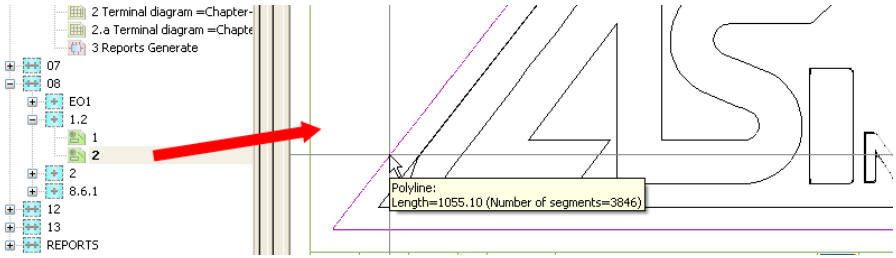


Fig. 8.20 Read-in example of a DXF file

Just to remind you one more time: These pages are graphical pages only, without any (electrical engineering) logic. When schematic pages are exported in the DXF/DWG format, the pages are "dismantled" into graphics.

If these DXF/DWG files are then imported (e. g. you import the previously exported schematics), then after importing they are still graphics files and therefore all the imported elements on the pages are also graphics. These imported DXF/DWG files are **not** converted back to logic pages.

8.2 Image files

In addition to DXF/DWG export and import for various CAD systems, EPLAN can also export project pages as image files.

Schematics and all other pages belonging to the project, such as reports, cover sheets, or the table of contents, are normally always exported as image files when they need to be archived.

The DXF/DWG formats, for example, are not suitable for long-term archiving because these formats may change over time. Image formats on the other hand will remain unchanged in the future and are recommended for long-term archiving.

8.2.1 Exporting image files

You export image files in a similar manner to exporting DXF/DWG files.

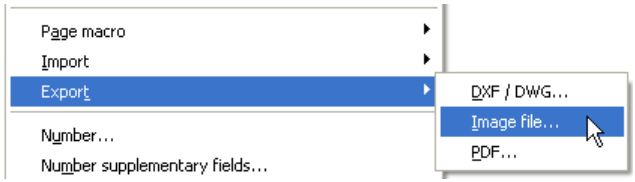


Fig. 8.21 Export/Image file menu

You first select the pages to be exported in the **page navigator**. If you wish to export the entire project, then you just need to select the name of the project in the page navigator.

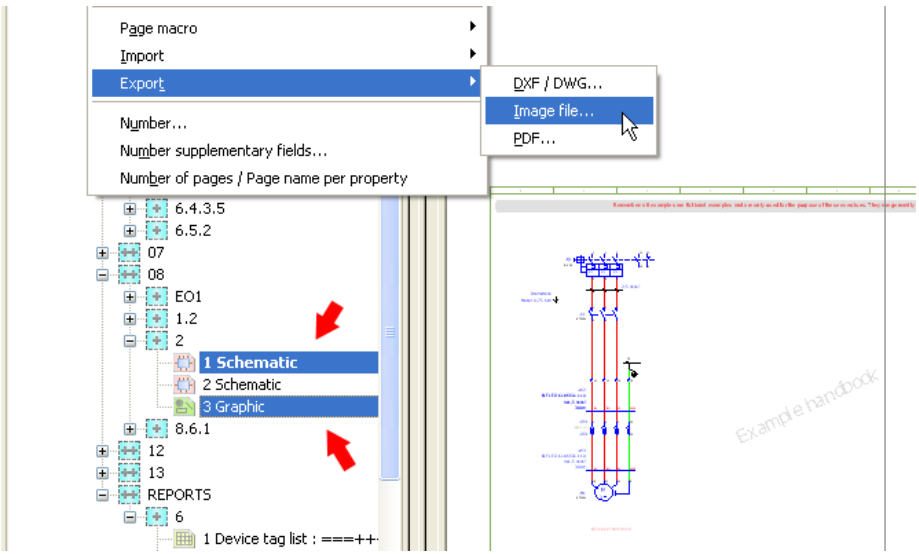


Fig. 8.22 Selected pages in the page navigator

NOTE: The **Image file export** function also only allows selections from a single project. You cannot select pages for export from different projects at the same time.

You use the **PAGE/EXPORT/IMAGE FILE** menu to open the **Image file export** dialog.

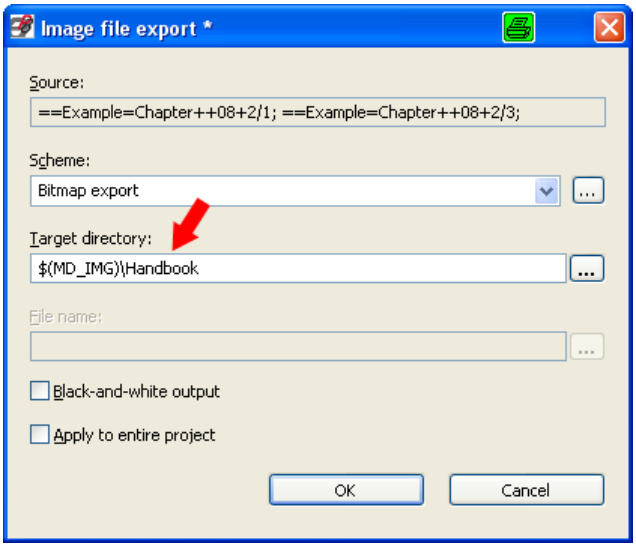


Fig. 8.23 Image file export dialog

In this dialog, you can now select and apply an existing scheme that defines the output. EPLAN also allows you to define the output settings, such as compression or the color depth of the generated image files, via your own scheme.

To create your own scheme for outputting image files, click the ... button next to the **Scheme** selection field. EPLAN opens the **Image file export** settings dialog.

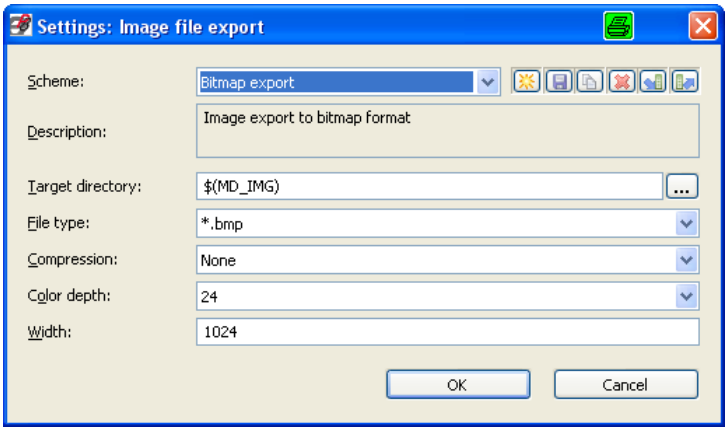


Fig. 8.24 Settings for image file export

Click the graphical **NEW** button to create a new scheme. In the subsequent **New scheme** dialog, you define a name for the new scheme and an optional description. When you confirm and apply your entries by clicking **OK**, the **Image file export** dialog is displayed again. The new scheme name is now loaded by default and you can edit it.

Make sure to click the **SAVE** button to save the scheme after you have made your settings, such as the selection of the file type, the compression, or color depth. You can now exit the dialog via the **OK** button.

EPLAN now returns to the **Image file export** dialog. In addition to allowing selection of a different scheme or output directory, this dialog has two further options.

The first option affects the output of the pages themselves: Should the output be generated in color or black and white? Selecting **Black and white output** generates image files without color information.

Exceptions to this are embedded images, such as a company logo in a plot frame or cover page form. These are usually output in the original colors. If the black-and-white output option is not used, then EPLAN generates the output pages in color. In this case, "color" means that the output in the image file looks the same as that which you see on the screen.

The second option **Apply to entire project** affects the range of the output pages. At this point you can decide to export all the project pages to image files, even if you have only selected some pages and not the whole project for exporting. You do not need to cancel the export, you can simply activate the **Apply to entire project** option.

TIP: You can directly enter a new directory into the *Target* field (output directory). EPLAN can also create a *[New directory]* automatically on export. If EPLAN cannot find the output directory then it creates one.

If all settings have been made, EPLAN can start the export. Exit the dialog via **OK**. EPLAN now generates the individual image files and stores them in the set output directory.

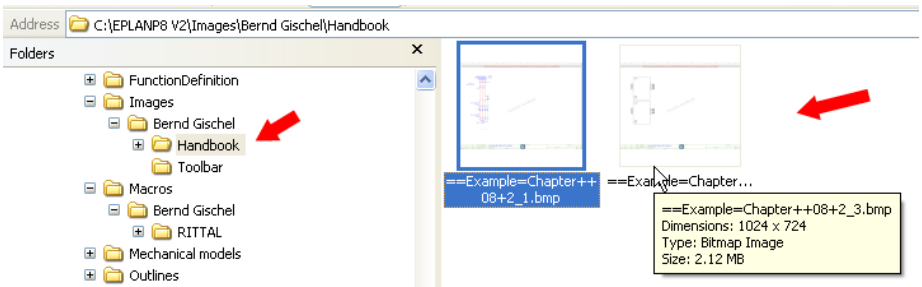


Fig. 8.25 Exported schematic pages as images

8.2.2 Insert image files (import)

You cannot import image files as pages in EPLAN. The **PAGE/IMPORT** menu has no *import image files* function. You cannot compare this type of function with the function for importing DXF/DWG files. They are completely different functions.

NOTE: EPLAN handles the import of image files in a different way. Image files are not imported; they are inserted and their size or resolution can be changed later.

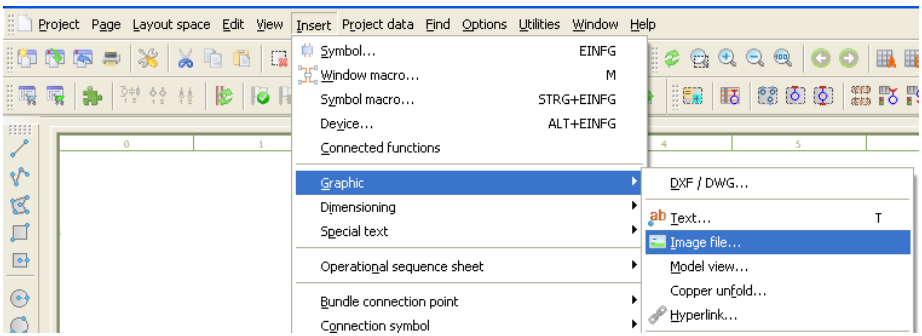


Fig. 8.26 Insert image files menu

When you select the **INSERT/GRAPHIC/IMAGE FILE** menu, EPLAN opens the **Select image file** dialog. As usual in EPLAN, the corresponding default user directory is displayed (in this case for images). Other directories can, of course, be selected.

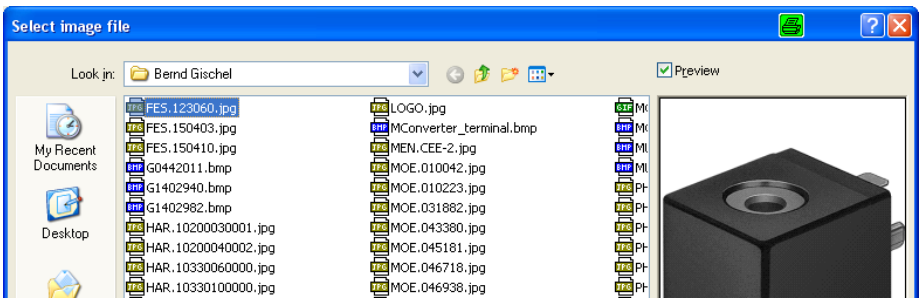


Fig. 8.27 Dialog for the selection of image files

EPLAN can insert numerous different image formats. The corresponding image format is preselected in the **Select image file** dialog in the lower area.

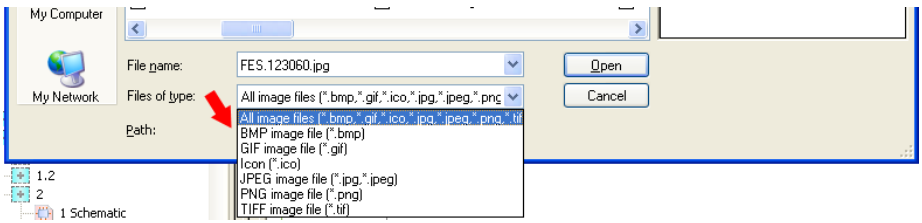


Fig. 8.28 Selection of the image format

File types such as **.bmp* and **.jpg* are the most common, but space-saving formats such as **.png* are also possible. There are no limitations on the directory structure or file names used. Special characters or deeply branched directory structures (including spaces) are no problem for EPLAN.

After selecting the desired image file (only one image file at a time can be inserted), click the **OPEN** button. The important **Copy image file** dialog is displayed.

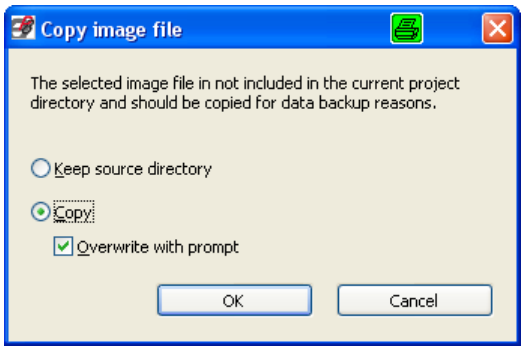


Fig. 8.29 Copy image file

TIP: You should take the message in this dialog seriously. If the image file is not copied to the project directory (the **Keep source directory** option), then the possibility exists that this "linked" (referenced) image file will not be saved during a data backup (depending on the settings). It is, therefore, a good idea to always use the **Copy** option in this dialog.

If necessary, you can enable the **Overwrite with prompt** option. If an image file with the same name already exists in the target directory, then a security message is displayed before overwriting. You can still cancel the copy operation at this point.

If you click **OK** in this dialog, then EPLAN copies the image file into the *\IMAGES* project directory and returns to graphical editing. You now define the first corner of the image by pressing the **ENTER** key, or by clicking the left mouse button.

Insert an image into a page

EPLAN displays a window that you can enlarge by pulling with the mouse or via the cursor keys. The size of this window defines the initial size of the image on the page.

Clicking the left mouse button or pressing **ENTER** again causes EPLAN to display the **Image properties** dialog. Here you can make settings defining the **size in coordinates** or in **percent** of the original size.

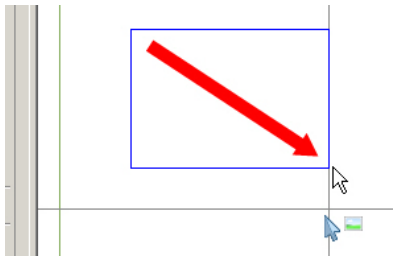


Fig. 8.30 Resizing the window

The *Keep aspect ratio* setting is important. This avoids distortion of the image. This setting should always be activated.

Properties of an image

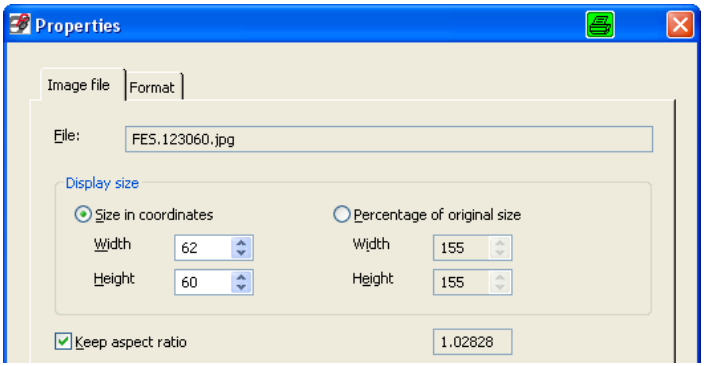


Fig. 8.31 Formatting options dialog

After you confirm the settings by clicking **OK**, the image is inserted (placed).

To edit images at a later date, for example to change the size or enter something into the **Description** field, you just select the graphic with a left click. EPLAN changes the display and "draws" a box with small handles around the graphic.

Each of these handles can be used with the mouse to make the graphic larger or smaller. You can, of course, specify the size of the image directly in the image properties at any time. A new layer in EPLAN also lets you move the graphic into the background where you can no longer use edit functions such as delete, move, etc. if the *Background* option in layer management is selected. This layer, *EPLAN705.Graphic.Background*, is selected or set in the properties dialog on the **Format** tab in the **Layer** selection.

To open the **Properties** dialog, you first select the graphic as described above and then press the **CTRL + D** shortcut key. This function for calling up the properties also exists in the right-click popup menu.

8.3 Print

The print command for pages in EPLAN is accessed via the **PROJECT/PRINT** menu item. The **Print** dialog offers a number of settings that will now be briefly described. The page print settings should be adjusted initially to suit your particular printer.

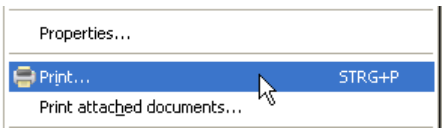


Fig. 8.32
The Print menu

8.3.1 The Print dialog and its options

If you wish to print pages, then you first select them in the **page navigator** and then prepare them via the **PROJECT/PRINT** menu. EPLAN then displays the **Print** dialog.

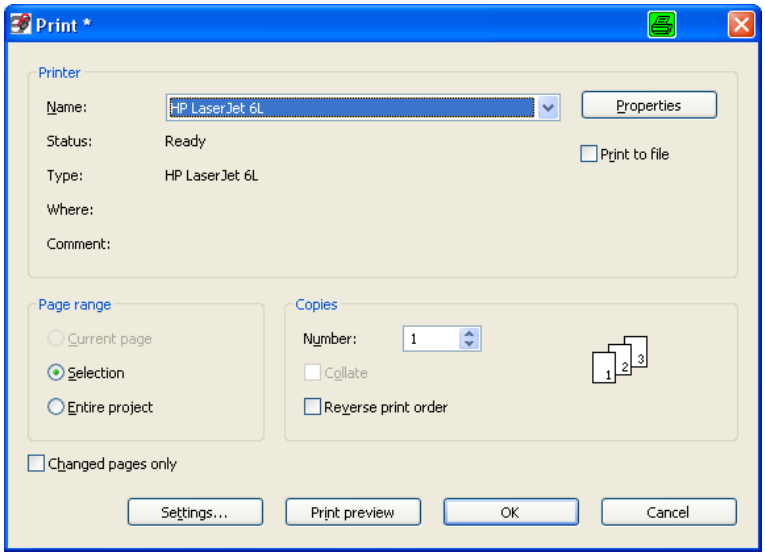


Fig. 8.33 The Print dialog

With just a few exceptions, the **Print** dialog is very similar to the conventional, familiar Windows print dialogs. Familiar options are the selection of the installed printer and the option allowing printing of multiple copies with different output sorting options.

8.3.1.1 Page range/Changed pages options

At this point, in the **Page range** options, you can select the **Selection** option to print only the currently selected pages in the page navigator or change your mind and select the **Entire project** option. The **Current page** option is automatically set to deactivated as soon as more than one page has been selected in the page navigator. The **Changed pages only** setting prints only those pages that have changed since the last print operation.

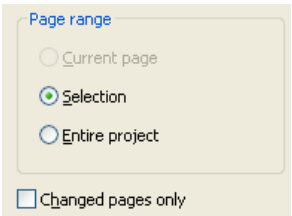


Fig. 8.34
Print options

8.3.1.2 The Settings button

The **Settings** button contains a number of print settings for the pages themselves. Here you can define the print size, whereby **Print to scale** means: If a page was assigned the A3 plot frame, for example, then it is printed 1:1 in A3 format. Printers that are smaller than A3 format (i. e. generally A4 printers) will only print the upper left corner of such pages.

The **Scale to page** setting and the **Keep aspect ratio** parameter should be used by default. This causes all larger dimensions to be scaled down to the printer model and a complete page is always printed. Scaling usually is to an A4 page size.

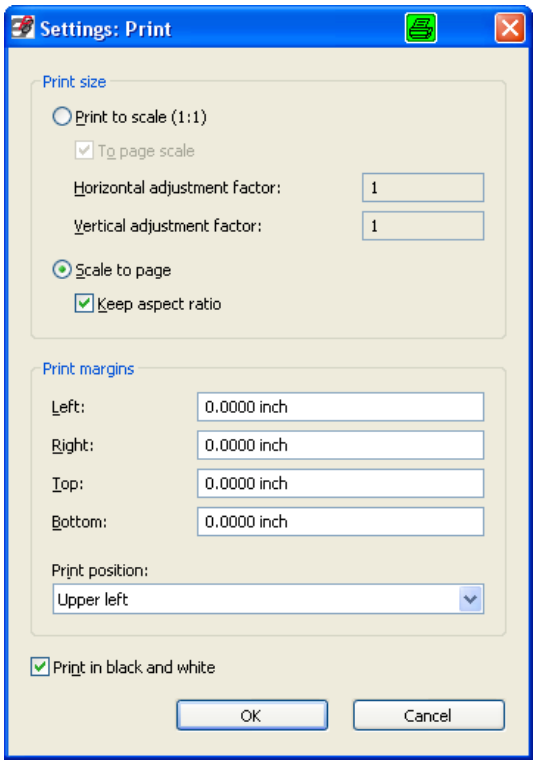


Fig. 8.35
More page settings

NOTE: The **Print margins** settings should initially be adjusted to suit the installed printer(s).

Unfortunately, one cannot provide generally applicable suggestions for the print margins because there are simply too many printer models with widely varying technical properties. Here you can only try things out and make a number of tests with the connected printer.

Selecting the **Print position** allows you to position the page on the printed sheet of paper. EPLAN offers a number of different print positions. It is a good idea to use the **Upper center** setting with an upper print margin of 10 mm.

The **adjustment factors** offer an additional setting to compensate for small deviations in printouts to scale. Here, too, it is a good idea to experiment a bit, because these values depend on the printer model as well as on the printer driver in use. It is therefore not possible to specify generally applicable values here.

8.3.1.3 The Print preview button

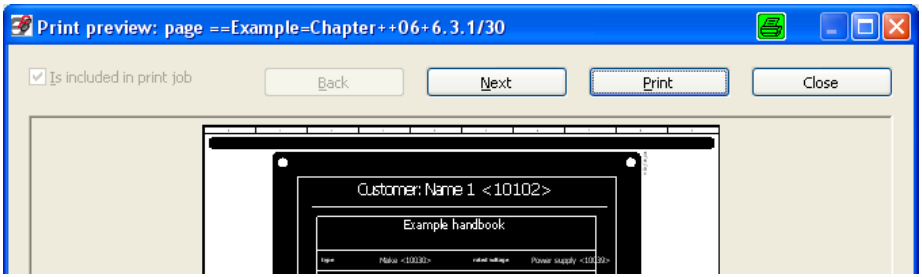


Fig. 8.36 Print preview

The **Print preview** button in the **Print** dialog provides a rough preview before performing the actual printing. However, not all the information on a page is displayed. This is not so important because, at this point, the print preview is mainly used to check the position of the page. If the print preview looks OK, you can print directly from print preview via the **PRINT** button. Without further prompts, EPLAN will start printing on the printer specified as the default printer in the operating system.

8.3.2 Important export/print setting

Aside from the simple print/export and the print settings in the **Print** dialog, there are some other important user settings related to printing and/or exporting the project. As the name implies, these settings are always user-defined.

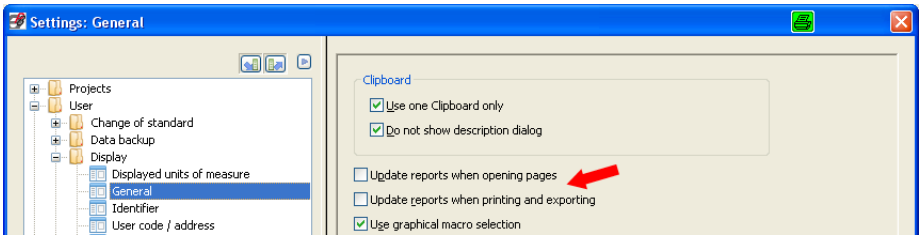


Fig. 8.37 Update reports when printing/exporting settings

To ensure that reports are current, the **Update reports when printing and exporting** setting should be activated. This prevents the project from being printed with outdated reports. This setting is located in the **OPTIONS/SETTINGS/USER/DISPLAY/GENERAL** menu.

8.4 Export and import of projects

In addition to exporting and importing image files or DXF/DWG files, EPLAN also allows entire projects to be imported and exported in XML format.

Fig. 8.38 Import and export project data

8.4.1 Exporting projects

Projects are exported from the **page navigator** via the **PROJECT/ORGANIZE/EXPORT** menu.

To export a project, you first select the desired project and then start the **EXPORT** function. When the *Export* function starts, the **Browse For Folder** dialog is displayed. This is where you define the directory where you want the exported project to be saved.

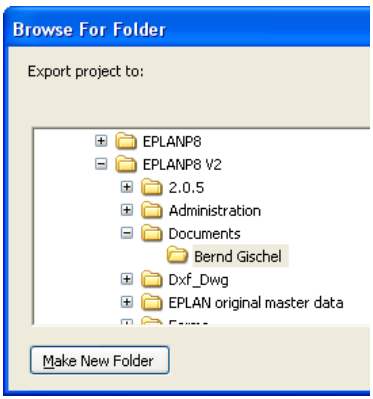


Fig. 8.39 Defining the export folder

NOTE: Using the page navigator, it is only possible to export **one** project at a time. If several projects are selected, then the *Export* function cannot be chosen (the entry is grayed out).

After you selected the export folder and click **OK**, EPLAN immediately starts the export without further prompts. The project is exported and is assigned the file extension **.epj*.

8.4.2 Importing projects

The counterpart to exporting is importing XML projects. In the graphical editor, you execute this function via the **PAGE/ORGANIZE/IMPORT** menu.

When you select the **IMPORT** function, EPLAN opens the **Import project** dialog. Here you select from the directory structure the directory containing the project (**.epj*) to be imported.

Fig. 8.40 Importing a project (*.epj)

The project is selected and applied via the **OPEN** button in the lower area of the dialog. EPLAN opens the **XML project** dialog and inserts the selected project into the **XML file** field.

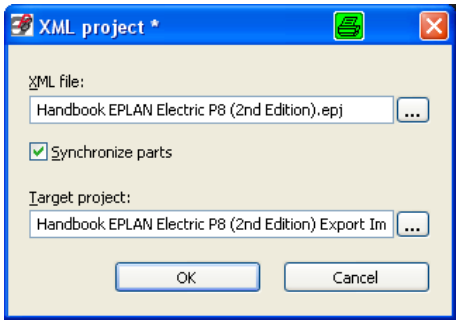


Fig. 8.41 Importing an XML project

The **Synchronize parts** parameter enables EPLAN to synchronize project parts with the parts master data during the import. You enter the name of the project into the **Target project** field and the imported project will be shown in project management under this name.

You can also use the ... button in this dialog to select a different directory. After you click **OK**, the import process starts, and EPLAN generates the project in the defined directory structure.

8.5 Print attached documents

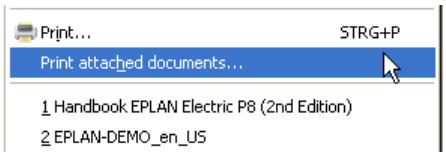


Fig. 8.42 Print attached documents

The **PROJECT** menu has a **PRINT ATTACHED DOCUMENTS** function that allows you to print all documents stored in the project, e. g. PDF documents, in a single step rather than individually.

When you select this function, EPLAN opens the **Print attached documents** dialog for the current project. In this dialog, you now have the option of excluding individual documents from printing. To do this, you remove the checkmark in the **Print** column.

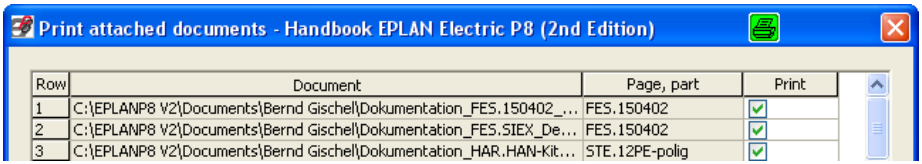


Fig. 8.43 Dialog for selection of attached documents

When you click the **OK** button, EPLAN sends all documents selected in the **Print attached documents** dialog, one after another, to the connected printer.

NOTE: External documents (*External document* page type), linked documents on report pages and linked documents on Internet pages cannot and/or will not be printed.

8.6 Import PDF comments

EPLAN can import comments (notes, etc.) stored in PDF documents (schematics generated from EPLAN) and store these in an extra layer. This allows, for example, a service technician on-site to electronically directly add his/her own comments to the PDF schematics. These can then be imported into EPLAN, viewed, edited and evaluated.

NOTE: To be able to again import a PDF with comments into EPLAN, you will need to use EPLAN's internal PDF output function. You can find it in the **PAGE/EXPORT/PDF** menu. Write protection (read only) must be turned off for the PDF output to be generated. This setting is in the **OPTIONS/SETTINGS/USER/DISPLAY/IDENTIFIER** menu. The *Read only* setting is located on the *General* tab.

8.6.1 Importing commented PDF documents

To add comments to a PDF, the PDF generated by EPLAN must be enabled for commenting. You can do that with Adobe Acrobat Professional (version >= 7) or the freeware PDF XChange Viewer (tested with version 2.0 Build 42.4).

The comment function for Adobe Reader (version 7 and higher) can be enabled in Acrobat Professional using the **ENABLE FOR COMMENTING IN ADOBE READER** menu item in the **COMMENTS** menu. With the freeware PDF XChange Viewer, all you need to do is open the PDF, enter the comments and then save the file.

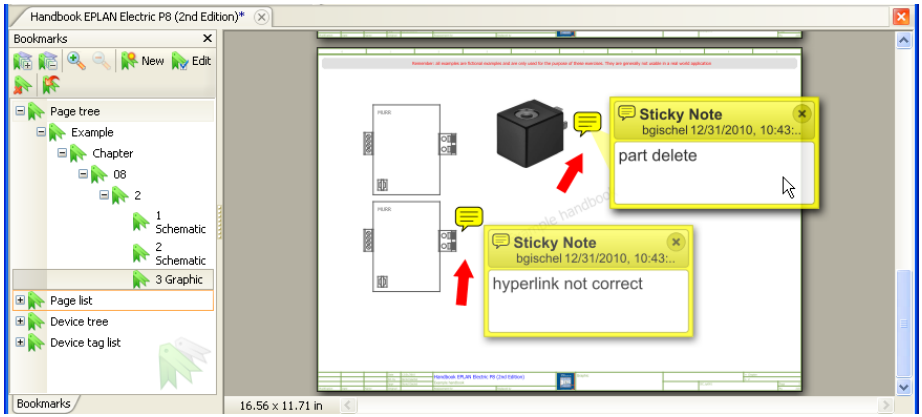


Fig. 8.44 Example of comment entries on a schematic page

After commenting the PDF document in Acrobat Reader or PDF XChange Viewer, it must be saved and closed. In EPLAN, you can now import (read in) this commented PDF document from the specified directory via the **PAGE/IMPORT/PDF COMMENTS** menu.

NOTE: To import a project, it must be in the same directory to which it was exported.

After the PDF document is imported, the **Import PDF comments** dialog containing import information is displayed.

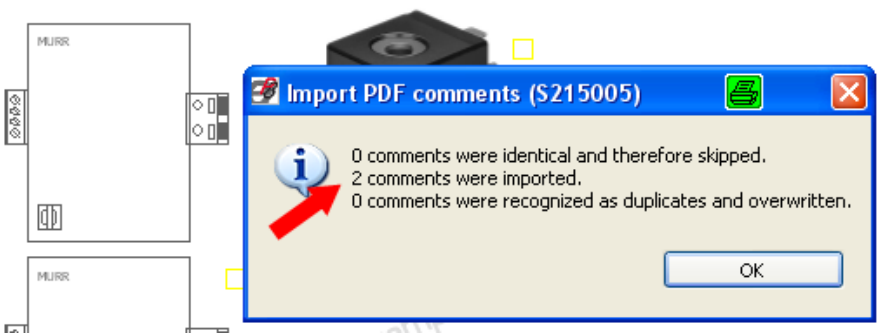


Fig. 8.45 Successful import

You can see the results of the import in the comments navigator. You can access the comments navigator from the **PAGE/COMMENTS NAVIGATOR** menu.

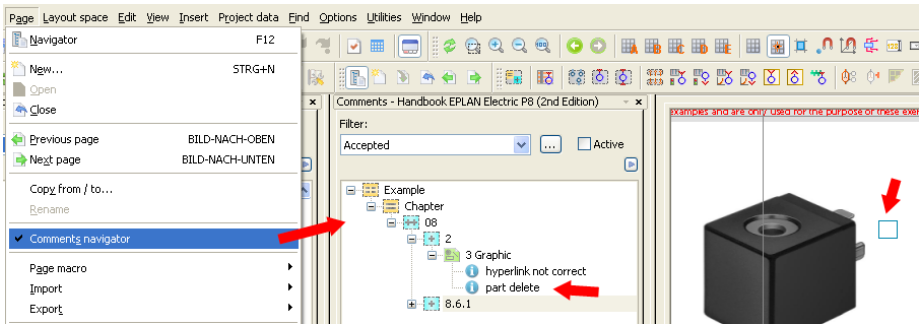


Fig. 8.46 The comments navigator

In the comment navigator, you can edit comments (e.g. set the status to "Rejected", "Accepted", etc.) in the comment properties.

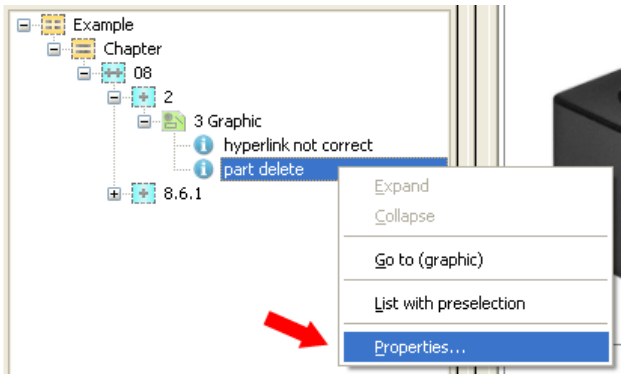


Fig. 8.47 Properties of a comment

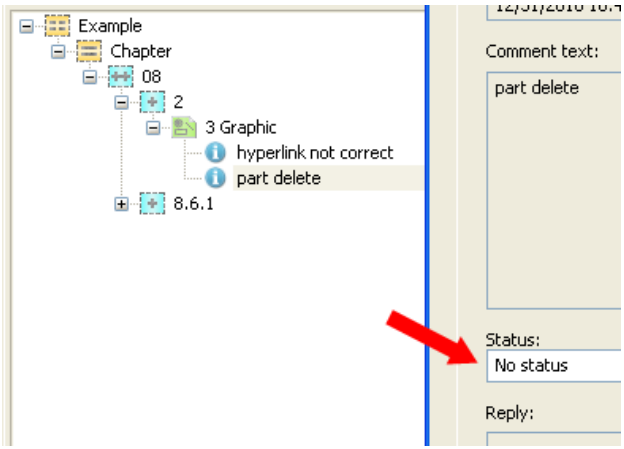


Fig. 8.48 Status of a comment

These properties are, of course, also available directly at the comments in the schematic. You can open them by double-clicking the yellow box.

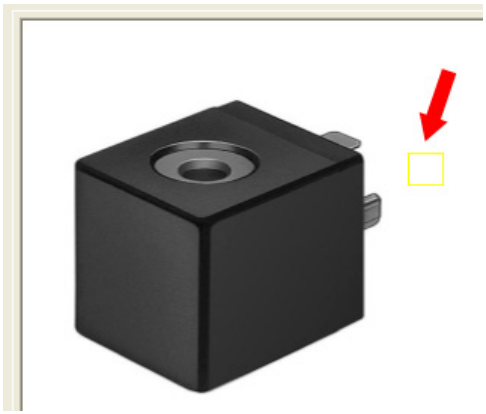


Fig. 8.49 Comment selection

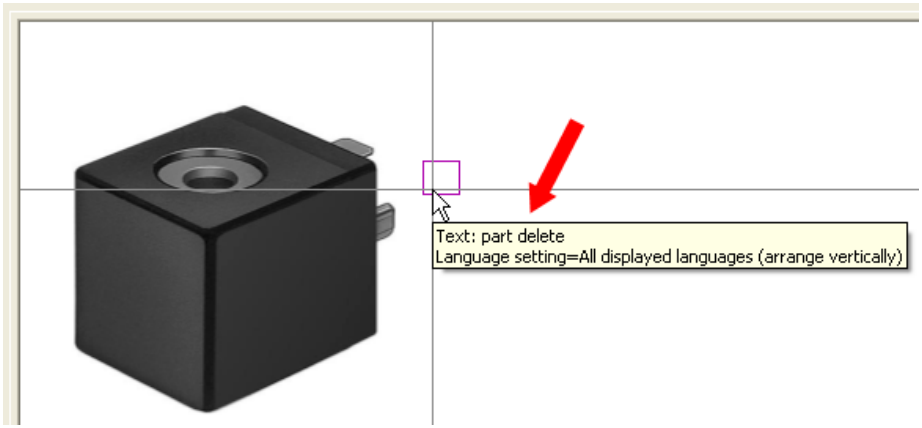


Fig. 8.50 Comment

8.6.2 Deleting PDF comments

Not only can you edit PDF comments (status of comments, etc.), you can also delete them in the usual way by selecting them and then calling the normal EPLAN delete function.

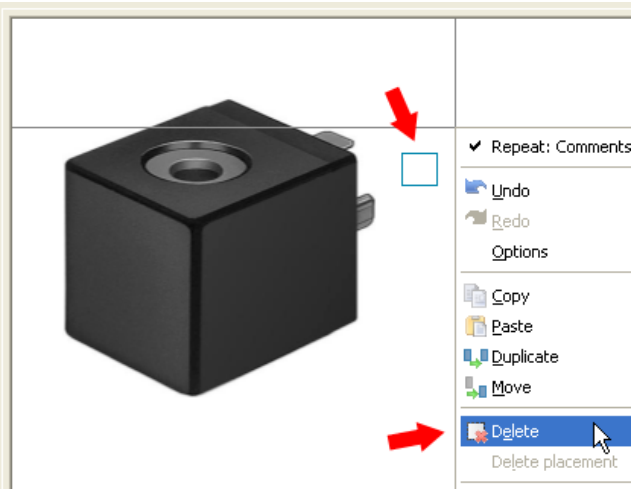


Fig. 8.51 Deleting comments

Another much more convenient method is to compress the project and create your own "Remove all PDF comments" scheme (via the **PROJECT/ORGANIZE/COMPRESS** menu and then modify an existing scheme or create a new one). This automatically and permanently removes all imported and unnecessary PDF comments from the project in a single step.

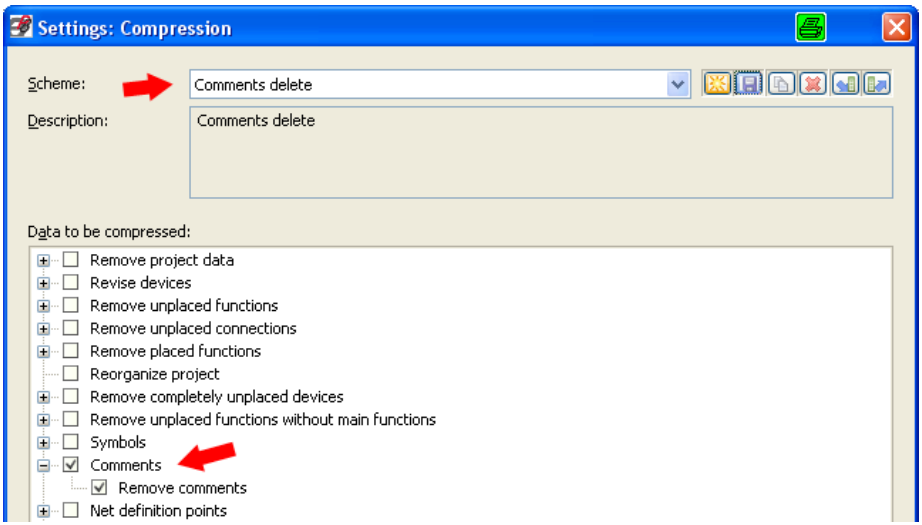


Fig. 8.52 Compression scheme to remove PDF comments

8.7 Generate PDF documents

To exchange documents without problems, the PDF format is usually used. The PDF format has become a "de-facto standard" for worldwide document exchange. It therefore makes sense to also use the PDF format for exchanging schematics and other schematic documents.

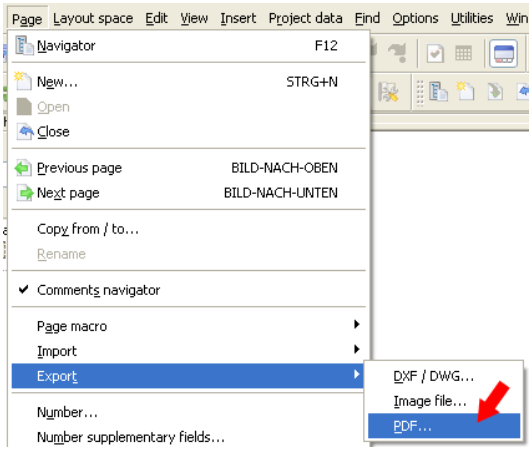


Fig. 8.53
Export as a PDF file menu

The PDF format integrated into EPLAN offers more than just the simple generation of PDF documents. A certain amount of "intelligence" can be included in the documents. In EPLAN, "intelligence" means that navigation elements, such as jump functions at cross-referenced components, etc., can also be included in the PDF. Besides the intelligent PDF, EPLAN can also generate a PDF in the PDF/A-1b format for longer-term archiving.

NOTE: If the PDF/A-1b format is used, linked documents and models will **not** be included in the PDF output.

The setting whether the PDF is a standard PDF or one in the PDF/A-1b format is defined in the settings for the PDF output. First select the **PAGE/EXPORT/PDF** menu and then open the scheme for the PDF output (click *More*) in the **PDF export** dialog.

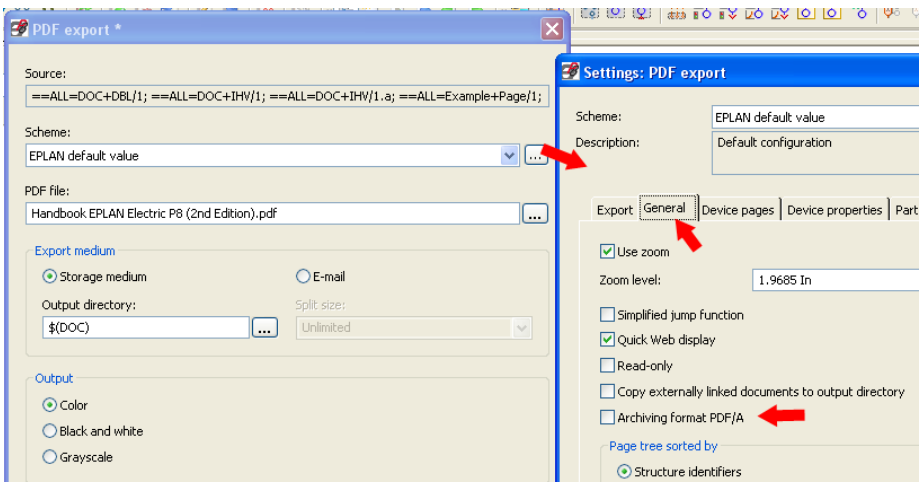


Fig. 8.54
Activate archiving format PDF/A-1b

In the now-open scheme with the settings for the *PDF export*, open the *General* tab and activate the *Archiving format PDF/A* option.

8.7.1 Export of PDF files

Generating PDF documents is child's play in EPLAN. In the same way as when exporting DXF or image files, you first select the desired pages or the entire project in the page navigator. The PDF export is also subject to the restriction that you can only export pages from one project at a time. You cannot export pages from different projects that are open in the page navigator at the same time.

After selecting the pages, you use the **PAGE/EXPORT/PDF** menu to open the **PDF export** dialog.

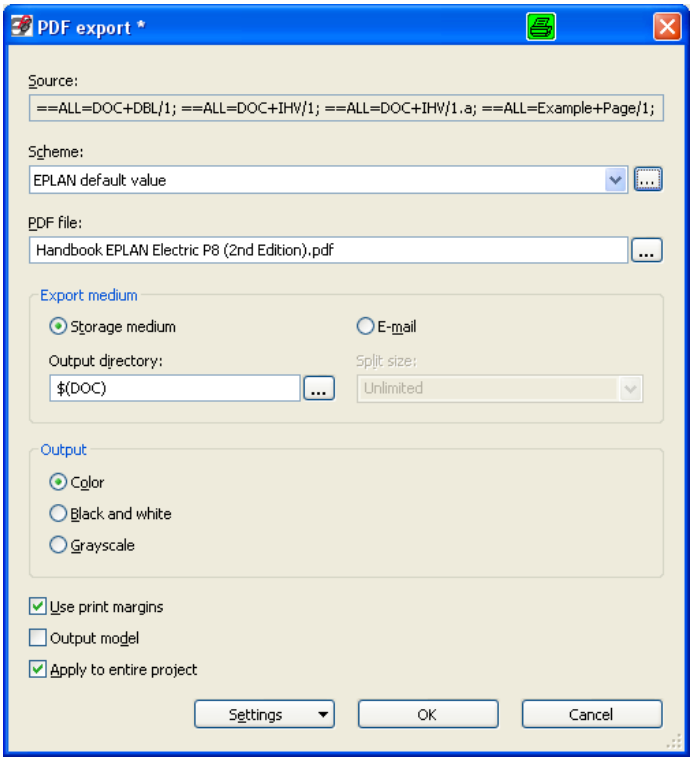


Fig. 8.55 PDF export dialog and its settings options

The **PDF export** dialog is similar to the DXF/DWG export dialog or the image file export dialog. EPLAN automatically enters the selected pages into the *Source (page/3D model)* field. If the project name was selected in the page navigator, then the Source field contains the project name, and the *Apply to entire project* option is grayed out.

Enter the name of the PDF file into the *PDF file* field. By default, EPLAN sets the target directory for the file name automatically to the project directory **\[PROJECT NAME]\ DOC** in the *Export medium* selection and, here, in the *Output directory* field. The PDF file to be generated also automatically receives the project name.

The target directory and the name of the PDF file can, of course, be freely chosen. Click the ... button next to the PDF file field. EPLAN opens the **PDF export** dialog. Here you can use the familiar Windows functions to change the directory, create new directories, or change the PDF file name.

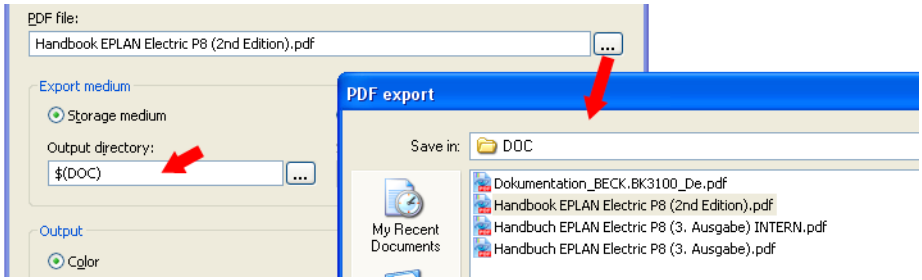


Fig. 8.56 Selection of the directory and file name for export

These modifications are saved with the **SAVE** button. EPLAN then applies the new settings of the PDF file name to the *PDF file* field.

In the settings of the **PDF export** dialog, it is also possible to use the *E-mail* option, including its settings (split size), directly as an export medium. If this setting is active, EPLAN will generate the PDF, open the e-mail client and attach the PDF (zipped) to a new mail.

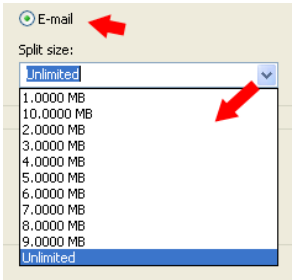


Fig. 8.57
Use E-mail option

The PDF file can now be generated. But EPLAN also provides other settings defining the format of the output: the settings *Color*, *Black and white*, *Grayscale* as well as *Use print margins*, *Output model* and *Apply to entire project*.

You can subsequently activate the **Apply to entire project** option if you have decided to output the entire project and not just the pages you selected. If the entire project has already been selected, this setting will appear grayed out.

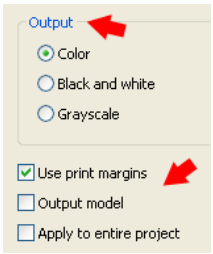


Fig. 8.58
Other settings

The **Color** output option generates a color PDF – as shown on the screen. Generally this looks quite nice, but it is unsuitable for printing on a black-and-white printer. The color settings of the background scheme *White* are applied. This setting can be found under the **SETTINGS/USER/GRAPHICAL EDITING/GENERAL** options.

When the **Black and white output** option is activated, a black and white PDF document is output. This means that all colored elements are printed in black on a white background. However, embedded image files are output in color.

The last output option, **Grayscale**, generates a grayscale PDF. You should test this option to see what best suits your needs. I recommend this option.

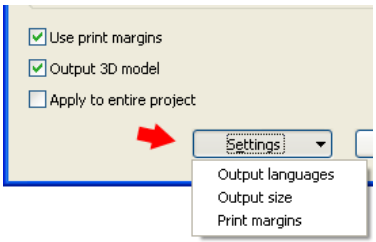


Fig. 8.59 Settings: Print margins

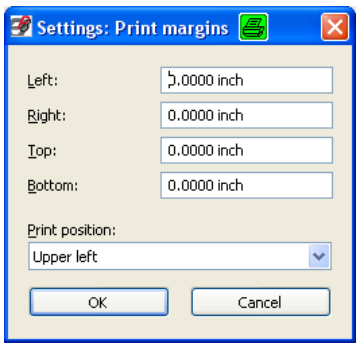


Fig. 8.60 Print margins dialog

The **Use print margins** setting uses the print margins defined in the workstation *print* settings. These are located under **OPTIONS/SETTINGS/WORKSTATION/GRAPHICAL EDITING/PRINT**. They can, however, be changed directly in the **PDF export** dialog at any time. To change these settings, click the **SETTINGS** button and enter your desired settings in the **PRINT MARGINS** menu item.

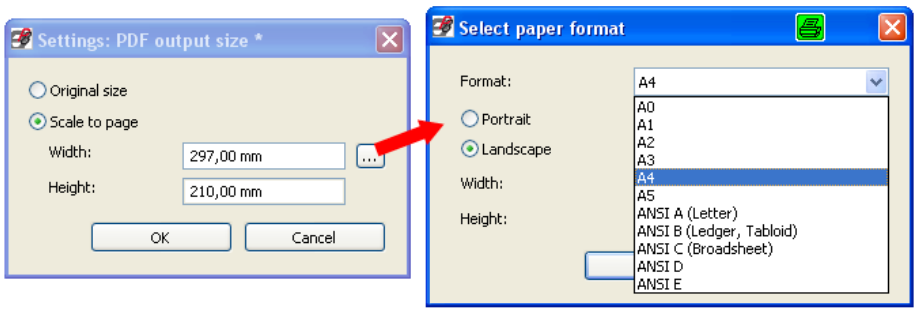


Fig. 8.61 Settings for PDF output size

The **Output size** setting lets you scale pages or use the original page size (the page size is determined based on the dimensions of the set plot frame) when outputting to a PDF.

The **Output model** option enables EPLAN to output the models in the project (3D panel view) at the same time.

When you have finished entering the settings, you can generate the PDF document by clicking **OK**. EPLAN generates the PDF document and saves it in the specified target directory. After the successful PDF export, there will be no other messages.

8.7.1.1 Other settings for the PDF export

Aside from the "simple" settings and options for the PDF export described, EPLAN offers a great number of additional settings on the basis of a PDF export scheme, which can be used to generate information and define how the PDF document is to be generated.

To open the scheme, click the *More* button in the **PDF export** dialog.

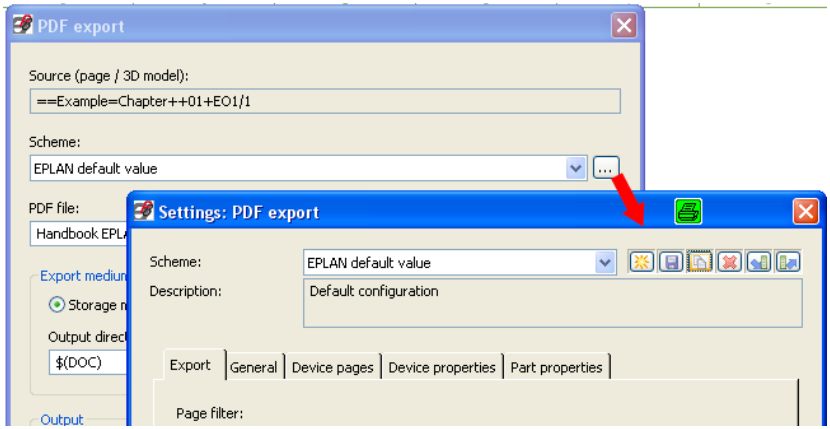


Fig. 8.62
Scheme for other detailed settings

This scheme has several tabs. The *Export* tab contains various settings about how and where the PDF is to be generated, as already described previously.

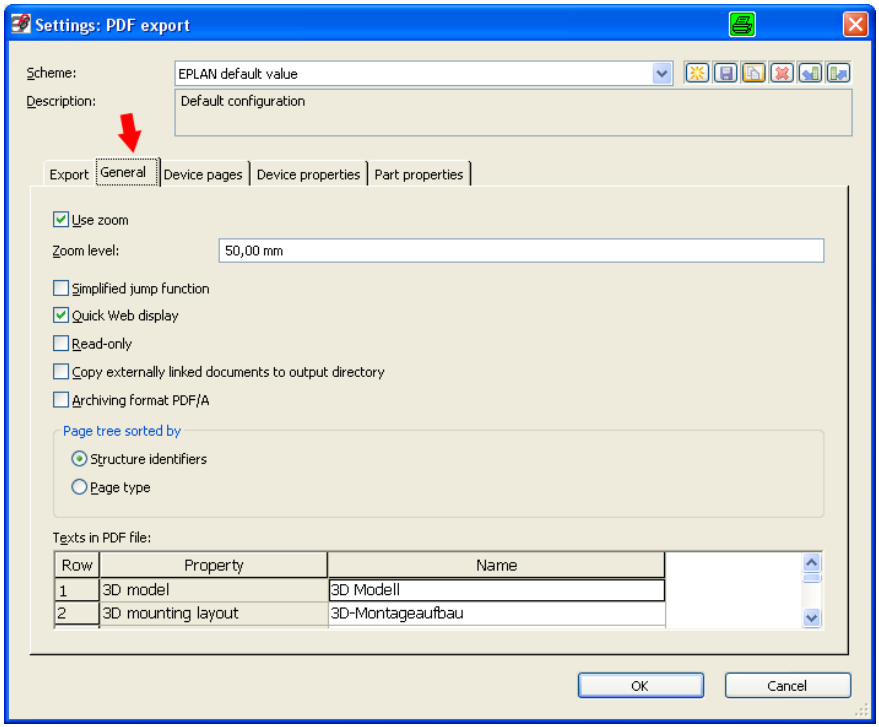


Fig. 8.63
General settings

The *General* tab contains settings and options regarding the basic functions to be given to the PDF. This includes, for example, *simplified jump functions*, *quick web display* and whether the PDF is to be sorted according to structure identifiers or page types.

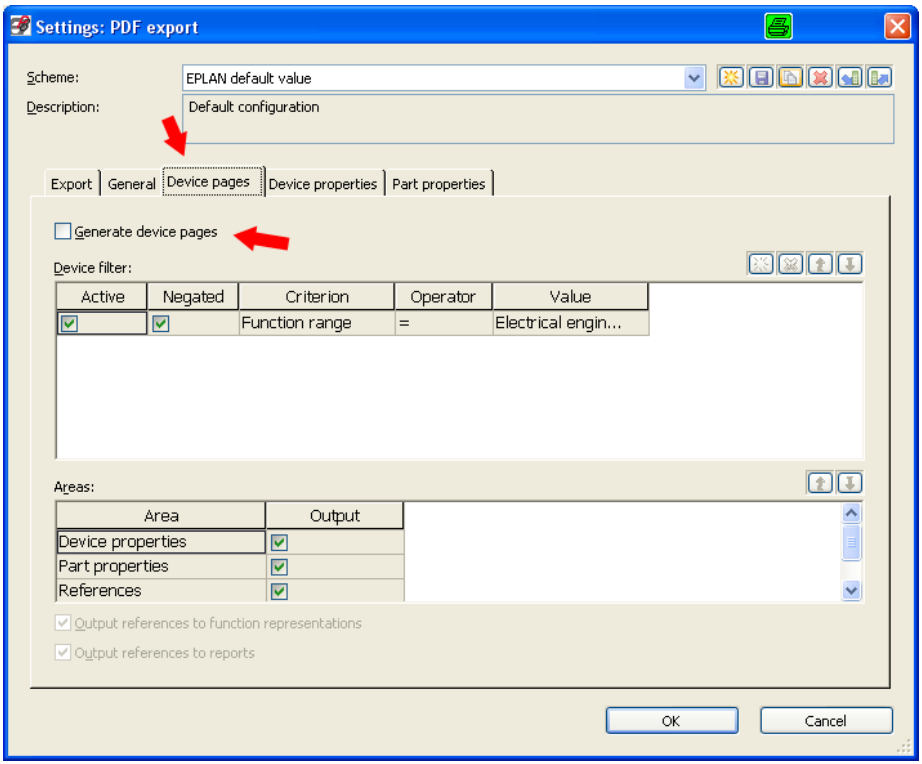


Fig. 8.64 Settings for device pages

The *Device pages* tab defines whether device pages are to be generated at all. If so, you can make other settings to refine the level of detail. For example: Is the output also to include *device properties* or *part properties*? To which *function ranges* should these settings not apply?

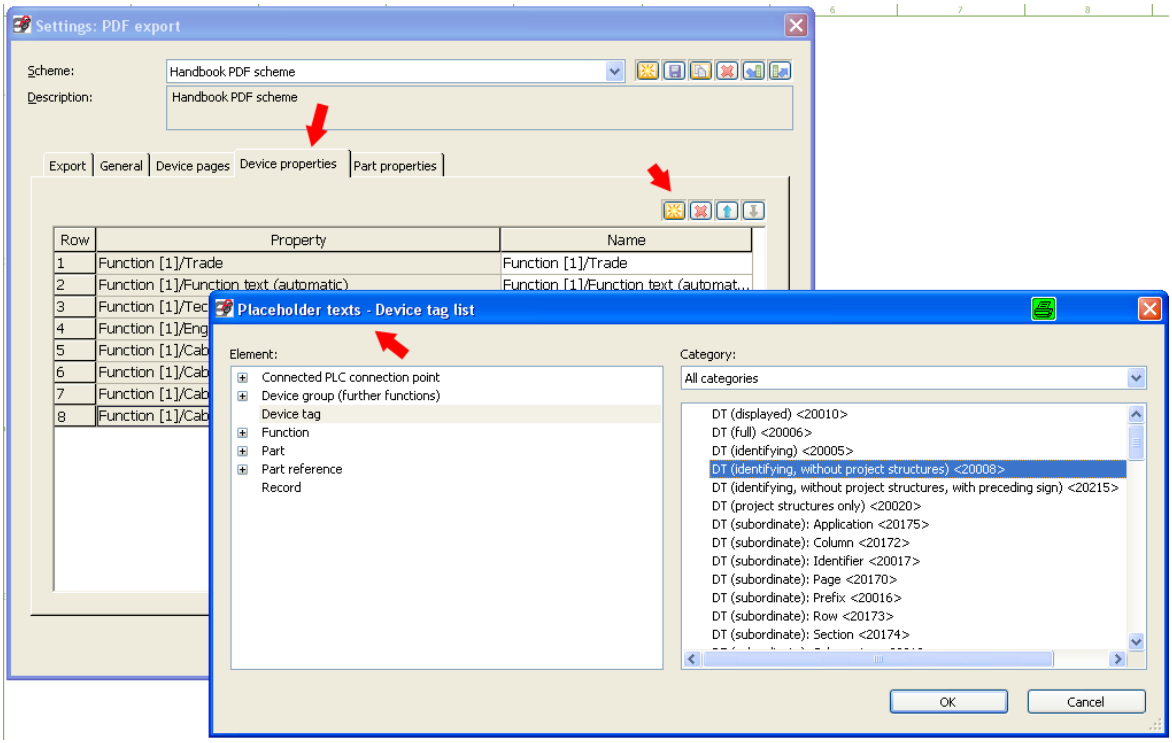


Fig. 8.65 Define device properties

Using the *Device properties* tab, you can output in the PDF very specific properties of the devices. Using the usual EPLAN approach, you can add *placeholders* here or delete them from the list.

The *Part properties* tab allows you to select the desired part properties to be written into the PDF. This could include, for example, only the *part number* and the first *designation* of a part.

If all settings have been changed accordingly and the scheme has been saved, this scheme will be available to all other projects. The settings contained in the scheme could then be used to let EPLAN generate the PDF.

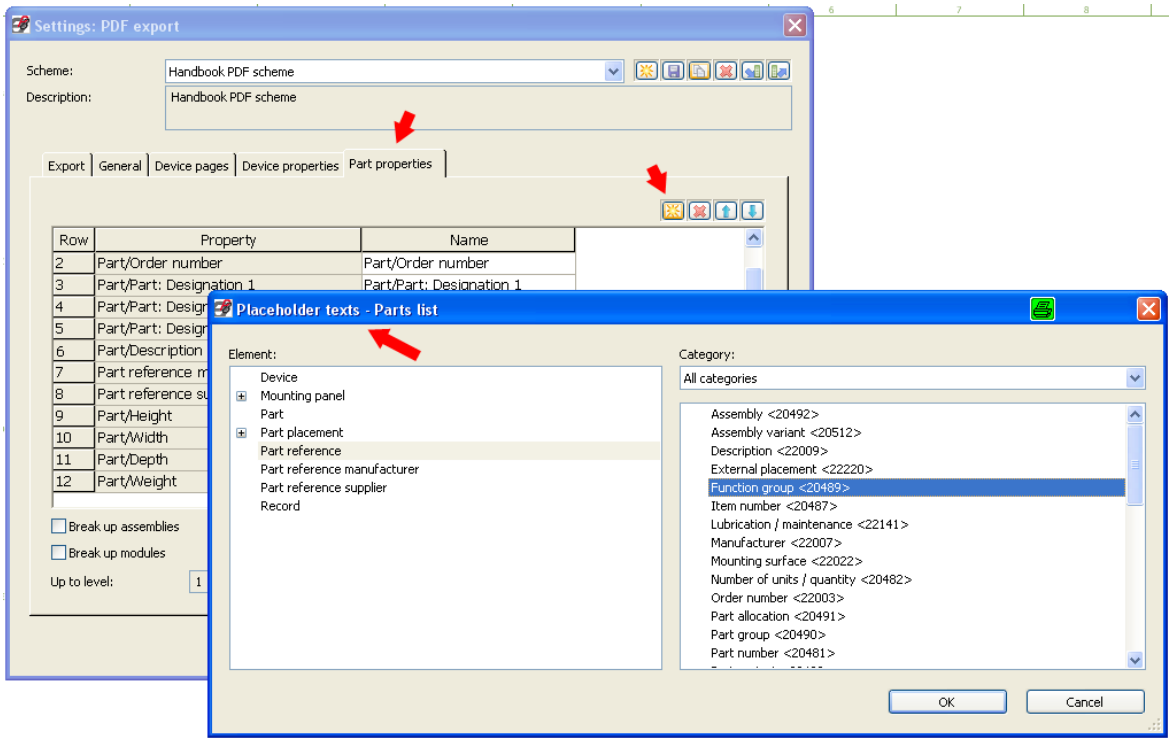


Fig. 8.66 Define part properties for the PDF export

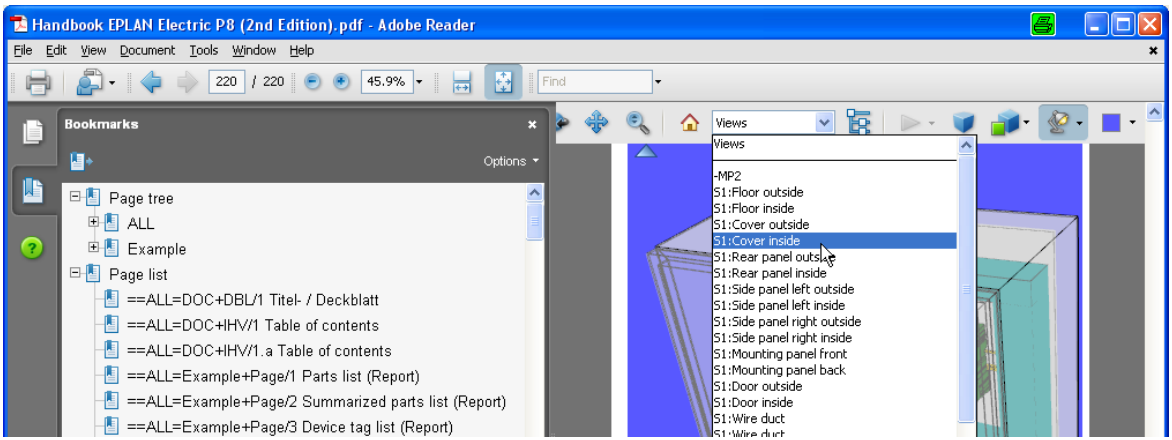


Fig. 8.67 Example: generated PDF, opened with Acrobat Reader

9 Data backup

An important feature in EPLAN is the data backup. Data backup not only includes zipping, unzipping, backing up, and restoring projects, there is also an e-mail function that provides a convenient way of electronically exchanging EPLAN projects between users.

Zipping, backing up, and restoring projects is not all that EPLAN is capable of, however. Individual files, such as a plot frame, the parts database or dictionary (foreign language database), can also be separately backed up and restored.

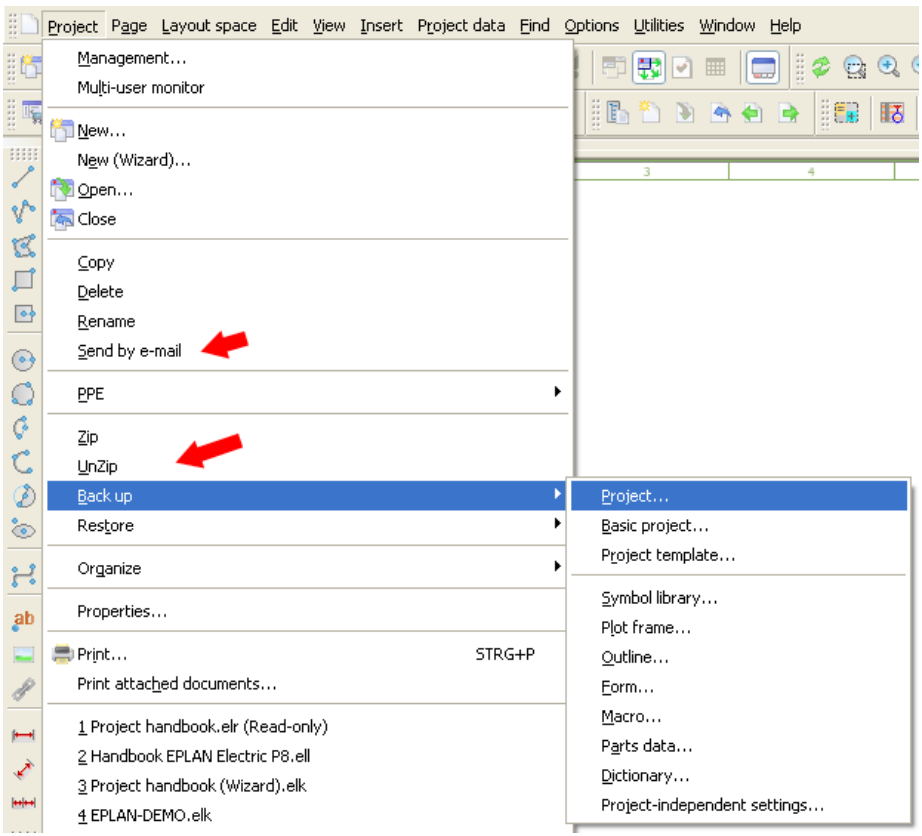


Fig. 9.1
Menu showing backup methods

9.1 Zipping and unzipping of projects

In EPLAN, zipping a project means that the entire project directory is compressed into a single space-saving project file (plus the project information file *ProjectInfo.xml*).

9.1.1 Zip projects

Zipping is started via the **PROJECT/ZIP** menu. After a confirmation prompt, EPLAN immediately zips the project. The *Zip* function itself does not require any special settings.

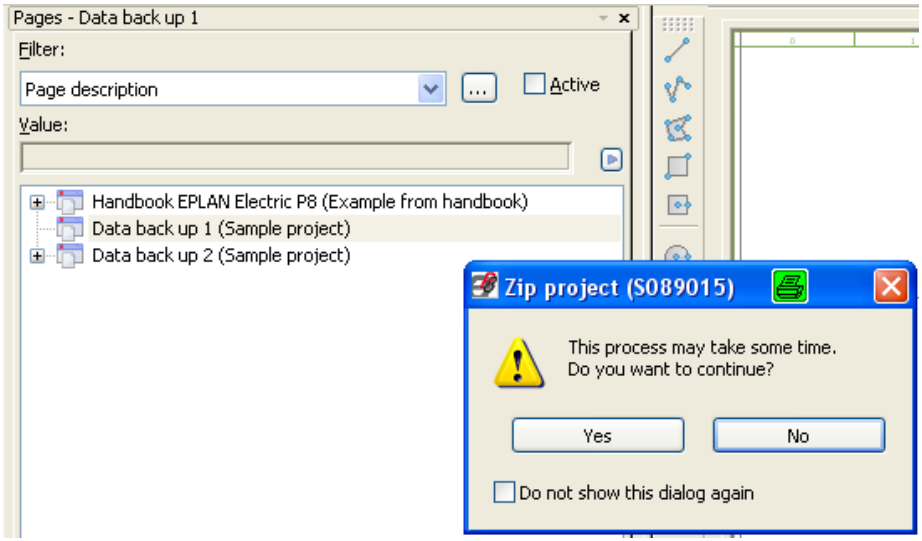


Fig. 9.2 Prompt whether the project is to be zipped

All data in the project directory is zipped. EPLAN leaves only the project properties information (*ProjectInfo.xml*) and the zipped project file *([Projectname].zw0*) in the project directory.

NOTE: Only one project at a time can be zipped via the **PROJECT** menu. Zipped projects are no longer displayed in the page navigator because after being zipped, they are automatically closed and visually removed from the page navigator.

Once EPLAN has completed the **ZIP** function (of the project), it opens a dialog with a message, and the project "disappears" from the page navigator.

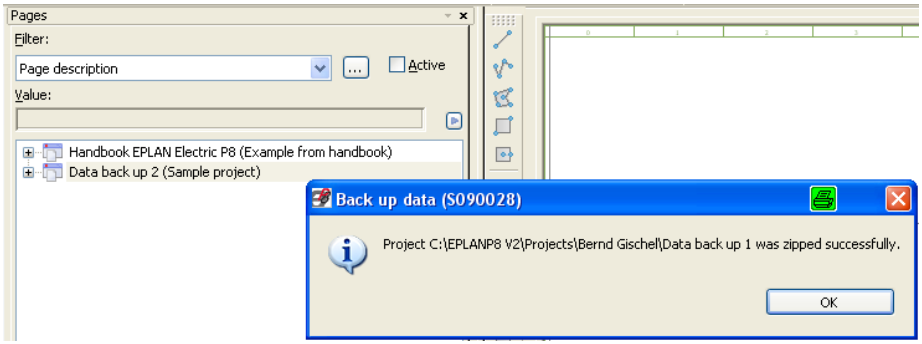


Fig. 9.3 "Project was successfully zipped." message

9.1.2 Unzip projects

A zipped project can be unzipped for editing in various ways.

One way is to select the project via the **PROJECT/UNZIP** menu and then unzip it by clicking the **OPEN** button. When the project has been unzipped, it will not be displayed in the page navigator. The unzipped project must first be opened via **PROJECT/OPEN**.

Another way is to select the zipped project as usual via **PROJECT/OPEN**. Here, too, the project is unzipped, but unlike in the previous option, it is opened in the page navigator immediately after being unzipped.

When a project is unzipped, EPLAN automatically opens the **Restore** dialog. The project is unzipped and then the page navigator is opened. Depending on the unzipping method, the unzipped project is added to any other, possibly open, projects there. The project can immediately be edited.

9.2 Backing up and restoring projects

Compared to zipping a project, the **BACKUP** and **RESTORE** options have many more settings for customizing the data backup and restoration processes.

9.2.1 Back up projects

Unlike the *Zip* function, the *Backup* function is used to file off projects or to be able to sent them to external customers. This is the main difference between these two methods. A zipped project can, of course, also be sent to customers, but it makes more sense to use the *Backup* function for this.

When working on a project, the backup function can also be directly called from the page navigator via the **PROJECT/BACKUP/PROJECT** menu.

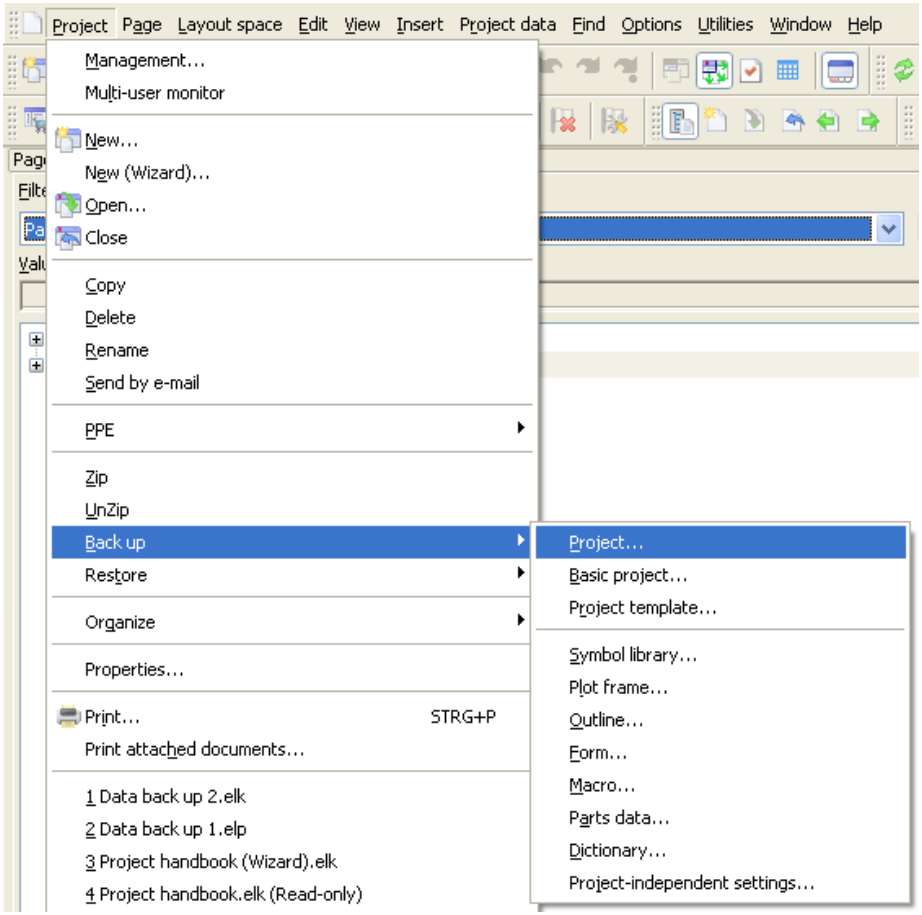


Fig. 9.4 Backup menu and its options

9.2.1.1 Backup options (in the page navigator)

Before starting the **BACKUP** function, you first select the project to be backed up in the page navigator. Now you can back it up via the **PROJECT/BACKUP/PROJECT** menu. EPLAN then displays the **Back up projects** dialog. The dialog allows you to define further settings affecting the backup scope for the project.

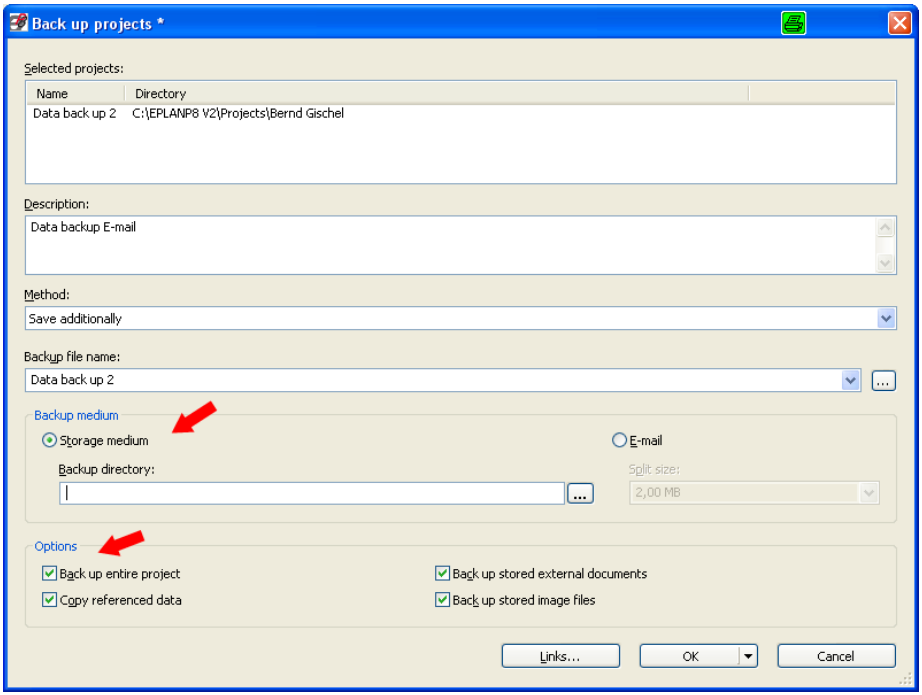


Fig. 9.5 Back up projects dialog

The **Selected projects** field contains the project name that was selected for backup in the page navigator. EPLAN applies it automatically and, next to it, displays the additional directory information for the project.

NOTE: The *Description* field is informal. This means that you can optionally enter something here. It has no effect on the actual backup process. The description is displayed in the dialog when restoring a project.

The **Method** selection field defines the manner in which the backup is to occur. This will also affect whether or not you can continue to edit the project.

EPLAN provides the user with the following options.

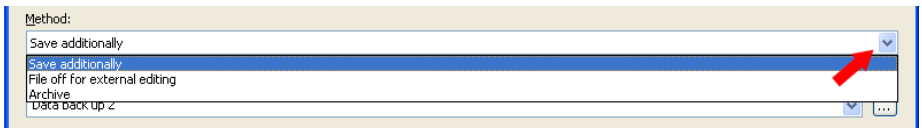


Fig. 9.6 Possible options for the backup method

Backup method

- **Save additionally:** The project is **additionally** saved on another drive (as **.zw1*) or prepared for sending as an e-mail attachment. It still remains in the original project path and is neither deleted nor changed. This is the option that I recommend.
- **File off for external editing:** The project is also additionally saved (as **.zw1*) and the original project is write-protected so that no further changes are possible. This method is useful when the project (e. g.) is to be filed off to a customer because he/she wishes to make changes and no other changes should be made to the original project until it is restored.
- **Archive:** The project is not additionally backed up; it is filed off to a different drive (as **.zw1*). The original project, except for the project information (*ProjectInfo.xml*), is deleted from the original project path.

Backup medium

In addition to the selection of the backup method, EPLAN also provides two backup media options for the project to be backed up.

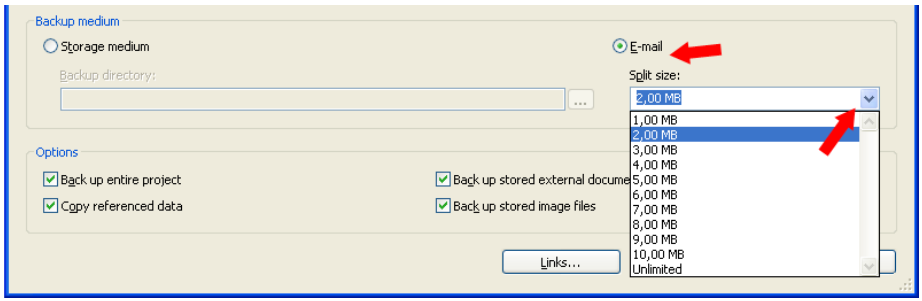


Fig. 9.7 Backup medium selection

- **Storage medium:** Usually another drive – you can use any directory structure with absolutely no limitations. If supported by the operating system, the project can be directly backed up to a CD/DVD burner.
- **E-mail:** The project is backed up and given a **.zw1* file extension. The installed e-mail client is then opened and the project added as an attachment. EPLAN provides the option of dividing the e-mail file into several files (parts) (some variables have been preset via a selection list, but these can be changed in any way; that is, even 6.5 MB is possible). This is useful when the recipient cannot receive large attachments. The recipient **must** then save all the files (parts) in the received e-mails in the same directory in order to restore the project later on.

A project was backed up using the **Save additionally** method and the **E-mail** backup medium. The split size was set to 2 MB.

Backup via e-mail

When you click **OK**, EPLAN starts the data backup. If the project directory differs from the default project directory, a message to this effect appears.

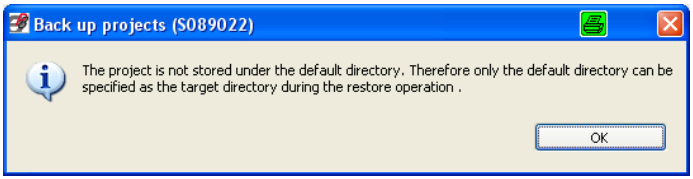


Fig. 9.8 Note indicating that the directory differs from the default

EPLAN then generates several files internally, opens the installed e-mail program and attaches the first part (Part 2 of [Total]) to a new e-mail.

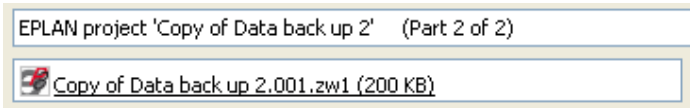


Fig. 9.9 First e-mail generated automatically by EPLAN

The following information is automatically entered as the subject: the project name, information on which part is attached, and the total number of parts to be sent.

EPLAN does not continue with the data backup until the e-mail has been sent. In this way, EPLAN automatically generates all parts and the corresponding e-mails. You only need to enter the recipient and send the e-mail.

The last part (Part 1 of [Total]) is the most important part. When restoring the project later on, EPLAN first looks for this part in the selected directory, and all the other parts must also be in this directory.

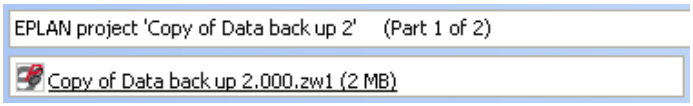


Fig. 9.10 Second e-mail containing the last part of the project backup

This completes the e-mail backup. After the final part has been sent, EPLAN displays a success message.

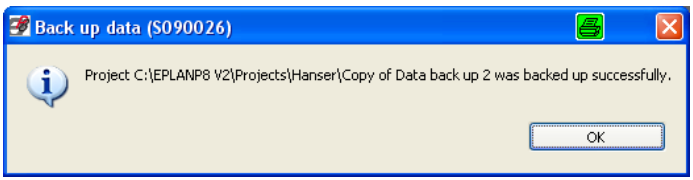


Fig. 9.11 Message that all parts have been successfully sent

NOTE: If errors exist in any of these parts, then EPLAN cannot restore the project later on. If this occurs, then EPLAN displays a message window telling the user that the project cannot be restored. In this case, the only solution is to request another data backup.

To prevent a backed-up project from becoming unnecessarily "big" (data volume), EPLAN offers the user four additional options in the **Back up projects** dialog to reduce the data volume.

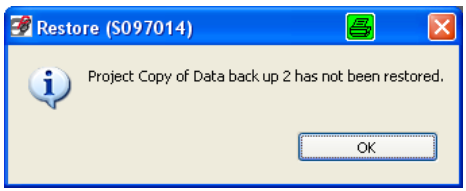


Fig. 9.12
Error message when the project could not be restored

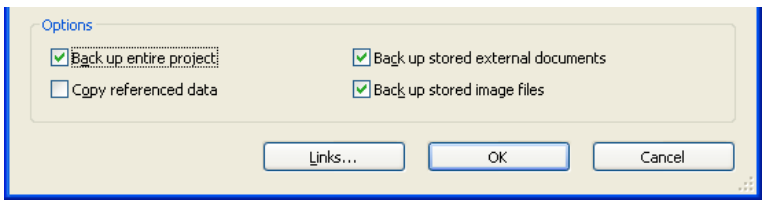


Fig. 9.13 Reduce data volume using these options

- **Back up entire project:** Backs up the entire project, including all data in the project directory and all subdirectories.
- **Copy referenced data:** If this option is set, EPLAN backs up everything, including referenced data. This increases the amount of data in the project. The options for stored external documents and stored image files must also be activated.

NOTE: Referenced data is data that is not stored in the project, but is available only as reference. In short: EPLAN only knows the link (file path) to where, for example, the graphic is stored on the server.

- **Back up stored external documents:** This option allows the user to include in or exclude from the data backup external documents stored in the project, such as PDF or Excel documents. However, only the external documents in the *\[project name]\DOC* directory are included in the backup.
- **Back up stored image files:** Similar to the previous option (external documents), you can define whether image files are included in or excluded from the backup process. This also only applies to image files in the *[project name]\Images* directory.

External documents or image files that are **not** in the project directory will **not** be automatically included in a backup. The previously mentioned **Back up external documents** and **Back up image files** options have **no effect** at this point.

If these files should also be backed up, then they must either be stored in the project (copied into the project directory on insertion during project editing) or they must be additionally and automatically stored in the project directory by EPLAN via the **LINKS** button.

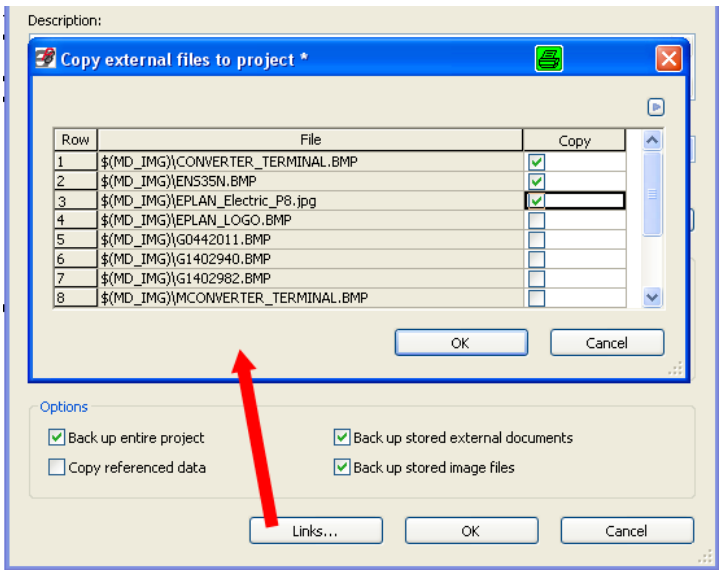


Fig. 9.14 Copying externally linked files to the project (back up projects)

For external documents and image files that are linked into the project via hyperlinks, there is another way of collecting them and copying them together into the project directory before performing a data backup. You use the **EDIT/OTHER/LINKS** menu to open the **Copy external files to project** dialog. This dialog is identical to the one that you can call up during the backup of projects.

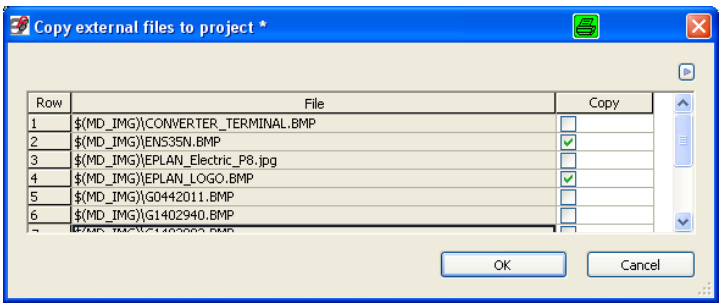


Fig. 9.15 Copying externally linked files to the project (graphical editor)

EPLAN now lists all linked external documents in this dialog. You then select the **Copy** function for all desired documents. When you confirm the dialog by clicking **OK**, all selected files are copied to the *\[project name]\DOC* or *\IMAGES* project directory.

9.2.2 Restore projects

Backed-up projects can be restored. First call the **PROJECT/RESTORE/PROJECT** menu and then select in graphical editing one or several projects you wish to restore.

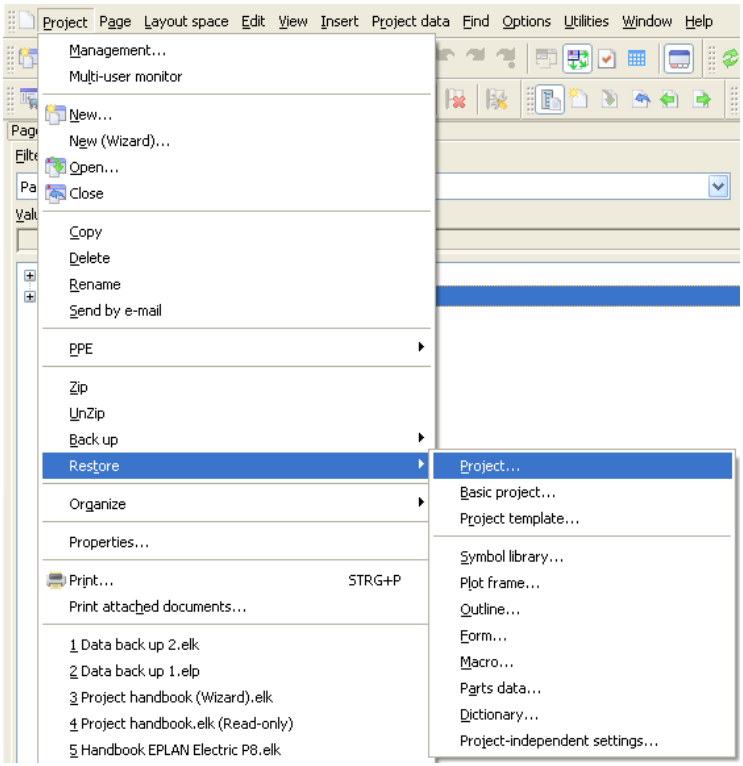


Fig. 9.16 Restore projects menu

Restore projects settings

The **Restore project** dialog opens with a number of setting and selection options.

- **Backup directory:** In the **Restore project** dialog, you can select the backup directory via the **[...]** button. You can use any type of backup directory, which is not bound to any specific structure.

> **TIP:** It is a good idea to define a permanent directory for the backup files and/or the e-mail files. It is also a good idea to integrate this directory into the normal EPLAN directory used for the installation.

- **Projects:** Here, you need to select the project to be restored.
- **Target directory:** Before restoring a project, you should first check the *target directory* field and, if necessary, set it to the correct project path. To change the target directory, click the **[...]** button to select a different directory.

- **Project name:** In this field, you can assign a new project name before you restore the project, or you can leave the current project name as is.

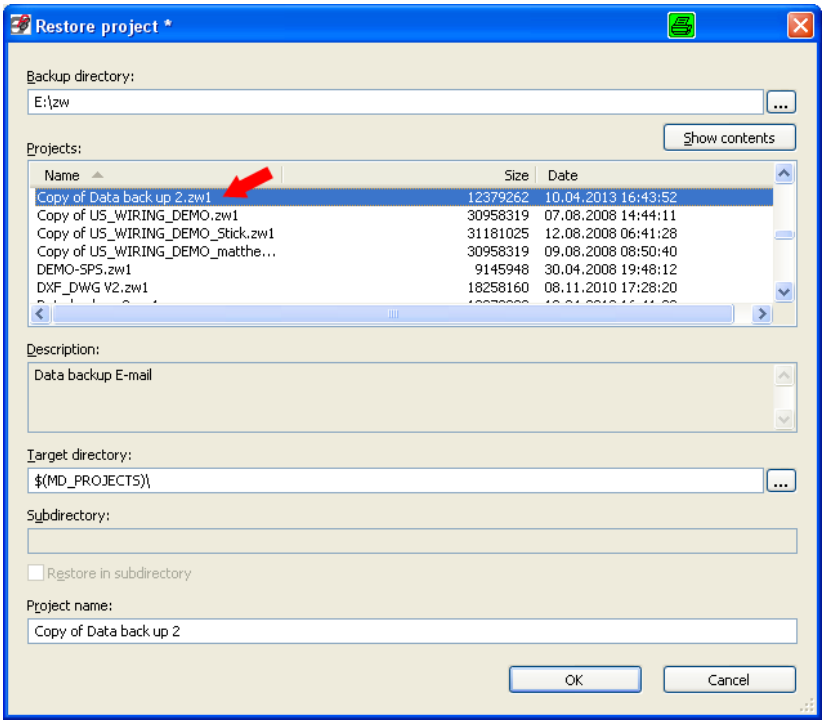


Fig. 9.17 Restore project dialog

Click **OK** or a double-click the project (in the *Projects* field) to restore the project using the set parameters. EPLAN then begins the restore process. No further query dialogs are displayed.

If a project of the same name already exists in the target directory, then EPLAN displays a warning message asking if the target project should be overwritten. If you click the **YES** button, EPLAN restores the project and overwrites the old project.

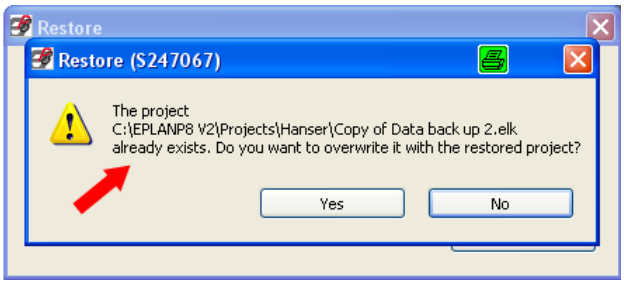


Fig. 9.18 Possible message if a project already exists

EPLAN can also overwrite opened projects if you wish. After a prompt, EPLAN closes the project and then restores it.

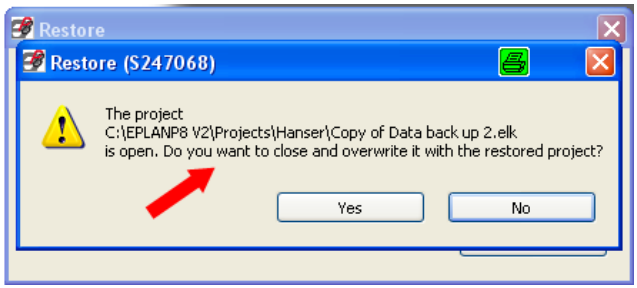


Fig. 9.19 Another message if a project already exists

Once the project has been fully restored, EPLAN opens it in the **page navigator**.

If you click the **NO** button, EPLAN cancels the restore process and displays a message stating that the project was not restored.

EPLAN writes the reason for cancelling in the system messages. The reason can be viewed by clicking the **SYSTEM MESSAGES...** link. EPLAN opens the **System messages** dialog. System messages can be saved or you can close the dialog. If you close the **System messages** dialog, you are returned to the previous dialog. Simply click **OK** to return to graphical editing.

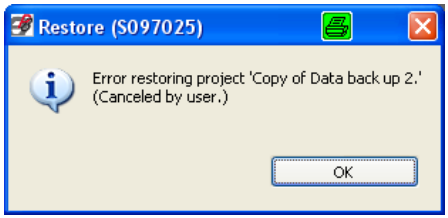


Fig. 9.20
Note displayed when canceled by user

9.3 Other important settings

There are also default settings related to the backing up and restoring of data, such as for defining backup directories or for cleaning projects of old data that is no longer required. It is also possible to automate the process of backing up data and cleaning projects.

9.3.1 Default settings for project backup (global user setting)

You can define global default settings for projects. These are then applied to the specific project. You can adjust these and other default settings in EPLAN via **OPTIONS/SETTINGS/USER/DATA BACKUP/DEFAULT SETTINGS/PROJECTS**. Here you can set the target directory, the split size of e-mail attachments and other options such as back up entire project,

save external documents and image files, and the default setting for the backup description field. These settings are then applied when a project is backed up.

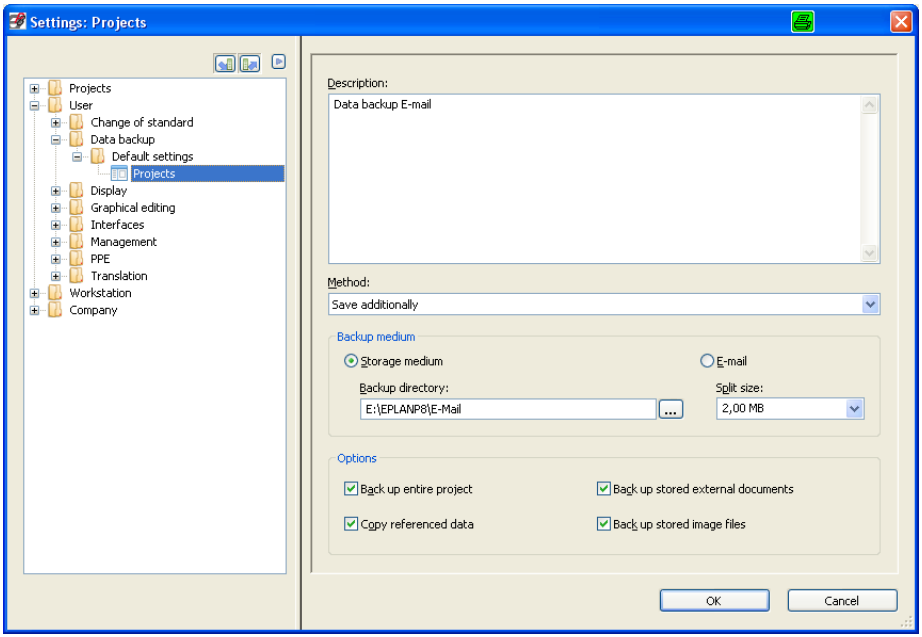


Fig. 9.21 Default settings for backing up projects

TIP: If you subsequently change these settings yourself when backing up a project, EPLAN remembers these settings precisely for this project. The global user settings then no longer apply. The updated data backup settings for a project always have priority over the data backup settings in the user settings and will always be used in the future for this project.

The backup directory and the split size of e-mail attachments can, of course, be changed later, also for projects that have already been backed up. The new settings are then applied again for the next backup process.

9.3.2 Compress project (remove unnecessary data)

While editing a project, it can become "filled" with a great deal of data. Unplaced devices may be inserted or forms (master data) may be created for test purposes only. Since EPLAN stores all data in the project automatically, even when used only once, a project may end up containing, for example, ten different terminal diagram forms or dozens of parts even though only one of these was actually used for reports at the end of project editing or the parts are no longer needed.

Compress project function

Data that has become unnecessary in the course of project editing can be removed in EPLAN via the *Compress project* function. You access this function via the **PROJECT/ORGANIZE/COMPRESSMENU.**

Fig. 9.22 Project/Organize/Compress menu

EPLAN then starts the **Compress project** dialog.

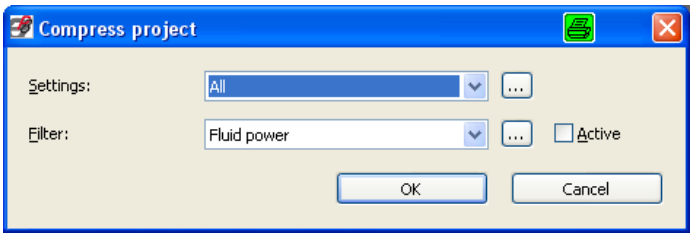


Fig. 9.23 Compress project dialog

The dialog offers several ways of affecting the data volume of the project.

You select the desired compression scheme to compress the data in the **Settings** selection field. EPLAN provides a number of default settings (finished schemes, which can be selected from a selection list), but here too, you can also define your own personal settings by creating and saving your own schemes. Click the **[...]** button next to the **Settings** selection field. The **Settings: Compression** dialog opens.

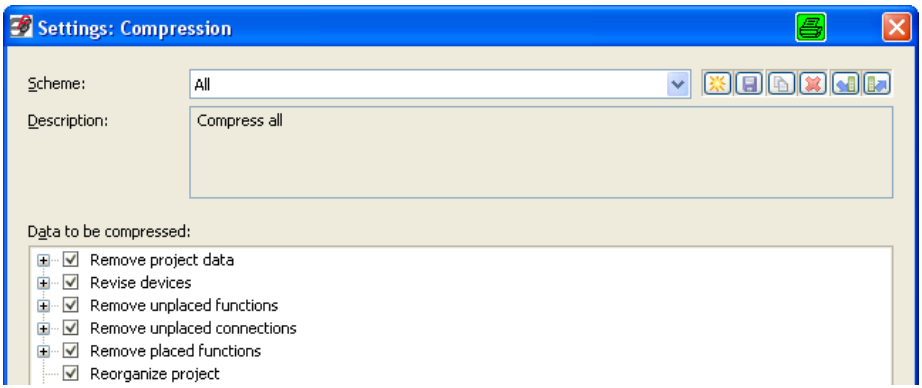


Fig. 9.24 Settings options for a compression scheme

Here, you can use the familiar graphical buttons to create a new scheme or copy and change an existing scheme.

In the lower **Data to be compressed** field, EPLAN offers a number of actions that can be performed during the compression.

One example is the item **Remove project data/Unused forms**. This setting removes all unused forms from the project and the project then contains only forms that are actually used.

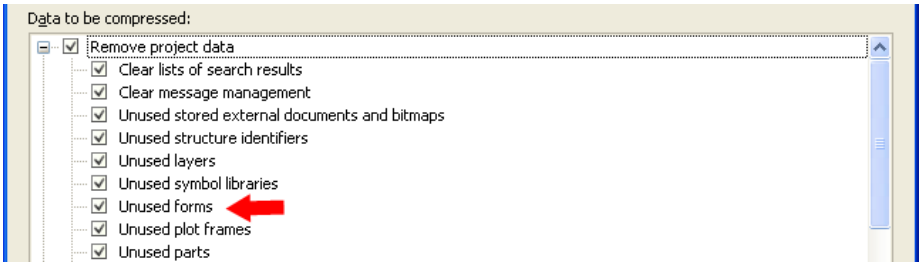


Fig. 9.25 Setting to remove unnecessary forms

NOTE: If, for example, forms documentation was created before the compression, all forms (including the unused ones) remain stored in the project because they are needed for the forms documentation. You should remove the reports here, especially the forms documentation, prior to compression.

The other, second, method is to use filters for the compression process. As usual, filters can be set and are either created or edited as schemes. In the **Filter** for compression you can, for example, exclude specific areas from the compression process, such as particular functions in the schematic or entire structure identifiers.

EPLAN thus provides various options for specifying what data is to be removed (using filters) and what areas or data the compression process should include.

9.3.3 Automated processing of a project

So far, we have looked only at individual steps. The **Organize/Compress** function was used to free the project of unnecessary ballast and the project was then backed up using the **Project/Back up/Projects** function.

The **Automated processing** function in EPLAN allows such actions to be combined together and automated. In this example, we will use the compress project and data backup actions.

The **Automated processing** function is accessed via the **UTILITIES/AUTOMATED PROCESSING** menu. It contains three menu items: **RUN ACTIVE SCRIPT**, **RUN** and **SETTINGS**.

Automated processing with scripts

Fig. 9.26 The Automated processing menu item and its options

The **SETTINGS** menu item opens the **Settings: Automated processing** dialog. As is customary in EPLAN, you can create and save new schemes in this dialog. All projects can use these schemes.

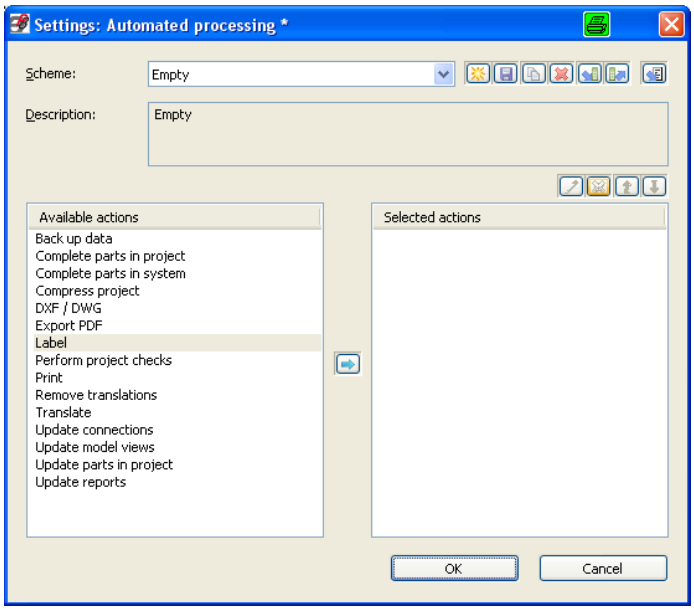


Fig. 9.27 The possible settings of automated processing

The **Automated processing** function can be used to export and import schemes. The schemes are in the *Scripts* directory (applies for a standard installation). The dialog contains the **Scheme** and **Description** selection fields.

You use the → button to add selected available actions to the **Selected actions** field. If further settings are available for this action, EPLAN opens the corresponding settings dialog for further settings for this action after you move it.

For the **Back up data** action, for example, the Back up data dialog will be opened. Here you can set special settings for this script.

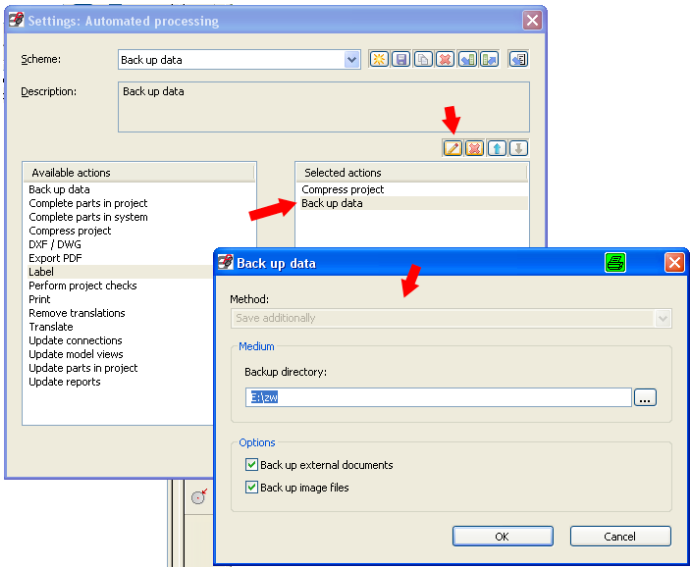


Fig. 9.28 Further dialog for data backup

If you now combine these settings, i. e. compress project followed by a data backup, then you can process it automatically.

Combining settings for scripts

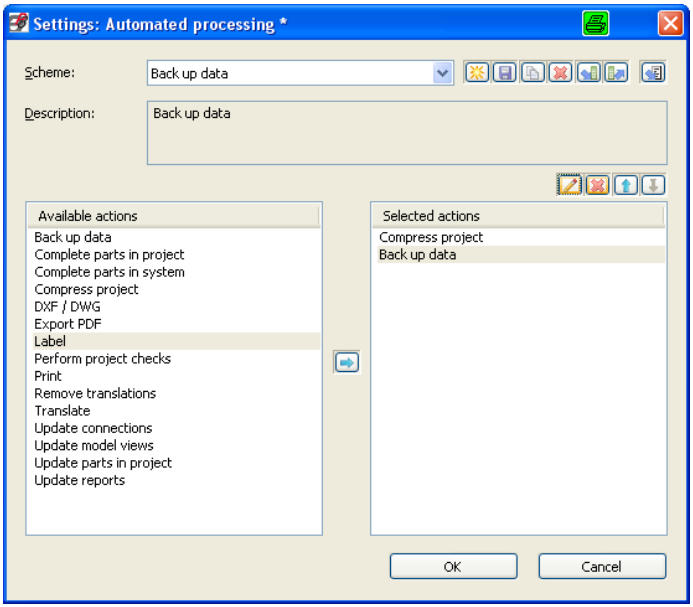


Fig. 9.29 Finished script with selected actions

This means that there are no more individual steps, you simply call up this one scheme and EPLAN fully automatically performs all the previously defined actions in a single step.

You can later edit the available actions via the top toolbar on the right. Select the corresponding action in the **Selected actions** field and open it with the *Edit* button.

NOTE: Created schemes are not stored on a project-specific basis. Using my customer code as an example, such a scheme would be located in the *\Bernd Gischel\Scripts* directory.

In general, the last edited scheme is stored in the settings and can be started via the **UTILITIES/AUTOMATED PROCESSING/RUN** menu item. But you can also select and run other schemes here.

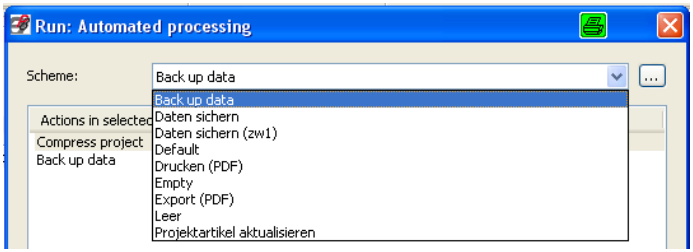


Fig. 9.30 Scheme selection

When the scheme is started, EPLAN executes the stored actions and displays the current progress in a dialog.

When the script (scheme) is finished, no other messages are displayed.

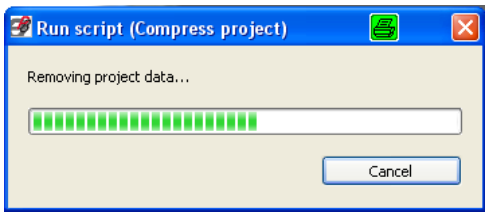


Fig. 9.31 Status of the script (scheme)

9.4 Backing up and restoring master data

As well as projects, you can also back up and restore master data in EPLAN. This is a great advantage because it provides a unified, simple method of exchanging data between EPLAN users. EPLAN uses file extensions (among other features) to distinguish between the different types of data backups.

File extensions of EPLAN backup files

- **zw1**: File extension for projects
- **zw2**: File extension for symbol libraries
- **zw3**: File extension for plot frames

- **zw4**: File extension for forms
- **zw5**: File extension for macros
- **zw6**: File extension for parts data
- **zw7**: File extension for dictionaries
- **zw8**: File extension for user and workstation data

9.4.1 Backing up master data

Backing up of master data generally always follows the same process. Master data is saved via the **PROJECT/BACKUP** menu item. During the backup process, you can select the master data type that should be saved.

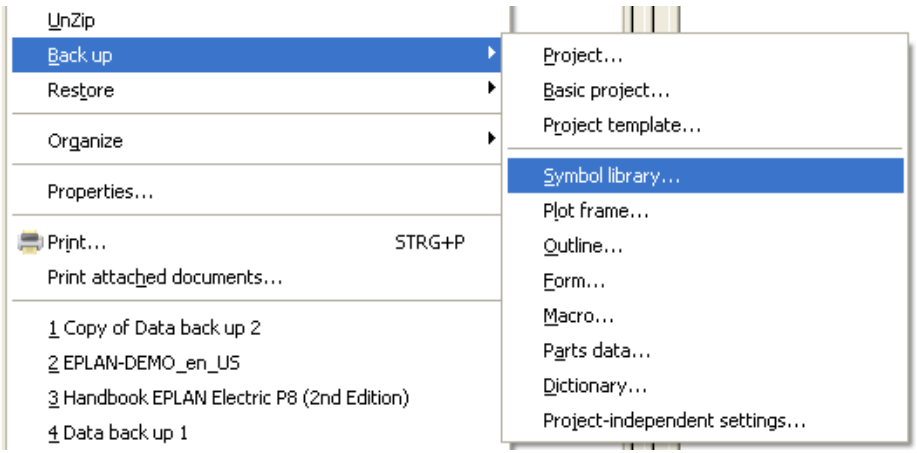


Fig. 9.32 Project/Backup/[Master data] menu

When master data is backed up via this menu item, EPLAN opens the corresponding **Back up [master data]** dialog, whereby **[master data]** represents the various types of master data such as symbol libraries, plot frames or the parts database. The title bar of the respective window also indicates which type of data backup dialog is currently being used.

The **Back up parts data** dialog here is used as a representative example, since all the dialogs have the same structure and are used in the same way.

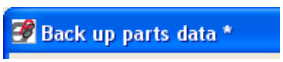


Fig. 9.33 Title bar

A data backup dialog for master data has a window with an entry indicating which master data is to be backed up. The example illustrates backing up of parts data.

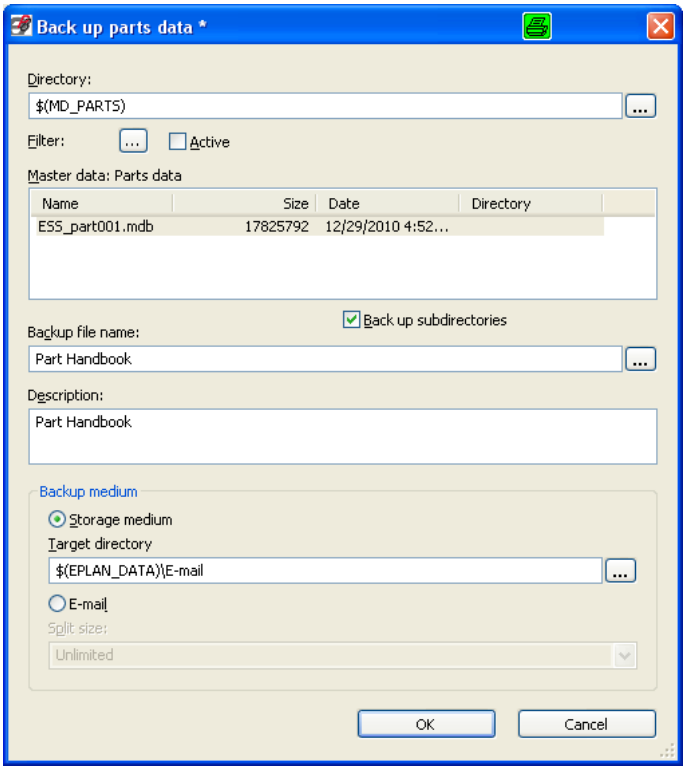


Fig. 9.34 Dialog as an example of how to back up master data

- **Directory:** A selection field defining the directory containing the master data to be backed up.
- **Backup file name:** An important entry since a backup cannot be performed without a name for the backup file. EPLAN uses this name as the file name for the backup file and appends the appropriate file extension.
- **Description field:** This field can (but does not have to) contain a description of the current data backup. There are no limits to the contents of this field. This description is later displayed in the *Description* field when restoring from the backup.

The fields/options such as **Storage medium** or **E-mail** have already been sufficiently described in the section on data backups for projects and will not be further discussed here.

When all settings are correct, you start the data backup process by clicking **OK**. EPLAN begins to back up the data and displays a final message after a successful backup.

This completes the backing up of master data.

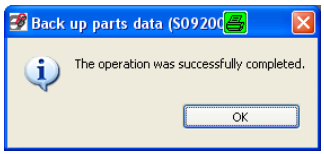


Fig. 9.35
Message that the data has been successfully backed up

9.4.2 Restoring master data

You use the **PROJECTS/RESTORE [MASTER DATA]** menu to restore master data. As described in Chapter 8, "Export, Import, Print", examples of master data are plot frames, forms, parts data and symbol libraries.

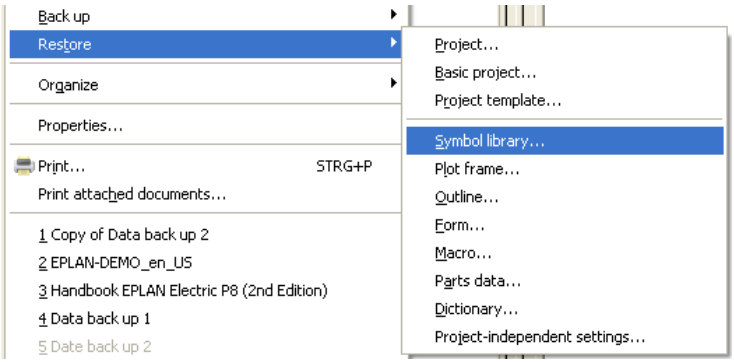


Fig. 9.36 Project/Restore menu

If you wish to restore this data in EPLAN, you use the **PROJECT/RESTORE [MASTER DATA]** menu to open that dialog.

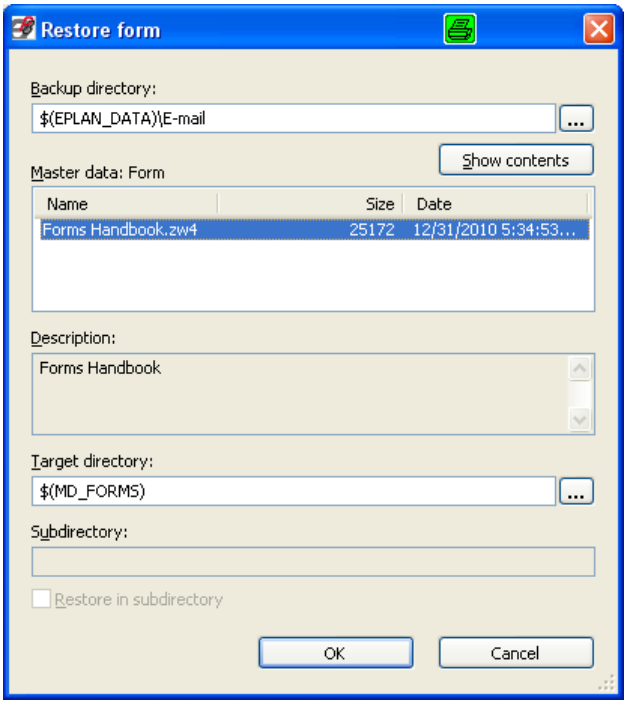


Fig. 9.37 Example of a Restore master data dialog

Again, EPLAN has standard dialogs for various types of master data, so that the dialog shown here – to restore master data – is identical to that of all other types of master data.

You set the **Backup directory** to the directory containing the data you wish to restore. You can use the ... button to select the appropriate storage location if this differs from the default storage folder.

Once the correct folder is selected, the **Master data: [Master data type]** field lists the available backups (files).

NOTE: You can only select one file here. Multiple selection is not possible.

The **Description** field displays the information entered when the master data was backed up. This field is purely informative.

To restore the master data to a different specific directory, you need to set up a suitable folder that differs from the default directory in the **Target directory** field. The default directory is the original source directory of the master data, i.e. the location where the backup was performed.

Once all entries match your requirements, you start to restore master data by clicking **OK**.

EPLAN restores the selected master data to the specified directory and then displays a message with information about the restore process.

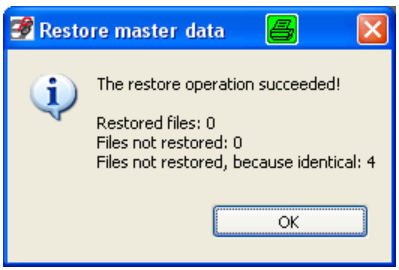


Fig. 9.38 Message that the data has been successfully backed up

9.5 Send project by e-mail directly

In addition to sending an e-mail as part of a normal backup process, in EPLAN it also possible to send a project directly via e-mail. You can access this function directly via the **PROJECT/SEND BY E-MAIL** menu item.

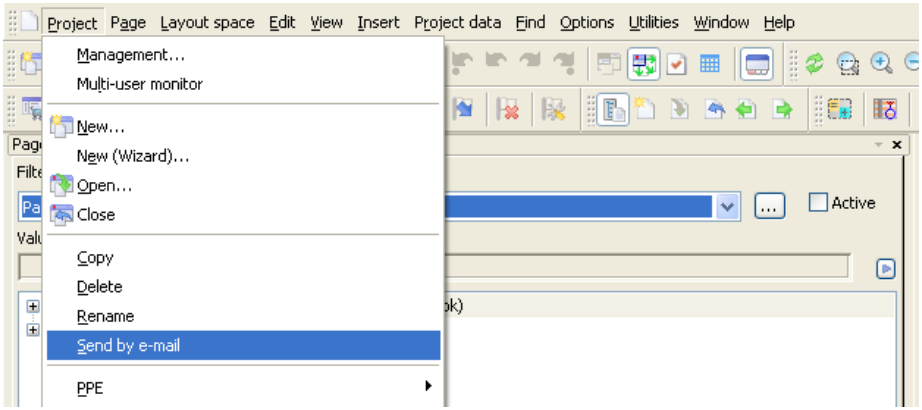


Fig. 9.39 Send project by e-mail directly

You select the desired project in the **page navigator** and then call up the **Send by e-mail** function. EPLAN displays a prompt.

Then, EPLAN handles the default settings for sending by e-mail (e. g. the stated size of split files) and begins to generate the backup files.

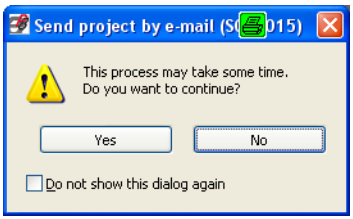


Fig. 9.40
First note regarding e-mail sending

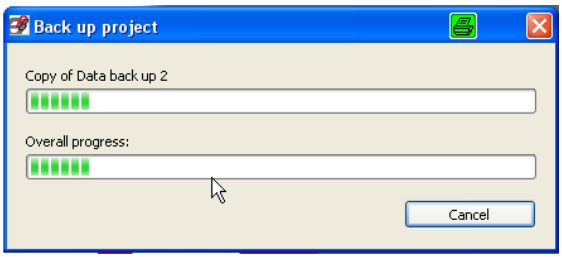


Fig. 9.41 Progress dialog for backing up project by e-mail

Depending on the size of the files, EPLAN now generates one or several e-mails with the split project files and then waits for the user to send these e-mails.

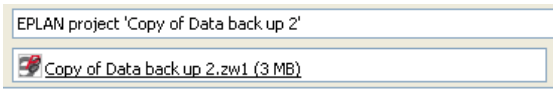


Fig. 9.42 Newly generated e-mail with the split project files

When all e-mails are sent, a message stating that they were successfully sent is displayed. The complete project has now been sent.

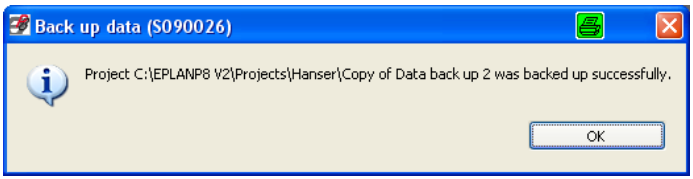


Fig. 9.43 Message that e-mails have been successfully sent

NOTE: You can send only one project at a time by e-mail from the **page navigator**. Multiple selection, i.e. sending several projects by e-mail at once, is not possible in the **page navigator**.

10 Master data editors

EPLAN supplies a large number of finished system master data for forms, plot frames and also symbols. These are normally sufficient for most applications. However, if special conditions for appearance exist or if particular project or device data is to appear in the forms and this is not possible with the standard EPLAN reports (forms) supplied, then some new system master data such as reports (forms) will need to be created from scratch.

NOTE: In EPLAN, forms are usually used for reports. This is why we can speak of reports or forms. But not all reports are forms. A report can also be an Excel table that was created with the *Labeling* function.

Master data menu

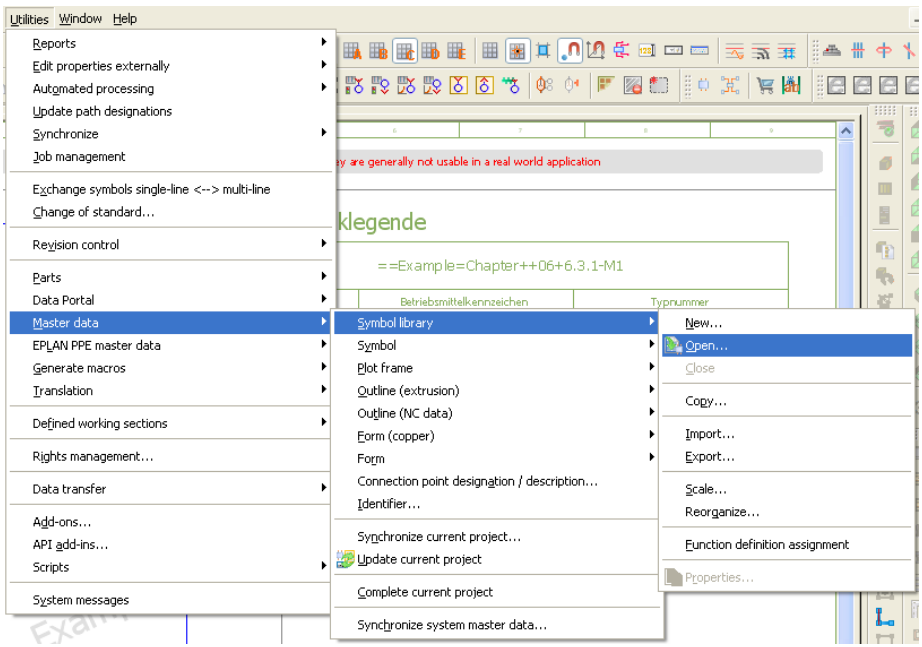


Fig. 10.1 The master data menu with the possible editors

NOTE: EPLAN always opens master data for editing on a temporary basis in the current project. It doesn't matter which project, but a project has to be open.

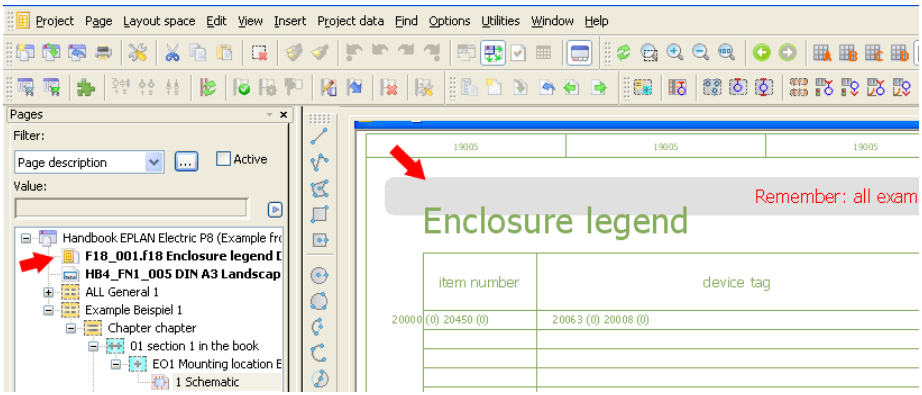


Fig. 10.2 Example representation for master data opened for editing

Generally, it is always system master data that is edited. Master data editors never modify project master data. It is the form or plot frame in the system master data directories that is modified and/or created and then saved in the system master data directories.

Fig. 10.3 Prompt when stored master data have been modified

If modified master data is to be used in the current project, then after it is modified the system master data must be synchronized with the project master data, i. e. the automatic prompt whether data is to be synchronized must be answered with **YES**. Otherwise, the modifications to the master data will not be used in the current project, because it has not been updated.

You access the various editors via the menu **UTILITIES/MASTER DATA/[TYPE OF MASTER DATA]. [TYPE OF MASTER DATA]** can be: symbol libraries in general, symbols, plot frames or forms.

Note: Creating outlines and forms will not be covered in this version of the manual.

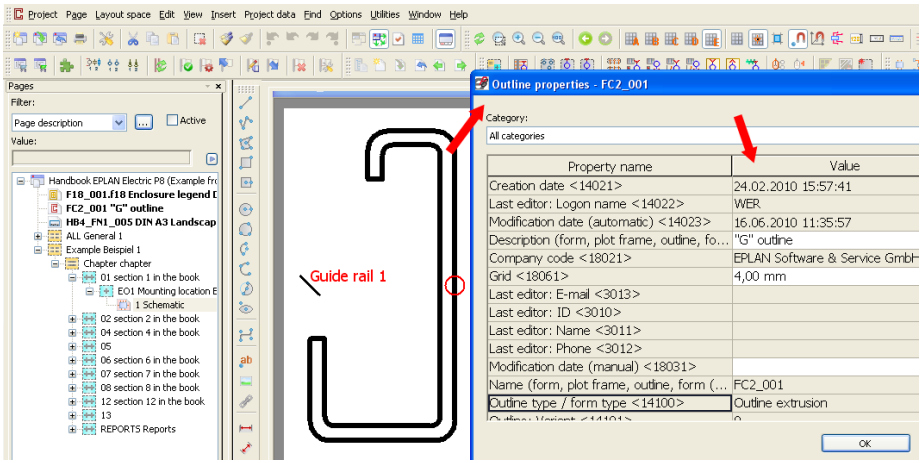


Fig. 10.4 Example of a finished outline (created by EPLAN)

NOTE: Before you start editing master data, you should keep the following in mind: Original master data supplied with EPLAN should never be modified. It is possible to modify this data, but if EPLAN master data is a good basis for your own master data then I recommended that you copy it and assign a new name. You can then modify this copy as you wish.

If you follow this recommendation, then, when performing updates etc., the new, possibly modified original EPLAN master data in your system data directories can simply be replaced with the EPLAN master data of the same name. This means that you can benefit from improvements or extended functions immediately, and you will not have to integrate them into your own master data.

TIP: It is fastest and easiest to use existing master data as a basis and then edit it. You can create the data from scratch. But this will take more time and requires a more thorough understanding of the master data editor.

This chapter provides a brief explanation of the basic function steps required for editing and understanding. Not all the details and possibilities of the form editor, plot frame editor, symbol editor or symbol library editor can be explained here. These editors are simply too complex for this because there is a (practically) infinite number of options for implementing various requirements in connection with the master data.

10.1 Preparatory measures

Although it is convenient to edit a project's system master data, this can also be "dangerous". The more you edit, the greater the "danger" of losing track of what you have done: What forms/plot frames have I already created, what do they look like, and which ones can I use as templates for a new form/plot frame?

The upshot is that you might have to click through many forms/plot frames, and you might not be able to see all the information you need on the small preview. You load the form, only to discover that it is not the one you used last week in another project.

TIP: If a multiple screen solution is used, it is best to enable the separate graphical preview. The window size of the preview can be modified, and you can view the forms/plot frames in a convenient size. The graphical preview is activated in the **VIEW/GRAPHICAL PREVIEW** menu.

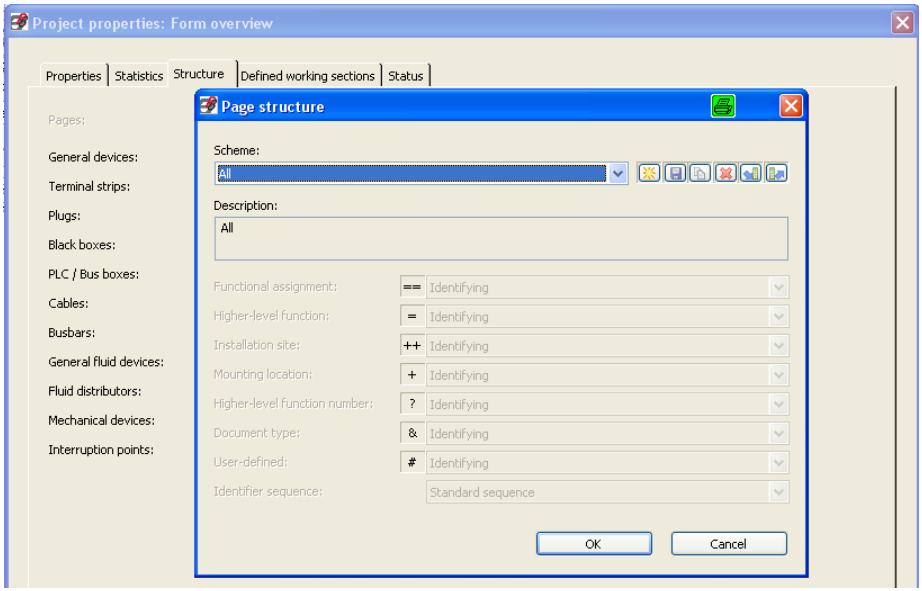
10.2 Overview of forms

To help prevent the onset of "master data chaos", you have various options at your disposal to display forms/plot frames in a structured manner.

10.2.1 First option – manual overview

A form overview would be a type of "form project", which is created and maintained manually. But it is really nothing other than a normal schematic project. Depending on your personal wishes and organization, you can store all forms and plot frames as extra pages in this project. This means all forms/plot frames are gathered in a project and created as separate pages, and the corresponding form or plot frame is assigned to such a page.

The selected page structure covers all identifiers, which allows deep structuring of the pages by customer and page type (in this case report type). However, this is a matter of personal taste and users can decide for themselves.



Possible structure of a project providing an overview of all forms and plot frames

Fig. 10.5 A possible page structure

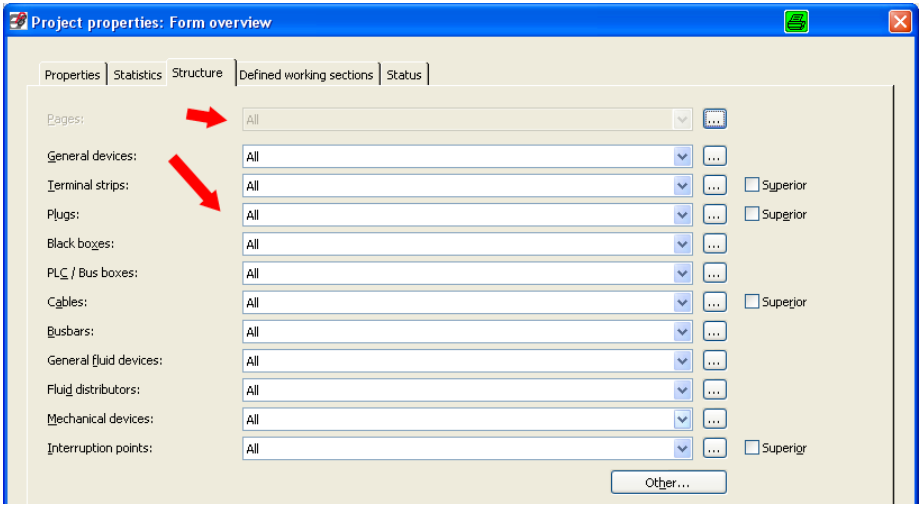


Fig. 10.6 Structuring of the remaining objects

Of course, you cannot directly edit the form or plot frame on these pages. We do not want to do this anyway. When the project is always maintained with the master data, you have a good overview of which forms/plot frames have been created over time, and, if needed, you can select a particular form or plot frame for subsequent editing.

Possible layout of the page structure with the various types of identifiers

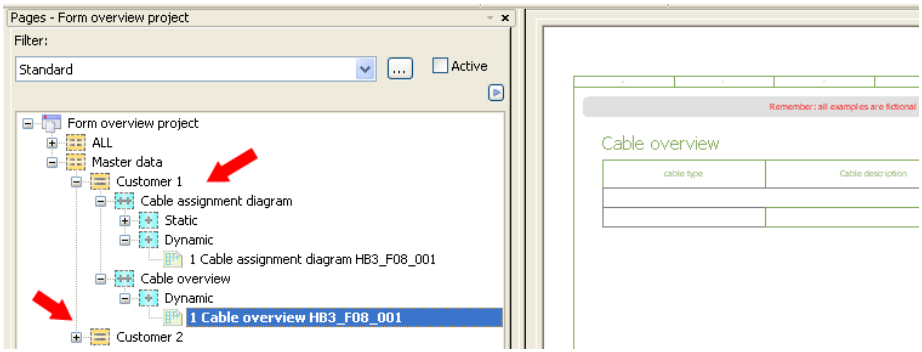


Fig. 10.7 Possible layout of the page structure with the various types of identifiers

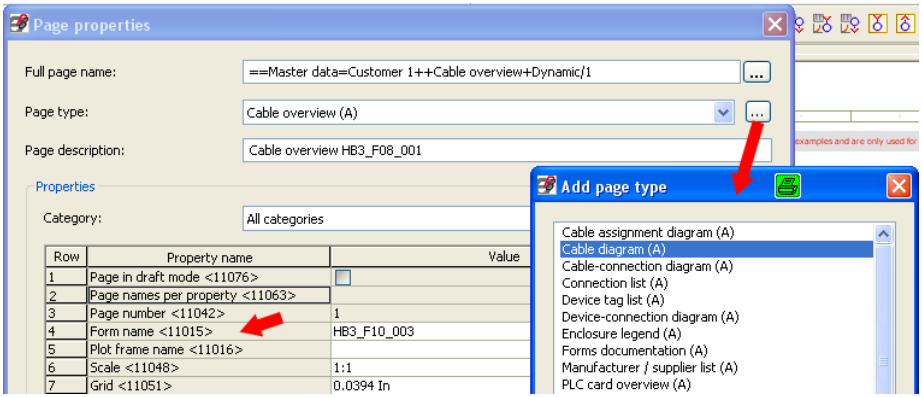


Fig. 10.8 Page properties of a page in the project

Pages are created via the usual steps. The relevant page types (e. g. here a cable diagram) are set for the pages, the page descriptions are filled, and the form is selected via the selection in the *<11015 Form name>* property. This stores it in the page and displays it.

10.2.2 Second option – automatic overview

A second option for documenting forms and plot frames is to use EPLAN's own media to create an overview.

In this case you also create a new project and store all master data (forms, plot frames) in the project from the corresponding customer system master data directory. The easiest way to do this is via Synchronize master data (**UTILITIES/MASTER DATA/SYNCHRONIZE CURRENT PROJECT** menu).

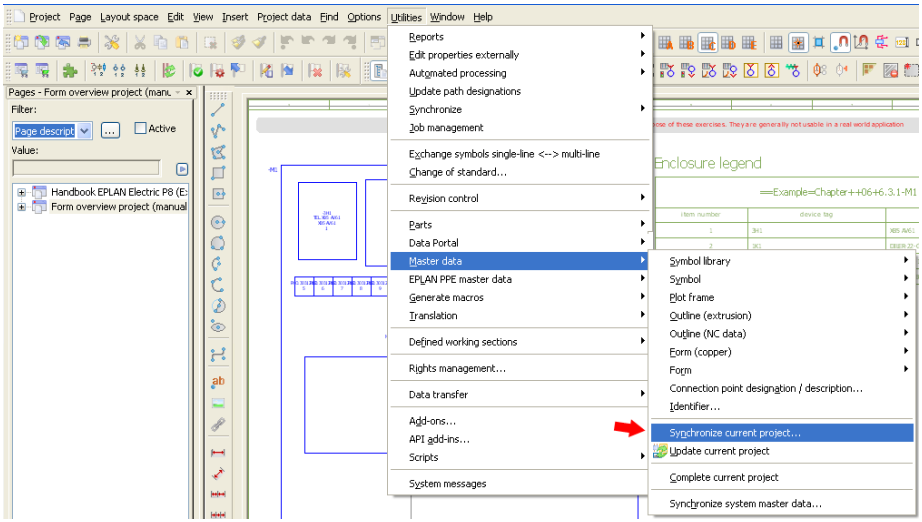


Fig. 10.9 Synchronize master data menu for a project

In the subsequent **Synchronize master data** dialog, you can then copy to the project all (non-)existing system master data.

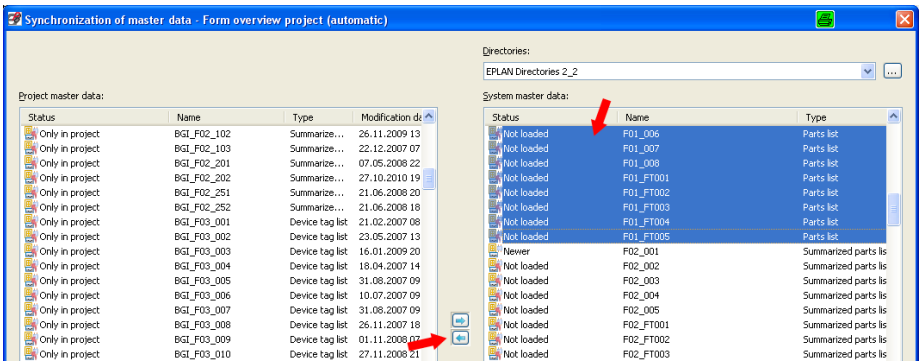


Fig. 10.10 Synchronize master data dialog

If all master data like forms and plot frames have been copied to the project, you can then generate the *Forms documentation* and *Plot frame documentation* reports. EPLAN automatically generates the corresponding project overview for all forms and plot frames in the project. You can also use this as an overview of the existing master data in the system directories.

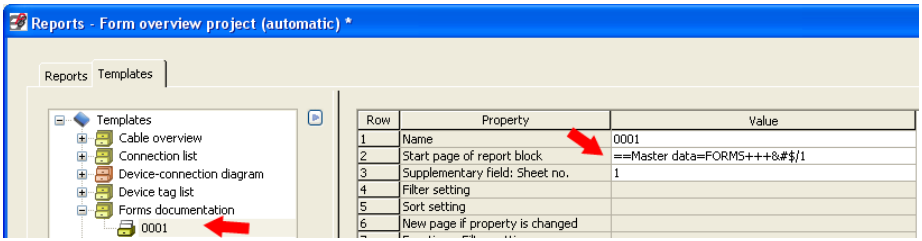


Fig. 10.11 Templates for the forms and plot frame documentation reports

A possible final result is shown in Fig. 10.12.

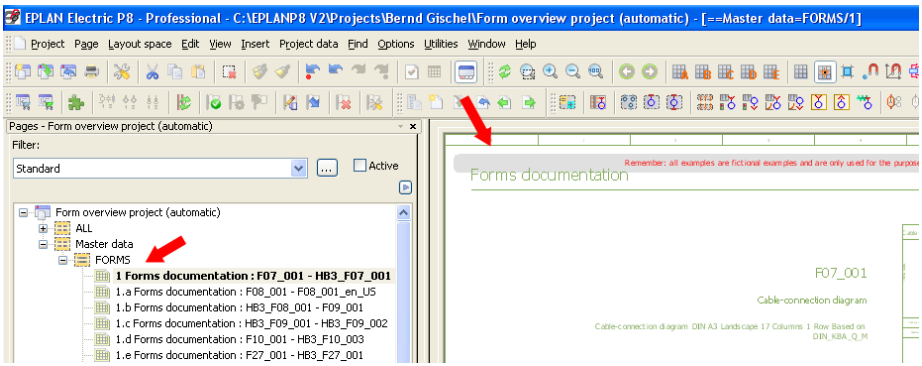


Fig. 10.12 Page navigator view

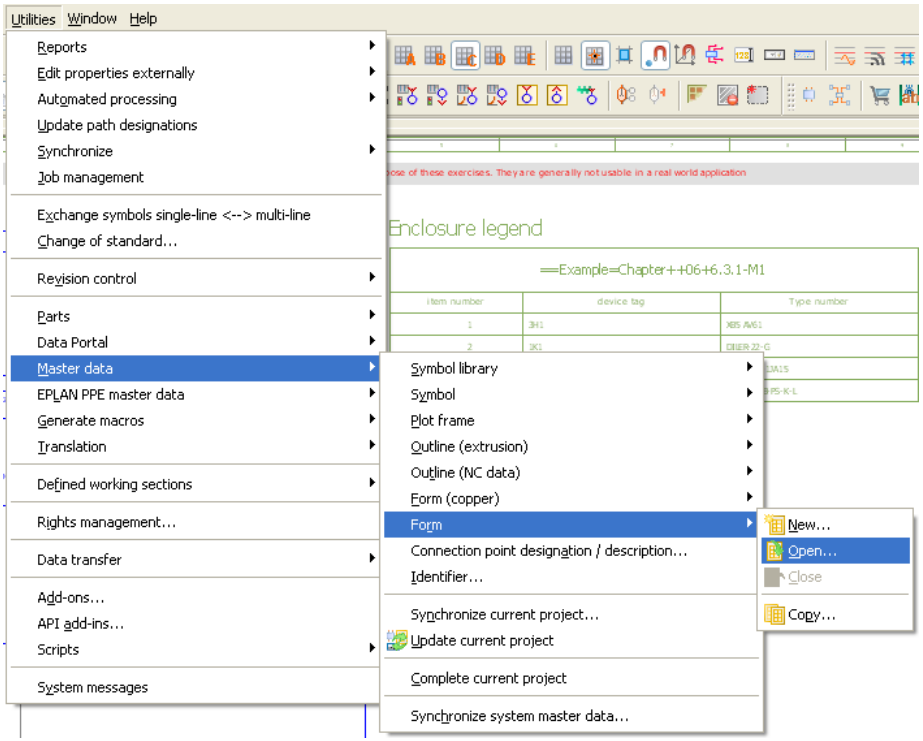
Unfortunately, there are no other sorting options for forms documentation or plot frame documentation. But it is sufficient for a quick overview.

The great advantage of automatic generation of forms documentation or plot frame documentation is clearly that the relevant pages are generated automatically by EPLAN, with literally with just a “click”. You can’t get any faster than that!

10.3 Forms

Forms are used to graphically insert evaluated project data into the project documentation. Examples of forms are terminal diagrams, cable diagrams, device tag lists or a table of contents.

The supplied master data does not always exactly match the existing project data. This may be due to page references that are too long, function texts that do not fit in the intended rows and columns, or the need to “spruce up” the display of PE terminals with a small graphical symbol. Consequently, reports will need to be adjusted or created from scratch.



Menu for editing forms

Fig. 10.13
Form editor menu

EPLAN provides a number of different ways of adjusting a form to the new layout. Forms can be created from scratch, edited (i.e. opening and changing existing forms) or copied and then modified.

The form structure can be accessed via the **UTILITIES/MASTER DATA/FORM** menu.

NOTE: The form editor does not have a *Save* function. If changes are made and the form editor is exited via the **CLOSE** function in the **UTILITIES/MASTER DATA/FORM** menu, then the form is automatically stored with these changes in the system master data.

The project master data, however, still contains the previous "old" form, which you could, if necessary, copy back to the system master data directory. This means that, as long as no (manual) synchronization of the master data (system master data with the project master data) is performed (e.g. via the **UTILITIES/MASTER DATA/SYNCHRONIZE CURRENT PROJECT** menu), then this method can be used to return the edited form in the system master data back to the state of the project master data.

After starting the function, EPLAN opens the **SYNCHRONIZE MASTER DATA [PROJECT NAME]** dialog. You now select the form in the project master data area and use the button to copy it into the system master data area.

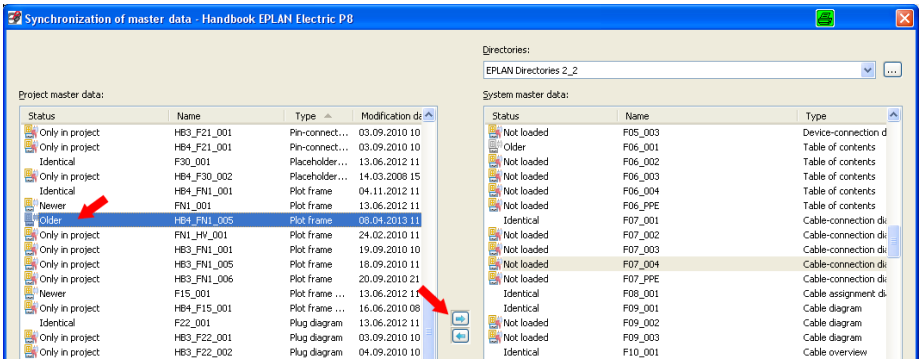


Fig. 10.14 Synchronize master data dialog

When existing master data is found, EPLAN opens the **COMPARE MASTER DATA** dialog. A decision must be made here as to whether the system master data should be replaced with the project master data.

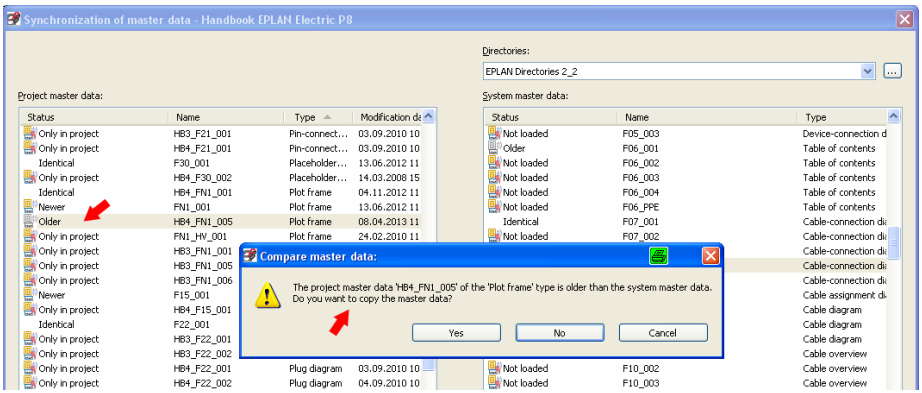


Fig. 10.15 Prompt asking whether the system master data should be overwritten

If you confirm the dialog with **YES**, the system master data is overwritten with the project master data and EPLAN displays a notification message that the system master data was updated.

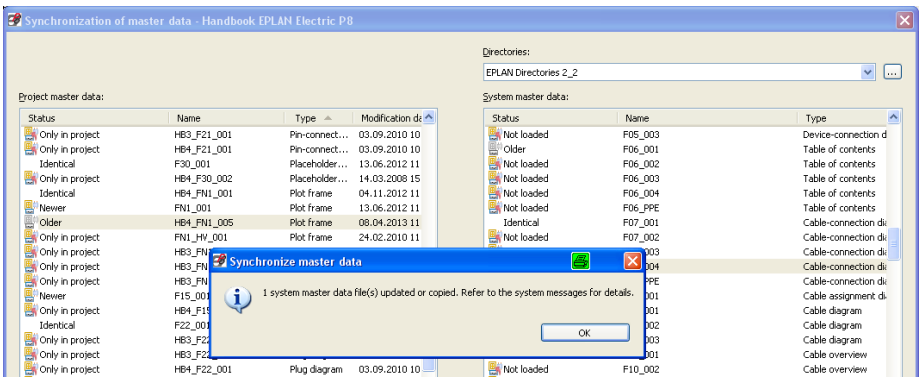


Fig. 10.16 Result message in the Synchronize master data dialog

NOTE: This method described is really only an "emergency plan" for restoring master data that was unintentionally changed. As already mentioned, EPLAN allows automatic synchronization of master data when projects are opened.

If master data is incorrectly reset to an old state, unwanted master data might be synchronized in existing projects (if the automatic master data synchronization has been activated). You should therefore be extremely careful when using this feature.

10.3.1 Create new form (from copy)

The easiest method is to create a form is based on an existing one. It makes no difference what type of page is currently open in the project. You call up the **UTILITIES/MASTER DATA/FORM/COPY** menu and EPLAN opens the **Selection/Copy form** dialog.

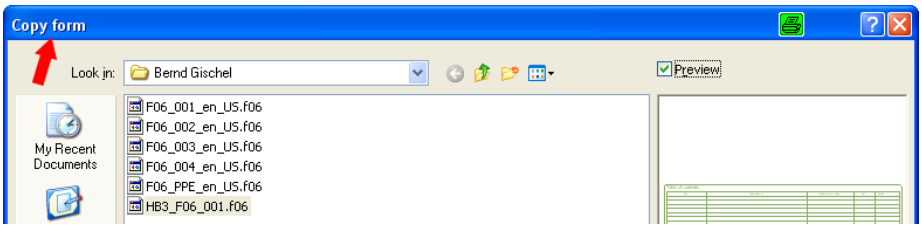


Fig. 10.17 Copy form dialog

In this dialog, you select the form to be copied. Confirm the selection by clicking the **OPEN** button in the lower area.

EPLAN then opens the **Create form** dialog. Here you enter the new form name in the **File name** field.

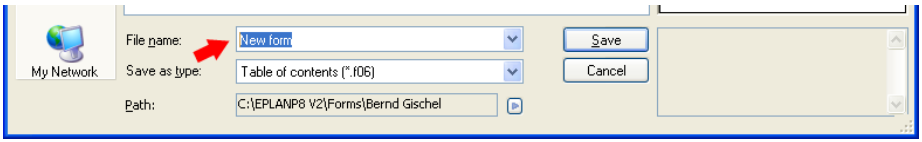


Fig. 10.18 Entry in the File name field

The form can now be saved via the **SAVE** button. EPLAN closes the dialog, opens a temporary page with a **cable diagram page type** and displays the copied form on this page for editing.

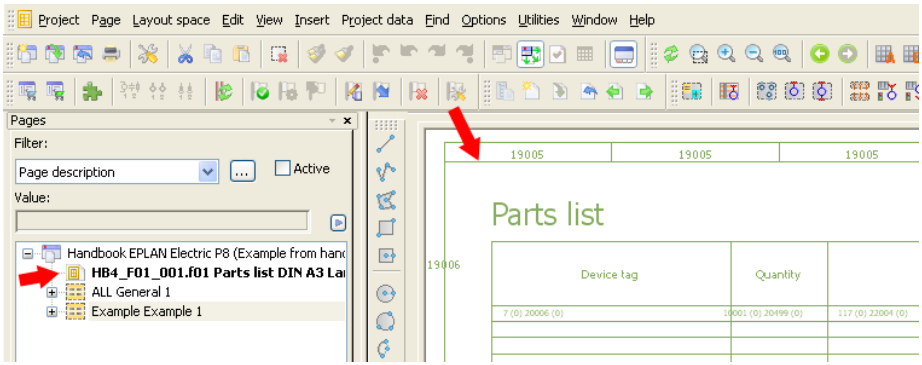


Fig. 10.19 Temporary page with the form in the page navigator

The form can now be edited. In the form editor, all graphical functions such as **Insert line** or **Insert circle** (via the **INSERT/GRAPHIC** menu) are available, as well as functions for inserting normal text.

The **INSERT** menu contains a number of other functions relating only to the form editor. The graphical functions have already been explained or are well-known functions in the graphical editor.

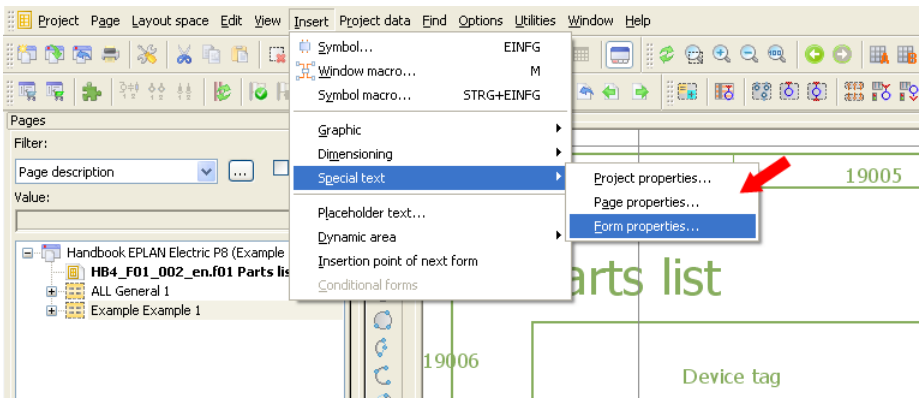


Fig. 10.20 Special texts

Special texts such as project, page and form properties can be placed in forms. The actual form texts (data for reports) are, however, called “placeholder text” in EPLAN. In contrast to the project and page-specific texts or form properties, these placeholder texts are only filled with project data after the reports are graphically generated.

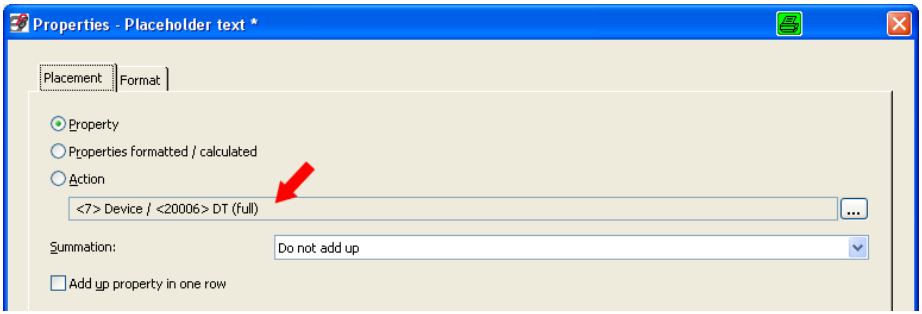


Fig. 10.21 Display placeholder text dialog

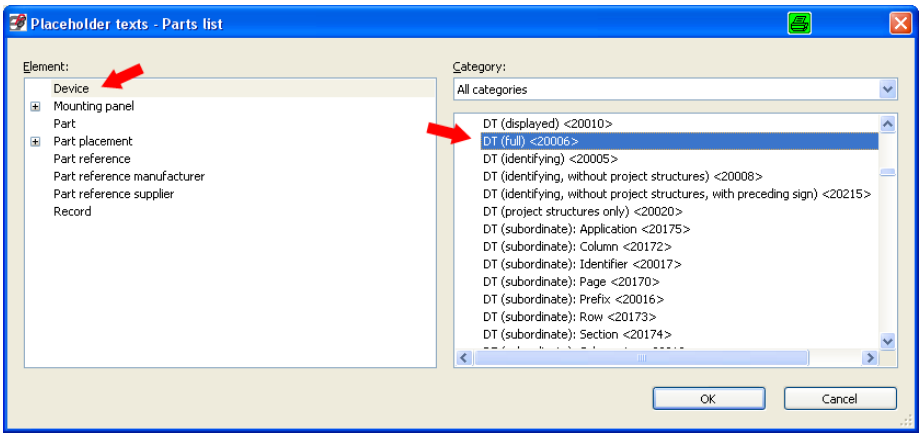


Fig. 10.22 Select placeholder text dialog

Insert placeholder texts

Open the INSERT/PLACEHOLDER TEXT menu, and in the **Placeholder text** dialog on the **Placement** tab, use the ... button in the subsequent **Placeholder texts – [Report type]** dialog to select a placeholder text.

The report shown here is an example – other placeholder texts apply to all other forms. In our example, the element in the left area is preselected (left selection area in the dialog), as is the desired property that is required in the right area of the dialog. You can now confirm this selection by clicking **OK**.

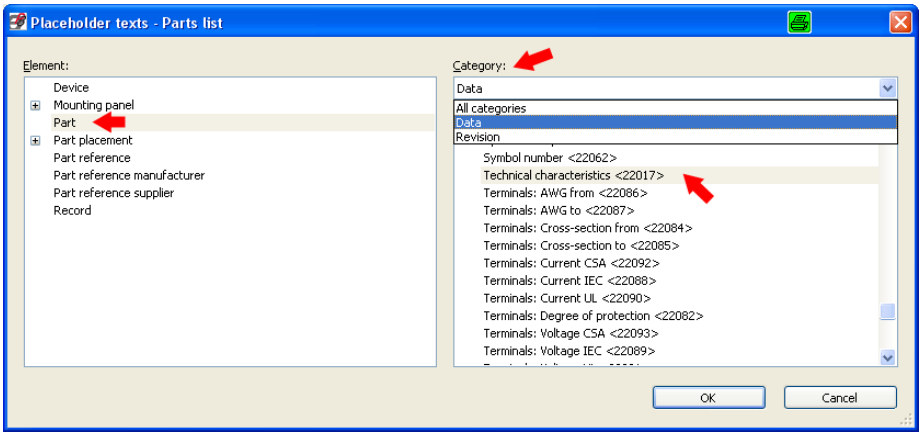


Fig. 10.23 Preselection (left) and selection (right) of the property

EPLAN closes the dialog and transfers the selected placeholder text to the Placeholder text dialog.

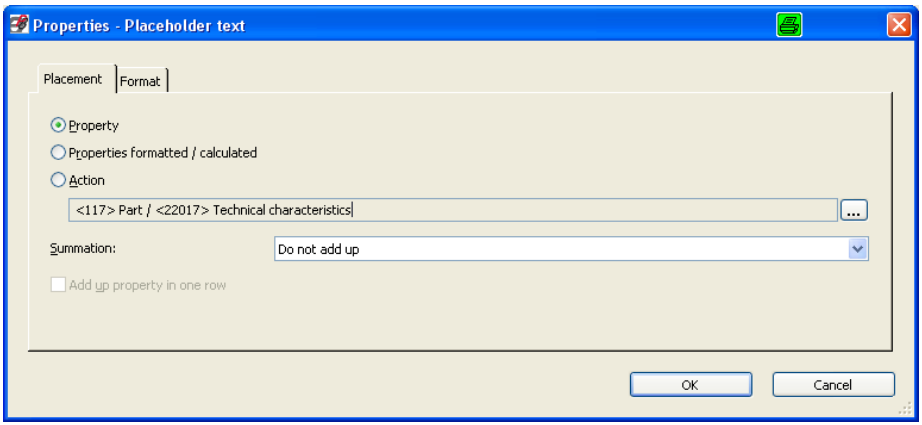


Fig. 10.24 Applied placeholder text

Confirm the **Placeholder text** dialog with **OK**. The placeholder text now hangs on the cursor and can be placed at any (practical) position in the form.

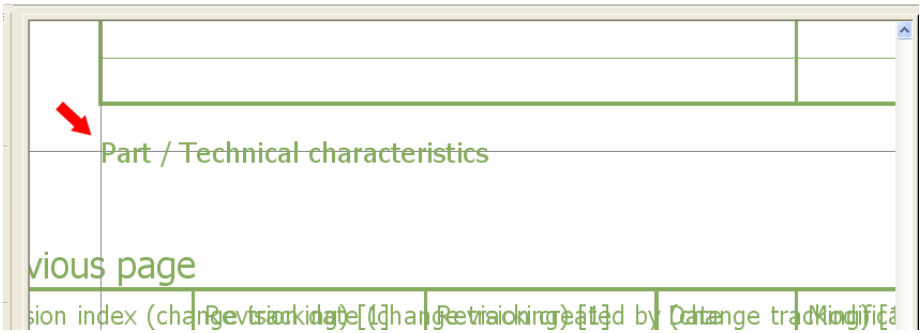
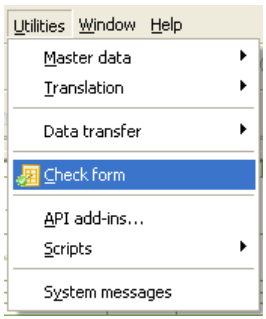


Fig. 10.25 Choice of placement

As with any other (free) text, placeholder text can be freely formatted. This allows the form to be appropriately "constructed". It is also possible to subsequently modify placeholder text, i. e. select a different one. You select the placeholder text and display its **properties** with a double-click or by clicking the popup menu.

You then proceed as described above. Use the ... button to call up the subsequent dialogs and select, apply and place other properties in the usual manner.

When you are done, the form can be checked. Using the function in the **UTILITIES/CHECK FORM** menu, EPLAN can check the form for errors. If the form is OK, the following message is displayed: "The form is OK."



Check form function

Fig. 10.26 Check form

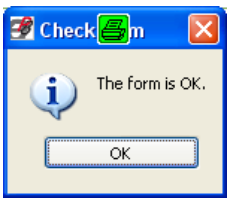


Fig. 10.27 Message that the form is OK

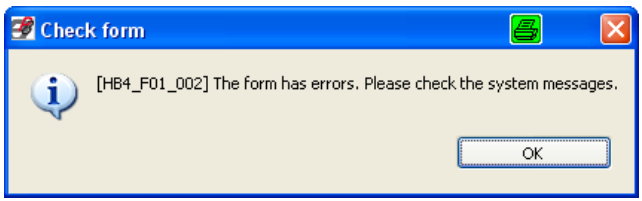


Fig. 10.28 Error in the form

If the form has errors, EPLAN indicates this and writes the errors to the system messages. These should be corrected to avoid incorrect system master data.

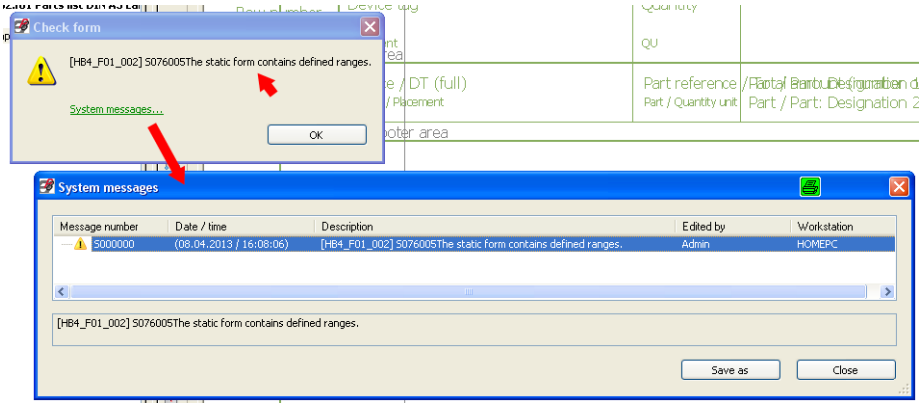


Fig. 10.29 System messages on incorrect form

Dynamic or static form handling

When handling forms, EPLAN distinguishes between dynamic forms, where only certain areas exist (e.g. a data area that contains an X amount of data), and static forms, where the amount of data is permanently "wired" into the form.

Dynamic forms expand their graphical display area depending on the amount of project data. The maximum amount of data that a dynamic form should display is defined in the form properties. In our example (the procedure is the same for all forms), you can access the form properties via the key combination **CTRL + M + D** or in the page navigator via the **PROPERTIES** function in the right-click popup menu.

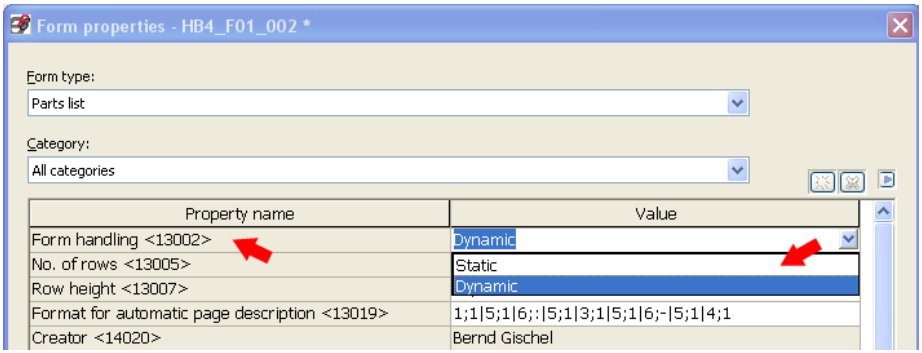


Fig. 10.30
Form properties dialog

Form properties

EPLAN opens the **Form properties** dialog. All relevant form data is defined here. How should the form be handled (dynamically or statically), which entries in the *automatic page description* property should be written when project data is evaluated, or what is the maximum amount of data (number of rows) that the form should display before a page break or new report page is created?

An example of a dynamic form is shown in Fig. 10.31. The rows are dynamically extended depending on the volume of data. If a terminal strip has five terminals, then the form will contain five rows with data. A terminal strip with twelve terminals will show twelve (data) rows in a report.

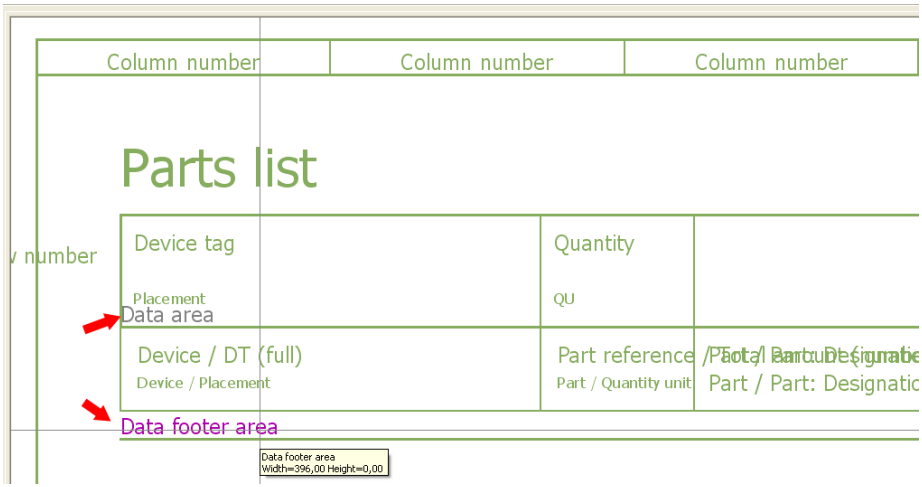


Fig. 10.31
Example of a dynamic form

In principle, static forms have the same properties as dynamic forms. In static forms, the data areas are fixed, i. e. permanently defined, and the maximum number of data rows is entered in the form properties – in the same way that the graphic for the form was created with the maximum number of data records.

An example of a static form is shown in Fig. 10.32. The difference can be clearly seen. Whereas a dynamic form has a data row that is dynamically expanded, a static form already has all data rows as a finished graphic. The data is written to these graphical rows when generating a report.

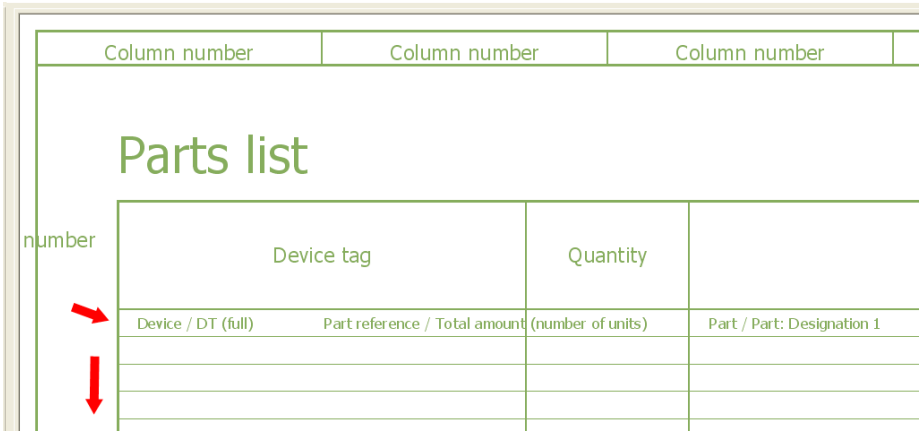


Fig. 10.32 Example of a static form

If the form is OK, then you can exit the form via the **UTILITIES/MASTER DATA/FORM/CLOSE** menu. It is then permanently "saved".

NOTE: After the very first editing, a copied form is automatically stored in the system master data and (when used) also stored in the project master data. At this point, synchronization of the master data (system and project) does not need to be performed.

When the form is edited a second time, then it is only updated in the system master data. Master data synchronization must now be performed and confirmed in order to update/synchronize the project master data with the system master data.

10.3.2 Edit existing form

An existing form is selected for editing via the **UTILITIES/MASTER DATA/FORM/OPEN** menu in the **Open form** dialog. EPLAN then opens a temporary page containing the form that was selected for editing.

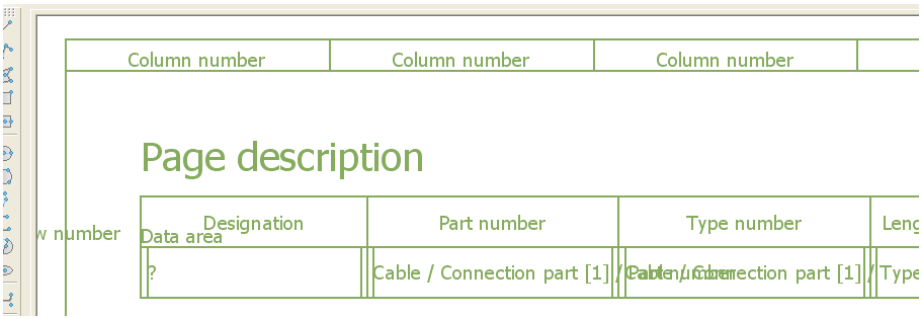


Fig. 10.33 Example: Form is open for editing

NOTE: Editing of the form is not explained here because the general procedure for editing an open form is the same as the procedure described in the last section 10.3.1.

10.3.3 Create new form

You create a new form via the **UTILITIES/MASTER DATA/FORM/NEW** menu. EPLAN then opens the **Create form** dialog. The new **Form name** must be entered in the **File name** field.

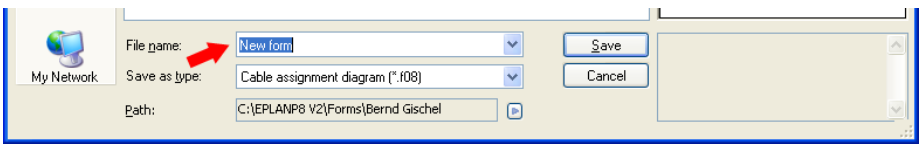


Fig. 10.34 Entering the form name

The **Create form** dialog can now be exited via the **SAVE** button. EPLAN opens, in the now familiar manner, a temporary page in the background and the **Form properties** dialog is displayed in the foreground. You can already begin making the first entries here. However, this is not absolutely necessary at this time and you can enter this data later.

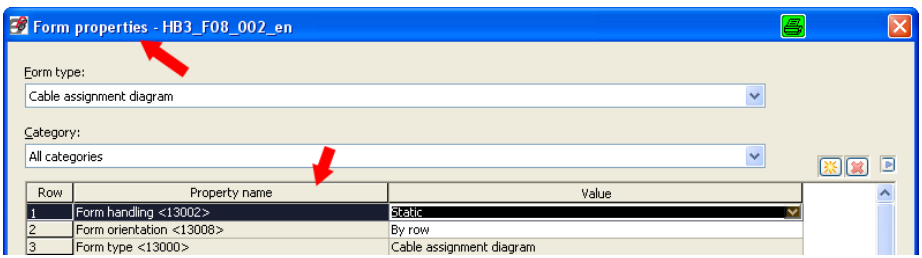


Fig. 10.35 Form properties dialog

As described in the previous sections, form editing is done using the various different functions, such as inserting graphical elements or texts.

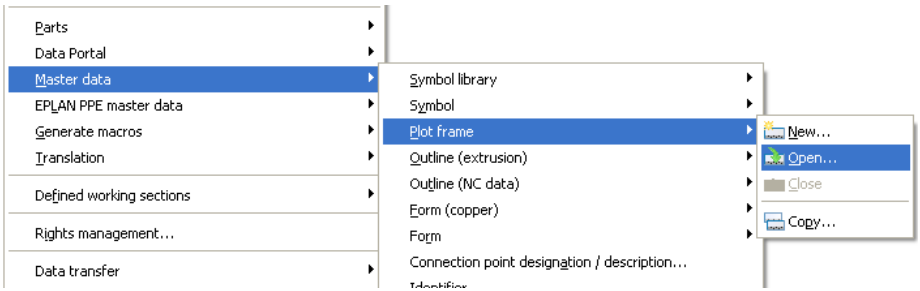
Creating a new form can be a time-consuming process because you start with "blank" form. It is therefore recommended that you copy a similar form and then modify this copy.

NOTE: The procedure for creating and editing a new form is identical to the procedure for editing existing or copied forms and is therefore not explained further.

10.4 Plot frames

Plot frames are a sort of frame around the actual project pages. They contain, for example, structure identifiers, column and/or row information and other data like page description, etc.

Editing plot frames in the plot frame editor is very similar to editing forms in the form editor. The editor is accessed via the **UTILITIES/MASTER DATA/PLOT FRAME** menu.



Menu for editing plot frames

Fig. 10.36 Opening the plot frame editor

As with the form editor, this menu has the items **NEW**, **OPEN**, **CLOSE**, and **COPY**. This menu functions in a similar way to the menu used when editing forms. We will therefore only deal with the features that are special to the plot frame editor.

NOTE: The plot frame editor does not have a *Save* function either. To restore plot frames that were accidentally modified, you should therefore use the *Undo* function in the plot frame editor or consider synchronizing existing project master data with the system master data.

10.4.1 Create new plot frame (from copy)

To copy and then edit a plot frame, in the UTILITIES/MASTER DATA/PLOT FRAME menu, select the COPY function. EPLAN opens the **Copy plot frame** dialog.

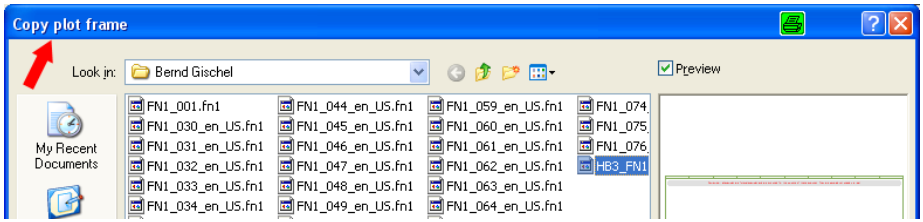


Fig. 10.37 Copy plot frame dialog

In this dialog, you select a suitable plot frame and confirm your selection by clicking the OPEN button in the lower area. EPLAN closes the **Copy plot frame** dialog and opens the **Create plot frame** dialog.

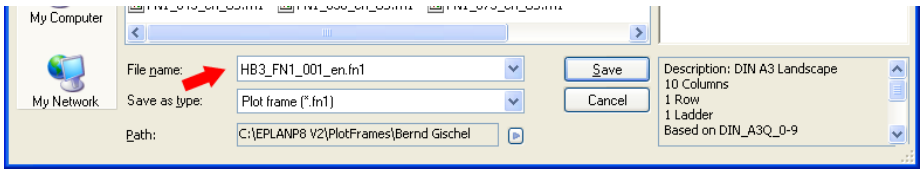


Fig. 10.38 Create plot frame dialog

Enter the new name of the plot frame into the **File name** field. When you click SAVE to close the dialog, EPLAN temporarily opens (in the current project) a page with the copied plot frame.

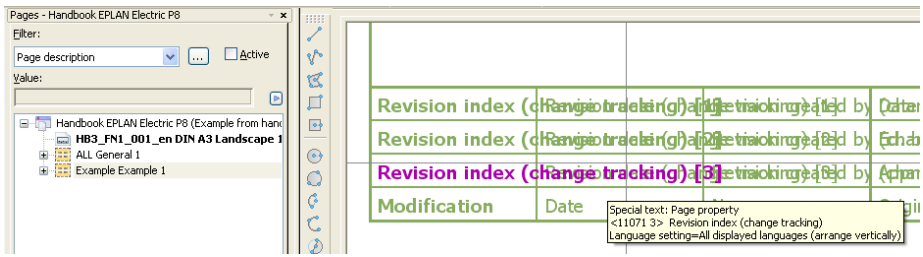


Fig. 10.39 Open plot frame in the page navigator

The plot frame can now be edited using the same functions as already described in the form editing section 10.3. Graphical elements and texts can also be placed in the plot frame editor.

In contrast to forms, it is possible (and, in part, for project planning necessary) to specify column text (paths) and row text for the plot frame. You can also embed a special text watermark in the plot frame. This special text is then automatically filled with data from revision management.

Special texts for plot frames

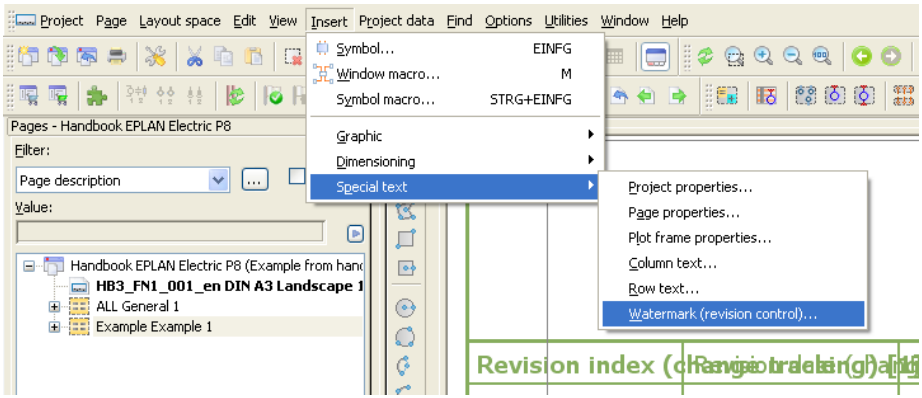


Fig. 10.40 Insert/Special text menu

Column and row texts are treated as normal text. This also includes the formatting (font size, color, etc.).

Column and row texts are placeholder texts that are defined by entries in the **Plot frame properties** dialog (via the **CTRL + M + D** shortcut key or in the **page navigator** by selecting the temporary page, opening the popup menu, and selecting the PROPERTIES item) and that are later automatically filled in the project.

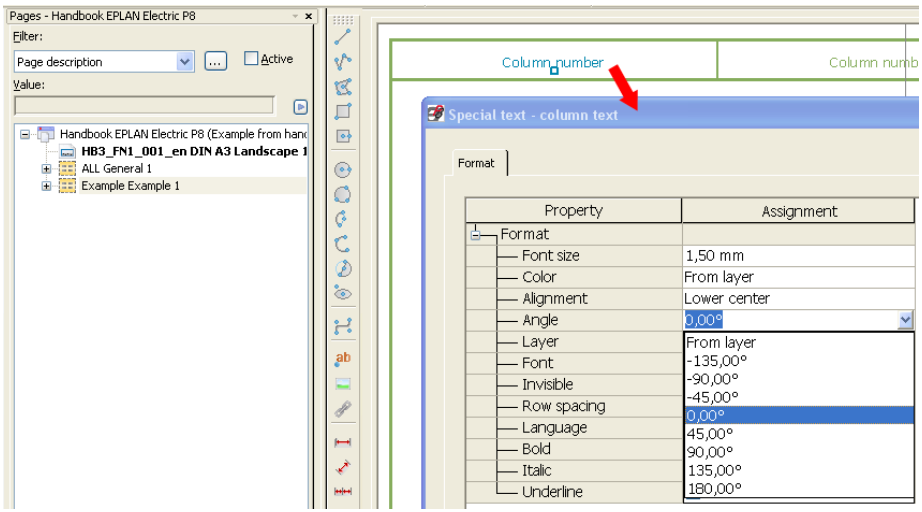


Fig. 10.41 Formatting options for column and row texts

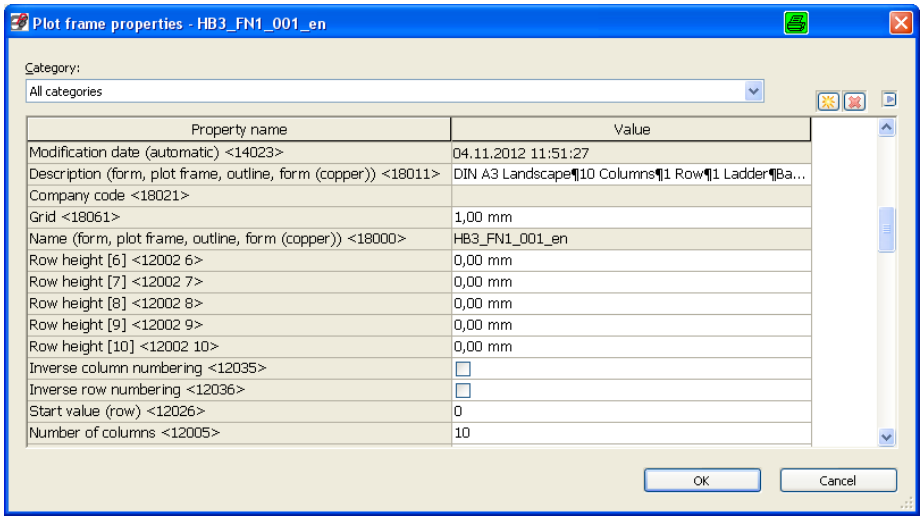


Fig. 10.42 Plot frame properties

Once all entries have been made or the plot frame has been graphically edited as desired, it can be closed ("saved") by selecting, in the **UTILITIES/MASTER DATA/PLOT FRAME** menu, the **CLOSE** function.

EPLAN closes and permanently "saves" the plot frame in the system master data. If the plot frame is needed for the project, it will be automatically stored in the project from the system master data. This functional process is identical to working with forms; the same is true of the synchronization in case of subsequent modifications to the plot frame.

10.4.2 Edit existing plot frame

In addition to copying plot frames, existing plot frames can also be edited. To do this, select the **UTILITIES/MASTER DATA/PLOT FRAME** menu and then select the **OPEN** function. EPLAN opens the **Open plot frame** dialog.

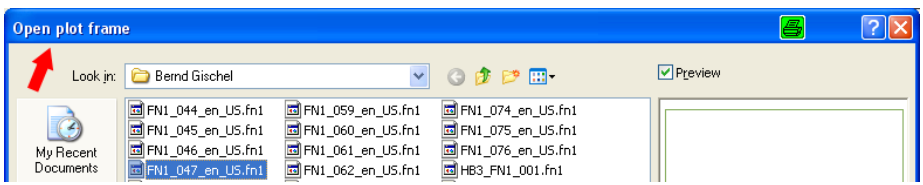


Fig. 10.43 Open plot frame dialog

Here you select the plot frame you wish to edit and open it via the **OPEN** button in the lower area of the dialog. EPLAN closes the dialog and opens the plot frame again as a temporary page in the page navigator of the current project.

It can now be edited using the familiar functions (graphics, text, etc.); the plot frame properties can also be edited or extended.

Close the plot frame using the **UTILITIES/MASTER DATA/PLOT FRAME/CLOSE** menu. If the plot frame has been used in the current project, you will be prompted whether you wish to synchronize with the project master data.

10.4.3 Create new plot frame

Instead of copying or editing existing plot frames, you can create a completely new plot frame via the **NEW** function in the **UTILITIES/MASTER DATA/PLOT FRAME** menu. EPLAN then opens the **Create plot frame** dialog.

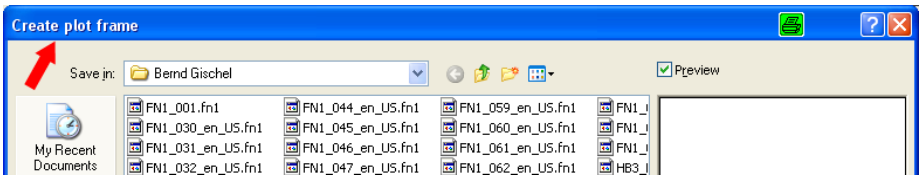


Fig. 10.44 Create plot frame dialog

You enter the new name of the plot frame in the **File name** field of the **Create plot frame** dialog.

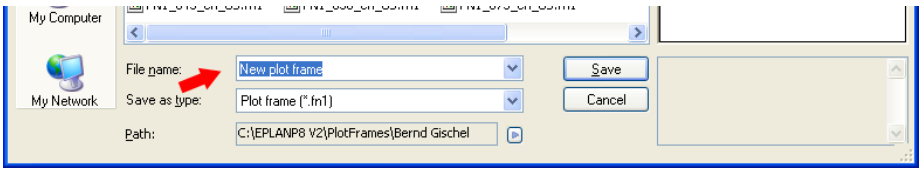


Fig. 10.45 New plot frame name in the File name field

Clicking the **SAVE** button creates the plot frame and EPLAN then opens the **plot frame properties** of the plot frame and also displays an empty temporary page in the page navigator of the current project.

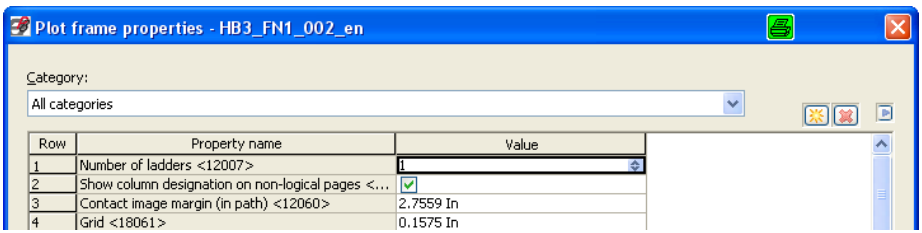


Fig. 10.46 Plot frame properties of the new plot frame

Now, you can edit the plot frame using the familiar functions.

NOTE: Instead of creating a new plot frame, you can also use a copied plot frame as a template for a new plot frame and then modify it accordingly.

This is done in the same way as editing an existing plot frame and will not be explained further here.

11 Old EPLAN data (EPLAN 5 . . .). . .

Generally speaking, importing data from older versions of EPLAN (EPLAN 5, EPLAN 21, PPE and FLUID) is a fast, time-saving way of continuing to work with these projects in EPLAN Electric P8. Conversion is relatively trouble-free, barring a few specific solutions. But it is still not the best solution.

TIP: It is a better to start new projects using EPLAN Electric P8 and not to base them on data from projects created with older versions.

It is also not absolutely necessary to convert all projects from old versions to EPLAN Electric P8. Since EPLAN will continue to support the earlier versions for a long time to come, there is no reason to panic. Older or currently running projects can still be edited or finished with the earlier versions.

This chapter only provides a very brief introduction to the import functions. More information can be found in the first edition of the reference handbook (ISBN 978-3-446-42266-9, only available in German), which has a detailed chapter on importing old data. Alternatively, you can use EPLAN Electric P8 online help, read the support documents on importing old EPLAN data to EPLAN Electric P8 found in the EPLAN support area, or use the many EPLAN Internet forums.

It is important to realize that there were numerous ways to create schematics in the old versions of EPLAN. It is therefore unavoidable that deviations will occur here and there in a converted (imported) project.

NOTE: If projects are imported into EPLAN Electric P8 from an old version, then the old project remains intact. After importing, you have two projects, the original project (that of the old version) and the same project (“new”) converted to the EPLAN Electric P8 data model. None of the old data is deleted or changed, so the original project can still be edited in the previous version without problems.

If a project is edited further in the previous version, then it is no longer the same as the project that was previously converted to an EPLAN Electric P8 project. Therefore it must be re-imported into EPLAN Electric P8.

11.1 Import options

EPLAN Electric P8 offers an import assistant for the various old versions of EPLAN, as well as for the different data from older versions. This can encompass entire projects or individual data sets, such as a form, a macro, parts data or cable data.

You will find this import assistant in the **UTILITIES/DATA IMPORT** menu in the menu items: **EPLAN 5/FLUIDPLAN DATA IMPORT** (for EPLAN 5 and fluid projects as well as for EPLAN 5/Fluid master data), **EPLAN 21** (for EPLAN 21 projects and EPLAN 21 master data), **EPLAN PPE** and **EPLAN CABINET** (for PPE old projects, PPE old data and Cabinet data).

In general, it is possible to import both entire projects and individual data using EPLAN Electric P8. Additional functions and calls (e. g. for EPLAN 5/fluidPLAN, EPLAN 21, EPLAN PPE and EPLAN Cabinet) were integrated into the individual data import menu items to enable you, for example, to import into EPLAN Electric P8 an individual plot frame or parts data exported from the older system.

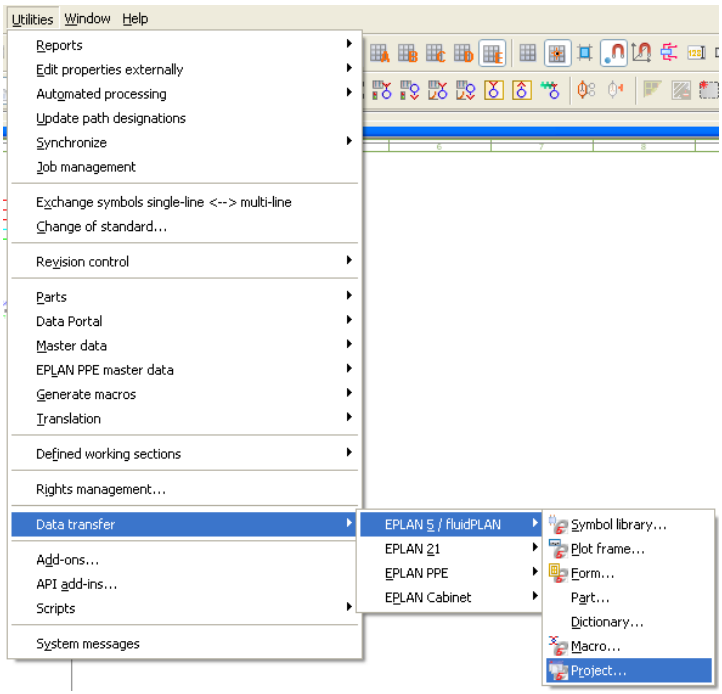


Fig. 11.1
Menus for (old) data imports

NOTE: The sequence of menu items in the menu is important during data importation. This sequence was not arbitrarily chosen; it represents the standard sequence of a data import.

An EPLAN 5 data import, for example, should begin with the symbol library. Then the plot frame should be imported followed by the forms, the parts data and the foreign language file (the dictionary in EPLAN Electric P8), and any macros. At the end, the actual EPLAN 5 project can be imported.

In addition to these individual steps, EPLAN Electric P8 offers a "one-click wizard" for importing data. To import old data, you start with the **UTILITIES/DATA IMPORT/EPLAN 5 – FLUIDPLAN/PROJECT** menu item, enter some information in the subsequent dialogs, and then let EPLAN Electric P8 automatically take care of the rest of the import procedure.

12 Extensions

In addition to the standard functions delivered with EPLAN Electric P8, there are a series of further function extensions that are covered by additional modules, called add-ons. These add-ons make it easier to work with projects and their data, or help in other ways to accelerate and optimize engineering. This chapter will explain some of these add-ons. In addition to those that are discussed here, there are also a series of further add-ons that can be requested directly from EPLAN.

12.1 EPLAN Pro Panel

The **EPLAN Pro Panel** module offers mounting layouts in 3D, which has significant benefits for the EPLAN user and subsequent users (workshop, customer, etc.)

12.1.1 What can EPLAN Pro Panel do?

EPLAN Pro Panel is a very useful alternative to conventional 2D mounting panel layouts. The 3D approach is based on an existing schematic, but of course it can also be started with the selected parts themselves, even without a schematic. This is made possible by the common data platform for all EPLAN products such as EPLAN Electric P8, EPLAN Fluid and EPLAN Pro Panel.

EPLAN Pro Panel supports the user during design and mounting layout, e. g. with a collision check to make sure that devices are not placed in the wrong location. EPLAN Pro Panel doesn't, of course, relieve users of (mental) work, but it does help to avoid wrong decisions (for example, misplacements), which saves time in the end.

It is not possible to describe all the EPLAN Pro Panel functions in this section. There are simply too many functions and too many ways to solve tasks using EPLAN Pro Panel. But we will explain the basics, the first steps – from a project with placed devices, to 3D layout

of a mounting panel, to the final step of creating the familiar 2D model view (similar to the familiar 2D panel layout).

12.1.2 General

There are many ways to use EPLAN Pro Panel. There are also some requirements in terms of hardware and the data needed to quickly get the results you want with EPLAN Pro Panel.

12.1.2.1 What are the hardware requirements?

In terms of hardware, we don't need powerful "CAD machines". The hardware used for EPLAN Electric P8, for example, is sufficient. The graphics card is the only exception. It needs to be up to handling the 3D feature. It must be able to handle OpenGL Version 2.1 or later. In theory, you don't have to have OpenGL, but without it something will be displayed incorrectly sooner or later, and the pleasure of designing with EPLAN Pro Panel will be lost.

And it goes without saying that using multiscreen solutions with EPLAN Pro Panel is better than a small one-screen solution.

12.1.2.2 What are the data requirements?

Basically, there are no major requirements on the data and its contents. Sophisticated 3D models are not necessary. Simple values for **width**, **height** and **depth** suffice for the start.

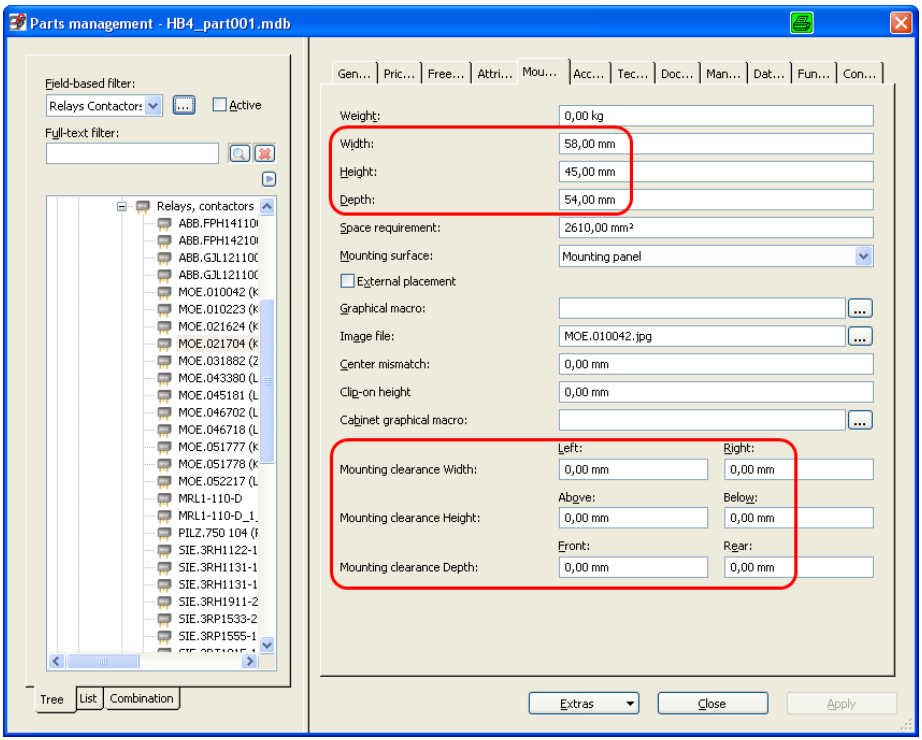


Fig. 12.1
Required technical data for a part

Ideally, data for **mounting clearances**, **center mismatch** and **clip-on height** should correctly entered.

If, instead of a single "faceless" body, you would like to have a detailed body with the smallest screw details in the layout, you can add a macro (**Graphical macro**). An image file is always advantageous, and not just for EPLAN Pro Panel.

These files can also be used in different reports (e.g. a parts list with the technical data and an image of the device), and their use is not limited to EPLAN Pro Panel.

12.1.2.3 What is possible with EPLAN Pro Panel?

Not only is a 3D display visually appealing (better understood by customers), it also offers useful "by-products", such as a 2D derivation of a 3D layout.

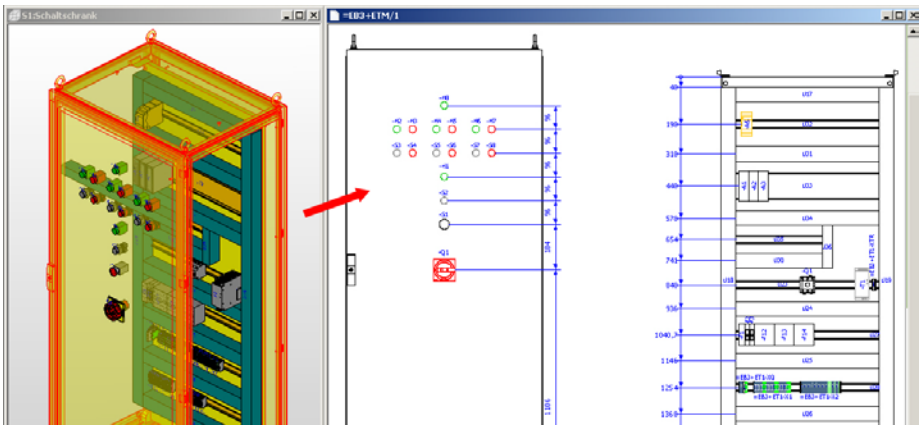


Fig. 12.2 3D representation and the 2D view derived from it

The clear advantage here is obvious. The enclosure is projected in 3D, and the 2D representation with its dimensions, distribution, front views and side views, and maybe additional follow-up documentation such as an enclosure legend, can be created automatically and transferred to the workshop for construction.

You do not have to produce two layouts; you create one 3D layout that is then changed or extended. The 2D derivation and all other automatic follow-up documents, such as reports, are simply updated.

12.1.2.4 General information on operation

In following sections, we will cover a few basic settings, notes on operation, and the meaning of some buttons, menu items, etc.

12.1.3 Toolbars in EPLAN Pro Panel

In addition to the menu functions, EPLAN Pro Panel also has toolbars that contain many of the functions. As usual in EPLAN Electric P8, toolbars can be called up and activated by right-clicking in the toolbar area.

Fig. 12.3 Various EPLAN Pro Panel toolbars

EPLAN Pro Panel comes with many toolbars, such as **3D VIEWPOINT** and **PRO PANEL OPTIONS**). You can also make your own toolbars, for instance an **INSERT PRO PANEL...** toolbar containing items for inserting enclosures and mounting rails or even placing locked areas. You then have a shortcut for the alternative route via the **INSERT/[DEVICE FOR PRO PANEL]** menu, in which **[DEVICE FOR PRO PANEL]** means mounting panels, mounting rails, etc. The following sections briefly describe some of the toolbars.

12.1.3.1 3D viewpoint toolbar

The first six buttons of this toolbar offer different *2D view directions*, e. g. on the enclosure, the terminal box, or other parts.

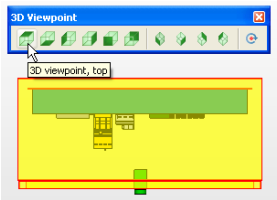


Fig. 12.4 View from above

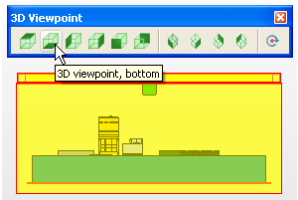


Fig. 12.5 View from below

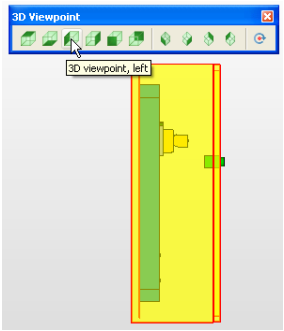


Fig. 12.6 View from left

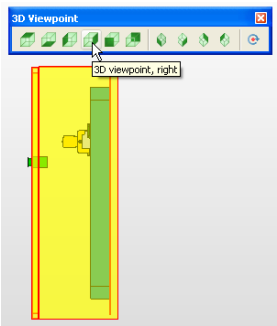


Fig. 12.7 View from right

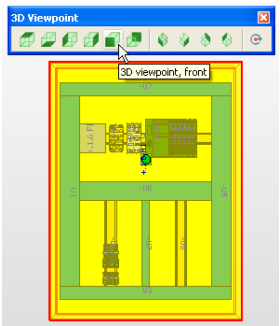


Fig. 12.8 View from front

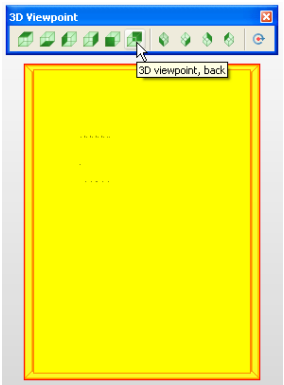


Fig. 12.9 View from rear

The next four buttons activate the *3D view* and its four different isometric viewpoints.

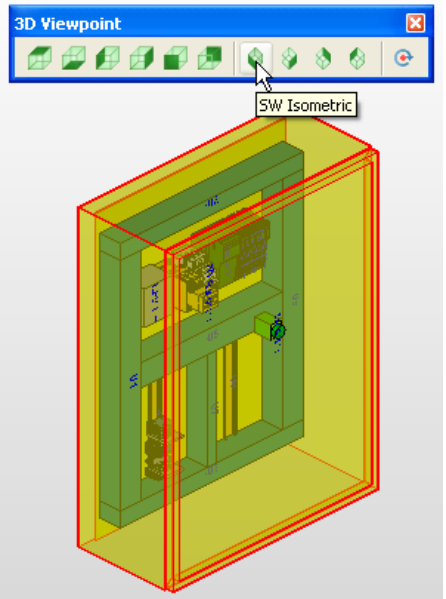


Fig. 12.10 South-West isometric view (SW)

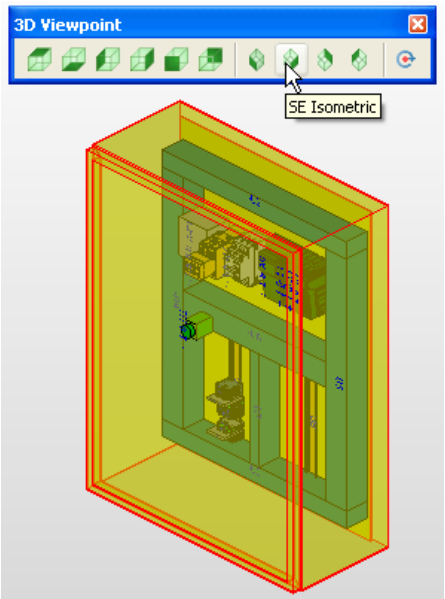


Fig. 12.11 South-East isometric view (SE)

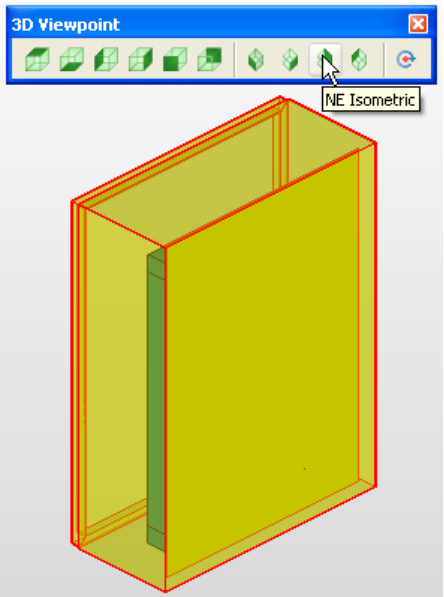


Fig. 12.12 North-East isometric view (NE)

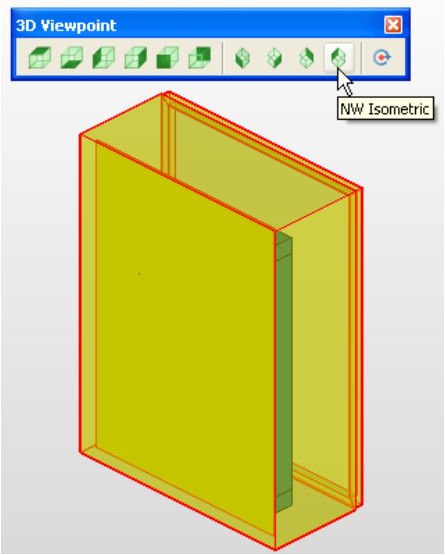


Fig. 12.13 North-West isometric view (NW)

The last button on this toolbar, **ROTATE VIEWING ANGLE**, allows you to hold down the left mouse button to freely and smoothly rotate an object to get the exact angle that you want or need.

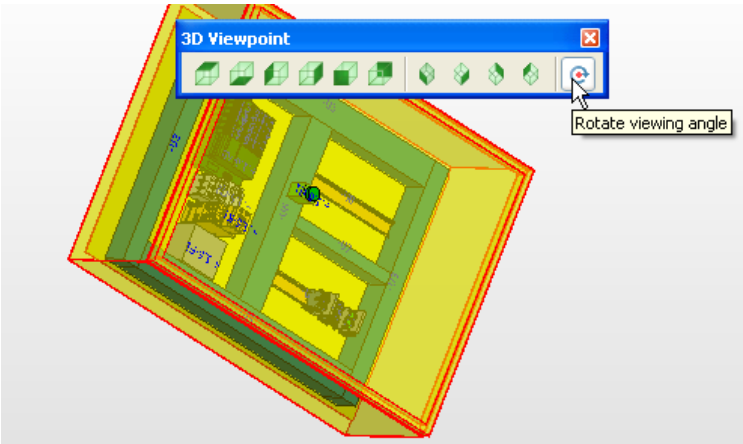


Fig. 12.14
Manually selecting the rotation angle

12.1.3.2 Pro Panel options toolbar

The **PRO PANEL OPTIONS TOOLBAR** contains functions such as various display and control functions.

When the **COLLISION CHECK** button is switched on, EPLAN calculates before placement whether items will collide, showing this behavior in color and not allowing the device to be placed.

The **DISPLAY MOUNTING CLEARANCES** button enables a quick visual check of whether there is still some more space around the placed device, or if the mounting clearances are already as minimal as possible.

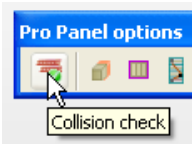


Fig. 12.15
Collision check

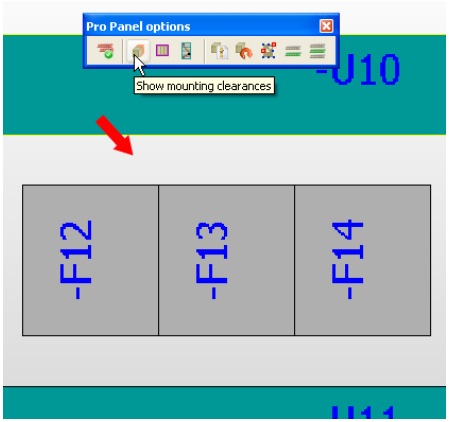


Fig. 12.16 Do not display mounting clearance

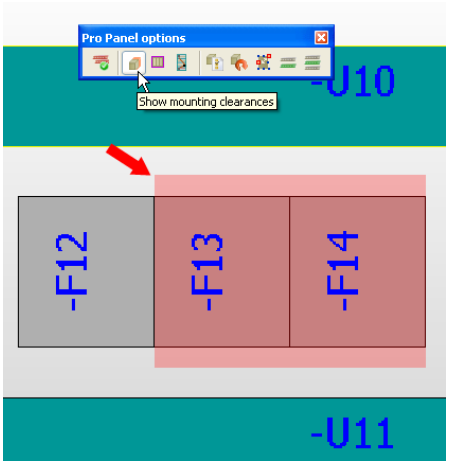


Fig. 12.17 Display mounting clearance

The **SIMPLIFIED REPRESENTATION** button activates a function that displays all devices in a simple "block" without any detailed graphics.

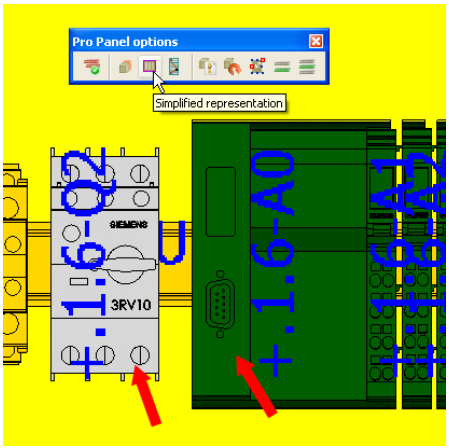


Fig. 12.18 Detailed representation

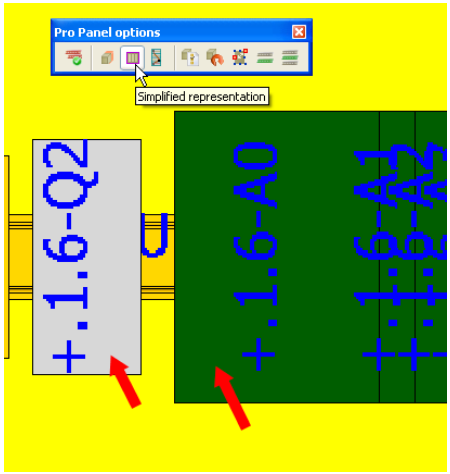


Fig. 12.19 Simplified representation

In Fig. 12.18, all devices (even if present and drawn in the macro) are displayed with lots of details, such as screws, setting ranges, switches, etc. If this is not necessary or if the display (the graphical rendering) becomes too slow, then you can switch on the simplified representation (Fig. 12.19). This also applies, for example, to large, extensive terminal strips where the focus is less on the individual terminals than on identifying the terminal strip as a whole in the mounting layout.

The **CONNECTION PREVIEW** button contains a preview of the devices that are connected to each other.

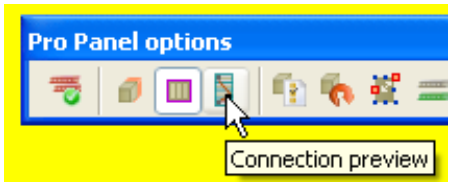


Fig. 12.20 Connection preview

All of the other buttons in the **PRO PANEL OPTIONS** toolbar, such as **PLACEMENT OPTIONS**, **CHANGE ROTATION ANGLE**, **CHANGE HANDLE**, **ADOPT LENGTH**, and **PLACE CENTERED**, are addressed in greater depth in subsequent sections. At this point, this is just a brief overview of the functions.

Now we will take a look at the **PLACEMENT OPTIONS** button. If a device is placed and is "hanging" on the cursor, you can use this button to call up the **PLACEMENT OPTIONS** dialog.

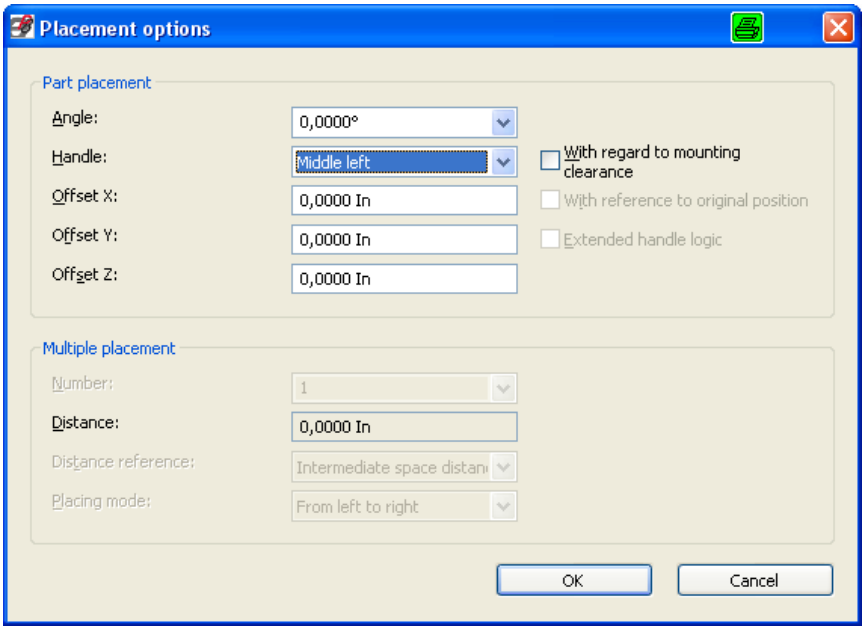


Fig. 12.21 Placement options dialog

The **PLACEMENT OPTIONS** dialog offers several ways to affect placement. In the *Rotation angle* selection, you can select a value from the selection list or simply enter a degree value; for example, you could place a device rotated by 50 degrees.

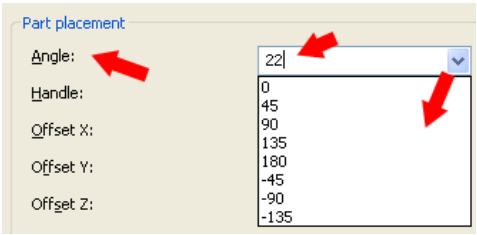


Fig. 12.22 Adjustable rotation angle

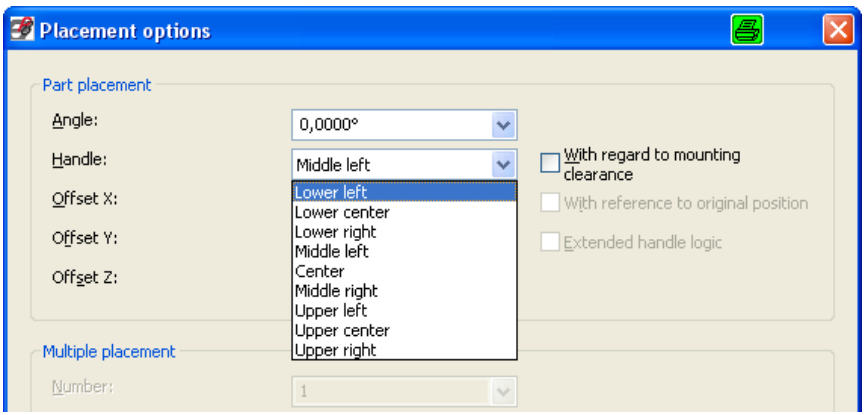


Fig. 12.23 Selecting a handle

It is also possible to preselect the *handle*. You can also apply a value here from the selection list. This selection list, however, cannot be extended. The handle is closely associated with the *With regard to mounting clearance* setting. If this setting is activated, the mounting clearance is calculated at the handle and the device is accordingly placed "further away" (i. e. with mounting clearance).

Fig. 12.24 Offset settings

Offset settings X, Y, Z: You can enter values directly here to achieve a precise placement at specific angles. This means that EPLAN Pro Panel does not allow a different placement, such as placing a device 50 mm from the edge. Manual deviation during placement, e. g. due to a "trembling mouse", is not possible.

The adjacent setting, *With reference to original position*, can only be actively switched on during actions such as copying, duplicating, etc. This allows you to enter an offset value directly, and EPLAN Pro Panel automatically calculates the additional placement (e. g. for a copy of a device) from the original position.

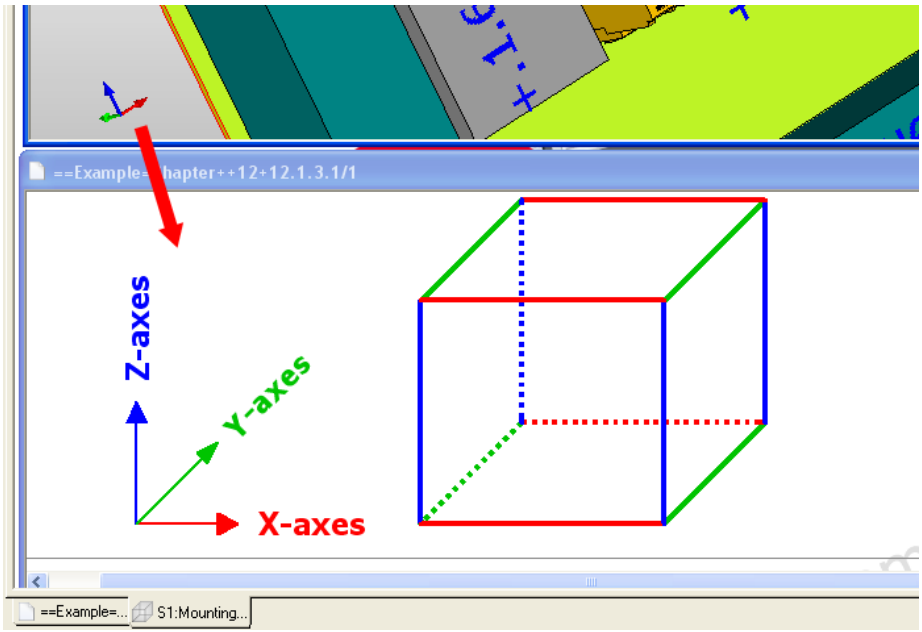


Fig. 12.25 Axes

For the offset settings, it is good to know that we are working in 3D space in the three drawing axes of X, Y, and Z. The three drawing axes are represented in EPLAN Pro Panel by colored lines that always run at right angles to each other. These axes provide useful orientation in 3D space, for example when placing objects.

NOTE: The Y axis is green, the X axis is red, and the Z axis is blue in EPLAN Pro Panel's 3D display.

The *Multiple placement* setting allows you to place several objects at once. All of the objects selected in the **3D mounting layout navigator** "hang" on the cursor, and after you place the first device on the mounting surface (e.g. a mounting panel), all of the other devices are automatically placed as defined in the *multiple placement* settings.

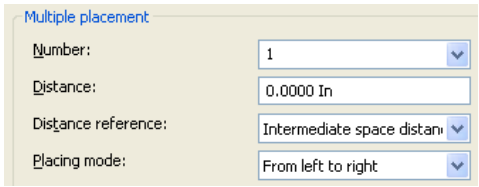


Fig. 12.26
Multiple placement

One example would be placing control elements in a door. EPLAN Pro Panel automatically calculates the desired distances from the first control element, without having to "manually" place every single control element yourself.

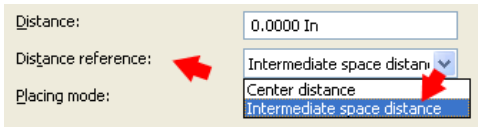


Fig. 12.27
Setting for distance reference and distance

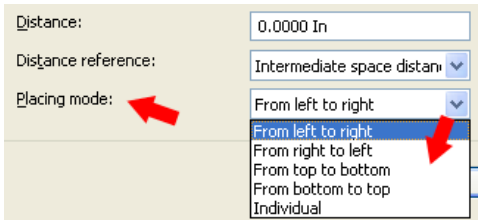


Fig. 12.28
Setting for placing mode

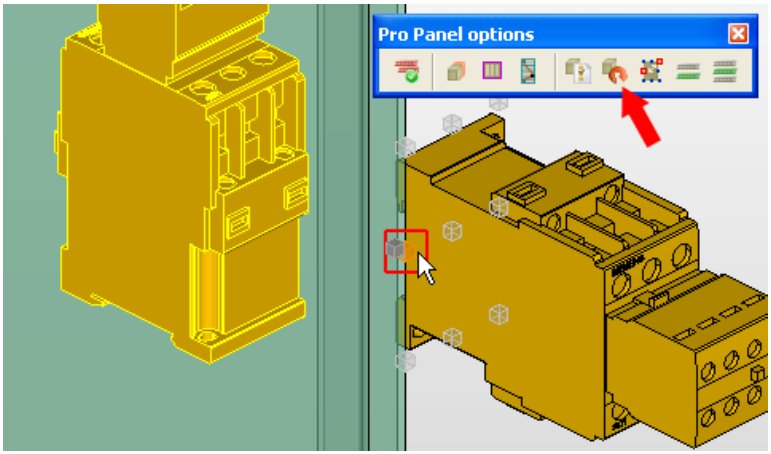


Fig. 12.29
Changing the rotation angle before placement

Another button, **CHANGE ROTATION ANGLE**, offers the option of easily changing the rotation angle before placing a device. However, you can only rotate in fixed 90-degree increments.

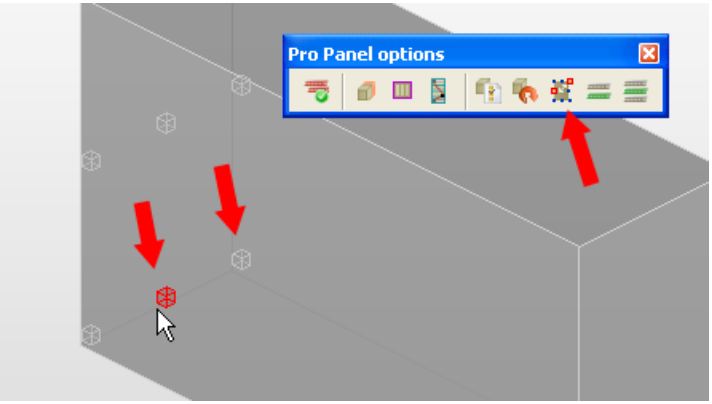


Fig. 12.30 Changing the handle

With the **CHANGE HANDLE** button (or just pressing the **A** key), you can quickly change an object's handle. Fig. 12.30 clearly shows that the contactor has many handles, both external and internal contacts located in the center, left and right. This allows the object to be handled in a number of ways and to be placed with the desired point.

If, for example, you want to place an additional mounting rail or cable duct that has the same length as an existing mounting rail or cable duct, you can use the **ADOPT LENGTH** button.

Fig. 12.31 Adopt length

The procedure is as follows: In the free space on a mounting panel, you want to place a cable duct next to a cable duct with the same length. Using the **INSERT/WIRE DUCT** menu, you select and apply the cable duct. The cable duct now "hangs" on the cursor.

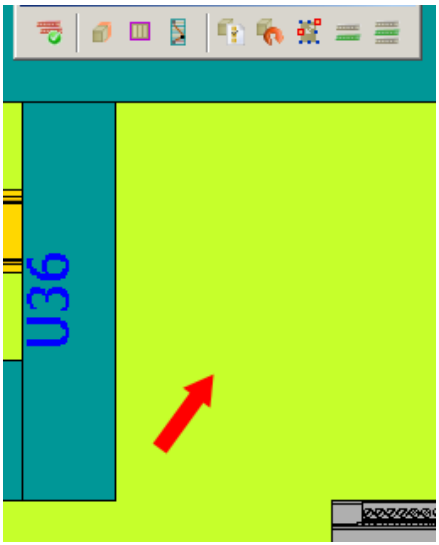


Fig. 12.32 Free space

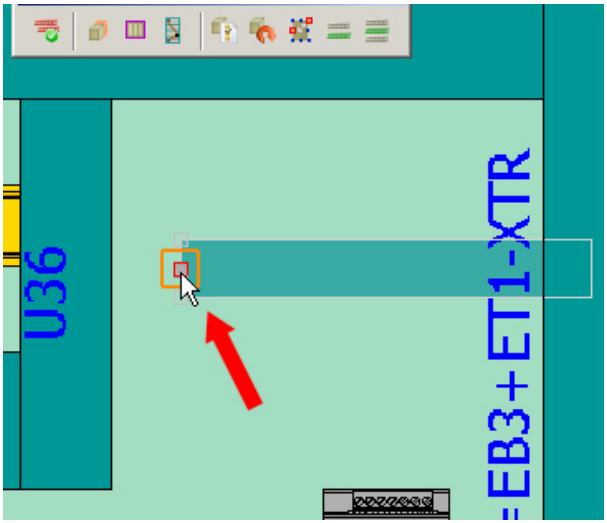


Fig. 12.33 Cable duct "hangs" on the cursor.

You want the new duct to be exactly the same length as duct U36 shown on the left (Fig. 12.33). Click the **ADOPT LENGTH** button (the new duct is still "hanging" on the cursor). The cursor now changes its appearance.

Cable duct (U36) is displayed, and EPLAN Pro Panel immediately uses its length and alignment data. The new cable duct, with its new changed alignment and length, now "hangs" on the cursor for placement. The duct can now be placed.

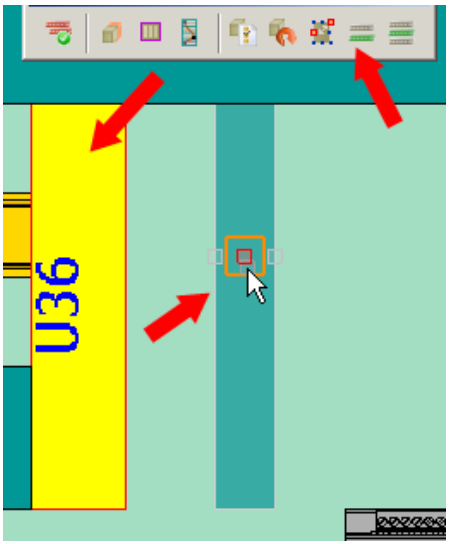


Fig. 12.34
Adopted data

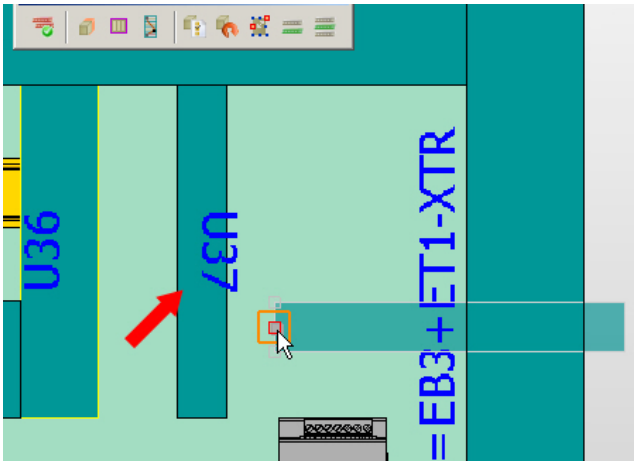


Fig. 12.35 Placed cable duct

You can now place the next duct or end the action.

The last button, **PLACE CENTERED**, located in the **PRO PANEL OPTIONS** toolbar, enables you to place a mounting rail or cable duct exactly centered between two other mounting rails or cable ducts. Of course, this requires that there are already at least two placed mounting rails or two placed cable ducts.

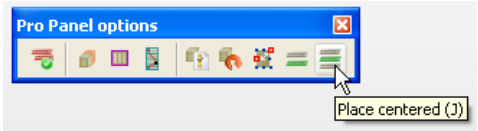


Fig. 12.36 The Place centered button

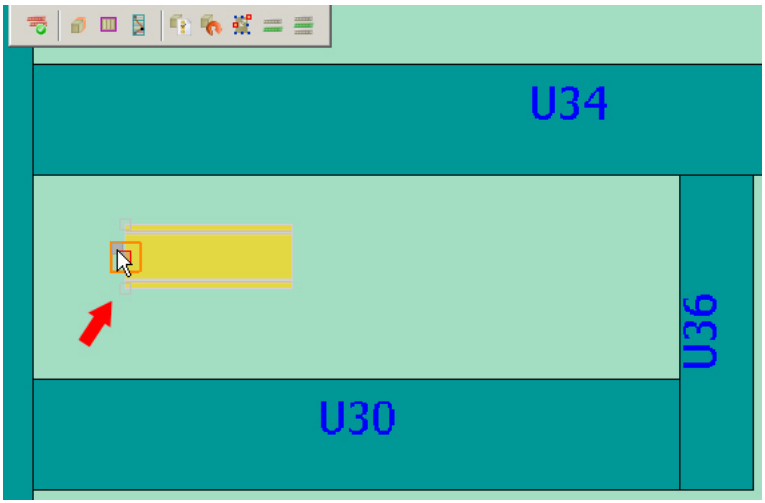


Fig. 12.37 Placing a mounting rail

Using the **INSERT/MOUNTING RAIL** menu from, you select and apply a mounting rail from parts management. The mounting rail now "hangs" on the cursor.

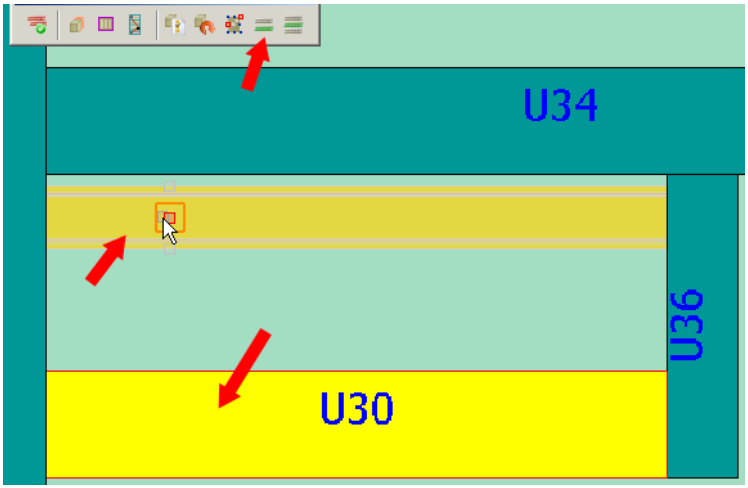


Fig. 12.38 Adopt existing length

Now you click the **ADOPT LENGTH** button and then the cable duct whose length the mounting rail should adopt (U30). EPLAN Pro Panel adopts the length of the cable duct for the mounting rail to be placed.

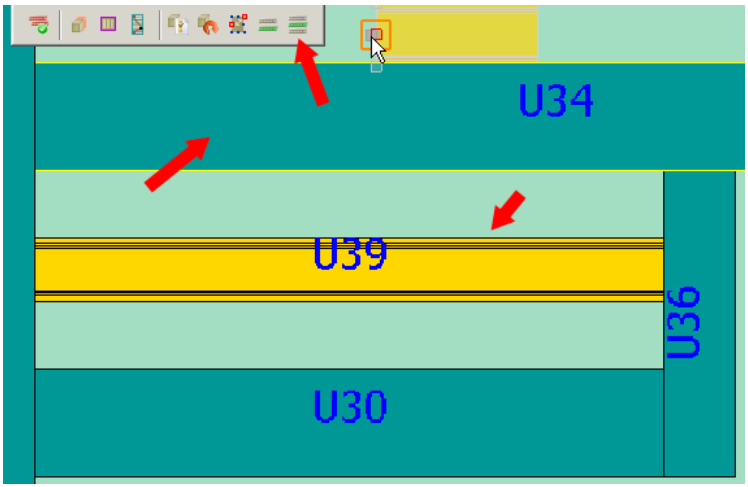


Fig. 12.39 Mounting rail placed centered

Now click the **PLACE CENTERED** button. The cursor changes its appearance and then the "counterpart", the upper cable duct, follows (U34). EPLAN Pro Panel then places the mounting rail automatically in the center between the two cable ducts. Now you can place the next mounting rail (it is still "hanging" on the cursor) or end the action.

12.1.3.3 Insert Pro Panel toolbar...

The last toolbar, **INSERT PRO PANEL...**, is a "customized toolbar" that is *not* part of the standard EPLAN Pro Panel delivery. EPLAN provides a similar menu bar, but it does not include a button for placing a locked area.

TIP: The **INSERT PRO PANEL...** toolbar is provided in the sample data for this manual and can be imported into EPLAN.

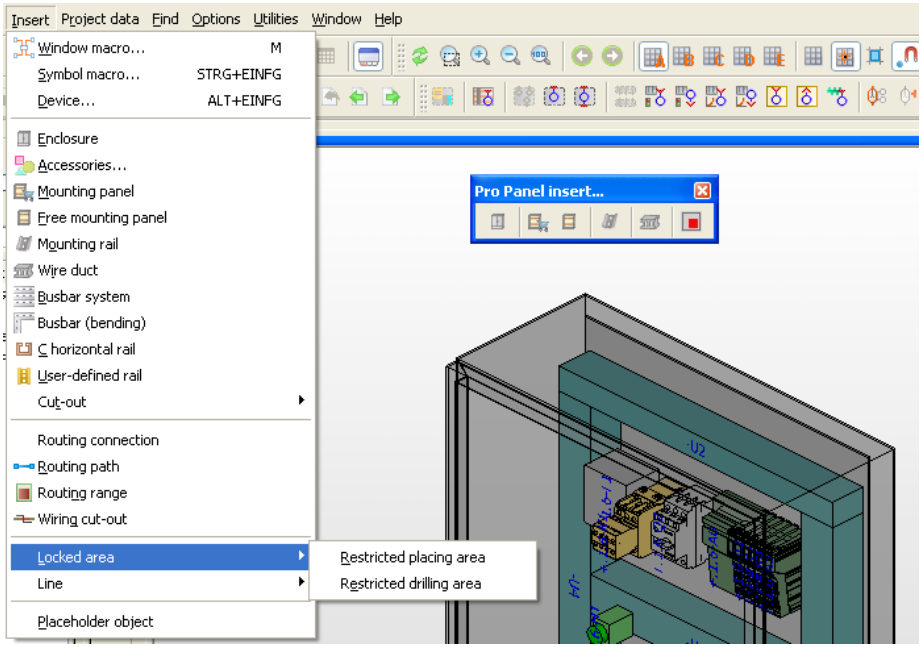


Fig. 12.40 Insert Pro Panel toolbar ...

This toolbar runs the menu functions via the **INSERT/ENCLOSURE** or **INSERT/MOUNTING PANEL** menu, etc. after you click one of the buttons, making working with EPLAN Pro Panel a bit faster since you don't have to go through the **INSERT** menu.

There is little to say about the **MOUNTING RAIL** function itself. If you click this button, **Parts management** opens in the correct area and only offers mounting rails for selection.

Fig. 12.41 Mounting rail button

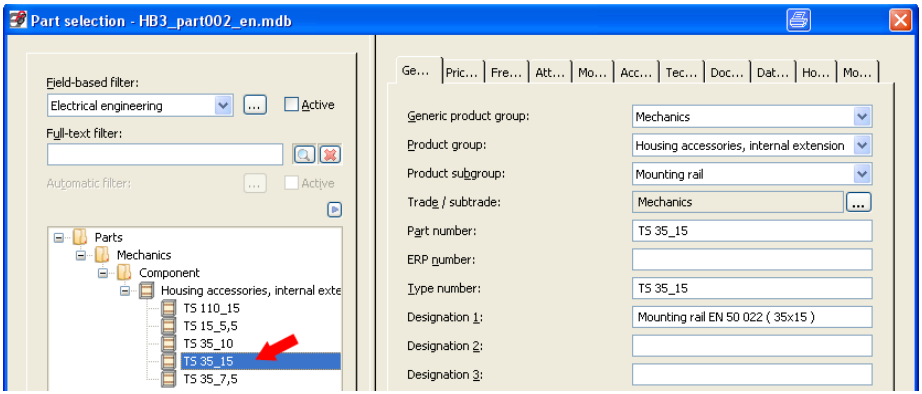


Fig. 12.42 Selecting a mounting rail

These can now be directly adopted. When you click the **OK**, EPLAN closes **parts management** and you can place the mounting rail.

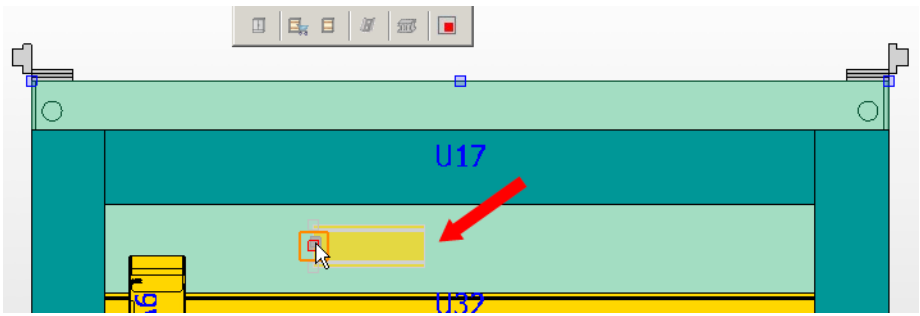


Fig. 12.43 Placing a new mounting rail

12.1.4 Menus in EPLAN Pro Panel

In addition to the toolbars, EPLAN Pro Panel offers other functions in new menus and adds new functions, etc. to existing menus. It would take up too much room to describe all the functions, but we will take a brief look at some of capabilities of the EPLAN Pro Panel add-on.

NOTE: Part of the menus and menu expansions overlap with the description of the functions of the individual toolbar buttons. Therefore they will only be mentioned here and not described in detail. More information is available in section 12.1.3, “Toolbars in EPLAN Pro Panel”.

12.1.4.1 Layout space menu

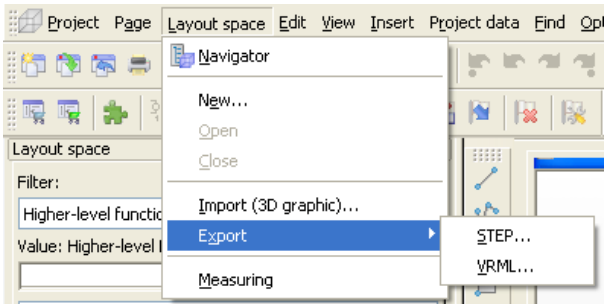


Fig. 12.44 Layout space menu

The **LAYOUT SPACE** menu includes general functions. The **LAYOUT SPACE/NAVIGATOR** menu item opens the layout space navigator. The **NEW**, **OPEN**, and **CLOSE** menu items create a new layout space or open and close an existing layout space. To use the **OPEN** and **CLOSE** menu items, the layout space navigator must be open; otherwise, these menu items are grayed out.

A layout space is essential to be able to display 3D models. If a layout space does not exist yet, then one must be created. This is done by clicking the **NEW** item in the **LAYOUT SPACE** menu.

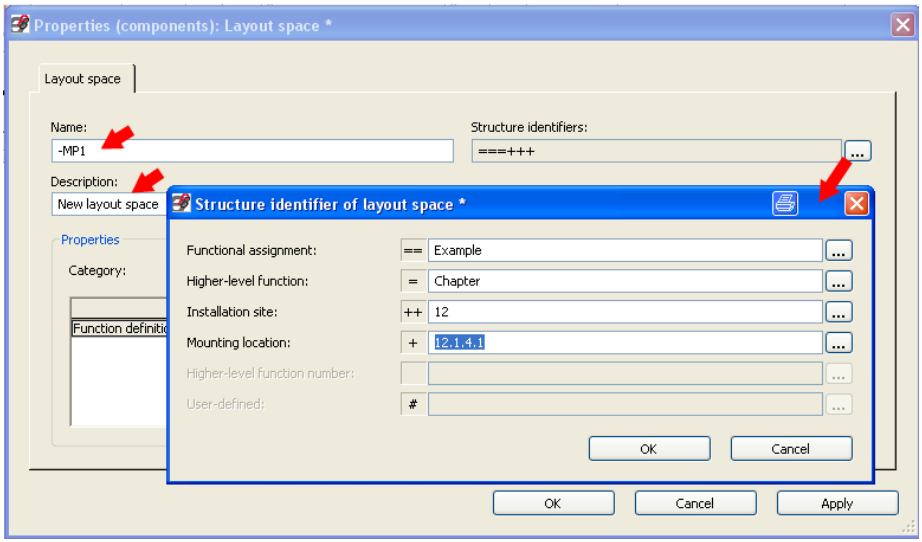


Fig. 12.45 Creating a new layout space

EPLAN now opens the **PROPERTIES (COMPONENTS): LAYOUT SPACE** dialog. Here you assign a **name** to the layout space. You can also assign **structure identifiers** other properties such as a **layout space description**.

When you click **OK** , the new layout space is created in the layout space navigator and is available immediately for 3D objects.

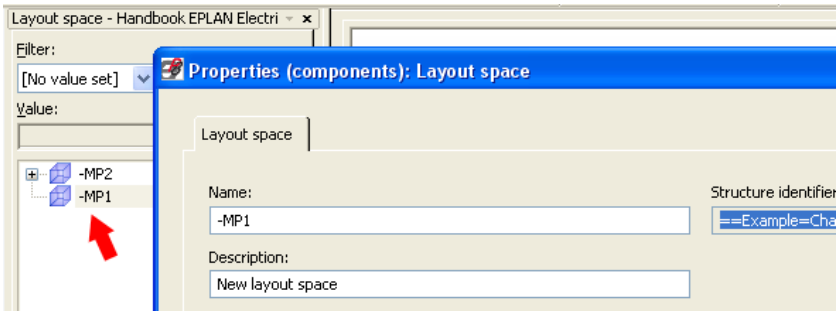


Fig. 12.46
Available layout space

The **LAYOUT SPACE/IMPORT 3D GRAPHIC** menu item enables you to import CAD data into EPLAN.

NOTE: The 3D CAD data to be imported must be saved in the neutral *STEP* format (file extension **.stp*, **.step* or **.ste*). Not all CAD data is suitable for use in EPLAN Pro Panel because some manufacturers process this data in a very detailed and complex way that can push a standard computer to its limits. Please take this into account before importing.

When you select the **LAYOUT SPACE/IMPORT 3D GRAPHIC** menu item, the **OPEN** dialog opens.

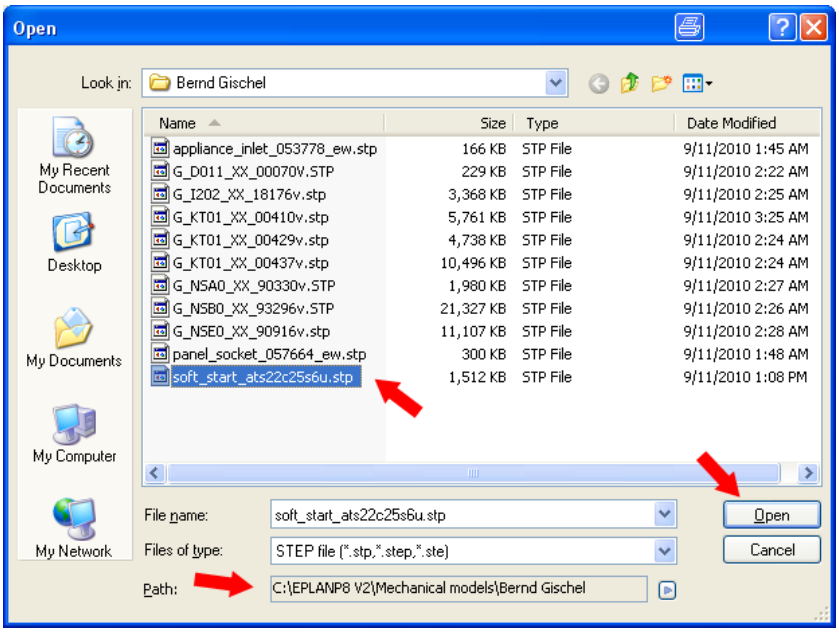


Fig. 12.47
Importing a 3D file

In this dialog, you select a file to import and confirm your selection by clicking the **OPEN** button.

EPLAN then creates the 3D representation without further prompts and displays it as a 3D object in the layout space.

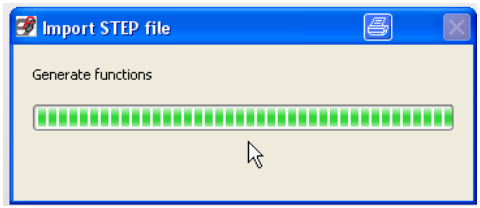


Fig. 12.48
Importing an external STEP file

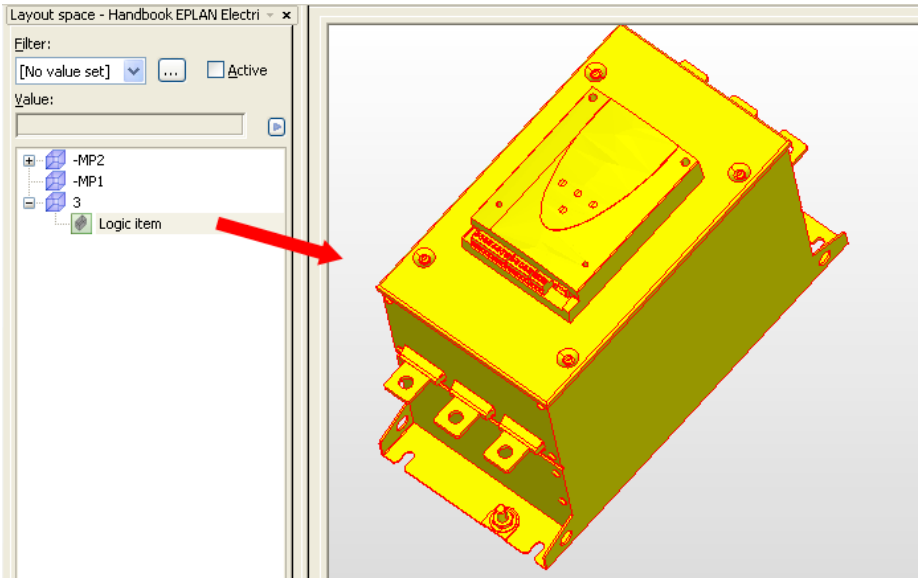


Fig. 12.49
Imported file as a 3D object

The **LAYOUT SPACE/EXPORT** menu item offers multiple export options, so that you can use external CAD tools to further process individual objects or the entire layout space.

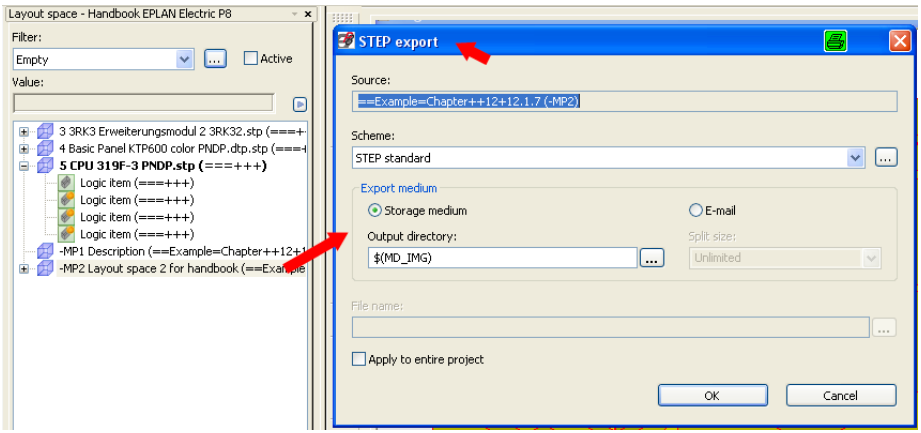


Fig. 12.50
Export dialog

You can use the **MEASURING** function in the **LAYOUT SPACE** menu, for example, to perform measurements on an equipped mounting panel.

When the **MEASURING** function is started, the **MEASURING RESULT** dialog opens and you can click the first measuring point with the cursor.

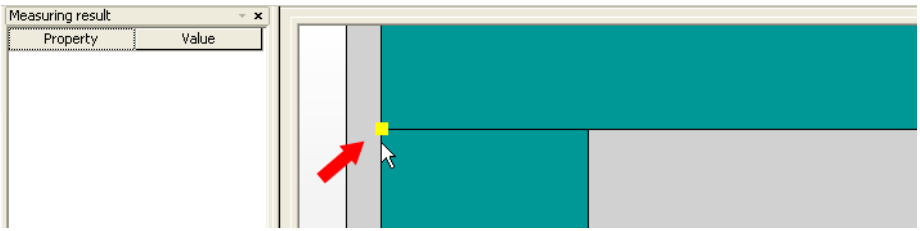


Fig. 12.51 Setting the first measuring point

Then you click the second measuring point. EPLAN Pro Panel determines the corresponding values and displays them in the **MEASURING RESULT** dialog.

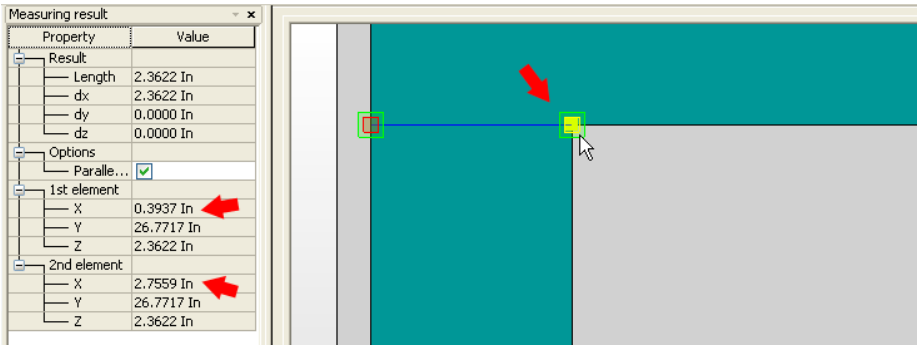


Fig. 12.52 The second measuring point was set and the length was determined.

12.1.4.2 Extended Edit menu

EPLAN Pro Panel has a new menu item, **DEVICE LOGIC**, in the **EDIT** menu. This menu item has additional submenu items.

The **DEVICE LOGIC** menu item has the following main task: Devices that are used for mounting layouts must have certain properties – such as the ability to be placed in the layout space, on other objects or where the handle points are, etc. You can use these menu items under Device logic to define the exact properties.

For correct placement, it is necessary to at least define a placement area for these objects. You can do this using the **EDIT/DEVICE LOGIC/PLACEMENT AREA** menu items and the functions connected to them.

This device logic area belongs to the range of graphical editing functions.

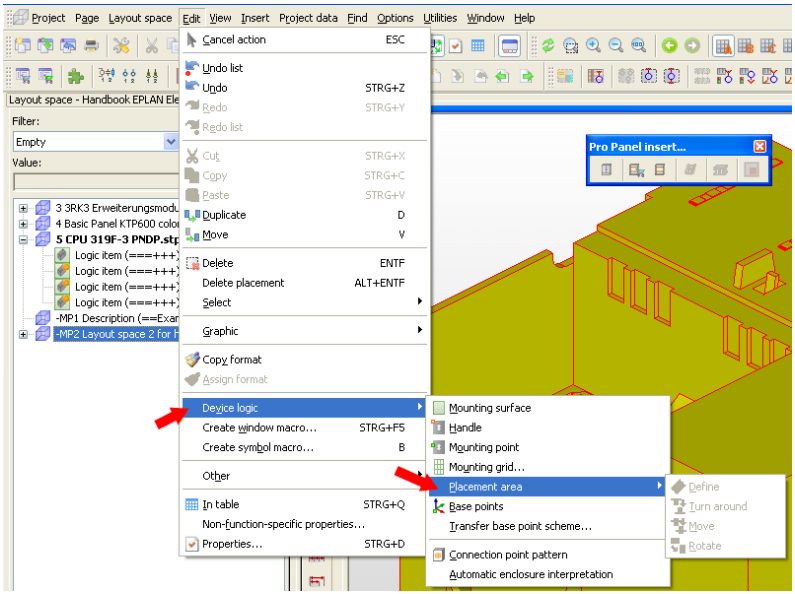


Fig. 12.53
Device logic menu

12.1.4.3 Extensions to the View, Insert and Options menus

A few menu items for working with objects in the layout space were added to the **VIEW** menu.

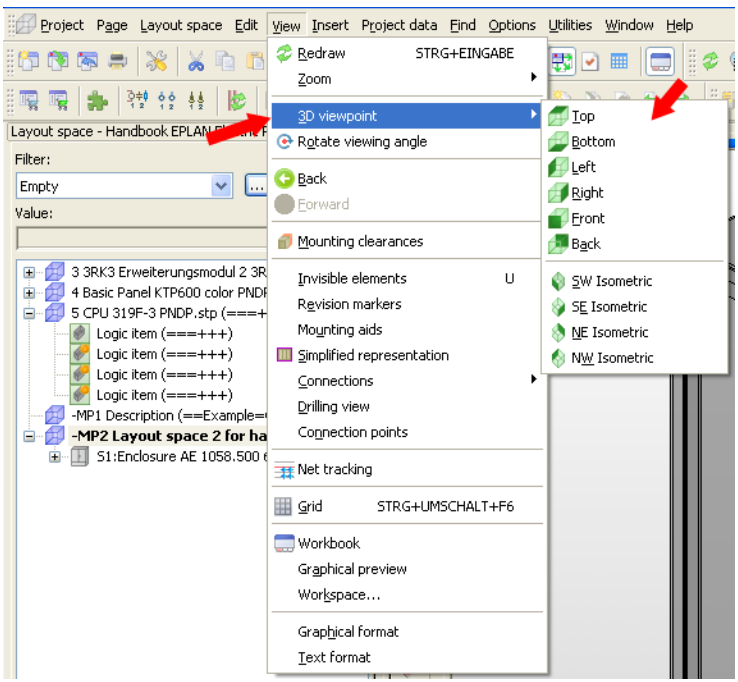


Fig. 12.54
Extensions to the View menu

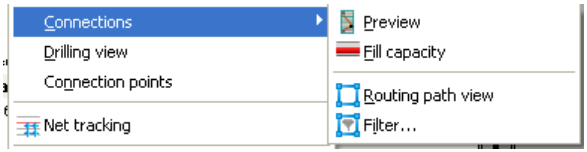


Fig. 12.55 Extensions to the View menu (2)

These menu items, such as **3D VIEWPOINTS** (see Fig. 12.54) and **SIMPLIFIED REPRESENTATION**, have the same function as the existing **3D VIEWPOINT** and **PRO PANEL OPTIONS** toolbars, which were described in greater detail in previous sections. Therefore, we will not again describe the functions or their functionalities.

Like the **VIEW** menu, the **INSERT** menu was also extended to include menu items for working with EPLAN Pro Panel.

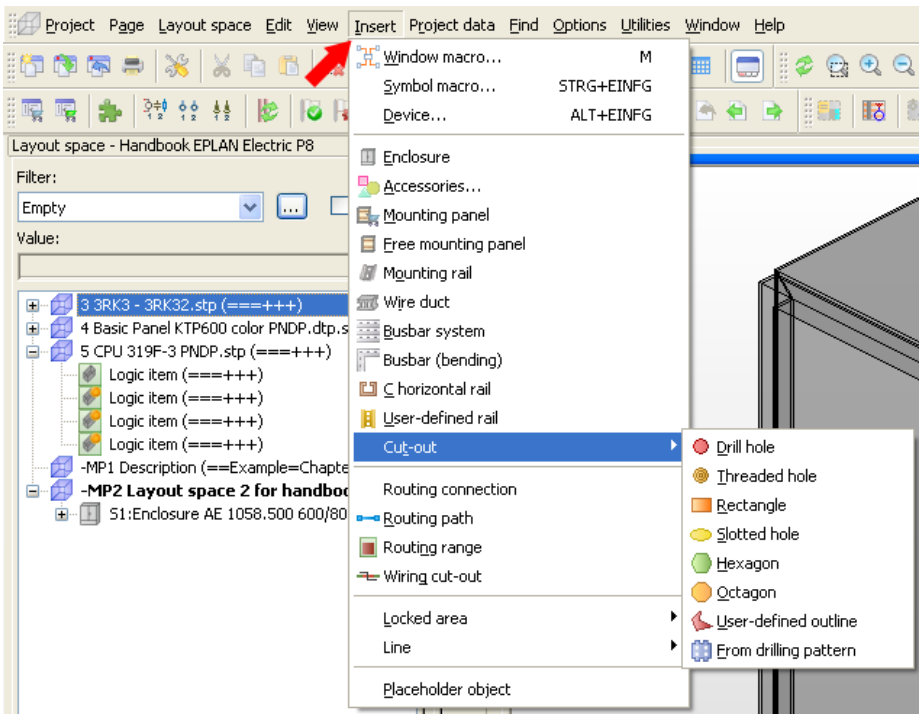


Fig. 12.56 Extensions to the Insert menu

Here again, the menu items included in the **PRO PANEL** toolbar were already explained in detail and will not be repeated here.

The same applies to the **OPTIONS** menu as to the two previous menus, **VIEW** and **INSERT**.

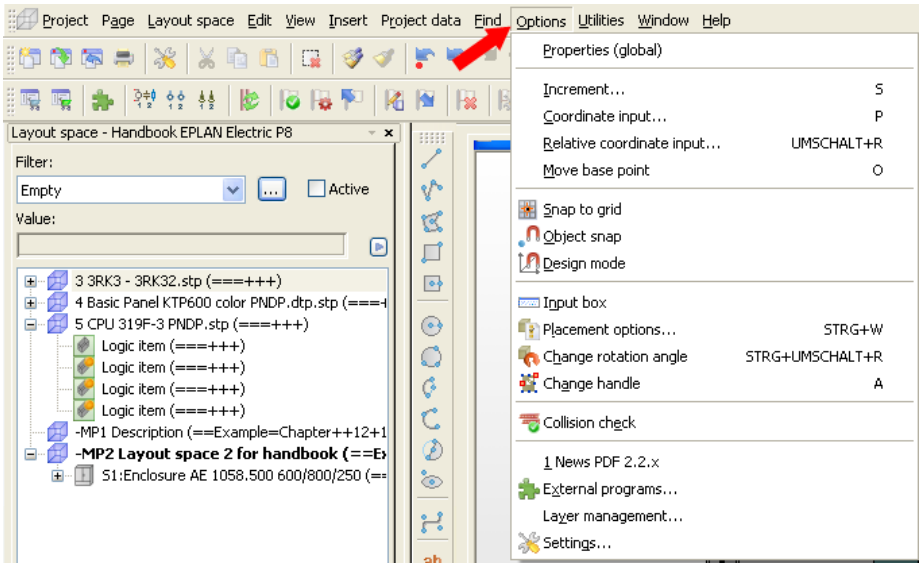


Fig. 12.57 Additional Pro Panel menu items

Functional extensions, such as **CHANGE ROTATION ANGLE** and **CHANGE HANDLE**, have already been explained in detail.

12.1.4.4 The Insert/Generate model view (2D) menu

Using the **INSERT/GRAPHIC/MODEL VIEW (PRO PANEL)** menu item, you can create a 2D view of a 3D mounting layout and its components (such as a mounting panel or a side panel).

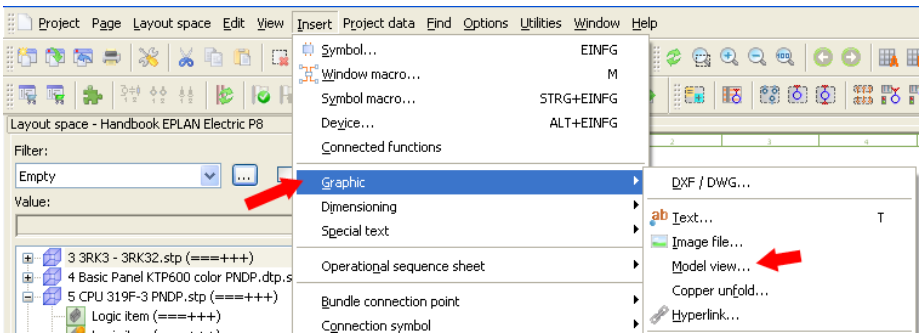


Fig. 12.58 Generating a model view (2D)

To generate a model view (a 2D representation) from a 3D representation, you first have to switch to normal editing. You open both the graphical editor and the page where the model view is to be placed.

NOTE: The page type where a 2D derivation is to be generated must be a **model view** page.

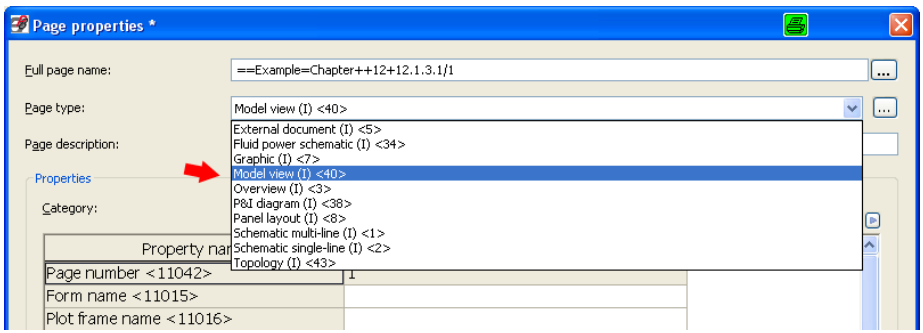


Fig. 12.59 Model view dialog

Now you click the **INSERT/GRAPHIC/MODEL VIEW (PRO PANEL)** menu item.

The cursor changes and you select the first corner of the model view. After placing the first corner, you select and click the second corner.

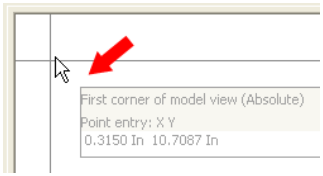


Fig. 12.60
Setting the first corner

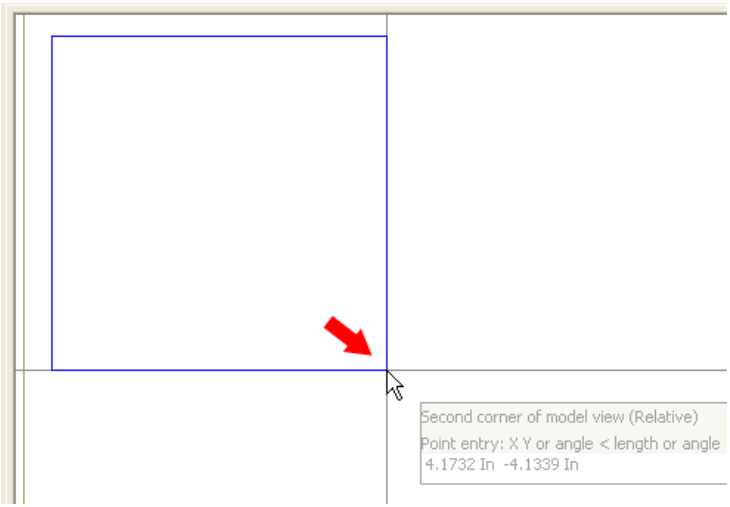


Fig. 12.61 Setting the second corner

EPLAN now opens the **MODEL VIEW** dialog. In this dialog, on the *View* tab, you define the **view name** and enter an optional **description** of the model view.

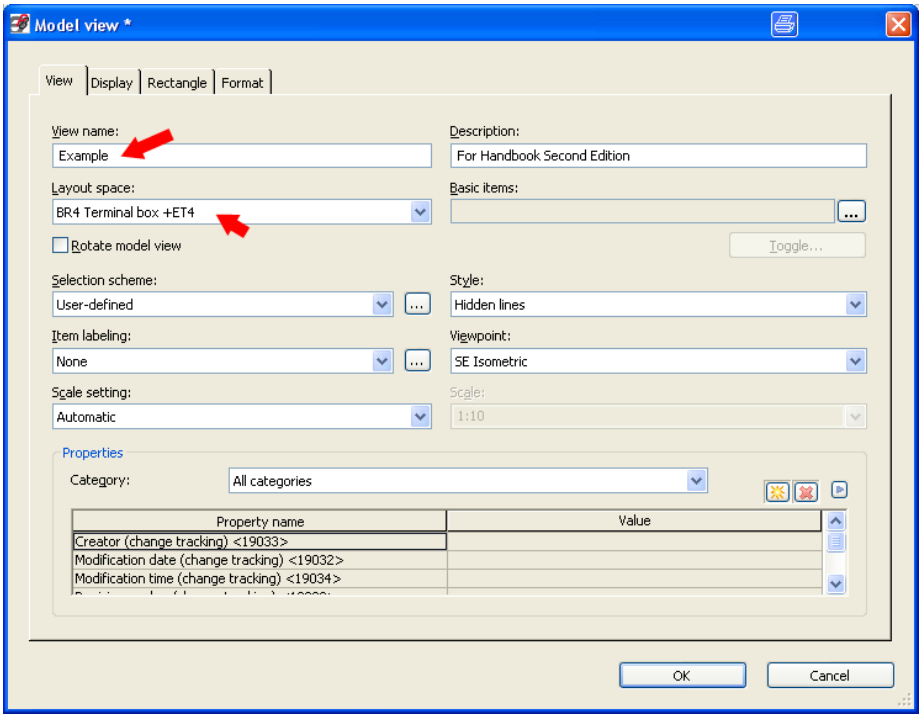


Fig. 12.62 Model view dialog

Then the **layout space** and the **basic items** must be selected. The layout space could be a **terminal box** or an **enclosure**. Basic items can be the **mounting panel**, a **flange**, or **side panels**. You can select all of these 3D objects from the selection lists.

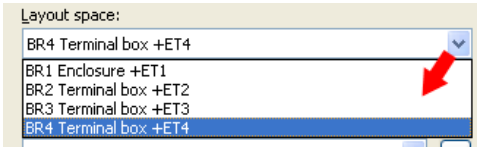


Fig. 12.63 Selecting the layout space

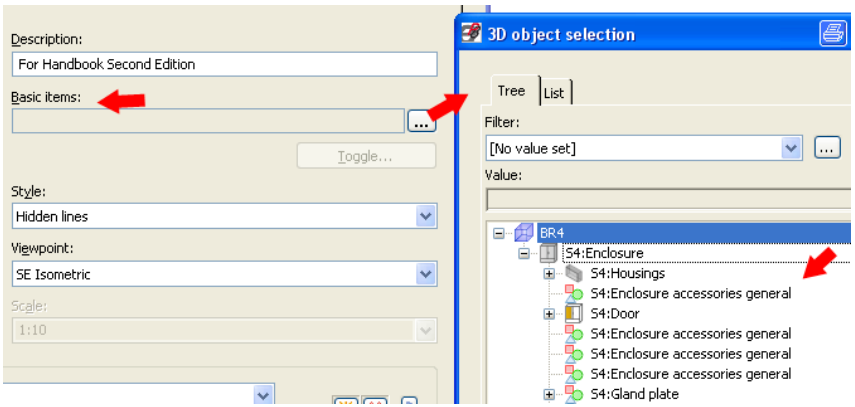


Fig. 12.64 Selecting objects

Once you have made all the described selections, you need to define the **viewpoint** (view) that should be generated and whether **item labeling** should take place, and if so according to what filter scheme.

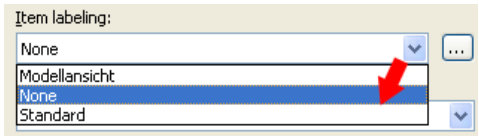


Fig. 12.65 Defining the item labeling

Once these two settings are completed, you can exit the **MODEL VIEW** dialog by clicking **OK**. EPLAN now generates the 2D model view using the defined settings and displays it with a progress dialog.

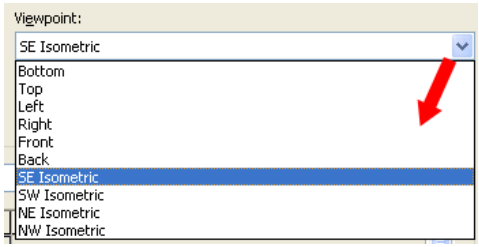


Fig. 12.66 Defining the viewpoint (of the object)

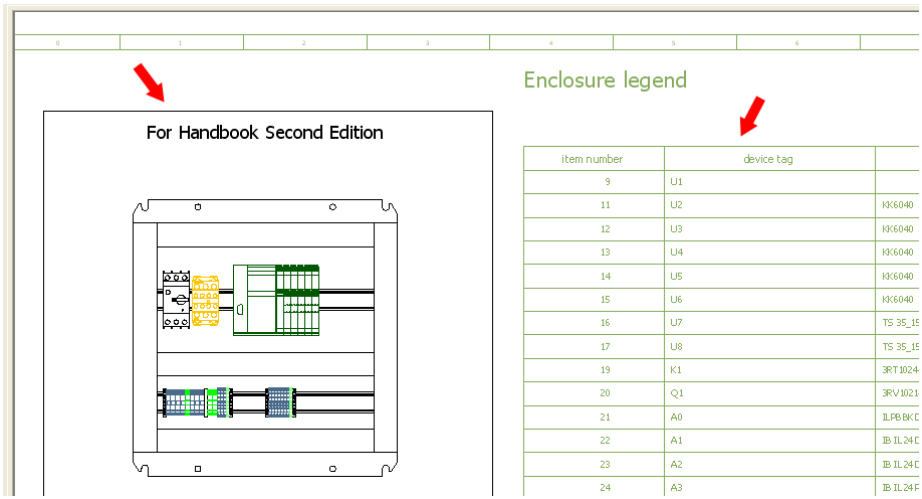


Fig. 12.67 Finished model view with an inserted enclosure legend

To open the 3D model from the 2D model view, select the 2D model view, then right click to open the popup menu and select **OPEN 3D VIEW**.

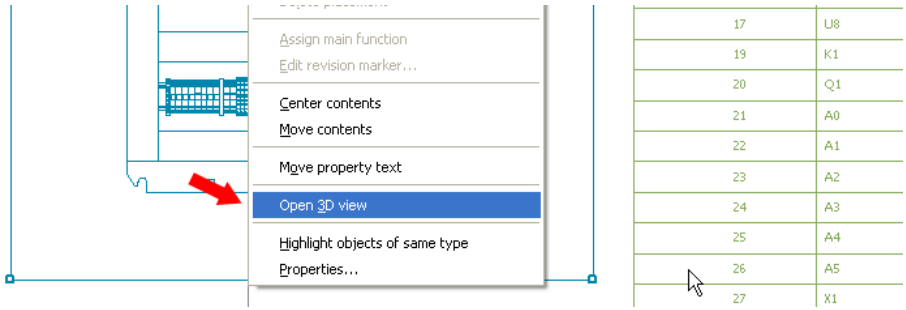


Fig. 12.68 Open 3D view menu item

EPLAN opens the 3D view.

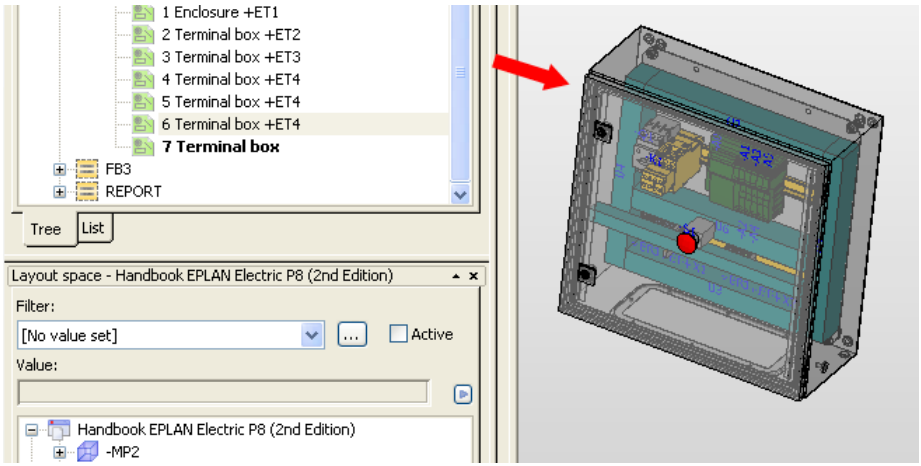


Fig. 12.69 Open 3D view (manual rotation angle)

12.1.4.5 Extensions to the Utilities/Reports menu

The model views that exist in a project can be updated using the **UTILITIES/REPORTS** menu.

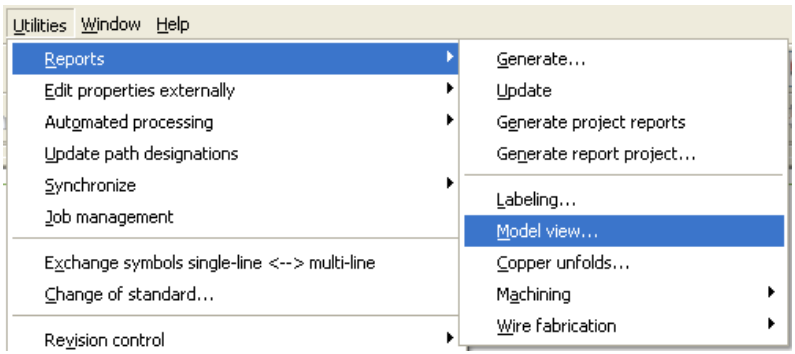


Fig. 12.70 Update model view

To update one or more model views in a project, you start the **MODEL VIEW** function found the **UTILITIES/REPORTS** menu. EPLAN opens the **MODEL VIEWS** dialog.

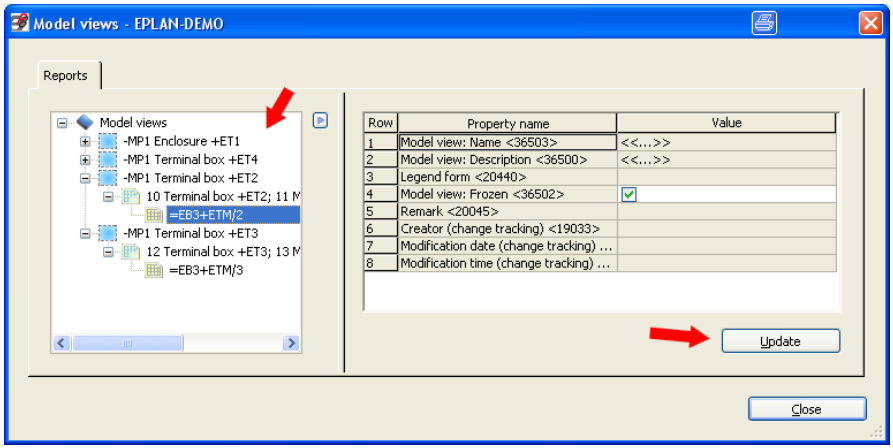


Fig. 12.71 The Model views dialog with the existing views in the project

To update one or more views, select the relevant reports (model views) in the left area of the dialog and then click the **UPDATE** button. All of the selected model views will then be brought up to date and have the same status as their "templates", the 3D views.

As is usual in EPLAN, you use a new **Enclosure legend** report to create a new report on the page with the module view. You can enter a legend form in the model view properties.

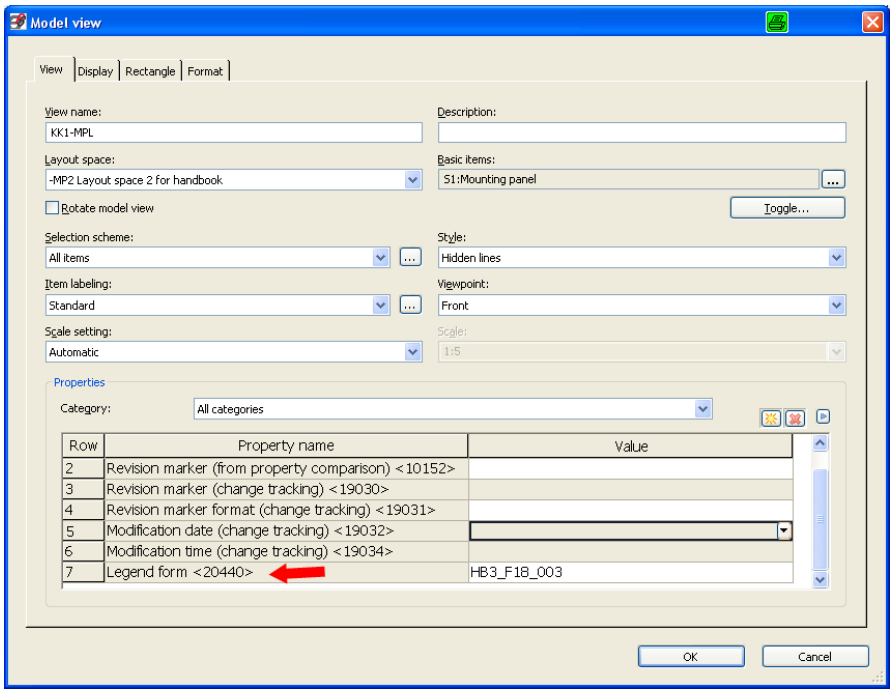


Fig. 12.72 Model view legend form

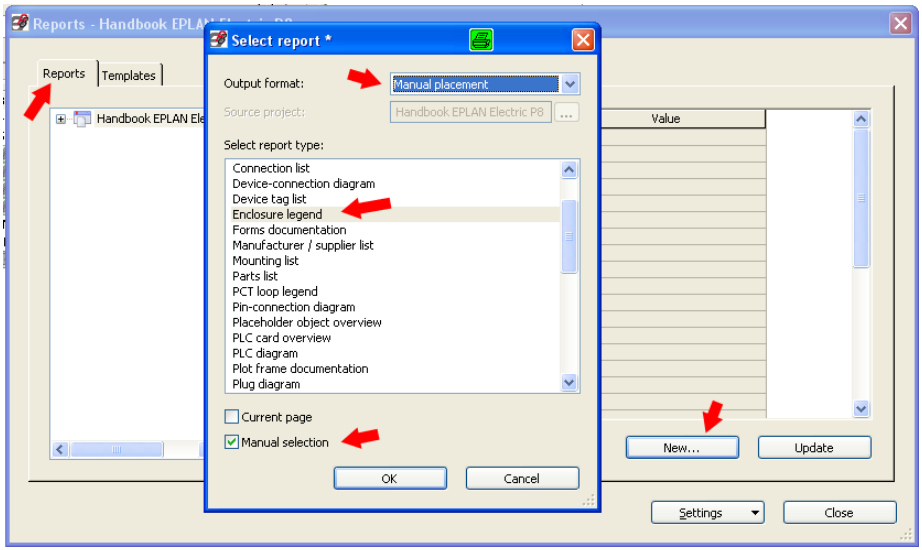


Fig. 12.73 Defaults for the output of legends

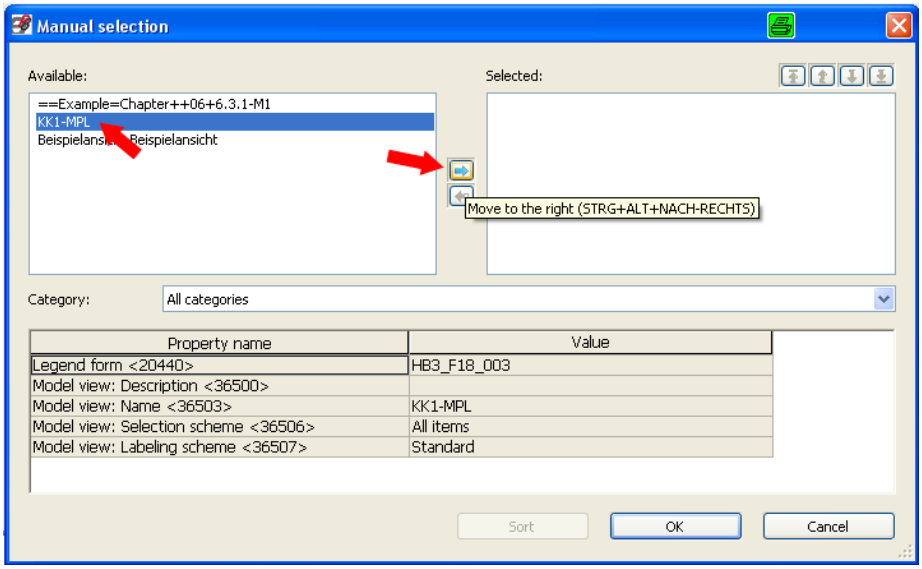


Fig. 12.74 Selecting a model view

After you click **OK** in the **MANUAL SELECTION** dialog, several prompts on filter and sorting criteria may be shown. Then EPLAN "hangs" the report on the cursor for placement. The report can now be placed, and later updated via the **REPORT/MODEL VIEW** menu.

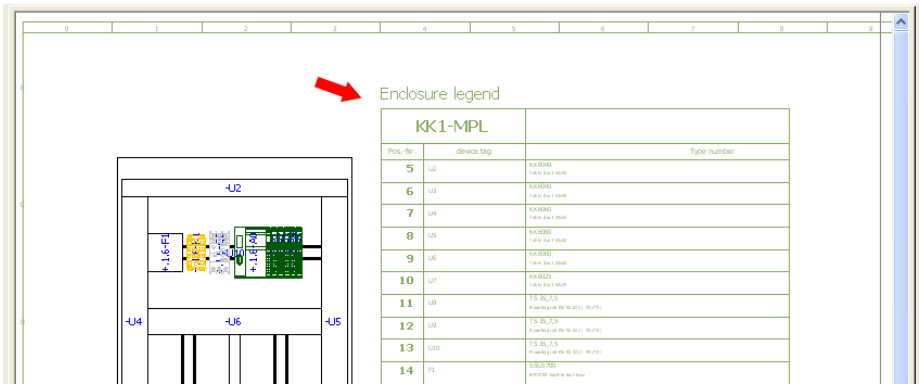


Fig. 12.75 Placed report

12.1.5 Navigators in EPLAN Pro Panel

In addition to the many toolbars, menus and menu extensions, EPLAN Pro Panel also has two new navigators.

12.1.5.1 Layout space navigator

As you know, a 3D view is not possible without a layout space. This means that the creation of layout spaces is mandatory. As described in section 12.1.4.1, this is done via the LAYOUT SPACE/NEW menu.

If a layout space was created, it will be in the **layout space navigator**. The **layout space navigator** can be opened via the LAYOUT SPACE/NAVIGATOR menu.

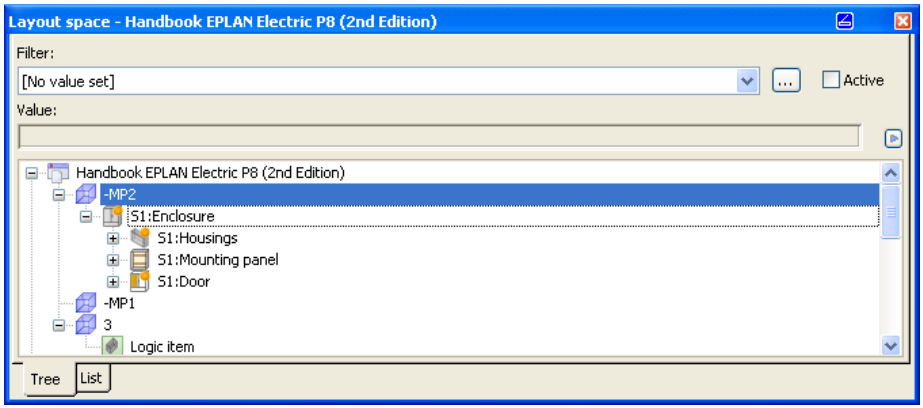


Fig. 12.76 Layout space navigator

The **layout space navigator** contains all of the objects that belong to a 3D layout. Depending on your needs, the **VIEW** can be set or changed in the **layout space navigator**. Simply right-click in the **layout space navigator** and select the **VIEW** menu item.

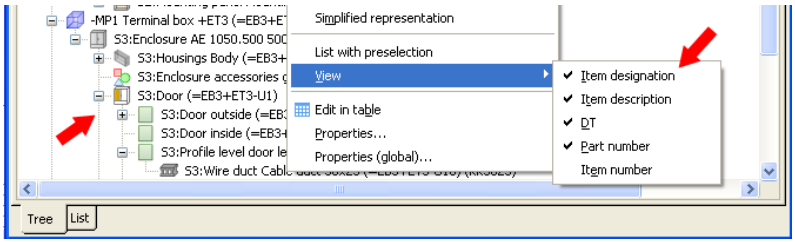


Fig. 12.77 The View entry in the popup menu

Besides the different views, the popup menu contains a number of additional functions for editing 3D models.

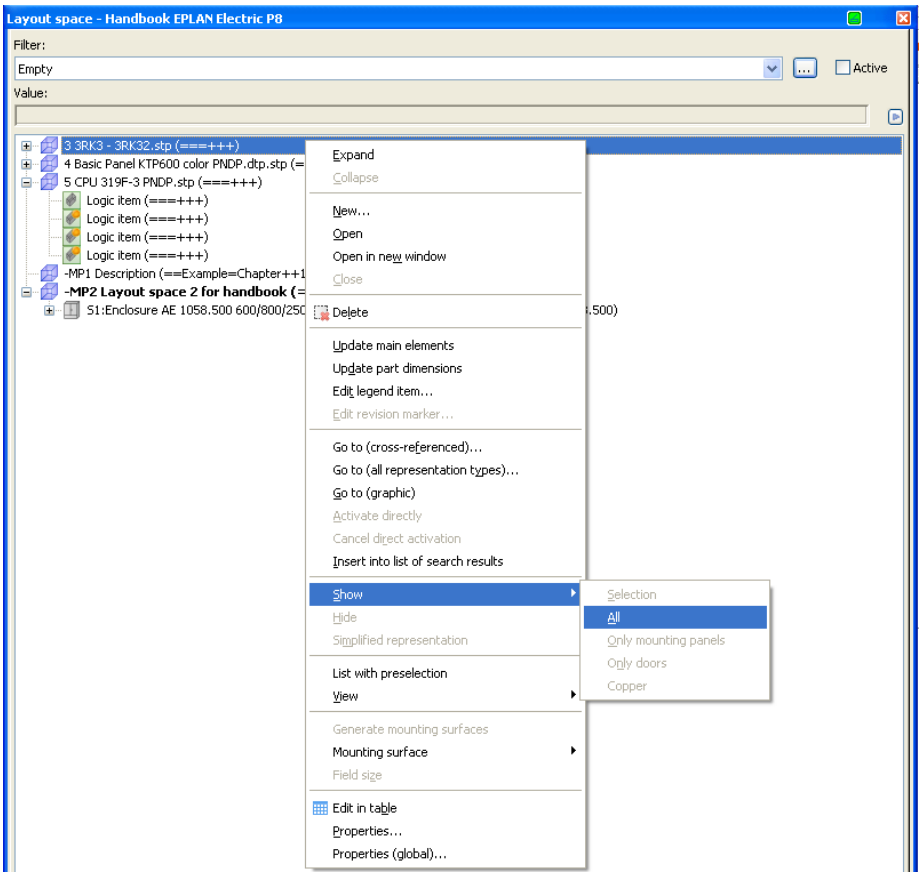


Fig. 12.78 Popup menu

In addition to the menu items found in the other navigators, such as **NEW** and **GO TO (GRAPHIC)**, the **Layout space navigator** has a few special menu items like the **ACTIVATE DIRECTLY** function that activates the 3D object that was selected in the **Layout space navigator**.

The **ACTIVATE DIRECTLY** function activates the selected mounting surface and all of the parts contained in it, and shows the mounting surface from the front in a 2D view. All of the objects that belong to this layout space are hidden.

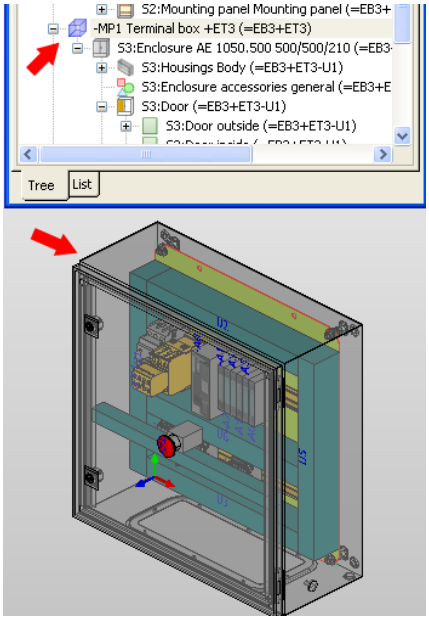


Fig. 12.79 Standard view

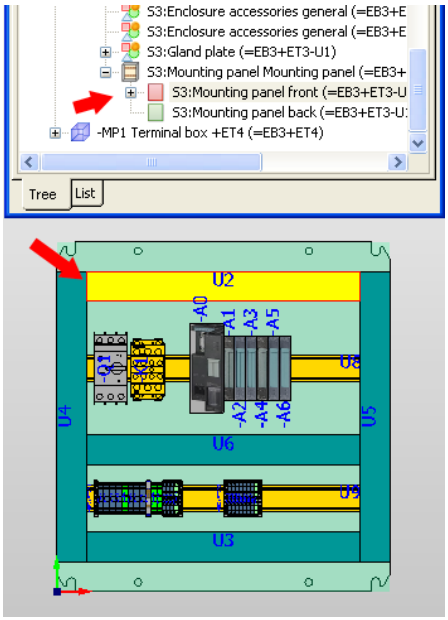


Fig. 12.80 Mounting panel directly activated

To undo the direct activation, select the **CANCEL DIRECT ACTIVATION** menu item in the popup menu. This sets the view back to standard.

As well as the functions named above, there are also the **SHOW** and **HIDE** functions. **HIDE** enables you to hide a layout space's complete contents or selected objects. **SHOW** lets you display all objects or just specific objects.

12.1.5.2 3D mounting layout navigator

Besides the **layout space navigator**, which displays among other things the end results of equipment and also contains all of the mechanical parts, such as enclosures, bases and orifices, there is also the **3D mounting layout navigator**.

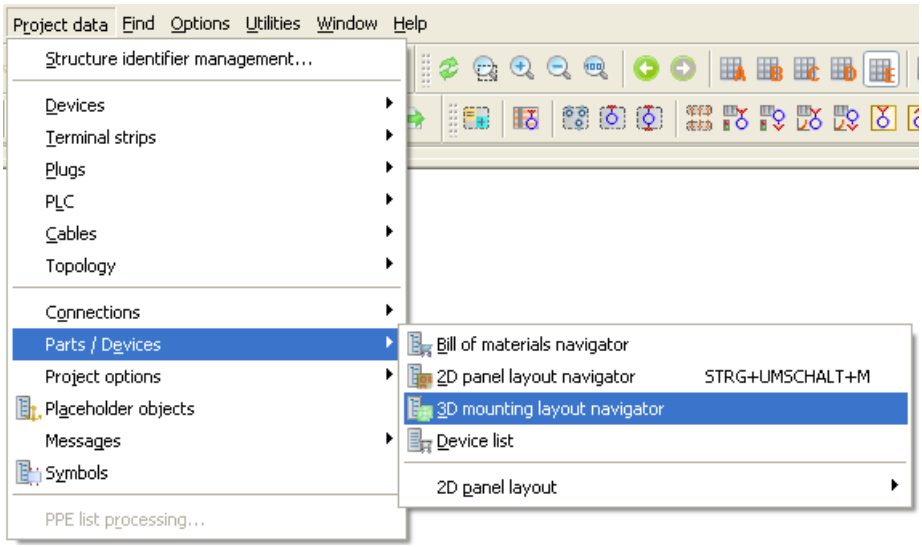


Fig. 12.81 Calling up the 3D mounting layout navigator

The **3D mounting layout navigator** has the device view of the project. This is also where all of the devices are listed and offered for placement in the layout space.

The navigator is started via the **PROJECT DATA/PARTS/DEVICES/3D MOUNTING LAYOUT NAVIGATOR** menu.

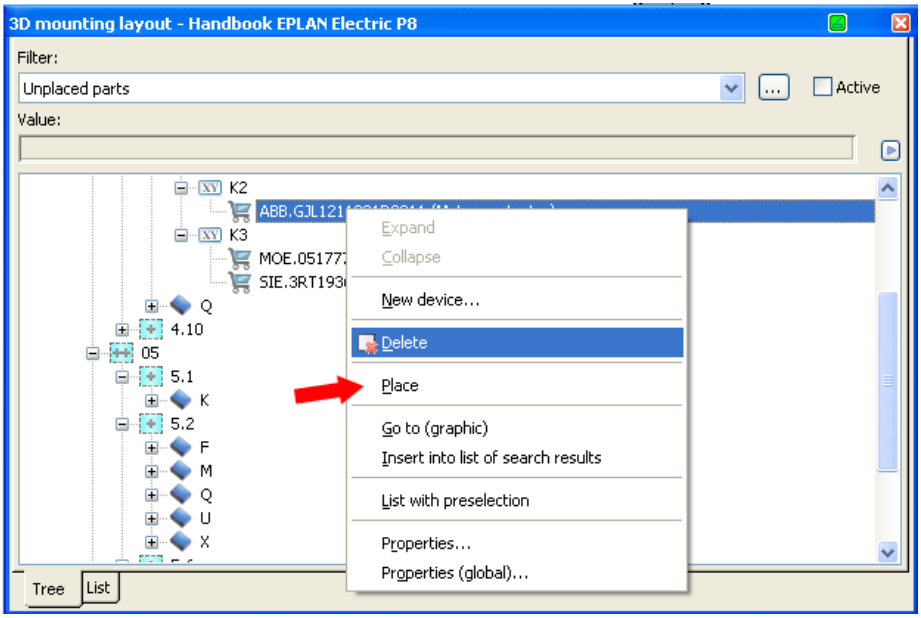


Fig. 12.82 The 3D mounting layout navigator

The desired devices from the **3D mounting layout navigator** can now be placed in the layout space.

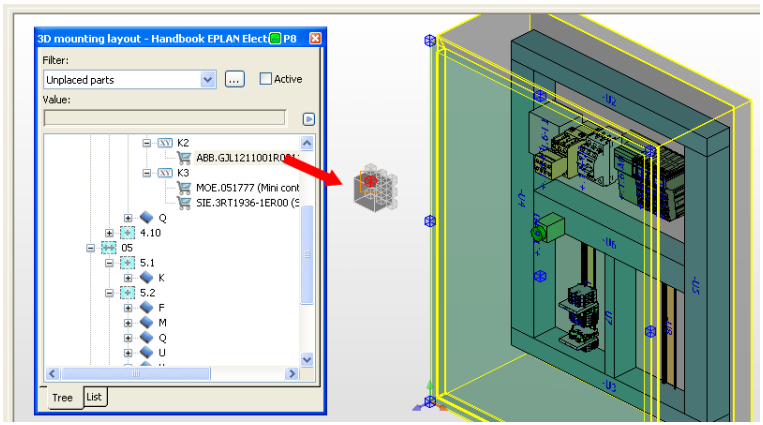


Fig. 12.83 An object to be placed "hangs" on the cursor.

You select one or more devices, right click to open the popup menu and then select **PLACE**. The selected devices now "hang" on the cursor and can be placed in the layout space.

NOTE: The **3D mounting layout navigator** only displays those devices that have a part entry. It is irrelevant whether this part has additional technical data, such as width, height, macro data, or the like.

If the parts do not contain the data needed to place them, EPLAN displays a message and stops the placement.

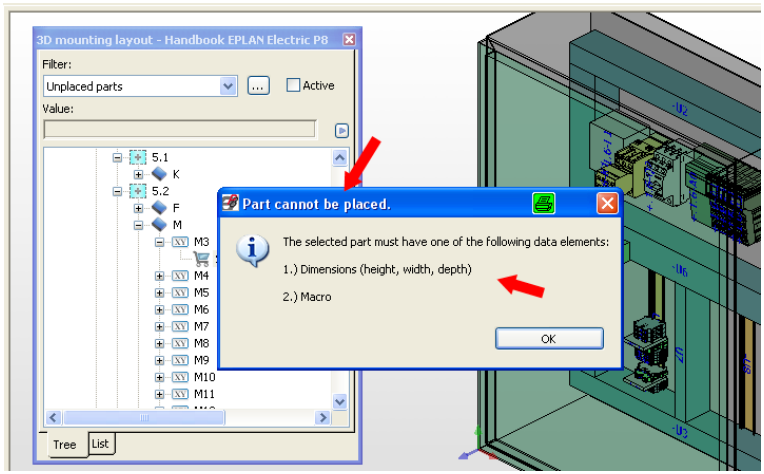


Fig. 12.84 Parts data required for placement is missing

12.1.6 Settings for EPLAN Pro Panel

There are a few general, project-specific and user-specific settings for EPLAN Pro Panel.

The **Project-specific settings** are entered in the OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/MANAGEMENT/3D IMPORT menu.

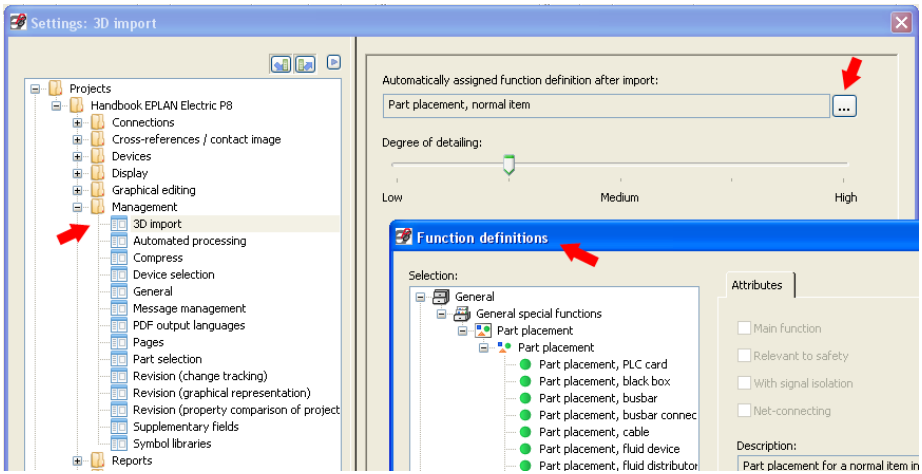


Fig. 12.85 Project-specific setting (1)

This is where the automatic function definition required for a 3D import is defined.

There is also a **second project-specific setting**. This setting can be used define the background colors of the model views. It is found under the OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/GRAPHICAL EDITING/GENERAL options.

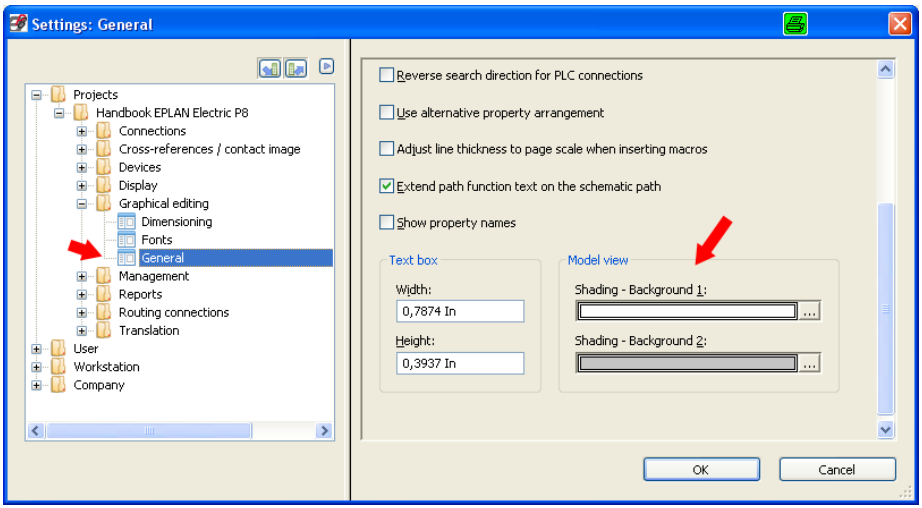


Fig. 12.86 Project-specific setting (2)

The **user-specific settings** for the 3D area are found in **OPTIONS/SETTINGS/USER/GRAPHICAL EDITING/3D**.

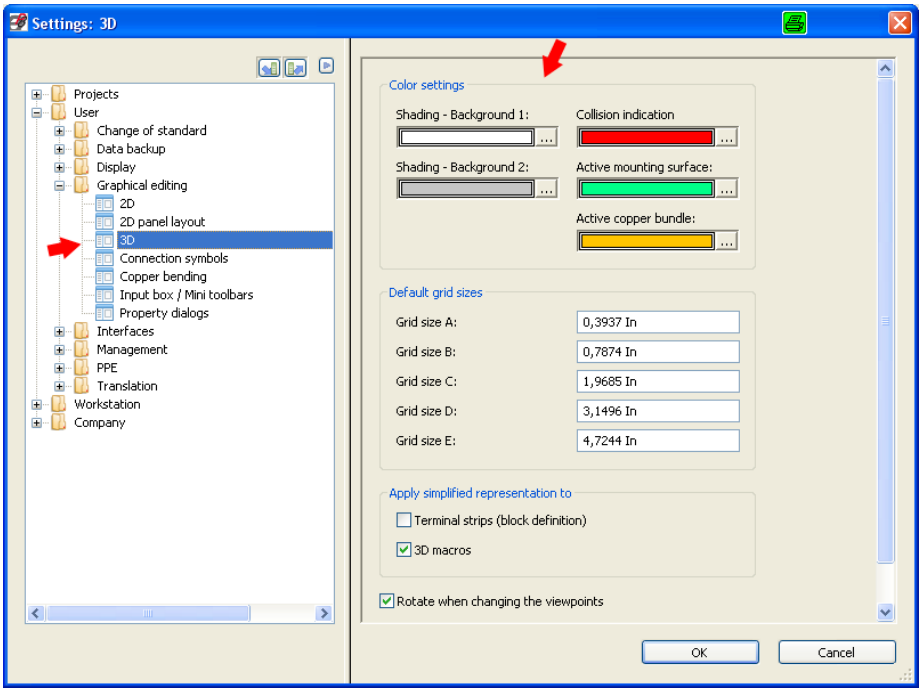


Fig. 12.87 User-specific setting (1)

You can select color settings for backgrounds or other grid sizes here. This is also where the default setting is located for how terminal strips (individual terminals) or 3D macros in simplified representation should be handled.

In addition to these settings, it is also possible to show an input box while working in the 3D area. This setting is located under **OPTIONS/SETTINGS/USER/GRAPHICAL EDITING/INPUT BOX/MINI TOOLBARS**.

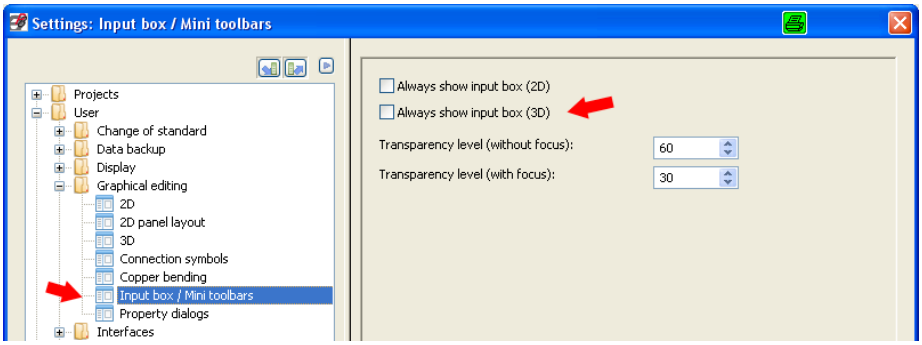


Fig. 12.88 User-specific setting (2)

12.1.7 A practical example with EPLAN Pro Panel

Now that you have been given a brief overview of the functions and how they work, this section uses a short and simple example to demonstrate how to use EPLAN Pro Panel. We will proceed step by step so that you can follow along.

NOTE: The prerequisite for a 3D layout is that the project has devices with associated parts that have the minimal requirements in terms of technical data, such as width, height and depth.

Step 1: You start by using the **LAYOUT SPACE/NEW** menu to create a layout space. Give the layout space a **designation**, a **description** and a **structure identifier**.

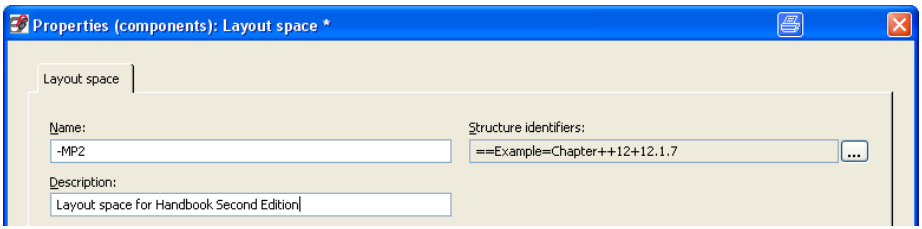


Fig. 12.89 A new layout space was created.

Step 2: Now open the layout space (**LAYOUT SPACE/OPEN** menu), select the desired layout space in the **layout space navigator**, and then use the **INSERT/ENCLOSURE** menu to select and apply terminal box AE 1058.500 from **parts management** and insert it into the layout space.

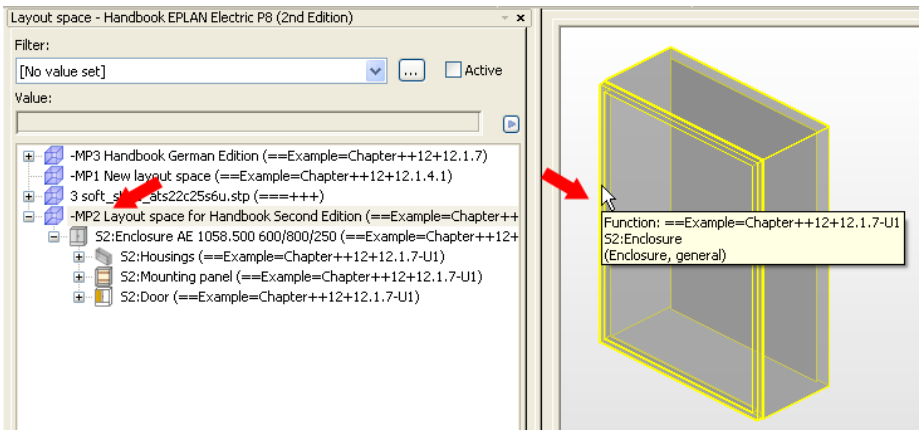


Fig. 12.90 Inserted terminal box

Step 3: Now you will place a few cable ducts on the mounting panel of the terminal box. First you directly activate the mounting panel in the **layout space navigator** so that the front view is visible in the 3D view. In the **layout space navigator**, select *mounting panel front* and then right click to open the popup menu and select ACTIVATE DIRECTLY.

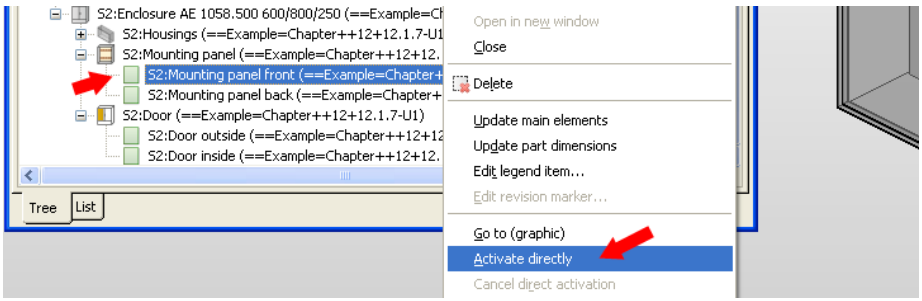


Fig. 12.91 Selecting the mounting panel front.

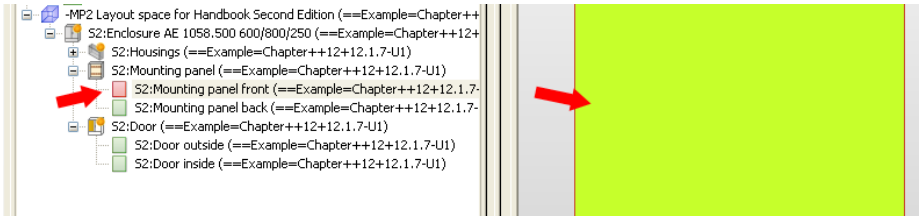


Fig. 12.92 The mounting panel front is now activated.

Step 4: Now you select a cable duct from **parts management** via the INSERT/WIRE DUCT menu and place it on the mounting panel. Use the popup menu to call up the PLACEMENT OPTIONS. The upper cable duct should be placed 50 mm from the upper edge and 10 mm from the left edge of the mounting panel. Then cable duct will be extended and placed 10 millimeters from the right edge.

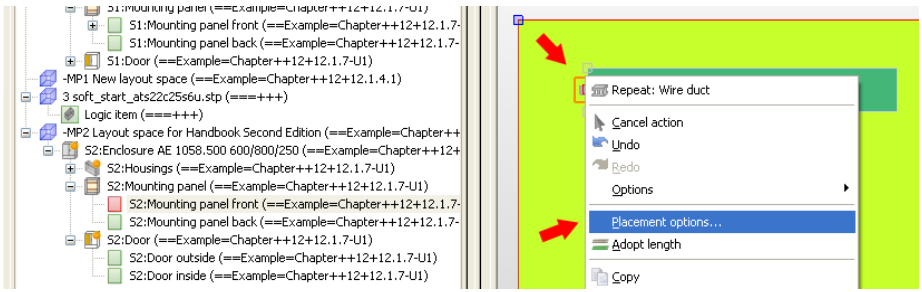


Fig. 12.93 Placing the cable duct via Placement options

Click the PLACEMENT OPTIONS menu item and for the first placement point, set the handle and the X and Y offsets that are to be used for calculating the offsets.

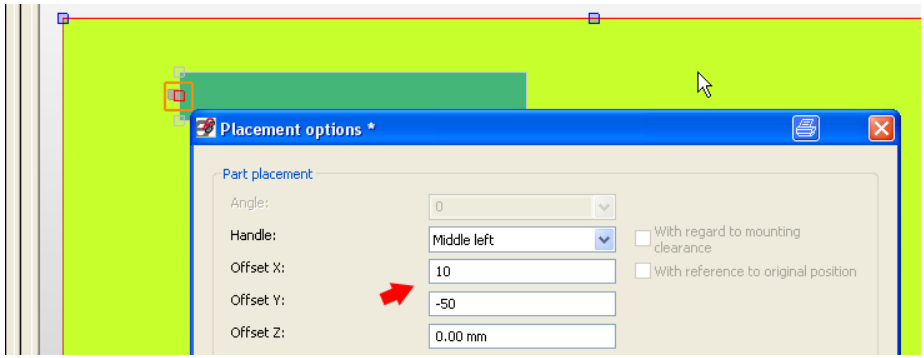


Fig. 12.94 Setting placement options for the cable duct (first handle)

Exit the dialog with **OK**. Now set the handle to the upper left corner of the mounting panel by clicking this corner.

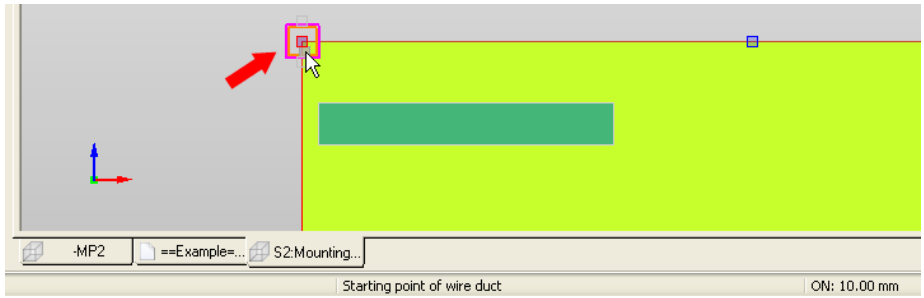


Fig. 12.95 The first handle is placed.

Using the popup menu, call up the **PLACEMENT OPTIONS** dialog again and enter the offsets for the second placement.

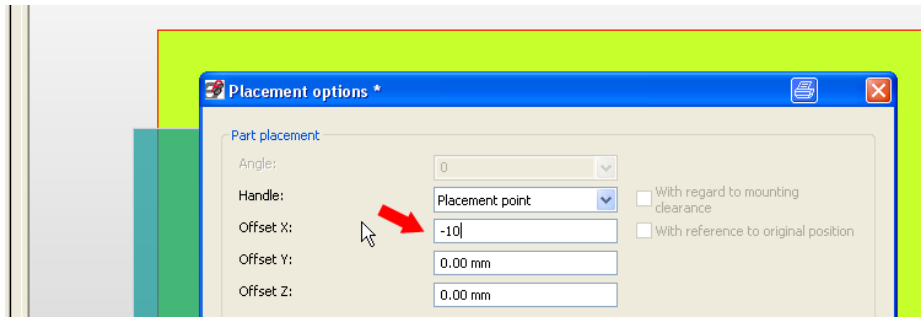


Fig. 12.96 Setting the second handle.

Now exit the **PLACEMENT OPTIONS** dialog and left-click the upper right corner of the mounting panel. EPLAN Pro Panel now places the cable duct as desired.

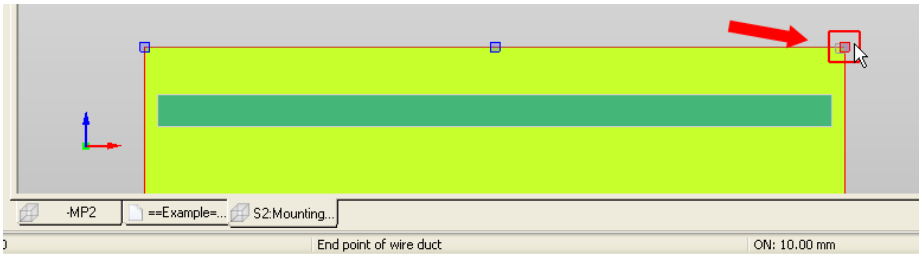


Fig. 12.97 The second handle is placed.

Step 5: The lower cable duct is placed with the same steps. The offset settings for the first handle are: Offset X = 10 mm; Offset Y mm; handle = lower left. The offset settings for the second handle are Offset X = −10 mm; Offset Y = 0 mm.

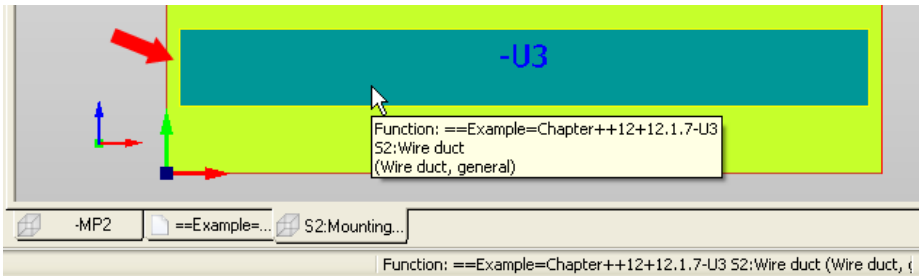


Fig. 12.98 Placed second duct

Step 6: Now you will connect the left and right side of the cable duct to the upper and lower cable ducts. Use the **INSERT/WIRE DUCT** menu to select and apply a duct from **parts management**. The cable duct now "hangs" on the cursor. Use the A key to switch to one of the outside handles and then left-click one end of the existing duct.

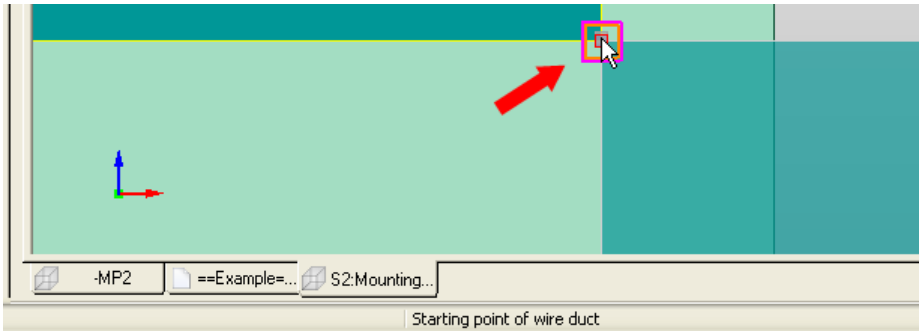


Fig. 12.99 Placing a new duct

Now drag the duct down with the mouse and place the duct on the external handles of the existing duct (left click). This also places this duct.

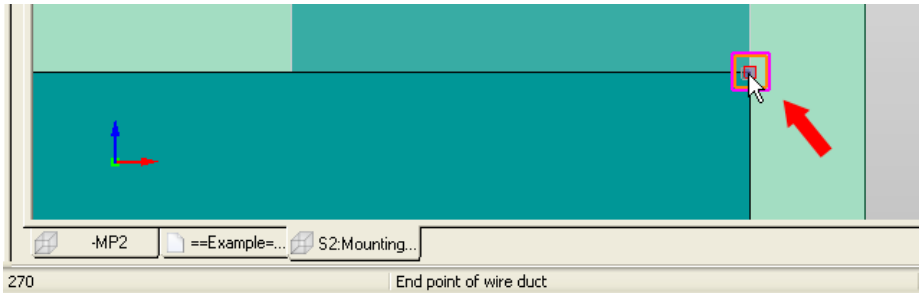


Fig. 12.100 The new duct is placed.

Step 7: Now you will place a cable duct in the center between the cable ducts. This is relatively easy because EPLAN finds the centered handles on the existing cable ducts on its own. Select a duct from **parts management** via the **INSERT/WIRE DUCT** menu and then place the duct on the center connecting point of the left duct.

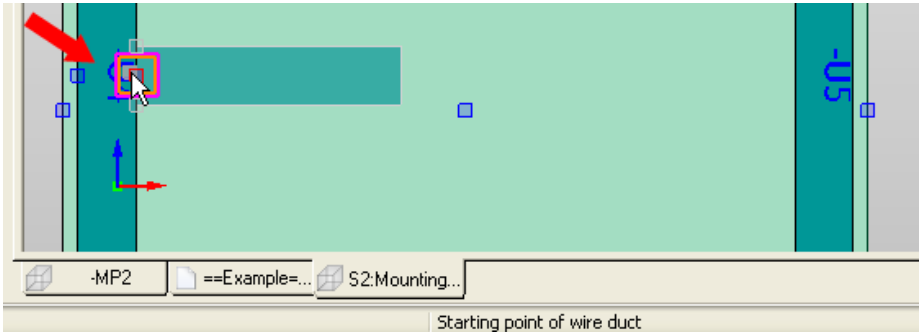


Fig. 12.101 New duct "snaps" into place on the handle

Now use the mouse to drag this duct to the right onto the right duct. It will again "snap" into place on the center handle.

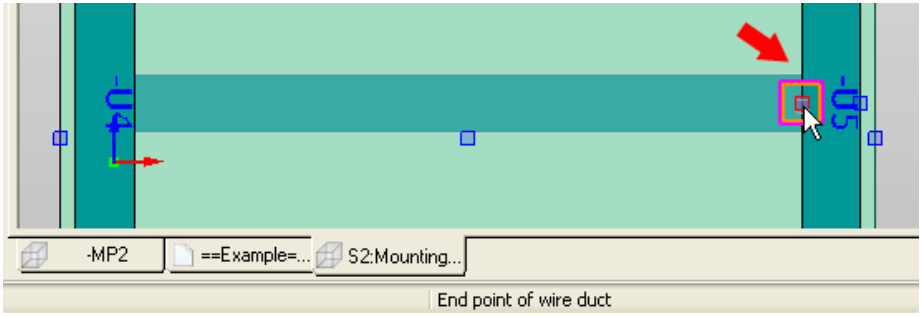


Fig. 12.102 Connecting the new duct with the right duct

This finishes the placement of the center duct, which sits precisely in the center between the upper and lower duct.

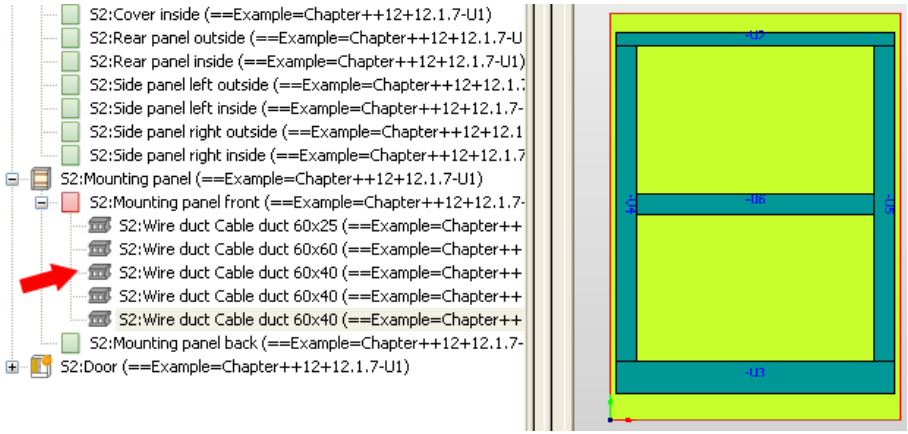


Fig. 12.103 Overall view of mounting panel

Step 8: Now you will place another duct in the lower area, exactly in the center between the left and right ducts. Select a cable duct from **Parts management**. This duct will be placed in the center of **cable duct –U6**.

The selected cable duct "hangs" on the cursor. Now open the **PRO PANEL OPTIONS** menu and click the **ADOPT LENGTH** button. Then click **cable duct –U4**.

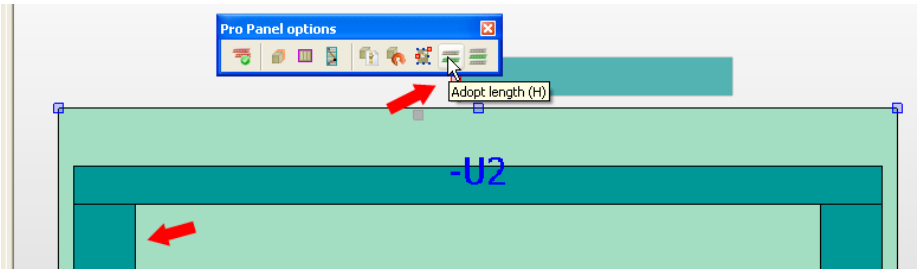


Fig. 12.104 Adopt length

In the **PRO PANEL OPTIONS** menu, click the **PLACE CENTERED** button and then click **cable duct –U5**. EPLAN Pro Panel now places the new duct exactly in the center.

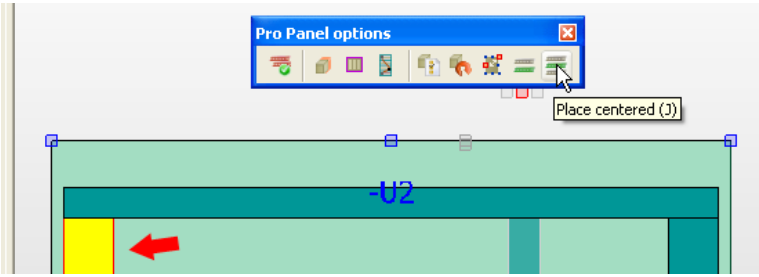


Fig. 12.105 Place centered

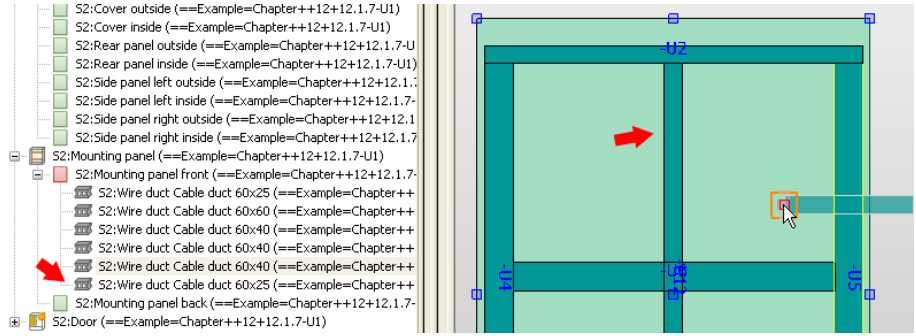


Fig. 12.106 Placed duct

Step 9: Of course, this duct is too long, and if the **COLLISION CHECK** in the **PRO PANEL OPTIONS** was turned on, it would not be possible to place the duct because it "cuts" the center duct. This is why you will now trim, or shorten, the upper part of the duct.

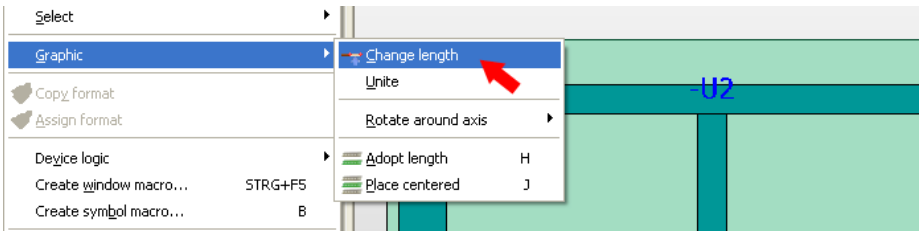


Fig. 12.107 Change length

Call up the **CHANGE LENGTH** menu item in the **EDIT/GRAPHIC** menu and click the center duct.

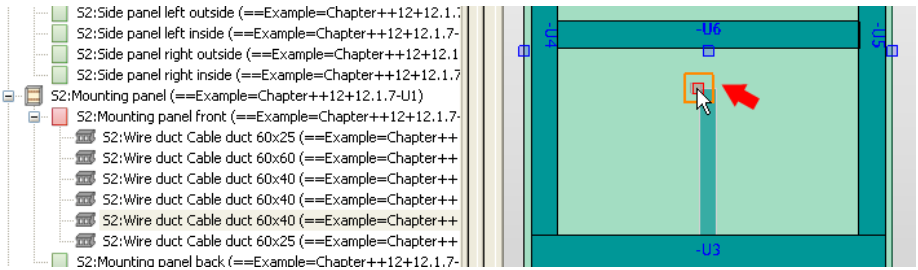


Fig. 12.108 Changing the duct length

You can easily change the length of the duct by clicking it and then dragging it with the left mouse button. Now drag the handle to the –U6 duct and let it "snap" into place there. This shortens the duct, and all of the desired cable ducts have now been placed.

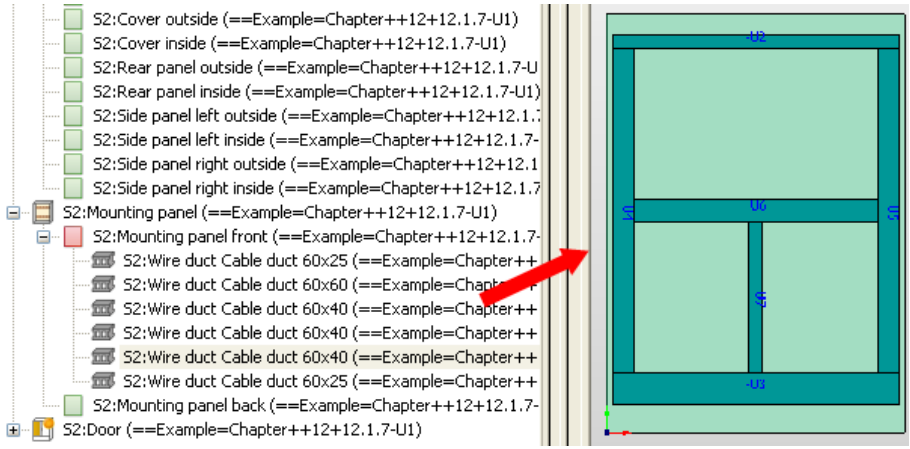


Fig. 12.109 Finished mounting panel equipped with cable ducts

Step 10: Now you will place a few mounting rails. Mounting rails are placed in the same way as cable ducts, i. e. you select them from **parts management** in the **INSERT/MOUNTING RAILS** menu and place them on the mounting panel. Use the handles on the ducts, or the **ADOPT LENGTH** and **PLACE CENTERED** functions.

The mounting panel is equipped with mounting rails is shown in Fig. 12.110.

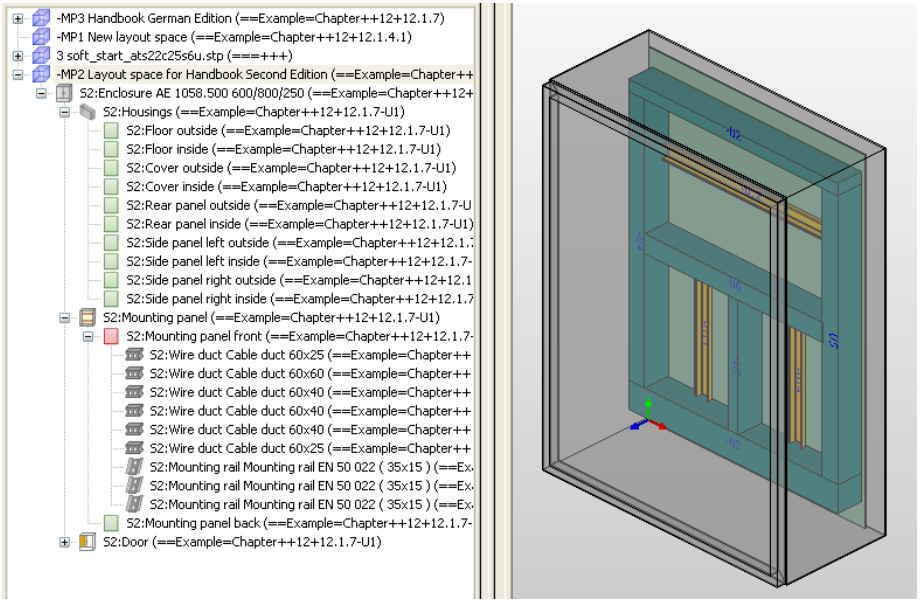


Fig. 12.110 Mounting panel equipped with ducts and mounting rails

Step 11: After you have placed the cable ducts and mounting rails on the mounting panel, you can add devices.

Open the **PROJECT DATA/PARTS/DEVICES** menu in the **3D MOUNTING LAYOUT NAVIGATOR**. Set the mounting panel to the **3D VIEWPOINT FRONT** view.

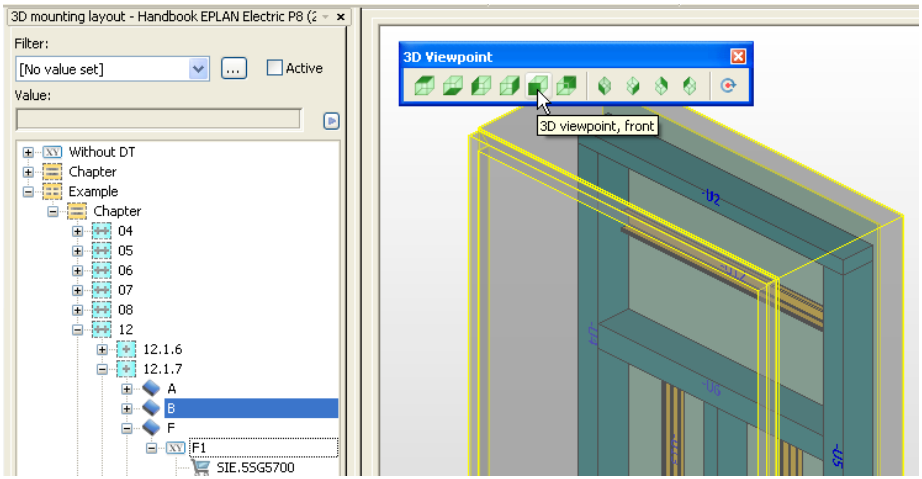


Fig. 12.111 Open 3D mounting layout navigator

Step 12: Now the devices that exist in the project and that are listed in the **3D mounting layout navigator** can be placed on the mounting panel.

Select the devices in the navigator and right-click to select the **PLACE** menu item from the popup menu.

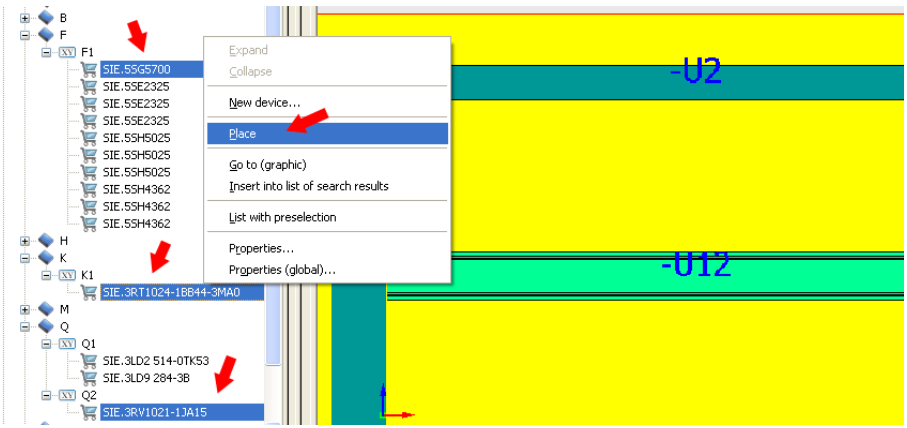


Fig. 12.112 Devices to be placed are selected in the navigator

When you click **PLACE**, the devices "hang" on the cursor and can be "snapped" onto the mounting rails. Before you place the devices, you can call up the **PLACEMENT OPTIONS** once more and enter the desired placement options, either by right-clicking to open the popup menu or via the corresponding button.

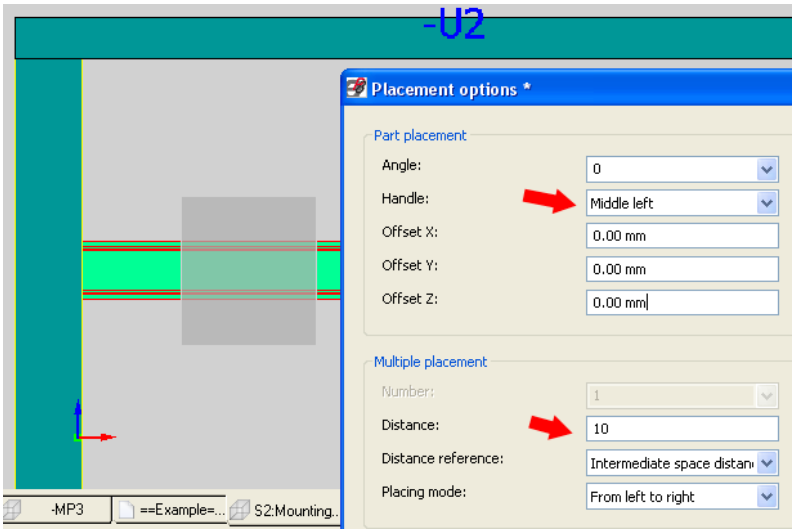


Fig. 12.113
Defining placement options

After you click **OK** and place the first handle on the center of the mounting rail (EPLAN "snaps" it in place), all of the devices with set placement options are placed. The same thing happens to the other devices that belong on this mounting rail (see step 13).

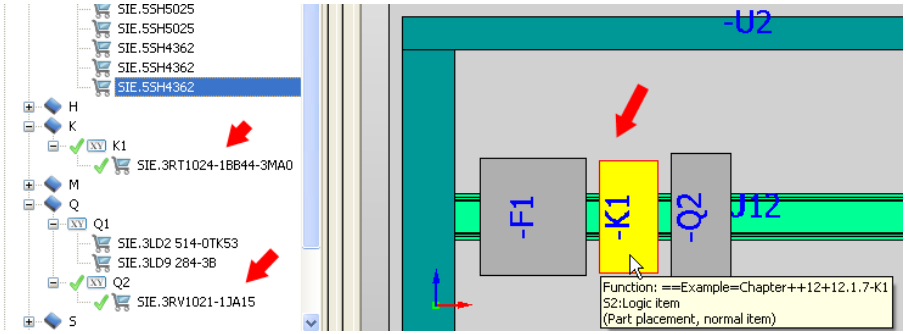


Fig. 12.114
The first placed devices

EPLAN displays a small green check mark next to the device in the navigator to visually indicate that the device has been placed.

Step 13: Now you can place the other devices. The terminal strips are to be placed upright in the lower area on the two shorter mounting rails. Select the terminals in the navigator and use the popup menu to call up the **PLACE** function.

Fig. 12.115
Prompt asking how the terminals should be placed

Before it places the terminals, the **PLACE PARTS OF A TERMINAL STRIP** dialog opens asking how the terminals should be placed. If you answer this dialog with **YES**, then after you place the first terminal on the mounting rail EPLAN will automatically place the rest of the selected terminals. If you answer with **NO**, then you can place all of the terminals one by one on the mounting rail.

For this example, answer the prompt with **YES**. Before you place the first terminal, use the popup menu to call up the **PLACEMENT OPTIONS**. You define the placing mode here: the terminals should be placed from bottom to top.

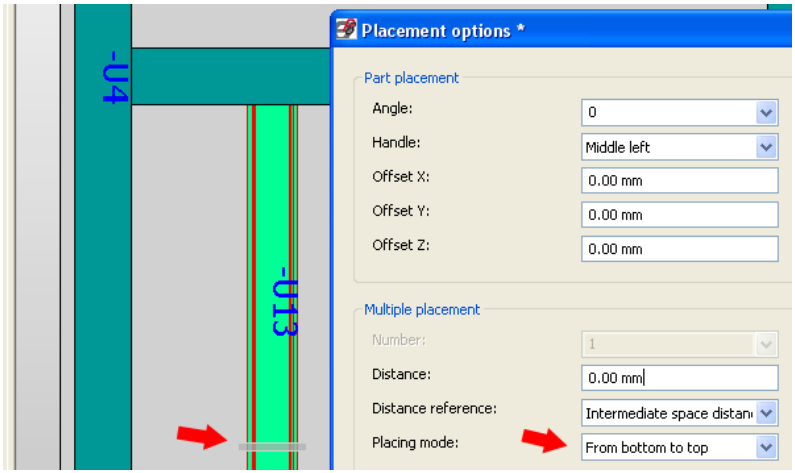


Fig. 12.116 Defining the placing mode for terminals

Then you can leave the dialog by clicking **OK**, and the first terminal can be placed. EPLAN Pro Panel then automatically arranges all of the other selected terminals at this first terminal. The same process applies to the next terminals. You select them and place them on the mounting rail. Now all of the terminals have been placed.

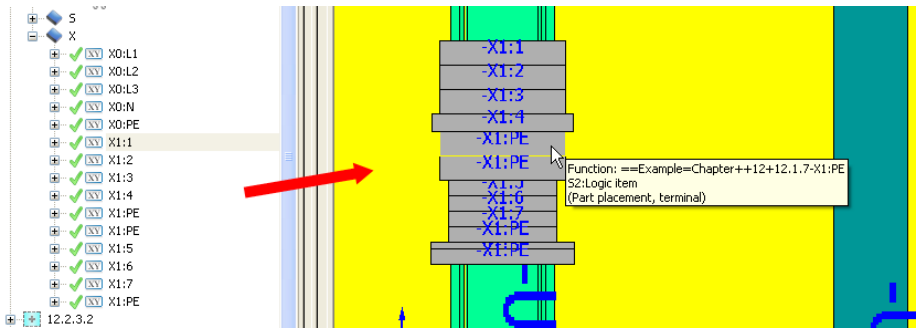


Fig. 12.117 Placed terminals

Step 14: Now you will add the signal lamp to the door. You directly activate the door's exterior in the **layout space navigator** and set the view to **3D VIEWPOINT FRONT**.

Now select the signal lamp in the **3D mounting layout navigator**, call up the **PLACE** function, and then right-click the **PLACEMENT OPTIONS**. Enter a distance of 300 mm from the upper edge of the door.

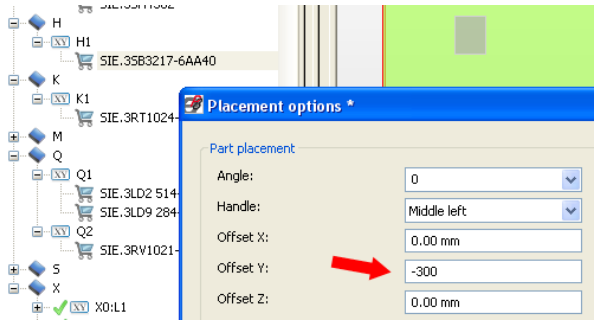


Fig. 12.118 Before placing the signal lamp

Confirmed the dialog with **OK**. Now you "grab" the middle of the upper edge with the "snap" function and left click. EPLAN places the signal lamp in the center at a distance of 300 mm from the edge.

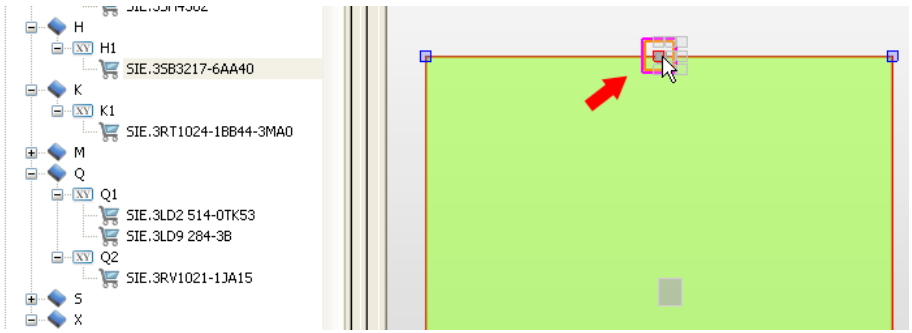


Fig. 12.119 "Grabbing" the center

Fig. 12.120 shows the result in the 3D view.

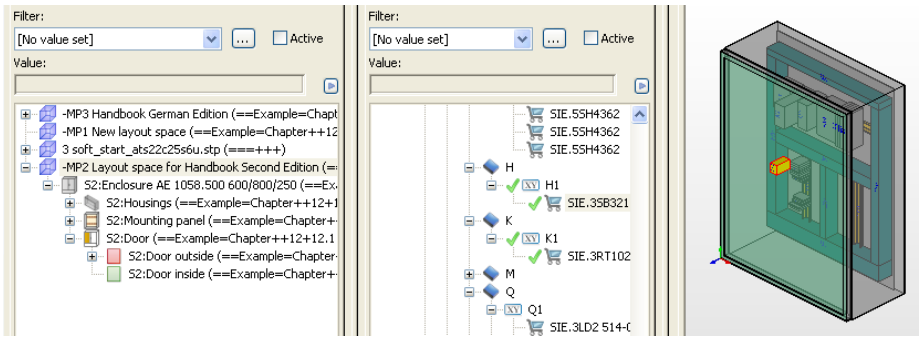


Fig. 12.120 Result: equipped enclosure

Step 15: To complete the documentation, you now generate a 2D model view from the 3D view. You can close the **Layout space** and the **3D mounting layout**. Then you switch to the graphical editor and create a **model view** page.

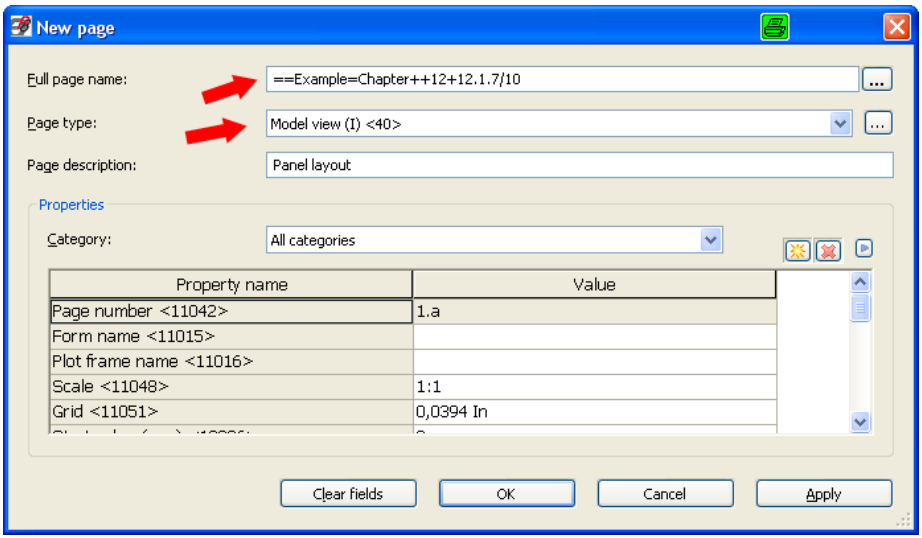


Fig. 12.121 Creating a new model view page type

Step 16: Open this page and use the **INSERT/GRAPHIC/MODEL VIEW (PRO PANEL)** menu to insert a **model view**. The **MODEL VIEW** dialog opens after placement.

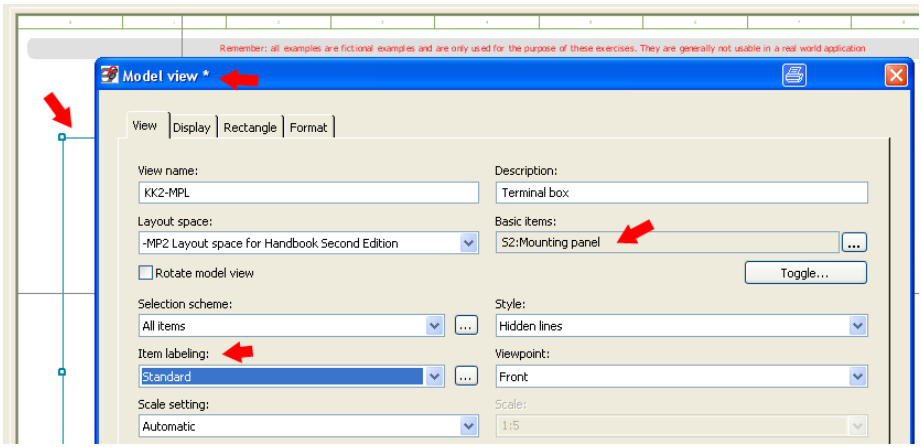


Fig. 12.122 Settings for generating the model view

In this dialog, you set the **view name**, the **description**, the **layout space** to be displayed, and its **basic items** (here the *mounting panel*). Set the scheme for **item labeling** to *default*. Click **OK** to close the dialog. EPLAN generates the 2D model view.

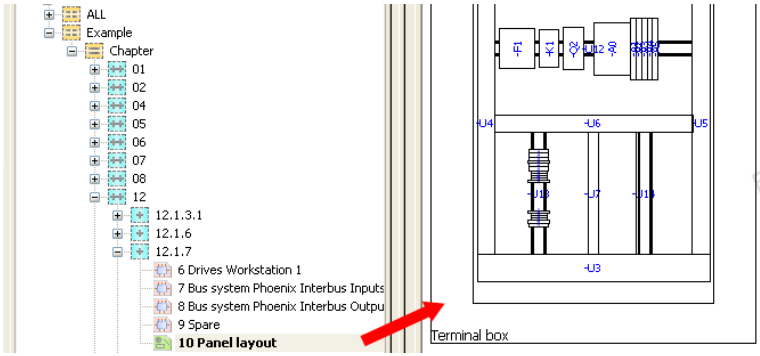


Fig. 12.123 Partial view of 2D model view

Step 17: Now you will place an enclosure legend on the page. To place an enclosure legend, open the **properties** of the **MODEL VIEW** (double-click the model view or use the keyboard shortcut **CTRL + D**) and choose a form for the *Legend form <20440>* **property name**.

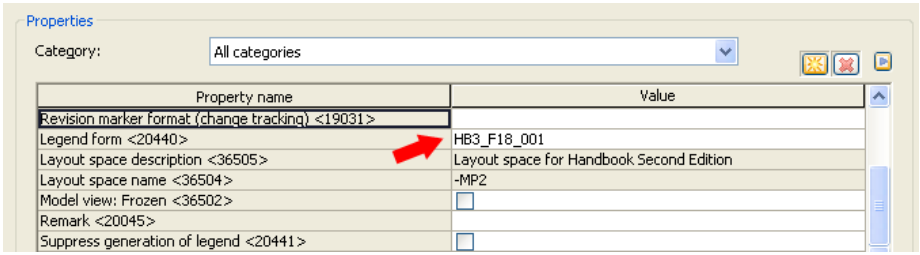


Fig. 12.124 Defining the legend form for this model view

Now you can manually place the **enclosure legend** report on this page using standard EPLAN Electric P8 procedure.

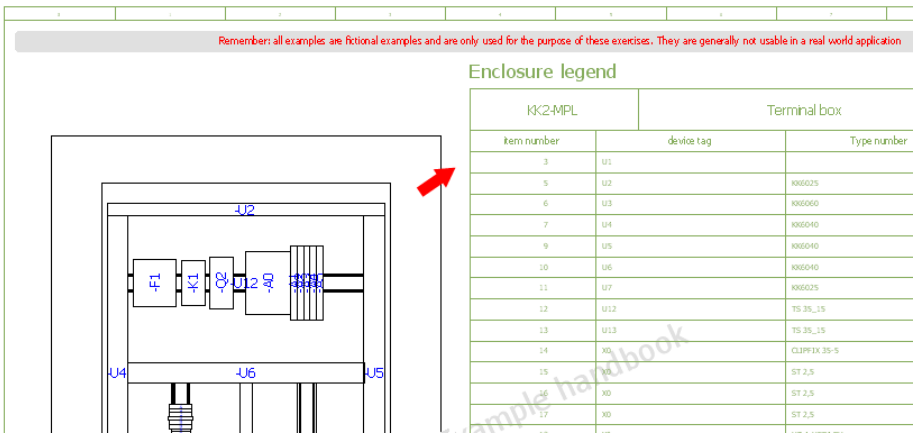


Fig. 12.125 Mounting panel with enclosure legend

12.2 EPLAN Data Portal

The previous sections mentioned the necessity of correct function templates (the individual function definitions of devices), device selection and finished macros. Function templates, among other things, are a major focus when working with EPLAN Electric P8. Users could, of course, fill their own parts with this additional information themselves, or they can use the EPLAN Data Portal solution.

12.2.1 What are the advantages of the EPLAN Data Portal?

The EPLAN Data Portal provides online access to parts data that can be directly used during current and subsequent project planning. This parts data is usually provided by the manufacturer, is subjected to an incoming inspection by EPLAN (whereby EPLAN as a software manufacturer cannot, of course, check the correctness of the data provided by the manufacturer), and is then provided online in the EPLAN Data Portal. This allows users to directly use parts (devices, macros, etc.) from the EPLAN Data Portal, without having to add them to, or create them in their own parts management.

12.2.1.1 Prerequisites for use

A valid Software Maintenance Contract must exist in order to use the EPLAN Data Portal. The EPLAN Data Portal can then be used as desired by providing data from the contract.

EPLAN distinguishes specific user groups on the basis of their Software Maintenance Contract levels, but this does not play a major role in the use of the EPLAN Data Portal. As always, the more comprehensive and more precise the Software Maintenance Contract, the greater the number of options are for using the EPLAN Data Portal.

The following sections will be limited to standard use.

NOTE: The EPLAN Data Portal is automatically installed when, for example, EPLAN Electric P8 is installed. Separate installation is not necessary.

12.2.2 Before the first start

An account must be created in order to use the EPLAN Data Portal. To do this, you start the **OPTIONS/SETTINGS/USER/MANAGEMENT/DATA PORTAL** menu item.

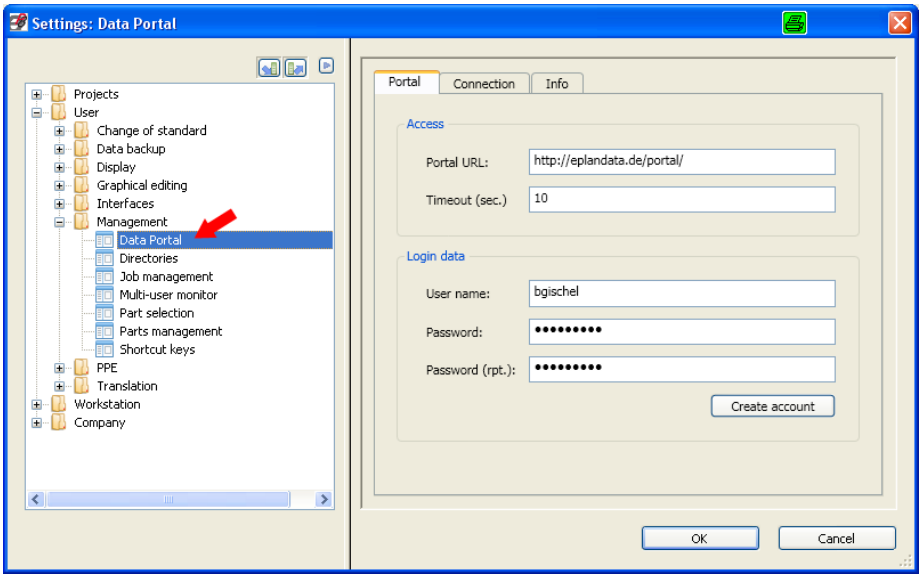


Fig. 12.126 EPLAN Data Portal settings

The settings for the EPLAN Data Portal are subdivided into several tabs. **Access** and **login data** are defined in the *Portal* tab (default setting already entered by EPLAN – does not need to be changed). This creates an account, and the user can use the Data Portal after completing the prompts.

The settings on the *Connection* tab normally do not need to be changed.

12.2.2.1 Data Portal navigator

The **Data Portal navigator** is the main focus of the EPLAN Data Portal. It is started via the UTILITIES/DATA PORTAL NAVIGATOR menu.

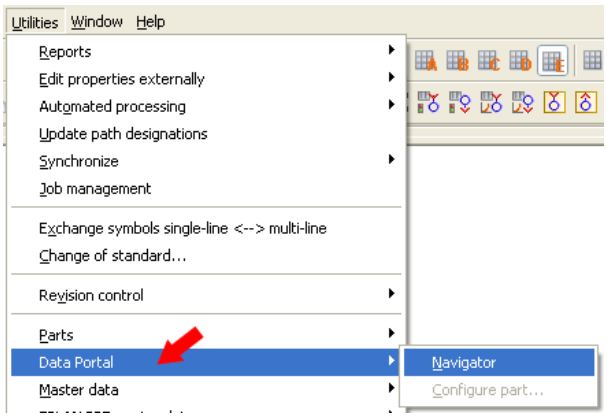


Fig. 12.127 The data portal navigator menu item

Like elsewhere in EPLAN, the navigator can be docked and undocked anywhere. This means that it can be freely placed on the desktop or docked to any desired position.

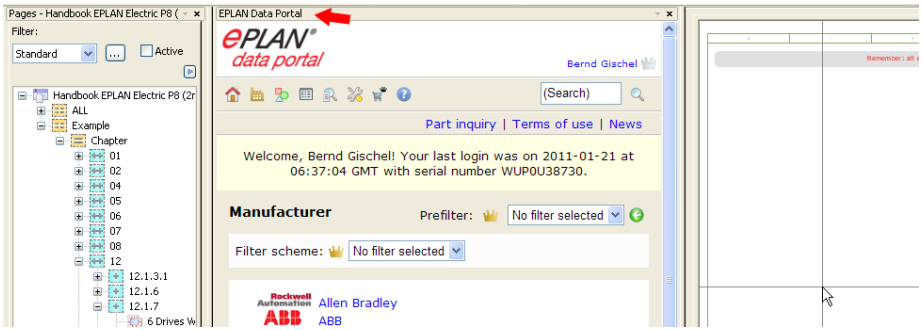


Fig. 12.128 The Data Portal navigator

The navigation/toolbar, with buttons for functions such as **DISPLAY MANUFACTURERS**, **SEARCH**, **SETTINGS** and **ONLINE HELP**, is located under the navigator dialog logo.

Fig. 12.129 The data portal navigator's navigation/toolbar

The navigation bar buttons have the following meanings:

	Open start page		Display manufacturer
	Display catalog		Display list
	Display search		Display settings
	Display shopping cart		Display help

Located a little bit "apart" from the other buttons is the button that takes you back to the previously displayed page in the navigator.

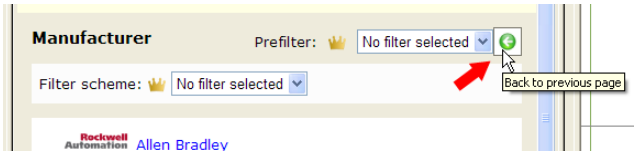


Fig. 12.130 Back button

Below the navigation/toolbar is the navigator's actual "data area". The first time the navigator is started, a list of the manufacturers in the EPLAN Data Portal is displayed here. You can click each manufacturer. The Portal opens the corresponding pages with the current manufacturer data. You can continue to navigate through this data.

Fig. 12.131 Manufacturers in the Data Portal

12.2.3 How the EPLAN Data Portal works

How does the EPLAN Data Portal work? It is very simple. While you are designing, you can use the Portal to insert missing parts data or macros onto schematic pages and/or parts management. This, of course, assumes that the missing data is actually available in the Portal.

12.2.3.1 Insert parts, macros and data from the Portal

A brief example will show how you can find a PNOZ device in the Portal and then insert it onto an open schematic page.

Requirement: The Data Portal navigator is started. You press the button to start the search function.

EPLAN opens the **Search** dialog. Here you can enter *PNOZ* in the **Designation** field and *Pilz* in the **Manufacturer** field. All other filters are not used or are set to the value *No filter*. Press **ENTER**, or click the **FIND** button at the very bottom of the **Search** dialog to start the search.

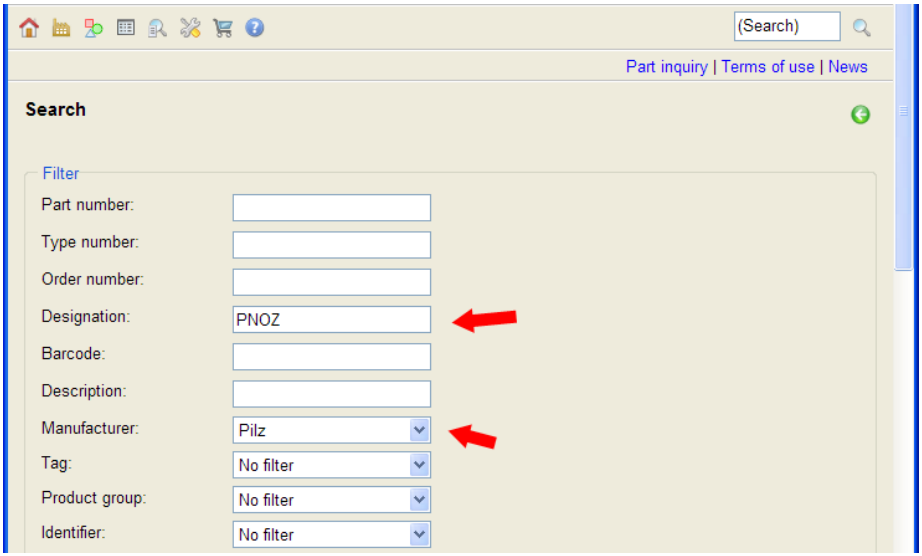


Fig. 12.132
Search function

If EPLAN finds parts matching the search term, the Data Portal lists these in an overview list. The list contains the parts found by the search as well as header data such as manufacturer, subgroup, languages and characteristics.

This list includes the devices, their part numbers, and other information such as type and order number, designation, the product group to which the device belongs, the assigned identifier, in what languages the device is available in the database, and what other characteristics the device possesses. Characteristics are additional information.

Simply stated, the more superior a part in the Portal is, the more characteristics it has in the Portal. Characteristics can be: the device has device data (function definitions/templates) or, for example, macros are available for the device.

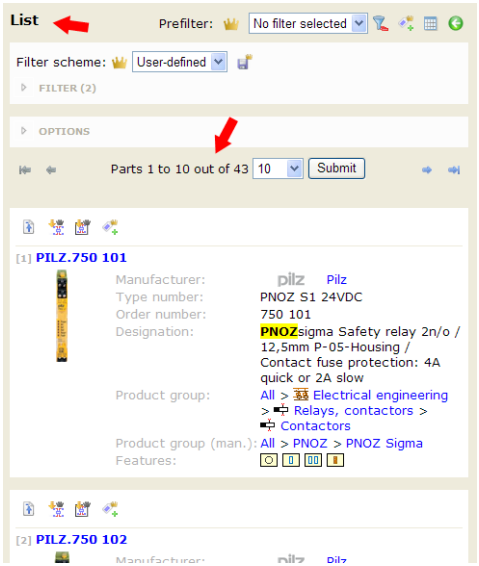


Fig. 12.133
Parts found

NOTE: The search results always depend on how the search and the various search criteria were set and also what data is actually available in the EPLAN Data Portal.

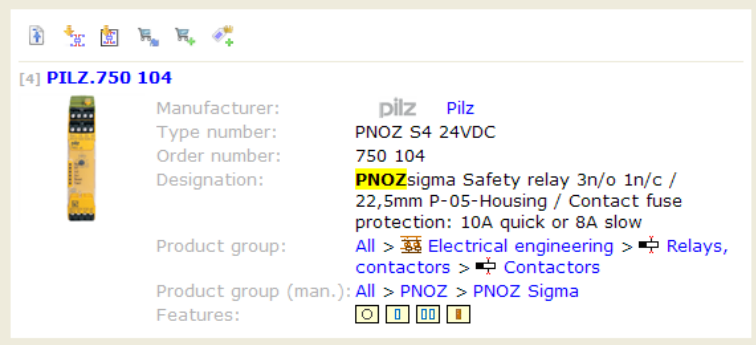


Fig. 12.134 Selected PNOZ device

In this example, we need part PILZ.750 104. This found device can now be directly used in the design.

The Portal list view has several buttons above the part that have the following meanings.

Back to beginning of page

This button is self-explanatory. It takes you back to the beginning of the page.

Insert macro <xyz> in graphical editor (on the schematic page)

If you click this button, EPLAN "hangs" the macro on the cursor, allowing it to be placed on the schematic page in the graphical editor.

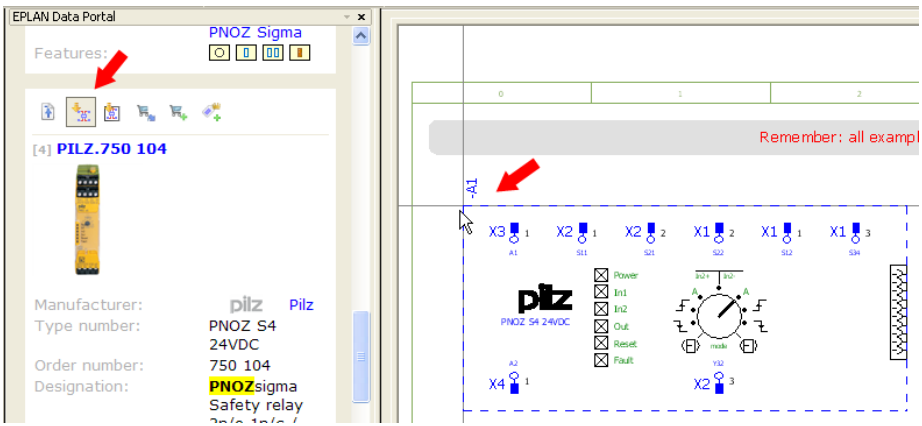


Fig. 12.135 Directly placing the selected macro in the graphical editor

Insert part <xyz> as device in the graphical editor (on the schematic page)

If you activate this button, first the **IMPORT OF PARTS** prompt appears.

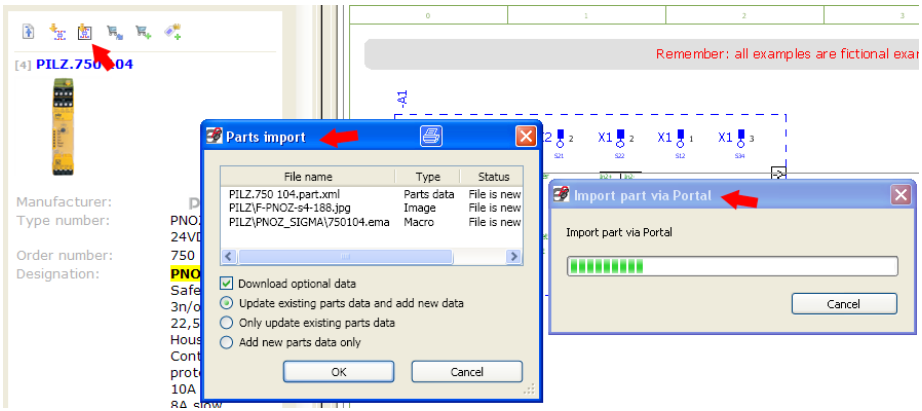


Fig. 12.136 Inserting the device in the graphical editor

You need to decide how the data is to be stored in the existing **parts management** and whether, for example, the *optional data* should also be imported.

When you click **OK** in the **IMPORT OF PARTS** dialog, EPLAN imports the data into the **parts management** and then "hangs" the macro on the cursor. Now you can place the macro on the schematic page.

Import part <xyz>

When this button is clicked, only the selected parts data is imported into the **parts management**. EPLAN restarts the **IMPORT OF PARTS** dialog and, depending on the settings, imports the corresponding parts data into parts management. The rest of the data (images, for example) is saved in the directory structure.

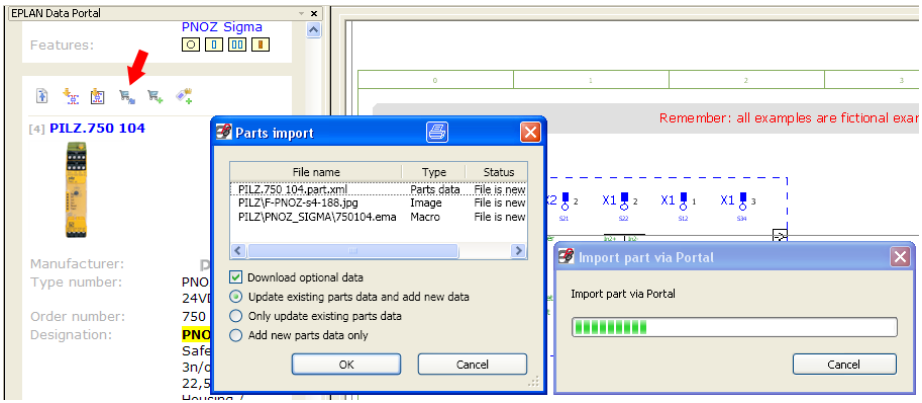


Fig. 12.137 Pure import of parts

After the import, EPLAN closes these dialogs and the graphical editor is active again.

Put part <xyz> in the shopping cart

Clicking this button adds the selected device to the shopping cart. The shopping cart is called up via the button.

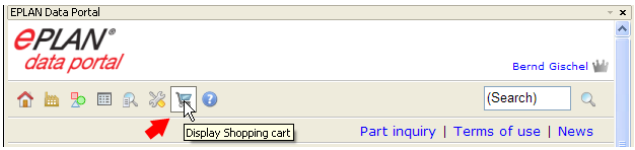


Fig. 12.138 Display the shopping cart

All devices in the shopping cart can be imported completely (collected).

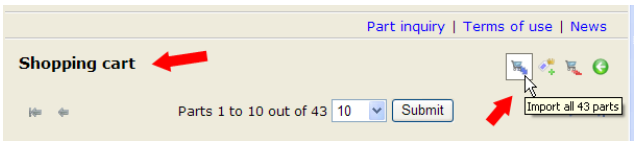


Fig. 12.139 Import the entire shopping cart

All of the parts can also be removed from the shopping cart.

Fig. 12.140 Empty the entire shopping cart

You can go back to the previous page by clicking this button.

Fig. 12.141
Back button

12.3 Project options

In EPLAN Electric P8, macros with value sets are very flexible, but they are limited in one aspect. Macros with value sets can only be used on the same page in EPLAN. Cross-page macros with value sets have not been available up to now. EPLAN closes this gap with the **Project options** module.

This section can only briefly touch on how to use project options, and only provides a short introduction to the procedure for creating project options. The large number of functions and the wide range of possibilities offered by the **Project options** module make it impossible to describe all the details of **project options** in the limited space available in this book. This section is intended therefore as food for thought, and to illustrate what is possible when using **project options**. The possible uses are (practically) limitless.

12.3.1 What are project options?

As the name indicates, project options are various options for (partial areas) of a project. This can mean that, for example, the PLC in a project is implemented with a Siemens PLC for one customer and with a Schneider PLC for a different customer. This cannot be implemented using VALUE SET MACROS, but it is possible using the **Project options** module. You simply display or hide the desired options.

Project options cannot be created across projects. They are only available for the project for which they were created. It is possible to create project option templates that can be used in other projects.

12.3.2 Terminology in the Project options module

Project options are a type of "generic term". To help to understand project options, the following section contains a brief explanation of the terms used.

- **Project options group:** This function allows grouping of project options. A project option group can contain several projects, but only one project option at a time can be switched on. All other project options in the project options group are then switched off. Project options groups can only be created in the **project options navigator**.
- **Project options:** These are partial areas of a project that can be switched on and off as desired. They can consist of one or more extracts, pages, page areas or unplaced objects. Project options also can only be created in the **project options navigator**.
- **Sections:** A section is always assigned to a project option. A section can consist of a partial circuit, a complete page or several pages. Unplaced objects are also possible in sections. In addition to using the **project option navigator**, sections can also be created from the PROJECT DATA/PROJECT OPTIONS/CREATE SECTION menu.
- **Project options navigator:** The project options navigator is used for managing the project options groups, for creating and managing the project options, and switching the options on and off. It is called up via the PROJECT DATA/PROJECT OPTIONS/NAVIGATOR menu. The typical, familiar tree and list views are available within this navigator. Custom filters can also be set for the view in the navigator.

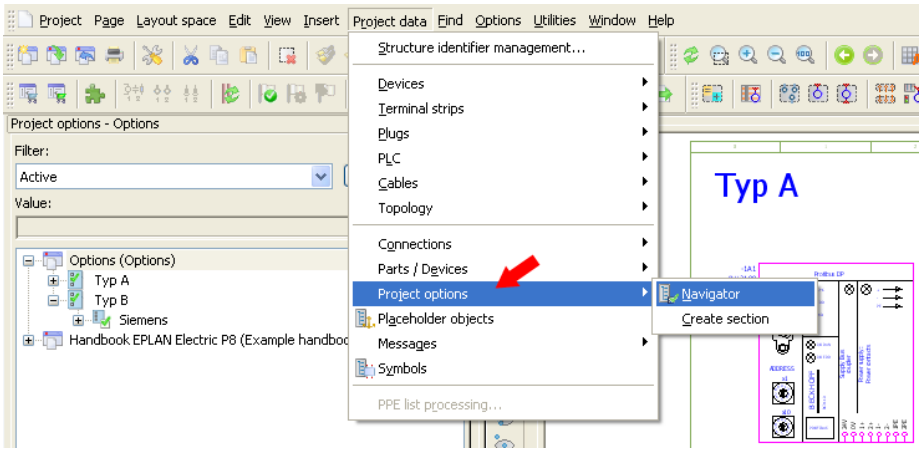


Fig. 12.142
Calling up the project options navigator

12.3.3 Creating options and sections

- A bit of preparation is required for you to be able to easily "switch" between different options in a project. **Project options groups** and their affiliated **project options** must be created, and finally there is the most important point: creating the **sections**.
- Section 12.3.3.1 explains in just a few steps the most important points for creating these options and their associated parts.

12.3.3.1 Project options group

A **project options group** is created in the **project options navigator**. You start the navigator via the PROJECT DATA/PROJECT OPTIONS/NAVIGATOR menu.

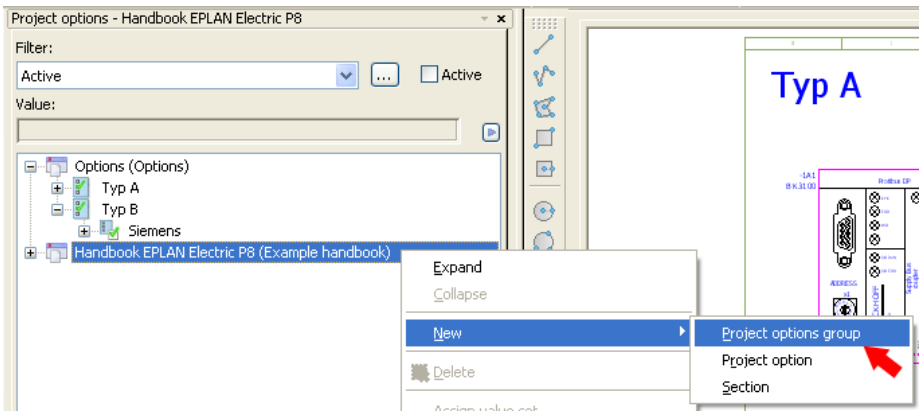


Fig. 12.143
Creating project options groups

Then you call up the **project options navigator** in the popup menu and select NEW/PROJECT OPTIONS GROUP. EPLAN opens the PROJECT OPTIONS GROUP dialog.

You have to enter a **name** and **description**. When you click **OK**, the group is saved and displayed in the navigator.

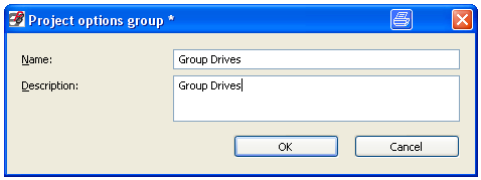


Fig. 12.144 New project options group

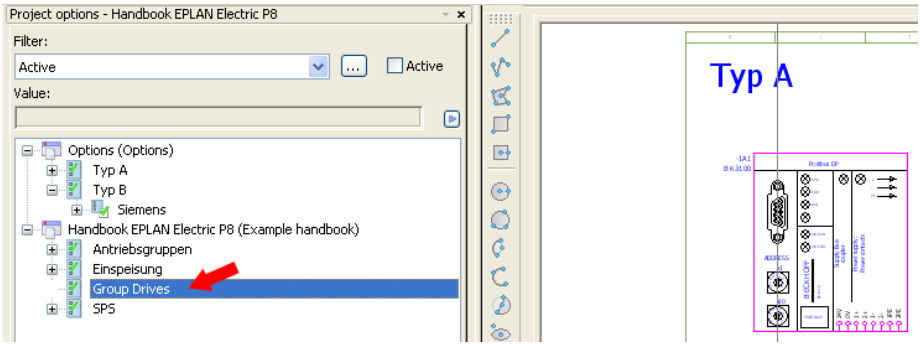


Fig. 12.145 View in the navigator

Examples of project options groups could be: *power supply*, *PLC*, *automatic operation*, *manual operation*, etc. The actual project options for power supply or PLC can be created beneath these project options groups.

12.3.3.2 Project options

Project options can also only be created in the **project options navigator**. Project options always belong to a project options group. This means that one of the project options has to be selected, or that a project options group must be selected when creating a project option. To do this, you open up the **project option navigator** and start the popup menu command NEW/PROJECT OPTION.

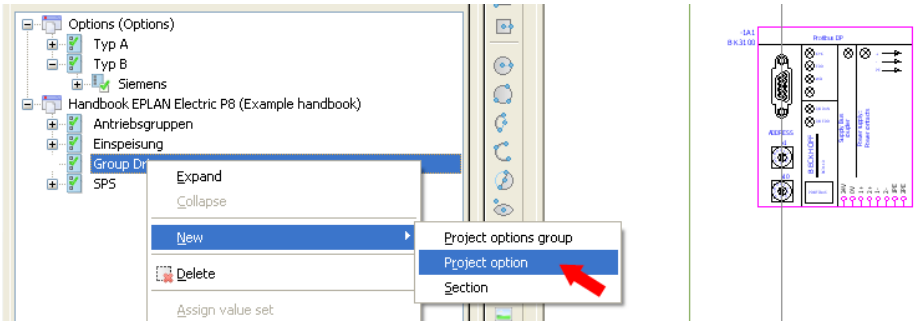


Fig. 12.146 Creating a new project option

EPLAN opens the **PROJECT OPTIONS GROUP** dialog. The project options group can be changed here, if desired. You have to assign the project option a **name** and can also enter a **description**.

When you click **OK**, the project option is saved and sorted under the selected project options group in the **project option navigator**.

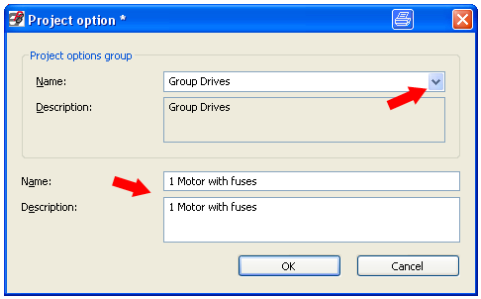
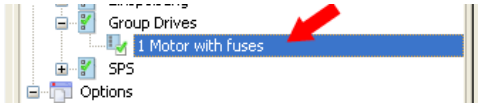


Fig. 12.147
Creating a new project option

Several project options can be assigned to a project options group. However, only one project option at a time can be active, i. e. switched on.

The icon in front of the project option name indicates whether a project option is switched on. If it is switched on, you will see a small green check mark.



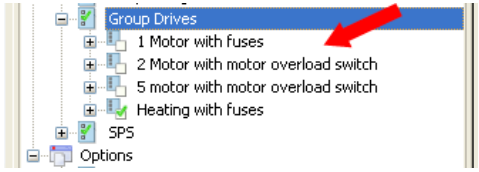


Fig. 12.148
Newly created project option

Fig. 12.149
Multiple project options

12.3.3.3 Sections

The actual **sections** can now be assigned within project options. As already described, sections can be partial sections, one or more pages, or unplaced functions.

To generate a section, in the **project options navigator**, you need to select the corresponding **project options group** and one of the **project options** below this group. Then you call up the popup menu and select **NEW/CREATE SECTION** to create the section.

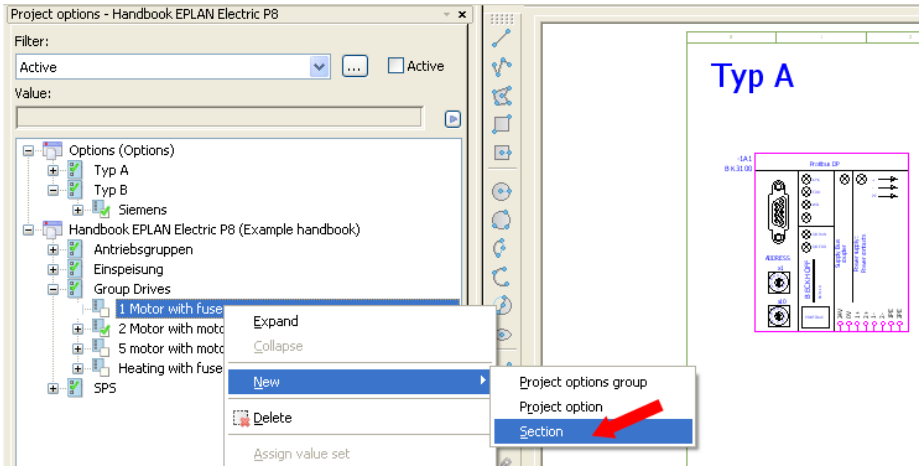


Fig. 12.150 Creating a new section

EPLAN starts the function. The objects belonging to this section must now be defined. This is done using windows. When you have finished defining the objects, you have to enter a **name** and **description** for the section in the subsequent SECTION dialog.

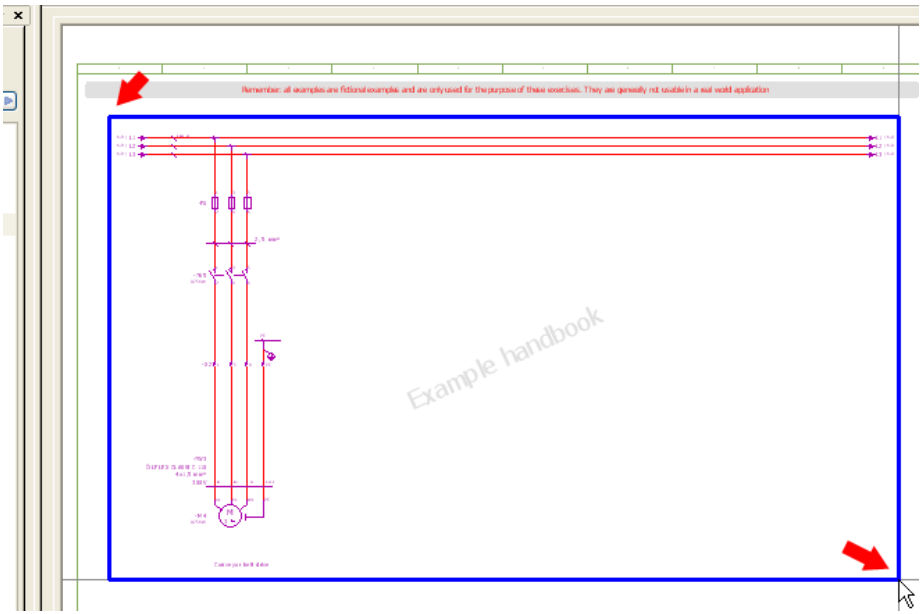


Fig. 12.151 Defining a section via a window

In the SECTION dialog, you once more have the opportunity to assign the section to a different project option or project options group. After you click the OK button, the section is saved and sorted under the associated **project options group** and **project option** in the **project options navigator**.

The next section can now be created. The first section can be switched to be transparent as a design aid. You do this by selecting the section and clicking the MAKE TRANSPARENT (ON/OFF) function in the popup menu (switched on). After this, the previous section must be switched off by clicking the green check mark in front of the project option name.

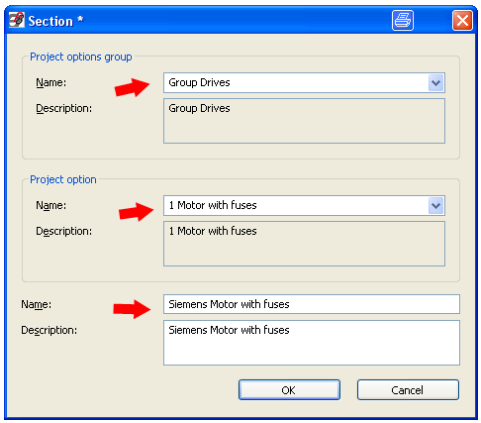


Fig. 12.152 Designating a new section

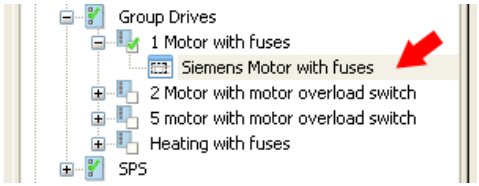


Fig. 12.153 Section in the navigator

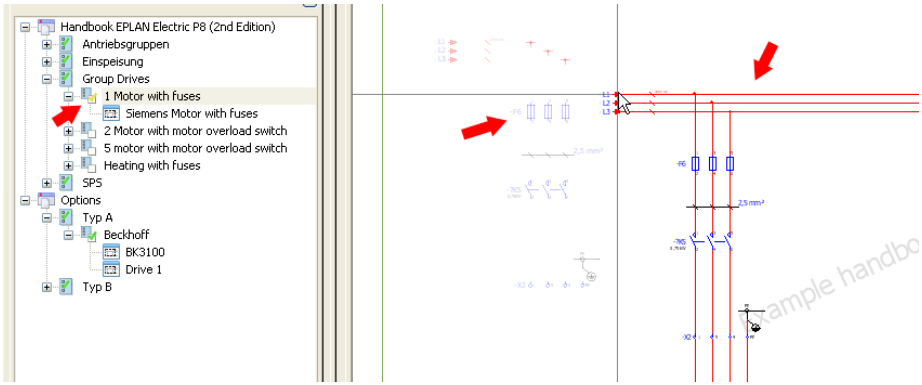


Fig. 12.154 Editing new circuit parts for a new section

EPLAN visually switches the previous section to transparent. This is indicated by the yellow check mark in front of the section name. The "new" section can now be created. You place and edit the desired elements. You can use the placements of the (previous) transparent option (of the section) to help with the placement of the new devices.

As an editing aid, you can also copy the previous section and place it over the transparent section. Now you can replace devices, add new devices added, or delete unnecessary devices and other elements. Completely different wiring can also be created, depending on what partial circuit is desired or required for this option.

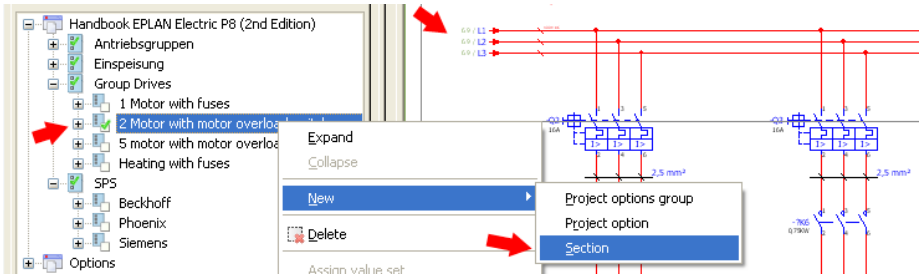


Fig. 12.155 Defining a new section

Once all changes for this section have been made, the section can be defined and saved. You again use windows to select the devices. Then you can use the **PROJECT DATA/PROJECT OPTIONS/CREATE SECTION** menu or the popup window of the **project options navigator** to create the new section.

You give the new section a **name** and **description** as before. The section is then saved. Both sections are now available as options in the project and can be switched on and off as desired.

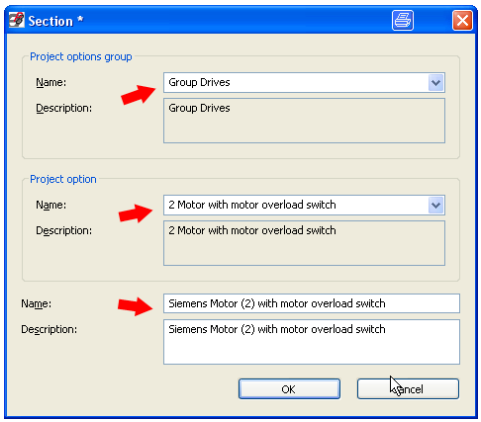


Fig. 12.156 Designating a new section

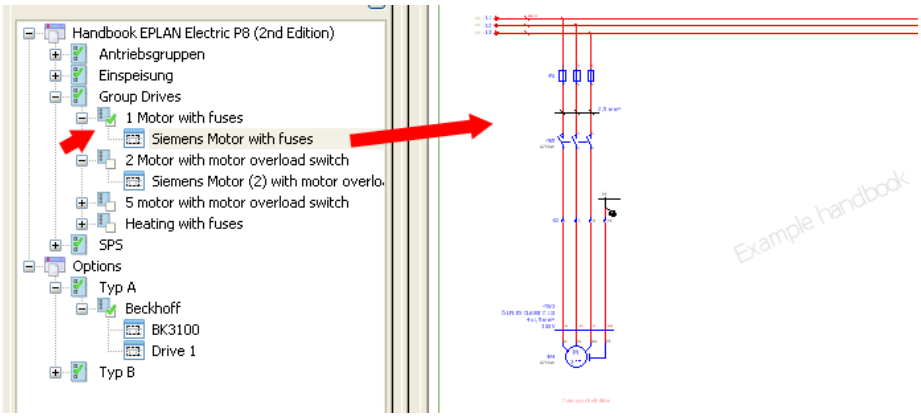


Fig. 12.157 Section with a motor

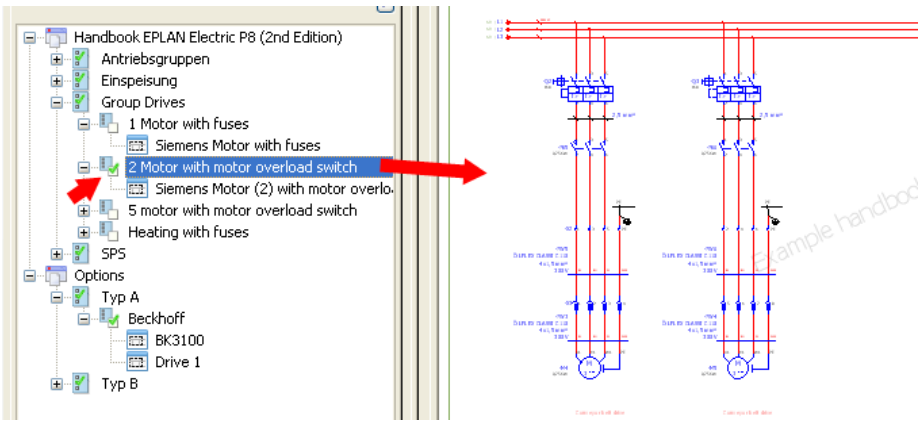


Fig. 12.158 The same page with another section

12.3.4 Generate options overview report

If several options are being used in a project, it may be helpful to have an overview of the options that are used and not used. EPLAN offers an easy option for automatically creating reports in a broad range of forms. All that is required is a form of the **project options overview** type and a report or report template.

Fig. 12.159 shows an example of how this kind of overview of the options used could appear in a report. Of course, you can use filters and sortings here so that, for example, your report only includes the active options.

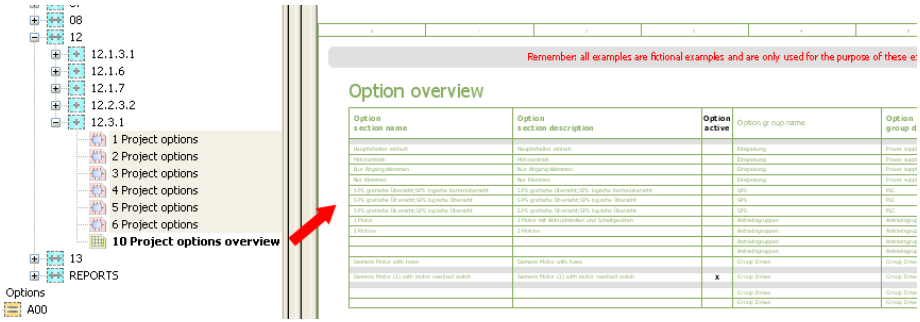


Fig. 12.159 Automatically generated options overview

13 FAQs

Despite the extensive online help that EPLAN provides, there will always be one or the other question that is not covered or whose answer does not precisely fit the situation at hand. This chapter therefore contains a number of questions frequently asked about EPLAN Electric P8 and their answers, divided into categories.

NOTE: I should point out that some questions may have several answers. EPLAN Electric P8 is known for offering different solution approaches; there is not "just one right way".

13.1 General

Question: What is the fastest way to rotate grouped graphical elements?

Answer: Select the grouped elements, then define the center of rotation (**EDIT/ROTATE** menu) and use the **OPTIONS/RELATIVE COORDINATE INPUT** menu (or the **SHIFT + R** shortcut key) and the subsequent **Relative coordinate input** dialog to enter directly the desired angle of rotation for the grouped element. Confirm the selection with **OK**. Click **OK** once more and EPLAN rotates the element by the entered angle.

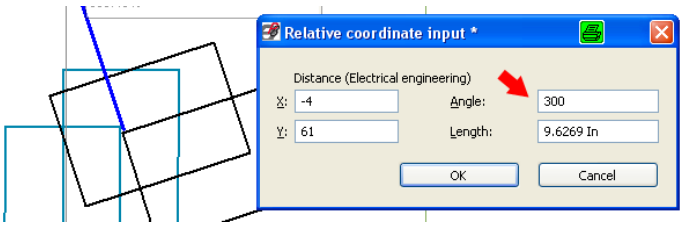


Fig. 13.1 Setting the angle in the dialog

Question: How do I enlarge or reduce graphical objects?

Answer: Select the object to be enlarged/reduced. In the **EDIT/GRAPHIC** menu, select the **SCALE** function. Specify the origin for scaling and confirm with **OK**.

NOTE: Only integer values or values with a decimal point are valid here.

Question: What is the meaning of the **PROPERTIES (GLOBAL)** function in the **OPTIONS** menu?

Answer: To edit properties globally, you have to activate the **OPTIONS/PROPERTIES (GLOBAL)** menu. This allows you to simultaneously (i. e. globally) edit a contactor and its contacts that are distributed throughout the schematic. For example, you could change the DT throughout the schematic.

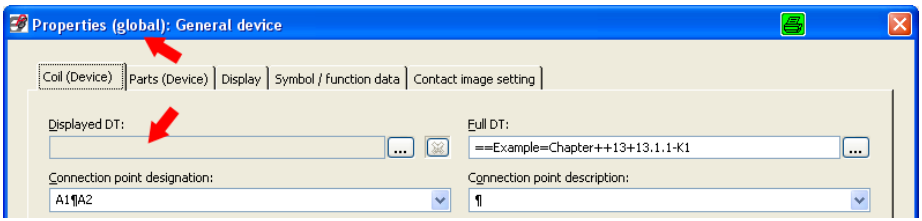


Fig. 13.2 Properties (global) option

When the **PROPERTIES (GLOBAL)** option has been activated, it is not possible to change the *Displayed DT* in the symbol properties. The field will appear grayed out. Only the *Full DT* field can be edited.

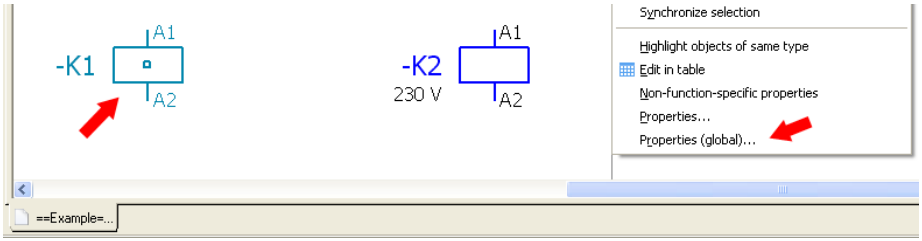


Fig. 13.3 Popup menu

The *Properties (global)* function can also be found in the right-click popup menu. To use this function, you must select, for example, a device. Then, you can call *Properties (global)* from the popup menu. Actually, you can select any device. It could be the coil, or even one of the coil's contacts.

Question: What is the difference between a *basic project* and a *template project*?

Answer: *Project templates* always contain all project-specific settings and the schemes of the project structure. Project templates have the file extension **.ept*. If a new project is

created on the basis of a project template, all project-specific settings will be imported and applied. The page structure in such a newly created project can be adjusted subsequently if no pages have been added to the project yet.

Basic projects contain both pages and all project-specific settings. They have the file extension **.ebp*. If a project is created on the basis of a basic project, all project-specific settings and the pages already contained in the basic project will be imported and applied. However, the page structure of the new project is defined by the pages of the basic project and, unlike project templates, cannot be changed.

The page numbering, though, can be changed for both types of projects. The main difference lies in how the page structure is modified. In one case it can be changed (new project from a project template), while in the other (new project from a basic project) it cannot be changed.

NOTE: As of version EPLAN Electric P8 1.9.x, basic projects always have the file extension **zw9*. The previous **epb* format still exists and can be used as a basis for new projects. However, it is no longer possible (this is also true of version 1.9.x and up) to create basic projects with the **epb* extension.

Question: How do I modify the line type of autoconnecting connections?

Answer: To modify the formatting of autoconnecting lines, you can set a *potential definition point* on the autoconnecting line and influence the formatting of the autoconnecting line in the symbol properties on the *Connection graphic* tab. A potential definition point can be inserted via the **INSERT/POTENTIAL DEFINITION POINT** menu.

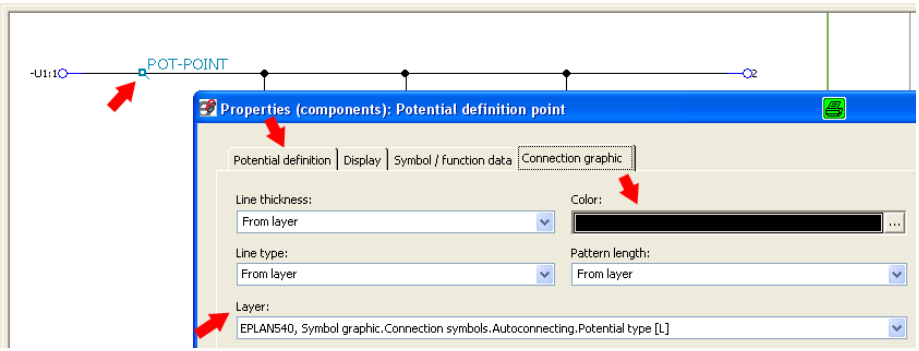


Fig. 13.4 Changing the formatting of autoconnecting lines globally

Since this involves a connection, the connections should then be updated (via the **PROJECT DATA/CONNECTIONS/UPDATE** menu). This way, all connections affected (e. g. PE connections) are updated, and they adopt the format of the potential definition point.

If you want to change the line type for a partial connection only, you must use a connection definition point. A connection definition point can be inserted via the **Insert/Connection definition point** menu, which can then be placed on the intended section of the connection.

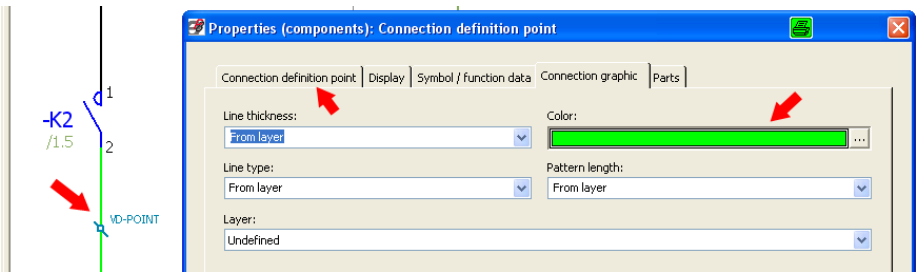


Fig. 13.5 Modifying the property of a partial connection

Question: Can macros with value sets be used across pages?

Answer: No. Macros with value sets can only be used on, or with, one page. If you wish to use macros with value sets across pages, you will have to use the **Project Options** add-on (to be purchased separately).

Question: Why do my interruption points in the project always refer to themselves on each page?

Answer: To ensure that cross-references work for interruption points, you must observe the following.

- *First page*: Insert an outbound interruption point
- *Second page*: Insert an inbound and an outbound interruption point
- *Third page*: Insert an inbound and an outbound interruption point
- *Last page*: Insert an outbound interruption point

Interruption points are always paired. The head of an interruption point must always come together with the tail of the next interruption point; always head to tail. Now, the cross-references (if all other properties are set to default) will work, and the interruption points will no longer result in wrong references.

Fig. 13.6 Default sorting of interruption points

By defining a sort code for interruption points, it is also possible to connect specific interruption points with each other in a targeted manner (requires identical DTs). Again: There must always be a pair with the same sort code.

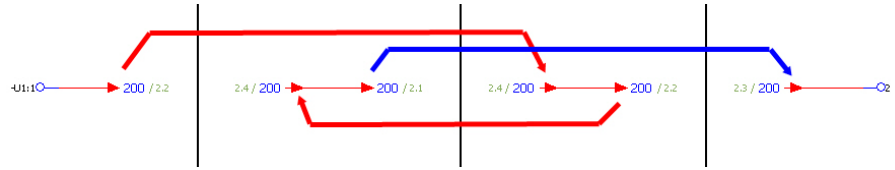


Fig. 13.7
Default sorting of interruption points

Question: Why do I sometimes see red exclamation marks on my devices in the device navigator?

Answer: If you see a red exclamation mark on a device, it means that a message has been received for this device via the message navigator.

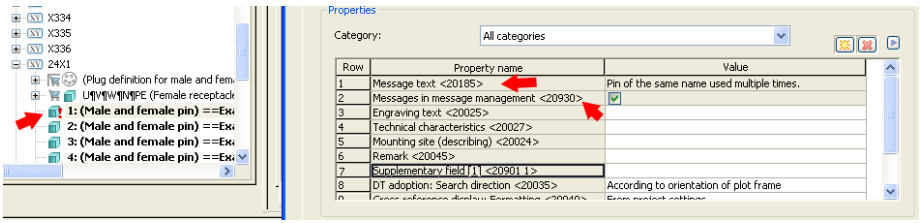


Fig. 13.8
Symbol properties

Simply open the message navigator, activate the *Selection* check box and select the device in the device navigator. The message or messages that have been generated for this device during the check run will be displayed.

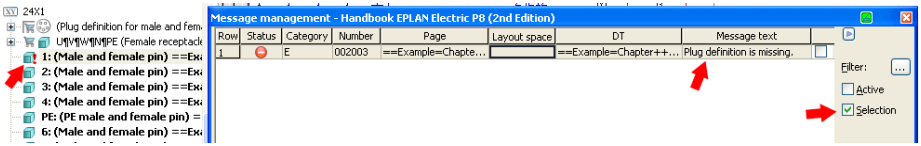


Fig. 13.9 Message in the message navigator

Question: Why does EPLAN Electric P8 not create connections, and why are no cross-references displayed?

Answer: In this case, the project in question is not of the **schematic project** type, but a project of the **macro project** type. These limitations are typical of macro projects.

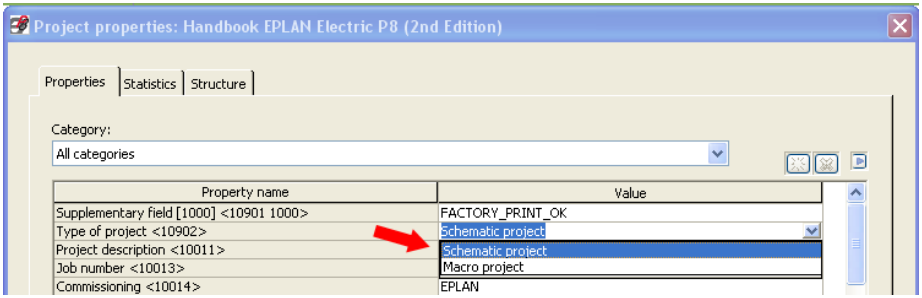


Fig. 13.10 Setting the project type

If this is not the type you want, you have to change the *Type of project <10902>* property in the project properties.

Question: How can I later change the project structure (page structure)?

Answer: You can't. The underlying structure of the project (page structure) cannot be later modified; under normal circumstances it can only be selected once (when creating the project).

NOTE: The page structure also cannot be changed when a new project is created from a basic project. This is also true if a new project is created based on a copy of another project. Here, too, it is not possible to change the page structure of the project, because it was defined as such in the project to be copied.

All other structures, such as devices, cables, etc. can, however, be later adjusted to the requirements of the project.

Question: Why doesn't copy and paste using CRTL + C and CTRL + V work anymore?

Answer: If copying with **CTRL + C** and inserting (pasting) with **CTRL + V** doesn't work (e. g. you select an object and then press **CTRL + C** to copy it and **CTRL + V** to paste it), then the **DESIGN MODE** in the **OPTIONS** menu has been activated. Simply deactivate it and you can again use the copy/paste functions.

Question: What is the difference between *Delete* and *Delete placement*?

Answer:

Delete: The selected object (device) is deleted completely (graphical placement and any device information). It is no longer available in the project data or in any of the navigators.

Delete placement: The object is only "graphically" deleted. It is no longer visible graphically (e. g. on a schematic page), but the object itself still exists in the project data with all its device information, e. g. in the device navigator, and can be placed "graphically" again later on or be retrieved from the relevant navigator.

Question: Why can't I enter data into the "Displayed DT" field anymore (it is grayed out)?

Answer: If in the symbol properties the *Displayed DT* field is grayed out (i. e. no data can be entered here), then the *Properties (global)* option has been activated. This must first be deactivated to be able to edit the *Displayed DT* field as usual.

Question: Where can I find a demo version of EPLAN Electric P8?

Answer: There is a P8 version for students, with a trial license of 270 days. It is called EPLAN Education and can be downloaded from the EPLAN website. As well as EPLAN Electric P8, it also contains EPLAN Fluid, EPLAN PPE and other programs depending on the stage of development. Installation instructions for the download are available.

NOTE: The data in the Education version is not compatible with the data in the industrial (purchased) version. And data from the industrial version cannot be edited with the Education version.

Question: How can I edit a single object within a grouping?

Answer: It is possible to edit a single object embedded in a grouping (without having to undo the grouping) by keeping the **SHIFT** key pressed and double-clicking the intended element. EPLAN then opens exactly this one object, and you can modify its data.

Question: How can I reduce five lines simultaneously by a specific length in EPLAN?

Answer: By using the stretch function. This function stems from the CAD area and works as follows (ideally, the **OBJECT SNAP** mode should be activated in the **OPTIONS** menu). Call the **STRETCH** function via the **EDIT/GRAPHIC** menu.

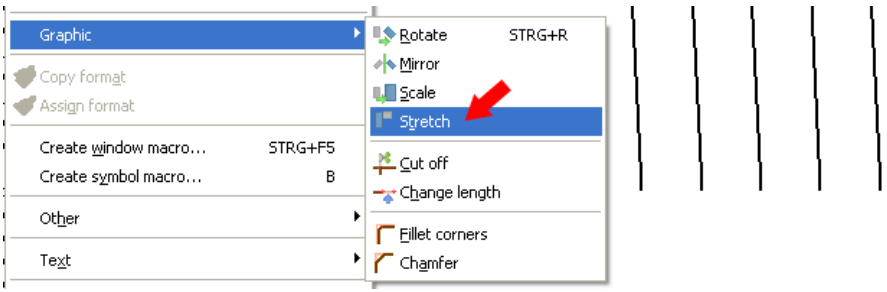


Fig. 13.11 Stretch function

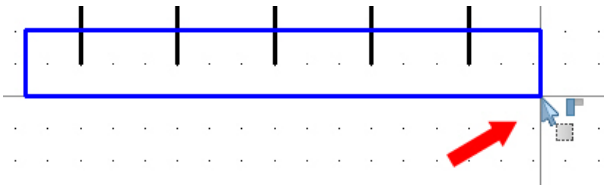


Fig. 13.12 Pulling open a window

Now, you select all line ends by pulling open a window (left mouse button) (the ends are "framed"). EPLAN marks the ends of the lines with a circle.

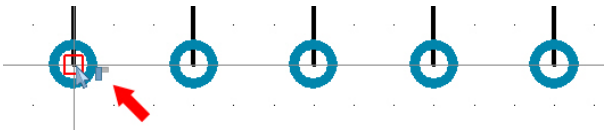


Fig. 13.13 Defining the starting point of the stretch

Then define the starting point of the stretch. Usually, this is done by clicking the end of a line. If object snap is enabled, the cursor will "snap" into place (red rectangle).

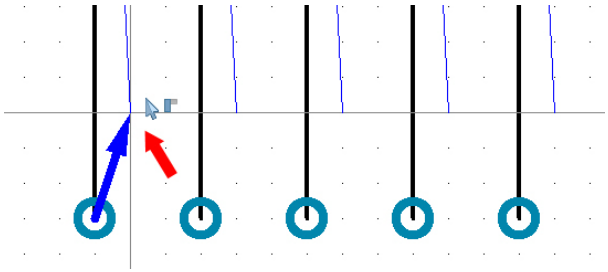


Fig. 13.14 Defining the end point of the stretch

Then you define the end point of the stretch (can be selected via the left mouse button; until it is placed EPLAN displays a cursor with a blue arrow). Any end point can be selected.

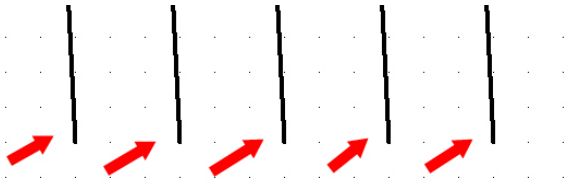


Fig. 13.15 All line data has been changed.

When you have selected the placement end point, confirm it via the left mouse button (click once). Now, the lines are "stretched" until this end point, i. e. shortened in this case. Of course, it also possible to extend them or to change the angle for all of them.

Question: How can the creator, customer or end customer data in the project properties be filled in automatically?

Answer: To avoid having to always manually enter the creator, customer or end customer data, such as address, name, etc. in the project properties, EPLAN provides an option that allows this data (which, of course, must already have been set up in parts management in the **Customer** area) to be written into the project properties by EPLAN automatically. Open the (optional) *Project management* in the **PROJECT** menu and select the intended project. Now click the **EXTRAS** button and then select the **READ CUSTOMER DATA** function. Here, you can select the data for the creator, customer or end customer from the **Select address** dialog. Click the **OK** button to import them into the project properties. EPLAN then fills automatically the corresponding project properties with the data stored in parts management.

Question: What is the difference between *Project properties* and *Project settings*?

Answer: *Project properties* are properties of the project, such as settings for the project structure or the types of devices used in it, such as terminal strips or general devices. Also part of project properties are specific properties that apply to the entire project globally, such as properties of the creator or the date of the project start. Project properties also include statistical information, such as the number of specific page types, etc. Project

properties are called via the **PROJECT** menu and the **PROPERTIES** menu item. Alternatively you can select the project in the **page navigator**, then open the popup menu and select **PROJECT** and then **PROPERTIES**.

Project settings are settings that affect the project but have nothing to do with the project "content", such as which forms are to used for project reports, which online DT numbering scheme should be used or how the project structure should be displayed in the page navigator. You use the **OPTIONS/SETTINGS** menu to call up the project settings. Then, the correct project must be selected in the **Projects** node (in case of several open projects).

Question: How can I interrupt an Autoconnecting line?

Answer: An Autoconnecting line can be interrupted by means of the ACBP symbol (Autoconnect break point). The symbol can be inserted by pressing **CTRL + SHIFT + U**, or can be selected in the **INSERT/CONNECTION SYMBOL/BREAK POINT** menu and then placed randomly like any other symbol.

Question: How can I change the position of the contact image?

Answer: The position of the contact image can be modified in different ways, e. g. in the page properties of the current page by changing the *Contact image offset <12061>* property. This value applies only to the current page and all symbols affected by it.

The position can be changed globally by adjusting the plot frame in use. To do this, adjust the *Contact image offset <12061>* property in the plot frame editor. This setting will then apply to the entire project.

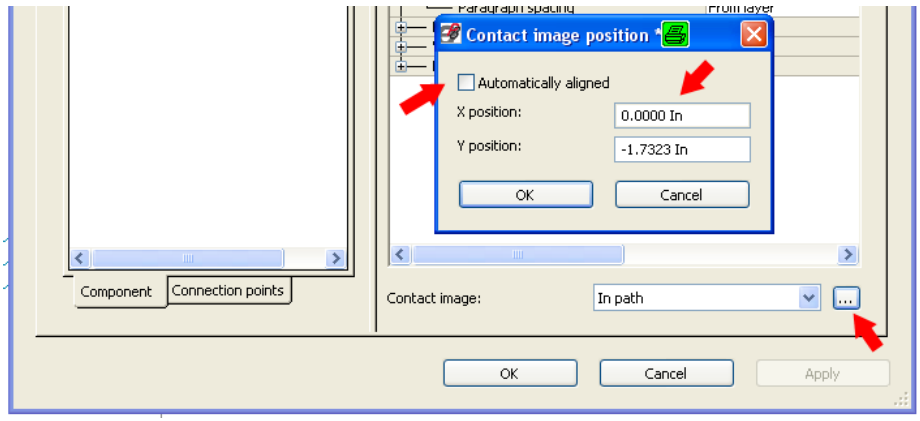


Fig. 13.16 Manually changing the position of the contact image

A third option involves calling the **Contact image position** dialog by using the More button directly on the affected symbol on the *Display* tab in the *Contact image* area and manually defining the X and Y positions.

It is also possible to select the symbol, press the **CTRL + B** shortcut key and click the contact image, while moving it into any position while keeping the left mouse button pressed.

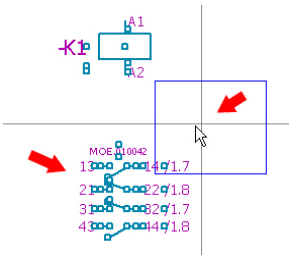


Fig. 13.17
Move

Question: What is the meaning of the "Superior" option in the project structure in case of different devices?

Answer: If the *Superior* option has been set, EPLAN, in the report, allocates to this device the structure identifier depending on whether the device has been assigned the "–" *preceding sign*.

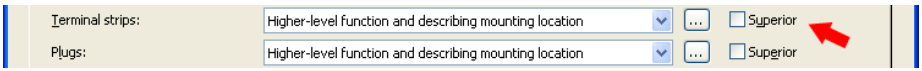


Fig. 13.18 Superior setting

If the device has the preceding sign "–", the device will be allocated the structure identifiers (of the page, etc.) as defined in the project structure. If the devices does not have the preceding sign "–", this does not happen.

For example, you can take one terminal strip and have it created globally across several mounting locations by omitting the "–" preceding sign. At the same time, it is also possible to generate a report on the remaining terminal strips, depending on the project structure defined, based on higher-level function and mounting location by placing the "–" preceding sign.

The *Superior* option is possible for the following device types: terminal strips, plugs, cables and interruption points.

Question: Where and how can I re-sort my structure identifiers?

Answer: To re-sort structure identifiers such as higher-level function or mounting location, you use the structure identifier management tool. It can be found in the **PROJECT DATA/STRUCTURE IDENTIFIER MANAGEMENT** menu. Here you will find editing options to re-sort or rename identifiers, to create new identifiers or to edit existing or create new descriptions of structure identifiers, and much more.

Question: How can I display the message(s) in message management for a specific object?

Answer: To view the relevant messages from message management for an item in the device navigator, you need to take the following steps.

1. Launch **PROJECT DATA/MESSAGES/MANAGEMENT** message management with the proper check run.
2. In the message management window, activate the check box next to **SELECTION**.
3. Start the device navigator.

4. Click the desired (defective) item in the device navigator.
5. Now you will see only messages in message management that concern this item.

Question: Where in EPLAN can I enter my own connection point designations or connection point descriptions for symbols?

Answer: For example, if you want to have for a horn (signal device) the existing connection point designations [x1 x2] as well as the connection point designation [L N] (in the symbol properties dialog of the horn), you need to take the following steps:

1. Via the **UTILITIES/MASTER DATA** menu, open the **CONNECTION POINT DESIGNATION/DESCRIPTION** menu item.
2. On the *Connection point designations*[1] tab, locate the corresponding function definition (here: *Signal device, acoustic, single*) in the *function definition* column.

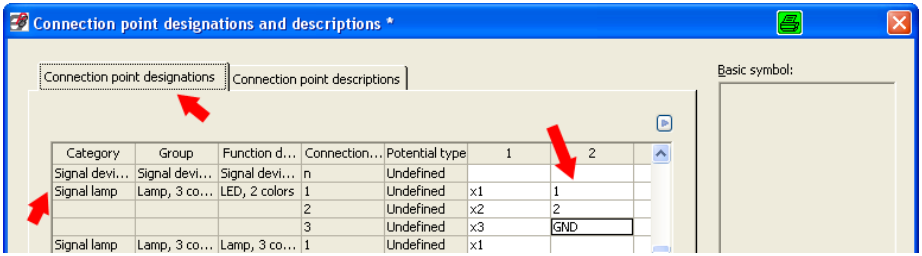


Fig. 13.19 Adding connection point designations

3. In column 2, enter the additional connection point designations per connection point.
4. Click the **APPLY** button and close the dialog with **OK**.
5. In the **UTILITIES/MASTER DATA** menu, select the **UPDATE CURRENT PROJECT** menu item. EPLAN now updates the project master data.
6. The new connection point designation [L N] is immediately available at the horn for selection.

Question: What is the difference between *[Update current project]* and *[Synchronize current project]*?

Answer:

Update current project: EPLAN automatically updates the project master data with the system master data. EPLAN will try, for instance, to store master data that is used in the project, but wasn't found. Manual editing is not possible at this point. Once this function is launched, there will be no more confirmation prompt (to abort the function, you would have to click the **CANCEL** button in the dialog). After the update, EPLAN will display a message.

1 For your own connection point descriptions, follow the same procedure. The only difference is that you must select the *Connection point descriptions* tab in the dialog.

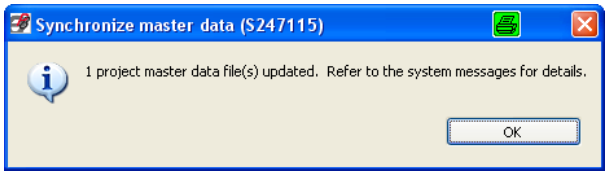


Fig. 13.20 Message regarding synchronization of master data

Synchronize current project: First you open the **Synchronize master data** dialog. In this dialog, master data can be exchanged in a targeted manner (in both directions, i. e. system to project or project to system). It is therefore possible to manually update specific master data (synchronize).

Both menu items are found in the **UTILITIES/MASTER DATA** menu.

Question: Can EPLAN create a macro from several pages?

Answer: Yes, EPLAN can do that. Such macros are known as page macros in EPLAN.

A page macro of a page can be created via the **CTRL+ F10** shortcut key. Alternatively, you can select the page in the page navigator and then select in the popup menu (right mouse button) the **CREATE PAGE MACRO** item. Then the information has to be stored in the **Save as** dialog.

The procedure for creating a page macro for multiple pages is similar. You select the pages in the page navigator and then press **CTRL + F10** or right click to call up the popup menu and call up the **Create page macro** menu item.

Question: EPLAN Electric P8 and Windows Vista or Windows 7 – is it possible?

Answer: EPLAN Electric P8 (and the rest of the EPLAN platform, such as EPLAN PPE, EPLAN Fluid, etc.) is approved for use with Windows XP Professional SP3 (32-bit) version 1.9.x and higher, Windows Vista Version 2.0 and higher, and as of version 2.0 for Windows 7.

The following Windows Vista versions are supported: Windows Vista Enterprise; Windows Vista Business N and Windows Vista Ultimate, in each case SP2 and 32-bit or 64-bit.

The following Windows 7 versions are supported: Windows 7 Enterprise; Windows 7 Business and Windows 7 Ultimate (32-bit or 64-bit).

Question: Why are blue boxes instead of graphical symbols shown in the reports?

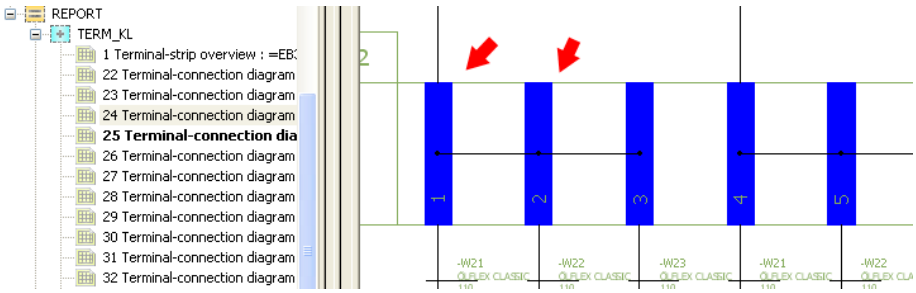


Fig. 13.21 Blue boxes

Answer: For graphical symbols, for example, from the GRAPHICS symbol library, to be displayed in reports, such as terminal or cable diagrams, the following user setting must be enabled. Under **OPTIONS/SETTINGS/USER/GRAPHICAL EDITING/2D** and here, in the selection field for the *Graphical symbol* color setting, you must select the *From symbol* setting.

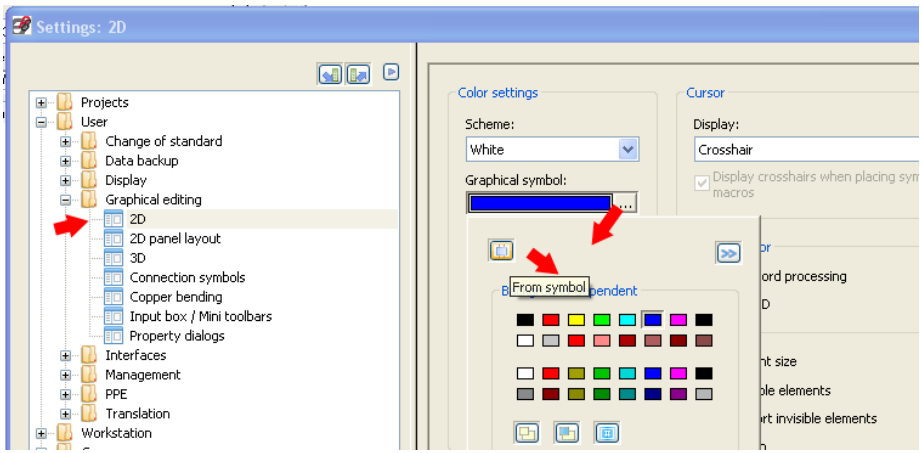


Fig. 13.22 Setting the color from the symbol

Now the graphical symbols will be displayed completely even in the reports.

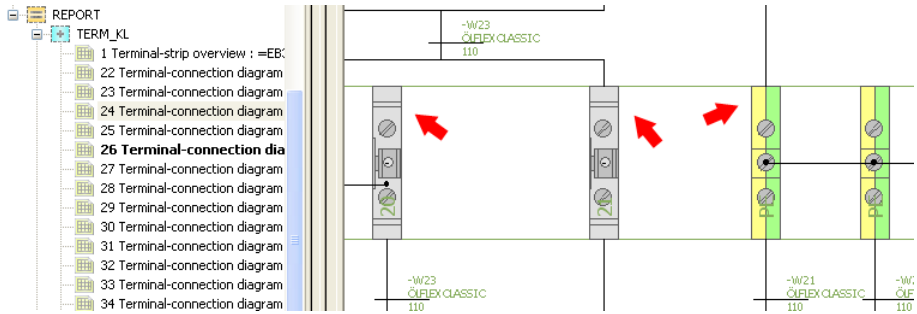


Fig. 13.23 Symbol color settings

Question: What is the difference between a window macro and a symbol macro?

Answer: Generally, they are not vastly different. Both macro types (window **.ema* and symbol macro **.ems*) can combine and store parts of a page or different objects in a macro, which can then be retrieved as a window or symbol macro.

A window macro is called via the **M** key or via the **INSERT/WINDOW MACRO** menu; a symbol macro via the **CTRL + INS** shortcut key or via the **INSERT/SYMBOL MACRO** menu. Both can be placed anywhere on the page.

Question: How can I change the EPLAN installation drive later on without reinstalling?

Answer: No reinstallation is required, for example, to change the installation drive from *C:\Program Files\EPLAN Electric P8* to *E:\EPLANP8 V2*. All you need to do is modify the installation paths in the *Install.xml* file.

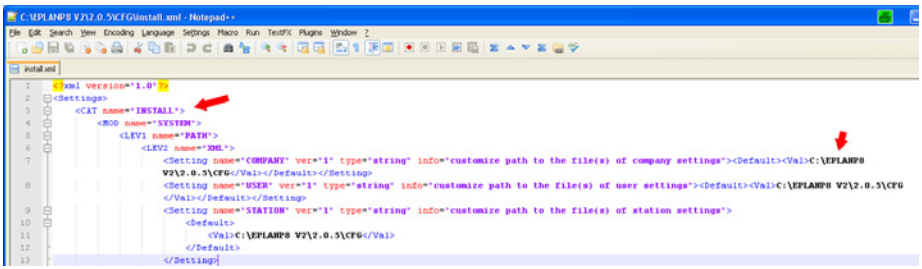


Fig. 13.24 Changing installation paths

The *Install.xml* file is located in the installation directory *Drive:\EPLAN8 V2\2.0.5\CFG* (depending on your installation, the path may be different).

Question: How can I change the company code later on without reinstalling?

Answer: No reinstallation is required; for example, in order to change the *Company code* from CompanyXYZ to BestCompanyInTheWorld. You simply have to change the *Company code* in the *Install.xml* file.

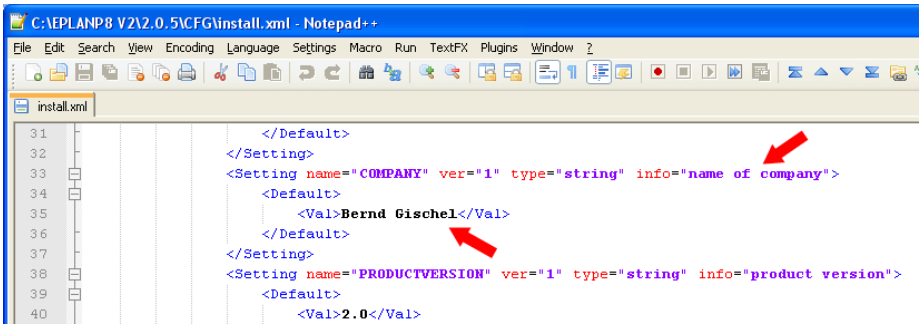


Fig. 13.25 Changing the company code

The *Install.xml* file is located in the installation directory *Drive:\EPLAN8 V2\2.0.5\CFG* (depending on your installation, the path may be different).

NOTE: Before EPLAN is restarted (after making changes to the company code in the *Install.xml* file), the rights database must be located in the new directory (new company code). Otherwise, EPLAN will not launch! The rights database is in the *E:\EPLANP8\Management\CompanyCode* directory (you need to replace the company code with your own company code).

Question: What is a main function?

Answer: A *main function* is the "leading" device with (distributed) devices displayed (e. g. the coil of a contactor) or a single switch. Only the main function, e. g. bears the device definition (parts data) of the device. Automatically, all other possible functions (such as the coil contacts) are not main functions or auxiliary functions.

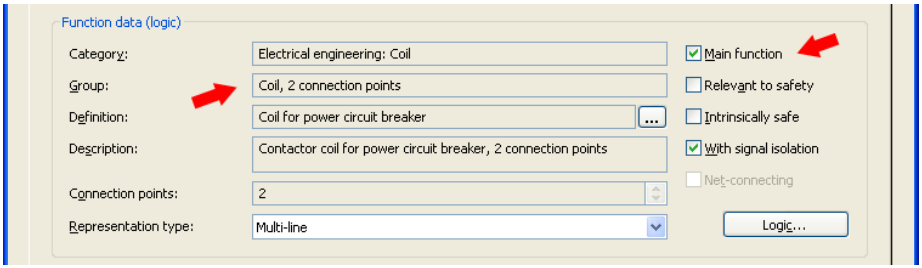


Fig. 13.26 Main function coil

In EPLAN, each device used can only have exactly *one* main function. This is true of all objects, whether they be contactors, motor overload switches or cable definitions.

Question: Where are the page sortings (subpage, characters) of reports stored?

Answer: To set up the desired sorting of report pages (subpages yes/no, numeric, letters uppercase/lowercase), go to the **UTILITIES/REPORTS/GENERATE** menu and open the **Reports** dialog. Here, click the **SETTINGS** button and then select the **OUTPUT TO PAGES** menu item. In the dialog, set the desired *Sorting* in the *Character* column and whether subpages (*subpage* column) are to be created. Finally close and exit the dialog with **OK**. Now the reports can be generated and updated.

Question: What project information appears in the **Open project** dialog on the right-hand side of the dialog?

Answer: The project information in the **Open project** dialog (on the right-hand side in the preview window) are taken from the project properties. This information cannot be affected or replaced with other project properties.

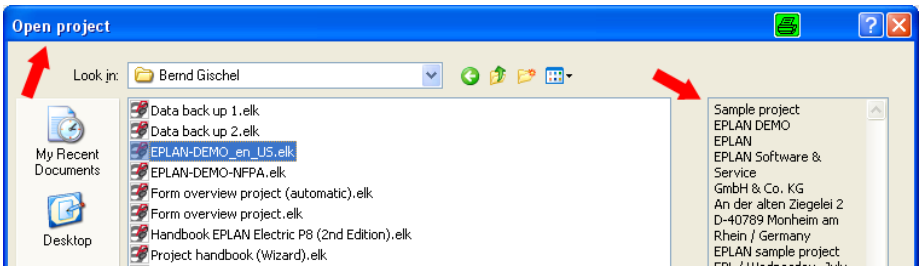


Fig. 13.27 Information in the Open project dialog

The following project properties are displayed (if they have been completed in the project properties): Project description <10011>; Job number <10013>; Commission <10014>; Company name <10015>; Company address 1 <10016>; Company address 2 <10017>; Supplementary field [1] <10901 1>; Last editor: Logon name <10022> and the Modification date <10023>

Question: How can I restore EPLAN to the values of the default installation?

Answer: To restore the default installation values of EPLAN (when confusing situations arise that cannot be explained or removed in any other way), EPLAN can be started by calling up: *"C:\Program Files\EPLAN Electric P8\2.0\BIN\W3u.exe" /setup*, whereby the program directory shown here must be adjusted to reflect your own installation directory.

NOTE: When you launch this call-up, you will lose all personal settings, such as set schemes, workspace, etc. These settings must be saved beforehand.

Question: The object snap for the dimension function does not work anymore.

Answer: If graphical objects on pages – e.g. for the dimension function – are no longer automatically captured by the mouse (the small red rectangle), then the **OBJECT SNAP** in the **OPTIONS** menu has been deactivated.

Once object snap is activated, you will be able to capture objects (points) again.

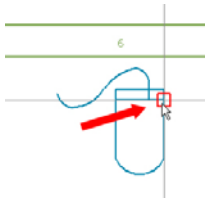


Fig. 13.28
Object snap on

Question: How can I create completely new connections or repair connections?

Answer: If connections (or their additional information) exhibit strange behavior, it is possible to recreate all project connections from scratch. To do so, keep the **SHIFT** and **CTRL** keys pressed and call from the **PROJECT DATA/CONNECTIONS** menu the **UPDATE** menu item. Now, all connections (and/or the connection database) will be recreated. Depending on the number of connections in the project, this may take some time, because EPLAN first deletes all connection data and then recreates them fully from scratch.

Question: How can I later change the sorting of the structure identifiers, such as higher-level function or mounting location?

Answer: To sort identifiers (structure identifiers), such as higher-level function or mounting location, you must start **STRUCTURE IDENTIFIER MANAGEMENT** from the **PROJECT DATA** menu. In the dialog that follows, using the corresponding tabs, you can change the sequence of the various identifiers used in the project (and also those not currently in use).

Question: Can EPLAN also process page names like "Seventy"?

Answer: Yes, this is possible. Usually the page name consists of a number (page number). For example, the name of the page can be "70" . EPLAN can also include alphanumeric components in the page numbers. Accordingly, a page can also be called "Seventy". However, any subsequent operations such as numbering pages will no longer function as desired since the pages have alphanumeric names.

The "number" property of the page is called page "name" and not "number" because it can be alphanumeric.

Question: How is it possible, for example, that a device connection point with the potential type PE has the same designation, without EPLAN showing a project check message?

Answer: In this case, the **LOGIC** (on the *Symbol/function data* tab and, here, the **LOGIC** button) of the device connection point must be adjusted. In the following **Connection point logic** dialog, there is the *Allow same connection point designations* setting. This setting must be activated for the relevant device connection points. This will keep EPLAN from objecting to functions having identical designations.

Question: Does the sequence of the stored symbol libraries affect the representation of the symbols?

Answer: No. The sequence of the stored symbol libraries does not affect EPLAN projects. EPLAN always takes symbols only from the correct symbol library. It relies on information such as the name of the symbol library, the symbol number and the symbol name. This requires, of course, that the same symbol libraries are/have been stored in the various projects.

Question: Why, after inserting a macro, do I see a bunch of red crosses instead of the symbol graphics?

Answer: After inserting a macro or similar things, if you see a bunch of red crosses instead of symbol graphics, the cause is often a missing symbol library in the project.

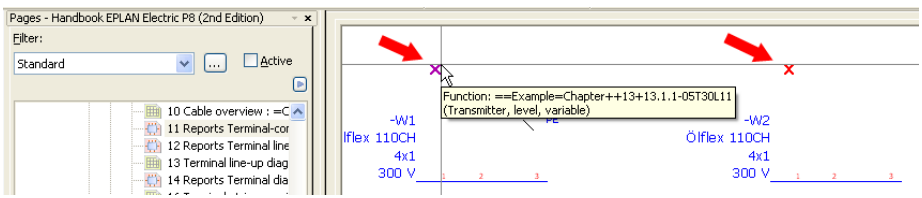


Fig. 13.29 Missing symbols

Since the symbol library/libraries is/are not stored (in the macro, etc.) when creating a macro or similar things (in another project), EPLAN needs the information to display the proper symbol graphic contained in the symbol library that was used in the project (where the macro was created).

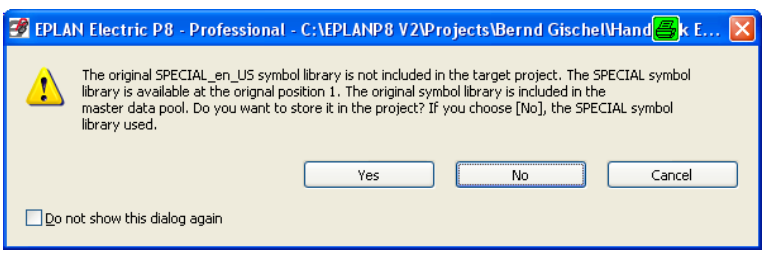


Fig. 13.30 EPLAN warning message when symbols are missing

This can be fixed by placing/storing the corresponding (missing) symbol library in the current project. Then, you must re-insert the macro, or simply select the symbol again.

Question: How can I insert special characters in EPLAN that do not exist in the **Special characters** dialog?

Answer: To insert special characters in texts or similar things in EPLAN, use the **CTRL + S** shortcut key to call the **Special characters** dialog. But the selection offered there is limited. If you want to insert a special character that is not included in EPLAN by default, you can copy it from another Windows application, for example, WORD, and then paste it into P8.

Question: A project was deleted by mistake. What can I do now?

Answer: If EPLAN was used to delete a project from the server, it is irretrievably lost, because Windows, the operating system, does not have a recycle bin (trash can) on the server. If the project was deleted from the local hard disk, you can restore the project from the Windows recycle bin.

NOTE: To restore projects deleted from the server, the only option is the data backup you (hopefully) created.

Question: Can I change the automatic distance of the contact image on component?

Answer: Yes. To change the automatic distance of the contact image on a component, you must call up the *Symbol properties*. Then switch to the *Display* tab, and in the *Contact image* area, deactivate the automation and enter a new, manual value for the X/Y position (offset).

Question: What do the odd frames around my symbols mean that suddenly appear when I, for example, move them?

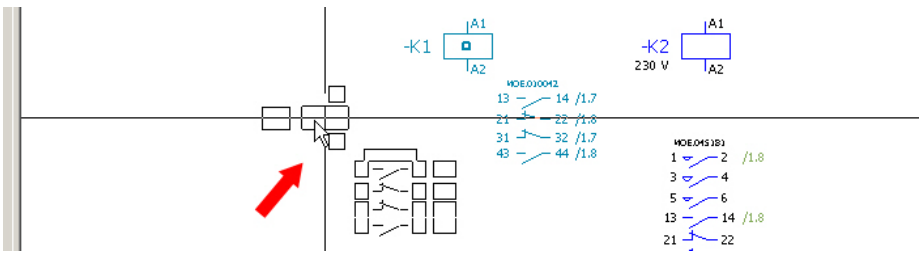


Fig. 13.31 Frames of the terminal server

Answer: If you copy, move, etc. symbols and they show odd frames during those actions, then the *Optimize terminal server* setting under **WORKSTATION/DISPLAY/GENERAL** has been activated. This setting, for example, makes the cursor change its appearance during actions like copying, moving, etc. To restore the previous behavior/appearance, deactivate this setting.

Question: Where can I find documentation on the EPLAN Electric P8 API interface?

Answer: The documentation on the API interface is currently available only to those who also purchase the API interface from EPLAN. You can also find information on using the API interface in the EPLAN Electric P8 online help.

Question: How do I access the EPLAN Data Portal?

Answer: Anyone who has a software maintenance contract will receive immediate (free) access to the Portal and the data contained therein after creating a user account.

Question: What is the easiest way to automatically rename a structure identifier, for example, a mounting location?

Answer: The easiest way to automatically globally rename a *structure identifier*, such as a mounting location+*OT12* to *+U23*, i. e. across the entire project, is to use *structure identifier management*. The mounting location used in the project, *+OT12*, will be found anywhere and renamed to the new structure identifier mounting location, *+U23*, without having to make manual changes. The structure identifier management tool can be accessed from the **PROJECT DATA** menu followed by the *Structure identifier management* menu item.

Question: In the *Select scope of menu* start dialog, what do the terms Beginner, Advanced and Expert mean?

Answer: The three settings – *Beginner*, *Advanced* and *Expert* - correspond to three fixed and defined user groups that are selectable if EPLAN is run without rights management.

Beginner refers to users with access to the functionalities that are absolutely necessary to draw schematics as well as work with macros and project data dialogs.

Advanced users can use other functions as well the previous ones (e. g. display options such as minimum text size, display of empty text boxes, etc.). They can also use data transfer.

Expert users have access to all functionalities, i. e. they can use the functions they need to prepare work such as system settings, edit master data, work with revisions and project options, back up data, etc.

NOTE: Conversely, this means: If one or the other function is missing, but according to the license should exist, check again to see the scope of menu with which EPLAN was launched.

Question: How and where can I change the view of the workspace?

Answer: Via the **VIEW/WORKSPACE** menu, it is possible to select from default views. In addition, you can create and save new workspace views of your own. As well, it is possible to save anew (overwrite) the current workspace with all its settings to a current scheme.

Question: How do I activate the mode for the direct editing of property texts?

Answer: In EPLAN, you can edit certain properties directly without having to call up the symbol properties dialog.

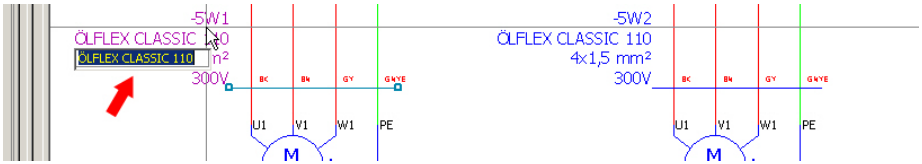


Fig. 13.32 Editing properties directly

Apart from the option of enabling and disabling this edit mode via the **OPTIONS** menu, you can also activate direct editing temporarily using the mouse or keyboard.

Using the mouse: As soon as you have selected text or a component, keep the left mouse button pressed for a short amount of time. A small window opens where you can edit the data directly.

Using the keyboard: Move the cursor near the text or component and press the [F2] key. A small window again opens for direct editing.

TIP: In the case of several properties on symbols, after activating the direct input mode, you can use Tab to "jump" to the individual properties and edit them.

Question: How and where can I change the color scheme (background color) of the graphical editor?

Answer: Under **OPTIONS/SETTINGS/USER/GRAPHICAL EDITING/GENERAL**, the color scheme of the graphical editor can be changed. The names of the color schemes indicate the color of the screen background: So, you can choose a *white*, *black* or *gray* background. The colors of the various elements are adjusted for the respective background. I recommend using the white background.

Question: How can I re-activate suppressed dialogs/messages?

Answer: To reactivate dialogs that have been deactivated before (e. g. Tip of the day), you must again set the check mark in the user settings. Go to the **OPTIONS/SETTINGS** menu and select in the now open dialog **USER/DISPLAY/USER INTERFACE** from the tree structure. There, on the right-hand side, you can select the *Reactivate suppressed messages* option. Then close the dialog again. All deactivated messages/dialogs are available again immediately. For some dialogs/messages, it may be necessary to restart EPLAN.

Question: How do I save a project or a page in EPLAN?

Answer: It is not at all necessary to save a project or page in EPLAN. Every change is directly "written" = saved (on the storage medium, such as a hard disk). As a result, additional manual saving is not necessary.

Question: How can the sequence of projects in the page navigator be changed?

Answer: It cannot be changed. The sequence of projects (in the **page navigator**) depends on the sequence in which they were opened.

Question: Can I hide the description of structure identifiers in the page navigator?

Answer: No. EPLAN automatically displays the description. It cannot be hidden.

Question: Is it possible to install EPLAN Electric P8 (all versions) side by side with a previous version, such as EPLAN 5 version (also all versions)?

Answer: Yes, absolutely, because both programs operate separately from each other.

Question: Is it possible to increase the maximum number of entries (12) in the clipboard?

Answer: No. The clipboard is limited to 12 entries. The clipboard >1 is activated as follows: Go to the **OPTIONS/SETTINGS** menu, open the **USER/DISPLAY/GENERAL** node and adjust the clipboard settings in the right area of the dialog.

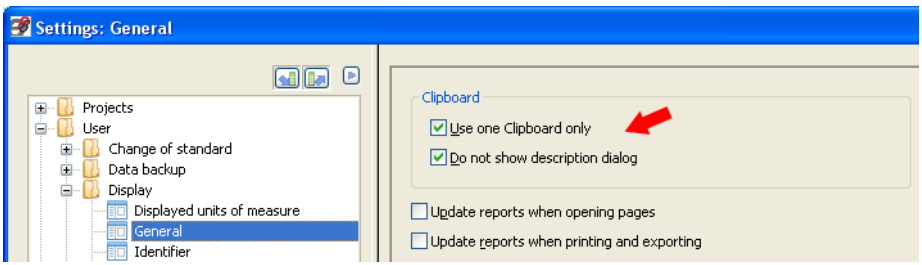


Fig. 13.33
Clipboard settings

Question: Can the *Update connections* function be assigned to a shortcut key?

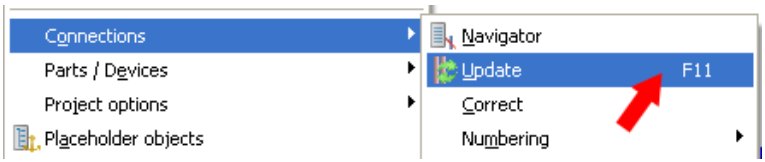


Fig. 13.34
User-created shortcut key F11

Answer: Yes, this is possible. In fact, it is recommended, because this function is used quite frequently. I recommend that you use the **F11** function key. EPLAN currently does not use this key, and it is also easy to reach and remember.

Question: How can I suppress the check message "Coil without contacts"?

Answer: If the project uses contactors/relays, etc. as spares (without using contacts), the check message P007009: Coil without contact" might be displayed.

Solution 1: This can be remedied only if you assign this coil a part with a correct function template, i. e. once you have made a device selection. Then, all spare coils in the project will be ignored for the purposes of the check run messages.

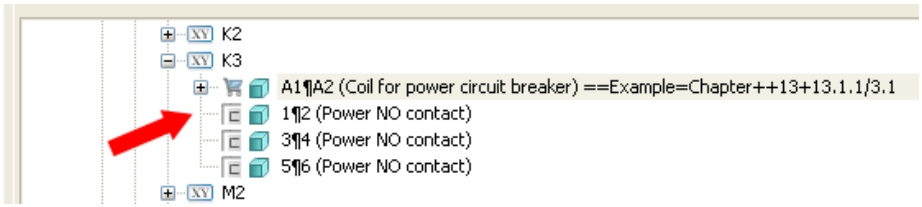


Fig. 13.35
Unplaced functions

Solution 2: Possible contact assignments (i. e. contacts like NO contact, NC contact, etc.) are added as new functions of the coil to the device navigator as unplaced functions.

Question: Can the PDF comment function in EPLAN only be used with Adobe Acrobat Writer (full version)?

Answer: No. You can use, for example, the *redlining* function (reimporting EPLAN comment entries contained in the generated EPLAN-internal PDF) with the freeware PDF-XChange Viewer (status from September 2010).

NOTE: The write protection for the internal EPLAN PDF output should be disabled (**OPTIONS/SETTINGS/USER/INTERFACES/PDF EXPORT** followed by the *General* tab). Otherwise, there could be problems when importing the commented PDF (cannot be imported). This also applies to the Acrobat software solution.

Question: What is the fastest way to find a specific structure identifier in the project?

Answer: Use the **F12** – standard shortcut key to launch the **page navigator**. In the navigator, use the right mouse button to select the **DETAILED SELECTION** menu item. The **Select pages** dialog opens. In this dialog, a prefilter is created, which is then used to filter the project's structure identifiers.

Question: Why can I not find a specific structure identifier in the project despite an activated prefilter?

Answer: If a structure identifier in the project cannot be located graphically via an activated prefilter in the **page navigator**, then it is part of an unplaced object.

Unplaced objects and their structure identifiers can be found, for example, in the **device navigator**. The arrangement/sequence of the structure identifiers is imported from the project settings. (under **DISPLAY/PROJECT STRUCTURE (NAVIGATORS).**

Question: How can lowercase letters be used/entered for a DT designation?

Answer: To be able to also enter lowercase letters for a DT, you need to open **OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/DEVICES/DT** and deactivate (remove check mark) the *Conversion to uppercase* setting.

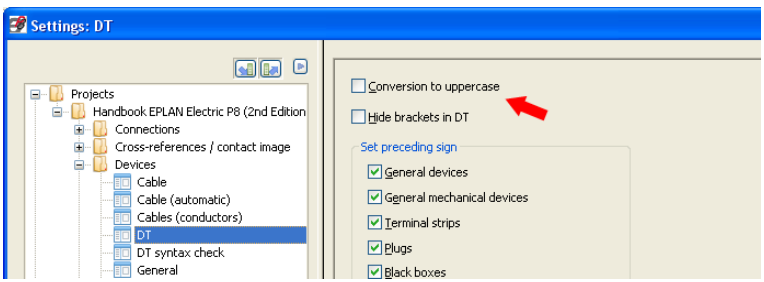


Fig. 13.36 Uppercase letters or lowercase letters

This setting applies to the following properties in the symbols dialog:

- Displayed DT
- Full DT

- Connection point designation
- Terminal designation
- Plug DT, Channel designation, and Address (for PLC connection points)
- Structure identifiers

NOTE: If the *Conversion to uppercase* is activated/deactivated, lowercase letters will/will not be automaticallyconverted to uppercase letters. The "conversion" only takes effect subsequent to the setting being activated/deactivated. This way, all "old" DTs entered remain in lower-case even if the setting is activated. Only the next new DT designation entered will be "converted" automatically to uppercase.

Question: How can I automatically insert a mechanical action line between two objects?

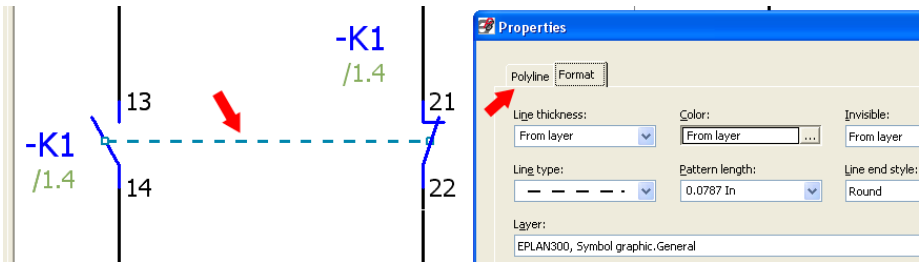


Fig. 13.37
Mechanical action line

Answer: In EPLAN, a mechanical action line is a purely graphical representation without any logical function. It is represented simply by a graphical line between two objects, or two contacts, which is then formatted according to the properties of the line.

Question: How can I open a document/image for a stored part directly from project editing?

Answer: To open a document (or image) stored for a part in parts management directly from project editing, you have to call up *properties* of the symbol and then switch to the *Parts* tab. Then, select the desired part and call the popup menu with the right mouse button.

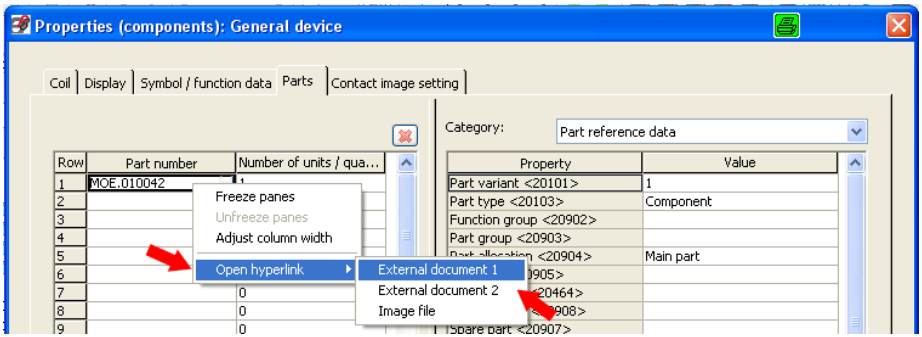


Fig. 13.38 Open hyperlinks

The desired information can be seen in this popup menu under the **HYPERLINKS** item. Here, you can find the stored external documents 1 to n (depending on the number on the part in parts management) as well as an image of the part.

NOTE: If no such information (such as external documents or an image) is stored on the part in parts management, the popup menu items will be grayed out and cannot be selected.

Question: What is the difference between a main function and a main function of superior device?

Answer: To understand this, you should picture a nested device. Device A (being the *main function*) is located, for example, in a black box B (also a main function). This makes black box B the *superior main function* of device A.

Question: What are the differences between *Save additionally*, *File off* and *Archive* in the data backup dialog?

Answer:

Save additionally: When backing up data, a complete (depending on the options selected) copy of the project is saved to another storage medium. In project management, it is also possible to back up multiple projects. Multiple selection is only possible in project management.

File off: If a project is to be passed on (e. g. to a customer), you can use the *File off* option. This creates a copy of the project (e. g. on a different storage medium), and the source project is set to read-only. This prevents changes in the meantime to this (filed off) project. If an attempt should still be made to edit such a filed off project, the user doing this will receive a message that the project has been filed off and locked for editing.

Archive: Finished (completed) projects can be archived in order to free up "space" on the hard disk. A copy of the project is saved to a different storage medium (e. g. the local hard disk), and the (finished) source project is deleted from the hard disk except for an information file.

Question: How can I start an Excel table via the external programs?

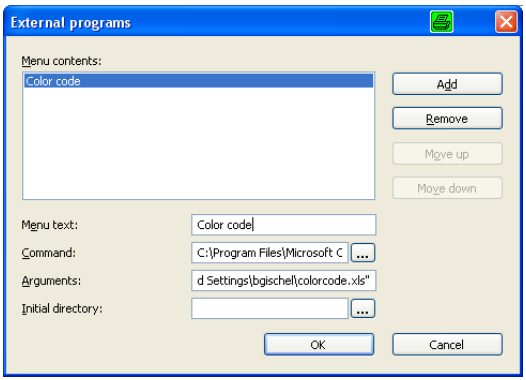


Fig. 13.39 Using external programs

Answer: To integrate an Excel table, etc. via the **EXTERNAL PROGRAMS** menu so that you call it up from the **OPTIONS** menu, proceed as follows:

1. Open the external programs via the **OPTIONS/EXTERNAL PROGRAMS**. menu.
2. Use the **ADD** button to create a new menu item.
3. Enter a suitable (subsequently accessible via the **OPTIONS** menu) *menu text*, for example "Color codes for EPLAN and other…" …".
4. In the *Command* field, enter the path to the *Excel.exe*, for example: *D:/Program Files/Microsoft Office/OFFICE11/EXCEL.EXE*.
5. In the *Arguments* field, enter the path to the Excel table: If there are blank spaces in the path name, then it in quotes, for example: "E:/EPLANP8 V2/Tools/ExcelColor codes for EPLAN E5, P8.xls"

Save the changes with **OK**. In the **OPTIONS** menu, the "Color codes for EPLAN and others…" menu item is immediately available and opens the Excel table if it is selected.

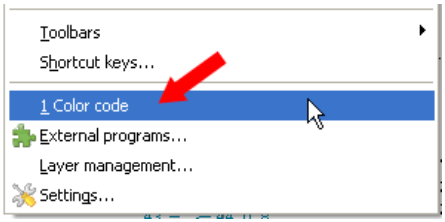


Fig. 13.40
New menu item

TIP: This menu item can also be assigned its own shortcut key.

Question: Why is the unit of the cross-section/diameter of a connection definition not displayed?

Answer: If the unit of the cross-section/diameter is not displayed, then the wrong property arrangement has been applied. The unit is not shown by default. A different property must be set to display the unit.

To display "mm^2" together with the cross-section information, you would need a property like *Cross-section/diameter with unit <31007>*. You need to add this property and format it on the *Display* tab of the **Properties** (…) dialog.

TIP: You can also manually set or selected the unit in the **Properties** (…) dialog, but this is not recommended.

You can also use the project default settings to define the *Unit* setting. Go to **CONNECTIONS/PROPERTIES** on the *Electrical engineering* tab and fill in the *Unit* field. This setting applies globally and is the recommended method.

Question: Is it possible to add information to the display window on the right in the *open project* dialog?

Answer: No. This information cannot be extended and is permanently defined by EPLAN.

Question: Where and how can I define a description for my structure identifiers?

Answer: To be able to sort structure identifiers or other descriptions, you have to enter them directly in **STRUCTURE IDENTIFIER MANAGEMENT** (**PROJECT DATA** menu).

Question: Can I change the row height (and thus the font size of the display), for example in the Properties (components) dialog or in the 'Edit in table' mode, etc., in order to be able to read it better?

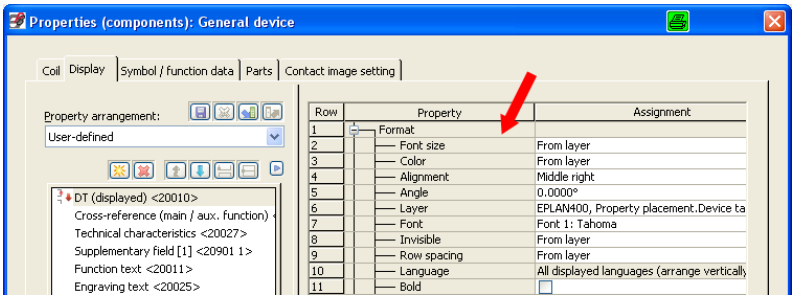


Fig. 13.41
Changing the row height – before

Answer: Yes, you can. To increase the row height (and thus the font size of the display), you click the relevant display, keep the **CTRL** key pressed and turn the scroll wheel of the mouse. Depending on the direction, the display will be increased or reduced.

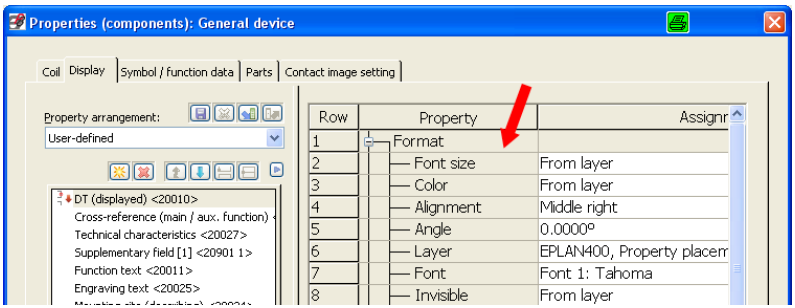


Fig. 13.42
Changing the row height – after

Question: Can I assign the *Find new DT* function in the **Properties (components) …** dialog in the *Displayed DT* field (right mouse button) to a shortcut key?

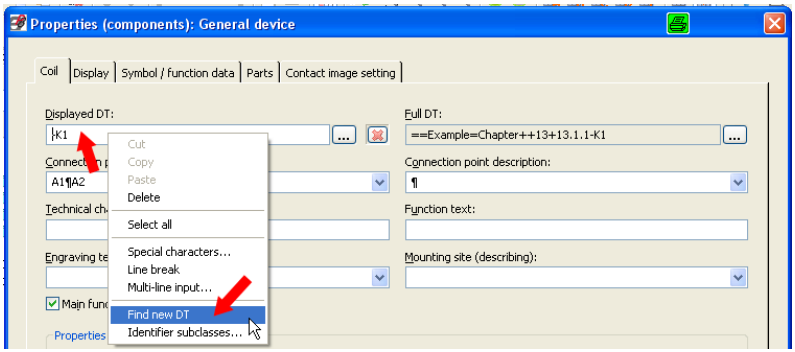


Fig. 13.43
Find new DT

Answer: You cannot use your own shortcut keys, but if you put the cursor in the *Displayed DT* field and press **CTRL + N** shortcut key (you must keep the **CTRL** key pressed), EPLAN will execute the function shown in the popup menu (**FIND NEW DT**).

Question: I received a check run message that is not relevant to me.

Answer: If checks are not necessary, you can deactivate them in the test scheme. Open the test scheme for editing, locate the corresponding check run messages and set the type of check to "No".

Question: Is there a way to fully reset the manual page sorting in the page navigator?

Answer: Yes, this is possible. You must take the following steps to fully reset the manual page sorting to default sorting:

1. Open the **page navigator** and select *List view* (tab at the bottom end of the page navigator).
2. Click the right mouse button while in the page navigator and select **MANUAL PAGE SORTING**.
3. Again right click in the dialog and select **MANUAL PAGE SORTING**.

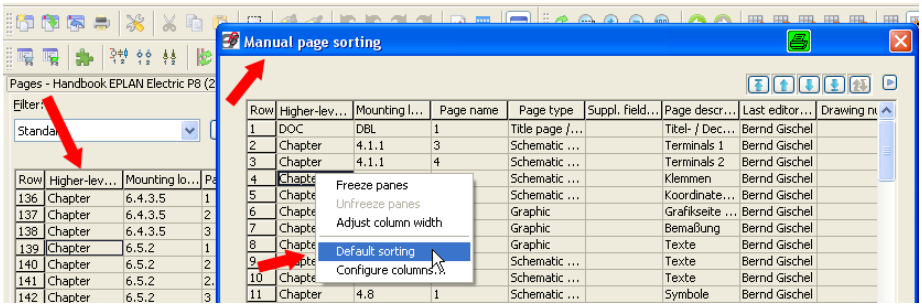


Fig. 13.44 Resetting manual page sorting

EPLAN resets manual page sorting in the page navigator to the default sorting.

Question: Is it possible to automatically find objects placed outside the plot frames?

Answer: Yes. EPLAN has a *check run message* for this. This must be activated in order for EPLAN to find objects placed outside the plot frame automatically. It is found in the check run settings under *ID 010 Cross-references* and has the message number "010001" with the message text "The placement lies outside the evaluation range of the plot frame".

NOTE: The check run message captures only logical objects. Free objects like texts, etc. are not captured by this check.

13.2 Parts

Question: While selecting a device, I noticed my part had a typing error in Designation 1. Can I remove this typing error directly without having to start parts management?

Answer: Yes, this is possible. Under *User/Management/Parts management*, **MODIFICATION ALLOWED DURING SELECTION** must be activated.

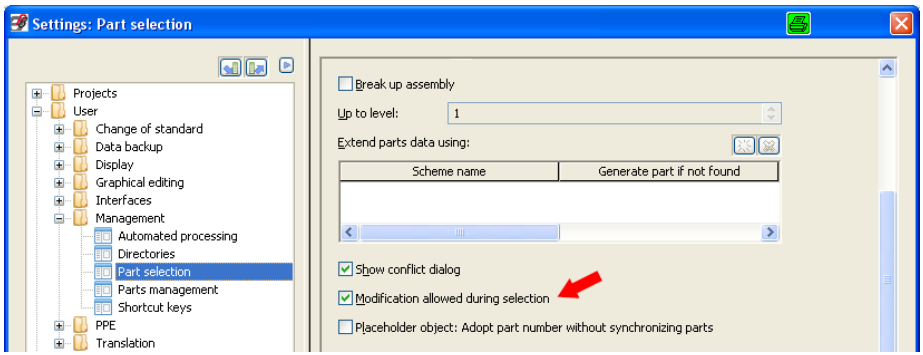


Fig. 13.45 Changing parts data setting during selection

Question: Is it possible to synchronize the stored parts data automatically upon opening the project?

Answer: Yes, this is possible. The *Synchronize stored parts when opening* setting in the project settings must be activated.

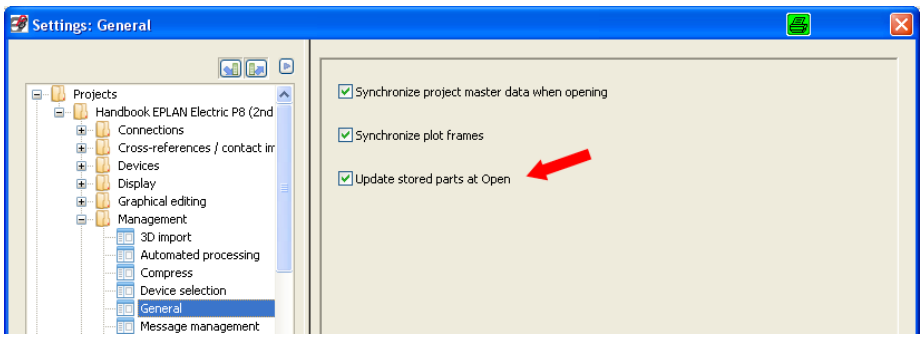


Fig. 13.46 Updating stored parts when opening the project

Question: Can I store more than one document on a part in parts management?

Answer: Yes, absolutely. EPLAN allows you to store up to 20 documents on a part in parts management. To do so, they must be entered in parts management for the desired part on the *Documents* tab and/or selected from the directories and imported.

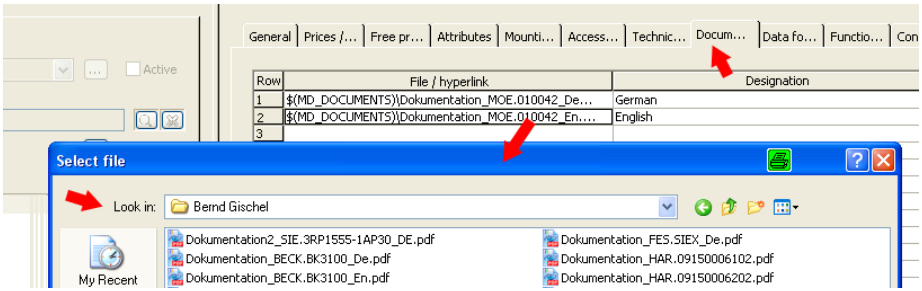


Fig. 13.47 Documents tab

Question: Why can I not enter a part on my function (the Parts tab is missing)?

Answer: Only *main functions* can have parts data. A function must be a main function in order to support parts data. If this feature is missing (the *Main function* setting has not been checked, e. g. on the *Symbol/function data* tab), the *Parts* tab will not be available either.

Question: What is the difference between device selection and part selection?

Answer: In the case of *part selection*, EPLAN does not check whether the part "matches" the device in the project when the part on the device is selected via the More button [...]) for the *Part number* on the *Parts* tab. To put it simply, you could assign a PLC card to a motor overload switch.

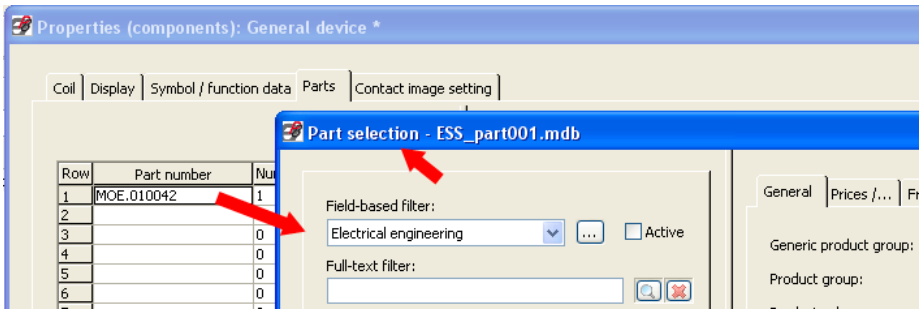


Fig. 13.48 Simple part selection

In the case of *device selection*, when you click the device selection button EPLAN automatically checks which device functions already exist and only displays parts (devices) that match the function definitions. That is to say, with device selection, it is not possible to assign to a motor overload switch the PLC card device.

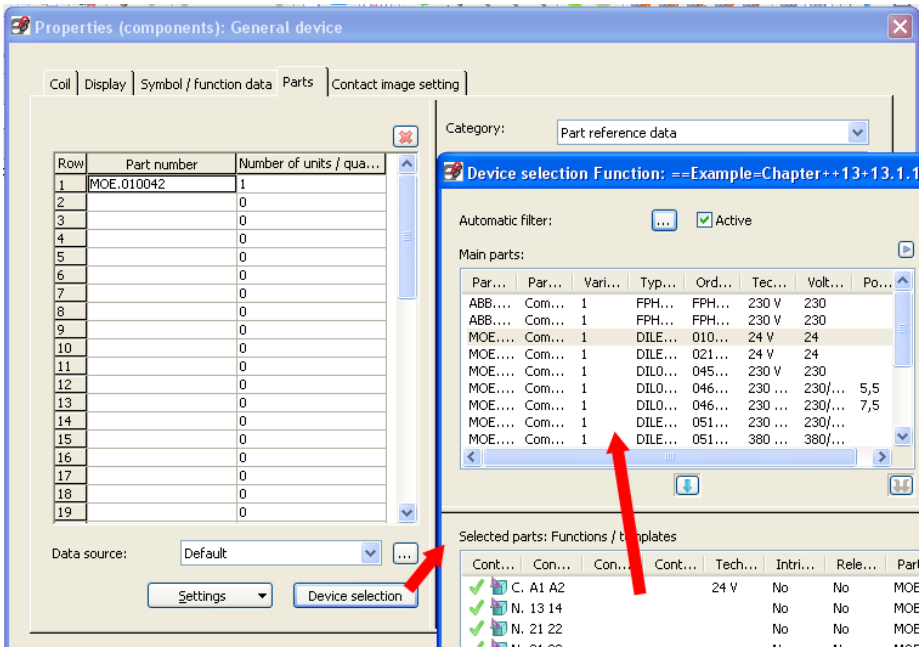


Fig. 13.49 Device selection

Question: How can I export all (or individual) parts in the project to a new parts management?

Answer: To export project parts used (stored) to a new parts management, you have to first create a new database in parts management. Open parts management and click **EXTRAS**. Select the **NEW DATABASE** function. Apply the new database as your current database and exit parts management.

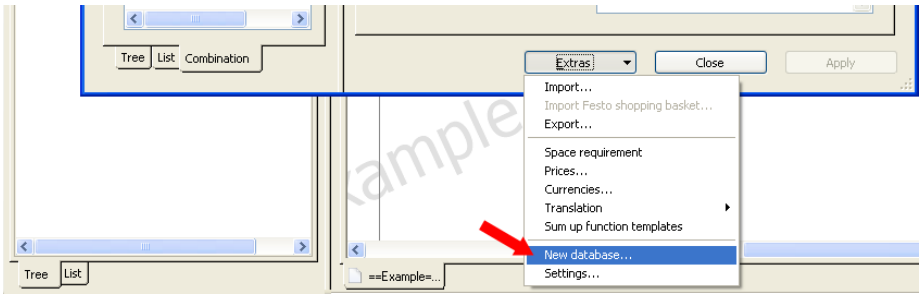


Fig. 13.50 Creating a new database

In the **UTILITIES/PARTS** menu, select the **SYNCHRONIZE CURRENT PROJECT** function. EPLAN opens the **Synchronization of parts** dialog. In this dialog, select the desired or all parts and use the arrow button to move them to the new database on the right.

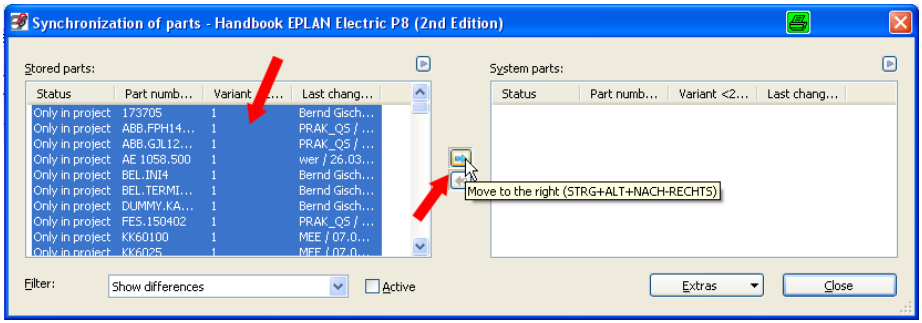


Fig. 13.51 Storing project parts in the new database

This way, all project parts are stored in a separate parts database.

Question: In parts management, how do I create new product groups, generic product groups or subgroups?

Answer: For reasons related to the data exchange, this is generally impossible. New generic product groups, product groups as well as new product subgroups cannot be created in the EPLAN parts management by the user. These groups are fixed and defined by EPLAN.

Question: In assemblies, how can I enter the function templates of individual devices?

Answer: You do not have to do this manually. EPLAN does this automatically. Simply click the **EXTRAS** button in parts management and select the **SUM UP FUNCTION TEMPLATES** menu item. EPLAN now collects the function templates of the individual devices and automatically enters them in the assembly's *Function template* tab.

Question: In parts management, is it possible to copy an existing function template of a part?

Answer: Yes, this is possible. To transfer (copy) a suitable function template of an existing part to another part that does not have a function template, you must do the following:

1. Start parts management.
2. Select a part with a suitable function template in the tree or list view.
3. Right click to open the popup menu and select the **COPY** function.
4. Now select the desired part (the one without a function template) in the tree or list view.
5. Right click to open the popup menu and select the **PASTE FUNCTION TEMPLATES** function.
6. EPLAN now transfers the function template to the part.

NOTE: If the part already has a function template, a confirmation prompt before pasting will ask you whether the existing function template should be overwritten or not. Make sure to confirm this.

Question: Is it possible to create from stored parts data a device list for an external project?

Answer: Yes, this is possible. To do so, you must export the parts data using a suitable labeling scheme and then import them in the external project as a device list.

Question: What is the meaning of the identifier on a part in parts management *Technical data* tab)? When a device is inserted, it is apparently not always transferred.

Answer: The identifier, which can be entered for the part, allows you to filter based on these identifiers in parts management (e.g. for part selection in graphical editing). It has no other meaning.

Question: How can I translate the *Designation 1 text* of parts?

Answer: To translate a part's *Designation 1* text, proceed as follows:

1. Start parts management.
2. Click the *Designation 1* field of the desired part.
3. Right click to open the popup menu and select **TRANSLATE**.
4. When EPLAN opens the **Found words** dialog, select the correct translation.
5. Click **OK** to apply the translation.

The *Designation 1 text* has now been translated and can be used in multilingual forms and other reports.

NOTE: The following condition must be met prior to any translation: The text to be translated must already exist in the dictionary.

Question: How can I make sure that parts to be provided with delivery are not listed in the reports?

Answer: To prevent, for example, parts supplied by the customer from appearing in reports, all you need to do is to give this part a characteristic that you can use later on for filtering purposes.

You could, for example, use the *Part group* in the part reference data for this characteristic. You could enter CS for customer-supplied or any other input here. You can later use this characteristic to filter out these parts so that they do not appear in reports.

TIP: In parts management on the *Technical data* tab, you can preset the *Part group* field to CS for parts that are generally customer-supplied. This way, the *Part group* is filled automatically with the entry CS on the device when the part is assigned.

Question: Where is a project-specific parts database stored?

Answer: The project-specific parts database is stored in the project directory *\...*.edb* under the file name *part.mdb*.

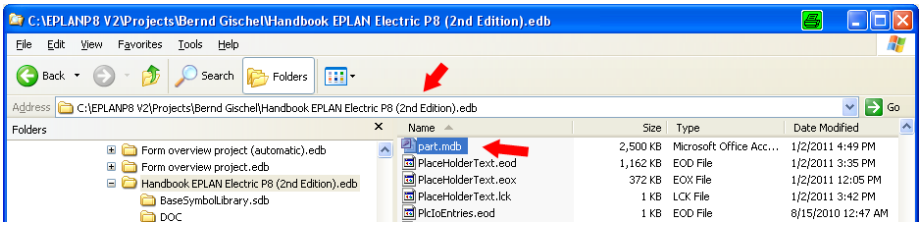


Fig. 13.52 Project-specific parts database

Question: There are many parts missing from my parts management. Where can I obtain the desired parts?

Answer: EPLAN provides only a limited selection of various parts (as sample parts) in the parts database *ESS_part001.mdb*. There are several ways to obtain additional parts. You can use the EPLAN Data Portal or visit different manufacturers' websites to look for EPLAN parts data. Finally, though, you can and will have to create the missing and/or desired parts in parts management yourself.

13.3 Terminals, plugs

Question: What is a main terminal?

Answer: A main terminal is comparable to a main function in EPLAN. A main function can support further functions, such as a contactor and its contacts. These functions can be taken from the navigators and used or placed.

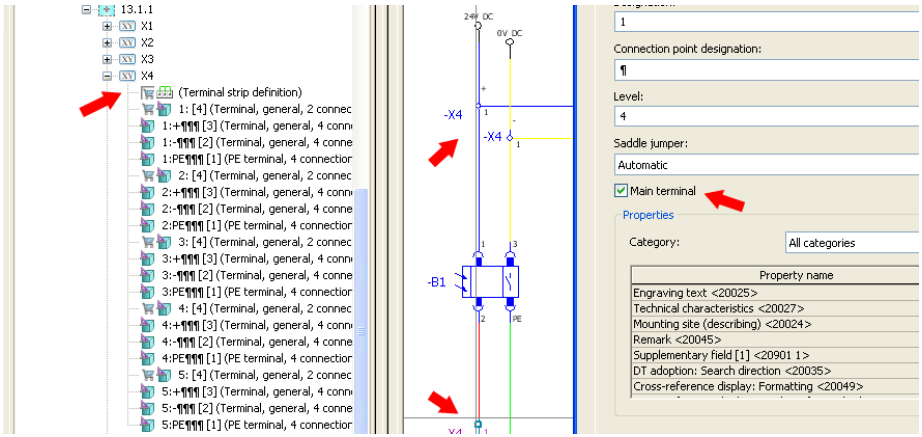


Fig. 13.53 Main terminals and auxiliary terminals in the navigator

A main terminal works the same way. An example of this is the initiator terminal. Apart from the switched connection point, there are also other connection points on the terminal, such as plus, minus or a PE connection point. These functions also exist in the navigators and can be used.

Question: How can I position terminals, i. e. define a sequence?

Answer: Terminals, for example, have properties like *Sorting <20809>* and *Sorting (graphical) <20810>*. On the basis of these two properties, terminals can be positioned freely on a terminal strip independent of their terminal designation. However, one property set excludes the other.

The *Sort code (terminal/pin) <20809>* property (free sorting (input)) also allows inputs with decimal points like 0.1. The following applies to the *Sorting (graphical) <20810>* property: If this property has been set, EPLAN will position this terminal in the terminal sequence behind the terminal on the left.

Question: Can one and the same terminal strip be output in two different reports?

Answer: No, this does not work (unless you freeze the first report). Since terminal strips are function-specific reports, there is no way in EPLAN to output one and the same terminal strip twice in a terminal diagram. Function-specific reports may and can occur in a project only once.

Question: Why do the terminal strip parts entered not appear in the parts list?

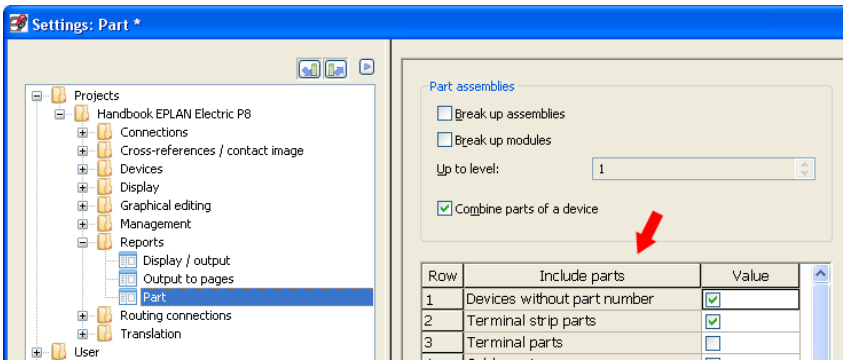


Fig. 13.54 Include parts

Answer: If articles like *terminal strip parts*, *terminal parts*, *cable parts*, *cable project parts*, *cable conductor parts*, *connection parts*, etc. are to be included in reports, for example, in the parts list, you must set the check mark in the **VALUE** column for the relevant parts (to be included) in the report settings under *Include parts*. The same is true of the desired option to output a parts list with all devices, even if they do not have any parts data (Device without part number).

Question: How can I assign "superior" function text to a terminal strip?

Answer: To assign function text to a terminal strip (for example, so that a report can be created on it in a terminal-strip overview), you must enter the function text at the terminal strip definition.

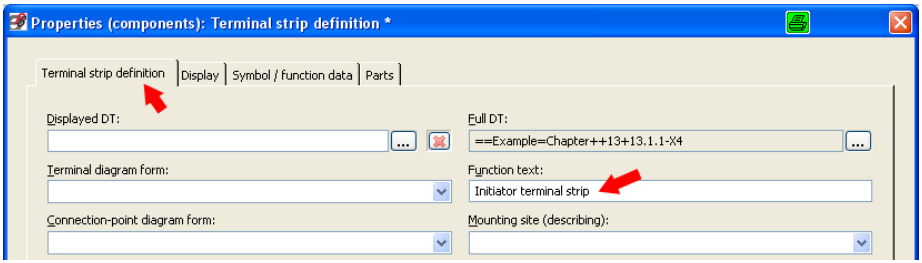


Fig. 13.55 Function text on terminal strip definition

If the terminal strip definitions do not exist, under **PROJECT DATA/TERMINAL STRIPS** you can call the **CORRECT** function and have the terminal strip definitions be generated automatically by EPLAN.

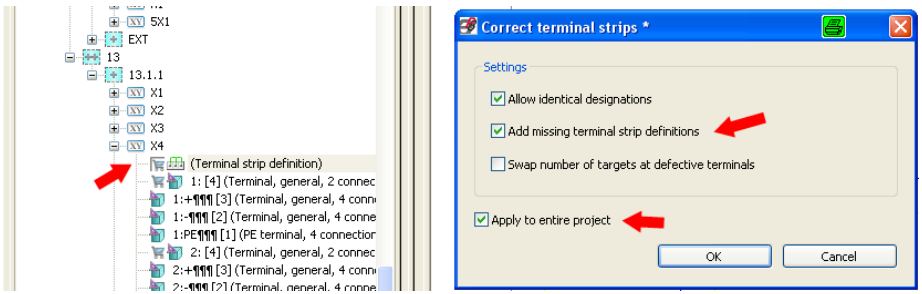


Fig. 13.56 Correcting a terminal strip

NOTE: For the correction function to work, you must select at least one terminal in the project. This can be any terminal if you want to apply the correction to the entire project.

Question: How can I generate plug definitions automatically?

Answer: To generate plug definitions automatically, call up **PROJECT DATA/PLUGS/CORRECT** and to select the *Add missing plug definitions* setting. If the function is to be executed for the entire project, the *Apply to entire project* setting must be selected.

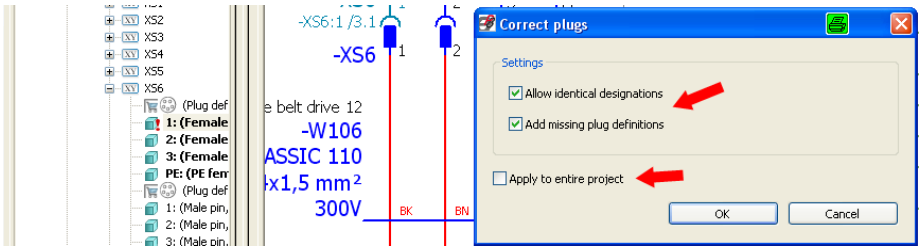


Fig. 13.57 Correcting plugs

After you click **OK**, EPLAN generates the definitions as unplaced functions (visible in the corresponding navigators, and from here they can also be placed on pages if necessary).

Question: How is it possible for terminals to have the same terminal designation, without EPLAN outputting an error message?

Answer: To avoid error messages in the check run for terminals sharing the same terminal designation, e. g. in the case of PE terminals and the PE terminal designation, you must activate the *Allow same designations <20811>* property in the symbol properties of the terminal.

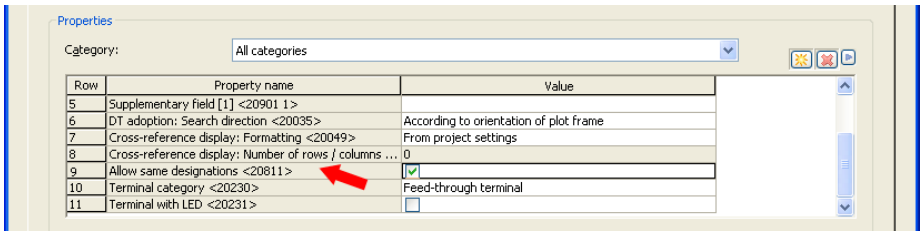


Fig. 13.58 Allow same terminal designations property

Question: How can I remove superfluous terminal strip or plug definitions from the project automatically?

Answer: To avoid having to delete superfluous terminal strip definitions (or plug definitions) from the terminal strip or plug navigator manually, you create a scheme in the **PROJECT/ORGANIZE/COMPRESS** menu that will remove this superfluous data from the project automatically.

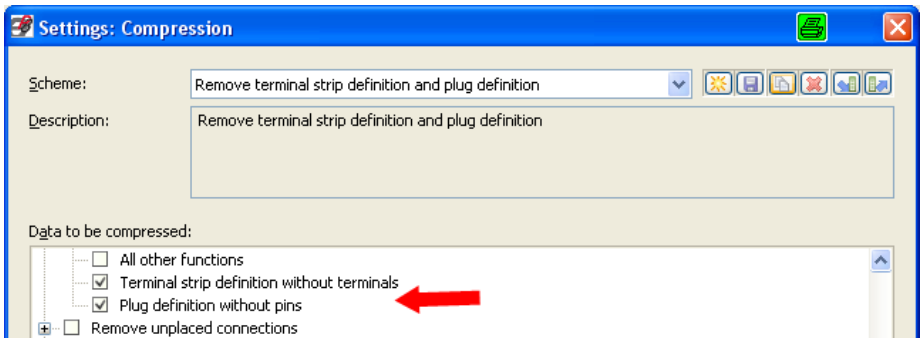


Fig. 13.59 Compression scheme

Question: How can several terminal diagrams be output on a report page in a space-saving manner?

Answer: To output several terminal diagrams (terminal strips) on a single report page, several conditions must be met. The form to be used must be a dynamic form. As well, in the **REPORTS** dialog (**UTILITIES/REPORTS/GENERATE** menu) under the **SETTING/OUTPUT TO PAGES** button, a check mark must be placed for the respective report in the *Combine* column.

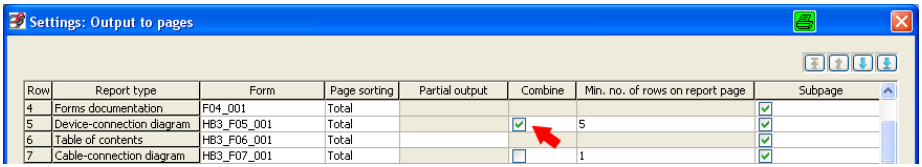


Fig. 13.60
Activate combination for a report

In addition to the *terminal diagram* report, the *Combine* option is also available for the following reports: Device connection diagram *.f05; Cable diagram *.f09; Cable-connection diagram *.f07; Terminal diagram *.f13; Terminal-connection diagram *.f11; Terminal line-up diagram *.f12; Enclosure legend *.f18; PLC diagram *.f19; Plug diagram *.f22; Pin-connection diagram *.f21, Symbol overview *.f25, Mounting list *.f32 and Topology: Routing path list *.f35.

13.4 Cables

Question: Why can't EPLAN execute a correct device selection of cables (incl. assignment of conductor information) on open interruption points?

Answer: An open interruption point in EPLAN is not automatically considered a logical connection. This can be seen very well when you place the cursor on the Autoconnecting line and then view the tool tip. This is why device selection and the automatic placement of conductors is supported only on valid (connected) logical connections.

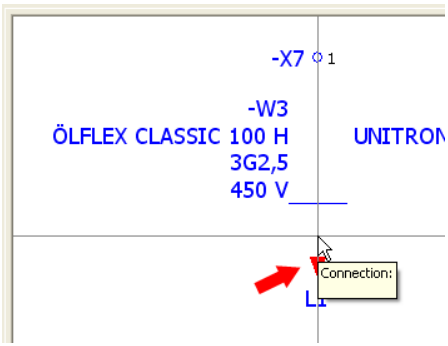


Fig. 13.61
Open connection

Question: What are the identifying properties of a cable definition?

Answer: The following properties of a cable are identifying properties: The actual *function definition*, the *shield designation* (Shielded by), the *pair index*, the *potential type*, the (conductor) *color* and/or *number* and the *cross-section* and/or *diameter*.

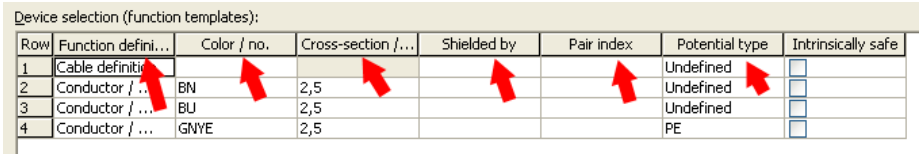


Fig. 13.62
Identifying properties of a cable part

Accordingly, these properties must match the function definitions of a device selection when the cable and/or its connection data were previously edited manually (e. g. the cable conductors or the pair index may have been inserted manually).

If one of these properties does not match the cable (in case of a device selection) in parts management, the device selection will not be able to find a suitable cable.

Question: Why are cable details, e. g. conductors of a cable, listed multiple times in the cable navigator?

Answer: If cable details such as conductors or the shopping cart are listed in the cable navigator multiple times, it indicates that several cable definitions (or lines) (displayed in a distributed manner) exist in the project and that they are all marked as *main functions* containing one part each.

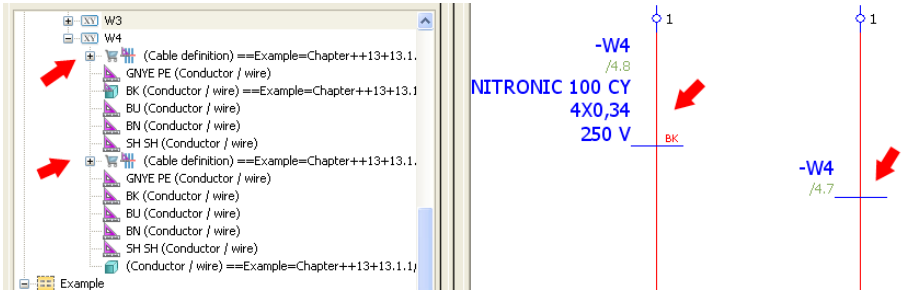


Fig. 13.63 Duplicate cable definition in the navigator

In the case of cables, too, EPLAN only allows for one main function. However, it does not matter to EPLAN where, with which representation type or on which cable definition the main function and the corresponding parts data are located.

Question: How can the conductors of a cable be assigned automatically in EPLAN?

Answer: To have EPLAN automatically "fill" in the conductor information for a cable (like color or cross-section), you need to create the cable as a part complete with all conductors in parts management.

If, when inputting the cable in the schematic, you use device selection to select and apply the cable, then when you exit the *Symbol properties cable* dialog, the conductors (crossing the connections) will be labeled by EPLAN automatically based on the parts management data.

Question: How can I place free conductors of a cable on terminals, without displaying them in the project?

Answer: To place free conductors of a cable on terminals, without representing them in the diagram, use the **INTERCONNECT DEVICES** function in the **PROJECT DATA/DEVICES** menu.

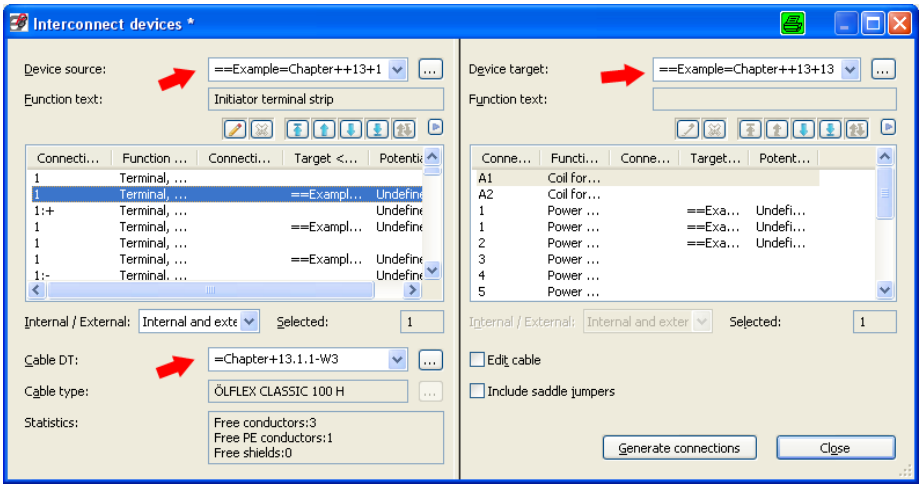


Fig. 13.64 "Virtual" interconnection of devices

Here, you can select the desired terminal strips/devices and the desired cable. Then, the corresponding terminals/devices are selected, followed by the **GENERATE CONNECTIONS** button to establish the connections.

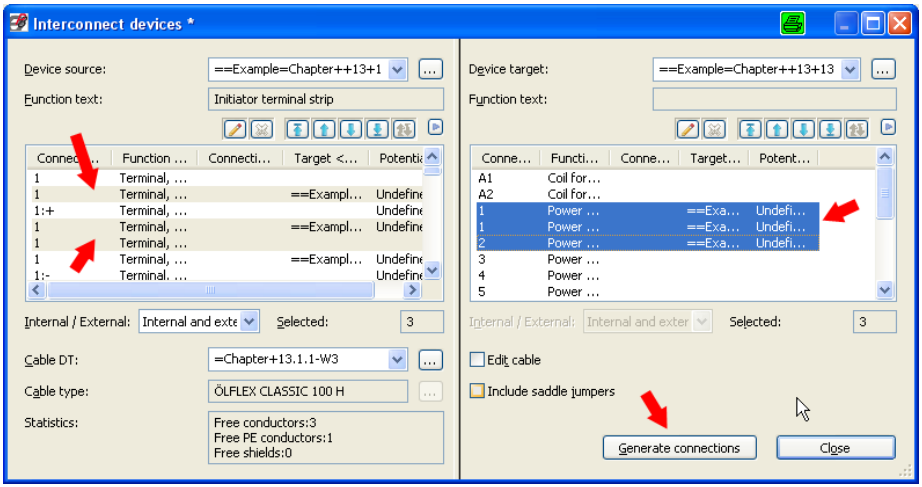


Fig. 13.65 Select devices and generate connections

A "virtual" connection is displayed visually in EPLAN with a small red slash.

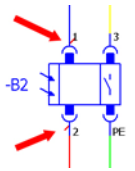


Fig. 13.66 Selecting

Question: How does EPLAN define the source and target of a cable connection?

Answer: When creating reports, EPLAN goes from right to left and from bottom to top and then determines the source and target. This means that when the cable runs from left to right, then the source would be on the left and the target on the right. The same is true of bottom to top. In this case, the source would be at the bottom and the target at the top.

13.5 PLC

Question: How can identical address ranges be used with different PLCs in a project?

Answer: To assign identical address ranges within a project (e.g. when using several CPUs), it is absolutely necessary to "attach" a CPU name to the individual PLC main functions. This way, EPLAN can then differentiate, without a check run message, between several identical addresses on the basis of the CPU (name) having been assigned.

Row	Property name	
12	Version <20418>	
13	Start address of PLC card <20419>	
14	Address range <20432>	
15	CPU: Name <20433>	Station 77
16	CPU <20167>	☑
17	Bus coupler <20164>	☐

Fig. 13.67 CPU name property

The CPU name is entered in the *CPU: Name <20433>* property of the PLC box of the main function (for all main functions of the PLC components that are part of this CPU).

Question: Do I have to assign to the PLC connection points (inputs or outputs) or PLC boxes a CPU name?

Answer: It depends on the number of CPUs (PLC stations) in use in the project. If there is only one PLC station, it is not necessary to assign a CPU name (*CPU: Name <20433>* property on the main function). But if there are several PLC stations, a CPU name (again only on the main function) should always be assigned. This will also allow you to output separate assignment lists.

13.6 Properties, layers

Question: Can I assign in the project properties my own property names?

Answer: Yes, this is possible. Open the project settings (**OPTIONS/SETTINGS** menu) and in the **MANAGEMENT** branch, open **SUPPLEMENTARY FIELDS**. In the *Project* tab, you can now assign your own names to *User supplementary field n <4000n>*.

Question: What information is located in the *<10200> (Number of pages per page type)* project property?

Answer: This property/value is determined automatically from the project. Nothing is being added here; the values are derived from the project properties on the *Statistics* tab.

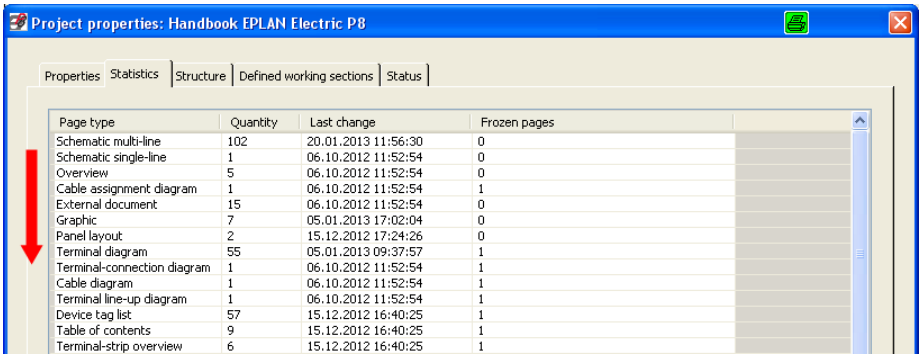


Fig. 13.68 Statistics of the different page types

This is how the *Number of pages per page type 10200 [n]* property breaks down:

[1] Schematic multi-line
[2] Schematic single-line
[3] Overview
[4] Cable assignment diagram
[5] External document
[6] Logo cad TRIGA drawing (no longer used)
[7] Graphic
[8] Panel layout
[9] Terminal diagram
[10] Terminal-connection diagram
[11] Cable diagram
[12] Terminal line-up diagram
[13] Device tag list
[14] Table of contents
[15] Terminal-strip overview
[16] PLC card overview
[17] Title page/cover sheet
[18] Symbol overview
[19] Connection list
[20] Potential overview
[21] Cable overview
[22] Parts list
[23] Plug overview
[24] PLC diagram
[25] Device connection diagram
[26] Cable-connection diagram
[27] Pin-connection diagram
[28] Plug diagram
[29] Enclosure legend
[30] Summarized parts list
[31] Structure identifier overview
[32] Forms documentation
[33] Plot frame documentation
[34] Fluid power schematic
[35] Revision overview
[36] Option overview
[37] Placeholder object overview
[38] P&I diagram
[39] Manufacturer/supplier list
[40] Model view
[41 – 42] Reserved
[43] Topology
[44 – ?] Reserved for subsequent page types

Question: How can I carry over the property arrangements quickly from one device to another?

Answer: To transfer the painstakingly created arrangements of properties from one device to another, you can use the **COPY FORMAT** and **ASSIGN FORMAT** functions in the **EDIT** menu in EPLAN.

To transfer a property arrangement, first select the (completely) formatted device. Then go to the **EDIT** menu and select the **COPY FORMAT** command. Then select the device to which the previously copied format is to be transferred. Now go to the **EDIT** menu and select the **ASSIGN FORMAT** command. The format of the property arrangement has been transferred to the device.

TIP: Block functions such as multiple selection can be used here. You can select multiple devices on a page and simultaneously transfer the form to them. This also works in the navigators. In this case, you need to select the devices to be modified and assign the format.

NOTE: Formats can be transferred only to similar (same function) devices. You could not, for example, transfer a cable format to a motor since it has a totally different function.

Question: On which layer is the "anchor" (placeholder object) for a macro with value sets located?

Answer: The layer of the so-called anchor (placeholder object; symbol 323/PLHO) of a macro with value sets is located on *EPLAN322 Symbol graphic.Macro.Placeholder object*. Here, for example, you can make settings whether the symbol is to be printed or whether it is to be visible in the diagram.

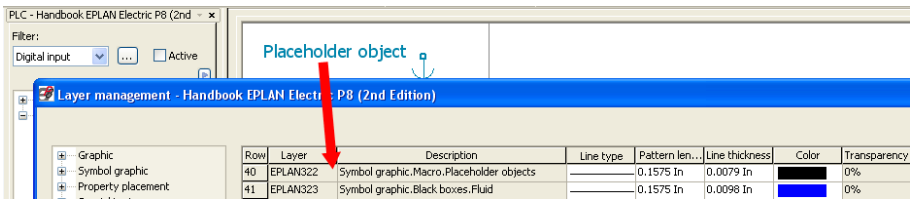


Fig. 13.69 Properties of the EPLAN322 layer

The symbol with the symbol number 323 (symbol name PLHO) originates from the SPECIAL symbol library and cannot be modified, because the SPECIAL symbol library is protected against modifications by EPLAN.

Question: On which layer is the part definition point symbol (symbol number 80, symbol name PDP) located?

Answer: The part definition point symbol (**INSERT – PART DEFINITION POINT** menu) is located on the layer *EPLAN321 Symbol graphic.Part definition points*.

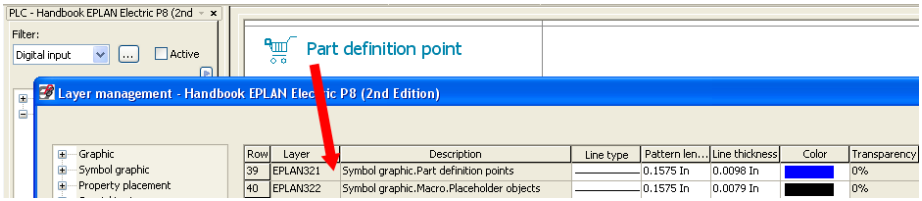


Fig. 13.70 Properties of LAYER321

The symbol with symbol number 80 (symbol name ADP) is also from the SPECIAL symbol library. This symbol cannot be changed since the SPECIAL symbol library is write-protected by EPLAN.

Question: How can I change the thickness (line thickness) of a dimension line (dimensioning functions)?

Answer: To change the thickness of a dimension line for different dimensioning functions, you can/must edit the *EPLAN107 (Graphic.Dimensions)* layer in layer management.

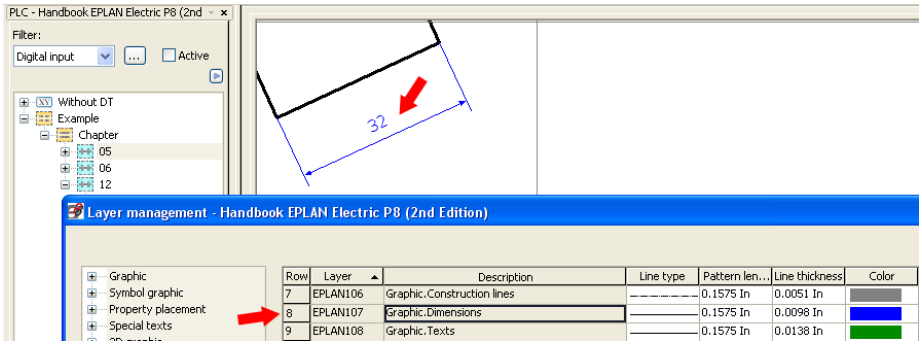


Fig. 13.71 Properties of the EPLAN107 layer

Question: What is the meaning of the three different options for the *cross-reference display <20021> property*?

Answer: There are three options for the *Cross-reference display <20021>* property.

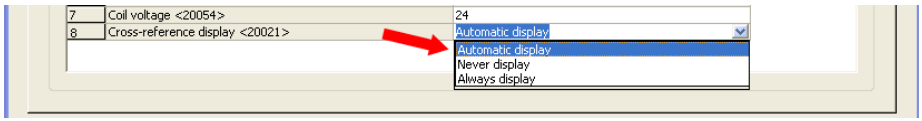


Fig. 13.72 Differences in cross-reference display <20021>

They regulate the display of automatic cross-references between the main function and its auxiliary functions and function as follows:

Automatic display: A cross-reference is displayed only if the *Displayed DT* field is filled with a correct DT, i. e. when it is not empty.

Never display: A cross-reference is never displayed; existing cross-references are suppressed in the display.

Always display: Cross-references are generally displayed.

Independent of the displays, there is always a logical connection between the main function and the auxiliary functions. All navigation commands, such as **GO TO...**, work as usual.

Question: Where can I change the text size of the part number or the type number above-/below the contact image?

Answer: This information (formatting options) is located on the layer *EPLAN480 Property placement.Part number* (for the part number) and on the layer *EPLAN481 Property placement.Type number* (for type designation).

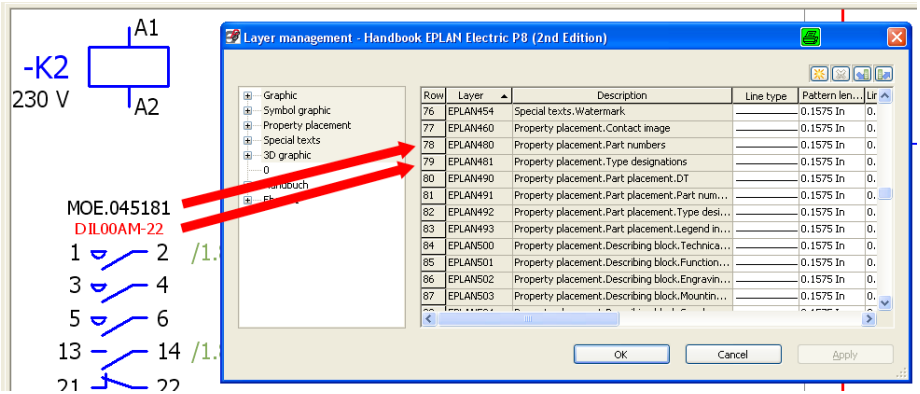


Fig.13.73 Properties of the EPLAN480 and EPLAN481 layers

Question: Do the layer management settings generally apply globally, or are they project-specific?

Answer: The layer management settings (called via the **OPTIONS/LAYER MANAGEMENT** menu) always refer to the current project.

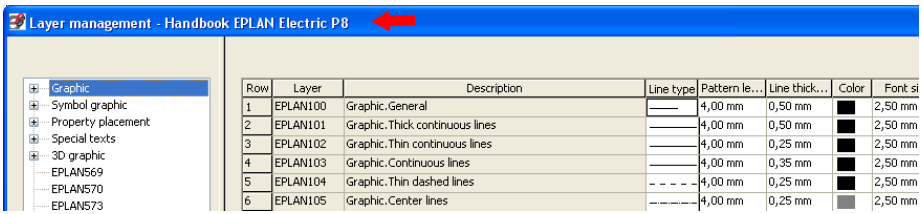


Fig. 13.74 Layer management and the project

Question: On which layer are the color settings for e. g. the alignment box of texts located?

Answer: The alignment box, for example, can be adjusted on the *Format* tab in the text dialog. It does not have a separate layer of its own. All settings (color, invisible, etc.) are controlled via the *EPLAN106 Graphic.Texts* or *EPLAN110 Graphic.Path function texts* layers.

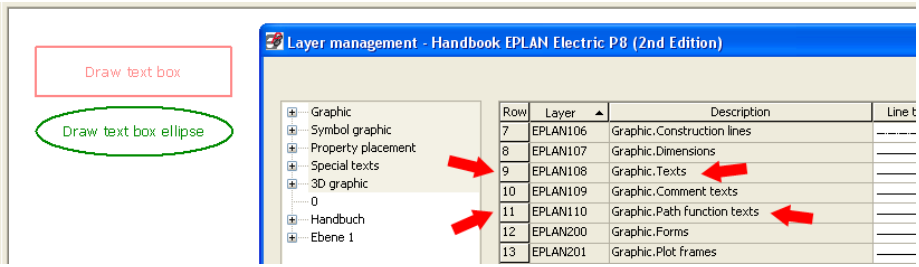


Fig. 13.75 Properties of the EPLAN108 and EPLAN110 layers

Question: How can I display the description of structure identifiers on devices?

Answer: If you wish to display the descriptions of structure identifiers on devices, you simply have to show on the desired device the corresponding property in the display of properties (*Display – Property arrangements – Components* tab).

The following properties are required in order to display structure identifier descriptions:

- *Description: Higher-level function <1130>*
- *Description: Higher-level function number <1730>*
- *Description: Installation site <1430>*
- *Description: User-defined <1630>*
- *Description: Document type <1530>*
- *Description: Mounting location <1230>*
- *Description: Functional assignment <1330>*

13.6.1 Master data

Question: How can I include row numbers in the plot frame?

Answer: You must enter the following plot frame settings in the plot frame editor:

- First, open the plot frame via the **UTILITIES/MASTER DATA/PLOT FRAME/OPEN** menu to edit it.
- Insert the *row texts* in the graphical editor with the desired number (**INSERT/SPECIAL TEXT** menu).

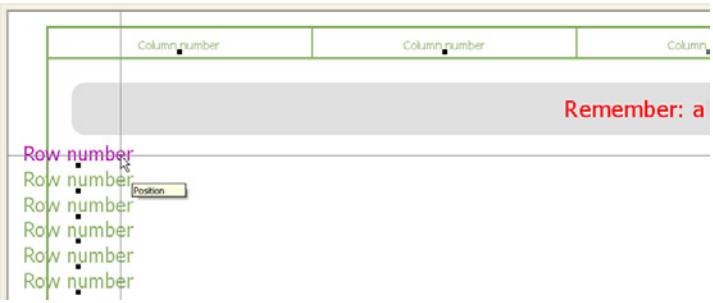


Fig. 13.76 Inserting row texts

- Use **CTRL + M** followed by **CTRL + D** to call the *Plot frame properties*.
- Enter the *number of rows*, define the *row height*, the *strings* and the *numbering format* for the rows and exit the plot frame properties by clicking **OK**.

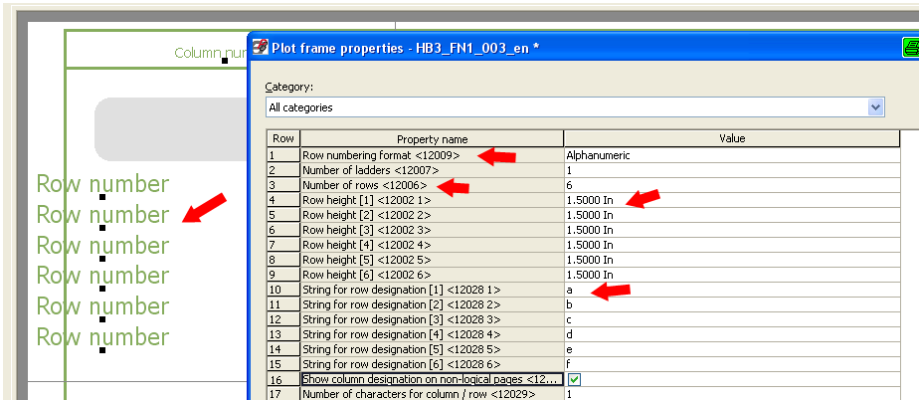


Fig. 13.77 Entering properties for the rows

- Select **UTILITIES/PLACE COLUMN AND ROW TEXT AGAIN**. EPLAN now automatically places the *column texts* and *row texts* anew on the basis of the widths and heights taken from the plot frame properties. If necessary, you can manually correct these distances or enter a graphical line if you want.

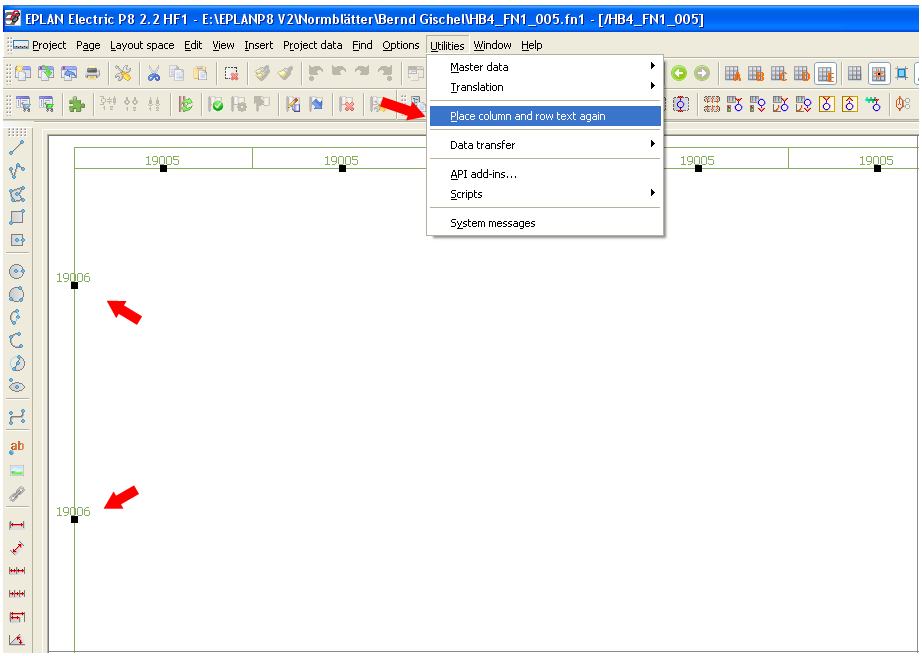


Fig. 13.78 Letting EPLAN determine the distances automatically

Editing of the plot frame is complete. It can now be closed and used.

Question: What is the difference between the symbols XBD and XBD2?

Answer: Given the correct "counter symbols", one combination does not generate a connection (XBD), while the other does (XBD2). This way, as well, the counterpart of the plug (female pin) is found and cross-referenced.

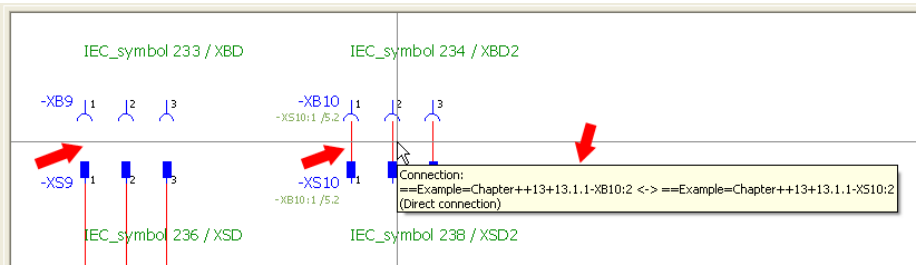


Fig. 13.79 Differences between XBD/XBD2 and XSD/XSD2 symbols

Question: Why is my report form, e. g. a terminal diagram, larger than my plot frame?

Answer: This is an imported EPLAN 5 project. The plot frame (a holdover from the old EPLAN 5 version) was left in the project settings, but reports are now created on the basis of the EPLAN Electric P8 report forms. Since plot frames imported from the old EPLAN 5 versions are smaller than the EPLAN Electric P8 report forms, the report forms are "written out", or placed, via the plot frame.

The only thing that helps is to select a new plot frame from the EPLAN system master data, such as *FN1_001.FN1*, in the project settings or to change an existing plot frame that corresponds to the old plot from (from EPLAN 5) and enter this changed plot frame in the project settings. The plot frame is set in the project settings: **OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/MANAGEMENT/PAGES**.

Question: Where can I exchange the global standard plot frame for another?

Answer: The global standard plot frame (considered a default setting for all pages) can be modified or replaced in the project settings under **MANAGEMENT/PAGES**.

NOTE: If a page has been assigned a plot frame directly (via the page properties, *plot frame name 11016* property), the global standard plot frame will not apply. In this case, it is always the plot frame assigned to the page that is displayed.

Question: Where can I assign a different legend form to a specific mounting panel (in the 2D panel layout)?

Answer: To select for a specific mounting panel a form other than the one globally set in the reports, select the mounting panel, open the properties of the mounting panel and then insert here the *Legend form <20440>* property (unless available).

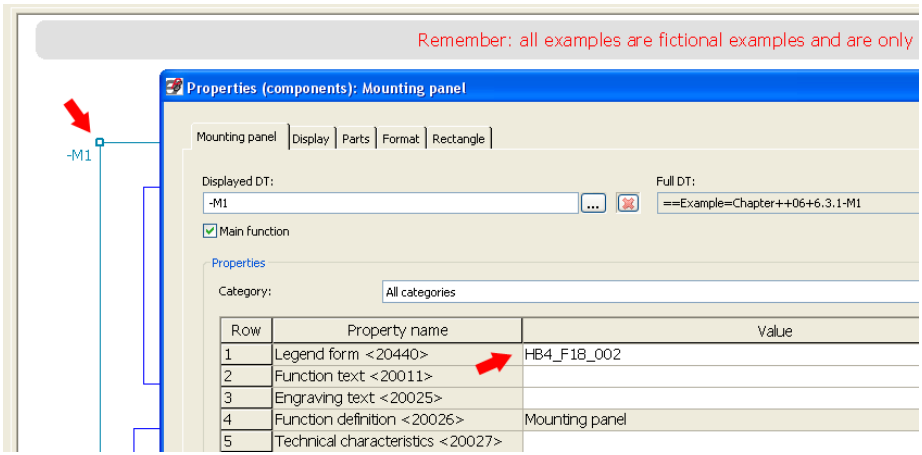


Fig. 13.80 Legend form entry

In the *Value* column, you can now enter a different form for this mounting panel.

Question: How can I assign to a form a separate plot frame as default, so that the standard plot frame is not imported from the project settings?

Answer: To assign to a report (a terminal diagram for example) generally a plot frame different from the standard plot frame of the project, you must do the following.

- Open the desired report (form) via **UTILITIES/MASTER DATA/FORM/OPEN**.
- Select and open the desired form in the **OPEN FORM** dialog.
- In the *Form properties* (accessible by pressing **CTRL + M** shortcut key followed by **CTRL + D**), set the *Use 'Plot frame to edit form' property for reports <13055>* property and then set the desired plot frame in the *Plot frame to edit form <13001>* property that should be automatically set in the future for this report (form).

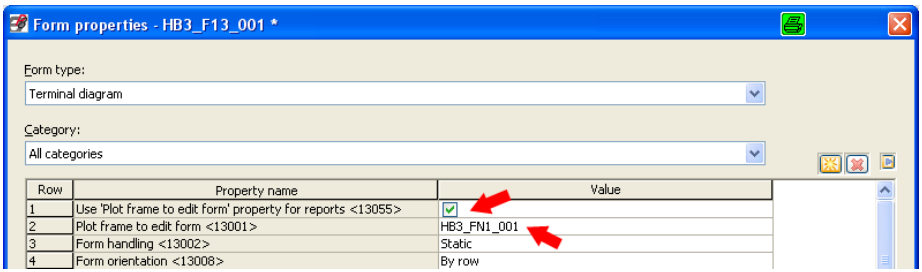


Fig. 13.81 Customizing form properties

- Close the form properties and then the form.

From now on, this form will always be automatically assigned the plot frame defined in the form properties for the purposes of graphical output.

Question: What form information is shown in the **Open form** dialog on the right-hand side, and is it possible to add to this information?

Answer: The following form information appears in the right-hand field of the **Open form** dialog.

- Description: Description (form, plot frame, outline) <18011>
- Company code:
- Creator: Bernd Gischel
- Creation date: 18.09.2010
- Last edited by: Bernd Gischel
- Modification date: 18.09.2010
- Form handling: Static form

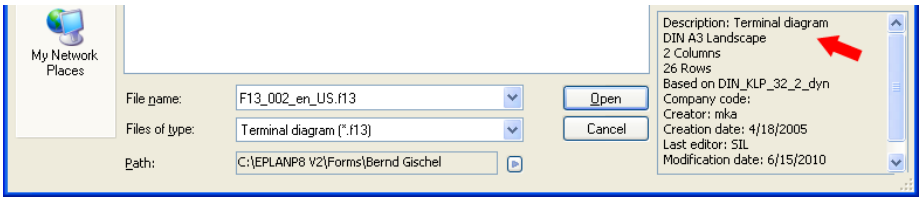


Fig. 13.82 Form properties in the Open form dialog

This information shown cannot be extended.

Question: What form information is shown in the **Open plot frame** dialog on the right-hand side, and is it possible to add to this information?

Answer: The following form information appears in the right-hand field of the **Open plot frame** dialog.

- Description: Description (form, plot frame, outline) <18011>
- Company code:
- Creator: Bernd Gischel
- Creation date: 18.09.2010
- Last edited by: Bernd Gischel
- Modification date: 18.09.2010

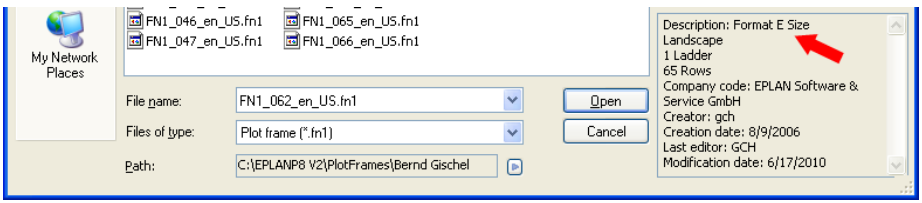


Fig. 13.83 Form properties in the Open plot frame dialog

Here, too, it is not possible to add further information to this display.

Question: How can I edit a form that is located only in the project received from a supplier and not in the system?

Answer: To edit a form (report) that is located only in the project master data and not in the system master data – e.g. a project sent by an external supplier who uses proprietary forms in the project – you must do the following:

1. Open the project that contains the form.
2. In the **UTILITIES** menu, select **MASTER DATA/UPDATE CURRENT PROJECT**. The **Synchronize master data** dialog opens.
3. Select the form not stored in the system master data in the left area of the dialog (the project master data) and use the button in the center to copy it to the EPLAN system master data on to the right.
4. Close the dialog. Now, the form (report) can be edited as usual in EPLAN (master data editor).

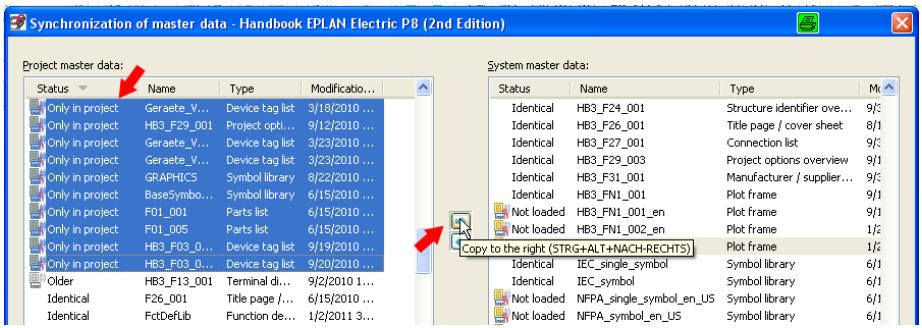


Fig. 13.84 Synchronizing master data when working with external master data

NOTE: This approach also applies to the remaining master data only stored in the project, such as plot frames, etc.

Question: How can I change the "fixed" project properties, e.g. on the cover/title page?

Answer: You can open the project properties directly via the **PROJECT/PROPERTIES** menu path and change the desired entries in the *Properties* tab.

Or you could select the project in the **page navigator** (you only need to select one page), call up the popup menu via the right mouse button and then also select **PROJECT/PROPERTIES**.

Question: When I generate a report, I receive the message "S079005 form does not exist." What can I do?

Answer: If EPLAN outputs this message when you generate a project report or a specific report, then you are missing the relevant form that was entered in the page outputs or report template.

In the **Reports** dialog, under the **SETTINGS** button, open the **OUTPUT IN PAGE** menu item. Here, select an existing form in the Report row, e. g. terminal diagram, from the *Form* column. Or you can select and apply an existing form in the relevant (report) template (in the **Reports** dialog; *Templates* tab) from the *Form* row.

Question: How can I set a symbol (or its symbol graphic) to invisible?

Answer: It is not possible to set a symbol (or the actual symbol graphic) to invisible. To set a symbol (or the actual symbol graphic) to invisible, the existing symbol must be exchanged for a symbol from the symbol libraries that precisely matches this symbol, with the exception that its symbol graphic properties are set to "invisible". If there is no "invisible" (counter) symbol in the symbol libraries, the symbol cannot be set to invisible.

Question: Is it possible to "convert" symbols to graphic, for example, to reduce or enlarge them for internal connections?

Answer: Yes, this is possible, and it is quite easy to do. You simply click the symbol to select it and then call up the **EDIT/OTHER/CONVERT COMPONENT INTO GRAPHIC** menu. EPLAN then "breaks down" the symbol and generates a separate handle for each element.

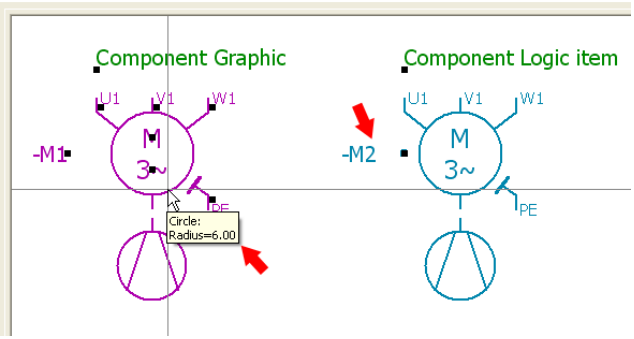


Fig. 13.85 Component converted into graphic

NOTE: This action can only be reversed immediately (i. e. as soon after the conversion as possible). If the project has been closed, or the **UNDO** list has been cleared for any other reason, you will no longer be able to "undo" the conversion (**GRAPHIC > COMPONENT/SYMBOL**).

Question: How do I remove a symbol library from the project?

Answer: Generally, this only works when all symbols in use from this symbol library have been removed from the project completely. Otherwise EPLAN will not let you remove the symbol library because these symbols might still be in use.

13.7 Data exchange

Question: Does EPLAN Electric P8 have an EXF interface like the previous EPLAN 5?

Answer: No, EPLAN Electric P8 does not have such an EXF interface. It isn't needed anymore because existing interfaces to programs like Excel allow you to easily and conveniently change data in EPLAN Electric P8 "externally".

Question: How do I convert EPLAN Electric P8 projects to the previous EPLAN 5 or EPLAN 21 versions?

Answer: Logically (i. e. with evaluable information), this is not possible at all. EPLAN Electric P8 projects are not backward compatible with EPLAN 5 or EPLAN 21. In purely graphical terms, for example in the DXF/DWG exchange format, this can be done. Of course, this may be subject to certain limitations regarding representation, line thickness or font size.

Question: Is there a viewer for EPLAN Electric P8?

Answer: Yes. There is a viewer for EPLAN Electric P8. It can be purchased from EPLAN.

Question: Should imported EPLAN 5 or EPLAN 21 projects be used as a basis for new projects in EPLAN Electric P8?

Answer: Although the import function is quite good (special or totally exotic "distorted" legacy projects aside), I personally would not use imported EPLAN 5or EPLAN 21 projects as a basis for future EPLAN Electric P8 projects.

Of course EPLAN 5 or EPLAN 21 projects could be enhanced to include the new EPLAN Electric P8 options, and at first this appears to require less effort than creating a new EPLAN Electric P8 project from scratch. But converted data can be tricky or not really be optimized for the new system (in this case, EPLAN Electric P8)

An example would be the additional auxiliary contacts of a motor overload switch, which in EPLAN 5 had to be drawn on the actual motor overload switch to obtain a cross-reference from the pair cross-reference. This approach is also possible in EPLAN Electric P8, but not necessary. It is not necessary, because a motor overload switch with auxiliary contacts can be created as a part including the function template, thus displaying the "contact image" on the motor overload switch automatically following device selection. This means that you do not have to set auxiliary contacts again, only to display the cross-reference to the auxiliary contact used on the motor overload switch (and its auxiliary contacts).

There are a number of examples where EPLAN Electric P8 has substantially more elegant and more convenient solutions. This is why starting from scratch in EPLAN Electric P8 is the better approach.

Question: How can I export specific device information to Excel?

Answer: To easily export device information (properties) to Excel, you should use the **Labeling** module. The **Labeling** module is located in the **UTILITIES/REPORTS** menu. Generally, there are several file formats to choose from: Excel format **.xls*, text file format **.txt* and XML file format **.xml.*

EPLAN provides a series of complete *schemes* that can be used to export specific properties of certain devices, such as cables, terminals or general devices, to Excel. These schemes can be copied. The properties to be exported can then be modified on the *Label* tab. But it is also possible to create your own, new schemes on the basis of the various *report types* like terminal diagram, device tag list, etc.

The scheme (for exporting to Excel) contains many format output options on the various tabs (*Header*, *Footer* and *Label*). You can also set up filter and sorting options in the schemes (**Settings** tab).

Question: Can I use programs other than Microsoft Excel to export and import EPLAN Electric P8 data?

Answer: No, this is not possible. Other program packages, such as OpenOffice, etc., cannot be used for functions such as the export and import of labeling or for external edit functions. This data exchange generally requires Microsoft Excel.

13.8 Reports

Question: Is it possible to generate a report for a device tag list with the graphical overview of complex devices, for example, as demanded by energy supply companies?

Answer: Yes, EPLAN can do that. One possibility is to use conditional forms.

You must do the following in EPLAN:

1. Generate a graphical overview of the device. Ideally, you should create a separate symbol library where such overviews can be created and saved.

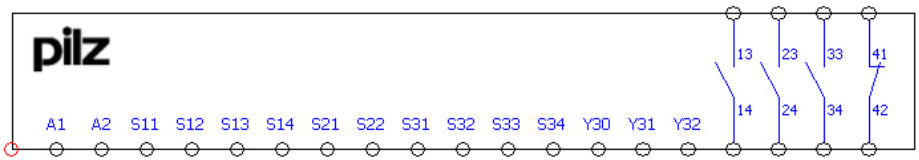


Fig. 13.86 Graphical overview

2. On the *Data for reports* tab in parts management, assign this symbol to the part for which a report with this graphical overview is to be generated in the device tag list.

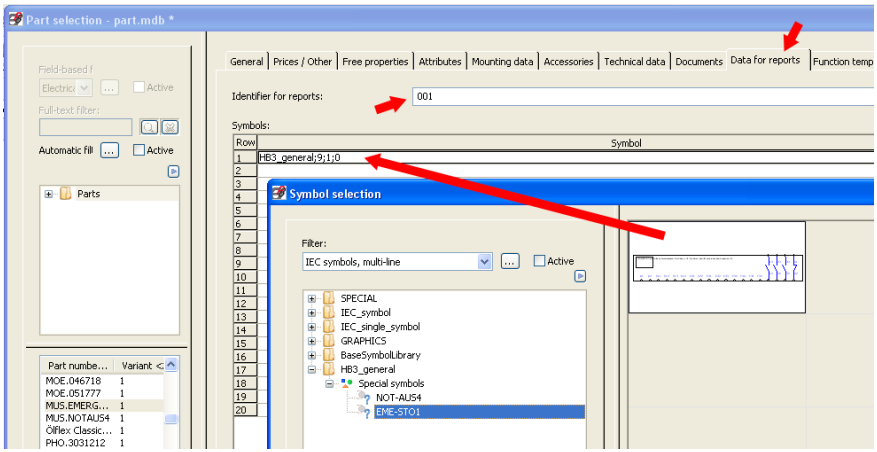


Fig. 13.87
Assign symbols to part

3. Then, you need to create corresponding device tag list forms, so that the representation can later be displayed on them as well. You can start with a basic form that displays all other devices without detailed representation as before.

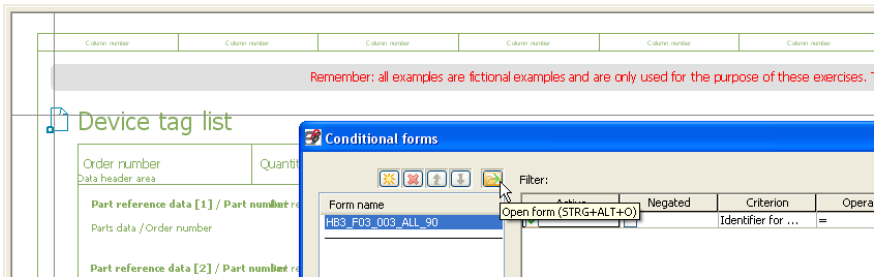


Fig. 13.88
Basic form editing

4. Into this standard form you place a conditional form (via the **INSERT/CONDITIONAL FORMS** menu) and use the button in the **Conditional forms** dialog to select the subform for the graphical detailed representation.

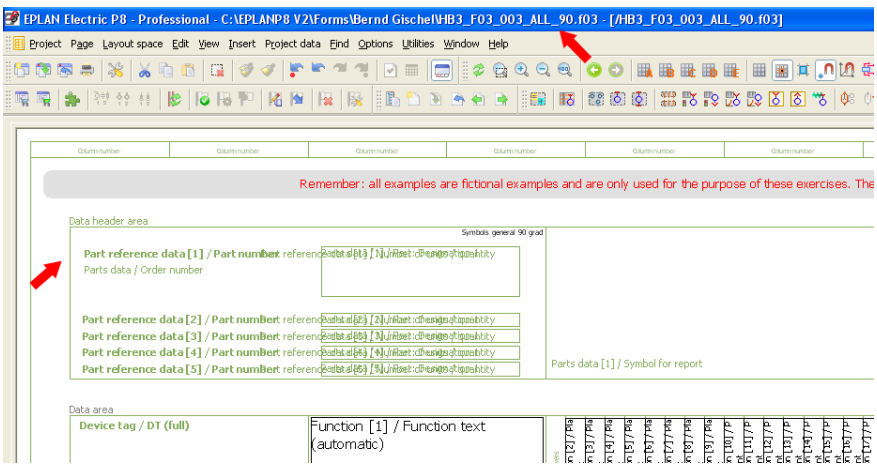


Fig. 13.89
Subform

5. The subform can be called conveniently and directly from the basic form for editing. Once the subform is closed again, EPLAN returns to the **Conditional forms** dialog. In this dialog, it is important to note that a *code* is now defined for the report, so that EPLAN, when generating a report, can correctly assign the part (with its code on the *Data for reports* tab) and the *code* of the conditional form (subform).
6. If all these preconditions have been met, you can generate a report of the device tag list. EPLAN generates the report and displays the devices accordingly.

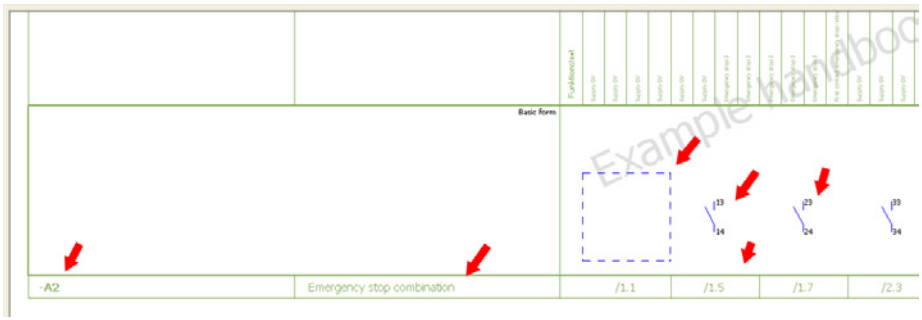


Fig. 13.90 Standard report without detailed representation

Fig. 13.91 Detailed report with symbol and special report

This “emergency stop combination” is, of course, a relatively simple example. It is also possible and conceivable to think of examples of complex devices with internal schematics or plug-in design, etc.

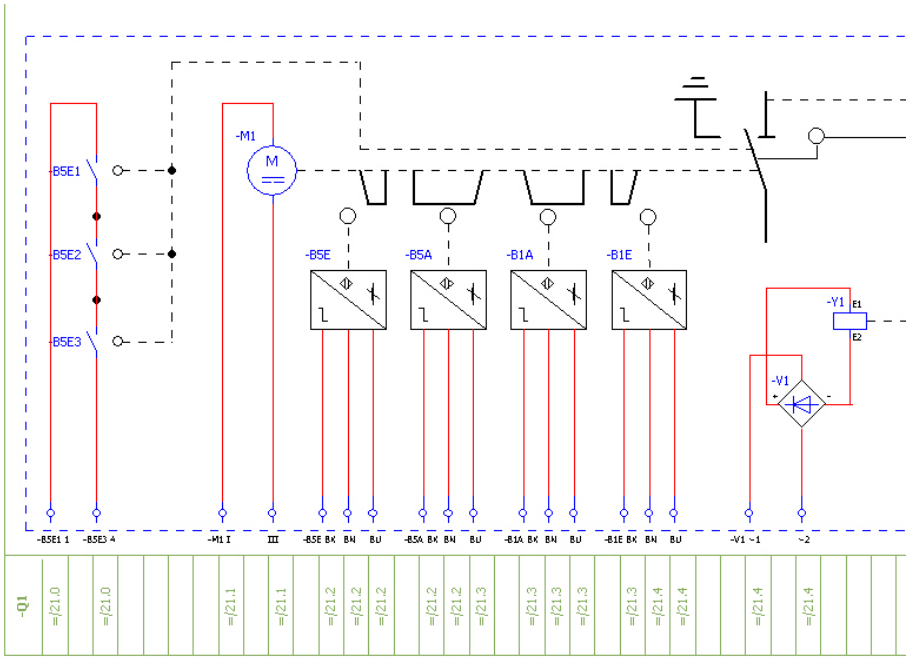


Fig. 13.92 Sample device in the device tag list

14 Creating a schematic project – step by step

Experience has shown that difficulties can arise at certain “editing points” in EPLAN Electric P8 even though the solution is fairly apparent. This chapter will take a step by step look at a few practical examples to help eliminate such stumbling blocks.

NOTE: In EPLAN Electric P8, there is not “one right” way to do something. When you go through the examples in this chapter, always remember the same result could be achieved in another way.

To use EPLAN, you first need to use an existing project template to create a schematic project that has a specific structure. Project planning in EPLAN roughly consists of the following steps:

1. Create a project
2. Generate pages
3. Create a schematic
4. Generate reports

14.1 Create a project

What is a project? In EPLAN, schematics, terminal diagrams, etc. are created and generated as pages in a project. Simply put, a project is a large database where all the parts of a project are stored.

What is a project structure? In EPLAN, a project structure is the structure that should be used for the project. Structures are identifiers, such as a higher-level function (=) or the installation site (++). All objects in the project are stored in such a project structure and sorted within this structure.

What is a project template? EPLAN needs a project template in order to create a new project. A project template defines certain settings, though they usually can be changed later.

EPLAN offers many ways to create a new project. In this example, we call up the **PROJECT/NEW** menu.

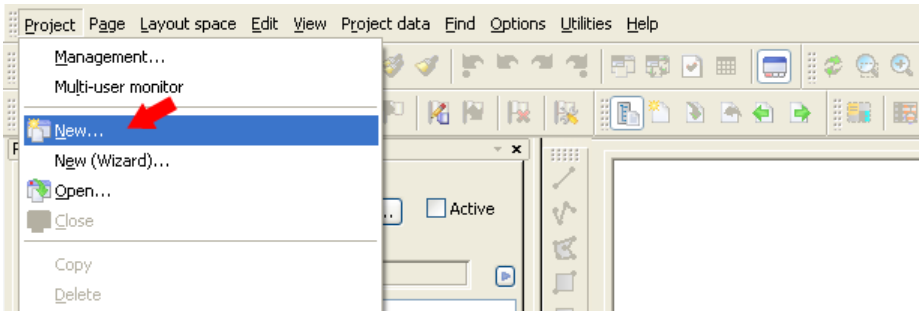


Fig. 14.1
Project/New menu

EPLAN opens the **CREATE PROJECT** dialog shown in Fig. 14.2 where settings need to be made.

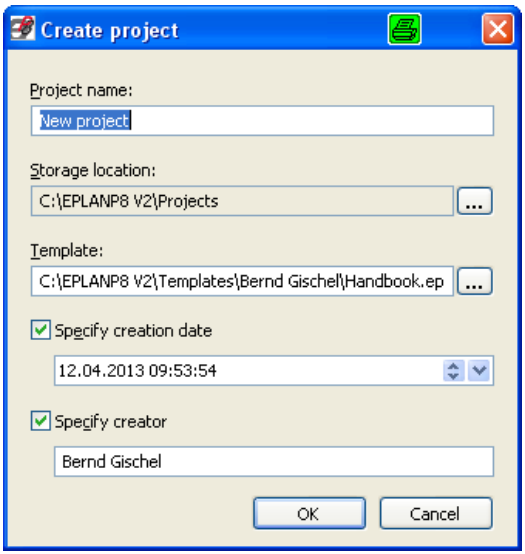
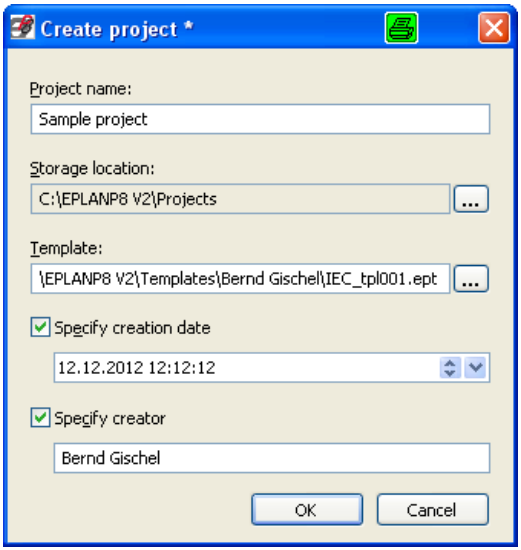


Fig. 14.2
Create project dialog

Table 14.1 The most important inputs for a new project

Setting	Comment	Example
Project name	Enter the name of the project here.	Sample project
Storage location	Use the **[...]** button to select the location where the project should be stored.	E:\EPLANP8\Projects
Template	Use the **[...]** button to select the template that the project should be based on.	IEC_tpl001.ept
Creation date	Enter the desired date and time.	12.12.2012 12:12:12
Creator	Enter the name of the person who is to be listed as the creator of the project.	Bernd Gischel



When you have entered all the information, you can close the dialog by clicking **OK**. EPLAN using the inputs to create a new project.

Fig. 14.3
Changed inputs in the Create project dialog

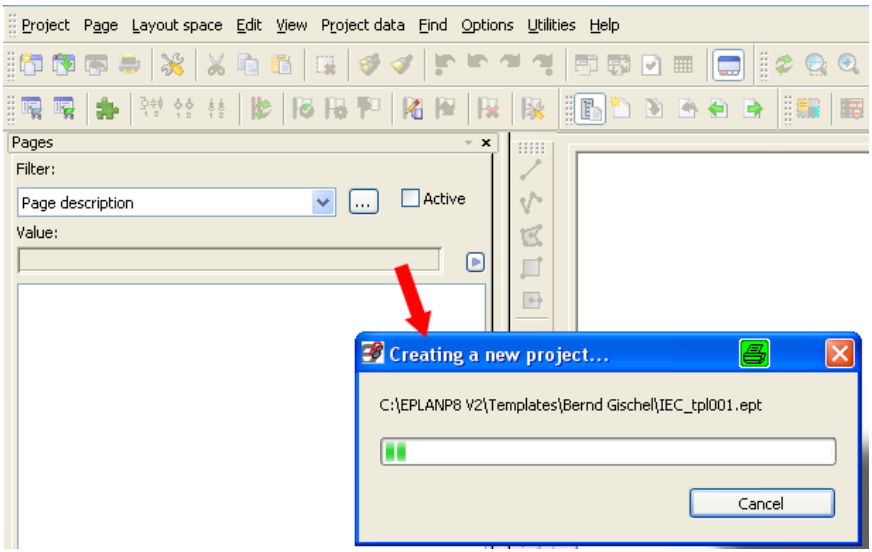


Fig. 14.4 A new project is being created

After creating the project, EPLAN automatically opens the **PROJECT PROPERTIES** dialog where you can enter other information for the project, such as a project description or a commissioning number.

NOTE: It is usually possible to change project properties later on. This means that these project properties – more or less descriptive information – are not initially required for a project.

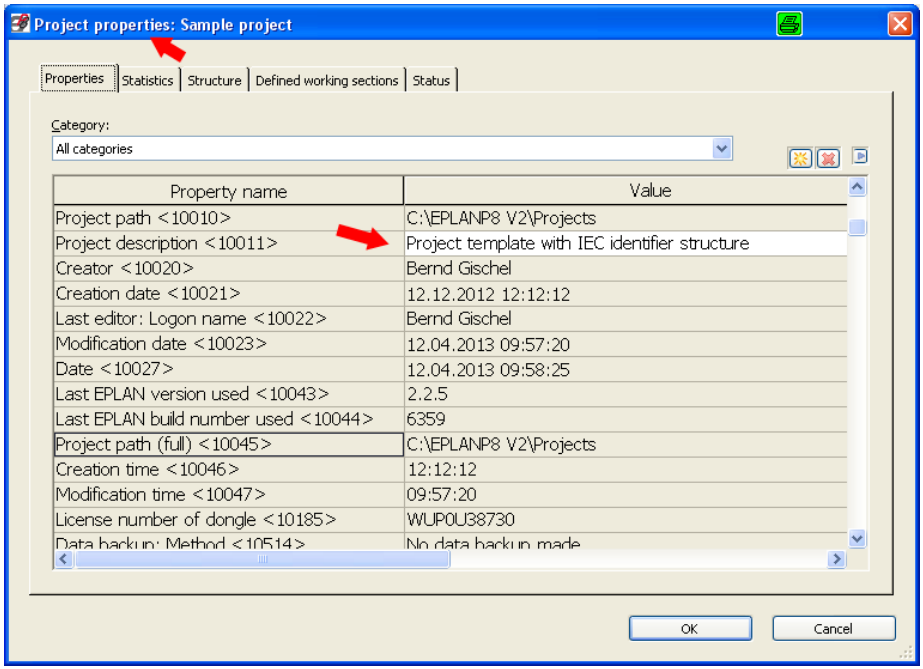


Fig. 14.5 Dialog for changing project properties

Now you can close the **PROJECT PROPERTIES** dialog. EPLAN then opens the new project in the **Page navigator**.

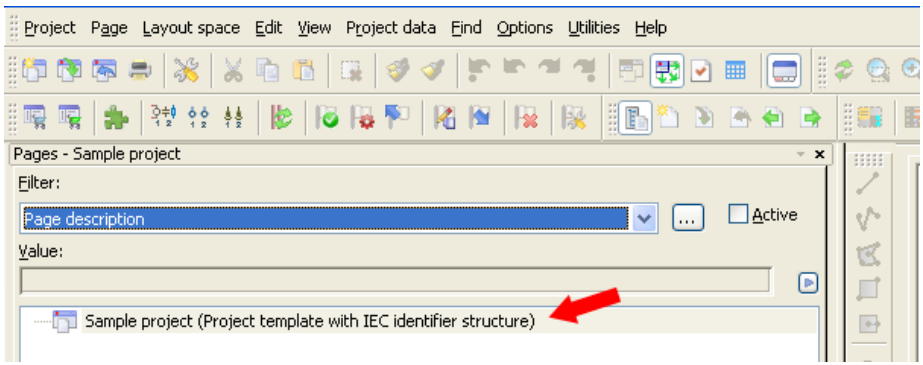


Fig. 14.6 Open project in the page navigator

14.2 Generate pages

To proceed further with the project, for example to edit it in the graphical editor, EPLAN needs pages (maybe even different types of pages) that can be called up later in the **Page navigator**.

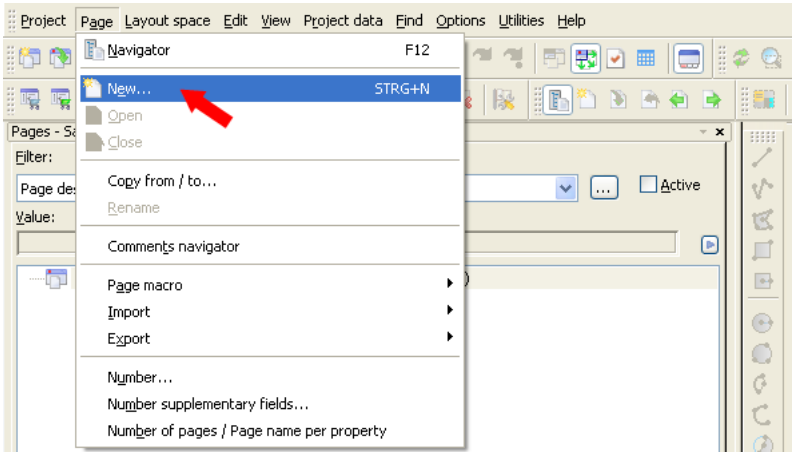


Fig. 14.7 Page menu

What are pages? Pages are needed in EPLAN, for instance, to create a schematic. You can use the **PAGES/NEW** menu to create different types of pages.

What are page types? EPLAN uses different types of pages. A schematic would usually use pages that have the page type *Schematic multi-line*. A cover sheet would have the page type *Title page/cover sheet* and a report, e. g. a terminal diagram, would have the page type *Terminal diagram*. EPLAN makes a distinction between automatic pages that EPLAN automatically generates (such as a *terminal diagram*) and interactive pages that can be manually created and edited (such as a *multi-line schematic*).

What is a graphical editor? The graphical editor is the part of the EPLAN program that is used to create and change a schematic or overviews. The graphical editor contains other functions that can be accessed using various menus such as **INSERT** and **EDIT**.

What is a page navigator? The page navigator is the center of action for page editing. The page navigator can be used to create and rename pages, or to change and even delete their properties.

Pages are created using the **PAGE/NEW** menu. Then you define the page properties (see Fig. 14.8 and Table 14.2).

NOTE: For a schematic project, you should always start with a cover sheet (page type *Title page/cover sheet*). The page type can be automatically generated, or (as an exception to the rule) manually created.

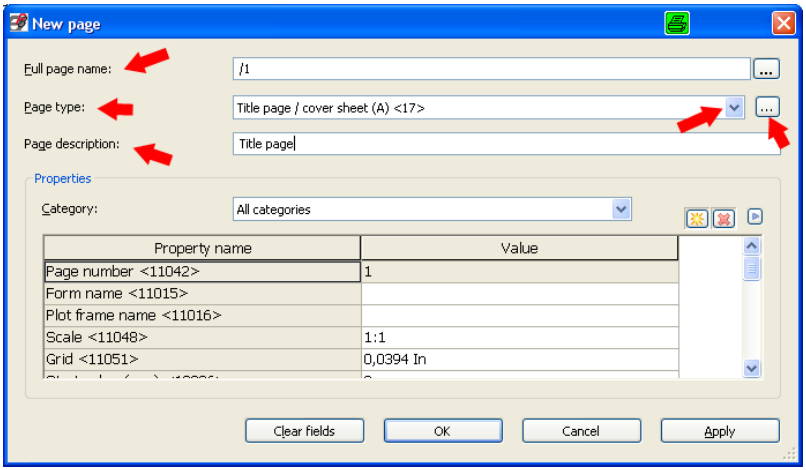


Fig. 14.8 Page properties dialog

Table 14.2 The most important inputs for a page

Setting	Comment	Example
Full page name	Specify the structure of the page here. You can also use the **[...]** button to select an entry.	"1" or "10"
Page type	Use the enter **[...]** button to select the page type (*Schematic multi-line, Title page/cover sheet,* etc.).	Title page/cover sheet" or "Schematic multi-line"
Page description	Enter a description of the page here.	Cover sheet for schematic documentation" or "Power supply"
Optional; Properties	*Optional;* form name (Note: is usually automatically filled in by EPLAN)	*Optional;* F26_001

When you have entered all the information, you can close the dialog by clicking **OK**. EPLAN using the inputs to generate a new page and displays it in the page navigator.

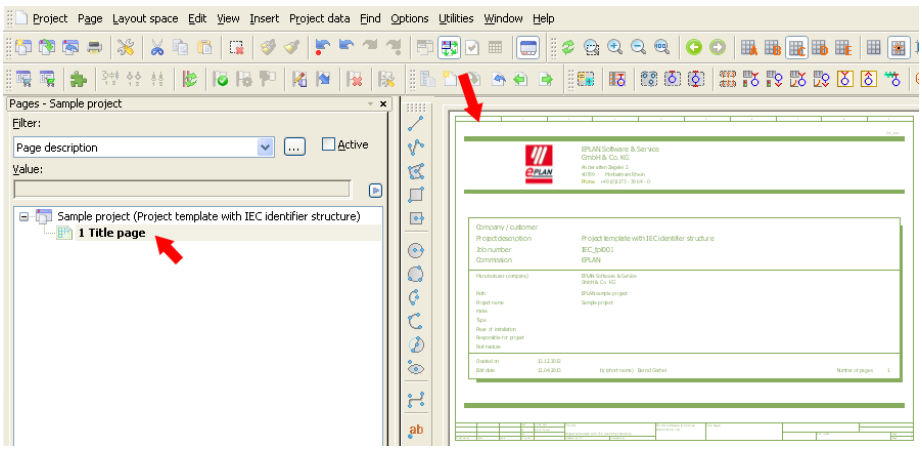


Fig. 14.9 First page – cover sheet

You use the same steps to create the next page.

- Call up the **PAGE/NEW** menu.
- Enter the **Full page name** in the **NEW PAGE** dialog.
- Select the **Page type** in the **NEW PAGE** dialog (Note: Many page properties, including the **Page type**, are taken from the currently selected page so you might have to use the selection list to change the page type.)
- Enter the **Page description** in the **NEW PAGE** dialog.
- Click **OK** to save and close the **NEW PAGE** dialog.

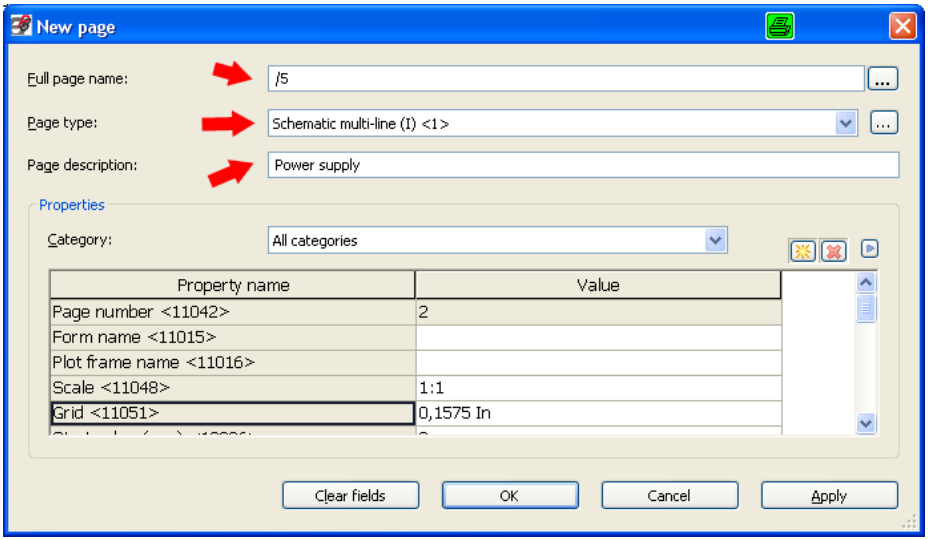


Fig. 14.10 New page with page type Schematic multi-line

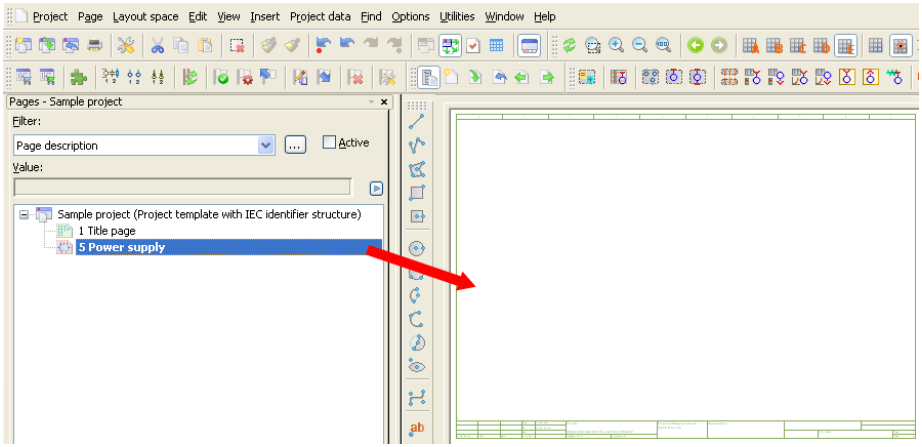


Fig. 14.11 Finished page in the page navigator

TIP: To edit the page properties, select the page in the open **Page navigator** and right click to open the **PROPERTIES**.

When you have finished creating all the pages you need, you can edit them. In the **page navigator**, double click the page you want to edit. EPLAN opens the page in the graphical editor.

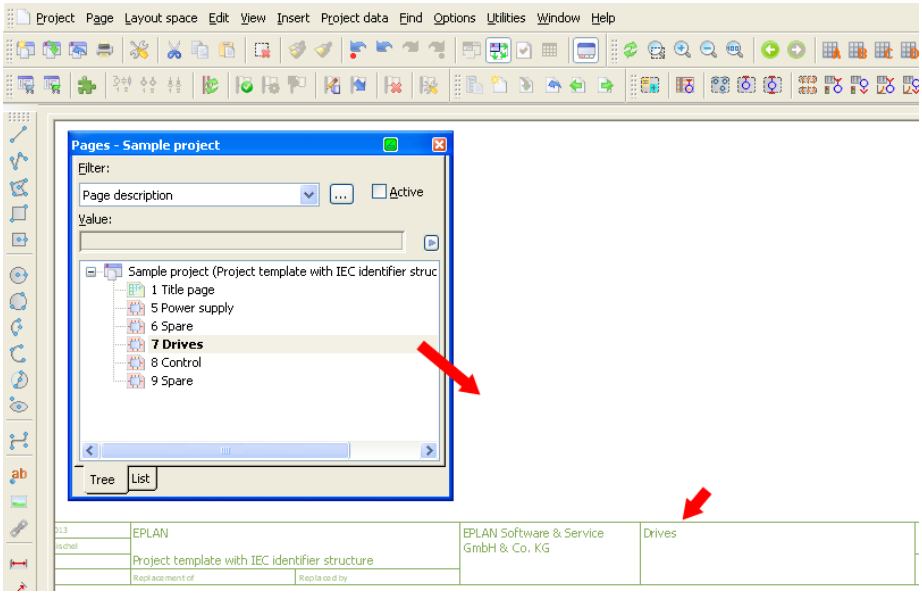


Fig. 14.12 Open page in the graphical editor

14.3 Create a schematic

A schematic mainly consists of various symbols, components, graphical elements and usually so-called connectors that are used, for example, to automatically implement auto-connecting connections.

What is a symbol? A symbol is a graphical representation that EPLAN uses to represent functions. A function can be a motor, a coil, a signal lamp, etc. A symbol is initially only a graphic or a graphical representation and does not possess any logical data.

What is a component? A component is a symbol that has an associated logical function. The function itself contains logical data, such as other function definitions. A coil for example, contains functions other than just the coil function, e.g. two NO contacts and

two NC contacts. A component is the combination of a graphical symbol (the coil) and its functions.

What are graphical elements? Graphical elements can be lines, circles and even rectangles. Graphical elements have no logical functions and are simply drawing elements that have no impact on the logical content of a schematic.

What are connectors? Connectors are special symbols that allow you to use existing automatic connects to "forward" logical data or to simply create a logical connection in the first place. Connectors include angles, T-nodes and interruption points.

What are autoconnection connections? One advantage of EPLAN is the so-called autoconnection connection. As soon as two connection points for a symbol or connector are precisely opposite one another, they are automatically (= autoconnecting) connected by a type of "line". EPLAN recognizes and outputs only these autoconnecting lines as logical connections. Connections that are simply a graphical line drawn between two connection points of a symbol are not logical connections.

14.3.1 Insert symbols

To insert a symbol, you press the **CTRL** key. EPLAN opens the **SYMBOL SELECTION** dialog.

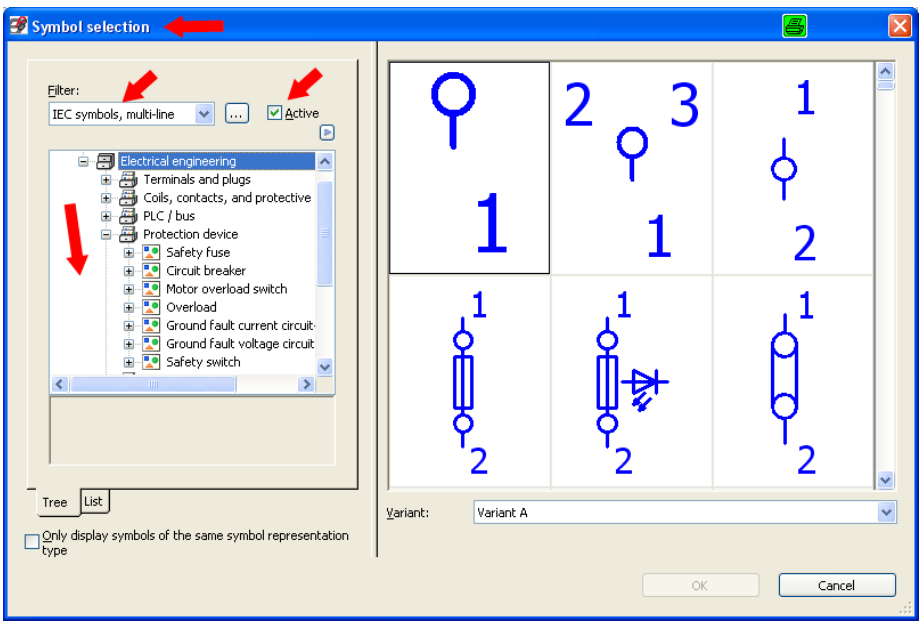


Fig. 14.13 Symbol selection dialog

In this dialog, you select the appropriate symbol and click **OK** to apply it. EPLAN "hangs" the symbol on the cursor, and it can then be placed anywhere on the page.

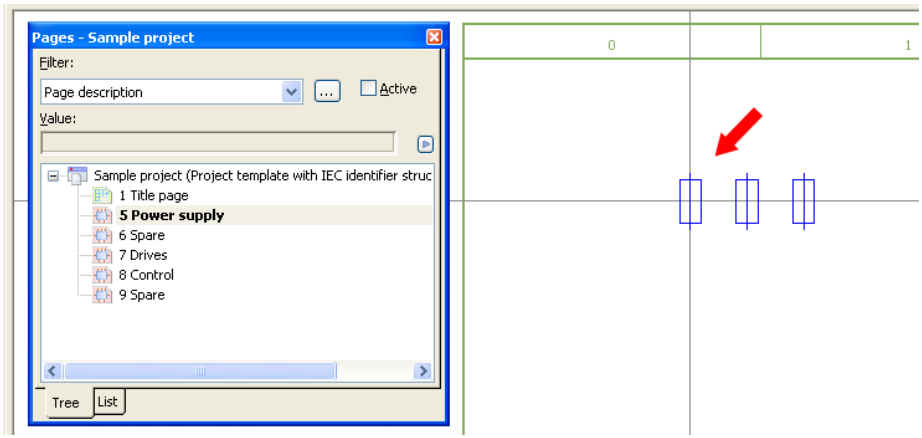


Fig. 14.14 Placing a symbol

After placement, EPLAN opens the **PROPERTIES (COMPONENTS) GENERAL DEVICE** dialog. Some of the properties may have default values. For example, the device tag (DT) if online numbering was activated in the project settings.

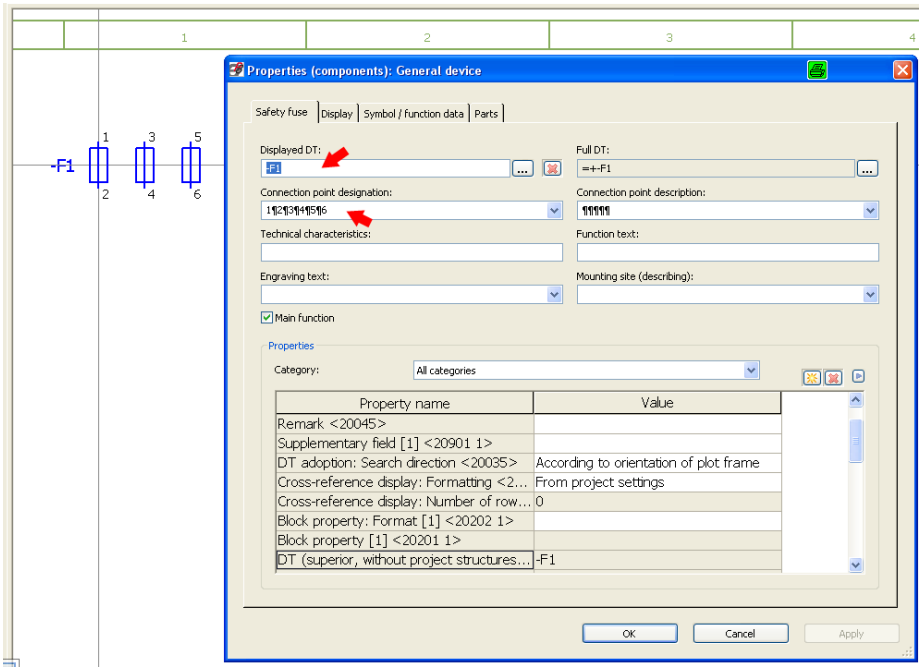


Fig. 14.15 Properties (components): General devices dialog

At a minimum, the following entries must be entered for the symbol in the dialog:

Setting	Comment	Example
Displayed DT	The actual device tag	–F1
Connection point designations	Designations for the connection points; for a coil this could be f A1 | A2	1 | 2 | 3 | 4 | 5 | 6 Note: You can enter the "|" separator by pressing CTRL + ENTER.
Optional; Technical characteristic	*Optional;* Technical characteristic of the device	*Optional;* 32A
Main function	Each DT can only have one main function. At least one must have a main function.	Click the box next to Main function to activate it

NOTE: EPLAN automatically creates the full device tag (DT) from the page identifier, including the displayed DT. You usually do not have to enter this manually.

When you have entered all the information, you can save and close the dialog by clicking **OK**.

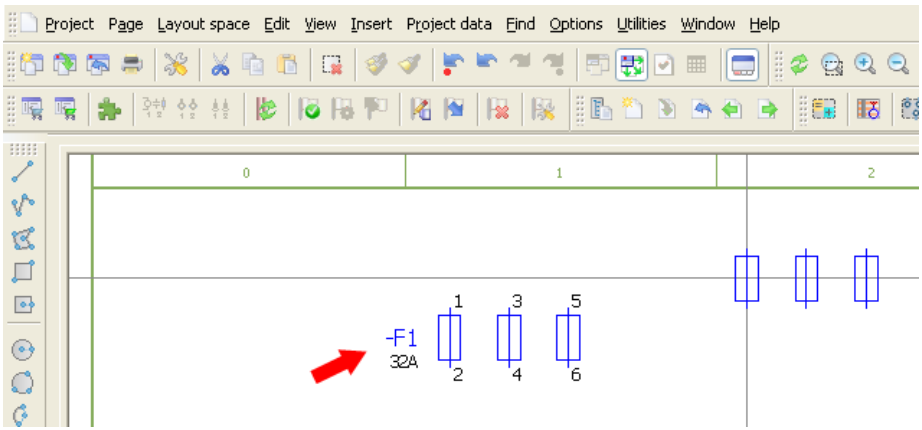


Fig. 14.16 Finished symbol with entered information

Now you can insert and place other symbols. When you are done, you can user connectors to connect them. For example, you could use the **INSERT/CONNECTOR** to select and place an angle.

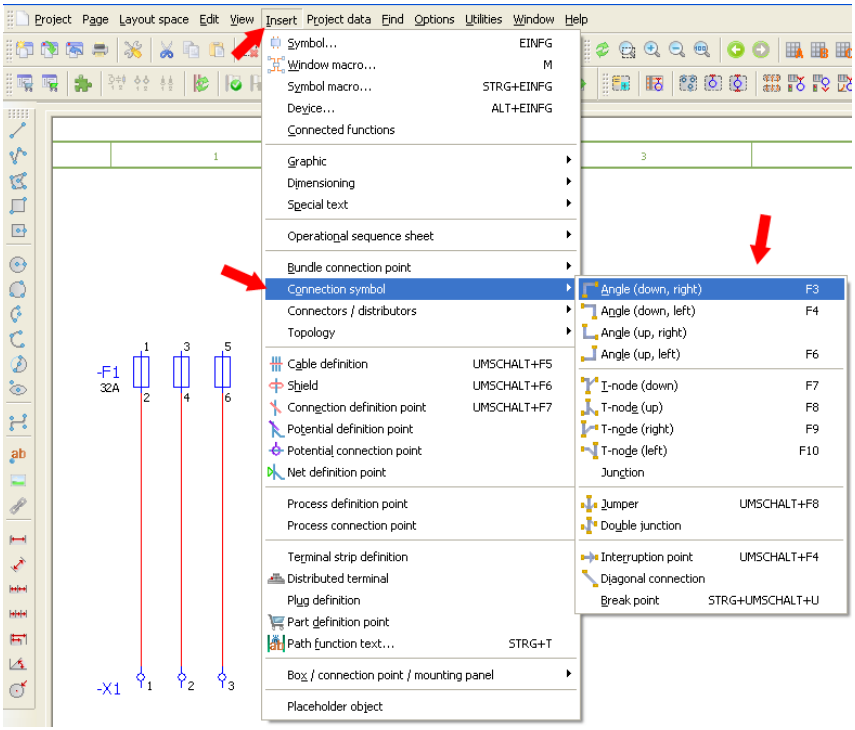


Fig. 14.17
Placing a connector

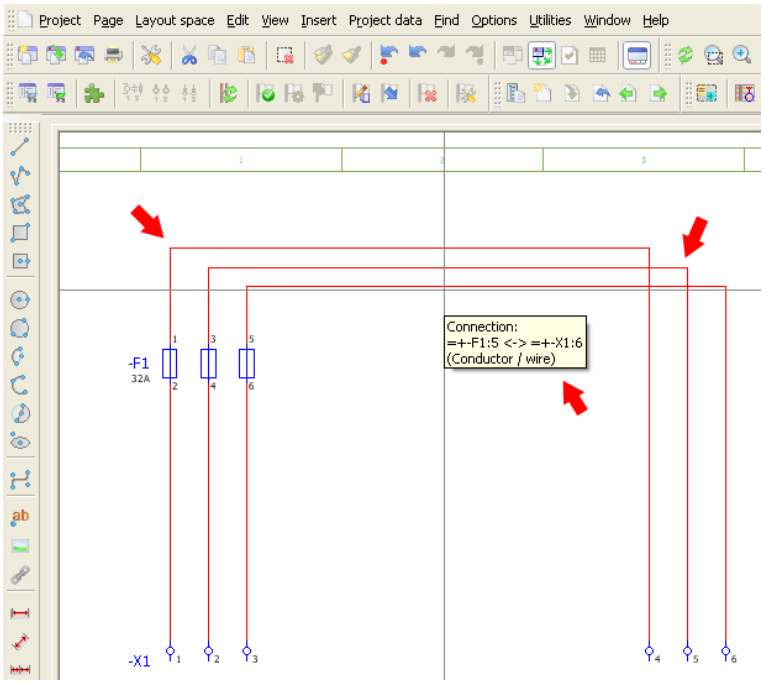


Fig. 14.18
Schematic with connectors

The terminals in the figures are inserting like normal devices using the **INSERT/SYMBOL SELECTION** menu or the **CTRL** key.

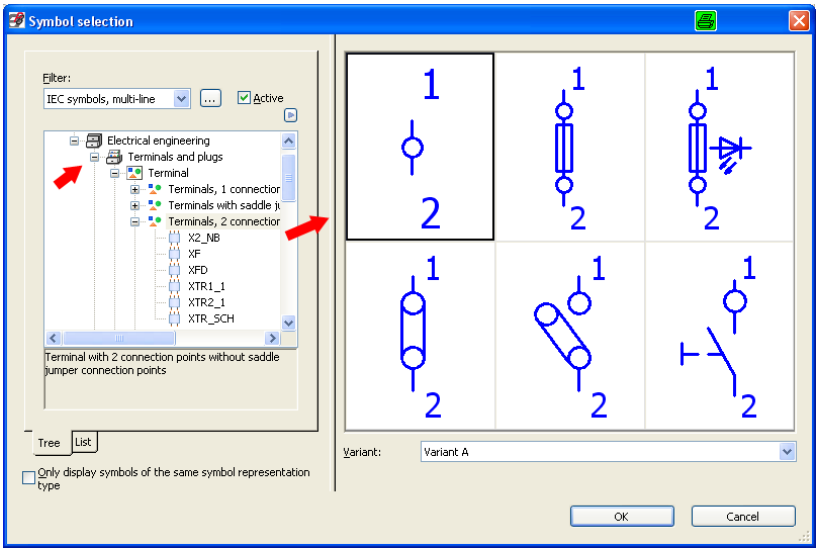


Fig. 14.19
Inserting terminals

After placement, EPLAN opens the **PROPERTIES (COMPONENTS): TERMINAL** dialog. In this dialog, you enter the **Displayed DT** and the **Connection point designation** (terminal or plug designation).

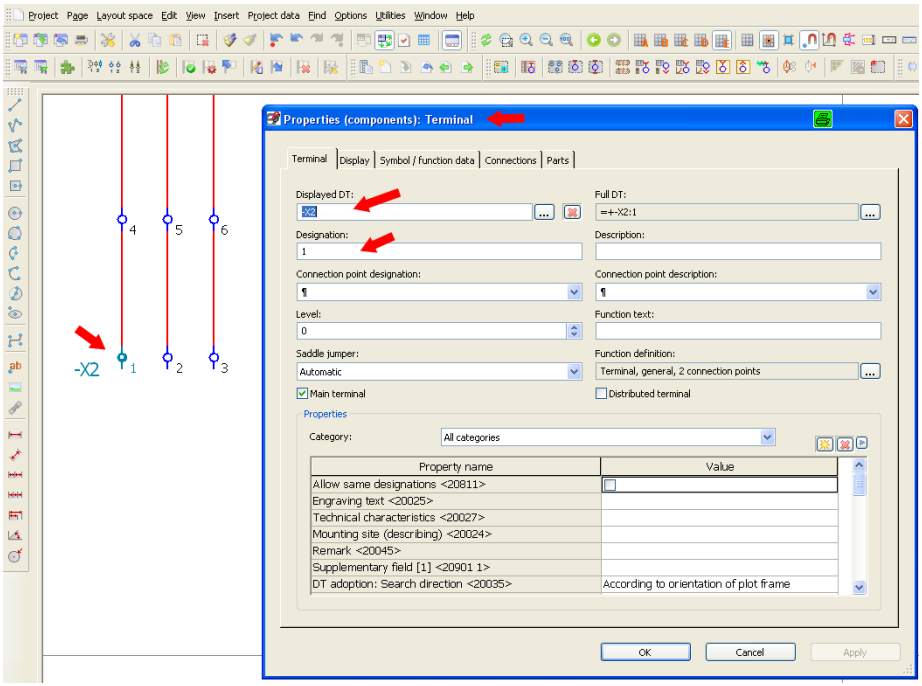


Fig. 14.20
Properties (components): Terminal dialog

14.3.2 Insert cables

In addition to devices like coils, terminals, etc., you can also insert cables. You place cables using **SHIFT + F5** or the **INSERT/CABLE DEFINITION** menu.

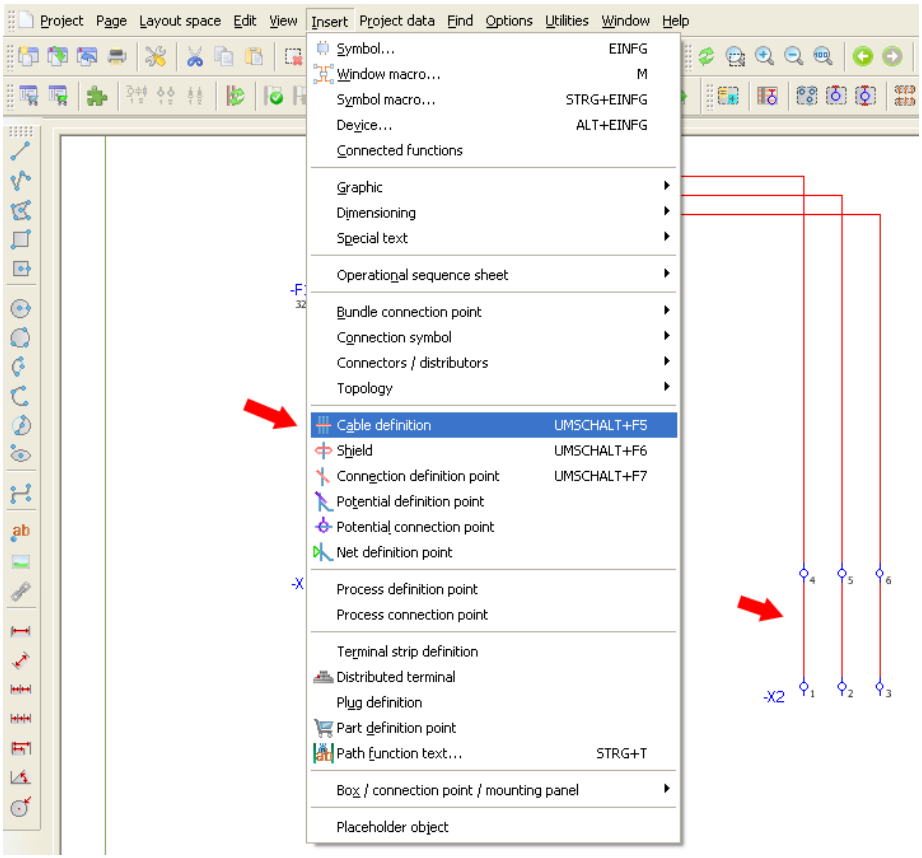


Fig. 14.21 Placing cables

When a cable is selected, the cable definition "hangs" on the cursor and you can now insert the cable between devices. You left click to select the starting point (usually on the left).

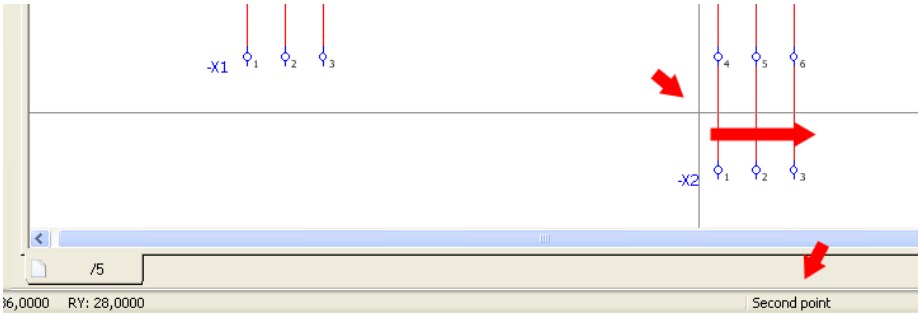


Fig. 14.22 Starting point for cable definition

Now you drag the cable via the connection and left click to set the second point. EPLAN then opens the **PROPERTIES (COMPONENTS)/CABLE** dialog where you enter data such as the **Displayed DT**.

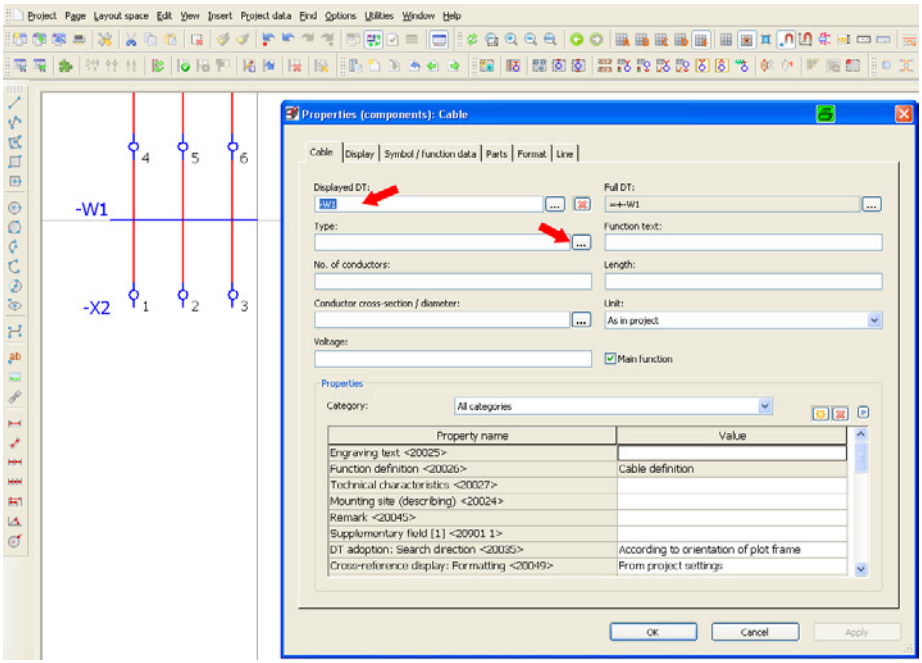


Fig. 14.23 Properties (components): Cable dialog

You can now add other cable data from **parts management**, such as the **Cable type**. The best way is to click the **[...]** button and select and apply the cable from parts management.

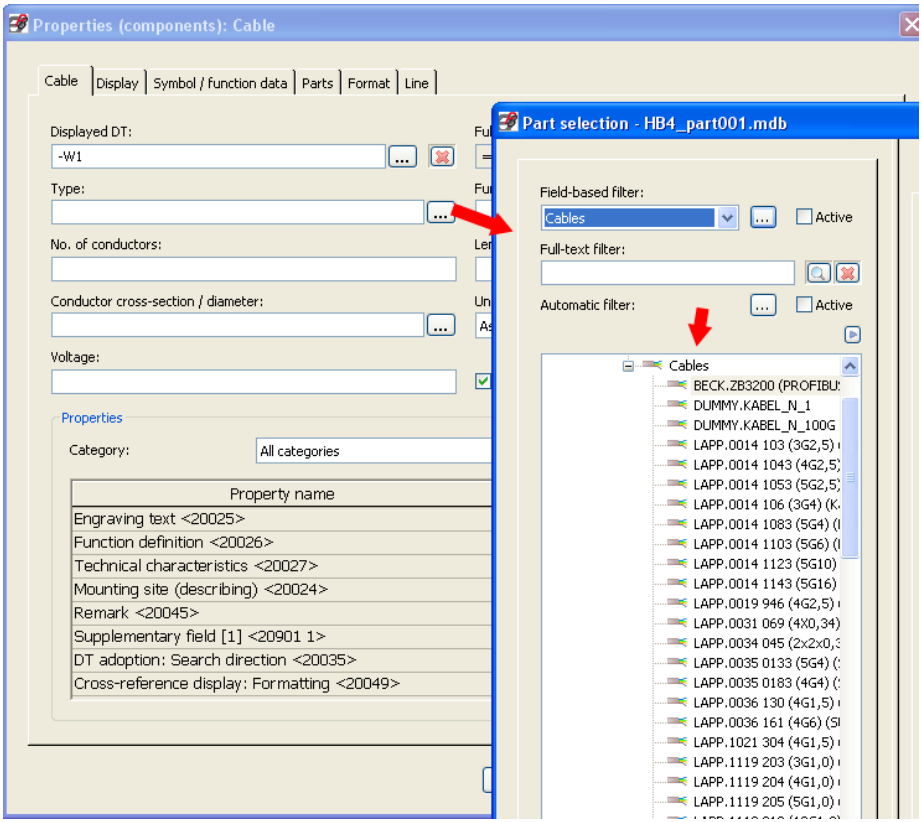


Fig. 14.24 Selecting a cable from parts management

When you click **OK**, parts management closes and EPLAN automatically adds the entered cable data such as cable type, number of conductors, etc.

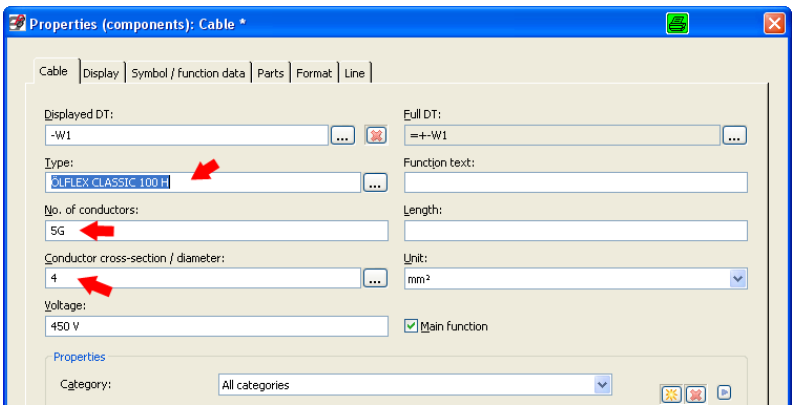


Fig. 14.25 Added cable data

When you click **OK** in the **PROPERTIES (COMPONENTS): CABLE** dialog, EPLAN closes the dialog and displays the cable and all its data in the schematic.

TIP: You can also manually enter additional cable data, such as cable type, number of conductors and conductor color.

To label the conductors, select the cable and then the **PROJECT DATA/CABLE/ASSIGN CONDUCTORS/REASSIGN ALL** menu item. EPLAN automatically assigns the conductors to the cable and labels them.

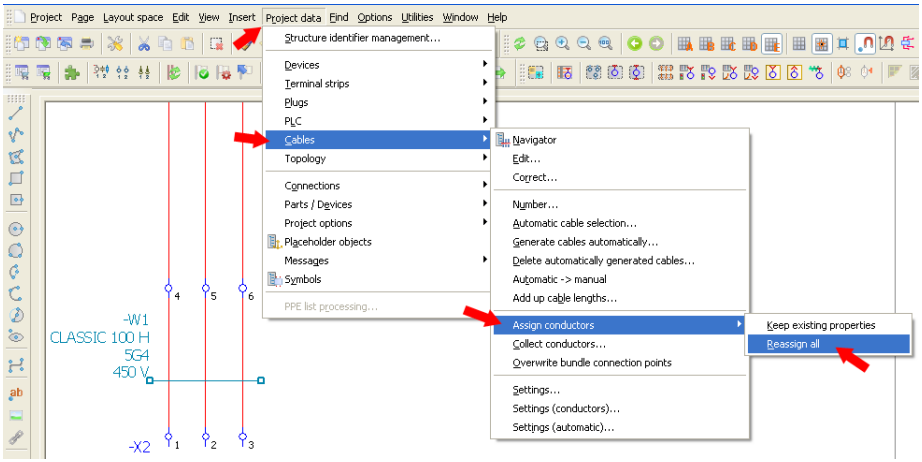


Fig. 14.26 Assigning conductors to the cable

Fig. 14.27 Assigned conductors

14.3.2.1 Add texts

In addition to symbols and components, a schematic also contains texts that describe or explain functions. There are two types of text in EPLAN: free text and path function text.

Free text is not output in reports such as a terminal diagram. EPLAN essentially ignores this type of text. It is only exists “visually” on the page.

Path function text can be output by EPLAN, for example in a terminal diagram. This is why this type of text is usually preferable to free text.

Free texts are inserted via the **INSERT/GRAPHIC/TEXT** menu.

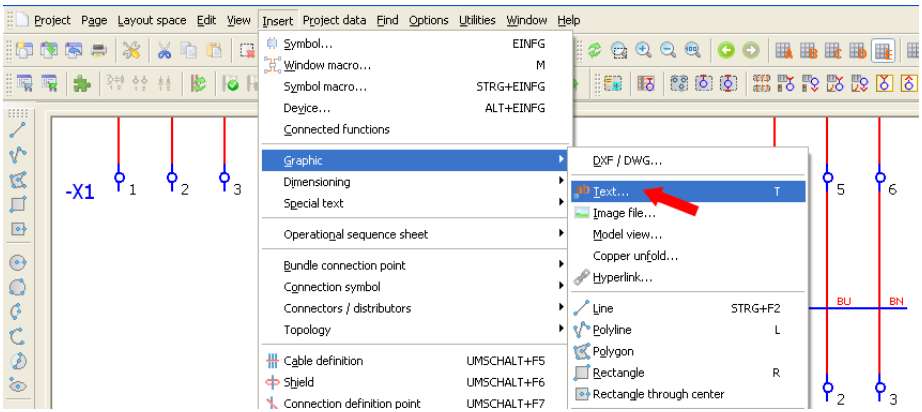


Fig. 14.28
Inserting free text

Path function text is inserted via the **INSERT/PATH FUNCTION TEXT** menu.

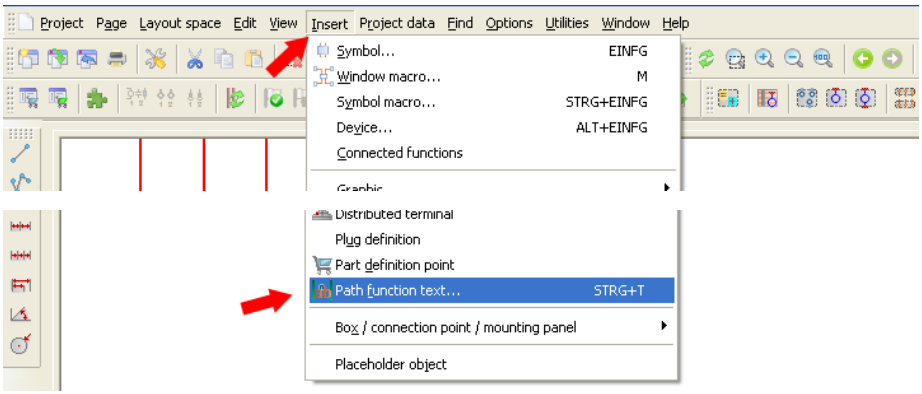


Fig. 14.29
Inserting path function text

After you have selected the type of text, EPLAN opens the **PROPERTIES/TEXT** dialog where you can enter the text and then place it.

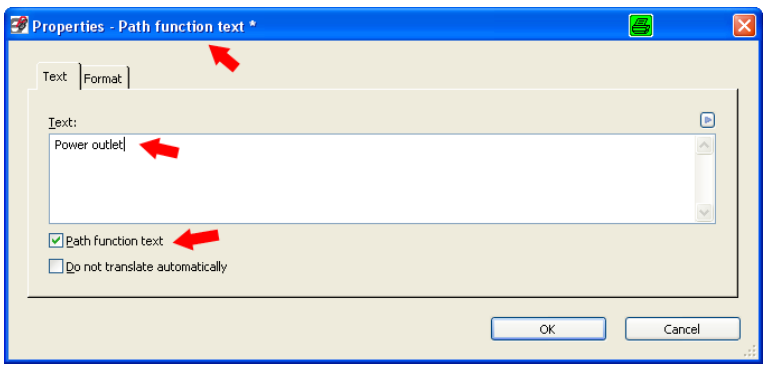


Fig. 14.30 Adding text

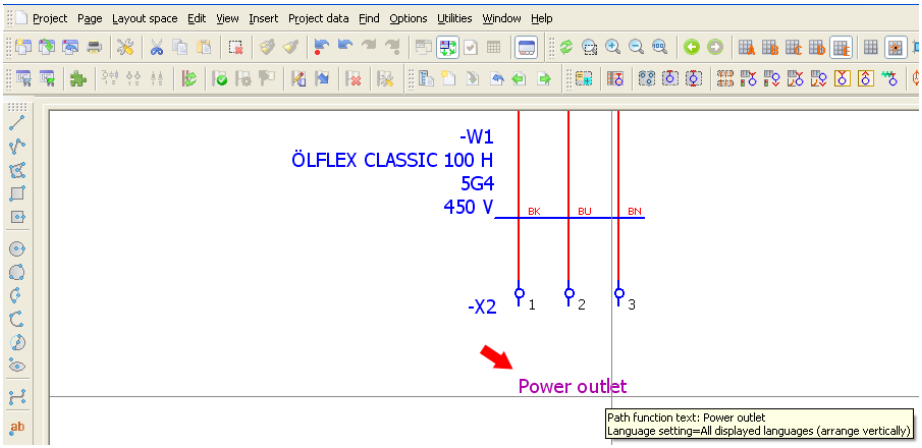


Fig. 14.31 Placed path function text

You have now finished creating a simple schematic.

14.4 Generate reports

In addition to schematics, projects can include reports, such as terminal diagrams, cable diagrams or even a table of contents.

What are reports? Reports are graphical lists that EPLAN fills with data from the project or schematic. Examples are terminal information for a terminal diagram or page information for a table of contents.

In EPLAN, reports can be created with just a few clicks. Select the **UTILITIES/REPORTS/CREATE** menu item.

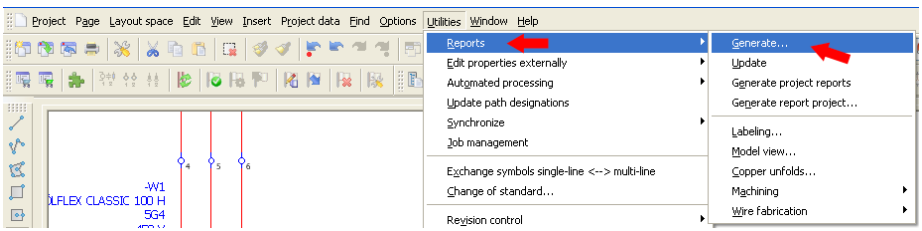


Fig. 14.32 Generate reports menu

EPLAN opens the **REPORTS** dialog. Click the *Reports* tab.

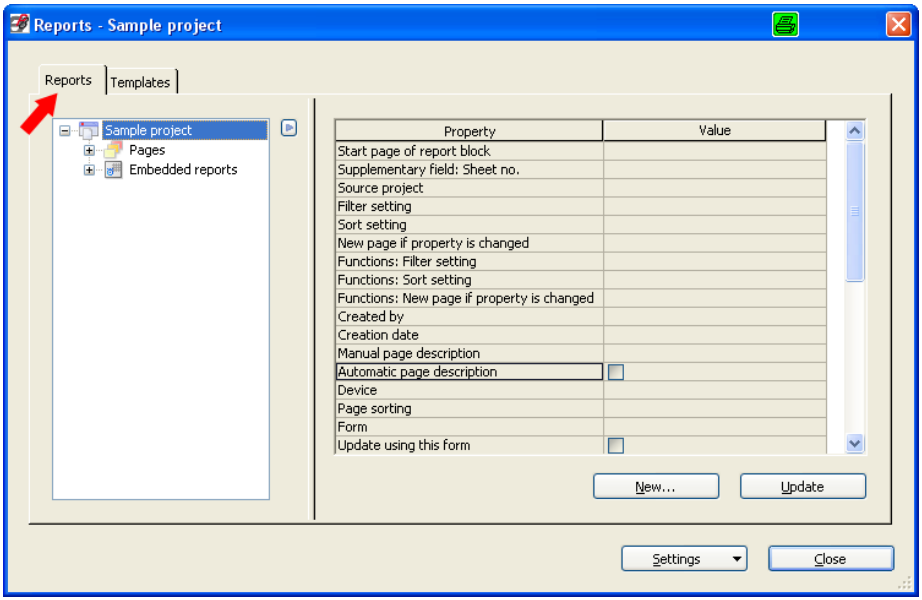


Fig. 14.33 Reports dialog

Now click the **NEW...** button. EPLAN opens the **SELECT REPORT** dialog. Here you select the desired **Report type** and **Output format**.

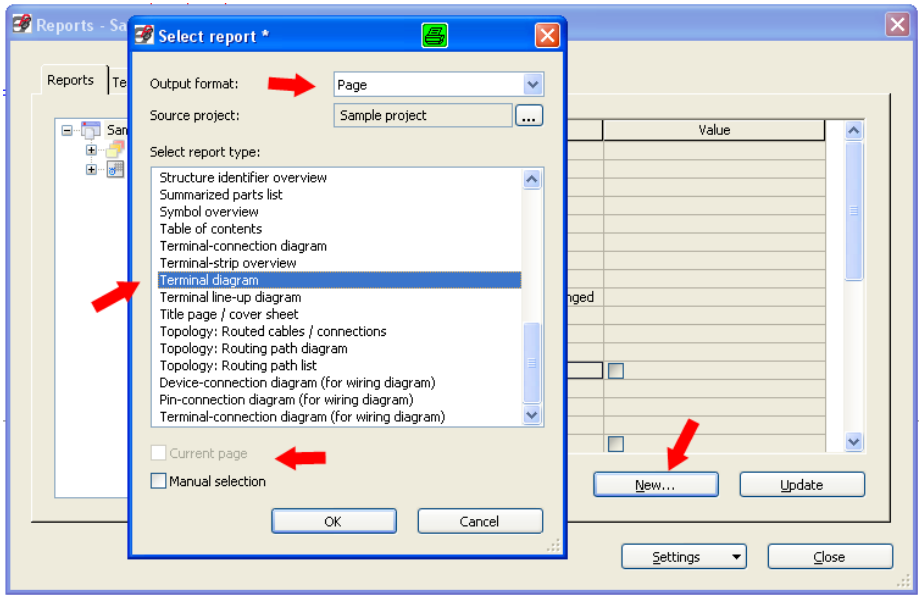


Fig. 14.34 Selecting a report

When you are done, click **OK** to exit the **SELECT REPORT** dialog. EPLAN opens the **FILTER/SORTING – TERMINAL DIAGRAM** dialog. Simply click **OK** to skip this dialog at the moment.

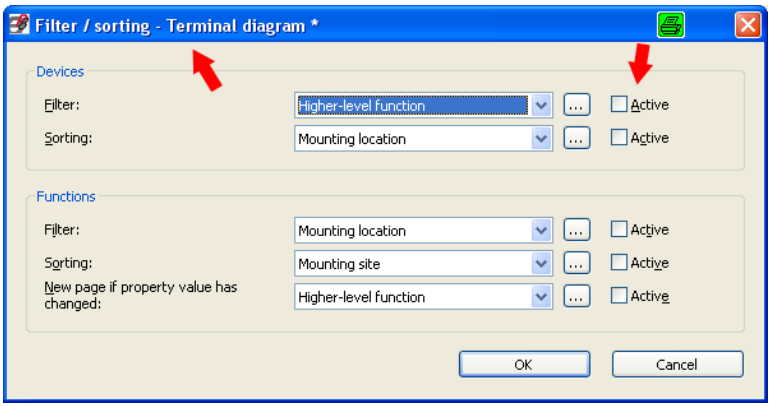


Fig. 14.35 Filtering and sorting options

EPLAN now opens the dialog where you can enter the initial data for the terminal diagram: the page (number) where the terminal diagram should be created and the page description.

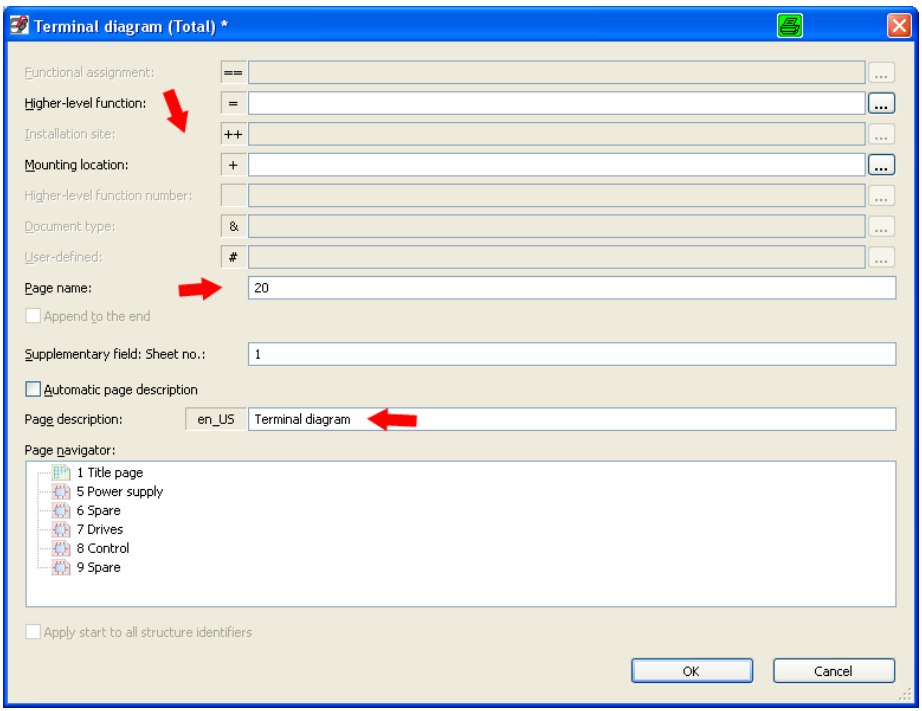


Fig. 14.36 Data for the terminal diagram page

After entering this data, click **OK** and EPLAN generates the diagram and automatically adds it as a new page or pages in the project.

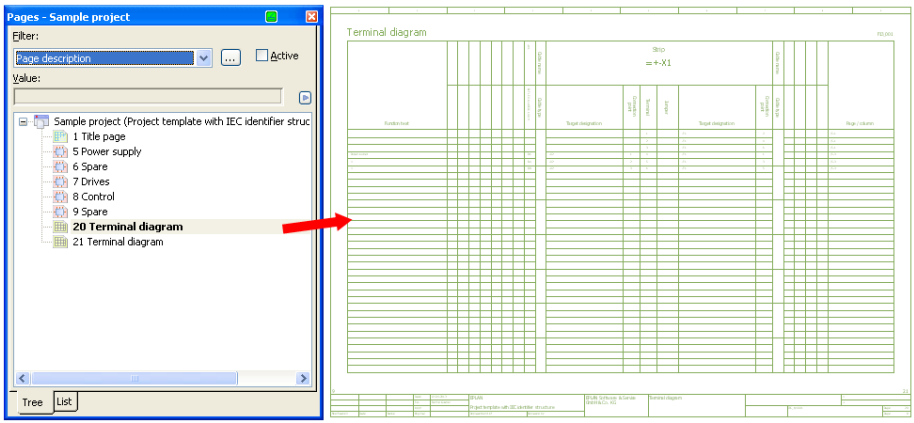


Fig. 14.37 Final report

14.4.1 Generating other report pages

You can of course generate more reports than just the terminal diagram. A table of contents, for example, that outputs a list of all the pages in a project. The procedure is the same as described in section 14.4. Fig. 14.38 shows a table of contents that was added to a project.

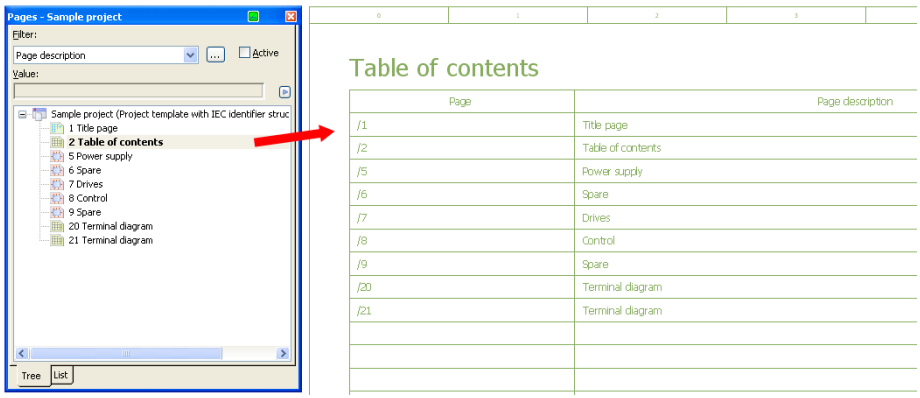


Fig. 14.38 Other reports

Index

Symbole

299
#H# *299*
#ID# *299*
#RO# *299*
#RW# *299*
<<...>> string *93*

A

Action line *531*
Actions *405*
Adapt page structure *82*
Add *128*
Address *223*
Addresses *221*
Adoption parameters *33*
Alignment *123*
alphanumeric *315*
Angle variant *131*
API *33*
API interface *527*
Apply *317*
Archive *396*
Area of application *241*
Arguments *533*
Assign conductors *581*
Assigning *166*
Assign main function *166*
Assignment *204*
Assignment lists *221*
Assign pages *364*
Asterisk *98*
AutoComplete *34*
Autoconnecting *511*
AutoCorrect *34*
Automated processing *24, 405*
Automatic cable generation *207*
Automatic cable selection *209*
Automatic generation *207*
Automatic page description *430*
Auxiliary contact *135*
Auxiliary function *166, 178*
Available *292*

B

Back *66*
Backed up *391*
Background *371*
Background color *528*
Backing up master data *409*
Back up *393*
Back up data *406*
Back up entire project *398*
Back up external documents *398*
Back up image files *398*
Backup directory *400, 402*
Backup drive *31*
Backup file name *410*
Backup function *394*
Backup scope *394*
Basic project *63, 510*
BIN *13*
Black and white *368*
black triangle *47*
Blank pages *265*
Block editing *194*
Block properties *341*
bmp *370*
Border *124, 130*
Brackets *22*
Break up assemblies *284*
Browse *262*
By column or row *21*
By position *315*

C

Cable assignment diagram *244*
Cable conductor *203*
Cable connection diagram *244*
Cable data *209*
Cable definition *213, 578*
Cable definition line *17, 207*
Cable diagram *245*
Cable editing functions *206*
Cable navigator *202*
Cable numbering *18, 205*
Cable overview *246*
Cable parts *261*
Cable properties *214, 245*
Cables *213*
Cable selection *210*
Cable type *207, 579*

Cancel *66, 206*
Category *292, 320, 323*
CD/DVD burner *396*
Center mismatch *445*
Center of rotation *509*
CFG *13*
Changed pages only *372*
Change of standard *32*
Change positions *43*
Characteristic *345, 497*
Characters *265*
Check form *429*
Check routines *320*
Check usage *316*
Circle *104*
Clipboard *29*
Clip-on height *445*
Close *91, 431, 436*
Closed *91*
Coil contact image *149*
Collapse *279*
Color codes *22*
Color depth *367*
Color designation *260*
Color information *368*
Color settings *32, 384, 521*
Column *22, 435*
Column configuration *53, 329*
Combine *263*
Combine main identifiers *260*
COM interface *33*
Comment function *530*
Company fonts *36*
Comparison criteria *345*
Comparison project *60*
Completed messages *324*
Component data *346*
Components *128, 572*
Compress *24, 404*
Compression *367, 405*
Conductor *203*
Conductor designations *212*
Configure *40, 43, 329*
Connection data *202, 210, 240, 258*
Connection definition point *17, 512*
Connection dialog *32*
Connection junction *19, 32*
Connection levels *243*
Connection list *253*
Connection point description *128*
Connection point designation *128, 218, 345*
Connection point designations *575*
Connections *22, 98, 170, 203, 214, 297*
Consecutive numbering *70*
Contact arrangement *22*
Contact image *22, 135, 138*
Contact image margin *139*
Contact image offset *517*
Contact image position *517*
Contactor coil *138*
Contacts *138*
Continued dimension *114*
Conversion *439*
Conversion to uppercase *17, 531*
Coordinate input *99, 102*
Coordinates *97, 101, 370*
Coordinate system *98, 103*
Copied *82, 423*
Copy *83, 85, 113, 314, 370*
Copy from/to *85*
Copy to project *399*
Correct *188, 198*
Correct terminal strips *188*
Create form *432*
Create plot frame *437*
Create report template *280*
Creating *57*
Creation date *566*
Creator *566*
Cross-reference *22*
Cross-reference DTs *168*
Cross-reference representation *21, 135*
Cross-references *135*
Currency *341*
Cursor position *97*
Cursor settings *32*
Customer-supplied *540*

D

Data area *430*
Data backup *391, 408*
Database *316*
Data entry fields *93*
Data portal *493*
Data records *258, 431*
Data rows *431*
Data to be compressed *405*
Data type *221*
Data volume *15, 404*
Default cable type *210*
Default directories *13*
Default settings *34, 46, 262, 402, 404*
Default values *139, 206*
Definition *132*
Delete *90, 113, 313, 314*
Deleted *90*
Delete report block *279*
Description *333, 371, 395, 410*
Description dialogs *29*
Designations *340*
Device connection diagram *243*
Device cross-references *22*
Device dialog *42*
Device groups *68*
Device navigator *164*
Devices *292*
Device selection *24, 211, 345*
Device structure *57*
Device tag *69, 164, 205*
Device tag list *242*
Device tag structure *19*
Device types *158*
Dialog elements *42*
Dialog language *30*
dictionary *34, 315*
Digital start address *224*
Dimensioning *113*
Dimension line termination *21*

Direct entry *126*
Directory *59, 299, 410*
Directory structure *13, 28*
Display *17, 19, 129*
Displayed DT *128, 201, 575*
Displayed languages *124*
Distortion *371*
Distributed view *219*
DOC *384*
Document type *311*
dongle *7*
Drawing directory *244*
DRX *103*
DRY *103*
DT numbering *71*
DWG *358*
DXF *358*
DXF/DWG dialog *358*
DXF/DWG directory *362*
DXF/DWG files *32, 359*
dynamic *251, 430*
dynamic forms *264, 430*

E

edc *296*
Edit *185, 203*
Edit externally *296, 301*
Editing in table *54*
Editor *289*
Edit properties *92*
Edit terminal strips *186*
elk *60*
ell *60*
elp *60*
elr *60*
els *60*
elt *60*
elx *60*
E-mail *391, 396, 410*
E-mail message split size *31*
Embedded reports *241*
Enclosure legend *249*
End point *516*
Engraving text *128*
Enlarge *111*
Enlarged *98*
Enlargement *360*
Entire project *210, 368, 372, 383*
epj *375*
Equals sign *259*
Error *11, 323*
Excel *239, 288, 290, 294*
Excel template *298*
Exchanging *296*
exclamation mark *513*
Expand *279*
Export *286, 357, 366, 375*
Exported *304*
Extension *124*
External application *302*
External files *288*
External format *239*
External PPC system *33*
External project *87*

F

Female pins *197, 200, 252*
Filed-off *60*
File off *396*
File type *296, 300, 360*
Filter *48, 126, 242, 326, 405*
Filter dialogs *48*
Filter setting *283*
Find *316*
Finish *66*
Font *36, 122*
Font size *122*
Footer *291*
Form *284*
Format *47, 122, 179*
Format elements *291, 293, 300*
Format of the subpage *265*
Formatting *46, 47*
Form editor *423, 426*
Form errors *429*
Form handling *430*
Form insertion point *267*
Form name *425*
Form overview project *418*
Form project *58*
Form properties *38, 241, 430, 432, 556*
Forms *415, 422*
Forms documentation *243*
Form settings *258*
Form structure *423*
Found words *315*
Frames *526*
Free properties *341*
Freeze report block *279*
Full DT *128*
Full name *340*
Full page name *78, 570*
Functional assignment *311*
Function data *55*
Function definition *24, 132, 136, 181, 193, 199, 225, 345*
Function group *343*
Functions *297*
Function template *345*
Function text *118, 128*
Function text synchronization *169*

G

General *300*
Generate connections *172*
Generated *77*
Generate file names *360*
Generate functions *193, 226*
Generate pin *201*
Generate plugs *200*
Generate project reports *287*
Generate report *269*
Global *510*
GRAFCET *17*
Graphic *77, 365*
Graphical *77*
graphical approach *12*
graphical buttons *42*
Graphical coordinate system *98*
Graphical elements *111*
Graphical functions *104, 426*

Graphical macro *445*
Graphical output *239, 262, 263, 269*
Graphical output pages *262, 264*
Graphical reporting *258*
Graphical reports *239, 268*
Graphics card *1*
Grid *92, 97, 99*
Grid sizes *31*
Group *112*
Group number *343*

H

Handle *47, 451*
Handle parts when outputting *261*
hardware *1*
hardware protection *7*
Hash *98*
Header *291*
Higher-level function *311*
Higher-level function number *311*
Highlighted *207, 209, 219, 296, 423*

I

Identical function text *259*
Identifier block *312*
Identifiers *18, 30, 70, 78, 126, 178, 299, 307*
Identifier set *18*
Identifying property *218*
IEC identifier structure *62*
Illogical values *299*
Image files *32, 365*
Image formats *365*
IMAGES *370*
Import *67, 286, 302, 357, 369, 375, 439*
Import of parts *499*
Increment *100*
Indirectly *128*
Information *98*
Initial data *585*
Insert *84, 113, 426*
Inserted *369*
Insertion mode *83*
Insertion point *99*
installation *1*
installation site *311*
Integer *265*
intelligence *118, 382*
interactive *77*
Interconnect devices *170, 546*
Interconnected *171*
Interfaces *32*
Internal *33*
Internal outputs *239*
Interruption points *512*
Item numbers *315*

J

jpg *370*
Jump functions *382*
Jump point *319*

K

Keep aspect ratio *371, 373*
Keep source directory *370*
Key combination *49*

L

Label *291*
Labeling *32, 239, 288*
Labeling data *289*
Labeling dialog *293*
Language *124*
Language display *130*
Language output *289*
Layer *122, 330*
Layer configuration file *331*
Layer designations *330*
Layer export *331*
Layer import *331*
Layer management *330, 552*
Layers *244, 251, 330*
Layer settings *331*
Layout space *472*
Legend form *555*
letters *24*
level *261*
License number *7*
Line *104*
Line thickness *24*
Links *399*
List view *53, 75, 84, 127*
Local *13*
Logic 1:1 *97*
Logical *77, 219*
Logical coordinate system *99*
Logic elements *100*
Login data *494*
Long-term archiving *365*
Lost *91*

M

Macro project *58*
Main form *263, 267*
Main function *166, 211, 575*
Main report *267*
Main terminal *541*
Male pins *200*
Managing *58*
Manual *139, 209, 240, 283, 319, 330*
Manual placement *250, 273, 275*
Manual selection *275*
Manufacturer *340*
Manufacturer data *496*
Master data *57, 408, 412*
Master data directories *49*
Maximum *430*
Message class *322*
Message description *319*
Message management *11, 24, 318*
Message number *323*
Message priority *319*
Message text *323*
Message type *319*
Method *395*

Minimum number *264*
Min. no. of report rows on report page *264*
Motor overload switch contact image *135*
Motor overload switches *22*
Mounting clearances *445*
Mounting data *342*
Mounting location *262, 311*
Mounting site *128*
Move *138, 204, 313, 406*
Move base point *103*
Moved *129*
Multi-line *21, 42, 77, 213*
Multiple designations *188*
Multiple placement *452*
Multiple selection *412*
Multiple starts *37*

N

Name *30, 283*
Naming conventions *57*
Navigation elements *382*
Navigator *157*
Negative values *99*
Nesting *70*
Network drive *28*
New *62, 181, 193, 199, 213, 225, 279, 291, 432*
New device *182, 227*
New form *425*
New functions *193, 200, 225*
New layer *332*
New page *78*
New page if property is changed *284*
New project *63*
Next *66*
Next forms *267*
No output *188*
Normal texts *117*
Note *323*
No type of check *323*
Number cable DT *215*
Numbered *71*
Numbering *18, 22, 71, 72, 167, 168, 178, 185, 204, 215, 364*
Numbering format *178*
Numbering function *178*
Numbering (offline) *18*
Numbering (online) *18*
Numbering options *205*
Numbering pattern *226*
Number/Name *131*
Number of decimal places *30*
Number (offline) *178*
Number plugs *199*
Number terminals *189*
Numeric *190*

O

object-oriented approach *12*
ODBC *33*
Offline *18, 178, 307*
Offline check *320*
Offline numbering *18, 178*
Offline type of check *323*
One language (variable) *124*
Online *12, 71, 240, 258, 307, 319*
Online assignment *18*
Online check *320*
Online/offline type of check *324*
online system *12*
Open *81, 86, 262, 370, 431*
Open in new window *81*
operation *59*
Options *289*
Order number *340*
Origin *98, 99*
Original position *451*
Original project *396, 439*
Output *17, 295, 361*
Output directory *360, 368*
Output file *289, 302*
Output format *271, 280*
Output settings *367*
Output sorting *372*
Output to pages *262*
Output type *289*
Overview *21, 34, 419*
Overview cross-references *21*
Overwrite with prompt *370*
Overwriting *61*
Overwritten *424*
own layers *332*

P

Page *22, 78*
Page break *260, 263, 430*
Page contents *82*
Page description *92, 284, 570*
Page editing *78*
Page handling *24*
Page name *21, 78, 524*
Page navigator *75*
Page number *83, 89, 265*
Page numbering *511*
Page-oriented *71*
Page properties *38, 84, 92, 253, 296, 426*
Page property texts *119*
Page range *372*
Pages *297, 357*
Page sorting *262, 284, 535*
Page structure *60, 65, 80, 88, 262, 364, 418*
Page type *21, 28, 77, 570*
Pair index *546*
Pairs *512*
Part *134*
Part assemblies *261*
Partial output *263, 284*
Part number *134, 340*
Part quantity list *242*
Parts data *339*
Parts database *24*
Part selection *24, 33, 134*
Parts list *241*
Parts management *164, 227, 336*
Parts master data *376*
Path areas *42*
Path function text *118, 221*

Path numbering *24, 71*
Path-oriented *71*
PDF export *383*
PDF file *384*
PDF format *382*
percent *370*
performance *29*
PE terminals *190*
phone *30*
P_ID *13*
Pin connection diagram *251*
PIN number *218*
Pins *197*
Place *197, 201, 213*
Place centered *455*
Placed *170, 241, 371*
Placeholder text *426, 435*
Place multi-line *183, 197*
Placing *263*
Plain text *117*
PLC address *219, 221*
PLC assignment lists *18*
PLC box *225*
PLC card *159, 218, 226*
PLC card overview *250*
PLC components *250*
PLC data *228*
PLC diagram *250*
PLC navigator *217*
PLC numbering *18, 72*
PLC overview *219, 220*
PLC-specific settings *224*
PLC terminal *218, 220*
Plot frame *25, 415, 433, 434*
Plot frame documentation *248*
Plot frame editor *433*
Plot frame properties *38, 139, 435*
Plug definitions *198*
Plug diagram *251*
Plug navigator *197*
Plug overview *252*
Plugs *197, 200, 252*
Plug/Socket strips *200*
png *370*
point wiring *20*
Polyline *104*
positions *139*
post-processing *419*
Potential overview *248*
Prefix *17*
Preselection *24*
Preview *85, 362*
Preview dialog *168*
Preview of result *179, 205*
previous versions *439*
Prices/Other *340*
Print *372*
Print command *371*
Printer *372*
Print margins *373*
Print position *373*
Print preview *374*
Print settings *373*
Print size *373*
Print to scale *373*
program directory *13*
program start *14*
Project *57, 66*
Project cables *244*
Project checks *24, 320*
Project data *157, 197, 240, 260, 262, 264, 274, 294, 422, 426*
Project database *36*
Project documentation *239, 244, 422*
Project editing *24, 60, 62*
Project file *392*
ProjectInfo.xml *392*
Project languages *124, 289*
Project management *14*
Project master data *16, 24, 32, 49, 58, 262, 266*
Project messages *307*
Project name *67, 91, 401*
Project options *500, 501*
Project options group *501*
Project options overview *508*
Project-oriented *71*
Project path *400*
Project properties *16, 38, 65, 392*
Project property texts *119*
Project selection *86*
Project setting *267*
project-specific *330*
Project structures *18*
Project template *60*
Project type *58, 59*
Project wizards *65*
Properties *37, 65, 92, 183, 283*
Property *129*
Property arrangement *45, 550*
Property dialog *99*
Property name *37, 128*
Property number *30, 37*
Property selection *39, 41, 92, 129*
purchase price *340*

Q

Quantity unit *340*

R

Range *28, 190*
Read back *302*
read-only *299*
read write *299*
Rectangle *104*
Reduce *111*
Reduction *360*
Relative coordinate input *102*
Remove project data *405*
Renaming *89, 296*
Repetition *259, 289*
Replace *313, 316*
Report pages *241, 265*
Report runs *258*
Reports *16, 29, 42, 118, 239, 269*
Report type *41, 241, 262, 270, 280*
Representation type *132*
Responsible for the project *72*
Restore *400, 401*
Restore back *400*
Restored *391*

Restore master data *411*
Restrictions *280*
Review *320*
Revised *60*
Revision control *24*
Revision overview *249*
Revision tracking *60*
Rights database *522*
Round *265*
row texts *435*
RX *97*
RY *97*

S

Save *384*
Save additionally *396*
Saved *304*
Scale *92, 111, 510*
Scaling factor *360*
schematic-oriented view *227*
schematic project *58, 565*
Scheme *69, 178, 205, 224, 290, 291, 300, 319, 321, 359, 363*
Script *24*
Script name *406*
Scrolling *32*
Sections *501*
Select *89, 170*
Select devices *171*
Selected *12, 275, 292, 296, 300, 361*
Select form *262*
Selection *82, 372*
Selection list *72, 128*
Select plot frame *436*
Separators *22, 32, 70*
Sequence *41, 190, 329, 525*
Sequence chains *17*
Sequence sheets *17*
Sequential numbering structure *62*
Service time *343*
Set data types *221*
Settings *15, 142, 290*
Settings – Company *35*
Settings – Project *16*
Settings – Station *35*
Settings – User *28*
shopping cart *500*
shortcut keys *34*
Single-line *21, 77*
Single-screen solution *1*
Size *370*
Snap *524*
Socket strip *198*
Sort *186, 315*
Sort code *512*
Sorting *77, 315*
Sorting into *364*
Sorting sequence *80*
Sort setting *283*
Source *171, 203, 215, 358, 383*
special characters *17, 526*
special texts *38, 40, 119, 426*
Split size *396*
Standard *186*
Standard base point *103*
Standard layers *330*
Standard plot frame *71*
Start application *293, 302*
Starting point *515*
Start page *283*
Static *102, 430*
Static forms *264, 430*
Status *406*
Status bar *97*
STEP *460*
storage location *57, 67*
storage medium *396, 410*
Stored *25, 37, 49, 58, 71, 262, 403, 431*
Storing *15*
Stretch *515, 516*
Structure *65, 127*
Structure identifier management *77, 307*
Structure identifier overview *252*
Structure identifiers *17, 19, 24, 30, 78, 281, 307*
student version *514*
Style version *360*
Sub-entries *330*
Subpages *24, 265*
Subsequent data *267*
Subsequent editing *296*
Summarized *262*
Summarized parts list *242, 267*
Superior *69, 518*
Supplementary field *25*
– Sheet no. *283*
Supplier *340*
Switched on *100*
Symbol *572*
Symbol data *131*
Symbol/Function data *131*
Symbol library *16, 25, 126*
Symbol library assignment *33*
Symbol list *126*
Symbol names *126*
Symbol overview *252*
Symbol preview *126*
Symbol project *58*
Symbol properties *21, 38*
Symbol representation type *127*
Symbols *21, 126*
Symbol selection *126, 127, 162, 573*
Symbol variants *126*
Synchronization *58, 416*
Synchronize *23, 169*
Synchronize function texts *170*
Synchronize master data *241, 425, 520*
Synchronize parts *376*
System master data *16, 24, 48, 58, 266, 415, 416*
system messages *30*

T

TAB *296*
Tabbed page list *81*
Table of contents *244*

Tabs *66*
Target *171, 203, 215*
Target and point wiring *20*
Target designations *364*
Target directory *400*
Target file *293*
Target project *64*
Target system *360*
Target wiring *19*
Technical characteristic *128*
Technical data *343*
Template *67, 267, 269, 280, 294, 566*
Template project *510*
Temporary *58, 60, 416, 434, 437*
Temporary page *425, 431*
Terminal connection diagram *246*
Terminal editing *159*
Terminal line-up diagram *247*
Terminal number *218*
Terminal parts *261*
Terminal schemes *191*
Terminal strip definitions *188, 544*
Terminal strip navigator *185*
Terminal strip overview *248*
Text dialog *46*
Text expansion *124*
Text file *291*
Text input field *121*
texts *116*
title page/cover sheet *253*
T-node *20, 32*
Total *262*
Total + higher-level function *263*
Transferred *170*
translation *26, 117, 122, 124, 315*
translation run *28*
tree structure *76*
Tree view *52, 84, 127*
two-screen solution *1*
types of check *318*

U

UNDO command *91*
UNDO function *296, 433*
Ungroup *112*
Unit *22, 30*
Unplaced *170, 183, 197, 201, 202, 225, 242, 403*
Unzipped *391*
Update *279*
Update pages *287*
upper-case letters *17*
Use available function data *143*
Used *202*
Use existing *160*
Use graphical coordinates *101*
Use original DT *209*
user-defined *46, 312*
user interface *28, 51, 96*
user settings *31*
user supplementary field *25*
using the mouse *51*
Utilities *239*

V

Value *128*
Value sets *512*
Variant *131*
Viewer *560*
Viewing angel *447*
Viewpoints *157*

W

Warning *323*
watermark *24*
Wearing part *343*
Which cables *210*
Window legend *250*
Window macros *58*
Wire *203*
Wizard *66*
Workbook *81*
Working area *96*
Workspace *29, 30*
Write back *219, 240, 298*
Write protection *396*
Writing back *206*

X

XML file *291, 296, 376*
X,Y *98*

Y

Y positions *102*

Z

Zipped *60, 391*
zw1 *408*
zw2 *408*
zw3 *408*
zw4 *409*
zw5 *409*
zw6 *409*
zw7 *409*
zw8 *409*

Bernd Gischel

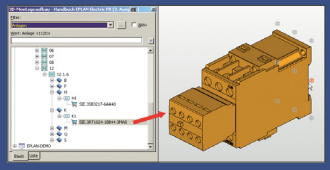
EPLAN Electric P8 Reference Handbook

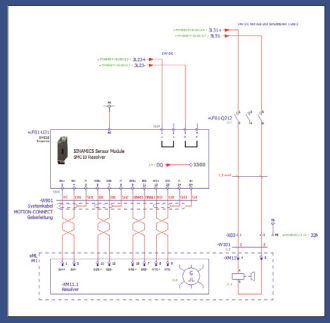
This reference book, now in its third edition, offers a comprehensive introduction to electrical engineering design with EPLAN Electric P8. Based on Version 2.2 of EPLAN Electric P8, this handbook gives you an introduction to the system basics before going into the range of functions offered by EPLAN Electric P8.

This book covers topics such as creating projects, the graphical editor (GED), using navigators, creating reports, parts management, message management, revision management, exporting, importing, printing, data backup, master data editors and importing old EPLAN data. It also covers add-ons such as EPLAN Pro Panel and EPLAN Data Portal.

Numerous examples show you the many ways you can use EPLAN Electric P8 and give you ideas of how to best solve everyday tasks. Practical information, such as a step-by-step procedure for creating schematic projects and a chapter with FAQs, is also included.

The examples used in the book are available as an EPLAN Electric P8 project at www.eplan-efficient-engineering.com/handbook





Bernd Gischel works for Dressel GmbH & Co. KG in Unna as EPLAN administrator and designer for electrical engineering systems. He moderates the EPLAN forums at CAD.DE and EPLAN.DE and is responsible for the »P8 Magic« knowledge and data portal for EPLAN Electric P8.

HANSER

www.hanser-fachbuch.de
Carl Hanser Verlag
ISBN 978-1-56990-476-3

9 781569 904763