Bernd Gischel

EPLAN Electric P8 Reference Handbook

Bernd Gischel

EPLAN Electric P8 Reference Handbook

4th edition

Hanser Publishers, Munich
Hanser Publications, Cincinnati

HANSER

The Author:

Bernd Gischel, Lünen, Germany

Translated by think global GmbH, Berlin, 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.
The final determination of the suitability of any information for the use contemplated for a given application remains the sole repsonsibility of the user.

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 2016
Production Management: Denise Jäkel
Coverconcept: Marc Müller-Bremer, www.rebranding.de, München, Germany
Coverdesign: Stephan Rönigk
Typesetted by Kösel Media GmbH, Krugzell, Germany
Printed and bound by Firmengruppe Appl, aprinta druck, Wemding, Germany
Printed in Germany
ISBN 978-1-56990-498-5
E-Book ISBN 978-1-56990-499-2

Contents

Preface 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 17
2.3.1 Settings – Project 18
2.3.2 Settings – User 34
2.3.3 Settings – Station 41
2.3.4 Settings – Company 41
2.4 EPLAN and multiple starts? 43
2.5 Properties 43
2.5.1 Project properties 45
2.5.2 Page properties 47
2.5.3 User-defined properties 48
2.5.4 Symbol properties (components) 52
2.5.5 Form properties 53
2.5.6 Plot frame properties 53
2.6 Buttons and popup menus 53
2.6.1 Device dialog buttons 54
2.6.2 Buttons in dialogs (configuring) 54

2.6.3 Buttons in dialogs such as filter schemes 55
2.6.4 Restore default function 56
2.6.5 Property arrangements (components) 56
2.6.6 Format properties 58
2.6.7 Buttons (small black triangles) 59
2.6.8 Dialogs for schemes 59
2.7 Master data 60
2.8 Operation 61
2.8.1 Using the keyboard 61
2.8.2 Using the mouse 63
2.9 User interface – more useful information 63
2.9.1 Using workspaces 63
2.9.2 Dialog display 64

3 Projects 69

3.1 Project types 70
3.1.1 Project types in EPLAN 71
3.1.2 Project templates and basic projects 72
3.2 Creating a new project 74
3.2.1 New project (from a basic project) 74

4 The graphical editor (GED) 79

4.1 Page navigator 79
4.1.1 Page types 81
4.1.2 The popup menu in the page navigator 84
4.1.3 Page navigator filter 103
4.1.4 Edit in table 105
4.1.5 List with preselection 107
4.2 General functions 108
4.2.1 Title bar 108
4.2.2 Status bar 109
4.3 Coordinate systems 110
4.3.1 Graphical coordinate system 110
4.3.2 Logical coordinate system 111
4.3.3 Fluid power and process-engineering coordinate systems 111
4.3.4 3D coordinate systems 111
4.4 Grid 112
4.5 Increments, coordinate input 113
4.5.1 Increment 113
4.5.2 Coordinate input 114
4.5.3 Relative coordinate input 115
4.5.4 Move base point 115

4.6 Graphical editing functions 116
4.6.1 Graphical objects: lines, circles, rectangles 117
4.6.2 Trim, chamfer, stretch and more 119
4.6.3 Group and ungroup 127
4.6.4 Copy, move, delete 128
4.6.5 Dimensioning 129
4.7 Texts 132
4.7.1 Normal (free) texts 133
4.7.2 Path function texts 134
4.7.3 Special texts 136
4.7.4 Properties – Text dialog 139
4.8 Components (symbols) 145
4.8.1 Insert components (symbols) 145
4.8.2 Properties (components) dialog – [device] tab 147
4.8.3 Display tab 148
4.8.4 Symbol/function data tab 150
4.8.5 Parts tab 153
4.9 Cross-references 154
4.9.1 Contact image on component 154
4.9.2 Contact image in path 157
4.9.3 Special feature: Pair cross-reference 159
4.9.4 Contact image distributed device list 162
4.10 Device selection settings 164

5 Navigators 165
5.1 Overview of the most important navigators 166
5.1.1 Additional navigators and modules 171
5.1.2 General navigator functions 174
5.2 Device navigator 174
5.2.1 Swap 176
5.2.2 Assign main function 176
5.2.3 Synchronize function texts 179
5.2.4 Numbering (Number DT) 180
5.2.5 Device protection 186
5.2.6 Import (device data) 187
5.2.7 New 190
5.2.8 New functions 192
5.2.9 New device 193
5.3 Terminal strip navigator 195
5.3.1 Edit (terminal editor) 195
5.3.2 Generate multi-level terminal 200
5.3.3 Number terminals 201

5.3.4 New 205
5.3.5 New functions 206
5.3.6 New terminals (devices) 207
5.3.7 View 210
5.4 Plug (female pin) navigator 212
5.4.1 Edit (plug/female pin editor) 212
5.4.2 Number pins 213
5.4.3 New 213
5.4.4 New functions 214
5.4.5 Generate plug definition (plug and/or female pin) 214
5.4.6 Generate pin (plug and/or female pin) 215
5.5 Cable navigator 217
5.5.1 Edit 217
5.5.2 Number (cable DT) 219
5.5.3 Automatic cable selection 221
5.5.4 Generate cables automatically 223
5.5.5 Assign cable connections 226
5.5.6 New 228
5.5.7 Number DT 229
5.6 PLC navigator 231
5.6.1 Write back connection point descriptions 233
5.6.2 Set data types 235
5.6.3 Export/Import data 235
5.6.4 Addresses/assignment lists 236
5.6.5 Address 239
5.6.6 New 241
5.6.7 New functions 242
5.6.8 New device 244
5.6.9 View 244
5.7 Part navigator (Devices/Parts) 246
5.7.1 Add project part 247
5.7.2 Add part 249
5.7.3 Insert device 249
5.7.4 Exchange part 250
5.7.5 Edit part 251
5.7.6 Device selection 251
5.7.7 Assign item number 252
5.7.8 Synchronize parts data 253
5.7.9 Export/Import 253
5.7.10 View 253
5.8 Macro navigator 255
5.8.1 Macros – a general description 255
5.8.2 Macro project 255

5.8.3 Macro box 256
5.8.4 Macro navigator/Generate automatically 264
5.8.5 Macros in general (without macro project) 268
5.8.6 Macros with value sets (without macro project) 272
5.9 Navigators – general functions 281
5.9.1 Place from navigators 281
5.9.2 Assign from navigators 284
5.9.3 Filters 285
5.10 Correction functions 287

6 Reports 289

6.1 What are reports? 290
6.2 Report types 290
6.3 Types of graphical reports 291
6.3.1 Report types (forms) 291
6.3.2 Special connection diagrams 310
6.3.3 Next forms 312
6.3.4 Conditional forms 315
6.4 Settings (output options) 317
6.4.1 The Display/output project setting 318
6.4.2 The Parts project setting 319
6.4.3 The Output to pages project setting 321
6.5 Generate reports 333
6.5.1 Reports dialog 333
6.5.2 Generate reports without templates 335
6.5.3 Popup menus in the Reports tab 345
6.5.4 Generate reports with templates 346
6.6 Other functions 354
6.6.1 Update 354
6.6.2 Generate project reports 355
6.6.3 Generate report project 355
6.6.4 Settings for automatic updates 355
6.7 Labeling 356
6.7.1 Settings 356
6.8 Edit properties externally 365
6.8.1 Export data 366
6.8.2 Import data 373

7 Management tasks in EPLAN 375
7.1 Structure identifier management 375
7.1.1 List of identifiers in structure identifier management 379
7.1.2 Graphical buttons 381
7.1.3 Sort menu 383
7.1.4 Extras menu 384
7.1.5 Configure protection (protect identifiers from changes) 386
7.2 Message management 388
7.2.1 The visual appearance of message management 389
7.2.2 Project checks 390
7.2.3 Message classes and message categories 391
7.2.4 Filters in message management 399
7.2.5 Various message edition options 401
7.3 Layer management 404
7.3.1 Standard layers 405
7.3.2 Export and import of layers 405
7.3.3 Create and delete your own layers 406
7.3.4 Uses of layers 408
7.4 Parts management 411
7.4.1 Structure of parts management 413
7.4.2 Tabs in parts management 413
7.5 Revision control 423
7.5.1 General 423
7.5.2 Generate new revision 424
7.5.3 Execute changes 425
7.5.4 Complete page(s) 429
7.5.5 Generate reports 431
7.5.6 Complete a project 432
7.6 Project management 434
7.6.1 Project management dialog 434
7.6.2 Project management buttons 436
7.6.3 Project management filters 441
7.6.4 Create projects (project management) 442

8 Export, import, print 445
8.1 Export and import of DXF/DWG files 446
8.1.1 Exporting DXF and DWG files 446
8.1.2 Import of DXF and DWG files 451
8.2 Image files 456
8.2.1 Exporting image files 456
8.2.2 Insert image files (import) 459

8.3 Print 463
8.3.1 The Print dialog and its options 463
8.3.2 Important export/print setting 466
8.4 Export and import of projects 466
8.4.1 Exporting projects 467
8.4.2 Importing projects 467
8.5 Print attached documents 468
8.6 Import PDF comments 469
8.6.1 Importing commented PDF documents 470
8.6.2 Deleting PDF comments 473
8.7 Generate PDF documents 475
8.7.1 Export of PDF files 476

9 Data backup 485

9.1 Zipping and unzipping of projects 486
9.1.1 Zip projects 486
9.1.2 Unzip projects 488
9.2 Backing up and restoring projects 489
9.2.1 Back up projects 489
9.2.2 Restore projects 495
9.3 Other important settings 498
9.3.1 Default settings for project backup (global user setting) 498
9.3.2 Compress project (remove unnecessary data) 499
9.3.3 Automated processing of a project 501
9.4 Backing up and restoring master data 505
9.4.1 Backing up master data 505
9.4.2 Restoring master data 507
9.5 Send project by e-mail directly 509

10 Master data editors 511

10.1 Preparatory measures 514
10.2 Clear overview of forms 514
10.2.1 First option – manual overview 514
10.2.2 Second option – automatic overview 517
10.3 Forms 519
10.3.1 Create new form (from copy) 522
10.3.2 Edit existing form 529
10.3.3 Create new form 529
10.4 Plot frames 531
10.4.1 Create new plot frame (from copy) 531
10.4.2 Edit existing plot frame 534
10.4.3 Create new plot frame 535

11 Old EPLAN data (EPLAN 5) 537
11.1 Import options 538

12 Extensions 541
12.1 EPLAN Data Portal 541
12.1.1 What are the advantages of the EPLAN Data Portal? 541
12.1.2 Before the first start 542
12.1.3 How the EPLAN Data Portal works 545
12.2 Project options 553
12.2.1 What are project options? 553
12.2.2 Terminology in the Project options module 553
12.2.3 Creating options and sections 554
12.2.4 Generate options overview report 563

13 FAQs 565
13.1 General 566
13.2 Parts 599
13.3 Terminals, plugs 606
13.4 Cables 610
13.5 Properties, layers 611
13.5.1 Master data 615
13.6 Data exchange 622
13.7 Reports 624

14 Creating a schematic project – step by step 627
14.1 Create a project 627
14.2 Generate pages 630
14.3 Create a schematic 634
14.3.1 Insert symbols 635
14.3.2 Insert cables 640
14.3.3 Add texts 643
14.4 Generate reports 645
14.4.1 Generating other report pages 648

Index 649

Preface

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.

Version 2.5 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 2.5 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 2.5.

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, as it does with this one, 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 2.5, 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.

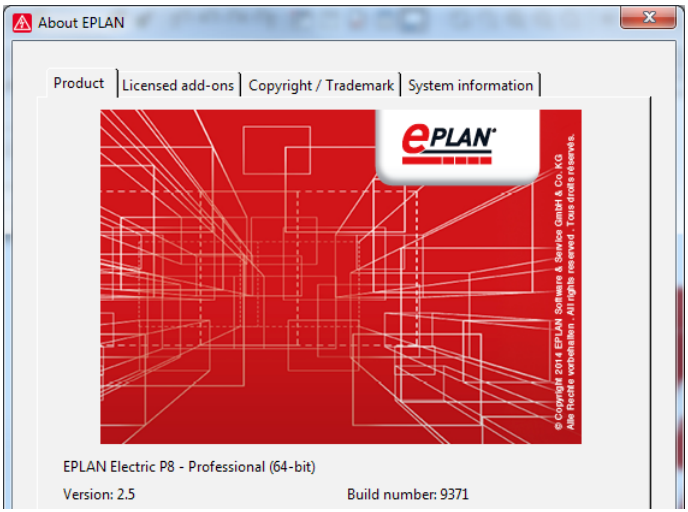
And finally, I would again like to thank EPLAN Software & Service GmbH & Co. KG for their consistent and very friendly support and collaboration in compiling some of the information for this edition of the EPLAN Electric P8 Reference Handbook.

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 2.5 was used.



Help / Info / Product tab

NOTE for users of previous versions: Certain parts of the functions described here may exist in EPLAN Electric P8 Versions 1.7 to 1.9 and 2.0 to 2.4, but their use, settings and range of functionality may differ from the current Version 2.5.

The examples used in the book are available as an EPLAN Electric P8 project at *www.eplan-efficient-engineering.com/handbook*.

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.

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.

NOTE: Starting with EPLAN Electric P8 Version 2.5, only the 64-bit variant will be available. The use of Microsoft Office 64-bit is required to use Access databases. Microsoft Office 32-bit can still be used though, but the parts/translation/project database will have switched to SQL or SQL Express.

1.1 Hardware

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 during the installation, the new EPLAN Setup Manager (available since Version 2.1) 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).

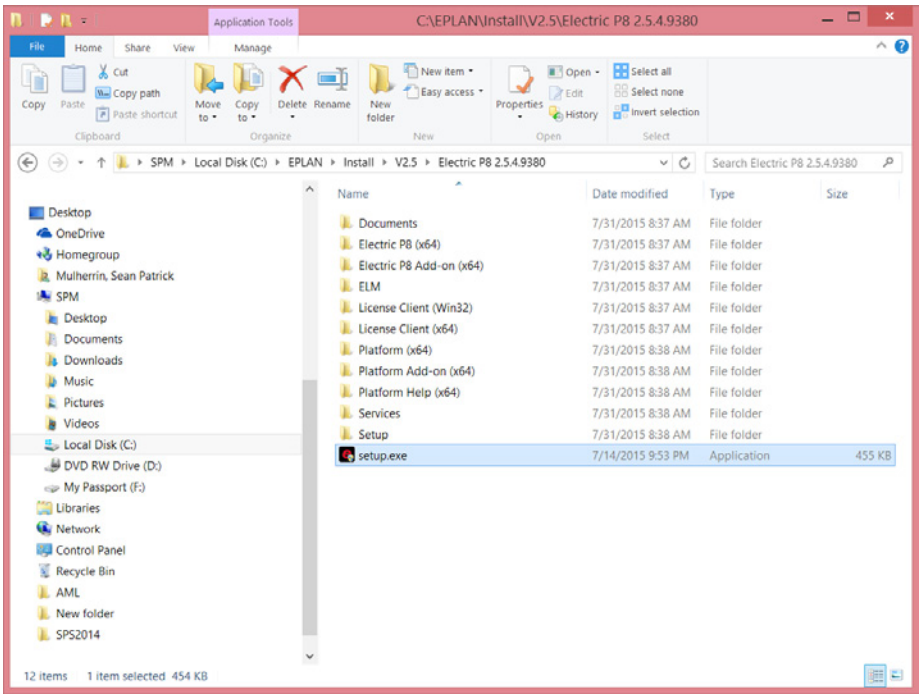


Fig. 1.1 Installation directory

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. But EPLAN will display a message if necessary.

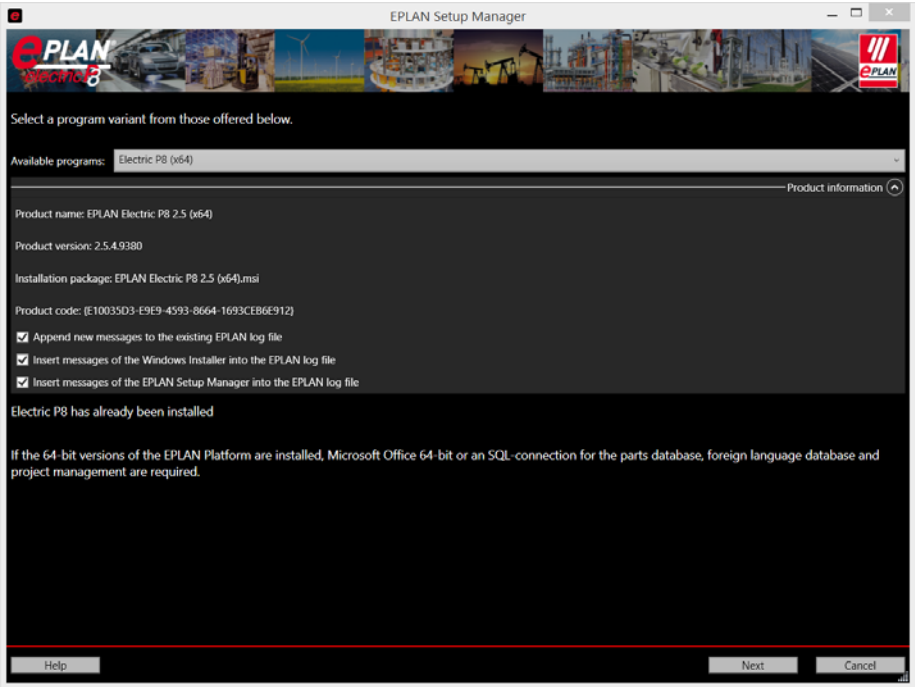


Fig. 1.2 The EPLAN Setup Manager's basic settings

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

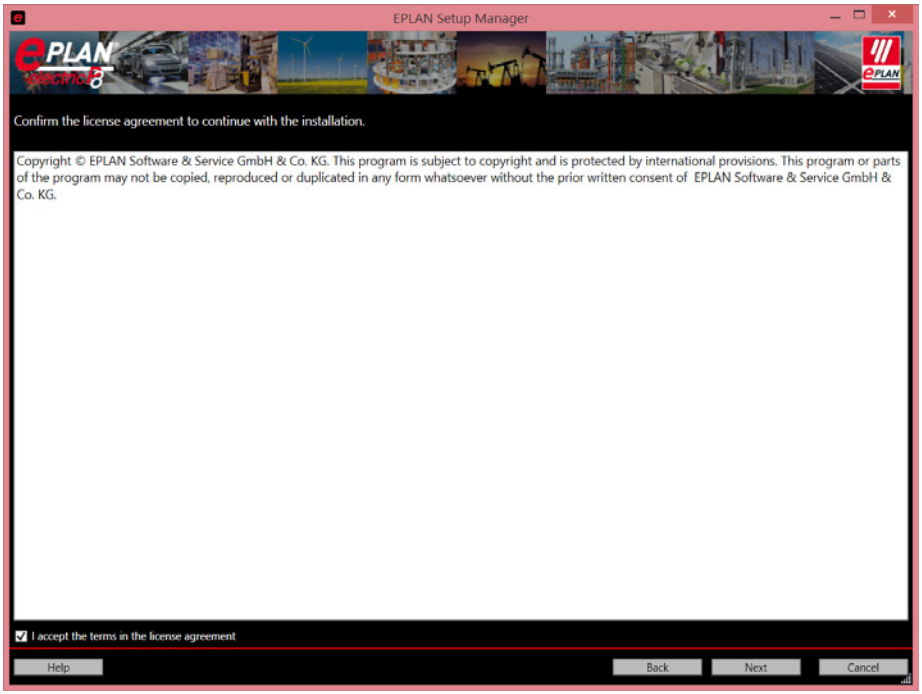


Fig. 1.3 Licensing agreement dialog

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 *measuring unit* for the system, the implementation type of the *help system* (online or local), as well as also 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.

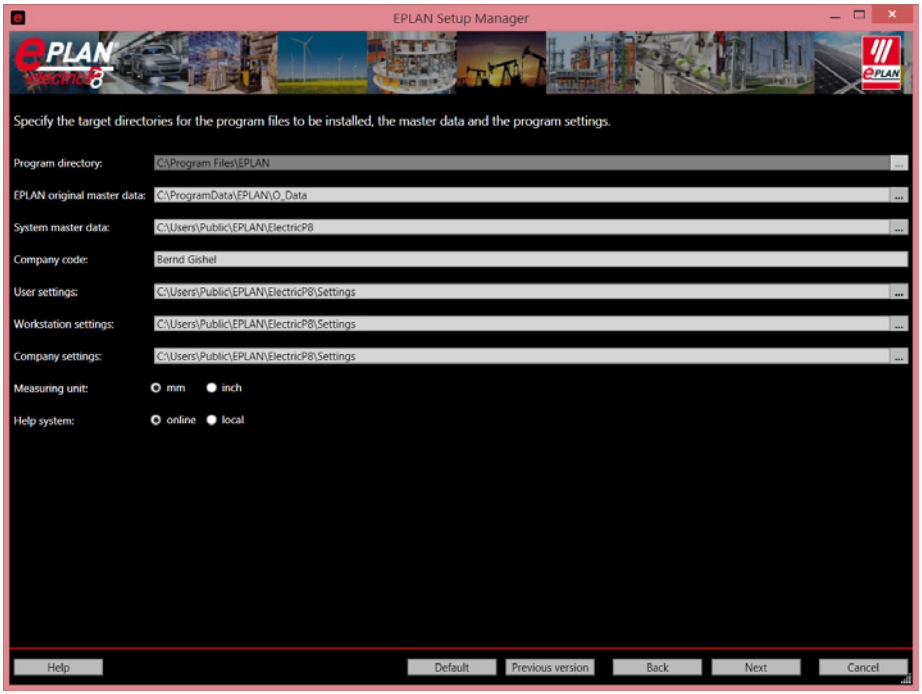


Fig. 1.4 Definition of target directories

These suggested directories can be kept or changed to the previous version (to do so, click the **PREVIOUS VERSION** button). I always change these directories (and of course the company code) to my own target directories.

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.

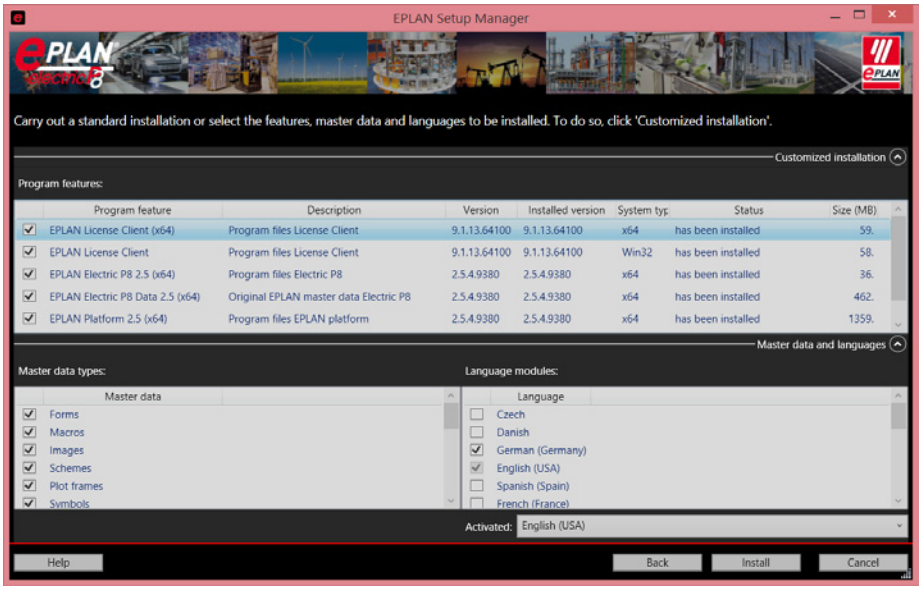


Fig. 1.5
Scope of components to be installed

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

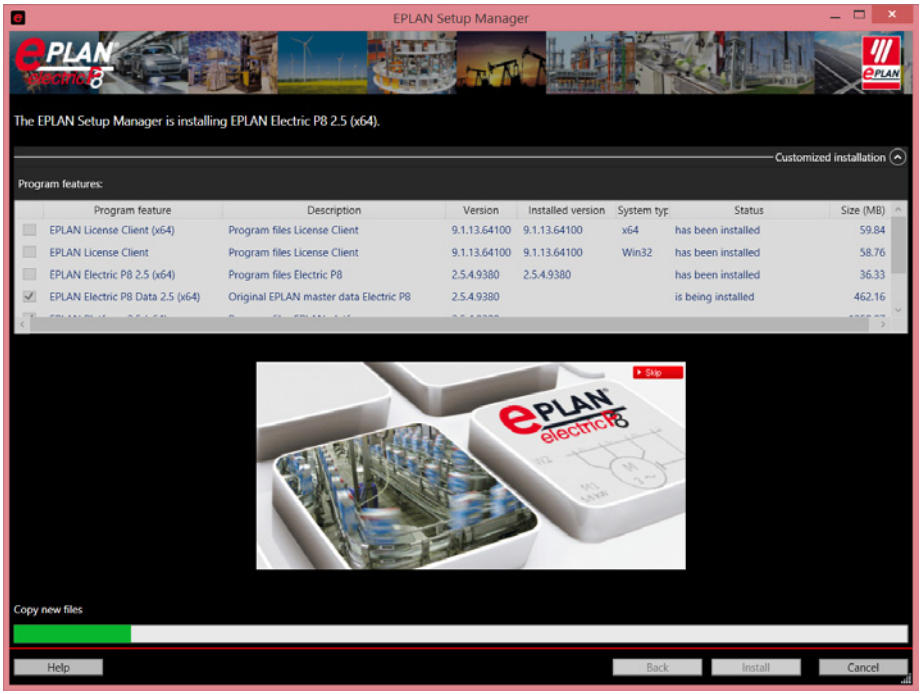


Fig. 1.6
EPLAN being installed

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 on purpose 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. 1.7
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 dialog prompting or requesting a selection of the appropriate license is displayed before the program starts.

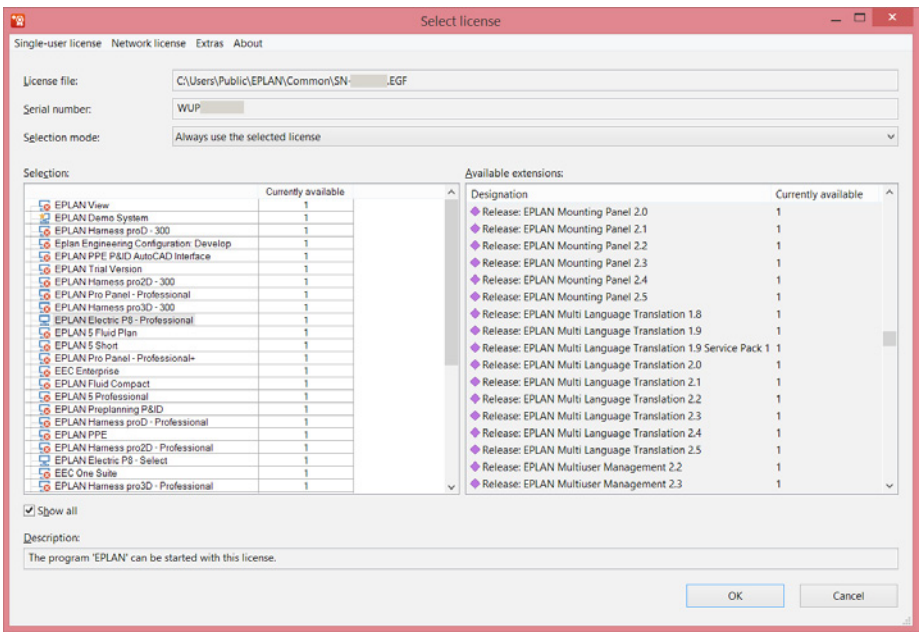


Fig. 1.8
Select License dialog

Once a selection has been made (here: EPLAN Electric P8), confirmed with **OK**, and if there is no validation, a one-time dialog requesting the validation code (license number) of the corresponding dongle (hardware protection) is displayed. This may be done via an online query or by entering the validation code received manually.

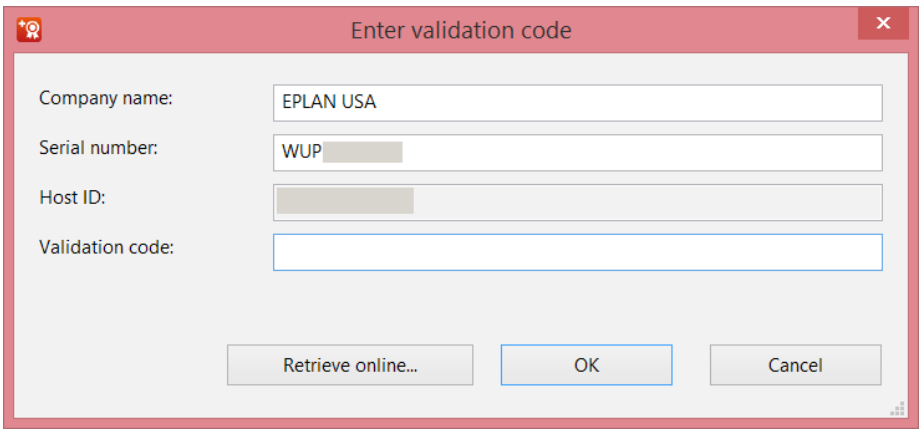


Fig. 1.9 Entering validation code and activating EPLAN

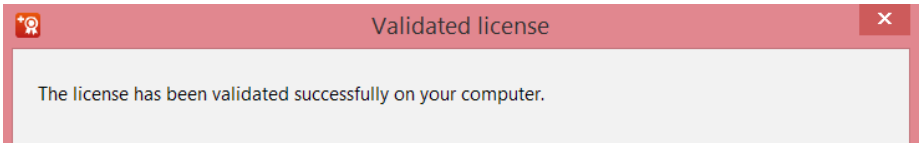


Fig. 1.10 License validated successfully

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.

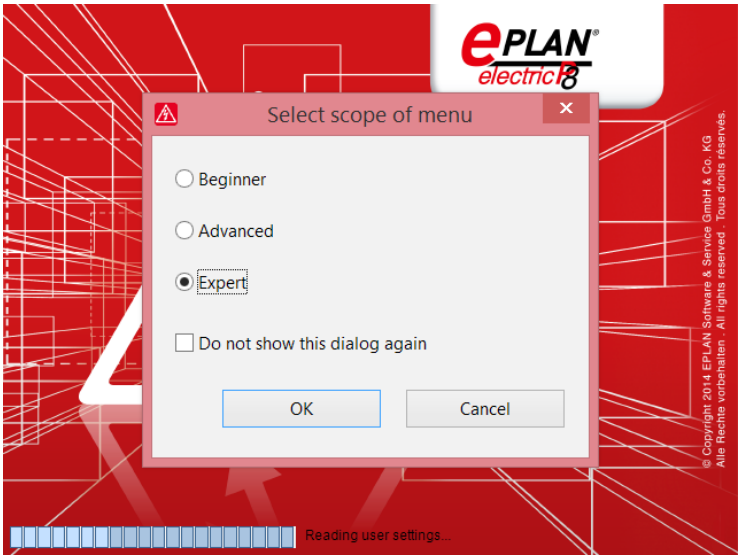


Fig. 1.11
Starting up EPLAN for the first time

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.

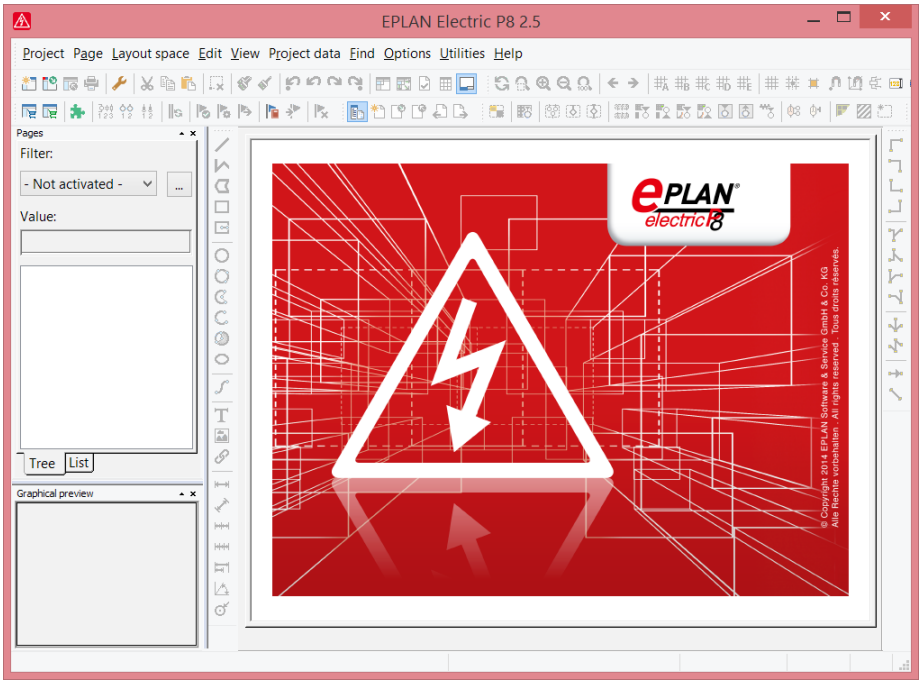


Fig. 1.12
Starting up EPLAN for the first time

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 2.5 as necessary.

1.3.1 Parallel operation with previous versions

Because EPLAN has again in Version 2.5 made some changes to the databases, which are no longer compatible with the previous versions (Version 2.3 and down), we recommend that you install Version 2.5 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 2.5 and reformat it for Version 2.5, 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 in this chapter are the directory structure, data storage, user, workstation and project settings, 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, optimized over the years.

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, depending on the setting within the check run selected. 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 (no message) or not and which priority a message (error, warning, or note) should have. It is also possible 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. 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.

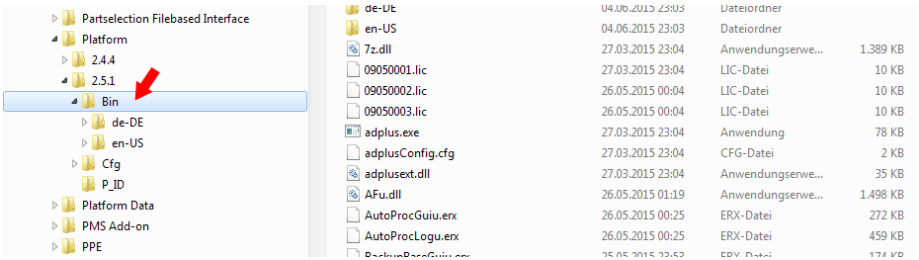


Fig. 2.1 Program directory with subdirectory

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

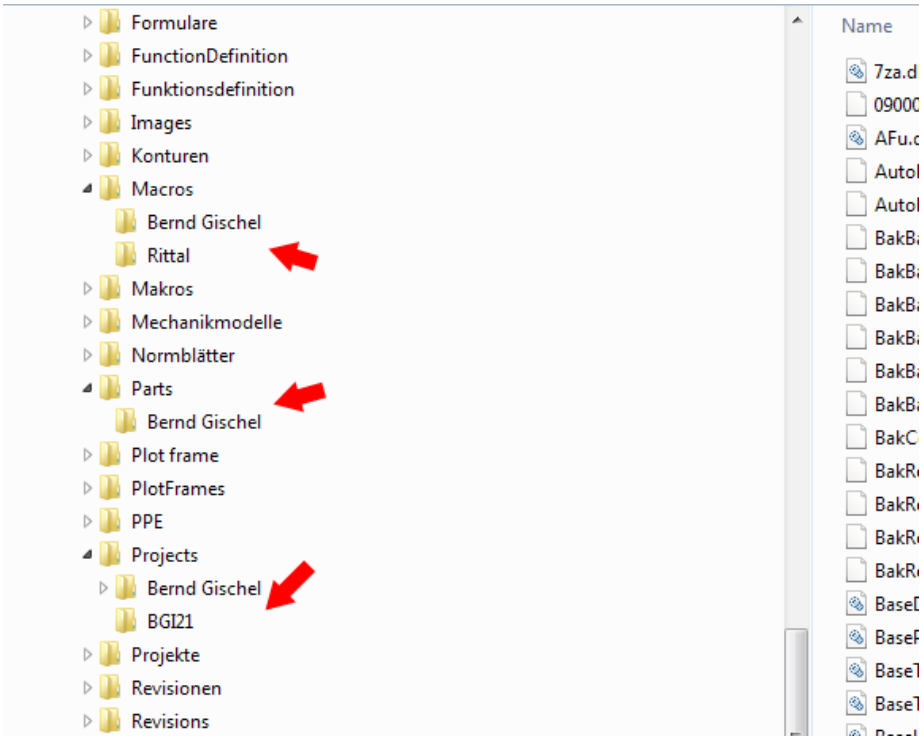


Fig. 2.2
Default customer directories

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 created, among other things, as default directories and filled with the relevant data during installation (the most important directories are listed):

- *Part:* contains the parts databases (Microsoft Access **.mdb* format) and the configuration files for importing and exporting parts
- *Images:* contains all images, such as images for parts data
- *Documents:* contains documents such as *PDF* documents or Excel tables
- *Dxf_Dwg:* contains CAD drawings in *DXF* or *DWG* format
- *Forms:* contains all forms (own system master data)
- *Function definition:* contains the files for the function definitions
- *Macros:* contains all macros, such as window macros (**.ema*), symbol macros (**.ems*), page macros (**.emp*) or also 3D macros for EPLAN Pro Panel
- *Mechanical models:* contains mechanical data, such as 3D models
- *Plot frames:* contains all plot frames (own system master data)
- *Projects:* default directory for projects
- *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 (own system master data)
- *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

In addition, during installation EPLAN stores a directory with the original EPLAN master data for subsequent synchronization with its own master data. This directory can be chosen freely during installation.

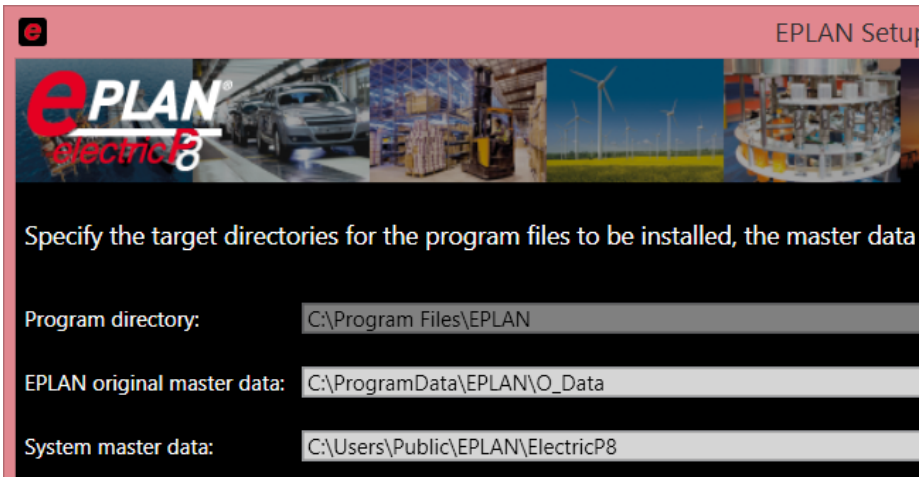


Fig. 2.3
Example: directory for EPLAN original master data

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.

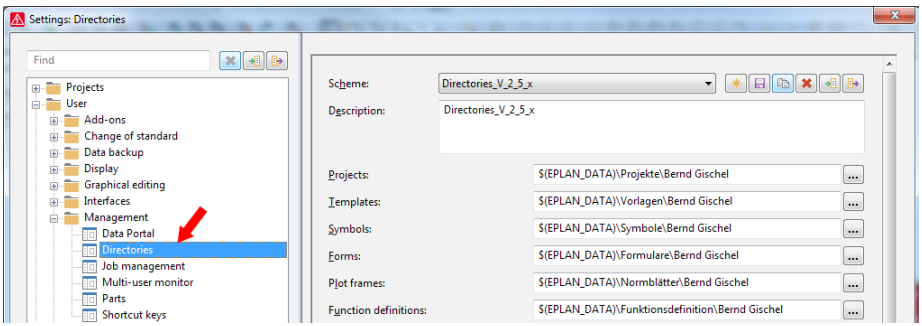


Fig. 2.4
Directory settings

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: "J:\EPLANP8x64\Platform\2.5.2\BIN\Eplan.exe" /Variant: "Electric P8"/PathsScheme:OwnSchemeName (without prefixes and suffixes). This makes it possible to use customer directories with different system master data selectively.

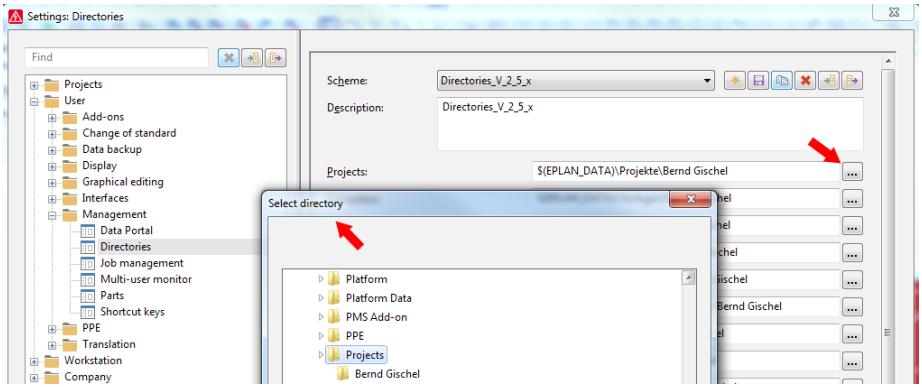


Fig. 2.5 Selecting other directories

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.

To make finding settings faster, EPLAN has integrated a search function with the **Settings** dialog.

The search itself is quite simple. Simply enter the desired term (the property you are looking for), such as "minimum", and EPLAN will instantly list suitable terms.

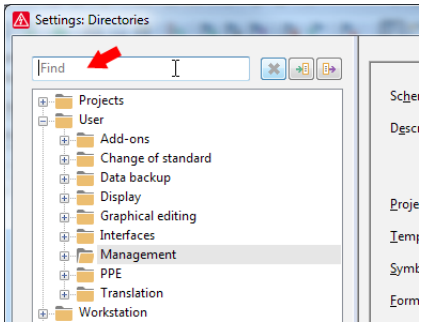


Fig. 2.6
Search field

Upper- or lowercase does not matter here. EPLAN simply looks for the text. It may be part of a word, and the word does not even have to begin with it.

Operations (AND/OR) and placeholders like * or ? do not work with this search function. Nor are numbers found, but only pure texts contained in the settings and/or their dialogs.

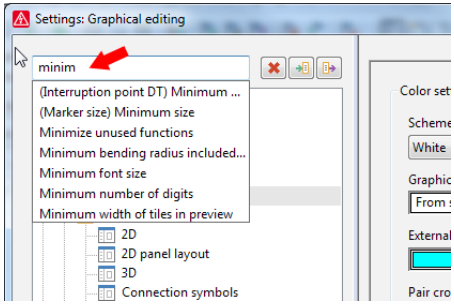


Fig. 2.7
Example: input of search term

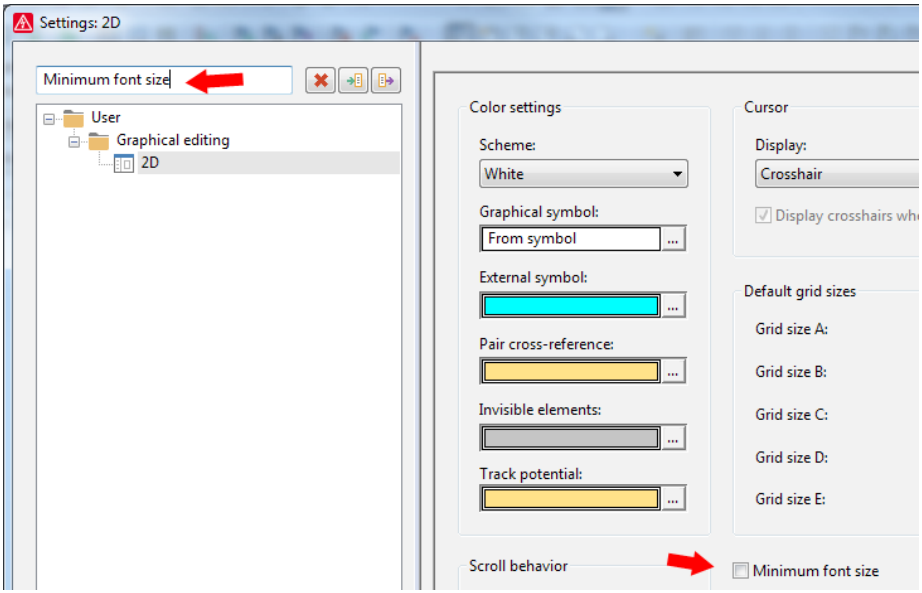


Fig. 2.8 Search results, minimum font size

If you click on the searched term, EPLAN opens the corresponding setting, and all other setting options in the tree structure are hidden. To display them again, the search must be 'emptied' first. This is done by clicking the **DELETE** button.

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.

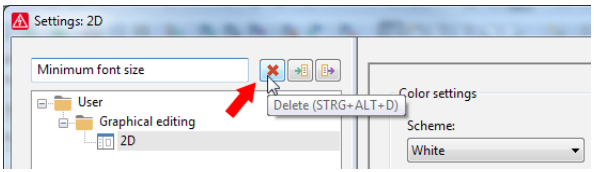


Fig. 2.9 Search results, minimum font size

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 set-

tings 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 manually 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 a number of important 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).

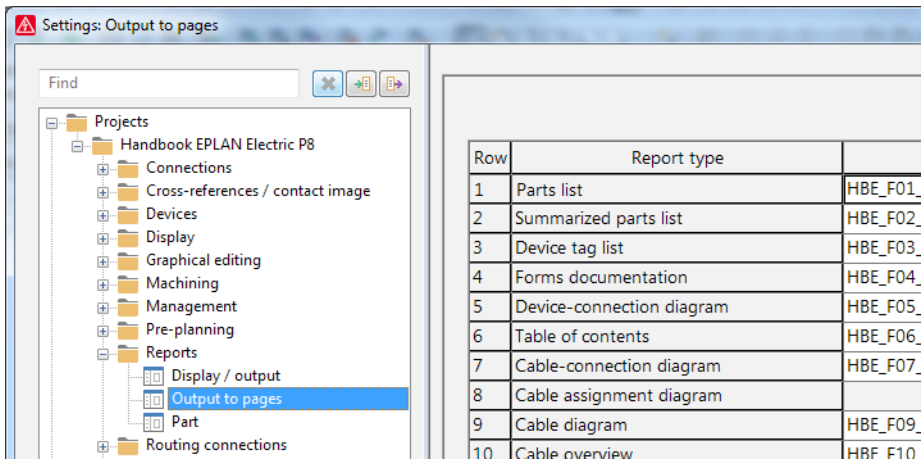


Fig. 2.10 Settings for reports

- **Display/Output:** Settings that affect the output or display of project 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.

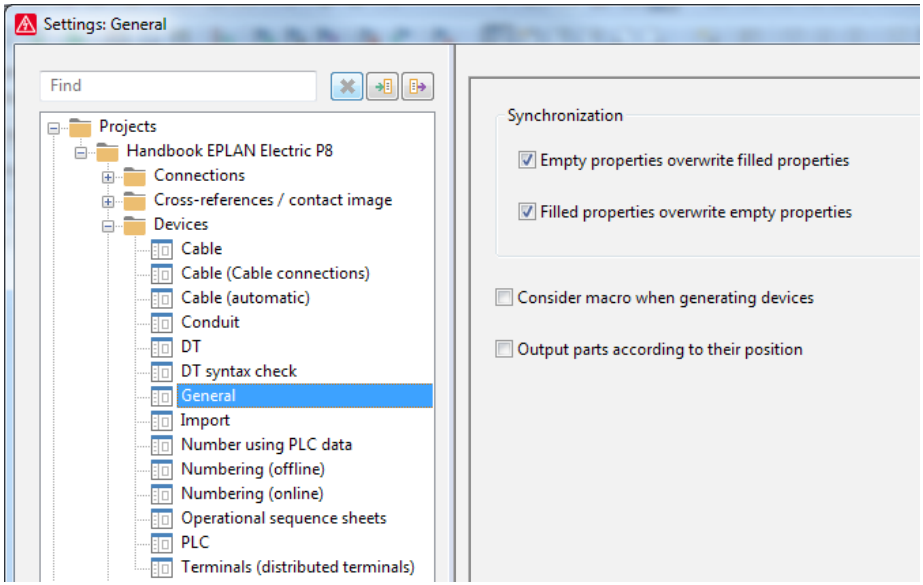


Fig. 2.11 Settings for devices

- **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 DIN EN 60848).
- **General:** Contains settings for the synchronization of functions, determining whether empty properties here may overwrite filled properties or vice versa. This is also where you find settings to specify whether a macro stored at the part is to be considered when generating new devices (such as in a navigator, via the New Device menu entry).
- **DT:** Contains general settings, including the parameter for *conversion to uppercase* for devices. If this 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 syntax check:** These settings define the possible *special characters* for structure identifiers and for the input of device tags (electrical engineering/fluid trade)
- **Import:** EPLAN allows for the import of device data. Schemes are used to define the allocation, that is, which external data belong to which EPLAN property.
- **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). A general standard cable can also be specified as a default for the project.
- **Cables (cable connections):** These settings allow for default settings; such as defining the symbols that EPLAN should use when inserting a connection definition in connec-

tion with cable definition lines and when inserting shields. These settings apply to multi-line and single-line representation types.

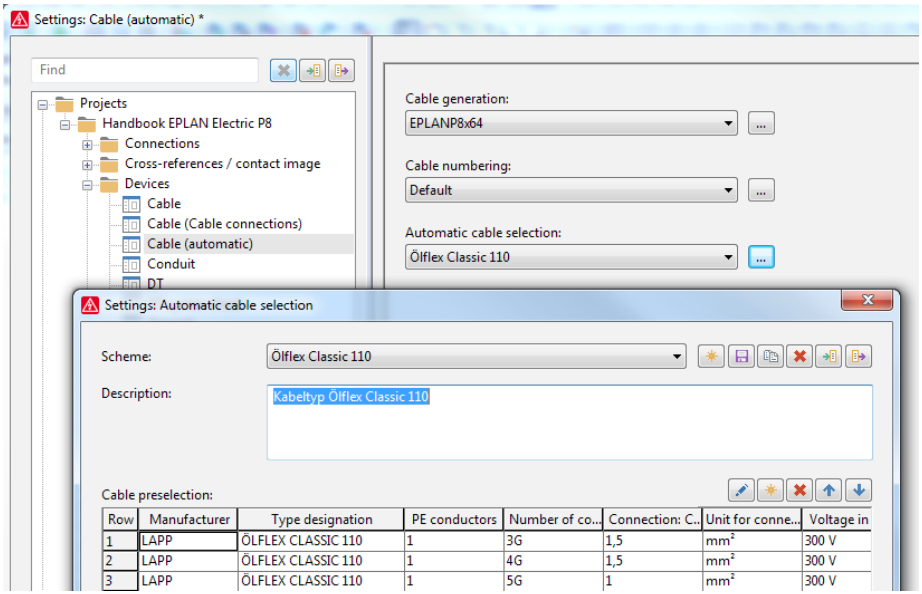


Fig. 2.12 Automatic cable selection (cable node (automatic))

- **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)? This also contains the option for optimizing jumpers.
- **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).
  Note: A conduit definition line is inserted in the same way as a cable definition line.
- **PLC:** The defaults for such output as PLC assignment lists, the settings for the extended path function text import (extension of the path in order to import also "adjacent" function texts "on the left or at the top"), 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.

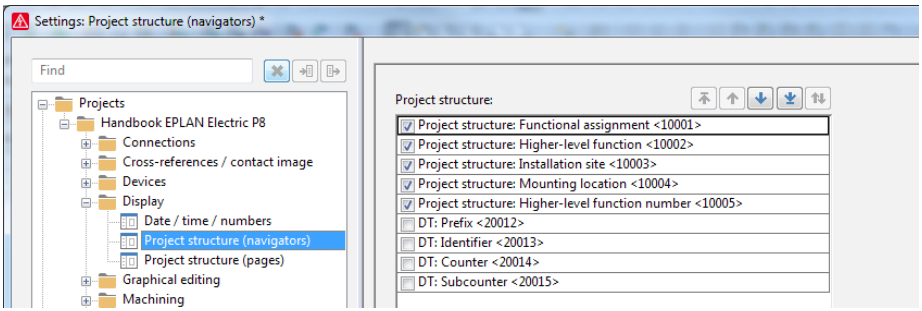


Fig. 2.13 Setting the project structure and format display

- **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. Here, you can also define whether the substructures of the structure of a device should be represented as ungrouped or grouped.

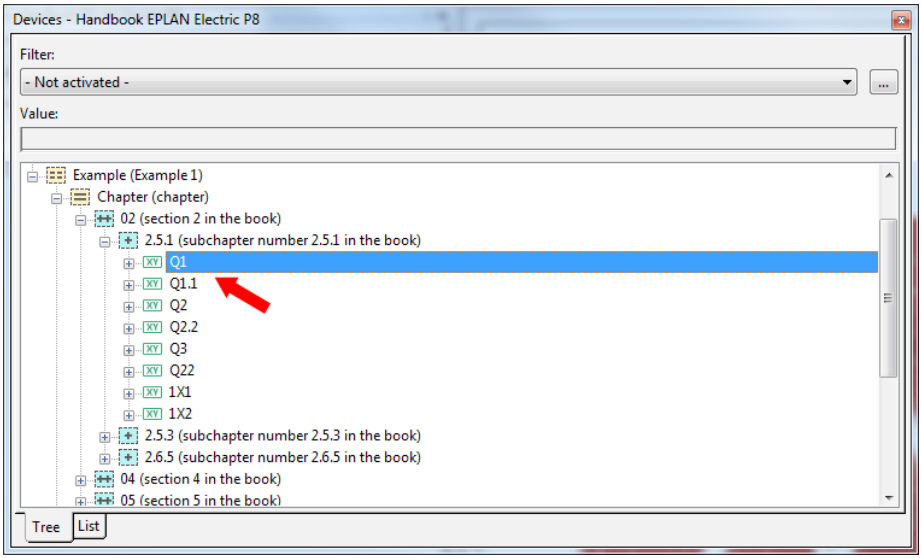


Fig. 2.14 Grouped structure of the device (display only)

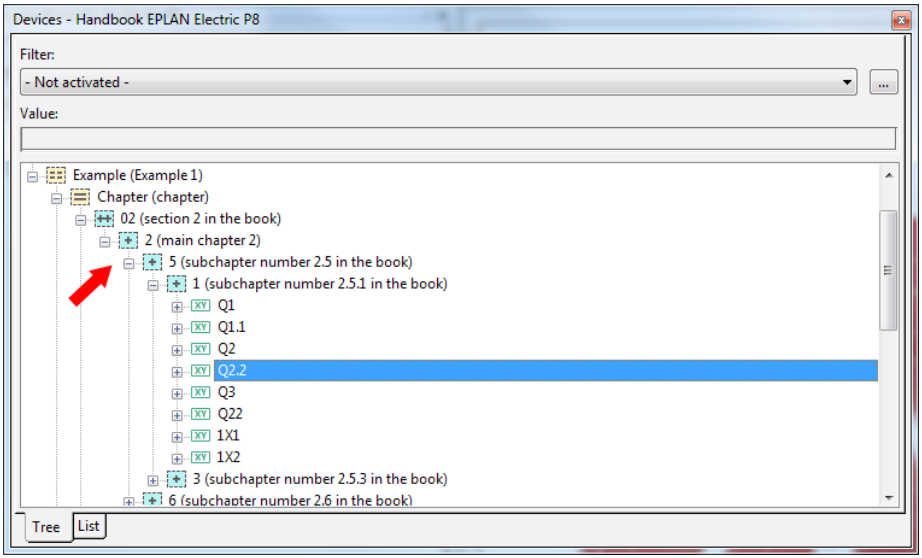


Fig. 2.15 Ungrouped structure of the device (display only)

- **Project structure (pages):** These settings are used to adjust the display sequence, especially in the page navigator (Page/Navigator menu) or via the F12 function key. Like in the previous node, there is a setting here to represent the structures as ungrouped.

NOTE: These settings change only the representation or display of the project structure or devices in the navigators. The page structure and/or the device tag structures are not changed.

2.3.1.4 Setting Projects [project name]/Graphical editing

This section covers selected general settings for graphical editing.

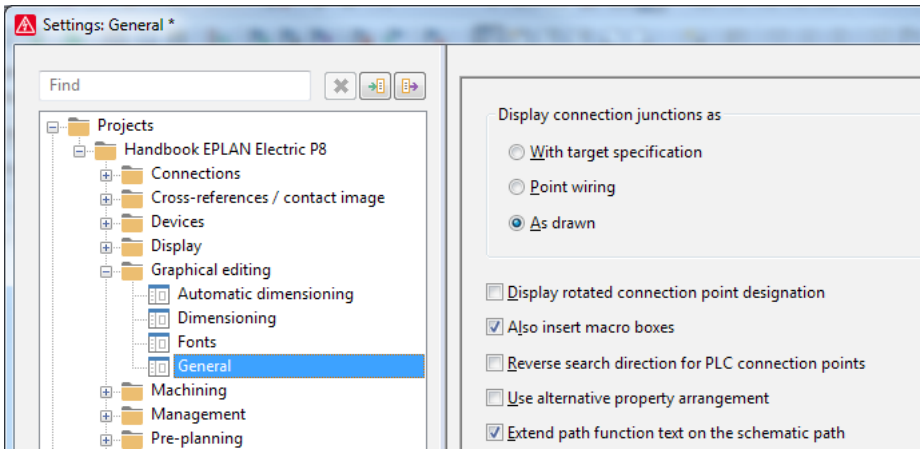


Fig. 2.16 Settings for graphical editing

- **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.

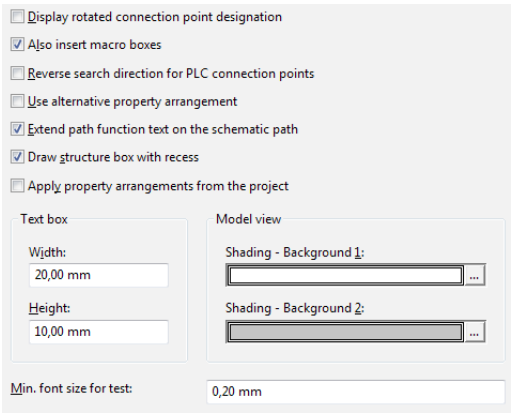


Fig. 2.17
Additional general settings

- **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 on the page.
- **Reverse search sequence for PLC connection points:** If this setting is activated, EPLAN will look for the path function texts against the connection point direction of the symbols. Usually, for example, EPLAN would run a downward search for the associated path function text at a PLC input placed at the bottom of the page and connected upward, and only if EPLAN finds no path function text will it continue the search upward. If this setting is then activated, EPLAN will look for a path function text immediately in the connection point direction.
- **Use alternative property arrangement:** Setting to achieve a representation according to the GOST standard. When this setting is activated, the properties to be displayed are represented on the right or at the top right of the symbol – in contrast to the 'regular' left-side representation.
- **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 a path (the path and/or column width to be searched is obtained from the selected plot frame).
- **Draw structure boxes with recess:** This setting allows for EPLAN structure boxes to be drawn automatically with a recess. Instead of the usual rectangle, for example, a recess is drawn in which the DT is placed, depending on the property arrangement.

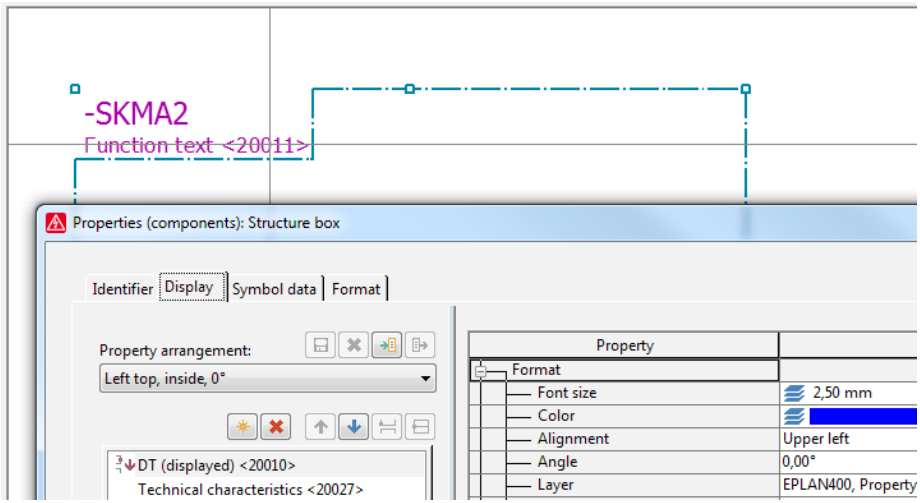


Fig. 2.18
Structure box with recess

- **Import property arrangements from the project:** If parts with configured, user-defined property arrangements are copied from a project, they are compared when being inserted into another project. Depending on the setting of this parameter, they are applied to the new project (including the given property arrangement name), or you do not overwrite the existing property arrangements, in which case you will then see the property arrangement name "user-defined" instead of the name you used in the source project.
- **Text box size:** Default width x height settings for the size of the text box (can be selected individually in the **Properties (Texts)/Format Tab** dialog).

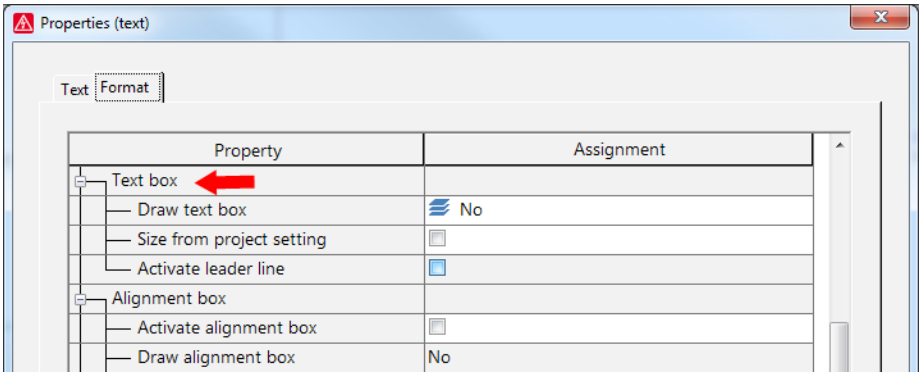


Fig. 2.19
Setting of text box in the Properties (Text) dialog

- **Model view:** Settings for the background colors of the model views of panel layouts.
- **Minimum font size for checks:** This setting is used to configure the check value (Other message class, message number 022027; Text too small message text) of the font size that must not be undershot in the project.
- **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*).

NOTE: The settings in the "Automatic Dimensioning" node do not apply to the 2D panel layout. This is why they will not be explained further here.

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

These settings offer many parameters defining the representation of references, such as for interruption points, functions, such as motor overload switches, contactors, as well as 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).

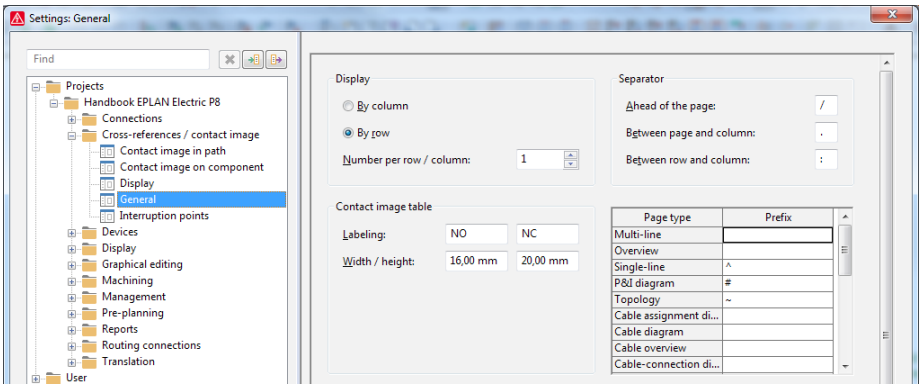


Fig. 2.20 Cross-references/contact image settings

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, the separator between the name of the interruption point and the cross-reference, the representation of cross-references, as well as settings for the representation of the target display at the interruption points.
- **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:** Global project setting defining between which representation types (multi-line, single-line overview, topology, etc.) device cross-references should be displayed. These settings can be configured for both directions.
- **Contact image on component:** Default setting for displaying cross-references at components, such as motor overload switches, circuit breakers, as well as the default setting for a form to be embedded for the contact images. The form type is the distributed device list *.f45.
- **Contact image in path:** Default setting for displaying, for example, cross-references of the unfolds (contact image) of contactor coils. Just like with the previous setting, here, too, a global form for the contact image can be configured instead of the default settings.

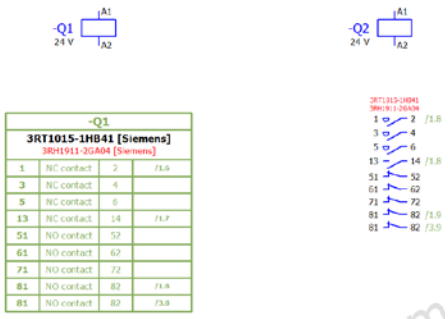


Fig. 2.21
Example: contact image with and without a form

2.3.1.6 Setting Projects [project name]/Connections

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

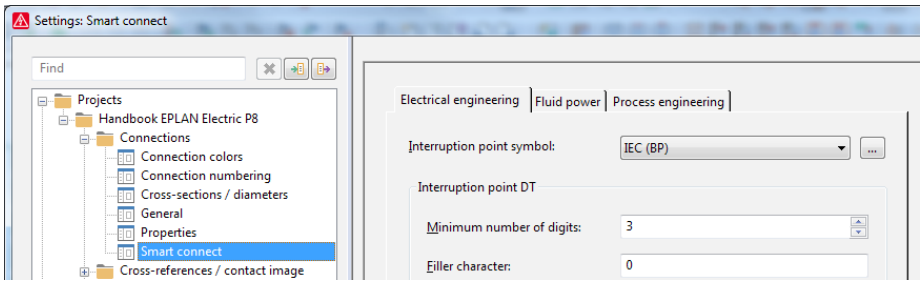


Fig. 2.22 Connection settings

- **General:** Some settings for determining the source and target
- **Properties:** Project-wide default settings for specific properties of a general connection, such as a global default value (e.g. 0.75), the unit (mm^2), or also a global default value for the color of a connection. These settings can be configured for the various trades (Electrical engineering, Hydraulics, Pneumatics, and many others).
- **Cross-sections/diameters:** Extendable table with a project's default settings for cross-sections and diameter values.
- **Smart connect:** Defaults for settings such as using a special interruption point symbol and for interruption point designations of 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 (scheme) connections and their information, such as source and target DT, conductor color and cross-section.

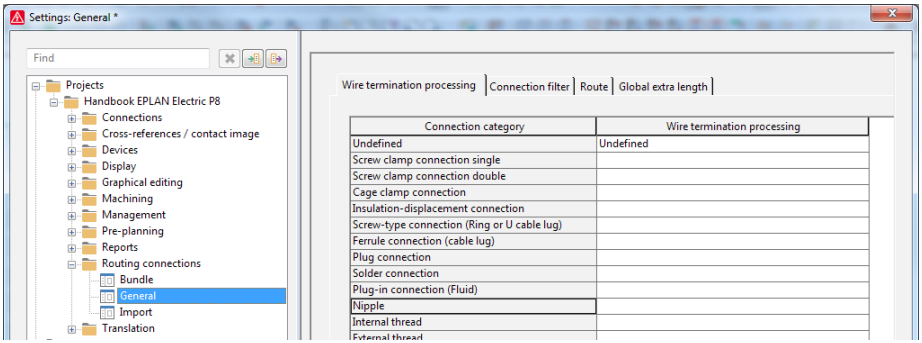


Fig. 2.23 Settings for routing connections

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.

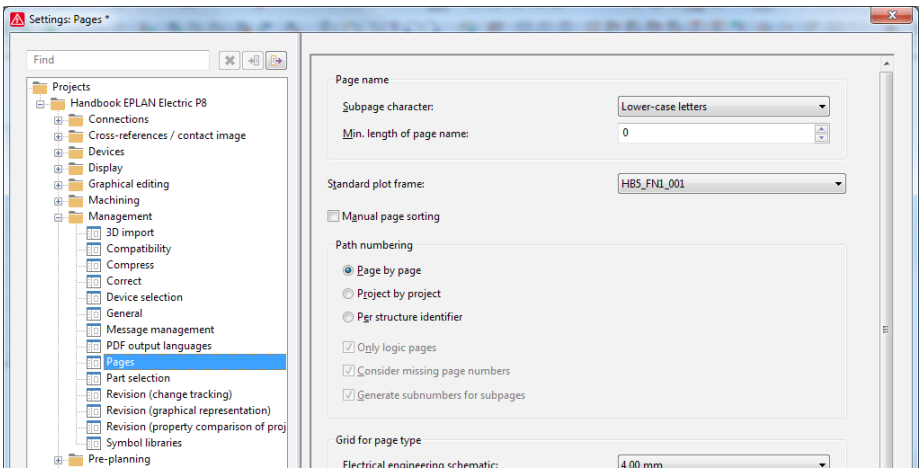


Fig. 2.24 Settings for managing the project

- **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
- **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.?
- **Compatibility:** Settings for the 'downward compatibility' of legacy projects and/or legacy functions. These settings should ideally no longer be used when editing a project.
- **Compress:** Default scheme used for compressing the project
- **Correct:** Default setting of a scheme to correct project data, such as terminals, plugs, cables and connections
- **Message management:** An important setting that defines the extent and type of error checking 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 error checking 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 check run. 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.

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.

- **General:** This is where you enter project language settings, settings defining the handling of texts when translating during data entry. Settings for displaying translation languages (displaying translations in the project) are also configured here. You also have to define a source language here.

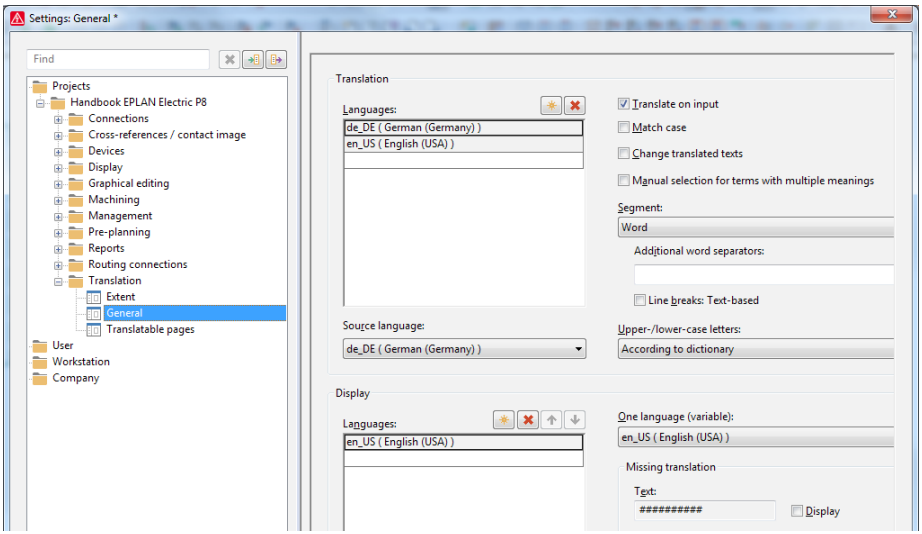


Fig. 2.25 Translation

- 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.

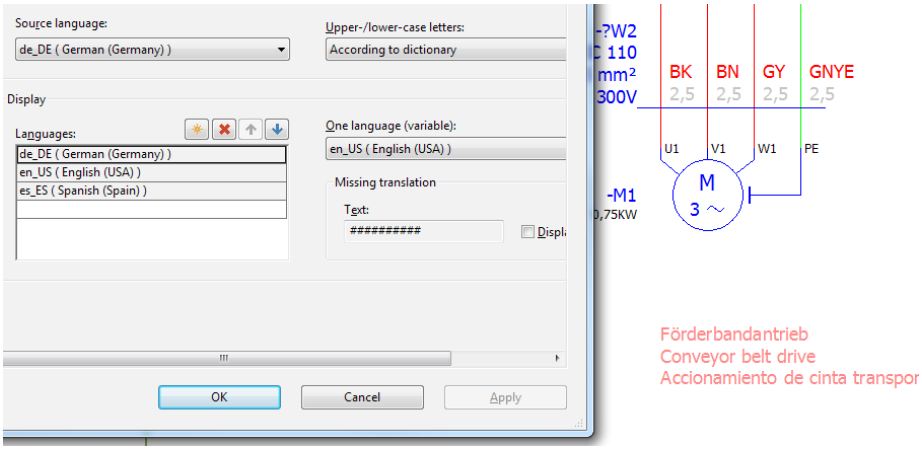


Fig. 2.26
Setting the translation sequence to “English, German, Spanish”

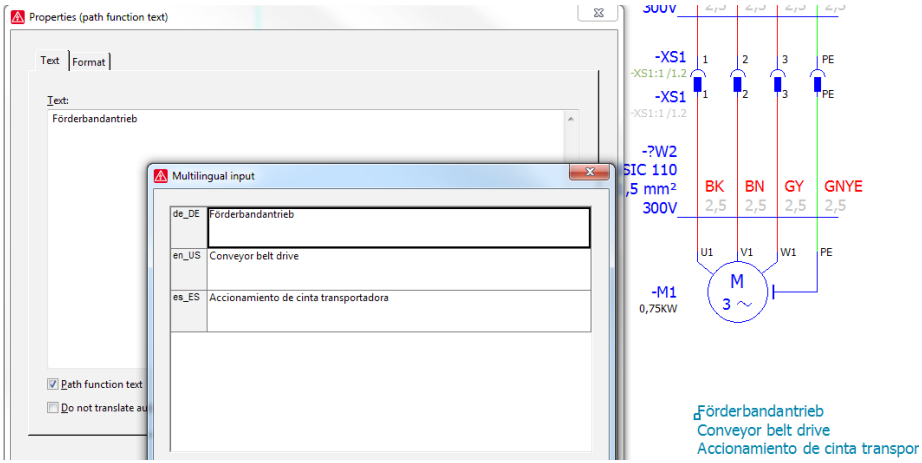


Fig. 2.27
Display in graphical editing

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

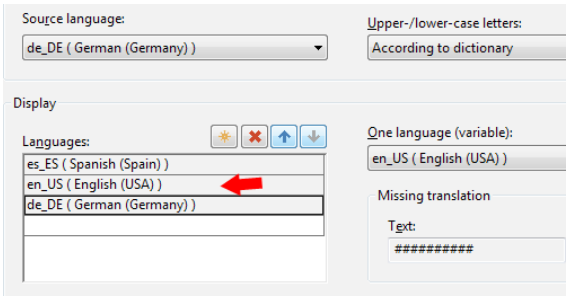


Fig. 2.28
Changing the translation sequence to “Spanish, English, German”

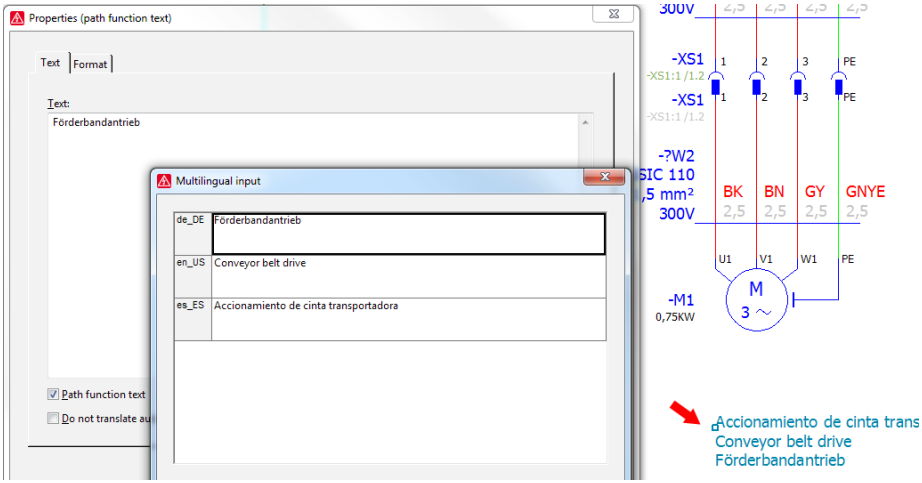


Fig. 2.29 Display in graphical editing

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.

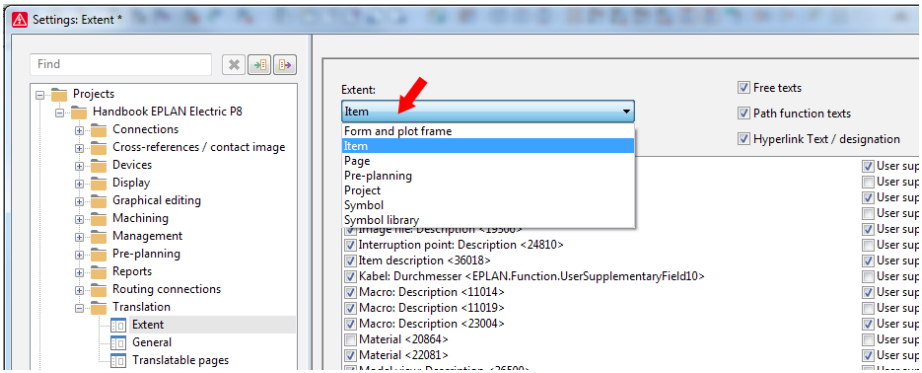


Fig. 2.30 Selection of settings for translation range

- **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.

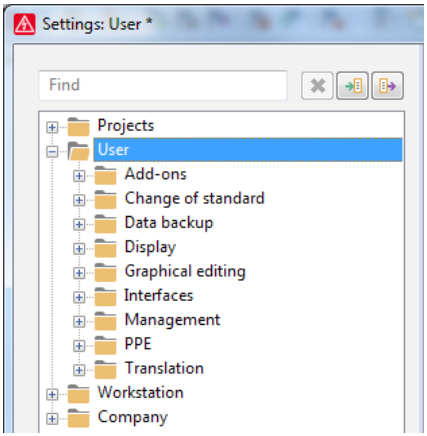


Fig. 2.31
User settings

The *User — Add-Ons* node contains settings for extension modules to be purchased separately. Since EPLAN has a fairly large selection of extension modules, which may vary for each license, this will not be discussed further at this point.

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.

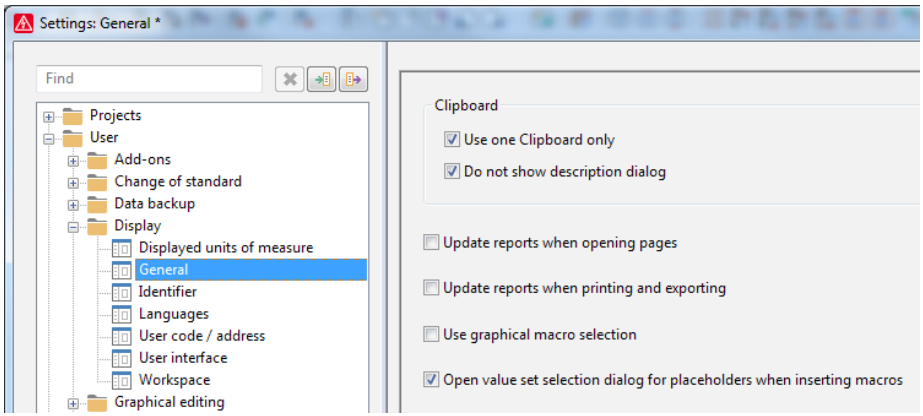


Fig. 2.32 General display settings

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 further settings, some of which are explained briefly below.

- **Update reports when opening pages:** If this setting 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.

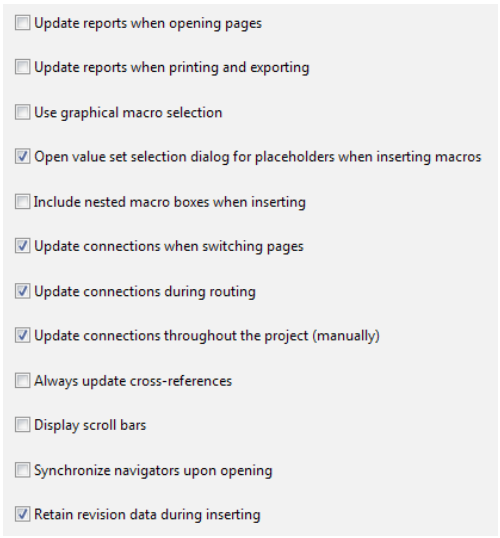


Fig. 2.33
Additional general settings

- **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 node also contains these 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.
- **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

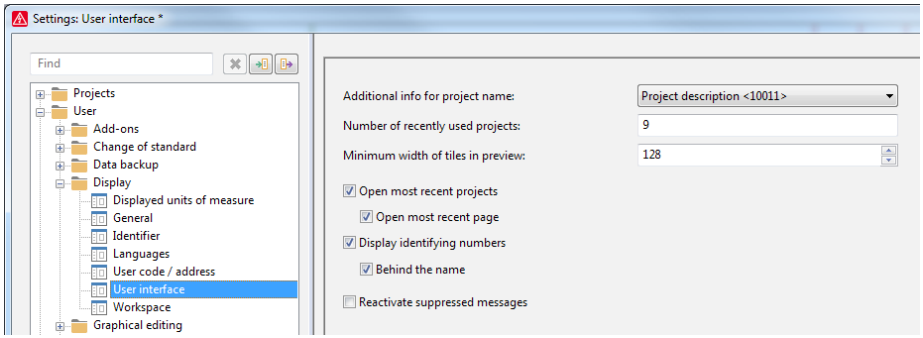


Fig. 2.34
Extended interface settings

- **User interface:** General settings, such as regarding project information and/or the display of preview information.

  Further user interface settings are possible here: Should the last projects be reopened when the program starts, or should the property number (identifying numbers) 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.
- **Languages:** Language setting of the EPLAN system and online help system setting. This is used, for example, to define whether the online help system is to be used locally or online (requires Internet connection).

2.3.2.2 Setting User — Data backup

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

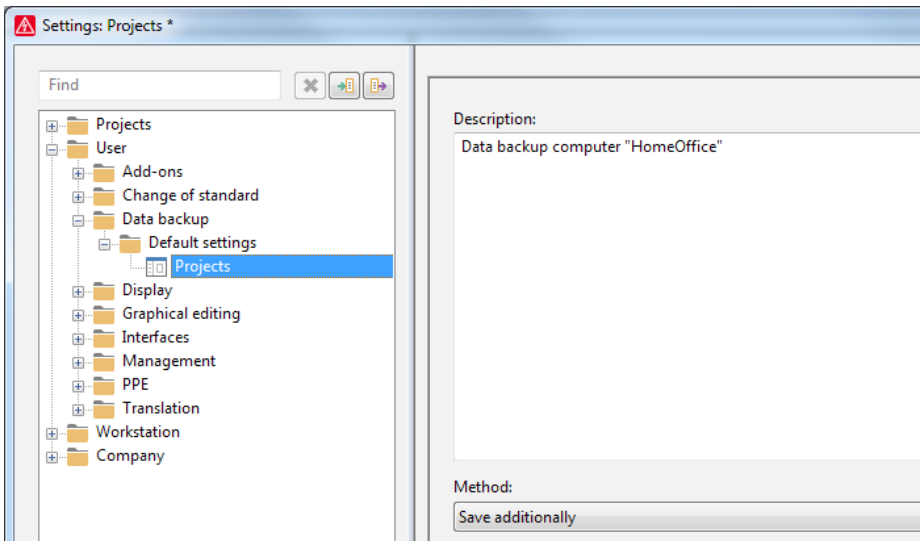


Fig. 2.35
Data backup settings

- **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 connection symbols (T-nodes, etc.).

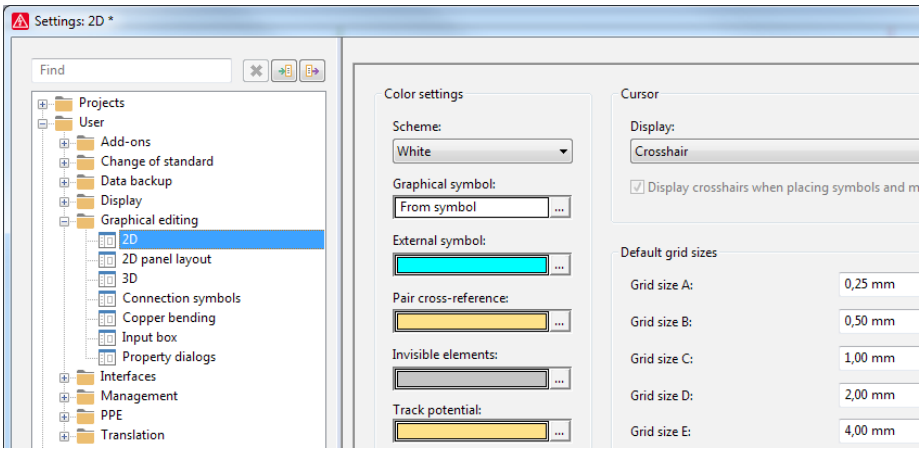


Fig. 2.36
Settings for graphical editing

- **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. The 4; 2; 1; 0.5 and 0.25 levels have proved particularly practical, especially when separate keyboard shortcuts are used for the grid settings.

- **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 of standard. There will not be any entries in this node either until a change of standard has occurred.

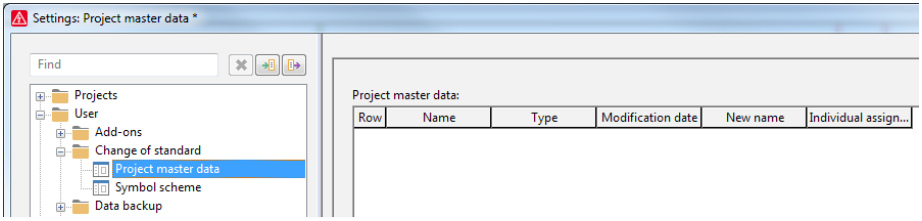


Fig. 2.37
Change of standard

- **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.

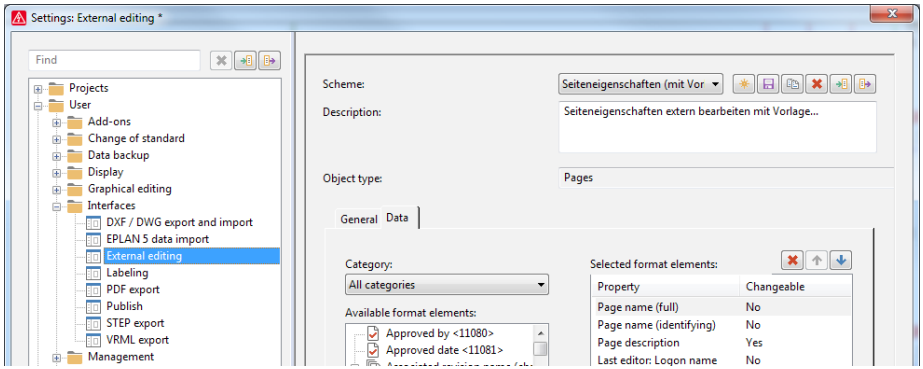


Fig. 2.38 Various interface settings

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

- **Labeling:** This is where you define default settings for the most frequently used schemes, in order to transfer data to the labeling system.
- **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.
- **Edit externally:** Definition of various schemes 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.
- **Publish:** Settings for EPLAN's own publication EPDZ format. This allows for EPLAN projects to be viewed using the EPLAN View app.

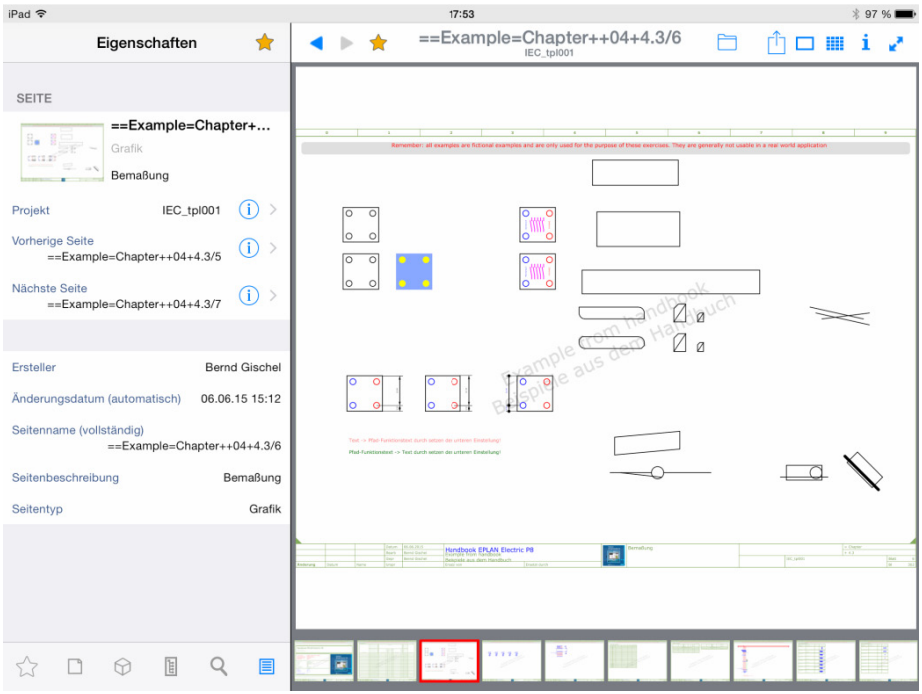


Fig. 2.39 Example: EPLAN project published in the EPLAN View app (iPad)

2.3.2.6 Setting User – Management

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

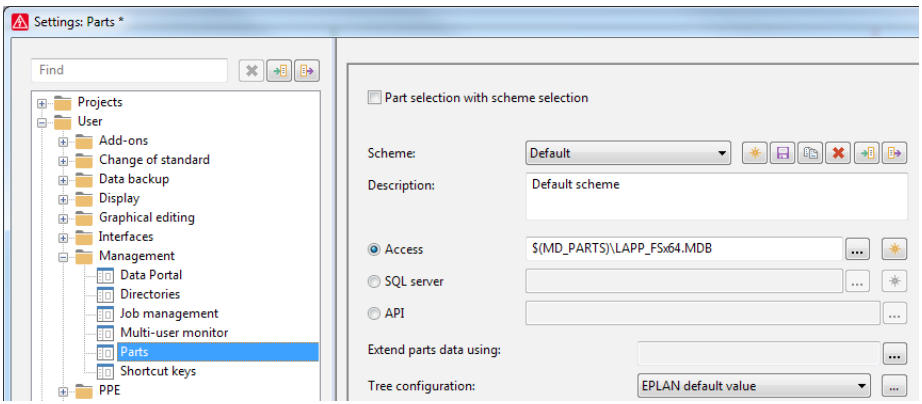


Fig. 2.40 Management settings

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

- **Data portal:** Settings for the user, the portal and the connection settings to the portal.
- **Shortcut keys:** 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.

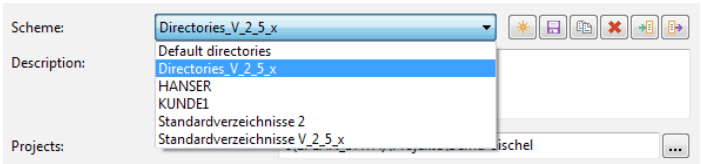


Fig. 2.41 Default directories

2.3.2.7 Setting User — Translation

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

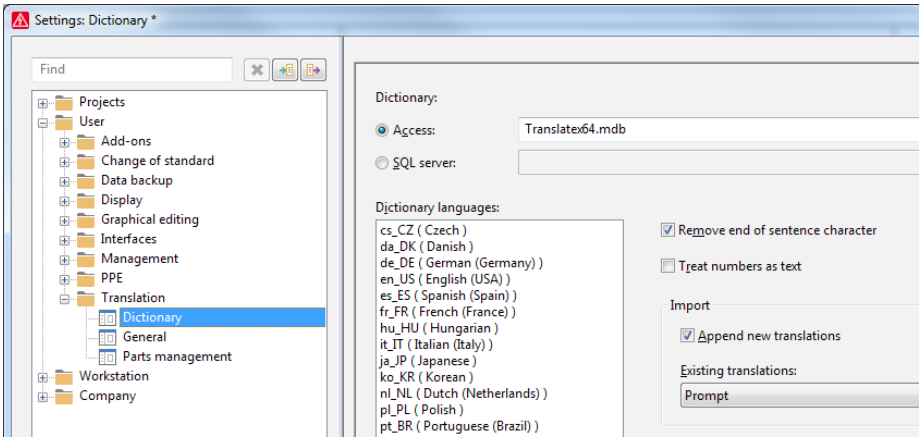


Fig. 2.42 Translations and foreign languages

- **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.
- **Parts management:** Definition of how describing part texts are to be translated in the translation run of parts management.
- **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.

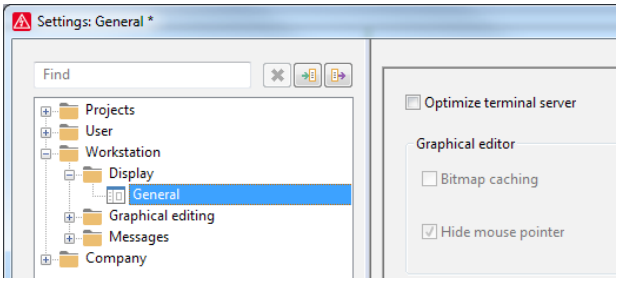


Fig. 2.43 Station settings

- **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.

- **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.

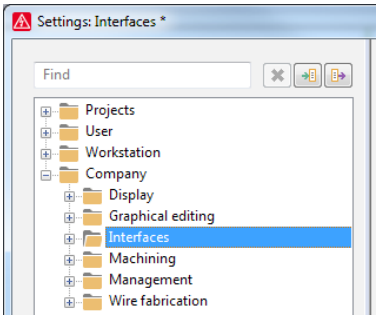


Fig. 2.44
Company settings

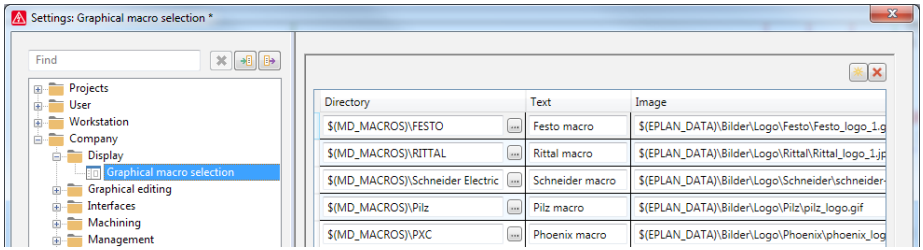


Fig. 2.45
Defining the settings for graphical macro selection

- 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.

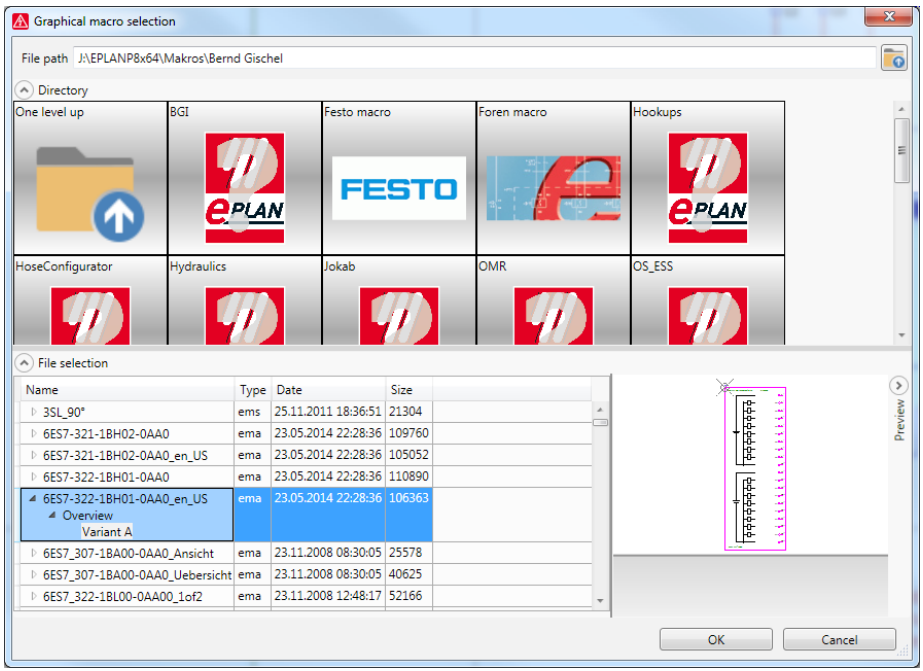


Fig. 2.46
Graphical macro selection dialog (started from the graphical editor)

- **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

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.

The different types of properties in EPLAN

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.

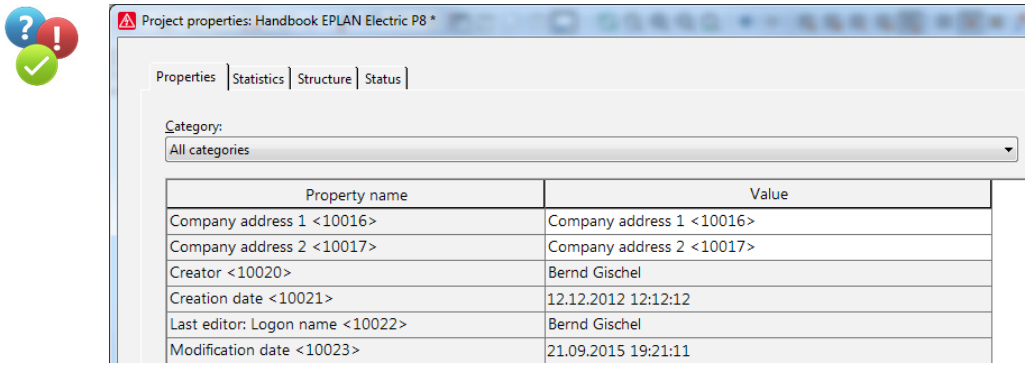


Fig. 2.47 Project properties

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

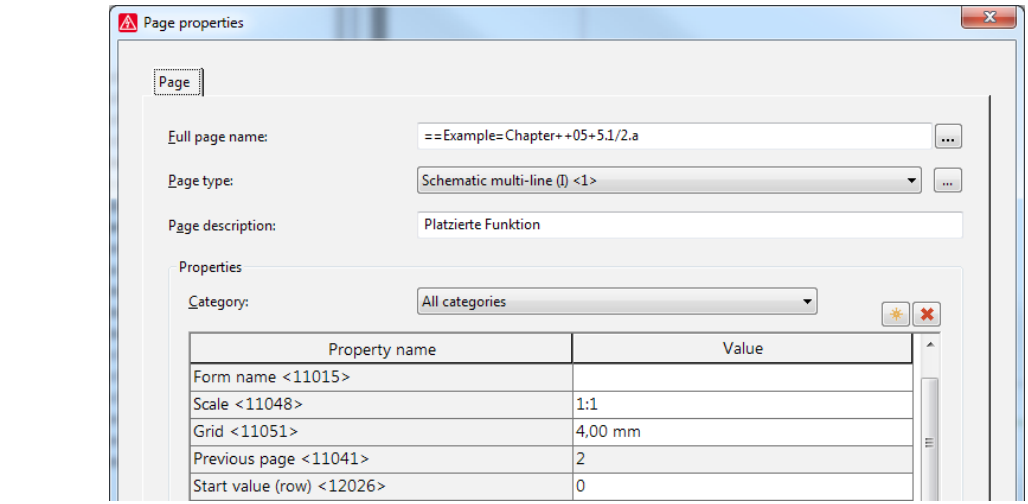


Fig. 2.48 Page properties

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.

As an addition to the above fixed, not extendable, properties, EPLAN has integrated the user-defined properties that can be edited and configured for your own requirements.

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.

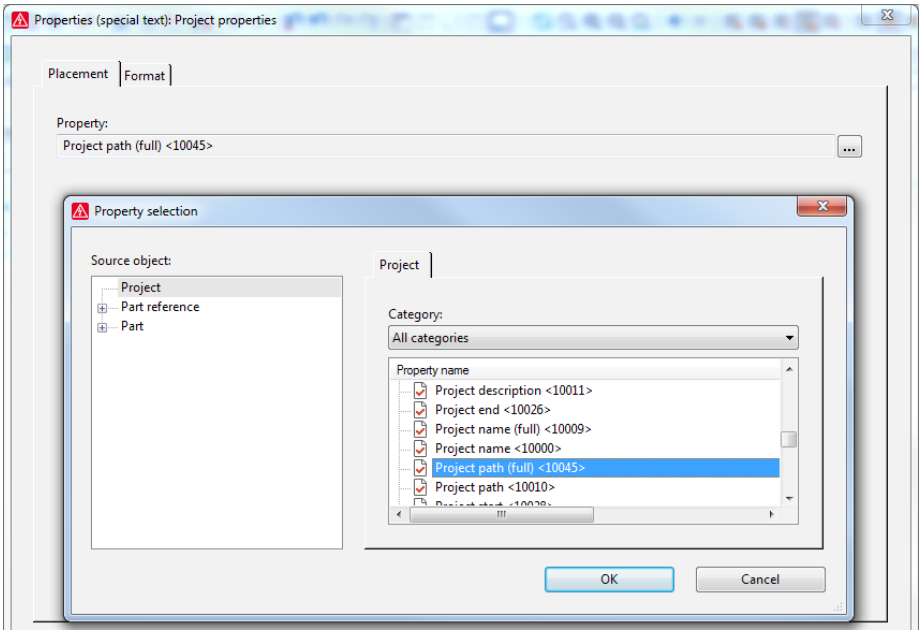


Fig. 2.49 Selecting and inserting a project property

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.

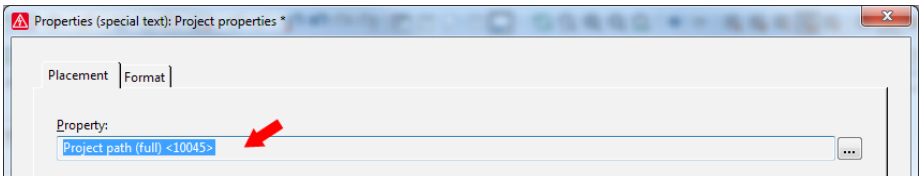


Fig. 2.50 Transferred project property

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.

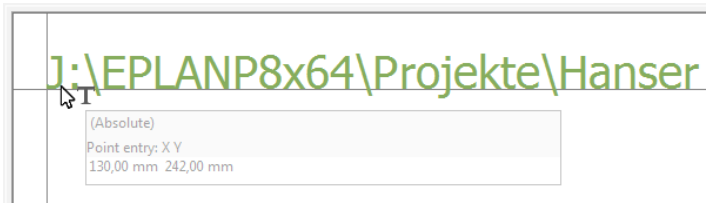


Fig. 2.51 Placing project properties

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.

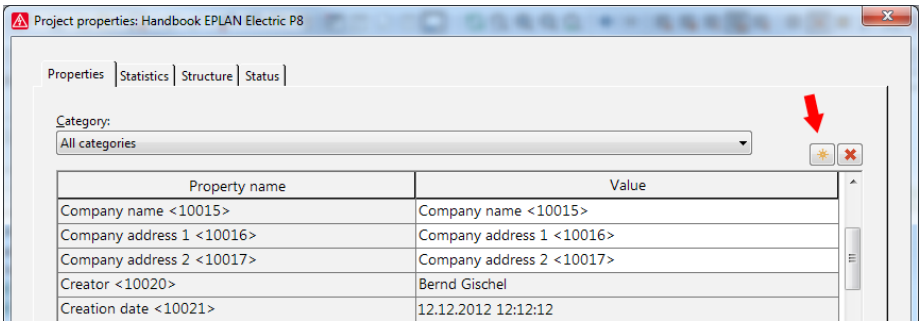


Fig. 2.52 Add project property

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.

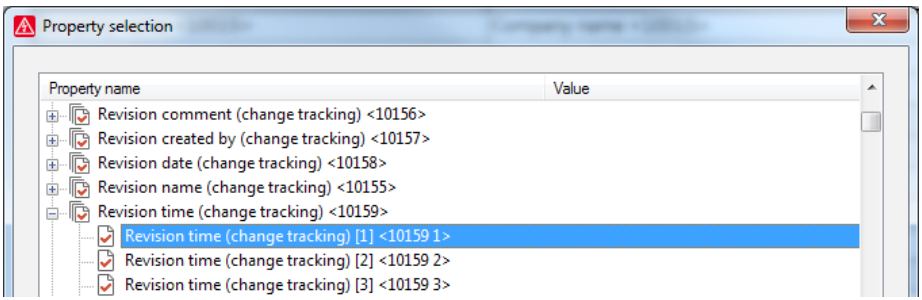


Fig. 2.53 Property selection

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

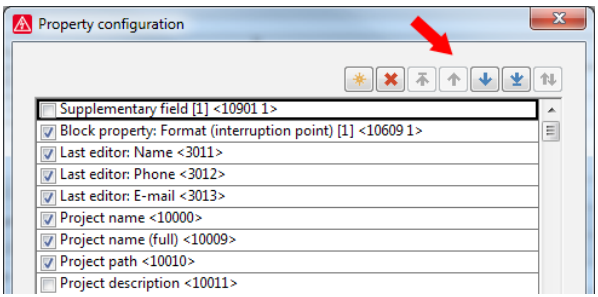


Fig. 2.54 Property configuration

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.

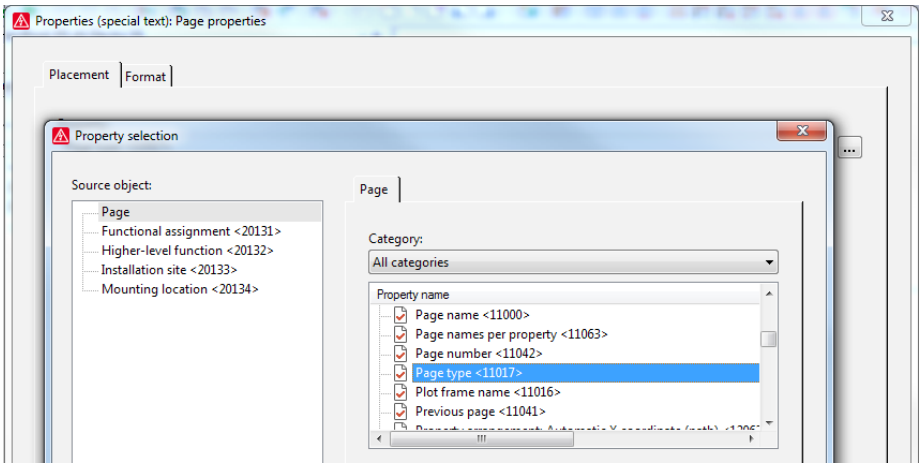


Fig. 2.55 Add page properties

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

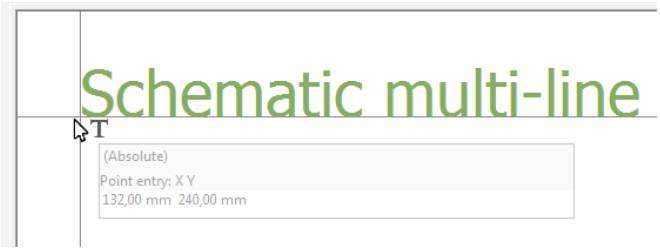


Fig. 2.56
Placing a page property

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.

2.5.3 User-defined properties

User-defined properties are properties that the user can create and edit freely. User-defined properties are always created in a project-specific manner, but can be easily integrated with or added to other projects by means of EPLAN's integrated import and export functions. User-defined properties are assigned to specific areas (project, page, etc.). Using default selection lists, self-extending selection lists or free entries, they can be created and subsequently selected easily.

User-defined properties are edited using the **CONFIGURE OPTIONS/PROPERTIES** menu.

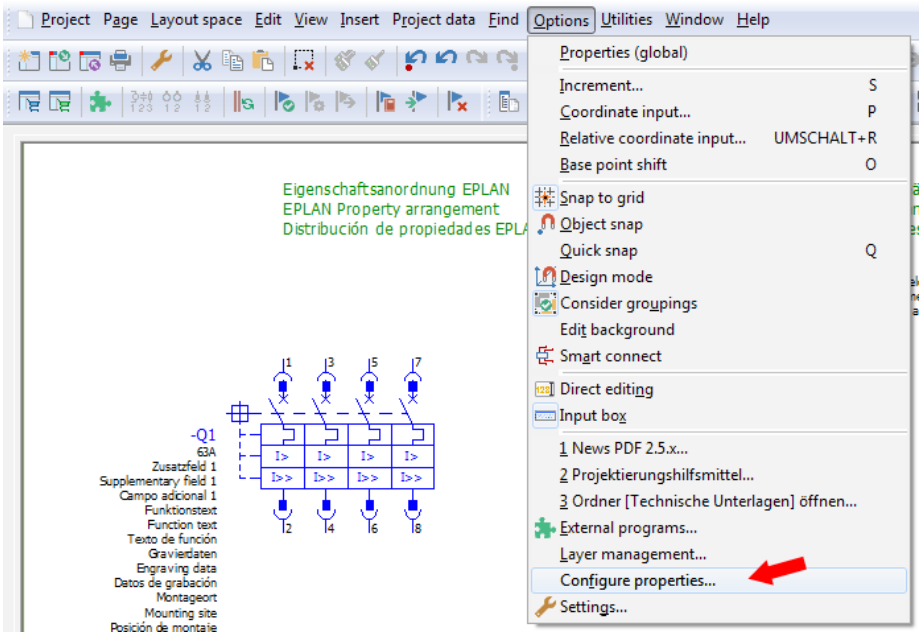


Fig. 2.57
Menu call: configuring (user-defined) properties

Afterwards, EPLAN opens the **Configure properties** dialog. If there are no user-defined properties yet, the left area of the dialog will remain empty.

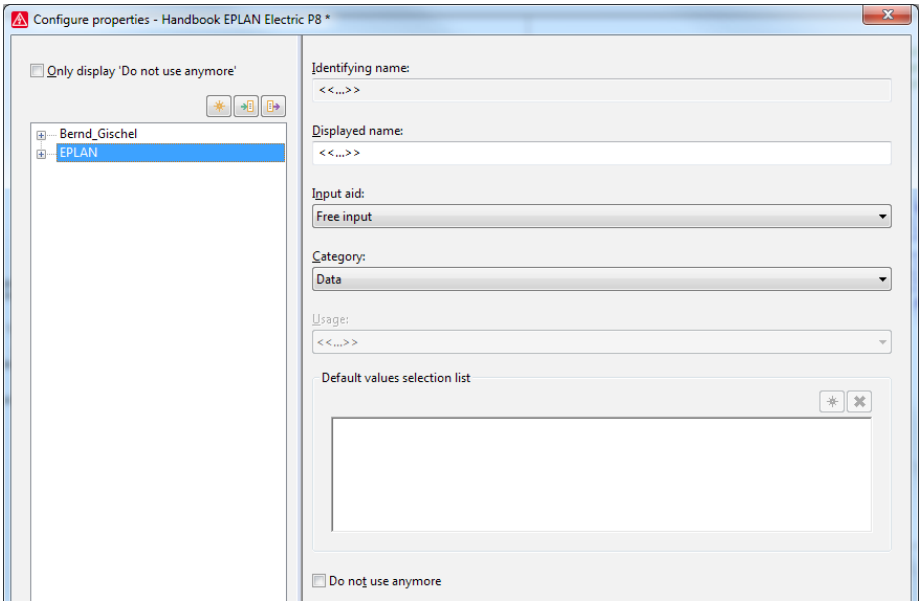


Fig. 2.58 Dialog for configuring properties

Click the **NEW** button (sun symbol) in the left area to create a new user-defined property and configure it for subsequent use. EPLAN opens the **Generate user-defined property** dialog.

Now, the new property can be specified. Depending on the input, EPLAN differentiates several levels. Each new level must be separated by a period.

In the example, the identifying name is thus *Level1*, followed by Level2, then Level3, etc.

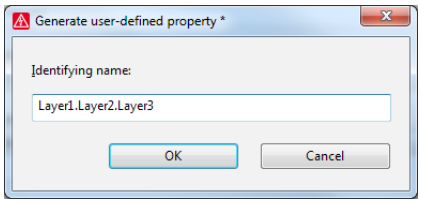


Fig. 2.59 Dialog Generate user-defined property

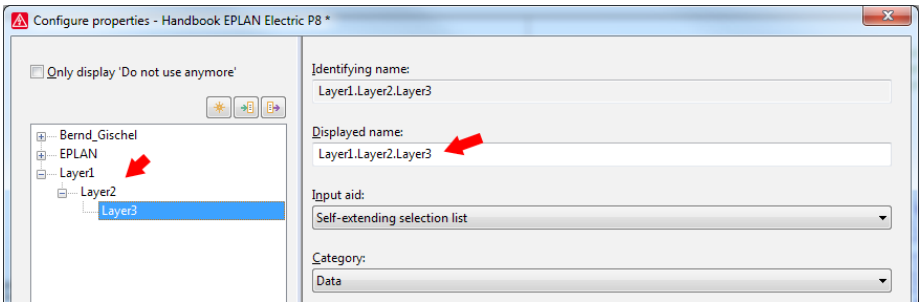


Fig. 2.60 Structure of the identifying name

This is how a structure can be established. Similar user-defined properties are sorted into the structure correctly.

The user-defined property can now be configured further. The following settings are possible:

- **Displayed name:** This can be edited freely and allows for multiple languages
- **Input aid:** Selection of one of the three options (free entry, self-extending selection list, selection list)
- **Category:** Selection of a category (such as devices, data, etc.) to be assigned to the user-defined property
- **Usage:** Selection of objects (project, page, etc.) to which the user-defined property is to be apply

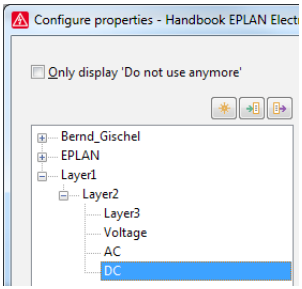


Fig. 2.61
Structure

NOTE: Once the user-defined property has been saved, the Usage selection can no longer be changed.

For example, a selection list (5V DC, 12V DC, etc.) with values can be specified for a user-defined property (control voltage DC) for the Data category and Project (Usage) object to limit the freedom in creating the project.

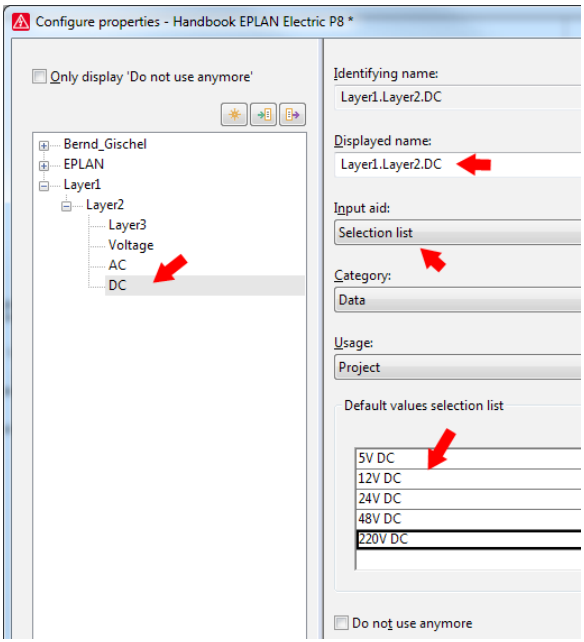


Fig. 2.62 Default values from a selection list

The user-defined property is added via the **NEW** button, as is common EPLAN practice.

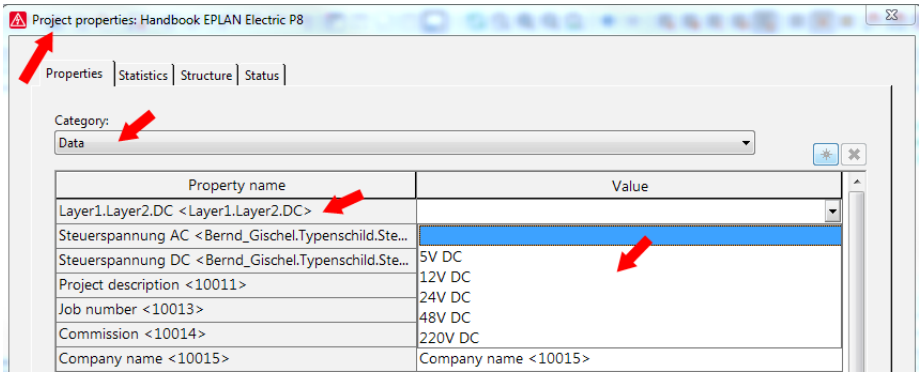


Fig. 2.63 Selection default value

2.5.3.1 Remove user-defined properties

To remove a user-defined property from a project: in the **Configure properties** dialog (**CONFIGURE OPTIONS/PROPERTIES** menu), select the *Do not use anymore* checkbox for this property.

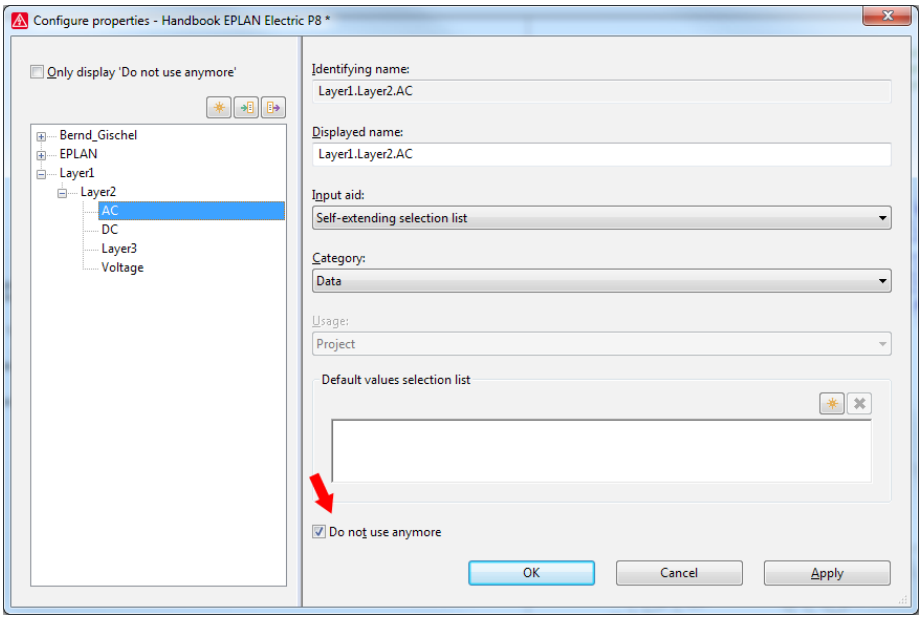


Fig. 2.64 Checkbox Do not use anymore

Entire nodes can be selected as well before selecting the *Do not use anymore* checkbox.

These selected user-defined properties can be removed from the project definitively with one compression run.

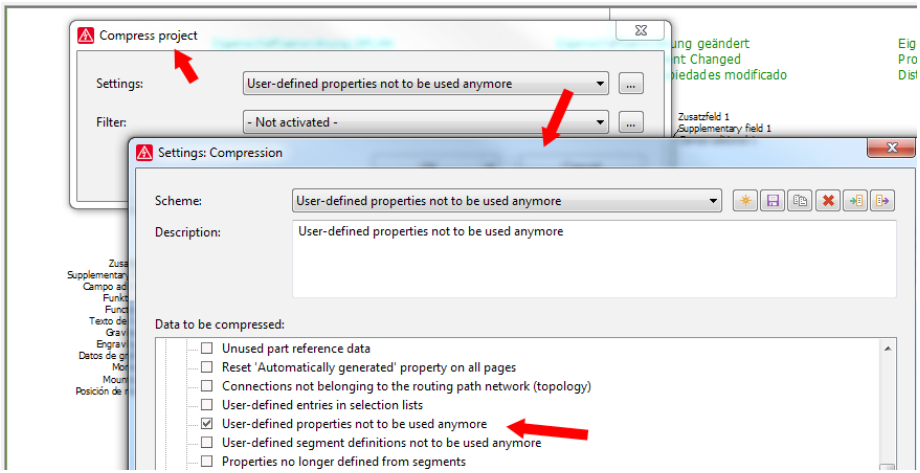


Fig. 2.65 Compression run

2.5.4 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.

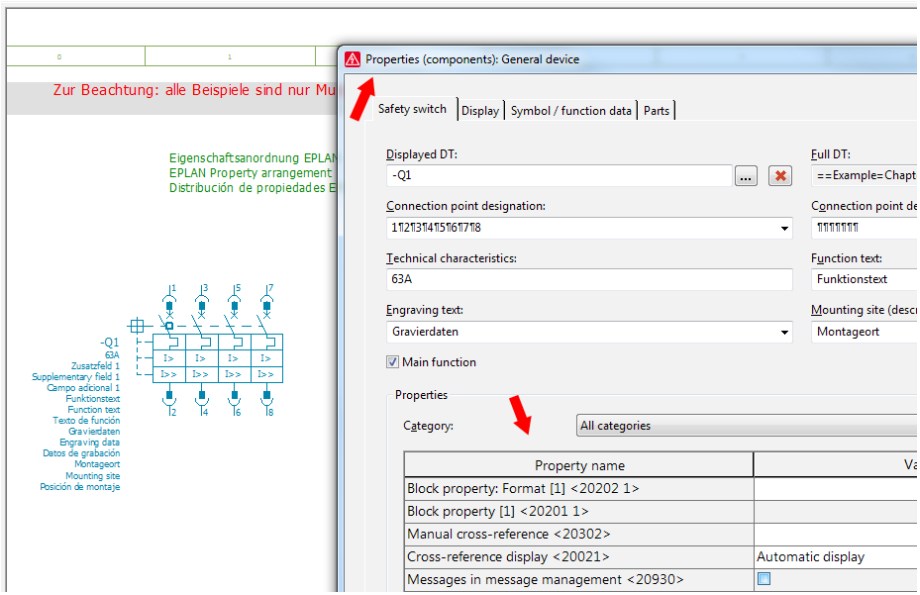


Fig. 2.66 Properties of a symbol (component)

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. You call up the popup menu using the right mouse 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.

2.5.5 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 be edited only in the **Form editor**.

2.5.6 Plot frame properties

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

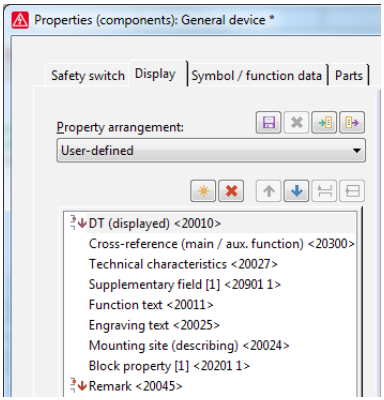


Fig. 2.67
Various buttons

	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.

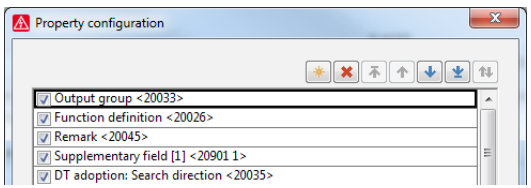


Fig. 2.68
Buttons in the property configuration dialog

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

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

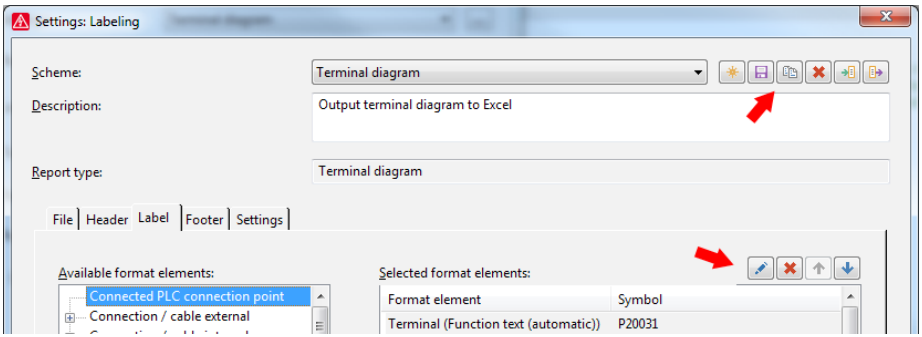


Fig. 2.69 Buttons in various settings dialogs

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

	Edit entry
	Copy entry

2.6.4 Restore default function

Depending on the dialog, you may find the *Restore default* function on the right mouse button. To call this function, click the corresponding field (such as Path selection) and then right-click. 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.

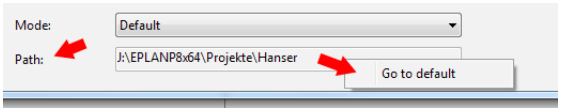


Fig. 2.70 Restore default function

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.

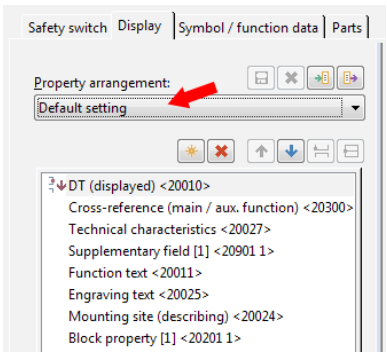


Fig. 2.71
Default setting

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

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.

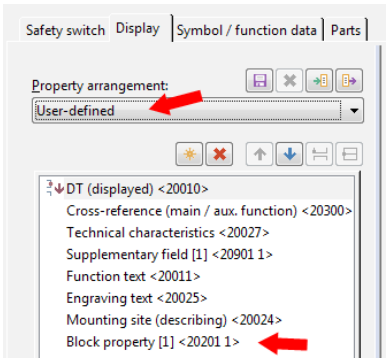


Fig. 2.72
User-defined arrangement

If a property has been changed, then EPLAN no longer regards it as a default value and the display in the *Property arrangement* selection field changes to *User-defined*. This user-defined arrangement can also be saved via the **SAVE** button and then called up or set at similar symbols (functions). 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.

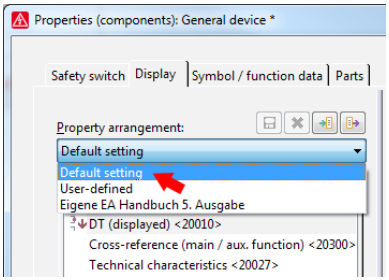


Fig. 2.73 Undo

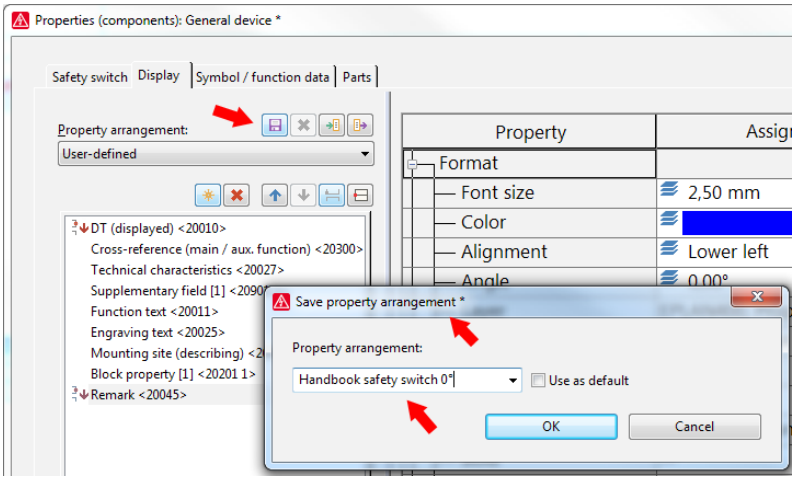


Fig. 2.74 Saving your own property arrangement

Property arrangements can only be transferred to the same "types" of devices (same function template), 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.

The property arrangements can also be imported or exported in order to continue using them in other projects. Use the **PROJECT/ORGANIZE/PROPERTY ARRANGEMENTS** menu for import/export.

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.

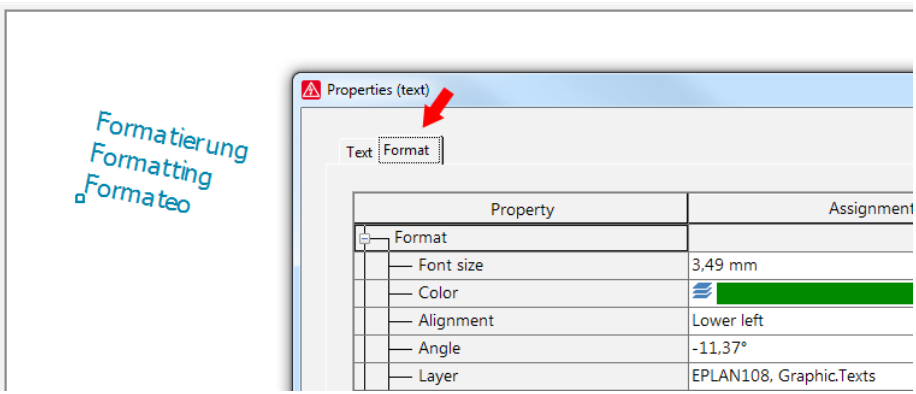


Fig. 2.75
Formatting options for texts (example)

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.

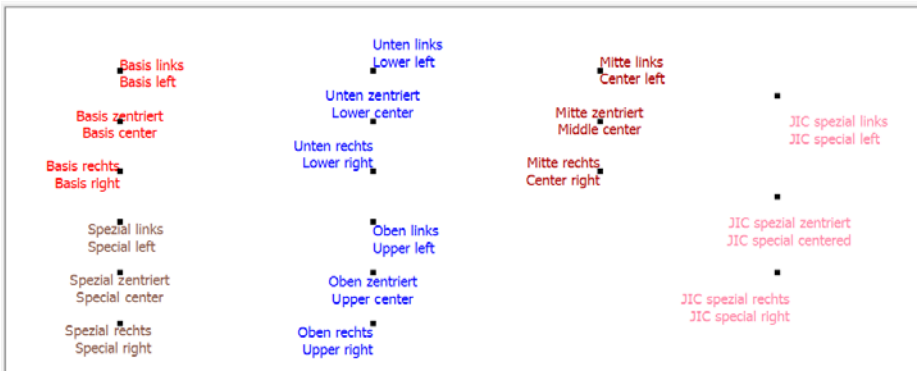


Fig. 2.76
Examples of different formats

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).

Additional representation in dialogs, including the option of changing formatting (font size, angle, etc.):

This symbol at the start of a line (in the *Value* column) means that the value entered is the original value from the corresponding level and has not been modified. If the value is changed, this symbol will disappear.

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.

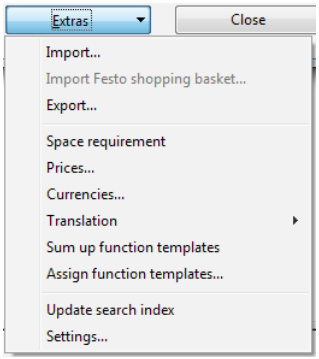


Fig. 2.77 Parts management

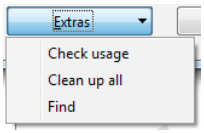


Fig. 2.78 Structure identifier management

2.6.8 Dialogs for schemes

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

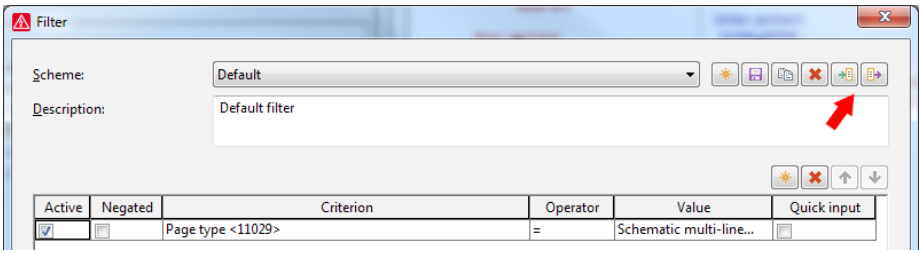


Fig. 2.79 Import and export options for schemes

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*	Example: Scheme prefix for cable numbering
LB..xml*	Example: Scheme prefix for labeling
WS..xml*	Example: 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.

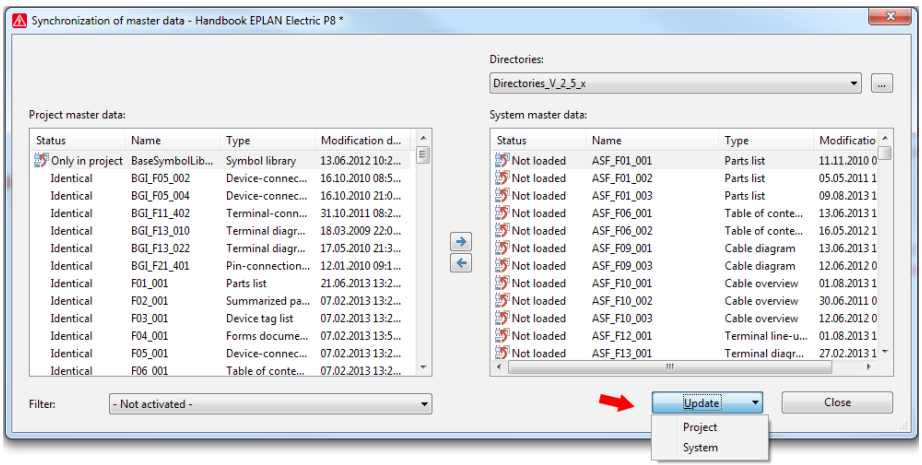


Fig. 2.80 Master data synchronization

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.

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.

Selected shortcuts

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**.

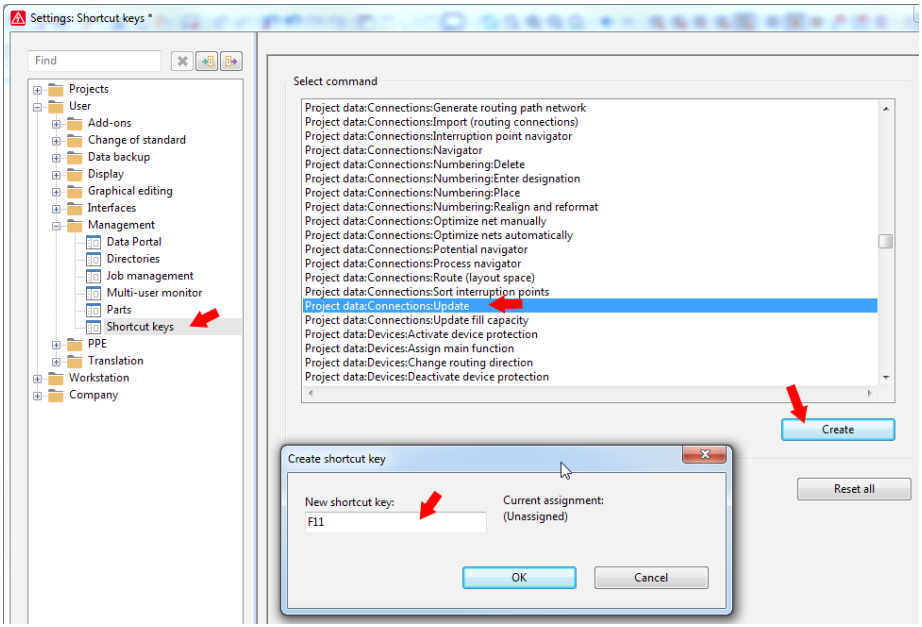


Fig. 2.81 Assigning a shortcut key to a function

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**.

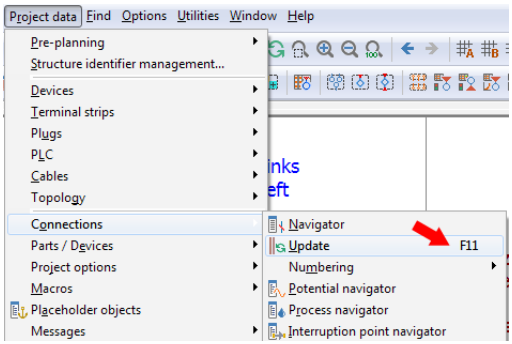


Fig. 2.82
Assigned shortcuts in the Connections menu

2.8.2 Using the mouse

EPLAN can be fully operated 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 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 not yet assigned in the *Name* field and a sensible description for the workspace in the *Description* field.

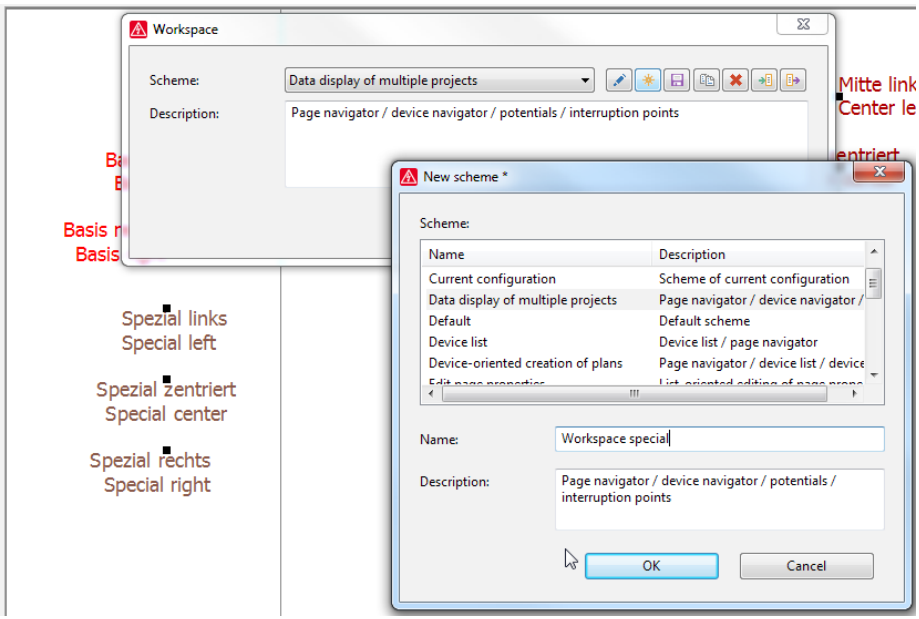


Fig. 2.83 Creating a workspace

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.

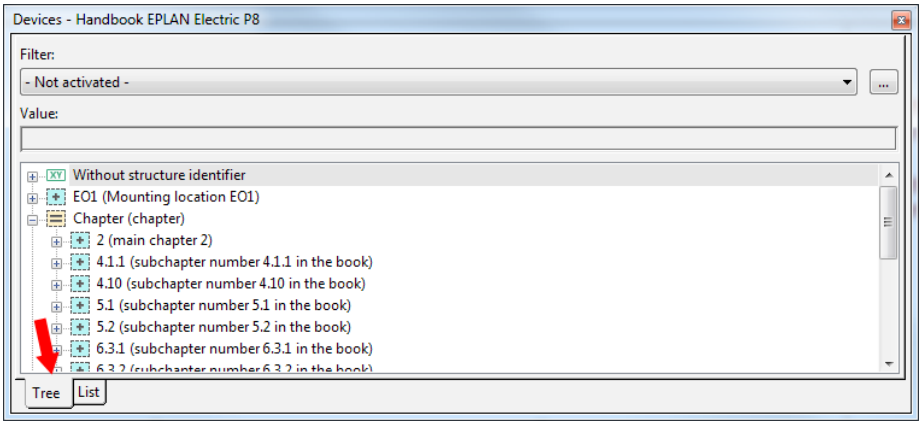


Fig. 2.84
Tree view

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.

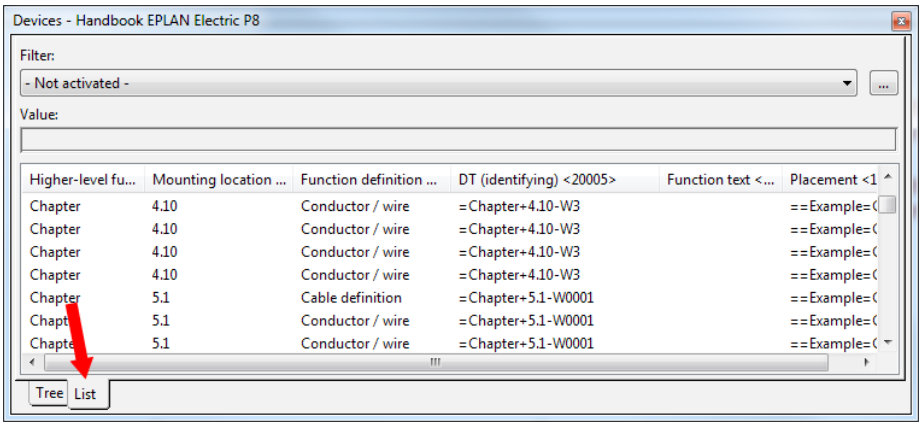


Fig. 2.85
List view

Unlike the tree view, it is possible to add further information to the list view. Right-click to call up the popup menu to select the *Configure representation* function. EPLAN opens the **Configure representation** 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.

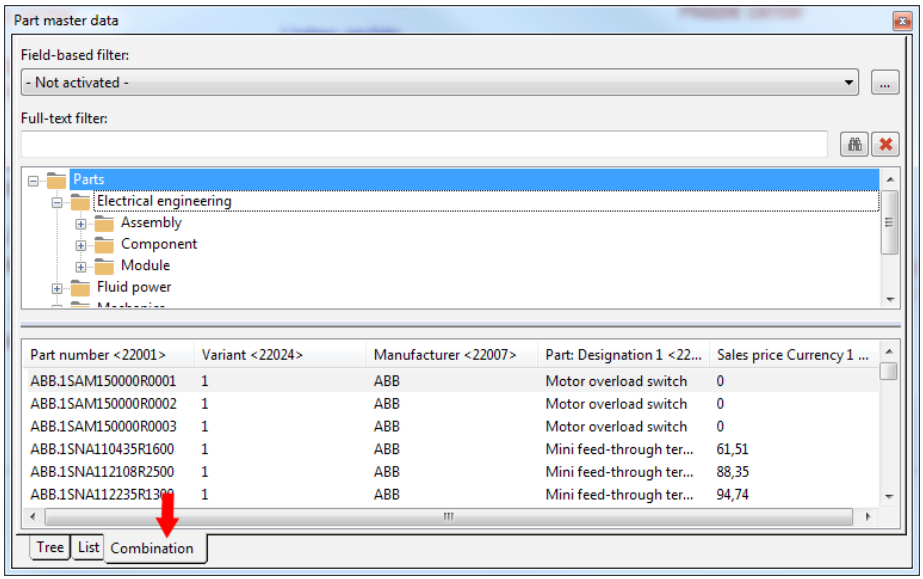


Fig. 2.86 Combined display

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.

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

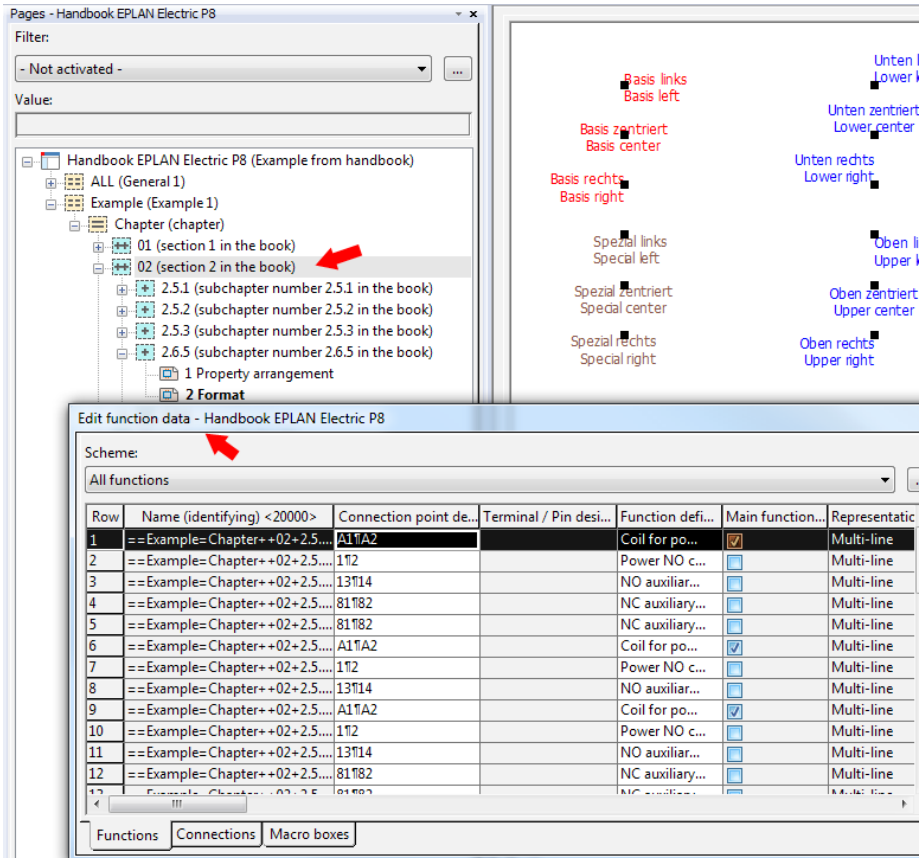


Fig. 2.87 Editing function data in table

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 locationof 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, during the time a project is edited, also all the master data, such as forms, parts data, function definitions, symbol libraries, etc., that is necessary for editing projects.

Global structure of a project

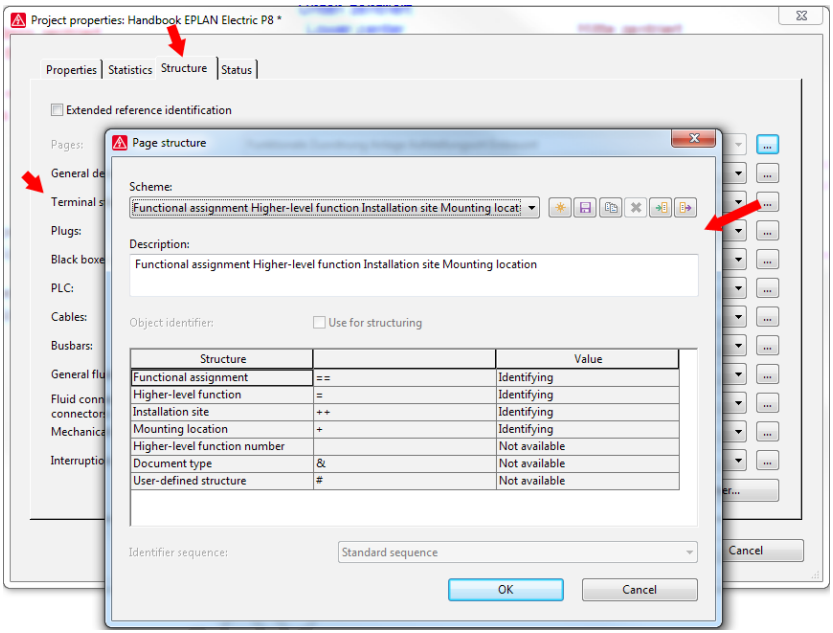


Fig. 3.1
Project properties – Structure tab

This data is all **completely** stored in the project. Data added later, such as newly used 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, using the macro navigator, 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*.

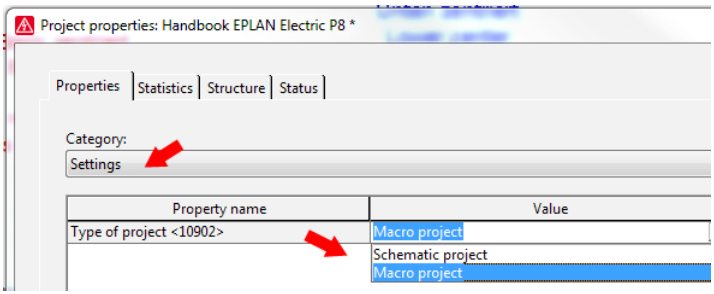


Fig. 3.2
Project properties – Properties tab

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 (always 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 with the project master data.

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 automatically anymore.

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

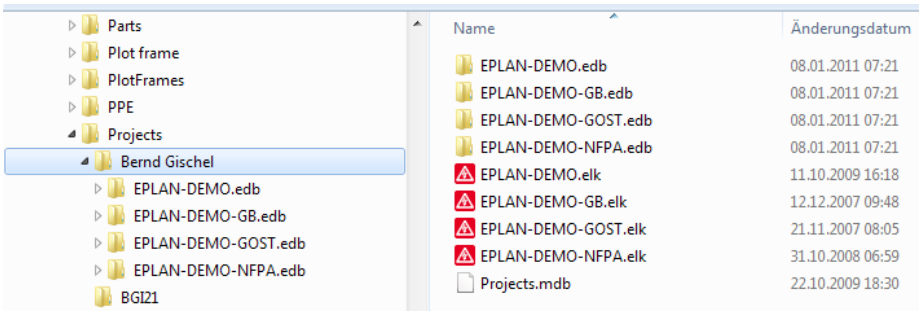


Fig. 3.3
Project directory with projects

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 directly.

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.

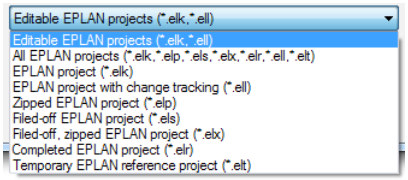


Fig. 3.4
Open project dialog with various file types

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* (***.zw9**) and *project templates* (***.ept**).

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

Opening a project template

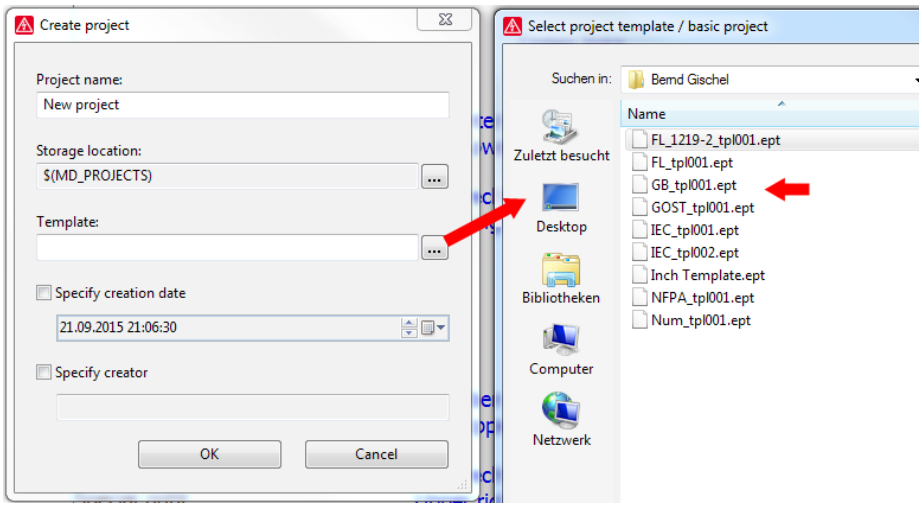


Fig. 3.5
*.ept project template

In contrast to a basic project, a new project created from a project template (via **PROJECT – NEW**) 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 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.

NOTE: Project templates with the *.epb file extension are old versions of project templates, which can no longer be generated. But they can continue to be used.

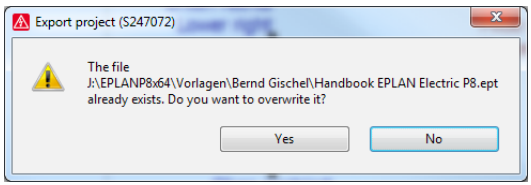


Fig. 3.6
Overwriting a project template

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.

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.

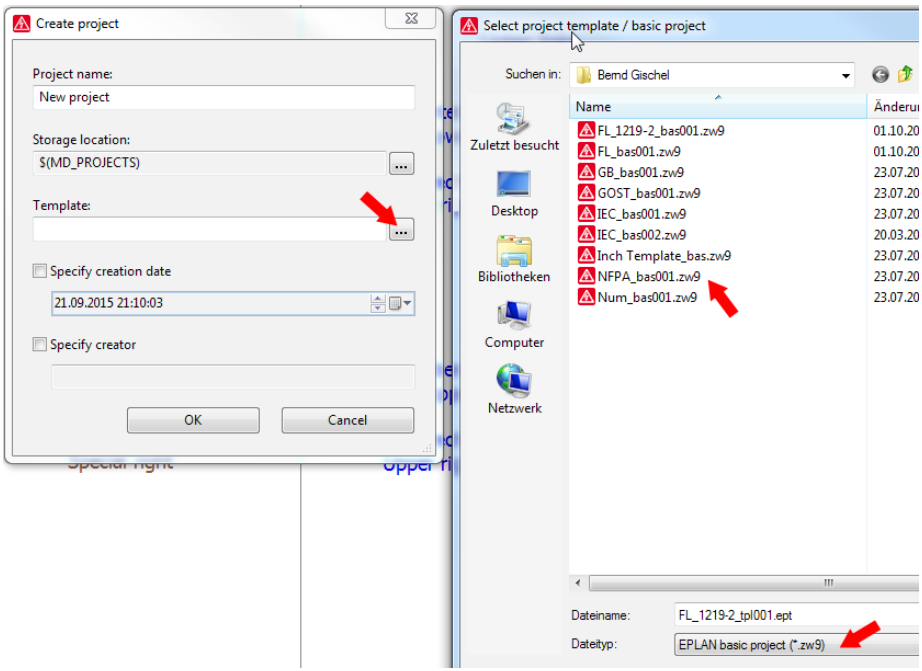


Fig. 3.7 Opening a basic project

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, by specifying the same file name.

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, while the supplied project template *Num_tpl001.ept* contains a sequential numbering structure.

Another 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.

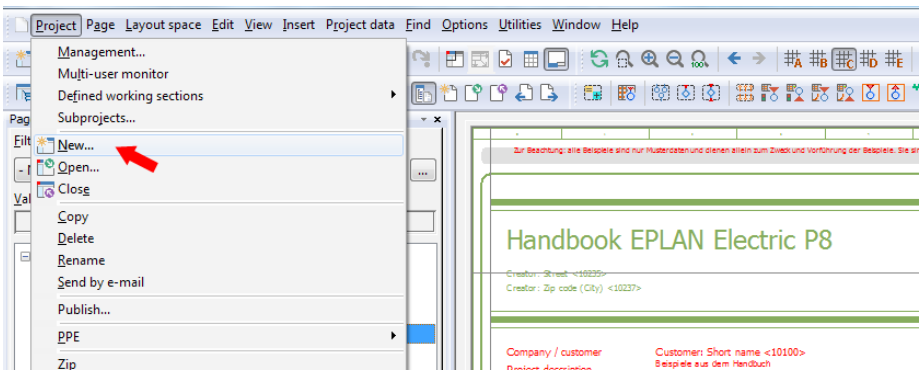


Fig. 3.8
Project/New menu

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

- **Project name** input field: The new project name (you need not worry about file extensions; these will be automatically added by EPLAN) 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.

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

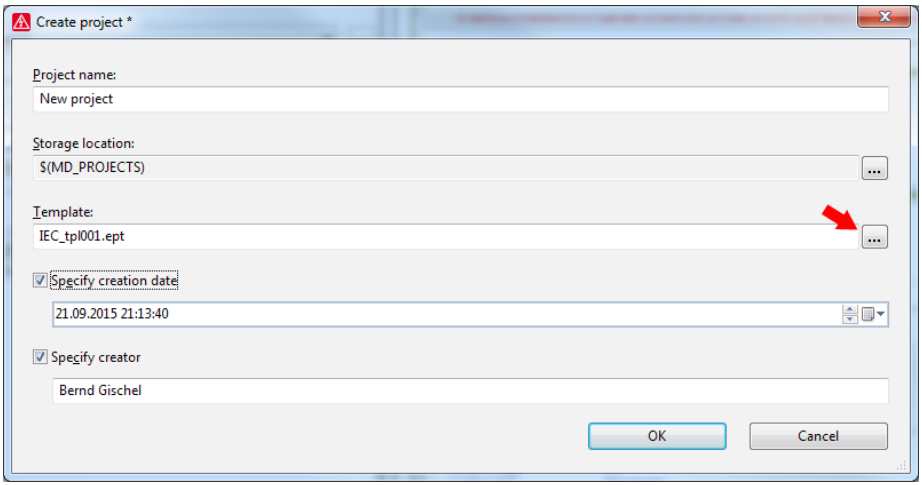


Fig. 3.9 Create project dialog

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.

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. In this example, the **Select project template/basic project** dialog opens, and 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.

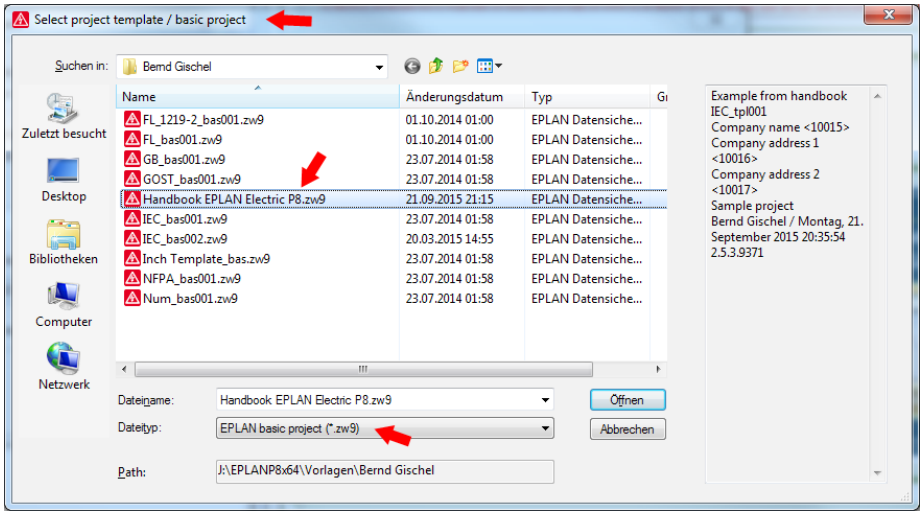


Fig. 3.10 Select project template / basic project dialog

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.

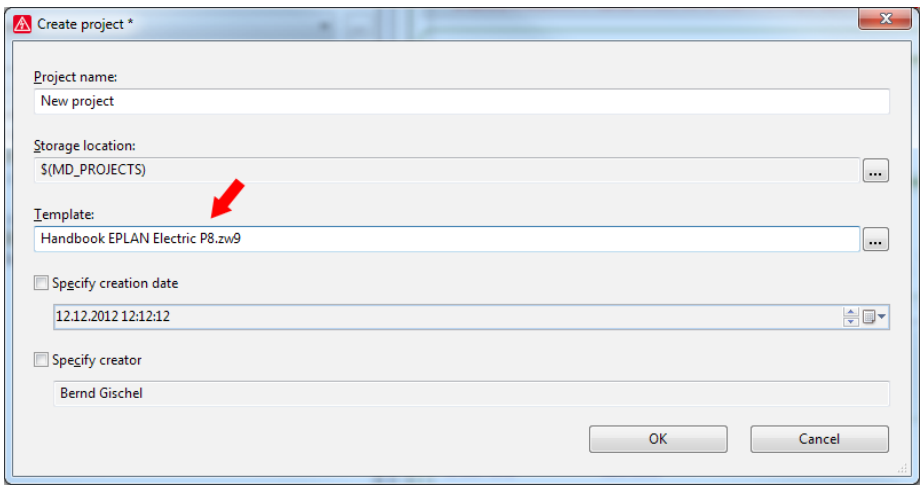


Fig. 3.11 Create project dialog, with the template

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).

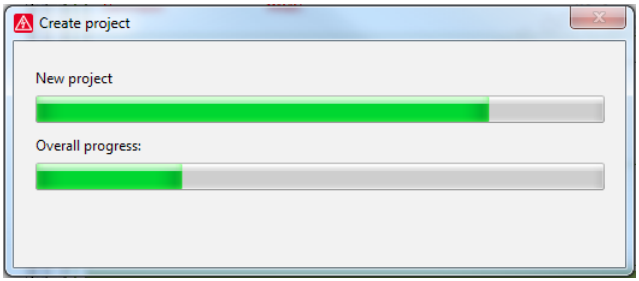


Fig. 3.12 The new project is being generated

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.

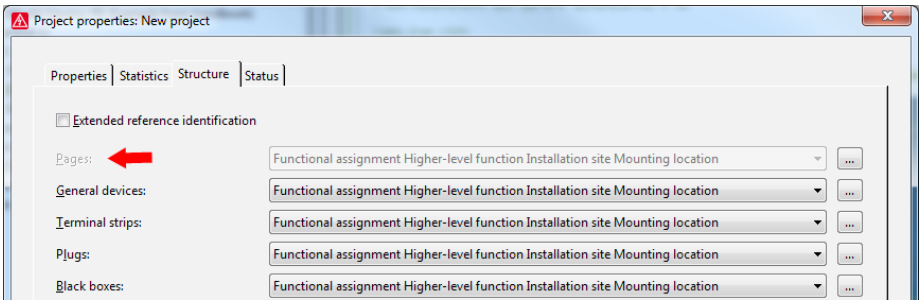


Fig. 3.13 Project properties – Structure tab

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.

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 for graphical project editing. 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 function templates, a project also contains individual page types, e. g. schematics, panel layout drawings or reports, such as terminal diagrams or connection lists, that can be divided into particular identifiers (structures) such as higher-level functions or mounting locations and installation sites.

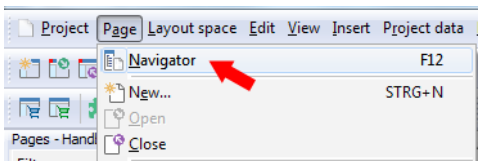


Fig. 4.1
Opening the page navigator

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 or via the F12 key. As is usual in EPLAN, the **page navigator** offers a choice between tree structure or list view.

Page navigator Tree/list

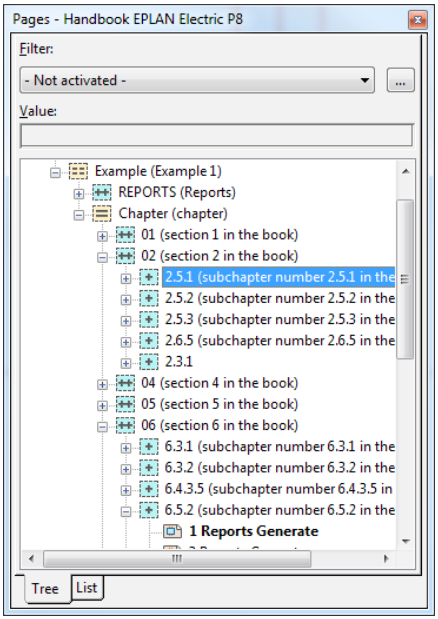


Fig. 4.2 Tree structure representation

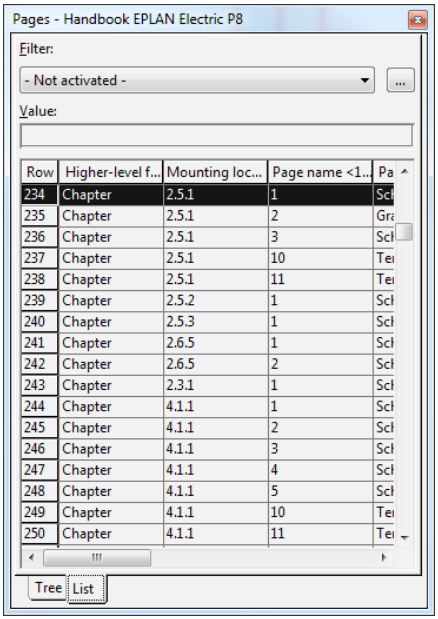


Fig. 4.3 List representation

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.

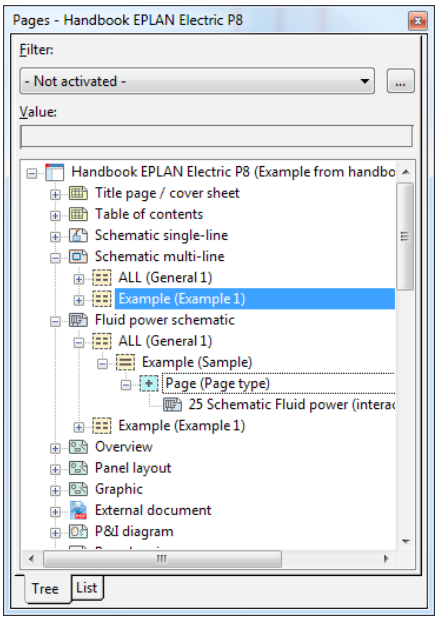


Fig. 4.4 Structured by page type

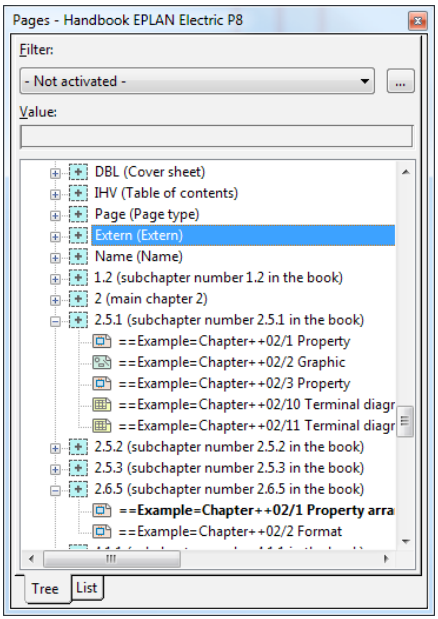


Fig. 4.5 Structured by mounting location

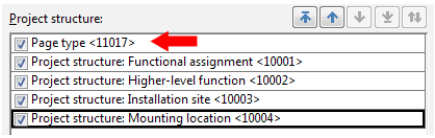


Fig. 4.6 Project setting by page type

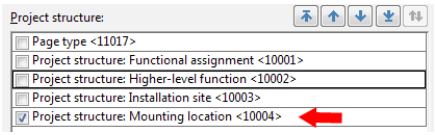


Fig. 4.7 Project setting by mounting location

This only affects the visual display of the pages in the **page navigator**. The sorting of the identifiers (higher-level function, mounting location, installation site, etc.) is defined as usual in *structure identifier management*, i. e. whether (e. g.) higher-level function = A11 is sorted before or after higher-level function = A32.

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.

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.

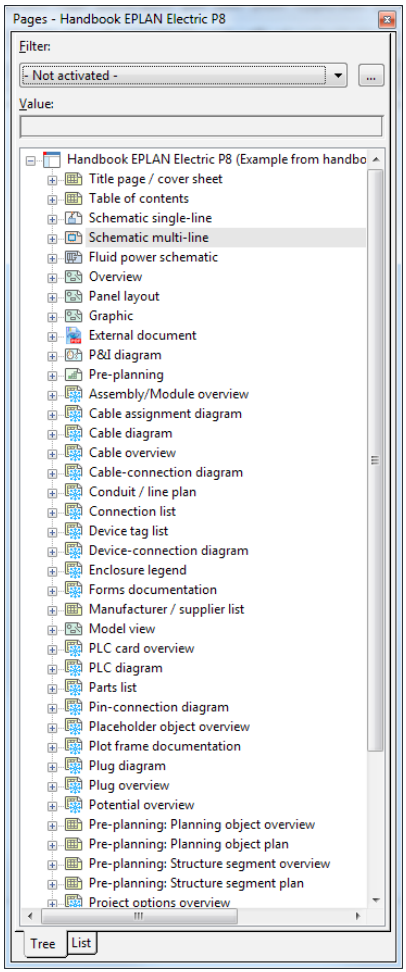


Fig. 4.8
Page types

Icon	Page type	Possible use	Interactive/ automatic
	Schematic multi-line	Multi-line schematics	Editing possible
	Schematic single-line	Single-line representation	Editing possible
	Fluid power schematic	Fluid power schematics	Editing possible
	P&I diagram	P&I diagrams	Editing possible
	Graphic	Graphical representation without logic	Editing possible
	Overview	PLC	Editing possible
	Topology	Routing path networks (2D)	Editing possible
	Pre-planning	Graphical pre-planning (no connection points and no connections possible)	Editing possible
	Table of contents	List of pages	Automatically generated page
	Manufacturer/supplier list	List of manufacturers/suppliers	Automatically generated page
	Title page/cover sheet	Cover sheet	Automatically generated page
	Terminal diagram	List of terminals	Automatically generated page
	Terminal-connection diagram	List of terminals with more graphical options	Automatically generated page
	Terminal line-up diagram	Shows terminals in a structure list, for example	Automatically generated page
	Terminal-strip overview	Overview of all terminal strips	Automatically generated page
	Cable diagram	List of a cable	Automatically generated page
	Cable assignment diagram	Representation of structure for cables with further information	Automatically generated page
	Cable-connection diagram	List of cables with more graphical options	Automatically generated page
	Cable overview	Overview of all cables	Automatically generated page
	Plug diagram	List of plug/female pin contacts	Automatically generated page
	Pin-connection diagram	List of plugs/female pins with more graphical options	Automatically generated page
	Plug overview	Overview of all plugs/female pins	Automatically generated page
	Panel layout	For the structure of mounting surfaces, such as enclosures, control panels	Editing possible

Icon	Page type	Possible use	Interactive/ automatic
	Enclosure legend	List of equipped devices of mounting surfaces	Automatically generated page
	Model view	2D drawings of 3D models generated on the basis of a layout space	Editing possible
	Connection list	List of connections	Automatically generated page
	Device tag list	List of devices	Automatically generated page
	Parts list	List of individual parts	Automatically generated page
	Summarized parts list	List of summarized parts, etc.	Automatically generated page
	PLC diagram	List of PLC connection points	Automatically generated page
	PLC card overview	PLC overview representation	Automatically generated page
	Device connection diagram	List of device connection points	Automatically generated page
	Symbol overview	Overview of symbols	Automatically generated page
	Potential overview	List of potential information	Automatically generated page
	Structure identifier overview	Overview of structure identifiers	Automatically generated page
	Revision overview	List of revision statuses	Automatically generated page
	Forms documentation	Overview of forms	Automatically generated page
	Plot frame documentation	Overview of plot frames	Automatically generated page
	Project options overview	List of project options	Automatically generated page
	Placeholder object overview	List of placeholder objects	Automatically generated page
	Topology: Routing path list	Output of routing paths and their data	Automatically generated page
	Topology: Routing path diagram	Output of routing path with cables, lines	Automatically generated page
	Topology: Routed cables/connections	Overview of routed cables, connections	Automatically generated page
	Pre-planning: Structure segment overview	List of structure segments	Automatically generated page
	Pre-planning: Structure segment diagram	List of structure segment properties	Automatically generated page

Icon	Page type	Possible use	Interactive/ automatic
	Pre-planning: Planning object overview	List of planning objects	Automatically generated page
	Pre-planning: Planning object diagram	List of planning object properties	Automatically generated page
	Pre-planning: Segment template overview	List of segment templates	Automatically generated page
	Pre-planning: Segment template plan	List of segment template properties	Automatically generated page
	Assembly/Module overview	Overview of assemblies and modules used	Automatically generated page
	Distributed device list	List of devices and associated, cross-referenced functions	Automatically generated page
	Bundle/conduit plan	List of bundles/conduits	Automatically generated page
	External document	Integration of external documents	Editing possible
	Logocad TRIGA drawing	Not used anymore; may still exist for legacy reasons	Editing possible

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.

Page menu and page navigator

You access the *popup menu* functions via the right mouse button in the **PAGE/NAVIGATOR** menu.

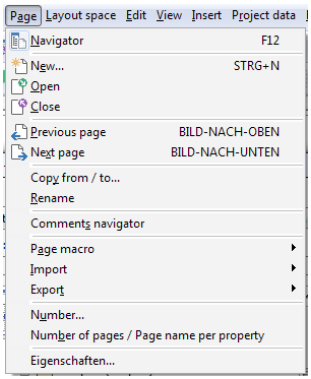


Fig. 4.9 Page menu

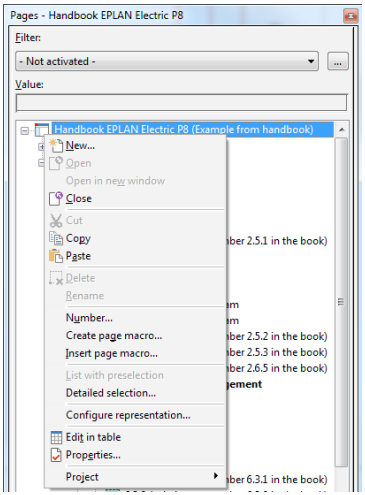


Fig. 4.10 Popup menu

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.

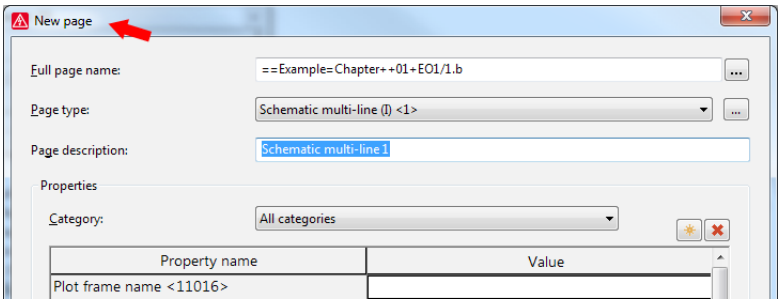


Fig. 4.11 New page dialog

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 (if used in the project's structures) has its own input field and own … selection button, which can be used to branch to the selection dialog for the selected identifier.

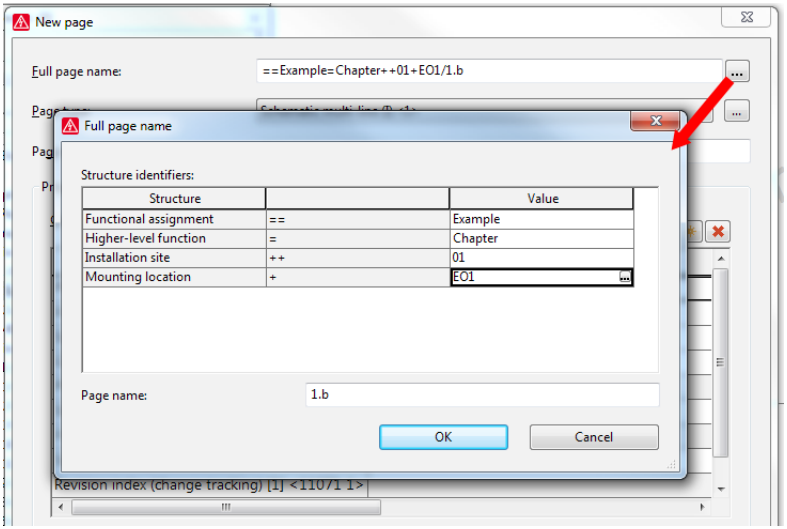


Fig. 4.12 Full page name dialog

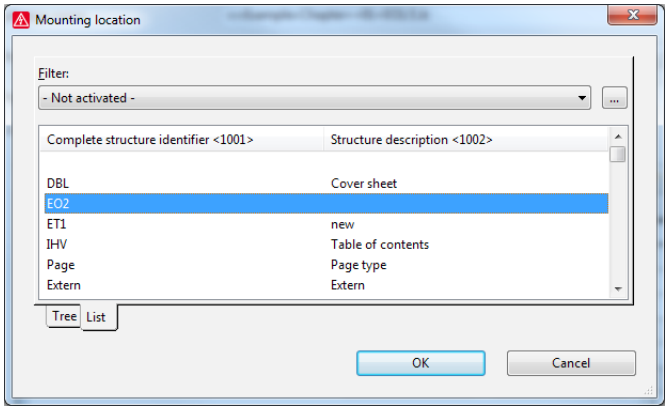


Fig. 4.13 Identifier dialog

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. This dialog, too, allows for switching between the tree and list views.

NOTE: A new identifier can also be directly entered into the **[Identifier type – Value]** input field in the **FULL PAGE NAME DIALOG**.

But keep in mind: 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.

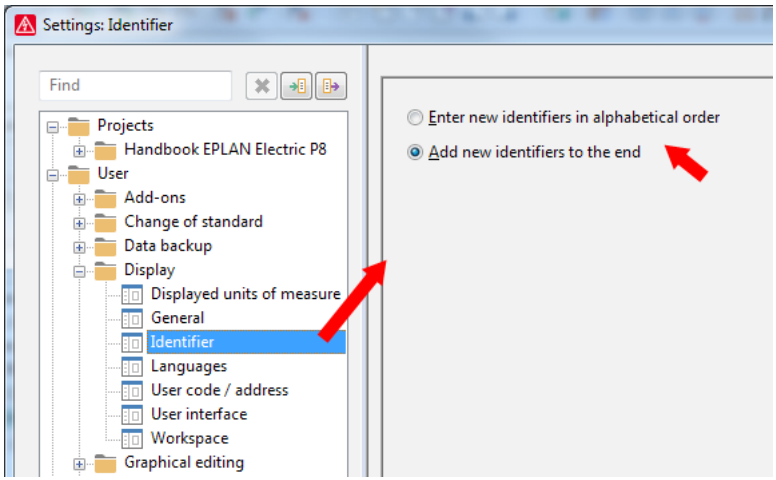


Fig. 4.14 Connection settings

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.

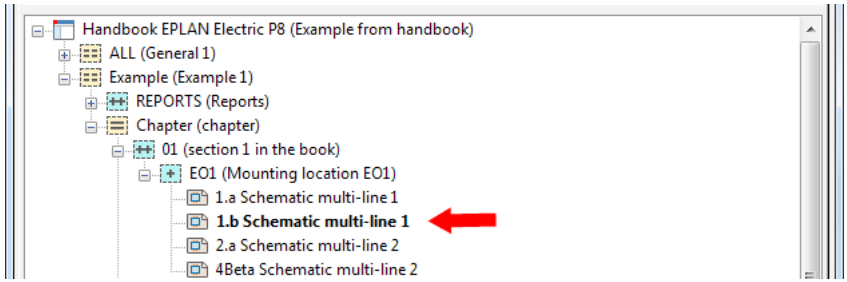


Fig. 4.15 Newly created page

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**.

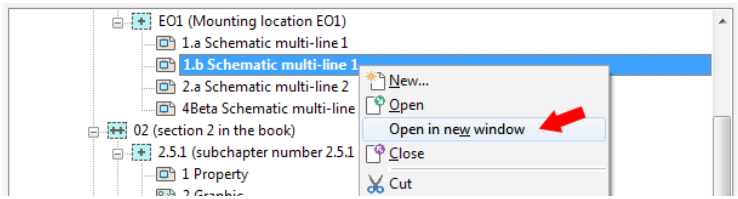


Fig. 4.16 Opening in new window

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.

Function: Open in new window

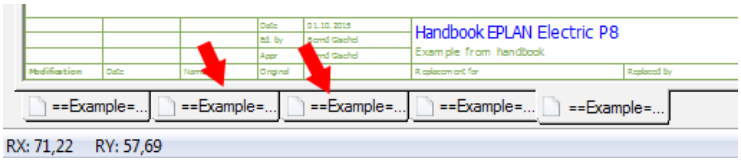


Fig. 4.17 Several open pages

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.

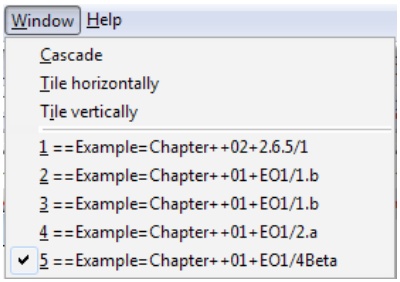


Fig. 4.18
Window menu

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.

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).

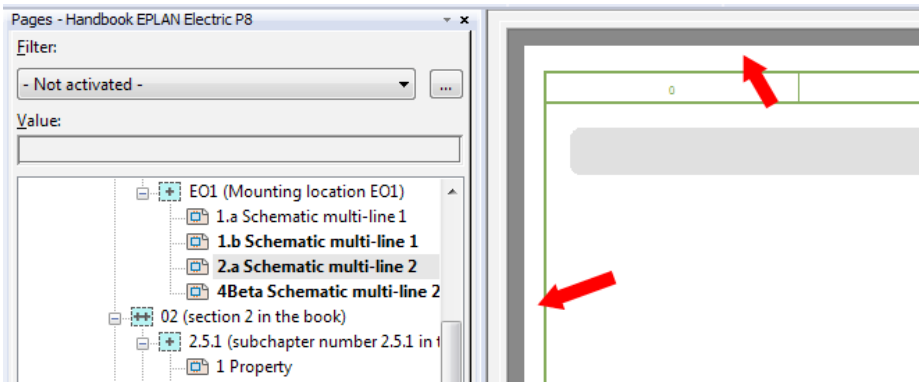
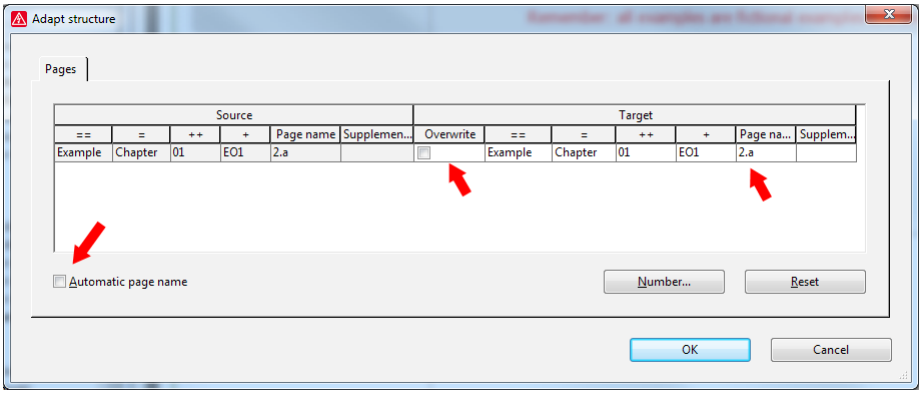


Fig. 4.19 Using CTRL + M to select a page

Use **CTRL + M** to select a page (outside the page navigator) in the graphical editor in order to copy it. 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) retrieves the copied page from the clipboard and pastes it 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 (without determining page name automatically)

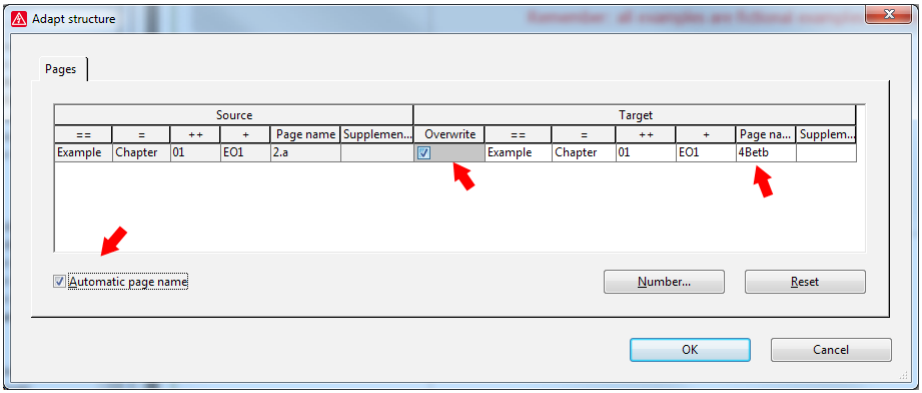


Fig. 4.21 Adapt structure dialog (with determining page name automatically)

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 the identifier structure selected. Once all entries are correct, you click **OK** to apply the page.

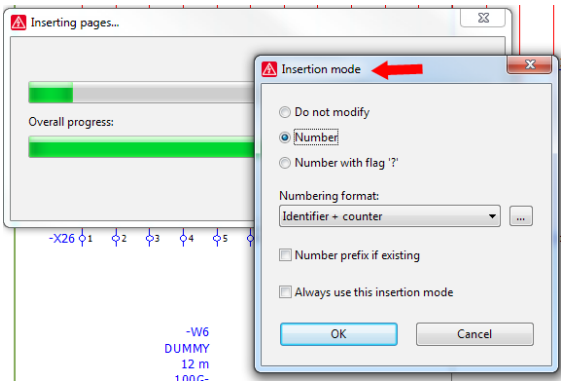


Fig. 4.22 Selecting the insertion mode

Depending on the settings and on whether there are any devices on the page to be copied), 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, numbers the devices according to the setting, and sorts the page into the page structure.

At this point, the **ADAPT STRUCTURE** dialog also lets you overwrite existing pages (using the **Target – Overwrite** column).

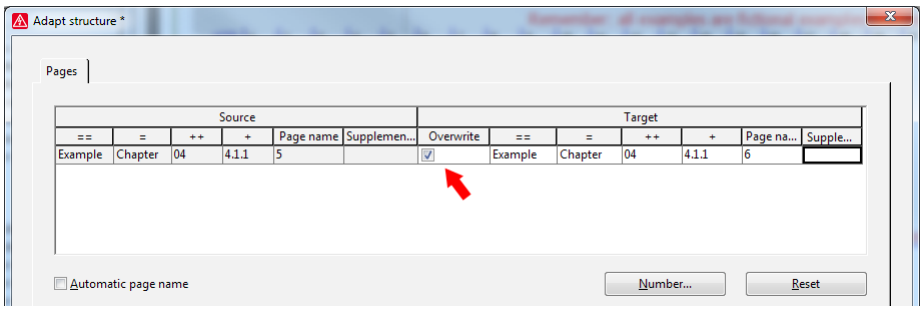


Fig. 4.23
Selecting Overwrite option

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).

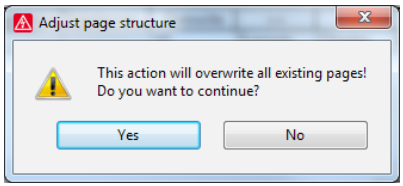


Fig. 4.24
Note regarding the Overwrite action

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.

Copying pages in the page navigator

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.

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.

Page navigator List view

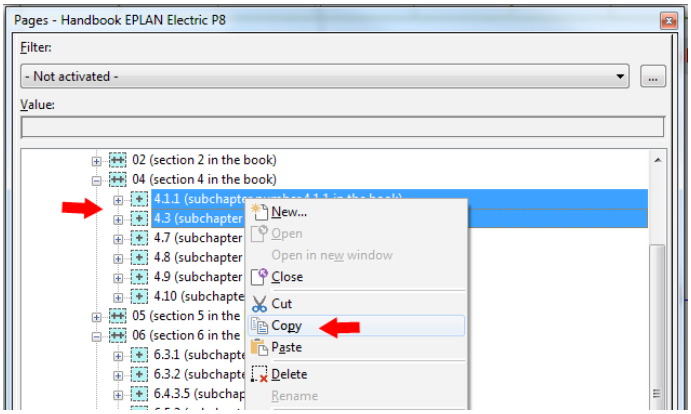


Fig. 4.25
Copying in the tree view

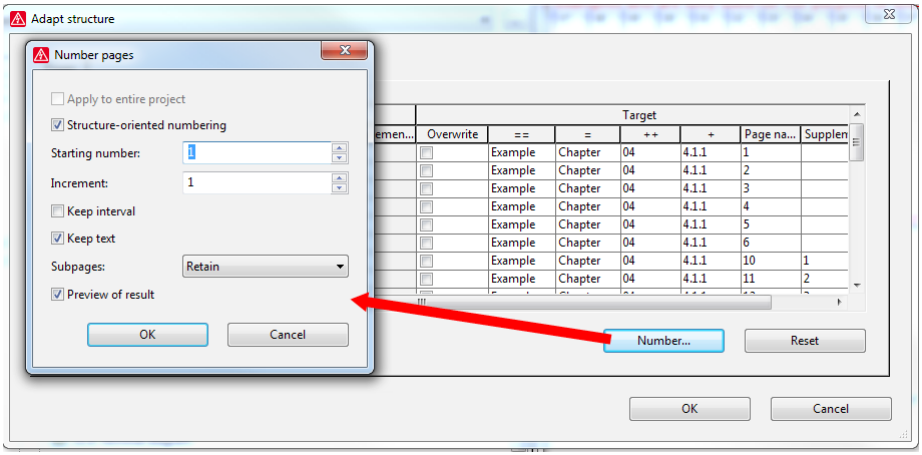


Fig. 4.26 Adapt structure dialog with Number pages option

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.

4.1.2.3.1 Number button

The option to re-number pages immediately when inserted offers some advantages. Page numbers do not have to be adjusted manually. Instead, if so desired, EPLAN can number the page numbers in an ascending order, for example, sorted by structure identifiers.

TIP: The preview of results should always be selected as well. This way, you can see the numbering results right away and, if necessary, reverse them via Cancel/Restore, without the changes being written to the project.

Two small examples will illustrate this. For setting 1, the start number 1 is selected, and no structure-based numbering. Text and subpages are retained.

After you click the **NUMBER PAGES** dialog, EPLAN numbers all pages (across identifier structures).

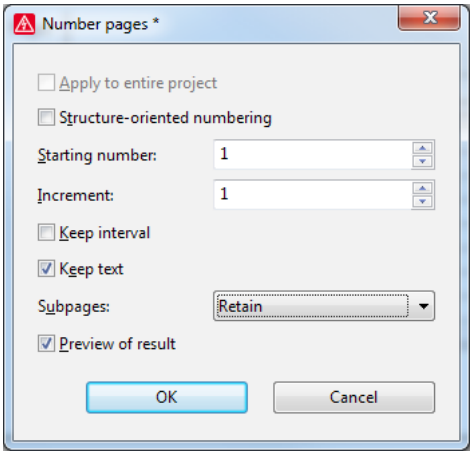


Fig. 4.27
Number pages dialog, setting 1

Target						
Overwrite	==	=	++	+	Page na...	Supplen
	Example	Chapter	04	4.1.1	5	
	Example	Chapter	04	4.1.1	6	
	Example	Chapter	04	4.1.1	7	1
	Example	Chapter	04	4.1.1	8	2
	Example	Chapter	04	4.1.1	9	3
	Example	Chapter	04	4.1.1	10	4
	Example	Chapter	04	4.1.1	11	5
	Example	Chapter	04	4.1.1	12	6

Fig. 4.28
Numbering result, setting 1

A second example illustrates setting 2. Here, the settings are structure-oriented numbering, start number 1, do not retain text, and the subpages are to be converted to main pages.

After you click the **OK** button, EPLAN breaks up the subpages, removes the alphabetical elements of a page number, and numbers the pages in an ascending order by structure identifiers, starting at 1.

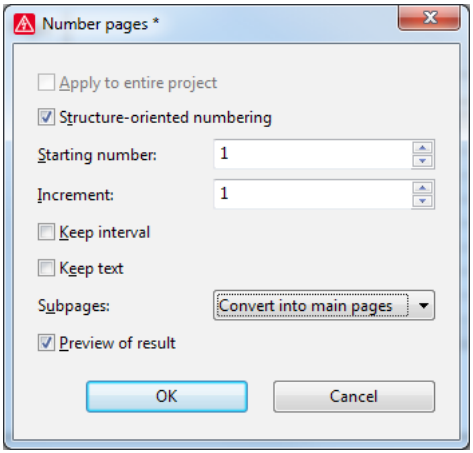
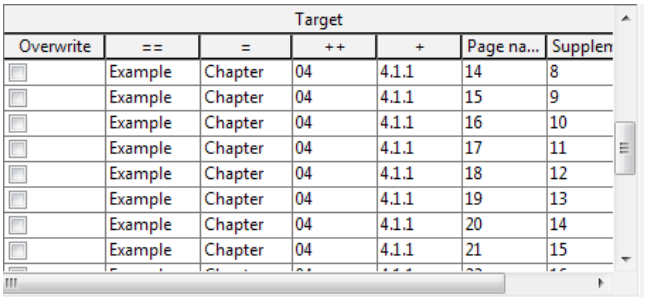


Fig. 4.29
Number pages dialog, setting 2



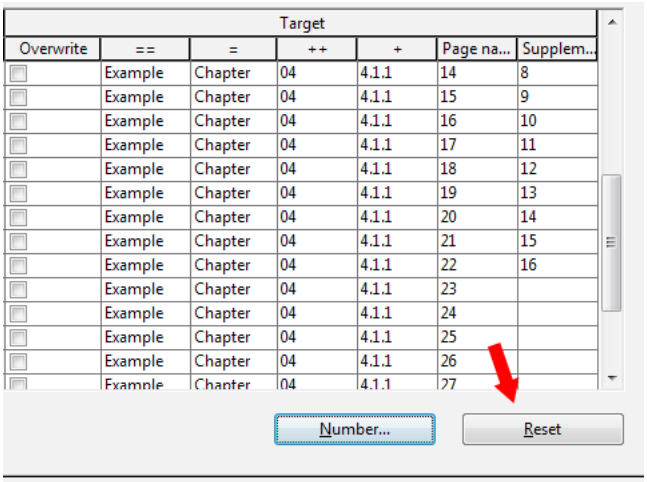
Target

Overwrite	==	=	++	+	Page na...	Supplen
	Example	Chapter	04	4.1.1	14	8
	Example	Chapter	04	4.1.1	15	9
	Example	Chapter	04	4.1.1	16	10
	Example	Chapter	04	4.1.1	17	11
	Example	Chapter	04	4.1.1	18	12
	Example	Chapter	04	4.1.1	19	13
	Example	Chapter	04	4.1.1	20	14
	Example	Chapter	04	4.1.1	21	15

Fig. 4.30
Numbering result, setting 2

4.1.2.3.2 Restore button

The **RESTORE** button restores the numbering to the values prior to the numbering.

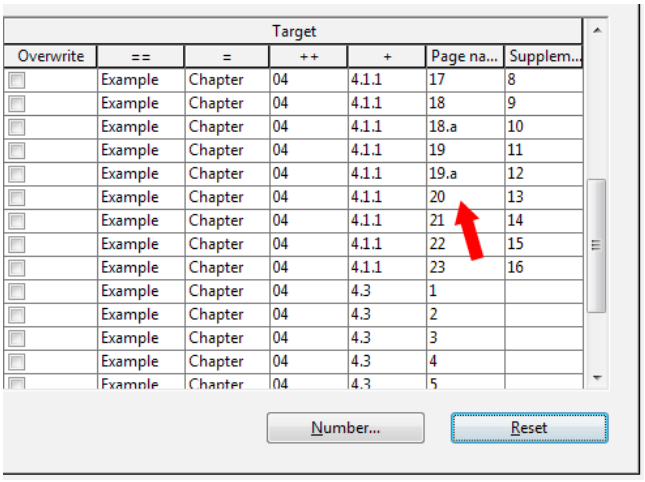


Target

Overwrite	==	=	++	+	Page na...	Supplem...
	Example	Chapter	04	4.1.1	14	8
	Example	Chapter	04	4.1.1	15	9
	Example	Chapter	04	4.1.1	16	10
	Example	Chapter	04	4.1.1	17	11
	Example	Chapter	04	4.1.1	18	12
	Example	Chapter	04	4.1.1	19	13
	Example	Chapter	04	4.1.1	20	14
	Example	Chapter	04	4.1.1	21	15
	Example	Chapter	04	4.1.1	22	16
	Example	Chapter	04	4.1.1	23	
	Example	Chapter	04	4.1.1	24	
	Example	Chapter	04	4.1.1	25	
	Example	Chapter	04	4.1.1	26	
	Example	Chapter	04	4.1.1	27	

Number... Reset

Fig. 4.31
Page numbering prior to restore



Target

Overwrite	==	=	++	+	Page na...	Supplem...
	Example	Chapter	04	4.1.1	17	8
	Example	Chapter	04	4.1.1	18	9
	Example	Chapter	04	4.1.1	18.a	10
	Example	Chapter	04	4.1.1	19	11
	Example	Chapter	04	4.1.1	19.a	12
	Example	Chapter	04	4.1.1	20	13
	Example	Chapter	04	4.1.1	21	14
	Example	Chapter	04	4.1.1	22	15
	Example	Chapter	04	4.1.1	23	16
	Example	Chapter	04	4.3	1	
	Example	Chapter	04	4.3	2	
	Example	Chapter	04	4.3	3	
	Example	Chapter	04	4.3	4	
	Example	Chapter	04	4.3	5	

Number... Reset

Fig. 4.32
Page numbering after restore

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.

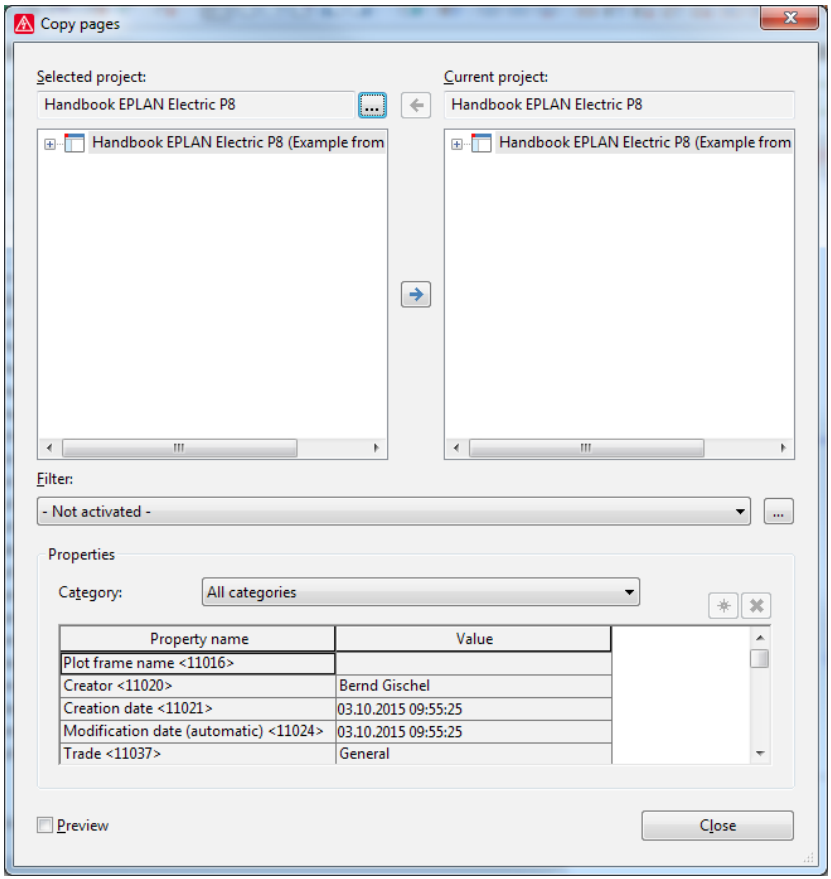


Fig. 4.33 Copy pages dialog

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.

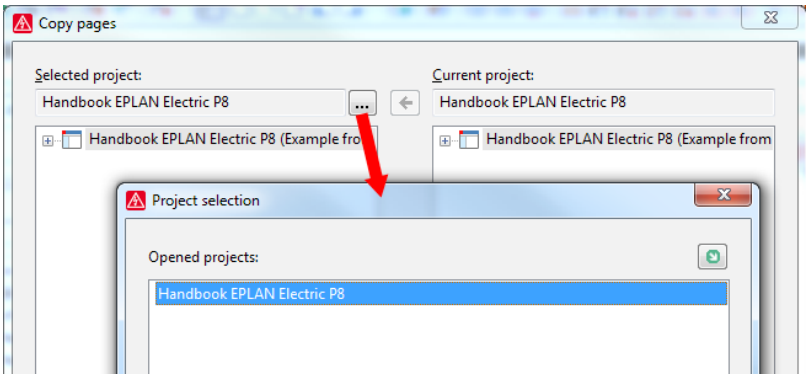


Fig. 4.34 Project selection dialog

When you click the button (**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.

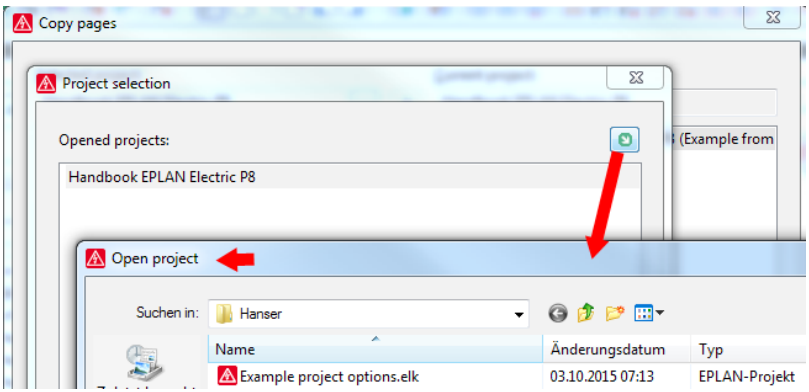


Fig. 4.35 Open project dialog

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

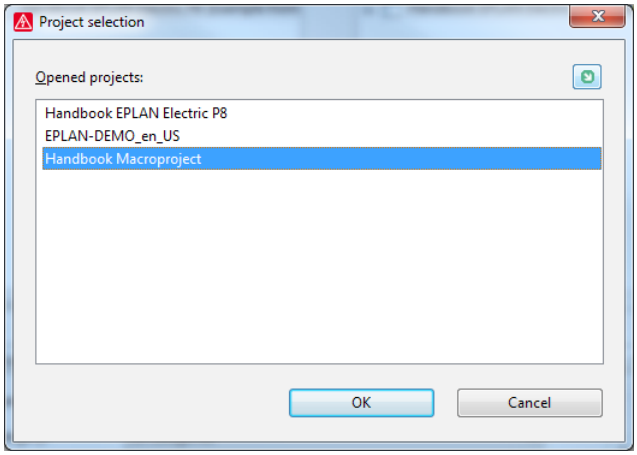


Fig. 4.36 Opened projects

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.

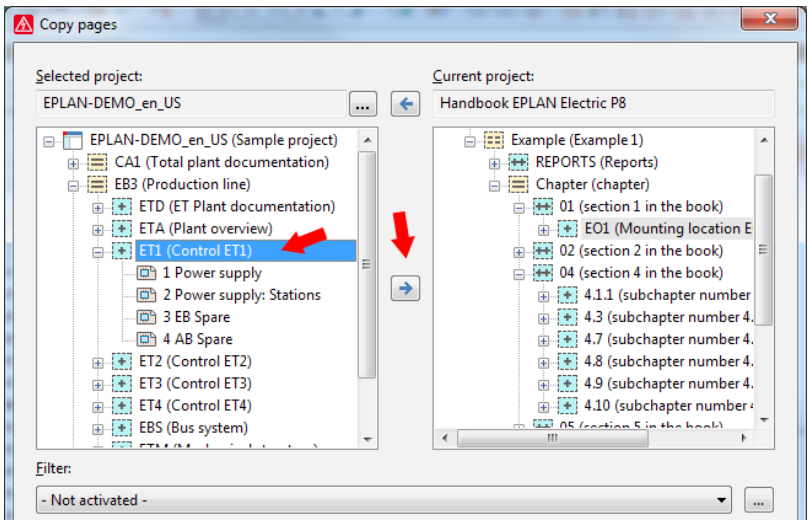


Fig. 4.37 Copying pages from a project

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.

Adapt structure

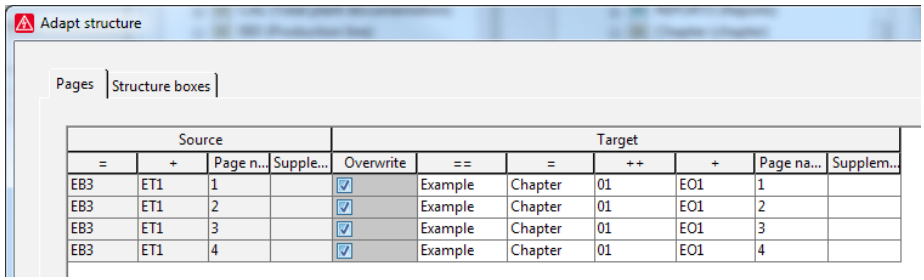


Fig. 4.38 Inserting pages into the page structure

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

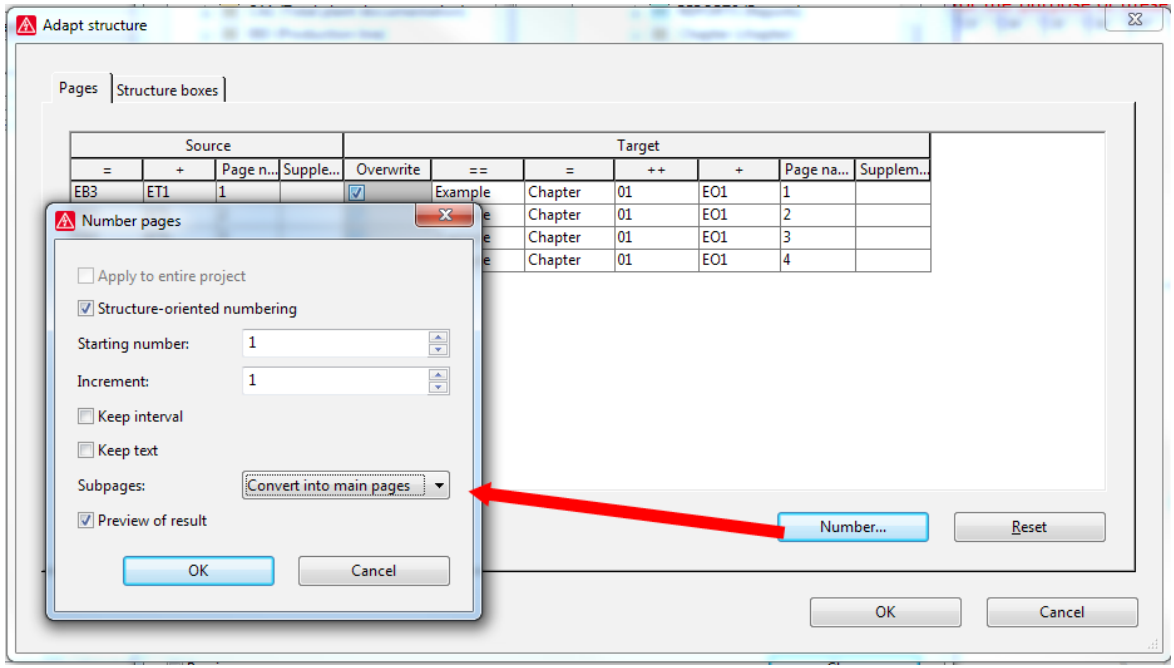


Fig. 4.39 Number pages

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)

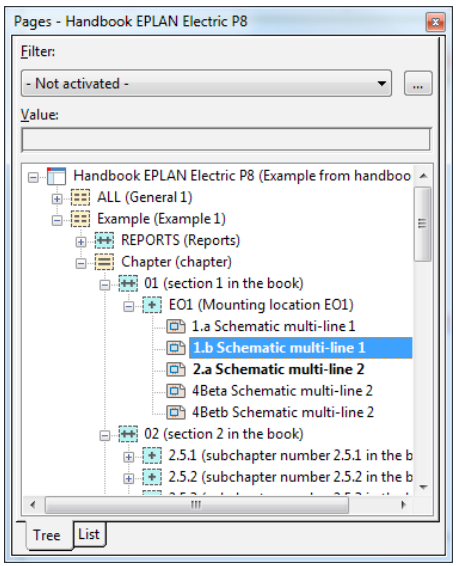


Fig. 4.40 Selecting a page

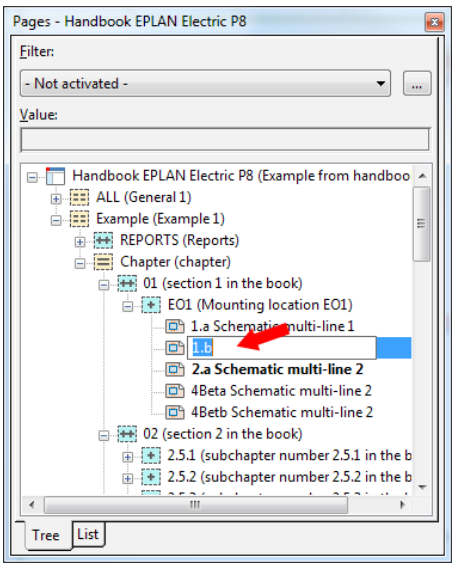


Fig. 4.41 Direct editing via F2 (page name with subpage)

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: When deleting pages, it is important to know that there is a difference in how pages are selected in the page navigator.

In Fig. 4.42, the four pages have been selected to be deleted. Only these four pages will be deleted in this case. The warning message in the **DELETE PAGES** dialog is very clear: "Do you want to delete Page … out of 4 selected pages?"

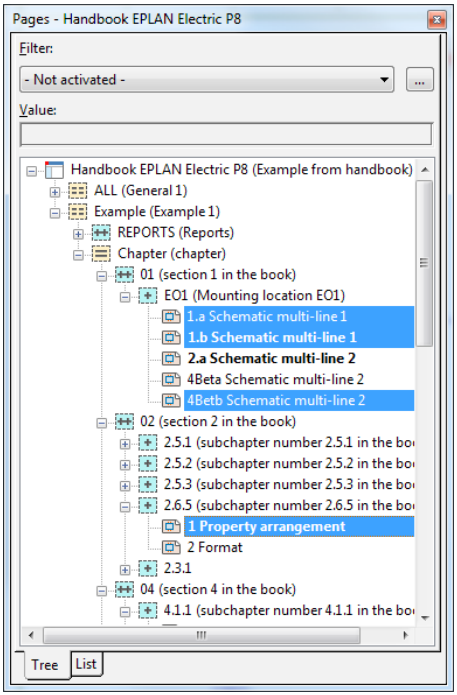


Fig. 4.42 Selection 1: 4 pages

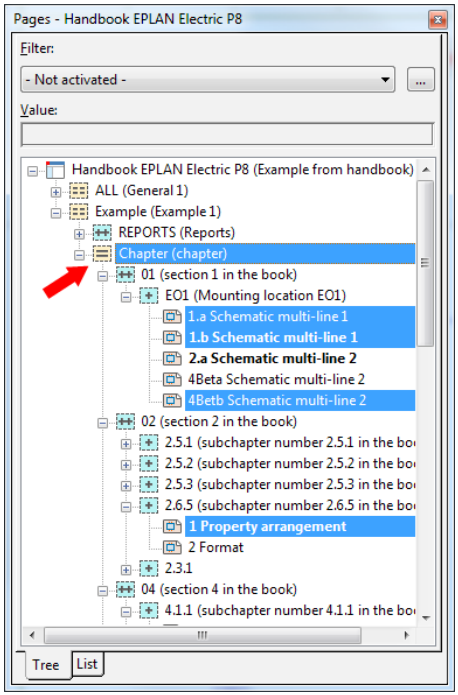


Fig. 4.43 Selection 2: Entire higher-level function (chapter)

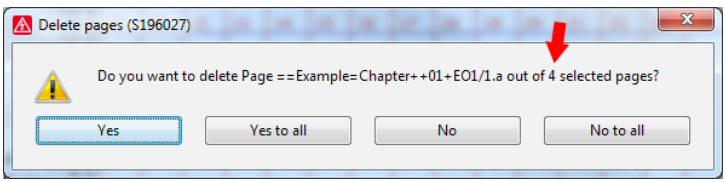


Fig. 4.44 Warning message: Delete 4 pages

In Fig. 4.43, 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**.

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.

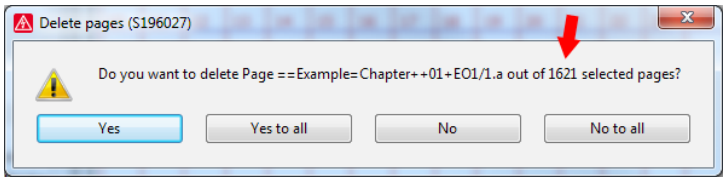


Fig. 4.45 Warning message: Delete 1621 pages? (instead of the desired 4)

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 when editing a project. 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*.

WARNING: 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 individually 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.

TIP: To close several or all opened pages in one go, simply select the project name and call the **CLOSE** command via the popup menu. This will close all opened page with a single click.

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 again the page and then **CTRL + D** calls the **EDIT PROPERTIES** function.

EPLAN then opens the **PAGE PROPERTIES** dialog.

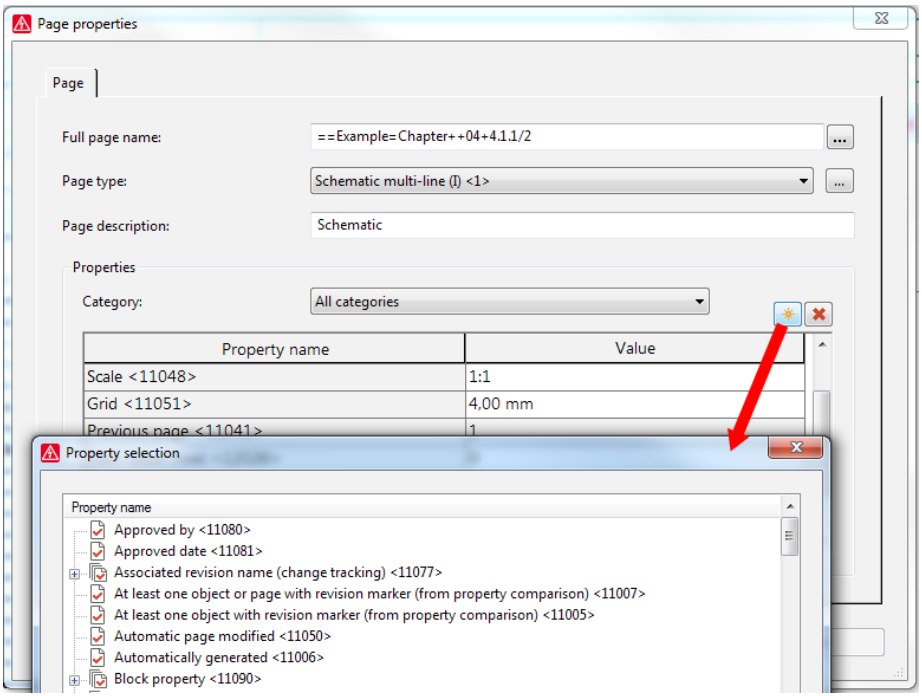


Fig. 4.46 Edit page properties dialog

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.

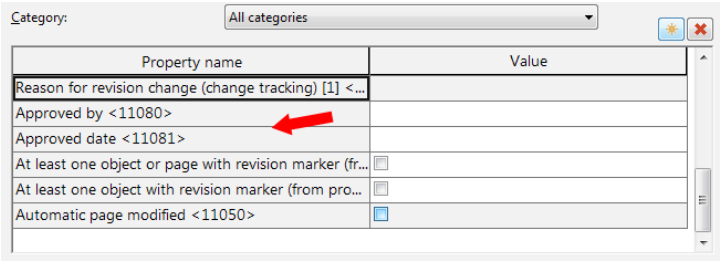


Fig. 4.47 Newly added properties

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.

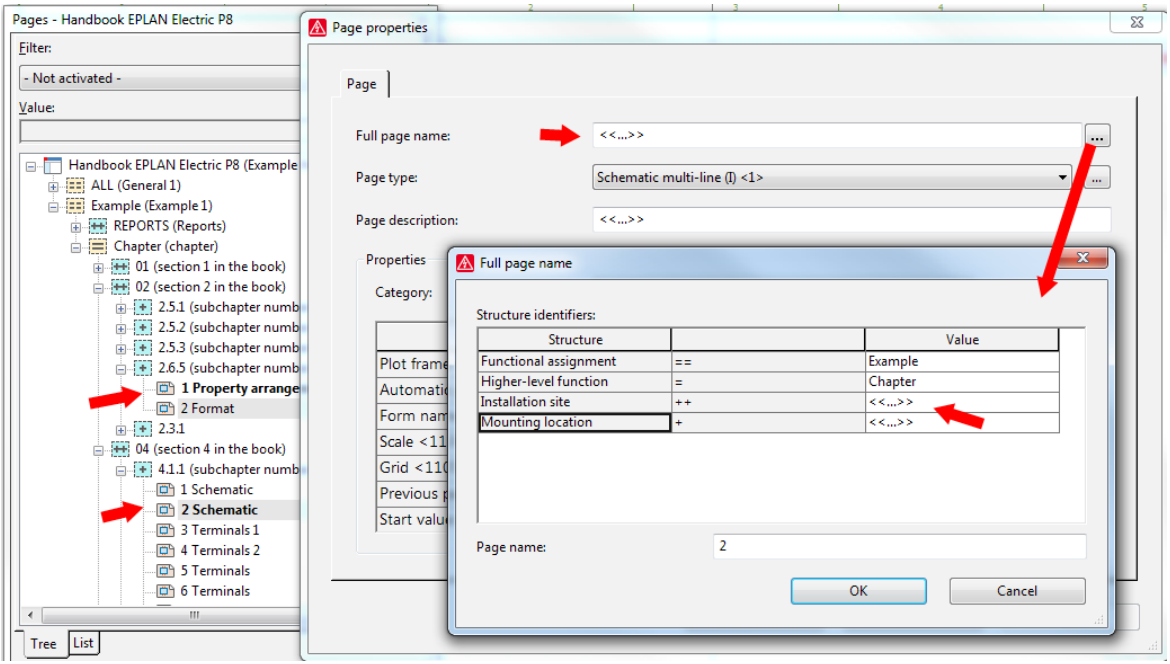


Fig. 4.48 Editing the properties of multiple pages

All properties that are the same in the selected pages are displayed in “plain text”. Examples would be the identifier (structure), 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 (without a prompt) 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, graphical element) in the plot frame to have EPLAN open the page properties.

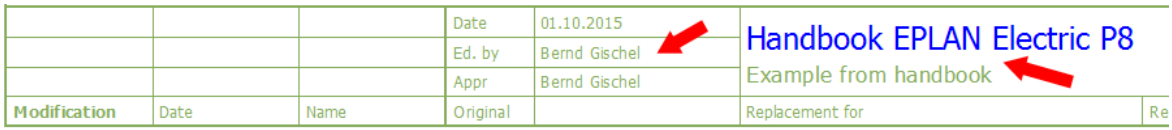


Fig. 4.49 Double-click any object in the plot frame to open page properties

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

Fig. 4.50 Empty areas do not work

4.1.3 Page navigator filter

In addition to the functions previously described, it is now also possible to directly activate a filter for previously created schemes and use the quick input functions in the page navigator and in other navigators.

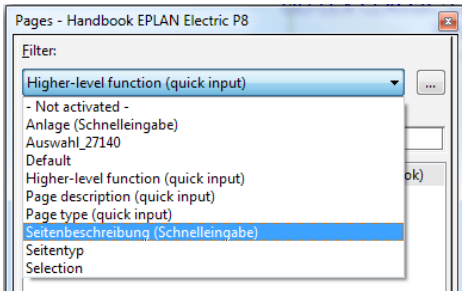


Fig. 4.51 Filter options

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.

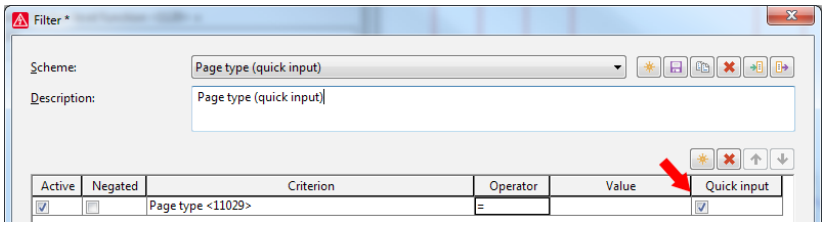


Fig. 4.52 Filtering by quick entry

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

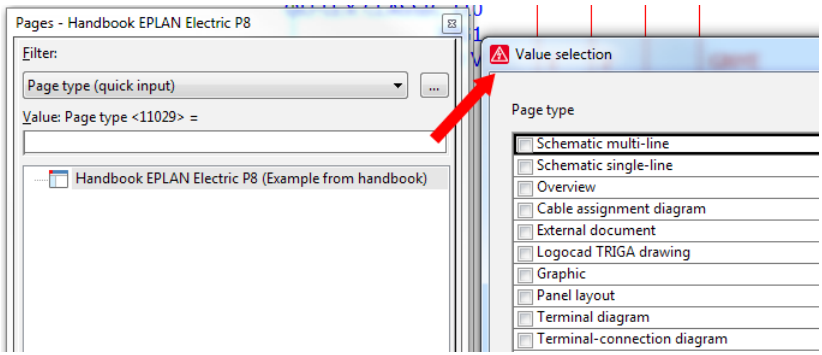


Fig. 4.53 Active page type filter

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, with one criterion, like here in the **Page type**. You simply activate the required value, here the to-be-filtered **page type**.

EPLAN also offers further filter options. The detailed selection. It is not limited to filtering page properties, but you can also activate further filters, for example, for specific identifiers and similar elements in the page contents.

Apart from the functions described here, there are a number of other options in the page navigator. Some of these will be explained briefly in the following sections.

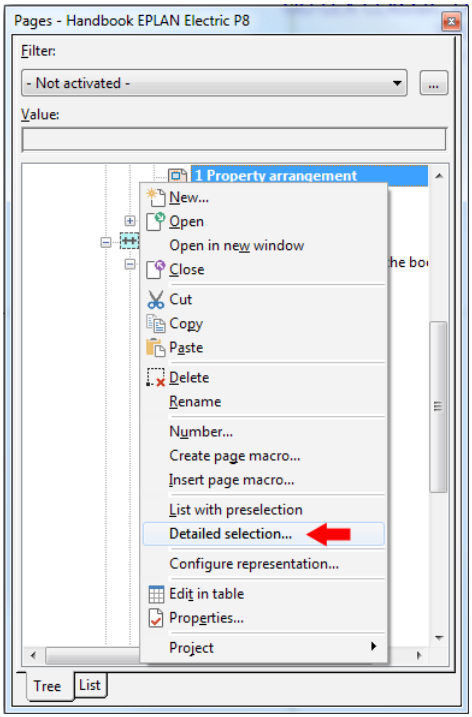


Fig. 4.54 Detailed selection

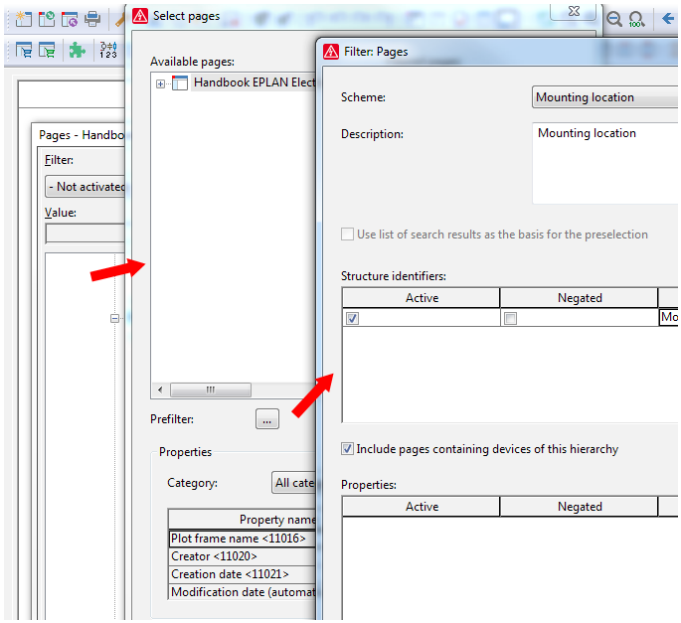


Fig. 4.55 Filter options

4.1.4 Edit in table

The edit in table function allows the user to edit/revise page property data quickly and conveniently, similar to the way it is done in Excel. To use this function, select the pages to be edited in the page navigator, and then call the **EDIT IN TABLE** function via the popup menu.

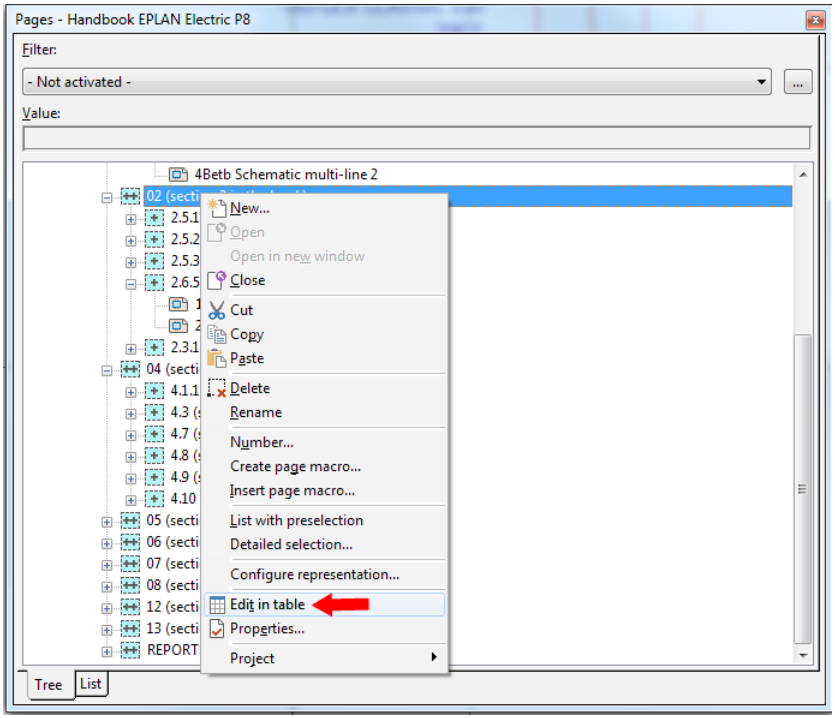


Fig. 4.56 Edit in table popup menu

Afterwards, EPLAN opens the **EDIT FUNCTION DATA** dialog. Aside from the actual edit function, this dialog also offers again the option to activate specific filters, thus editing quickly select properties of the objects contained on these pages.

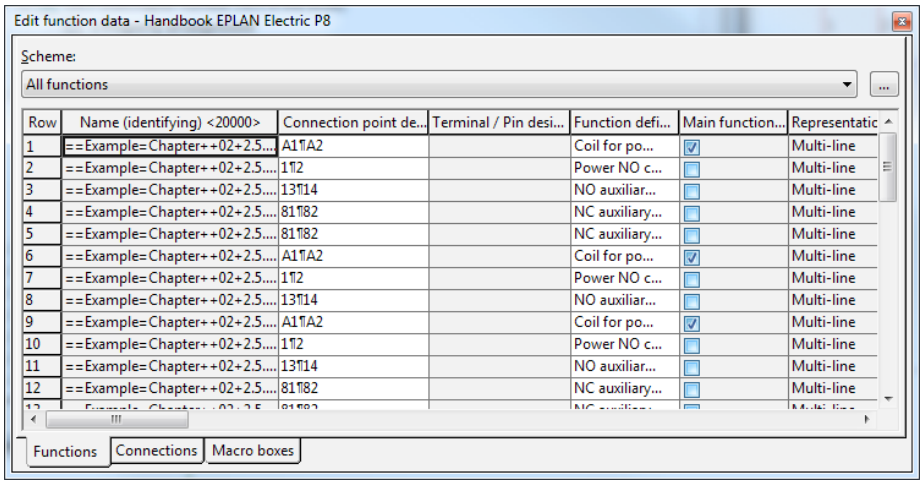


Fig. 4.57 Edit function data dialog in the page navigator

4.1.5 List with preselection

The **LIST WITH PRESELECTION** function in the page navigator makes it possible to filter specific areas quickly and without further filters.

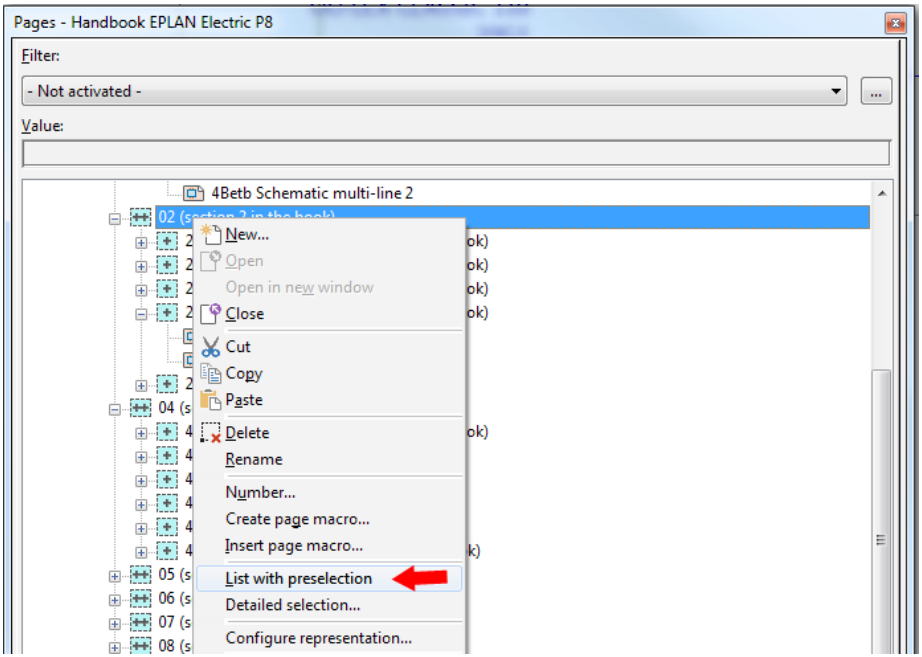


Fig. 4.58 Activating List with preselection function

For example, if you want to filter a specific identifier quickly, without creating an extra filter scheme, it is sufficient to select the identifier in the page navigator and right-click the **LIST WITH PRESELECTION** function.

EPLAN will then activate a "Selection" filter that is generated automatically. The page navigator will then only show the elements (pages in this case) that were selected before.

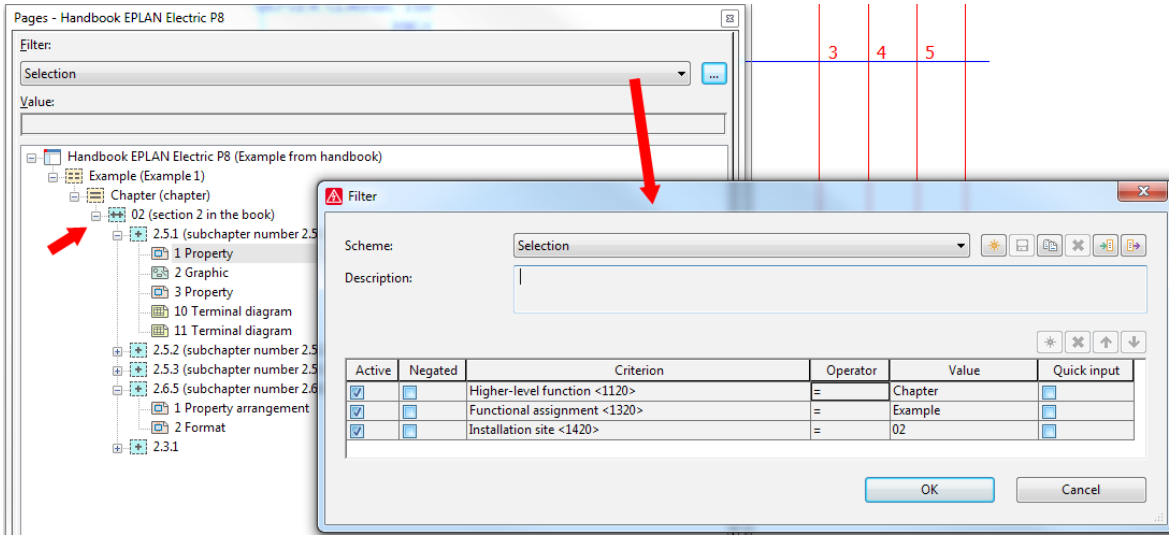


Fig. 4.59 Activated (automatic) "Selection" filter

4.2 General functions

The graphical editor can be divided into several areas. The main area is the working area for editing a 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 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, which is displayed, by default, with the full path designation.

If so desired, a project property can be configured together with the project name. To do so, go to (**OPTIONS/SETTINGS/USER/DISPLAY/USER INTERFACE** menu) and modify the *Additional information on project name* item.

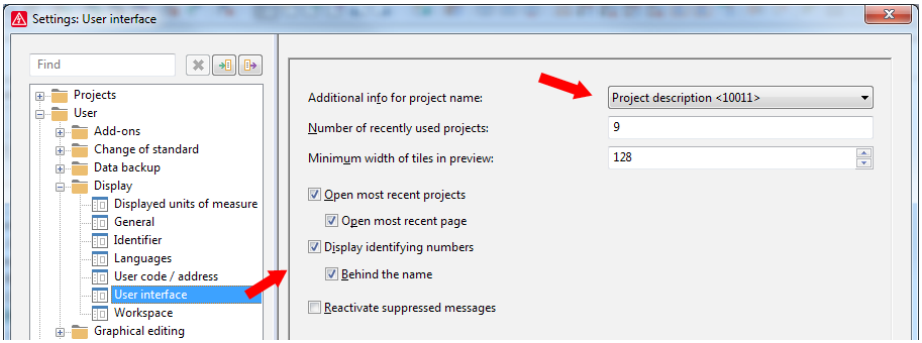


Fig. 4.60 Customizable display of the title bar

The following entries are possible here: user supplementary field 1, commissioning, project description, project number, supplementary field [1], or no project properties In Fig. 4.61, the **None** setting has been selected.

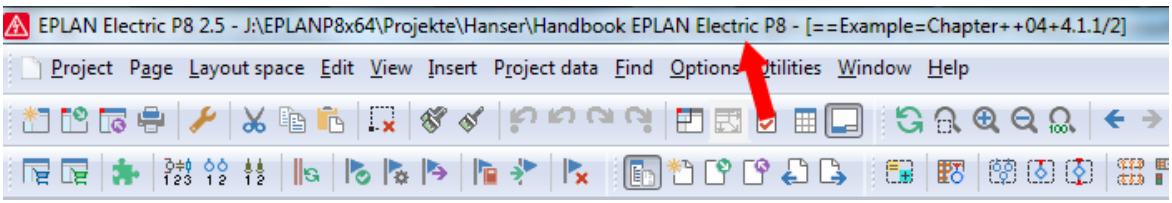


Fig. 4.61 Sample setting

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

4.2.2 Status bar

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

Fig. 4.62 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.

In the status bar, EPLAN also displays supporting information for the function currently in use (in this case, a graphical element 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)]
+	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.

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. Which coordinate system is used depends on the respective page type.

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.

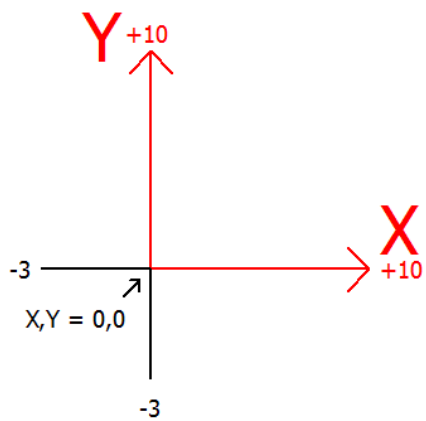


Fig. 4.63
Graphical

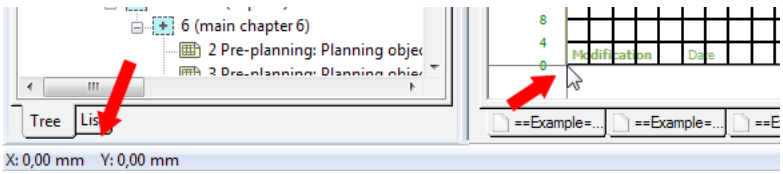


Fig. 4.64 Display in the status bar

4.3.2 Logical coordinate system

The logical system has its origin (RX,RY = 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.

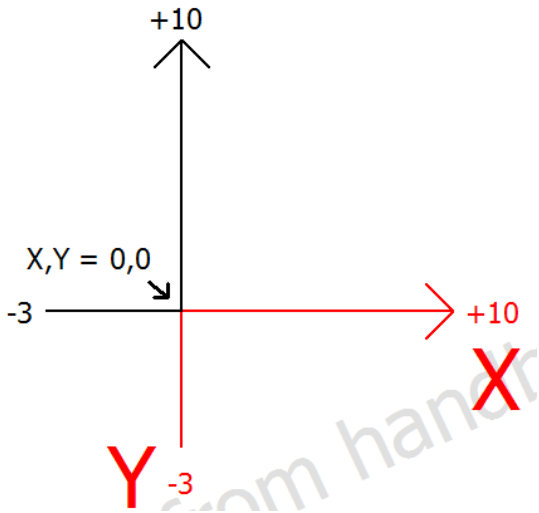


Fig. 4.65 Logical

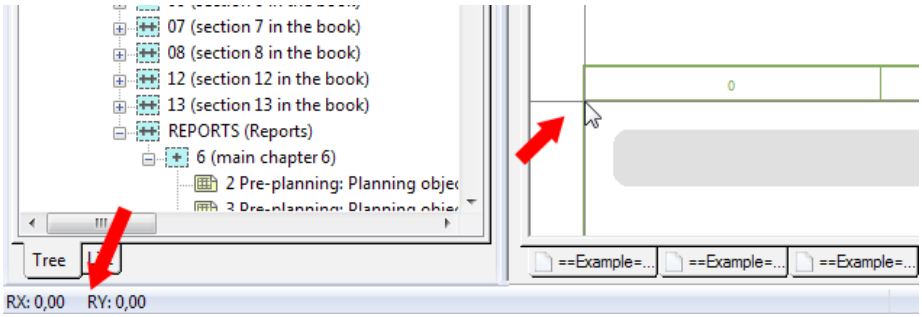


Fig. 4.66 Display in the status bar

4.3.3 Fluid power and process-engineering coordinate systems

Fluid power and process-engineering coordinate systems are two more examples that should be mentioned. For both systems, the origin is in the lower left (RX,RY = 0.0).

4.3.4 3D coordinate systems

Apart from the 2D coordinate systems mentioned so far, there is also a 3D coordinate systems in EPLAN (EPLAN Pro Panel add-on). It bears mentioning here for completeness' sake.

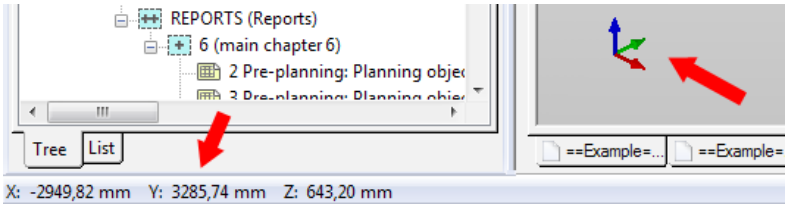


Fig. 4.67 Display in the status bar

4.4 Grid

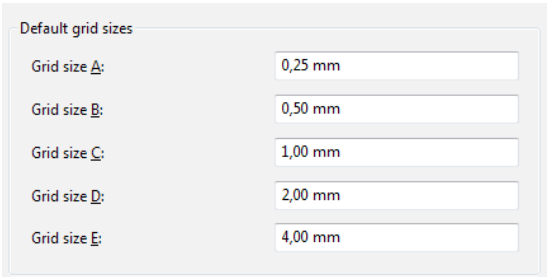


Fig. 4.68
Grid settings

In EPLAN, 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.

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

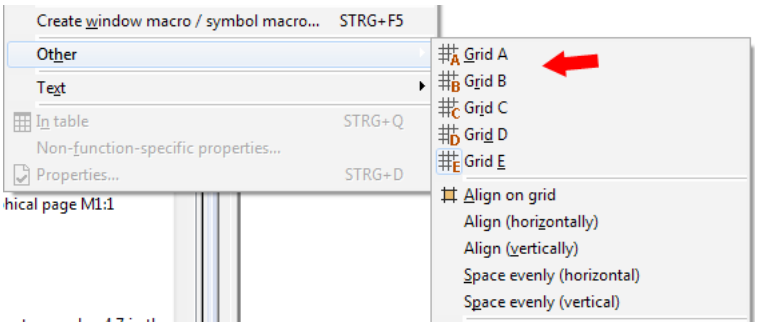


Fig. 4.69 Other menu item, with grid settings

Some of the grid settings should be assigned their own shortcut keys. This would allow for the page grid to be switched quickly, without having to switch to the grid all the time via the menu.

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

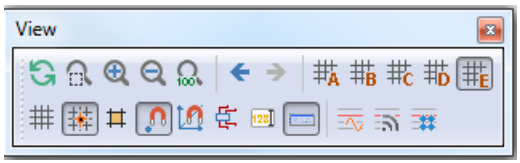


Fig. 4.70
Toolbar view

	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.

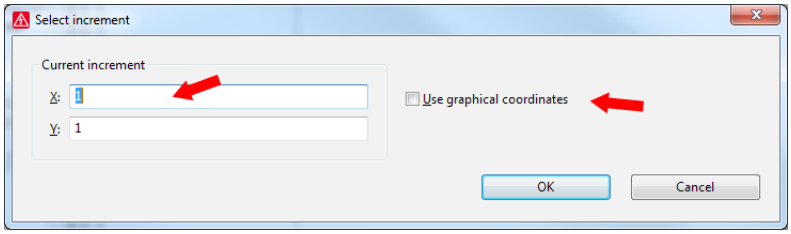


Fig. 4.71 Integer increment setting

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.

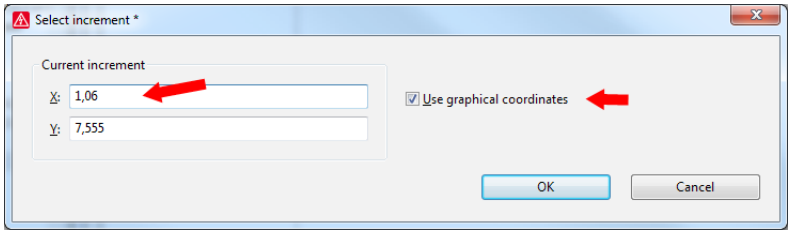


Fig. 4.72 Real-number increment setting

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

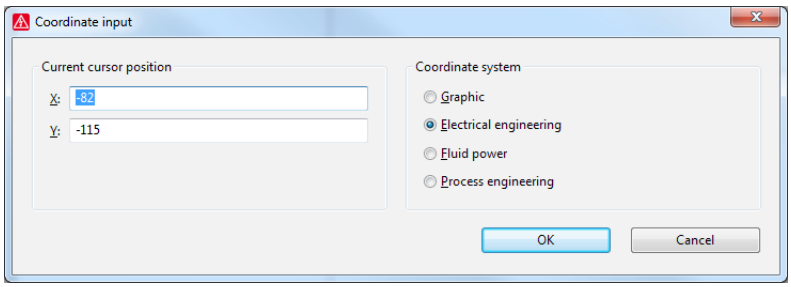


Fig. 4.73 Coordinate input

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

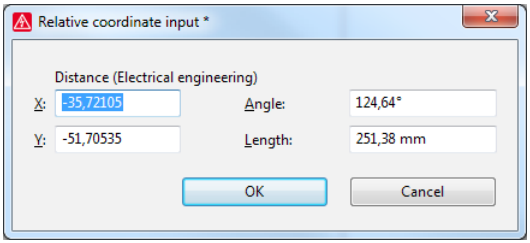


Fig. 4.74
Relative coordinate input

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, lower or upper left).

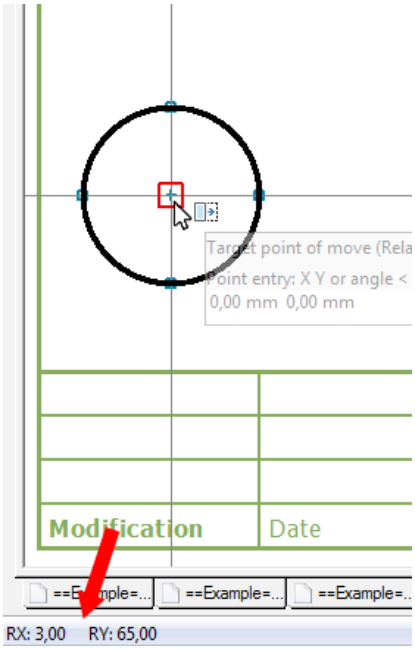


Fig. 4.75 Standard base point

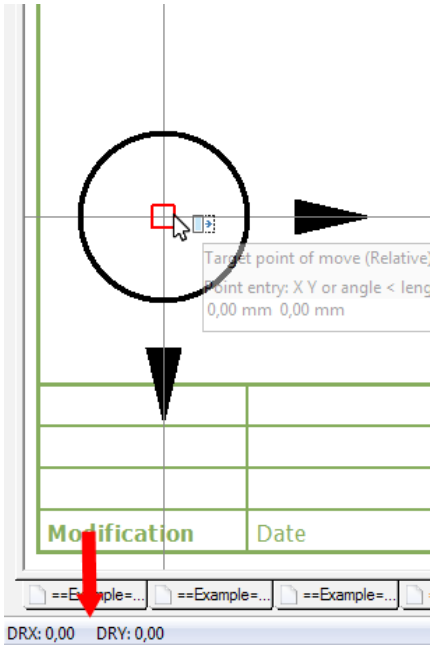


Fig. 4.76 New base point

In the example (center of the circle), the base point is RX = 3.00 mm and RY = 65.00 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.

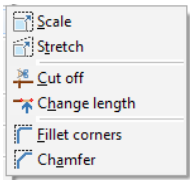


Fig. 4.77 Edit / Graphic menu

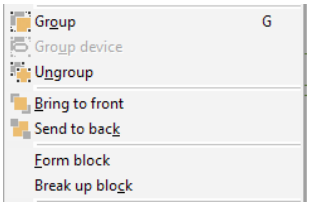


Fig. 4.78 Edit / Other menu

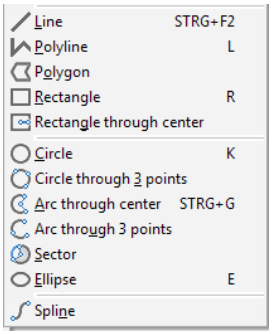


Fig. 4.79 Insert / Graphic menu

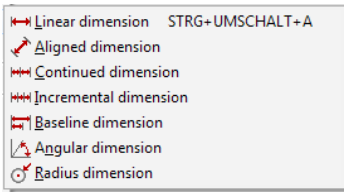


Fig. 4.80 Insert / Dimensioning menu

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).

NOTE: After changing the grid, it is important not to forget to set the grid to the default grid again. Otherwise, it can happen quickly that schematic pages end up being created in the wrong grid, and then everything would have to be restored to the default grid.

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. EPLAN drawings are always accurately scaled. That is, 1 mm in reality is always 1 mm, regardless of the page's scale!

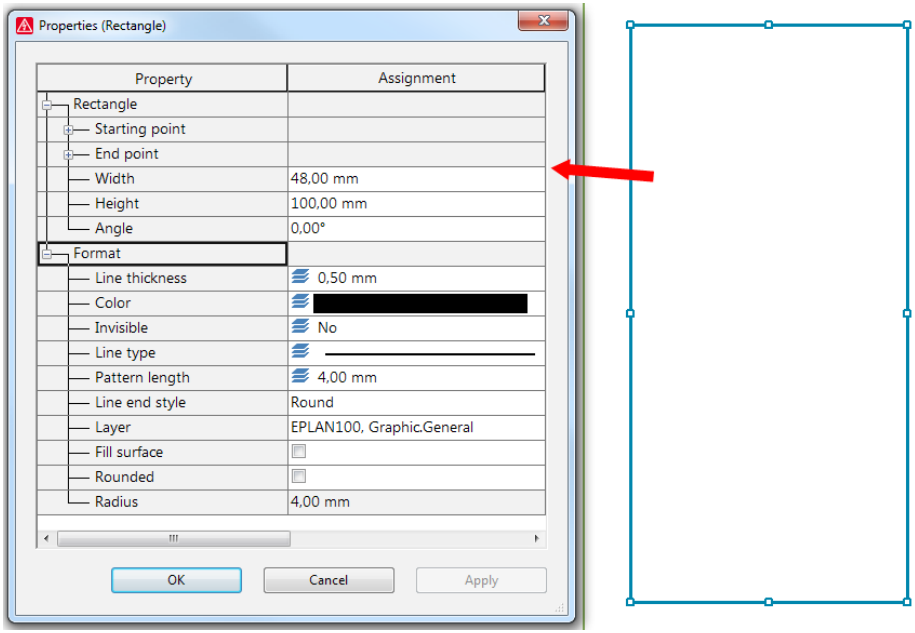


Fig. 4.81 Drawing a simple rectangle with manual width and height

Option 2: You start drawing the rectangle and define its size using the relative coordinate input (shortcut key **SHIFT + R**).

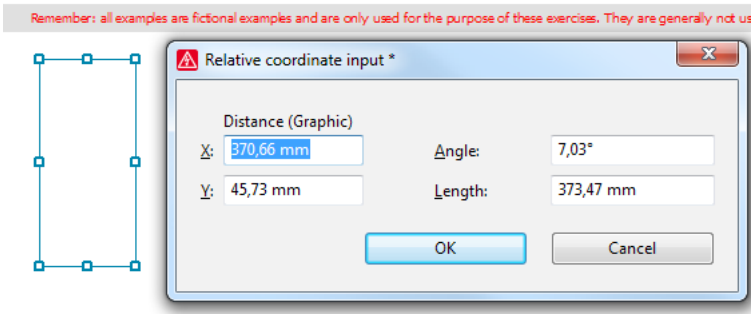


Fig. 4.82 Drawing a rectangle on the basis of relative coordinate input

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.

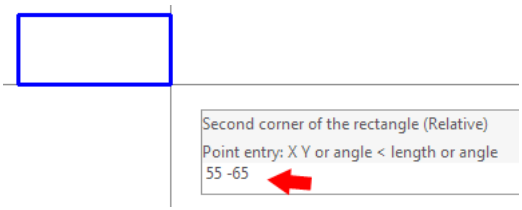


Fig. 4.83
Drawing a rectangle by direct input in the input box

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.

Of course, some functions are not only intended for graphical editing, but also for 'regular' EPLAN work, such as the **ROTATE** function.

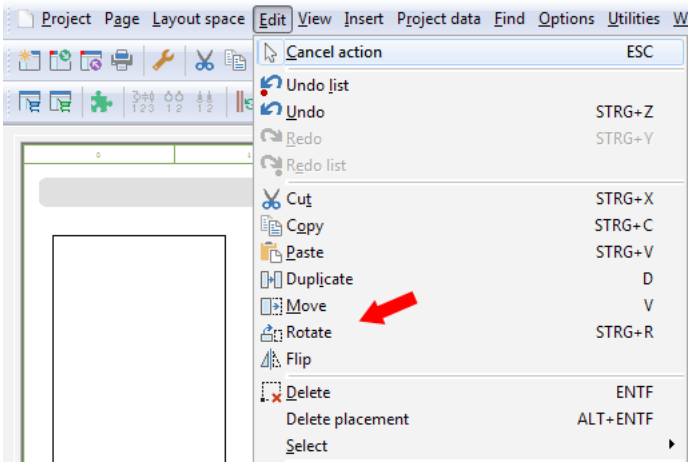


Fig. 4.84 Functions in the Edit menu

NOTE: The examples were partially carried out with the input box (in the **OPTIONS** menu) activated.

4.6.2.1 Rotate

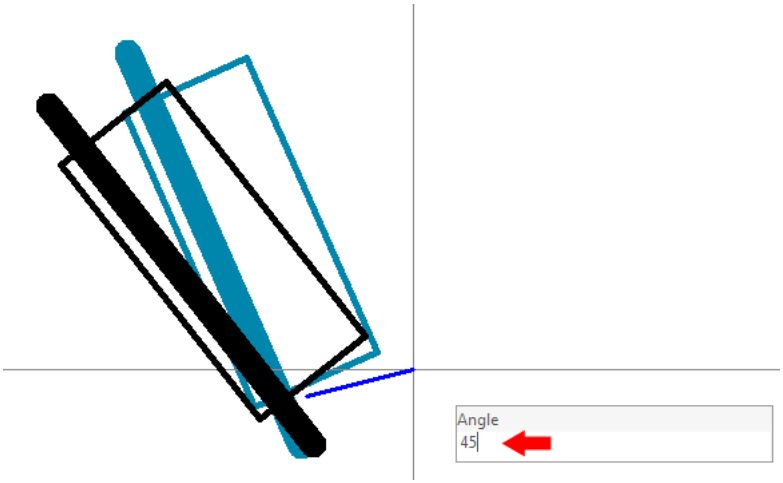


Fig. 4.85
Rotate

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.

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

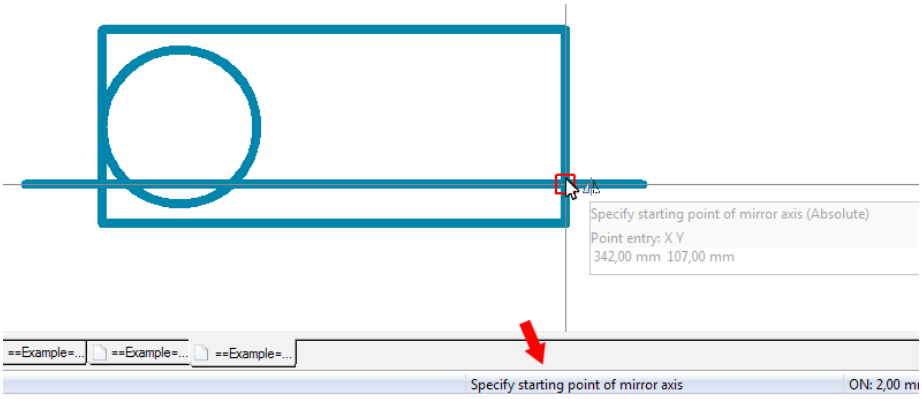


Fig. 4.86
Defining the starting point of the mirror axis

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*.

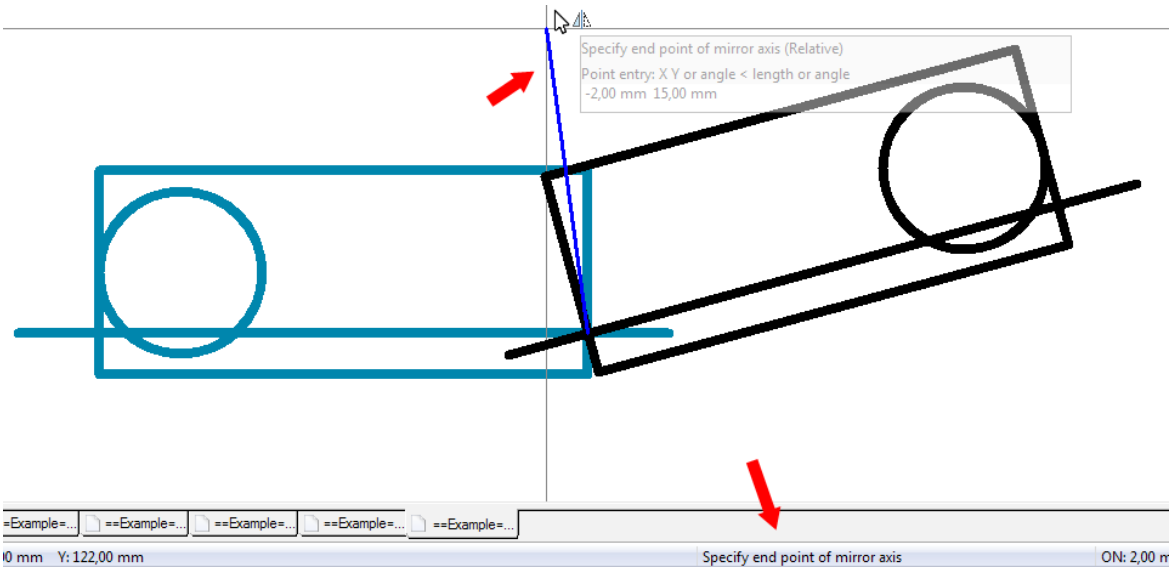


Fig. 4.87 Defining the end point of the mirror axis

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.

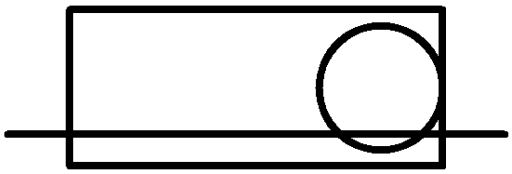


Fig. 4.88
Mirrored object

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.

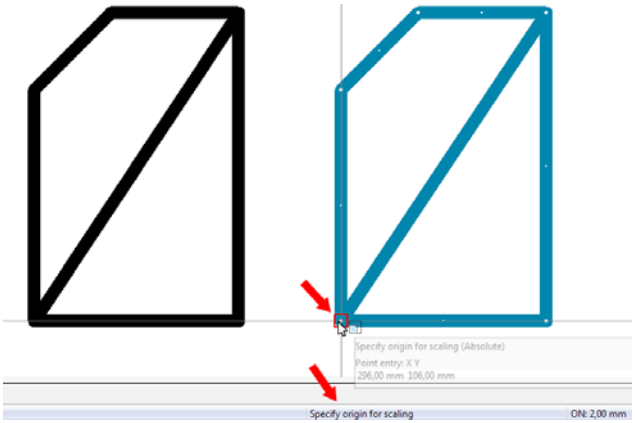


Fig. 4.89 Defining the origin for scaling

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

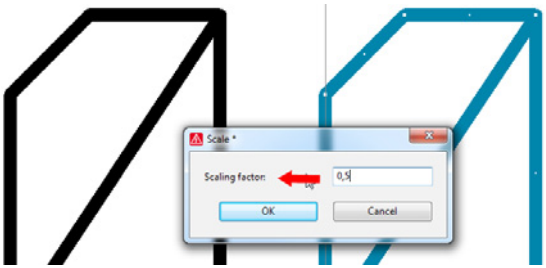


Fig. 4.90 Setting the scaling factor

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

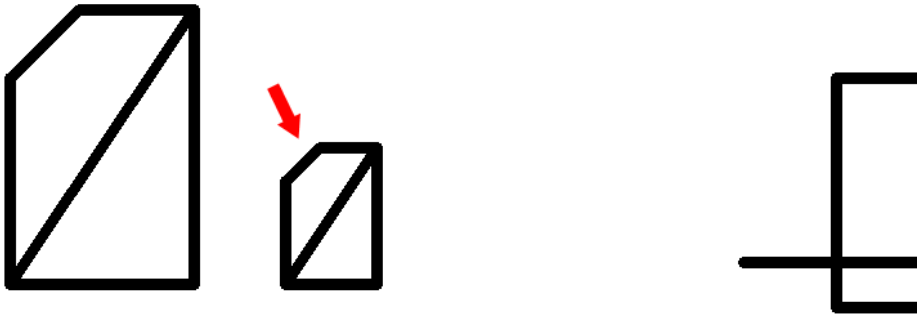


Fig. 4.91 Scaled object

When using the **Scaling factor** note that EPLAN expects a factor less than 1 to reduce (half the size smaller, for example, which is 0.5) and a factor greater than 1 to enlarge (for example, larger by a quarter, which is 1.25). 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.

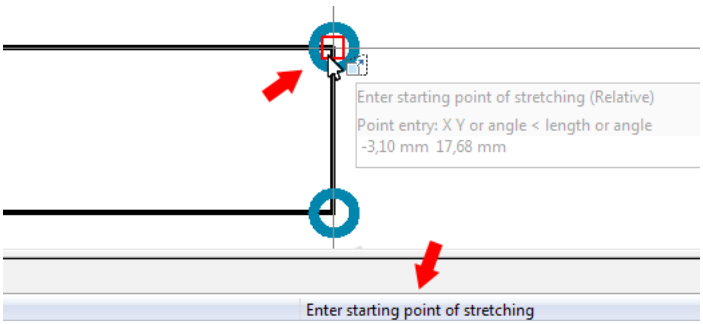


Fig. 4.92
Defining the starting point of the stretch

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

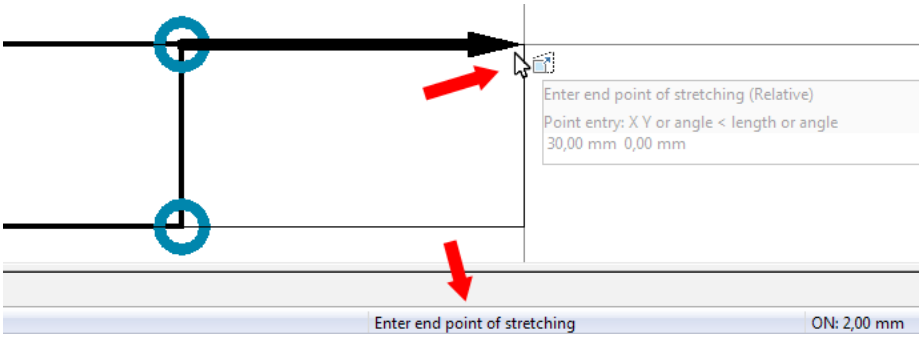


Fig. 4.93
Defining the end point of the stretch

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

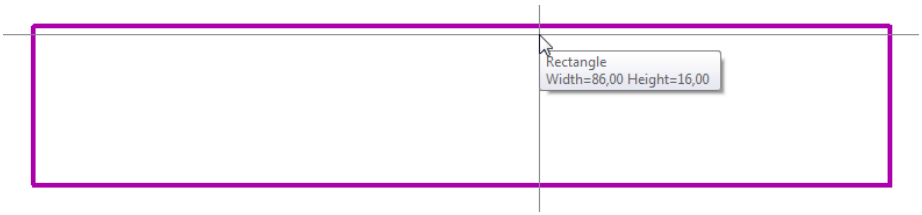


Fig. 4.94
Stretched object

Of course, rectangles cannot only be changed in length, but also in any direction.

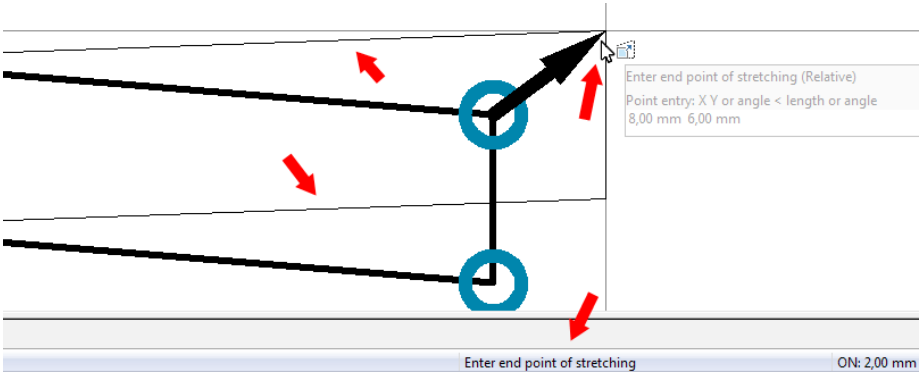


Fig. 4.95 Lengthening or shortening a different object (polylines)

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.

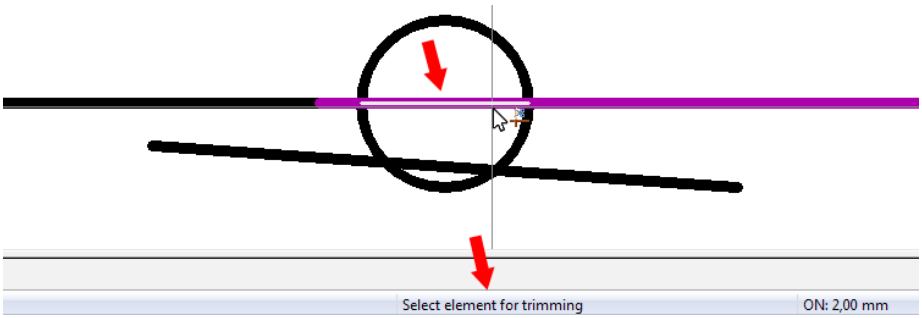


Fig. 4.96 Selecting the element to be removed

If, as shown in Fig. 4.96, you click the small section within the circle, EPLAN will cut it out. Then, the next section can be cut out, or the current **CUT OFF** function is terminated by pressing the **ESC** key.

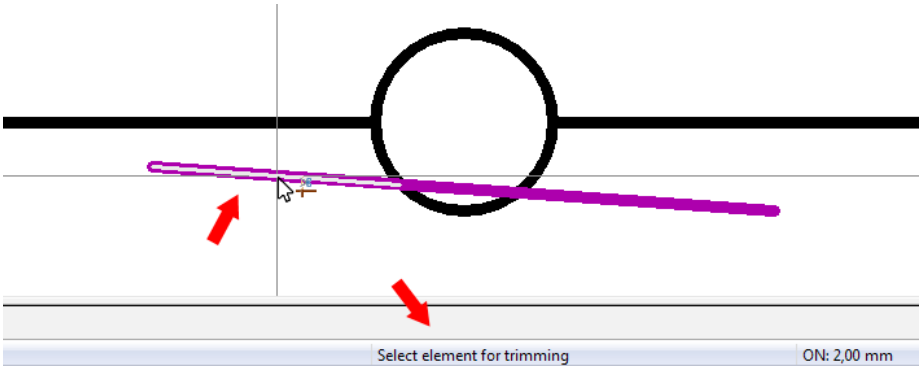


Fig. 4.97 Cut-off section

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 length 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.

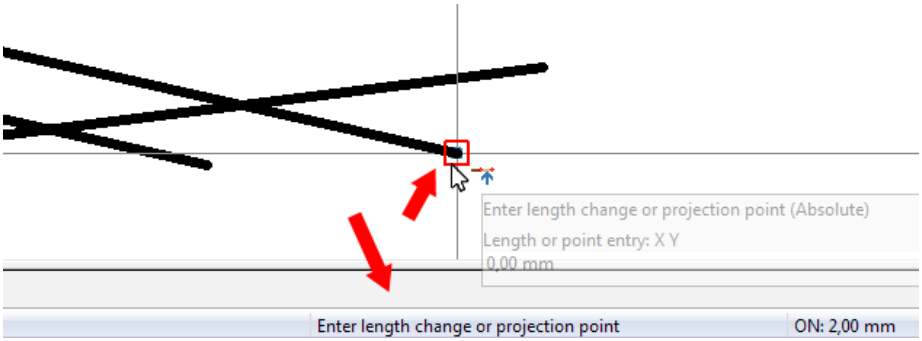


Fig. 4.98 Changing the length of a line

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.

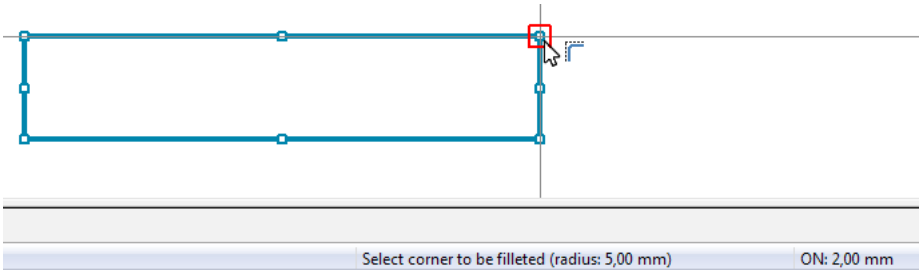


Fig. 4.99 Filleting corners

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

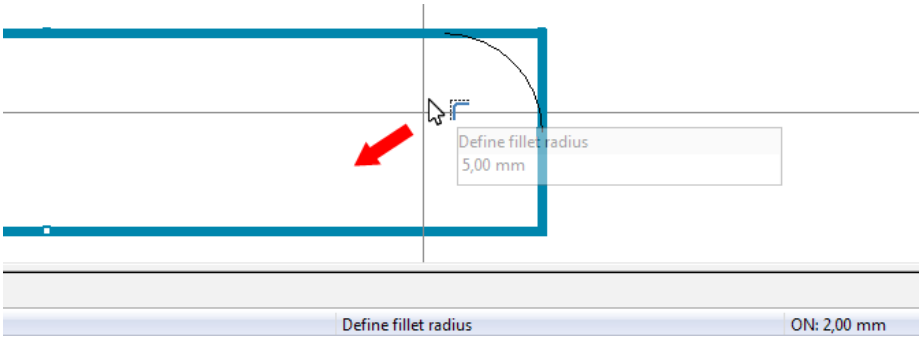


Fig. 4.100 Filleting the next corner

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

EPLAN always remembers the radius most recently defined and applies it to subsequent actions.

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.

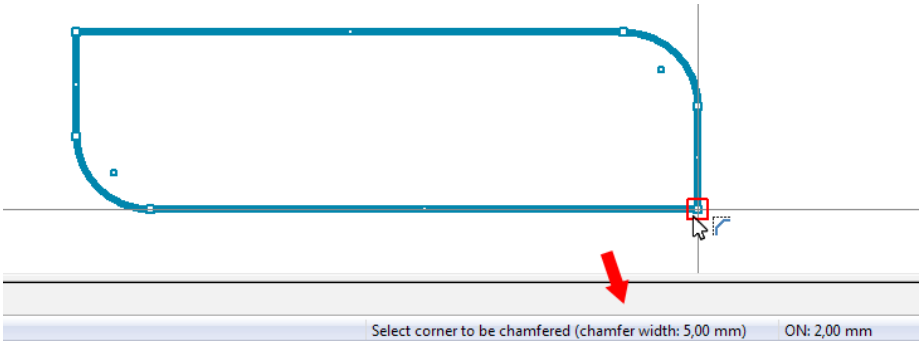


Fig. 4.101 Chamfering a corner

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.

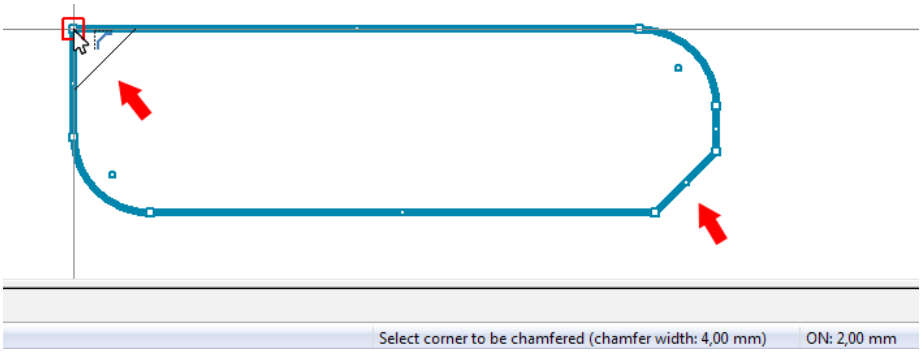


Fig. 4.102 Chamfer next corner

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).

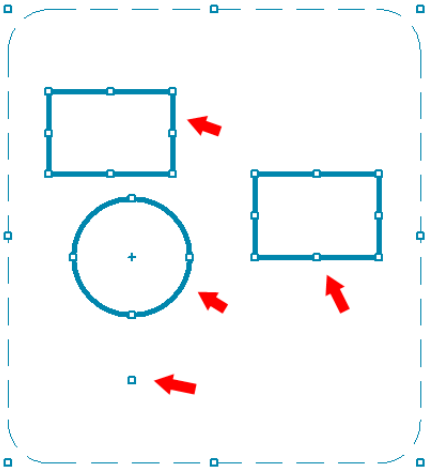


Fig. 4.103 Individual objects

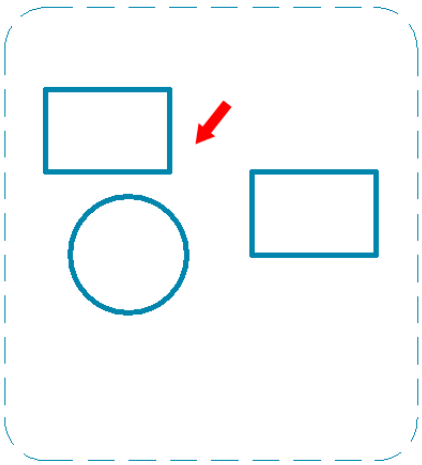


Fig. 4.104 Grouped objects

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 if you click anywhere on the grouped elements). 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.

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.

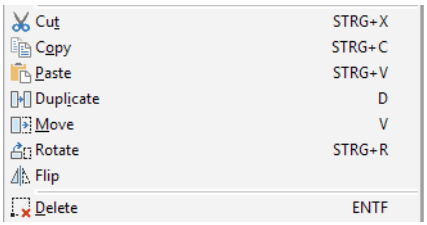


Fig. 4.105
Editing functions

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.

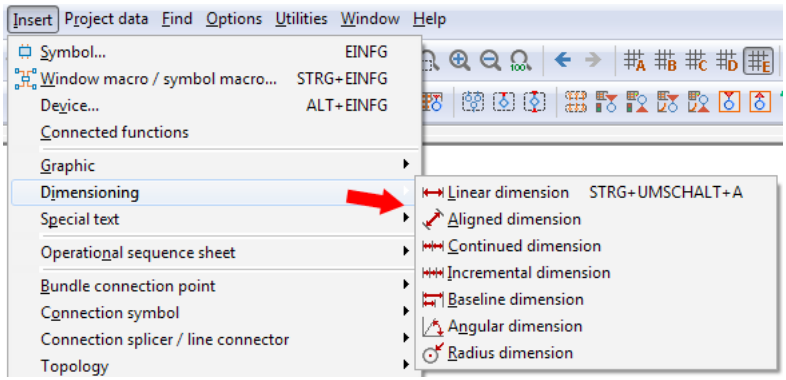


Fig. 4.106 Dimensioning functions

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.

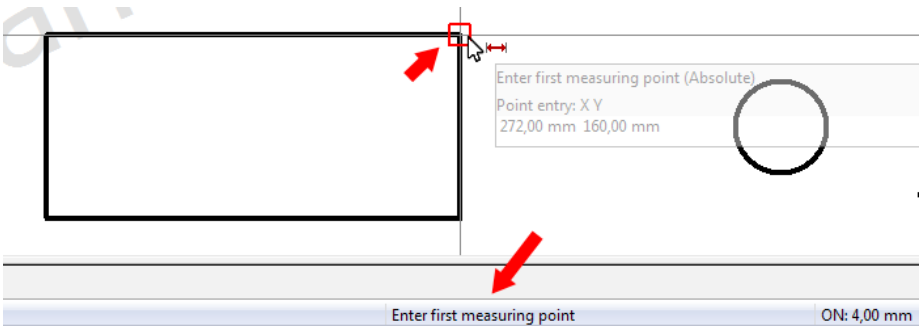


Fig. 4.107 Dimensioning starting point

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.

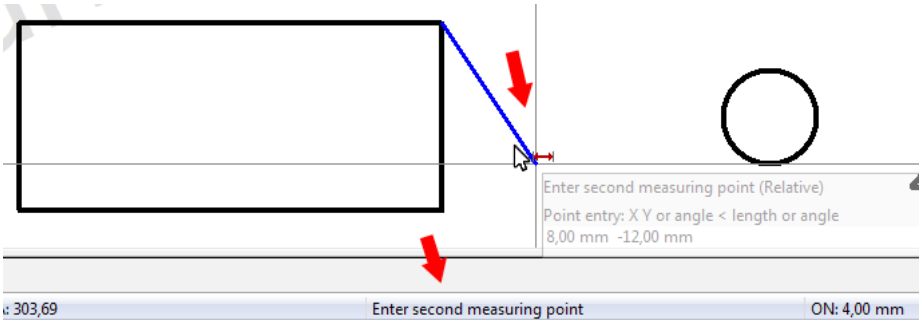


Fig. 4.108 Defining the second dimension point

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

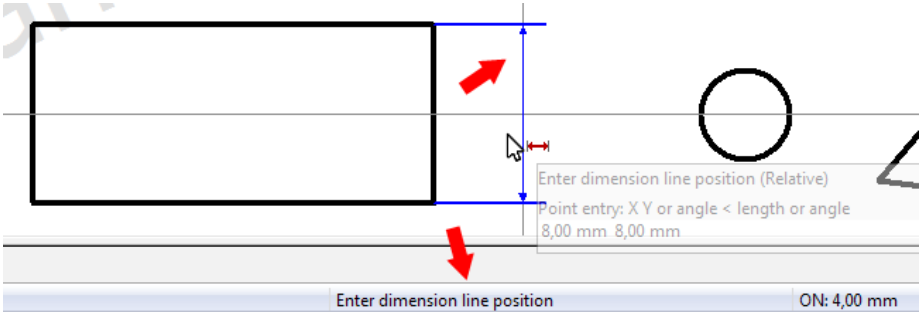


Fig. 4.109 Defining the position of the dimension line

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.

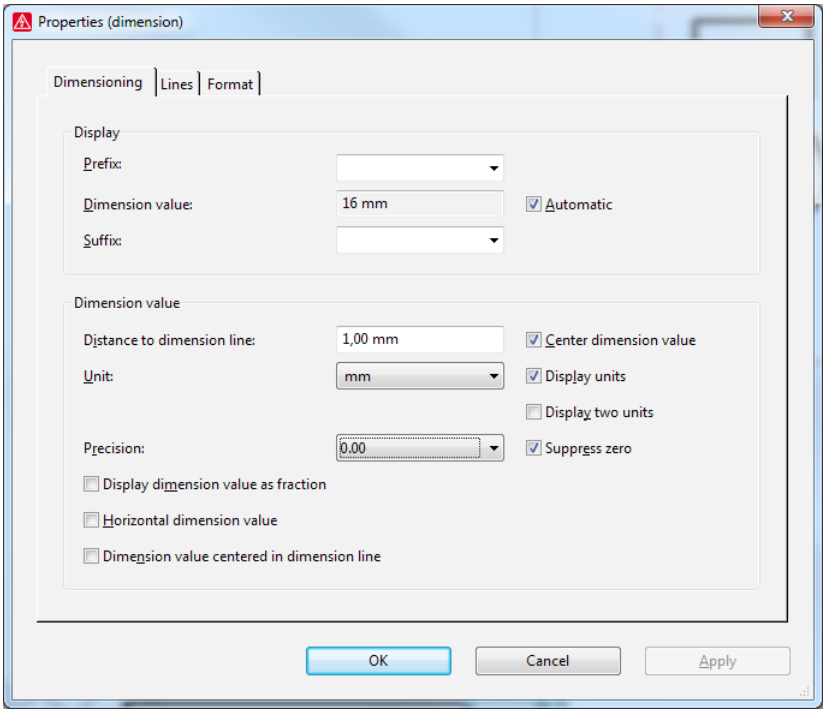


Fig. 4.110 Setting the current dimension

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

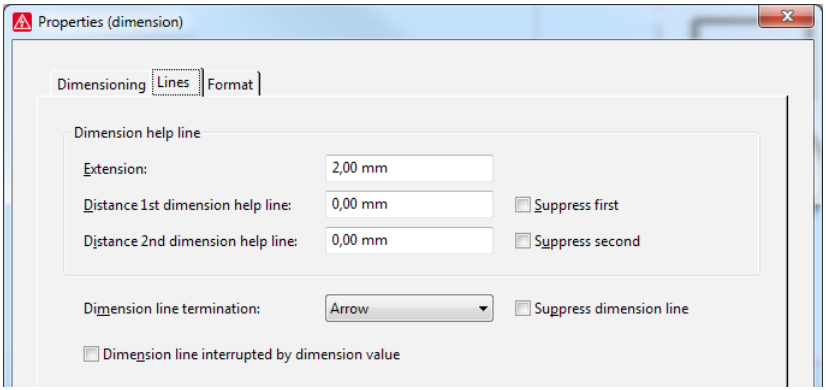


Fig. 4.111 Setting the dimension lines

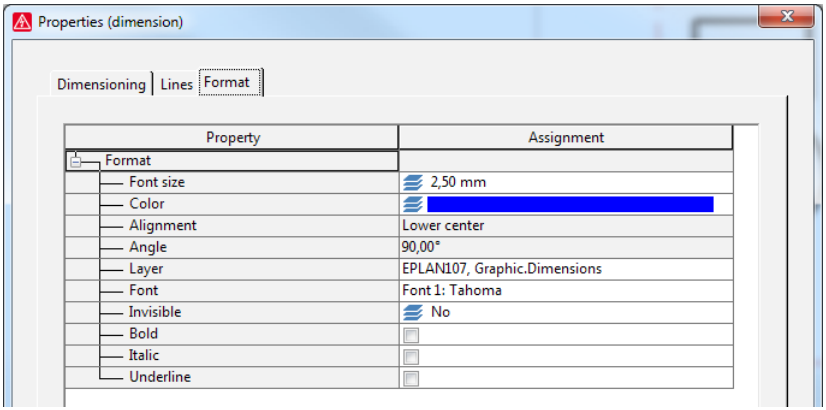


Fig. 4.112 Setting the format (dimension values, etc.)

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.

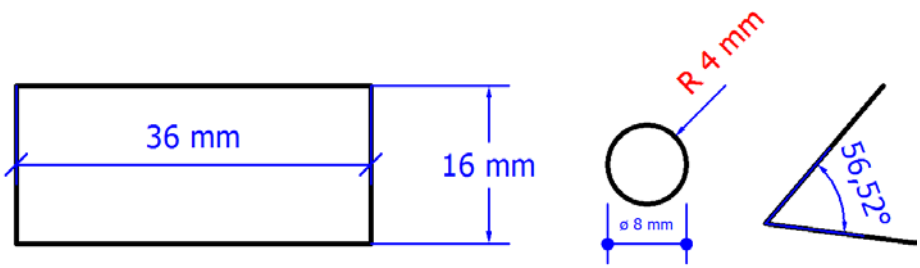


Fig. 4.113 Examples of different dim ensions

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 (**INSERT/GRAPHIC/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).

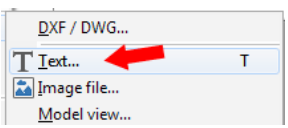


Fig. 4.114 Free graphical text

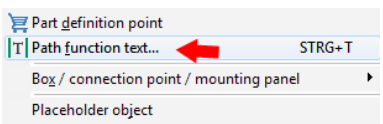


Fig. 4.115 Functional text in the Insert menu

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 an encoded text file to know which text entry the number refers to.

4.7.1 Normal (free) texts

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.

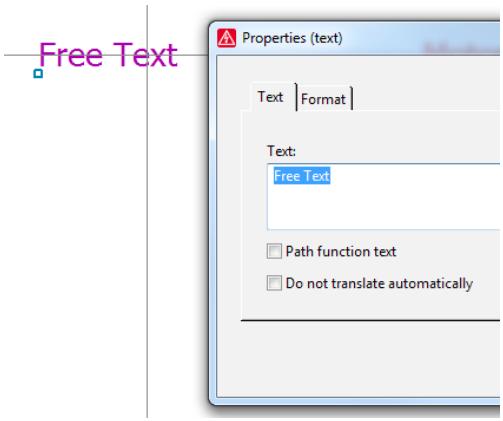


Fig. 4.116 Free text

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

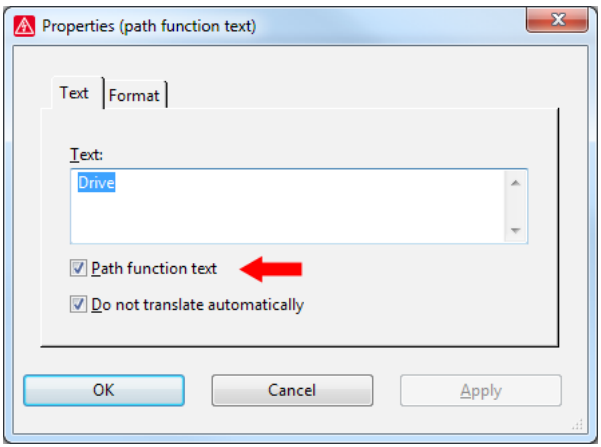


Fig. 4.117 Path function text selection box

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

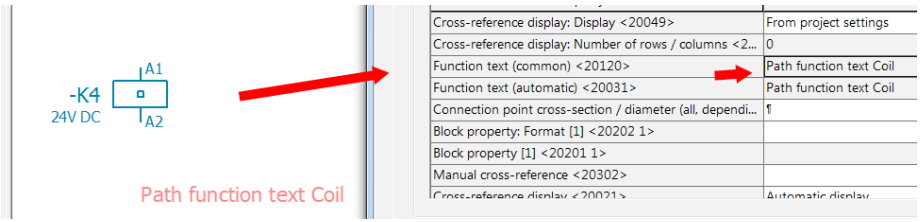


Fig. 4.118 Path function text

EPLAN uses the path function text for embedding in different reports (depending on the structure of the form) or for generating further reports. 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.

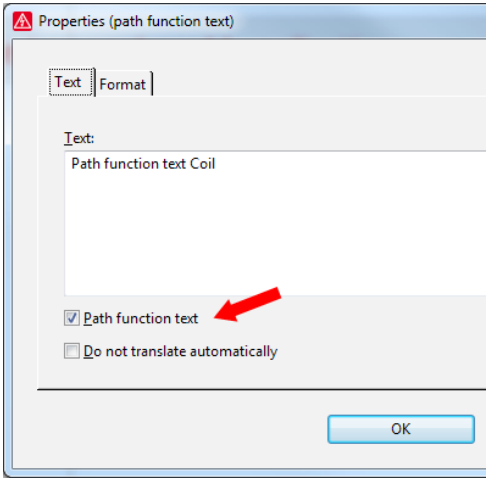


Fig. 4.119
Path function text

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 and save 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* (always to the left and right).

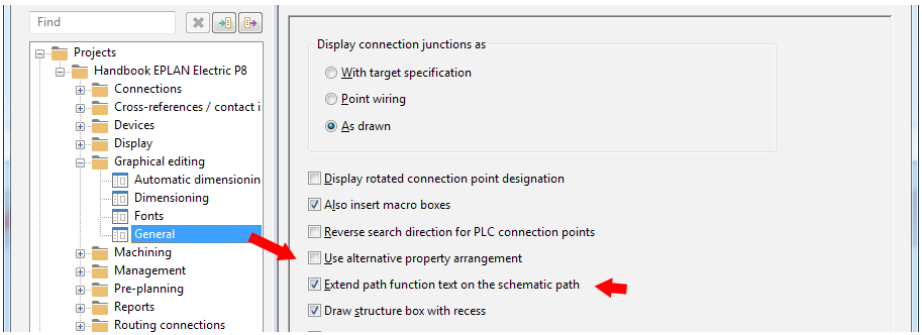


Fig. 4.120 Extend path function text setting

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*.

Apart from this extension setting, there is a second setting for path function texts that, however, concern only the PLC connection points. This setting is a project setting (*Device/PLC* node) and is deactivated or activated there.

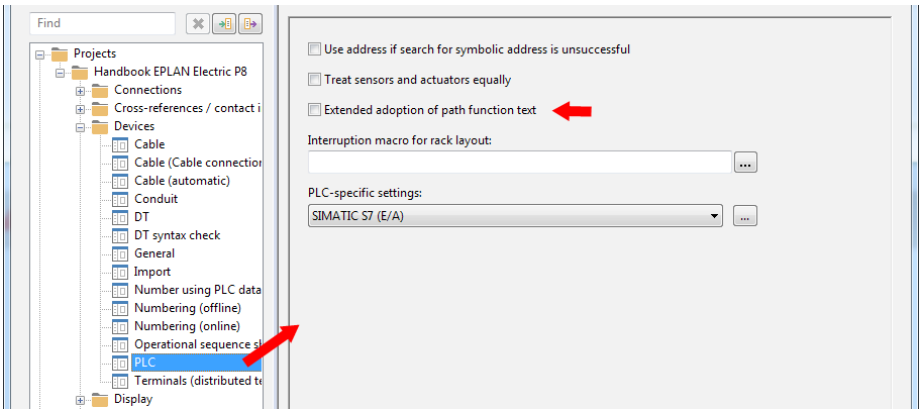


Fig. 4.121 PLC-extended function text import project setting

This is how it works when this setting has been activated: If EPLAN does not find a function text in the path, and no function texts (automatic) of sensors/actuators connected either, EPLAN will search the previous paths (search direction to the left) for function texts and import the first one found.

This setting should be checked should EPLAN transfer 'wrong' function texts to PLC connection points. It may be that a function text is missing in fact, or the existing text is not a function text.

4.7.3 Special texts

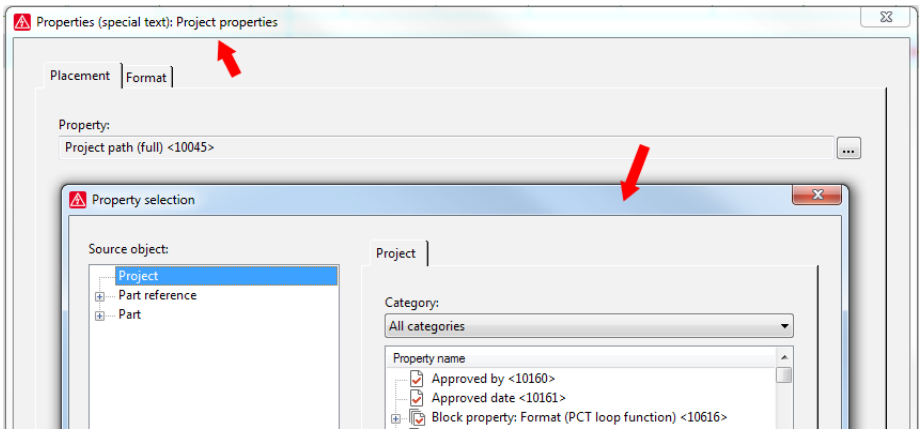


Fig. 4.122 Special texts

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.

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

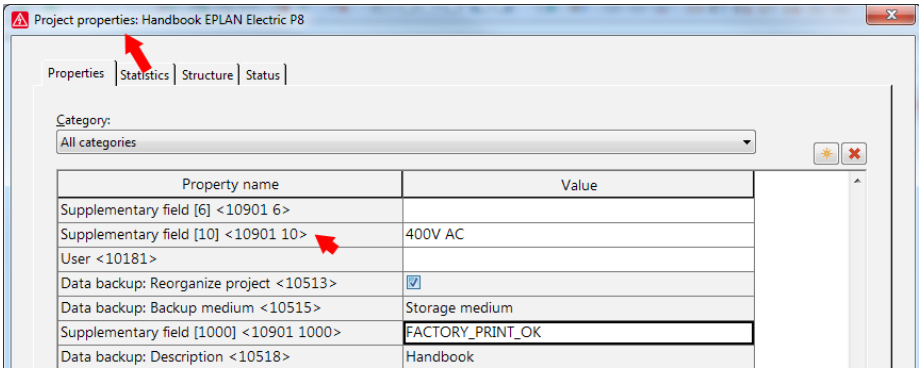


Fig. 4.123 Supplementary field [10] <10901 10> project property

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**.

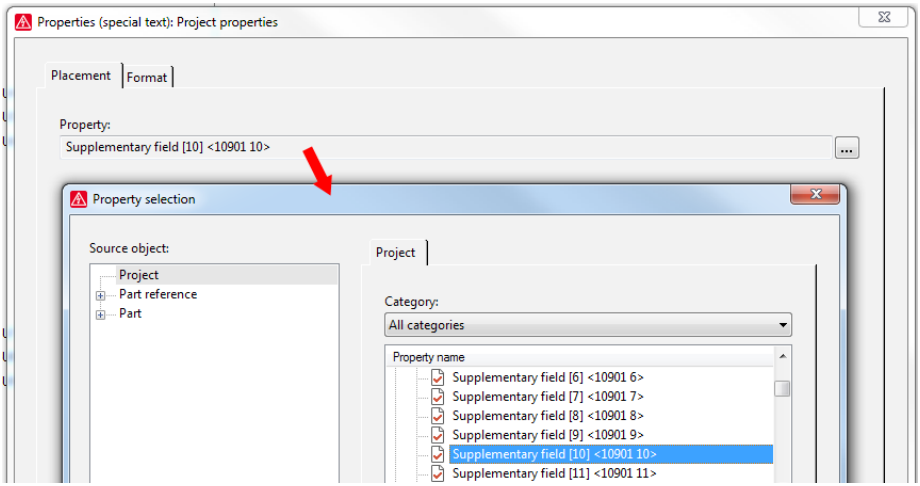


Fig. 4.124 Selecting project properties

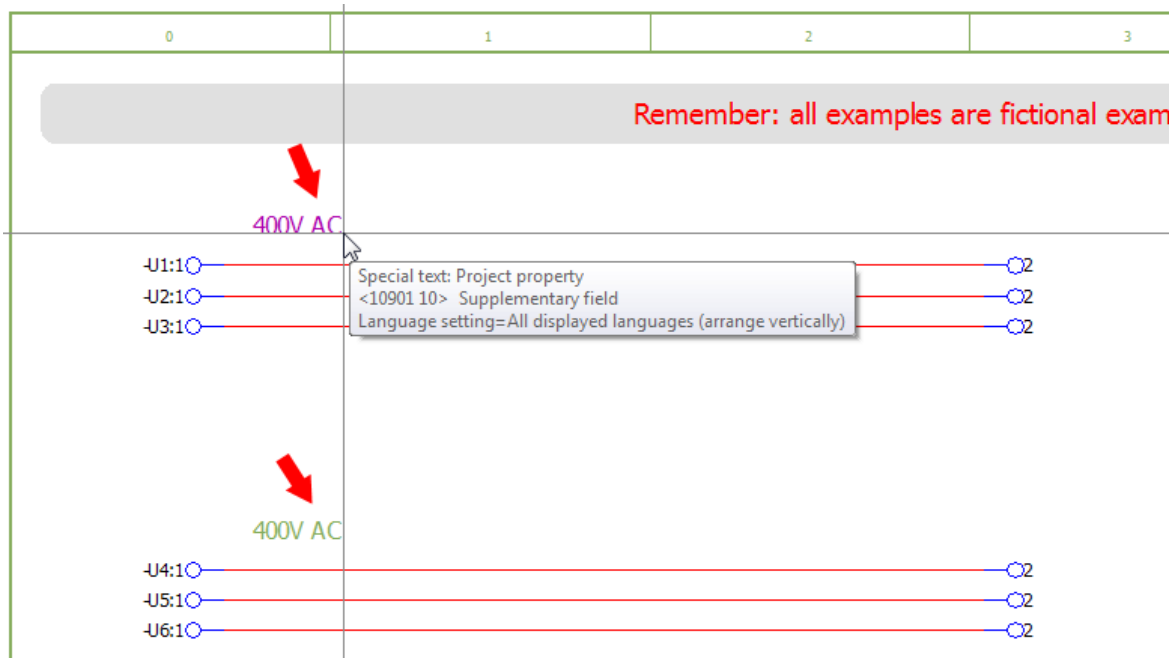


Fig. 4.125 Placed project property

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.

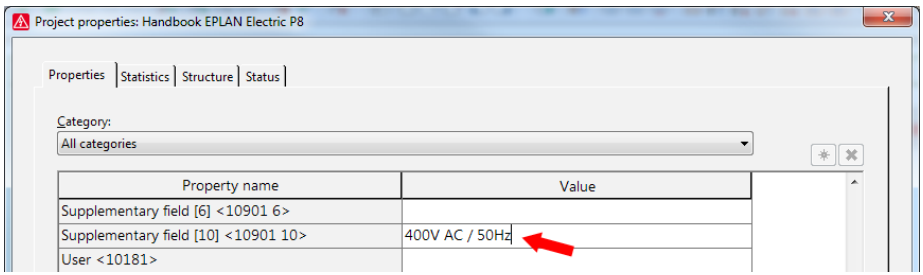


Fig. 4.126 Changed input in the project property

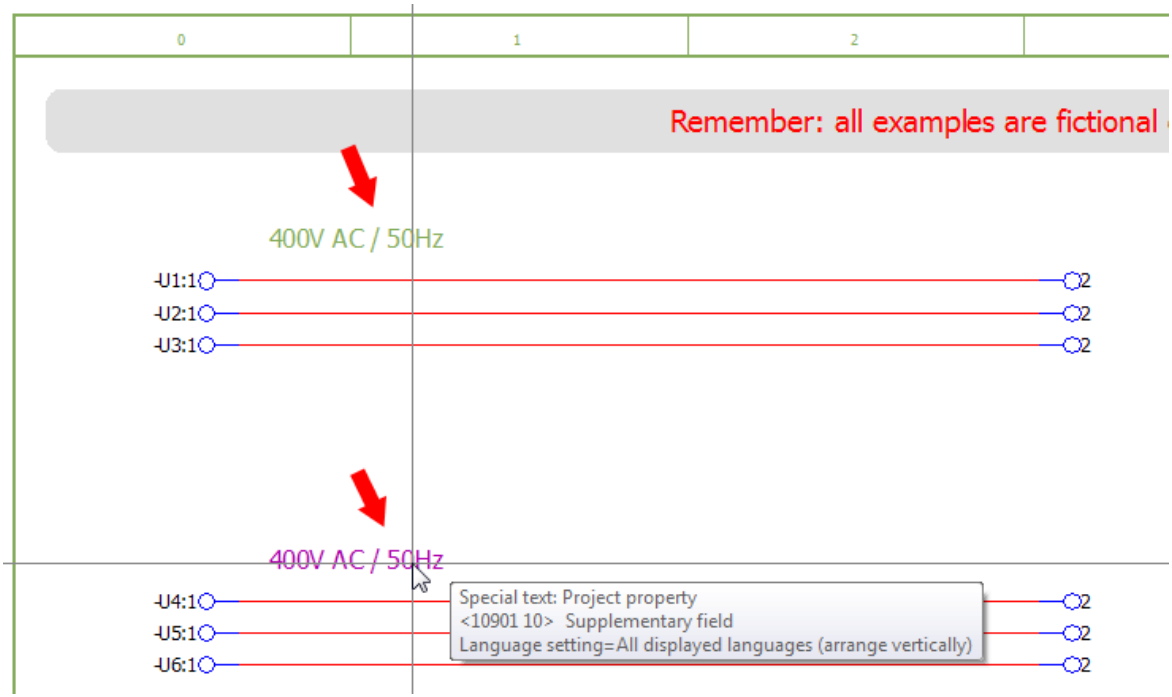


Fig. 4.127 Changed display in the project

4.7.4 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.

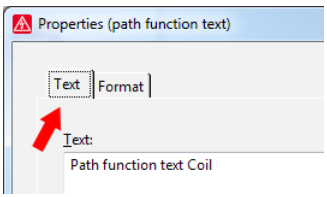


Fig. 4.128
Text input field

You call up the popup menu using the right mouse 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**.

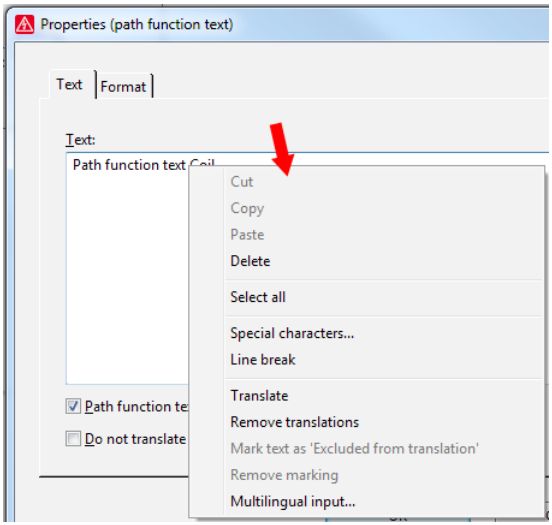


Fig. 4.129
Popup menu

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*.

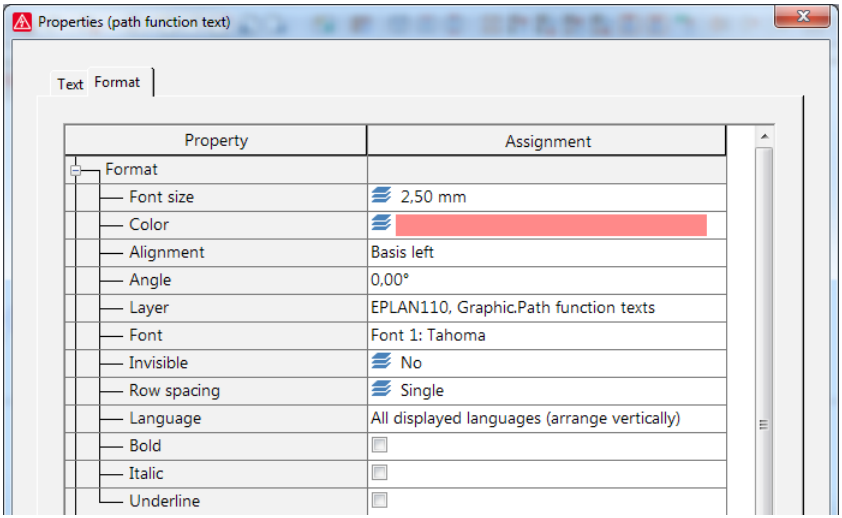


Fig. 4.130 Formatting options for texts in the Format area

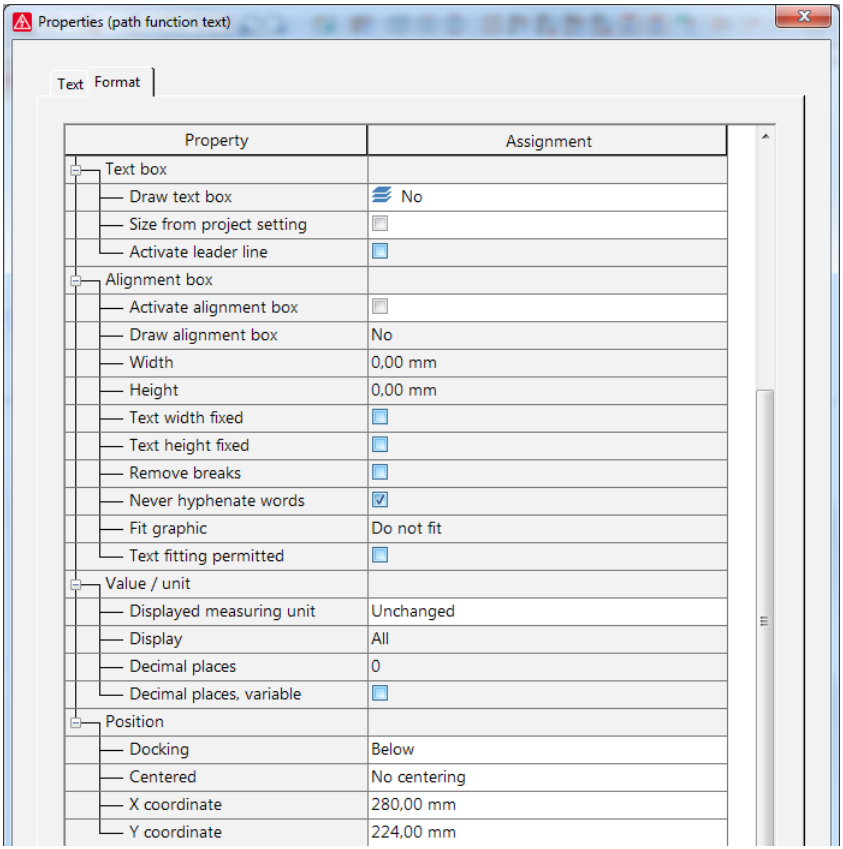


Fig. 4.131 Further options (Text box, Alignment box, Value / unit and Position)

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

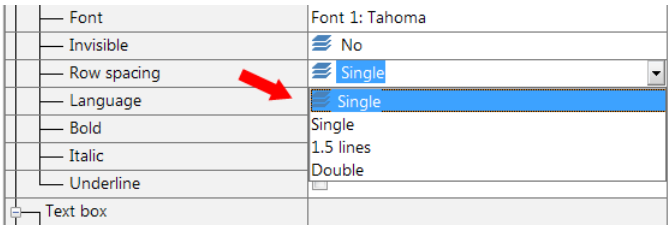


Fig. 4.132 Row spacing format property

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. If you enter your own value, thus overwriting the layer value, the symbol in front of the original entry disappears.

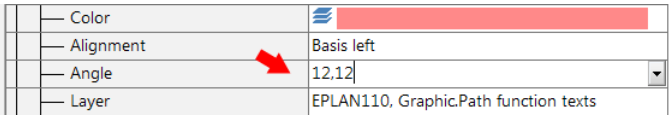


Fig. 4.133 Free input fields

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

4.7.4.2.1 Format tab – Format node – Language

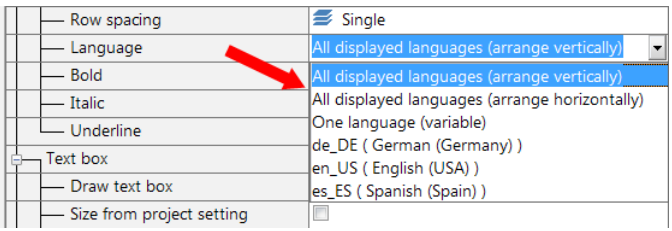


Fig. 4.134 Language selection list

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.

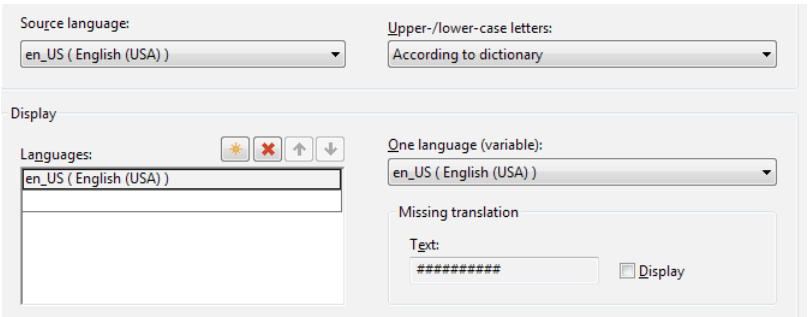


Fig. 4.135 Some translation settings

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. 4.136 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.

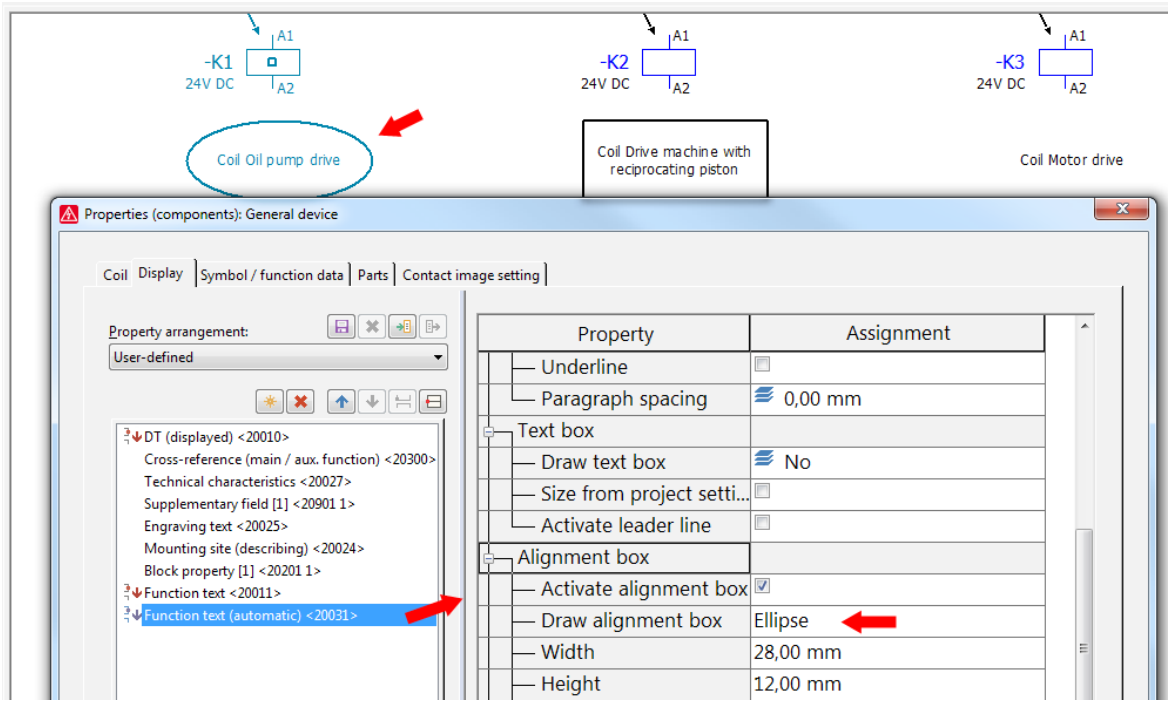


Fig. 4.136 Examples of an activated alignment box

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.

Property	Assignment
Format	
Font size	1,80 mm
Color	[redacted]
Alignment	Middle center
Angle	0,00°
Layer	EPLAN501, Property placement.Descri...
Font	Font 1: Tahoma
Invisible	No
Row spacing	Single
Language	All displayed languages (arrange vertic...
Bold	
Italic	
Underline	
Paragraph spacing	0,00 mm
Text box	
Draw text box	No
Size from project setting	

Fig. 4.137 Large row height

Property	Assignment
Format	
Font size	1,80 mm
Color	[redacted]
Alignment	Middle center
Angle	0,00°
Layer	EPLAN501, Property placement.D...
Font	Font 1: Tahoma
Invisible	No
Row spacing	Single
Language	All displayed languages (arrange ...
Bold	
Italic	
Underline	
Paragraph spacing	0,00 mm
Text box	
Draw text box	No
Size from project setting	
Activate leader line	
Alignment box	
Activate alignment box	
Draw alignment box	Ellipse
Width	28,00 mm
Height	12,00 mm

Fig. 4.138 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.

If you want to return to the intended EPLAN row height setting, simply click **FONT SIZE 100%** in the popup menu. EPLAN then restores the default setting.

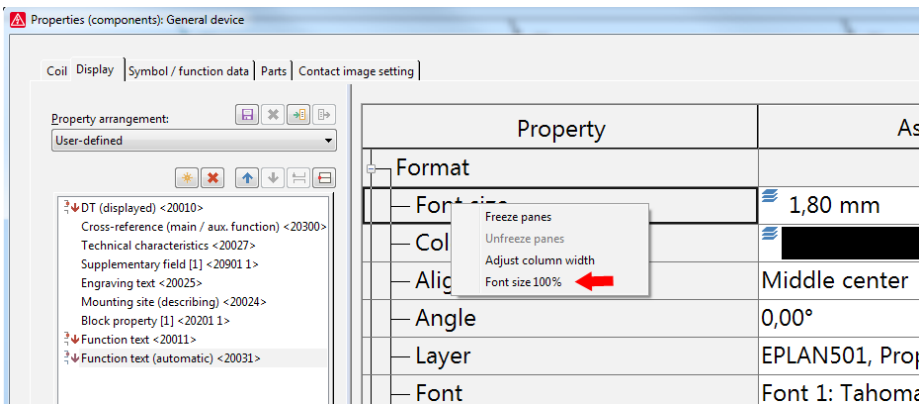


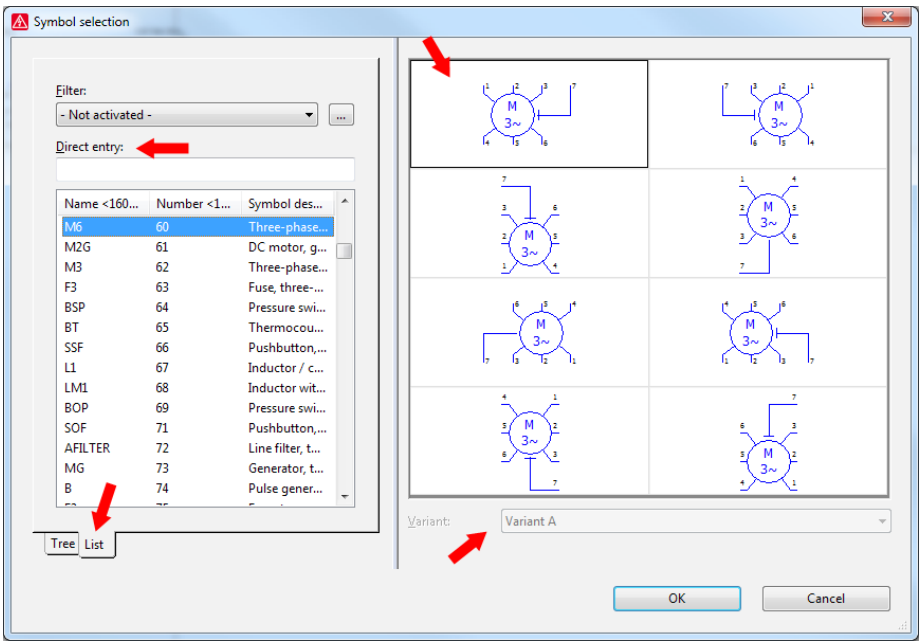
Fig. 4.139 Restore default font size 100 % setting

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. 4.140 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

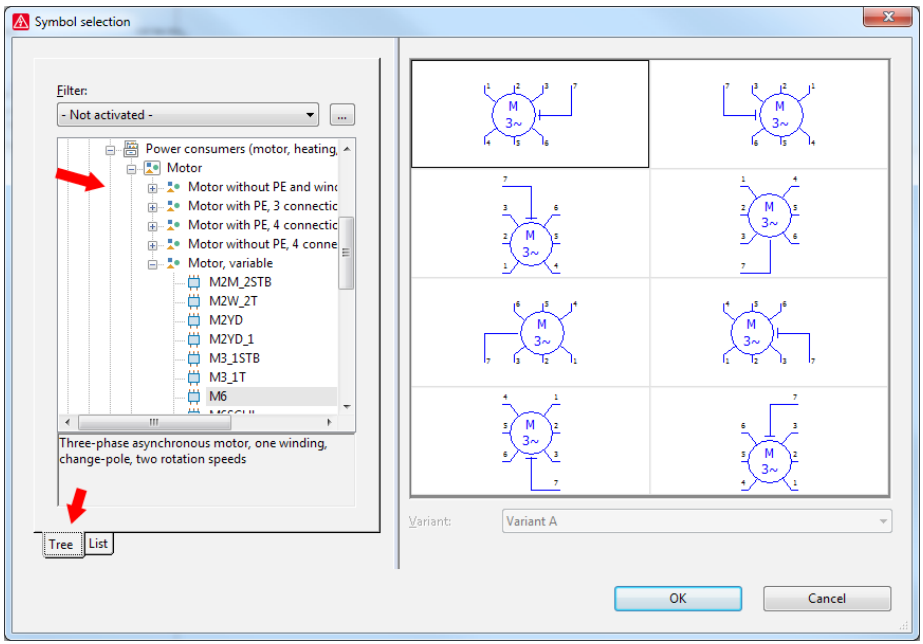


Fig. 4.141 Symbol selection via tree view

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.

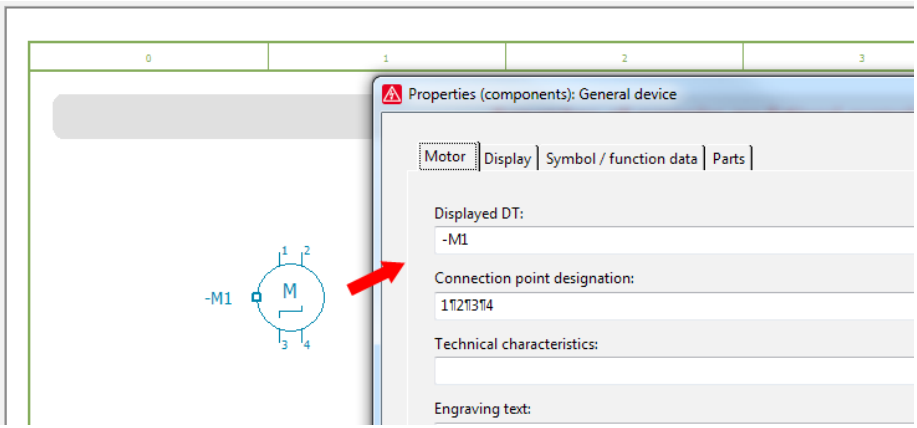


Fig. 4.142 Properties (components) dialog

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.

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).

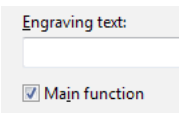


Fig. 4.143 Main function

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.

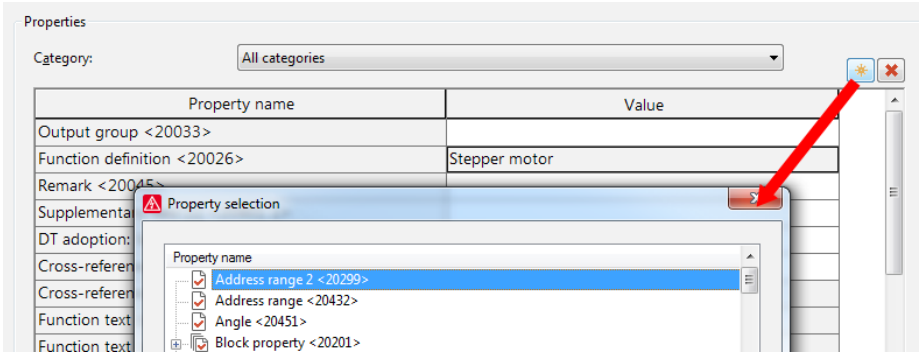


Fig. 4.144 Add new properties

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 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.

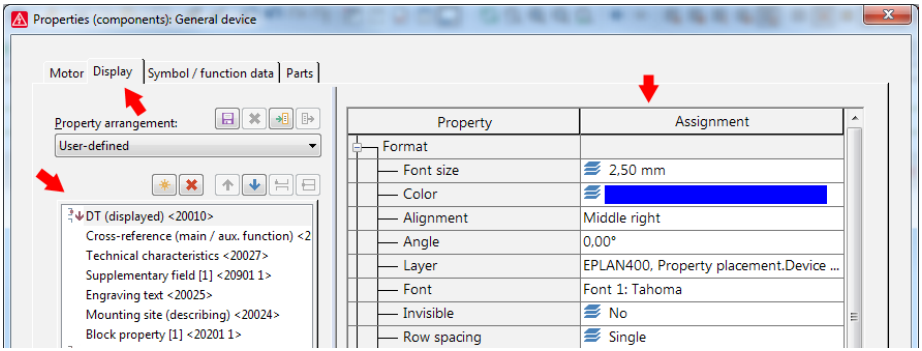


Fig. 4.145 Display tab and its options

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. 4.146, 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.

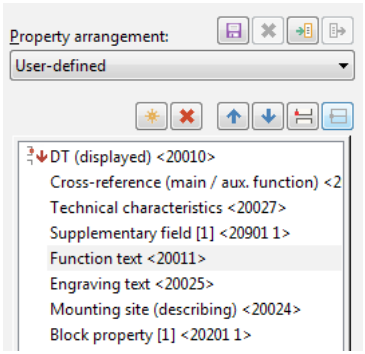


Fig. 4.146
Related properties

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".

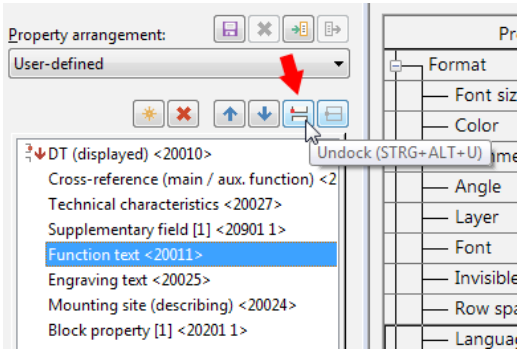


Fig. 4.147
Undocking a property

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.

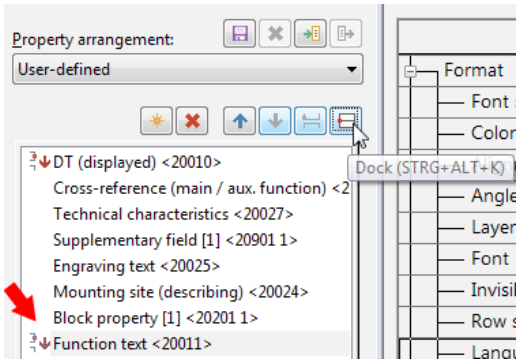


Fig. 4.148
Undocked and moved property

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.

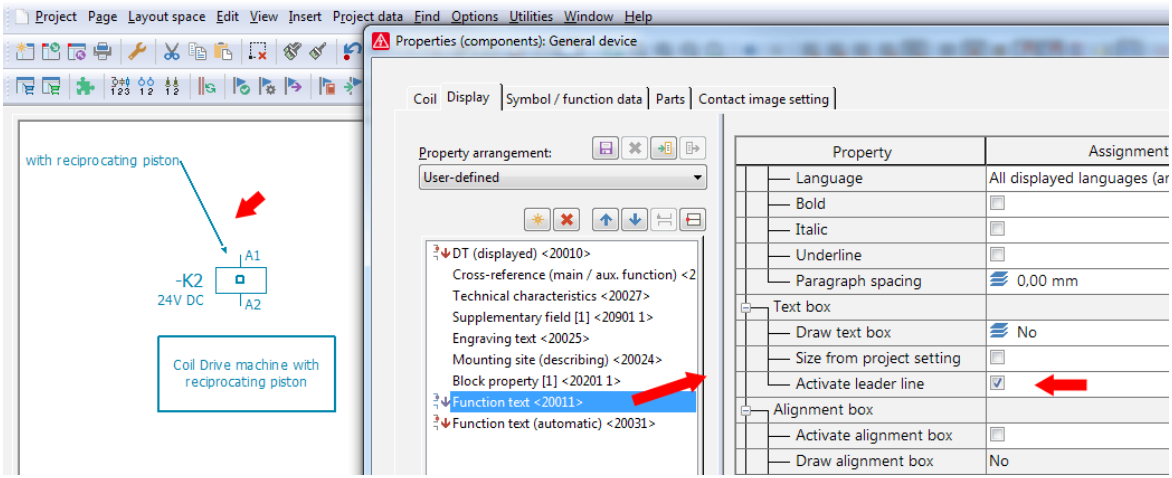


Fig. 4.149 Example of activated leader line

4.8.4 Symbol/function data tab

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

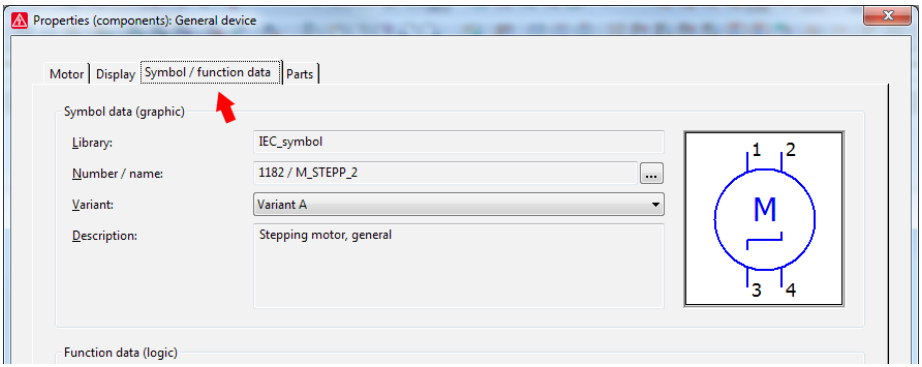


Fig. 4.150 Graphical information for the symbol

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.

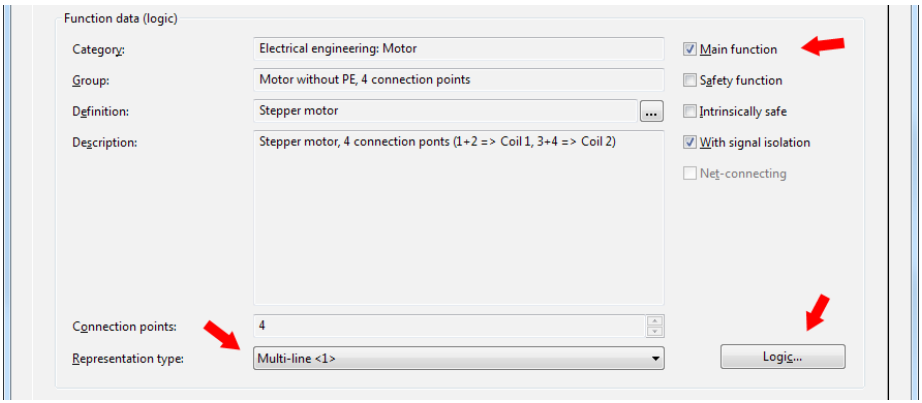


Fig. 4.151 Logical symbol functions

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.

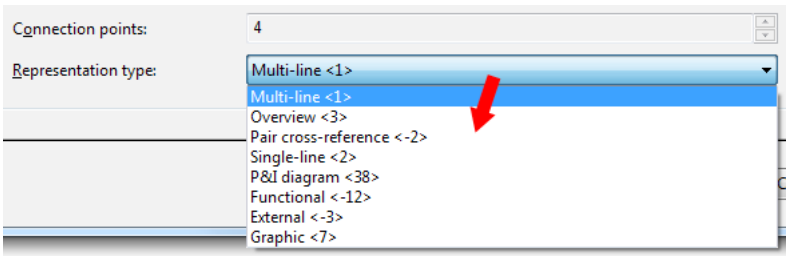


Fig. 4.152 Representation type selection

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.

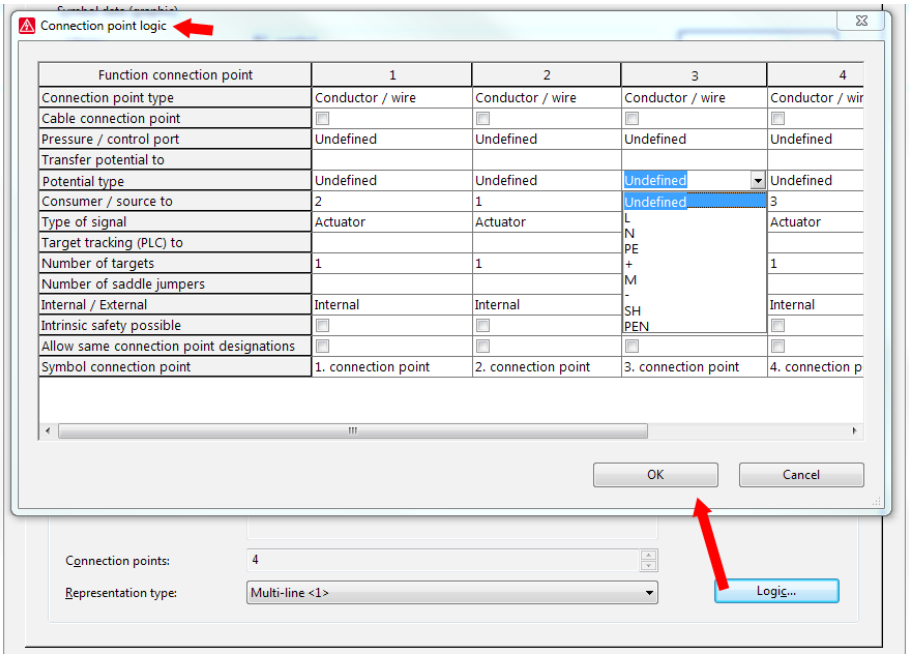


Fig. 4.153 Logical functions of individual connection points

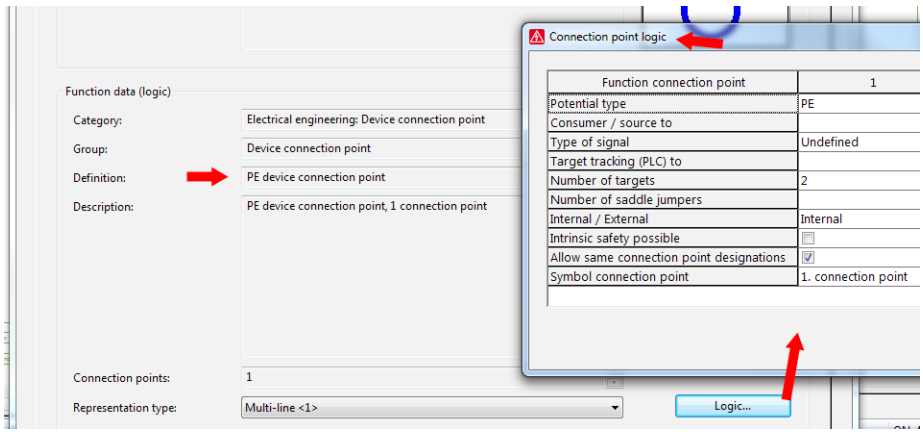


Fig. 4.154 Allow same designation

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 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 ... button that appears to switch to parts management.

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.

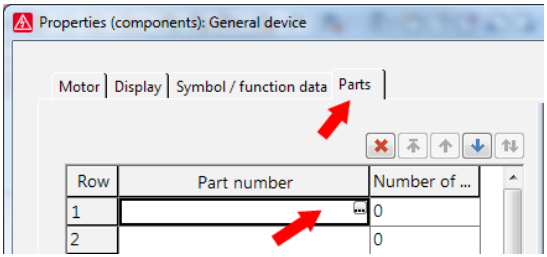


Fig. 4.155
Simple part selection

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.

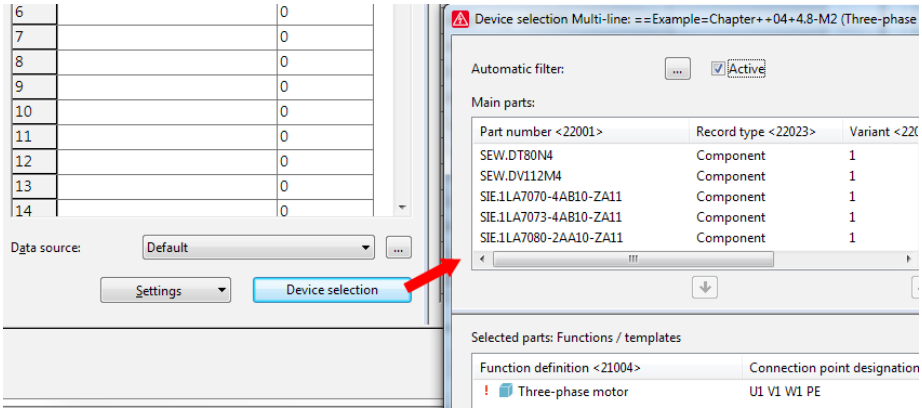


Fig. 4.156 Device selection

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.

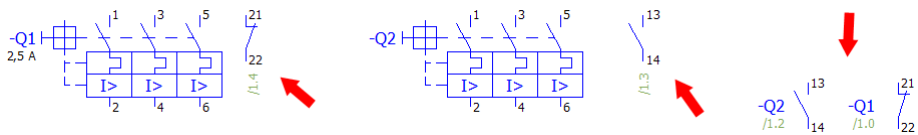


Fig. 4.157 Automatic cross-referencing

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.

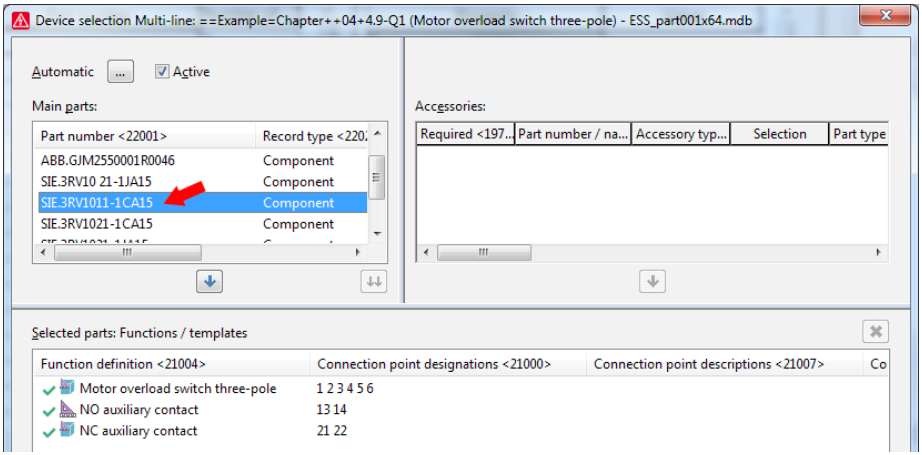


Fig. 4.158 New device selection

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.

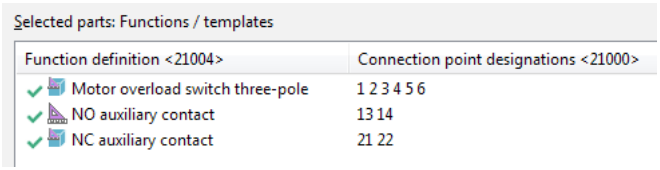


Fig. 4.159 Selected part with its function template

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.

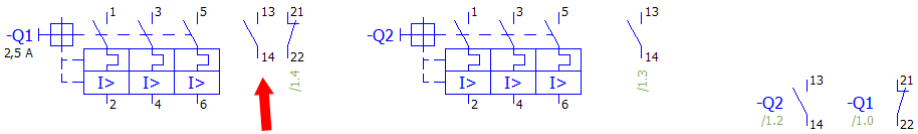


Fig. 4.160 Complete contact image

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.

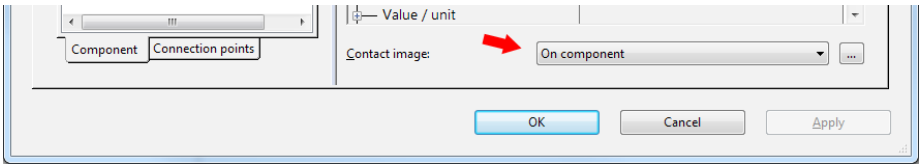


Fig. 4.161 Contact image setting

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.

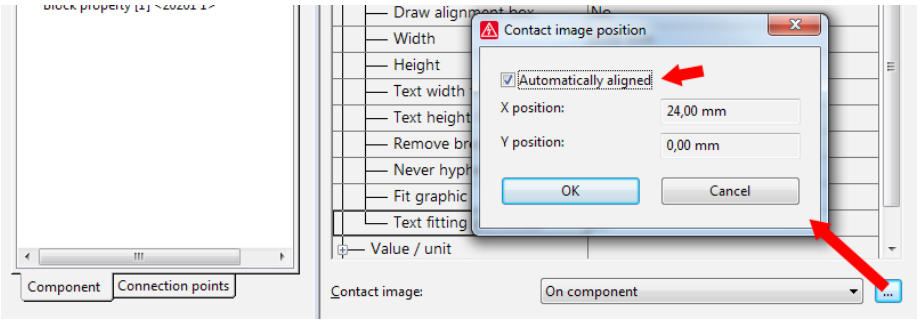


Fig. 4.162 Manual editing of the contact image position

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.

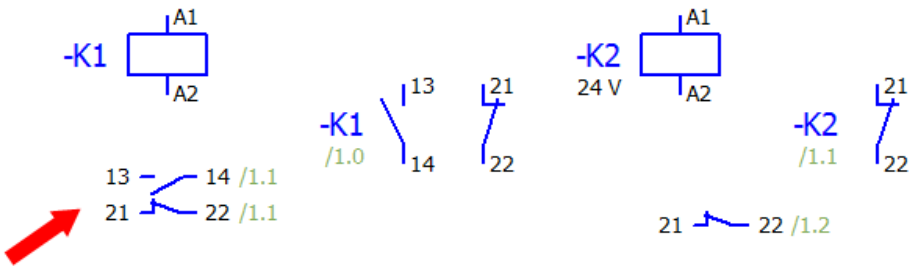


Fig. 4.163 Automatically generated contact image of the functions in use

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.

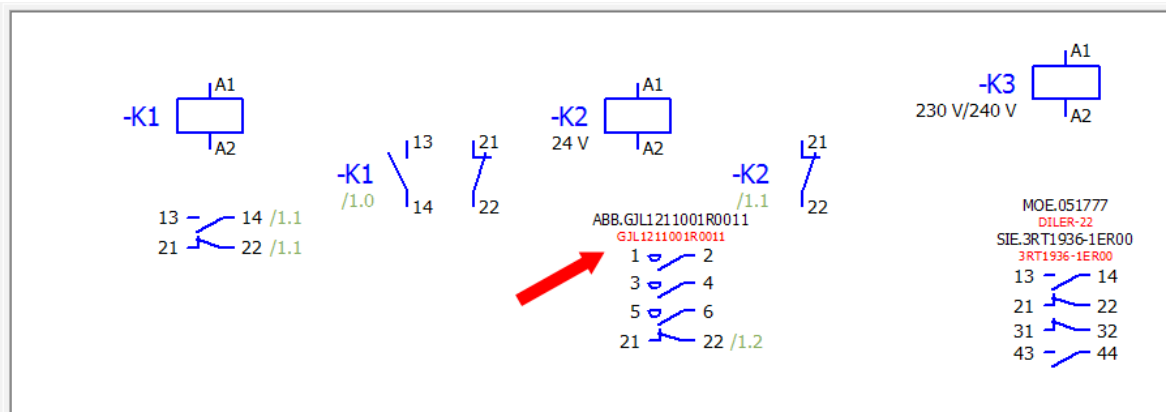


Fig. 4.164 Complete contact image following device selection

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*).

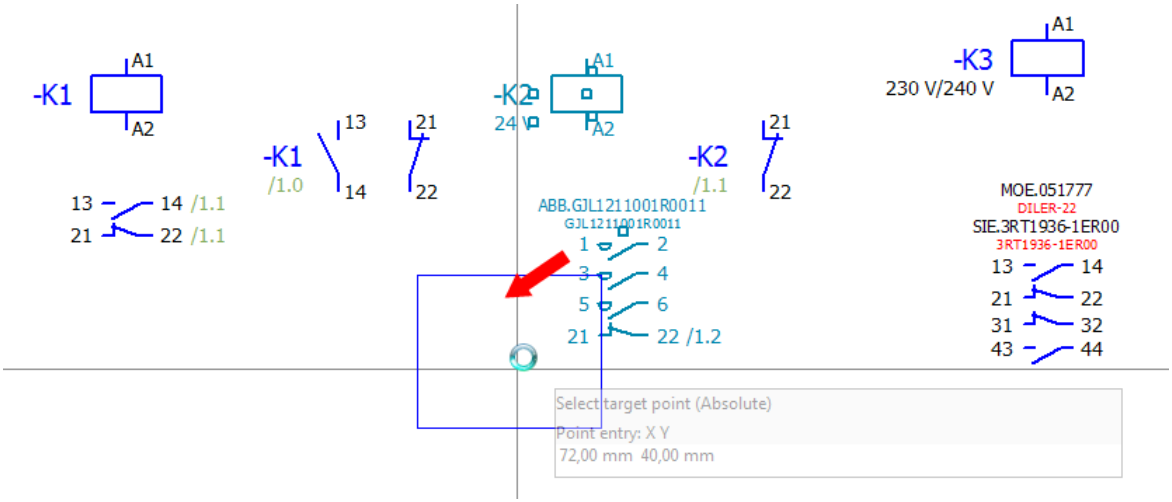


Fig. 4.165 Moving the contact image manually

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.**

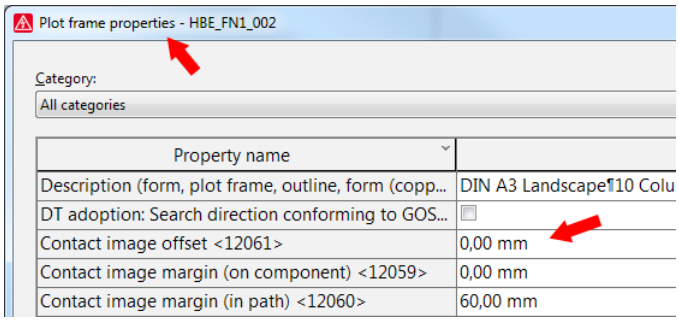


Fig. 4.166 Properties of contact image offsets

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).

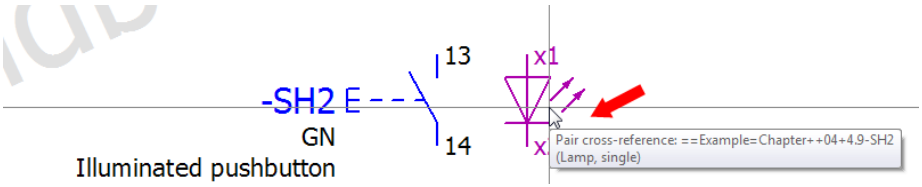


Fig. 4.167 Insert and designate symbols

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**.

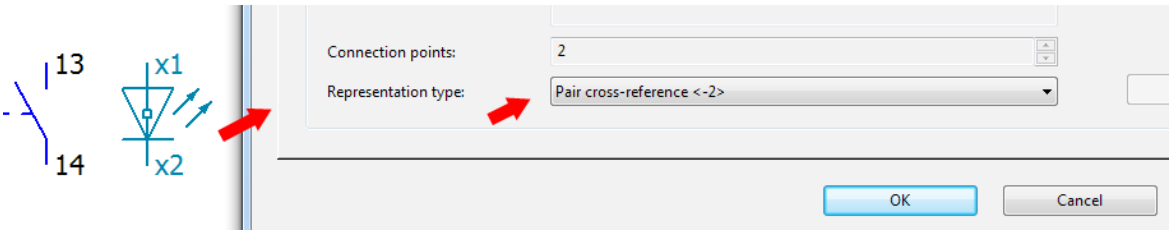


Fig. 4.168 Setting the pair cross-reference representation type

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**.

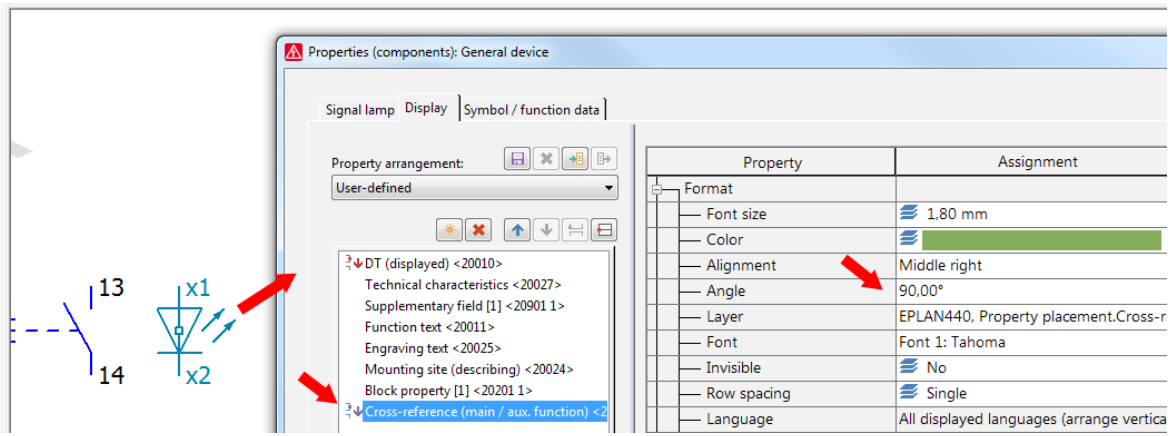


Fig. 4.169 Setting appropriate values for the cross-reference

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** popup menu or via the symbol selection.

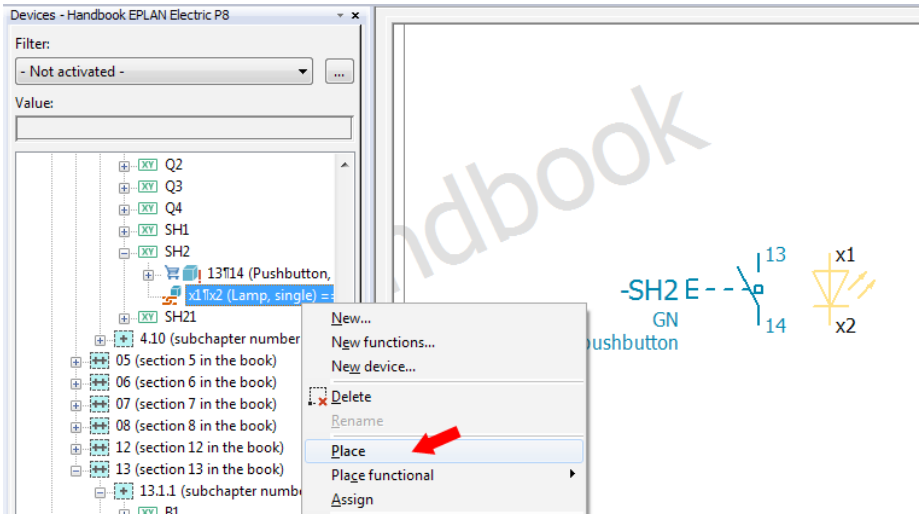


Fig. 4.170 Place multi-line function from the device navigator

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.

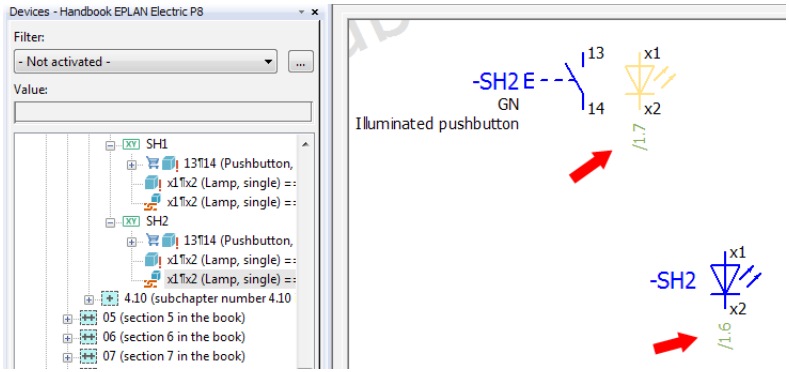


Fig. 4.171 Cross-reference with pair cross-reference representation type

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

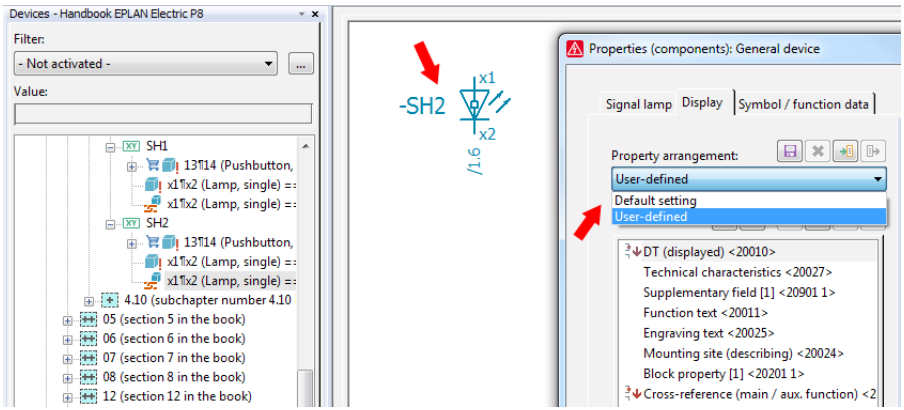


Fig. 4.172 Default property arrangement setting

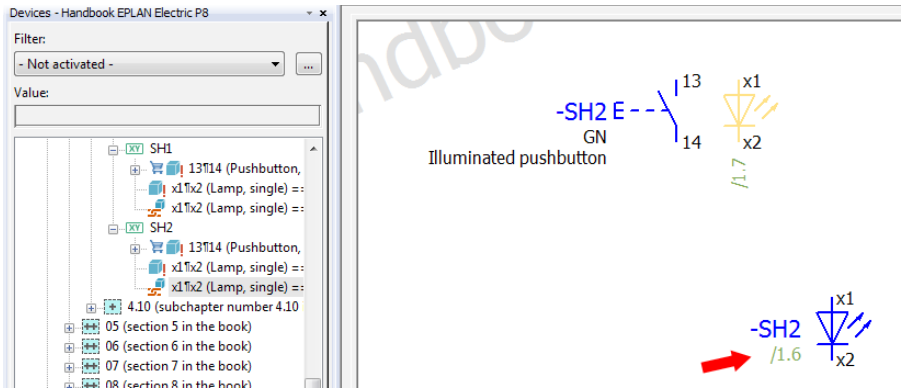


Fig. 4.173 Result

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 Contact image distributed device list

Aside from the usual methods of creating contact image types (in the path or on the component), EPLAN offers a further variant of contact images. Contact image based on a special form, distributed device list *.f45.

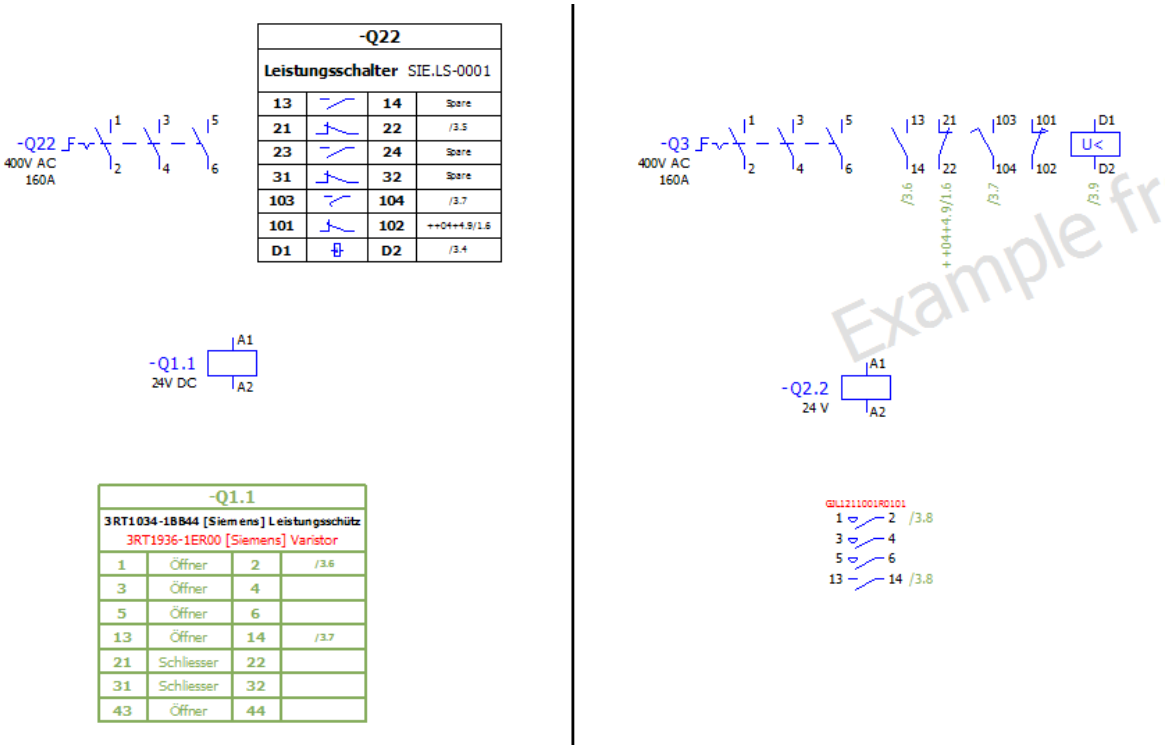


Fig. 4.174 Examples of form-based contact images

These contact images are based on the form distributed device list *.f45. The use of this type of contact images can be set directly in the symbol properties. For this, open the *Contact image setting* tab and select the **EMBEDDED FORM** option.

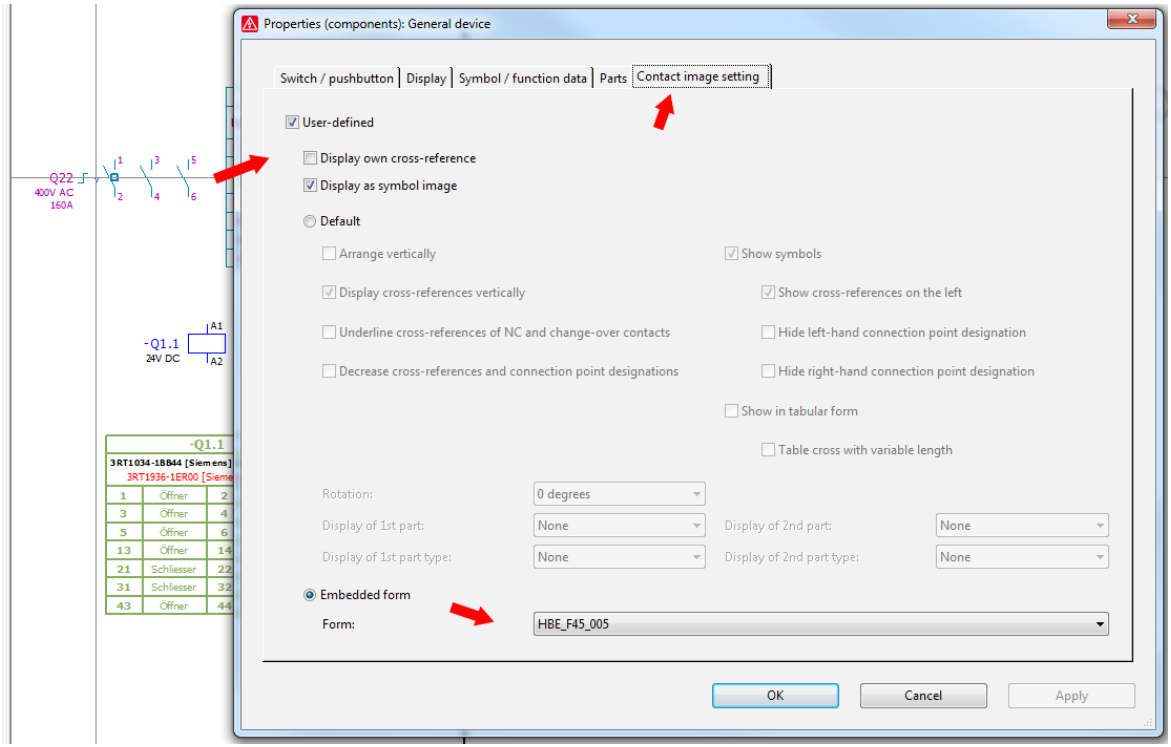


Fig. 4.175 Contact image setting tab

Now EPLAN allows you to select a form from the **FORM SELECTION LIST**, or if you click **BROWSE**, EPLAN opens the **SELECT FORM** dialog.

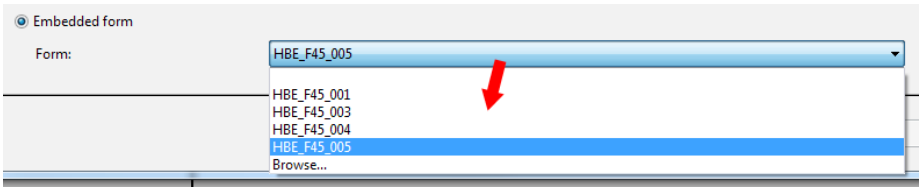


Fig. 4.176 Select form

The dialog can be closed after the selection. EPLAN now realizes the form with the desired information. Of course, the position of the contact image can be modified in any way you choose, as is also possible with the other types of contact image display.

NOTE: If there is no **CONTACT IMAGE SETTING** tab in the **PROPERTIES (COMPONENTS):** (...) dialog, you must use the Display tab (lower right) to set the display of the contact image from **NONE** to **ON COMPONENT** or **IN PATH**.

4.10 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.

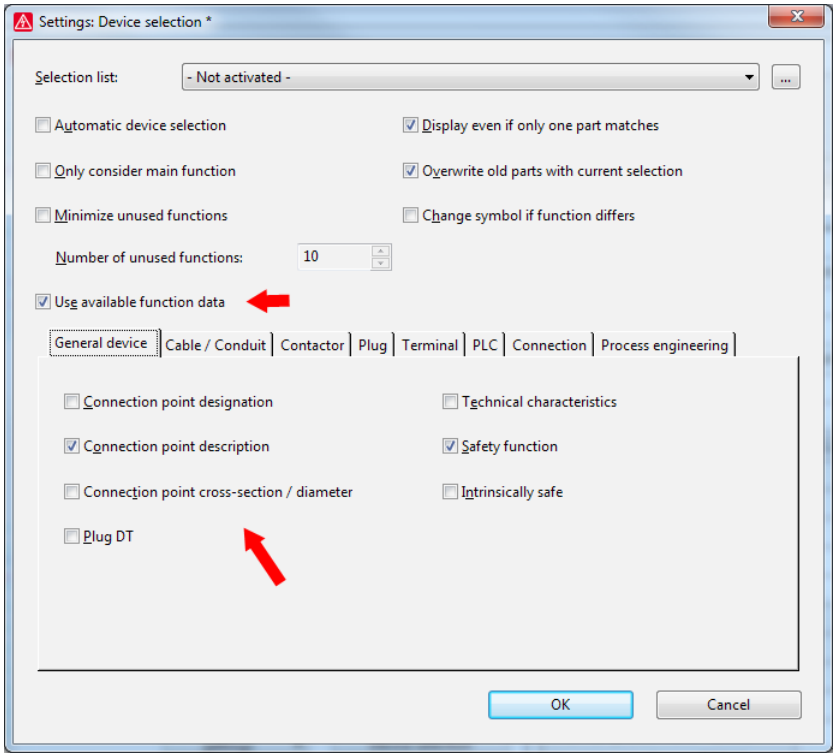


Fig. 4.177 Device selection settings

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.

5 Navigators

The graphical editor with its aids, such as the 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.

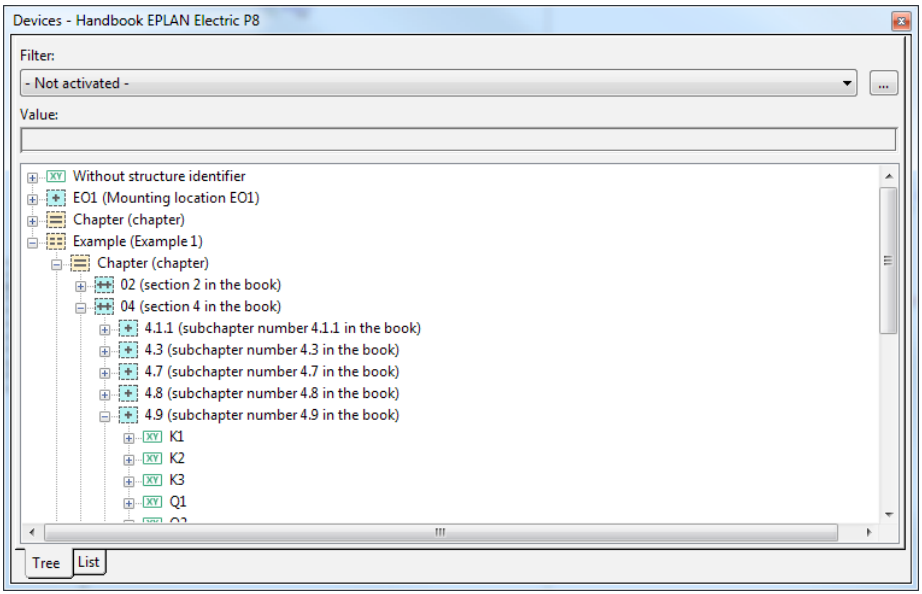


Fig. 5.1 A navigator and its view of devices

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).

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.

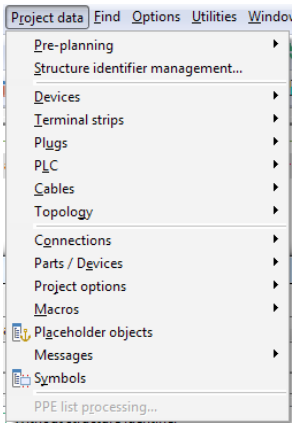


Fig. 5.2
Navigator variety

The **Device navigator** shows the view of the *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.

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.

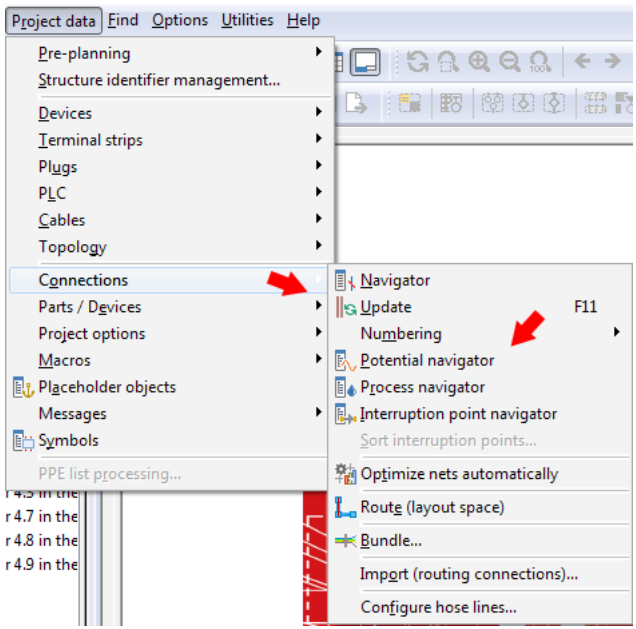


Fig. 5.3 Additional navigators for connections project data

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.

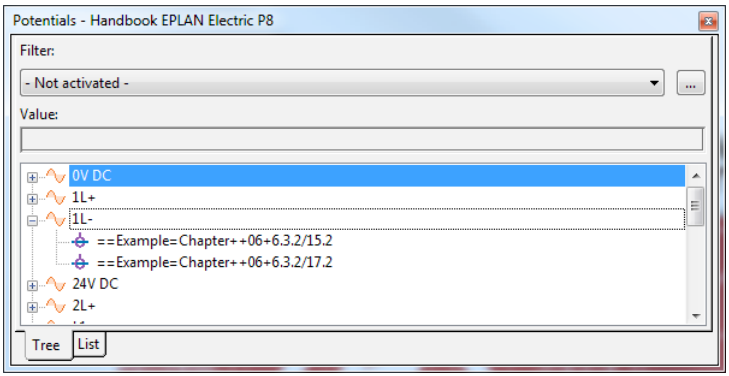


Fig. 5.4 Potential navigator

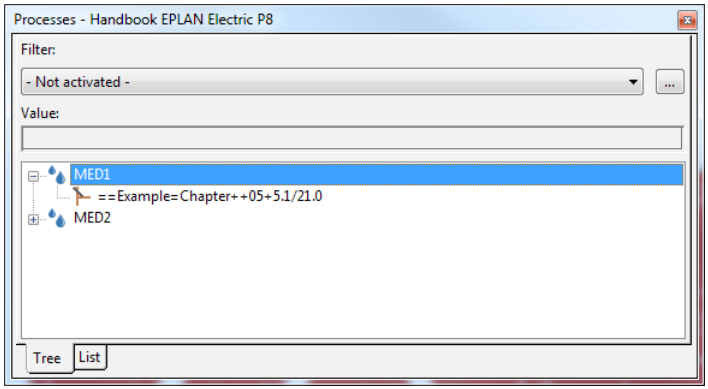


Fig. 5.5 Process navigator

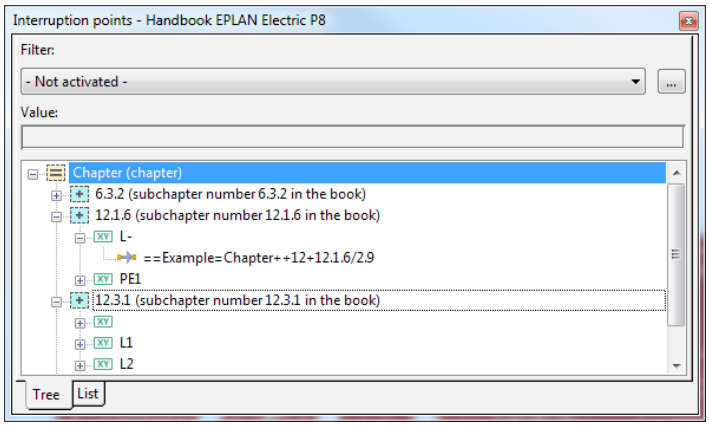


Fig. 5.6 Interruption point navigator

There are also 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.

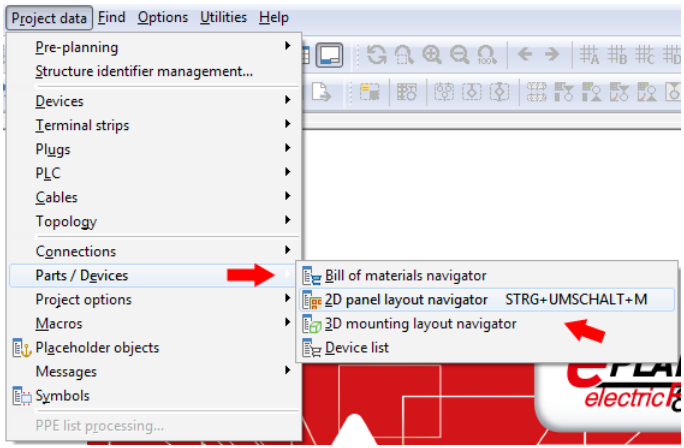


Fig. 5.7 Navigators in the parts / devices menu

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.

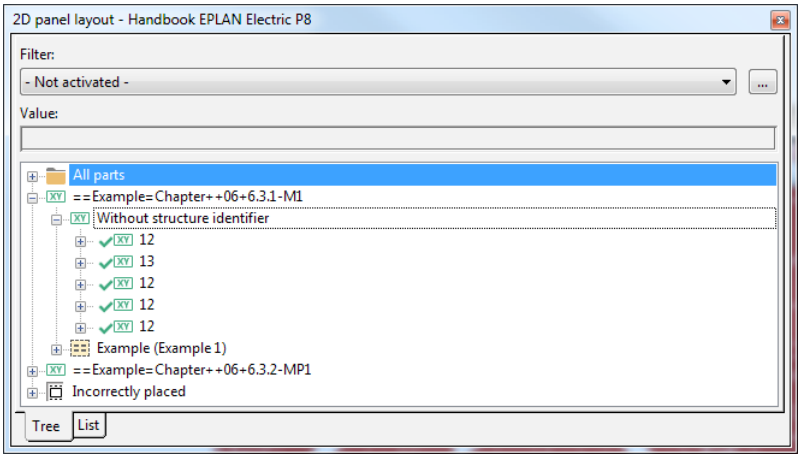


Fig. 5.8 2D panel layout navigator

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.

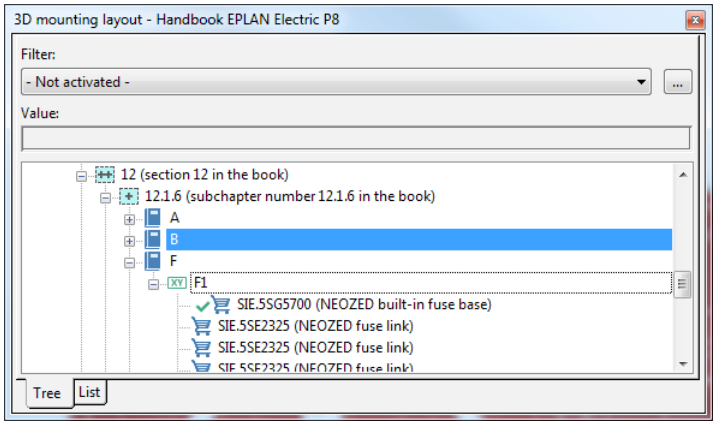


Fig. 5.9 3D mounting layout navigator

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.

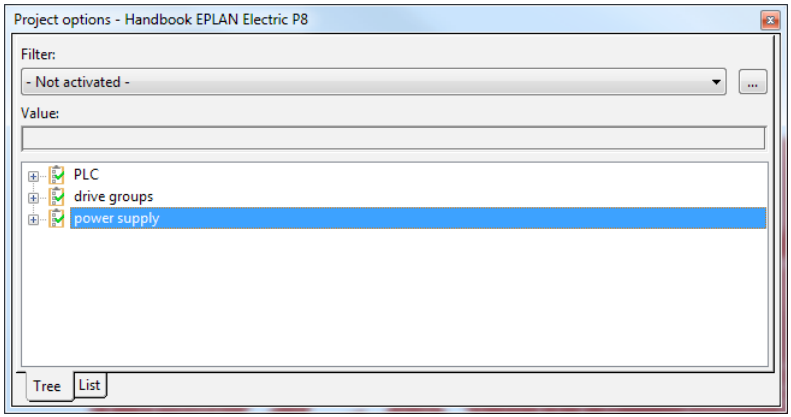


Fig. 5.10 Project options navigator

The **macro navigator** displays and manages conveniently macros of all types (page, window, symbol or 3D macros) in one single macro project.

Note: So far, the navigator serves no other functions in a schematic project.

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

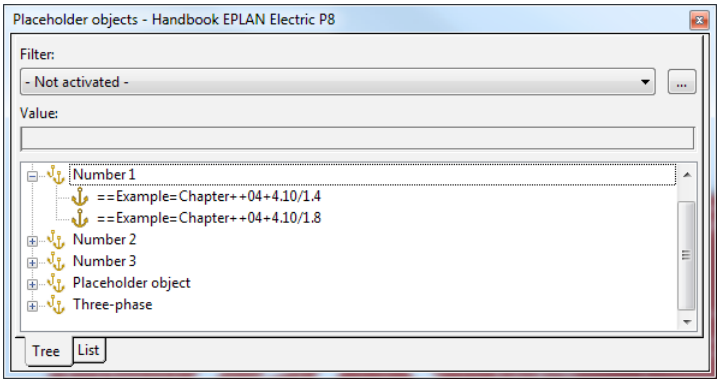


Fig. 5.11 Placeholder objects navigator

5.1.1 Additional navigators and modules

The following entry is not a navigator in the sense of those under section 5.1: **Symbol selection** in the **PROJECT DATA/SYMBOLS** menu. This entry allows symbol selection to stay open all the time like a navigator.

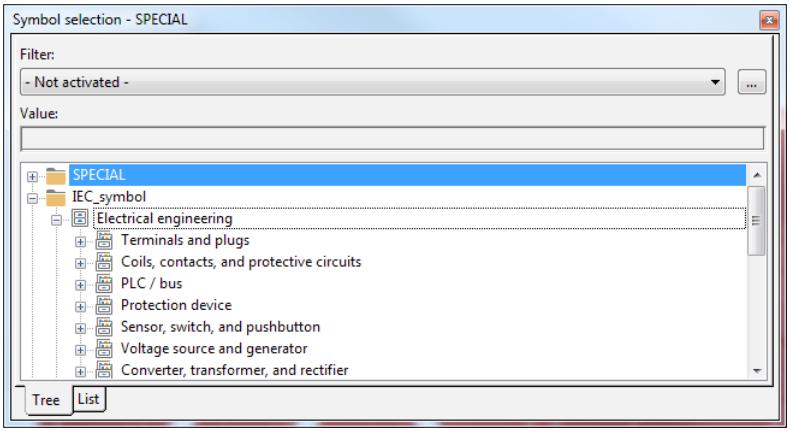


Fig. 5.12 Permanently open symbol selection

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** menu.

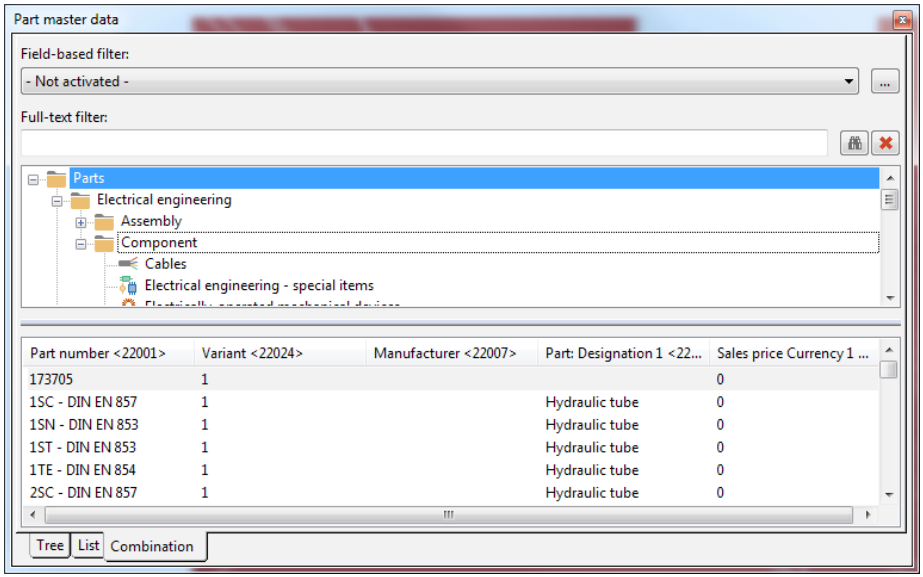


Fig. 5.13 Part master data navigator

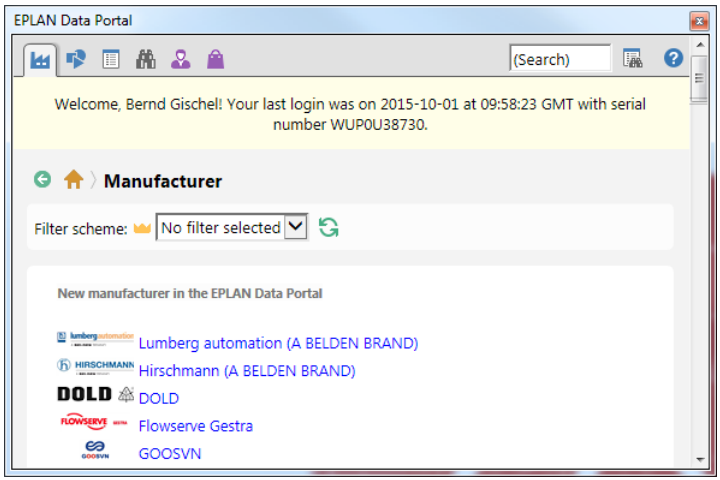


Fig. 5.14 EPLAN Data Portal

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**.

In addition, the **PAGE** menu also contains the **COMMENT NAVIGATOR**, which displays and manages entries made externally in the PDF documentation.

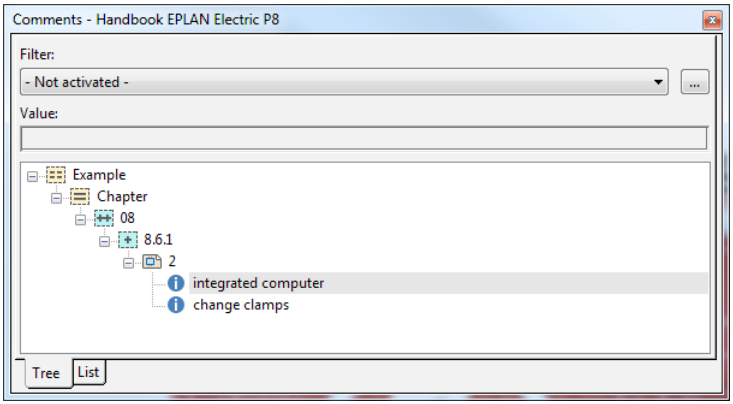


Fig. 5.15 Comment navigator

One should also mention other navigators in passing, which can be used in connection with the add-ons (sold separately) in the **PROJECT DATA** menu.

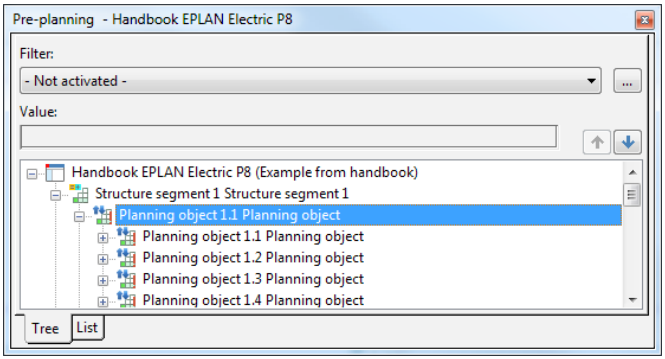


Fig. 5.16 Pre-planning navigator

The same menu also contains the **Segment template navigator**.

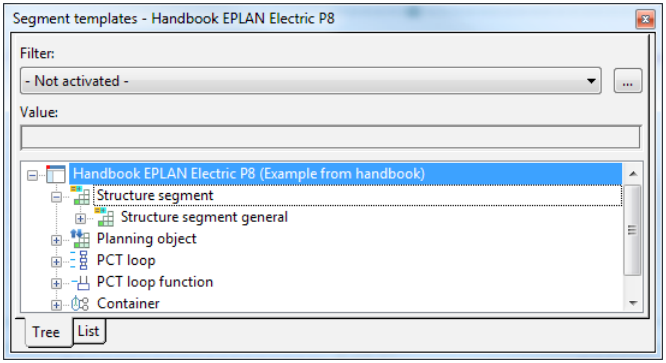


Fig. 5.17 Segment template navigator

Apart from these navigators for the pre-planning of projects, the **PROJECT DATA** menu also contains the **Topology navigator**.

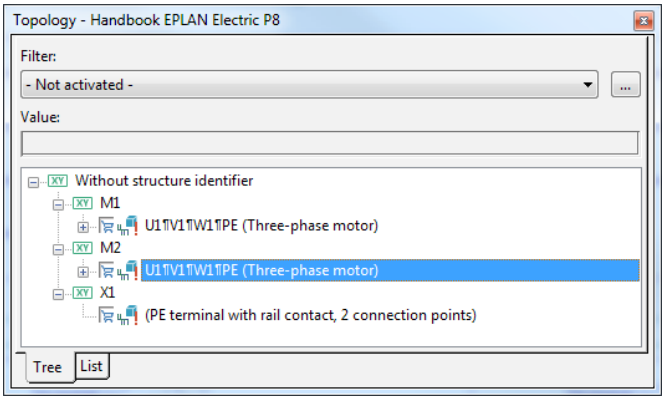


Fig. 5.18 Topology navigator

5.1.2 General navigator functions

NOTE: Functions that are identical or very similar in terms of the approach in the respective navigators are described in section 5.9.

This concerns the following functions in the popup menu of the respective navigator:

- Place
- Assign
- Filter and filter with quick entry

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.

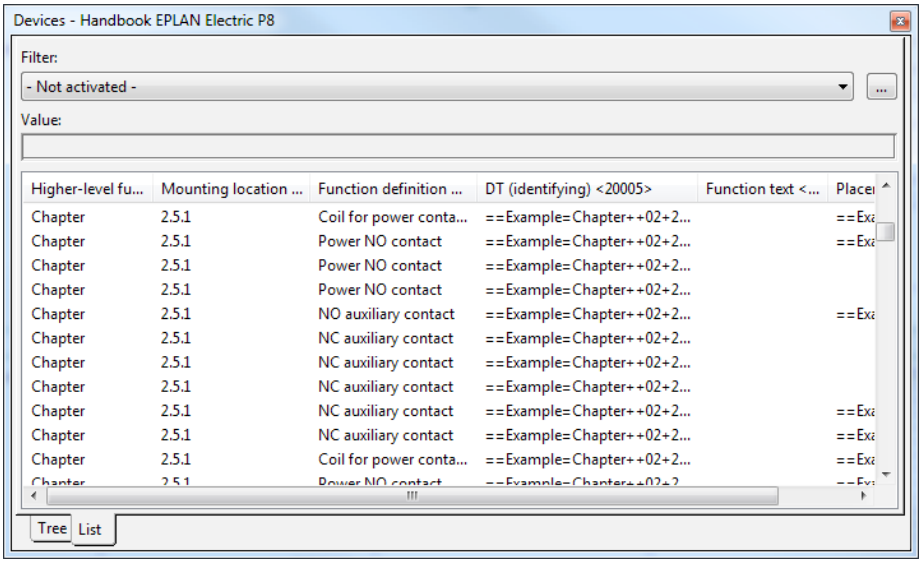


Fig. 5.19 List view in the device navigator

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).

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 or because they are more or less self-explanatory, 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.

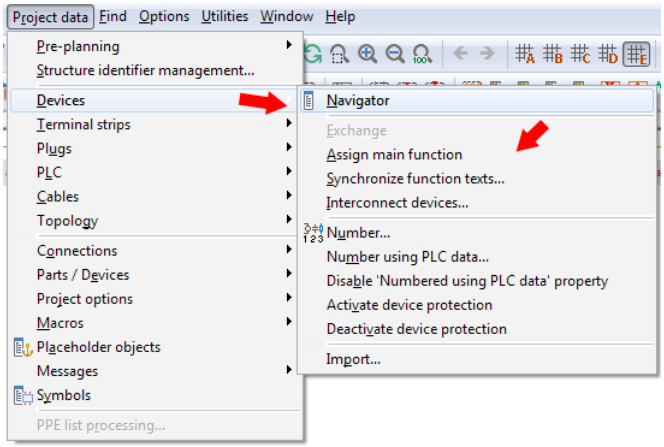


Fig. 5.20
Shortcut key in the device navigator

5.2.1 Swap

This function allows to swap the device tags (the DT) for two selected devices. It is easy: the two devices in question are selected (it makes no difference whether this involves the main function or auxiliary function), and then the **SWAP** command is executed.

Now EPLAN swaps out both DTs for all functions affected.

5.2.2 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. 5.21
Main function setting in the Properties (components) dialog

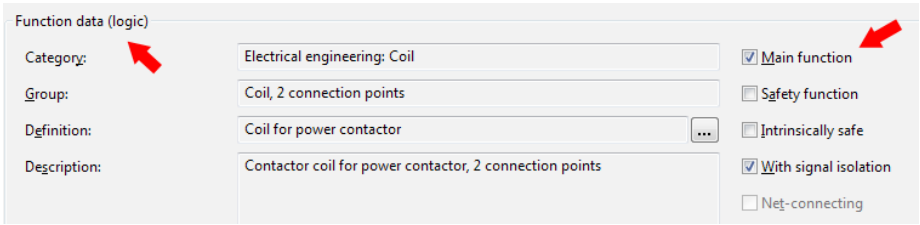


Fig. 5.22
Main function setting on the Symbol / function data tab

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.

In general, therefore, every independent item should initially be a main function. However, if other functions 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 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.

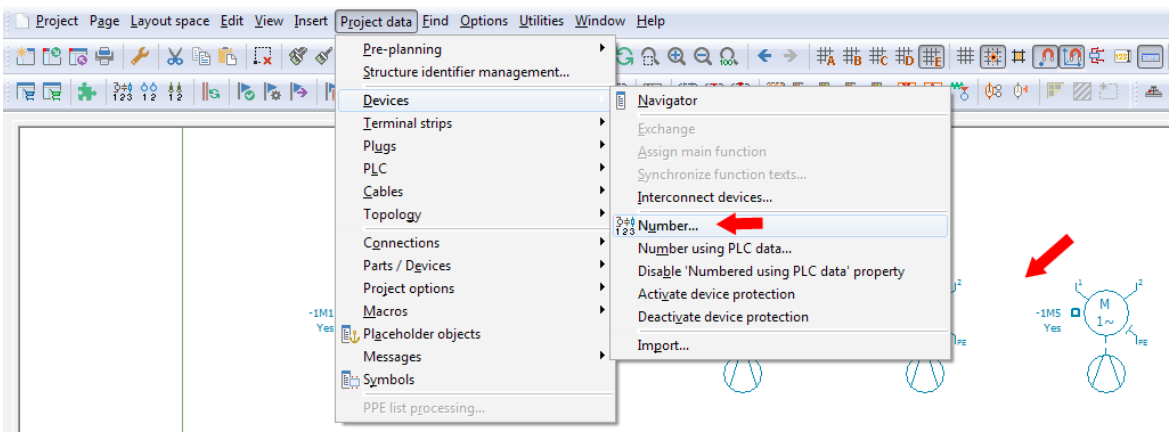


Fig. 5.23 Selected devices

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.

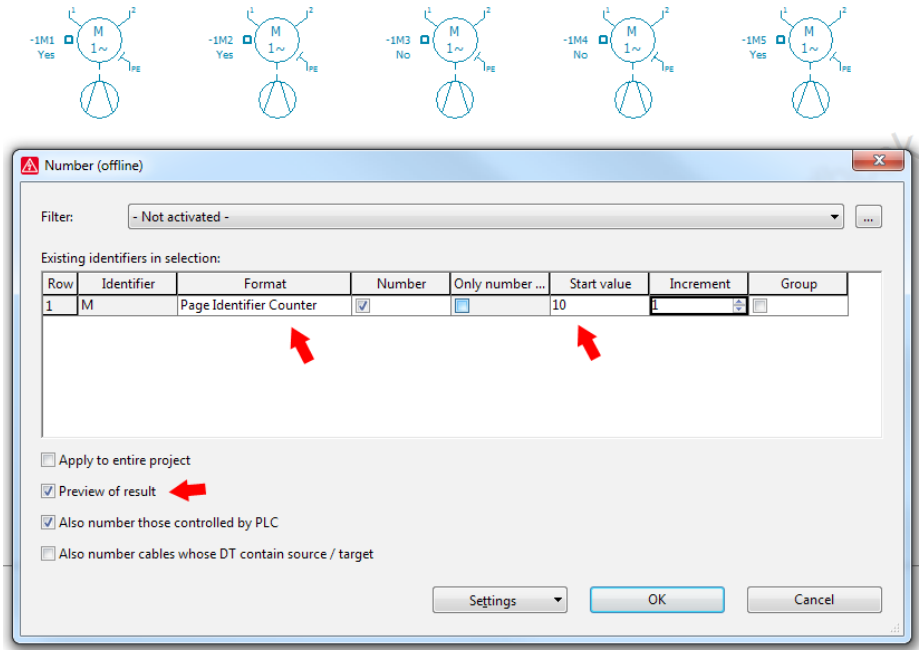


Fig. 5.24 Definitions before numbering

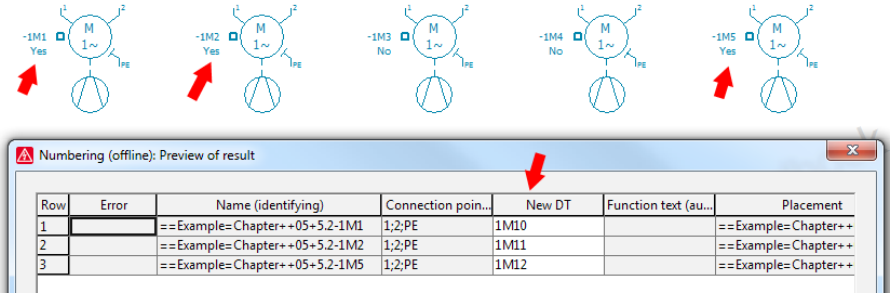


Fig. 5.25 Preview of result before final numbering

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. 5.26. All except a few devices were numbered. All devices without a main function were not numbered.

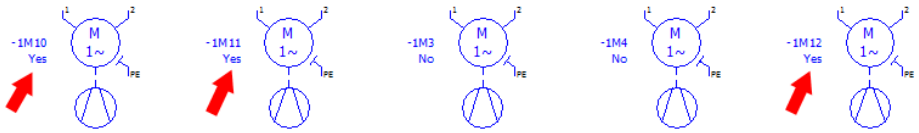


Fig. 5.26 Result of numbering

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.

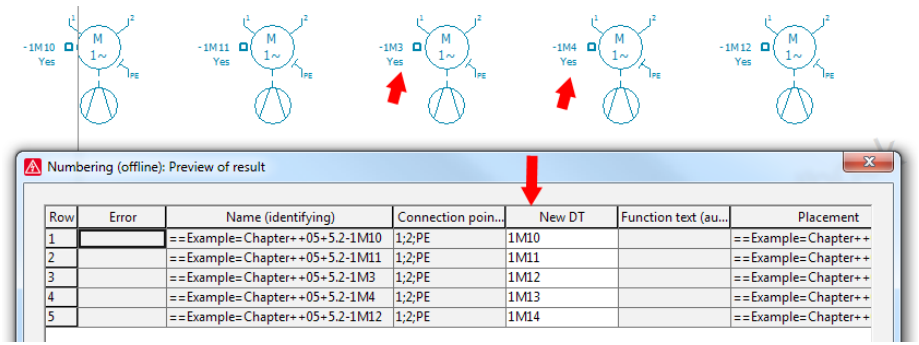


Fig. 5.27 All devices are now numbered.

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

5.2.3 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.

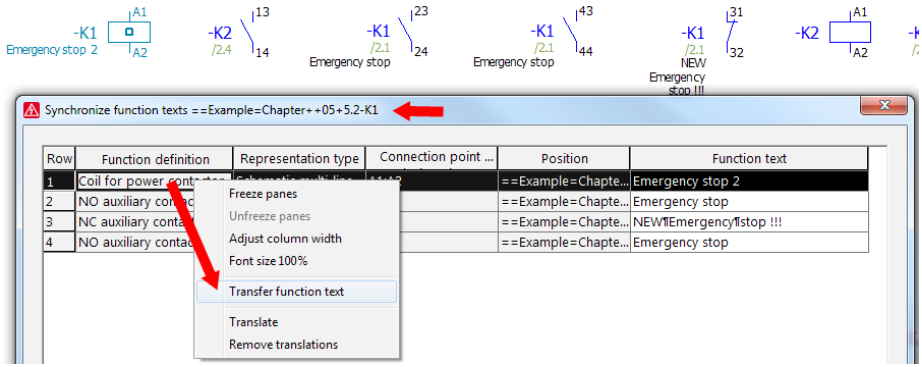


Fig. 5.28 A selected element and the Synchronize function texts dialog

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.

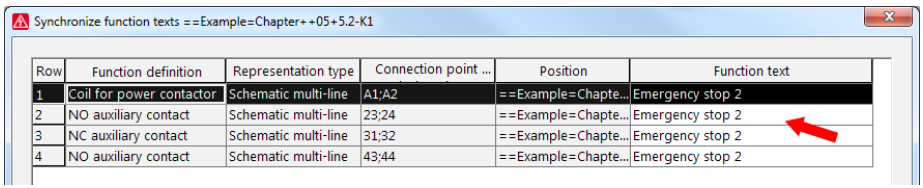


Fig. 5.29 Transferred function text in the dialog

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.

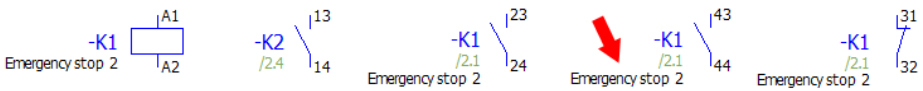


Fig. 5.30 Transferred function text in the schematic

5.2.4 Numbering (Number DT)

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.

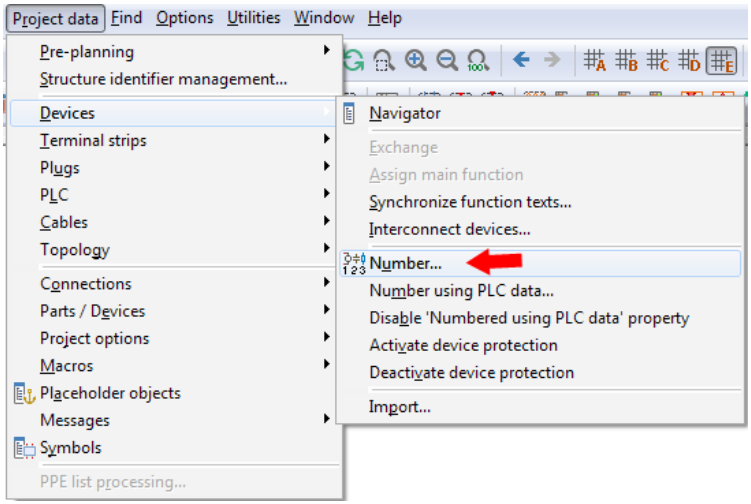


Fig. 5.31 Numbering devices

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 there are only auxiliary functions), that is, if there is no main function, then this object will not be included in the numbering. If these items are selected and numbering is then attempted, EPLAN displays an appropriate message.

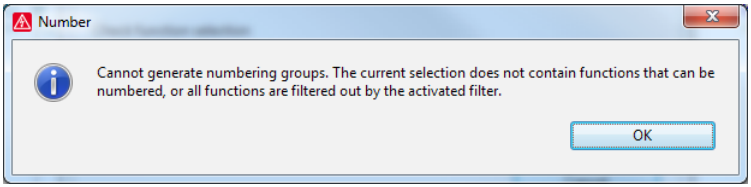


Fig. 5.32 Message

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.

Number (offline)

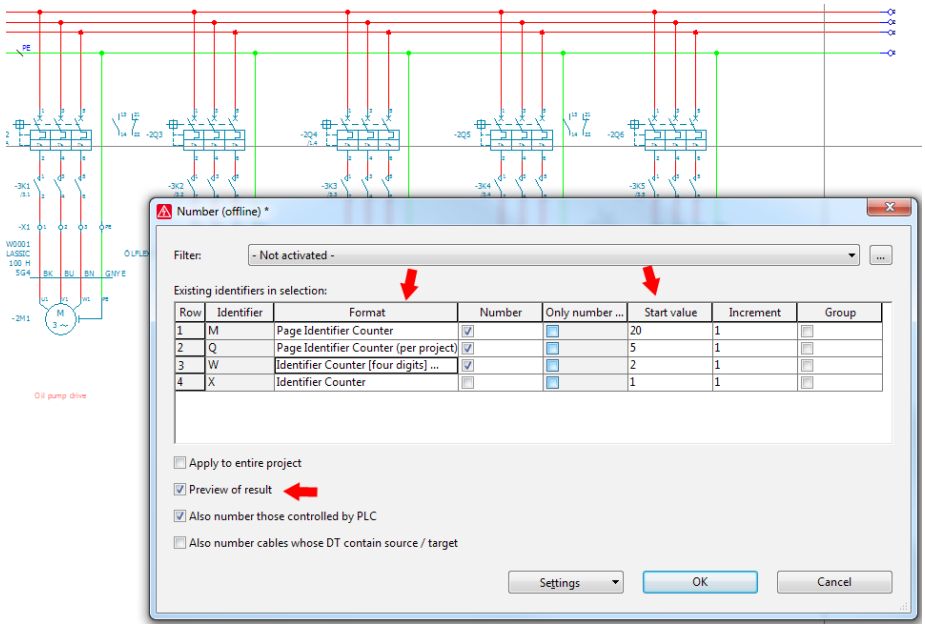


Fig. 5.33 Settings in the Number (offline) dialog

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 recommended to always use the *Preview of result* setting. This enables you to check one more time that everything is numbered (and intervene) 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.

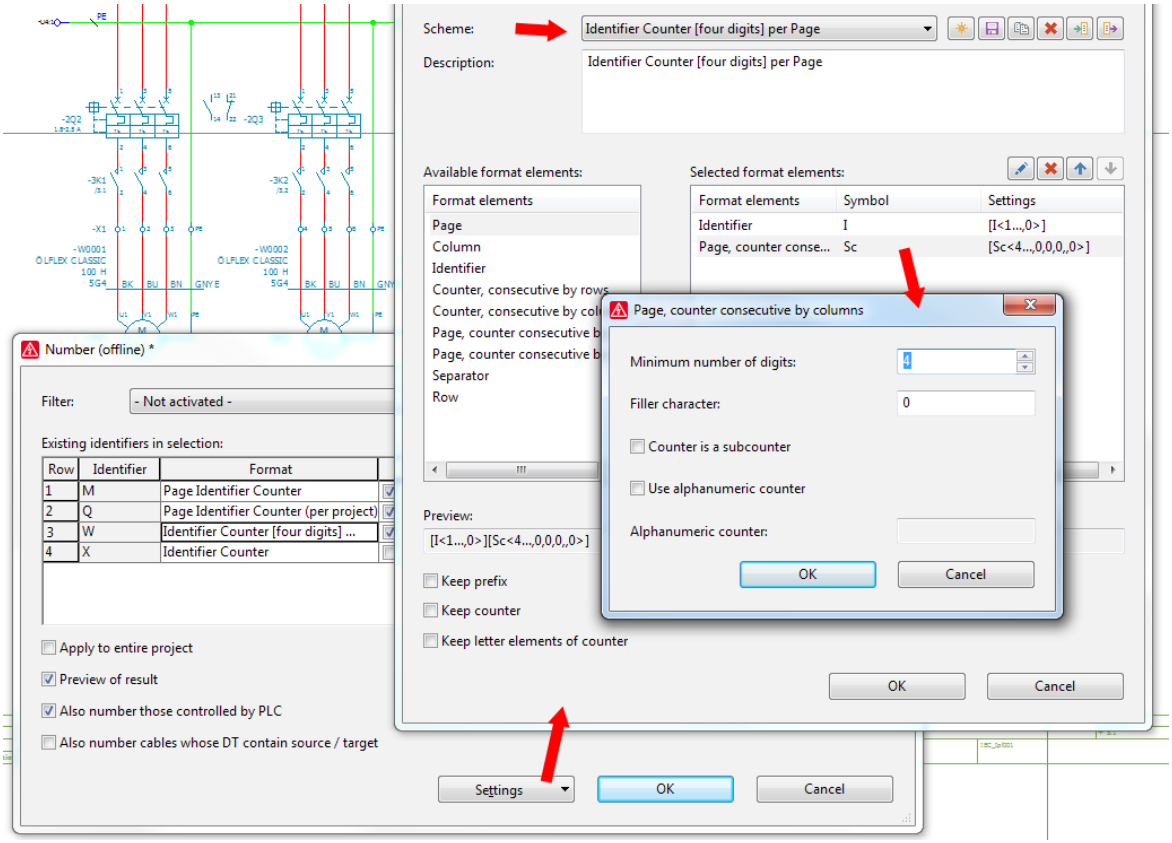


Fig. 5.34 Settings: Numbering (offline) dialog

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*.

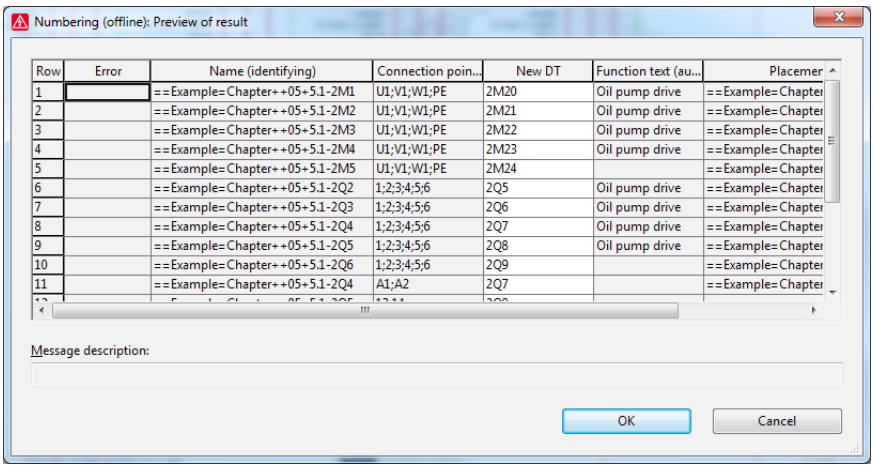


Fig. 5.35
Preview of result dialog

If the **PREVIEW OF RESULT** (if used) is confirmed, then EPLAN writes the modified device tags back to the schematic.

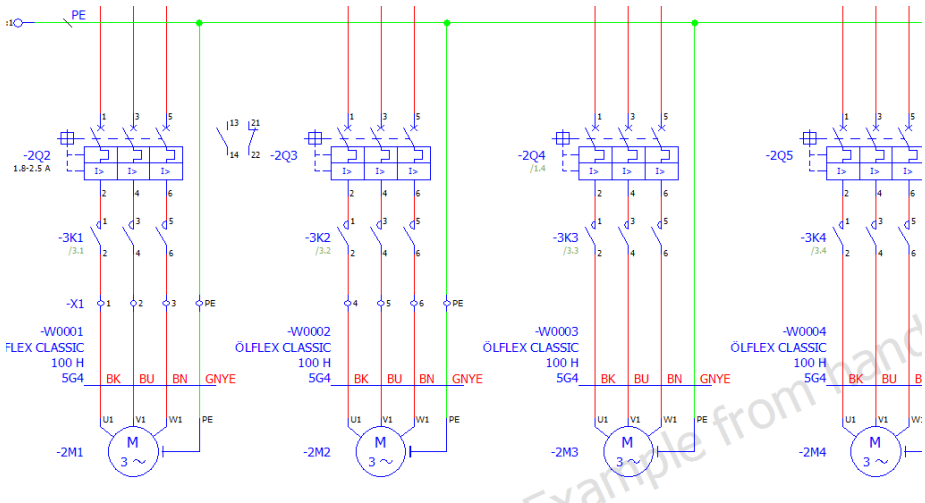


Fig. 5.36 Before numbering

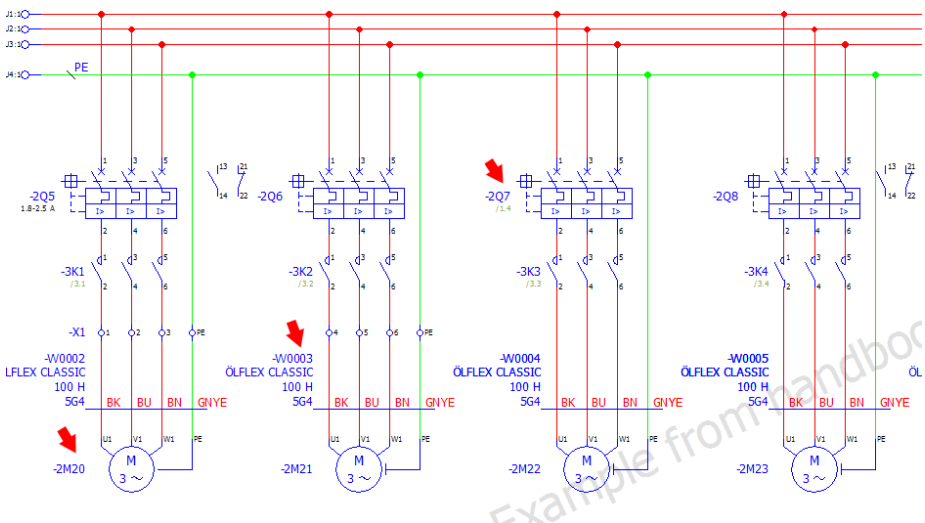


Fig. 5.37 After numbering

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.

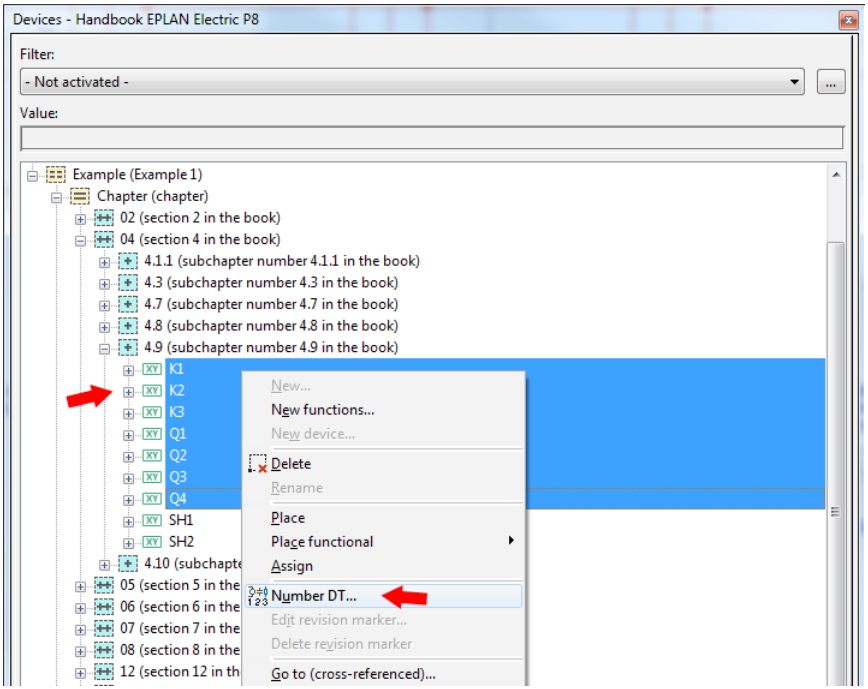


Fig. 5.38 Number DT in the popup menu

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).

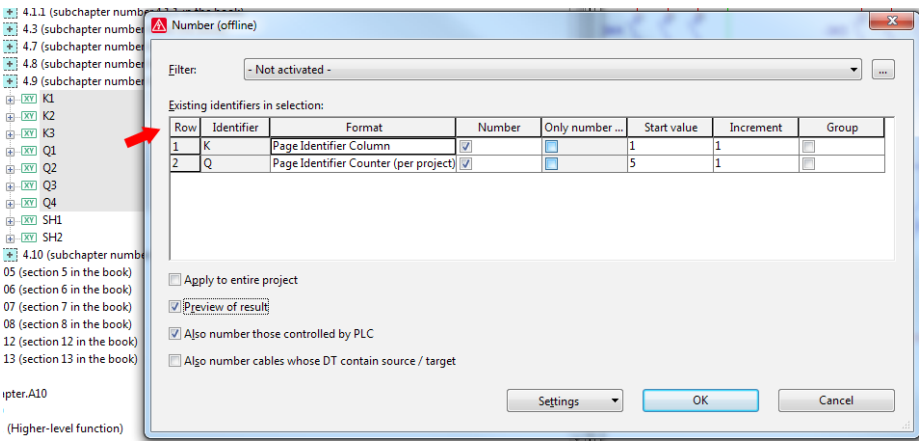


Fig. 5.39 Number (offline) dialog in the device navigator

5.2.5 Device protection

Among other things, device protection makes it possible to protect devices – and, in particular, the part assigned – against modifications. This way, unwanted changes, part or device selections can be prevented.

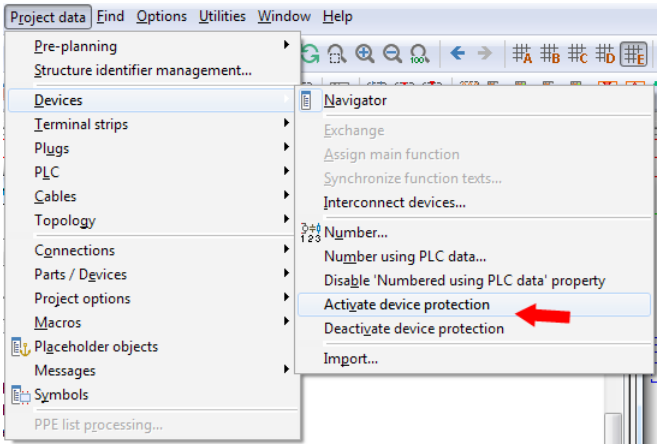


Fig. 5.40 Enabling / disabling device protection

Protected devices can be deleted in a project, but always remain in the project as unplaced devices.

5.2.5.1 Enable

The device protection is enabled simply by selecting the devices to be protected. Then, it is enabled in the **PROJECT DATA/DEVICES/ENABLE DEVICE PROTECTION** menu. EPLAN will then identify such protected devices using a modified icon in the navigators.

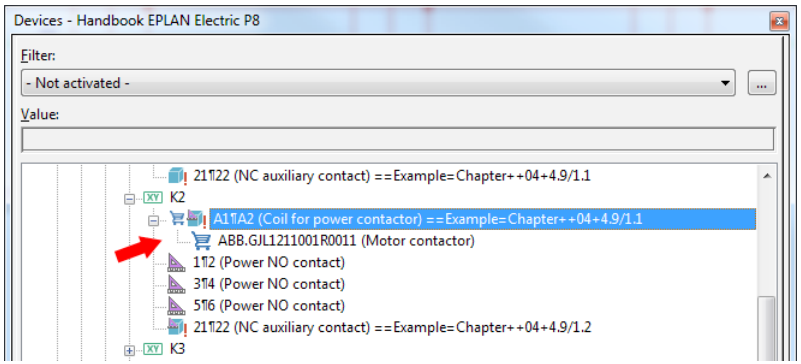


Fig. 5.41 Object without device protection

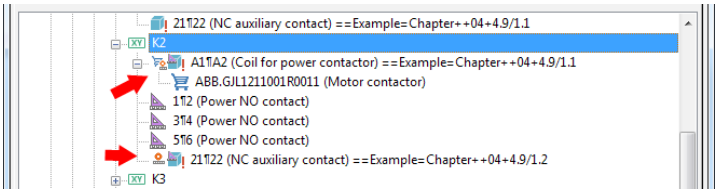


Fig. 5.42 Object with device protection

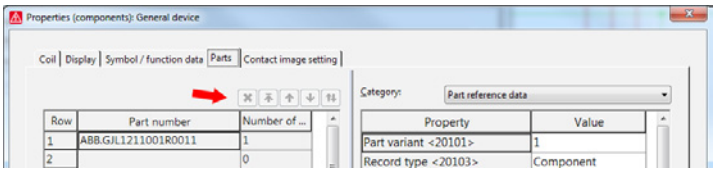


Fig. 5.43 Protected part (Parts tab)

5.2.5.2 Disable

The protection is easily disabled again by selecting the objects and then clicking the **PROJECT DATA/DEVICES/DISABLE DEVICE PROTECTION** menu.

5.2.6 Import (device data)

The **IMPORT** function in the device navigator allows for external data lists to be imported in EPLAN.

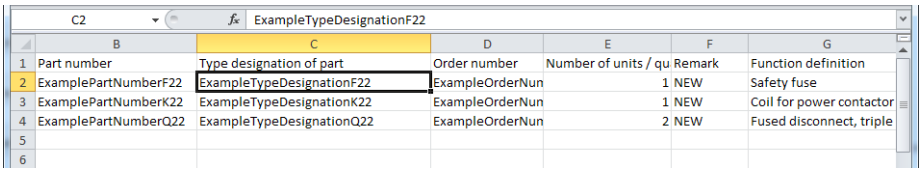


Fig. 5.44 Example of data list (Excel)

An import scheme must be used in EPLAN in order to import the data. EPLAN data fields and external data fields are assigned in this scheme.

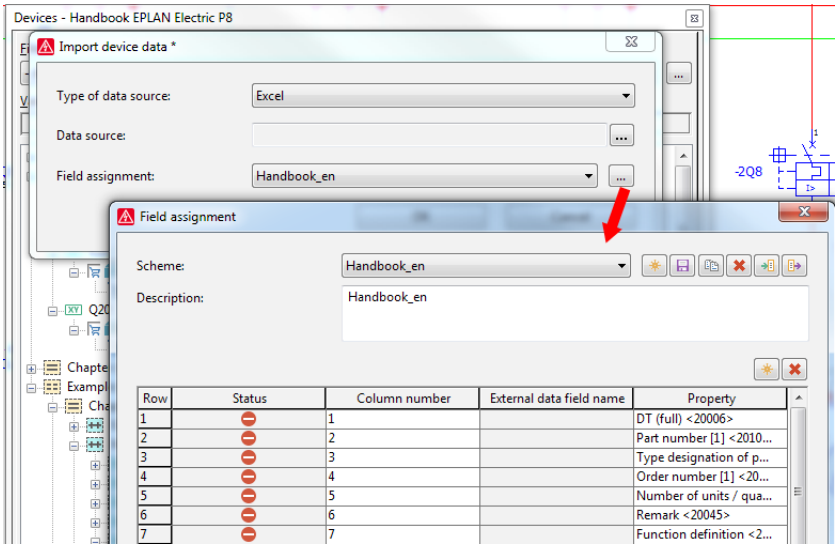


Fig. 5.45 Example of import scheme

A default import scheme can be configured in the settings under **OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/DEVICES/IMPORT**.

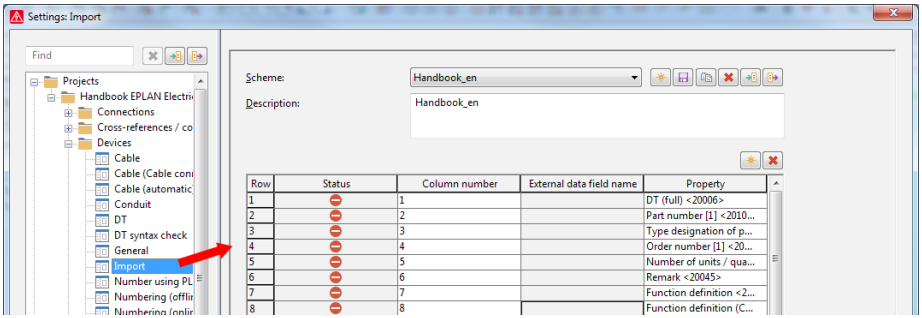


Fig. 5.46 Scheme in project settings

After selecting **IMPORT**, EPLAN opens the **IMPORT DEVICE DATA** dialog. Here, the type of data source, the data source itself and, of course, the import scheme (field assignment) are configured.

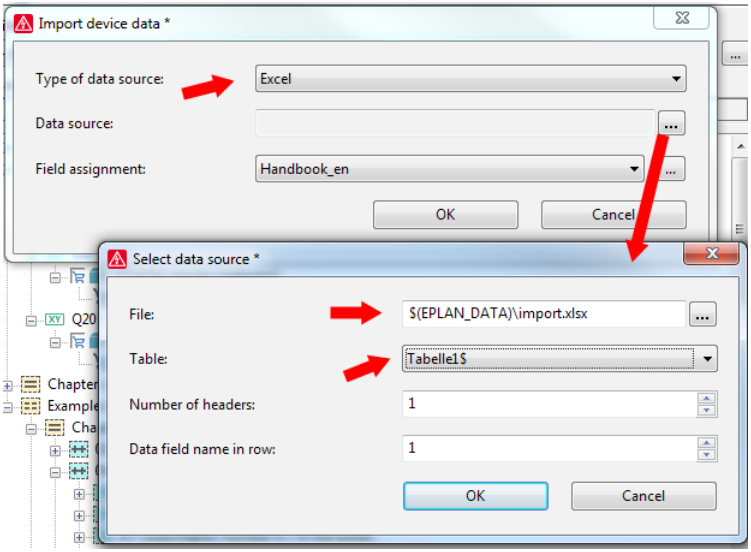


Fig. 5.47 Configuring import data

Once done, the dialog is confirmed with **OK**. EPLAN imports and opens the **SYNCHRONIZE DEVICES** dialog. Changes can be made in this dialog.

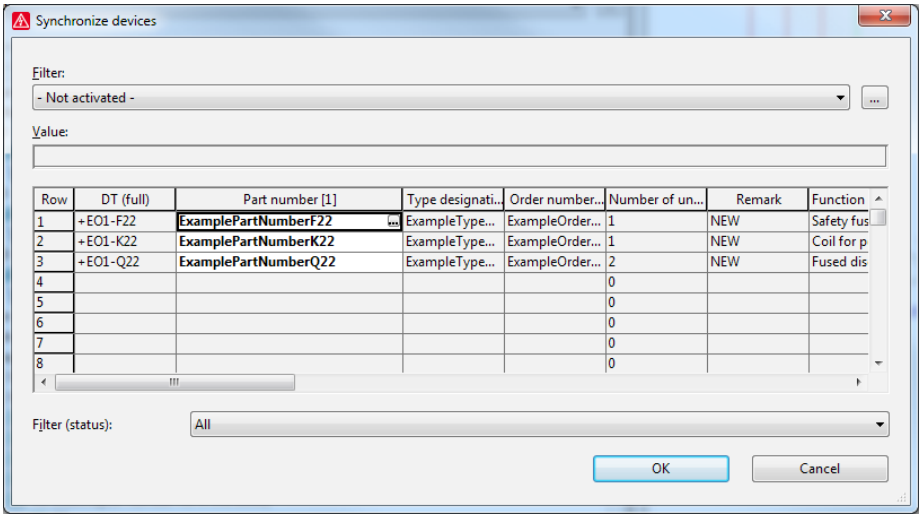


Fig. 5.48 Synchronize devices dialog

After clicking **OK**, EPLAN applies any changes made, imports the data, and displays a message about the successful import.

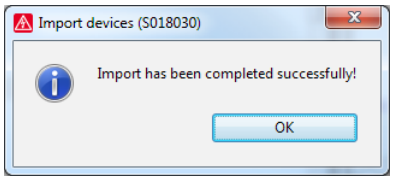


Fig. 5.49
Message

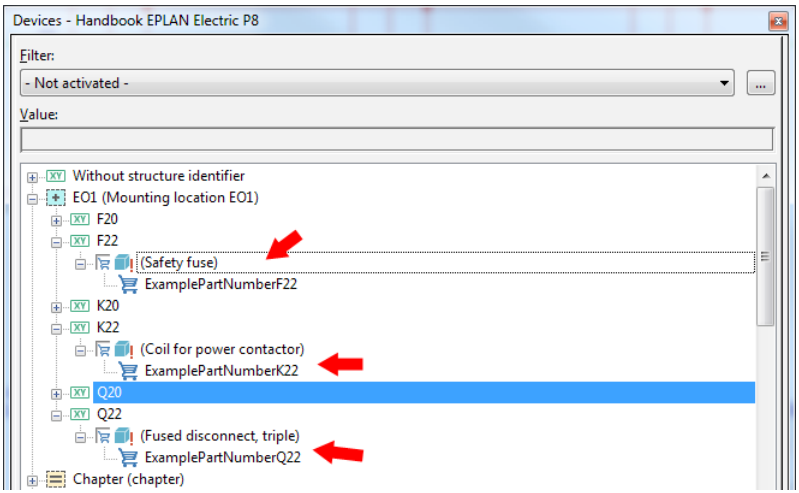


Fig. 5.50 Imported device data

5.2.7 New

The **NEW...** function inserts a new function definition, for example an additional auxiliary NC contact for a motor overload switch.

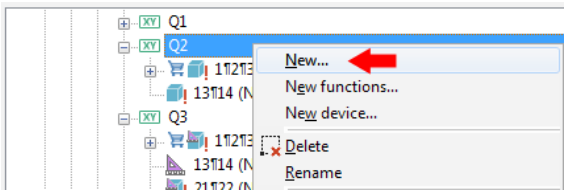


Fig. 5.51
New... menu call

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.

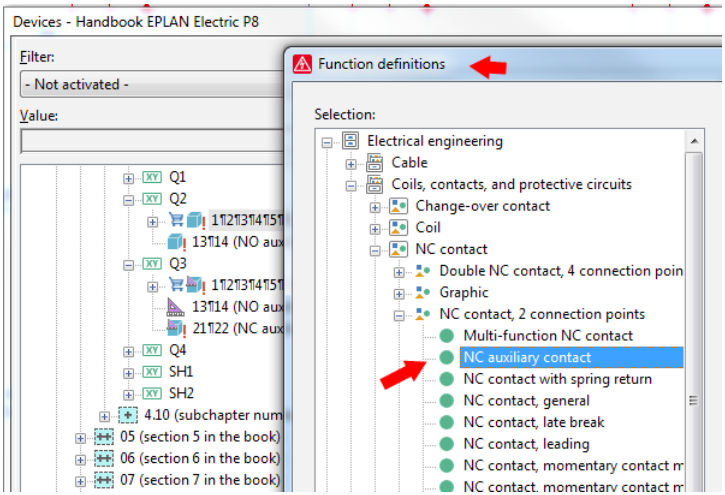


Fig. 5.52 Function definitions dialog

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 *Connection point designation* or the *Device tag*, which is already set by default to the DT selected in the device navigator.

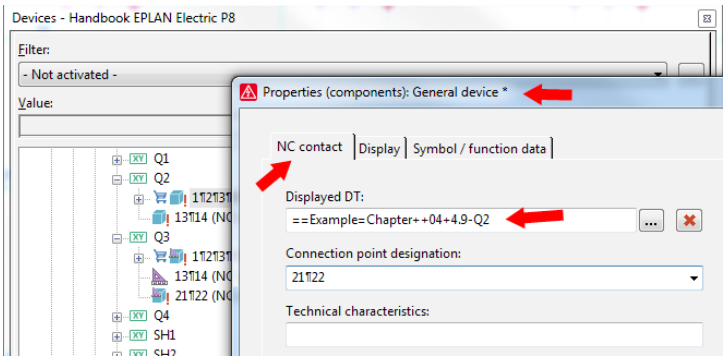


Fig. 5.53 Properties (components) dialog

When you click **OK** to exit the dialog, the *NC auxiliary contact* is assigned to the selected motor overload switch in the device navigator.

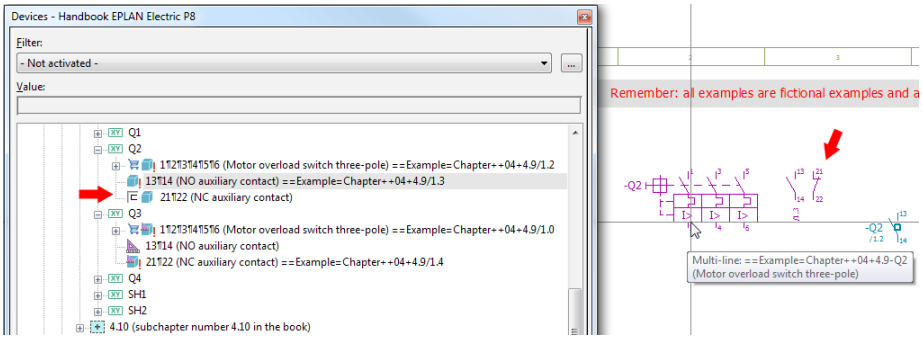


Fig. 5.54 New function on motor overload switch

5.2.8 New functions

To assign one or several new functions to a device, you must select the corresponding device in the device navigator and then call the **NEW FUNCTIONS...** function via the popup menu.

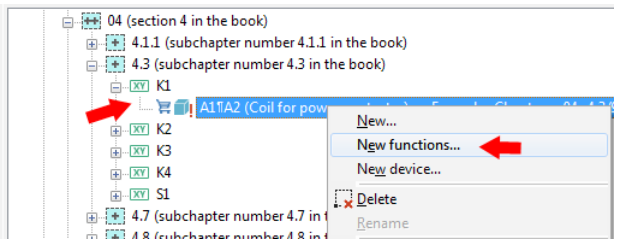


Fig. 5.55 Create new functions

EPLAN starts the **GENERATE FUNCTIONS** dialog and immediately fills the *Full DT* field with the DT selected. Now, using the **MORE** button, the desired function definition is selected from the **FUNCTION DEFINITIONS** dialog and applied to the **GENERATE FUNCTIONS** dialog by clicking **OK**.

Then, a numbering pattern must be defined. If no pattern is defined, EPLAN will not insert any new function (nor can the dialog be exited via the **OK** button). If a pattern is defined, EPLAN will number the new functions automatically when they are added. EPLAN will also count the corresponding number of new functions and transfer this information to the DT as well.

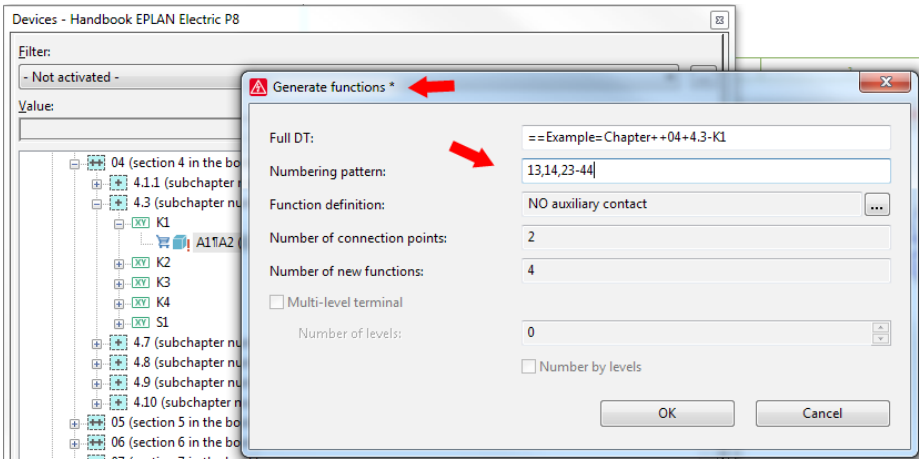


Fig. 5.56 Generate new functions

Once all these entries have been made correctly, you can exit the dialog by clicking **OK**. EPLAN closes the dialog and transfers the new functions.

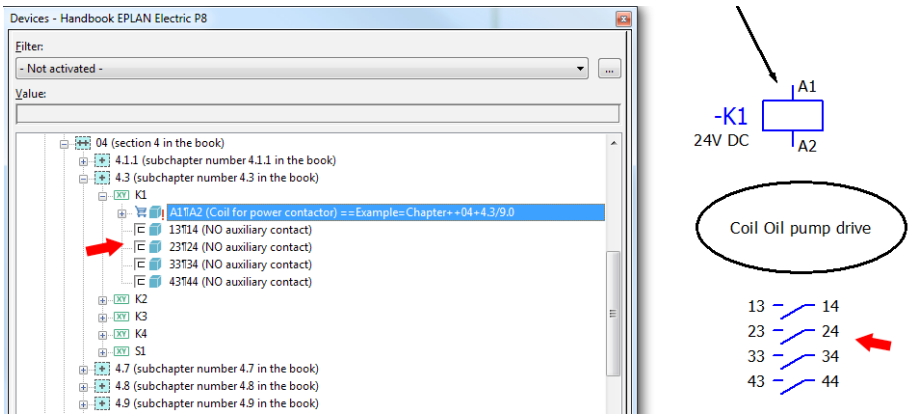


Fig. 5.57 Device with new functions

5.2.9 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.

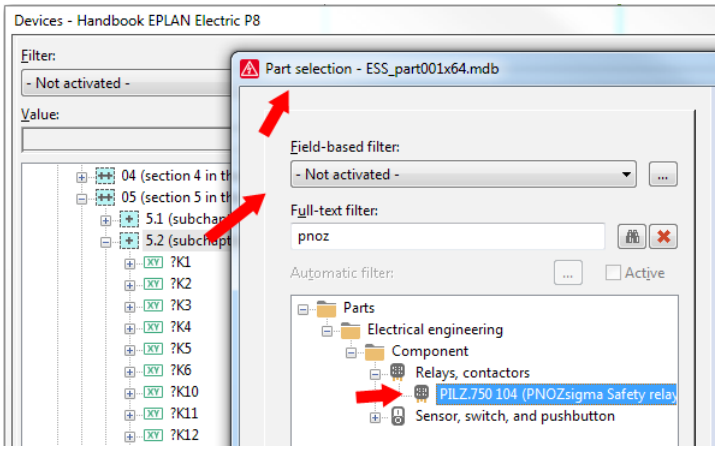


Fig. 5.58 Select device

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.

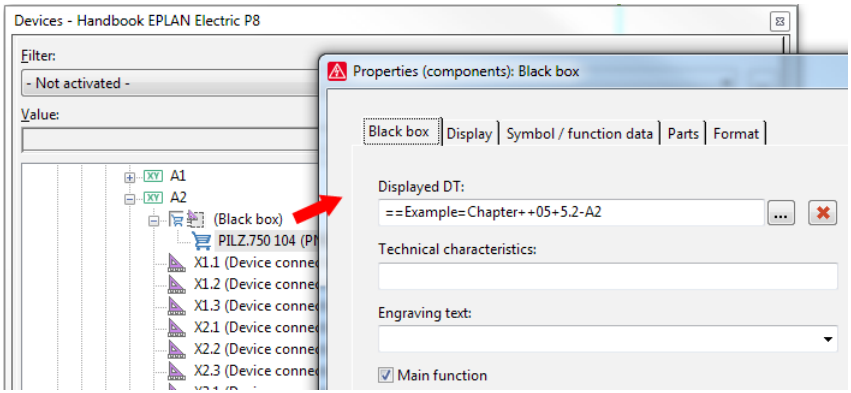


Fig. 5.59 Subsequent device editing

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.

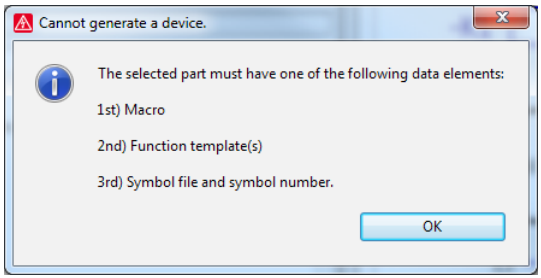


Fig. 5.60 Prompt if a device cannot be generated

5.3 Terminal strip navigator

There is not much to say about terminal strips. The **terminal strip navigator** helps the user perform many repetitive actions, for example numbering terminals.

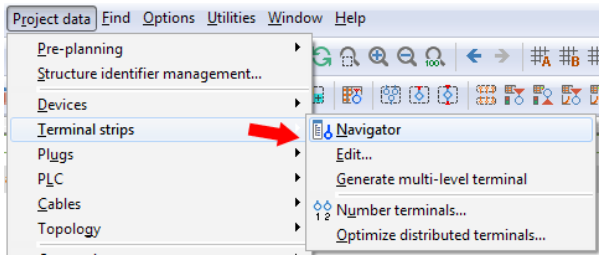


Fig. 5.61
Terminal strip menu

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 Edit (terminal editor)

5.3.1.1 General information about the terminal editor

The **EDIT** function in the **PROJECT DATA/TERMINAL STRIPS** menu opens the **EDIT TERMINAL STRIPS** dialog. This dialog contains different functions that can be executed or used via the popup menu and/or the upper buttons for editing terminals. Examples of these functions are *Number terminals*, *Move*, *Add strip accessories, and delete* or *Generate 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.

NOTE: Most functions available in the terminal editor are identical to those that can be called in the terminal strip navigator. Further explanation will therefore not be provided.

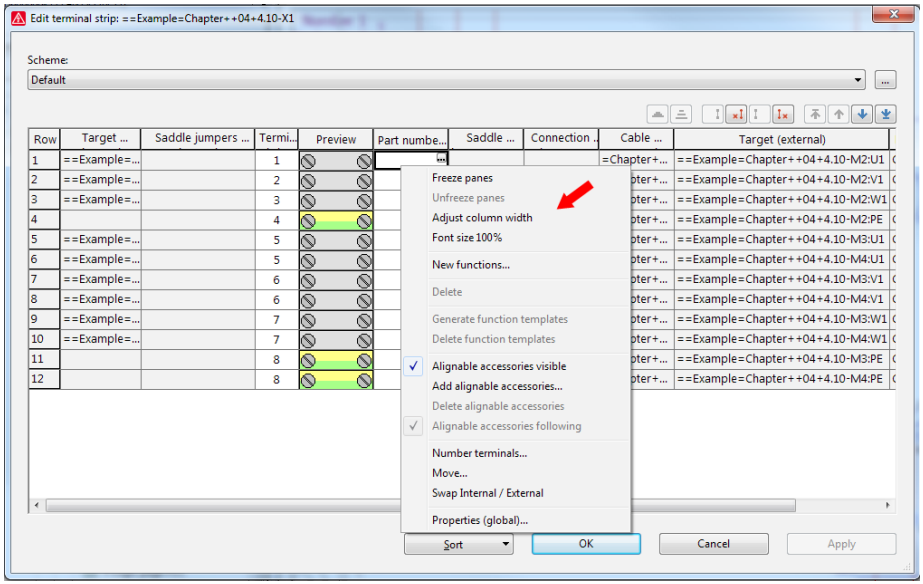


Fig. 5.62 Edit terminal strips dialog

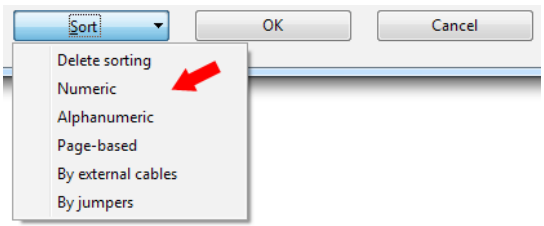


Fig. 5.63
Sort button and its functions

Additional functions for editing terminals are located in the lower area of the dialog after clicking the **SORT** button.

The **SORT** button opens another menu containing functions for sorting terminals. The **DELETE SORTING** 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.

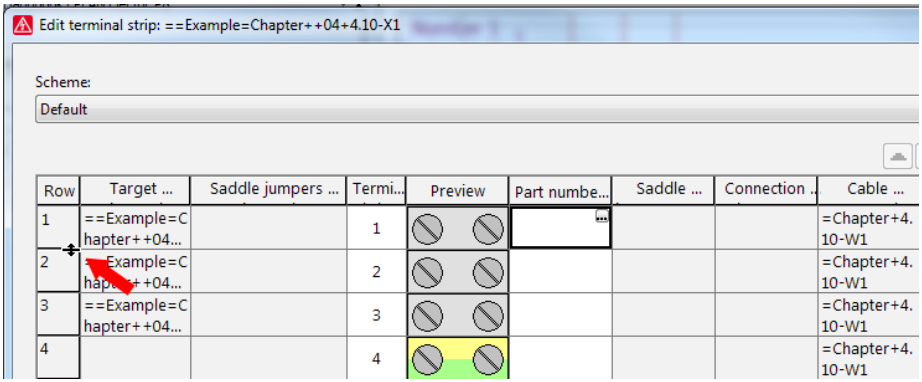


Fig. 5.64 Change row height

EPLAN opens the **EDIT TERMINAL STRIPS** dialog. The terminals are sorted here according to the standard. Click the **SORT** button and select the **DELETE SORTING** item. EPLAN sorts the terminal strips as follows: The existing device position and sort code at the terminal are deleted.

NOTE: This will also break up defined terminal devices. Thus, terminals are sorted by their designation (1, 4 or 17) and/or based on Sorting (graphical).

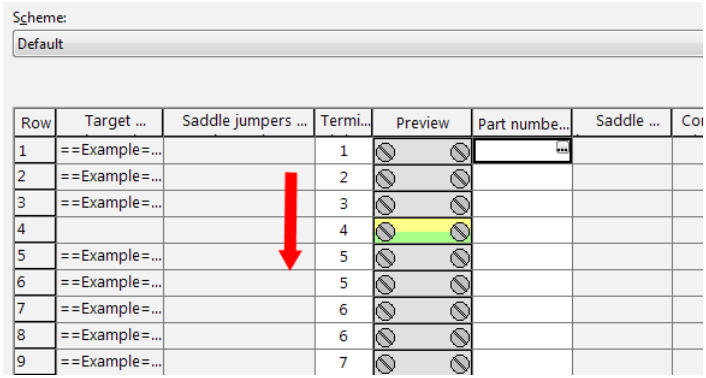


Fig. 5.65 Newly sorted terminal strip

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.

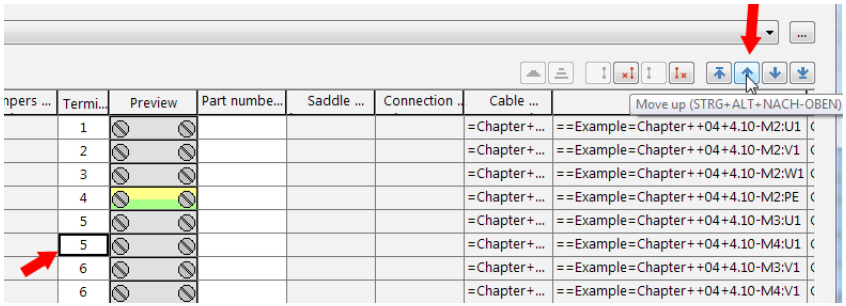


Fig. 5.66 Manually moving a terminal

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.

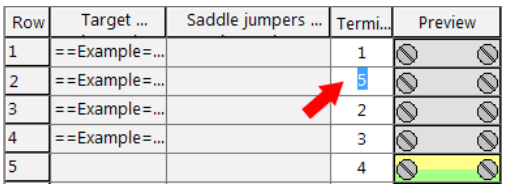


Fig. 5.67 Renaming terminal designation

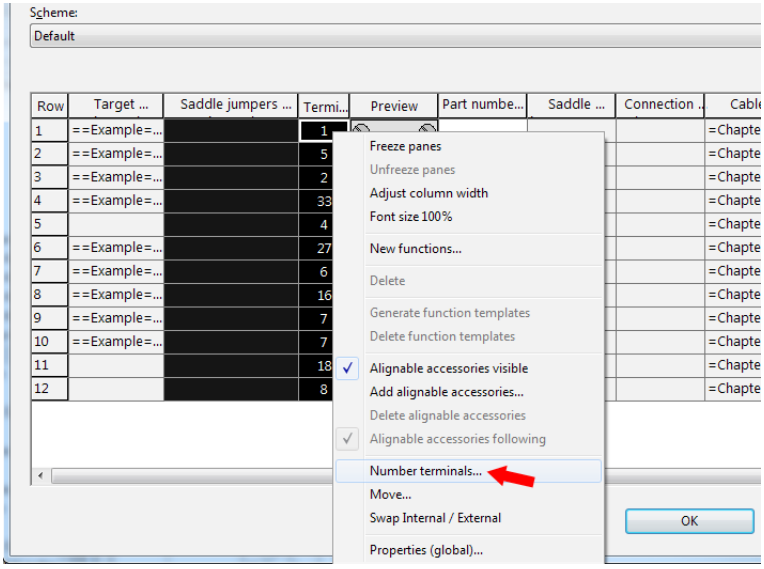


Fig. 5.68 Numbering terminals

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 to be made. EPLAN can then number the terminals.

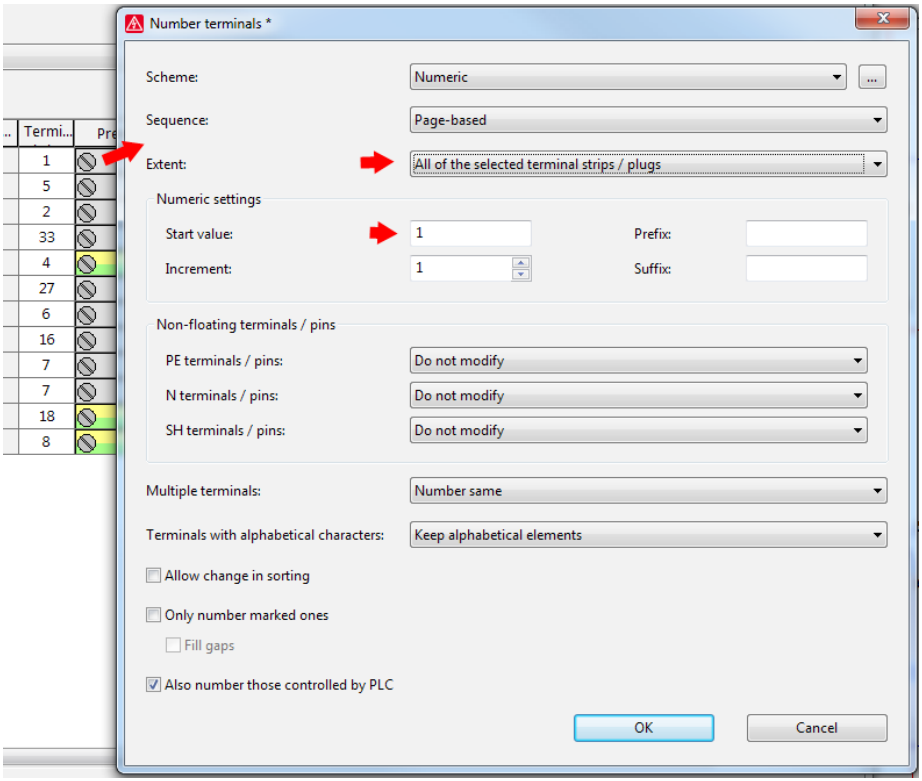


Fig. 5.69 Number terminals dialog

The newly numbered terminal strip will then look like the one in Fig. 5.70.

Row	Target ...	Saddle jumpers ...	Termi...	Preview	Part numbe...	Saddle jum...
1	==Example=...		1			
2	==Example=...		2			
3	==Example=...		3			
4			4			
5	==Example=...		5			
6	==Example=...		6			
7	==Example=...		7			
8	==Example=...		8			
9	==Example=...		9			
10			10			
11	==Example=...		11			
12			12			

Fig. 5.70 Numbered terminal strip

5.3.2 Generate multi-level terminal

Multi-level terminals can also be generated conveniently in the terminal editor. It is quite easy to do. Select the terminals to be used for a multi-level terminal.

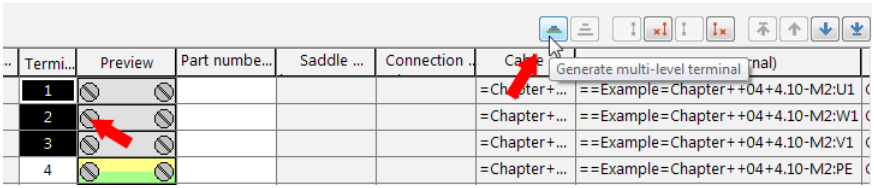


Fig. 5.71
Selecting terminals

Then, click the button (**GENERATE MULTI-LEVEL TERMINAL**). EPLAN groups the selected terminals to form a multi-level terminal.

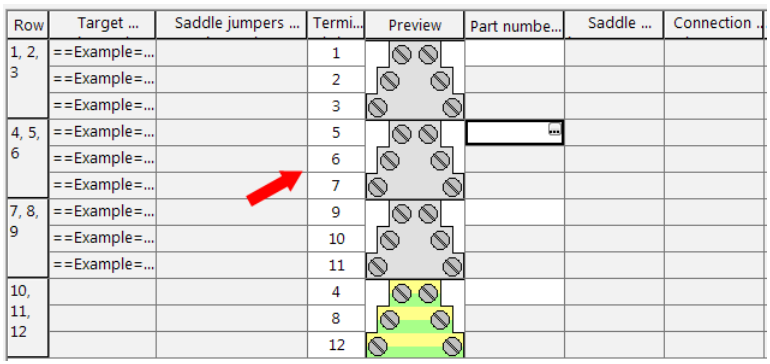


Fig. 5.72
Generated multi-level terminals

During this action, EPLAN allocates the levels and assigns a device position each to these multi-level terminals (Property **Terminal: Device position <20367>**).

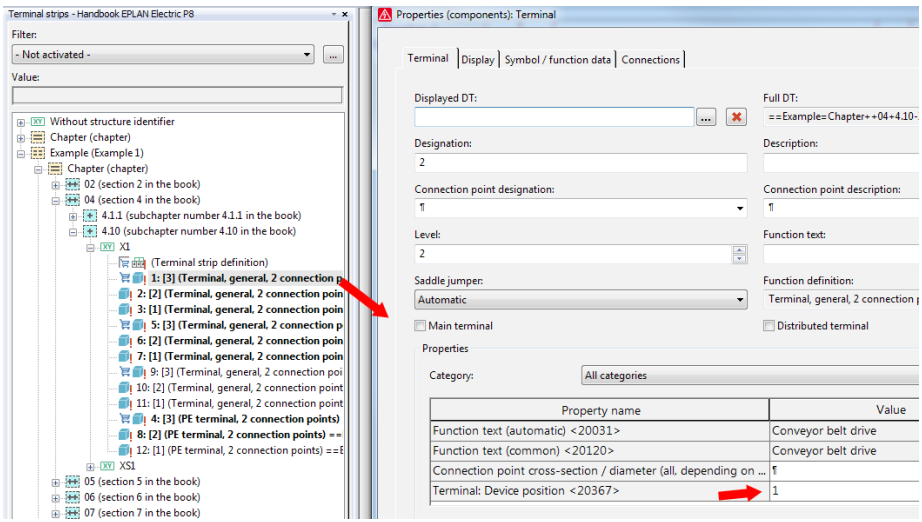


Fig. 5.73
Property Terminal: Device position

There are also other options in the terminal editor, but they are self-explanatory:

- Generate saddle-jumpers for selected terminals (externally or internally)
- Delete saddle-jumpers for selected terminals (externally or internally)
- Individual part selection per terminal
- Swap internal/external connection points for selected terminals

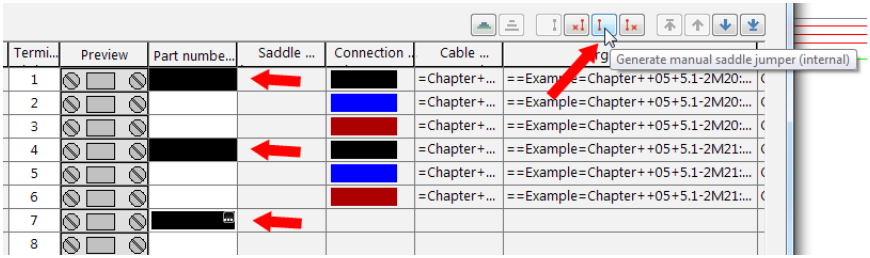


Fig. 5.74 Other options in the terminal editor

5.3.3 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.

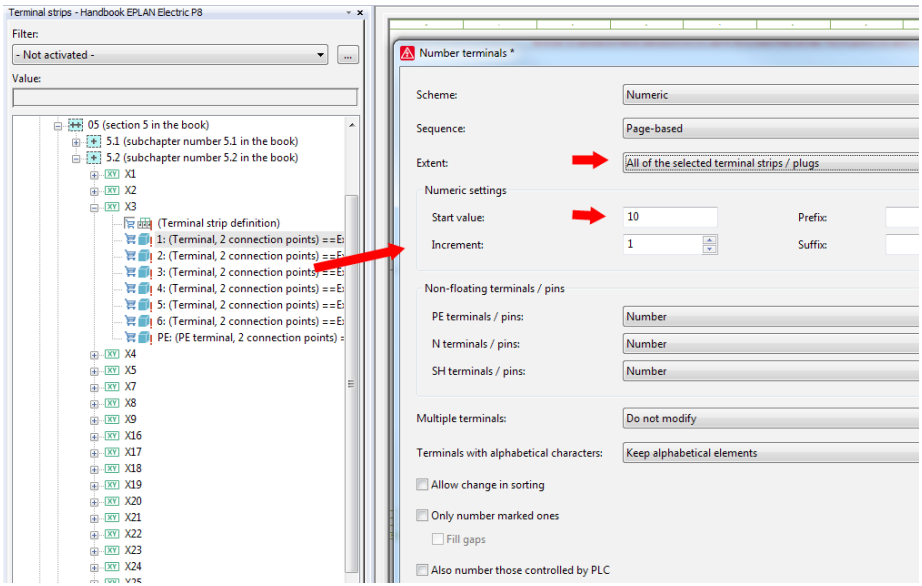


Fig. 5.75 Number terminals dialog

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.

You then start the **NUMBER TERMINALS** dialog via the **NUMBER TERMINALS** item in the **PROJECT DATA/TERMINAL STRIPS** menu.

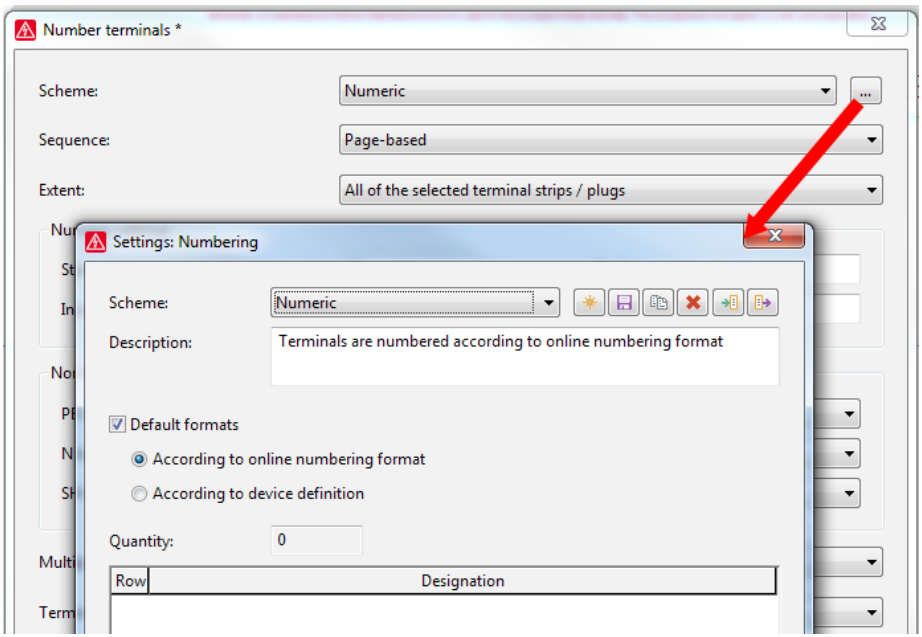


Fig. 5.76 Number terminals dialog with selection of numbering scheme

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 also 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 extent, and the handling of PE terminals.

When you click **OK** in the **NUMBER TERMINALS** dialog is clicked, EPLAN numbers the terminal strips as specified.

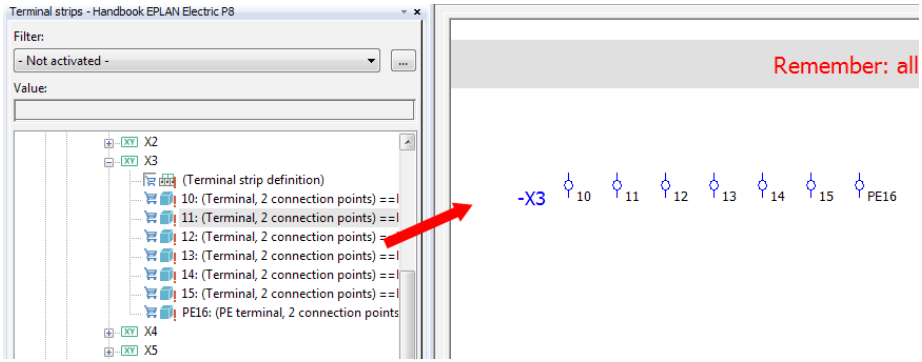


Fig. 5.77 Newly numbered terminals

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.

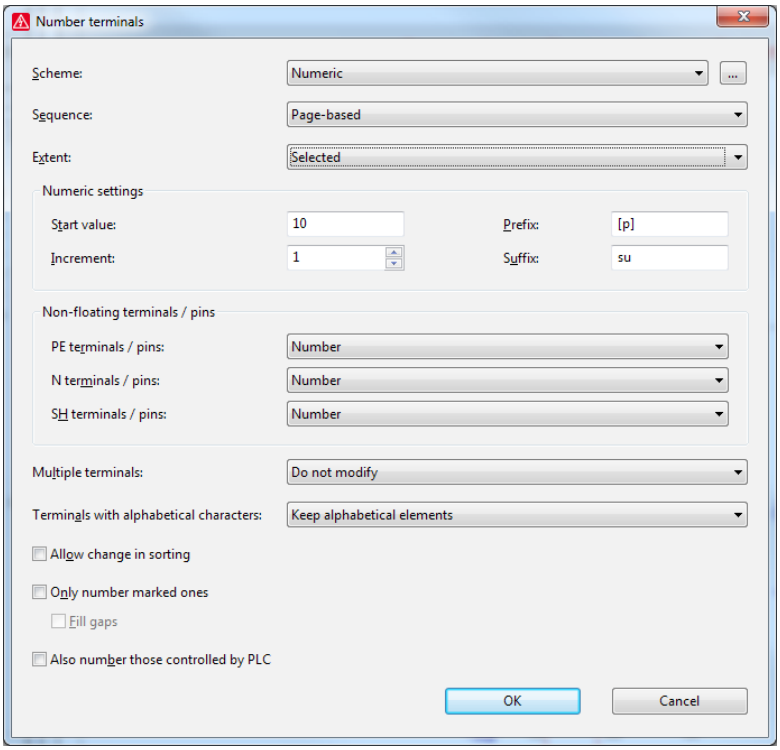


Fig. 5.78 Number terminals dialog

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.

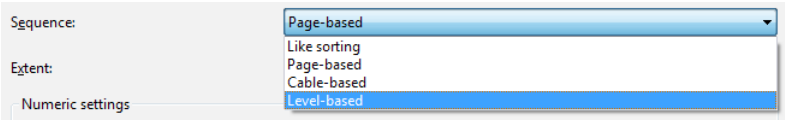


Fig. 5.79 Sequence setting

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?

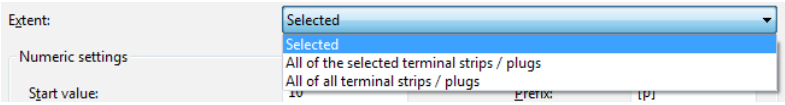


Fig. 5.80 Extent setting

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?

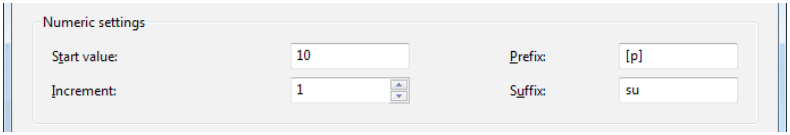


Fig. 5.81 Numeric settings

Numeric settings: This is where the start value, increment, prefix ([n] character preceding the terminal designation) and suffix ([n] character after the terminal designation) are set.

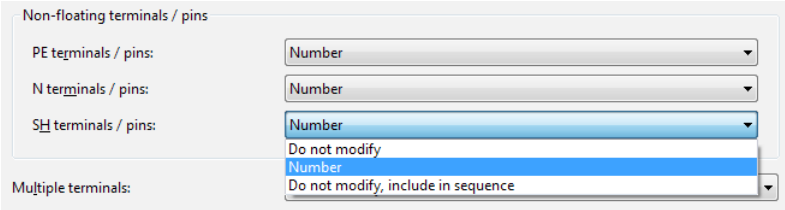


Fig. 5.82 Settings for specific terminal types

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 and associated function definition are crucial here!

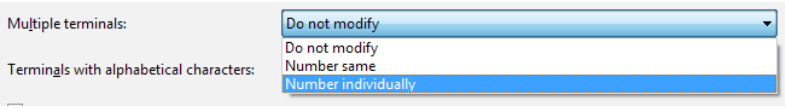


Fig. 5.83 Settings for multiple terminals

Multiple terminals: How should multiple terminals be handled during numbering operations? Should they be left unchanged, numbered all the same, or numbered individually?

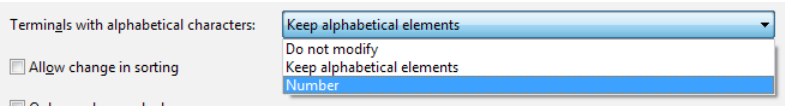


Fig. 5.84 Settings for alphabetically designated terminals

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 four more options.

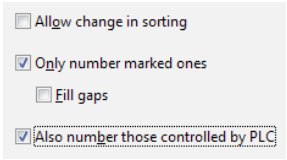


Fig. 5.85 Other options

Allow change in sorting: This affects the terminal storing on the terminal strip. An active option re-sets the (empty) sort codes. But manual sort codes are retained. A non-active option leaves the sorting as it was previously assigned.

Only number marked ones: Only those terminals with the ? placeholder are numbered. Depending on the *Fill gaps* setting, gaps between the terminal numbers (where they already exist, they will continue to be applied) are filled, or, if this option is not used, the terminals are completely renumbered.

The last setting, *Also number those controlled by PLC*, only numbers those terminals that are controlled by PLC.

5.3.4 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**. Only one new terminal is inserted each time!

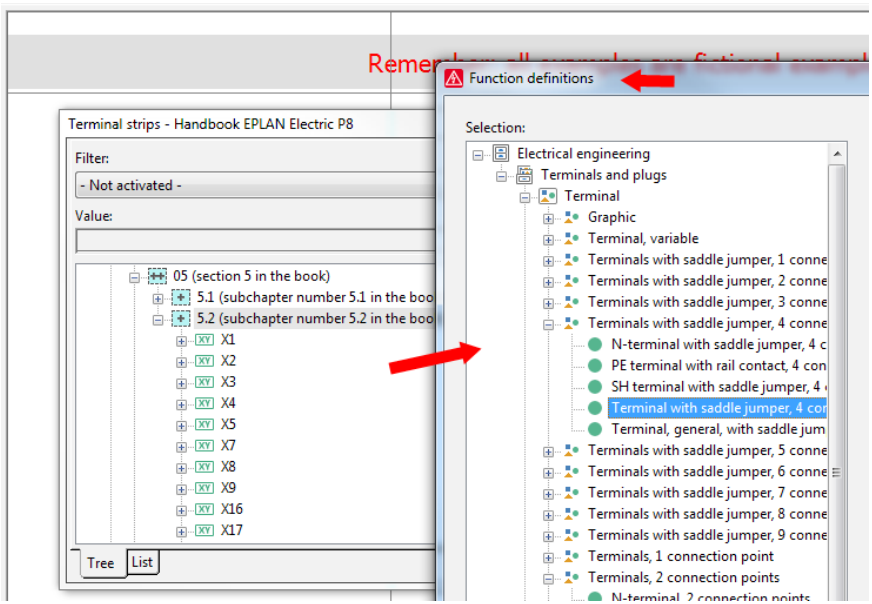


Fig. 5.86
Inserting new terminals

5.3.5 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.

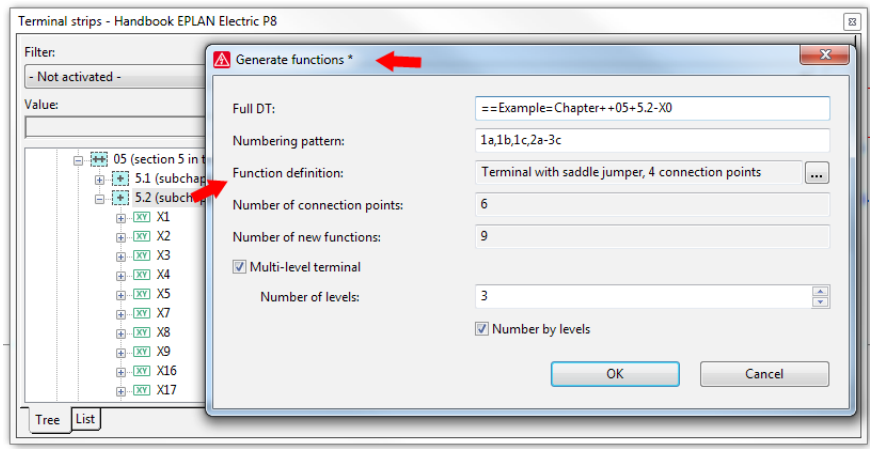


Fig. 5.87
Generate functions dialog

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.

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.

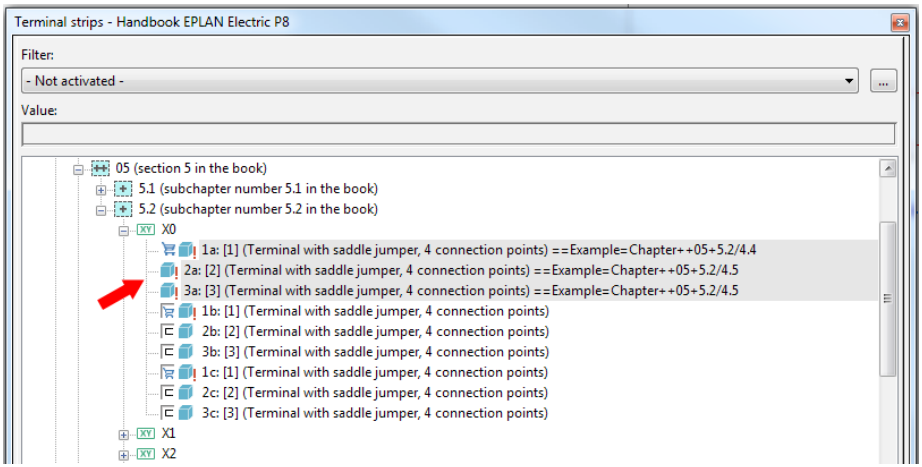


Fig. 5.88 New terminal strip with placed and unplaced terminals

NOTE: The newly created terminals do not have any parts data. This data must (can) be assigned later using device selection.

5.3.6 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.

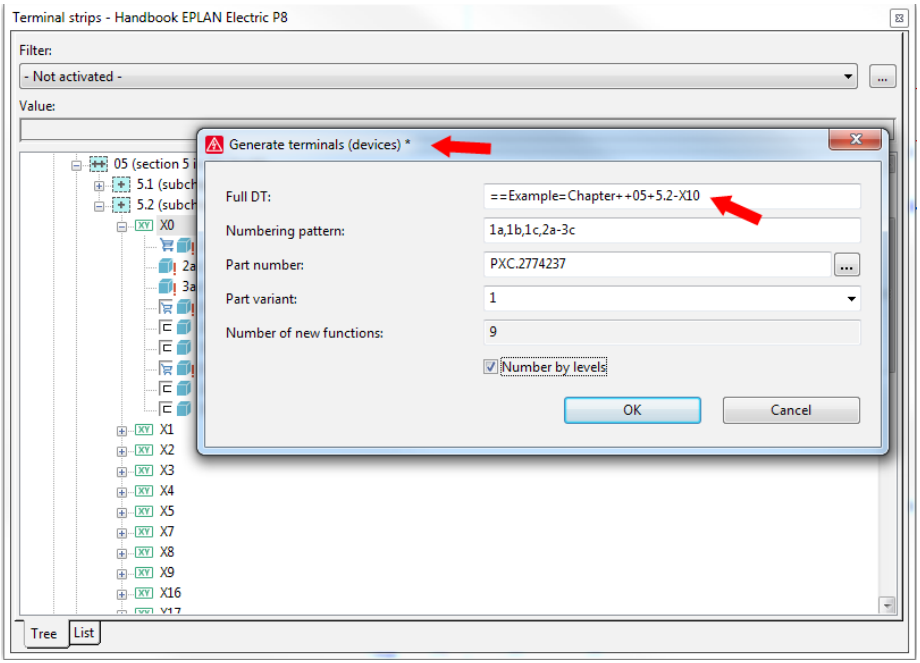


Fig. 5.89 Generate terminals (devices) dialog

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.

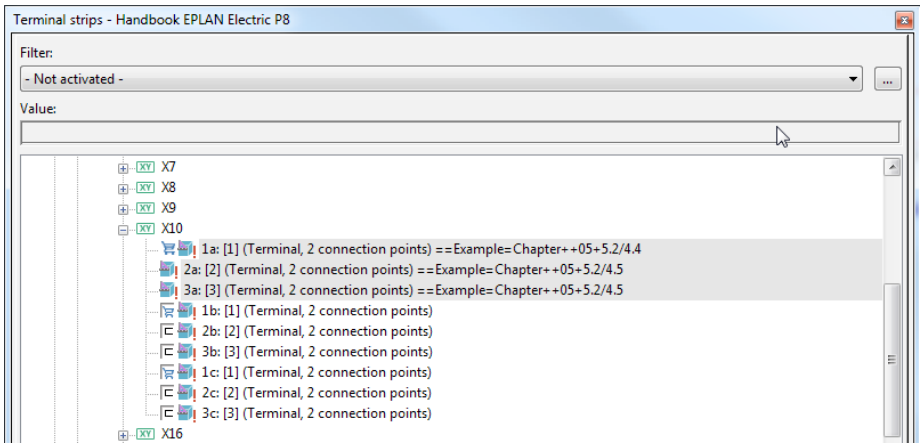


Fig. 5.90 Generated, new terminals (devices)

In contrast to the last section, these new terminals are real devices. They already have parts data.

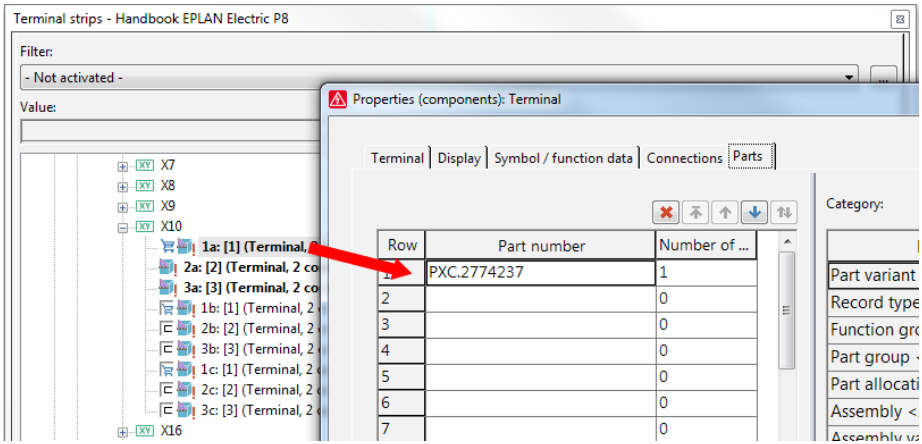


Fig. 5.91 Terminal parts data

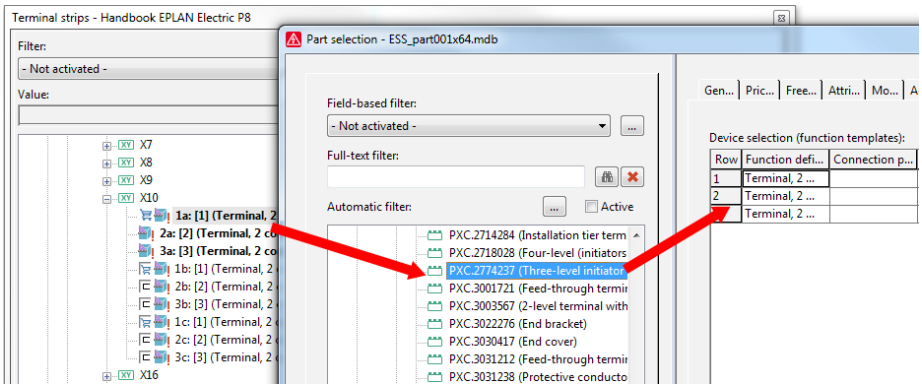


Fig. 5.92 Function template of the terminal part

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. These are so-called **auxiliary terminals**.

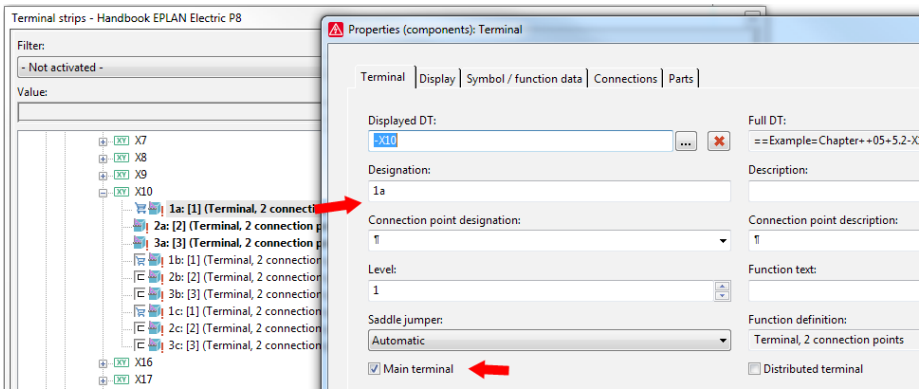


Fig. 5.93 Main terminal

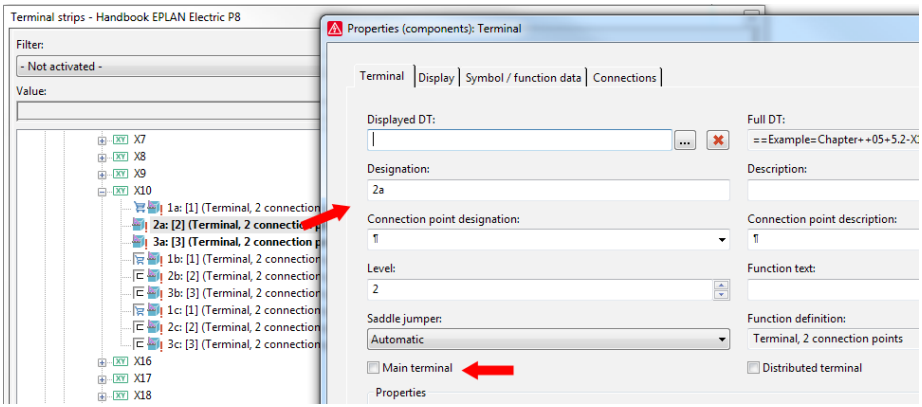


Fig. 5.94 Auxiliary terminal

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 View

The terminal strip navigator provides two different views of its devices (here: terminals and terminal strips). You switch to this view via the popup menu of the right mouse button.

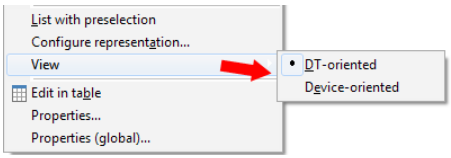


Fig. 5.95 Selection of view

5.3.7.1 DT-oriented

The DT-oriented view is the familiar view of the DT of a terminal strip.

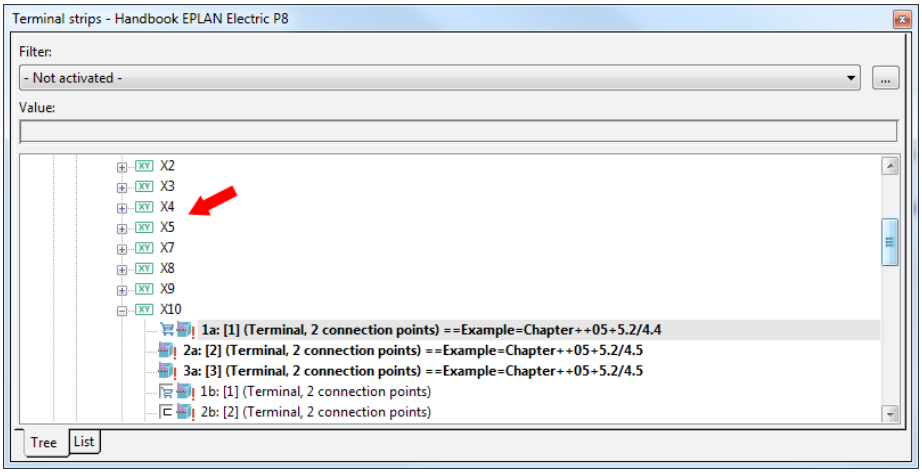


Fig. 5.96 DT-oriented view

5.3.7.2 Device-oriented

The second, device-oriented view switches the view to the devices, such as multi-level terminals.

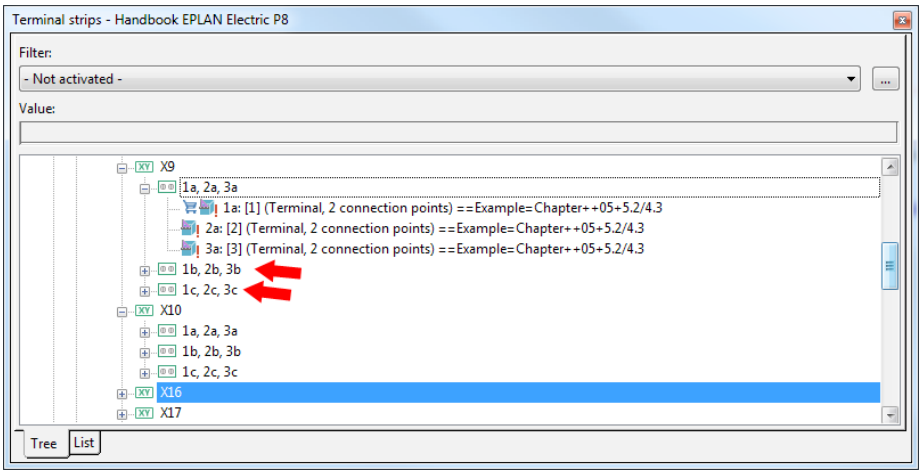


Fig. 5.97 Device-oriented view

5.4 Plug (female pin) 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 Edit (plug/female pin editor)

In addition to the actual **plug navigator** (which also includes the female pins), **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.

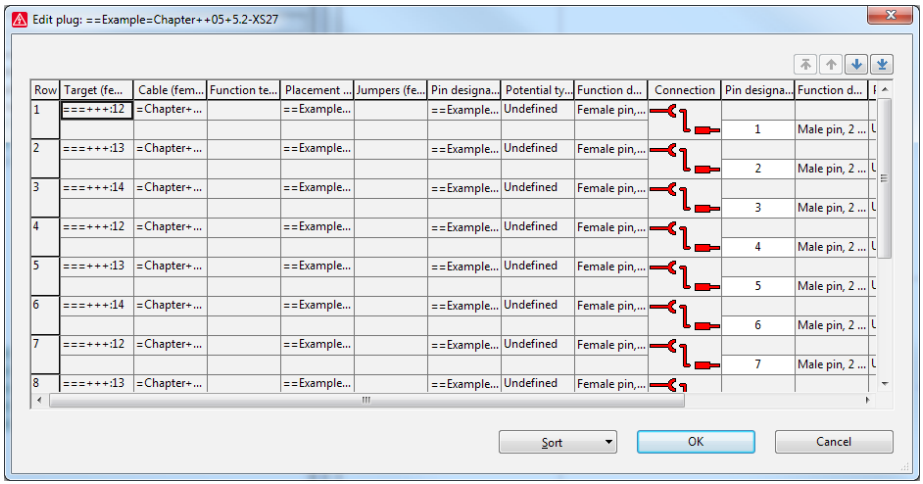


Fig. 5.98 Edit plug dialog

5.4.2 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, of course.

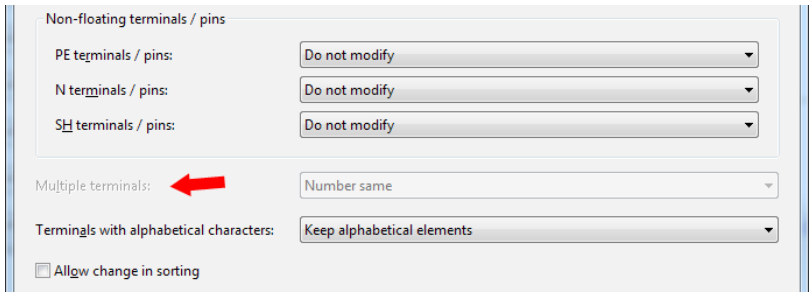


Fig. 5.99 Number plugs dialog

5.4.3 New

The popup menu of the **plug navigator** (accessible by right-clicking) contains the **NEW…** menu item. The procedure here is identical to the **NEW…** function in the **terminal navigator**. Here, too, a new function definition can be selected. In contrast to terminals, this is only possible for plugs and sockets.

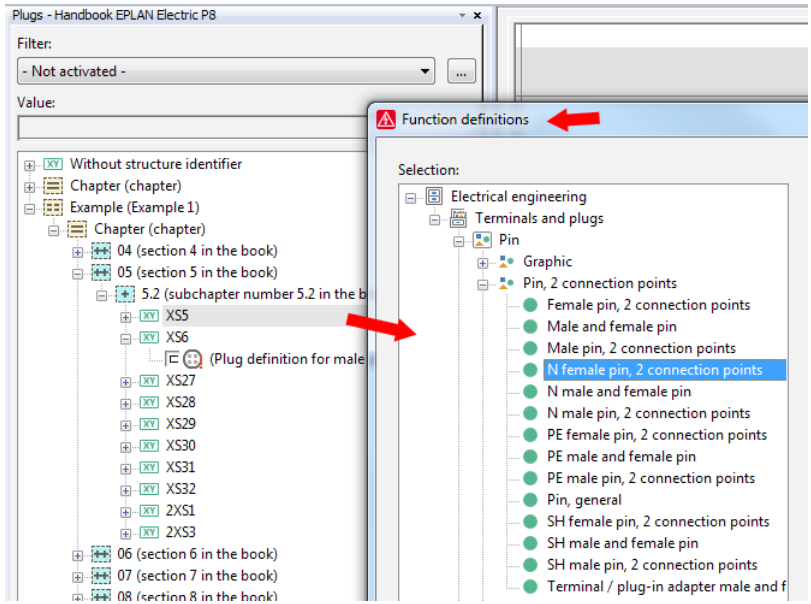


Fig. 5.100 Creating a new, individual plug or female pin

5.4.4 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.

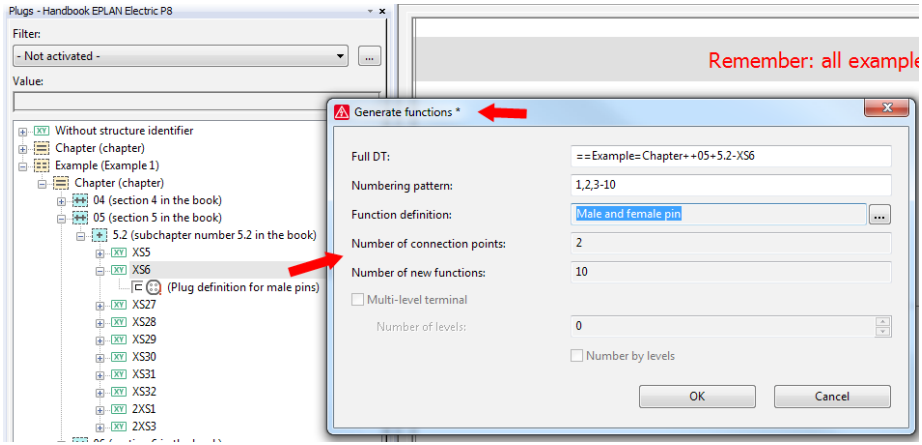


Fig. 5.101 Example of generating new functions

In contrast to generating new terminal functions, here it is not possible either to create multi-level terminals. The remaining procedures are the same as in the terminal strip navigator.

5.4.5 Generate plug definition (plug and/or female pin)

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.

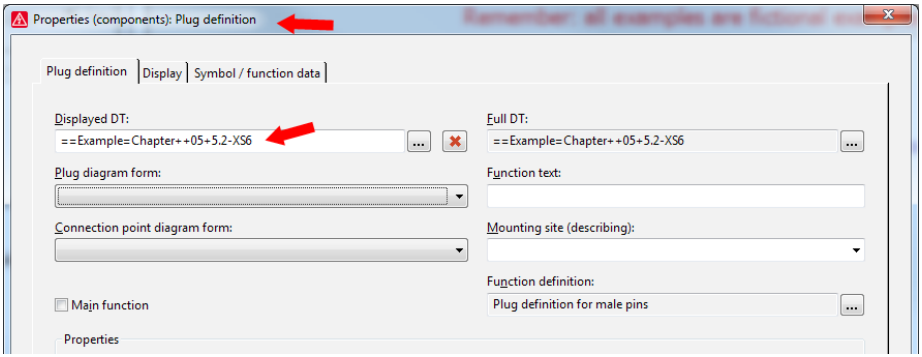


Fig. 5.102 New definition for male pins

5.4.6 Generate pin (plug and/or female 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.

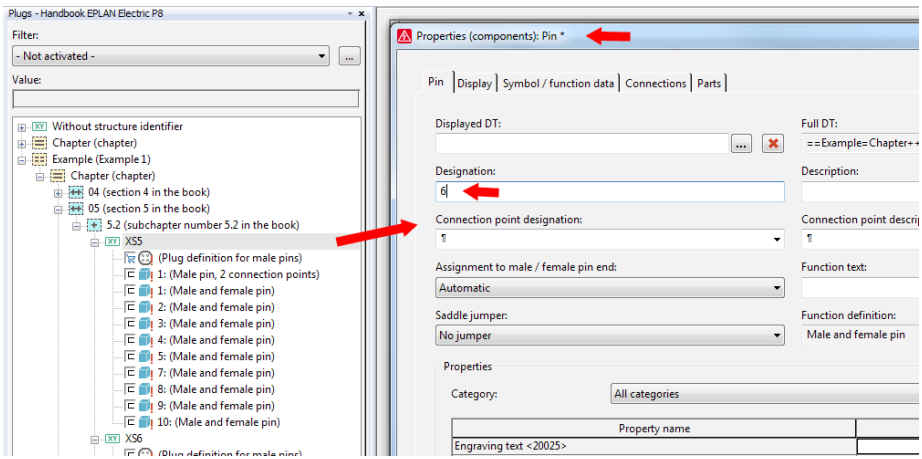


Fig. 5.103 Creating a male pin

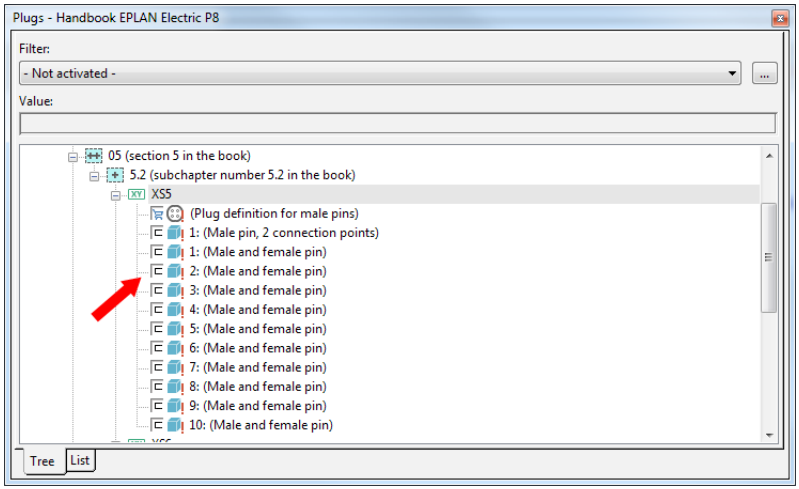


Fig. 5104 Unplaced male pins

In the **Displayed DT** field, use ... to **BRANCH** to 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 male pin by clicking **OK**.

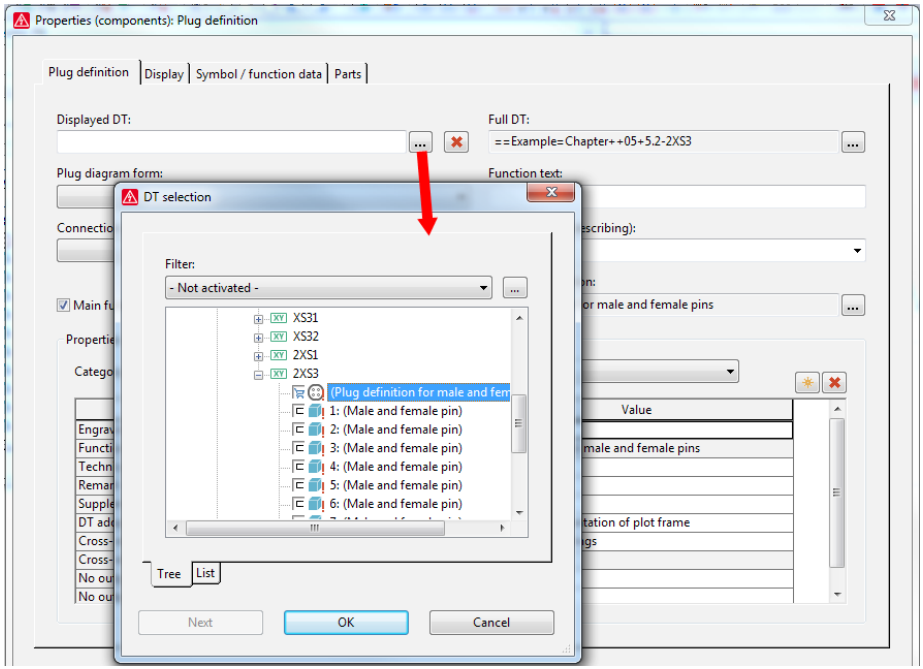


Fig. 5.105 DT selection

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 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.

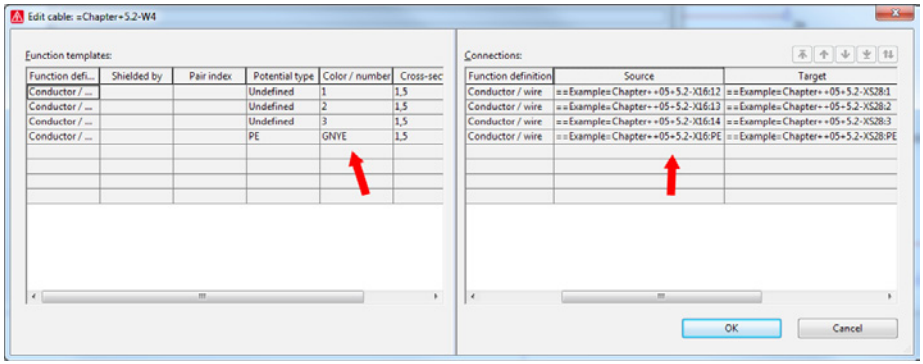


Fig. 5.106 Assignment conductor to source and target DT

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 in Fig. 5.107 and Fig. 5.108.

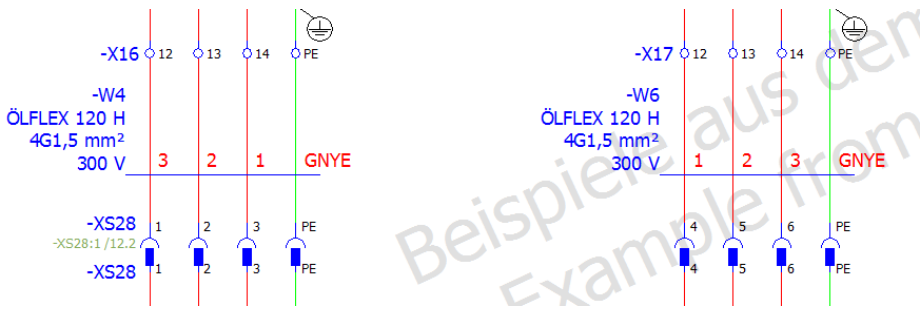


Fig. 5.107 Conductor assignment before the Edit cable dialog

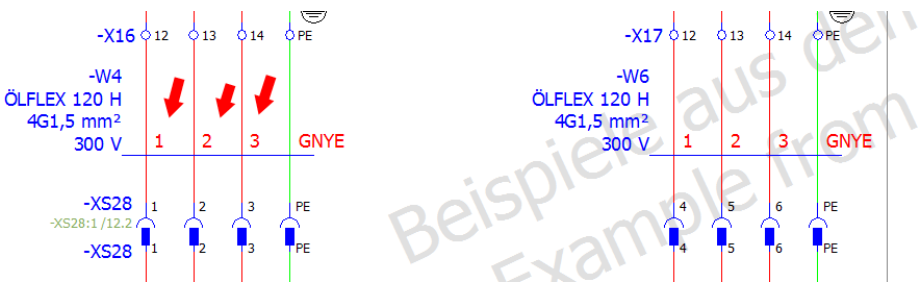


Fig. 5.108 Conductor assignment after exiting the Edit cable dialog

5.5.2 Number (cable DT)

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.

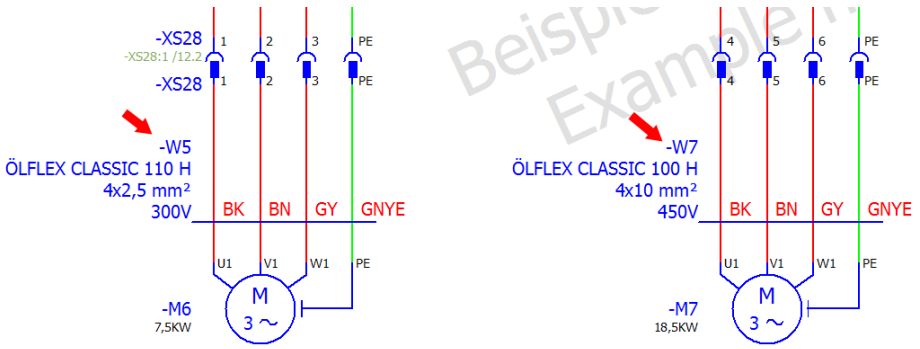


Fig. 5.109 Selected cables

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.

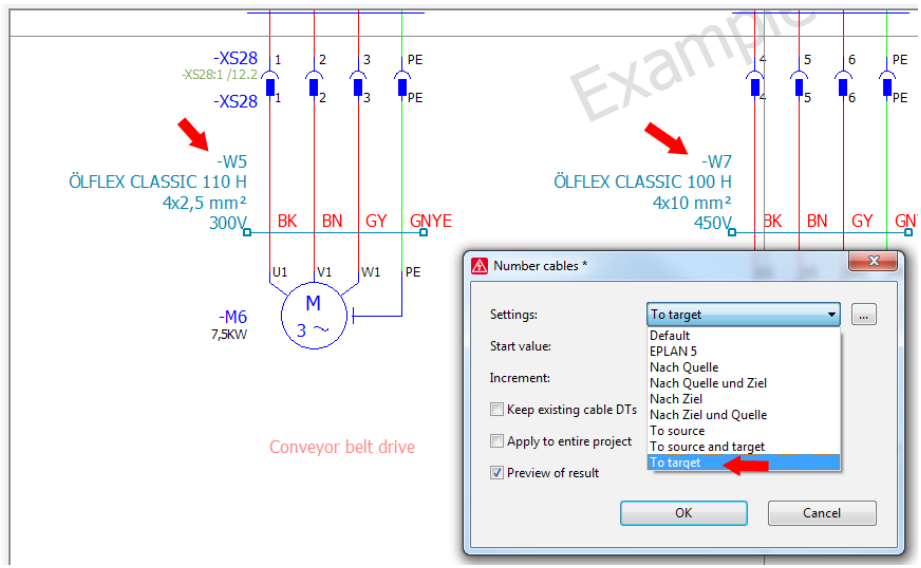


Fig. 5.110 Number cables dialog

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.

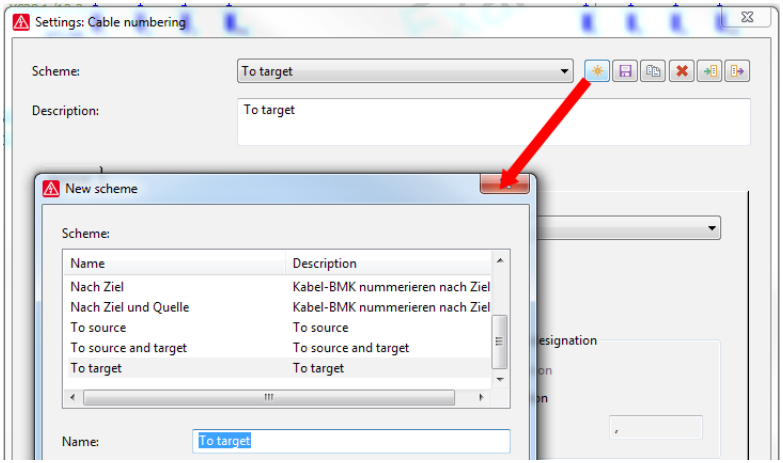


Fig. 5.111 Settings / Cable numbering dialog

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.

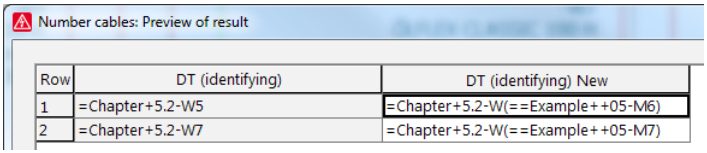


Fig. 5.112 Preview of result dialog

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.

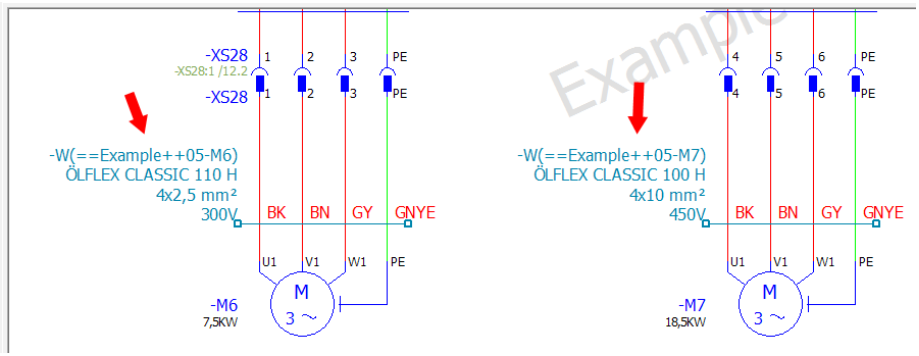


Fig. 5.113 Modified cable device tags

The default settings for the schemes of 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).

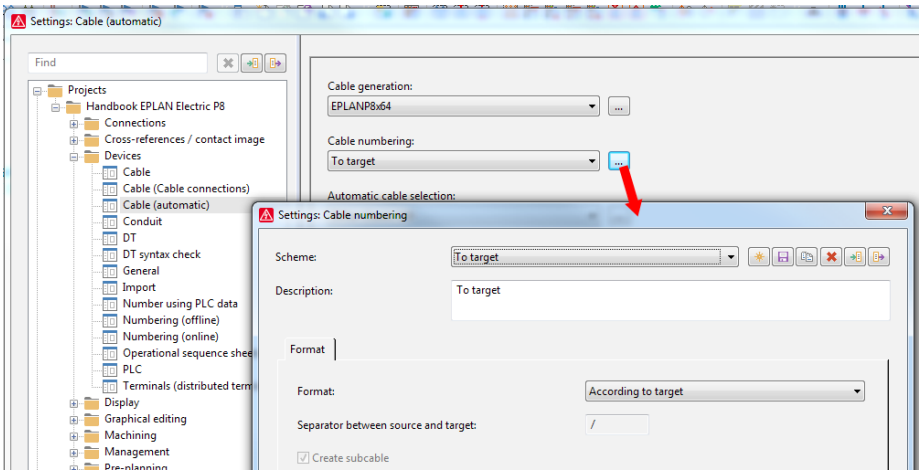


Fig. 5.114 Default settings for cable-editing functions

5.5.3 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.

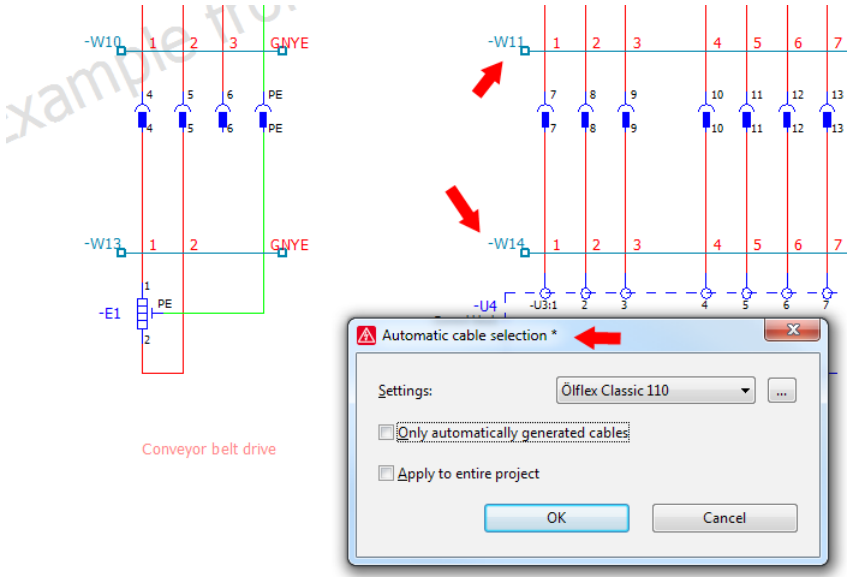
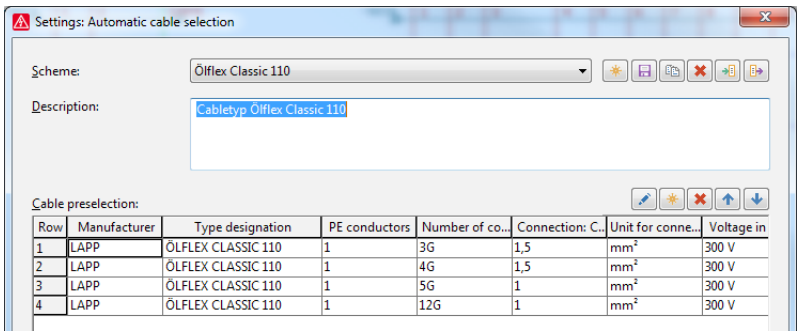


Fig. 5.115 Automatic cable selection dialog

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.

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.



Row	Manufacturer	Type designation	PE conductors	Number of co...	Connection: C...	Unit for conne...	Voltage in
1	LAPP	ÖLFLEX CLASSIC 110	1	3G	1,5	mm²	300 V
2	LAPP	ÖLFLEX CLASSIC 110	1	4G	1,5	mm²	300 V
3	LAPP	ÖLFLEX CLASSIC 110	1	5G	1	mm²	300 V
4	LAPP	ÖLFLEX CLASSIC 110	1	12G	1	mm²	300 V

Fig. 5.116 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.

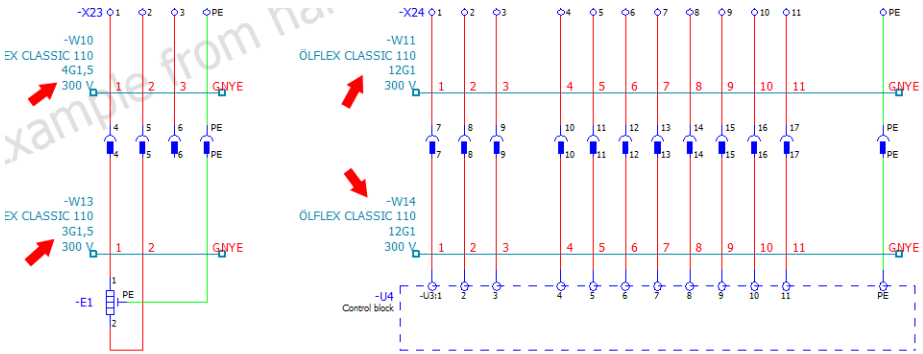


Fig. 5.117 Automatically generated cable data

5.5.4 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 connections 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.

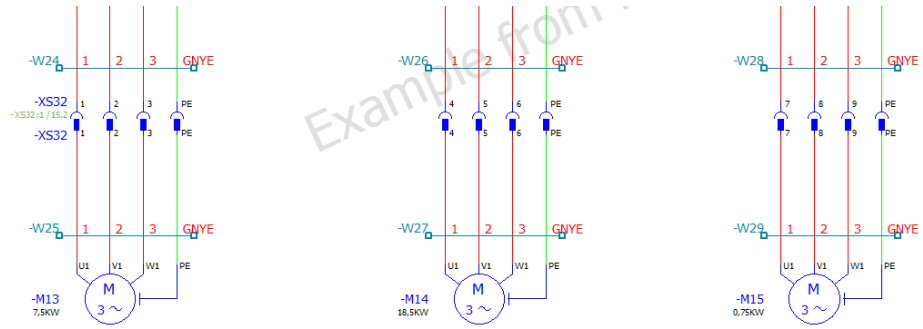


Fig. 5.118 Selecting existing cables

The **GENERATE CABLES AUTOMATICALLY** function in the **PROJECT DATA/CABLES** menu is then called up.

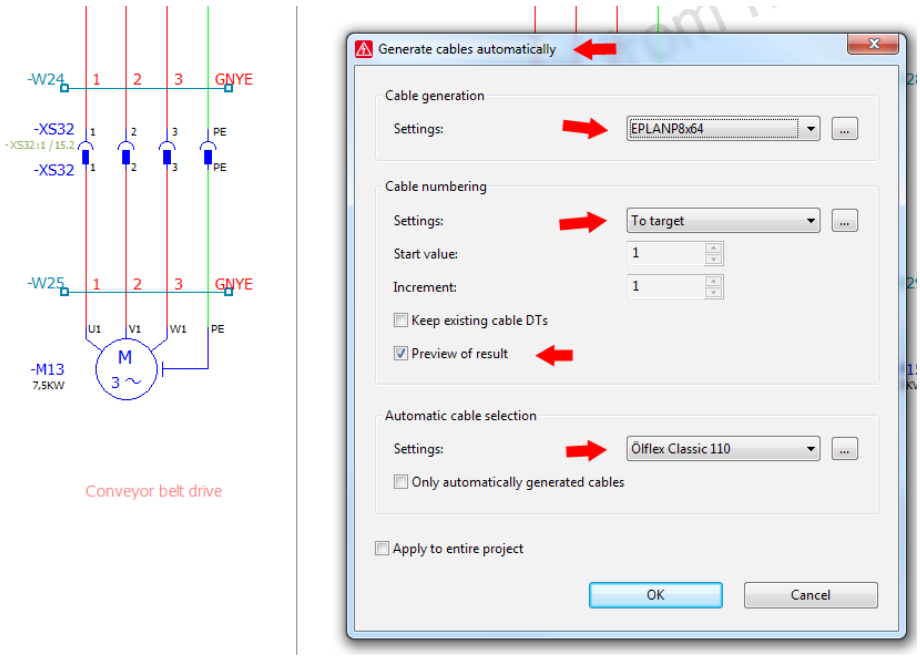


Fig. 5.119 Generate cables automatically dialog, with its settings

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.

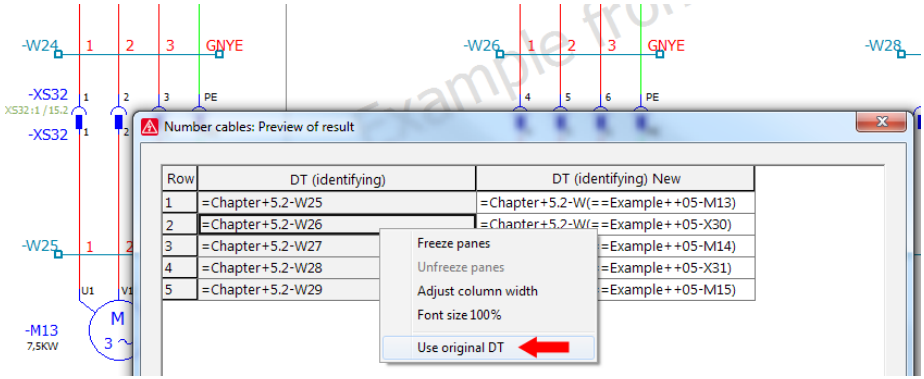


Fig. 5.120 Preview of result

As in other Preview of result dialogs, you can use the popup menu 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.

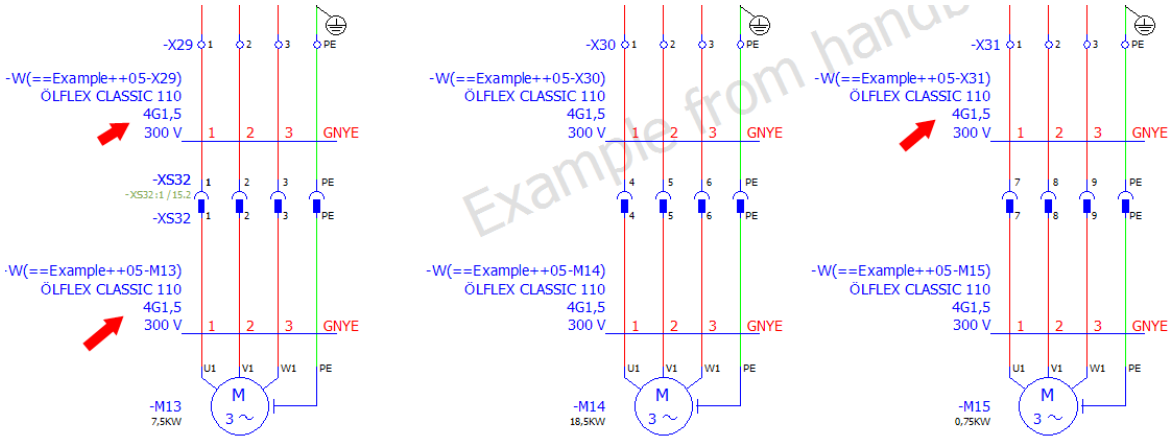


Fig. 5.121 Automatically generated cable data

5.5.5 Assign cable connections

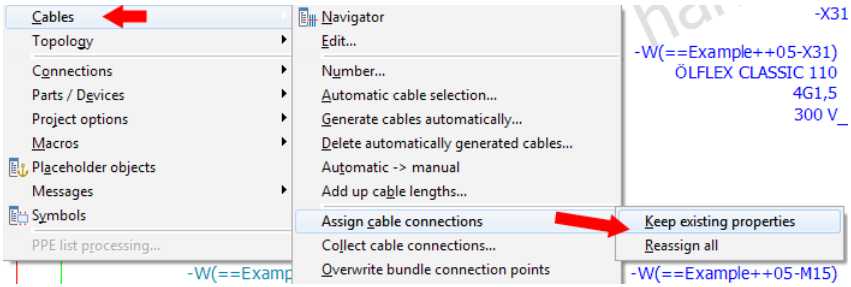


Fig. 5.122 Assign conductors menu item in the Cables menu

In the **PROJECT DATA/CABLES/ASSIGN CABLE CONNECTIONS** 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).

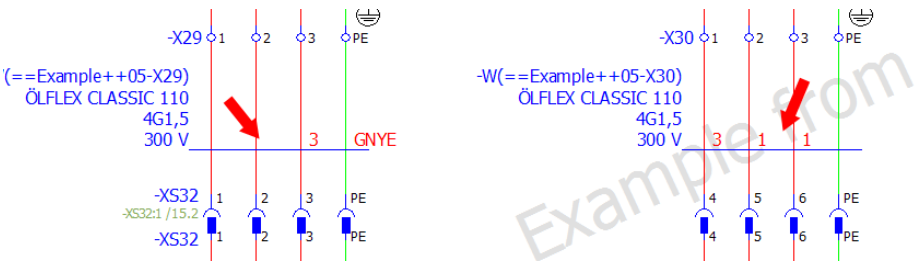


Fig. 5.123 Cables are selected by main function

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 unde-

fined connection definition points (conductor designations), whereby existing conductor designations are not overwritten.

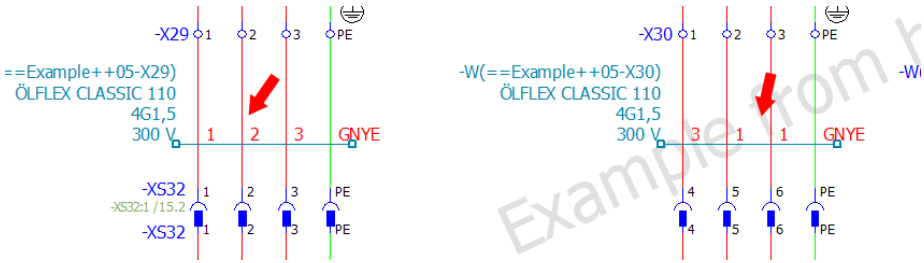


Fig. 5.124 Previously undesignated conductors were given designations.

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.

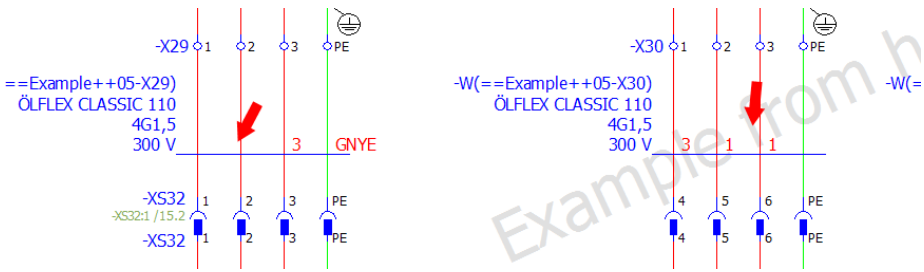


Fig. 5.125 Selecting cables

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.

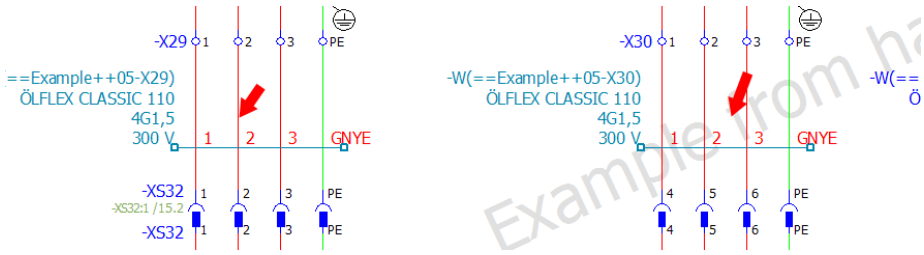


Fig. 5.126 All conductors were reassigned.

5.5.6 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.

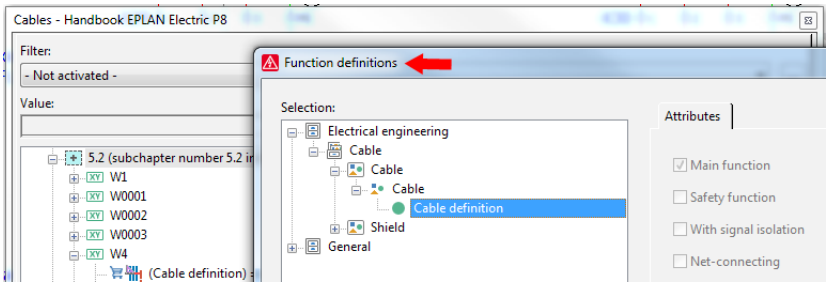


Fig. 5.127 Creating new cable definition

After the import, EPLAN creates a new cable definition and opens the **PROPERTIES (COMPONENTS): CABLE** dialog. Here, the other fields (cable type, etc.) can be filled. If all data has been entered, you can close the dialog by clicking **OK**.

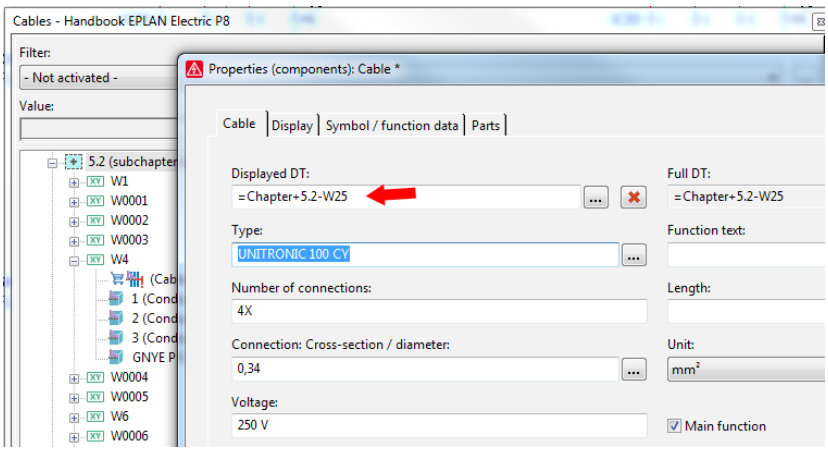


Fig. 5.128 Properties (components): Cable dialog

EPLAN creates the new cable and places it in the structure of the project.

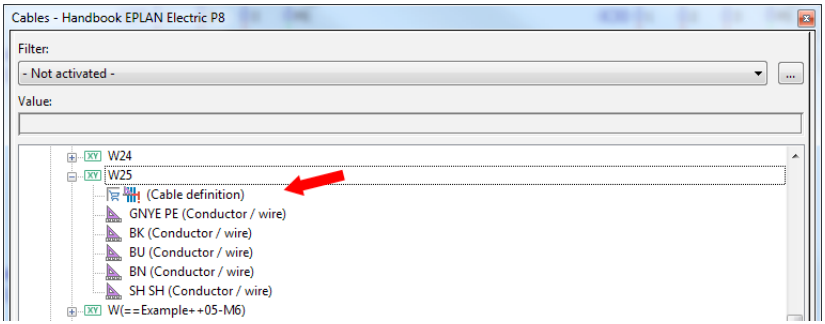


Fig. 5.129 New cable

5.5.7 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 DT**, 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.

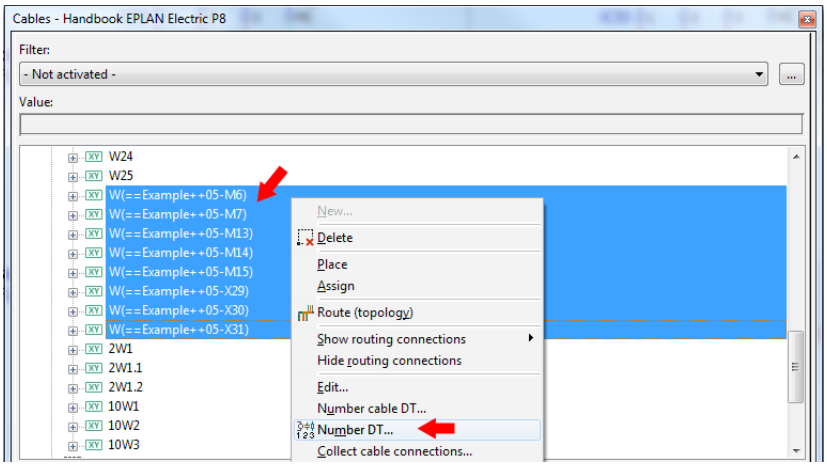


Fig. 5.130 Selected cables in the cable navigator

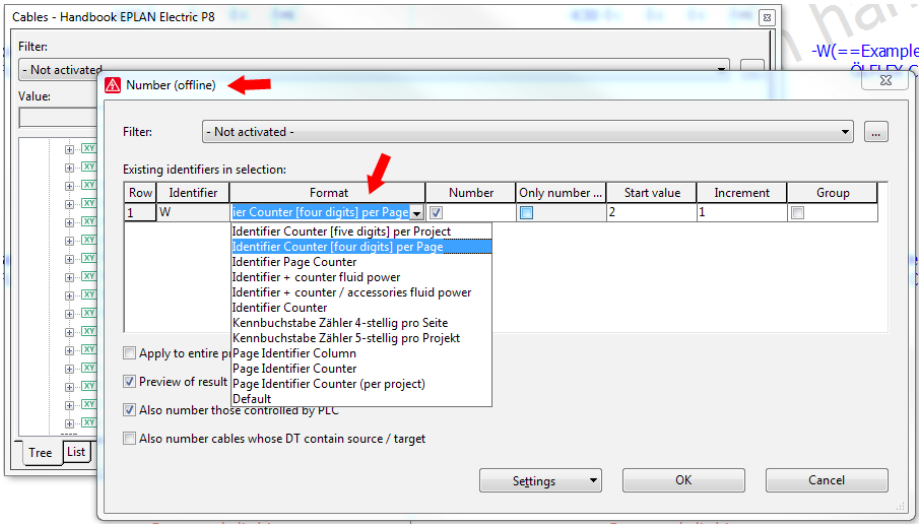


Fig. 5.131 Number (offline) dialog

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.

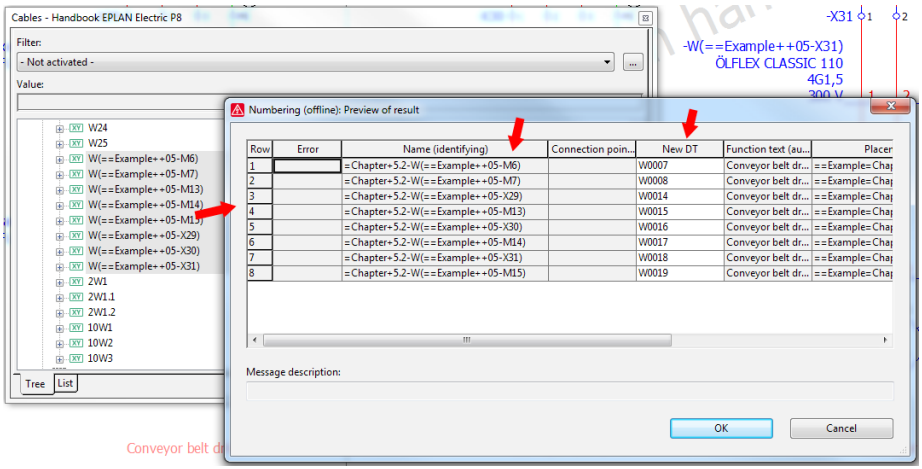


Fig. 5.132 Preview

Fig. 5.133 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*.

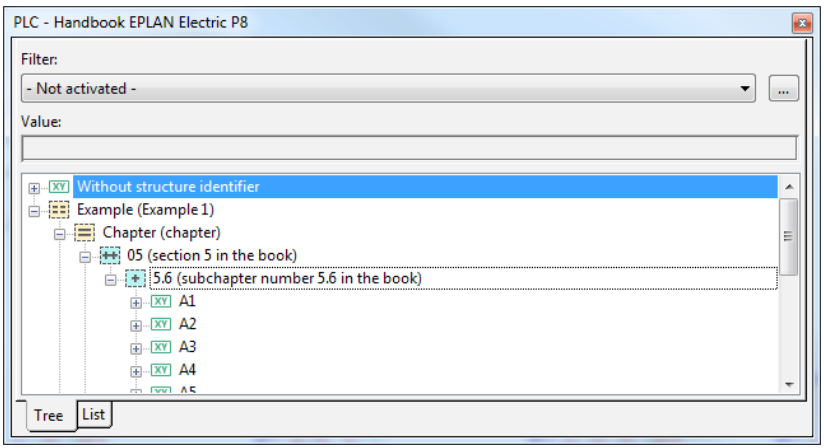


Fig. 5.134 PLC navigator

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**.

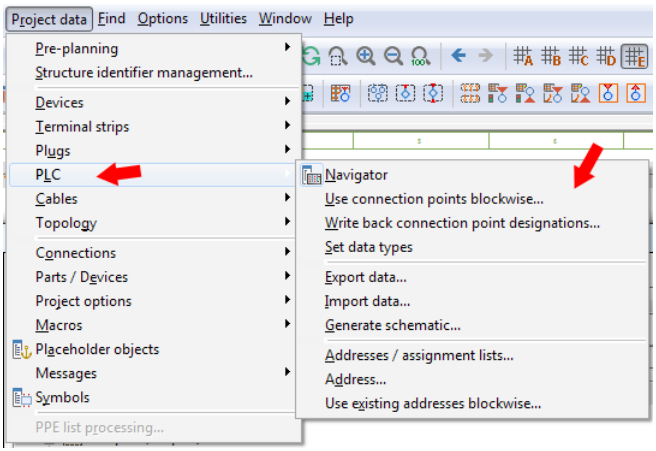


Fig. 5.135 Project data / PLC menu

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.

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.

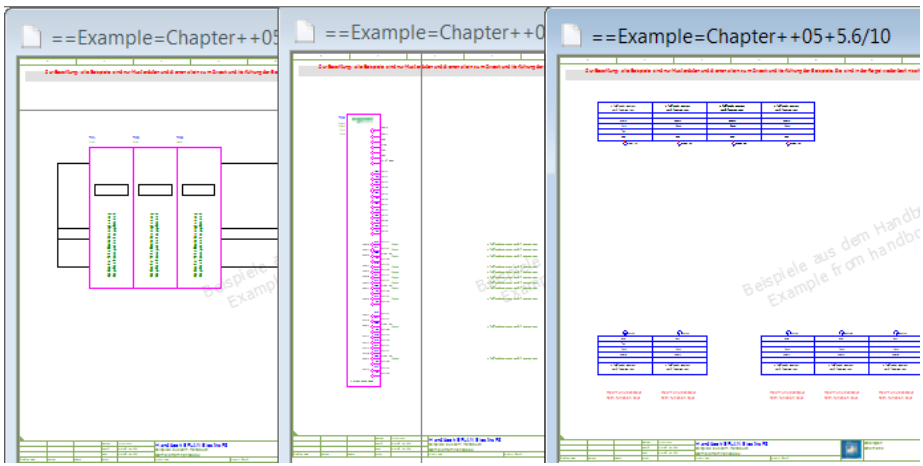


Fig. 5.136 Graphical overview / Logical overview / Distributed representation

The various ways of editing PLC data are described in sections 5.6.1 to 5.6.9, 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 Write back connection point descriptions

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.

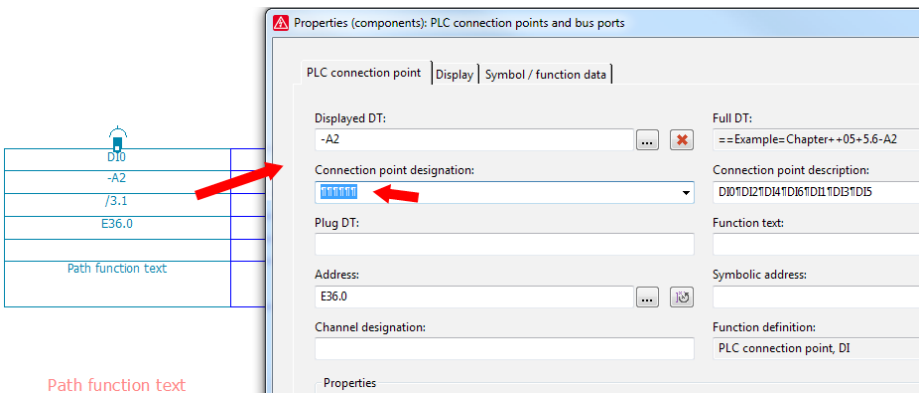


Fig. 5.137 Added distributed PLC data in schematic

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.

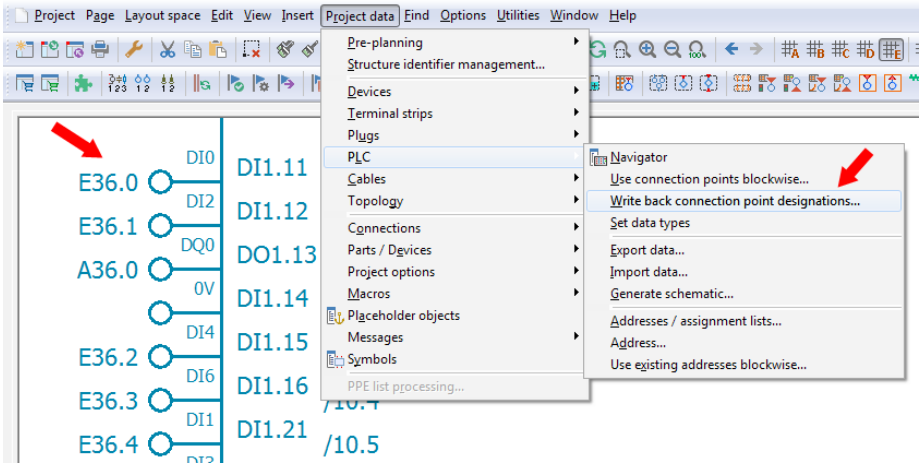


Fig. 5.138 Running the Write back connection point designations function

EPLAN opens the **WRITE BACK CONNECTION POINT DESIGNATIONS** dialog. Here, select the desired CPU and appropriate options. 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.

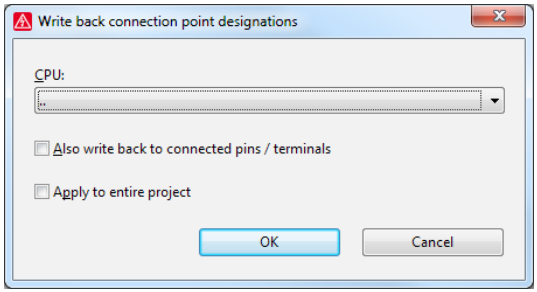


Fig. 5.139 Write back connection point designations dialog

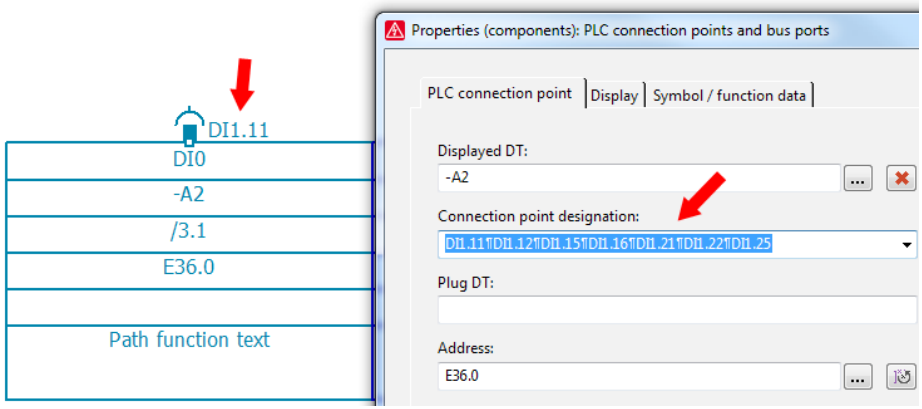


Fig. 5.140 Written-back connection point designations

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 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.

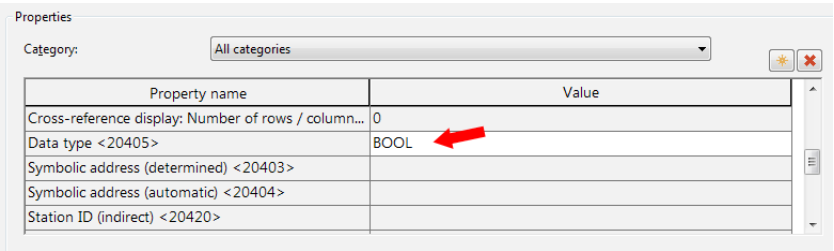


Fig. 5.141 Data type set

5.6.3 Export/Import data

This menu item allows for PLC data to be exported/imported after various PLC controls.

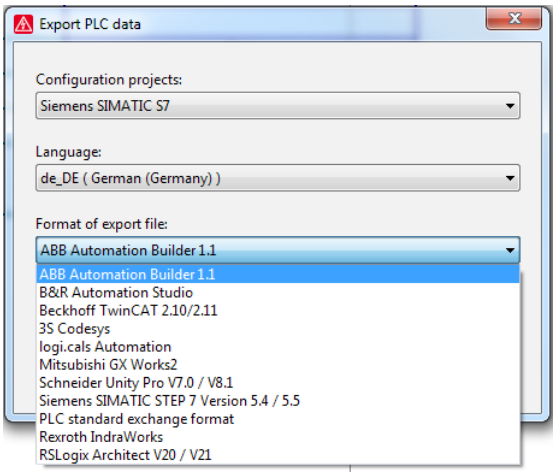


Fig. 5.142
Export PLC data dialog

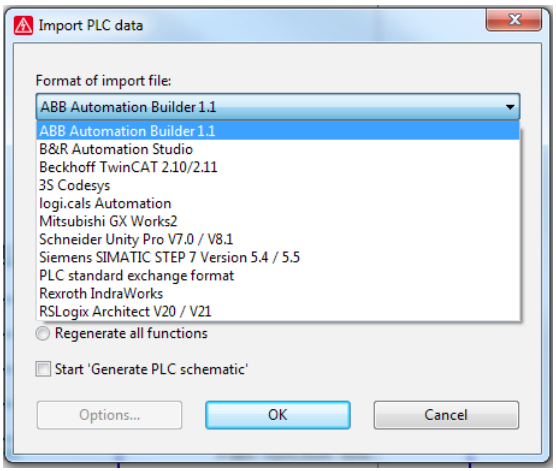


Fig. 5.143
Import PLC data dialog

5.6.4 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 and CPUs used in the diagram).

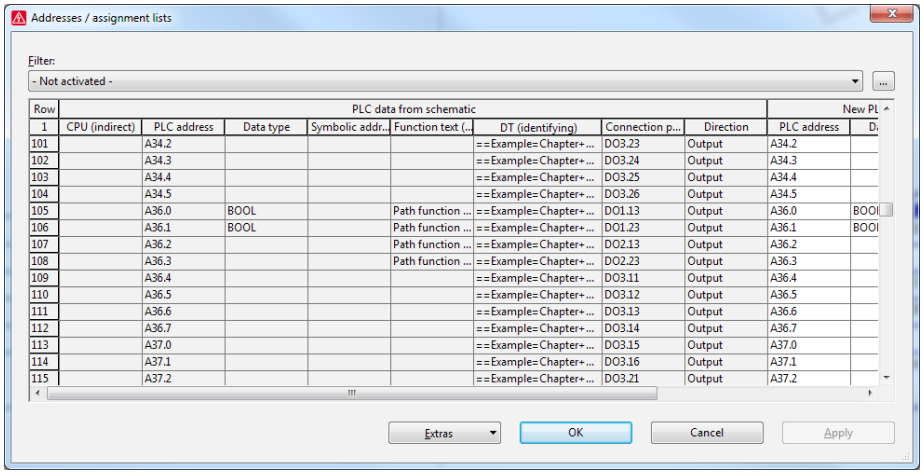


Fig. 5.144 Addresses/assignment lists dialog

In this dialog, the properties like the *PLC address*, the *Symbolic address* and the defined *Function text* of the respective CPU area selected can also be changed manually. 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). But before such an action, the CPU must be defined via a filter. If this has not been done, EPLAN will display a message.

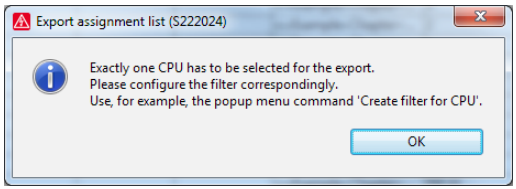


Fig. 5.145 Message about missing CPU filter

CREATE FILTER FOR CPU can be selected from the popup menu. EPLAN reads out the CPUs and creates a suitable filter automatically.

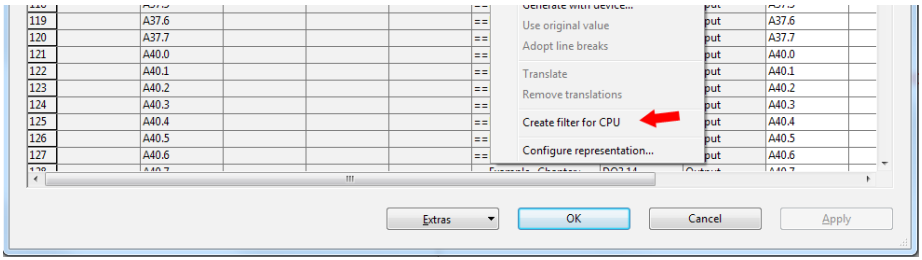


Fig. 5.146 Popup menu with Create filter for CPU command

After clicking it, EPLAN creates the filter.

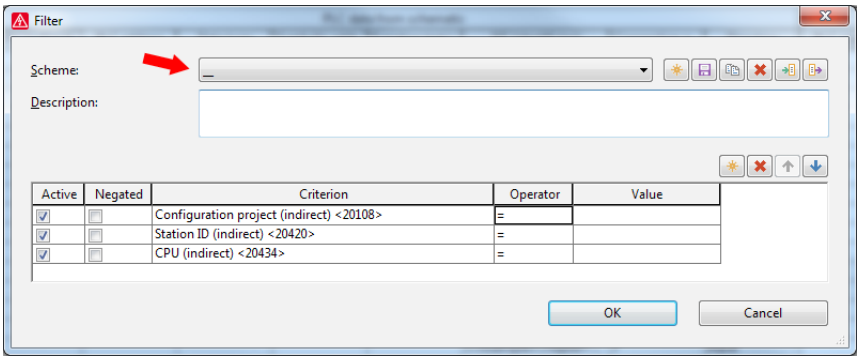


Fig. 5.147 Automatic filter generated by EPLAN

Click the **EXTRAS** button in the lower area of the dialog, and EPLAN opens another menu.

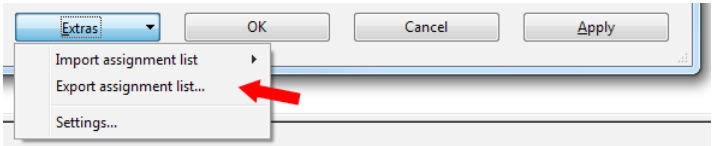


Fig. 5.148 The Extras button and its functions

This allows *assignment lists* to be *exported* or *imported*, including new CPU assignment lists. Different settings may be required, depending on the PLC and its software.

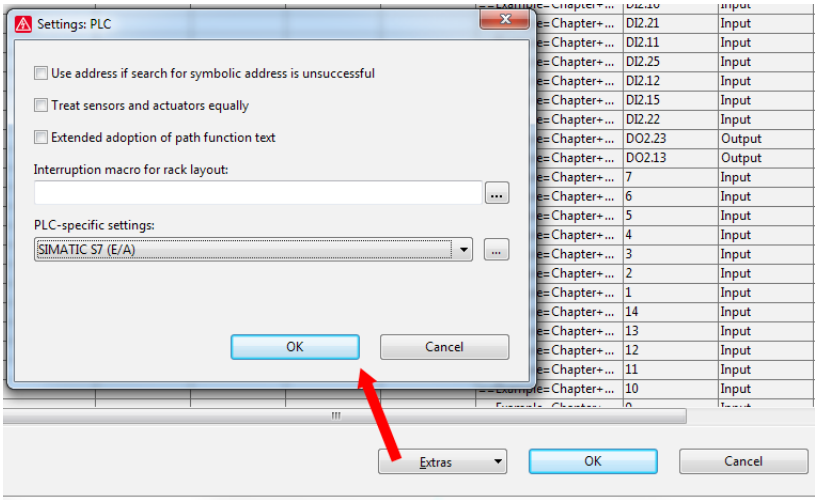


Fig. 5.149
PLC settings

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*.

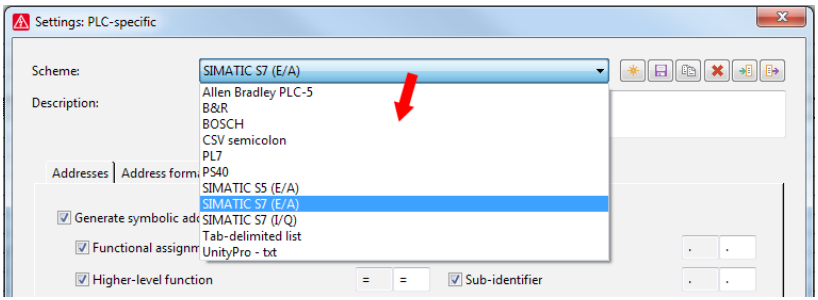


Fig. 5.150
Various PLC schemes

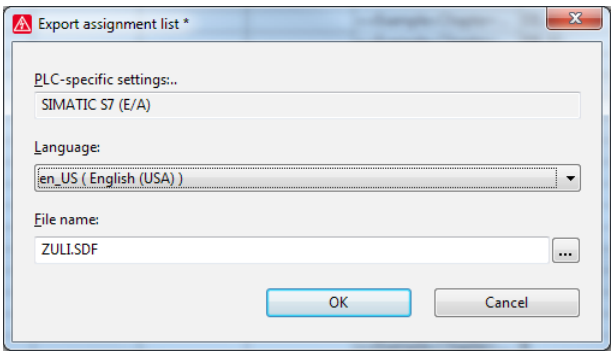


Fig. 5.151
Export assignment lists dialog

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. Depending on the existing CPUs (there might be several controls in the project), select the correct option before an export in the **ADDRESSES/ASSIGNMENT LISTS** dialog.

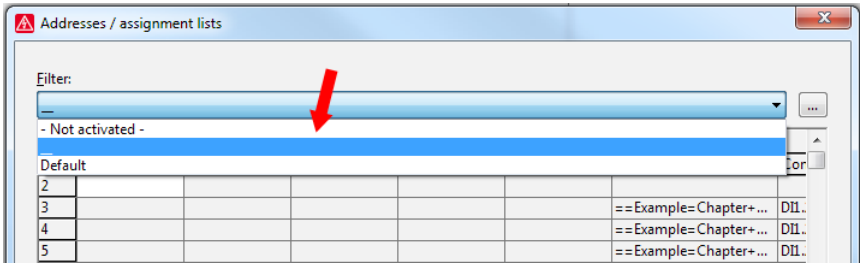


Fig. 5.152 CPU filter

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.

5.6.5 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 sufficient to select the PLC card in the PLC navigator (but it should no longer have any messages from message management, that is, it must be “error-free”). This ensures that all of the functions, those of the logical PLC overview and the PLC functions represented in distributed view, are included and subsequently 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.

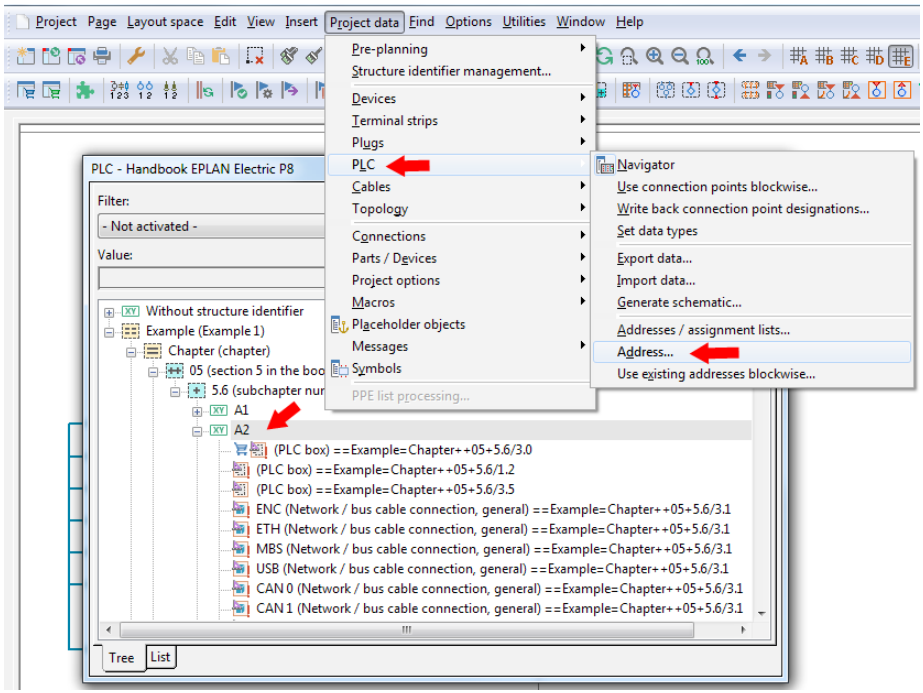


Fig. 5.153 Selecting the card to be addressed and calling up addressing

EPLAN now opens the **READDRESS PLC CONNECTION POINTS** dialog.

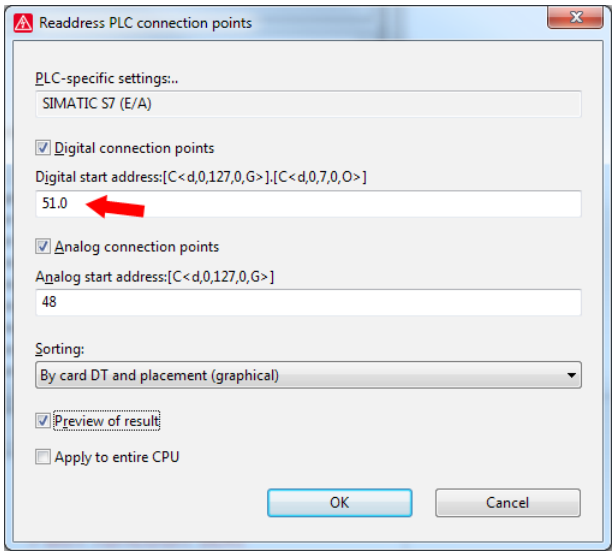


Fig. 5.154 Defining start addresses

In the **Digital start address** and **Analog start address** fields, the new address is entered *without* a prefix. EPLAN obtains the relevant 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 primarily during addressing, and ignore the setting of the *PLC-specific settings* scheme.

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, and you can use the **CANCEL** button to stop the process if necessary.

Address PLC connection points

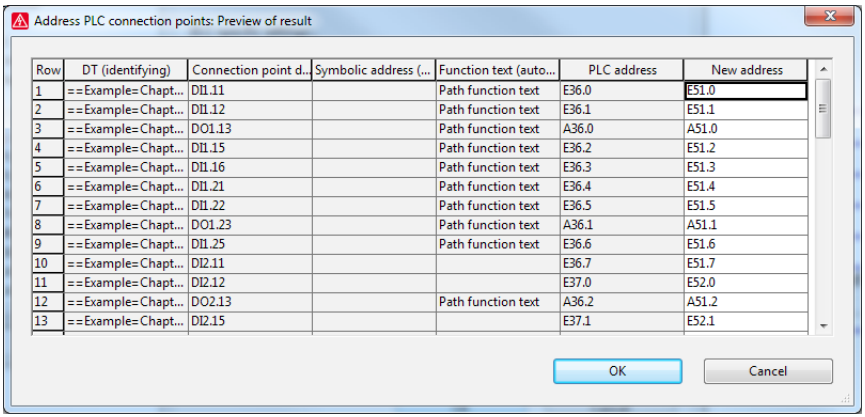


Fig. 5.155 Checking readdressing

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.

5.6.6 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.

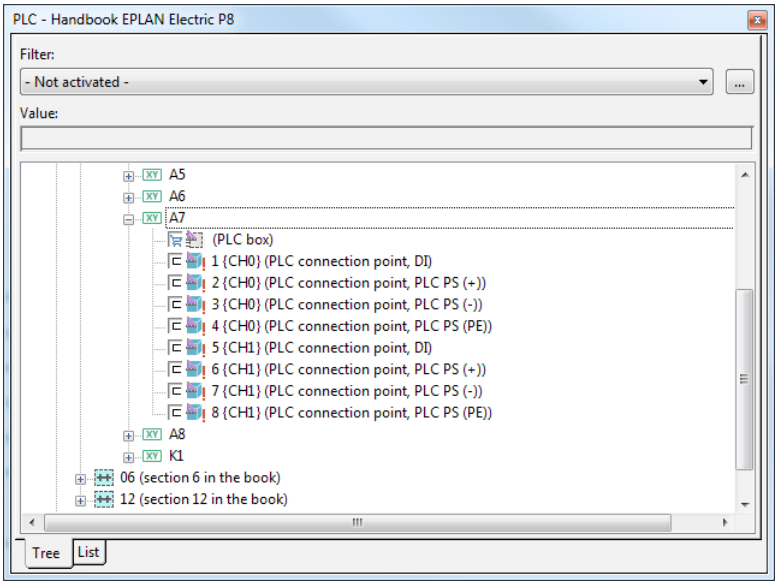


Fig. 5.156 New, unplaced PLC card in the navigator

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.

5.6.7 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.

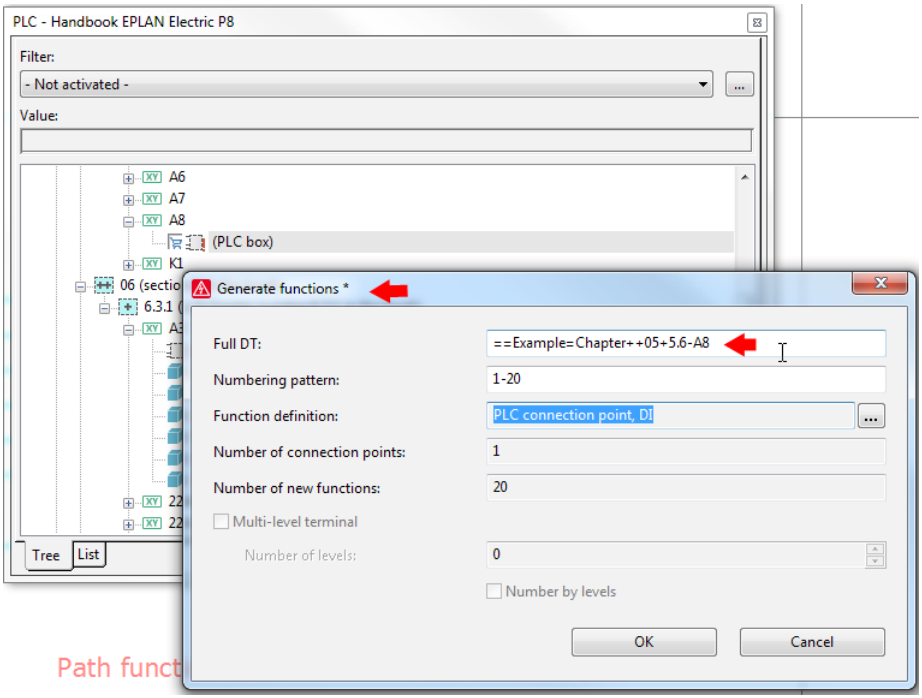


Fig. 5.157 Generating new PLC functions

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* (via the **PROJECT DATA/PLC/ADDRESS** menu) and their *Connection point designations*, ready to be placed.

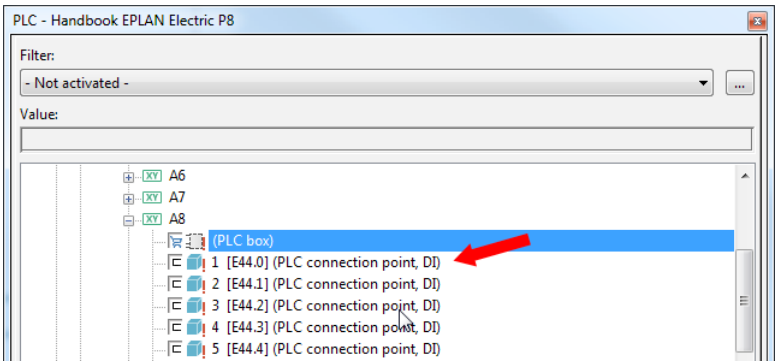


Fig. 5.158 New PLC functions

5.6.8 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**.

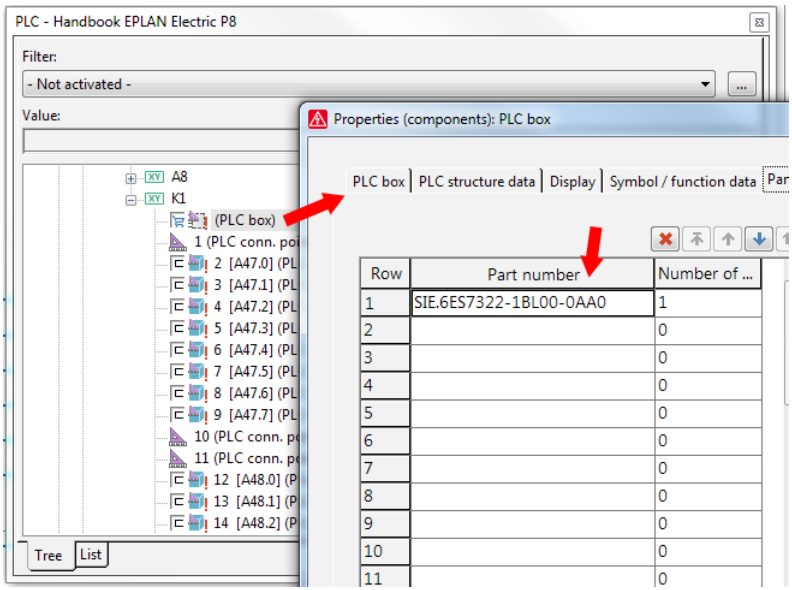


Fig. 5.159
New, unplaced device in the navigator

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.

5.6.9 View

The **PLC navigator** in EPLAN provides different *views* of the PLC data: **DT-oriented**, **Address-oriented**, **Channel-oriented**, and **Rack-oriented**.

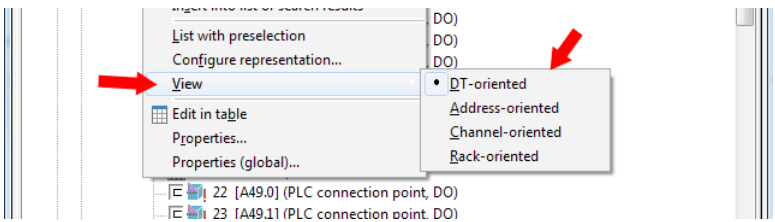


Fig. 5.160
Popup menu selection of views

5.6.9.1 DT-oriented

The DT-oriented view shows the DT and all functions below it. In the DT-oriented view, the PLC data is listed according to its device tag.

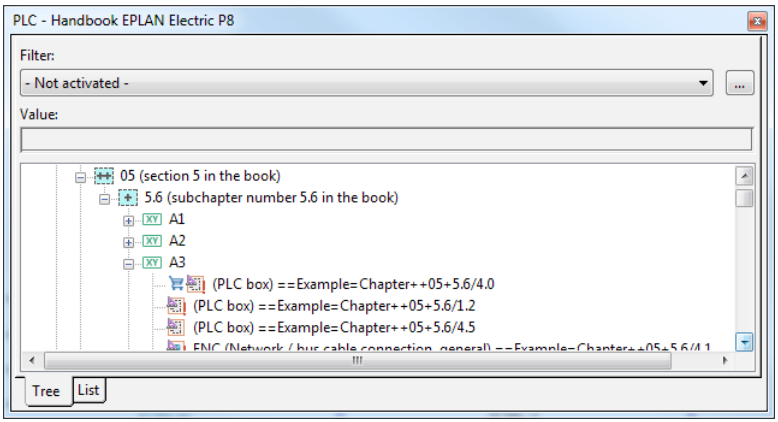


Fig. 5.161 DT-oriented view

5.6.9.2 Address-oriented

In the address-oriented view, the PLC navigator organizes the PLC data by address.

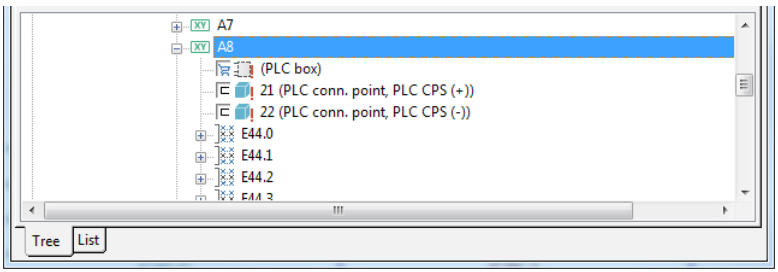


Fig. 5.162 Address-oriented view

5.6.9.3 Channel-oriented

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.

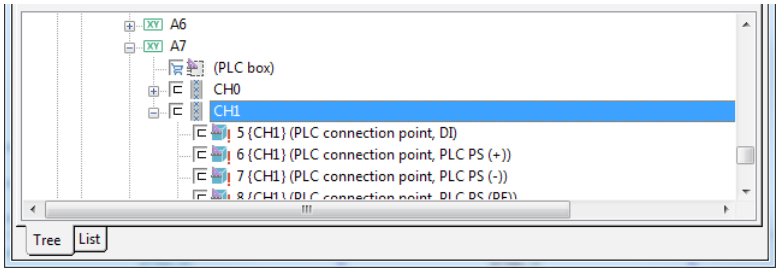


Fig. 5.163 Channel-oriented view

5.6.9.4 Rack-oriented

In the **Rack-oriented** view, the PLC data is displayed based on its assignment of devices to the racks.

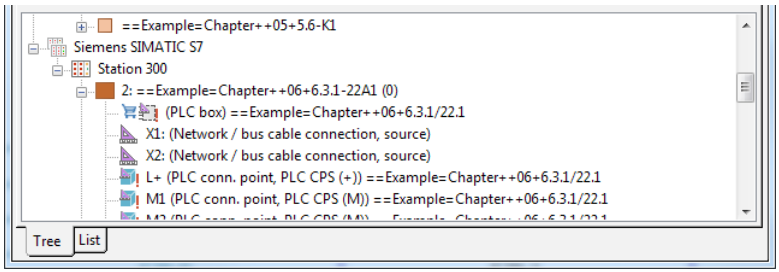


Fig. 5.164 Rack-oriented view

5.7 Part navigator (Devices/Parts)

The **bill of materials navigator** has a view of all devices with or also (configurable in OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/REPORTS/PARTS) without part numbers that exist in the project. The bill of materials navigator is started via the PROJECT DATA/PARTS/DEVICES menu.

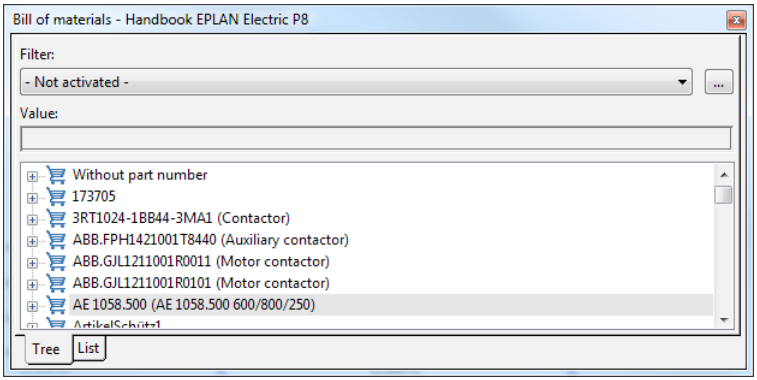


Fig. 5.165 Bill of materials navigator in the tree view

The **bill of materials navigator** offers different functions in its popup menu to make editing the project parts easier.

5.7.1 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*.

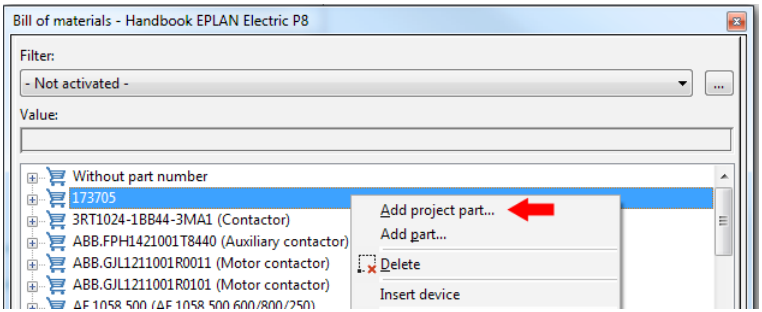


Fig. 5.166 Add project part (popup menu)

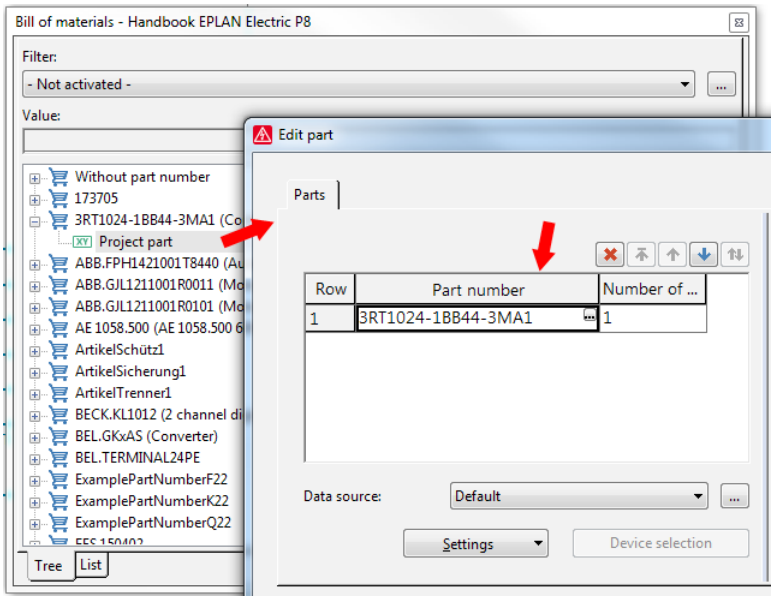


Fig. 5.167 Added project part

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.

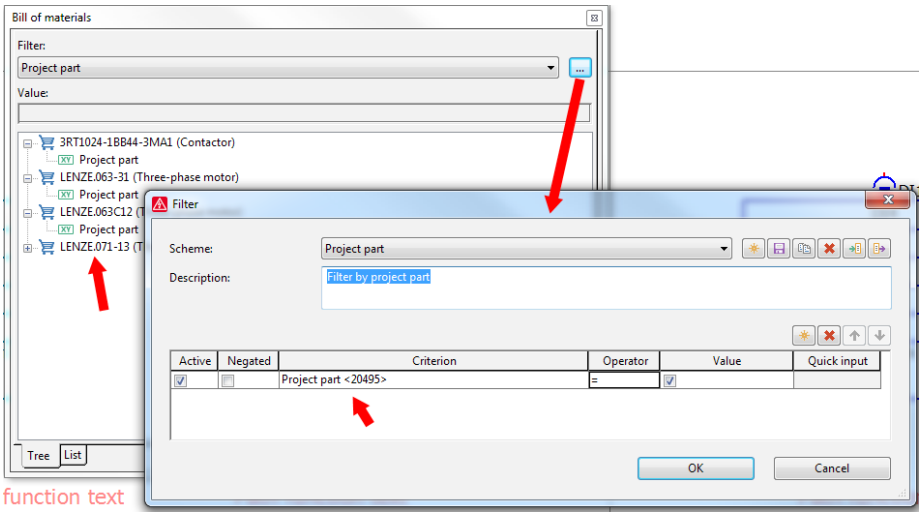


Fig. 5.168 Filter for project part

5.7.2 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.

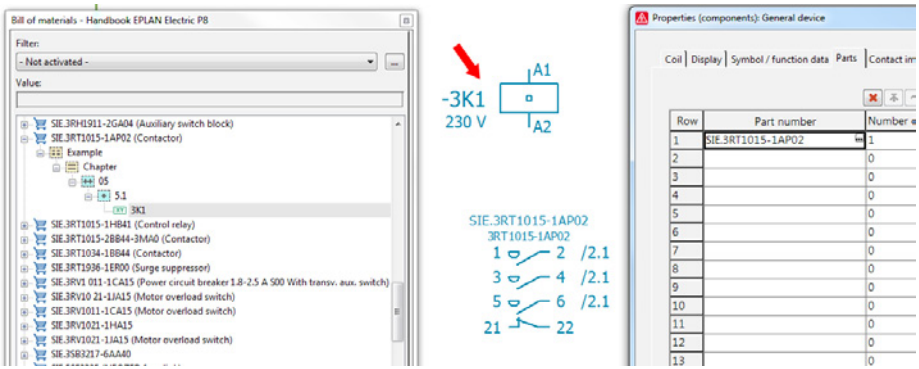


Fig. 5.169 Before running the Add part function

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**.

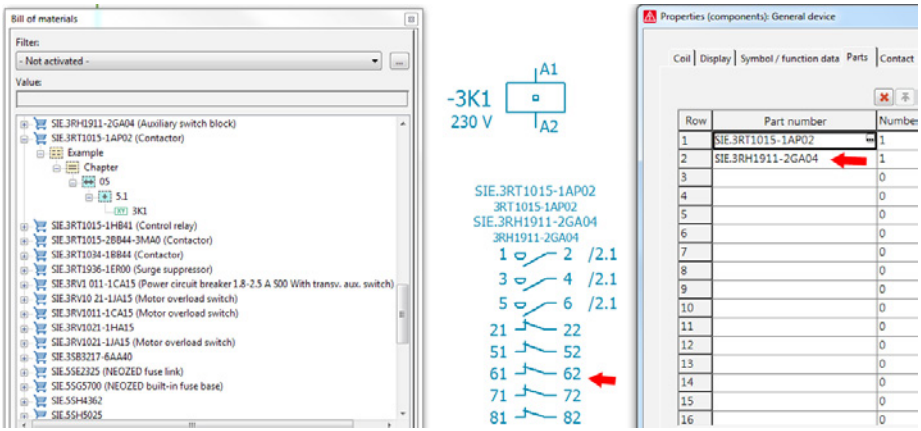


Fig. 5.170 A part was added.

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.

5.7.3 Insert device

The **INSERT DEVICE** function does just that: it adds a selected device to the graphical editor. Select the desired part, click **INSERT DEVICE** from the popup menu, and then the device hangs at the cursor with an automatic DT numbering, ready to be placed.

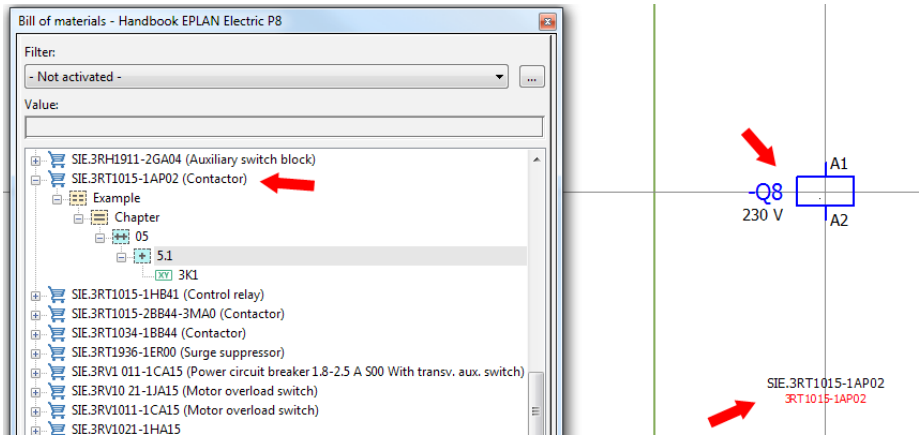


Fig. 5.171 Insert device

5.7.4 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. You may see the **CONFLICT** dialog. Here, you can specify the data to be overwritten or retained.

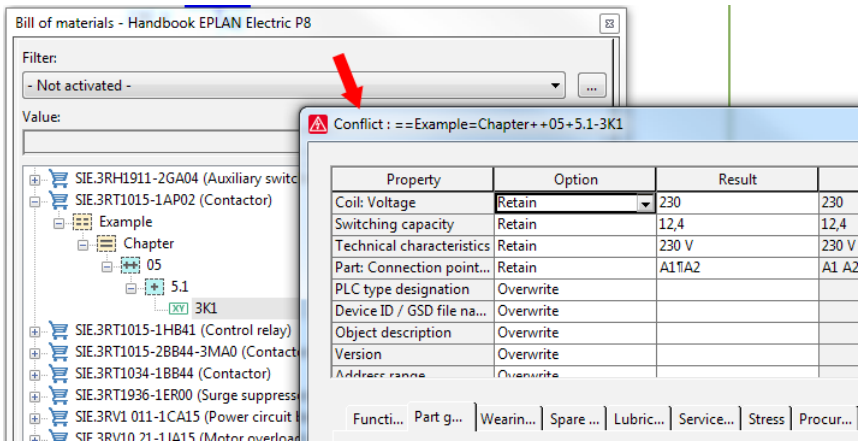


Fig. 5.172 Conflict dialog

EPLAN instantly exchanges this part with the selected one after you click the **OK** button.

5.7.5 Edit part

The **EDIT PART** function makes it possible to edit the part reference data of a part, to select a different part, or to change the quantity.

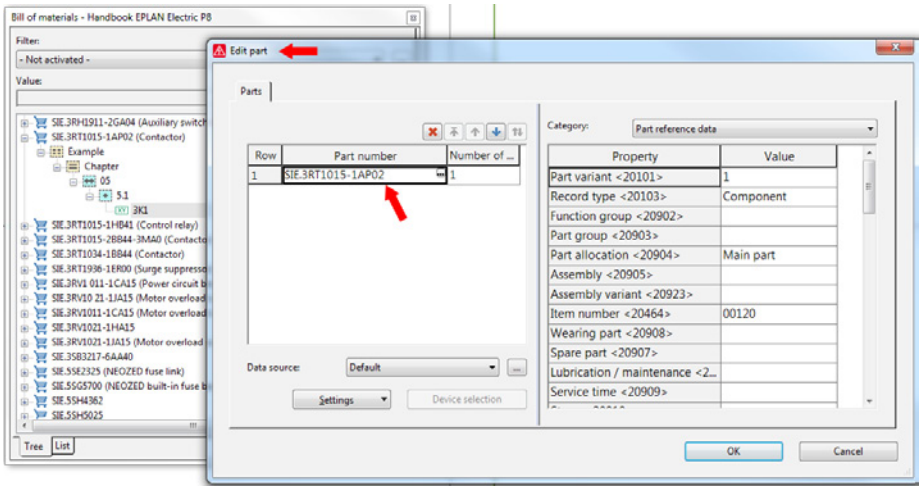


Fig. 5.173 Edit part reference data of a part

5.7.6 Device selection

This menu item belongs to the topic of bulk editing. For example, you can use this to perform a new *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.

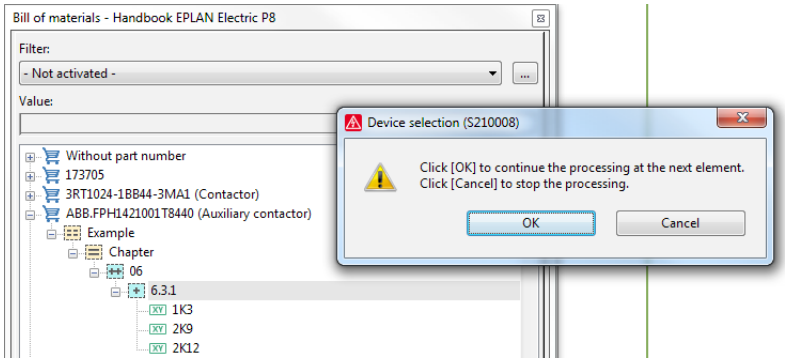


Fig. 5.174 Cancel option in device selection

5.7.7 Assign item number

This menu item opens the **NUMBER ITEMS** dialog. Here, you can implement a new item numbering of the selected item or of all project items.

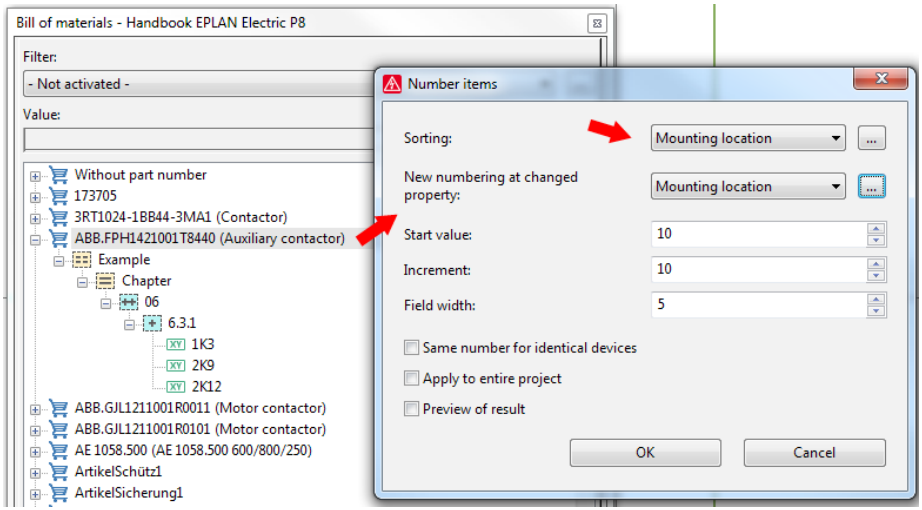


Fig. 5.175 Renumbering items

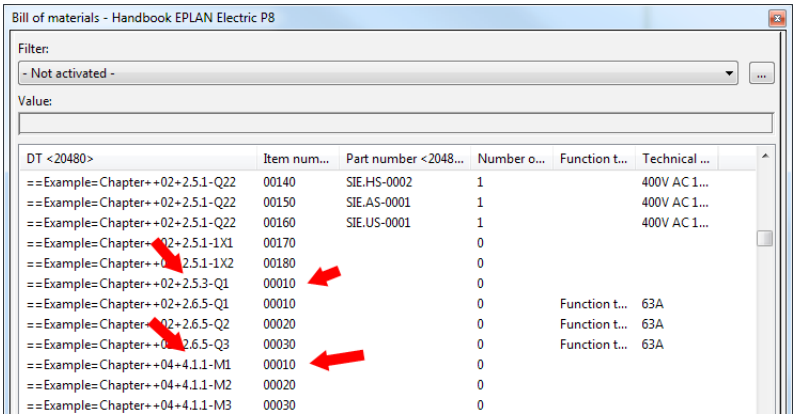


Fig. 5.176 Result of new item numbering

5.7.8 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 master data and overwrites the saved parts data in the project if necessary.

5.7.9 Export/Import

This menu item allows for bill of materials data of the entire project to be exported and/or re-imported (file format **.xml*). Both functions, however, only work within a project.

5.7.10 View

Like the other navigators, the bill of materials navigator offers fundamentally different viewpoints of the project and its data. Generally, there are four different views: *Part number*, *Part number/Structure identifier*, *Manufacturer*, and *Supplier*.

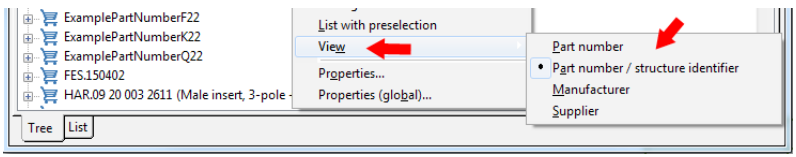


Fig. 5.177 View options

5.7.10.1 Part number

Example of **part number** view. It starts with the part number, followed by the identifier of the device, and finally the actual full DT.

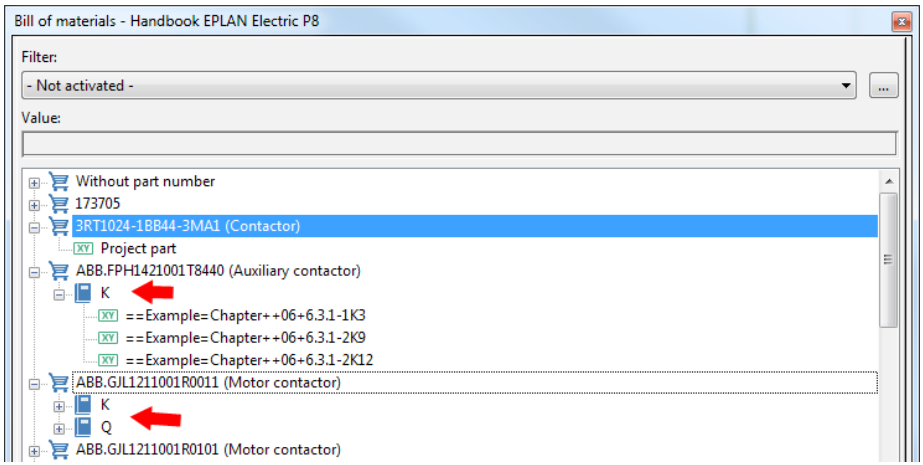


Fig. 5.178 Part number view

5.7.10.2 Part number/Structure identifier

EPLAN realizes the combined view of **part number** and **structure identifier** as follows. It starts again with the part number, followed by the structure identifiers (split by structure), and finally the DT.

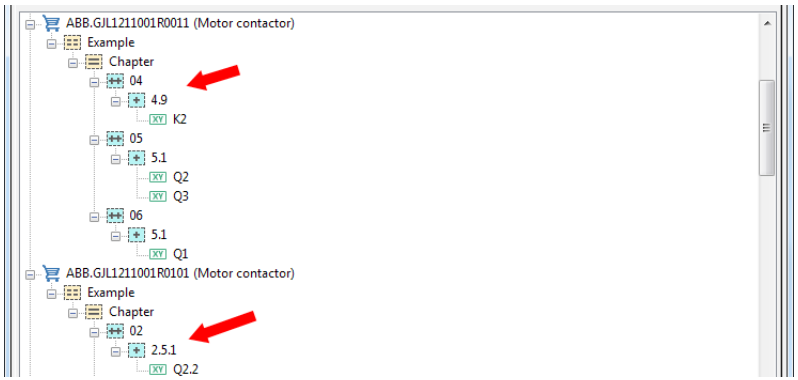


Fig. 5.179 Part number and structure identifier view

5.7.10.3 Manufacturer or Supplier

The **Manufacturer** and **Supplier** views are explained quickly. Depending on the view setting, the manufacturer or the supplier is at the top (both sets of data originate from parts management). This is followed by the part number, identifier and, finally, the full device tag (full DT).

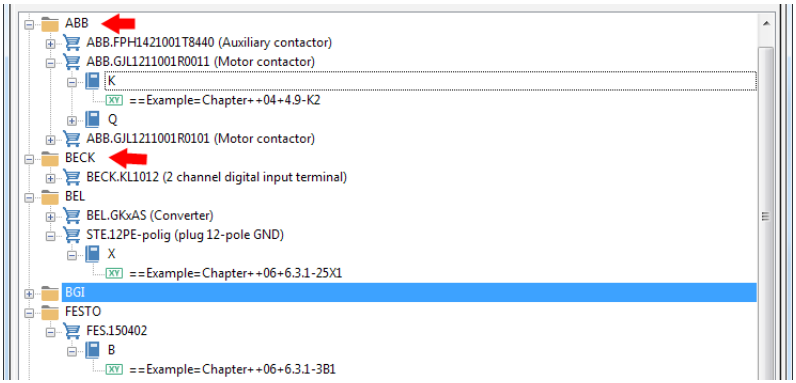


Fig. 5.180 Manufacturer/supplier view

5.8 Macro navigator

The macro navigator provides a complete overview of all macros, their variants and further information in a macro project.

NOTE: So far, the macro navigator serves no other functions in a schematic project.

5.8.1 Macros – a general description

Essentially, macros are a collection of various data that, once processed, is to render project editing more efficient and more convenient. Depending on the requirements, it is greatly advantageous to have maintained and documented macros and/or collections of macros.

Macros can also be created and used without a macro project. This option is explained further in sections 5.8.5 and 5.8.6. But having a macro project affords the user many advantages, which I will attempt to show in the following sections.

5.8.2 Macro project

A macro project is created like any regular schematic project. The differences of a macro project, when compared to a schematic project, are the following: The **Type of project**

<10902> project property is set to macro project; there are no cross-references, and no connections are generated.

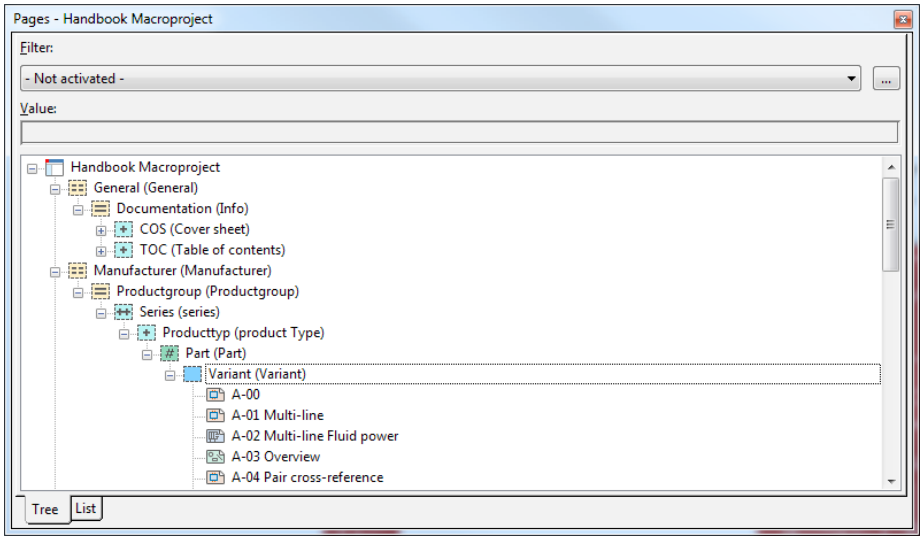


Fig. 5.181 Macro project sample structure

You should consider the structure of the macro project thoroughly before creating and generating macros and their variants. These should be a fit for the structure of the workflows – what is planned with whom and when, etc. If the macro project is built accordingly, maintenance and subsequent extensions do not pose problems, because the structure of the macro project has been well thought out.

"Wild" structures are advised against – having stored "some" macros "somehow" "somewhere" in "crazy" project structures. This causes nothing but confusion, and never produces the desired results, such as time and cost savings in project editing.

The next sections refer, always in a general manner, to the approach in a macro project, and illustrate some, but not all, options to be considered in the use of macros in connection with a macro project.

5.8.3 Macro box

The assignment and data of window and symbol macros are defined by means of a macro box. On the basis of special entries, it will be possible to generate from this the macros in desired structures at a later time. Generally, the macro box is inserted into the macro generated.

Depending on the setting, this macro box can be inserted later on in a schematic project or when using the macro, for example. This setting is found in the **OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/GRAPHICAL EDITING/GENERAL** menu.

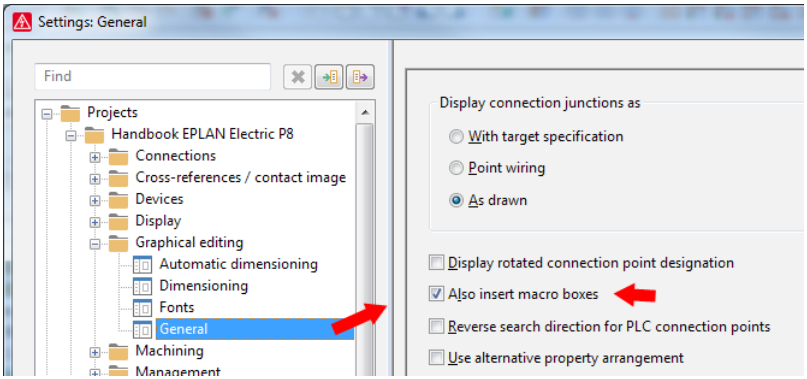


Fig. 5.182 Setting for inserting macro box

The macro box should always be inserted as well (recommended). For any subsequent editing or adjustment of data regarding the macros already used and inserted, this will be nothing but beneficial. For example, an edited macro variant can be synchronized (replaced) quite easily later on, because EPLAN remembers the macro's assignment.

NOTE: There are no macro boxes with page macros. Here, the data is set in the page properties.

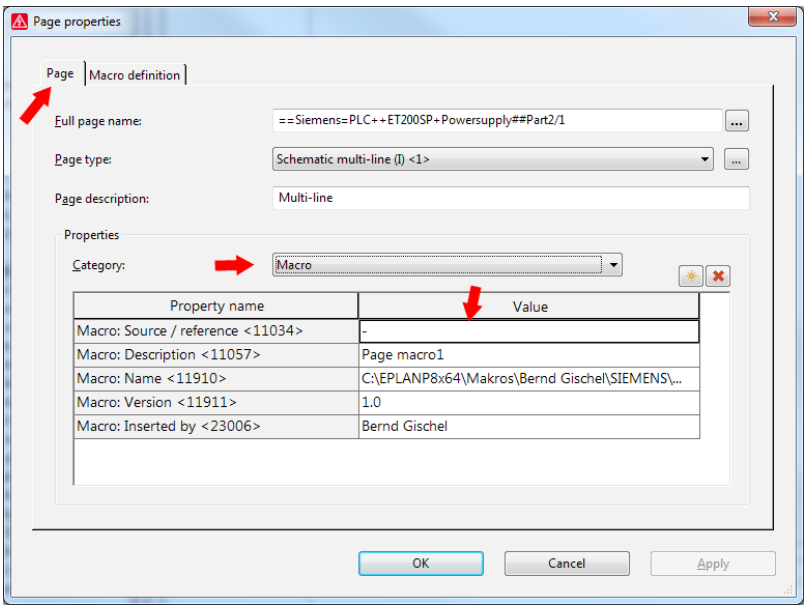


Fig. 5.183 Page properties data for page macros

5.8.3.1 General information on macro boxes

The macro box itself is located on a specific layer: EPLAN308. As the user knows, this can be hidden using the layer management tools, such as Print.

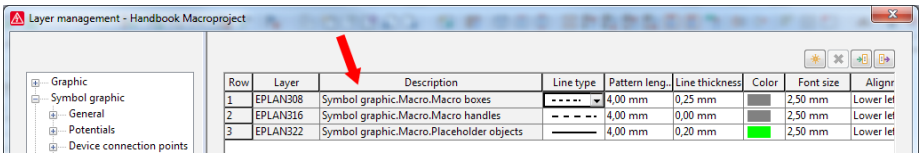


Fig. 5.184 Layers of a macro

It is also possible to let EPLAN remove from a project the inserted macro boxes in one go, using the compression function and a suitable scheme (**PROJECT/ORGANIZE/COMPRESS** menu).

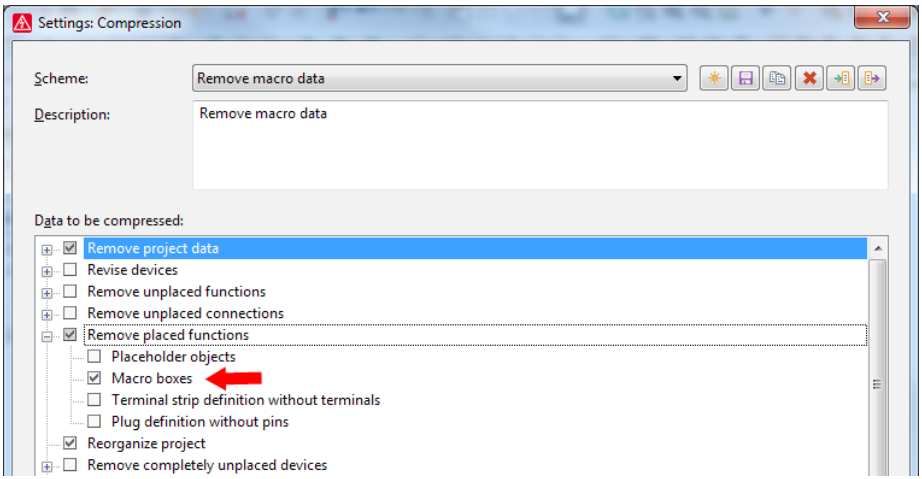


Fig. 5.185 Compression function

5.8.3.2 Macro box tab

Once the circuit, partial circuit or part (for example, an overview or panel layout representation) has been drawn, the macro box can be inserted.

The examples shown constitute the easiest representations without any further technical background. The primary purpose here is to illustrate the way in which the macro project/macro box and macro navigator are used.

Macro boxes are inserted via the **INSERT/BOX/CONNECTION POINT/MOUNTING PANEL/MACRO BOX** menu.

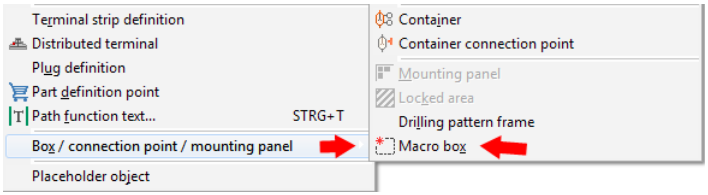


Fig. 5.186 Calling Insert macro box

The macro box is drawn across the objects like a black box.

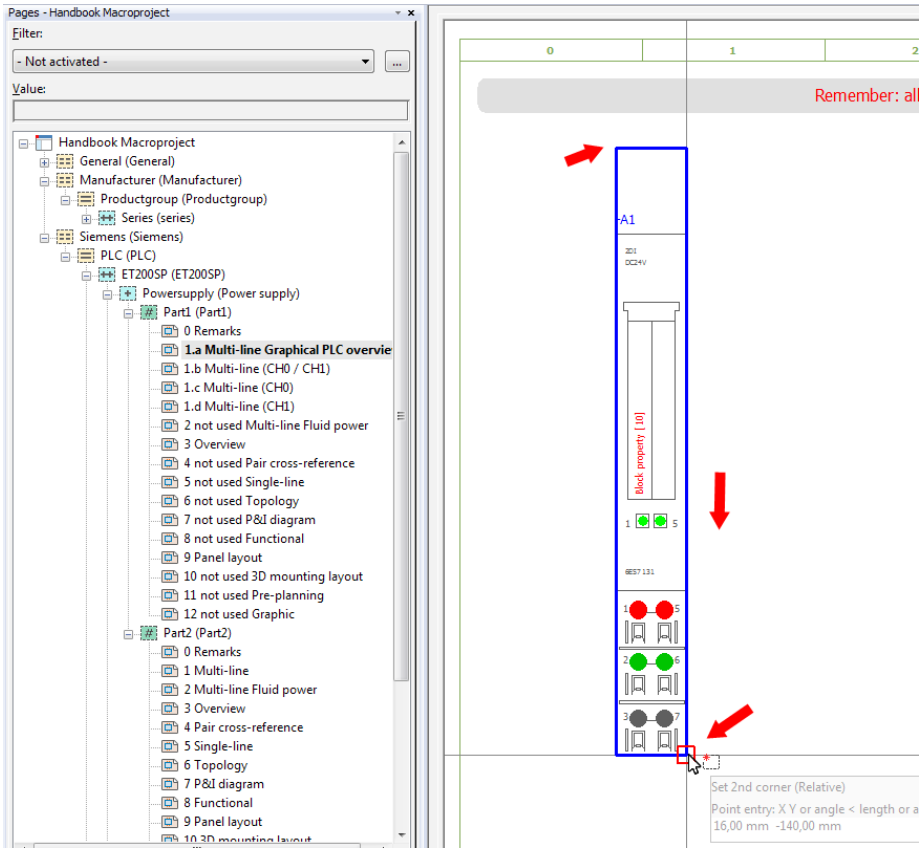


Fig. 5.187 Assigning objects to the macro box

After the assignment (bordering) of all objects, the second corner can be clicked. Now, another macro box can be inserted, or the action can be canceled.

Then, you can left-double-click the macro box. EPLAN opens the symbol properties of the macro box.

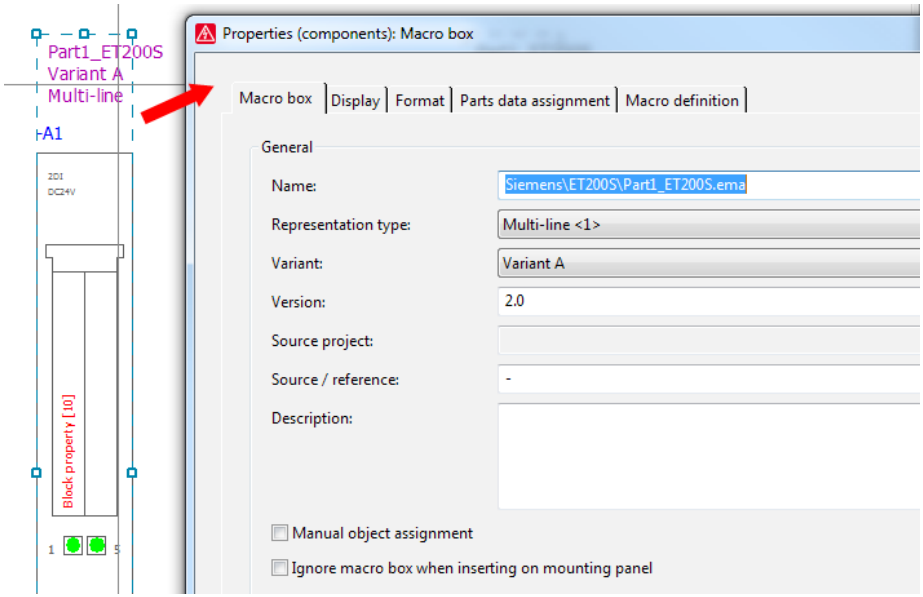


Fig. 5.188 Macro box dialog Macro box tab

5.8.3.2.1 Name, variant and more

Now, you can configure the settings for the macro on this tab. A name (macro name) is required. The other entries can, but need not, be made.

All fields should be filled as carefully as possible (recommended). This way, it will be easier to trace changes later on.

Path information can be included in the name (macro name). By default, EPLAN uses the default macro system directory. A specific structure (directory structure) can be entered here as well. This does not have to be an existing structure. EPLAN will create it automatically when generating the macro.

If no file extension is defined, EPLAN will use the file extension *.ema (window macro) automatically. If you specifically want a symbol macro output, then you must use the file extension *.ems with the macro name.

In the case of page macros, information, such as the macro name or any deviating directory for saving the page macro, is taken from the page properties, because there are no macro boxes here (i. e. page macros).

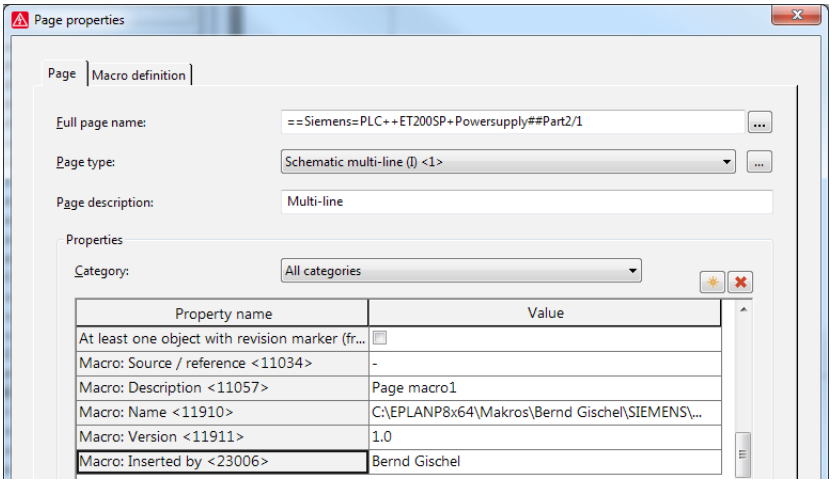


Fig. 5.189 Page properties dialog

5.8.3.2.2 Handle active

The handle is usually in the upper left. Using this option, the handle can be modified in any manner whatsoever. For this purpose, the option is enabled, which means you can make manual changes to the X and Y positions. EPLAN, then, indicates this “move” by means of a special symbol.

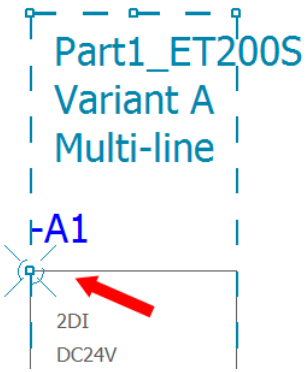


Fig. 5.190
Handle moved symbol

5.8.3.2.3 Manual object assignment

The MANUAL OBJECT ASSIGNMENT option allows for objects outside the actual macro box to be incorporated into the extent of the macro box.

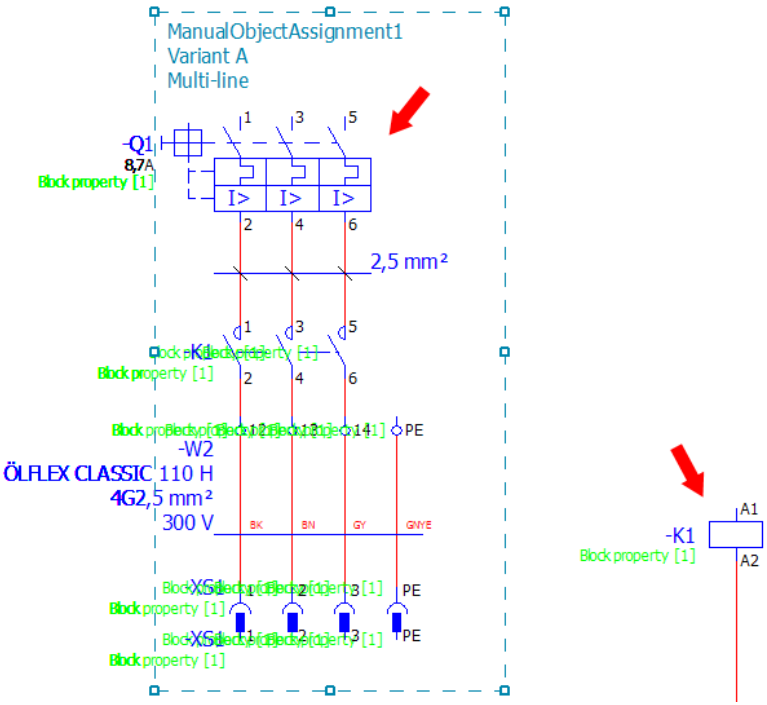


Fig. 5.191
Macro box without manual object assignment

To integrate the exterior objects with the macro box, you must click the macro box, then right-click to open the popup menu of the macro box, followed by the **ASSIGN OBJECTS TO MACRO BOX** menu item.

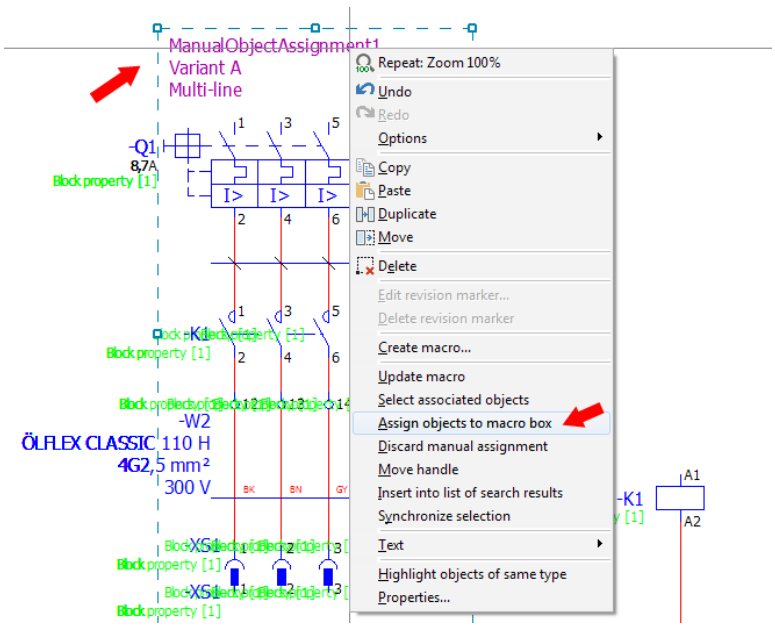


Fig. 5.192
Macro box popup menu

Now, select all elements using the mouse, then close this action with the second clicked point. All objects are assigned to this macro box now.

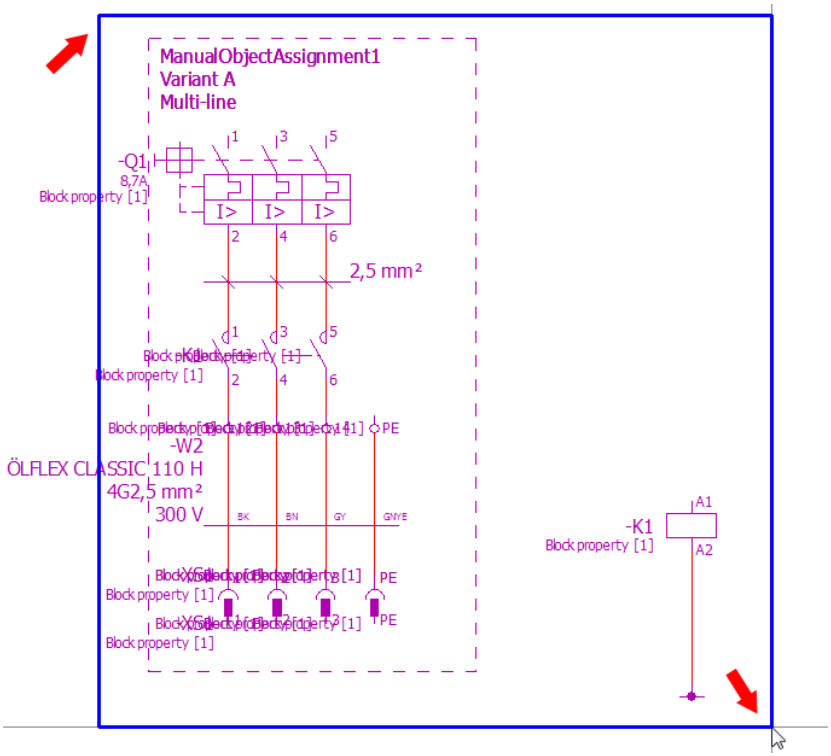


Fig. 5.193 Assigning exterior objects to the macro box

To verify, you can call the popup menu of the macro box and click the **SELECT ASSOCIATED OBJECTS** menu item. EPLAN selects the corresponding objects.

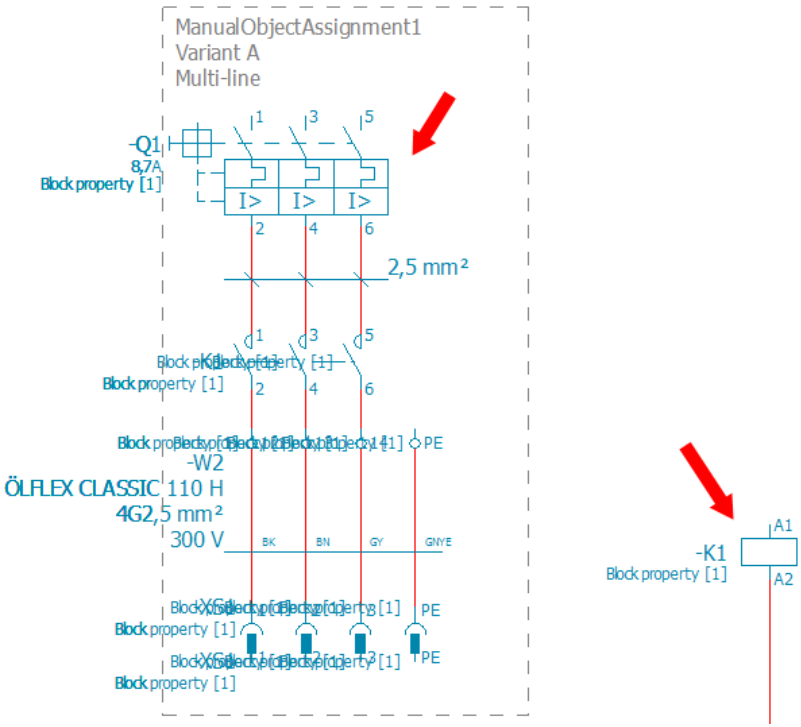


Fig. 5.194 All objects associated with the macro box

This assignment can be broken up again by calling the popup menu of the macro box and selecting the **DISCARD MANUAL ASSIGNMENT** menu item.

5.8.3.2.4 Ignore macro box when inserting on mounting panel

The **IGNORE MACRO BOX WHEN INSERTING ON MOUNTING PANEL** option ensures that the macro box will be ignored when placing such a macro (Panel layout representation type).

5.8.4 Macro navigator/Generate automatically

Once the macros and corresponding data, such as macro name, have been assigned and/or entered, go to the **PROJECT DATA/MACROS** menu and click **GENERATE AUTOMATICALLY**.

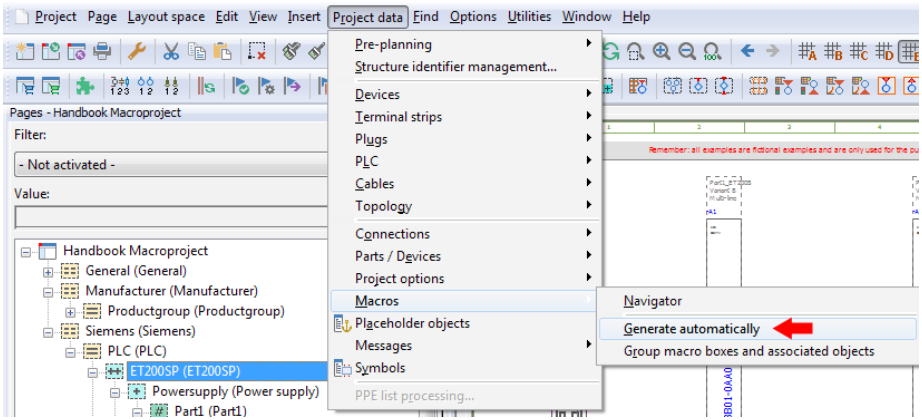


Fig. 5.195 Generating macros automatically

Before the actual generation, EPLAN displays a prompt as to whether only the selected macros or all macros in the macro project are to be generated.

When you click the **YES** button, EPLAN generates the macros automatically in the directories specified. This concludes the process of creating and generating macros. Now, these macros are ready for use.

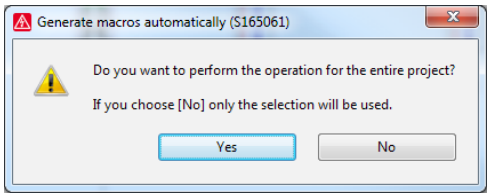


Fig. 5.196 Prompt

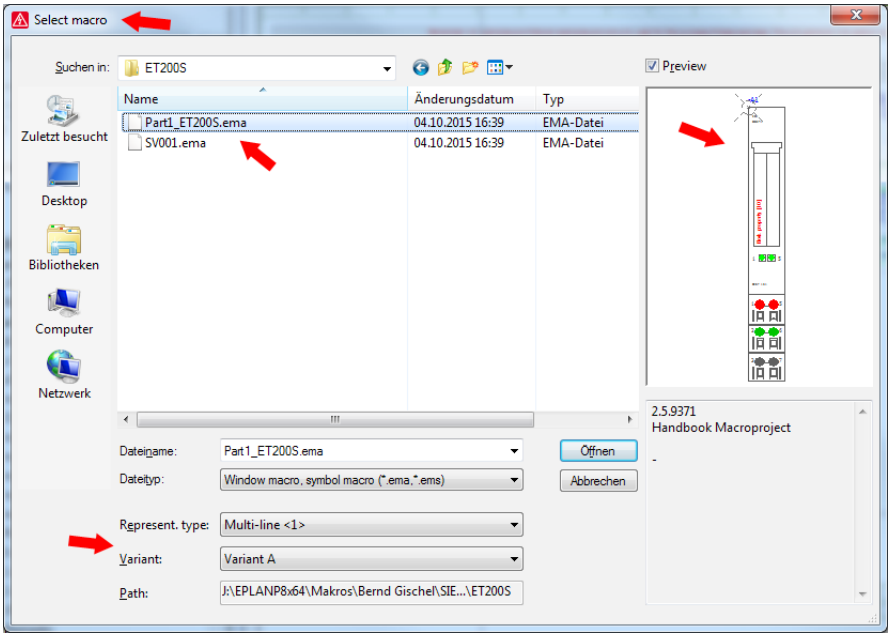


Fig. 5.197 Using a macro

If the macro is changed (in the macro project, of course!), you can have the macros re-generated and, subsequently, updated easily in the schematic project used. In other words, there is no need to delete anything, re-insert the macro or re-number the DT, etc.

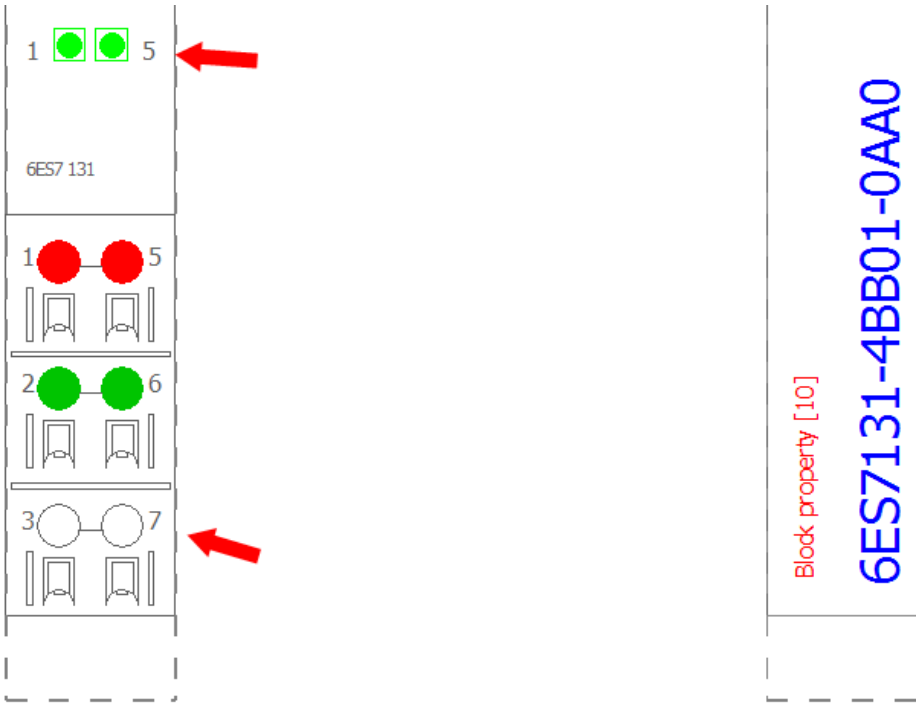


Fig. 5.198 Modified macro in the macro project

To update the macro, click the macro box and call **UPDATE MACRO**. EPLAN instantly changes the macro, showing it in the new version. This requires, of course, that the macro box has always been inserted as well (this option can be set in the **OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/GRAPHICAL EDITING/GENERAL** menu).

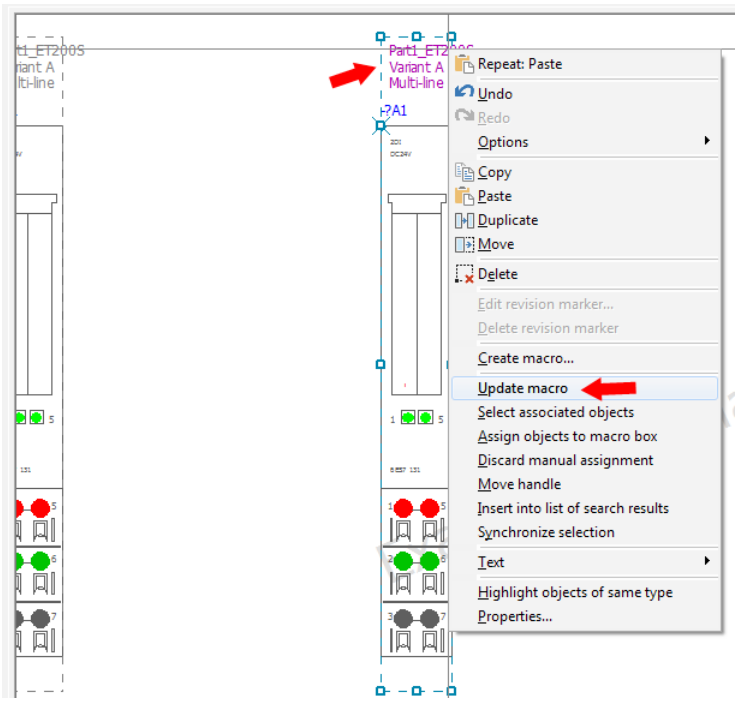


Fig. 5.199 Macro box popup menu – Update macro

Fig. 5.200 Successfully updated macro

5.8.5 Macros in general (without macro project)

Of course, apart from the macro project and/or generating the macro automatically, you can also create and save macros manually in a purely conventional manner. The following sections deal with this method, but it needs to be said that it is better to create, maintain and generate macros in a macro project, and to use only these macros later on.

Furthermore, there is always the option of generating macros from schematic projects, transferring them to a macro project, and then generating and maintaining them in the manner described.

5.8.5.1 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.

5.8.5.2 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 the shortcut key **CTRL + F5** or use the **EDIT/CREATE WINDOW MACRO/SYMBOL MACRO** menu to start the function. EPLAN opens the **SAVE AS** dialog. First, you must define a meaningful file name for the macro. The macro directory is usually the company-specific macro directory, but this can be changed if desired.

Window macros are inserted into a page using the **M** key or the **INSERT/WINDOW MACRO/SYMBOL MACRO** menu. 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/symbol 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/symbol macro can also be the contents of the entire page (all objects on the page). Window macros have the file extension **.ema*.

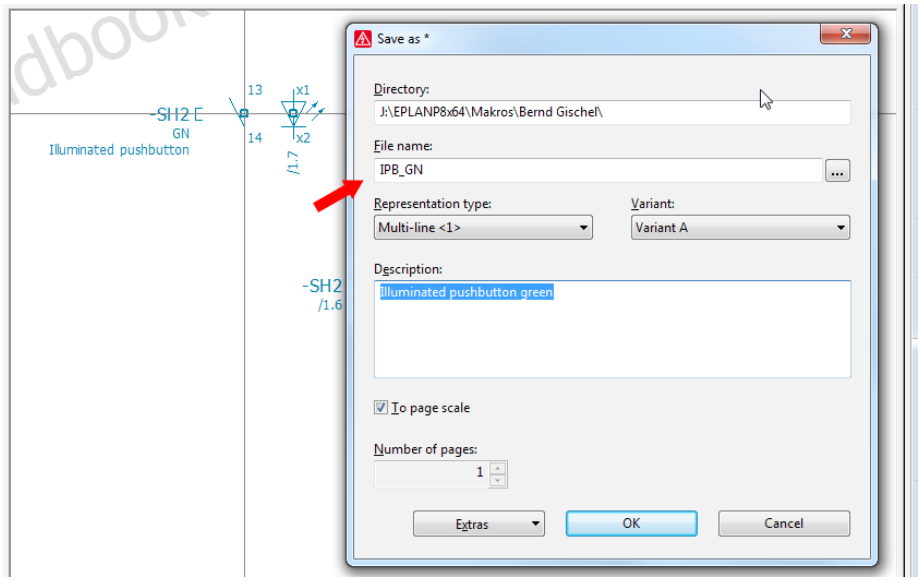
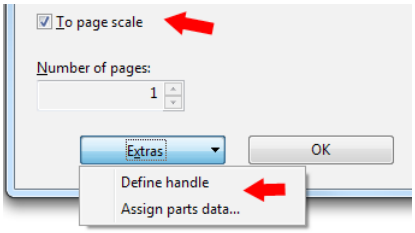


Fig. 5.201 Saving a macro

5.8.5.2.1 Define handle setting

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.



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

Fig. 5.202
Extras settings

The cursor changes when doing this, as can be seen in Fig. 5.203. 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.

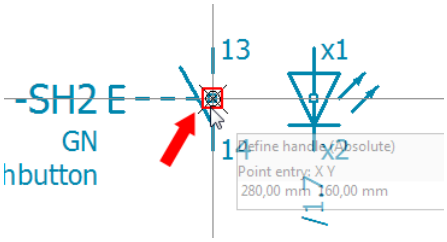


Fig. 5.203
Defining new handle

5.8.5.2.2 Define representation type setting

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

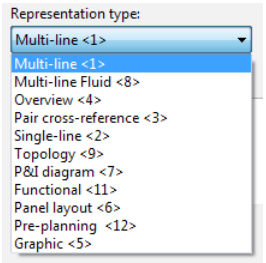


Fig. 5.204
Representation types

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.

5.8.5.2.3 Define variant setting

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

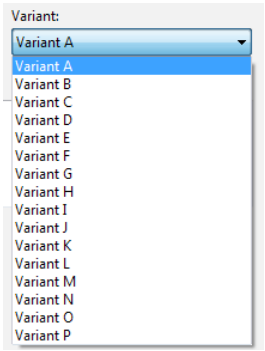


Fig. 5.205
Variants

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 176 possible variants (macro variants) within one macro. For example, you could save different macro variants of a PLC card in one macro.

5.8.5.2.4 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.

5.8.5.3 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 also inserted via the **INSERT/WINDOW MACRO/SYMBOL MACRO** menu.

5.8.5.4 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 5.8.5.2 “Window macros” 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.

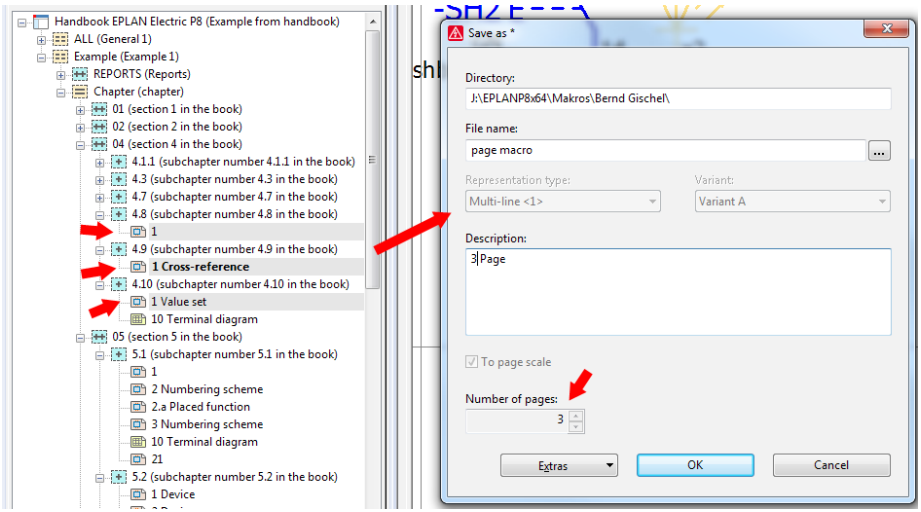


Fig. 5.206
Generating page macro

In addition to the individual page macro (directly on the page), in the **page navigator** it is also 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.

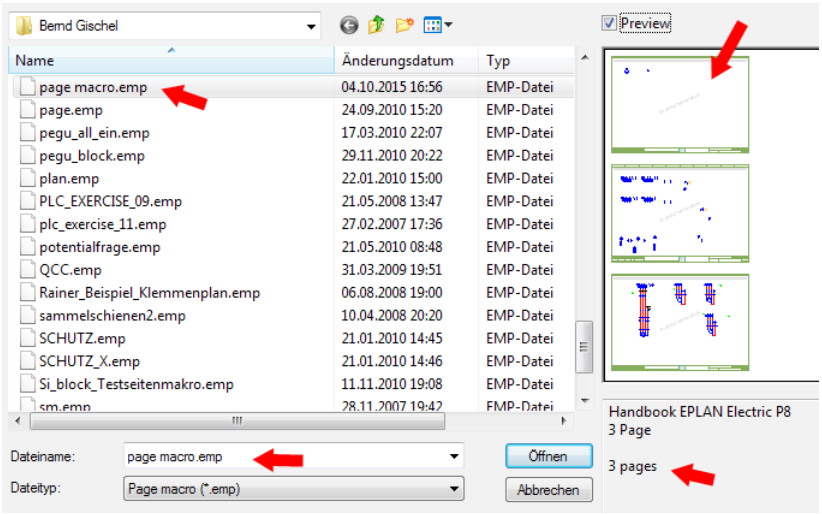


Fig. 5.207
Inserting page macro

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.

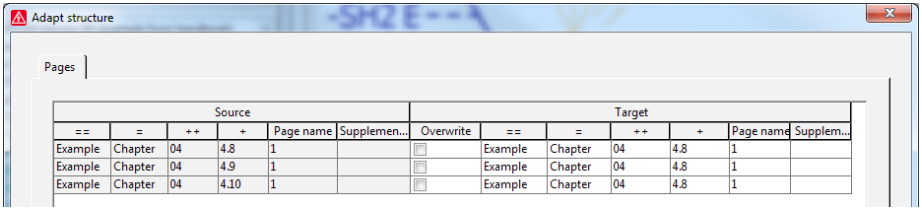


Fig. 5.208 Adapting structure

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

5.8.6 Macros with value sets (without macro project)

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**. These value sets are activated via a specific symbol – the placeholder object. This symbol is used to switch between value sets.

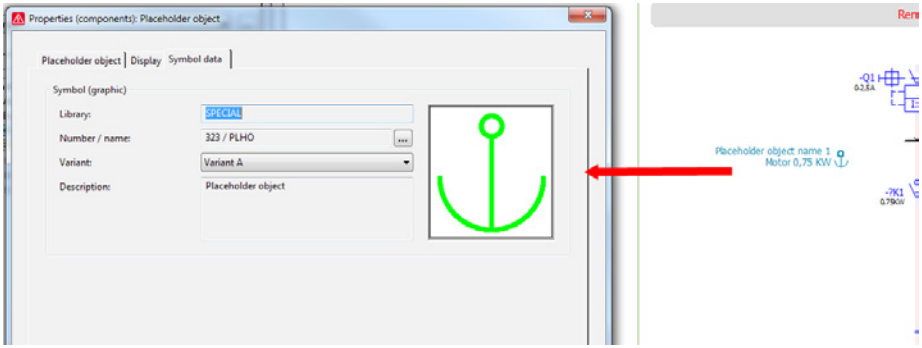


Fig. 5.209 Placeholder object symbol

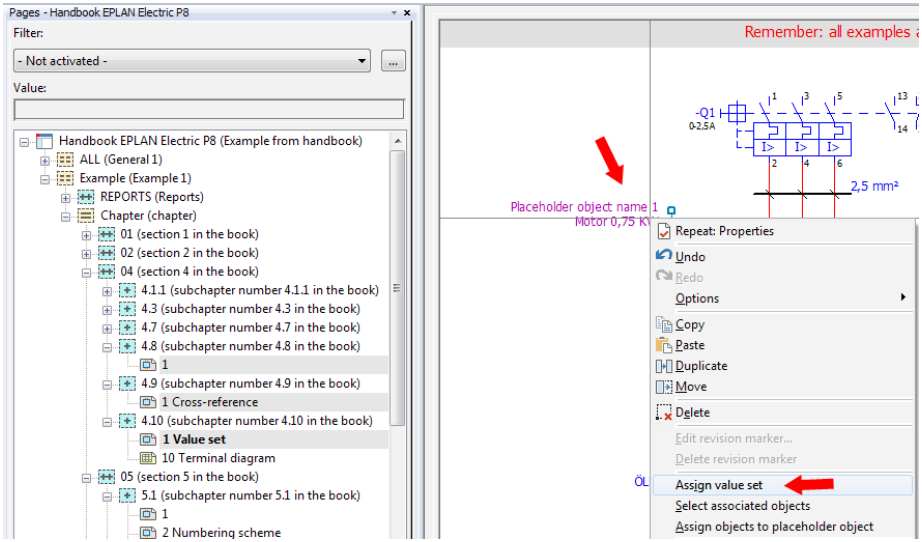


Fig. 5.210
Assigning value set

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.

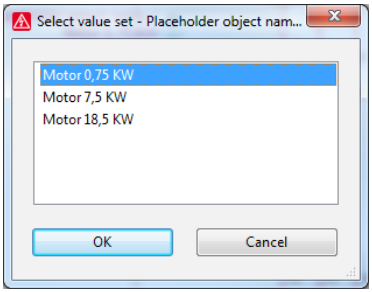


Fig. 5.211
Selecting value set

5.8.6.1 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.

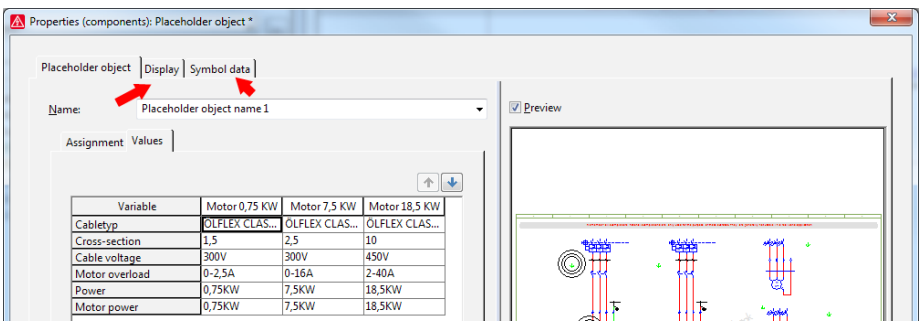


Fig. 5.212
Placeholder object symbol properties

5.8.6.2 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.

5.8.6.3 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>**.

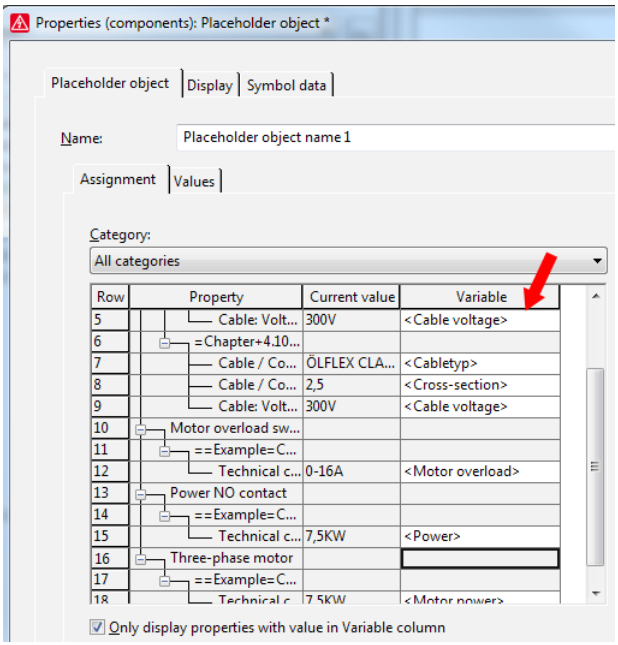


Fig. 5.213 Example of variables

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.

5.8.6.4 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.

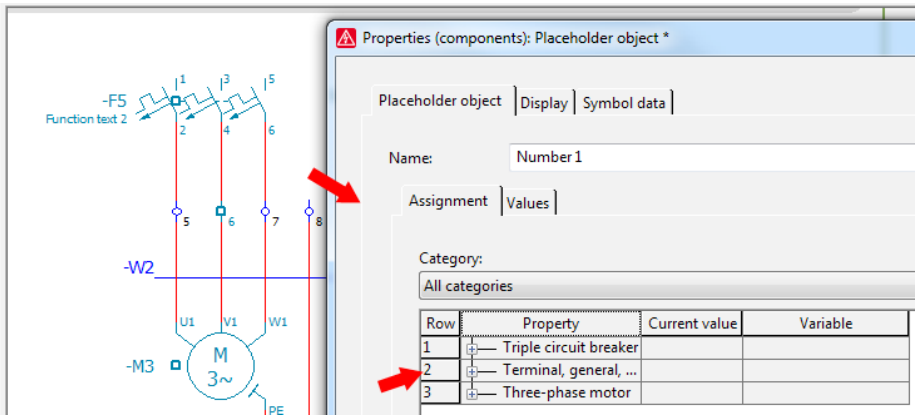


Fig. 5.214 Assigning individual objects to the placeholder object (option 1)

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).

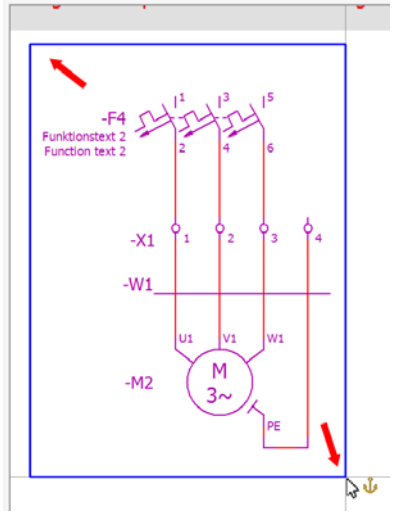


Fig. 5.215 Selecting objects with a window (option 2)

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.

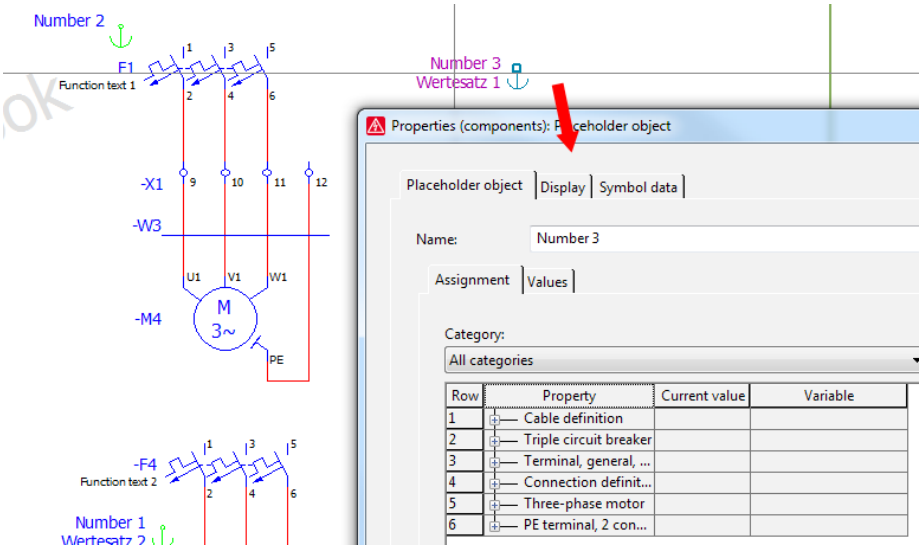


Fig. 5.216 Editing placeholder object

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.

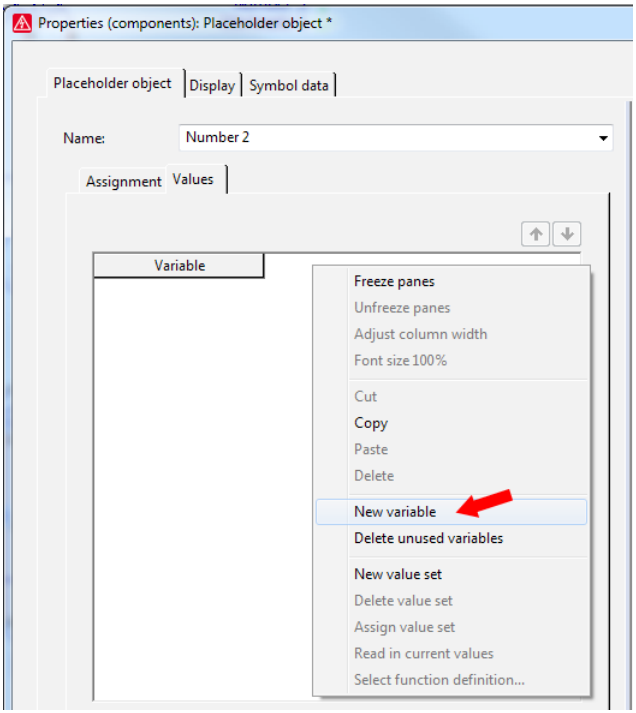


Fig. 5.217 Creating new variable

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.

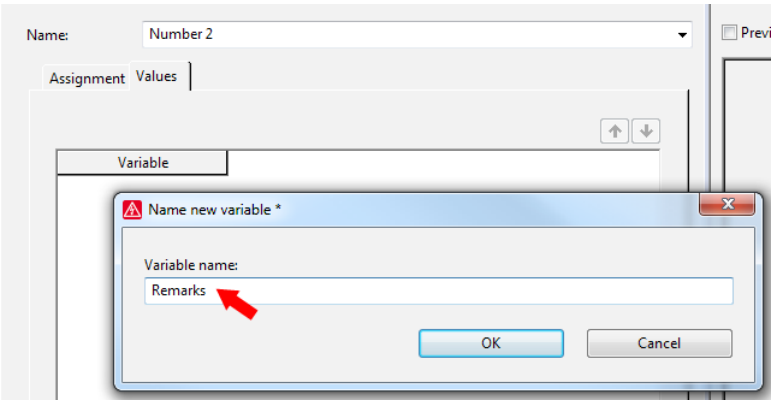


Fig. 5.218 Naming new variable

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

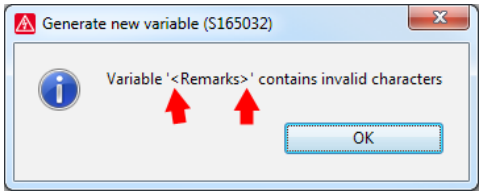


Fig. 5.219
Messages about characters not permitted (»<« or »>«)

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.

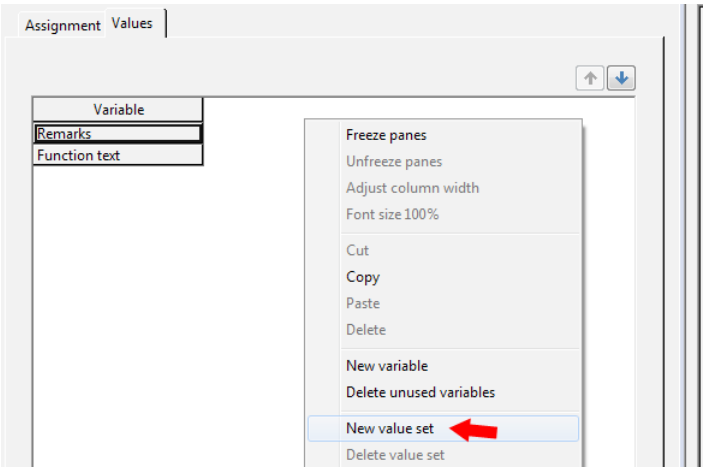


Fig. 5.220 Creating new value set

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.

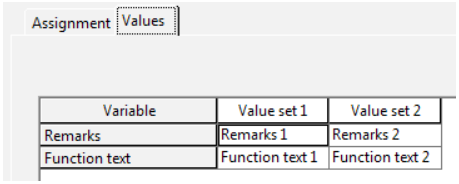


Fig. 5.221
Filling value sets with data

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.

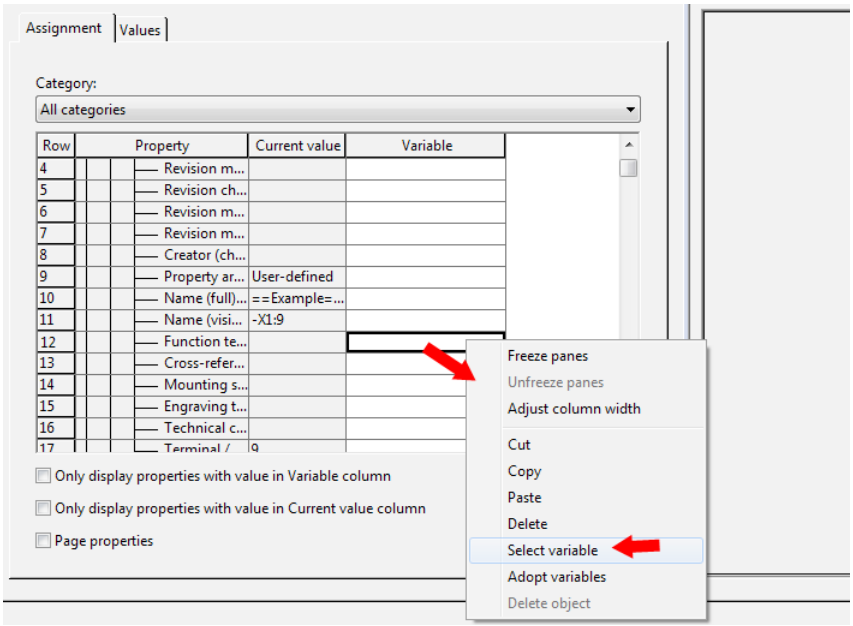


Fig. 5.222 Selecting variable

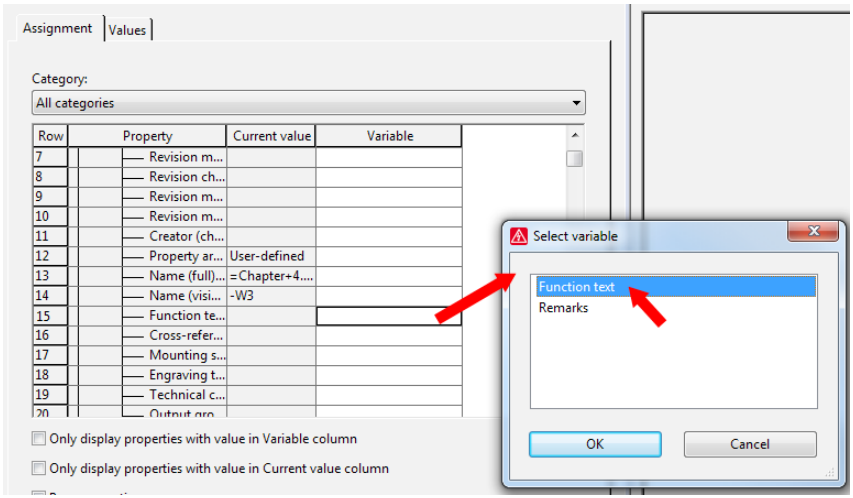


Fig. 5.223 Applying variable

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.

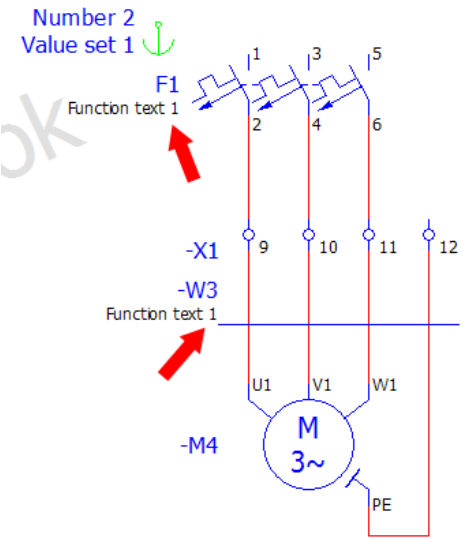


Fig. 5.224 Example value set 1

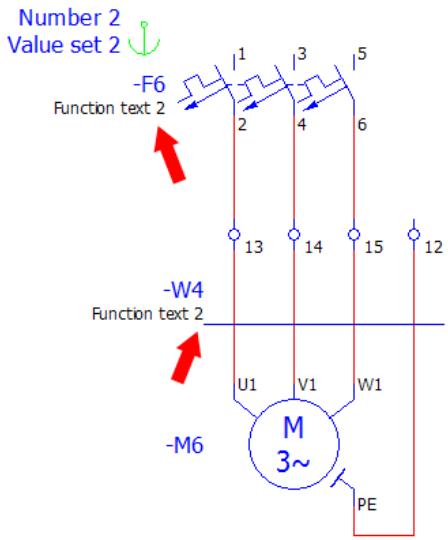


Fig. 5.225 Example value set 2

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.

5.9 Navigators – general functions

Most navigators and their popup menus contain functions of the same functional extent. Some of these functions are described one time below. Depending on the type of navigator, the view of certain devices may change (for example, the terminal strip navigator views terminals, while the cable navigator looks at the cable devices), but the functions of *Place* and *Assign* do the same things in all of these navigators. This is why these explanations have been limited, for the most part, to the device navigator.

5.9.1 Place from navigators

In the example in section 5.2.9, 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 in the popup menu of the **device navigator**.

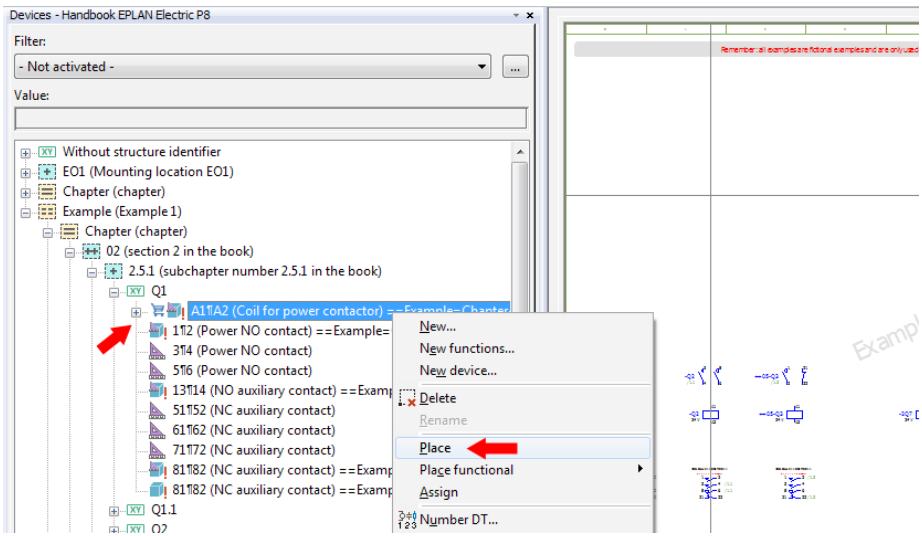


Fig. 5.226 Placing device

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.

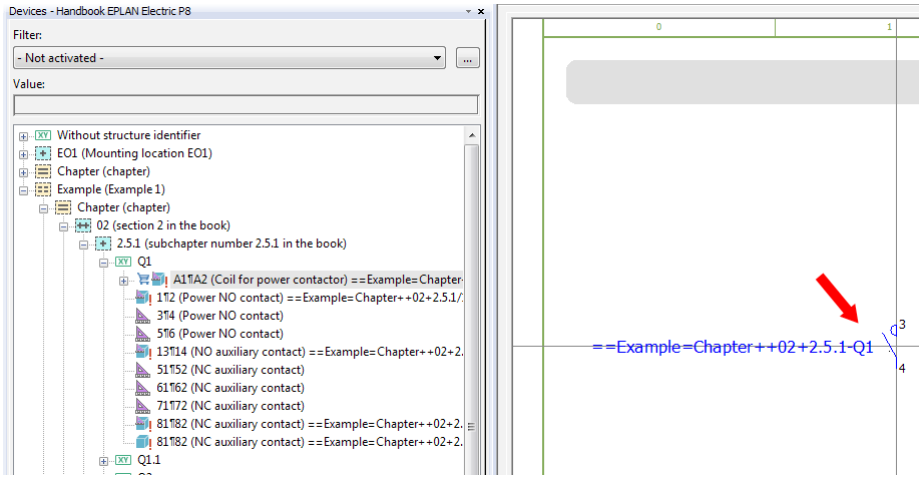


Fig. 5.227 Placed device

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.

EPLAN always places the device in accordance with the page type. For example, if the page is a *Schematic multi-line* page type, the *Schematic multi-line* representation type will be placed. If you do not want that, you must press the backspace key before placing the symbol on the page. EPLAN then opens the **PLACE DEVICE** dialog.

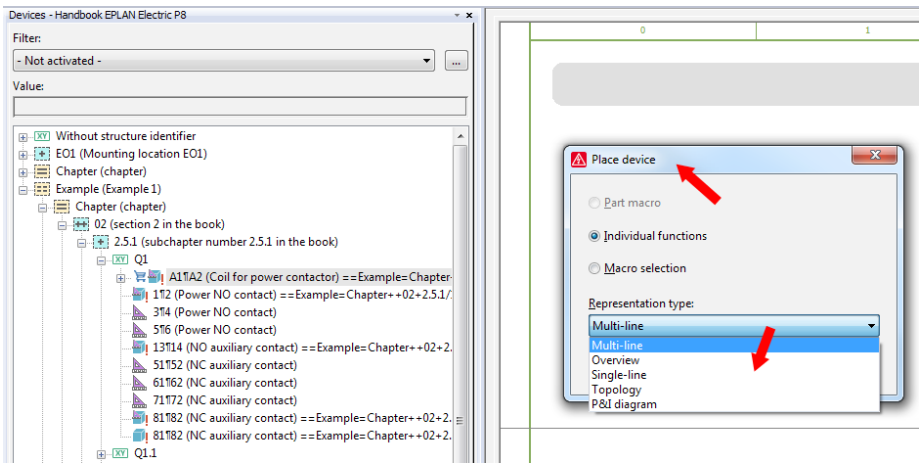


Fig. 5.228 Place device dialog

There are several options in this dialog:

Part macro: If the device has a part macro, you can select it here.

Individual functions: This option allows for all functions to be placed individually. Here, before placement (the **PLACE DEVICE** dialog is closed), use the **N** key to browse the functions. Thus, in the example, this is the circuit breaker, followed by the auxiliary NO contact, and finally the auxiliary NC contact.

Macro selection: Use this option to select a completely macro to be placed.

If only one of the options is selected and confirmed via **OK**, it is still possible to change the symbol before the actual placement. Again, the backspace key needs to be pressed. EPLAN then opens the **SYMBOL SELECTION** dialog.

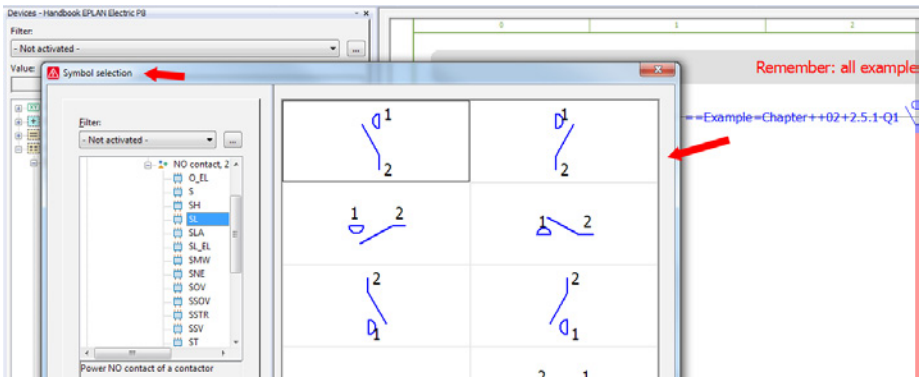


Fig. 5.229 Changing symbol before placement

If a symbol has been selected, the device can be placed fully, that is, all functions individually, for example. It is also possible here to rotate the symbol before placement by using the **TAB** key, that is, by changing the variant.

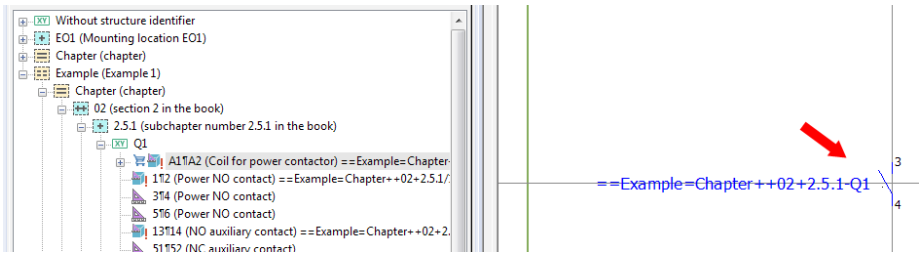


Fig. 5.230 Selecting variant before placement

5.9.2 Assign from navigators

A number of navigators have the **ASSIGN** function in their popup menu.

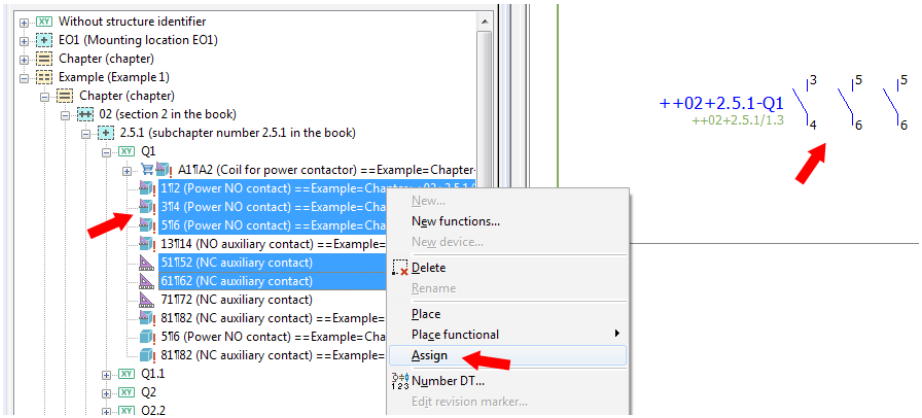


Fig. 5.231 The Assign menu item in the device navigator

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).

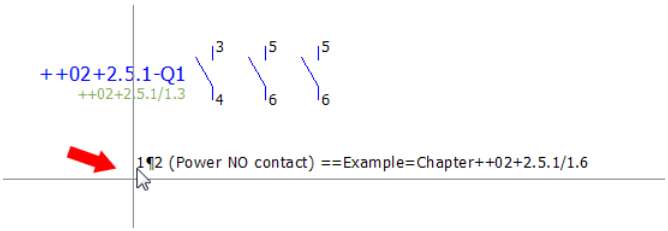


Fig. 5.232 The assigned properties "hang" on the cursor.

You then click the individual power contacts to be "equipped" in the schematic one after another, using the left mouse button.

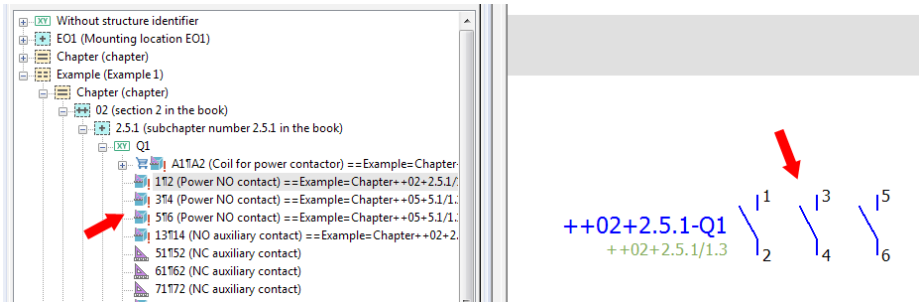


Fig. 5.233 Assigned device tags

EPLAN transfers one DT after another to the power contacts without requiring this to be manually added at each power contact. This is a very useful function that saves time and ensures the error-free transfer of information. It is also available in other navigators (in addition to the device navigator).

5.9.3 Filters

Filters are useful in displaying specific data in the navigators. There are really only two types of filters – filters without quick entry and those with quick entry.

5.9.3.1 Default filter

A default filter consists of one or several criteria that must be “fixed” in the filter.

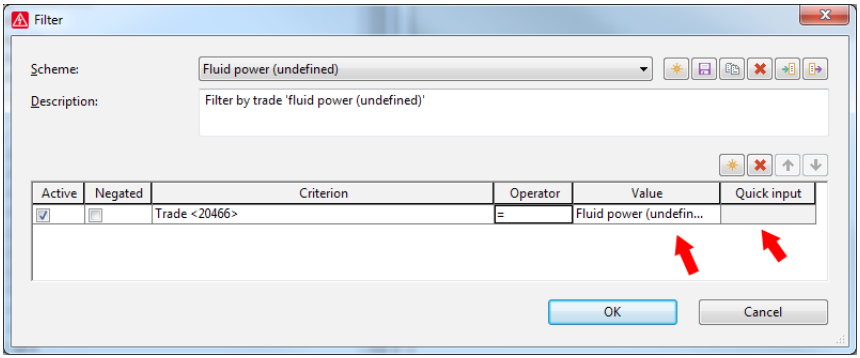


Fig. 5.234 Default filter, without quick entry

5.9.3.2 Quick entry filter

To simplify work, most navigators can also filter by a scheme according to different values. In other words, quick entry. This way, it is not necessary to create a filter for each case. Simply create a general filter with the option of activating quick entry. The actual filtering is then done via the selection field directly in the navigator.

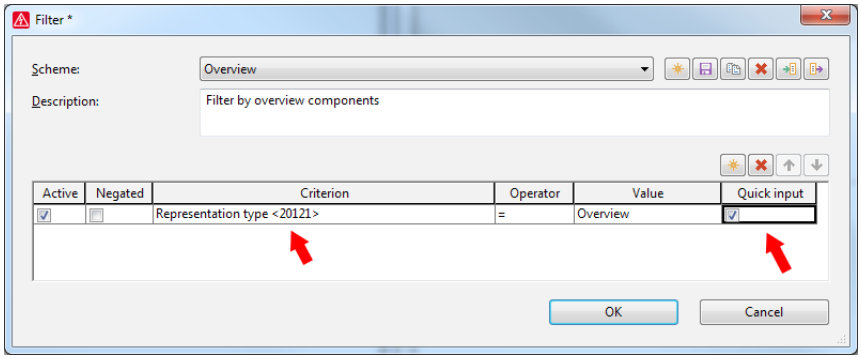


Fig. 5.235
Filter with quick entry option

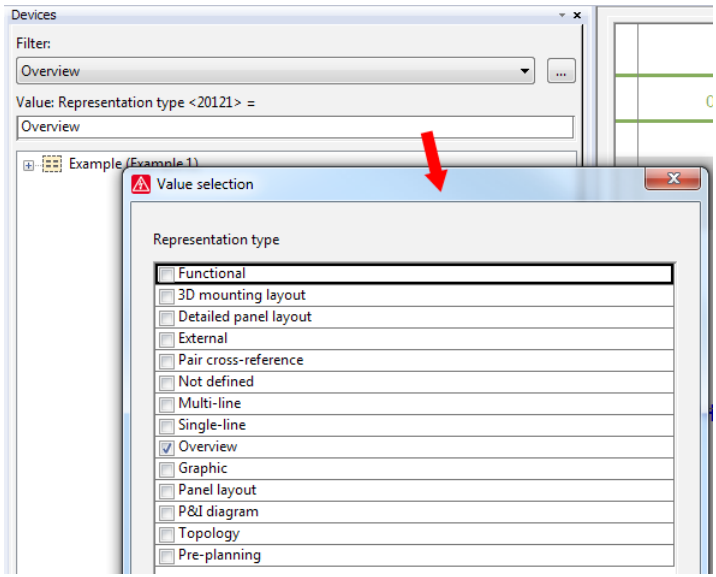


Fig. 5.236
Filter with quick entry

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, 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 resulted 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.

NOTE: Not all criteria are suitable for the quick entry option when filtering.

5.10 Correction functions

EPLAN offers a number of different correction functions. In versions prior to EPLAN P8 V2.5, the correction functions were located in the menus of the respective navigators.

Now, these correction functions can be found in one place, in the **PROJECT/ORGANIZE/CORRECT** menu. In this menu, you can save and execute in your own filter schemes possible corrections for the areas *Terminals*, *Plugs*, *Cables*, and *Connections*.

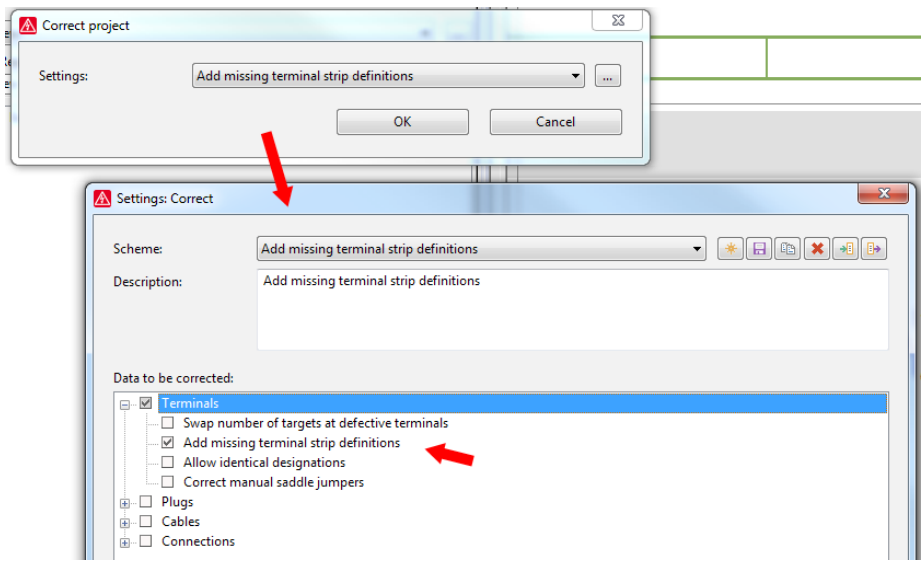


Fig. 5.237 Example of scheme for correcting terminals

6 Reports

This chapter deals with reports: Which ones are necessary in regular cases, 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 from using reports? Some of the possibilities offered by external editing and labeling are also touched upon using simple, short examples.

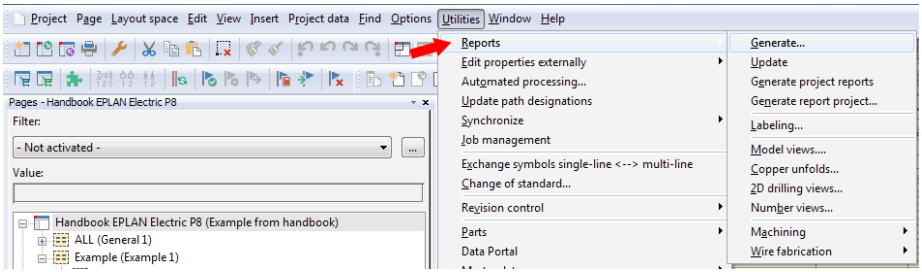


Fig. 6.1 Reports menu

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 (reports = 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.

The **UTILITIES/EDIT PROPERTIES EXTERNALLY** menu item contains the functions 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.

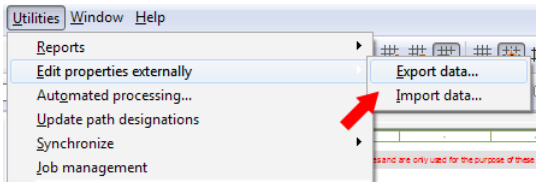


Fig. 6.2 Edit properties externally menu

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* (embedded report), 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, which itself evaluates only specific data. 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 currently 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.

- **F01** Parts list
- **F03** Device tag list
- **F13** Terminal diagram
- **F22** 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.

TIP: Some of the following reports are special reports for further add-ons (sold separately), such as Pre-planning or the Topology add-on (FieldSys). I list them here for the sake of completeness.

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 Design
Mini contactor	MOE.010042	1	==Example=Chap
Auxiliary contactor	SIE.3RH1122-1BB40	1	==Example=Chap
Auxiliary contactor	ABB.FPH1421001T8440	0	==Example=Chap

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 numbe
MOE				
	010042	2	Mini contactor	DILER-22-G
SIEMEN				
	3RH1122-1BB40	0	Auxiliary contactor	3RH1122-1BB
ABB				
	FPH1421001T8440	2	Auxiliary contactor	FPH1421001T

ple from handbo

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, such as part number, placement of main and auxiliary functions, technical part data, graphical preview of the component (symbol) used, and much more.

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 XB5 AV61	Indicator light Complete device, round, indicator light	/3.0	
-3K1 SIE.3RT1015-1BB41 3RT1015-1BB41	Contactor Contactor	/3.1	

Example fr

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 connection points and the internal/external devices connected to these.

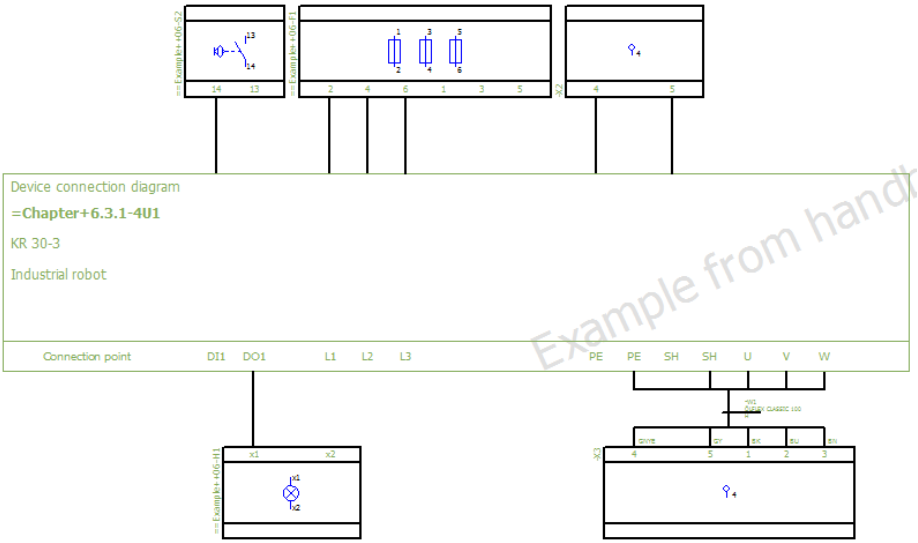


Fig. 6.7 Sample device connection diagram (file extension *.f05)

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 (information regarding the existing project pages, such as page description, page's creator, current modification date, etc.).

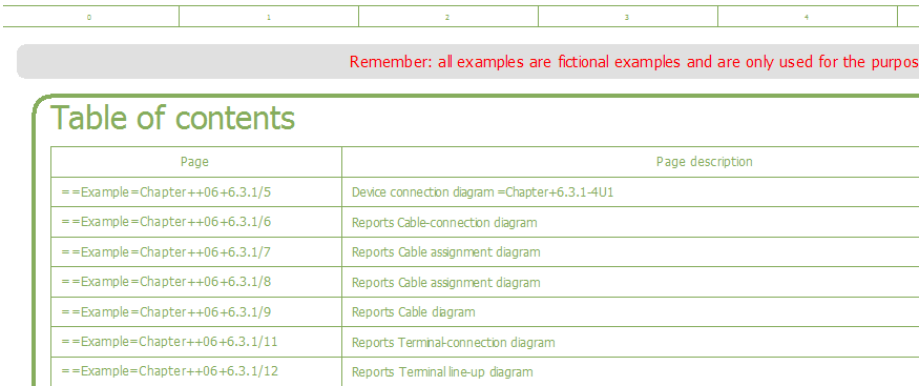


Fig. 6.8 Sample table of contents (file extension *.f06)

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.

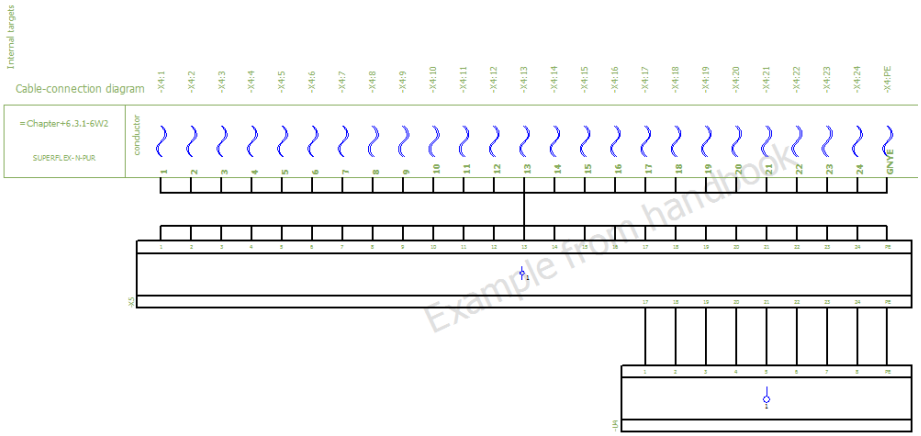


Fig. 6.9 Sample cable connection diagram (file extension *.f07)

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.

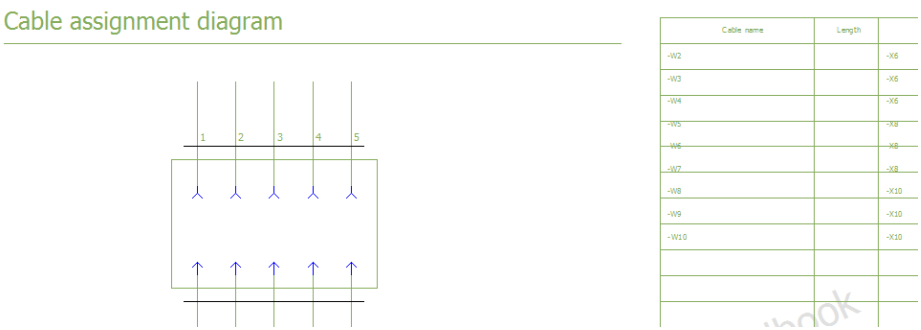


Fig. 6.10 Sample cable assignment diagram (file extension *.f08)

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.

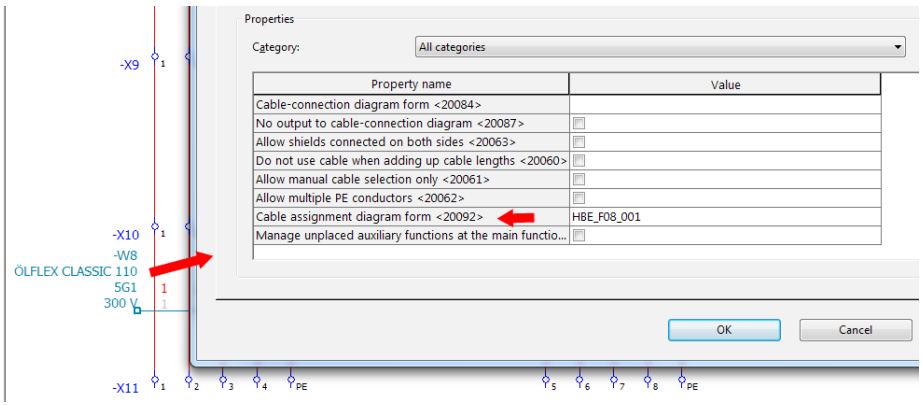


Fig. 6.11 Cable assignment diagram form entry on a cable

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 cable part in parts management (*Cable data* tab).

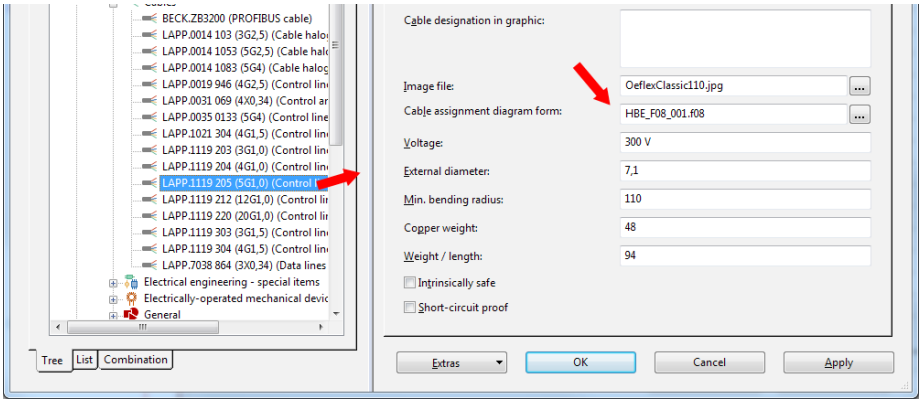


Fig. 6.12 Cable assignment diagram form entry in parts management

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		ÖLFLEX CLASSIC 110		
function text			no. of conductors		12G	cross-section	
function text	Page / column	Target designation from	Connection point	conductor	Target designation to	Connection point	
	==Example++06/9.0	-X12	1	1	-X13	1	==Ex
	==Example++06/9.1	-X12	2	2	-X13	2	==Ex
=	==Example++06/9.1	-X12	3	3	-X13	3	==Ex
=	==Example++06/9.1	-X12	4	4	-X13	4	==Ex
=	==Example++06/9.1	-X12	5	5	-X13	5	==Ex

Example from 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-section
ÖLFLEX CLASSIC 110	Control line, 3G1.5	+5.2-W13	3G	1,5
	Control line, 12G1.0	+5.2-W14	12G	1
UNITRONIC 100 CY	Control and signal line, 4x0.34	+5.2-W15	4X	0,34
ÖLFLEX CLASSIC 110	Control line, 5G1.0	+5.2-W16	5G	1
ÖLFLEX CLASSIC 110 H		+5.2-W(==Example++05-M6)	4	2,5
ÖLFLEX CLASSIC 100 H		+5.2-W(==Example++05-M7)	4	10
ÖLFLEX CLASSIC 110		+5.2-W(==Example++05-M8)	4	1,5
	Control line, 4G1.5	+5.2-W(==Example++05-M9)	4G	1,5
	Control line, 4G1.5	+5.2-W(==Example++05-M10)	4G	1,5

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.

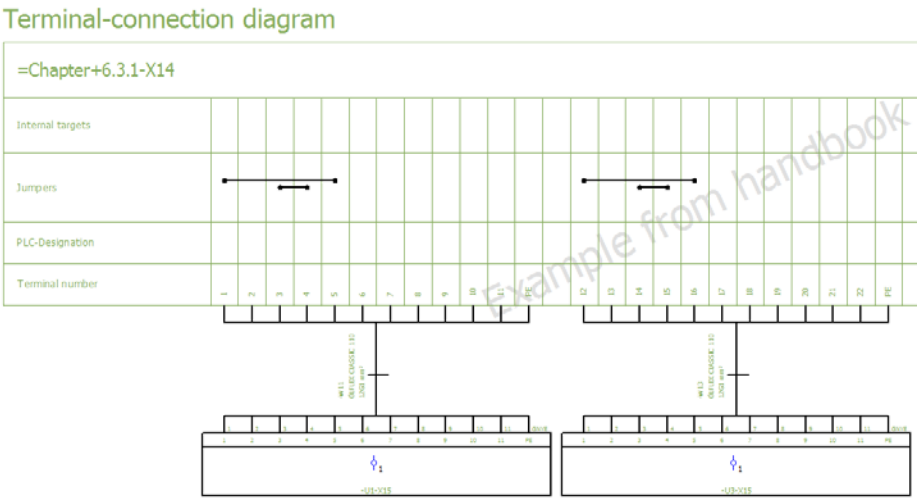


Fig. 6.15 Sample terminal connection diagram (file extension *.f11)

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.

6.3.1.11 Terminal line-up diagram *.f12

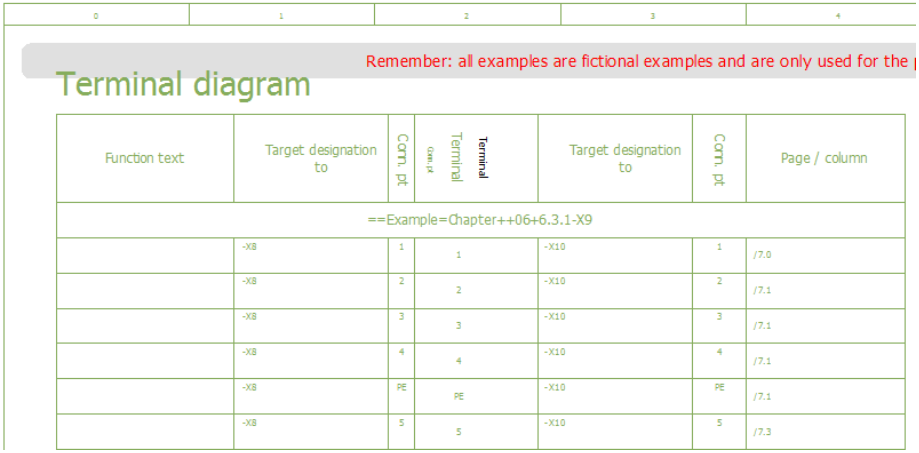
The terminal line-up diagram generates a graphical report for each terminal strip with the associated terminal (strip) parts.

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.



Terminal diagram

Remember: all examples are fictional examples and are only used for the

Function text	Target designation to	Conn. pt	Terminal	Target designation to	Conn. pt	Page / column
==Example=Chapter++06+6.3.1-X9						
	-X8	1	1	-X10	1	/7.0
	-X8	2	2	-X10	2	/7.1
	-X8	3	3	-X10	3	/7.1
	-X8	4	4	-X10	4	/7.1
	-X8	PE	PE	-X10	PE	/7.1
	-X8	5	5	-X10	5	/7.3

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
++04+4.1.1-X19		1	25	1
++04+4.1.1-X20		1	25	1
++04+4.1.1-X21		1	25	1
++04+4.1.1-X22		1	7	1
++04+4.1.1-X23		1	7	1
++04+4.1.1-X25		1	50	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.

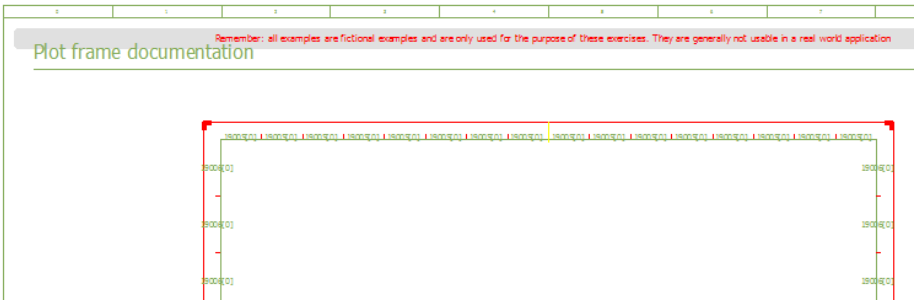


Fig. 6.19 Sample plot frame documentation (file extension *.f15)

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

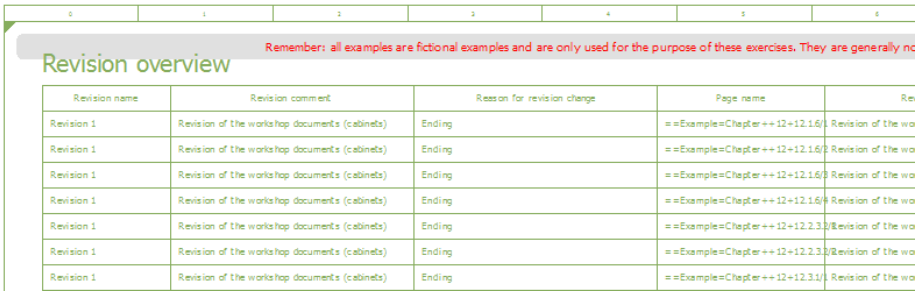
Name of potential	Potential value	Frequency	Potential type	Placement
0V DC			-	/17.3
24V DC	24V DC		+	/17.2
L1	400	50		++08+8.6.1/2.6
L1				++04+4.10/1.2
L1				++12+12.1.6/1.6
L2	400	50		++08+8.6.1/2.6

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

Remember: all examples are fictional examples and are only used for the purpose of these exercises. They are generally no

Revision name	Revision comment	Reason for revision change	Page name	Re
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.1.6/1	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.1.6/2	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.1.6/3	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.1.6/4	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.2.3.1/1	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.2.3.1/2	Revision of the wo
Revision 1	Revision of the workshop documents (cabinets)	Ending	==Example=Chapter++12+12.3.1/1	Revision of the wo

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

KK1-MPL		
Pos.-Nr.	device tag	Type number
5	U2	KK6040 Cable duct 60x40
6	U3	KK6040 Cable duct 60x40
7	U4	KK6060 Cable duct 60x60

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 a completely new report page. 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.

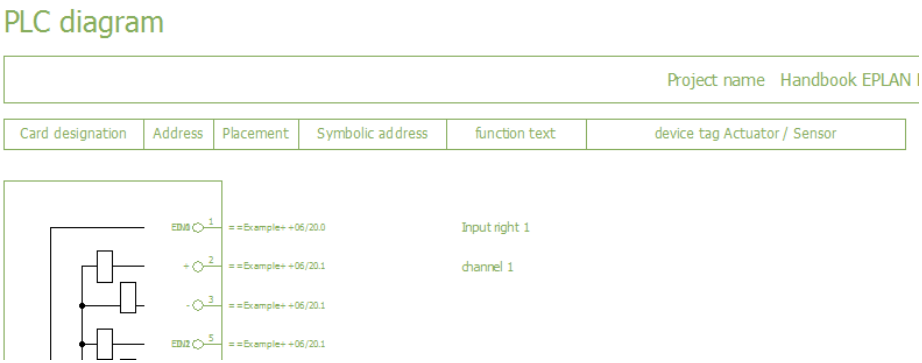


Fig. 6.23 Sample PLC diagram (file extension *.f19)

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.

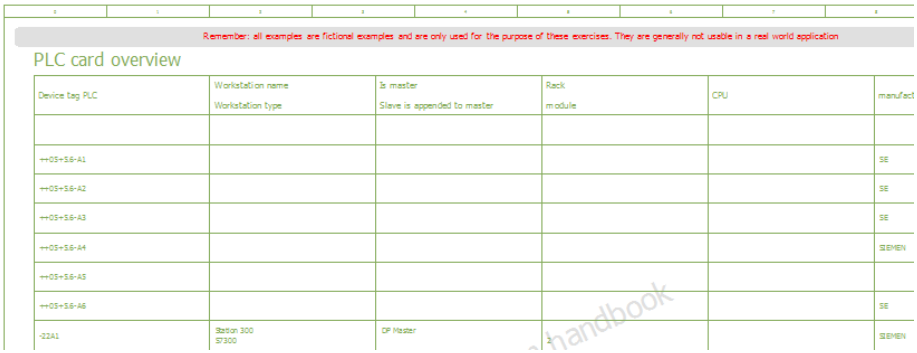


Fig. 6.24 Sample PLC card overview (file extension *.f20)

You can, of course, display each PLC component on its own graphical report page of the PLC card overview type. 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).

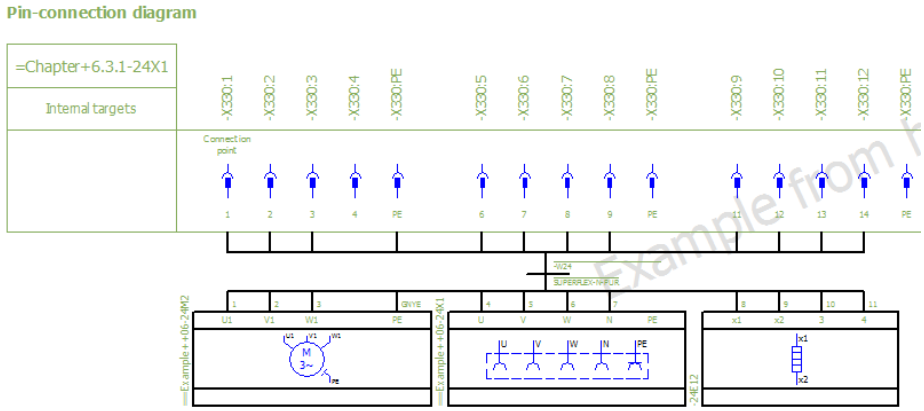


Fig. 6.25 Sample pin connection diagram (file extension *.f21)

A graphical output usually also includes a pin connection diagram. These can also be set to dynamic in the settings so that several plug 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.

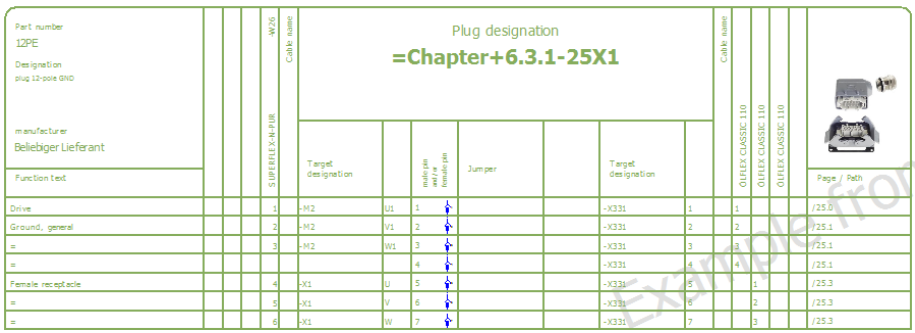


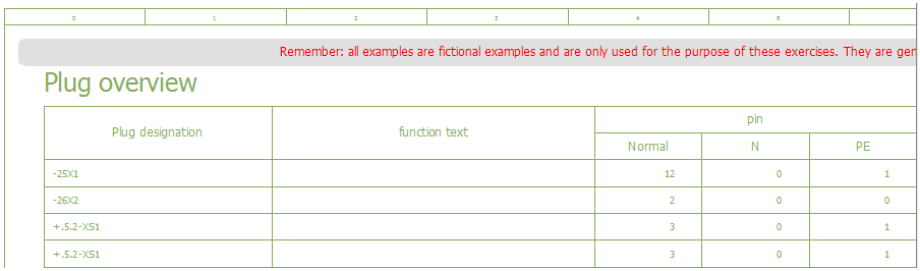
Fig. 6.26 Sample plug diagram (file extension *.f22)

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.



Remember: all examples are fictional examples and are only used for the purpose of these exercises. They are ger

Plug overview

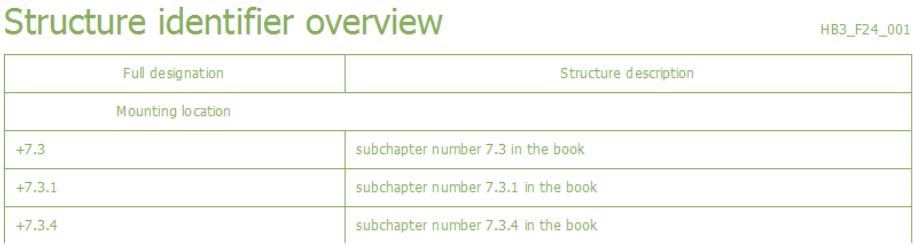
Plug designation	function text	pin		
		Normal	N	PE
-25X1		12	0	1
-26X2		2	0	0
+.5.2-XS1		3	0	1
+.5.2-XS1		3	0	1

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
Mounting location	
+7.3	subchapter number 7.3 in the book
+7.3.1	subchapter number 7.3.1 in the book
+7.3.4	subchapter number 7.3.4 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>**).

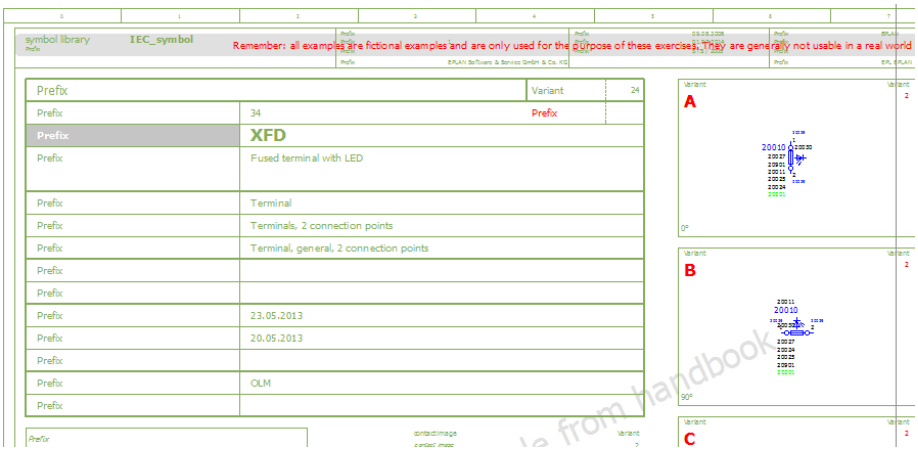


Fig. 6.29 Sample symbol overview (file extension *.f25)

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.

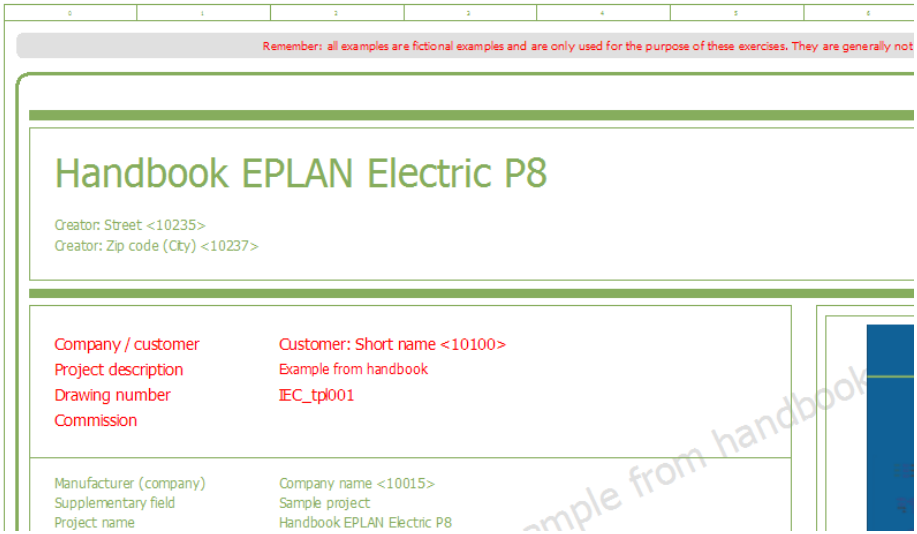


Fig. 6.30 Sample title page/cover sheet (file extension *.f26)

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.

Connection list

HB3_F27_001

connection	Target 1	Target 2	Color	cross-section
001	-X332:1	-X333:1		
002	-X332:2	-X333:2		
003	-X332:3	-X333:3		
004	-X332:4	-X333:4		

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. It bears mentioning here for completeness’ sake. Graphical forms have the file extension *.f28 and can be entered into the page properties as a “placeholder form graphics”.

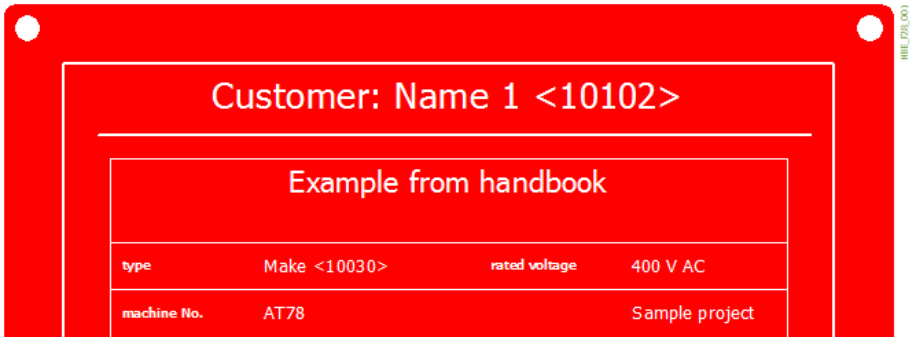


Fig. 6.32 Sample graphical form (file extension *.f28)

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.

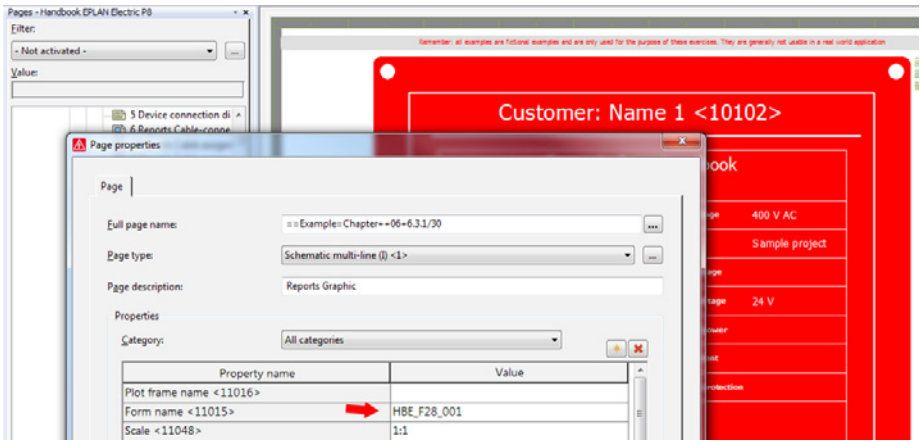


Fig. 6.33
Entering a form in the page properties

6.3.1.29 Project options overview *.f29

The project options overview report contains information on possible project options.

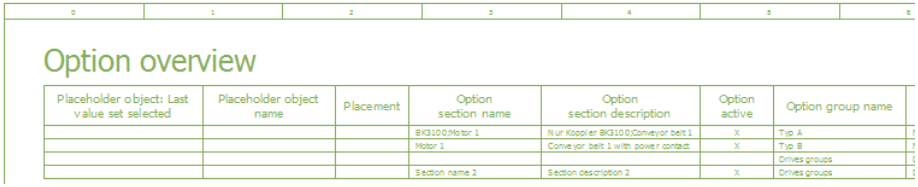


Fig. 6.34
Sample project options overview (file extension *.f29)

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.

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.

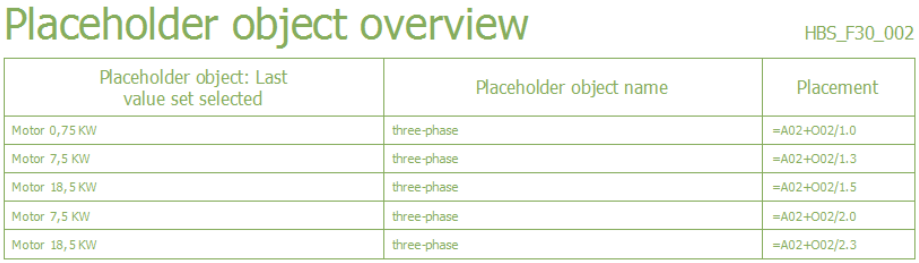


Fig. 6.35
Sample placeholder object overview (file extension *.f30)

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.

Remember: all examples are fictional examples and are only used for the purpose of these exercises. They a

Manufacturer / supplier list

Company	short name	Street	ZIP code
ABB	ABB	Eppelheimer Str. 82	69123
Beliebiger Lieferant	BEL		
	BGI		
Festo	FESTO	Ruiter Str. 82	73734
Kuka	KUKA	Zugspitzstr. 140	86165
Lenze	LENZE	Hans-Lenze-Str. 1	31855
Mennekes	MEN	Aloys-Mennekes-Str. 1	57399
Moeller	MOE	Hein-Moeller-Str. 7-11	53115

Fig. 6.36 Sample manufacturer/supplier list (file extension *.f31)

EPLAN's usual filter and sorting options are available here as well.

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.

6.3.1.38 Pre-planning: Structure segment overview *.f38

In the Pre-planning: Structure segment overview report, all structure segments and necessary expenditure are output from the subordinate (integrated) planning objects.

6.3.1.39 Pre-planning: Structure segment plan *.f39

The Pre-planning: Structure segment plan report evaluates each structure segment's entered properties.

6.3.1.40 Pre-planning: Planning object overview *.f40

The Pre-planning: Planning object overview report provides an overview of all planning objects and the necessary expenditure (hardware, software, etc. in hours).

6.3.1.41 Pre-planning: Planning object plan *.f41

The Pre-planning: Planning object plan report outputs the data for a planning object that has been entered in the planning object.

6.3.1.42 Pre-planning: Segment template overview *.f42

The report provides an overview, for each segment template, of all data in segment templates.

6.3.1.43 Pre-planning: Segment template plan *.f43

This report generates for each segment template the data that has been entered in the segment template.

6.3.1.44 Assembly/Module overview *.f44

The report lists all assemblies and modules, including their detailed data (such as part numbers, designations, descriptions, etc.), used in the project. Apart from this data, it is also possible to specify the quantity of use or the macro of the assembly and/or module.

Assembly/Module overview

SIE.3-polige Neozed-Sicherung 25A
3-pole Neozed fuse 25 A kpl.

Dash number	Quantity	Part number	Designation	Description
5SG5700	1	SIE.5SG5700	NEOZED built-in fuse base	
5SE2325	3	SIE.5SE2325	NEOZED fuse link	400 V GL D02/25 A, Folding box
5SH5025	3	SIE.5SH5025		
5SH4362	3	SIE.5SH4362		

Dash number	Quantity	Part number	Designation	Description
HAUPTSCHALTER	1	XXX.HAUPTSCHALTER	Main / Emergency stop switch	
HS11	1	XXX.HS11	Normalhilfsschalter 1S + 1Ö	

Fig. 6.37 Example Assembly/Module overview (file extension *.f44)

6.3.1.45 Distributed device list *.f45

This report outputs device properties and the associated, cross-referenced functions. For example, it can be used to generate graphical contact images with further functions that the “regular” EPLAN contact image does not offer.

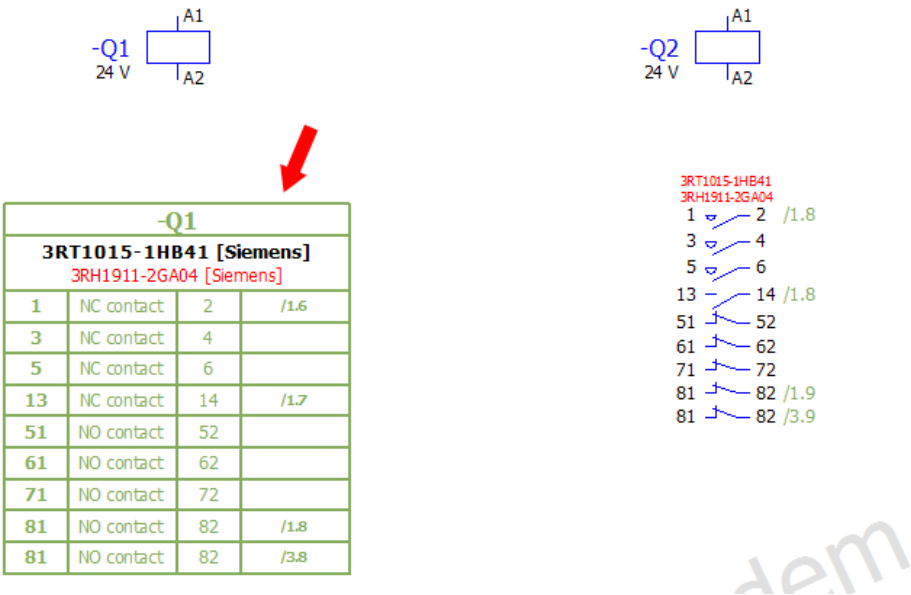


Fig. 6.38 Example Distributed device list (file extension *.f45)

6.3.1.46 Bundle/conduit plan *.f46

This report evaluates properties of bundles/conduits (Fluid power connections, such as tubes or pipes), and lists them in a clear overview.

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.

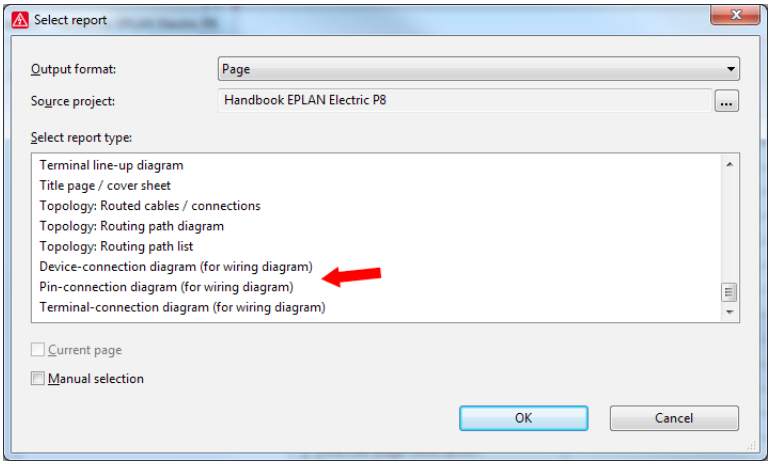


Fig. 6.39 Special connection diagrams

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. An example of such connection diagrams can be seen in Fig. 6.40 and Fig. 6.42.

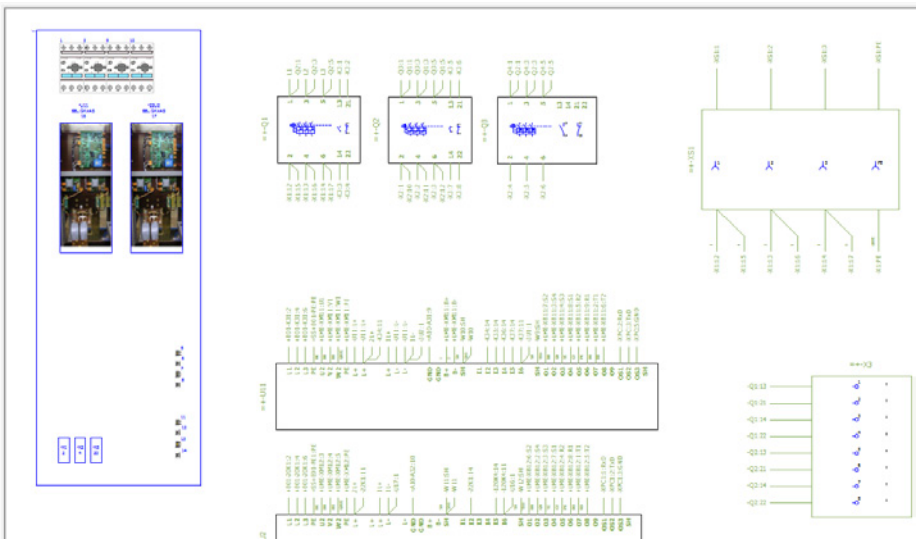


Fig. 6.40 Connection diagrams (here device connection points)

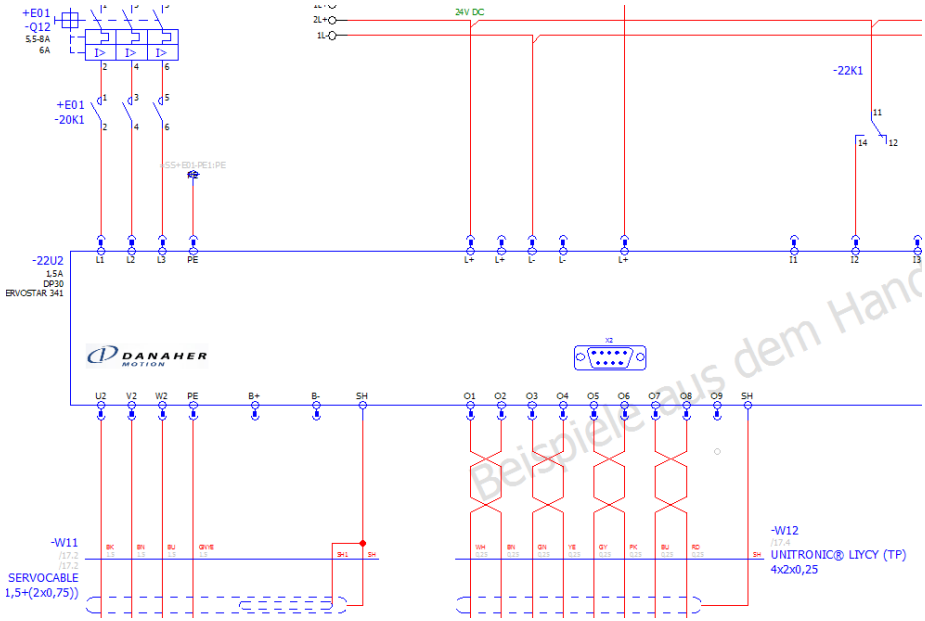


Fig. 6.41 Extract from the schematic for device 22U2

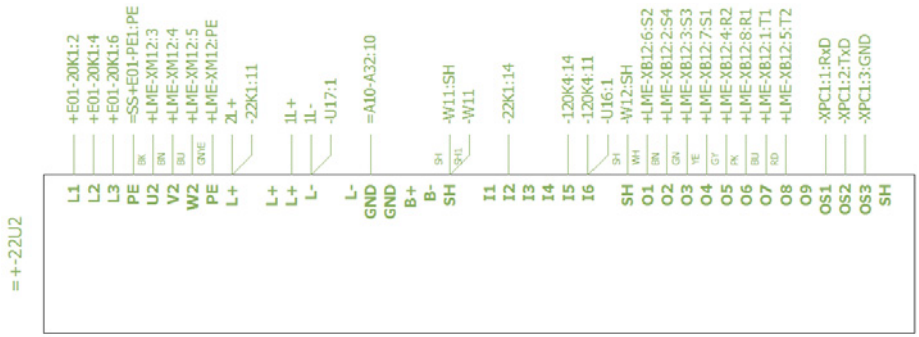


Fig. 6.42 Connection diagram showing detached representation of a device's (22U2) connection points

6.3.3 Next forms

For certain report types (e. g. cable assignment diagrams or terminal-strip overviews, etc.) it is possible to specify a next form. This is how it works. When generating a report, the next form is output after the current form.

It is inserted either on the same page as the current form or on a following page. This is specified via the insertion point of next form.

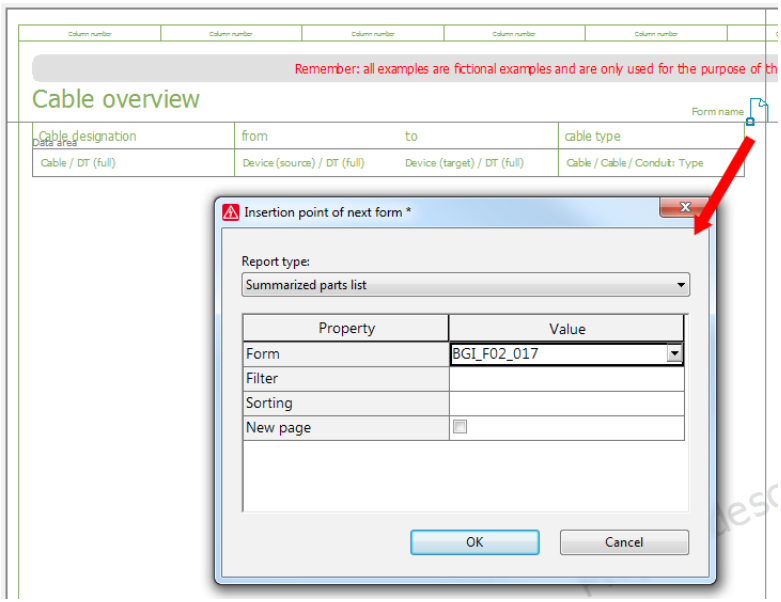


Fig. 6.43 Defining properties in the insertion point of next form

6.3.3.1 Insertion point of next form

An insertion point of next form can be placed in a form in such a way that EPLAN generates another report precisely from this point forward. In this way, reports can be combined, such as a cable overview and a summarized parts list that combines and represents as a report the parts of these cables.

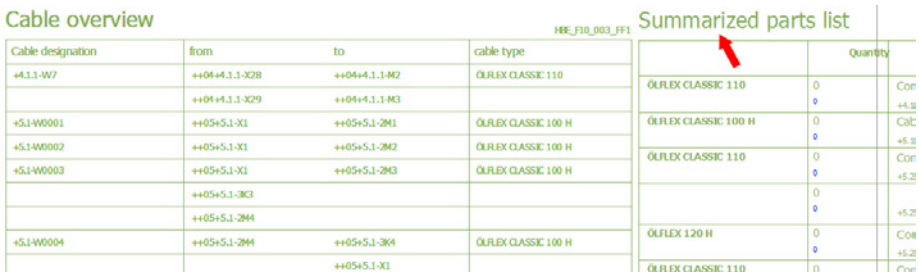


Fig. 6.44 Example of combined report

In fact, several insertion points of next form can be inserted in a form. The report sequence is determined by the placement of the graphic.

The desired next report and the next form are entered at the insertion point of next form. Here, too, the user can define filters and sorting for the graphical output of the reports.

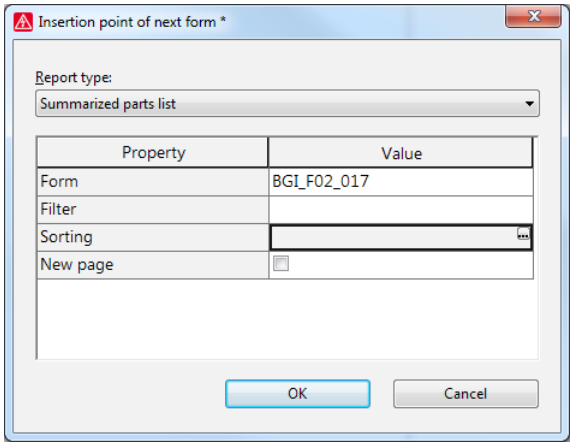


Fig. 6.45 Insertion point of next form dialog

6.3.3.2 Possible reports for next forms

Depending on the report type, there are several report types available for the next forms. The reports "Summarized parts list" and "Parts list" are the reports available for most forms.

The following table shows, for example, that a next form of the "Cable diagram" report type is also possible for the "Cable diagram" report type.

	Possible next forms			
Possible form types	**Summarized parts list *.f02**	**Parts list *.f01**	**Identical form type possible?**	**Additional reports possible?**
Assembly/Module overview *.f44			Yes	
Device tag list *.f03	Yes	Yes		
Cable assignment diagram *.f08	Yes	Yes		
Cable diagram *.f09	Yes	Yes	Yes	
Cable overview *.f10	Yes	Yes		
Terminal line-up diagram *.f12	Yes	Yes		Terminal overview
Terminal strip overview *.f14	Yes	Yes		
Terminal diagram *.f13	Yes	Yes	Yes	Terminal line-up diagram
PLC diagram *.f19	Yes	Yes	Yes	
PLC card overview *.f20	Yes	Yes		
Plug diagram *.f22	Yes	Yes	Yes	
Plug overview *.f23	Yes	Yes		
Connection list *.f27	Yes	Yes		
Topology: Routing path list	Yes	Yes		

	Possible next forms			
Possible form types	Summarized parts list *.f02	Parts list *.f01	Identical form type possible?	Additional reports possible?
Topology: Routed cables/ connections*.f36	Yes	Yes		
Pre-planning: Planning object overview *.f40	Yes			
Pre-planning: Planning object plan *.f41			Yes	
Pre-planning: Structure segment plan *.f39			Yes	

6.3.4 Conditional forms

Conditional forms are a type of embedded form linked to specific conditions and integrated into a dynamic "parent form" (report). But the conditional form must always be the same report type as its "parent form".

6.3.4.1 Insertion point of form: Conditional forms

Conditional forms are inserted in the form editor. First, the corresponding "parent form" must be opened. Then, select via the **INSERT** menu the **CONDITIONAL FORMS** menu item. Now, the insertion point of form hangs on the cursor and can be placed. After placement, EPLAN opens the **CONDITIONAL FORMS** dialog immediately. The appropriate form can be selected and the conditions be assigned in this dialog.

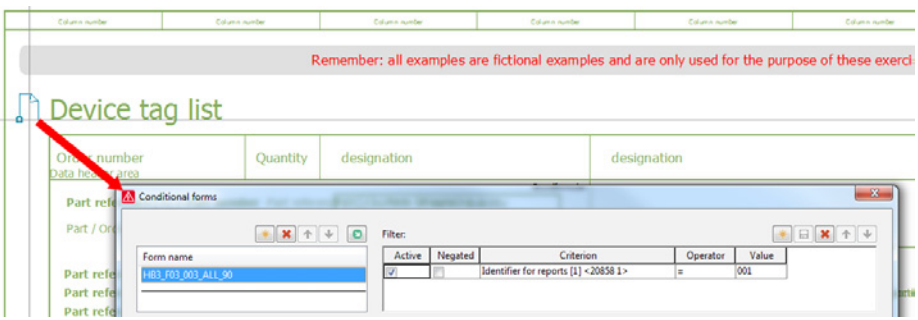


Fig. 6.46 Device tag list with embedded form (Form editor view), linked to a condition (criterion)

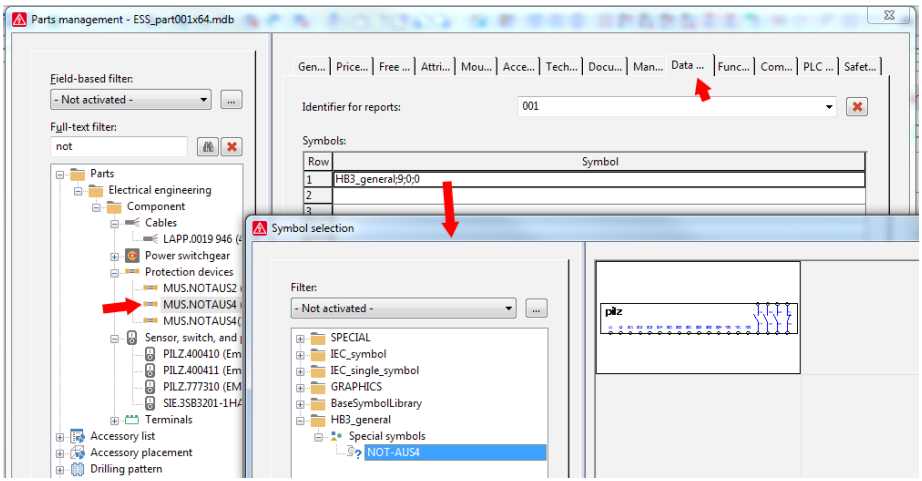


Fig. 6.47
Existing criterion from parts management at the part (Data for reports tab)

In this example, the conditional form would compare the identifier 001 at the part (*Data for reports* tab). If this identifier matches (i. e. it is identical), EPLAN is to represent the symbol from the linked symbol library in the report, instead of the multi-line components on the schematic page.

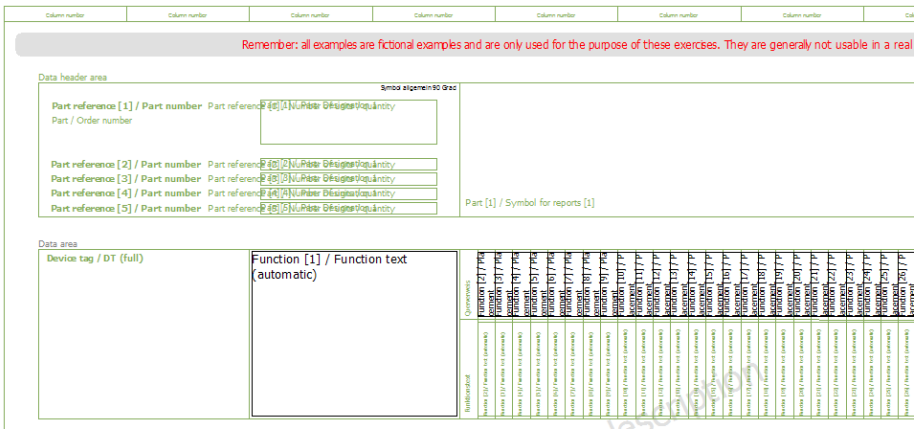


Fig. 6.48
Structure of the embedded form (Form editor view)

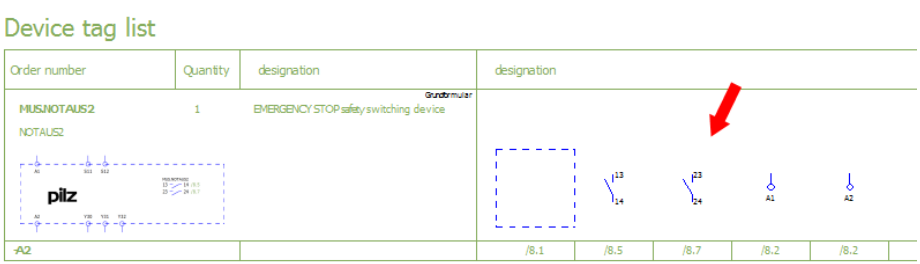


Fig. 6.49
Example of report (device tag list) without conditional form and without criterion

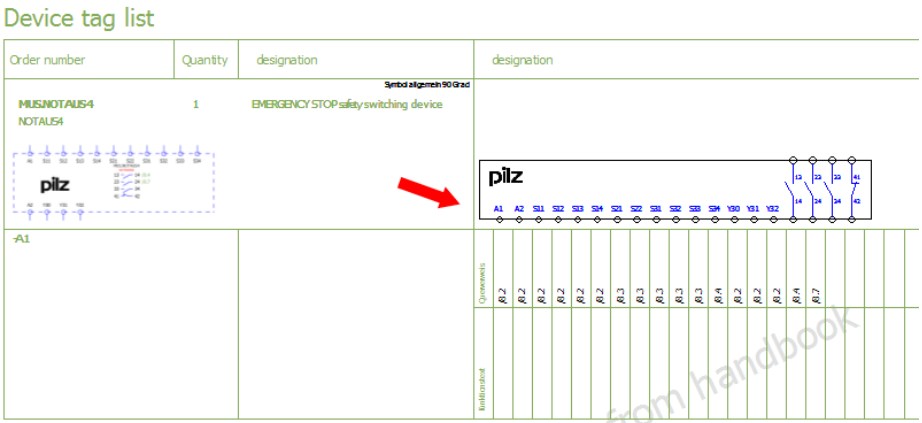


Fig. 6.50 Example of report (device tag list) with conditional form and criterion

6.3.4.2 Possible reports for conditional forms

The following “parent forms” (reports) can use conditional forms:

- Device tag list *.f03
- Cable diagram *.f09
- Terminal line-up diagram *.f12
- Enclosure legend *.f18
- PLC diagram *.f19
- Plug diagram *.f22

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, which is therefore always up to date. The system automatically updates (depending on the settings) the connections before EPLAN generates a report.

6.4.1 The Display/output project setting

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.

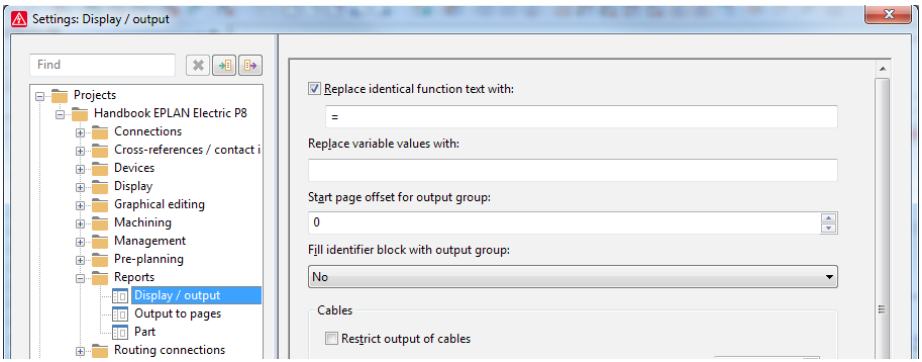


Fig. 6.51 Settings for the display and output of reports

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.

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.

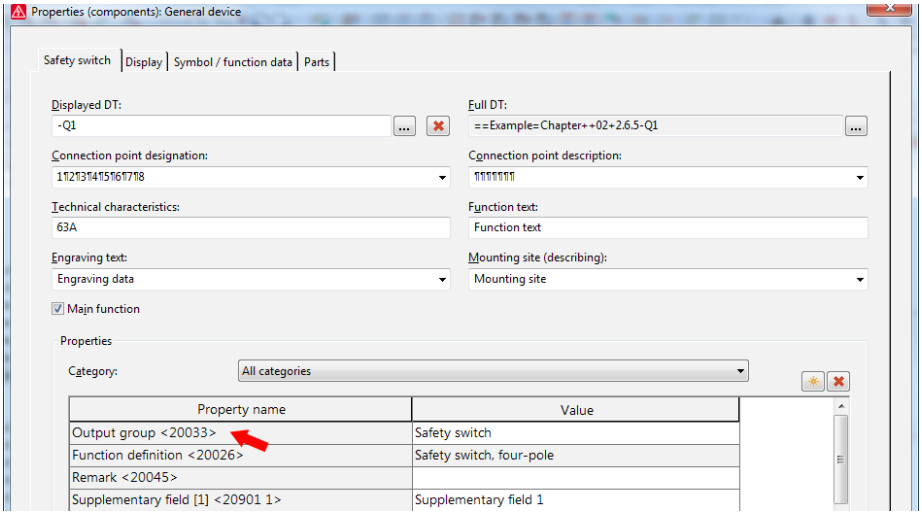


Fig. 6.52 Output group <20033> component property

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

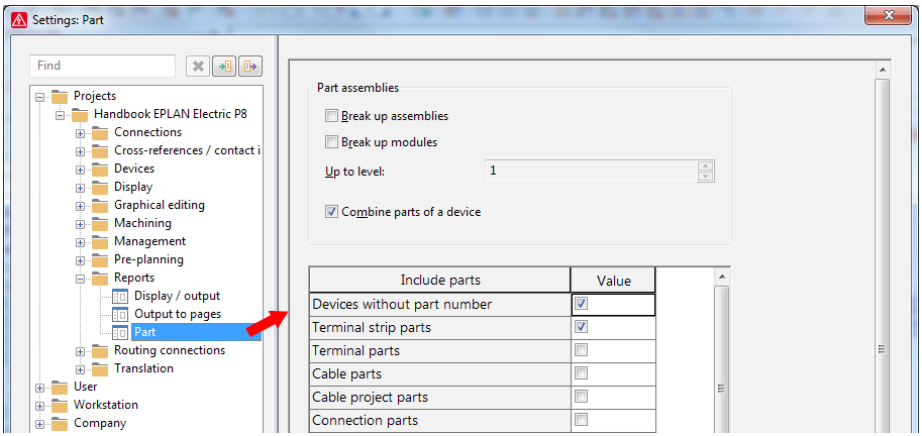


Fig. 6.53 Settings for the output of parts

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.

Parts list

designation	part number	Quantity	Device Designation
Designation 1 MDOUL 1	XXX.Artikelnummer MODUL1	1	=A01+E05-U3
Designation 1 MDOUL 1	XXX.Artikelnummer MODUL1	1	=A01+E05-12U4

Fig. 6.54 Parts list with attached representation (here: Module U3)

Parts list

designation	part number	Quantity	Device Designation
Motor overload switch	SIE.3RV1021-1CA15	1	-U3-F1
Contactor	SIE.3RT1015-1BB41	1	-U3-Q1
Auxiliary switch block	SIE.3RH1911-2GA04	1	-U3-Q1
Contactor	SIE.3RT1015-1BB41	1	-U3-Q2
Auxiliary switch block	SIE.3RH1911-2GA04	1	-U3-Q2

Fig. 6.55 Parts list with detached representation (here: Module U3)

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.

Parts list

Total

designation	part number	Quantity	Device Designation
Universal terminal with screw-type connection	PXC.3044076	7	=A01+E05-X7
Protective conductor terminal	PXC.3044092	3	=A01+E05-X7

Parts list

Total

designation	part number	Quantity	Device Designation
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7
Universal terminal with screw-type connection	PXC.3044076	1	=A01+E05-X7

Fig. 6.56 Example of report – once with terminal parts combined; once with all terminal parts evaluated individually

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

Generally, forms are used for outputting reports (forms). 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.

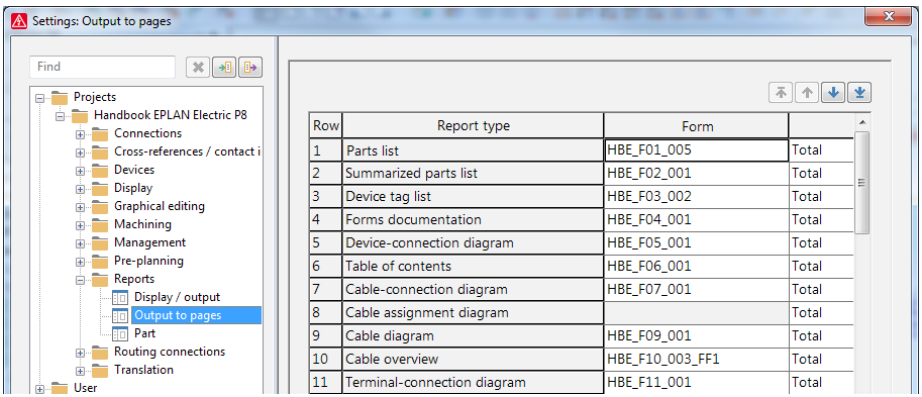


Fig. 6.57 Settings for the output to pages

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.

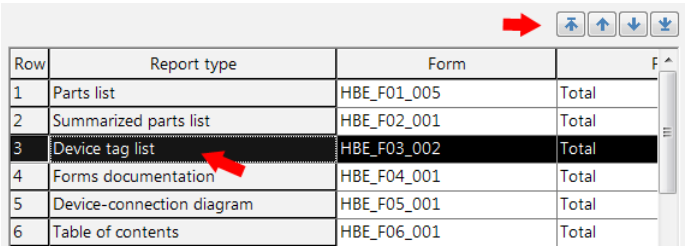
Report type
Parts list
Summarized parts list
Device tag list
Forms documentation
Device-connection diagram
Table of contents

Fig. 6.58
Report types

But it is possible to modify the sequence of rows (reports), thus, for example, always moving to the front only the required report types, and the reports less often or not at all needed to the end.

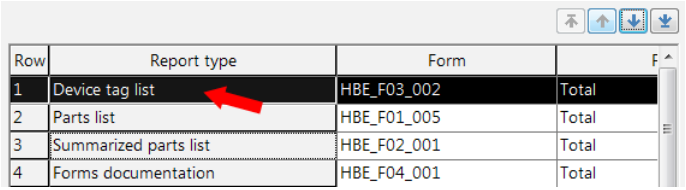
NOTE: A change in the report sequence may also entail a change of the sorting into the page structure. But this depends on the respective starting block specified.

The table of contents, however, is generally evaluated and generated at the end, because all changes to pages (number, arrangement in the page structure, etc.) must be documented and evaluated here.



Row	Report type	Form	F
1	Parts list	HBE_F01_005	Total
2	Summarized parts list	HBE_F02_001	Total
3	Device tag list	HBE_F03_002	Total
4	Forms documentation	HBE_F04_001	Total
5	Device-connection diagram	HBE_F05_001	Total
6	Table of contents	HBE_F06_001	Total

Fig. 6.59 Sequence of the arrangement prior to a move



Row	Report type	Form	F
1	Device tag list	HBE_F03_002	Total
2	Parts list	HBE_F01_005	Total
3	Summarized parts list	HBE_F02_001	Total
4	Forms documentation	HBE_F04_001	Total

Fig. 6.60 Sequence following a move

6.4.3.2 The Form column

The desired form can be set in the **Form** column. Click the **FORM** row. In the drop-down selection list that appears, you can import an existing entry or click the **BROWSE** entry. 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.

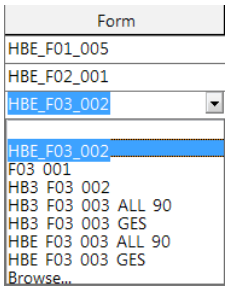


Fig. 6.61 Forms

TIP: After a selection, 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. Generally, of course, the starting block (the start page) of a report is also considered in the report template.

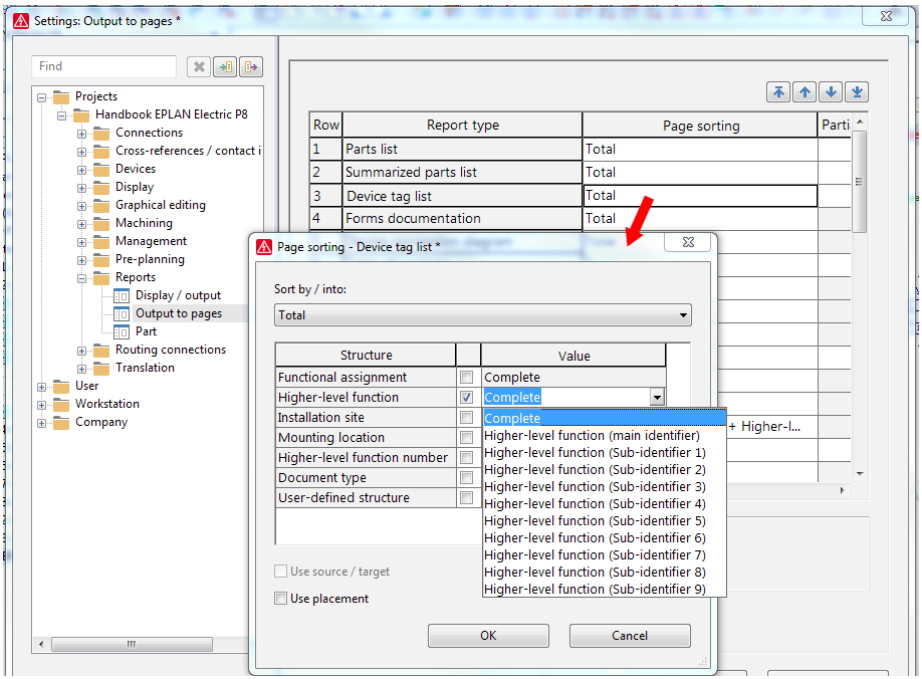


Fig. 6.62 Page sorting

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. A mounting location that may be specified on the start age would then be ignored.

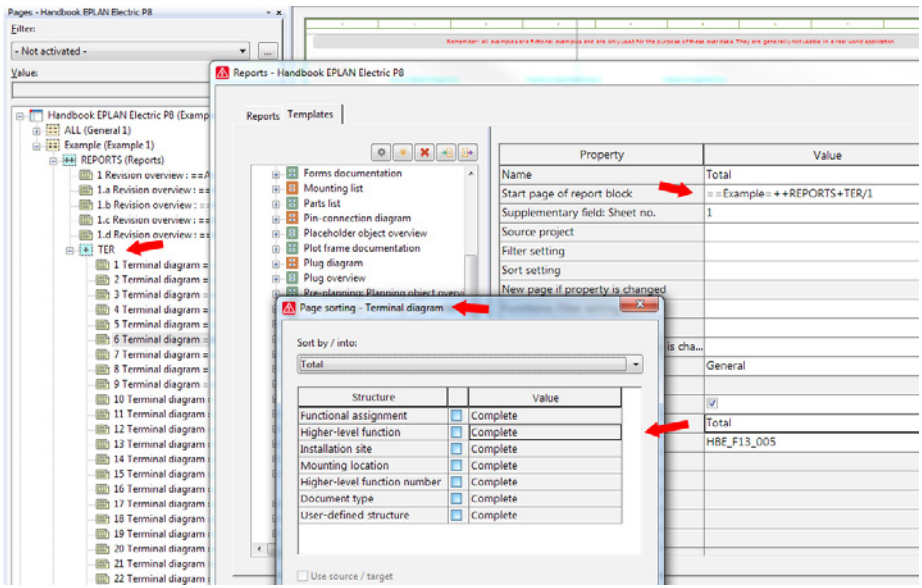


Fig. 6.63 Example of output by Total (specified via start page)

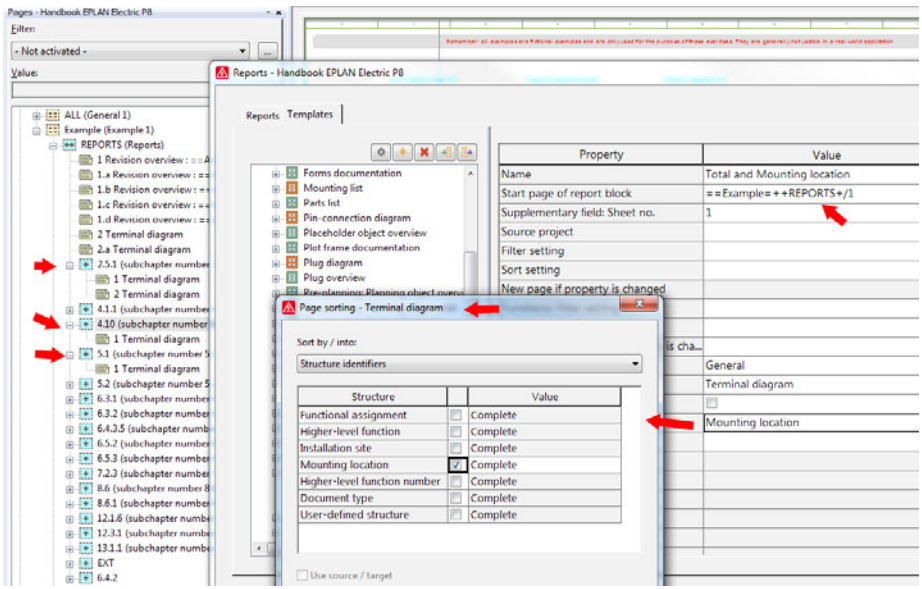


Fig. 6.64 Example of output by Total and Mounting location of the terminal strip

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.

6.4.3.4 Partial output column

Partial output is an interesting setting. For the report type where this is possible, you can automatically generate a (partial) table of contents for each higher-level function in addition to a main form such as a complete table of contents. 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) table of contents and a (partial) table of contents for each individual higher-level function. We explicitly point out here that the structure and graphical content of these forms can be different.

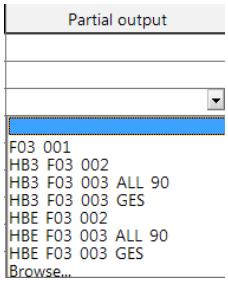


Fig. 6.65 Partial output

The total output of the table of contents lists all project pages, while the partial output of the table of contents lists only the pages in the respective identifier area assigned – in this case, the higher-level function.

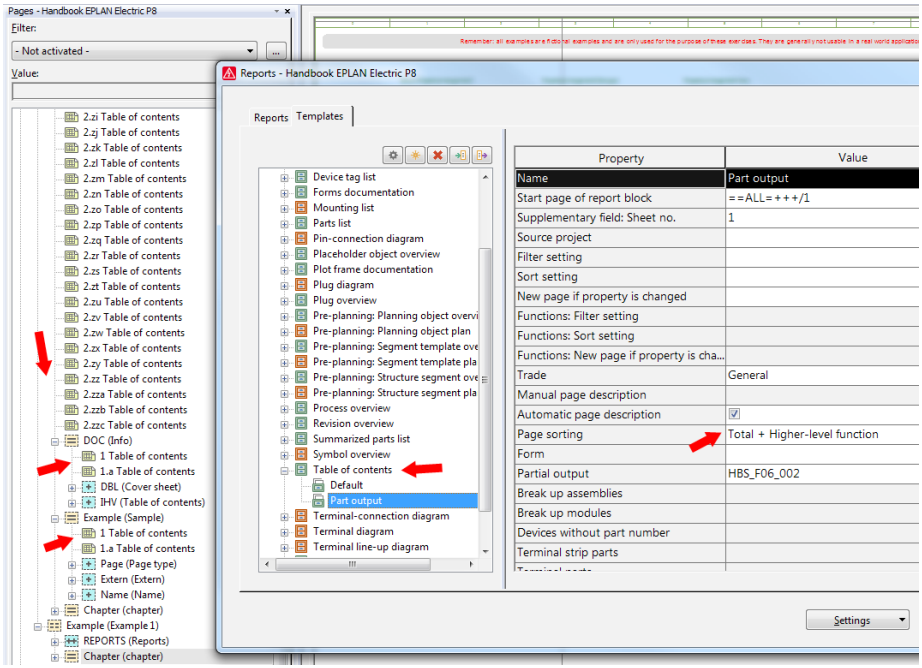


Fig. 6.66 Generated total and partial output of a table of contents

6.4.3.4.1 Possible reports for partial outputs

The following forms (reports) can use the partial output (other or similar forms, but of the same report type):

- Parts list *.f01
- Summarized parts list *.f02
- Device tag list *.f03
- Manufacturer/supplier list *.f31
- Table of contents *.f06
- Cable overview *.f10
- Terminal strip overview *.f14
- PLC card overview *.f20
- Plug overview *.f23
- Title page/cover sheet *.f26
- Topology: Routing path list *.f34
- Topology: Routed cables/connections *.f36
- Connection list *.f27
- Pre-planning: Planning object overview *.f40
- Pre-planning: Structure segment overview *.f38

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).

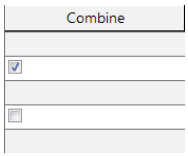


Fig. 6.67
Combining

The examples in Fig. 6.68 and Fig. 6.69 make this clearer. In Fig. 6.68, the **Combine** option was *not* selected. EPLAN generates one page of the *Terminal diagram* per terminal strip.

In Fig. 6.69, the **Combine** option was set to *active*. 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.

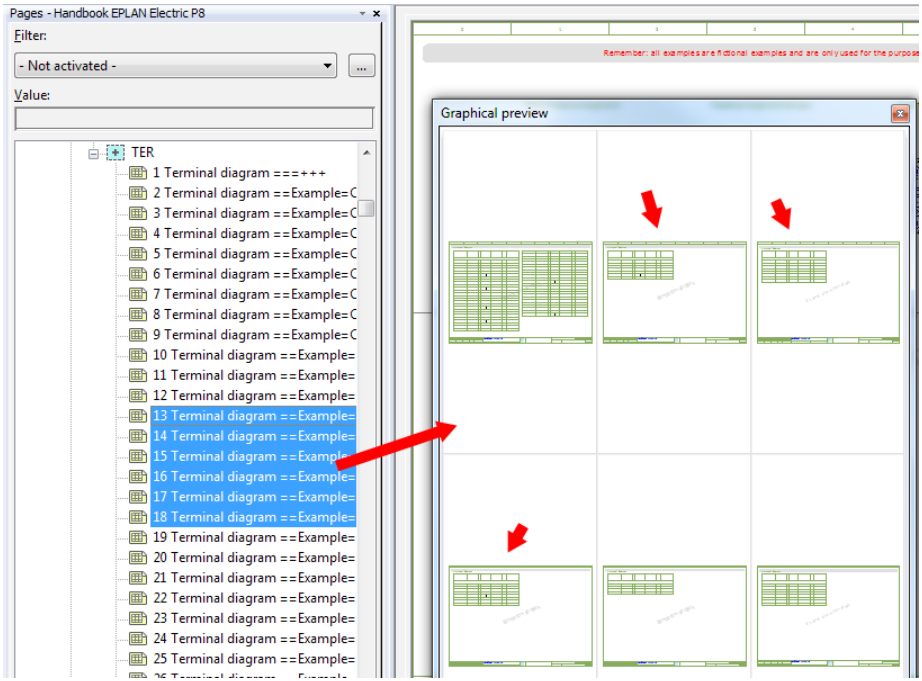


Fig. 6.68 Combine option not activated

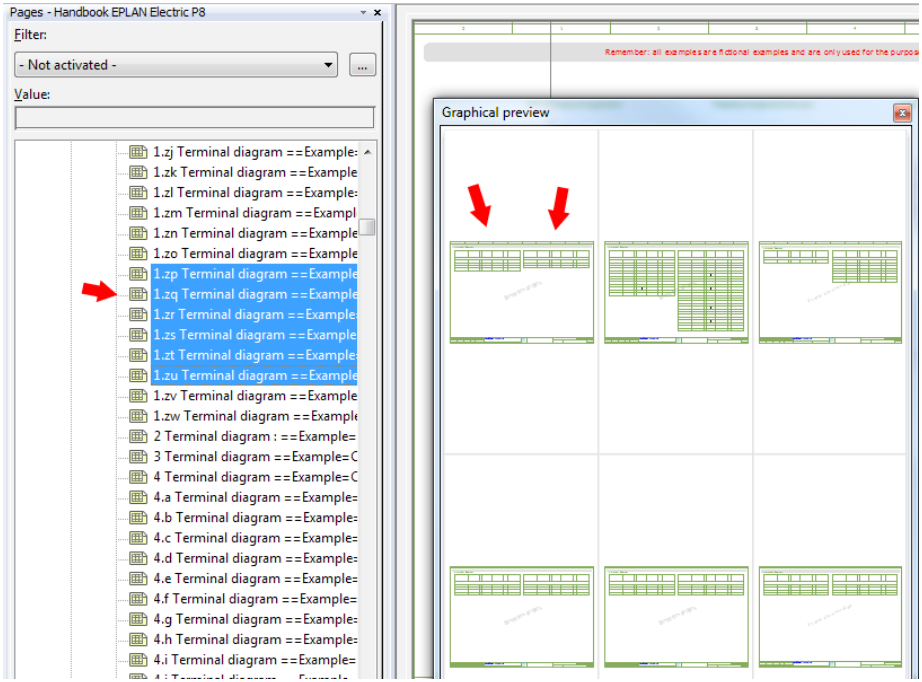


Fig. 6.69 Combine option activated

6.4.3.5.1 Possible Combine reports

The following forms (reports) can use the **Combine** option:

- Assembly/Module overview *.f44
- Distributed device list *.f45
- Device connection diagram *.f05
- Cable connection diagram *.f07
- Cable diagram *.f09
- Terminal connection diagram *.f11
- Terminal line-up diagram *.f12
- Terminal diagram *.f13
- Mounting list *.f32
- Bundle/conduit plan *.f46
- Enclosure legend *.f18
- PLC diagram *.f19
- Pin connection diagram *.f21
- Plug diagram *.f22
- Symbol overview *.f25
- Topology: Routing path diagram *.f35
- Pre-planning: Planning object plan *.f41
- Pre-planning: Segment template plan *.f43
- Pre-planning: Structure segment plan *.f39

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.

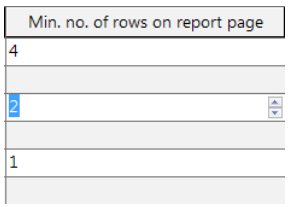


Fig. 6.70
Minimum number of rows

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 is not to change. This means that *subpages* must be created for the graphical output pages.

If this option is activated, the character definition, that is, the format structure of the subpage numbering (characters of the page number) must also be taken into account.

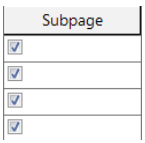


Fig. 6.71 Subpages

6.4.3.8 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.

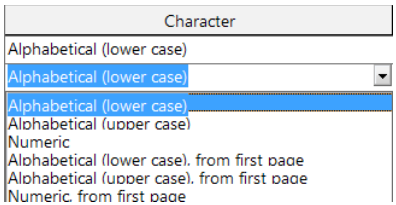


Fig. 6.72 Characters

Setting/Selection	Report pages start with
Alphabetical (lower case)	20; 20.a; 20.b; 20.c
Alphabetical (upper case)	15; 15.A; 15.B; 15.C
Numeric	301; 301.1; 301.2; 301.3
Alphabetical (lower case), from first page	5.a; 5.b; 5.c
Alphabetical (upper case), from first page	7.A; 7.B; 7.C
Numeric, from first page	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.

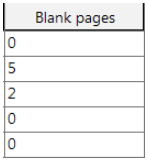


Fig. 6.73 Blank pages

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.

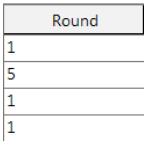


Fig. 6.74 Rounding

6.4.3.11 The Hide structure identifier if identical column

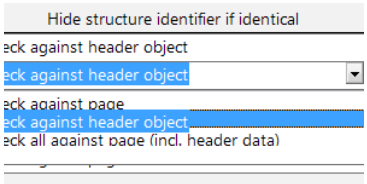


Fig. 6.75
Hide structure identifier if identical

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.

No selection is possible for the following reports, because structure identifiers cannot be abbreviated here – they are not even part of the report.

- Forms documentation *.f04
- Manufacturer/supplier list *.f31
- Table of contents *.f06
- Plot frame documentation *.f15
- Project options overview *.f29
- Revision overview *.f17
- Structure identifier overview *.f24
- Symbol overview *.f25
- Title page/cover sheet *.f26

There are four 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 all against page (incl. header data):* This setting, for example, applies to function-related reports, such as a terminal diagram. If structure identifiers or components of the structure identifiers of the devices that were output match the structure identifiers of the page, then the structure identifiers are not displayed in the header (the terminal strip designation) either.
- *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.

The following table provides a clear overview of which report can be used with which abbreviation rules. The usage is identified by x.

Report type	Check against page	Check against header object	Check all against page (incl. header data)	No
Assembly/Module overview *.f44	x	x	x	x
Parts list *.f01	x			x
Summarized parts list *.f02	x			x
Device tag list *.f03	x			x
Distributed device list *.f45	x	x	x	x
Device connection diagram *.f05	x	x	x	x
Cable connection diagram *.f07	x	x	x	x
Cable assignment diagram *.f08	x			x
Cable diagram *.f09	x	x	x	x
Cable overview *.f10	x			x
Terminal connection diagram *.f11	x	x	x	x
Terminal line-up diagram *.f12	x	x	x	x
Terminal strip overview *.f14	x			x
Terminal diagram *.f13	x	x	x	x
Mounting list *.f32	x	x	x	x
Bundle/conduit plan *.f46	x	x	x	x
Placeholder object overview *.f30	x			x
PCT loop list *.f33	x			x
Potential overview* .f16	x			x
Process overview *.f37	x			x
Enclosure legend *.f18	x	x	x	x
PLC diagram *.f19	x	x	x	x
PLC card overview *.f20	x			x
Pin connection diagram *.f21	x	x	x	x
Plug diagram *.f22	x	x	x	x
Plug overview *.f23	x			x
Topology: Routing path list *.f34	x			x
Topology: Routing path diagram *.f35	x	x	x	x
Topology: Routed cables/connections *.f36	x			x
Connection list *.f27	x			x
Pre-planning: Planning object overview *.f40	x			x
Pre-planning: Planning object plan *.f41	x	x	x	x
Pre-planning: Segment template plan *.f43	x	x	x	x
Pre-planning: Segment template overview *.f42	x			x
Pre-planning: Structure segment overview *.f38	x			x
Pre-planning: Structure segment plan *.f39	x	x	x	x

6.4.3.12 The Synchronize column

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 (entered here) having synchronizing *active* (check box set) are automatically synchronized with the system master data when the project is opened.

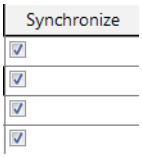


Fig. 6.76
Synchronize

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.

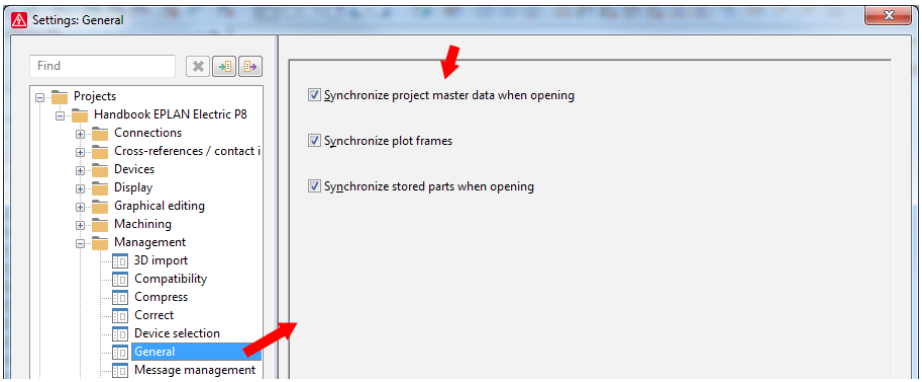


Fig. 6.77 Allow automatic synchronization of master data

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.

NOTE: The synchronize setting in the settings *Reports / Output to pages* always takes precedence over the global setting *Management / General* in the project settings.

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 Reports dialog

To generate graphical reports while editing a project, you use the EPLAN **UTILITIES/REPORTS/GENERATE** menu item to open the **REPORTS** dialog.

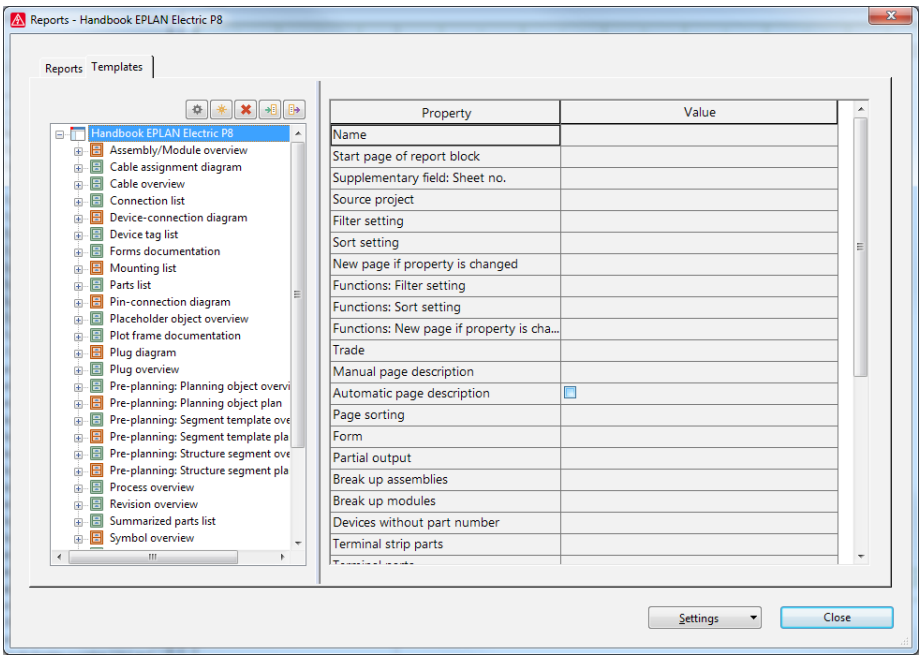


Fig. 6.78 Reports dialog

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.

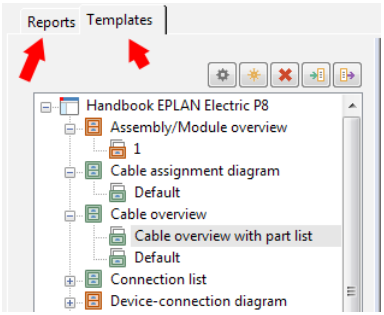


Fig. 6.79
Left area with tabs

Property	Value
Name	Cable overview with part list
Start page of report block	==Example=Chapter++06+6.3.3/10
Supplementary field: Sheet no.	1
Source project	
Filter setting	Cable typ
Sort setting	
New page if property is changed	
Functions: Filter setting	
Functions: Sort setting	
Functions: New page if property is cha...	
Trade	General
Manual page description	
Automatic page description	☑
Page sorting	From settings
Form	HBE_F10_003_FF1

Fig. 6.80 Right area with additional information

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.

The graphical symbols have the following meanings:

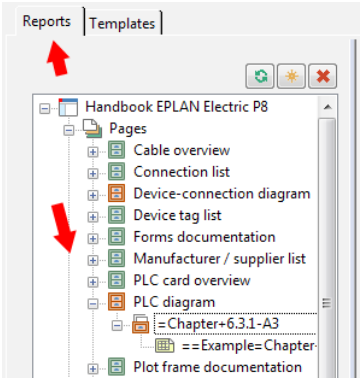


Fig. 6.81
Tree: Pages

Symbol	Node	Description
	Source project (if applicable)	Contains the project that has been entered as the source project for the project or the reports/templates (if applicable)
	(Report) Pages	Node contains generated reports
	Function-related report types	Node includes function-related reports, such as terminal diagrams
	Report type for overview reports	Node includes overview reports, such as a cable overview
	Report blocks (function-related reports)	Node includes reports that belong to a report block of function-related reports
	Report blocks (overview reports)	Node includes all reports that belong to a report block of overview reports
	Report pages/placed reports	Displays a report page or manual placement of an embedded report
	Embedded reports	Node includes all embedded reports that have been generated

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.

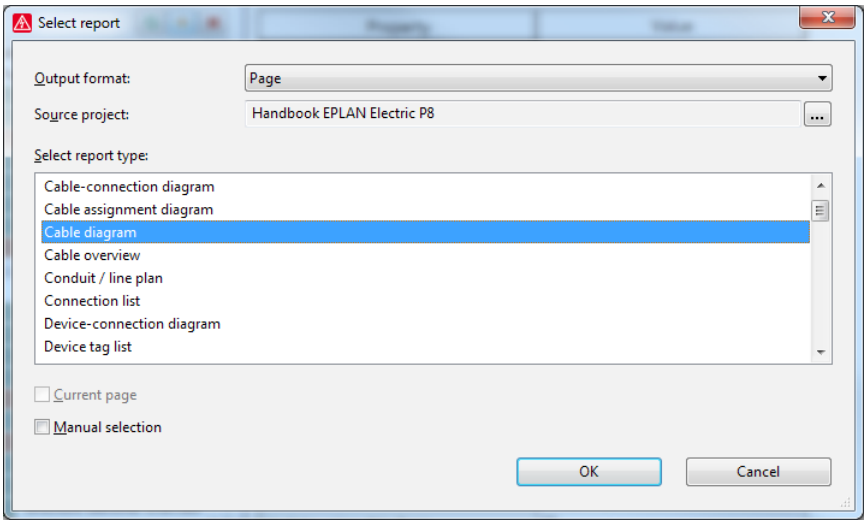


Fig. 6.82 Selecting a report

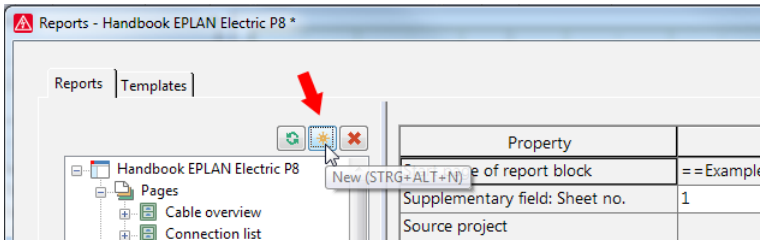


Fig. 6.83 Generating new report directly

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.

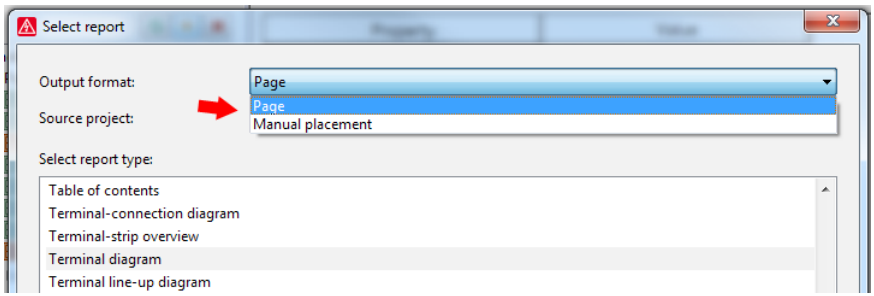


Fig. 6.84
Output format

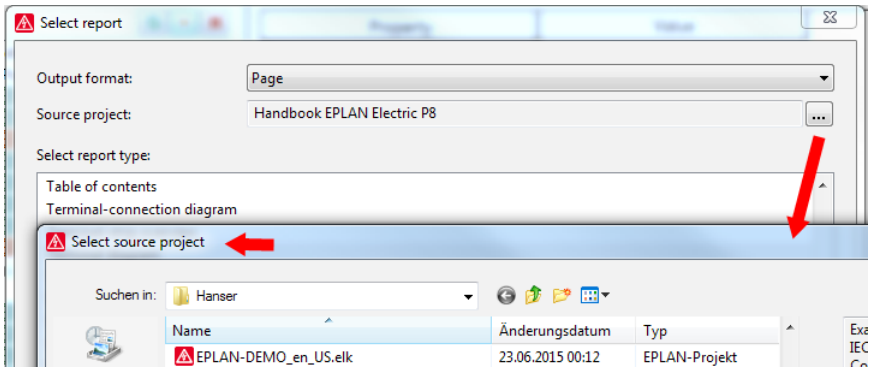


Fig. 6.85
Selecting a source project

There are two available *output formats*:

- Output to pages (new 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

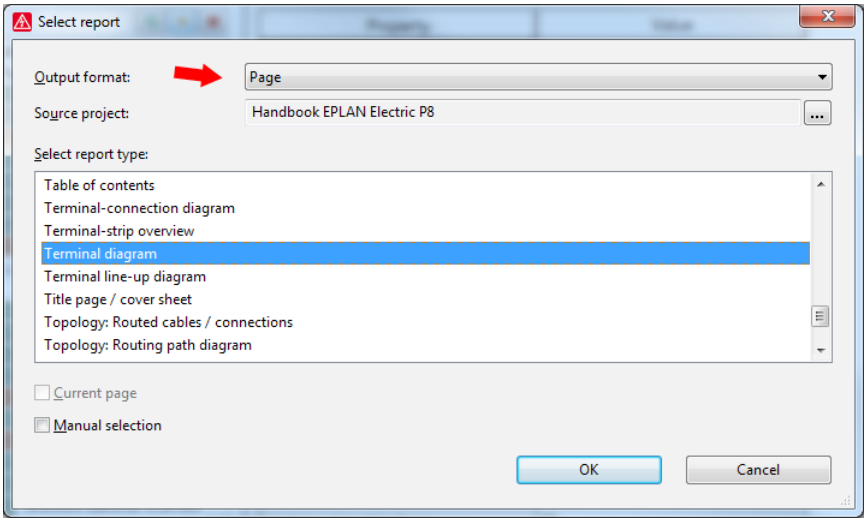


Fig. 6.86 Selecting a report

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).

If EPLAN is to use filters, all you need to do is select a corresponding filter scheme. Then, the filter scheme is automatically active. If the option is grayed out, then no filters can be used with this area.

If you do not want to use filters and/or sorting, you must select the – *Not activated* – scheme.

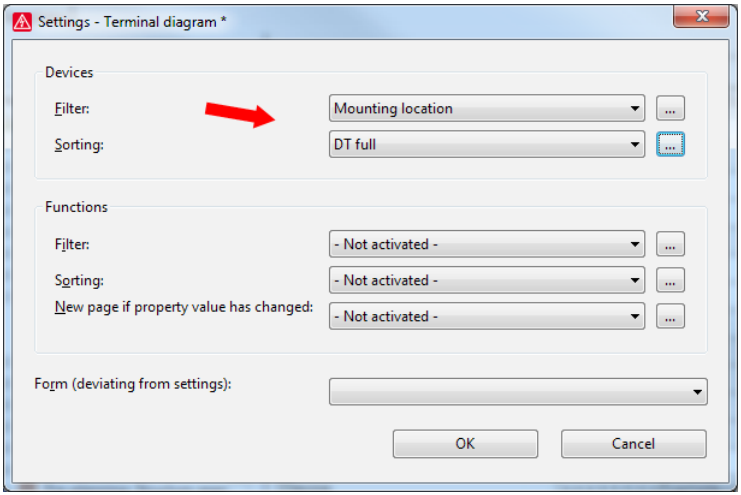


Fig. 6.87 Filter and sorting options for a report

EPLAN then opens the **[REPORT TYPE] (STRUCTURE IDENTIFIER)** dialog. Here you can define default values for sorting, such as the selection of *structure identifiers* (using the navigation in the page navigator) 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.

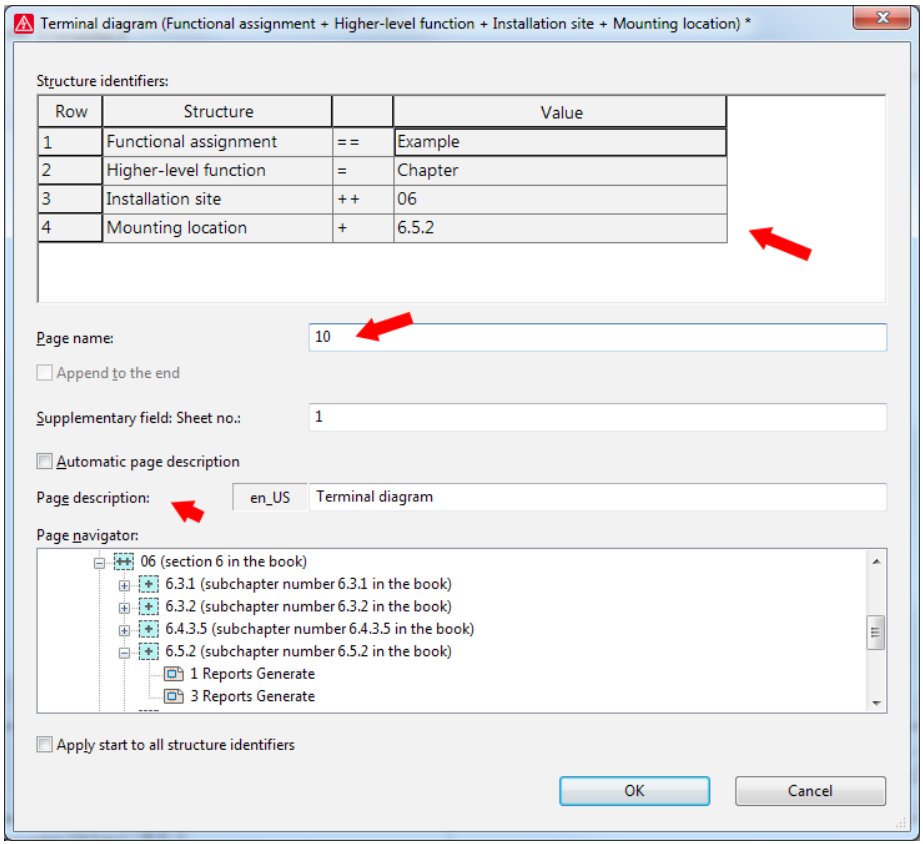


Fig. 6.88 Selecting the output

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.

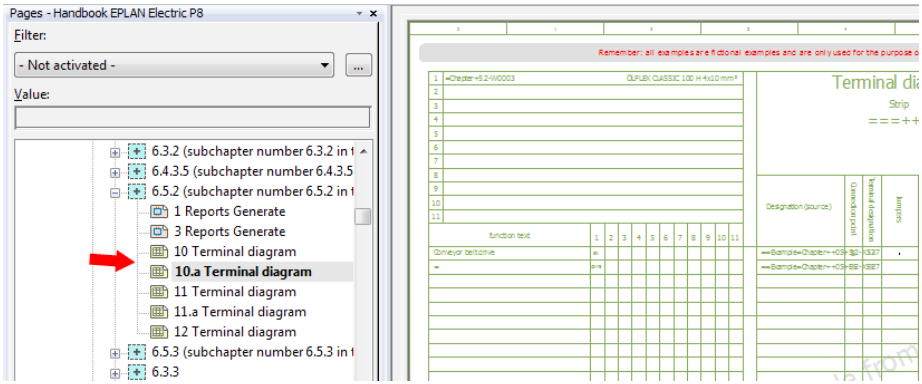


Fig. 6.89 Generated output of terminal diagram report

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.

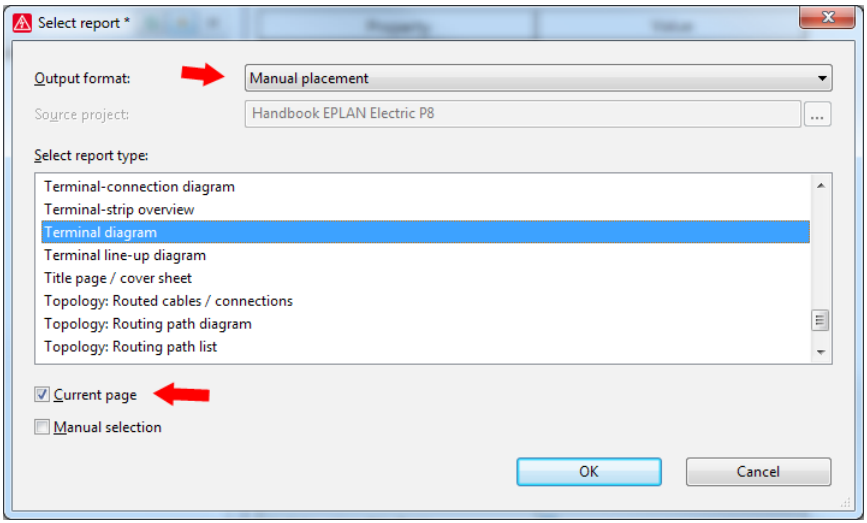


Fig. 6.90 Output options for data on the current page

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.

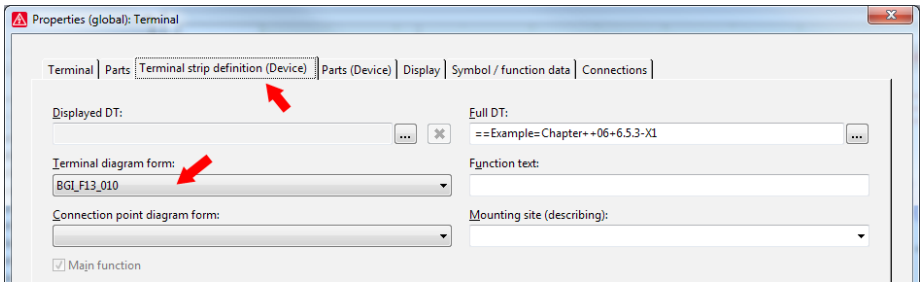


Fig. 6.91 Setting the default form

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.

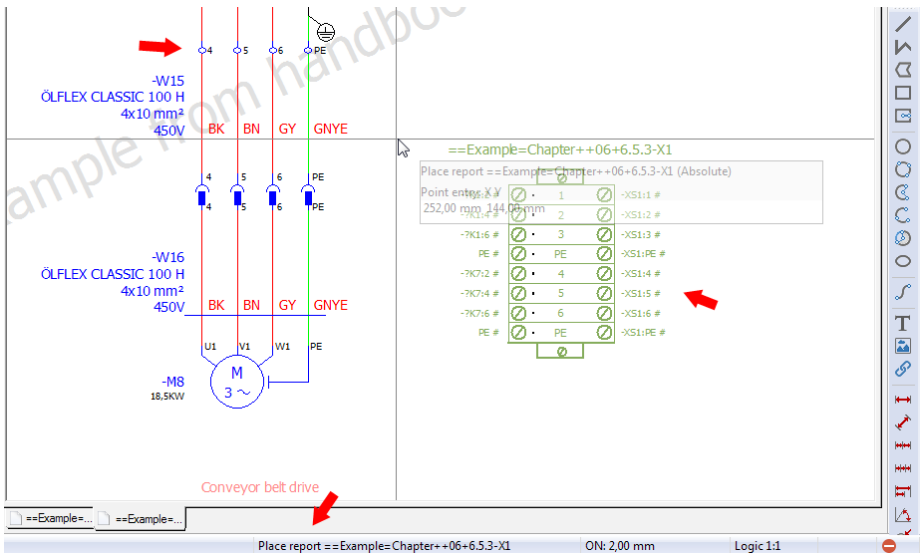


Fig. 6.92 Manually placing the report

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 specific 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.

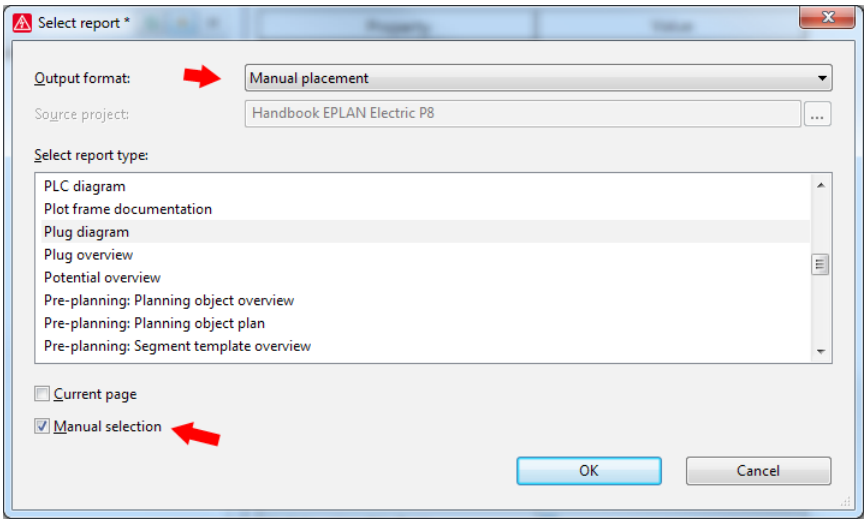


Fig. 6.93 Selecting a report

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**.

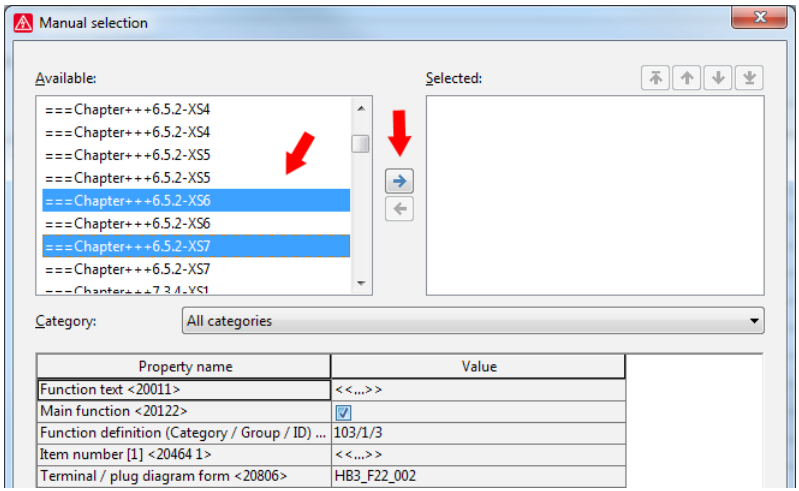


Fig. 6.94 Manual selection of project data

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.

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.

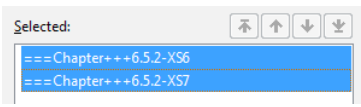


Fig. 6.95 Selected data

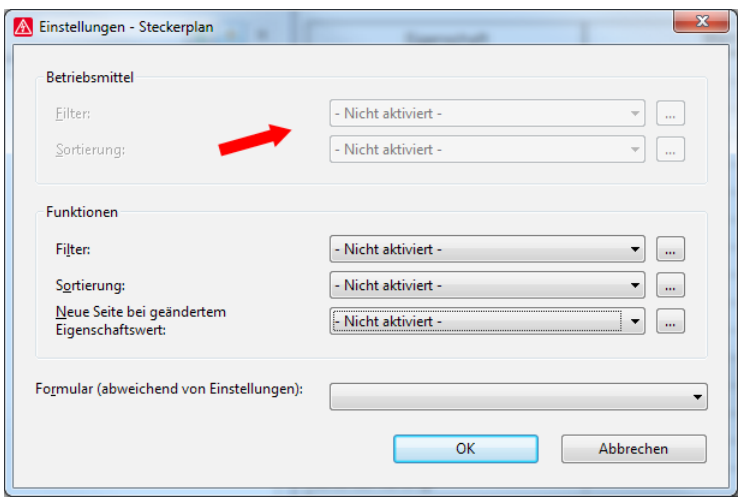


Fig. 6.96 Filter and sorting options for manually selected functions

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).

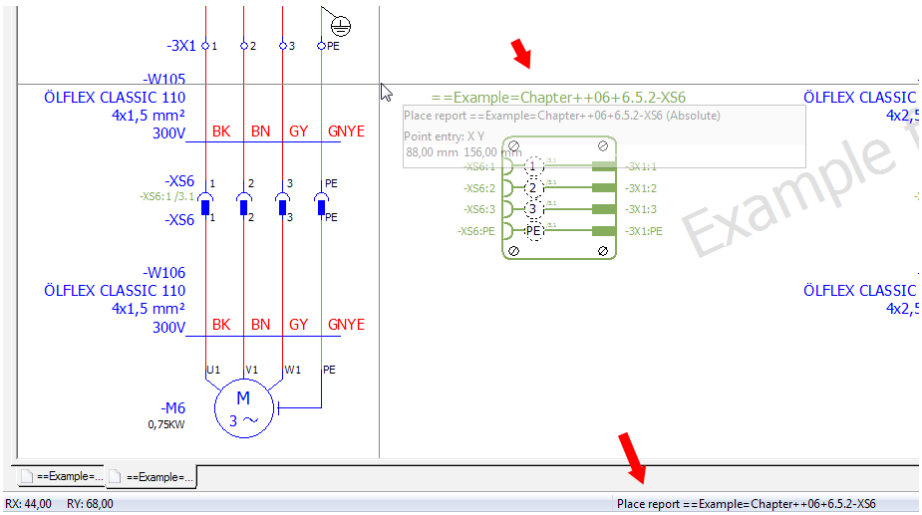


Fig. 6.97 Manually placing the report

When you click **OK**, EPLAN closes all dialogs. The reports hang on the cursor and can now be placed on the active schematic page.

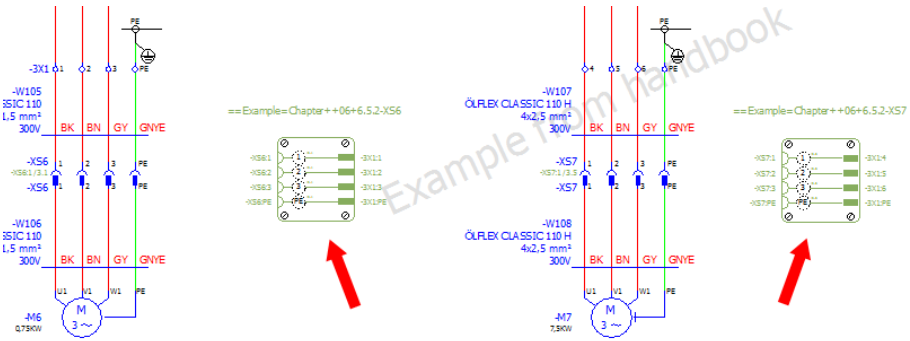


Fig. 6.98 Manually placed reports

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.

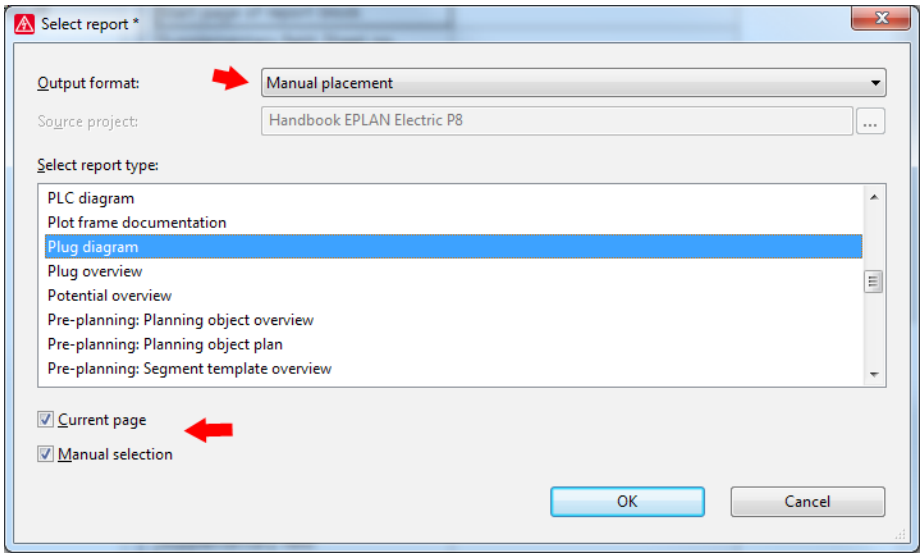


Fig. 6.99 Combining both options for the graphical output of the data

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.

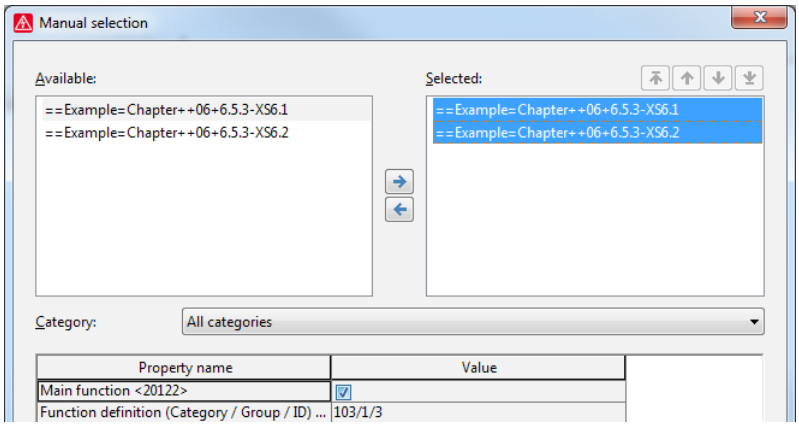


Fig. 6.100 Selecting page data

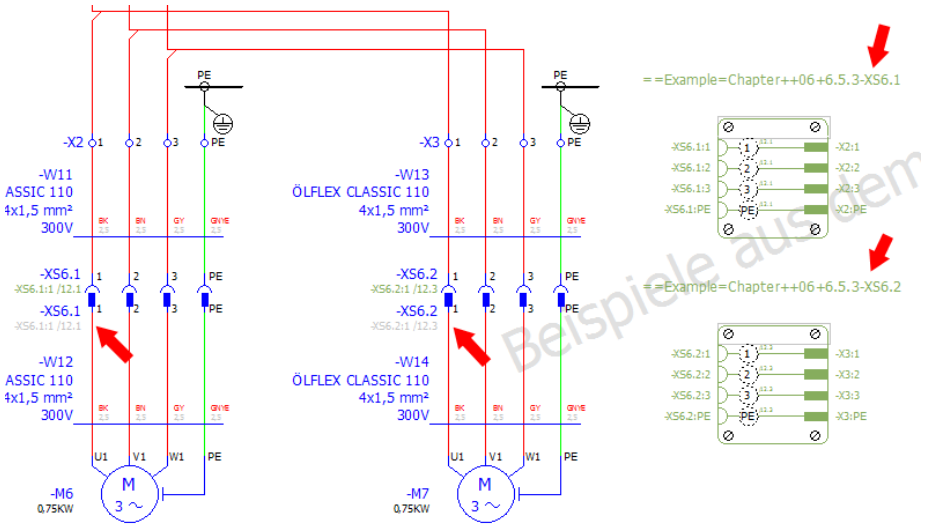


Fig. 6.101 Data has been placed

6.5.3 Popup menus in the Reports tab

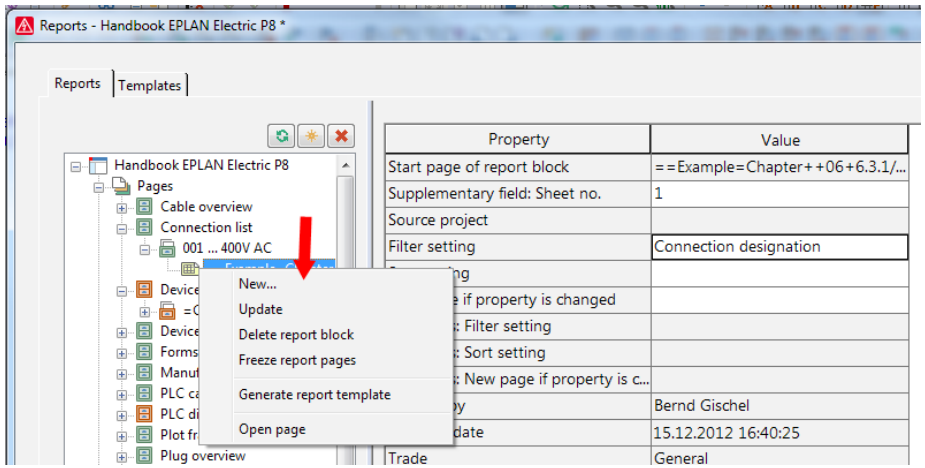


Fig. 6.102 Popup menu in the Reports tab

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 functions can be accessed, for example, 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 to confirm). The menu contains the following functions.

New: Creates a new report.

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 page: 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.

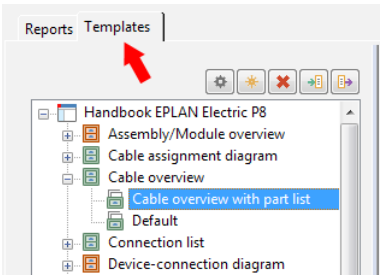


Fig. 6.103
The Templates tab

NOTE: The following restrictions apply to the use of templates to generate reports: The *output form* is fixed in the *Page* selection. A *manual selection* cannot be defined.

And how are templates created? The "how" involves a number of different settings for a template.

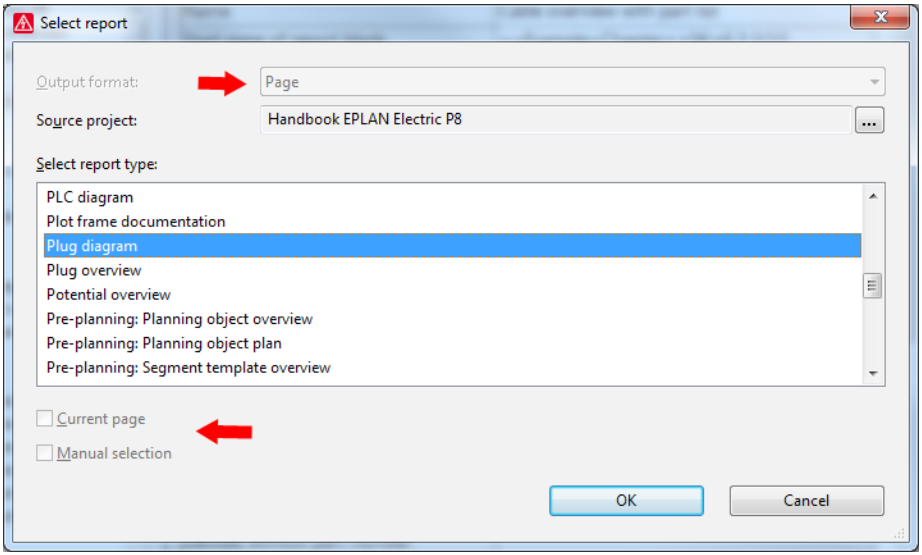


Fig. 6.104 Selecting a report type

To define the report type, you open the **REPORTS** dialog via the **UTILITIES/REPORT/GENERATE** menu and switch to the *Templates* tab. Use the **NEW** button to start the **SELECT REPORT** dialog. You now select the desired *report type* here. Confirm your selection with **OK**.

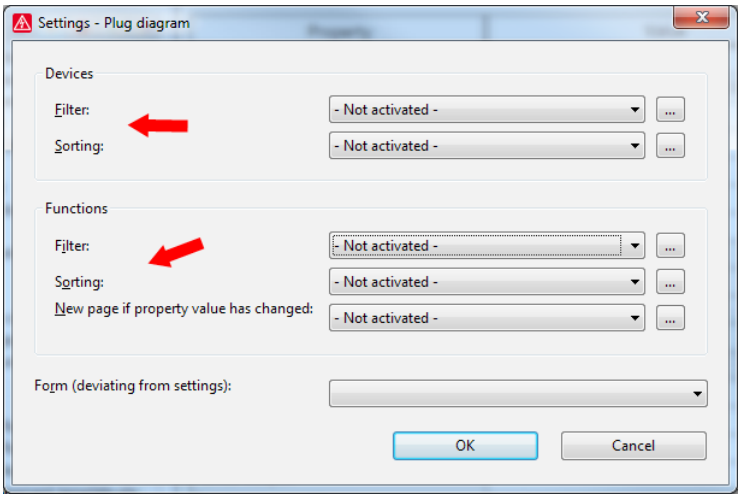


Fig. 6.105 The Filter/sorting dialog

As usual in EPLAN, you can use predefined filter and sorting options, or set up your own, in the **SETTINGS [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 **SETTINGS** dialog may look different, or a few of the options might not be displayed at all (grayed out).

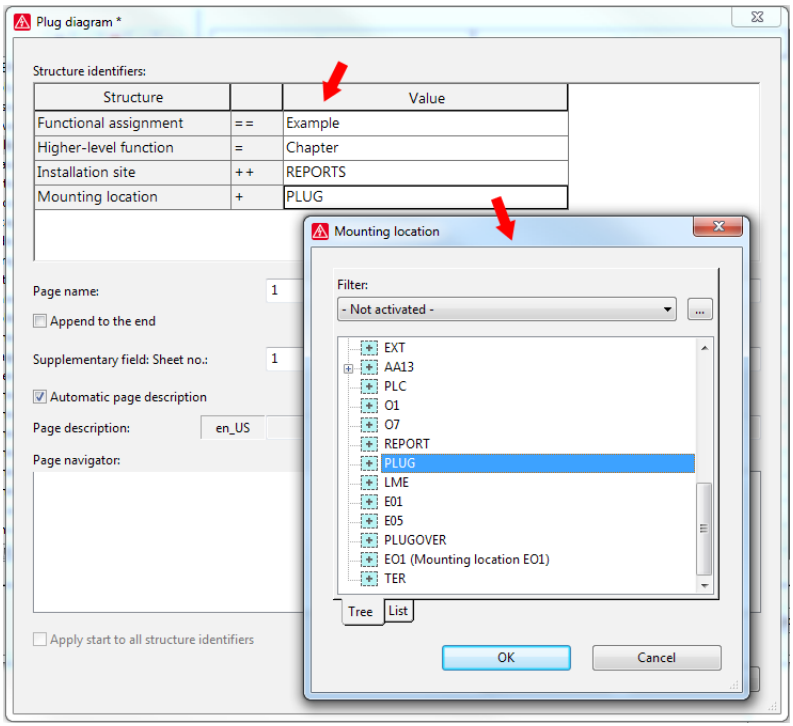


Fig. 6.106 Setting the structure of report pages

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.

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 initial template is given the default name 1. The second would be number 2, and so on.

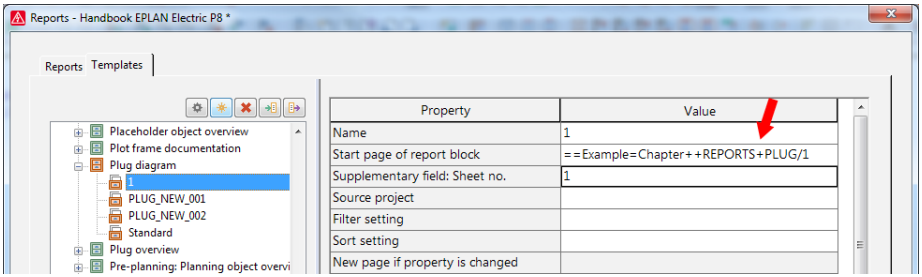


Fig. 6.107 Template addition complete

NOTE: If filters were selected previously, EPLAN assigns the filter's name to the template.

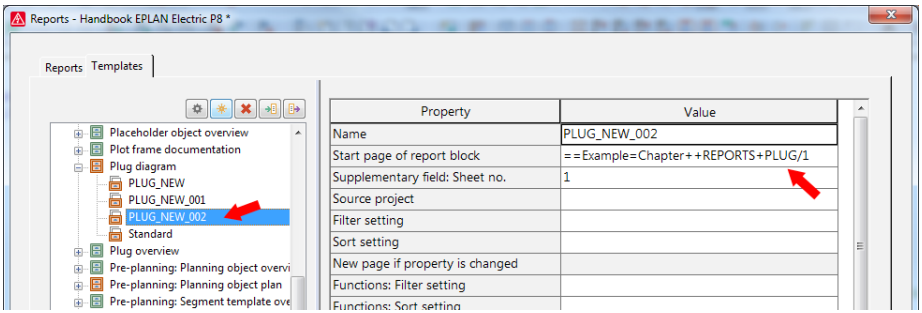


Fig. 6.108 Template name changed

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.

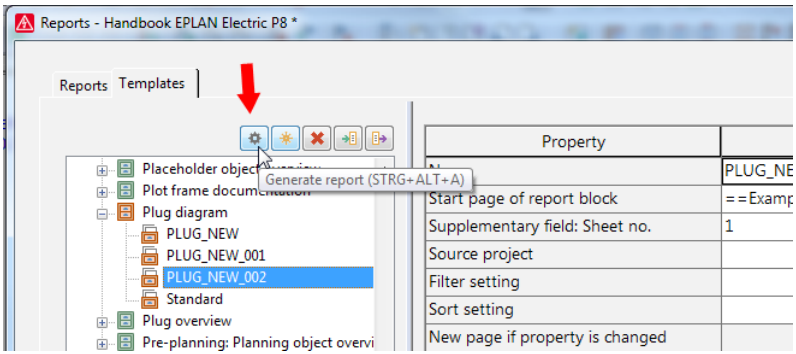


Fig. 6.109 Generate this template report

This completes the generation of a template. If you now select this template and click the 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 (some) properties.

Other properties of the Templates tab

Property	Value
Name	Full edition
Start page of report block	==Example=Chapter++REPORTS+6
Supplementary field: Sheet no.	1
Source project	
Filter setting	
Sort setting	
New page if property is changed	
Functions: Filter setting	
Functions: Sort setting	
Functions: New page if property is cha...	
Trade	General
Manual page description	
Automatic page description	☑
Page sorting	From settings
Form	HB4_F01_005
Partial output	HB3_F01_004

Fig. 6.110 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.

Source project: Displays the name of the source project that has (may have) been specified in the **SELECT REPORT** dialog in the *Source project* field. This field allows for different source project to be selected via the More button.

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 changes. This is not to be confused with breaks in the case of changes to functions.

Functions: Filter/Sort setting and *New page if property is changed*: These filters operate directly on the device functions, e. g. on a terminal.

Trade: Selection of and/or change to the trade. Generally, this property does not have to be changed, because EPLAN selects the correct one automatically.

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 filled with the **From settings** entry, because the basic values have already been set in the default settings (**OUTPUT TO PAGES SETTING**). However, this value can still be changed.

Automatic page description	☑
Page sorting	From settings
Form	HB4_F01_005
Partial output	HB3_F01_004
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
Cable connection part	From settings
Cable project parts	From settings
Connection parts	From settings
Plug parts	From settings
Pin parts	From settings
Busbar parts	From settings
Busbar connection point parts	From settings
Pre-planning part	From settings
Project filter	FACTORY_PRINT_OK
Template active	☑
Action	

Fig. 6.111 Further properties for report output

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.

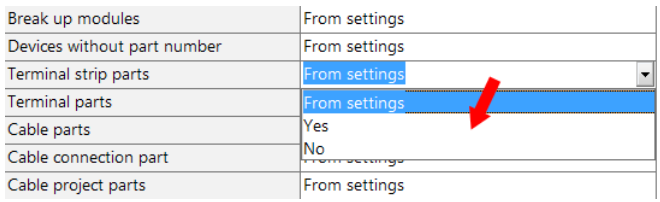


Fig. 6.112 Properties that affect parts output

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	☑
Action	

Fig. 6.113 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.

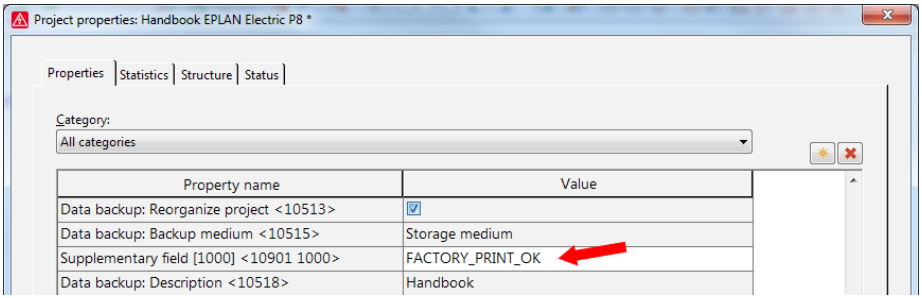


Fig. 6.114 Selecting a project property and defining its value

This means that if a defined project property's value matches the filter value, a report will be generated.

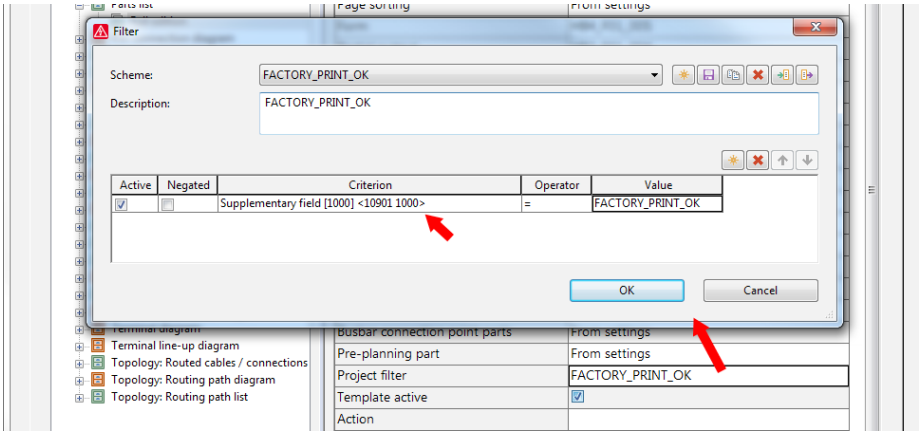


Fig. 6.115 Selecting a filter for the desired project property and its value

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 condi-

tion 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

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.

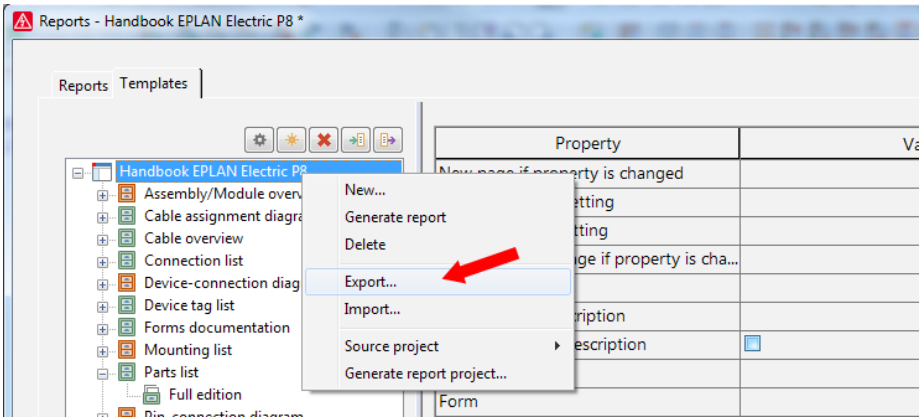


Fig. 6.116 Export/Import templates

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 (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

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.

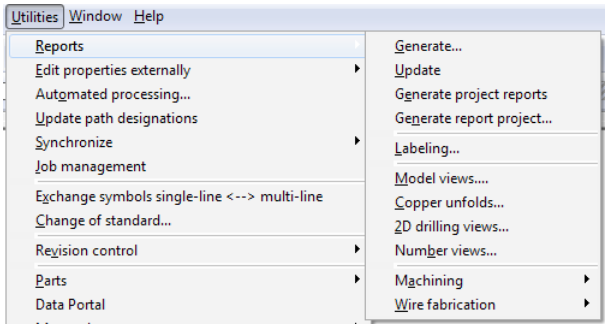


Fig. 6.117 Generate project (templates) reports

6.6.1 Update

An additional function that EPLAN offers is the *Update* function for *existing* reports. This function is also available in the **UTILITIES/REPORTS** menu via the button. In contrast to the *Generate project reports* function, the *Update* function always only affects the currently open report(s) 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 selected report is updated. If the update function is used on a schematic page, for example, EPLAN will display a prompt.

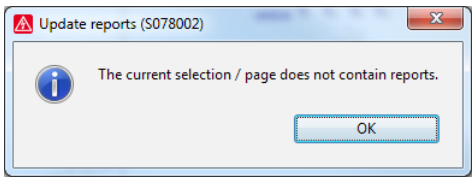


Fig. 6.118 Updating reports – no report selected

6.6.2 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.

6.6.3 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*.

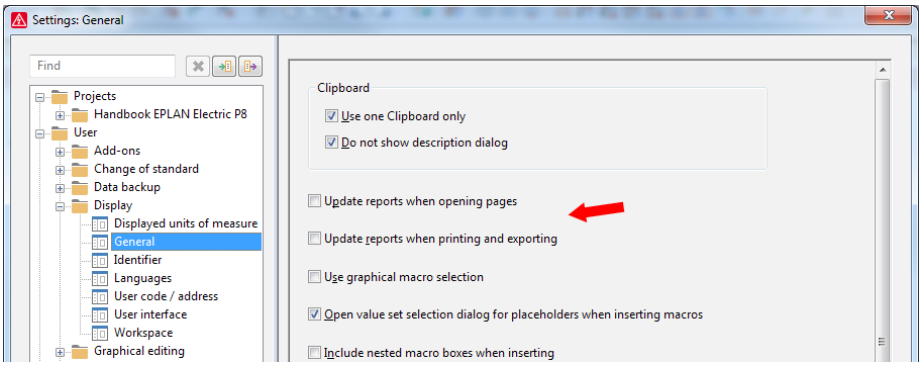


Fig. 6.119 Settings for automatically updating reports

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 and cannot be displayed any time sooner.

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.

6.7 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.

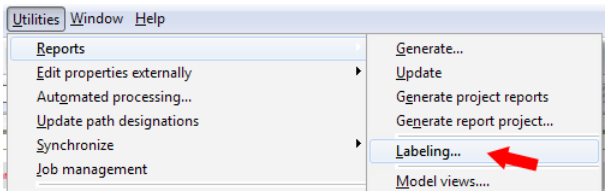


Fig. 6.120 Labeling

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.

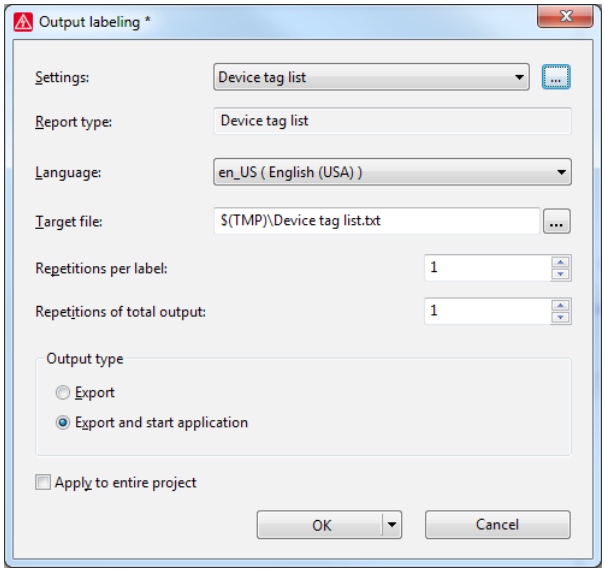


Fig. 6.121 Output labeling

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.

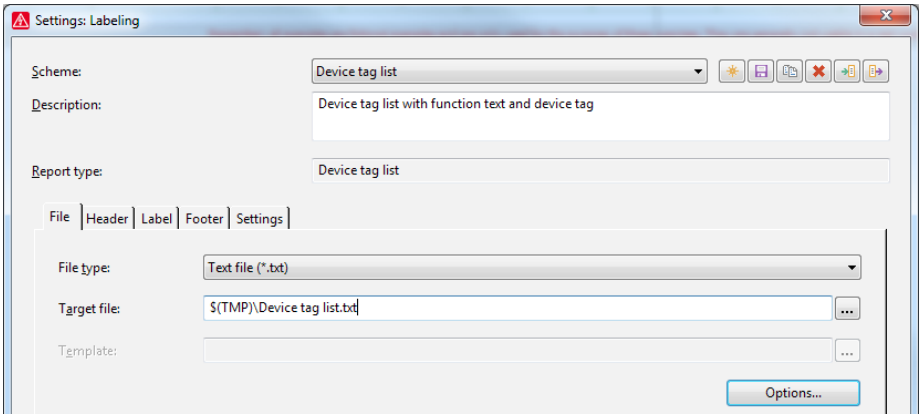


Fig. 6.122 The Labeling dialog with the scheme and the settings

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

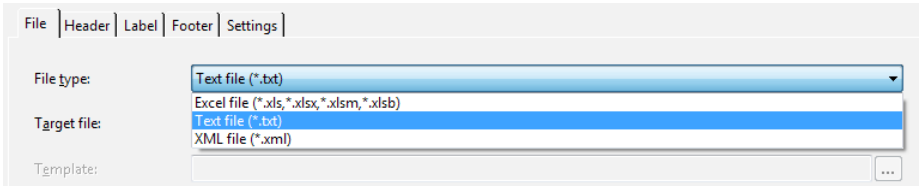


Fig. 6.123 Selection of the file type

*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.

*The 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.

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.

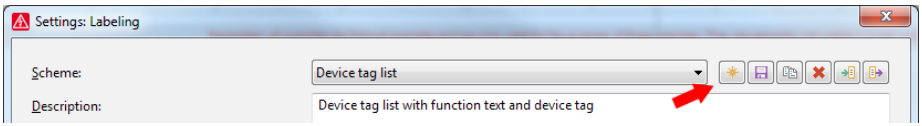


Fig. 6.124 Toolbar for editing the selected scheme

6.7.1.2 Labeling as text output

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.

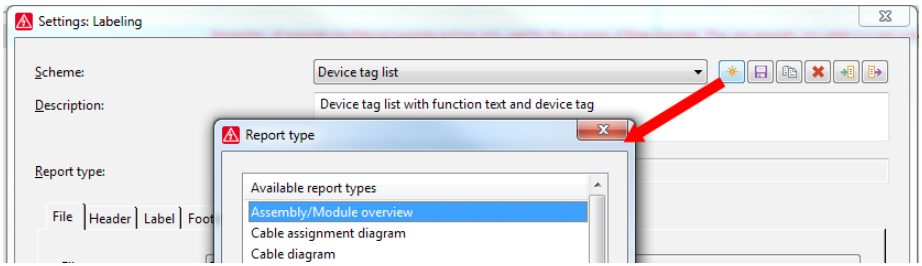


Fig. 6.125 Select report type

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.

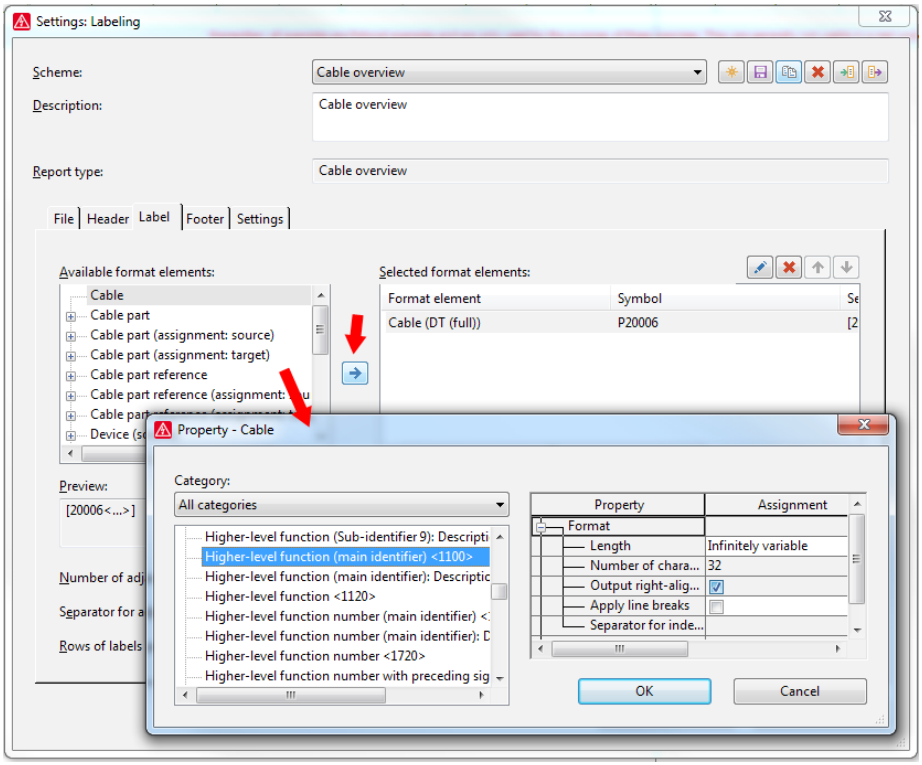


Fig. 6.126 Selecting desired properties

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** dialog. All available cable properties are listed here.

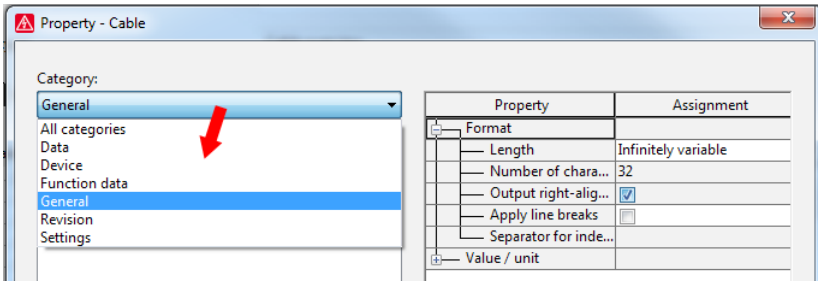


Fig. 6.127 Categories

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**.

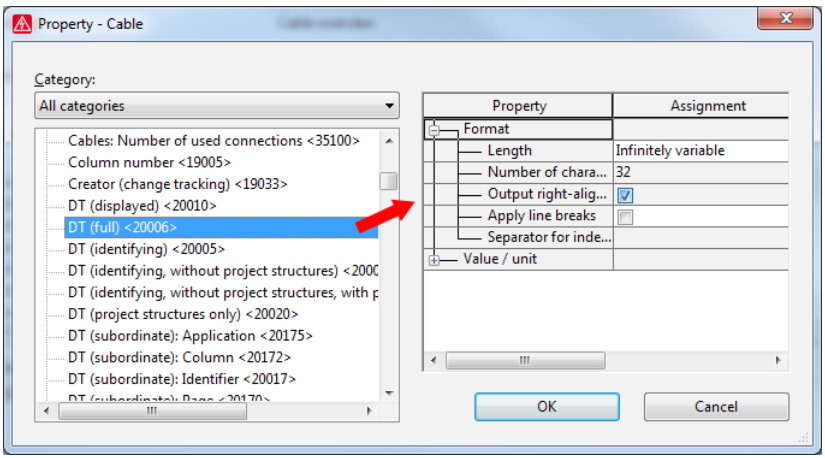


Fig. 6.128 Possible assignments for selected properties

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.

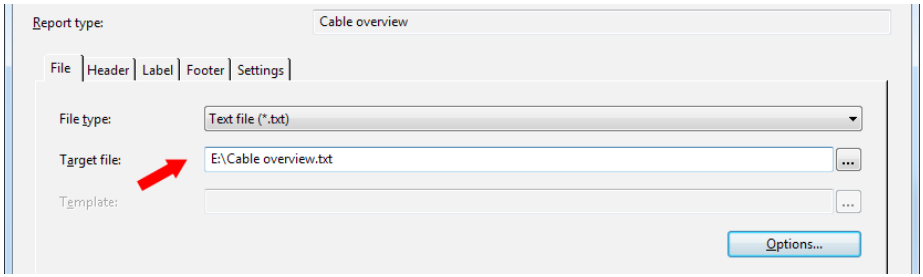


Fig. 6.129 Select target file

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. 6.130 (opened in a text editor).

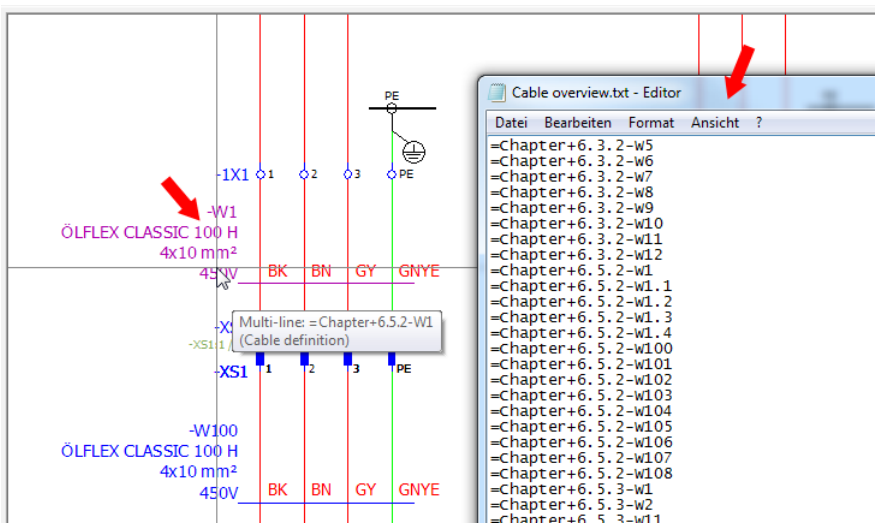


Fig. 6.130 Generated text file with information

6.7.1.3 Labeling as Excel file

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.

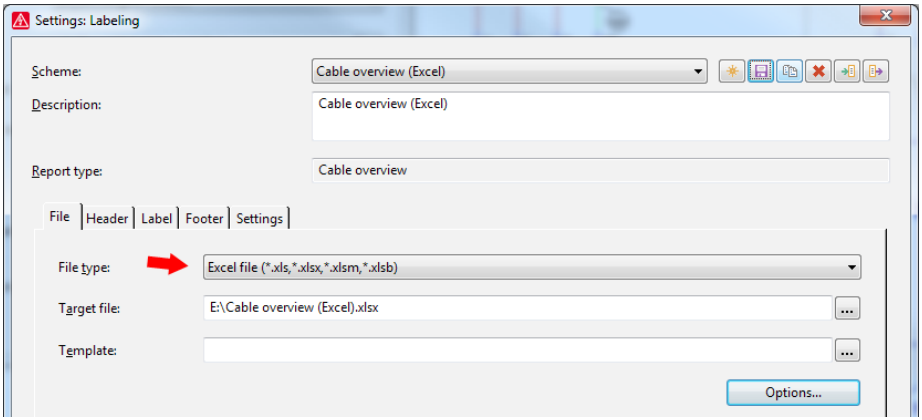


Fig. 6.131 Scheme for output to Excel

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.

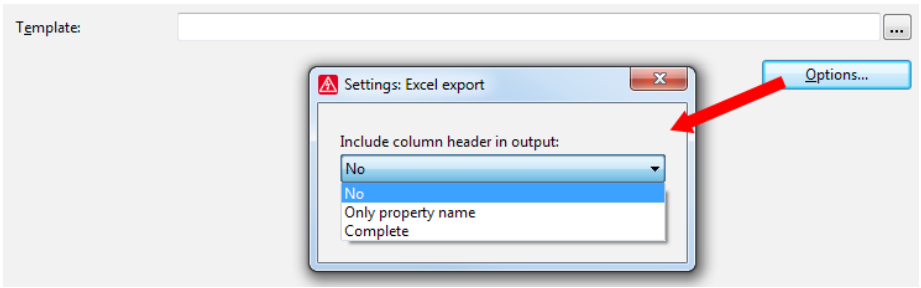


Fig. 6.132 Options button

The **OPTIONS** button on the *File* tab is important when outputting to Excel. EPLAN offers three settings here to *include column header in output.*

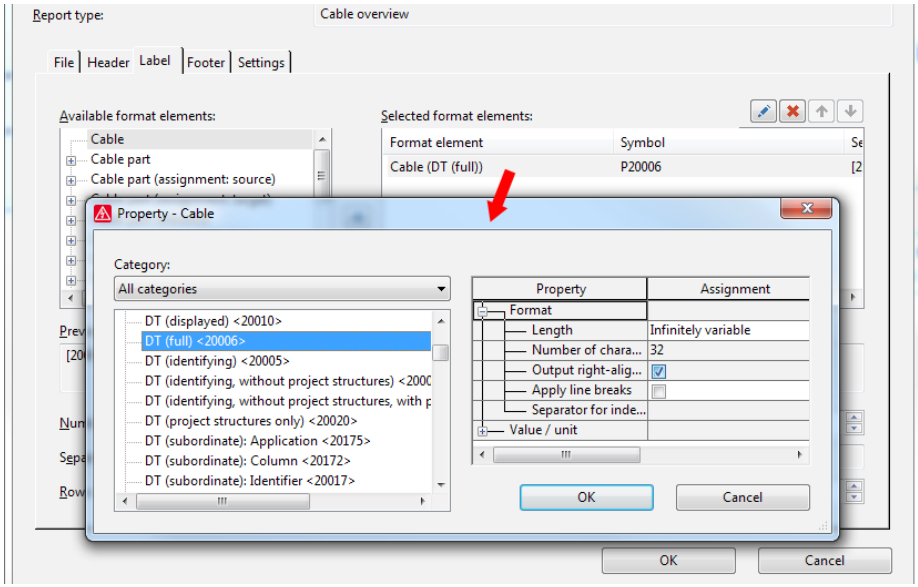


Fig. 6.133 Example of properties for column labeling

No: No output of column designation, and thus no output of column header.

Only property name: Only the property name is output (e. g. (full) DT).

Full: The full string (property name including other data) is generated as column headers (e. g. cable/(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.

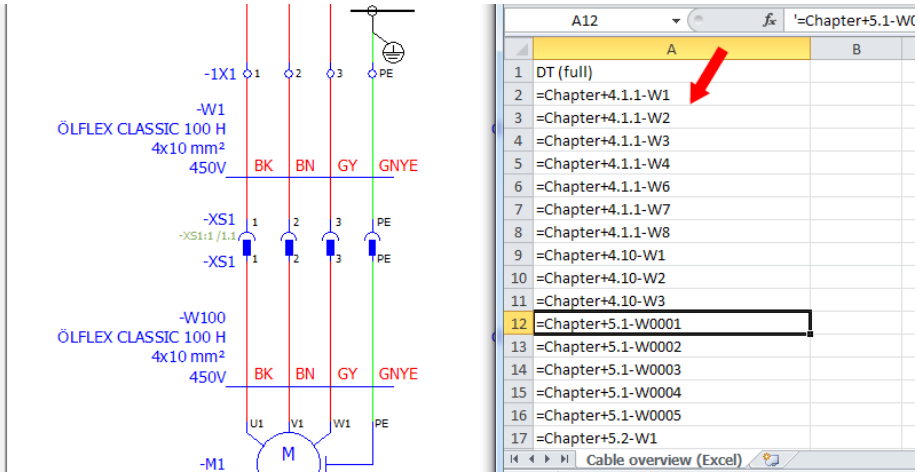


Fig. 6.134 Generated Excel file with the data from EPLAN

6.8 Edit properties externally

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.

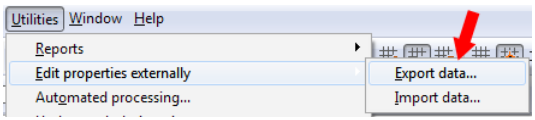


Fig. 6.135 Edit properties externally menu

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.

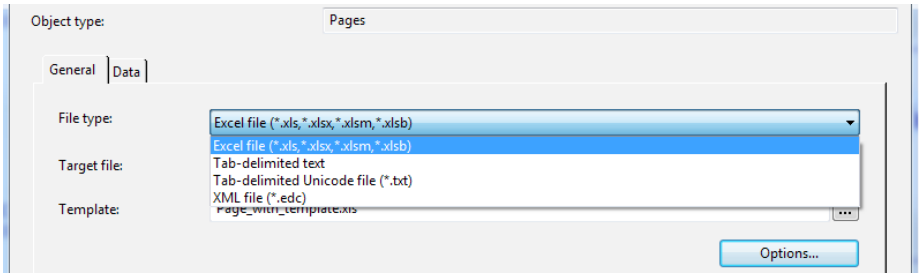


Fig. 6.136 File types for external editing

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

EPLAN allows for the transferring of project data to Excel. This relates to the following project data (extract):

NOTE: Generally, most properties are suitable for export. But not all properties can also be edited externally. This should be kept in mind.

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

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. 6.137.

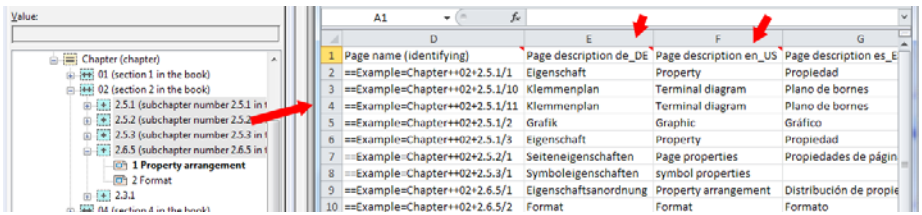


Fig. 6.137 Exported data without Excel template

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. 6.138.

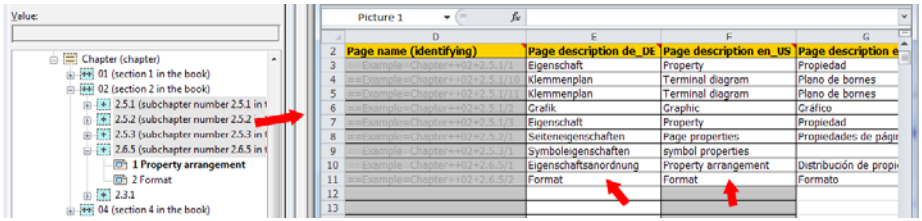


Fig. 6.138 Exported data with Excel template

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.

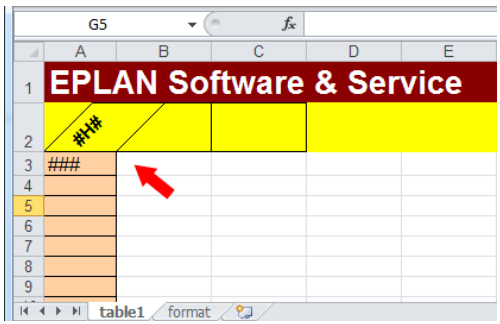


Fig. 6.139
Worksheet table 1

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.

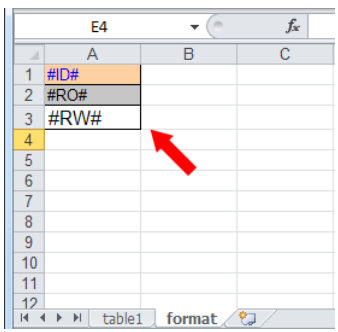


Fig. 6.140
The Format worksheet

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.

- **#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/EXPORT DATA** dialog. EPLAN opens the **EDIT EXTERNALLY** dialog with the scheme most recently used.

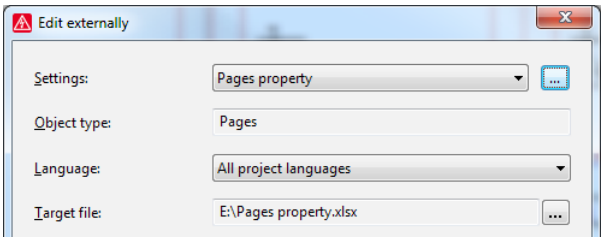


Fig. 6.141 Edit externally dialog

Use the More button to open the **SETTINGS/EXTERNAL EDITING** dialog.

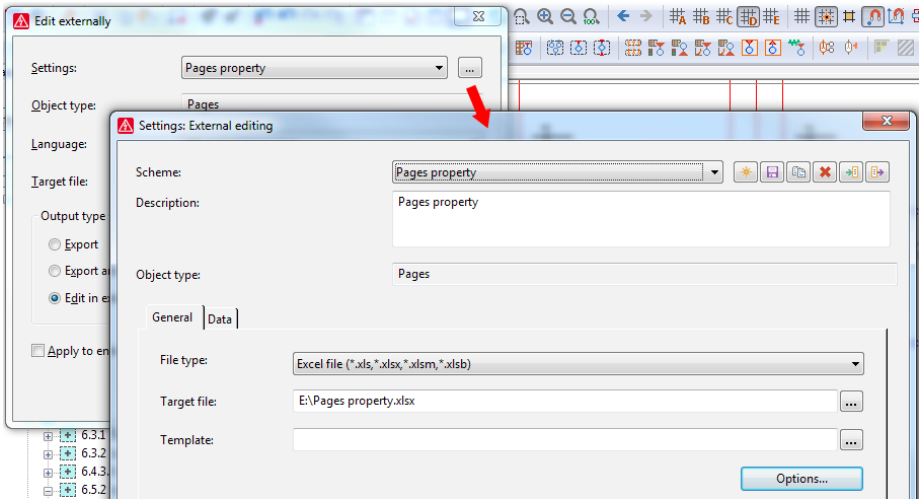


Fig. 6.142 Settings External editing dialog

Click the button to create a new scheme. This opens the **OBJECT TYPE** dialog. The desired type (e. g. pages) is selected and applied by clicking **OK**.

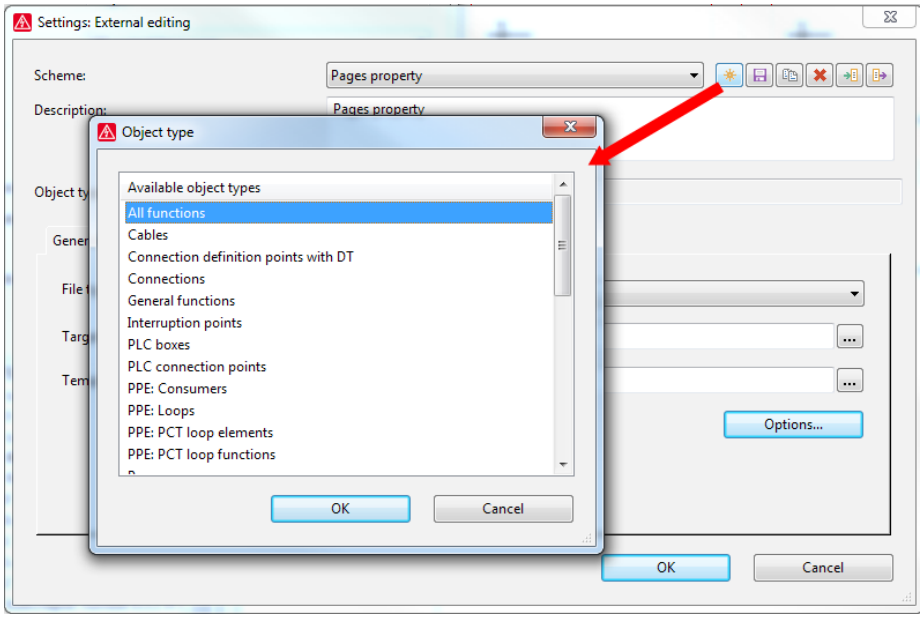


Fig. 6.143 Object type dialog

The **NEW SCHEME** dialog opens.

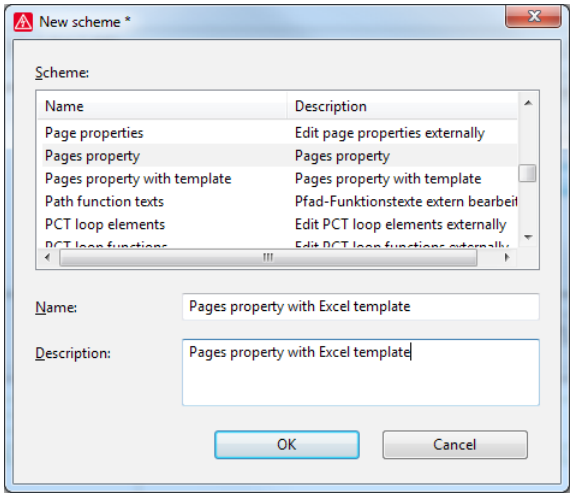


Fig. 6.144 New scheme dialog

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.

Settings for transferring data to Excel

In the *File type* selection field, you select the Excel file (**.xlsx*) 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.

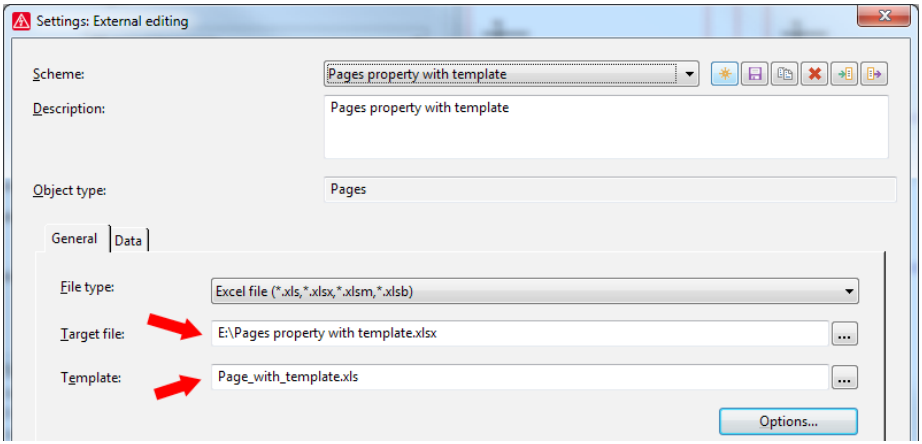


Fig. 6.145 Settings in the General tab

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. 6.146 for this example, a number of properties have been added to the *Selected format elements* field.

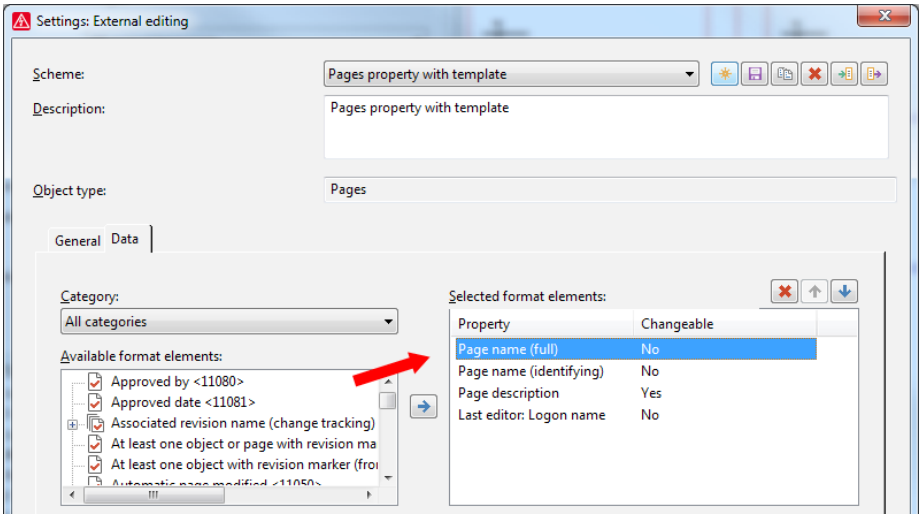


Fig. 6.146 Settings in the Data tab

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.

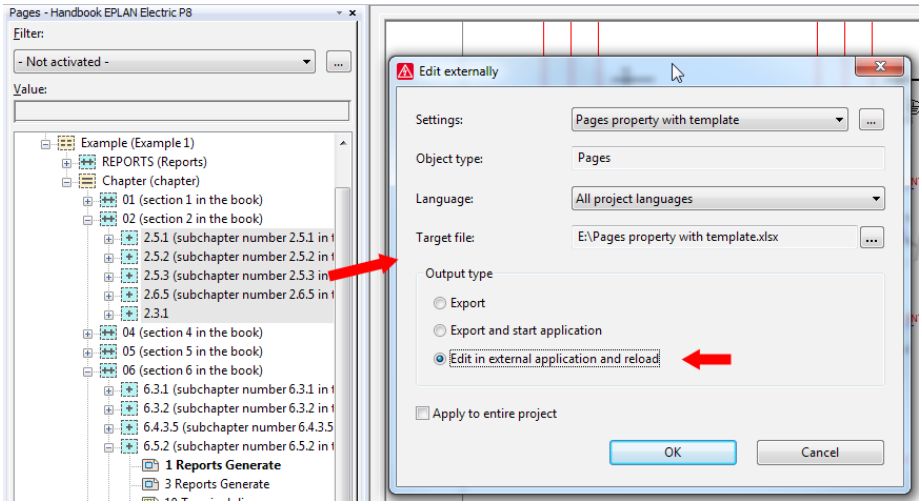


Fig. 6.147 Selected pages

Then, use the **UTILITIES/EDIT PROPERTIES EXTERNALLY/EXPORT DATA** menu to select and/or call the desired scheme. 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.

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 DATA**. 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.

Clicking the **OK** button generates the output file and then starts Excel. The loaded data can now be edited or changed.

After making all your changes, you save the Excel file (click the button once or press **CTRL + S**) and can now close Excel.

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.

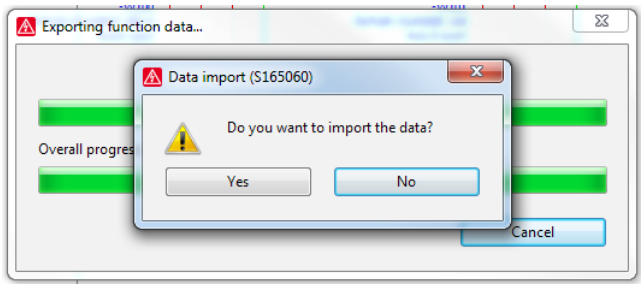


Fig. 6.148 Data import dialog

NOTE: It is not possible to undo this action (importing changed data).

6.8.2 Import data

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 DATA 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.

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 Electric P8 installed.

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.

Open the **UTILITIES/EDIT PROPERTIES EXTERNALLY/IMPORT DATA** 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.

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 error checking. You need to organize and manage parts data and the various layers of a project as well. This is why I use the general term “Management”.

7.1 Structure identifier management

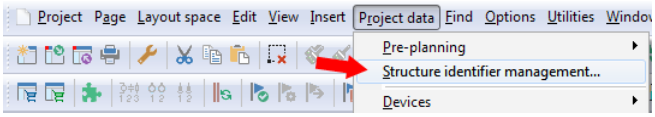


Fig. 7.1 Structure identifier management

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.

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.

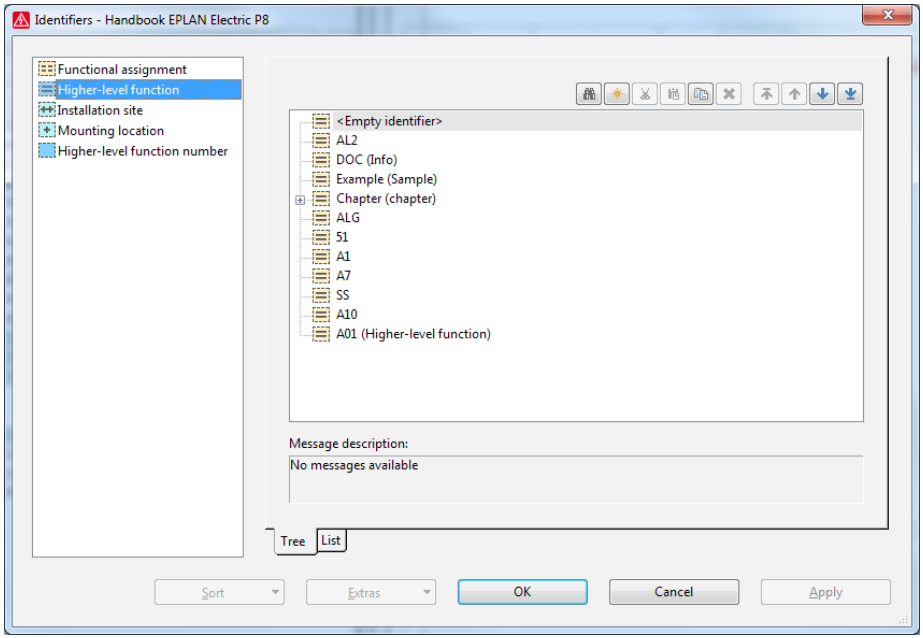


Fig. 7.2
Control center of structure identifier management (tree view)

The dialog consists essentially of two familiar views: the tree and list views. The following sections will focus primarily on the list view.

The tree view provides an easy and 'quick' overview of the identifiers entered, while the list view offers more options.

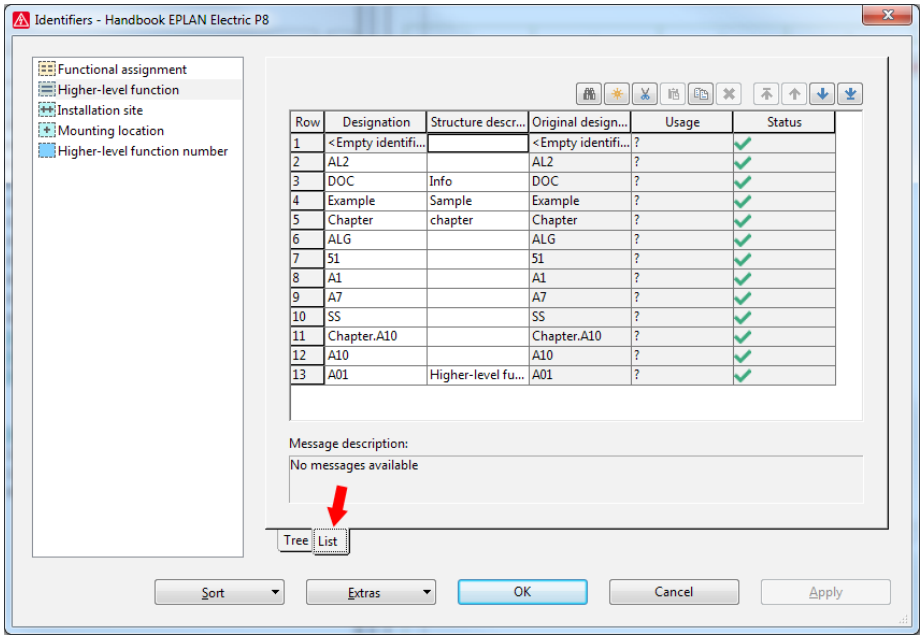


Fig. 7.3
Identifiers dialog in the list view

The possible structure identifiers of a project 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 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).

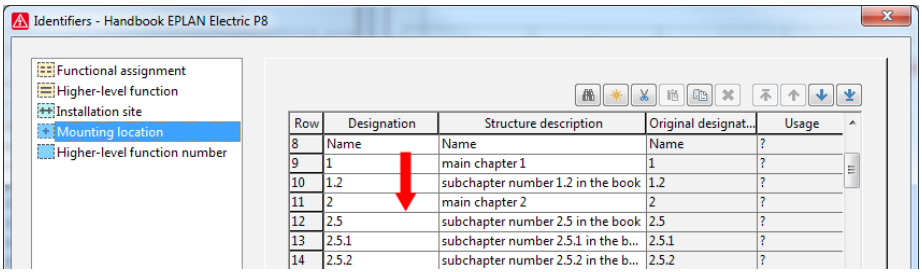


Fig. 7.4 Sequence of the mounting location

The page structure in the page navigator is now sorted according to the defaults in structure identifier management (Fig. 7.5).

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.

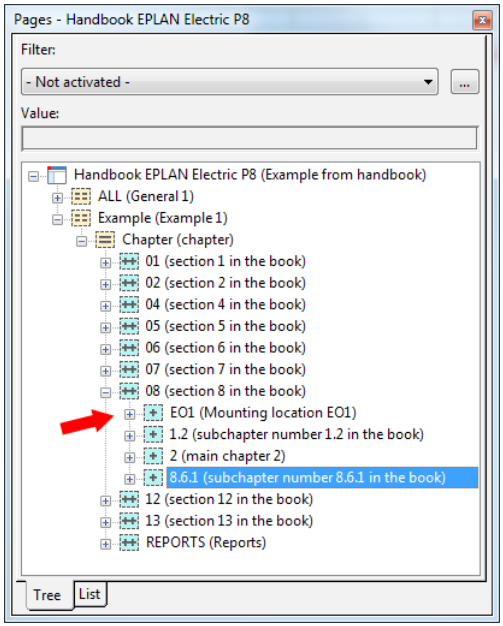


Fig. 7.5 Sequence in the page navigator

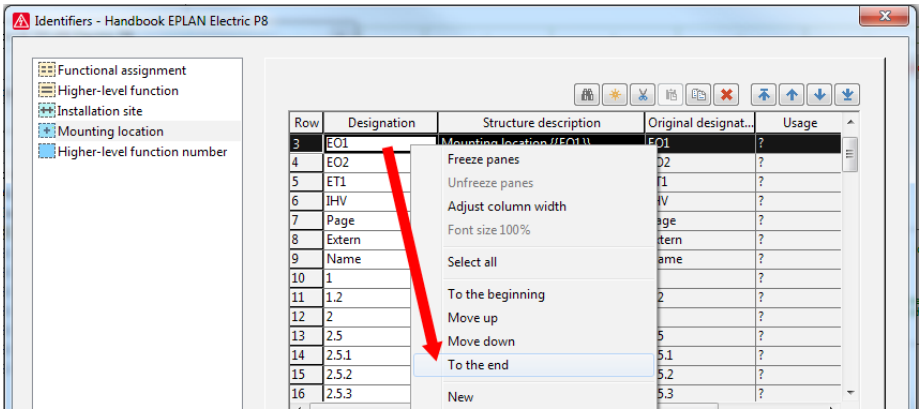


Fig. 7.6
Changing the sequence of structure identifiers

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.

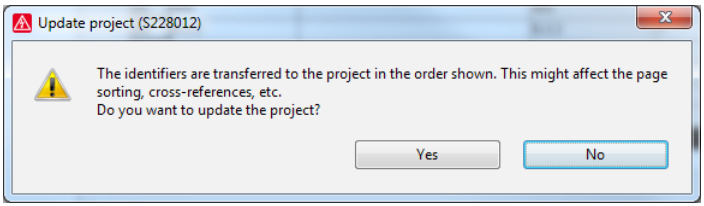


Fig. 7.7
Apply project data changes dialog

When you confirm, the changes are applied and the page structure is adjusted in the page navigator.

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.

Up to three detailed descriptions can be entered for each 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.

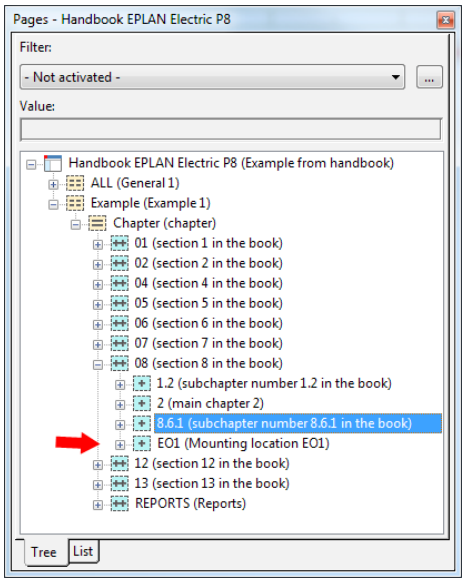


Fig. 7.8
Changed sequence

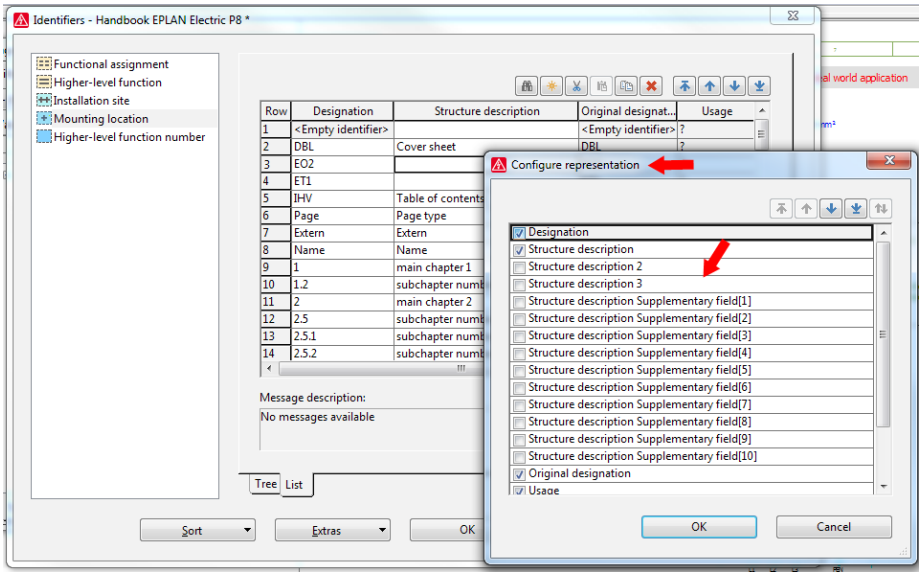


Fig. 7.9 Further descriptive options (properties)

7.1.1 List of identifiers in structure identifier management

The list of identifiers (left side in the **IDENTIFIERS** dialog) corresponds to the page structure entered, which was selected when the project was created.

NOTE: The list of identifiers has the same structure and is used in the same way. Therefore only one identifier in the list and its functions will be explained.

EPLAN can manage up to seven structure identifiers in one project, the so-called identifier blocks. The following structure identifiers are possible:

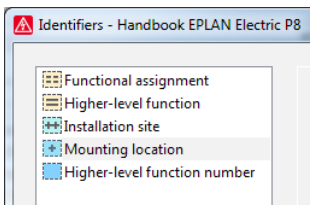


Fig. 7.10 List of identifiers in structure identifier management

- **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 'control panel' as a mounting location. 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**.

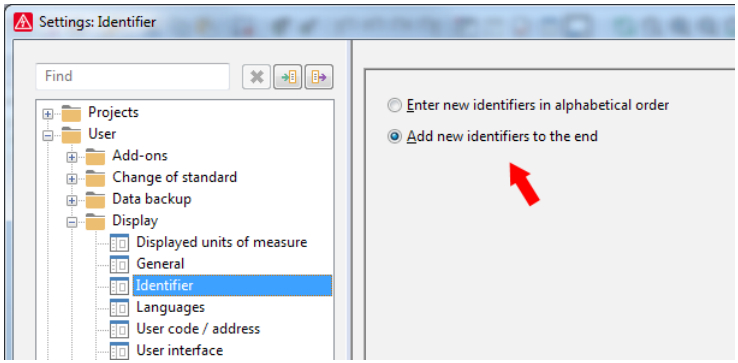


Fig. 7.11 Settings for Place identifiers

In order to assign **Descriptions** to these automatically sorted identifiers, structure identifier management must be started and the descriptions entered later.

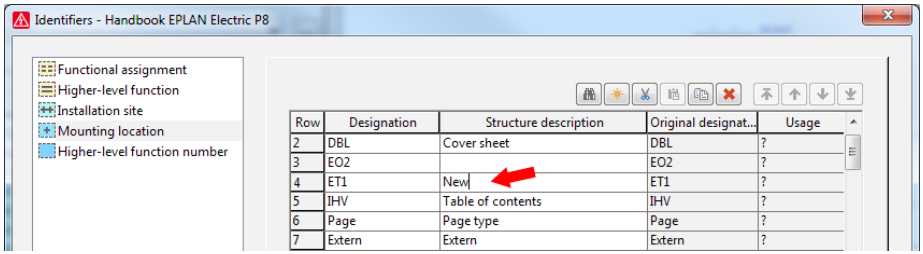


Fig. 7.12 Entering structure designations

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 **CONFIGURE REPRESENTATION**. EPLAN then opens the **CONFIGURE REPRESENTATION** dialog. In this dialog, you can select or deselect the appropriate properties for display.

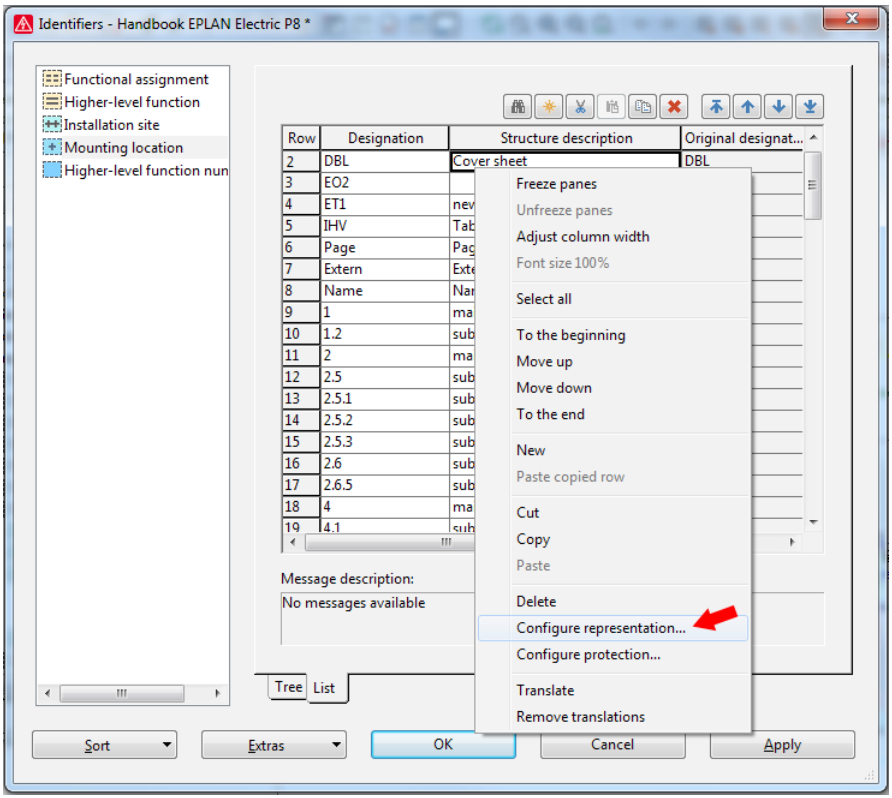


Fig. 7.13 Identifiers – Configure representation dialog

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) rows 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

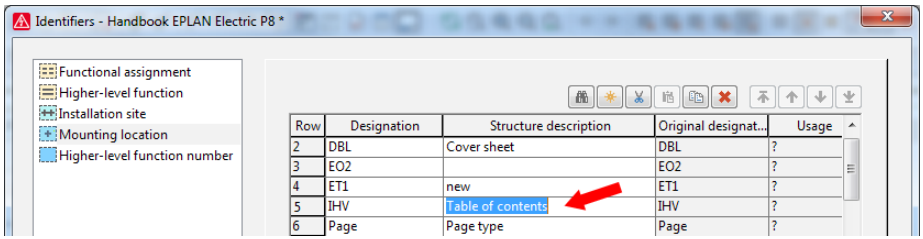


Fig. 7.14 Direct editing using F2

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 directly.

Either option opens up different right-click popup menus. But in either popup menu you can use similar functions, such as *Copy*, *Delete* or *Translate*.

Single selection

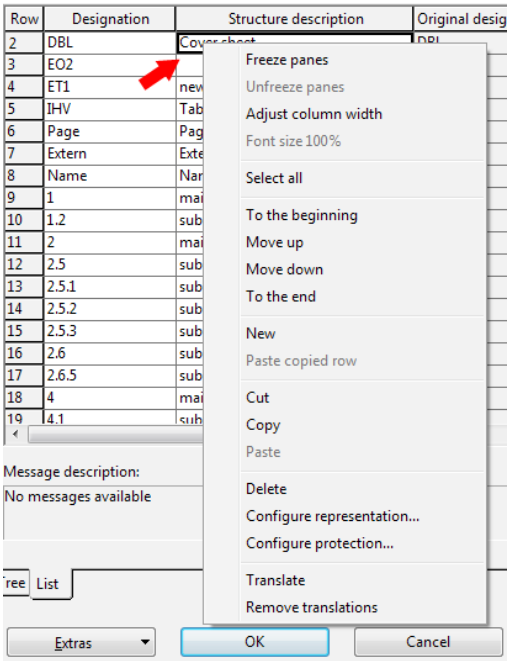


Fig. 7.15 Cell selected

Direct selection

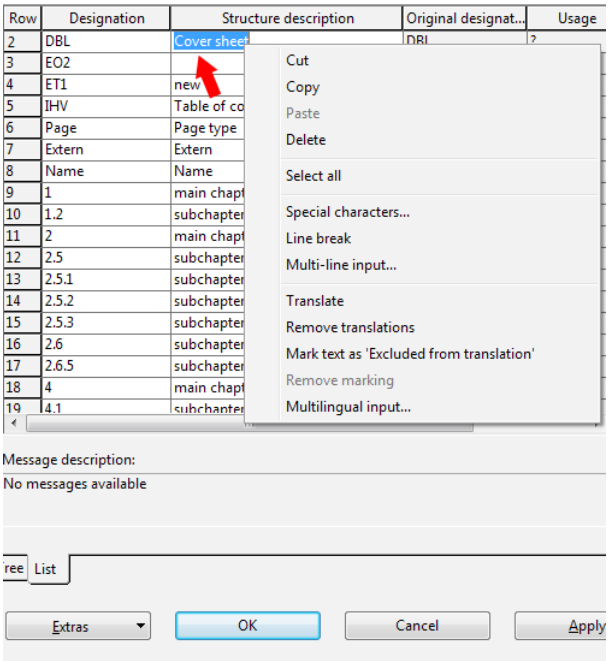


Fig. 7.16 Cell selected with F2

To translate column entries, for example, 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**.

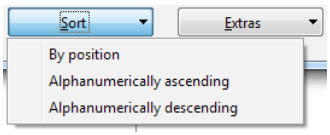


Fig. 7.17
Sort button

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.

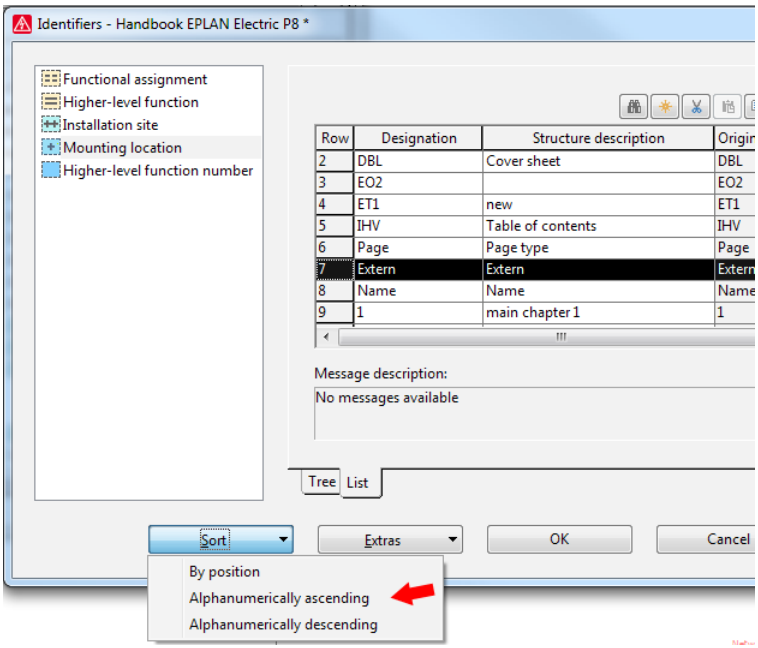


Fig. 7.18 Cursor position

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

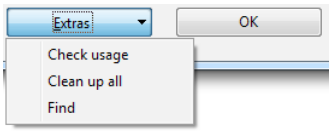


Fig. 7.19 Extras button

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 – even

individually, by selecting the row to be removed – from the database using the graphical **[DELETE]** button.

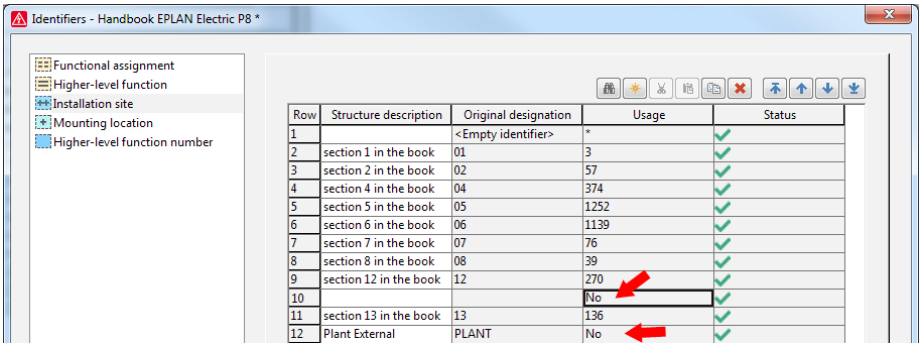


Fig. 7.20 Check usage

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!

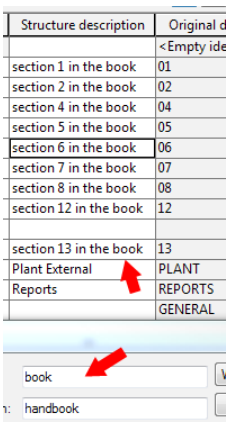


Fig. 7.21 Before replacement

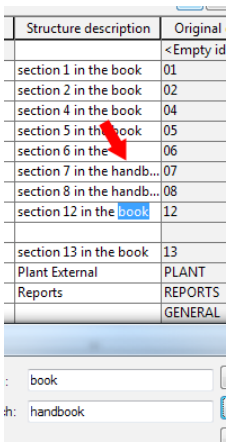


Fig. 7.22 After replacement

NOTE: An unwanted replace action can only be undone by exiting the **IDENTIFIERS** dialog with the **CANCEL** button.

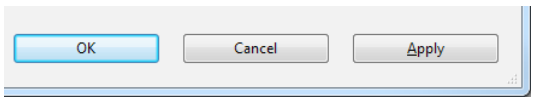


Fig. 7.23 More buttons

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, unlike other features in EPLAN, 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.

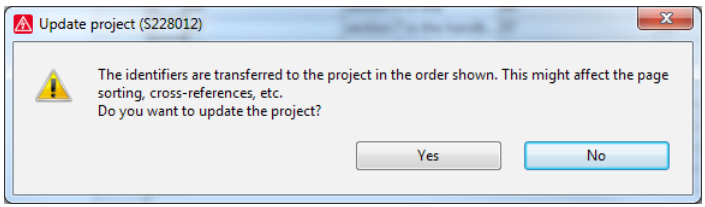


Fig. 7.24 Prompt after you click the **OK** button

NOTE: A kind of “interim save” (**APPLY** button) is helpful when editing the identifier structure so you do not lose all changes.

7.1.5 Configure protection (protect identifiers from changes)

Apart from the functions already described, EPLAN also allows for the option of preventing identifiers from being changed.

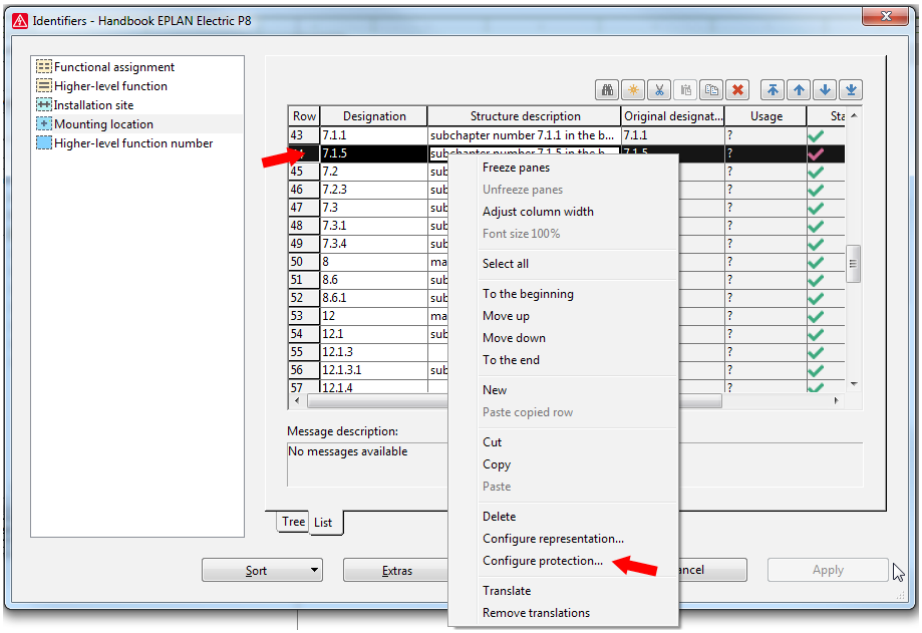


Fig. 7.25 Popup menu – Configure protection function

If you select the identifier and call the *Configure protection* function from the popup menu, EPLAN opens the **CONFIGURE PROTECTION** dialog.

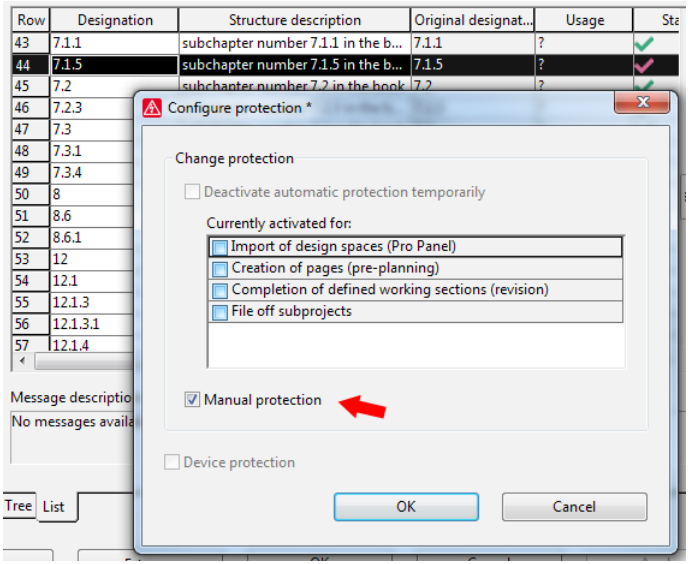


Fig. 7.26 Configure protection dialog

Here, you can now select the *Manual protection* check box. Once you click the **OK** button, this identifier will be protected from changes (change, delete, etc.) in structure identifier management.

EPLAN then gives this protected identifier a gray background.

Row	Designation	Structure description	Original designat...	Usage	Sta
43	7.1.1	subchapter number 7.1.1 in the b...	7.1.1	?	✓
44	7.1.5	subchapter number 7.1.5 in the b...	7.1.5	?	✓
45	7.2	subchapter number 7.2 in the book	7.2	?	✓
46	7.2.3	subchapter number 7.2.3 in the b...	7.2.3	?	✓
47	7.3	subchapter number 7.3 in the book	7.3	?	✓
48	7.3.1	subchapter number 7.3.1 in the b...	7.3.1	?	✓

Fig. 7.27 Protected identifier

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).

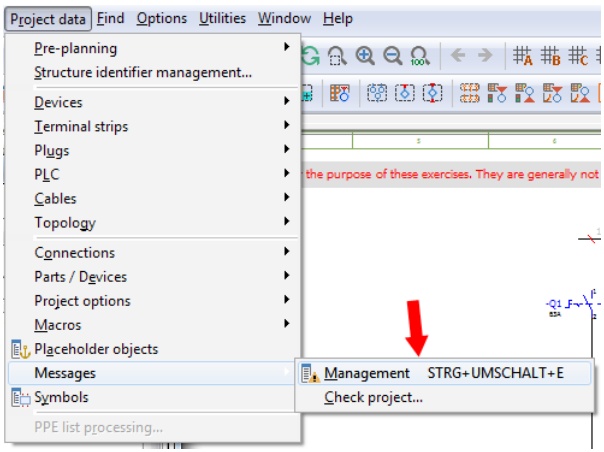


Fig. 7.28 Message management

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 online checks 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.

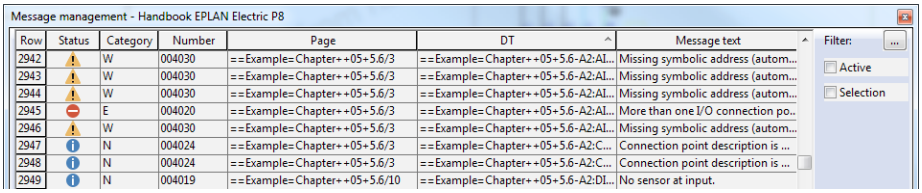


Fig. 7.29 Message management

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 check-run 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"

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.

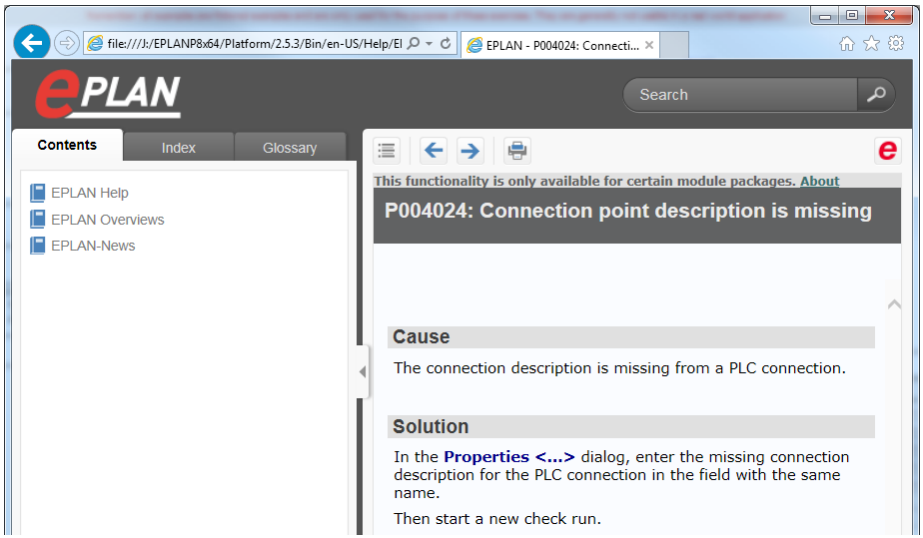


Fig. 7.30 Note on message in online help

7.2.2 Project checks

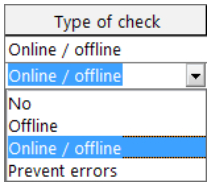


Fig. 7.31
Types of check

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 surely the first choice, when available, but is also the most computing-intensive because all the data and resulting messages must be continually checked and updated. This, of course, may have 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 and also working on the actual planning and design?

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.

Row	Number	Message text	Category	Type of check
1	001001	Terminal without designation.	Warning	Online / offline
2	001005	Terminal of the same name use...	Warning	Online / offline
3	001008	A terminal connection point has...	Warning	Online / offline
4	001010	Terminal strip definition missing	Note	Online / offline
5	001011	Terminal strip without terminals.	Warning	Online / offline
6	001012	Incorrect saddle jumper formati...	Error	Online / offline
7	001015	Too many wire jumper levels.	Warning	Online / offline

Fig. 7.32 Example of check run message

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.

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.

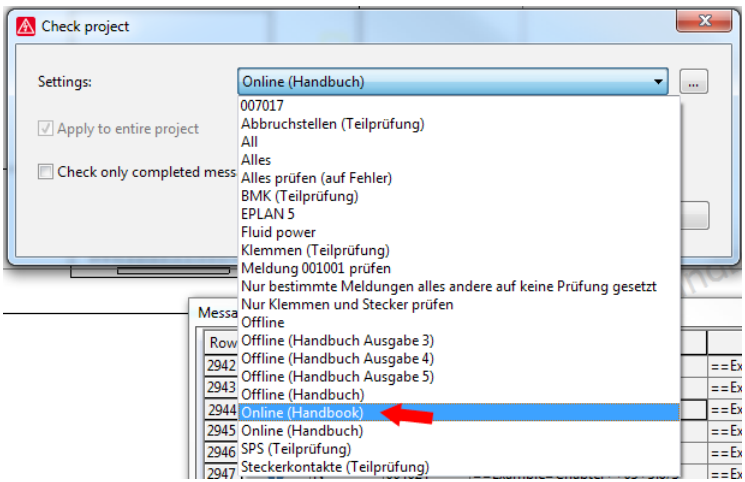


Fig. 7.33 Check project dialog

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

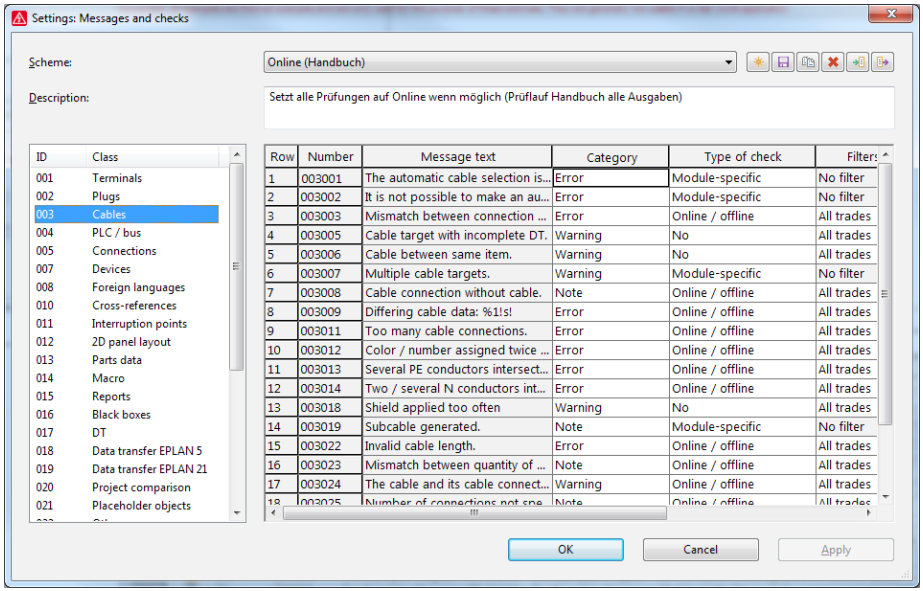


Fig. 7.34 Dialog for creating test schemes

EPLAN opens the **NEW SCHEME** dialog. Here you assign a suitably descriptive **Name** and **Description**. **Note:** The description can also be changed later. 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.

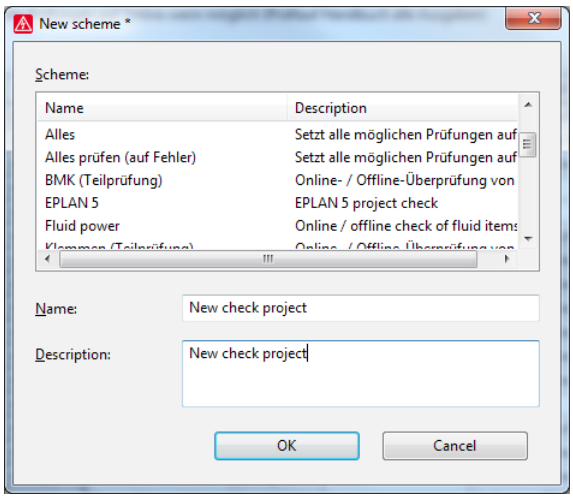


Fig. 7.35 Creating a new scheme

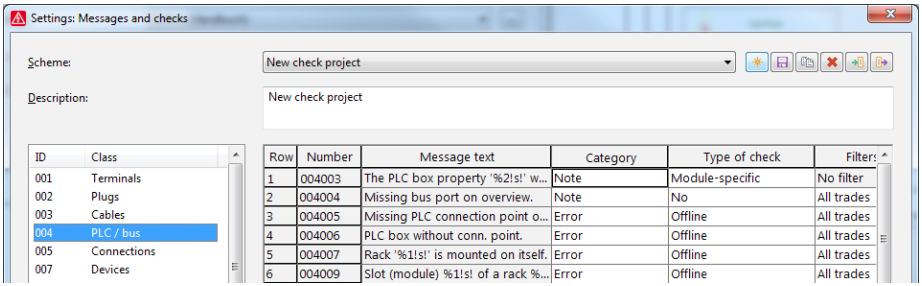


Fig. 7.36 Example of a new scheme

The individual messages can now be set and edited according to the desired specifications.

Every message belongs to a message class. There are currently 31 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.

Class number (ID)	Message class (class)
001	Terminals
002	Plugs
003	Cables
004	PLC / bus
005	Connections
007	Devices
008	Foreign languages
010	Cross-references
011	Interruption points
012	2D panel layout
013	Parts data import
014	Macro export
015	Reports
016	Black boxes
017	DT
018	Data transfer EPLAN 5
019	Data transfer EPLAN 21
020	Project comparison
021	Placeholder objects
022	Other
023	PPE
024	Fluid power
025	Project settings

Class number (ID)	Message class (class)
026	3D mounting layout
027	Topology
028	Pre-planning
029	Pre-planning depth
030	Subprojects
032	Drilling pattern
501	Part master data
999	External

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 (017) and check run message 003. The rest of the message number is a three-digit number.

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.

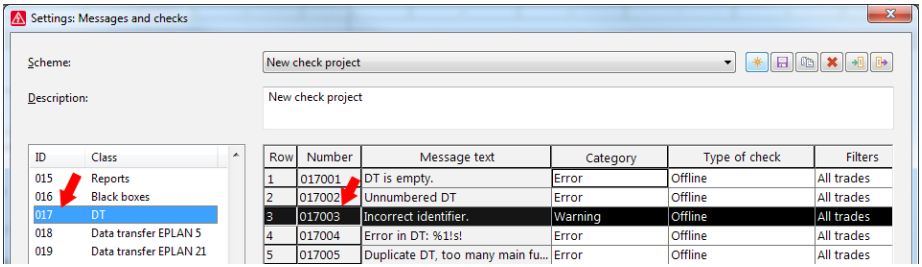


Fig. 7.37 Message text

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.

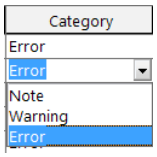


Fig. 7.38 Category

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.

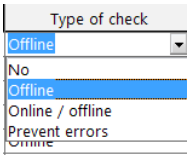


Fig. 7.39 Types of check

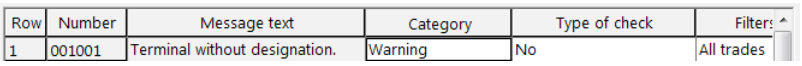


Fig. 7.40 Type of check set to “No”

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*. **Note:** The *Select* check box filter was set to active. Now, only messages are displayed that concern the highlighted selection.

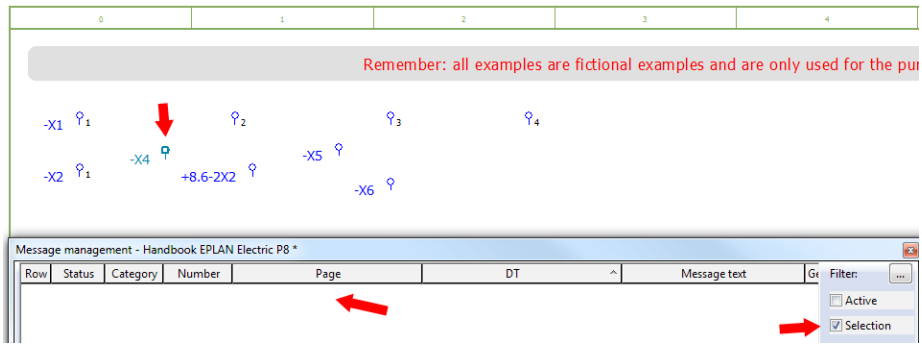


Fig. 7.41 Message management after completed check run

The check type **Offline** only generates messages when a check run is manually started.

Row	Number	Message text	Category	Type of check	Filters
1	001001	Terminal without designation.	Warning	Offline	All trades

Fig. 7.42 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*.

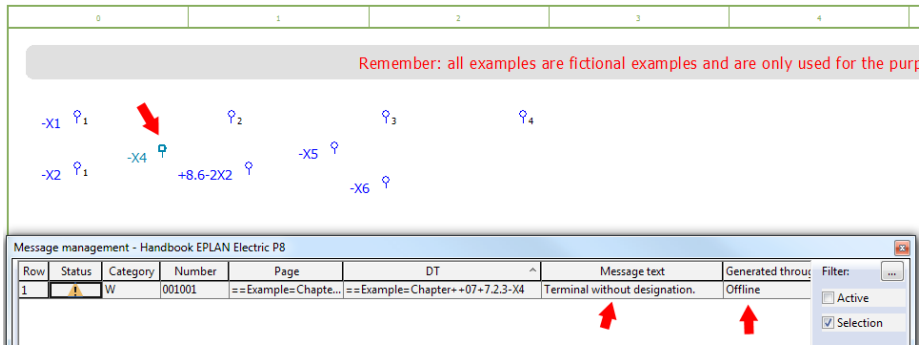


Fig. 7.43 Type of check set to “Offline”

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.

Row	Number	Message text	Category	Type of check	Filters
1	001001	Terminal without designation.	Warning	Online / offline	All trades

Fig. 7.44 Type of check set to "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*.

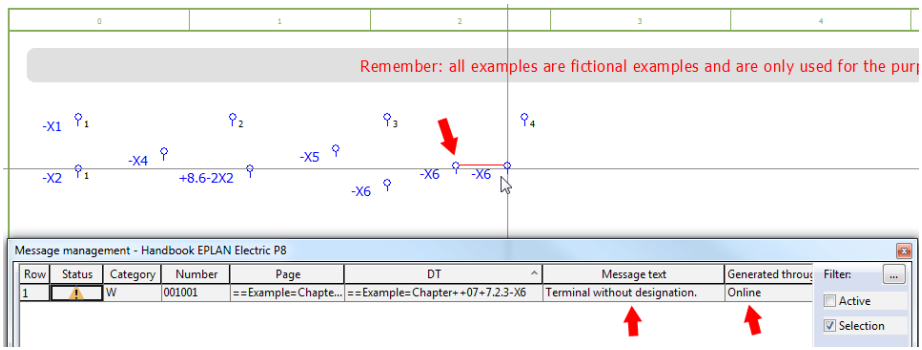


Fig. 7.45 Type of check set to "Online/offline", EPLAN reports the error immediately

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.

Row	Number	Message text	Category	Type of check	Filters
1	001001	Terminal without designation.	Warning	Prevent errors	All trades

Fig. 7.46 Type of check set to "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.

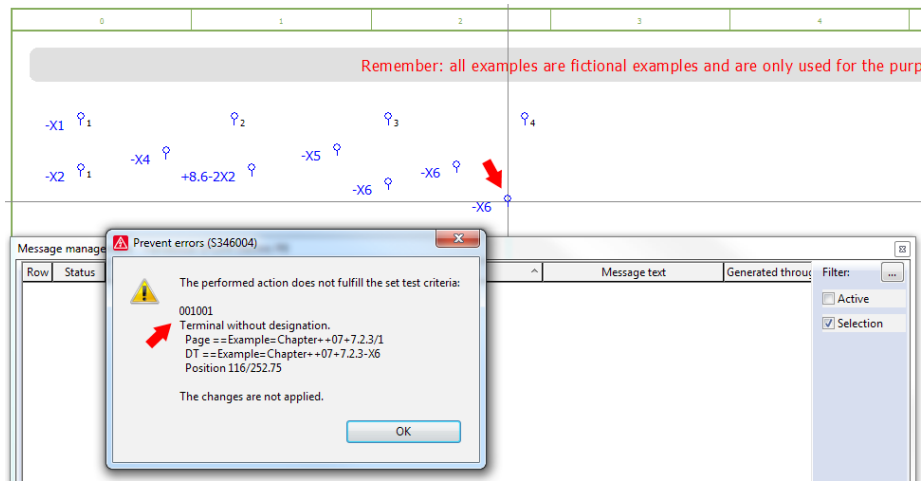


Fig. 7.47 EPLAN prevents the error on the basis of the type of check defined

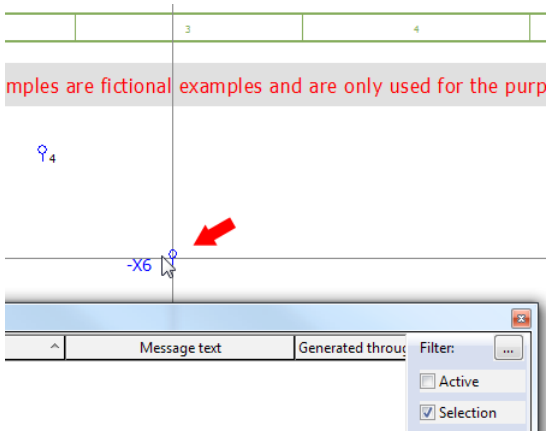


Fig. 7.48 Terminal can be repositioned

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 of correcting them later during a check run.

EPLAN cancels this action, and you can reposition and label the terminal. If the specifications are met (terminal designation exists), the terminal can be placed.

The **Module-specific check** type of check also generates messages during certain automated actions, but cannot be influenced any further.

In addition to message management, other modules also carry out checks in EPLAN and then generate messages in the **MESSAGE MANAGEMENT** dialog. These checks are displayed in the **MESSAGE MANAGEMENT** dialog, in the *Type of check* column, as "module-specific". If a module-specific check is not to be run, the type of check for this message must be set to "No".

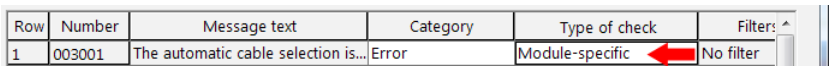


Fig. 7.49 Module-specific check

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.

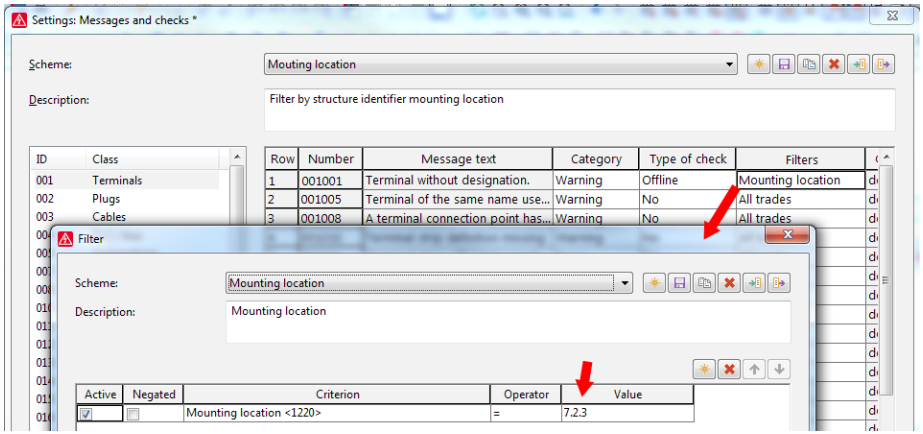


Fig. 7.50 Filter options for each check run message

These filters allow you to limit certain checks to specific areas or properties. In other words, checks can be targeted at specific objects..

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.

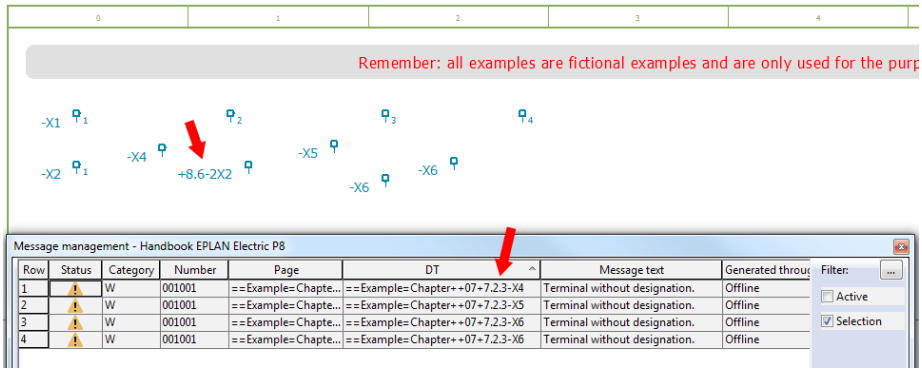


Fig. 7.51 Only checks defined by filter will be carried out

In the example, the terminals without a designation, whose mounting location was different than the one defined in the filter, were ignored by the check run. The terminals whose mounting location matches the filter, however, were 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 click the ... button to open the **FILTER: MESSAGES** dialog.

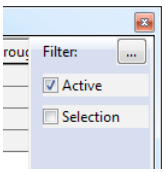


Fig. 7.52
Filter – Active

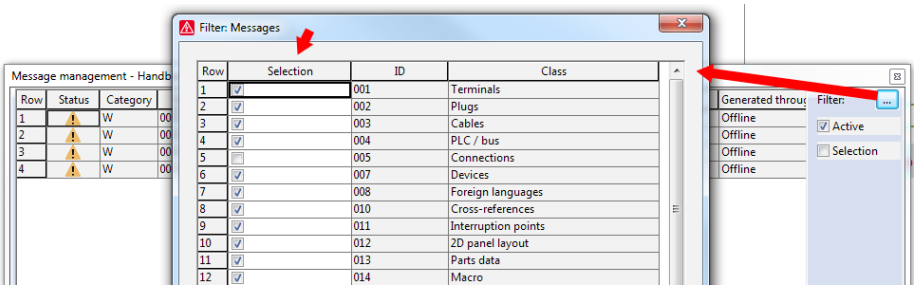


Fig. 7.53 Filter settings for message class selection

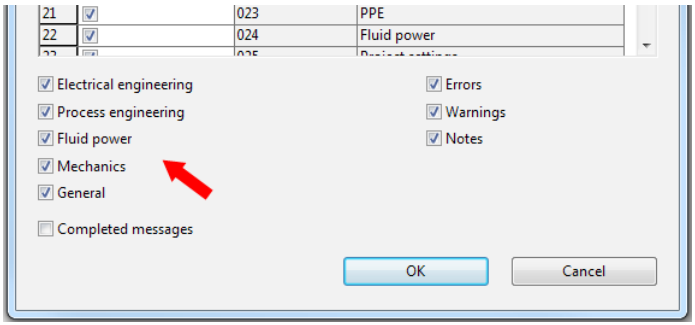


Fig. 7.54 Filter settings for the areas

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. 7.55 in section 7.2.4.2).

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 – Select 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.

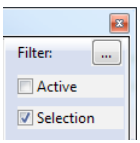


Fig. 7.55
Filter – Select

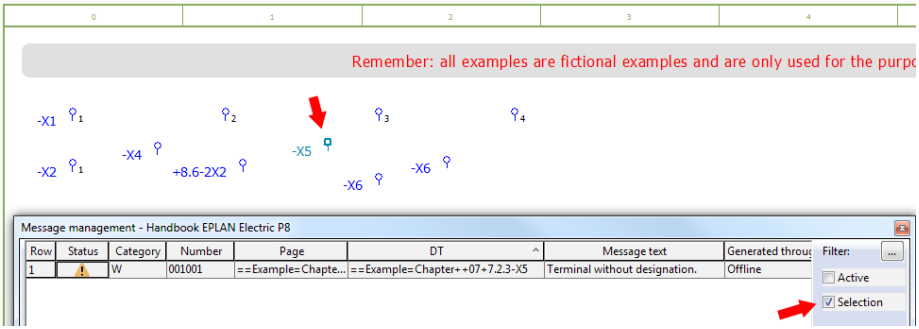


Fig. 7.56 Selection has been activated

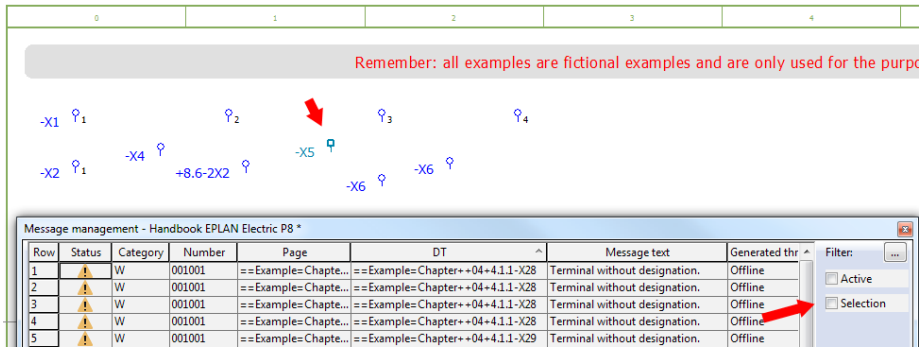


Fig. 7.57 Selection has not been activated

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.

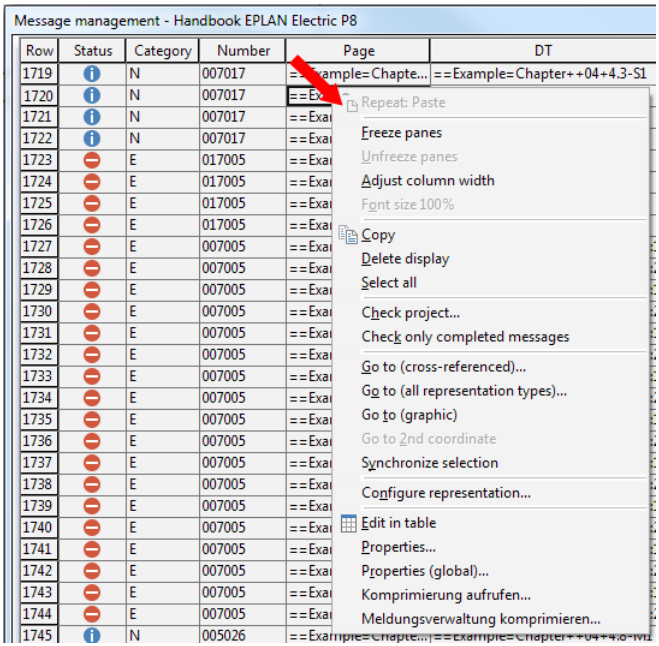


Fig. 7.58 Popup menu in the Message management dialog

The right-click popup menu provides several options/functions 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.
- **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.) 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. **Note:** Double-clicking a selected check with the left mouse button yields the same result. EPLAN jumps directly to the (checked) object in question (and opens the page).
- **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 function, familiar from other navigators, 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 REPRESENTATION**: This function allows the number of columns displayed in message management to be increased or decreased.

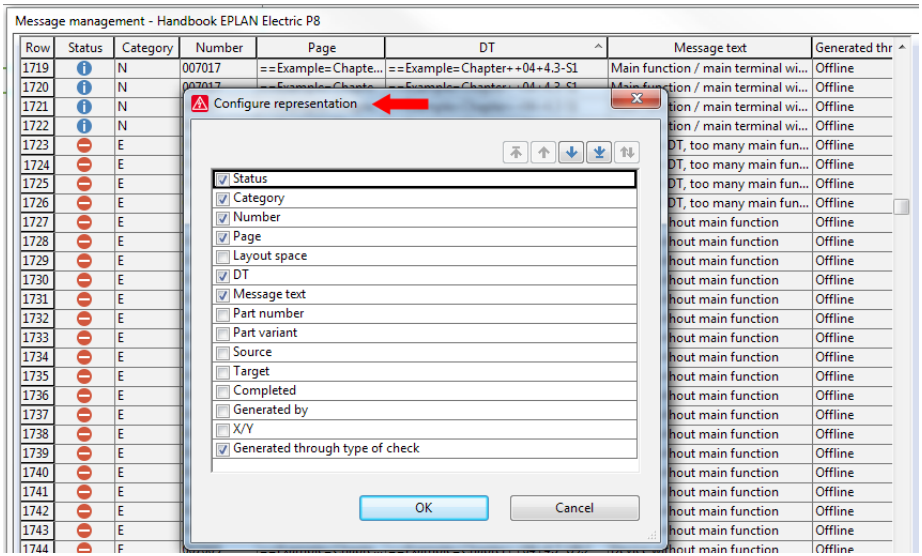


Fig. 7.59 Configure representation

When this function is executed, EPLAN displays the **CONFIGURE REPRESENTATION** 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.

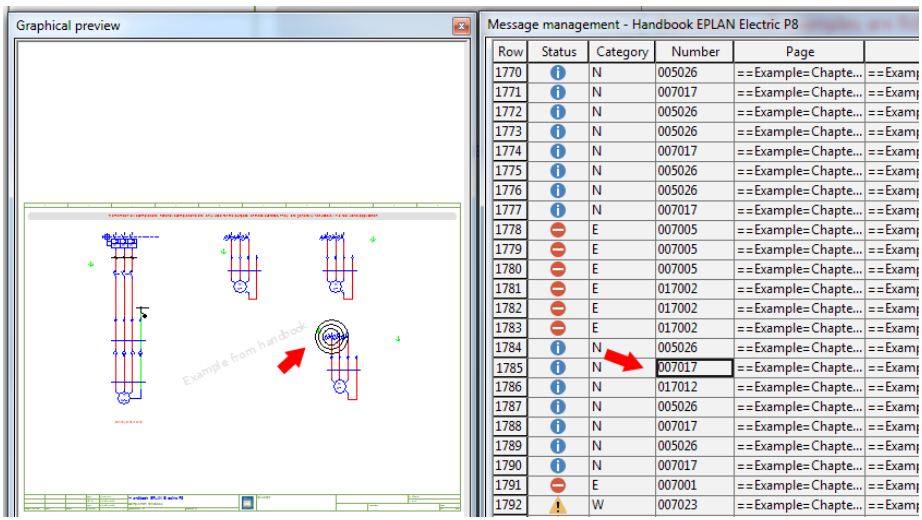


Fig. 7.60 Using the graphical preview to find errors

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, and developed it further from one version to the next.

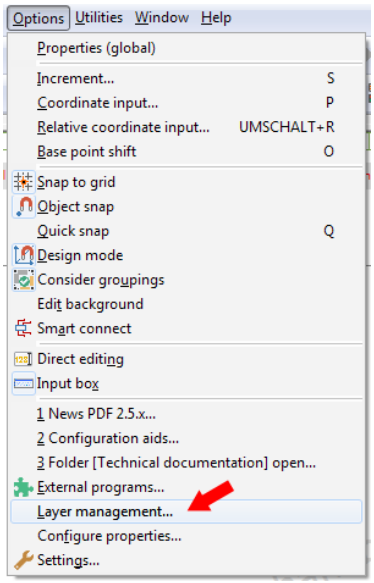


Fig. 7.61
Layer management

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 sub-entries (the actual layer information).

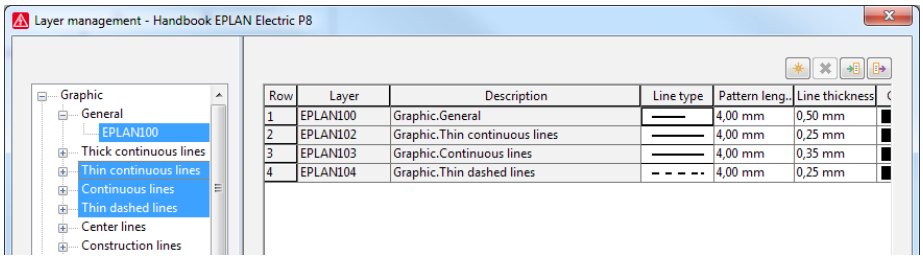


Fig. 7.62 Layer management dialog

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).

Row	Layer	Description	Line type	Pattern leng..	Line thickness	Color	Font size	Al
8	EPLAN306	Symbol graphic.PLC boxes	- - - - -	4,00 mm	0,25 mm		2,50 mm	Lowe
9	EPLAN307	Symbol graphic.Structure boxes	—·—·—	4,00 mm	0,25 mm		2,50 mm	Lowe

Fig. 7.63 Standard EPLAN layers

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. 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.

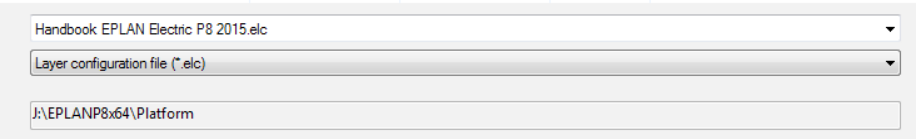


Fig. 7.64 Layer export

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.

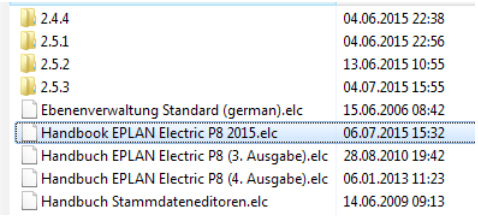


Fig. 7.65 Layer import

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.

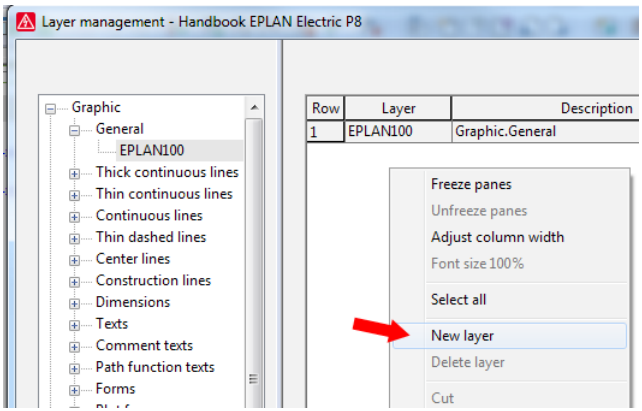


Fig. 7.66 Creating a new layer

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, call the *New layer* function. EPLAN generates a new layer with the name NEW_LAYER_1.

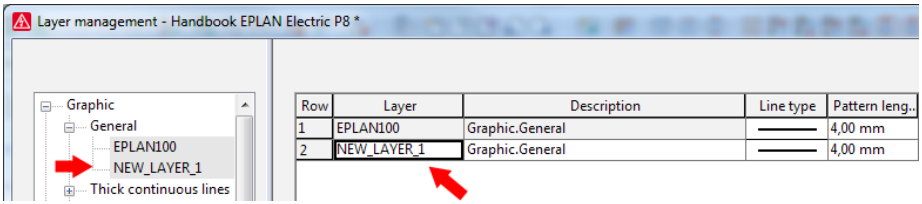


Fig. 7.67 New user-created layer

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 HBE-0100. The layer description will be changed to **Handbook.Graphic.Special**. Immediately after you save the layer, EPLAN will sort it as follows.

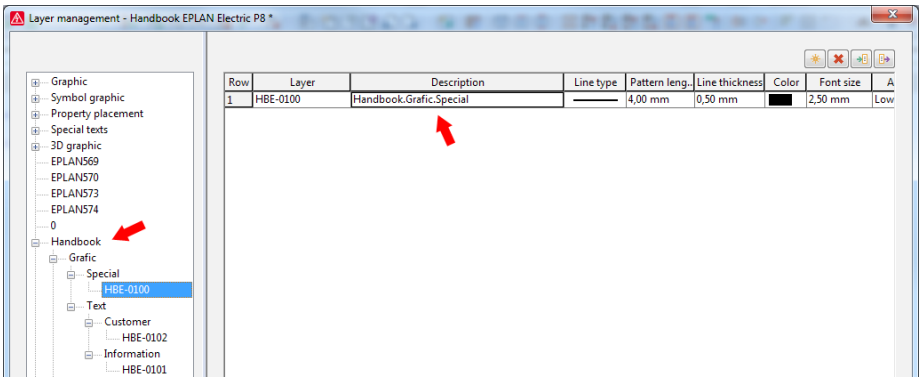


Fig. 7.68 Newly sorted layer

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, whether this layer should be in the background, or whether the layer and its elements should be scalable.

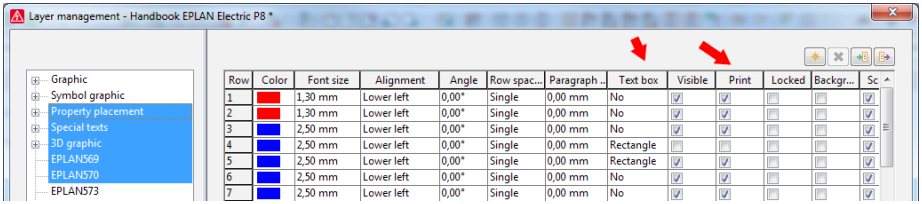


Fig. 7.69 Other layer settings options

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 HBE-0101 with the description **Handbook.Graphic.Texts.Workshop.Information** description was created. 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 leng..	Line thickness	Color	Font size	A
1	HBE-0100	Handbook.Grafic.Special		4,00 mm	0,50 mm		2,50 mm	Low
2	HBE-0101	Handbook.Grafic.Text.Information		4,00 mm	0,35 mm		3,50 mm	Low
3	HBE-0102	Handbook.Grafic.Text.Customer		4,00 mm	0,35 mm		2,50 mm	Low
4	HBE-0400	Handbook.Eigenschaftsplatzierung.Kabel.D...		4,00 mm	0,35 mm		2,50 mm	Low

Fig. 7.70
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.

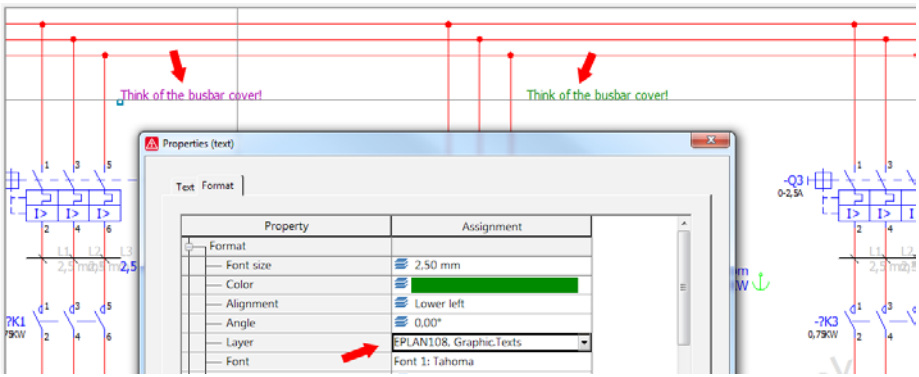


Fig. 7.71
Text with EPLAN default setting

Then, these texts are selected and placed on the newly created layer **Handbook.Graphic.Text.Information**.

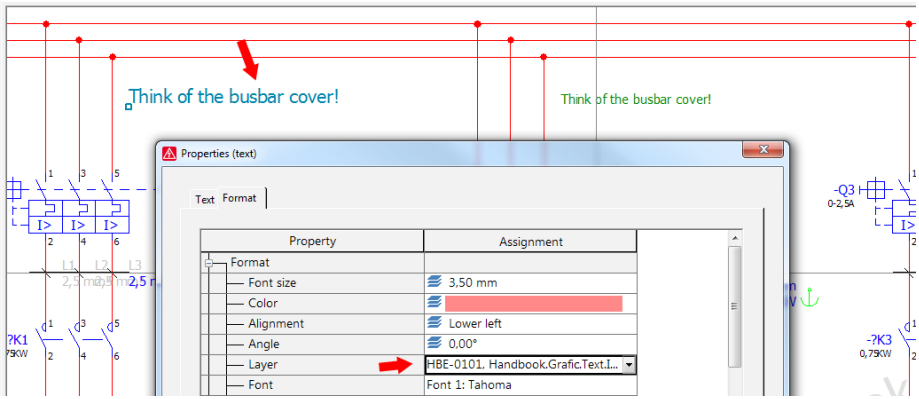


Fig. 7.72
Texts placed on the new layer

EPLAN handles the settings that pertain to this layer and automatically formats the texts accordingly. Via layer management, this workshop information can now be hidden, for example, prior to printing.

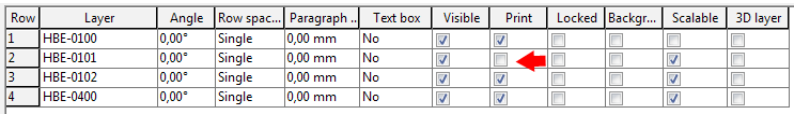


Fig. 7.73 Excluding a layer from printing

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 Editable layer properties

EPLAN provides the following properties/selections for layers, which can be changed globally in the layer management.

Property	Selection or input	Comment
Line type	Drop-down list	51 line types available
Pattern length	Free input	–
Line thickness	Free input	–
Color	Selection	Branching to color selection
Font size	Free input	–
Alignment	Drop-down list	Selection of 12 values
Angle	Free input	–
Row spacing	Drop-down list	Selection of 3 values
Paragraph spacing	Free input	–
Text box	Drop-down list	Selection of 4 values
Visible	Check box	Yes/No
Printing	Check box	Yes/No
Locked	Check box	Yes/No
Background	Check box	Yes/No
Scalable	Check box	Yes/No
3D layer	Check box	Yes/No

7.3.4.2 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.

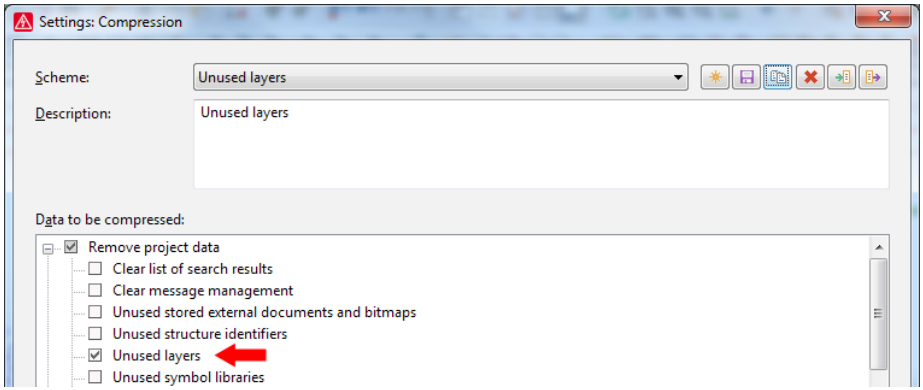


Fig. 7.74 Compression scheme to remove unnecessary layers

After the scheme is launched, EPLAN removes the unnecessary layers and then displays a summary of these removed layers.

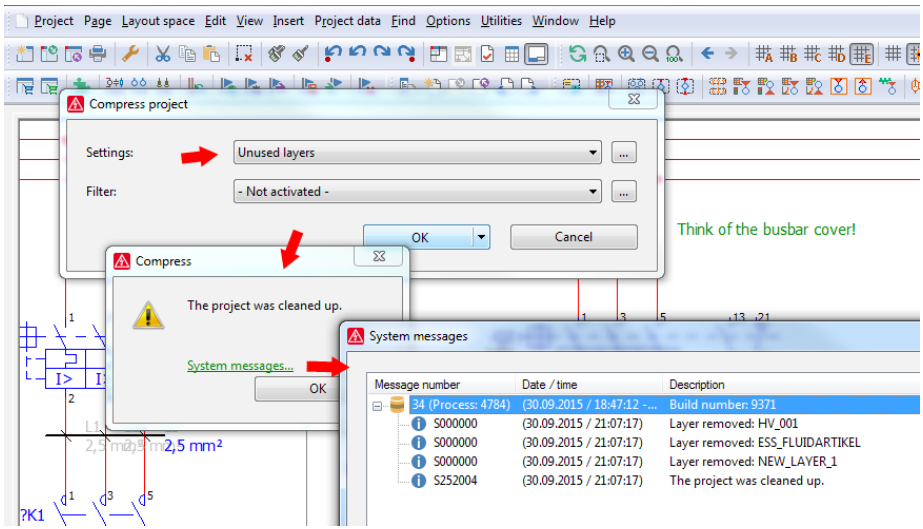


Fig. 7.75 System message

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.

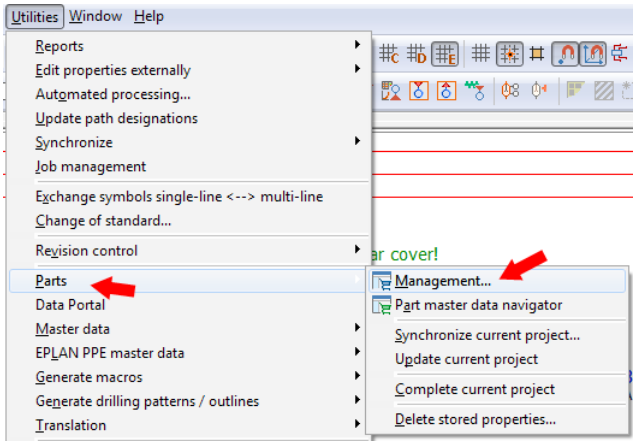


Fig. 7.76
Calling up parts management

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.

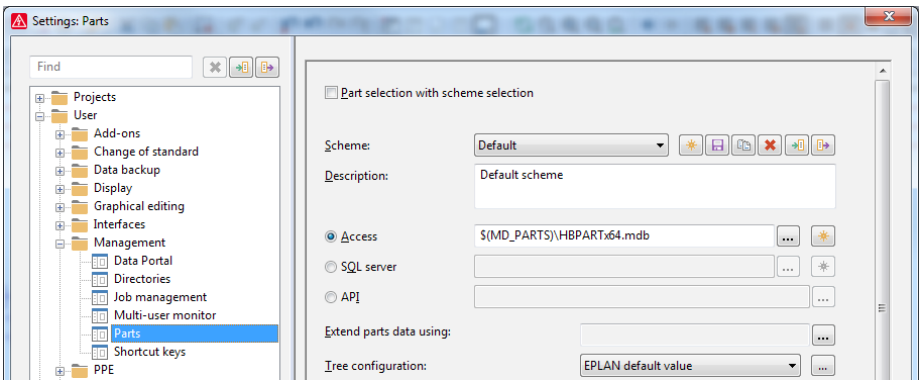


Fig. 7.77
Parts management user settings

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.

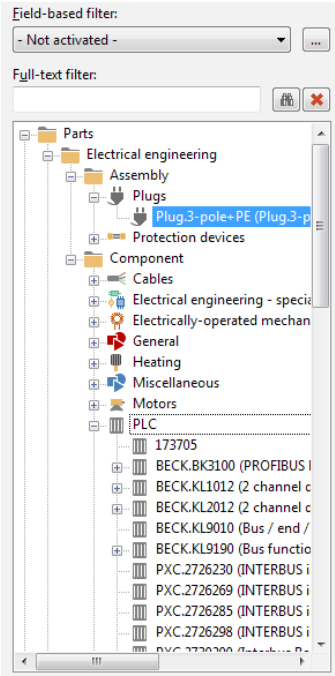


Fig. 7.78 Standard EPLAN view

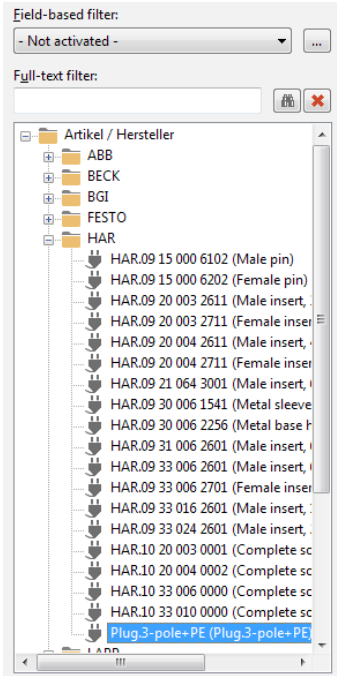


Fig. 7.79 View by manufacturer

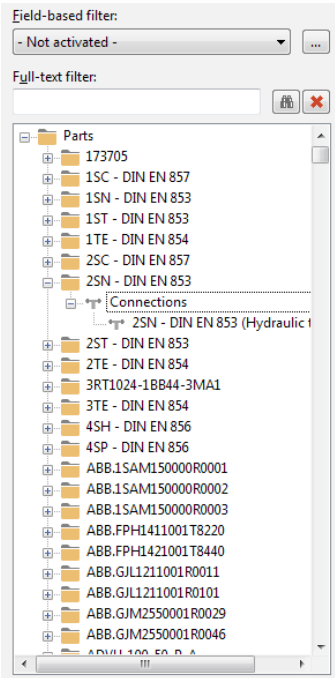


Fig. 7.80 View by manufacturer

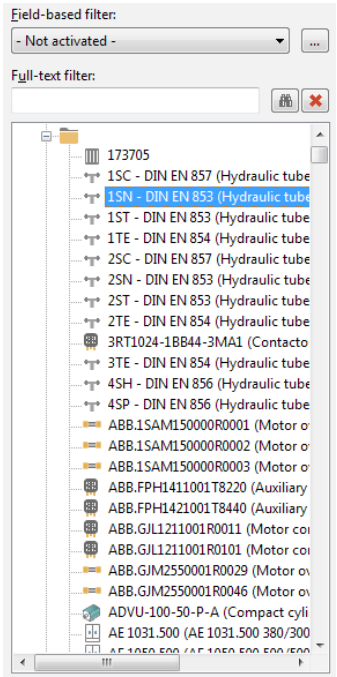


Fig. 7.81 View by ERP number

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. More information on this topic can be found in my book *EPLAN Electric P8 Artikelverwaltung* (ISBN 978-3-446-43332-8, only available in German).

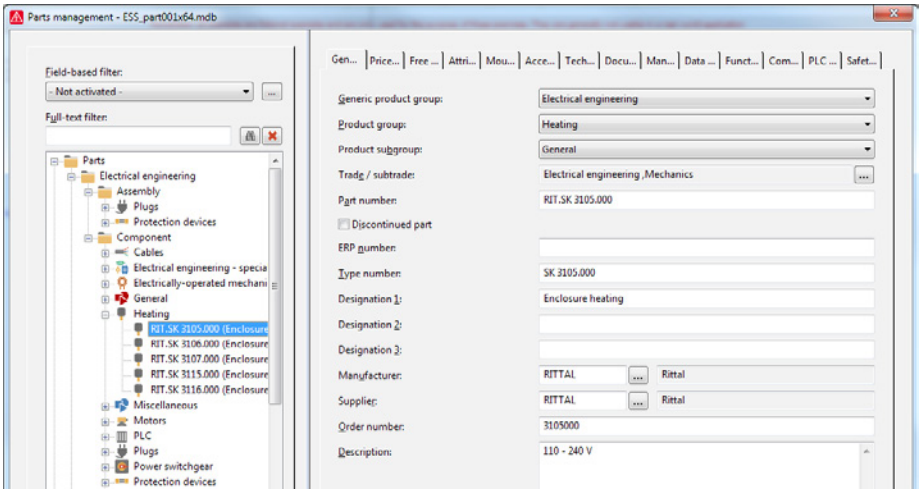


Fig. 7.82 Overview of the parts management dialog

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.

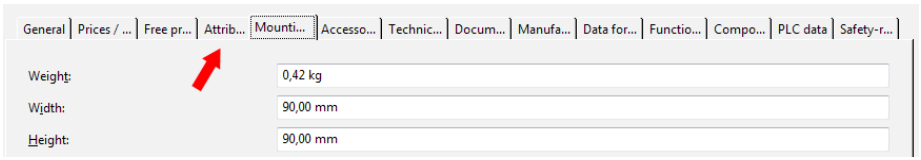


Fig. 7.83 The different tabs of a part

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.

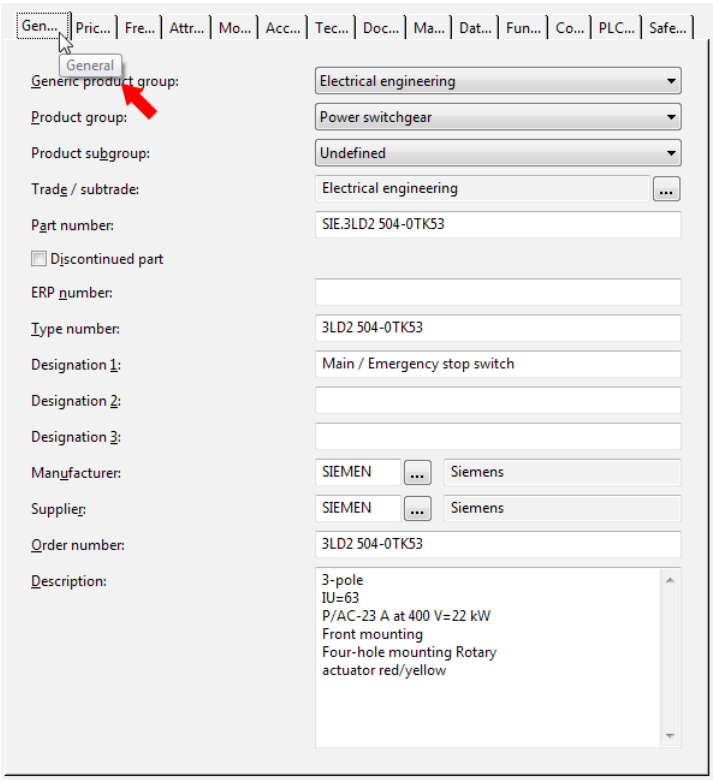


Fig. 7.84 General tab

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.

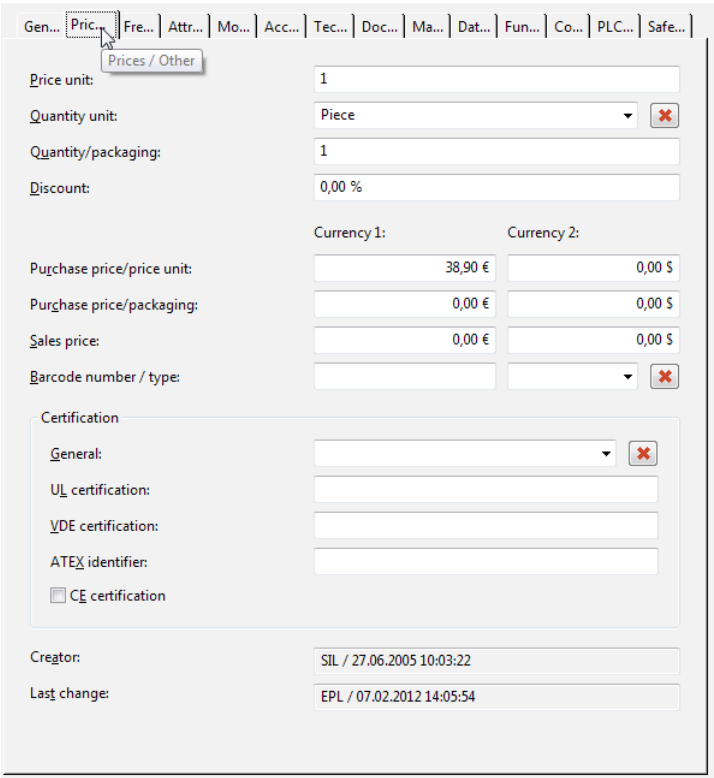


Fig. 7.85 Prices / Other tab

The *Prices/Other* tab contains commercial and organizational entries like *Quantity unit*, *Package sizes*, *Prices* and even linked documents for *Certification*.

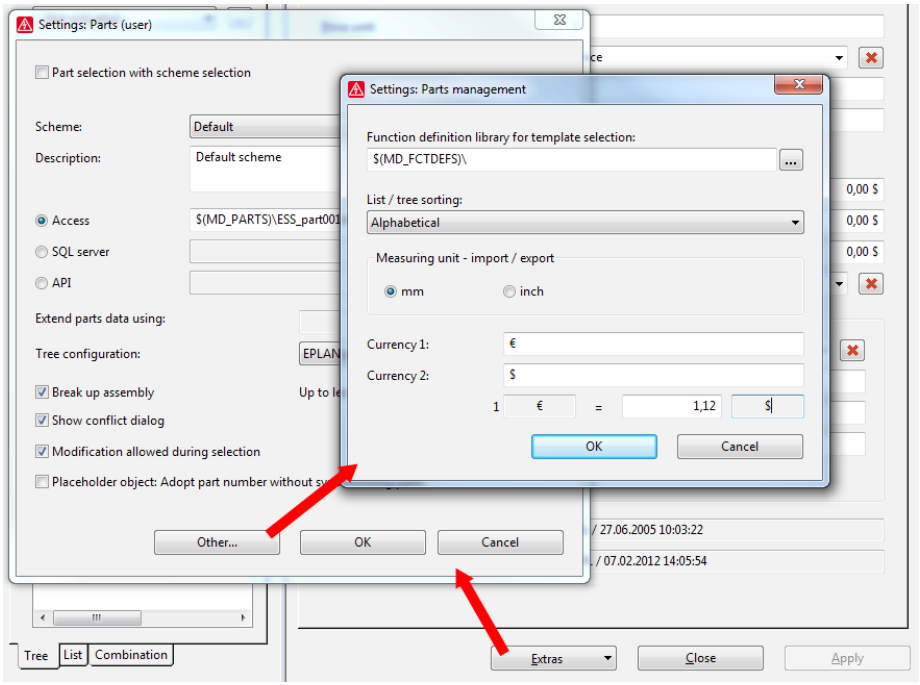


Fig. 7.86 Selecting a currency

To define the currency, you first click the **EXTRAS** button in parts management, then select the **SETTINGS** menu item, and then click the **MORE** button in the **SETTINGS/PARTS (USER)** dialog.

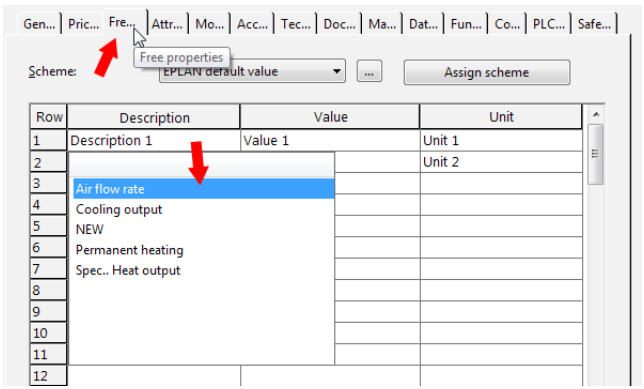


Fig. 7.87 Free properties tab

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.

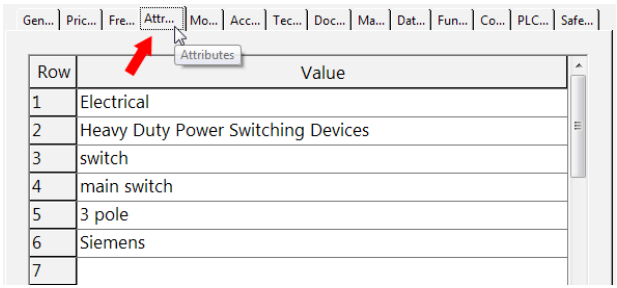


Fig. 7.88
Attributes tab

The *Attributes* tab allows you to store additional information (in the *Value* column) that is not available as standard in parts management or used elsewhere. 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.

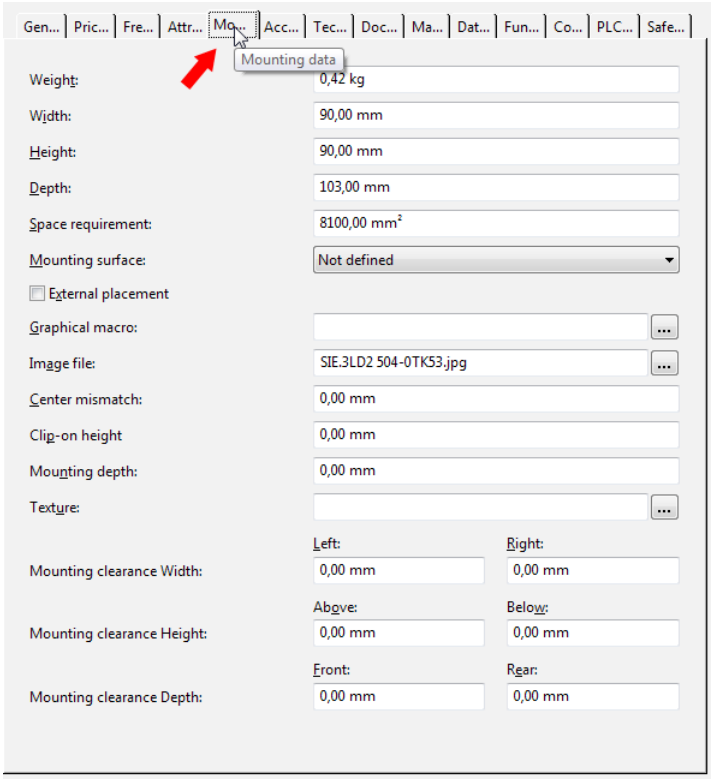


Fig. 7.89
Mounting data tab

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.

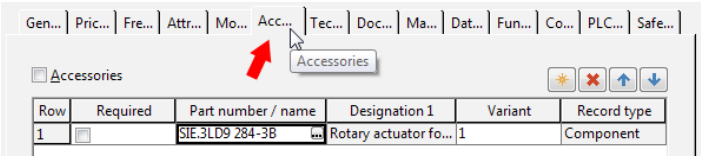


Fig. 7.90
Accessories tab

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.

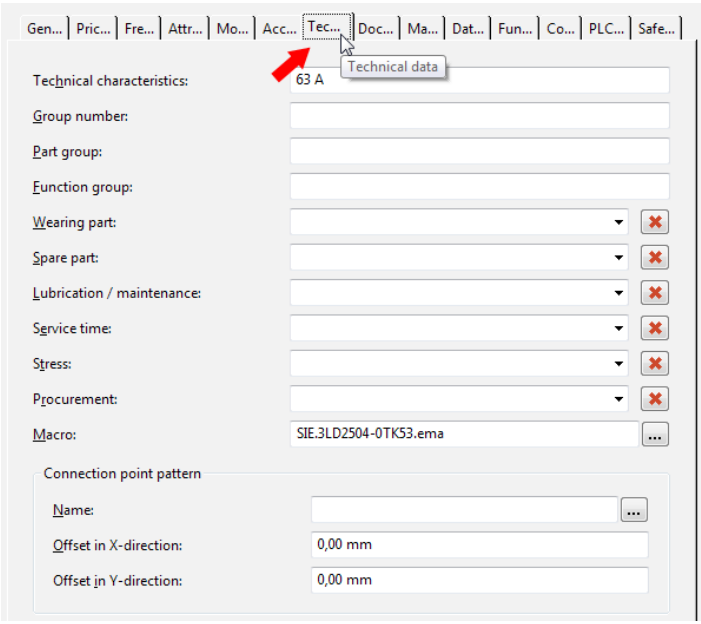


Fig. 7.91
Technical data tab

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.

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 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.

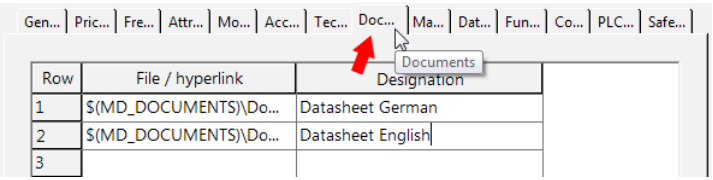


Fig. 7.92 Documents tab

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.

The value in the *Description* can be in multiple languages. This means that the field can be translated.

The *Manufacturing* tab contains a relevant drilling pattern for the selected part. The procedure here is similar to that for accessories lists. Here, too, the drilling pattern must be available before the actual assignment, that is, before it is assigned to the part.

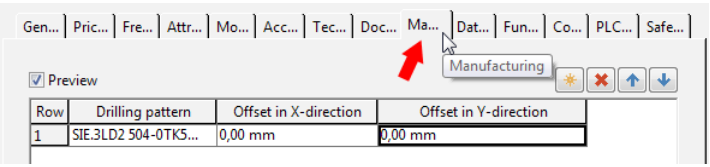


Fig. 7.93 Manufacturing tab

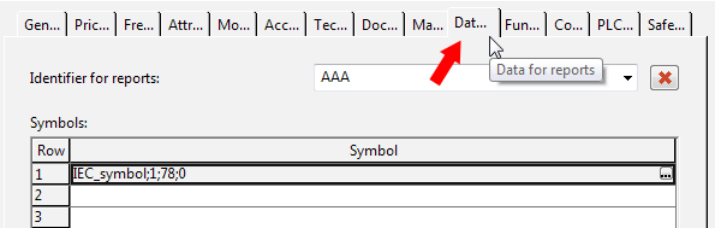


Fig. 7.94 Data for reports tab

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.

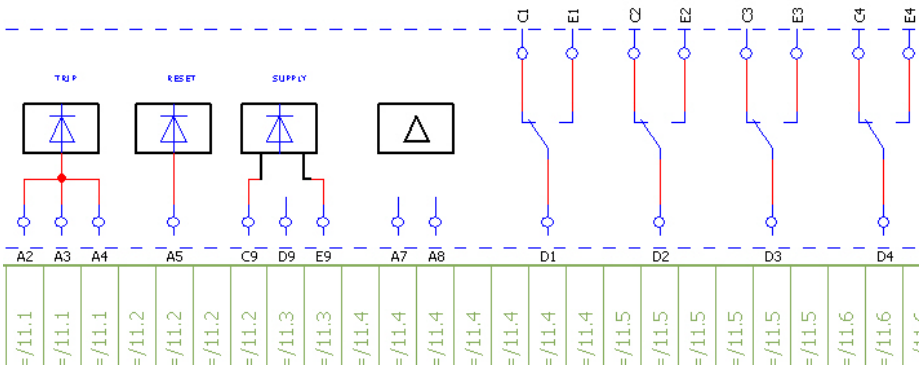


Fig. 7.95 Example view of a complex graphical report of a part

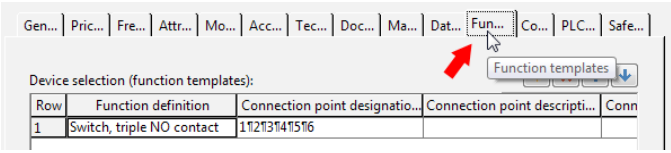


Fig. 7.96 Function templates tab

The *Function templates* tab contains key information about the actual part and for the subsequent device selection 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.

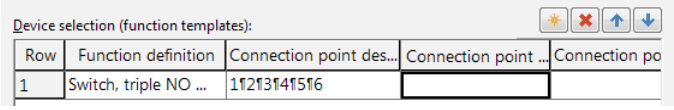


Fig. 7.97 The first columns of the function template

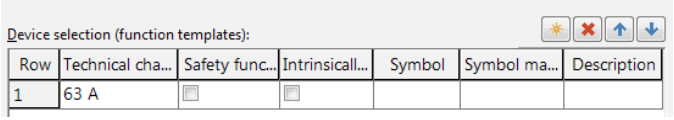


Fig. 7.98 Further function template columns

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*.

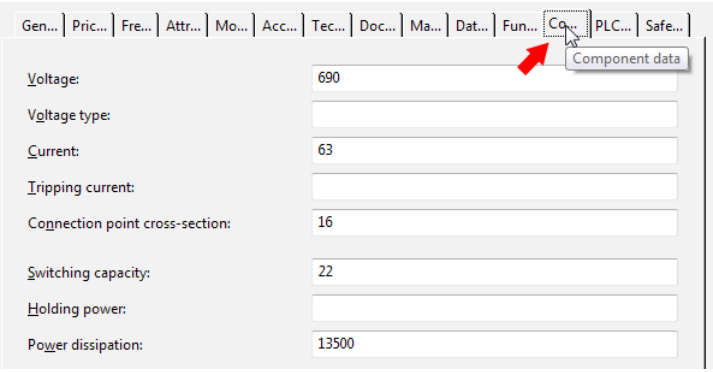


Fig. 7.99 Component data tab

The *Component data* tab 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.

The second-to-last tab, *PLC data*, contains information that is important for PLC devices. Here you can enter PLC type designations, enter object descriptions, or generate reports for them.

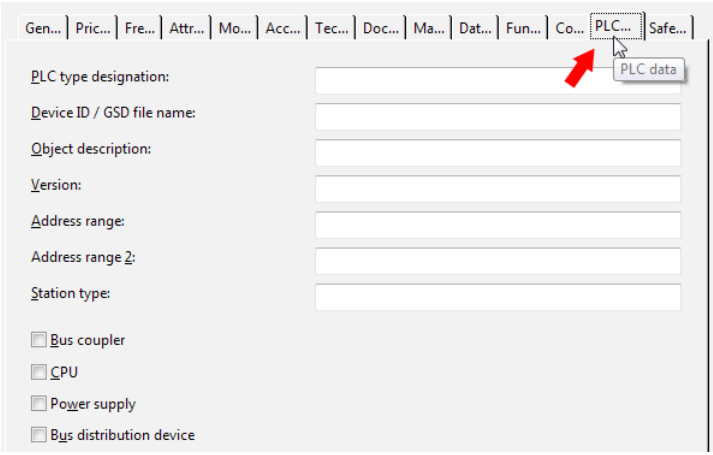


Fig. 7.100 PLC data tab

The last tab, *Safety-related values*, contains important information on a device's characteristic values (performance level, SIL, etc.), use cases, and much more.

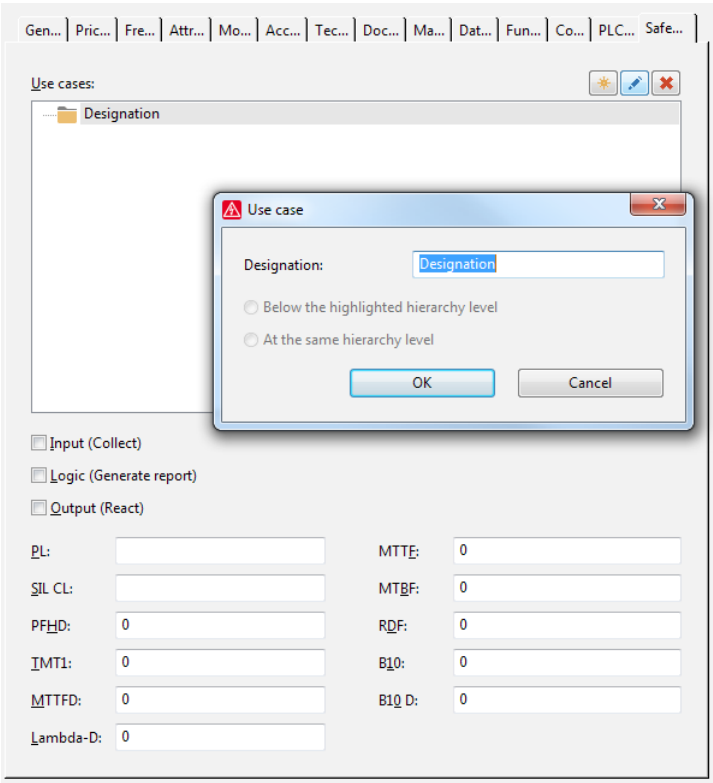


Fig. 7.101 Safety-related values tab

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**.

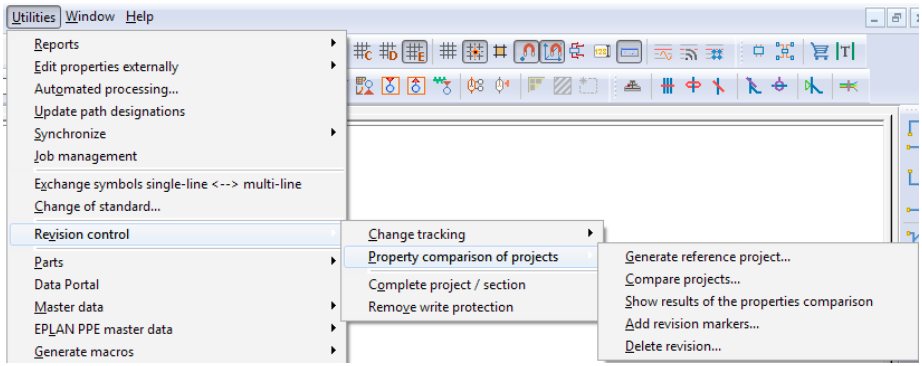


Fig. 7.102 Revision control with property comparison

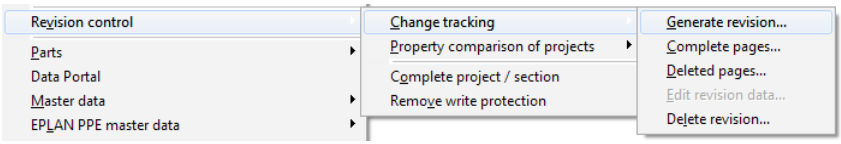


Fig. 7.103 Revision control with change tracking

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.

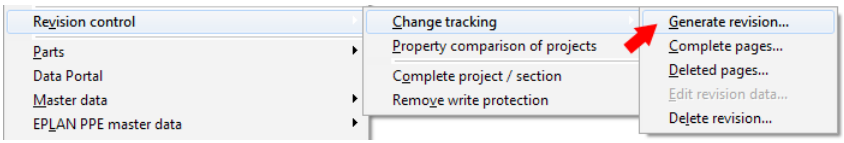


Fig. 7.104 Generating revision

EPLAN opens the **GENERATE REVISION** dialog. Here you must enter a *Revision name*. The *Comment* on the revision is optional. The *user*, *defined working section*, and the *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.

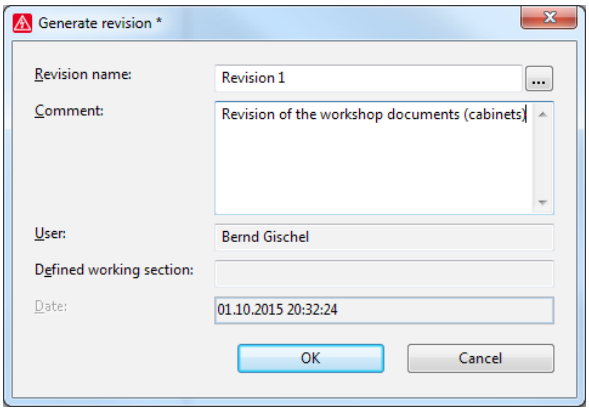


Fig. 7.105
Entering revision data

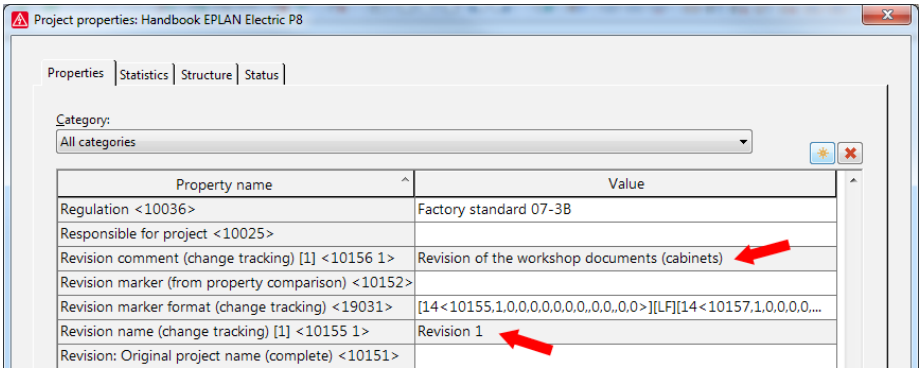


Fig. 7.106 Project property names of revision entries

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.

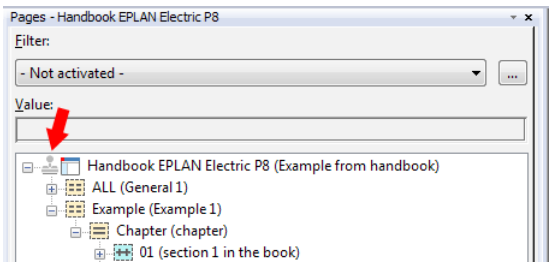


Fig. 7.107 Modified icon

7.5.3 Execute changes

After the revision has been created, the changes can be entered in the schematics.

If EPLAN is to query the index automatically with every change, you must set *Always prompt for description of page modification* in the settings (**OPTIONS/SETTINGS/PROJECTS/[PROJECT NAME]/MANAGEMENT/REVISION (CHANGE TRACKING)** menu).

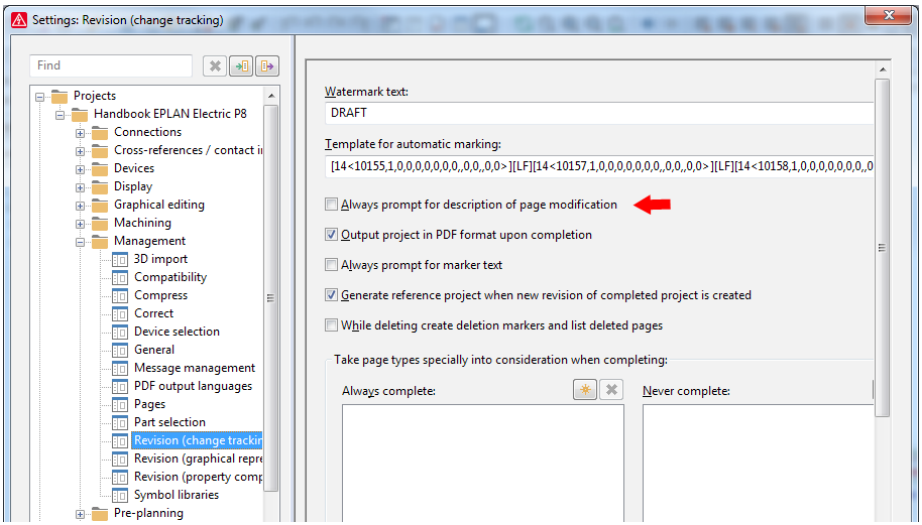


Fig. 7.108 Always prompt for page modification setting

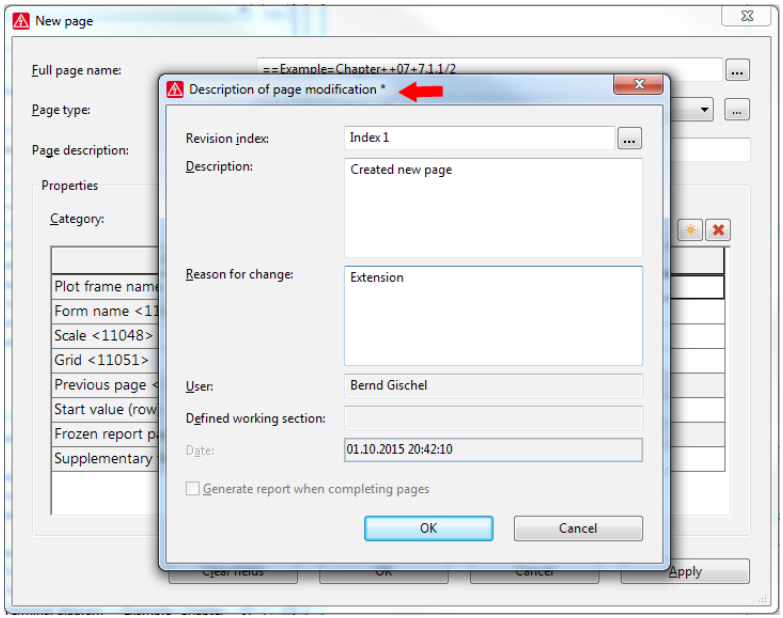


Fig. 7.109 Index query following a change

After each change, EPLAN prompts for 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. The *user*, *defined working section*, and the *change date* are also pre-set 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.

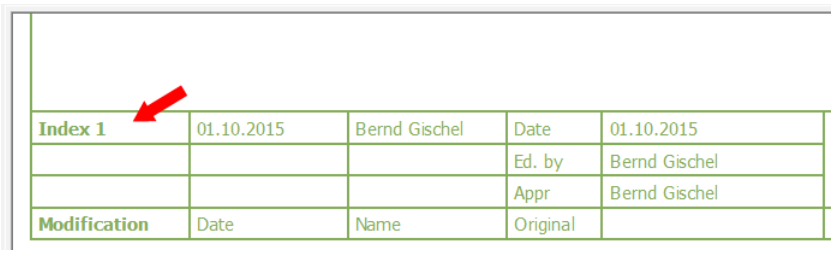


Fig. 7.110 Adding the revision index to the plot frame

The properties of the revision index and their additional data such as the description are pure page properties.

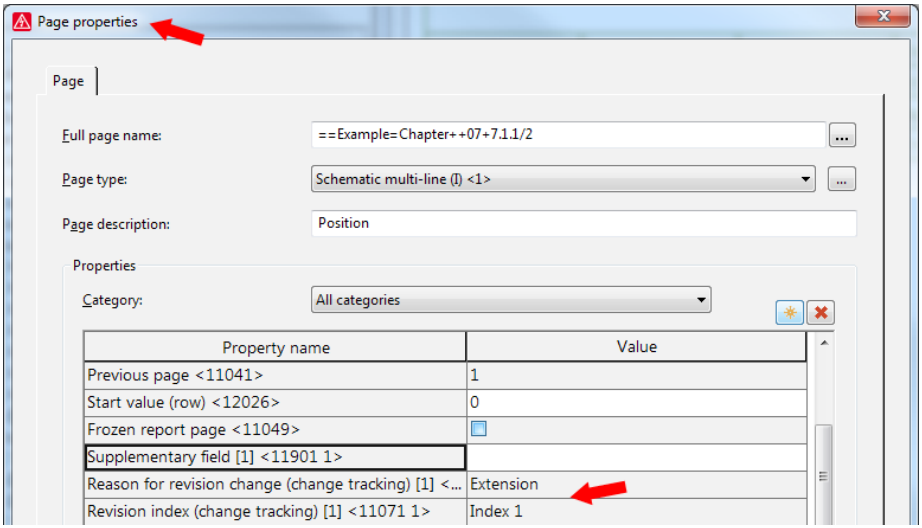


Fig. 7.111 Revision index in the page properties

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. This serves as a visual indication that this page and the modifications have not been completed yet.

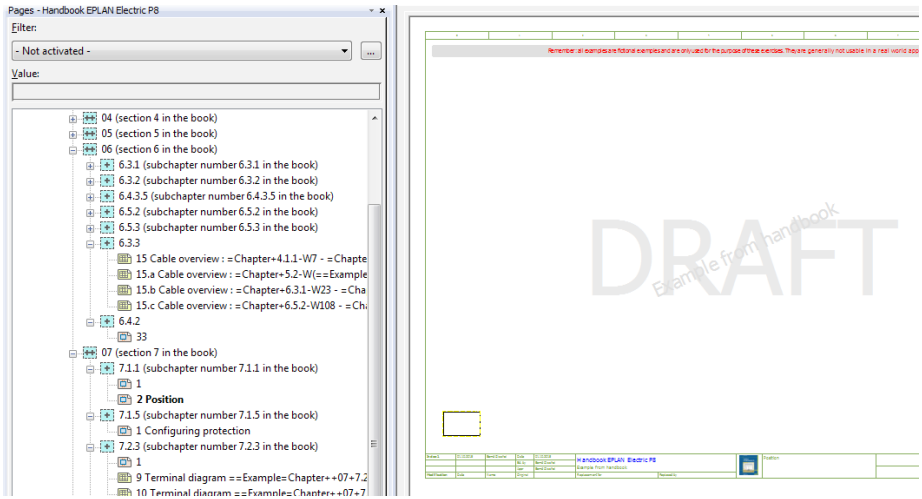


Fig. 7.112 Marking the page as DRAFT

EPLAN allows you to place revision markers for every modification. You can also use visually different revision markers for different types of changes.

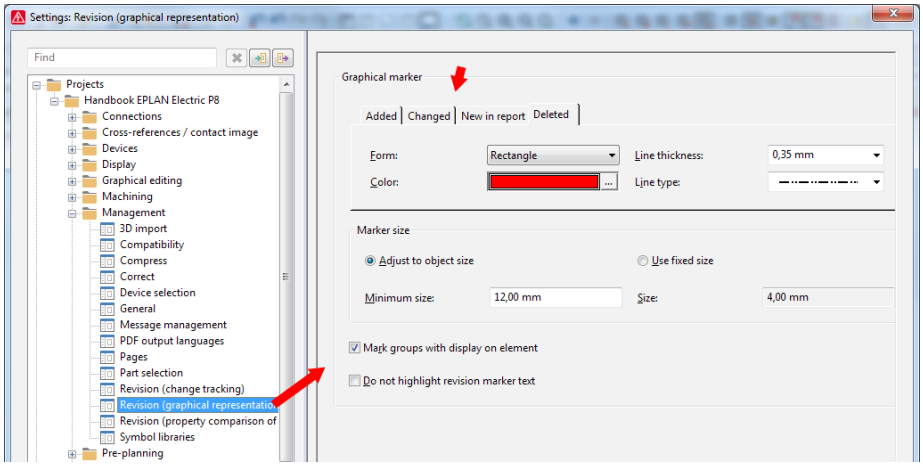


Fig. 7.113 Graphical markers for various modifications

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.

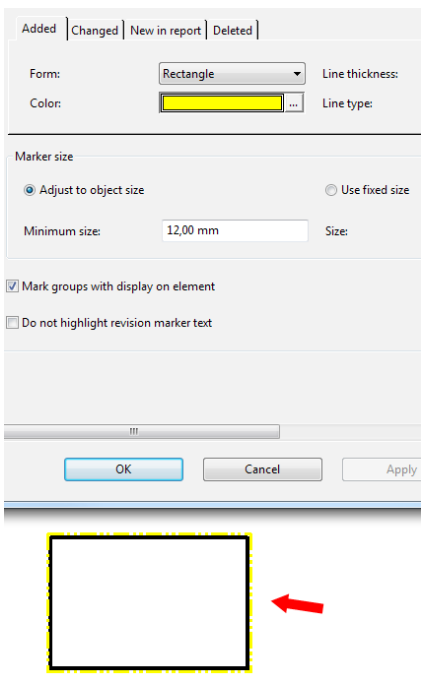


Fig. 7.114 Graphical marker for added objects

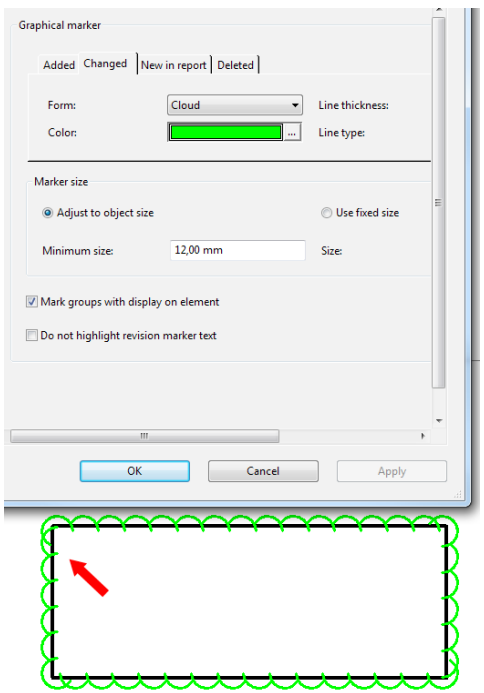


Fig. 7.115 Graphical marker for changed objects

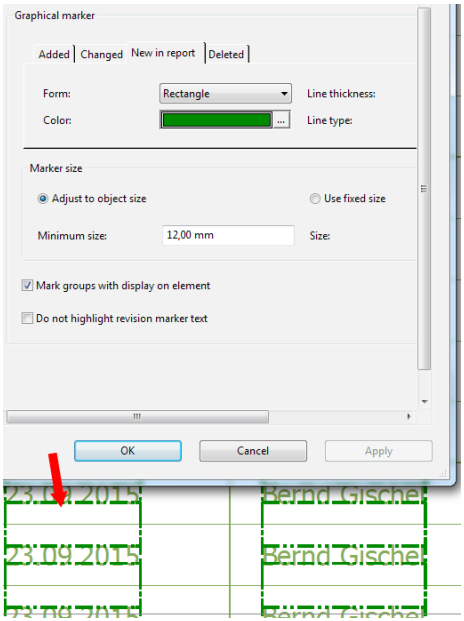


Fig. 7.116 Graphical marker for "New in report"

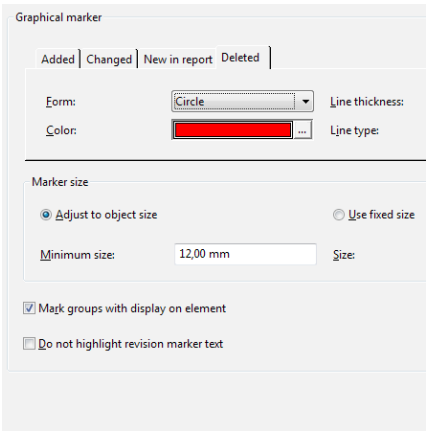


Fig. 7.117 Graphical marker for deleted objects

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.

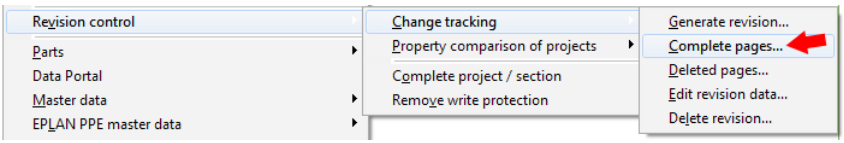


Fig. 7.118 Complete pages

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.

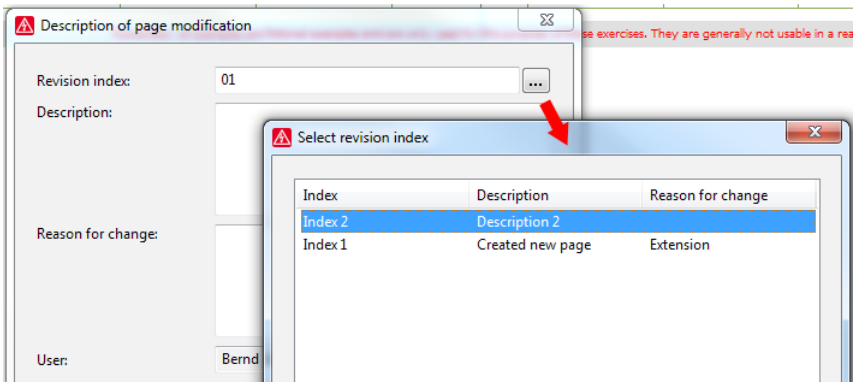


Fig. 7.119 Defining the description of page modification

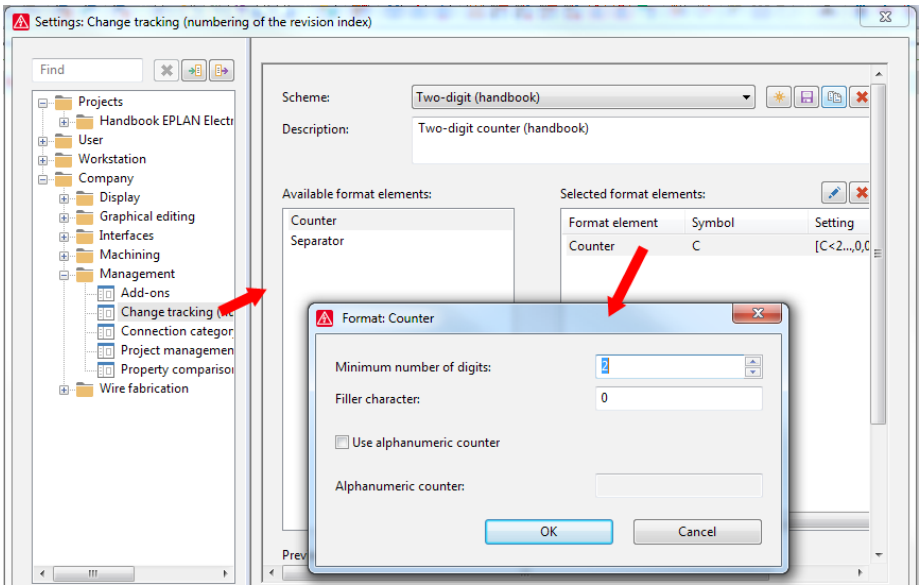


Fig. 7.120 Defining the numbering of the revision index

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.

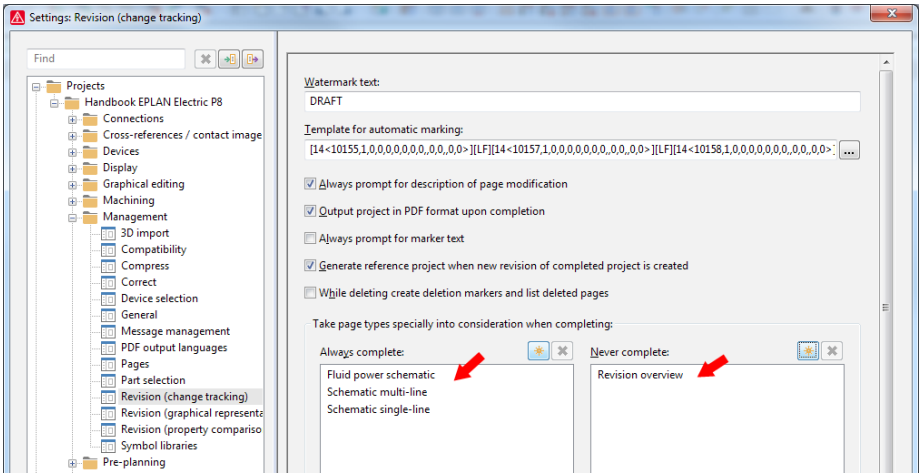


Fig. 7.121 Special handling of page types

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.

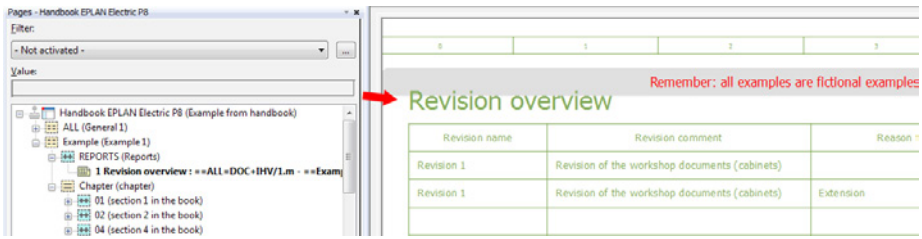


Fig. 7.122 Revision overview report

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.

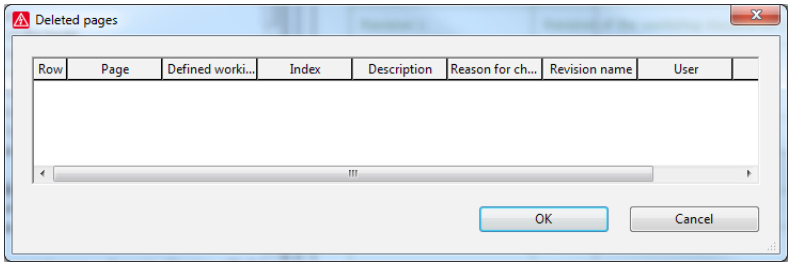


Fig. 7.123 Deleted pages dialog

7.5.6 Complete a 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/AREA** menu item.

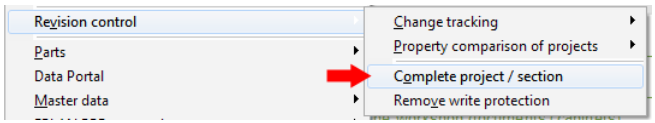


Fig. 7.124 Complete project/area

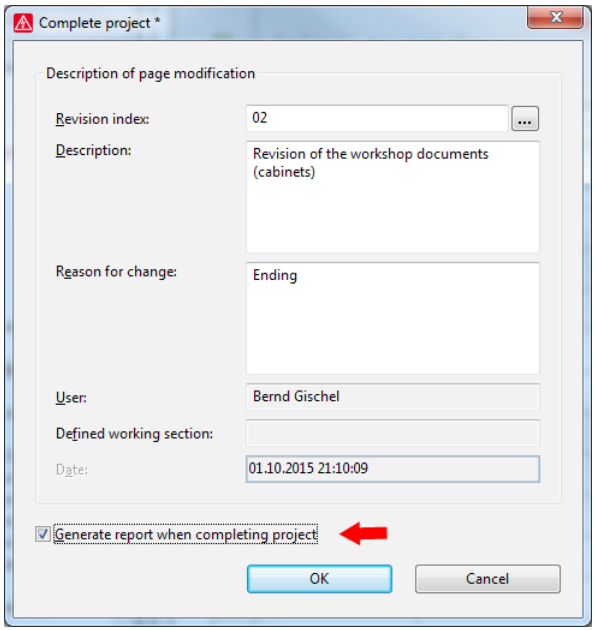


Fig. 7.125 Complete project

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.

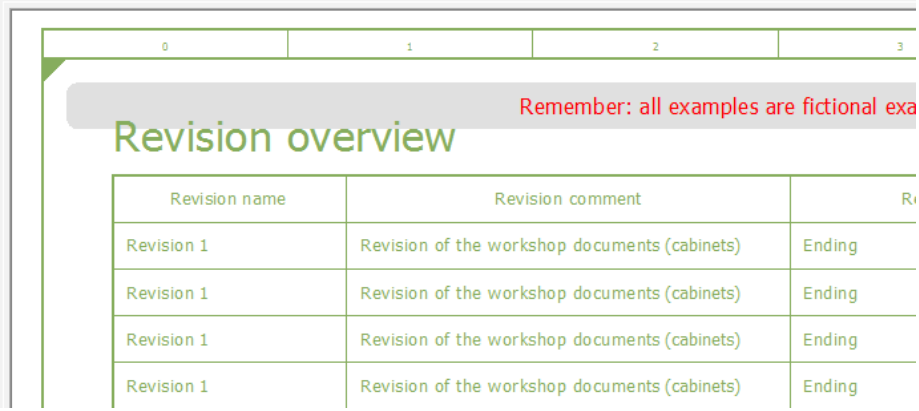


Fig. 7.126 Automatically completed project

A completed project is represented in the page navigator by a corresponding icon.

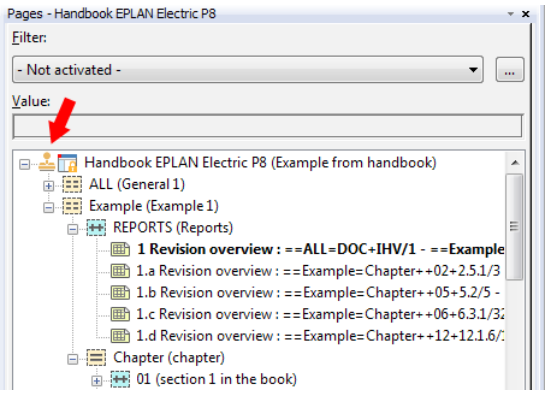


Fig. 7.127 Icon for a completed project

7.6 Project management

The previous sections already mentioned in part the EPLAN project management. This section will help gain a deeper understanding of project management, as well as show further options and functions in EPLAN.

7.6.1 Project management dialog

Project is opened via the **PROJECT/MANAGEMENT** menu item.

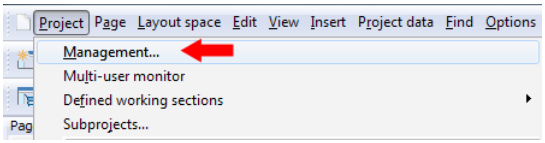


Fig. 7.128 Opening project management

EPLAN then opens the **PROJECT MANAGEMENT** dialog.

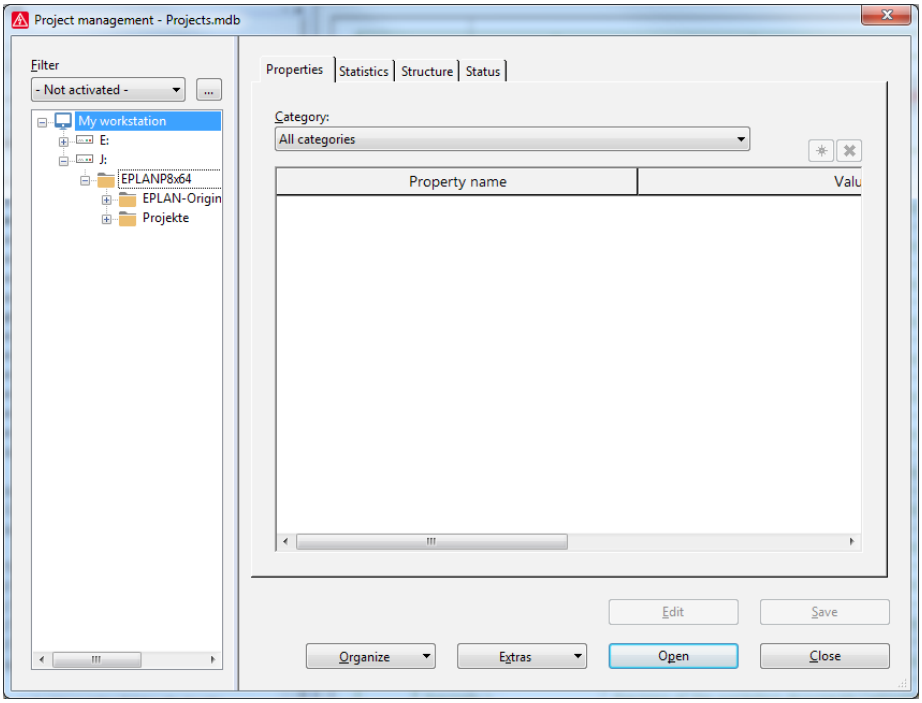


Fig. 7.129 Project management dialog

In the example, that would be the project database in the directory *\\EPLANP8x64 \Projects\Gischel\Projects.mdb*.

The storage location of the project database can be modified in the **OPTIONS/SETTINGS/COMPANY/MANAGEMENT/PROJECT MANAGEMENT DATABASE** setting.

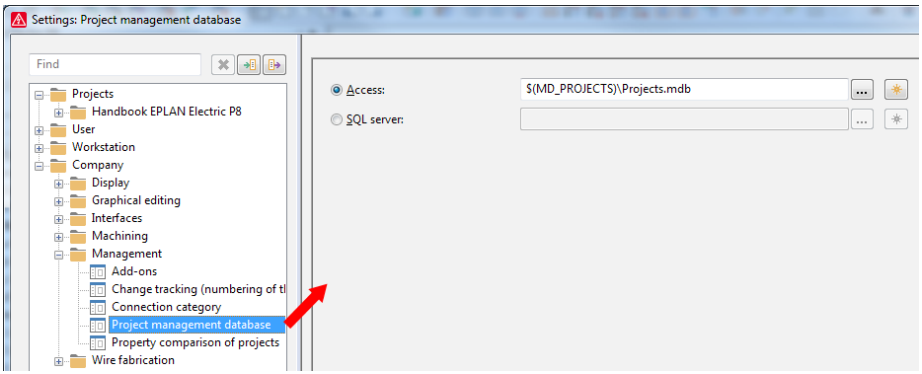


Fig. 7.130 Project management database storage path setting

The **PROJECT MANAGEMENT** dialog itself is divided into two key areas: the left area with an overview of the project directories and the projects on your own workstation (computer), or also on other workstations and the project tree.

Apart from the option to open or close the project tree by clicking the plus sign in front of the project directories, you can also use the *x* key of the numeric keypad to expand the project tree, or the – key to collapse it.

When closing project management, EPLAN saves the most recent representation of the directory tree and starts with the same representation when project management is called the next time.

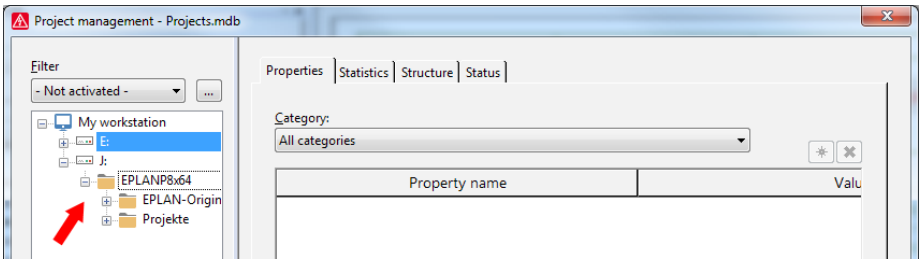


Fig. 7.131 Repeated opening of project management

The right area of the **PROJECT MANAGEMENT** dialog contains, at first sight, a display of information on the currently selected project or projects.

In the lower area, there are the buttons **ORGANIZE**, **EXTRAS**, **OPEN**, **CLOSE**, **EDIT**, and **SAVE**. I will now explain their most important functions.

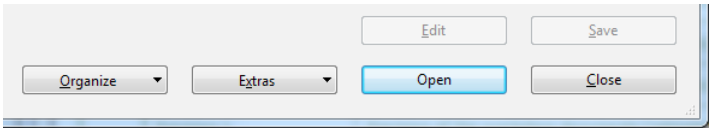


Fig. 7.132 Buttons in the Project management dialog

7.6.2 Project management buttons

7.6.2.1 Edit button

Use the **EDIT** button to configure the project settings of the currently selected project in the window on the left (project tree).

For example, this applies to the properties of the project (text entries) or structures of the various device groups. You should note that different settings, such as the page structure of the project, cannot be changed subsequently.

It is possible to select several projects and to edit their project settings collectively. This function is very similar to block editing and would apply the same changed entry to all projects selected.

- Example 1: Several projects selected, same settings for the selected projects in the user supplementary field 2.

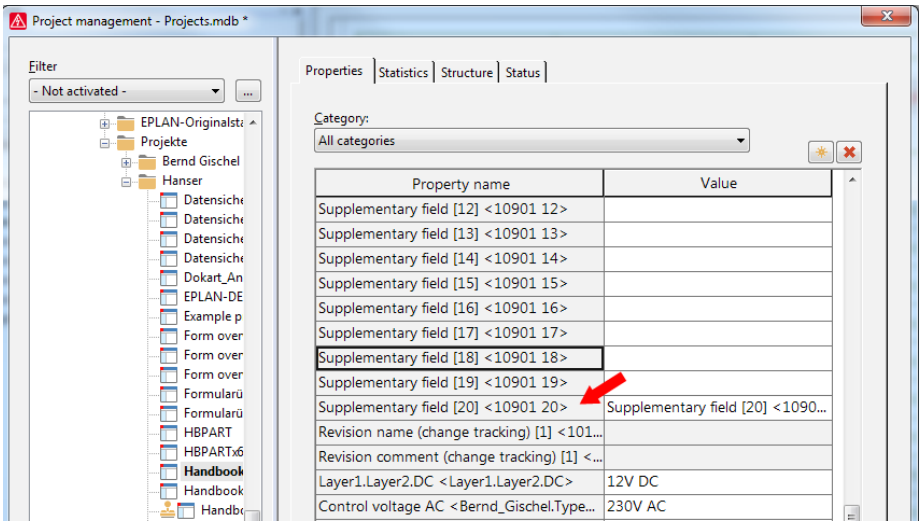


Fig. 7.133 Same properties of projects

- Example 2: Several projects selected, different settings for the selected projects in the user supplementary field 1.

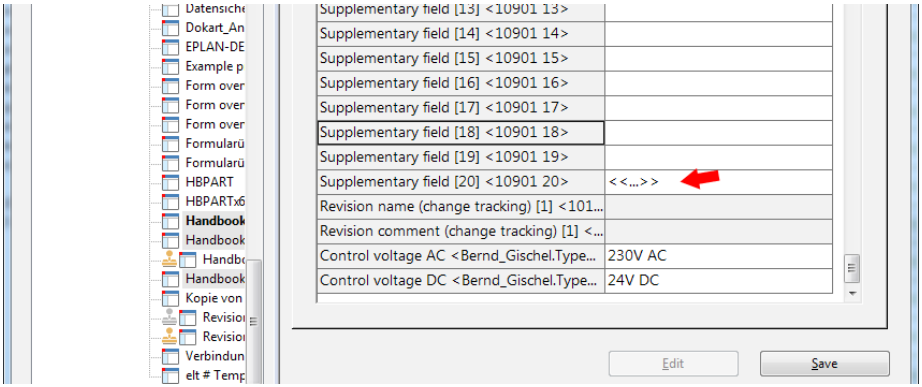


Fig. 7.134 Different properties of projects

While a change in the user supplementary field 2 is clear and unambiguous, a change in the user supplementary field 1 is daring, to say the lest, because the various project entries of the respective projects cannot be differentiated clearly.

Different entries in property fields (not only in project properties, but also overall in EPLAN) can always be identified by the string of characters <<...>>!

At this point, there is no option to undo input after saving (UNDO function)! You should therefore use the *Block editing* function very carefully! Only when all selected projects are supposed to be assigned the same entry can you make an input here without worrying about it.

7.6.2.2 Save button

The **SAVE** button is explained quickly. After changing project settings and/or project properties, you save them using this button.

If you click a different project in the project tree (on the left) after making changes, the changes will be lost, however. EPLAN will prompt you with the **PROJECT PROPERTIES** dialog whether the changes are to be saved.

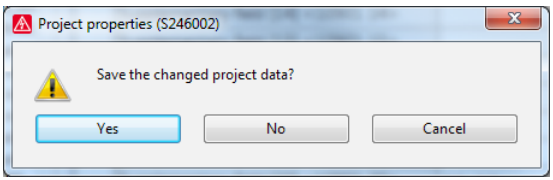


Fig. 7.135 Save project properties prompt

The prompt provides the usual options, such as **YES** (changes are saved), **NO** (changes are discarded, that is, you want to retain the original state of the project settings), or **CANCEL** (you continue to edit the current project settings).

7.6.2.3 Open button

Use the **OPEN** button to open and display the currently selected project(s). Project management is closed automatically in this case.

7.6.2.4 Close button

Use the **CLOSE** button to close project management. EPLAN jumps back to the graphical editor for further project editing of the current project.

Open projects (accessible in the page navigator) are represented in the project tree of project management (the setting depends on the Windows display settings) in a visually different manner.

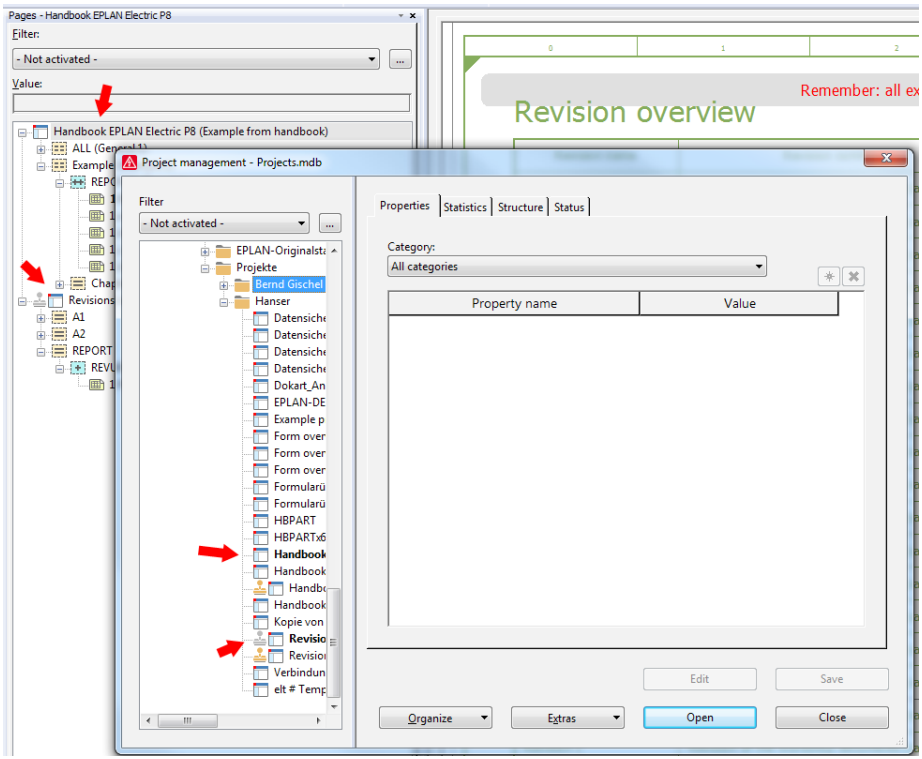


Fig. 7.136 Open projects in project management and in the page navigator

The default setting for the display is bold. In other words, projects already open in the page navigator appear in bold in the project tree of project management.

Projects already open in the page navigator cannot be opened again in EPLAN. If you still attempt it, EPLAN will display a warning that the project is already open.

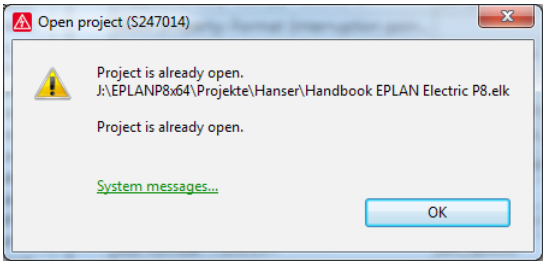


Fig. 7.137
Open project dialog

7.6.2.5 Organize button

In the **ORGANIZE** button, EPLAN provides further functions for the project editing and management of projects by way of a expandable popup menu.

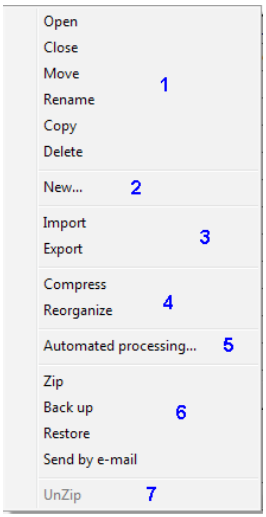


Fig. 7.138
Organize menu

In section **(1)** of the **ORGANIZE** popup menu, you find familiar functions to *open*, *move*, *rename*, *copy* or *delete* projects.

In section **(2)**, new projects are generated from a basic project or template project (as already described).

Projects can be imported and/or exported in section **(3)** of the popup menu. These projects are assigned, or must have, the **.epj* extension, and are available in the XML format.

Section **(4)** contains the functions *Compress* and/or *Reorganize* for projects.

The *Compress* function can be modified by means of configurable project parameters. The project parameters can be found in **SETTINGS/PROJECT/PROJECT NAME/MANAGEMENT/COMPRESS**. Here you can define many actions for compressing the respective project by using a scheme.

The *Reorganize* function is used to optimize the database. No other settings, such as via parameters, are possible here.

Section **(5)** contains the *Automated processing* function. This allows you to use ready-made schemes, or create schemes for automatic editing by calling the *New scheme* function.

This is done by way of the familiar procedure for schemes. The **SETTINGS...** button opens the **SETTINGS: AUTOMATED PROCESSING** dialog. Here you can, in the usual manner, generate and then define a new scheme with the desired actions. You can also import external schemes.

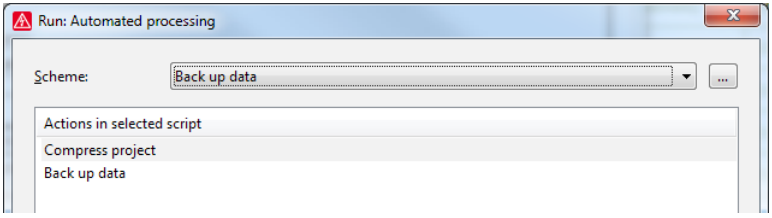


Fig. 7.139
Automated processing, scheme selection

Section **(6)** of the **ORGANIZE** popup menu in project management contains functions of *data backup* and *data restoration*, such as packing or backing up projects, or the *e-mail function*. The e-mail function requires that a functioning e-mail client and associated profile of the user have been set up.

In section **(7)**, a packed project (file extension *.elp) can be opened via the **UNPACK** command.

A packed project can also be opened in the project tree by double-clicking. EPLAN unpacks the project automatically, and then you can continue editing as usual.

Many of the functions accessed via the **ORGANIZE** popup menu can also be executed using the popup menu on the left (project tree).

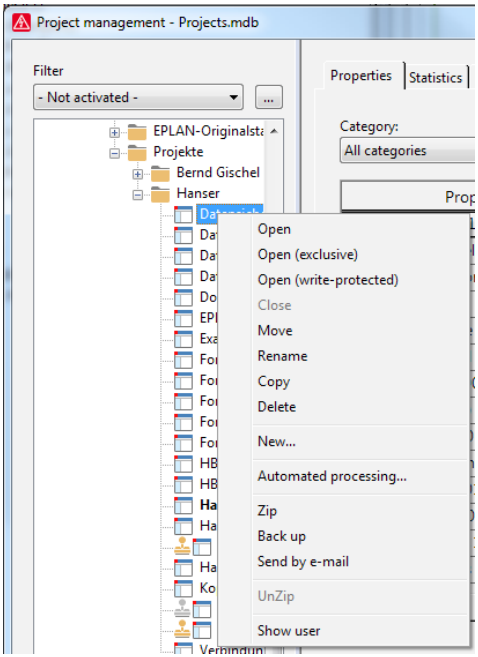


Fig. 7.140
Popup menu of the right mouse button

7.6.2.6 Extras button

That leaves us only with the **EXTRAS** button. This one also includes a number of popup menu functions, which will be explained briefly.

These are functions of the project database, as well as functions for comparing or reading project information.

Section **(1)** of the **EXTRAS** popup menu contains functions to complete the project database, import completely new project directories, or generate a project list.

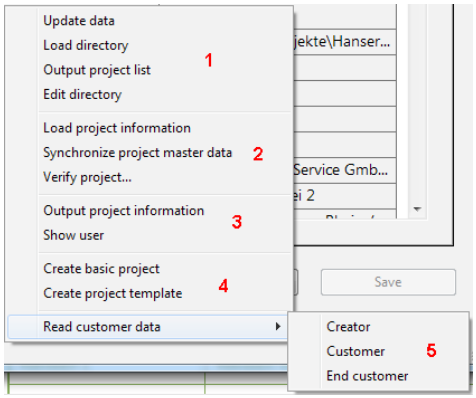


Fig. 7.141
Extras menu

Section **(2)** contains commands for reading out project information or, for example, for comparing a project with the set company parameters.

Among other things, section **(3)** provides an overview of which users are still in the selected EPLAN project.

Section **(4)** contains the familiar functions Create basic project and/or Create project template.

In section **(5)**, customer data can be imported from parts management into project properties. Select the project in question and open the *Read customer data* menu item. Then, you

can click one of the menu items, *Creator*, *Customer* or *End customer*, to transfer the data from parts management (customer data) to project properties automatically.

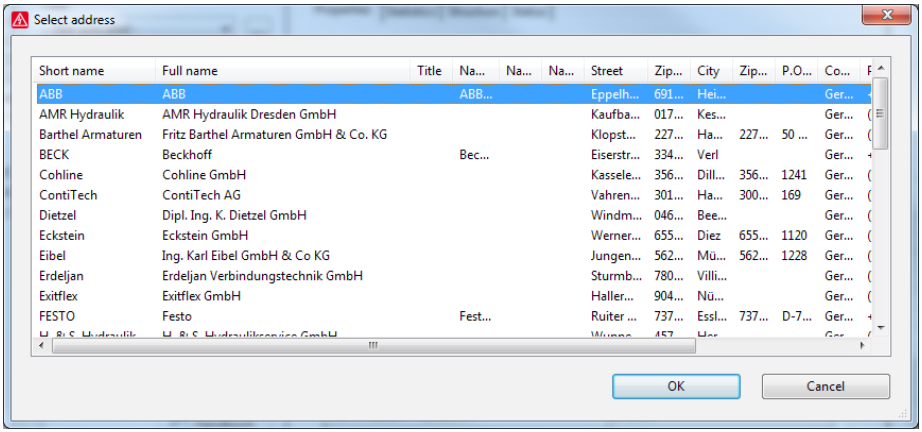


Fig. 7.142 Select address dialog

7.6.3 Project management filters

Over time, many projects are created. So as not to lose track, or to display only specific projects quickly, you can create filters in project management with a variety of possible report criteria via the project directories.

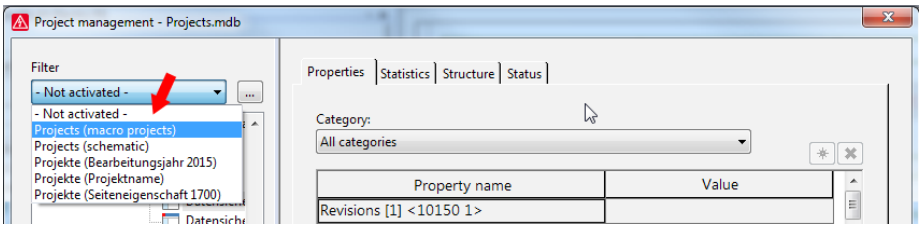


Fig. 7.143 Filter selection

You can select existing filter schemes in project management (upper left) using the *Filter* drop-down list. Of course, you can also create your own filters in the usual way.

If you have selected a filter scheme, but no projects are displayed, it is very likely that the criteria are not compatible with the set projects and/or project directories.

7.6.4 Create projects (project management)

Creating a new project is identical to creating one in the graphical editor. EPLAN works the same way here.

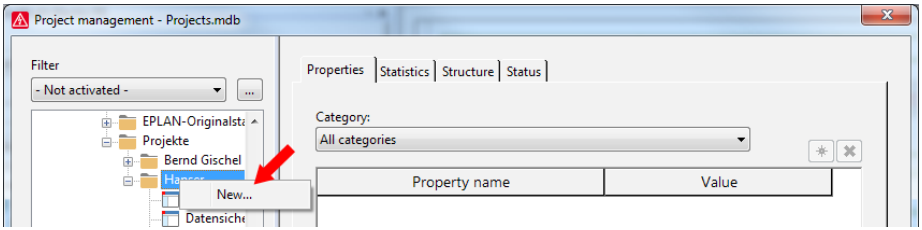


Fig. 7.144 Create new project in project management

In the left area of project management, you can use the popup menu or right mouse button to create a new project via the **NEW** menu entry and, in the subsequent dialog, on the basis of a project template or a basic project.

The further procedure, then, is identical.

7.6.4.1 Copy projects (project management)

Of course, apart from creating new projects, you can also copy projects in project management, and then adapt them to the required entries. EPLAN provides various options for copying a project.

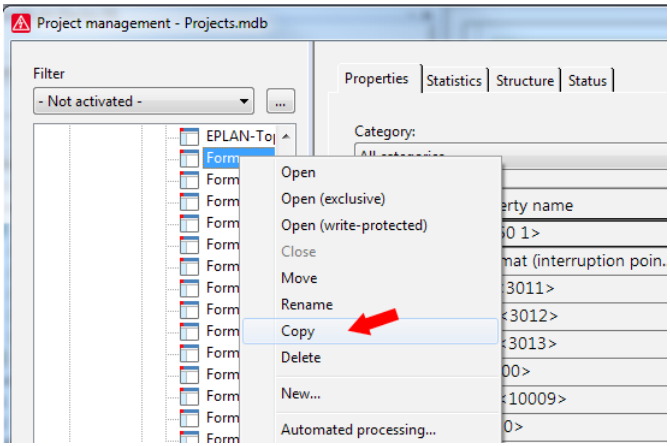


Fig. 7.145 Copy (project) popup menu

In the left area of the **PROJECT MANAGEMENT** dialog, select the project to be copied. Then, using the popup menu (right mouse button) or the **ORGANIZE** button, start the *Copy* function. EPLAN opens the **COPY PROJECT** dialog.

The **COPY PROJECT** dialog contains various options regarding how and to what extent the selected project is to be copied, and whether a specific creation date (project property 10021 creation date) and/or a specific creator (project property 10020 creator) is to be entered in the project copy automatically.

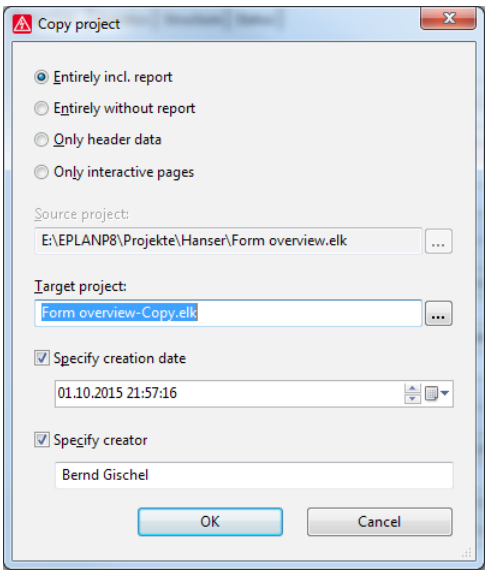


Fig. 7.146 Copy project dialog

You can also select and copy several projects in EPLAN. In the process, the **COPY PROJECT** dialog opens several times one after the other.

7.6.4.1.1 Entirely with report setting

The entire project, including header data, such as page structure and device structure, is copied together with all reports generated. This option generates an identical project copy.

7.6.4.1.2 Entirely without report setting

The entire project is copied, but without any existing (graphical) reports generated. After a project copy has been created, the reports must then be regenerated.

NOTE: With this copy option selected, any title page or cover sheet manually created will not be applied as a report either.

7.6.4.1.3 Only header data setting

The project is copied only with the existing header data, such as page structure and device structure. No existing pages or existing reports will be copied. The subsequent copy of the project is empty, which means that this project does not contain any pages.

7.6.4.1.4 Only logic pages setting

The project is copied with the existing header data, such as page structure and device structure, and also includes all logic schematic pages.

These include, for example, the following page types: External document, Graphic, P & I diagram, Schematic multi-line, Schematic single-line, Schematic (Fluid power), Panel layout, and Overview.

However, unplaced items or reports are not copied.

NOTE: Unplaced items are items that have not been placed in the project yet, that is, they have not been drawn yet and therefore have no graphical reference. They exist only in lists or in the navigators as devices – for example, unplaced terminal strip definitions.

7.6.4.2 Copy projects (Explorer)

Apart from the EPLAN copy function projects can also be copied “externally”. But this usually involves the mistake, for example, of only copying the project folder *.edb and forgetting about the project file *.elk for starting the project.

I strongly advise against it, because EPLAN cannot track such external copy actions automatically, nor does it register them in the project management database, for example. Projects should always be copied using the available EPLAN functions, and always within the EPLAN platform.

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

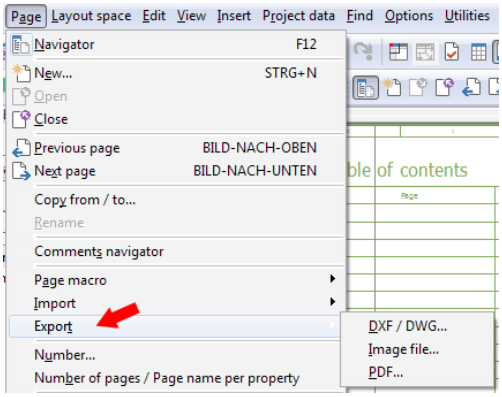


Fig. 8.1 Menu of miscellaneous export options

Import menu

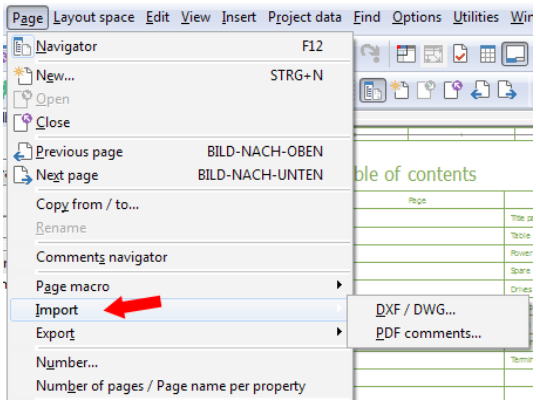


Fig. 8.2 Menu of miscellaneous import options

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 (via 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 and import 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).

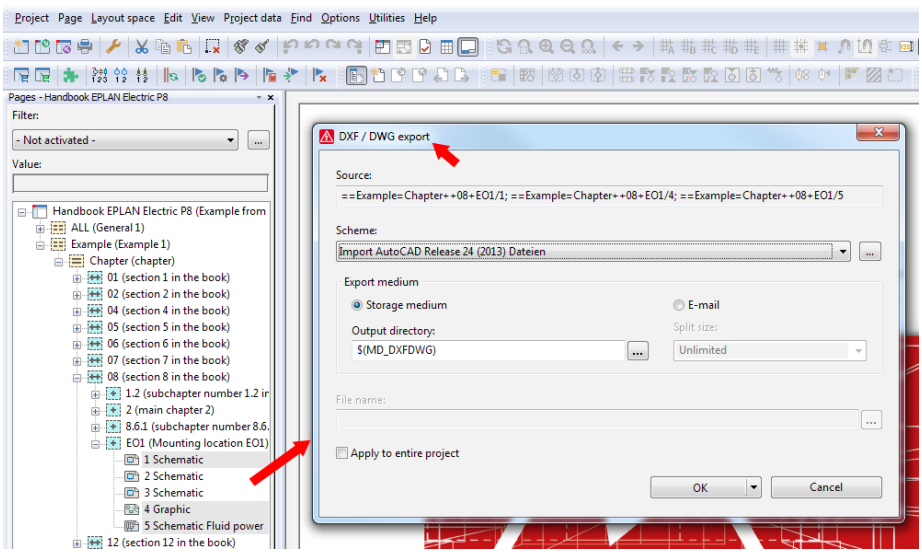


Fig. 8.3
Export pages to DXF/DWG format dialog

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.

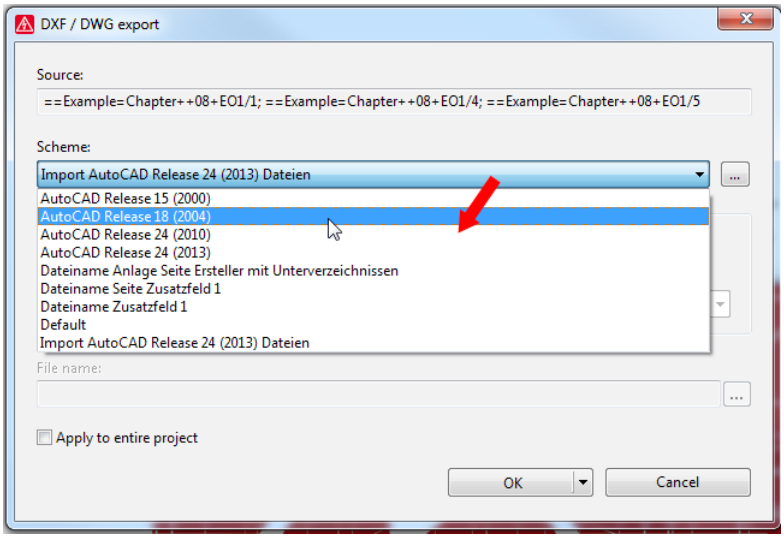


Fig. 8.4
Schemes can be adjusted in any manner

The **[...]** button opens the **Settings: DXF/DWG export and import** dialog, where the current scheme can be edited.

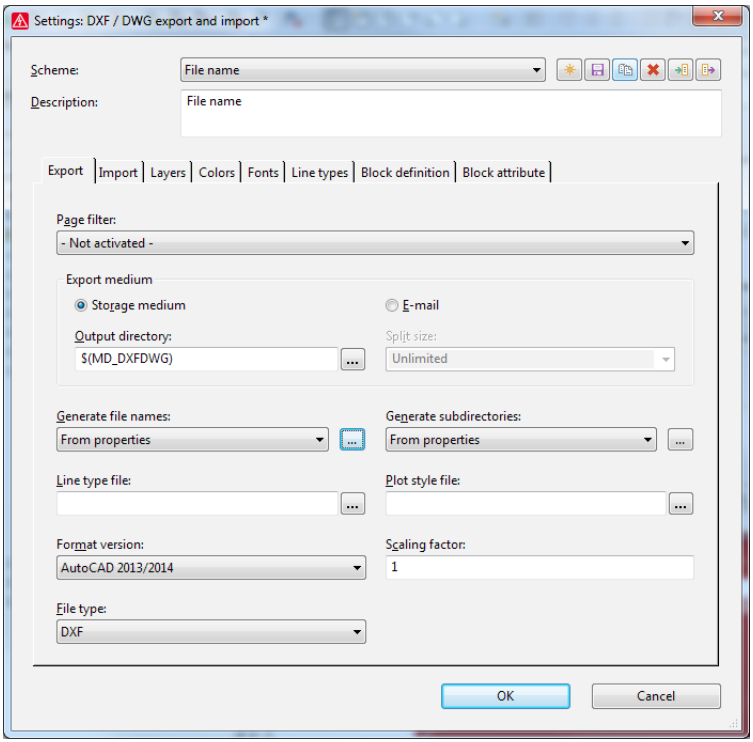


Fig. 8.5
Settings for exporting DXF/DWG files

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**.

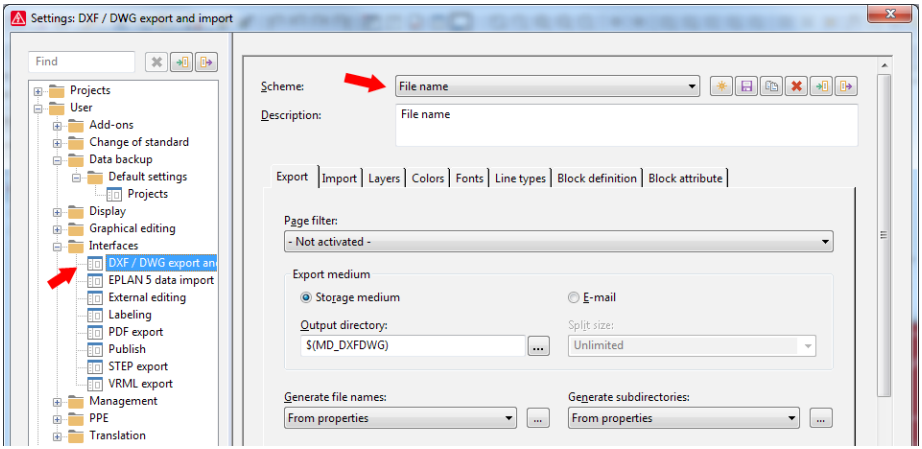


Fig. 8.6 Settings for exporting DXF/DWG files

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 (as usual, 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 in the current scheme 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.

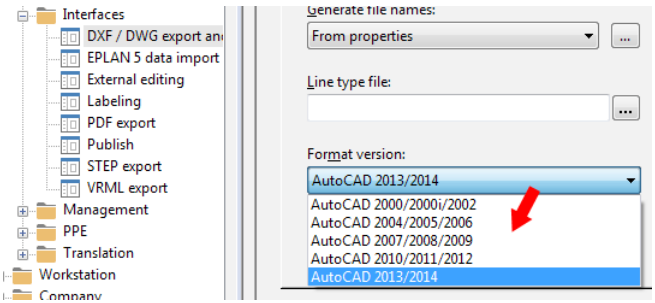


Fig. 8.7 Selection of formats

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.

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).

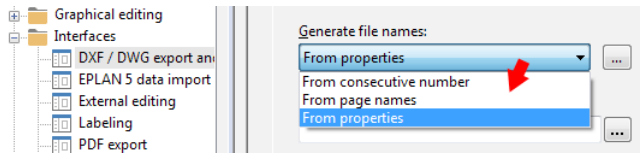


Fig. 8.8 File name options

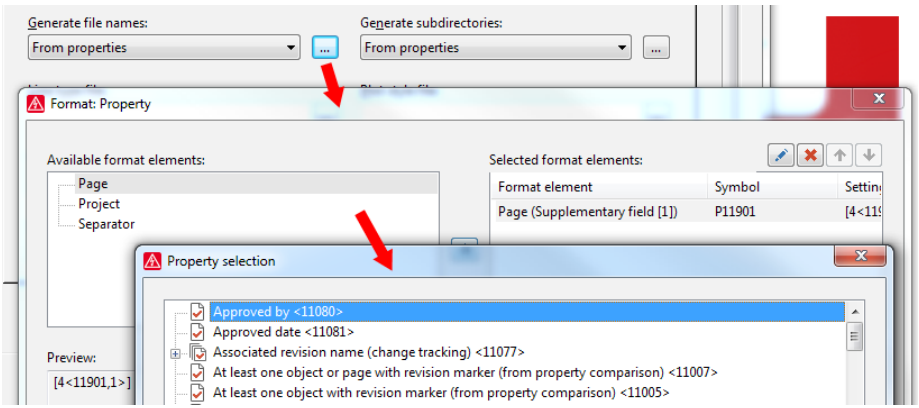


Fig. 8.9 Generating file names from properties (format elements)

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.

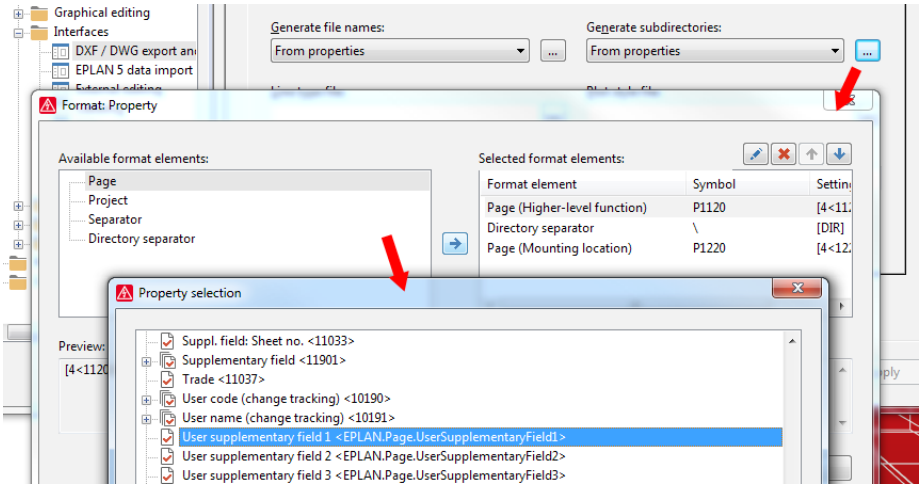


Fig. 8.10 Option of output in subdirectories

The two remaining settings, Line type file (*.lin) and Plot style file (*.ctb), are provided by the target system and must exist as files for EPLAN so that they can be included in reports as well.

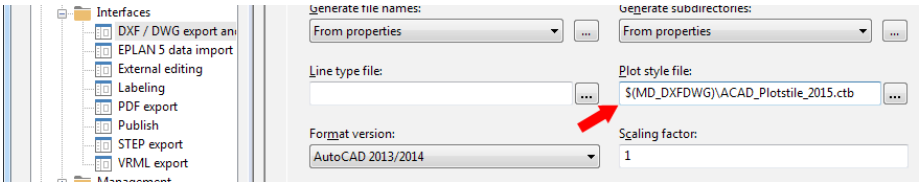


Fig. 8.11 Options to let the target system define types/styles

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 the selected pages 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.

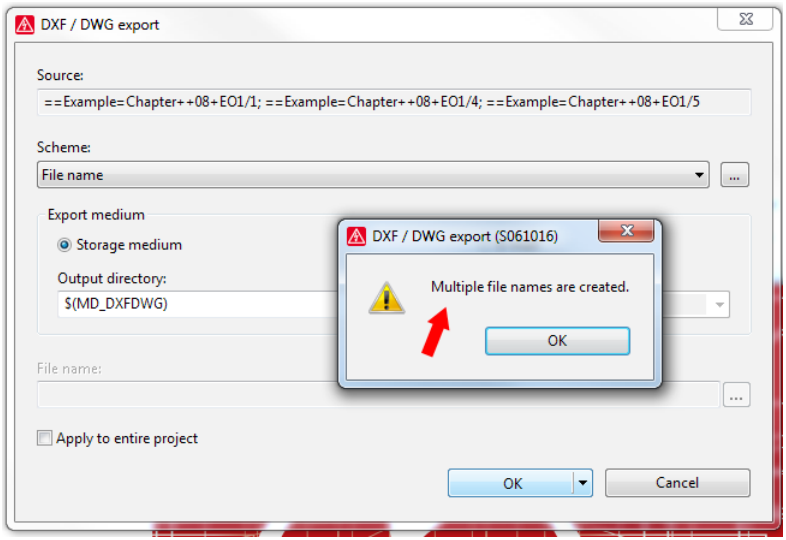


Fig. 8.12 Error message when generating the output

If the output goes through without any problems, the output will be generated. EPLAN then closes all dialogs, and again focuses on graphical editing.

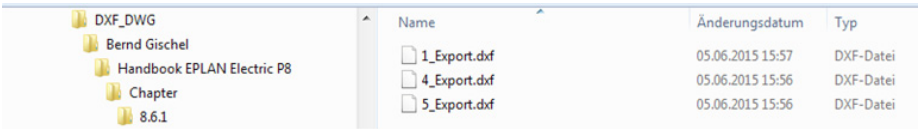


Fig. 8.13 Generated DXF files in the stored directory structure

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.

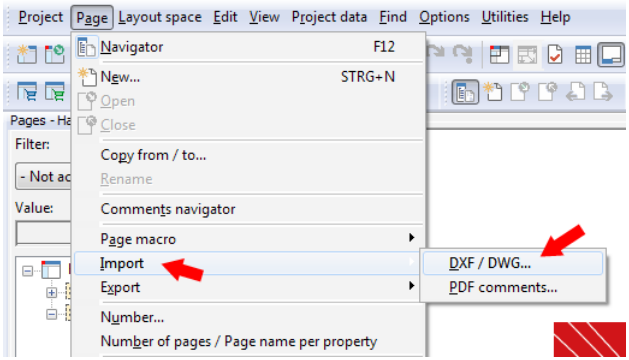


Fig. 8.14 Import DXF/DWG files

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.

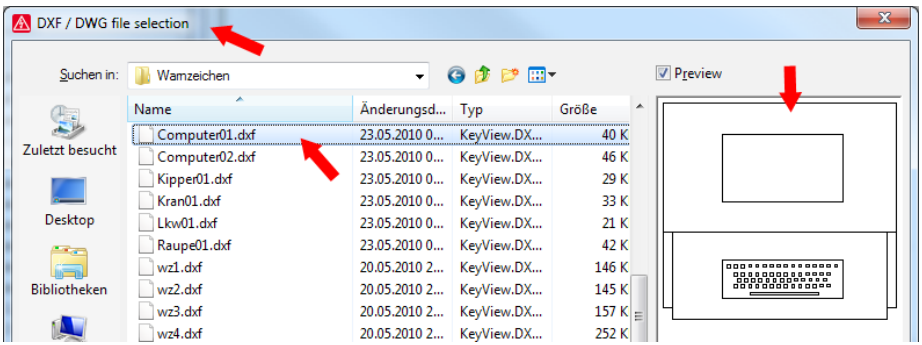


Fig. 8.15 Dialog for importing DXF or DWG files

The desired file type for the import (DXF or DWG) is set in the lower area of the dialog.

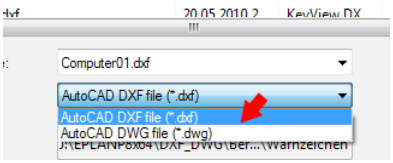


Fig. 8.16 Selection of the file type

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.

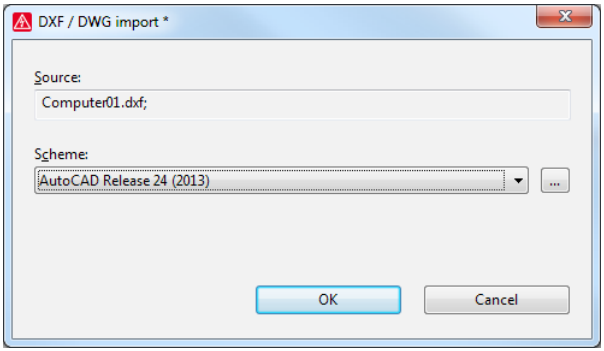


Fig. 8.17 Import dialog

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.

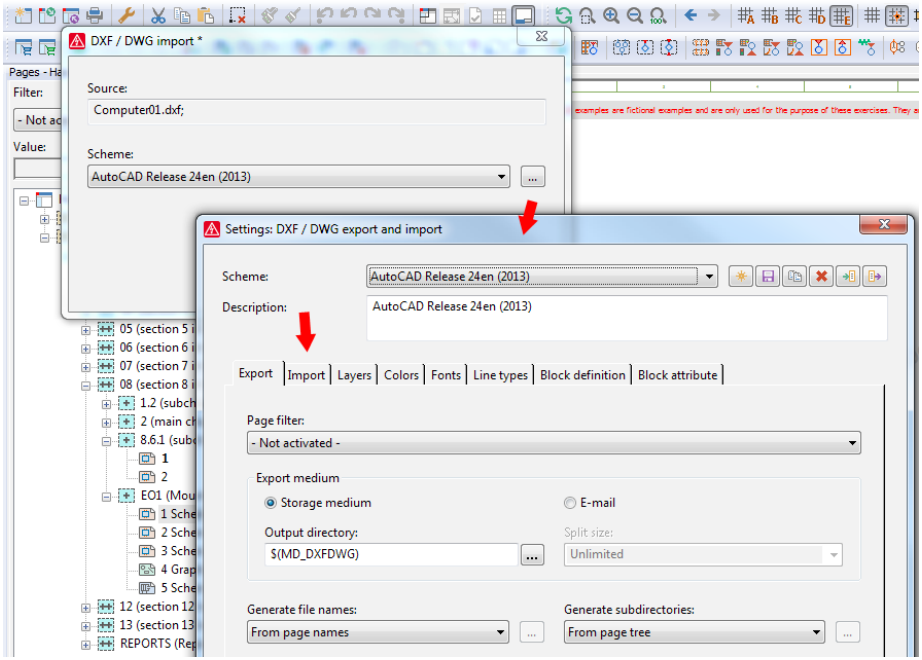


Fig. 8.18 Settings for importing DXF/DWG files

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.

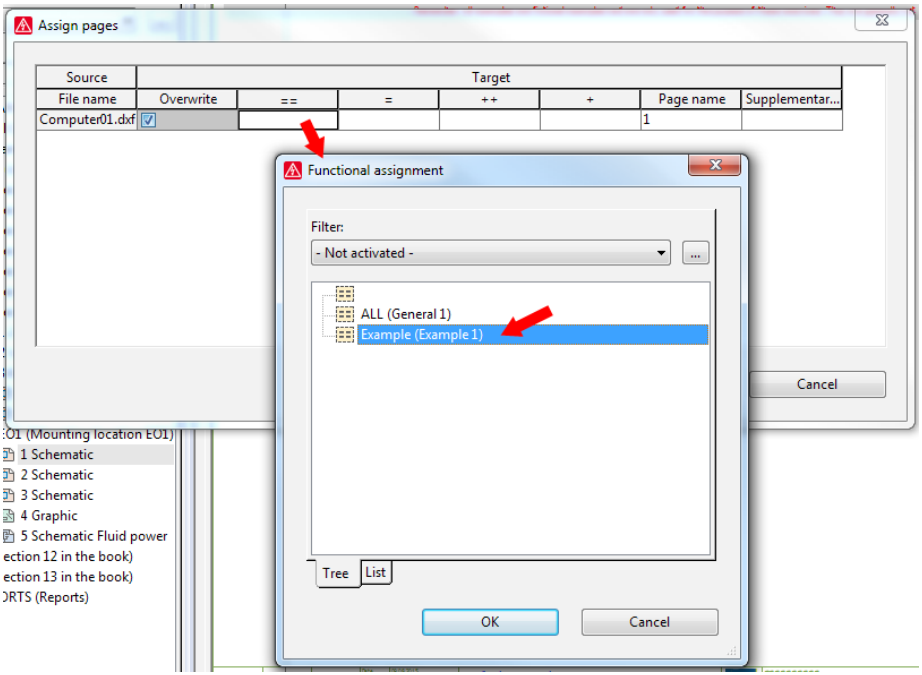


Fig. 8.19 Assign pages prior to the actual generation of the project pages

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.

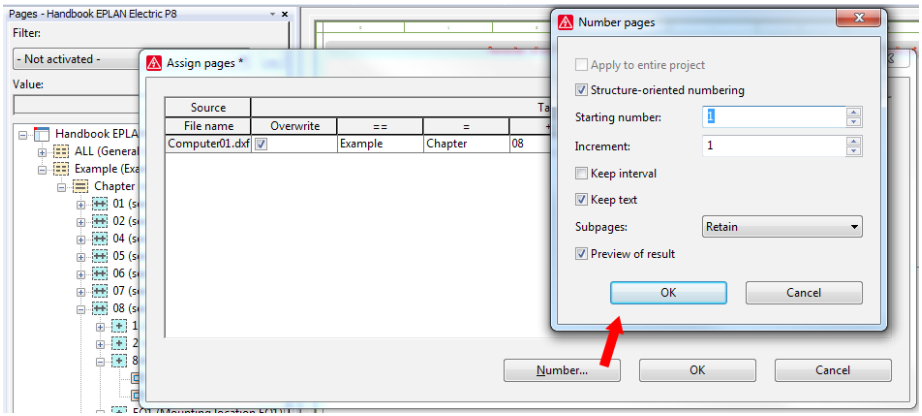


Fig. 8.20 Numbering pages automatically

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.

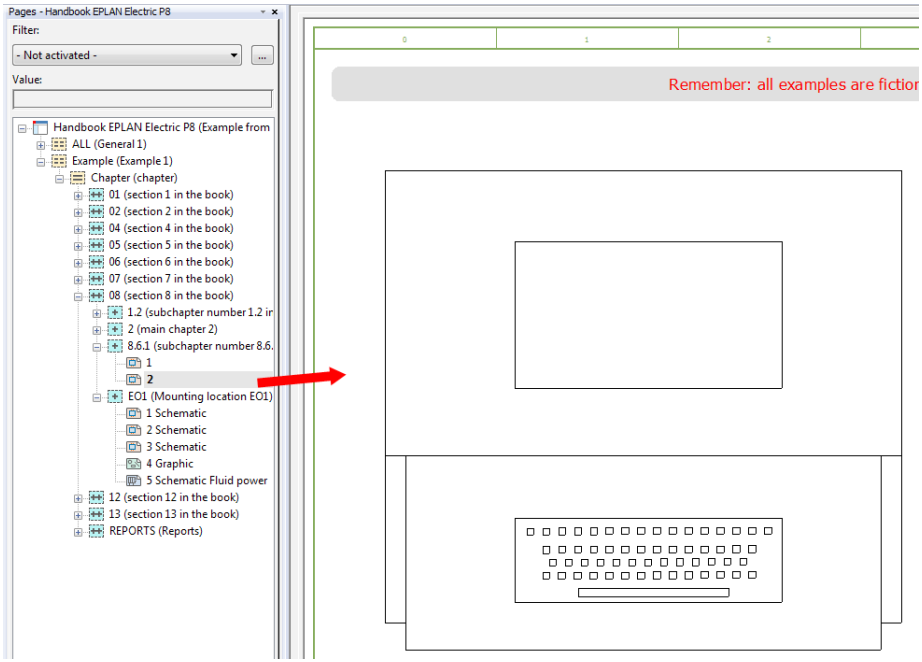


Fig. 8.21 Read-in example of a DXF file

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.

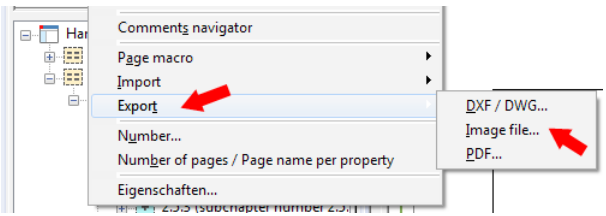


Fig. 8.22 Page / Export / Image file menu

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.

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.

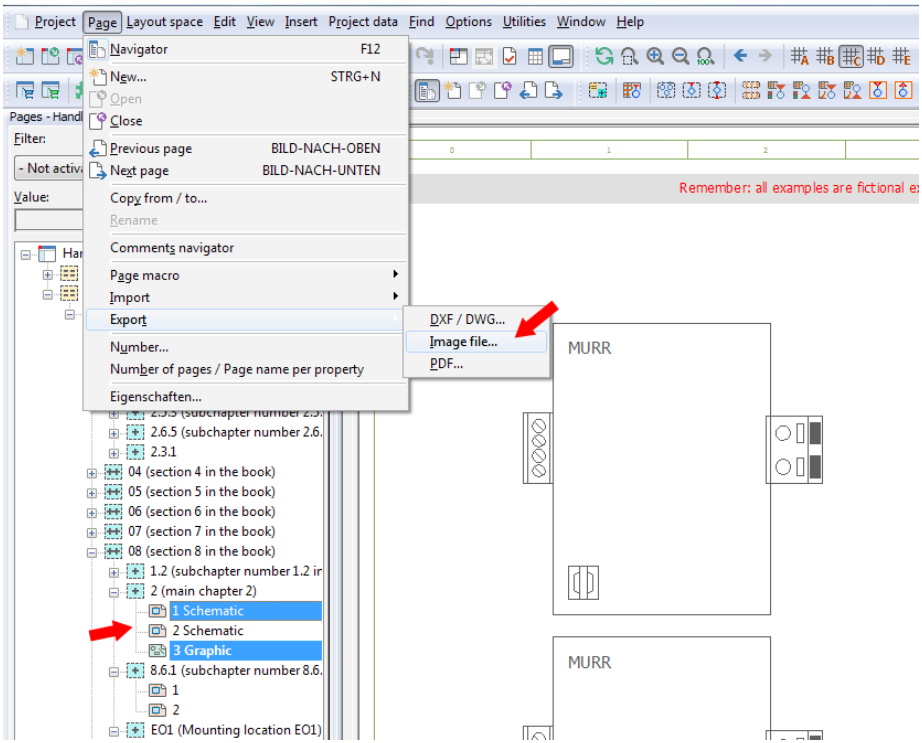


Fig. 8.23 Selected pages in the page navigator

You use the **PAGE/EXPORT/IMAGE FILE** menu to open the **Image file export** dialog.

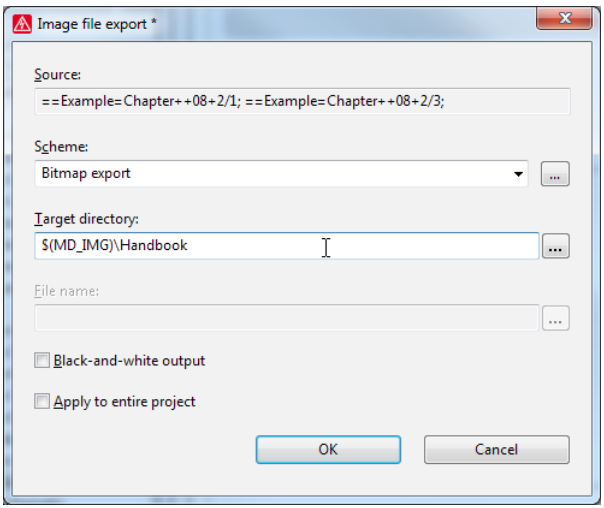


Fig. 8.24 Image file export dialog

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.

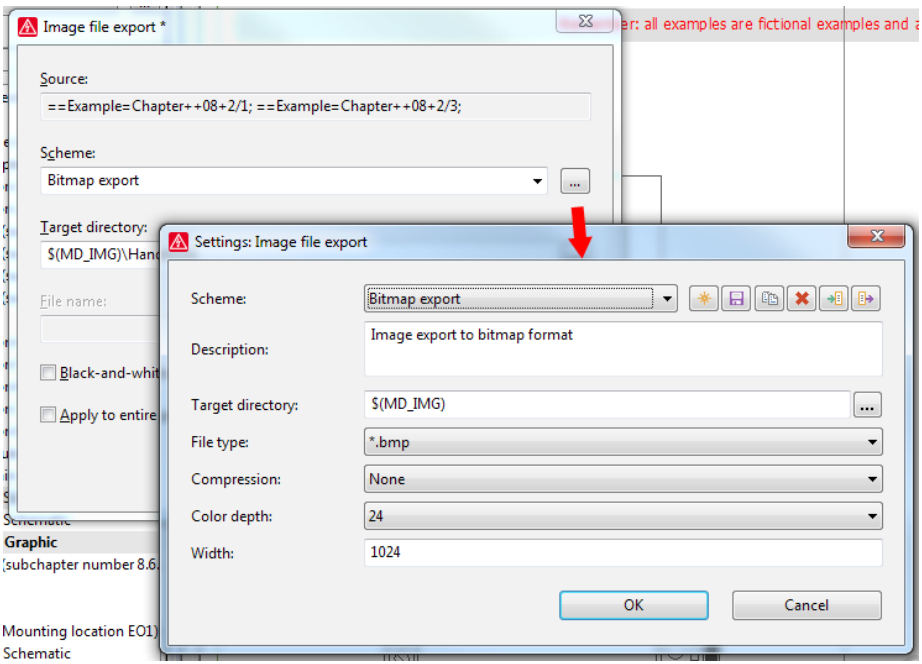
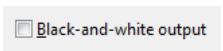


Fig. 8.25 Settings for image file export

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.

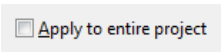


The first option affects the output of the pages themselves: Should the output be generated in color or black and white? Selecting **Black and white output** generates image files without color information.

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 directory* 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.

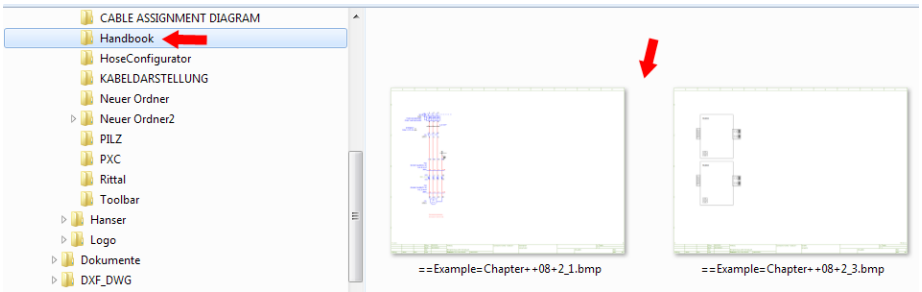


Fig. 8.26 Exported schematic pages as images

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.

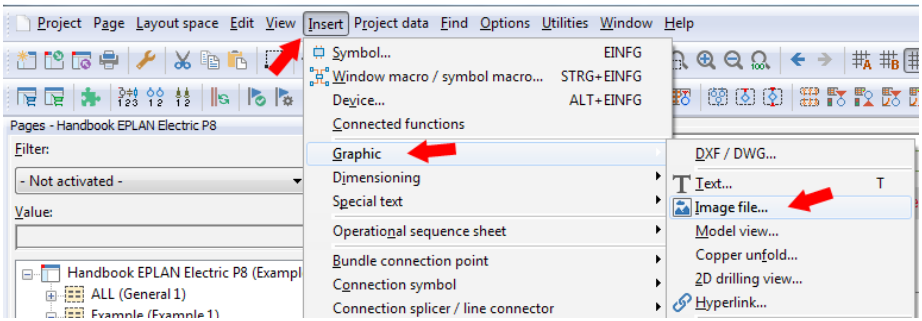


Fig. 8.27 Insert image files menu

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.

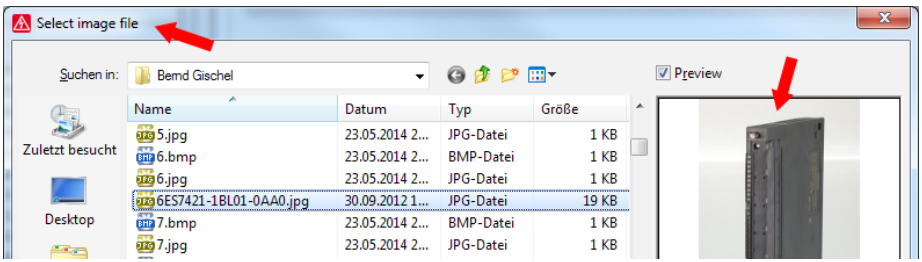


Fig. 8.28 Dialog for the selection of image files

EPLAN can insert numerous different image formats. The corresponding image format is preselected in the **Select image file** dialog in the lower area.

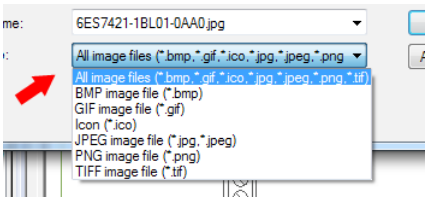


Fig. 8.29 Selection of the image format

File types such as **.bmp* and **.jpg* are the most common, but space-saving formats such as **.png* are also possible. There are no known limitations on the directory structure or file names used. Special characters permitted by the operating system 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.

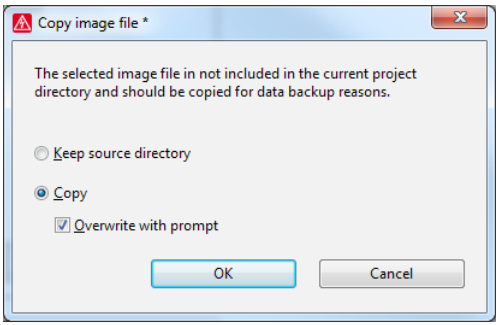


Fig. 8.30
Copy image file

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.

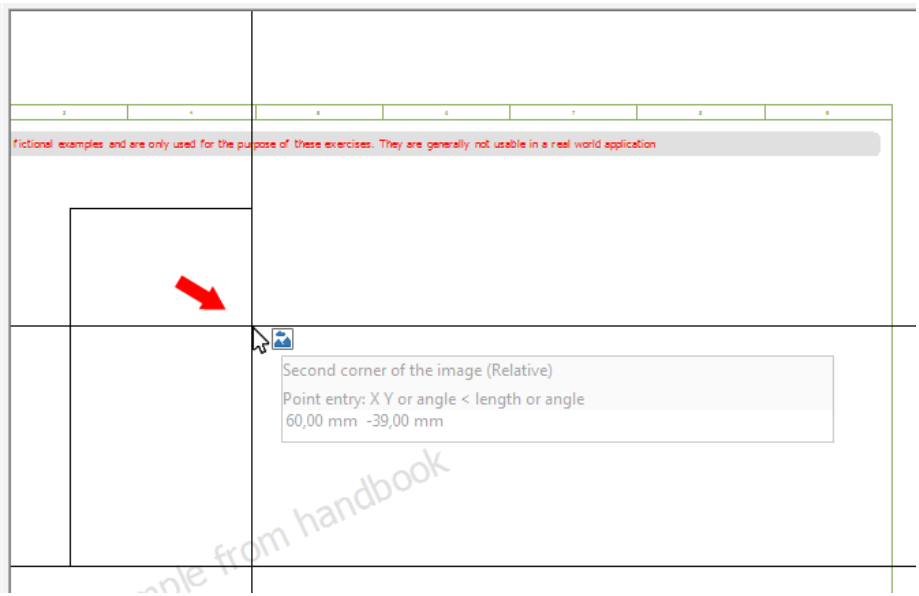


Fig. 8.31
Pulling open a window

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.

The *Keep aspect ratio* setting is important. This avoids distortion of the image. This setting should always be activated.

Properties of an image

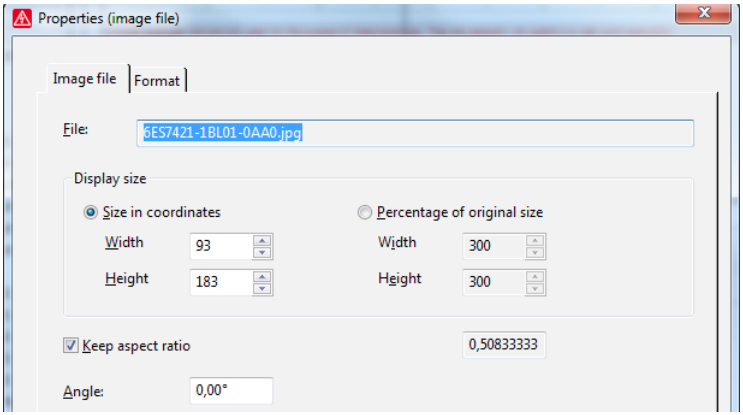


Fig. 8.32 Formatting options dialog

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.

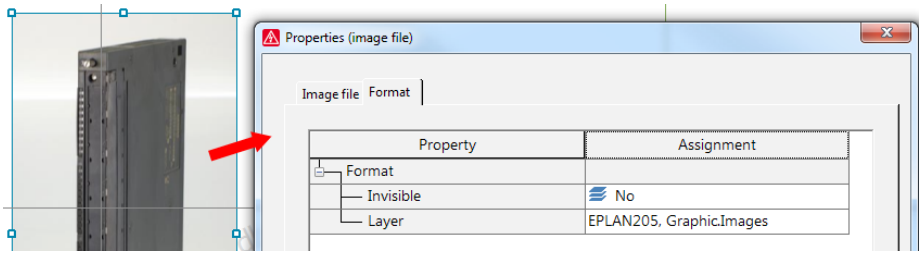


Fig. 8.33 Properties dialog

This function for calling up the properties also exists in the right-click popup menu.

8.3 Print

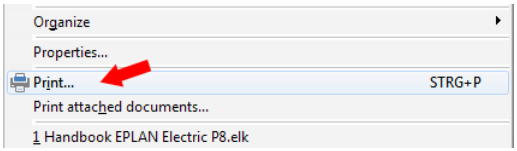


Fig. 8.34 The Print menu

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.

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.

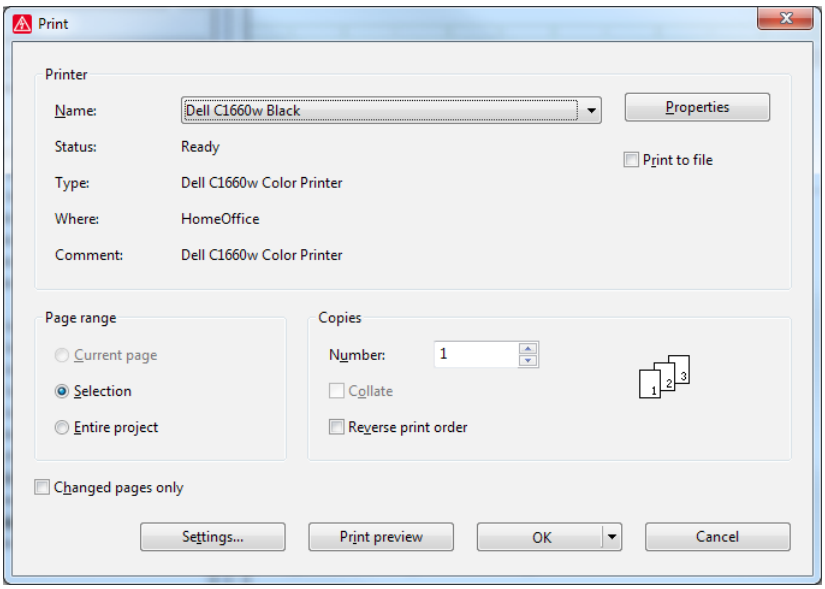


Fig. 8.35 The Print dialog

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.

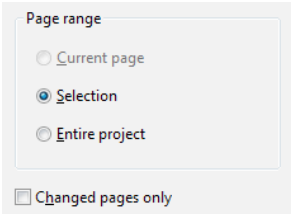


Fig. 8.36
Print options

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.

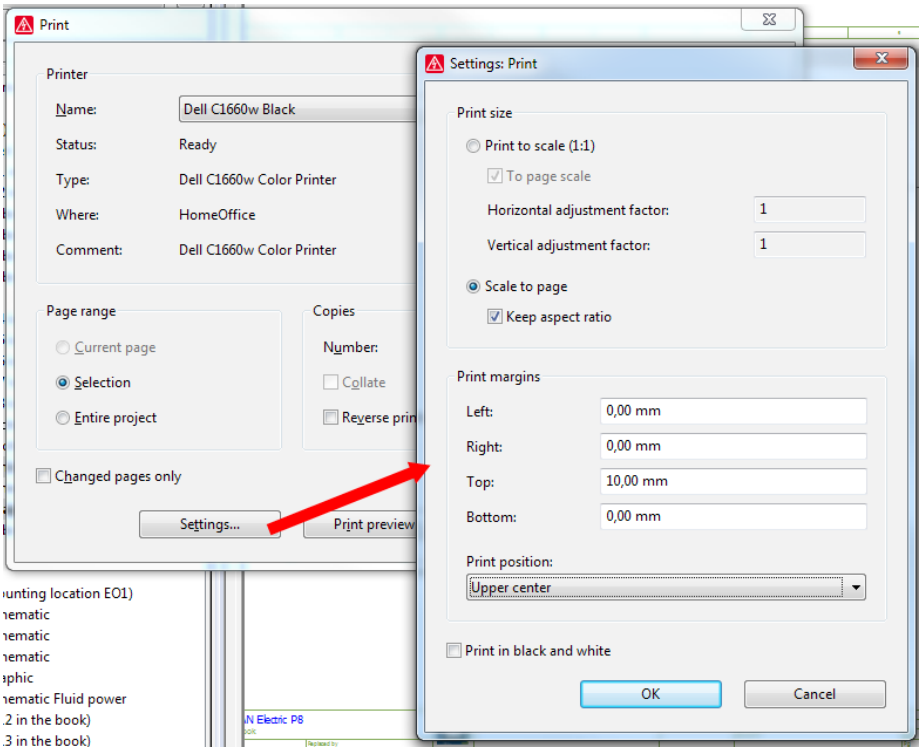


Fig. 8.37 More page settings

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.

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

The **Print preview** button in the **Print** dialog provides a rough preview before performing the actual printing.

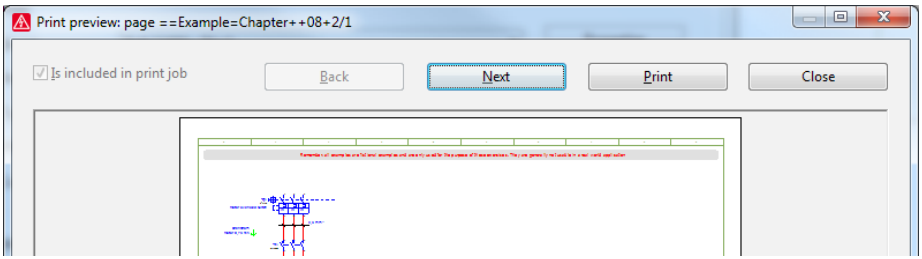


Fig. 8.38 Print preview

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.

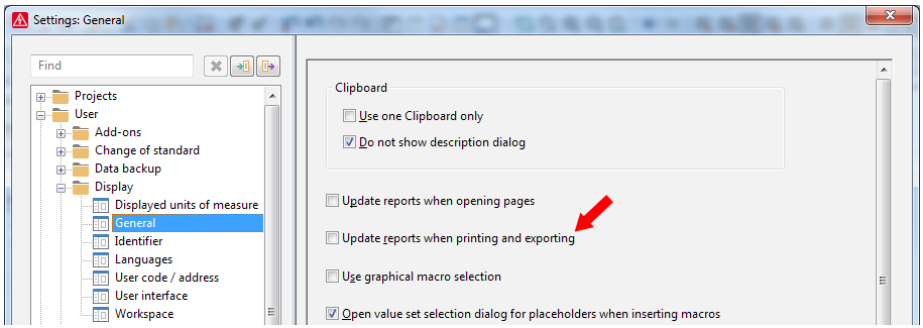


Fig. 8.39 Update reports when printing/exporting settings

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.

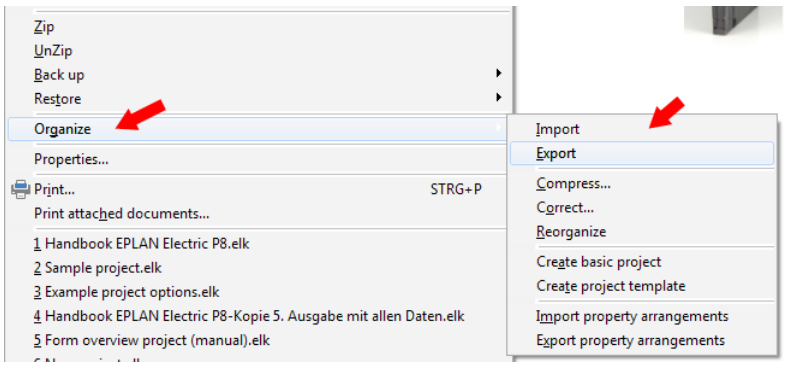


Fig. 8.40 Import and export project data

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.

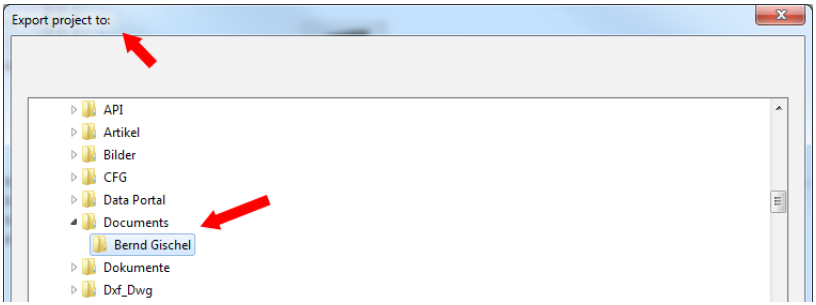


Fig. 8.41 Defining the export folder

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. EPLAN maps the progress in the *Export project* dialog. 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.

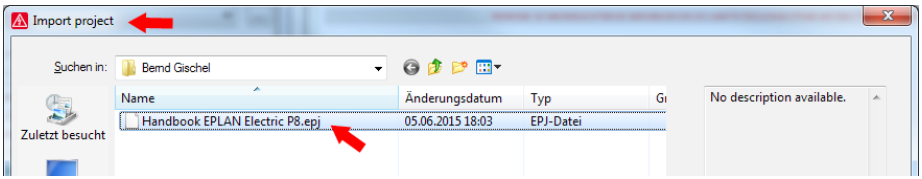


Fig. 8.42 Importing a project (**.epj*)

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.

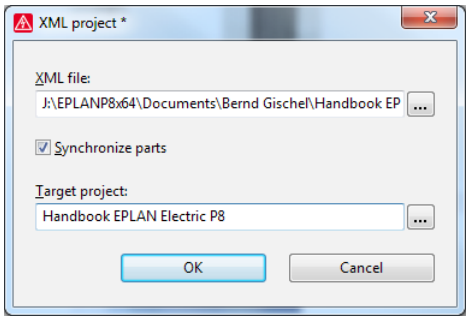


Fig. 8.43 Importing the EPL(XML) project

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 target directory for the project to be imported. After you click **OK**, the import process starts, and EPLAN generates the project in the defined target directory.

8.5 Print attached documents

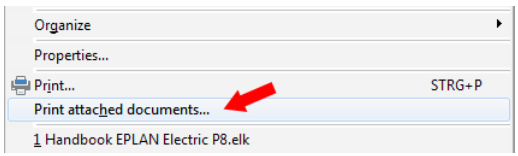


Fig. 8.44 Print attached documents

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.

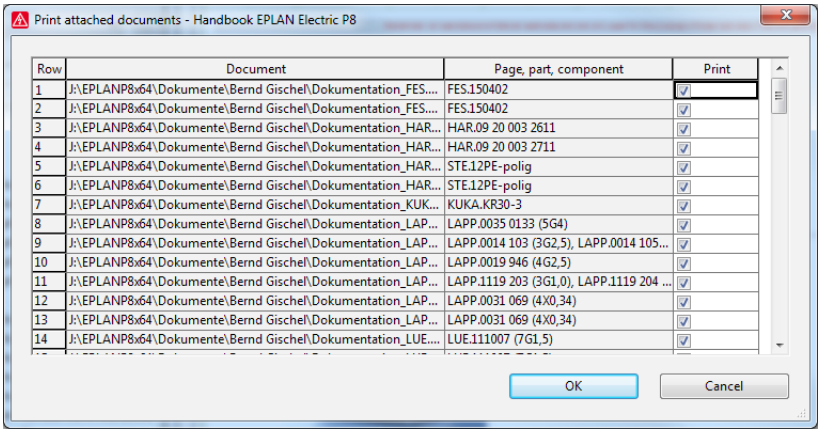


Fig. 8.45 Dialog for selection of attached documents

When you click the **OK** button, EPLAN sends all documents selected in the **Print attached documents** dialog, one after another, to the connected printer with a prompt for the printer to be selected.

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.

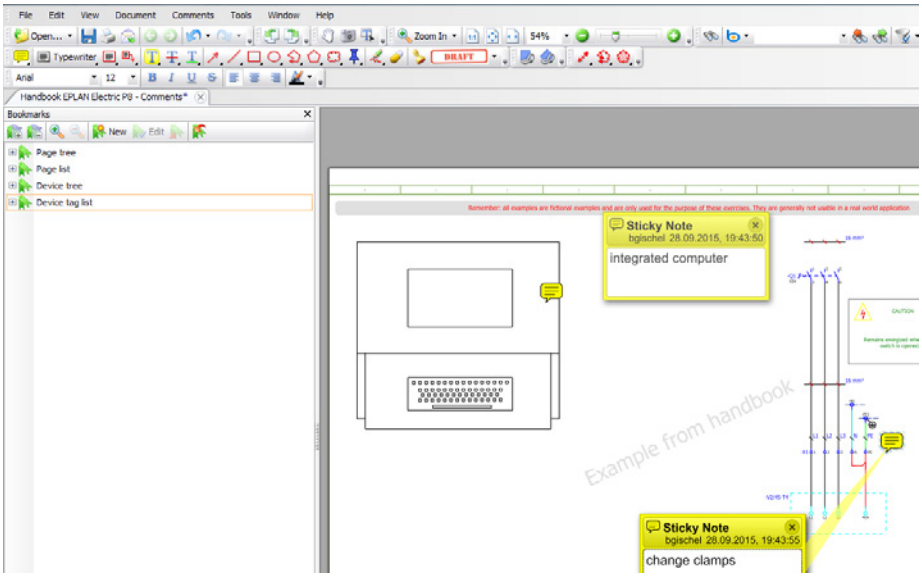


Fig. 8.46
Example of comment entries on a schematic page

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.

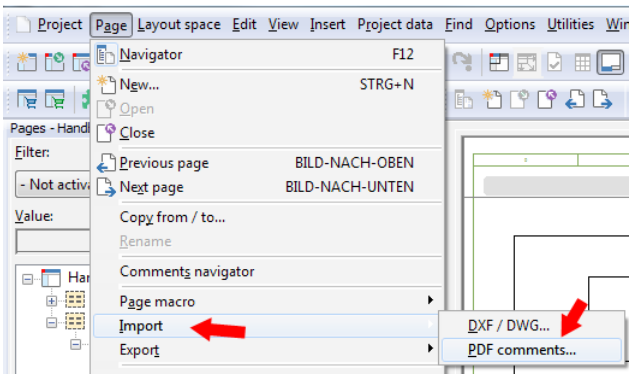


Fig. 8.47
Selecting PDF file to be imported

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.

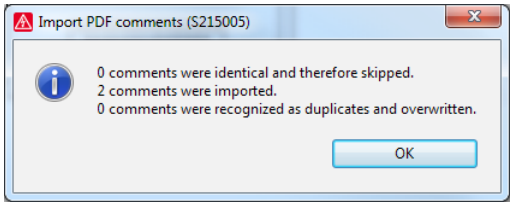


Fig. 8.48 Successful import

You can see the results of the import in the comments navigator. You can access the comments navigator from the **PAGE/COMMENTS NAVIGATOR** menu.

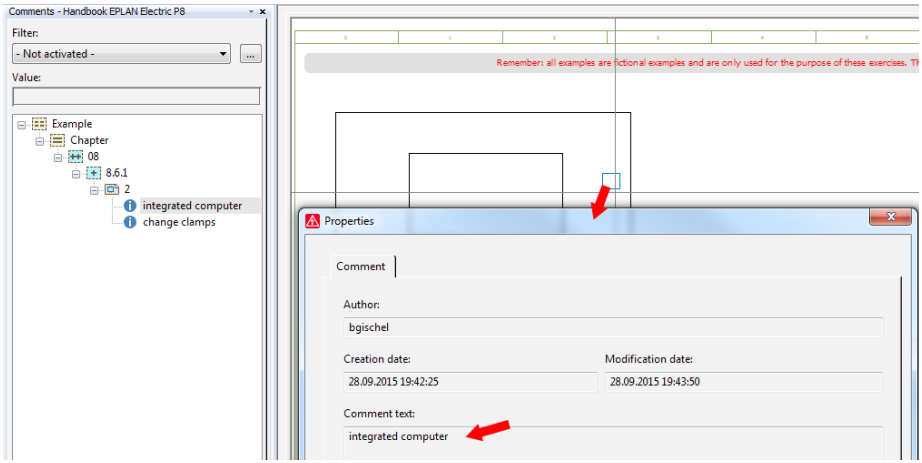


Fig. 8.49 The comments navigator

In the comment navigator, you can edit comments (e.g. set the status to “Rejected”, “Accepted”, etc.) in the comment properties.

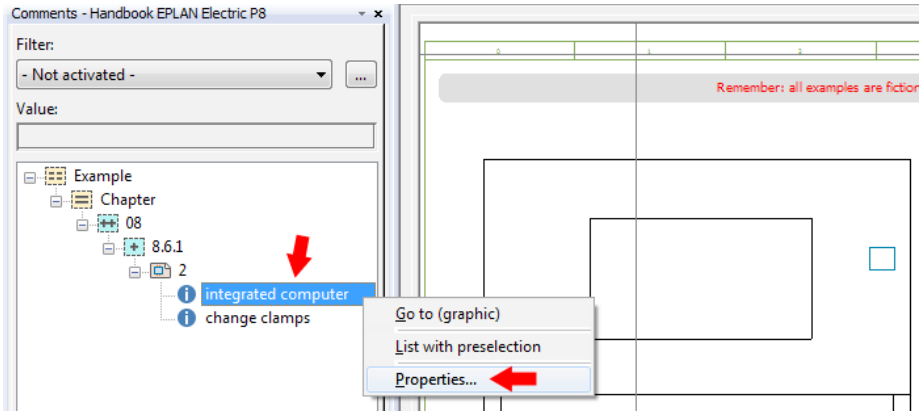


Fig. 8.50 Properties of a comment

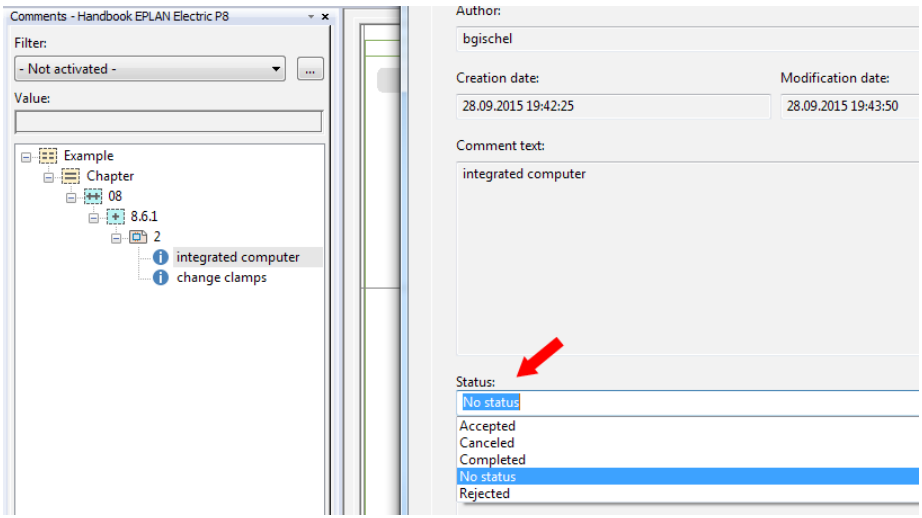


Fig. 8.51 Status of a comment

These properties are, of course, also available directly at the comments in the schematic. You can open them by double-clicking the yellow box.

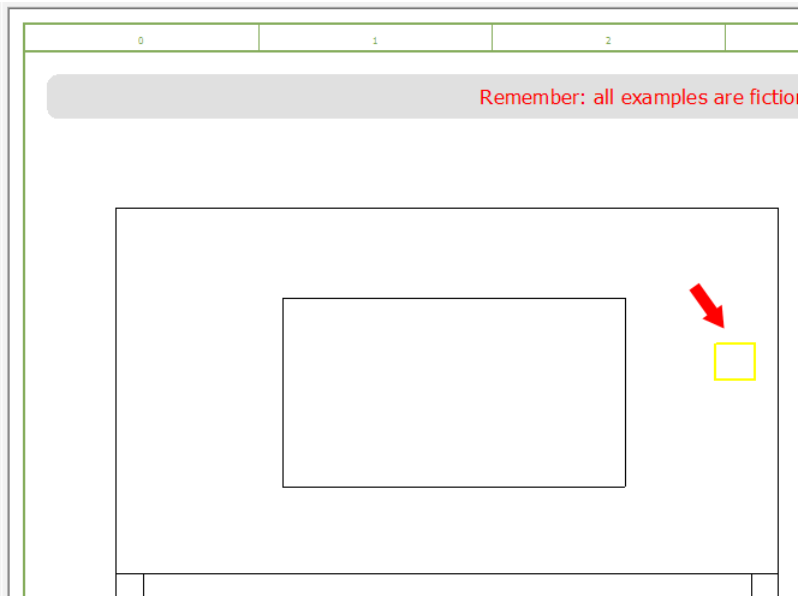


Fig. 8.52 Comment selection

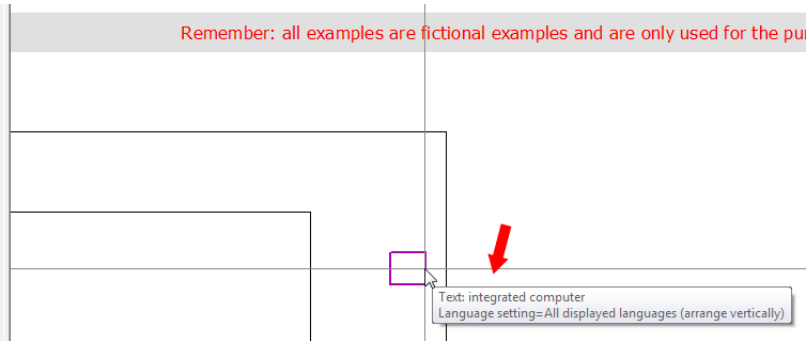


Fig. 8.53 Comment

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.

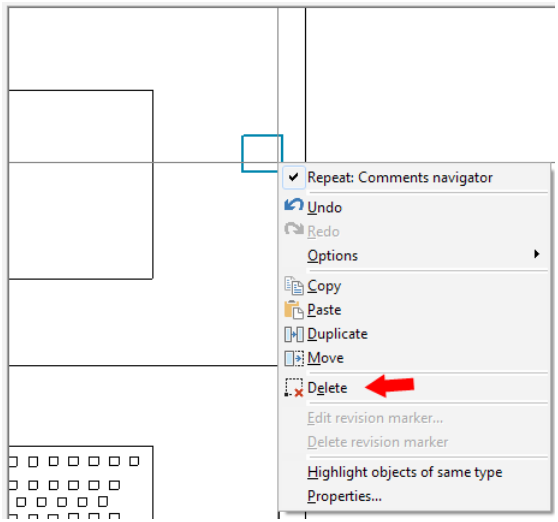


Fig. 8.54 Deleting comments

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.

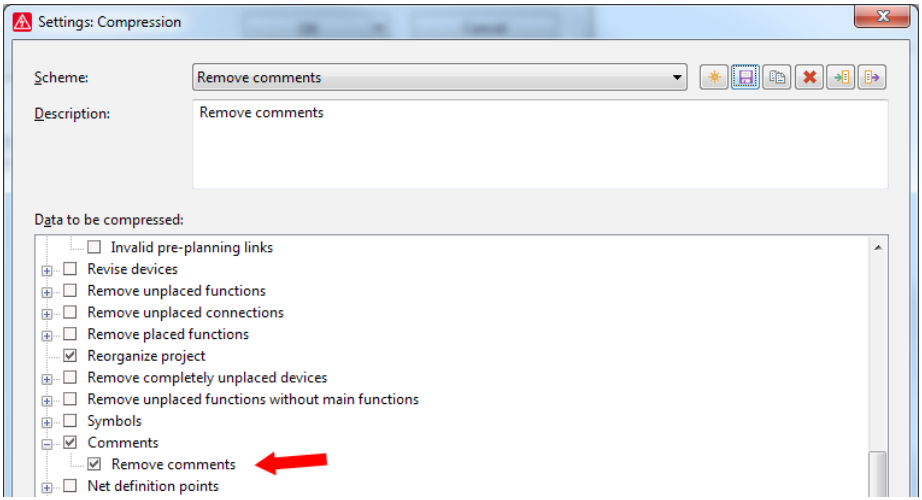


Fig. 8.55 Compression scheme to remove PDF comments

8.7 Generate PDF documents

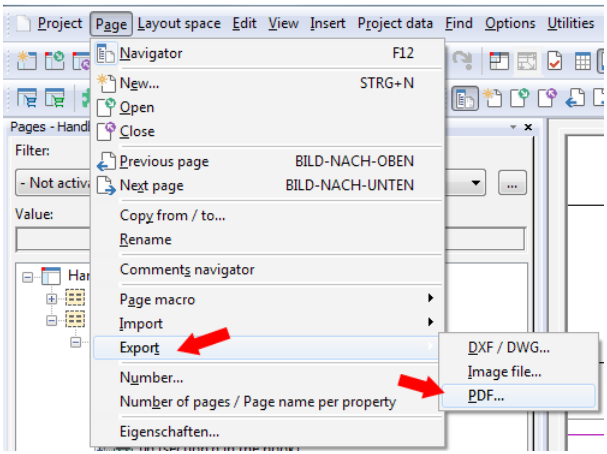


Fig. 8.56 Export as a PDF file menu

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.

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 format for longer-term archiving.

NOTE: If the PDF/A 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 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.

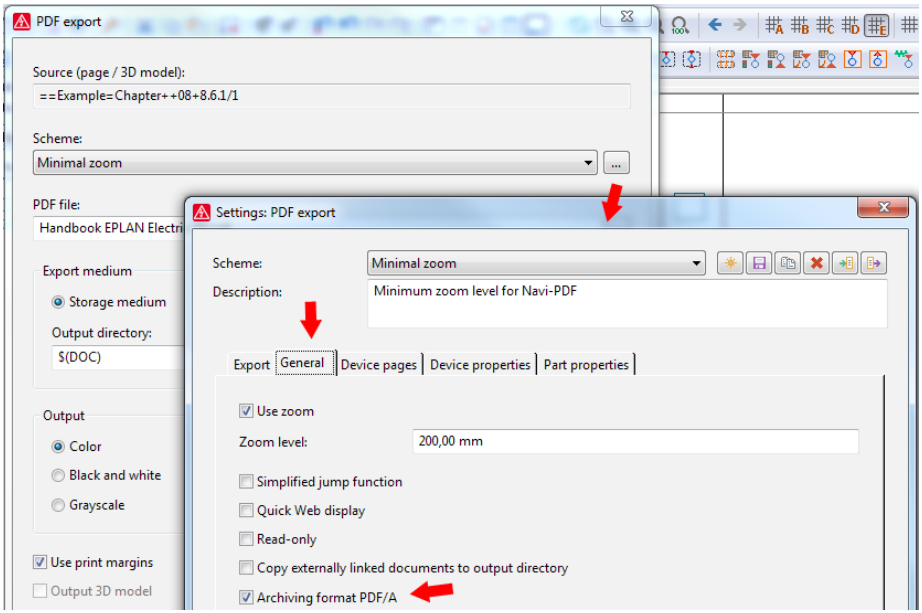


Fig. 8.57 Activate archiving format PDF/A

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.

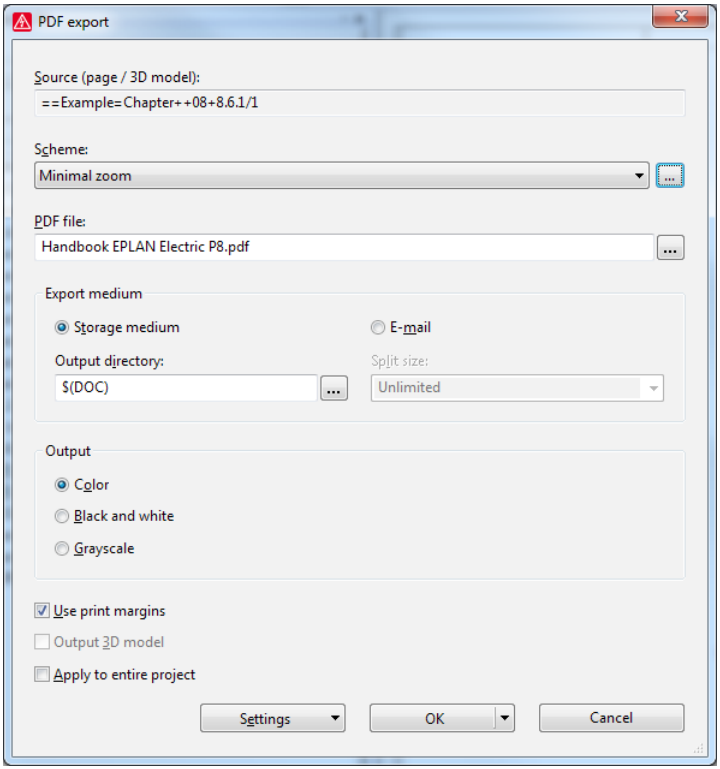


Fig. 8.58 PDF export dialog and its settings options

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.

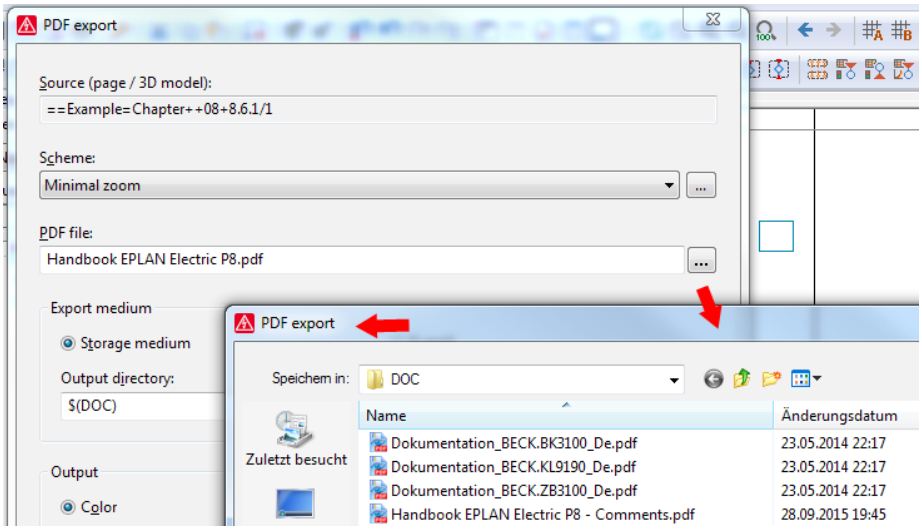


Fig. 8.59 Selection of the directory and file name for export

These modifications are saved with the **SAVE** button. EPLAN then applies the new settings of the PDF file name to the *PDF file* field.

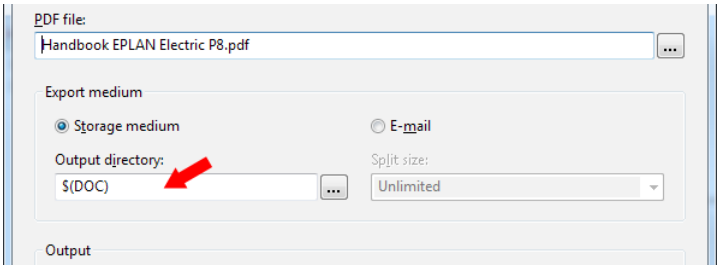


Fig. 8.60 Applied setting

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.

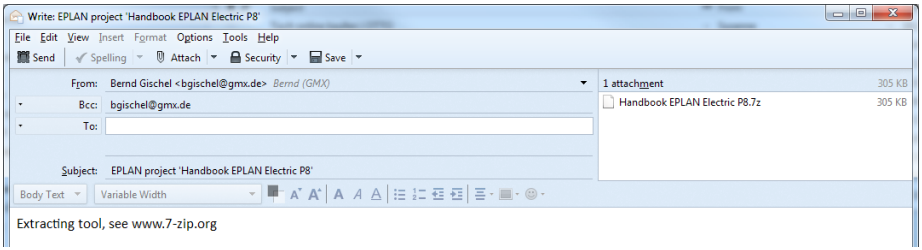


Fig. 8.61 Using the E-mail option

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*.

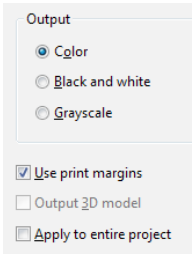


Fig. 8.62
Other settings

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.

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/2D options.

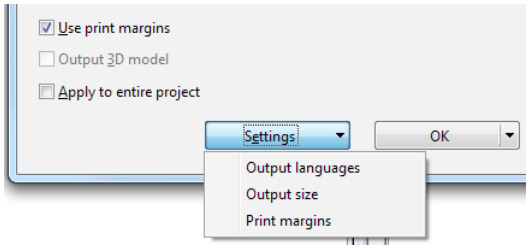


Fig. 8.63
Settings: Print margins

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.

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.

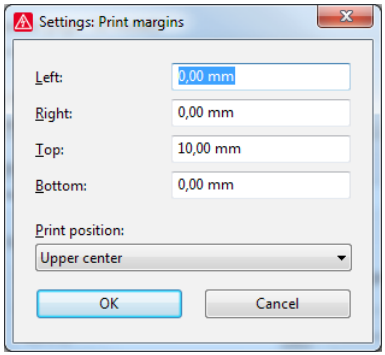


Fig. 8.64
Print margins dialog

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.

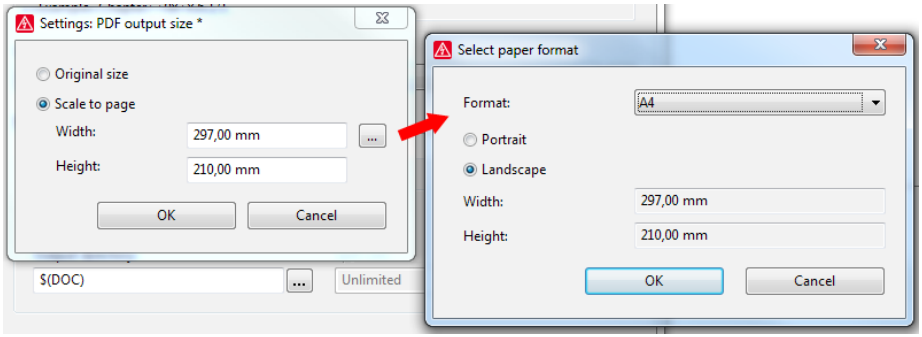


Fig. 8.65 Settings for PDF output size

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.

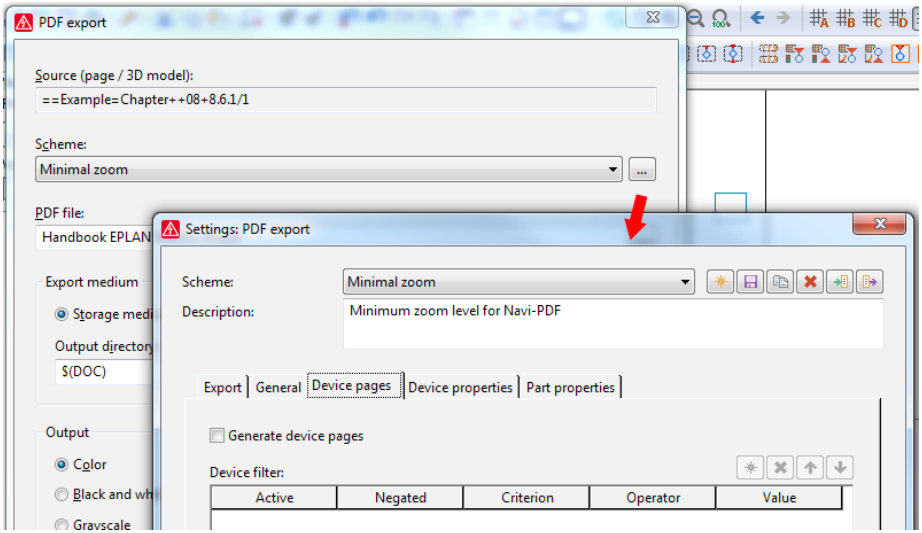


Fig. 8.66 Scheme for other detailed settings

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.

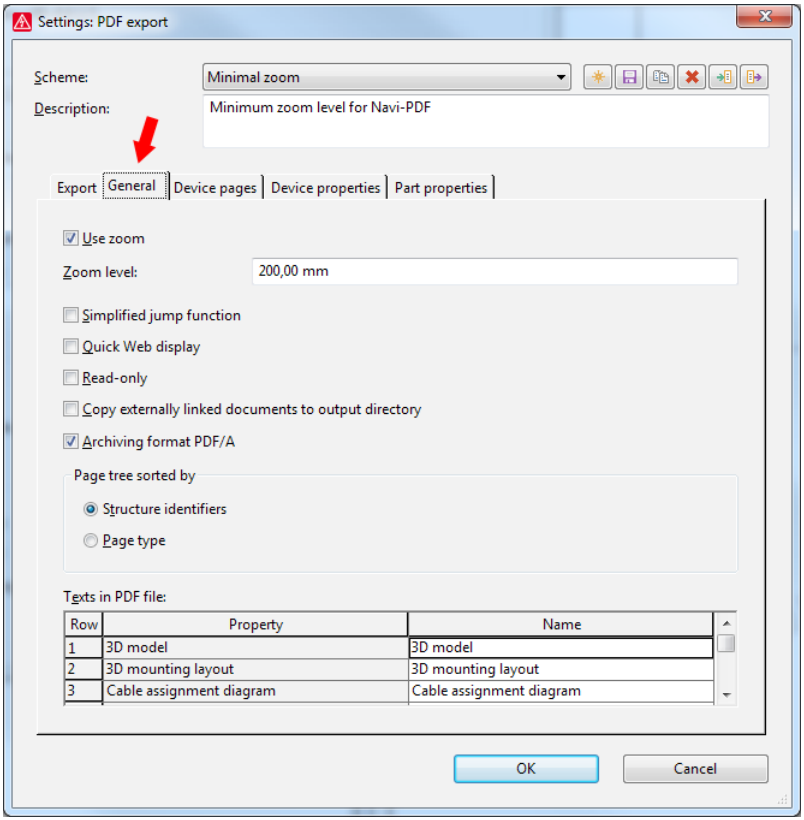


Fig. 8.67
General settings

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.

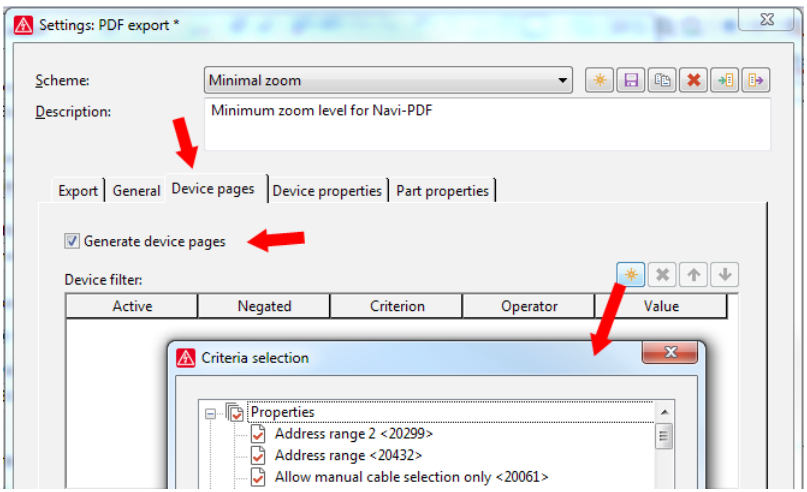


Fig. 8.68
Settings for device pages

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?

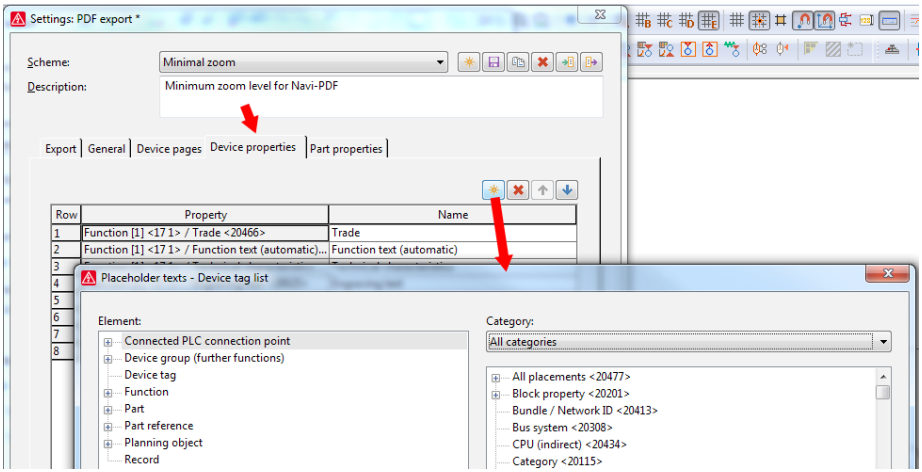


Fig. 8.69 Define device properties

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.

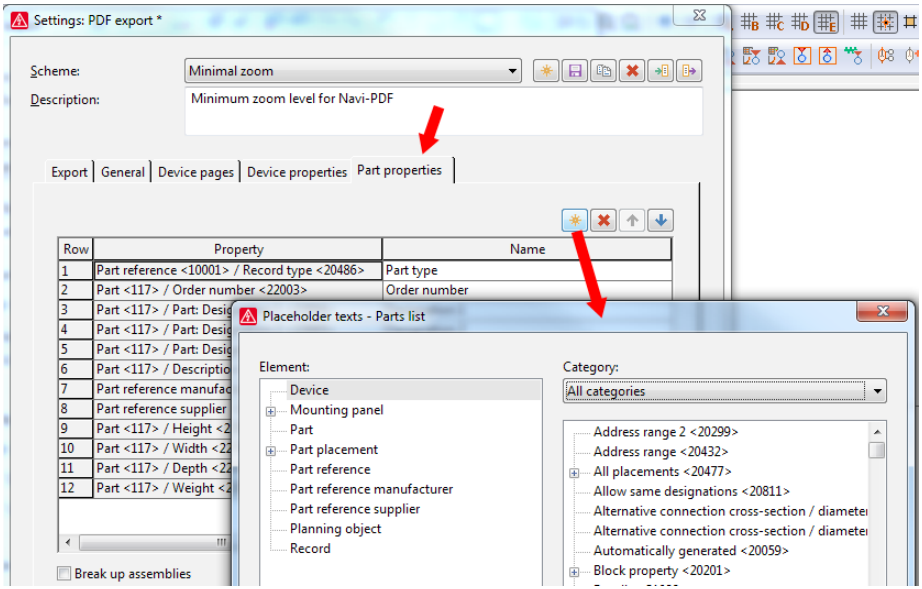


Fig. 8.70 Define part properties for the PDF export

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.

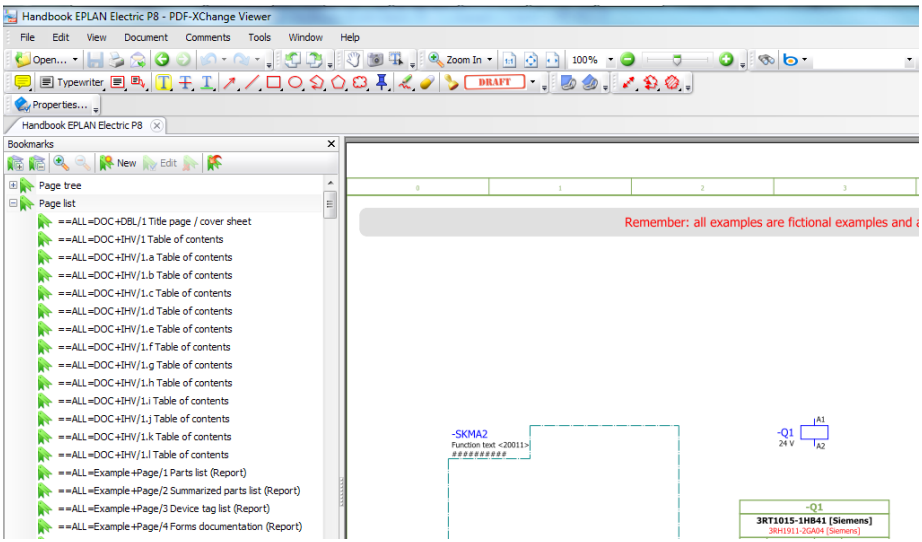


Fig. 8.71 Example: generated PDF

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, a form, the parts database or dictionary (foreign language database), can also be separately backed up and restored.

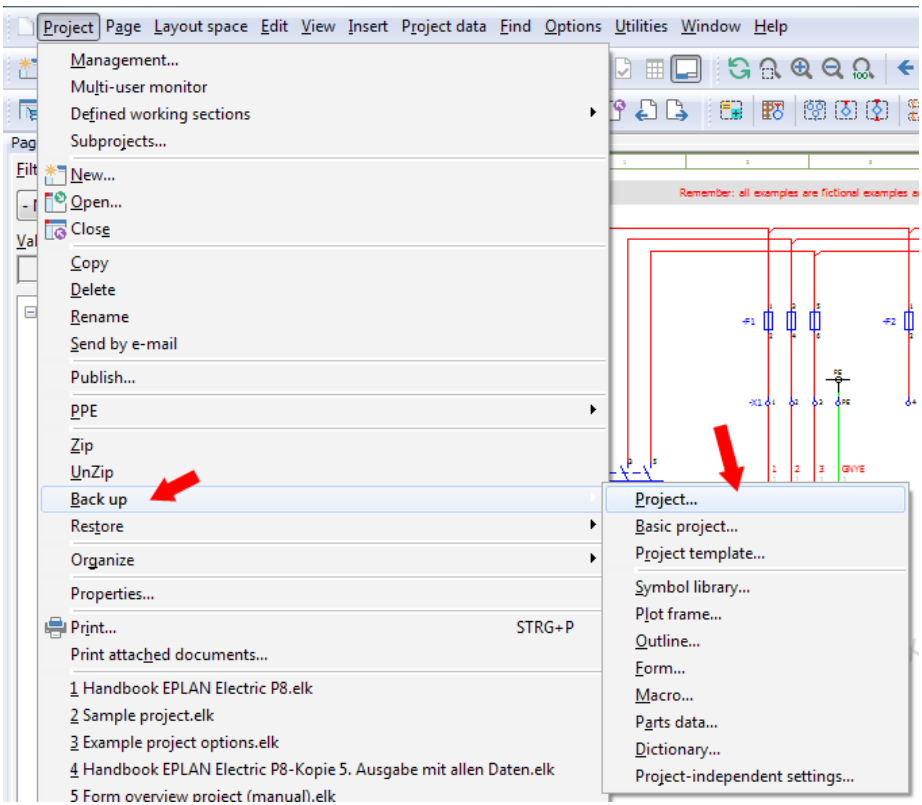


Fig. 9.1
Menu showing backup methods

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.

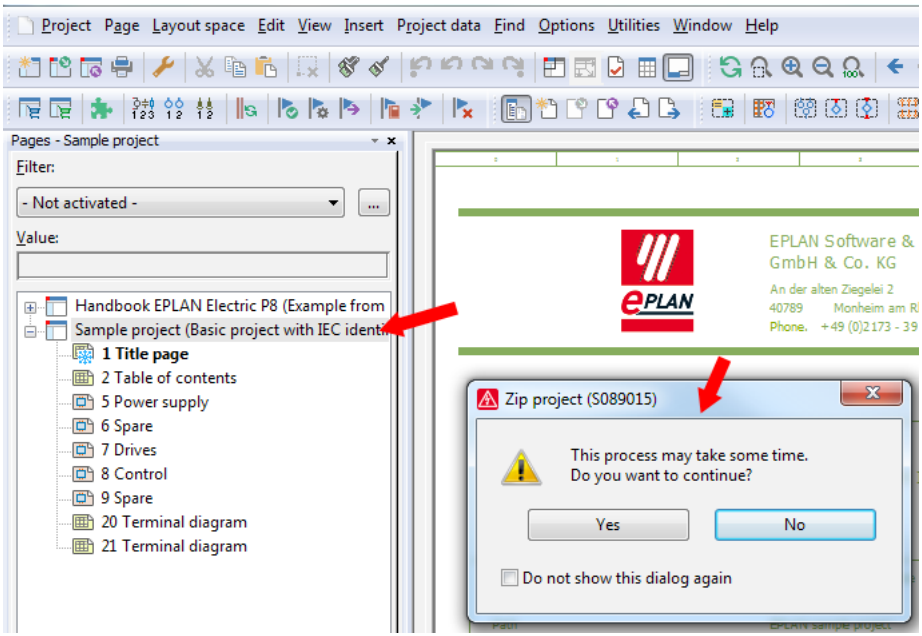


Fig. 9.2 Prompt whether the project is to be zipped

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 and, at a higher level, the project start file *Projectname.elp*.

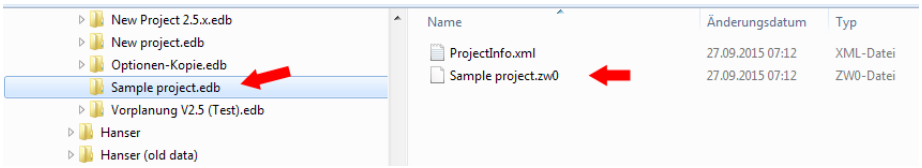


Fig. 9.3 Project data after zipping (packing)

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.

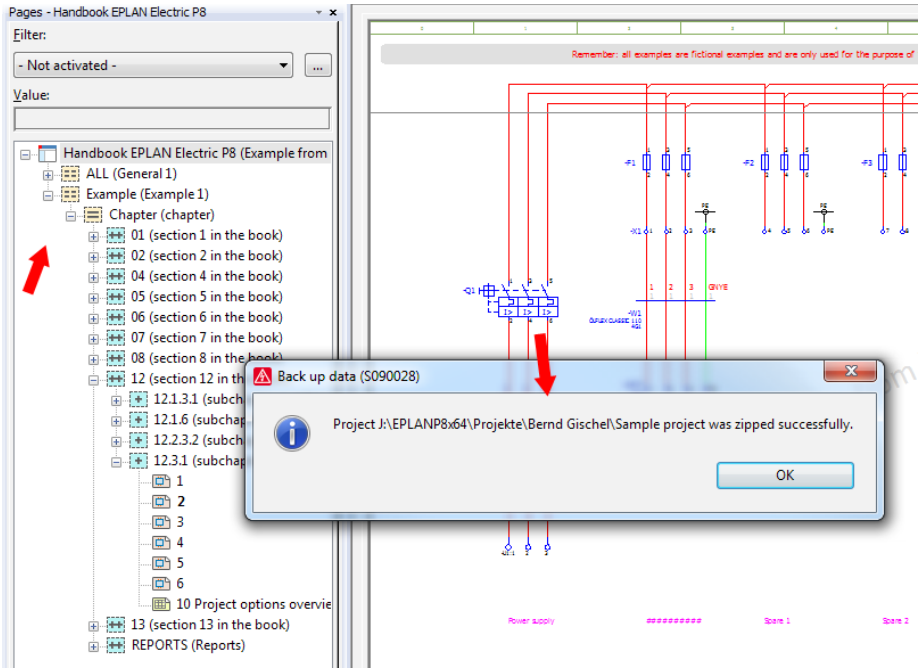


Fig. 9.4 "Project was successfully zipped." message

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**.

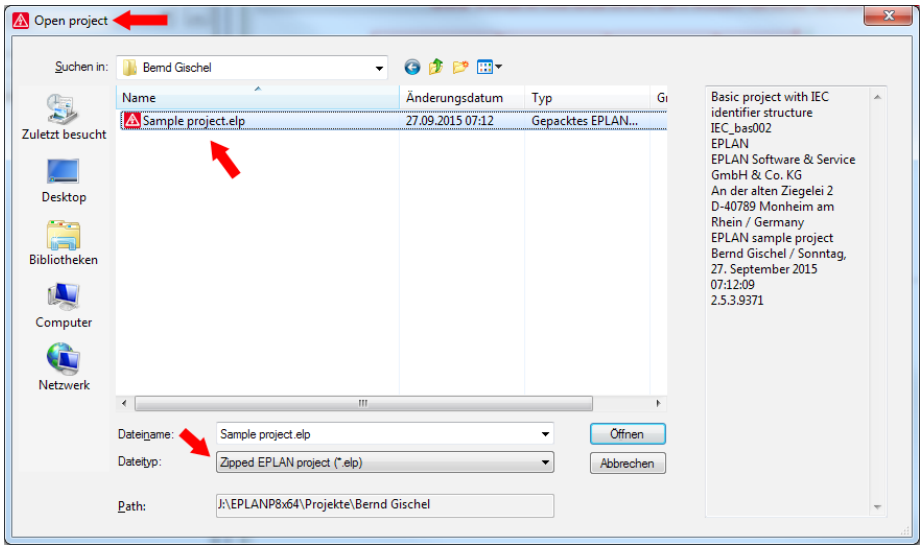


Fig. 9.5 Selecting project for unzipping (unpacking)

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.

NOTE: If zipped (packed) projects are not displayed, the appropriate file type **Packed EPLAN project (*.elp)** must be selected in the dialog.

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.

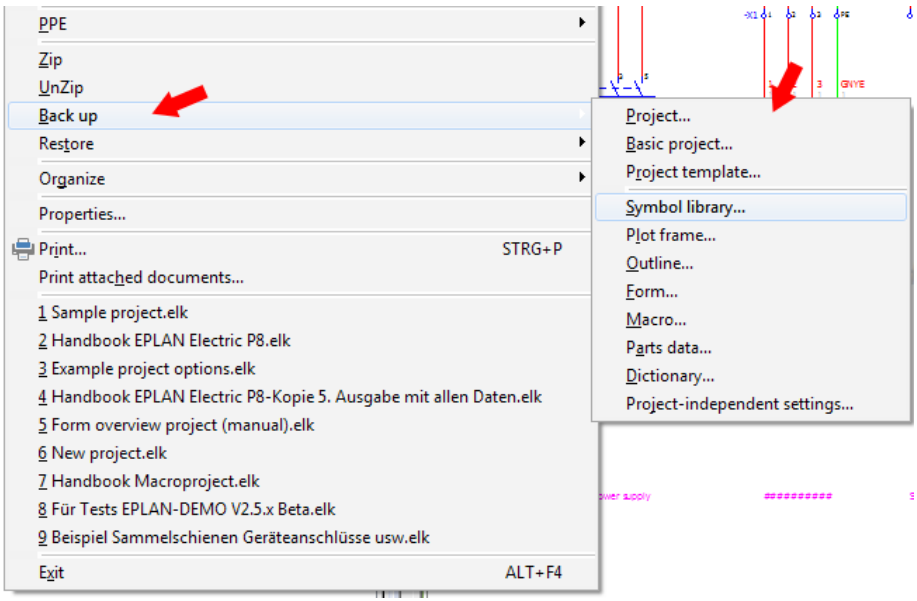


Fig. 9.6 Backup menu and its options

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.

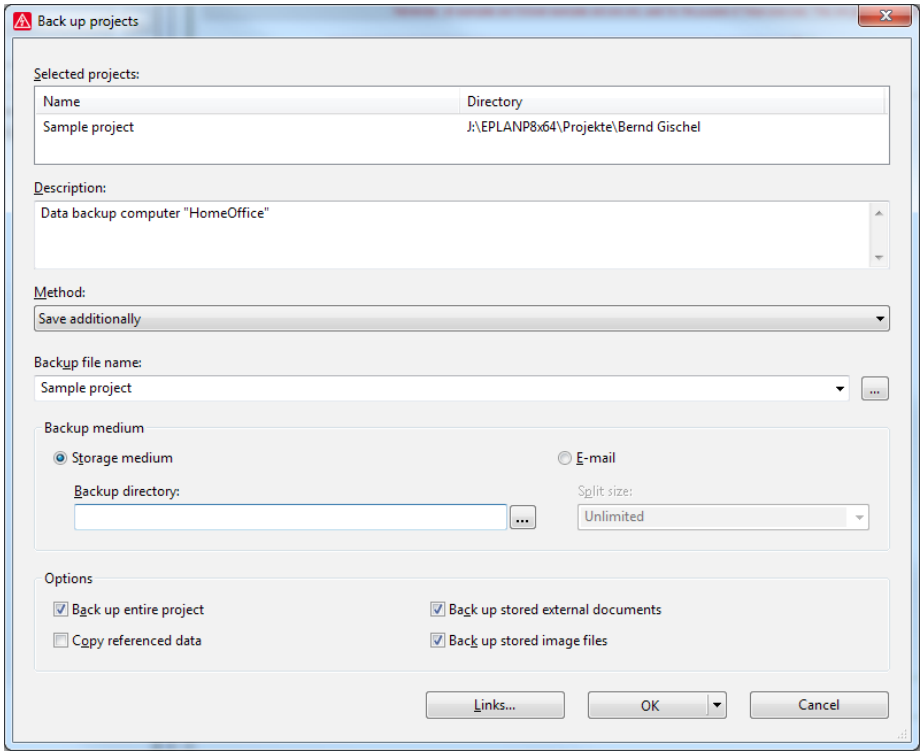


Fig. 9.7
Back up projects dialog

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.

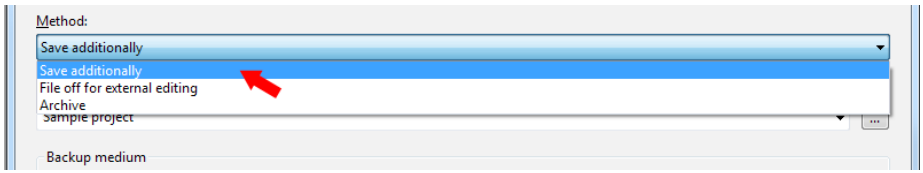


Fig. 9.8
Possible options for the backup method

Backup method

- **Save additionally:** The project is **additionally** saved on the storage medium (as **.zw1*) or, depending on the storage medium selected, is set up as an e-mail attachment. The project to be backed up 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.

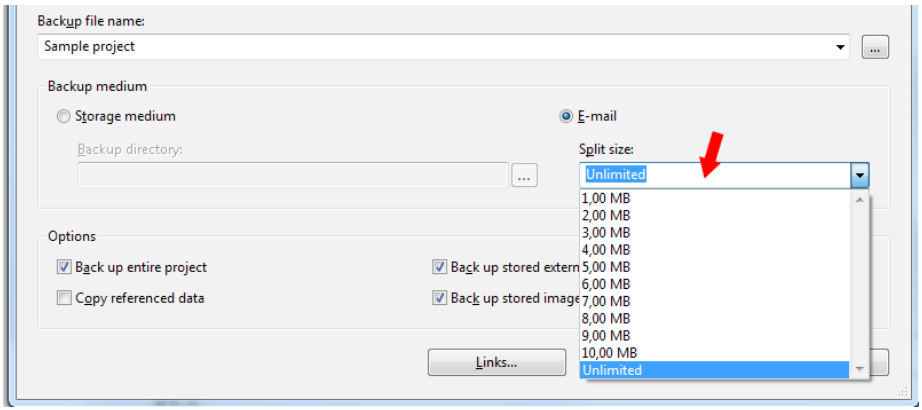


Fig. 9.9 Backup medium selection

- **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. EPLAN then generates several files internally, opens the installed e-mail program and attaches the first part (Part 3 of [Total]) to a new e-mail.

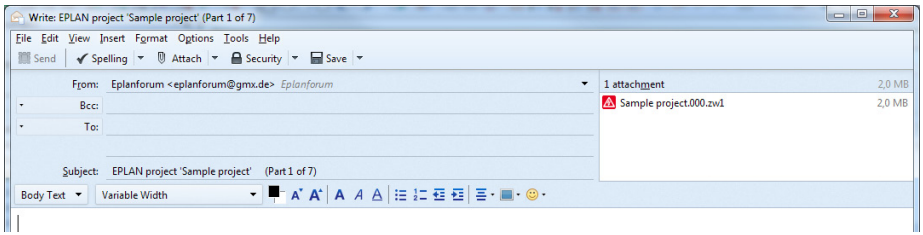


Fig. 9.10 First e-mail generated automatically by EPLAN

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.

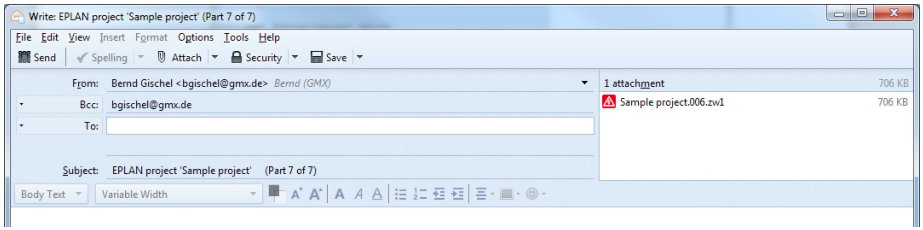


Fig. 9.11 "Last" e-mail containing the last part of the project backup

This completes the e-mail backup. After the final part has been sent, EPLAN displays a success message.

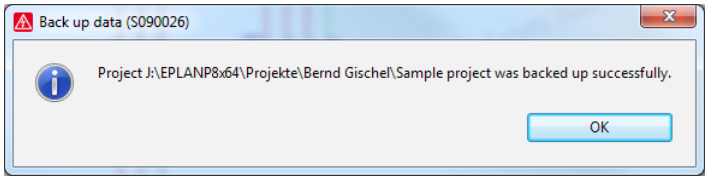


Fig. 9.12 Message that all parts have been successfully sent

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.

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.

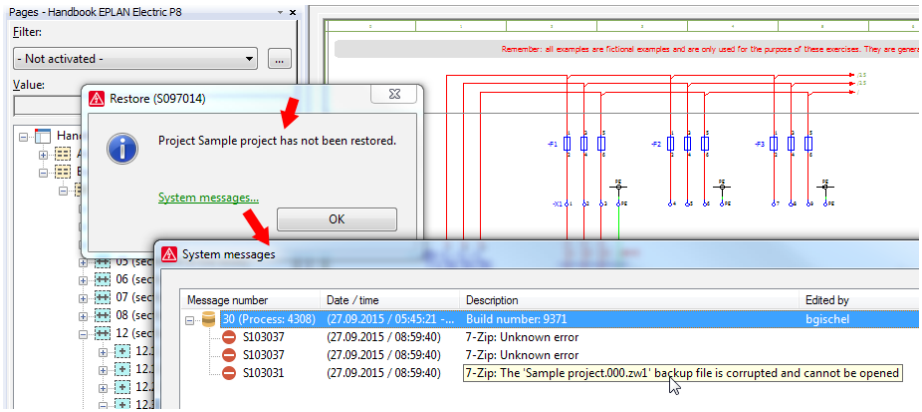


Fig. 9.13 Error message when the project could not be restored

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.

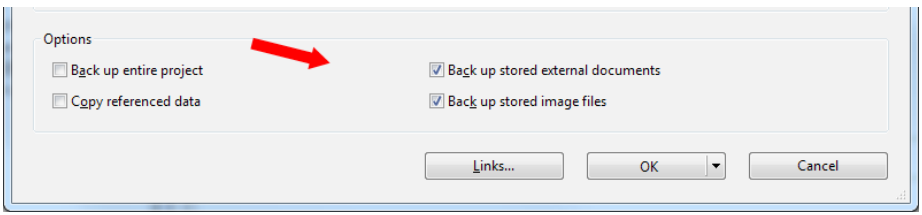


Fig. 9.14 Reduce data volume using these options

- **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.

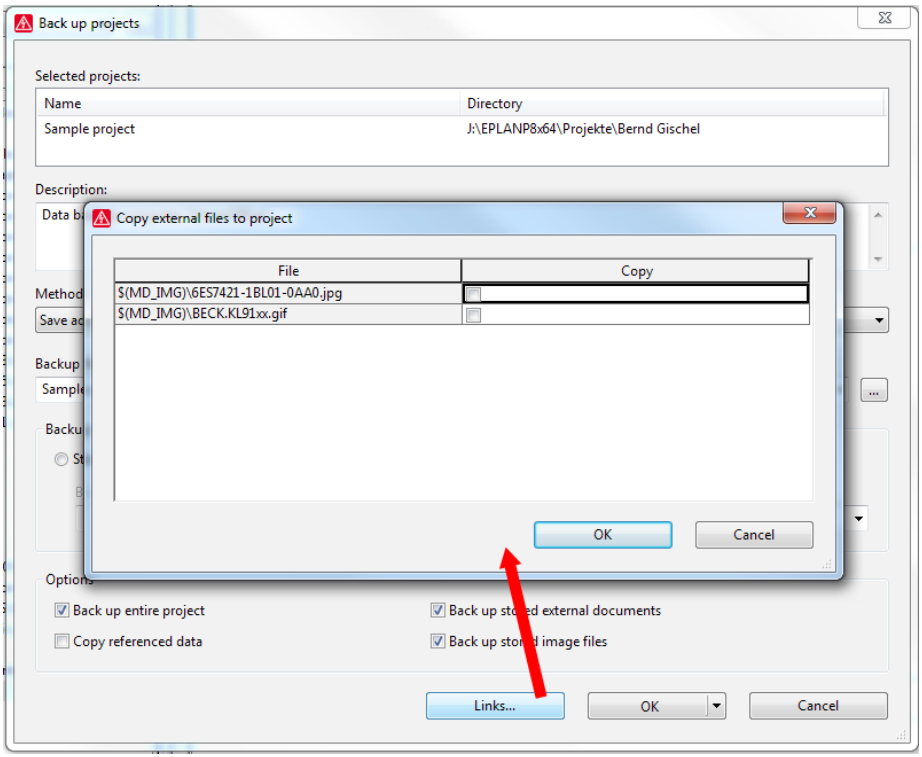


Fig. 9.15 Copying externally linked files to the project (back up projects)

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.

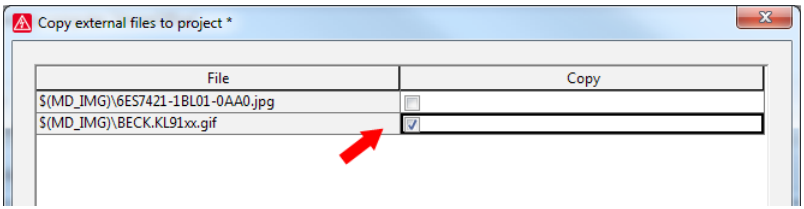


Fig. 9.16 Copying externally linked files to the project (graphical editor)

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 (restore back).

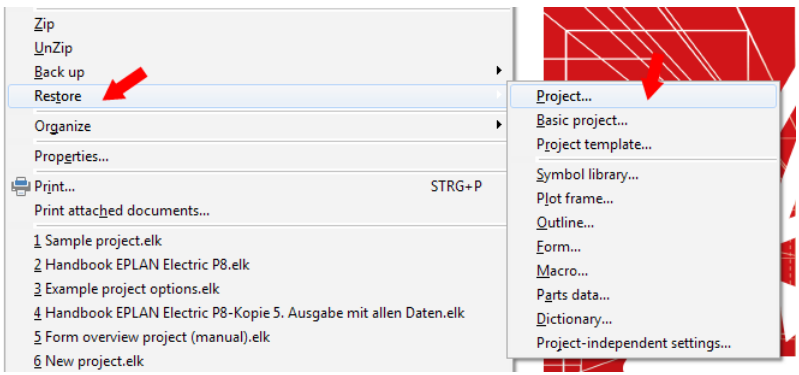


Fig. 9.17 Restore projects menu

The **Restore project** dialog opens with a number of setting and selection options.

Restore projects settings

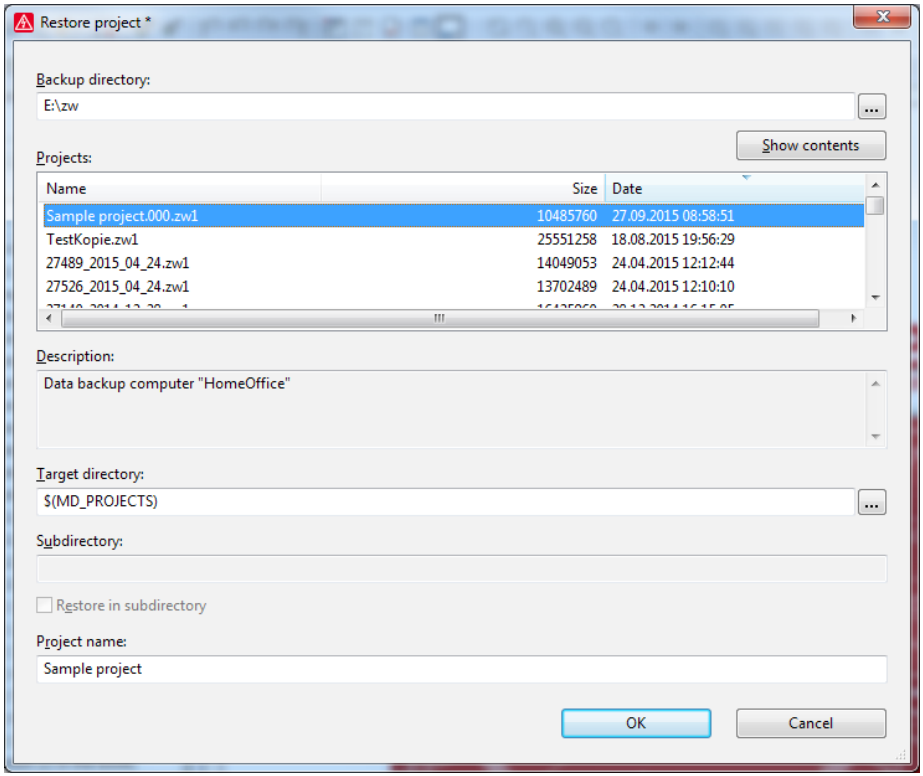


Fig. 9.18 Restore project dialog

- **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.

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.

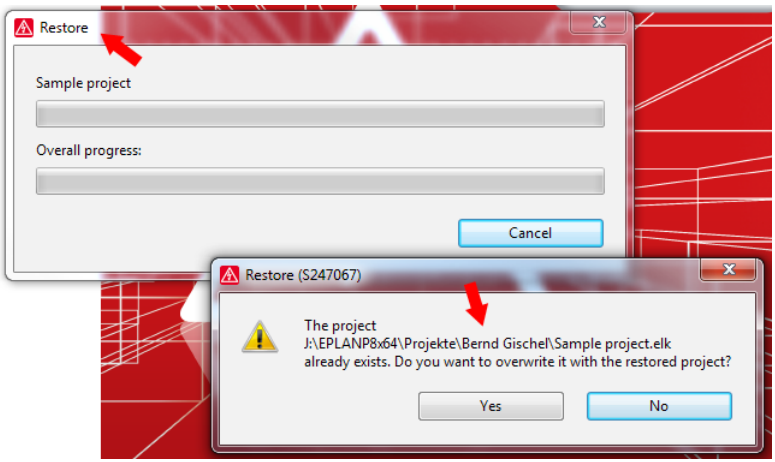


Fig. 9.19
Possible message if a project already exists

EPLAN can also overwrite opened projects if you wish. After a prompt, EPLAN closes the project and then restores it.

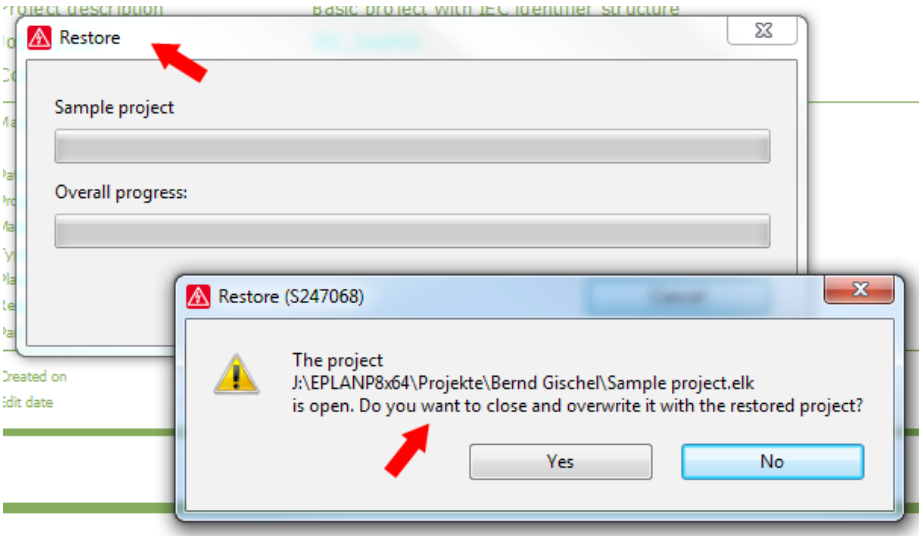


Fig. 9.20
Another message if a project already exists

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.

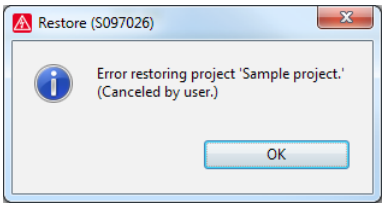


Fig. 9.21
Note displayed when canceled by user

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 when it is backed up for the first time. 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.

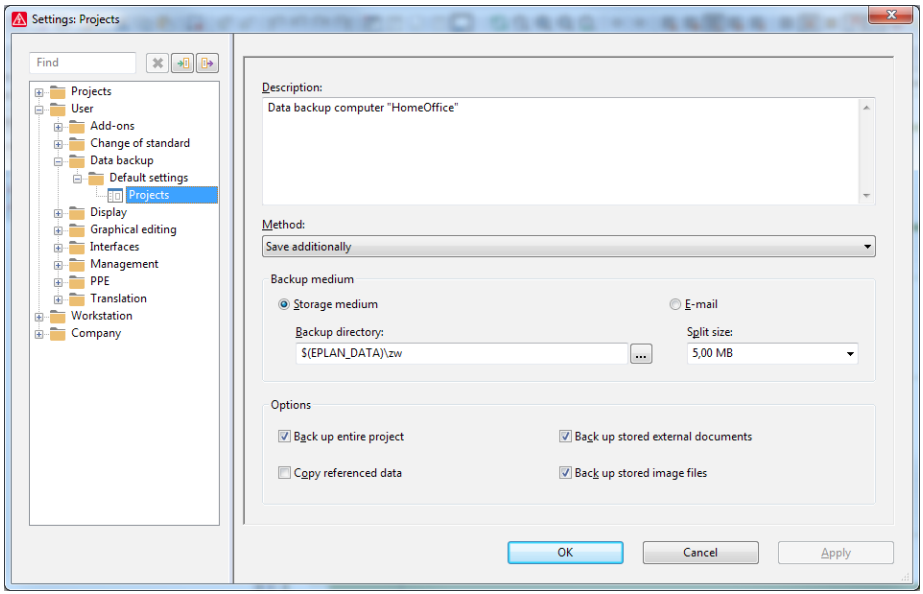


Fig. 9.22
Default settings for backing up projects

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 at first, 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/COMPRESS MENU**.

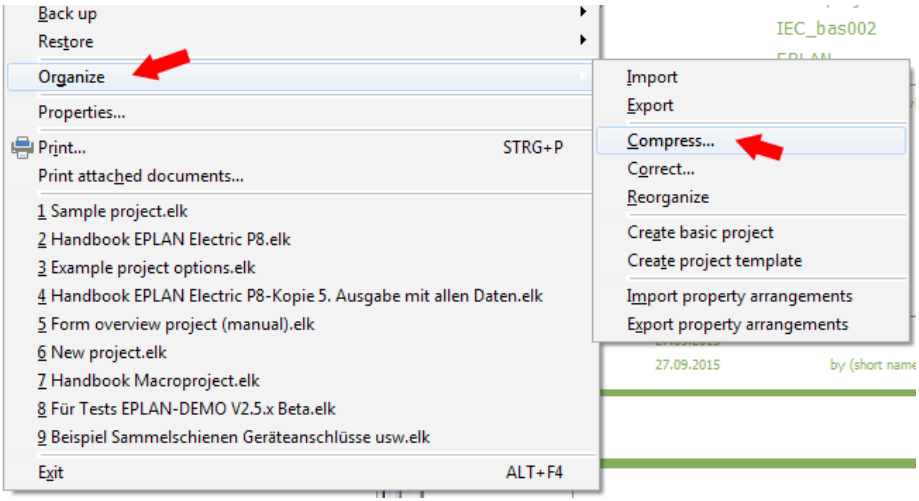


Fig. 9.23 Project / Organize / Compress menu

EPLAN then starts the **Compress project** dialog.

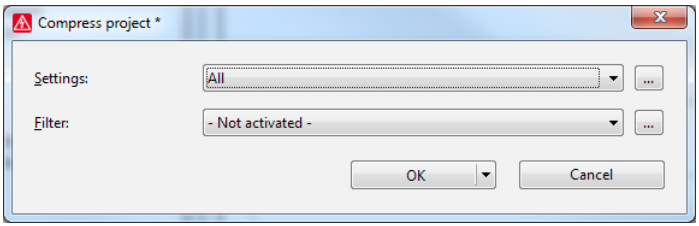


Fig. 9.24 Compress project dialog

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.

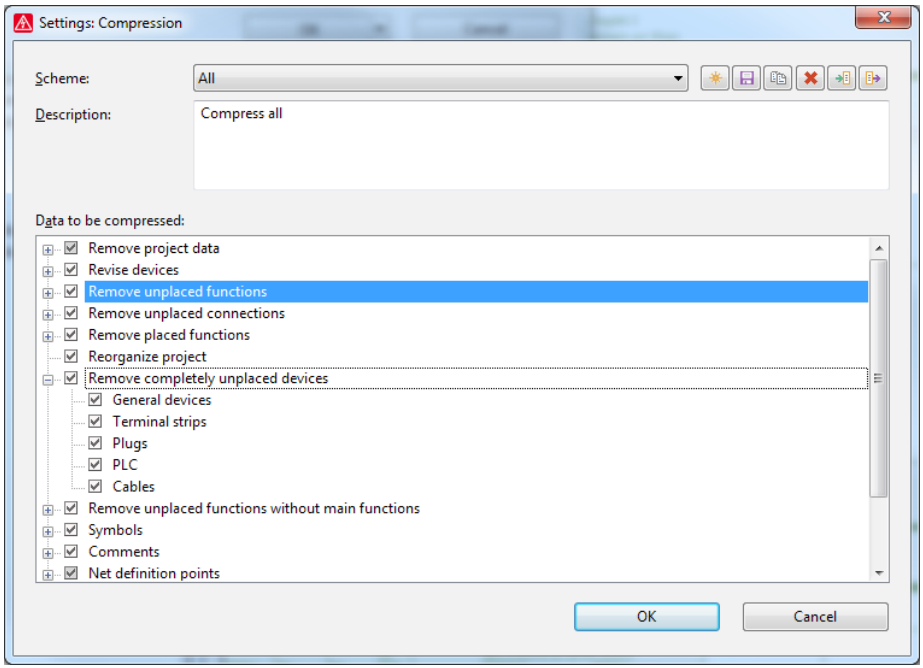


Fig. 9.25
Settings options for a compression scheme

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.

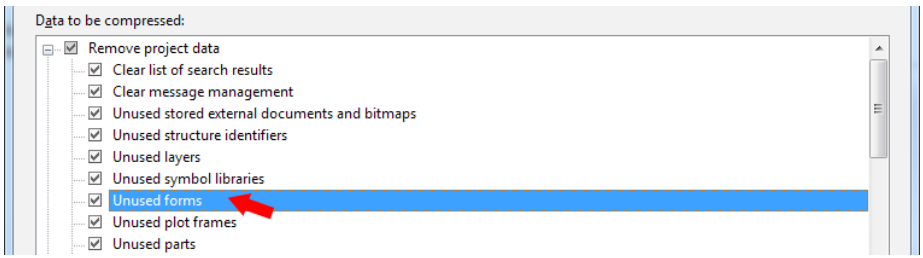


Fig. 9.26
Setting to remove unnecessary forms

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.

Automated processing with scripts

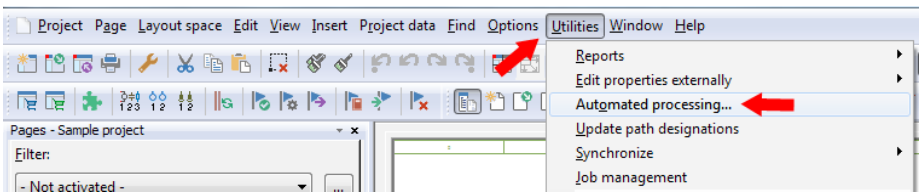


Fig. 9.27 The Automated processing menu item

Afterwards, EPLAN opens the **Run: Automated processing** dialog.

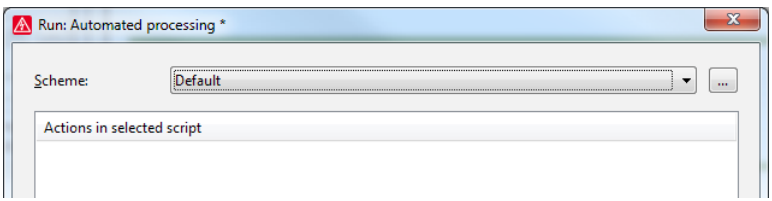


Fig. 9.28 Run: Automated processing dialog

Here, only an existing scheme can be selected, or a new one created.

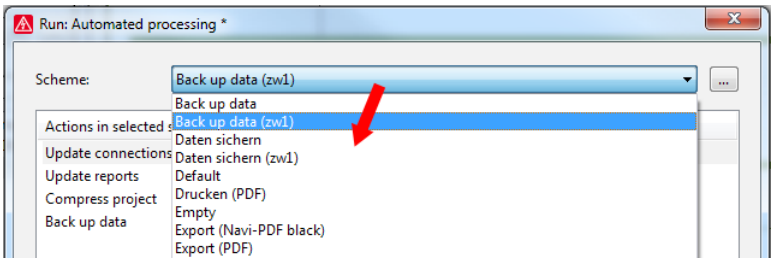


Fig. 9.29 Scheme drop-down list

The **[...]** button contains 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.

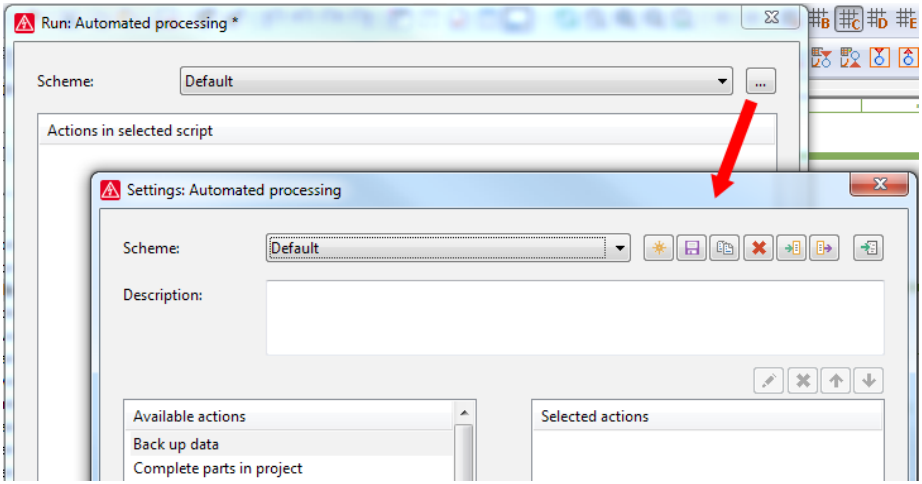


Fig. 9.30 The possible settings of automated processing

The **Automated processing** function is a scheme, as already mentioned. 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.

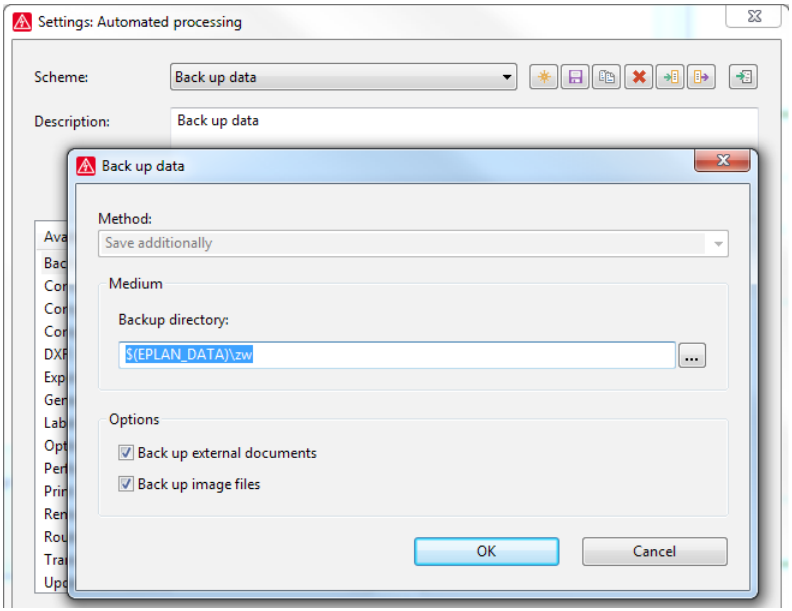


Fig. 9.31
Further dialog for data backup

Combining settings for scripts

If you now combine these settings, i.e. compress project followed by a data backup, then you can process it automatically.

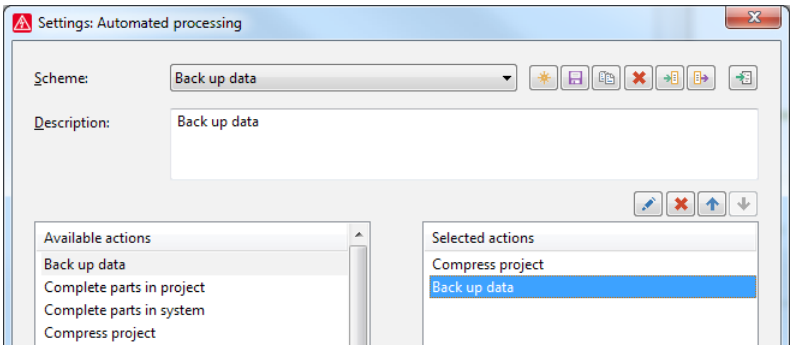


Fig. 9.32
Finished script with selected actions

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 following the export.

In general, the last edited scheme is stored and can be started via the **UTILITIES/AUTOMATED PROCESSING** menu item. But you can also select and run other schemes here.

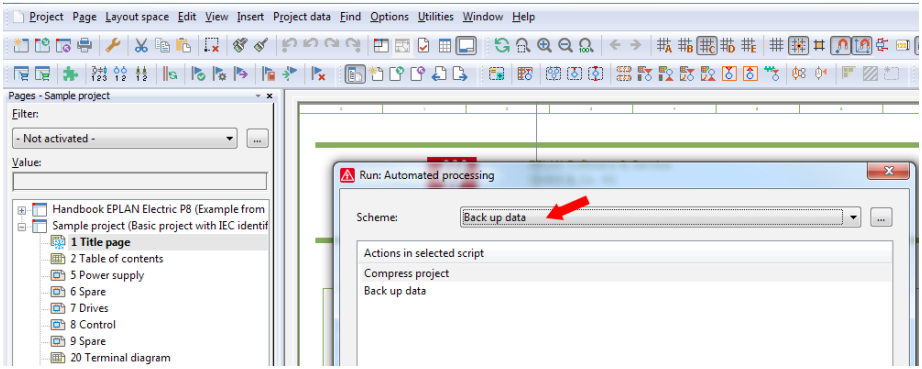


Fig. 9.33 Scheme selection

When the scheme is started, EPLAN executes the stored actions and displays the current progress in a dialog.

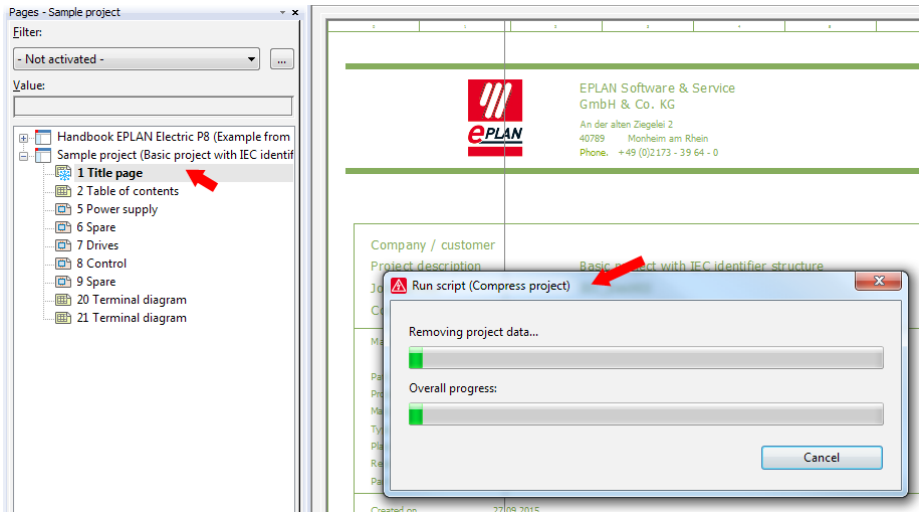


Fig. 9.34 Status of the script (scheme)

When the script (scheme) is finished, no other messages are displayed.

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 settings not specific to the project (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.

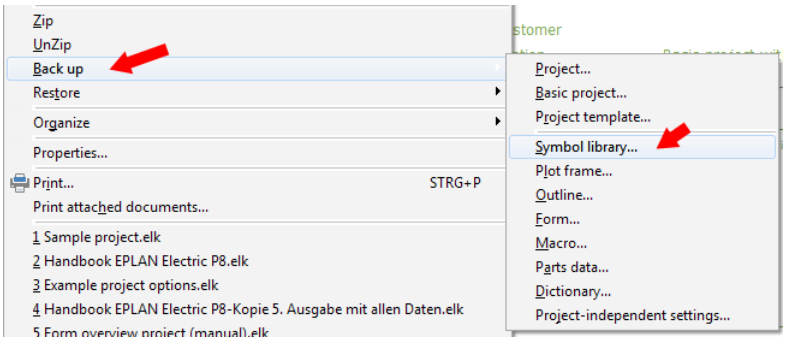


Fig. 9.35 Project/Backup/[Master data] menu

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.

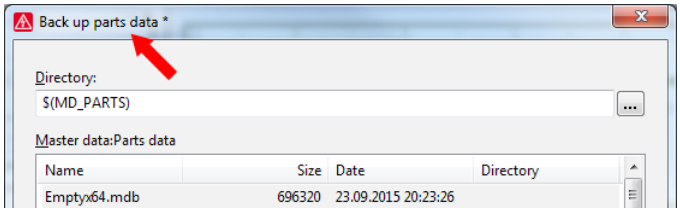


Fig. 9.36 Back up [Master data] dialog

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.

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.

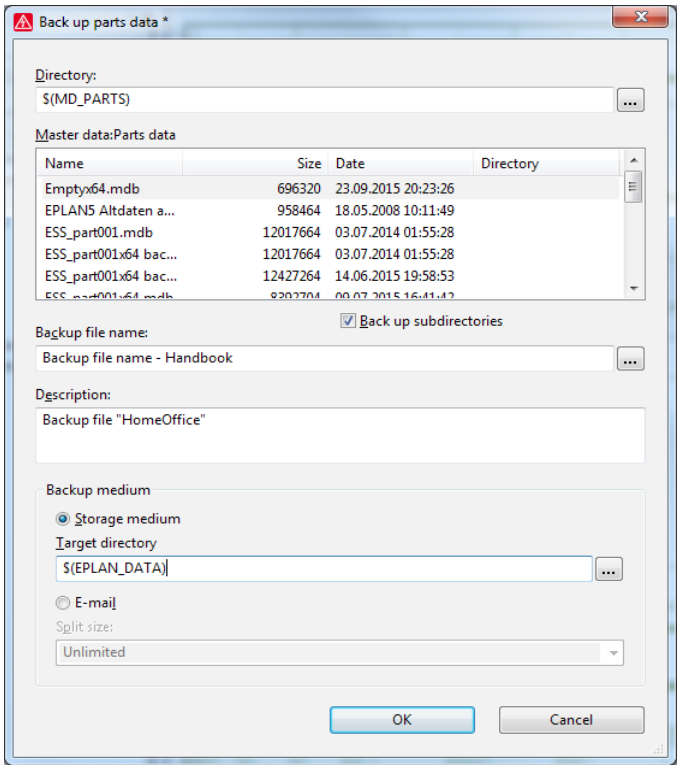


Fig. 9.37 Dialog as an example of how to back up master data

- **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. When the back up has been successful, a message will be displayed.

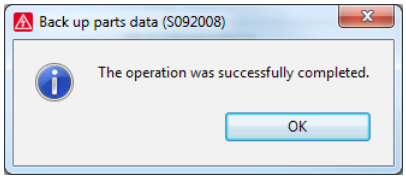


Fig. 9.38 Message confirming success

This completes the backing up of master data.

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.

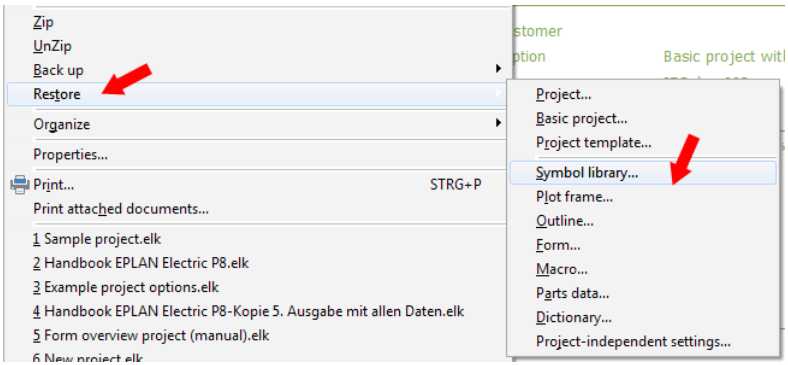


Fig. 9.39 Project/Restore menu

If you wish to restore this data in EPLAN, you use the **PROJECT/RESTORE [MASTER DATA]** menu to open that dialog.

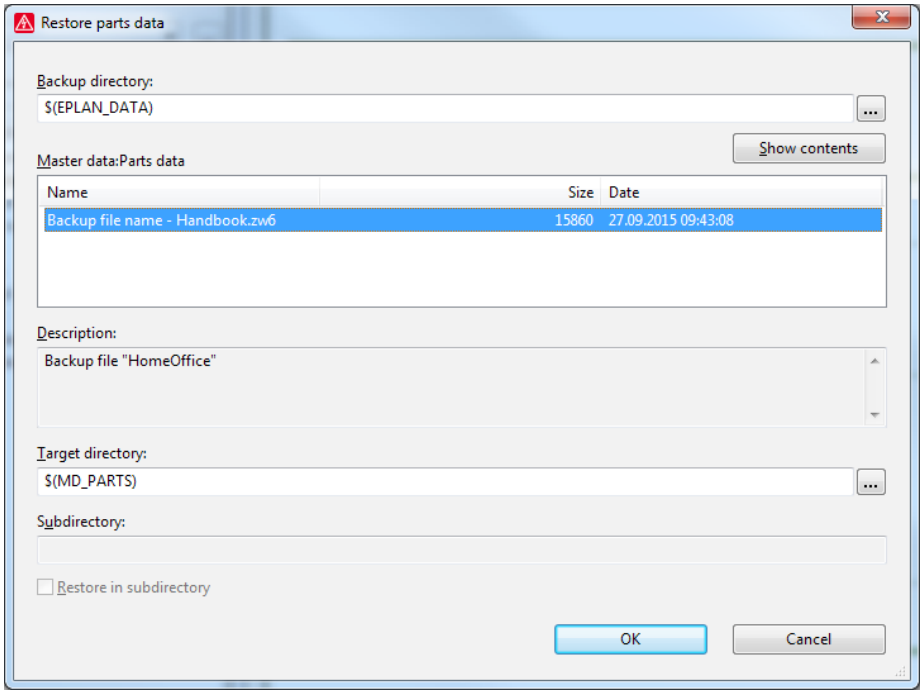


Fig. 9.40 Example of a Restore master data dialog

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 restore 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 back to the specified directory and then displays a message with information about the process of restoring the master data (or about the failure of the process).

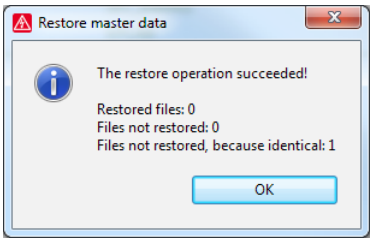


Fig. 9.41
Message confirming success

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.

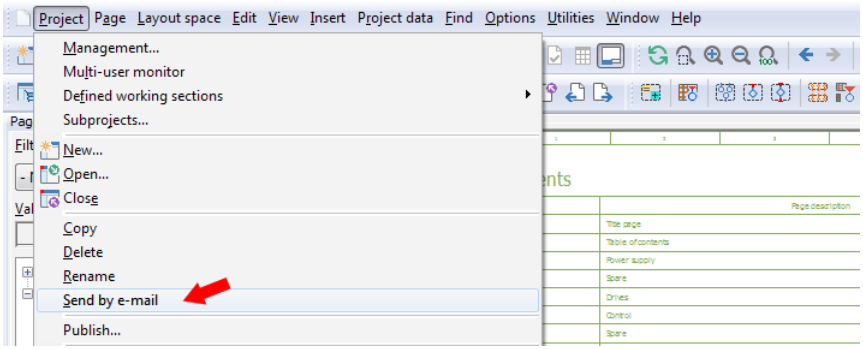


Fig. 9.42
Send project by e-mail directly

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.

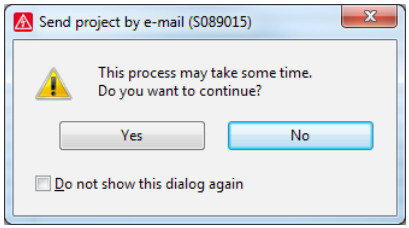


Fig. 9.43
First note regarding e-mail sending

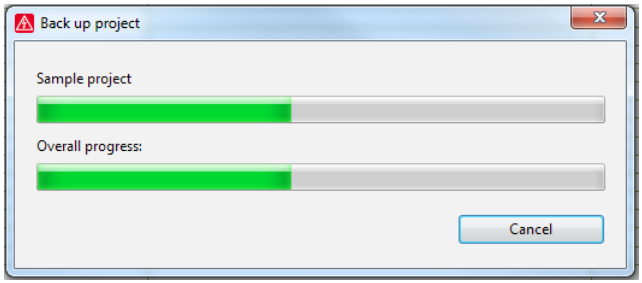


Fig. 9.44 Progress dialog for backing up project by e-mail

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.

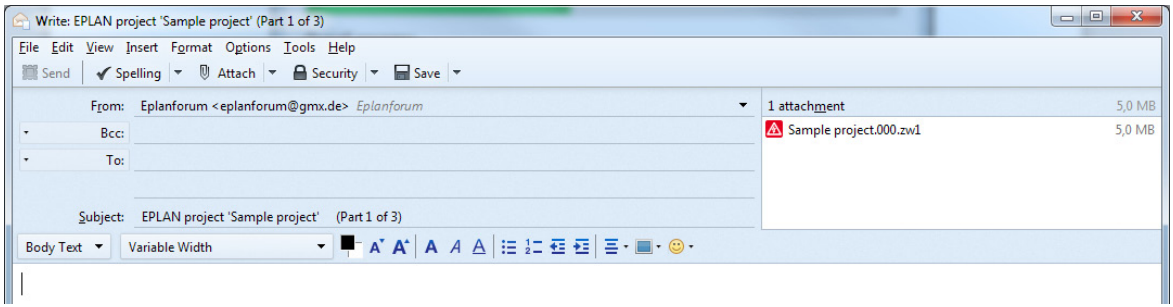


Fig. 9.45 Newly generated e-mail with the split project files

When all e-mails are sent, a message stating that they were successfully sent is displayed. The complete project has now been sent.

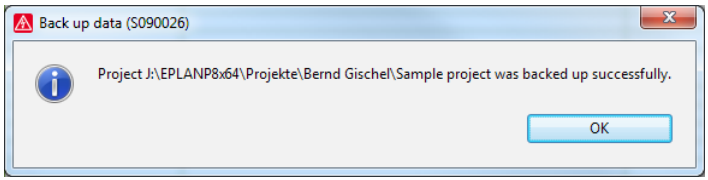


Fig. 9.46 Message that e-mails have been successfully sent

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, for example, with the *Labeling* function or *Edit properties externally.*

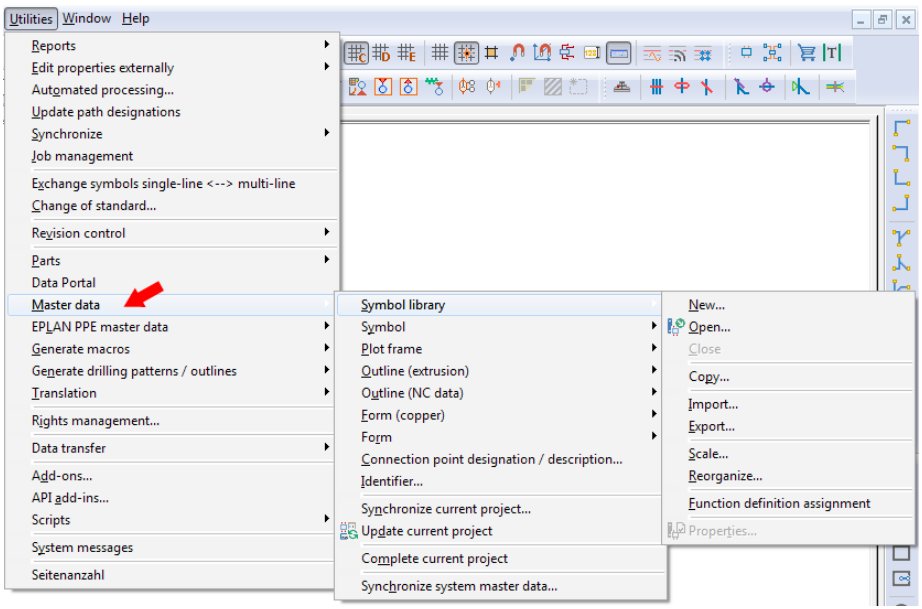


Fig. 10.1
The master data menu with the possible editors

NOTE: EPLAN always opens master data for editing on a temporary basis in the current project. It does not matter which project is involved. But a project must be open, otherwise master data cannot be selected for editing.

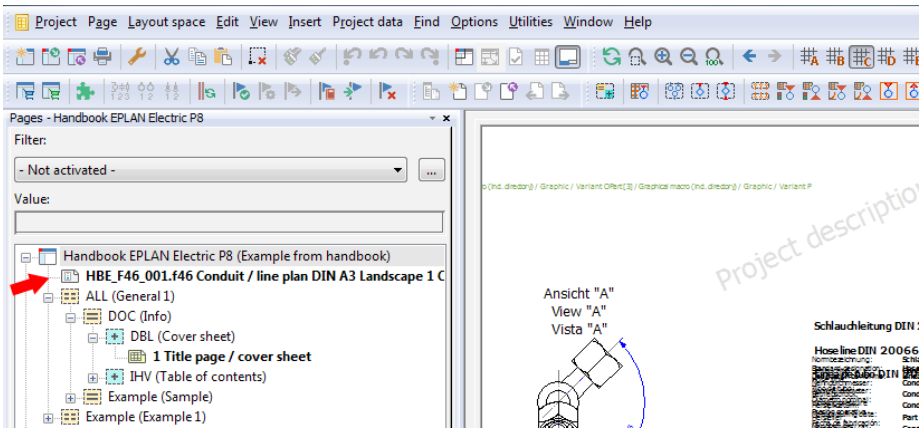


Fig. 10.2 Example representation for master data opened for editing

Generally, it is always system master data that is edited. Master data editors never modify project master data. It is rather 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.

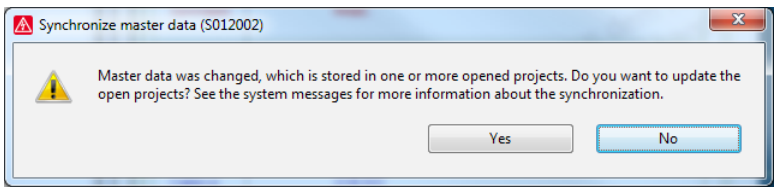


Fig. 10.3 Prompt when stored master data has been modified

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. In other words, 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]**. The **[FUNCTION]** can be symbol libraries in general, symbols, plot frames or forms.

NOTE: Editing outlines and forms will not be covered in this version of the manual.

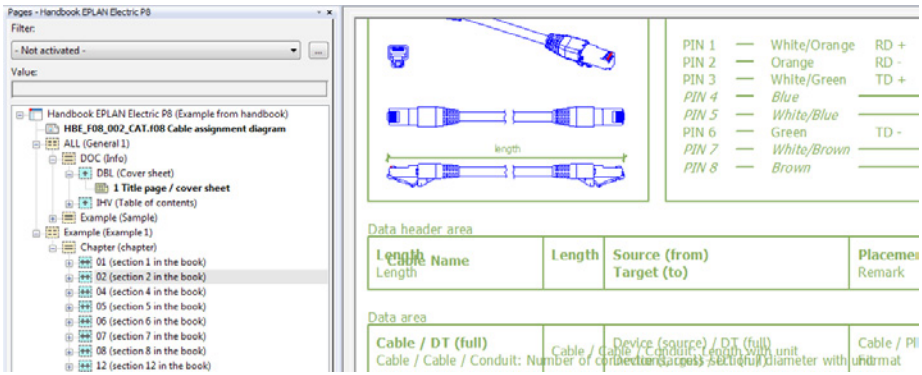


Fig. 10.4 Example of form structure

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.

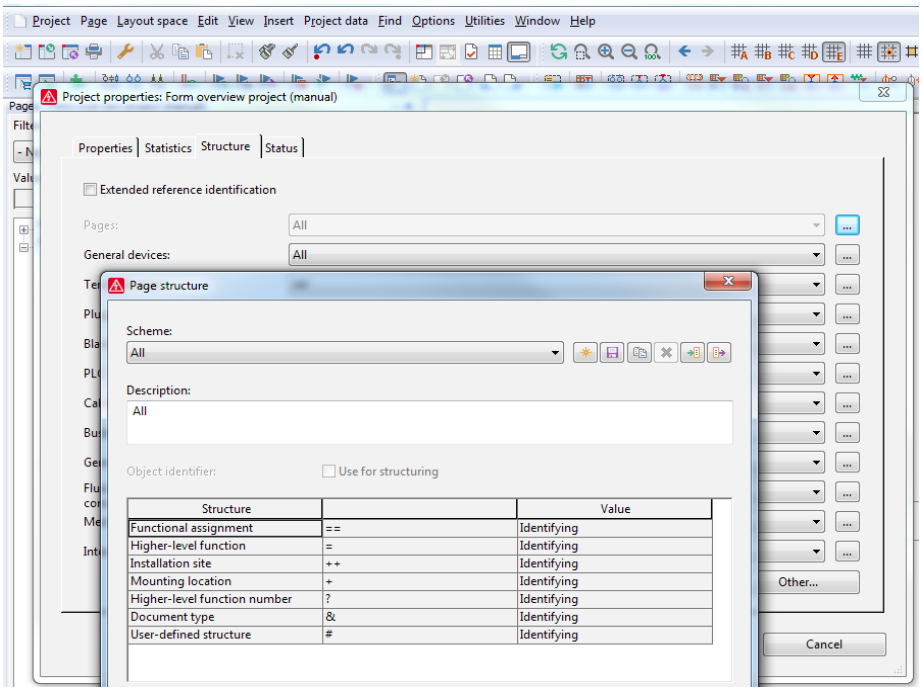
10.2 Clear overview of forms

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

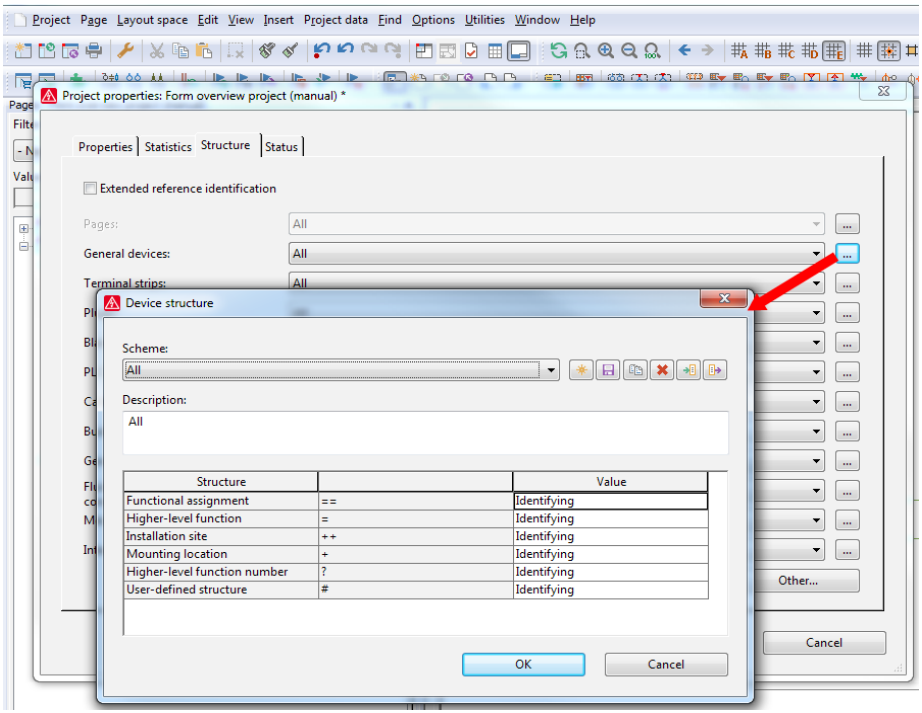


Fig. 10.6
Structuring of the remaining objects

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.

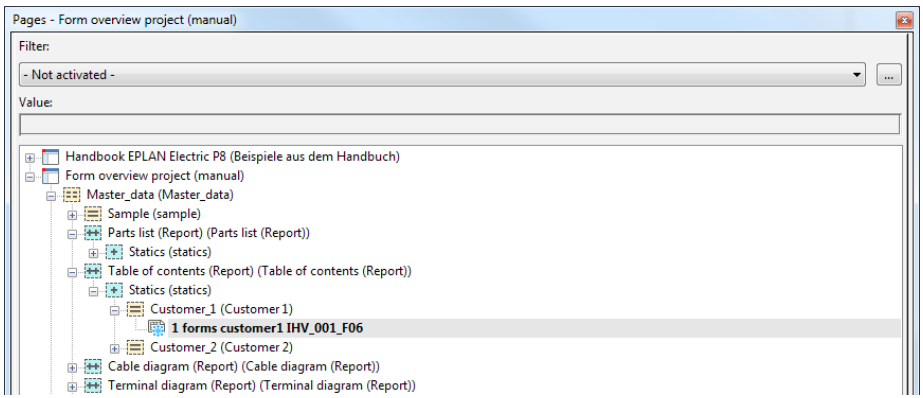


Fig. 10.7 Possible layout of the page structure with the various types of identifiers

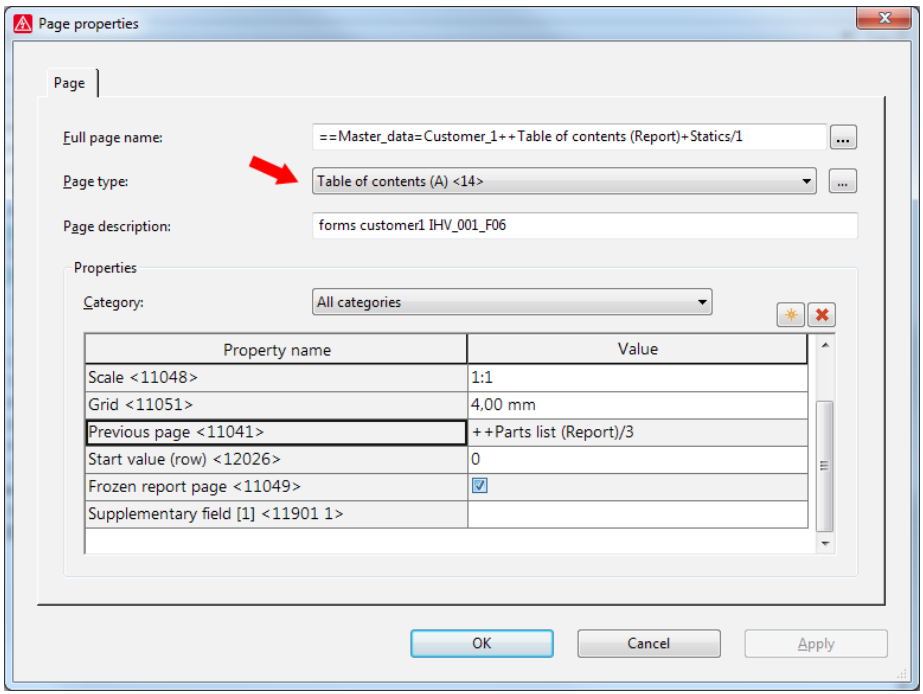


Fig. 10.8 Page properties of a page in the project

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).

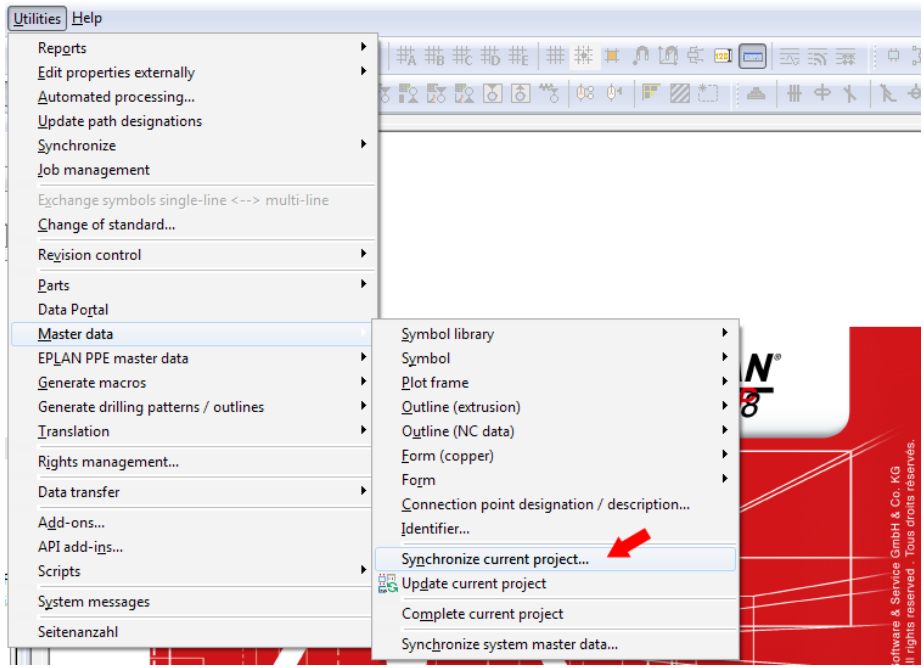


Fig. 10.9 Synchronize master data menu for a project

In the subsequent **Synchronize master data** dialog, you can then copy to the project all (non-)existing system master data. **Note:** No filter may be active.

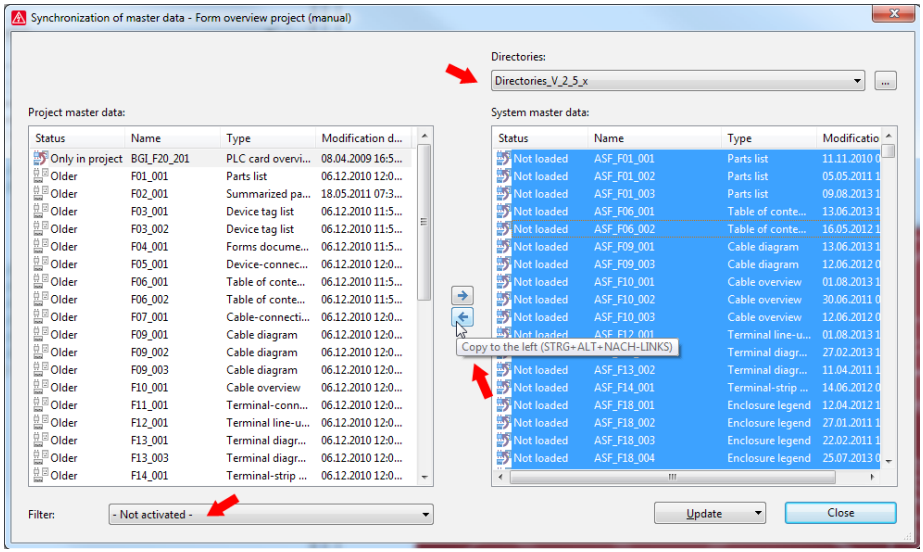


Fig. 10.10 Synchronize master data dialog

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.

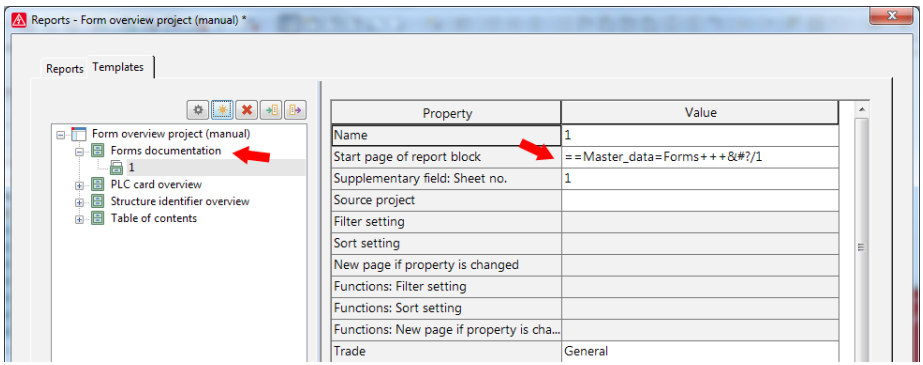


Fig. 10.11 Templates for the forms and plot frame documentation reports

A possible final result is shown in Fig. 10.12.

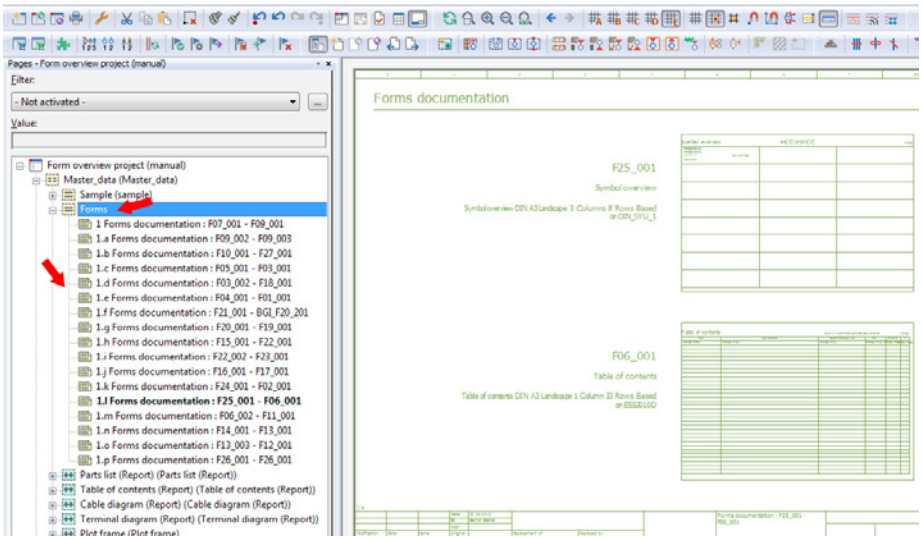


Fig. 10.12 Representation in the page navigator

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

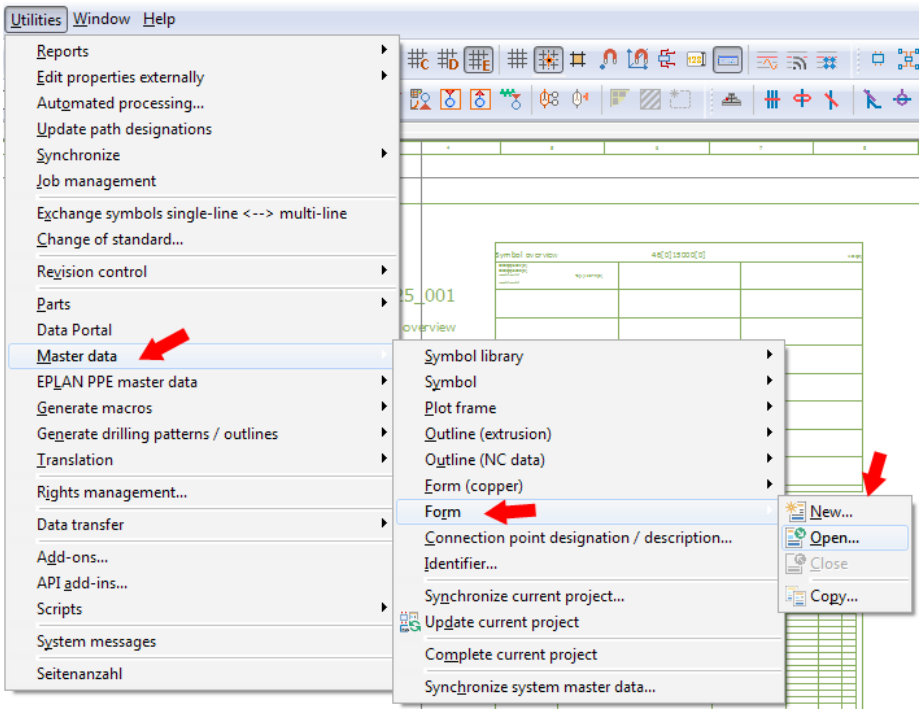


Fig. 10.13 Calling the form editor

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.

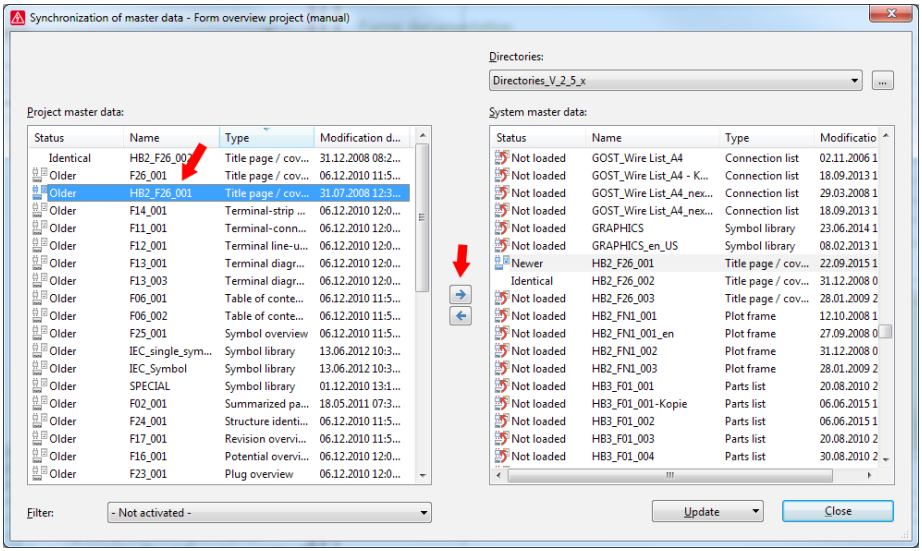


Fig. 10.14 Synchronize master data dialog

When existing master data is found, EPLAN opens the **COMPARE MASTER DATA** dialog. A decision must be made here for each master data file as to whether the system master data should be replaced with the project master data.

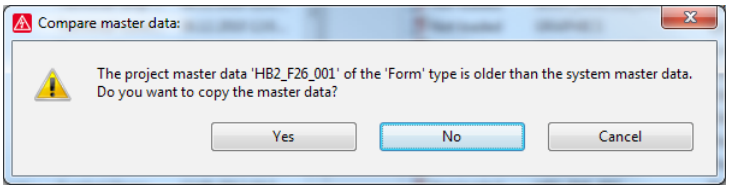


Fig. 10.15 Prompt asking whether the system master data should be overwritten

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. In addition, the system messages contain further information about the master data synchronization just completed.

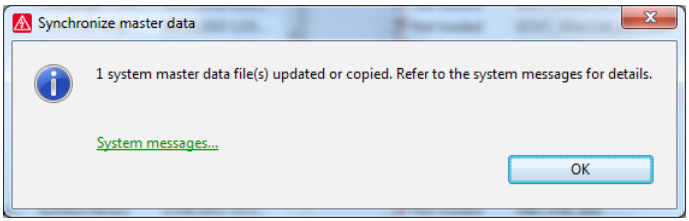


Fig. 10.16 Result message in the Synchronize master data dialog

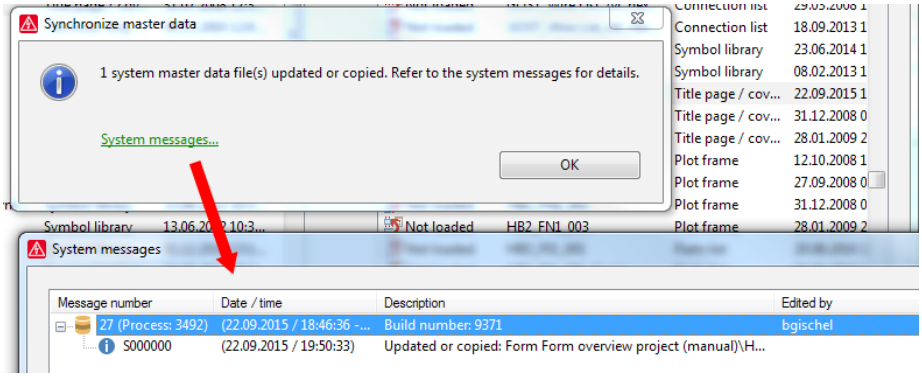


Fig. 10.17 System messages

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.

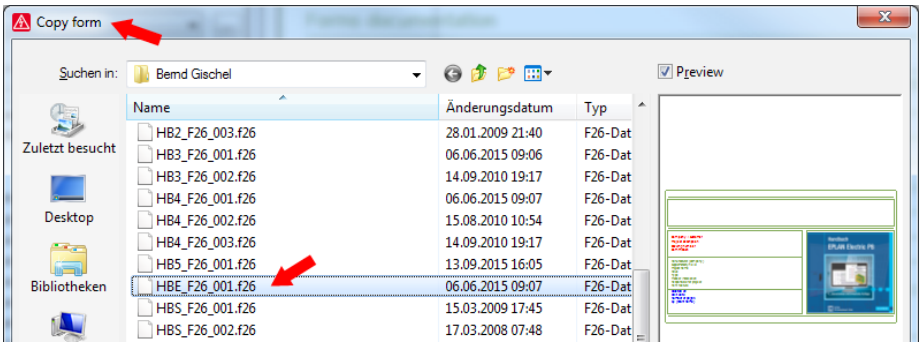


Fig. 10.18 Copy form dialog

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.

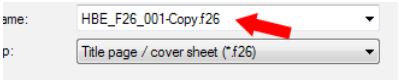


Fig. 10.19 Entry in the File name field

The form can now be saved via the **SAVE** button. EPLAN closes the dialog, opens a temporary page with the selected **[Form] page type** and displays the copied form on this page for editing.

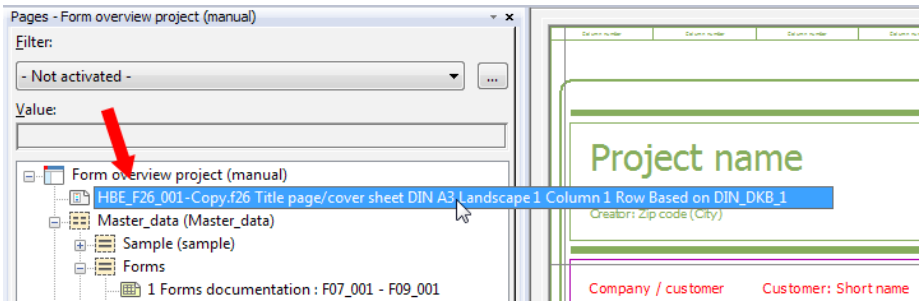


Fig. 10.20 Temporary page with the form in the page navigator

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.

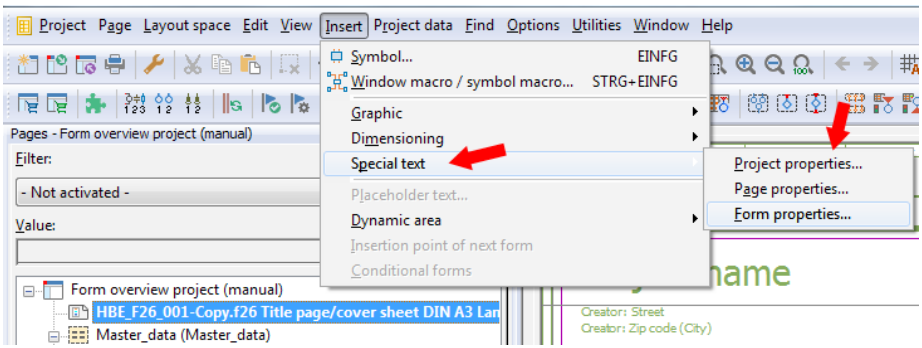


Fig. 10.21 Special texts

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.

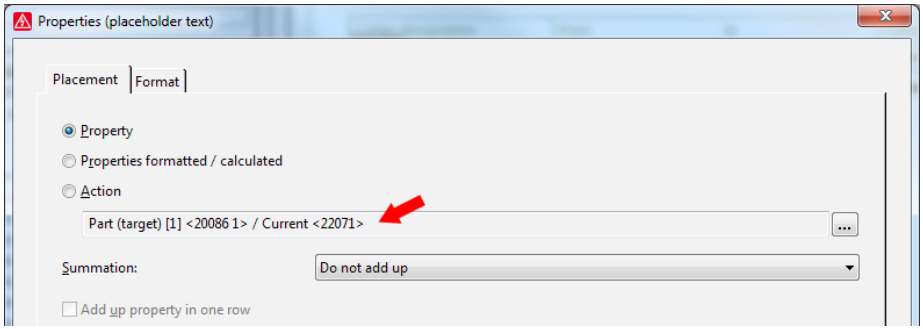


Fig. 10.22 Display placeholder text dialog

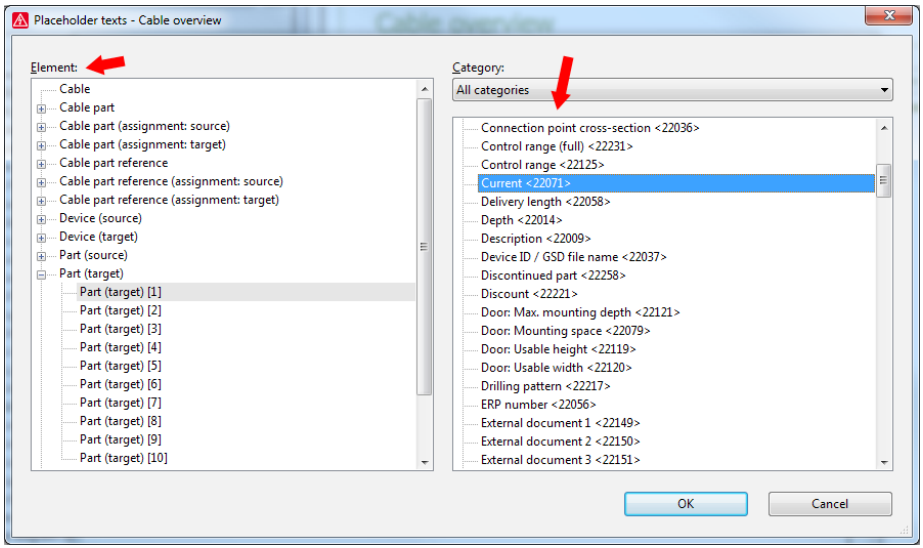


Fig. 10.23 Select placeholder text dialog

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**.

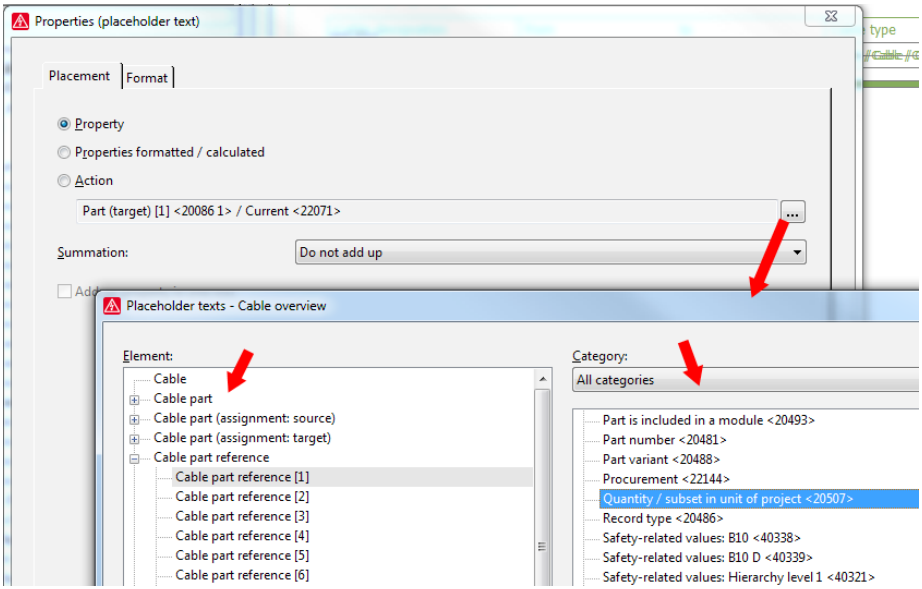


Fig. 10.24 Preselection (left) and selection (right) of the property

EPLAN closes the dialog and transfers the selected placeholder text to the Placeholder text dialog.

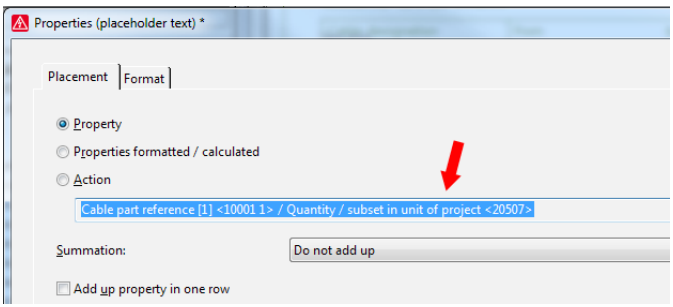


Fig. 10.25 Placeholder text applied

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.

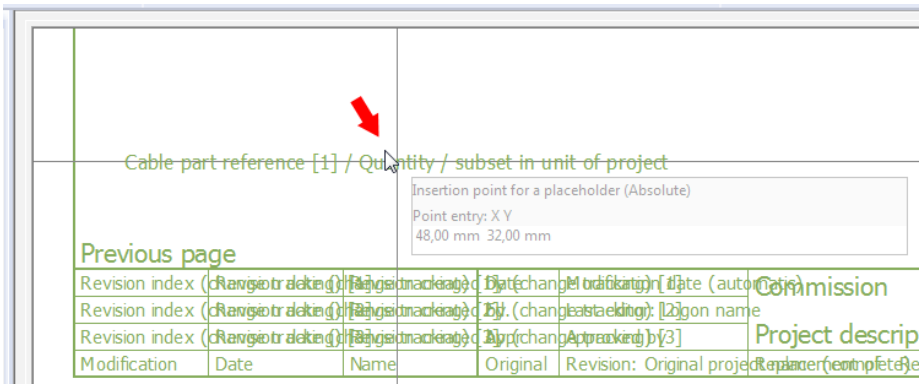
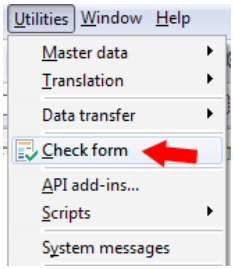


Fig. 10.26 Selecting the placement

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.



Check form function

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."

Fig. 10.27 Check form

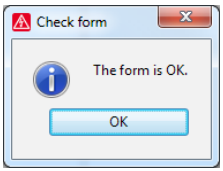


Fig. 10.28 Message that the form is OK

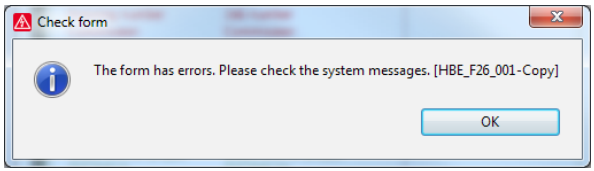


Fig. 10.29 Error in the form

If the form has errors, EPLAN indicates this and writes the errors to the system messages, but also indicates the error already in the subsequent dialog.

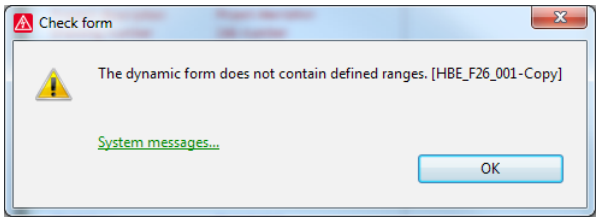


Fig. 10.30 Detailed error information

These should be corrected to avoid incorrect system master data.

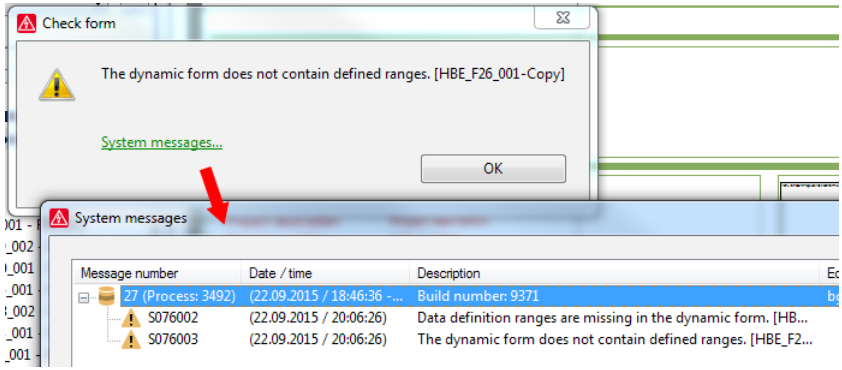


Fig. 10.31 System messages about the form errors

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.

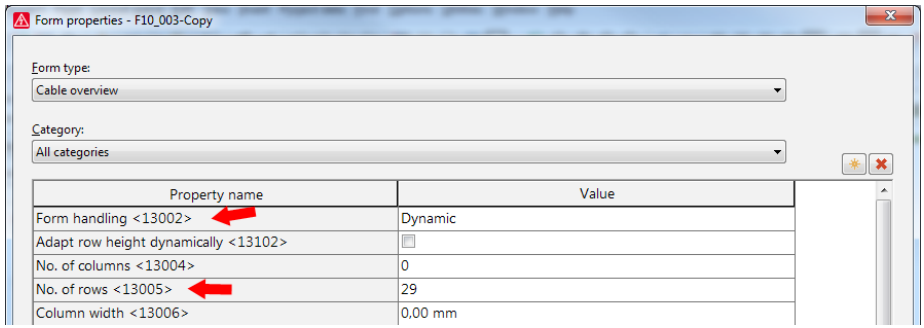


Fig. 10.32 Form properties dialog

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. 10.33. 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.

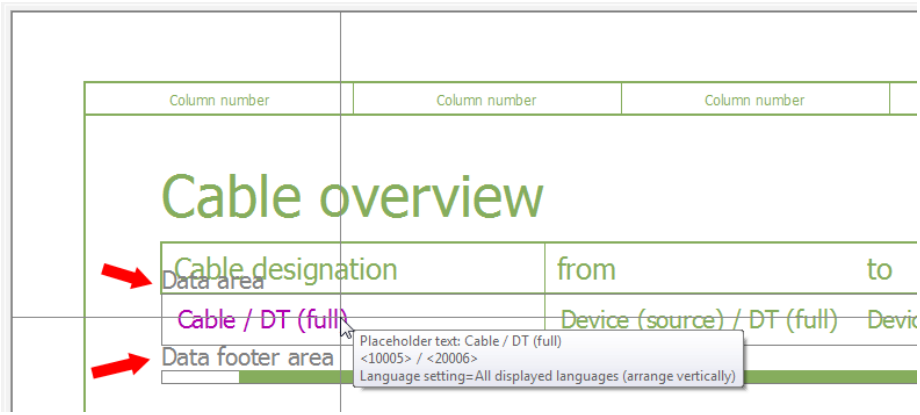


Fig. 10.33 Example of a dynamic form

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. 10.34. 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.

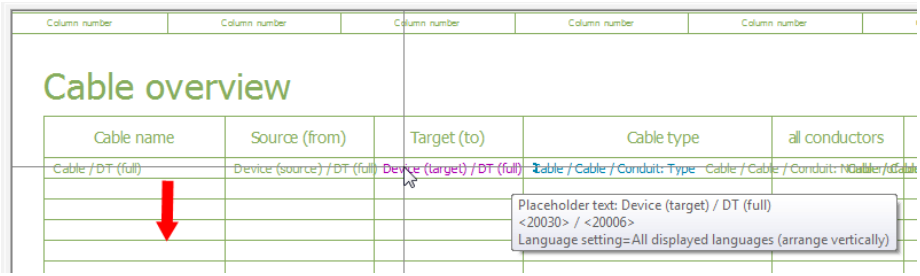


Fig. 10.34 Example of a static form

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.

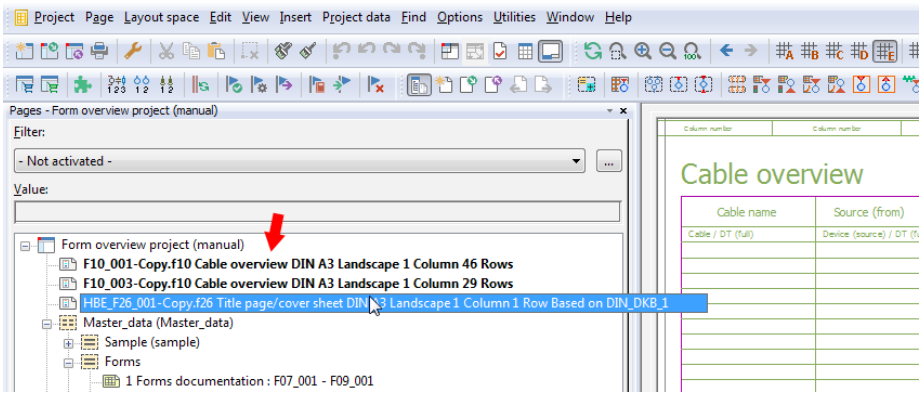


Fig. 10.35 Example: Form is open for editing

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 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.

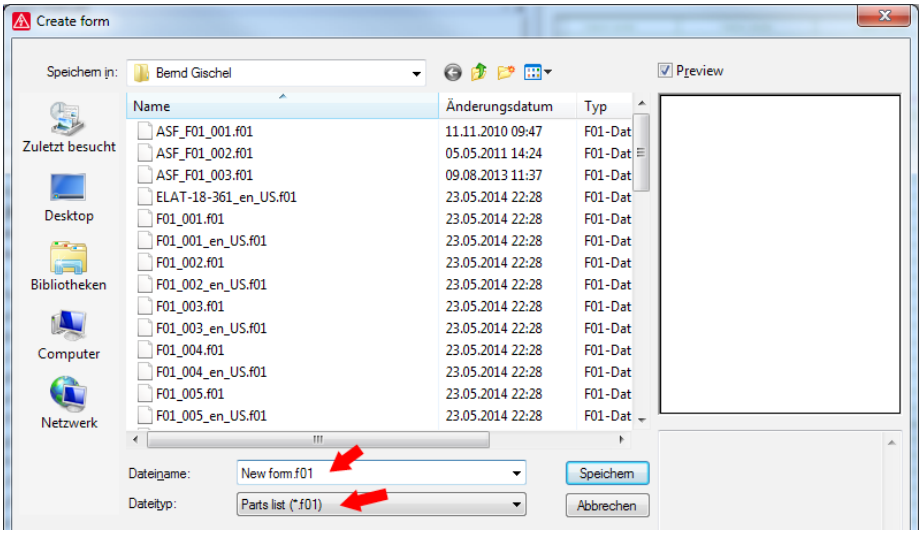


Fig. 10.36
Defining form name

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.

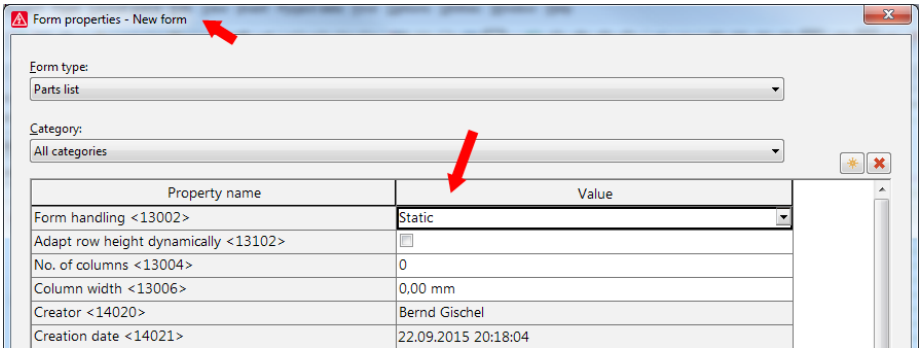


Fig. 10.37
Form properties dialog

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

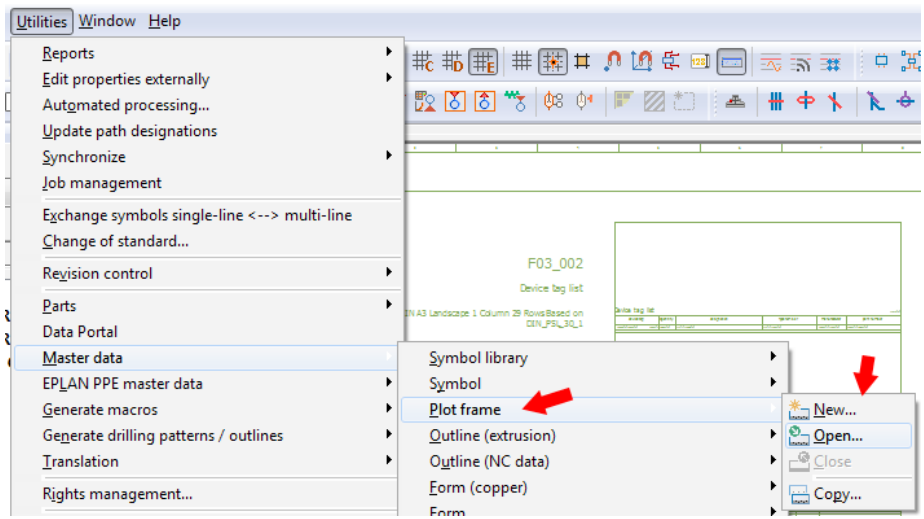


Fig. 10.38 Calling the plot frame editor

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.

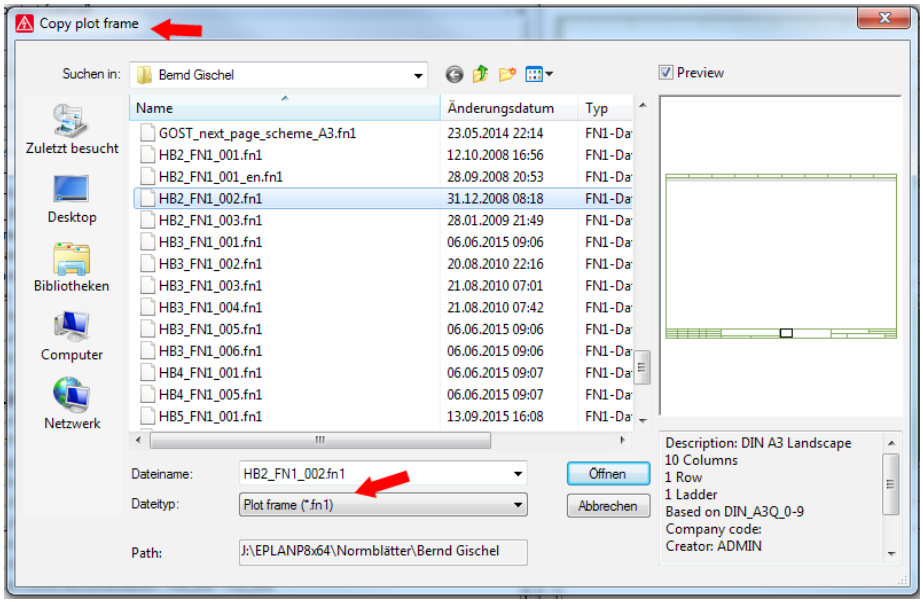
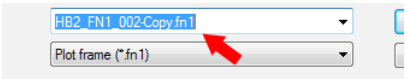


Fig. 10.39 Copy plot frame dialog

Fig. 10.40
Create 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.

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.

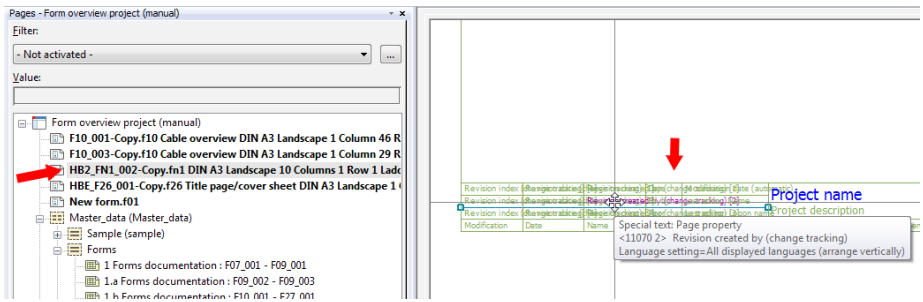


Fig. 10.41 Opened plot frame in the page navigator

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

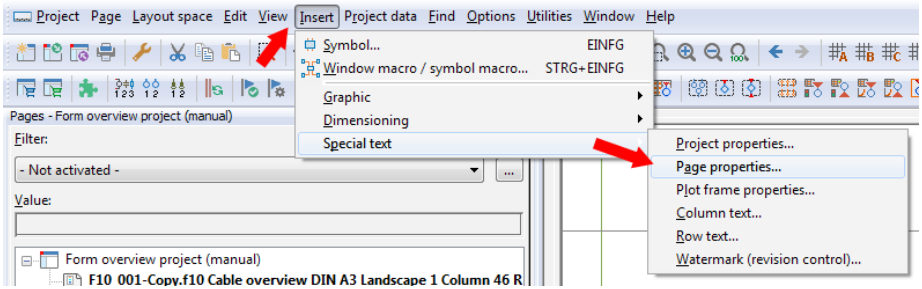


Fig. 10.42 Insert / Special text menu

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.

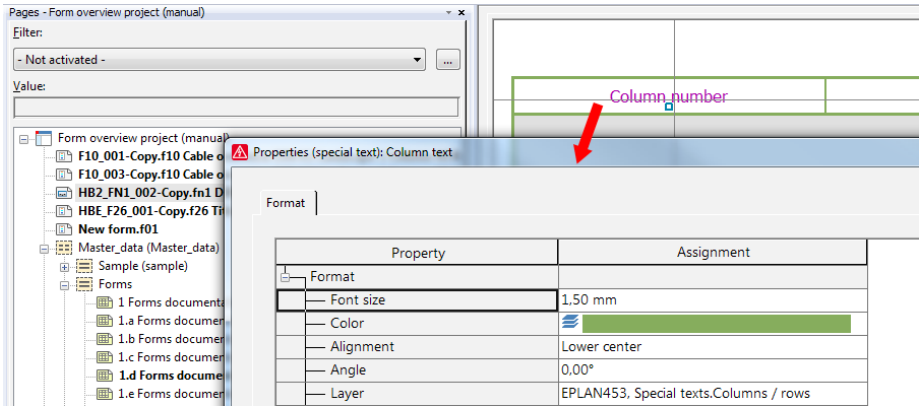


Fig. 10.43 Formatting options for the column and/or row texts

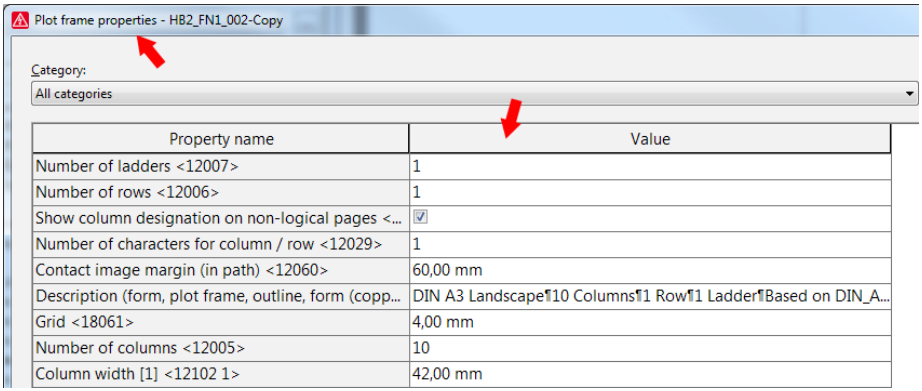


Fig. 10.44
Plot frame properties

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.

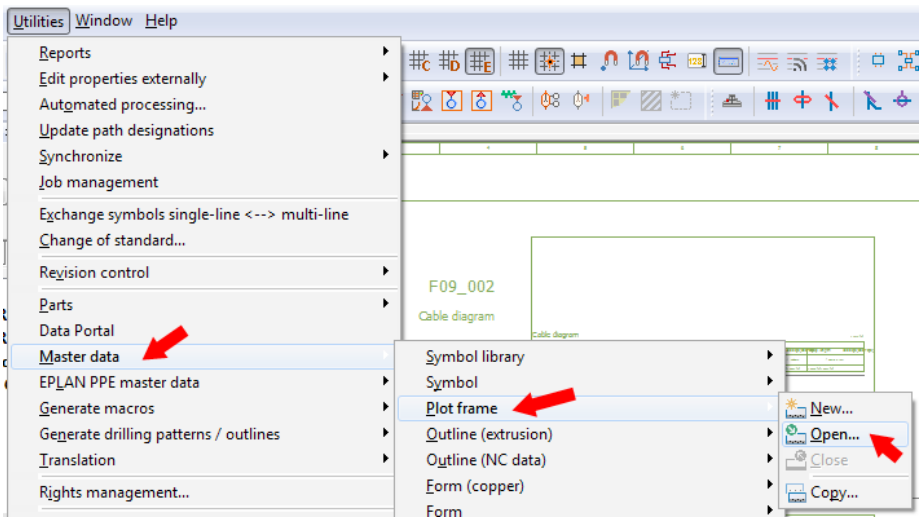


Fig. 10.45
Open plot frame dialog

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.

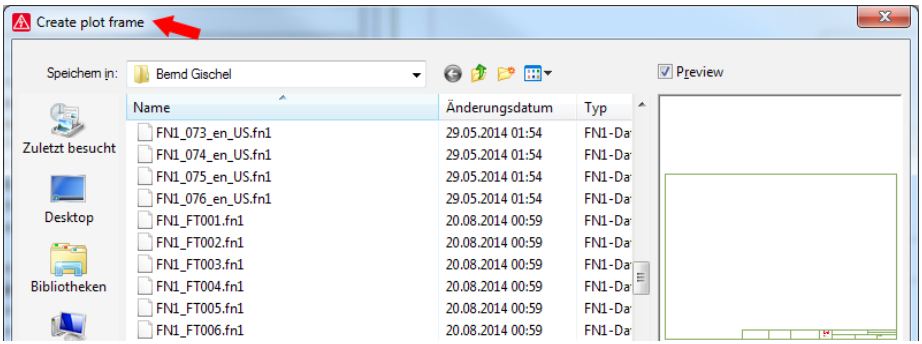


Fig. 10.46 Create plot frame dialog

You enter the new name of the plot frame in the **File name** field of the **Create plot frame** dialog.

Clicking the **SAVE** button creates the plot frame and EPLAN then opens the **plot frame properties** of the new plot frame and also displays an empty temporary page in the page navigator of the current project.

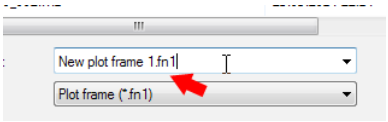


Fig. 10.47
New plot frame name in the File name field

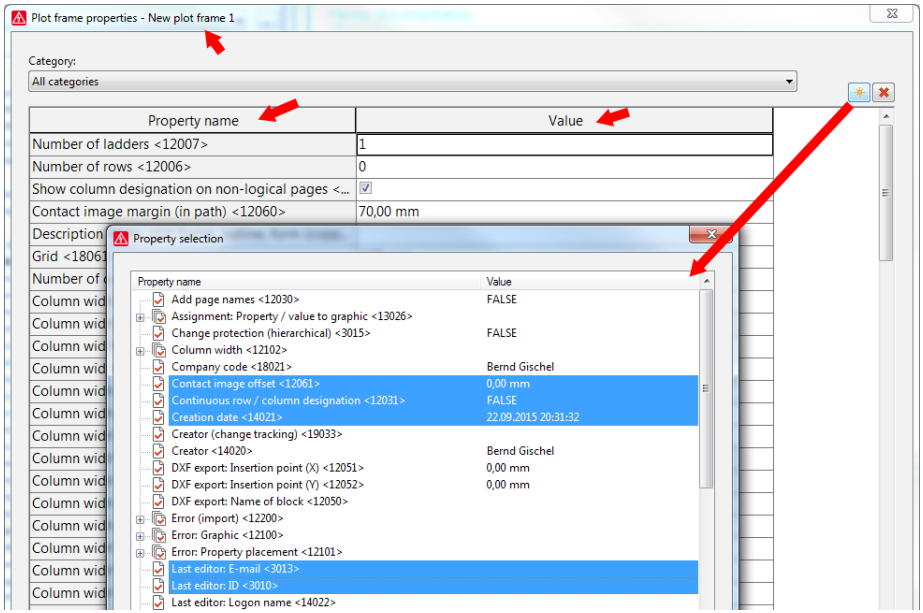


Fig. 10.48 Plot frame properties of the new plot frame

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.

NOTE: EPLAN supports the import of data from old versions of EPLAN 21, EPLAN PPE and EPLAN Cabinet up to EPLAN Electric P8 Version 2.3. From Version 2.4, this direct data import into the EPLAN Electric P8 interface will no longer be available.

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 previous versions, 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 5, 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 TRANSFER** menu in the **EPLAN 5/FLUIDPLAN** data transfer menu items (for EPLAN 5 and Fluid projects, as well as for EPLAN 5/Fluid master data).

In general, it is possible to import both entire projects and individual data using EPLAN Electric P8. Additional functions and calls were integrated into the individual data import menu items (EPLAN 5/fluidPLAN) to enable you, for example, to import into EPLAN Electric P8 an individual plot frame or parts data exported from the older system.

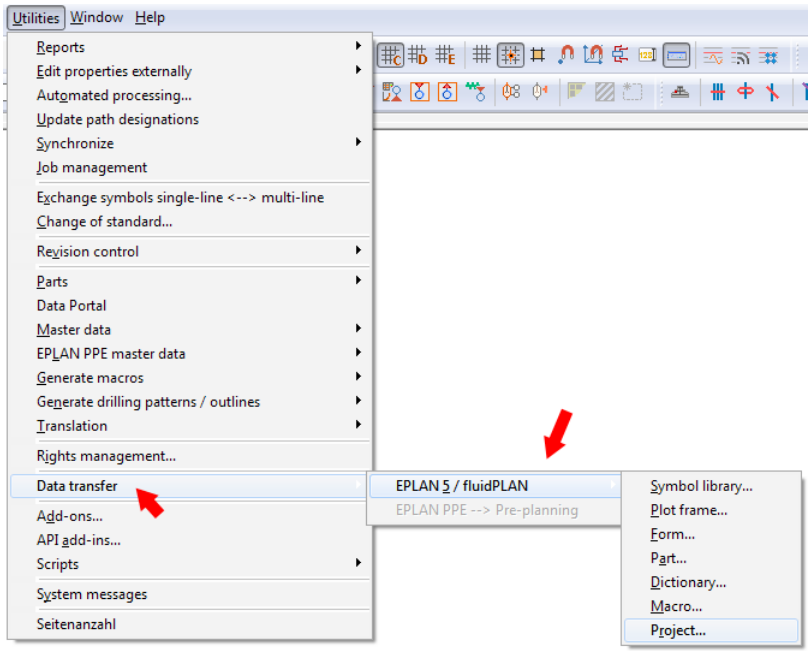


Fig. 11.1 Data transfer menu

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 Data Portal

The previous chapters 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.

12.1.1 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.

12.1.1.1 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.

12.1.2 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.

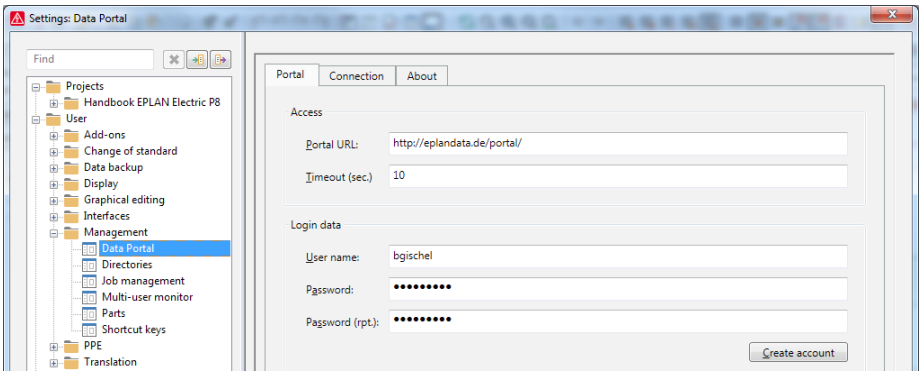


Fig. 12.1 EPLAN Data Portal settings

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.

If you do not have your own account yet, you must create it once. Define a user name and password, then re-enter the password a second time to confirm. Then, click the *Create account* button, and EPLAN will create the account.

The settings on the *Connection* tab normally do not need to be changed.

12.1.2.1 Data Portal navigator

The **Data Portal navigator** is the main focus of the EPLAN Data Portal. It is started via the UTILITIES/DATA PORTAL menu.

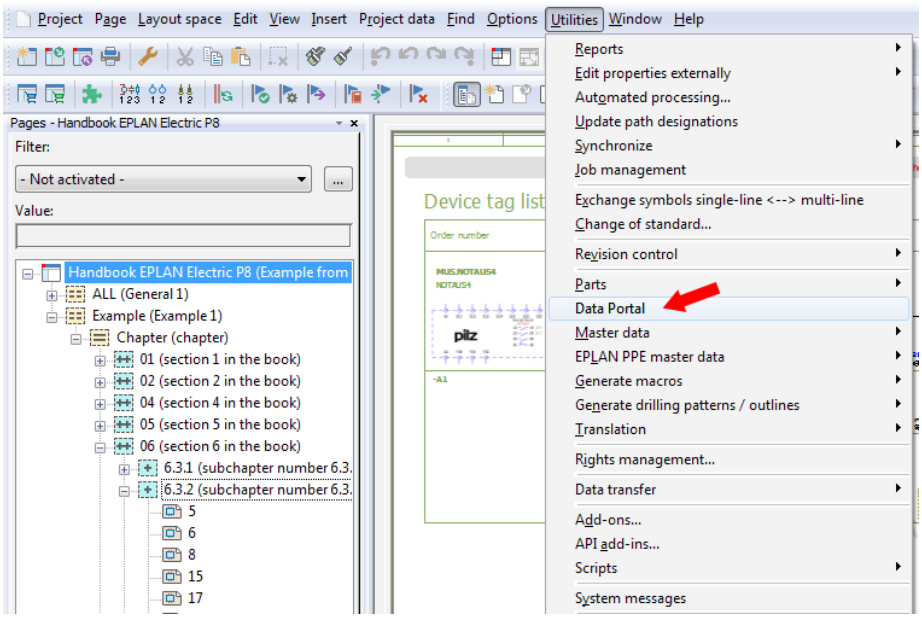


Fig. 12.2
Data Portal navigator menu item

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.

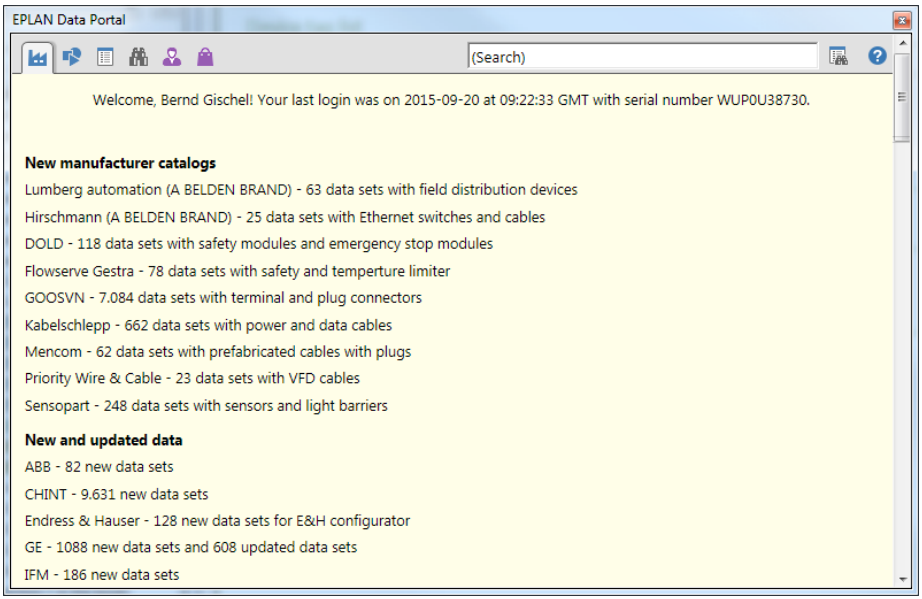


Fig. 12.3
Possible Data Portal view with optional information

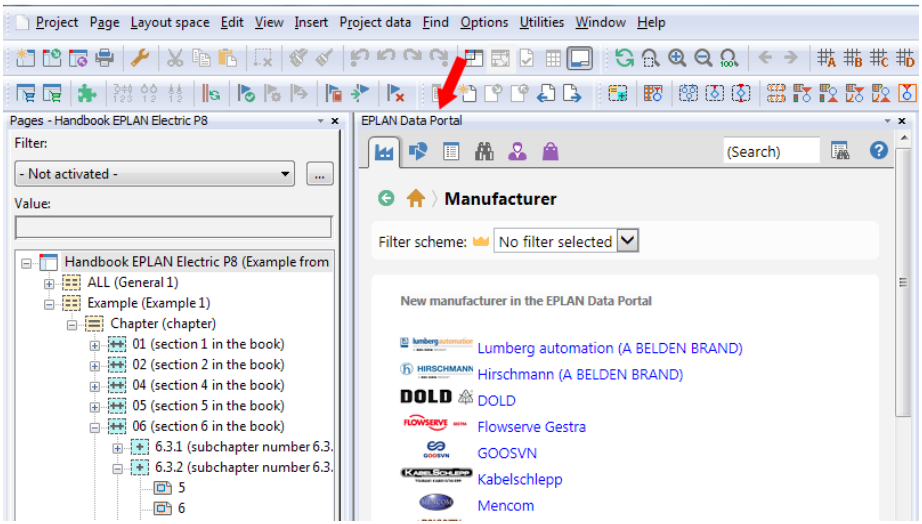


Fig. 12.4 The Data Portal navigator when docked

The navigation/toolbar, with buttons for functions such as **DISPLAY MANUFACTURERS**, **SEARCH**, **SETTINGS** and **ONLINE HELP**, is located under the navigator dialog logo.

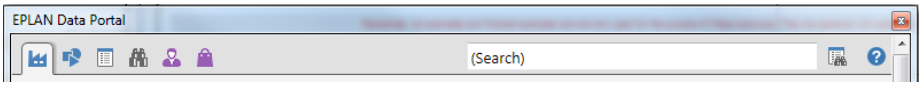


Fig. 12.5 The data portal navigator's navigation/toolbar

The navigation bar buttons have the following meanings:

	Open start page		Display manufacturer
	Display catalog		Display list
	Display search with search criterion		Display user data
	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. 12.6 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.

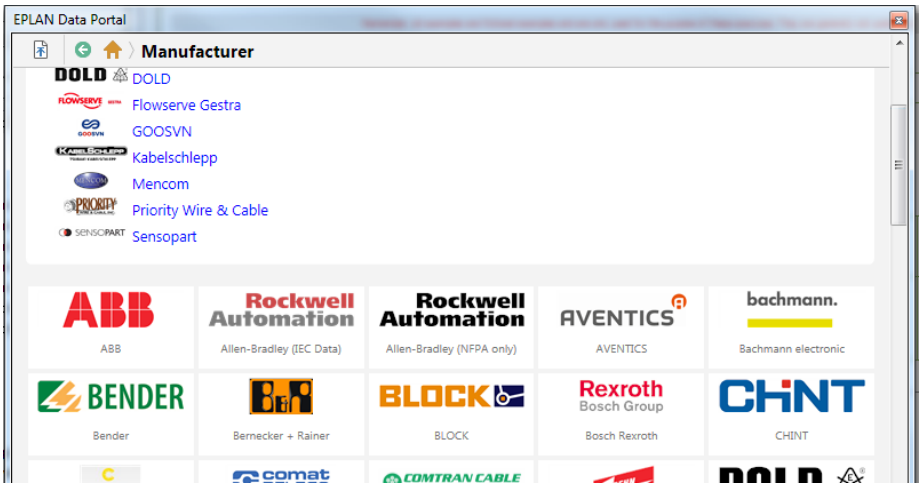


Fig. 12.7 Manufacturers in the Data Portal

12.1.3 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 import or insert missing parts data, macros or a device list of the parts contained in the shopping cart onto schematic pages and/or parts management. This, of course, assumes that the missing data is actually available in the Data Portal.

12.1.3.1 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. Then, you start the search.

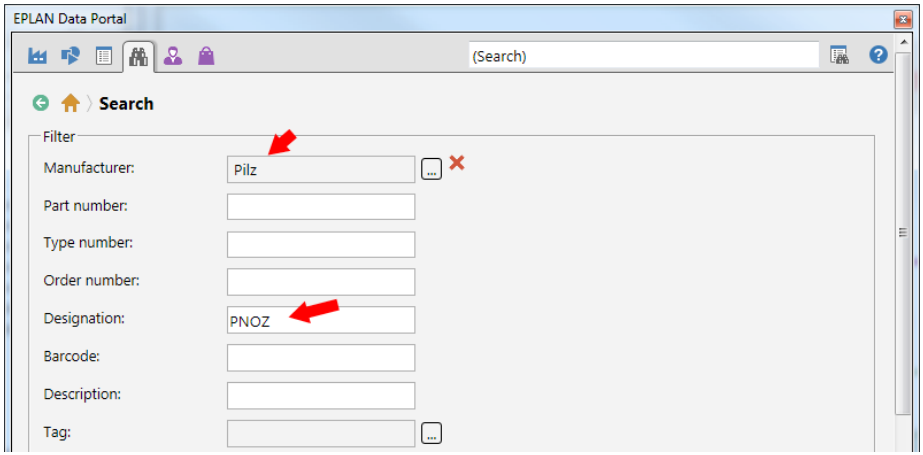


Fig. 12.8 Search function with criteria selection

EPLAN opens the **Search** dialog. Here you can enter in the **Designation** field the *PNOZ* value and, in the **Manufacturer** field, select the *Pilz* entry (via the *More* button) from the list of manufacturers. All other fields are not used and/or remain empty. Click the **FIND** button at the very bottom of the **Search** dialog to start the search.

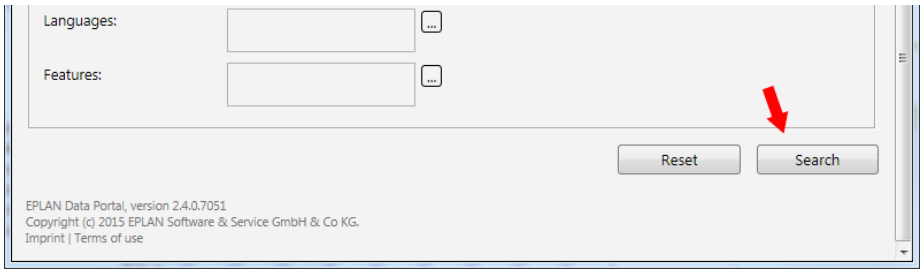


Fig. 12.9 Starting a search

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.

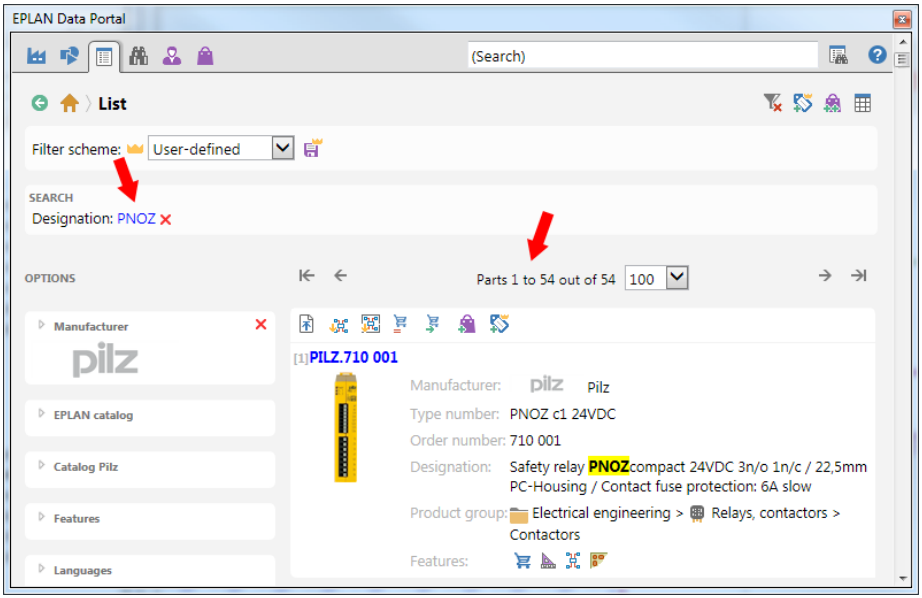


Fig. 12.10
Parts found

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, 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.

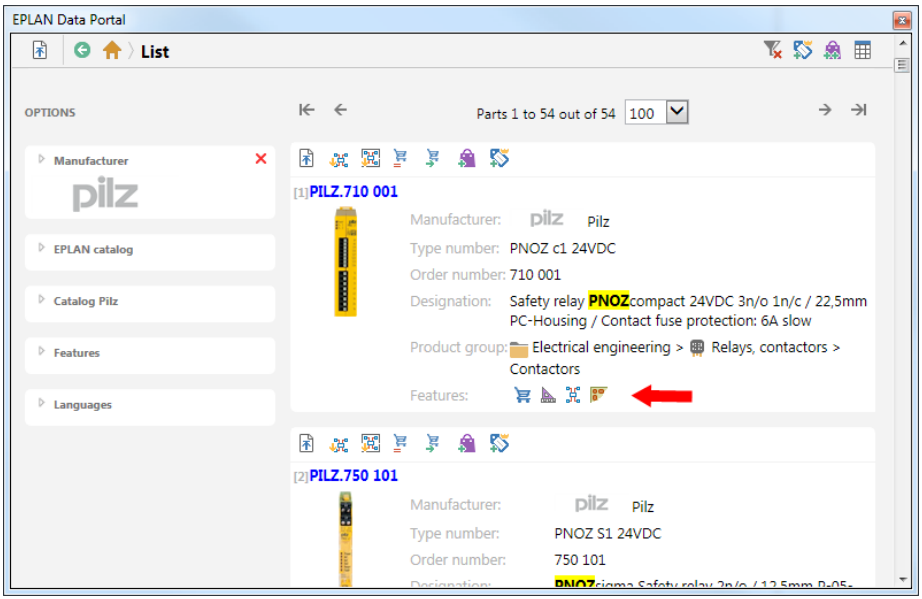


Fig. 12.11
Characteristics of a part

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.

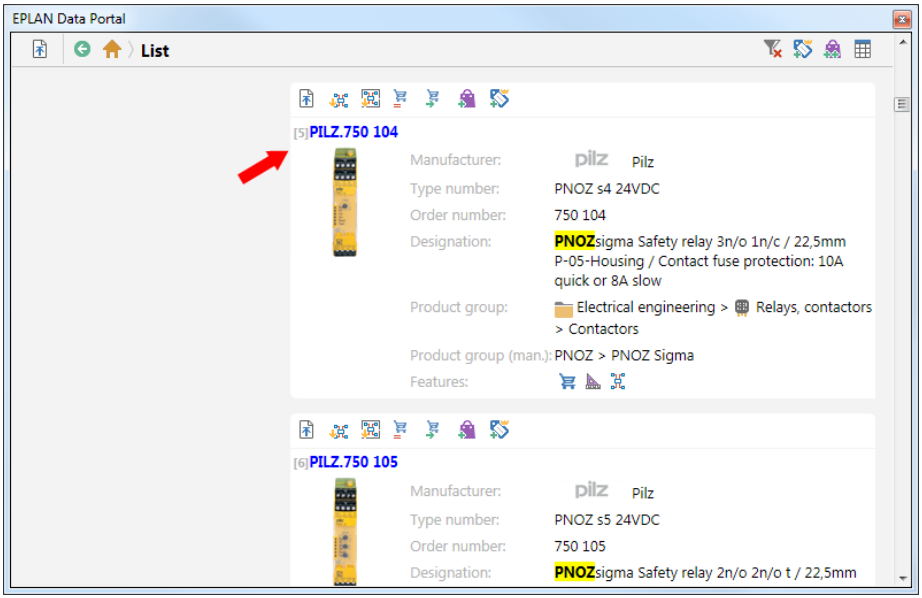


Fig. 12.12 Selected PNOZ device

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 of a filtered parts list.

	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.

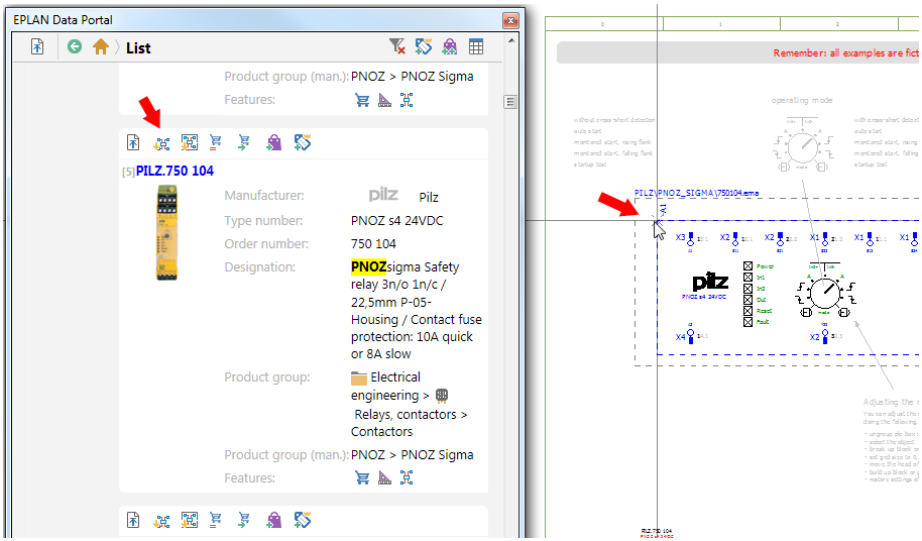


Fig. 12.13
Directly placing the selected macro in the graphical editor

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.

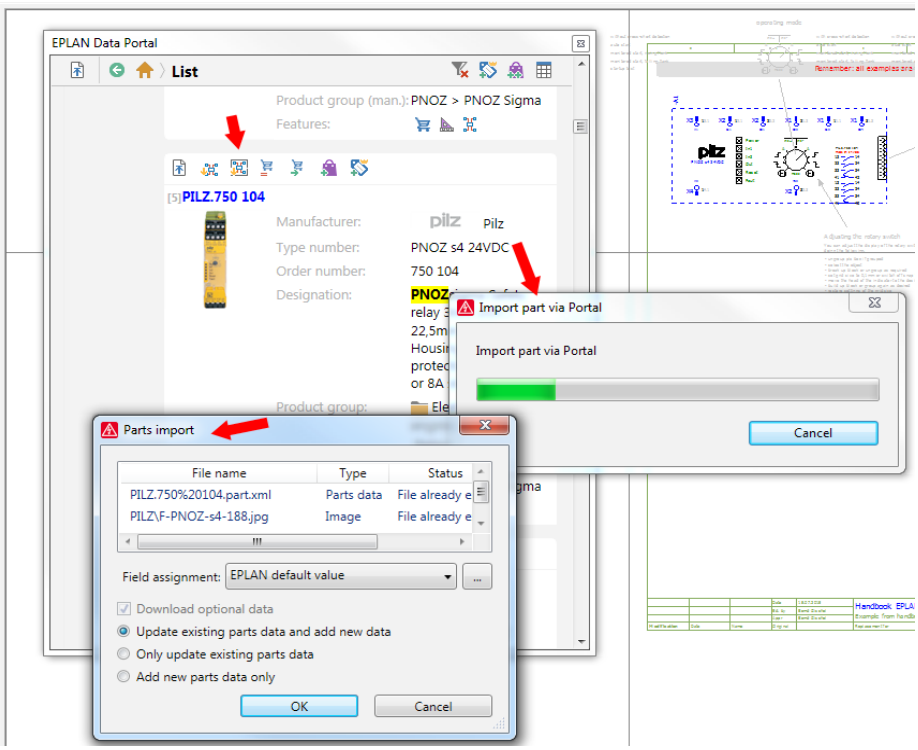


Fig. 12.14
Inserting the device in the graphical editor

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 automatically and, depending on the settings, imports the corresponding parts data into parts management. The rest of the data is saved in the directory structure.

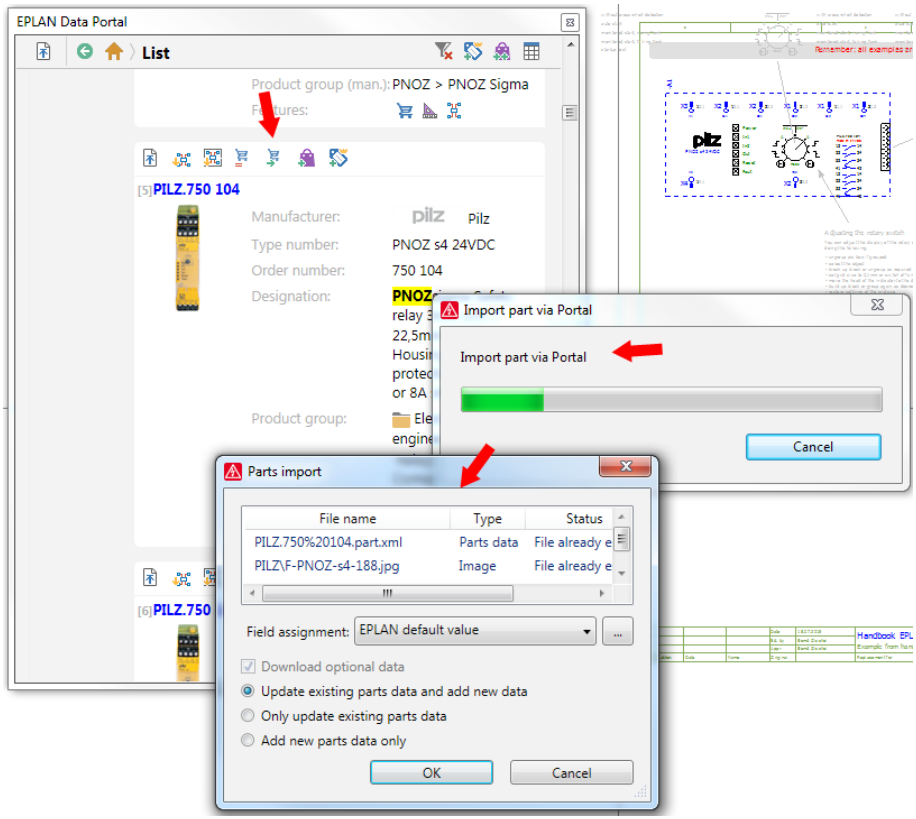


Fig. 12.15
Pure import of parts

After the import, EPLAN closes these dialogs and the graphical editor is active again.

	Put part <xyz> in the shopping cart (add)

Clicking this button adds the selected device to the shopping cart. The shopping cart is called up via the button in the toolbar.

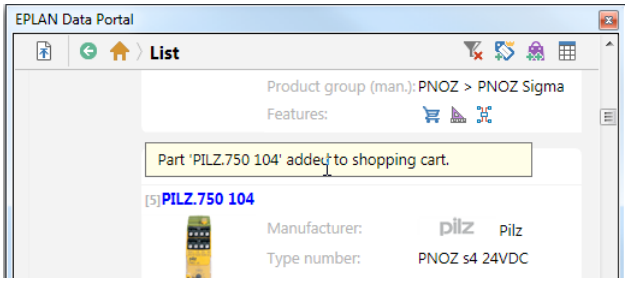


Fig. 12.16
Part added to shopping cart message

All devices in the shopping cart can be imported completely (collected).

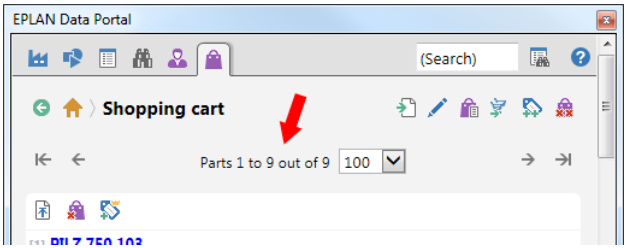


Fig. 12.17
Import the entire shopping cart

All of the parts can also be removed from the shopping cart.

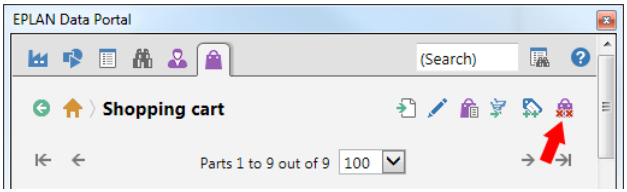


Fig. 12.18
Empty the entire shopping cart

NOTE: Apart from inserting parts, you can also assign to the selected part parts data of already placed devices in the Data Portal.

	Assign parts data

12.1.3.2 Generate device lists

Apart from the functions described, such as the import of parts data and inserting of parts as macros in the graphical editor, EPLAN Data Portal offers the function that allows you to import as device lists into the current project parts currently in the shopping cart.

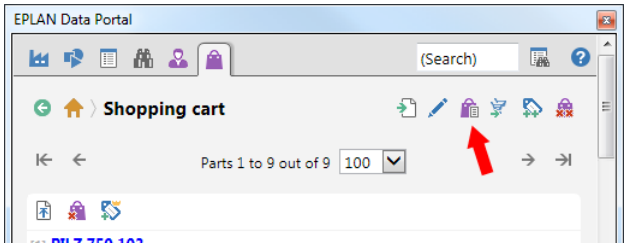


Fig. 12.19 Import shopping cart as device list button

Click the button for this purpose. EPLAN now starts the import automatically.

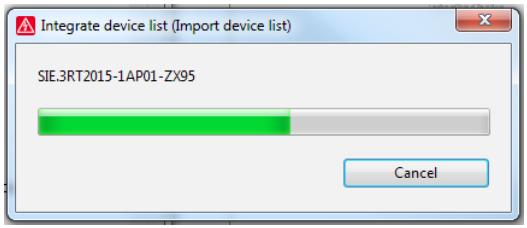


Fig. 12.20 Import device list dialog

When the import is complete, the device list will be available in the current project. The device list can be accessed via **PROJECT DATA/DEVICES/PARTS/DEVICE LIST**.

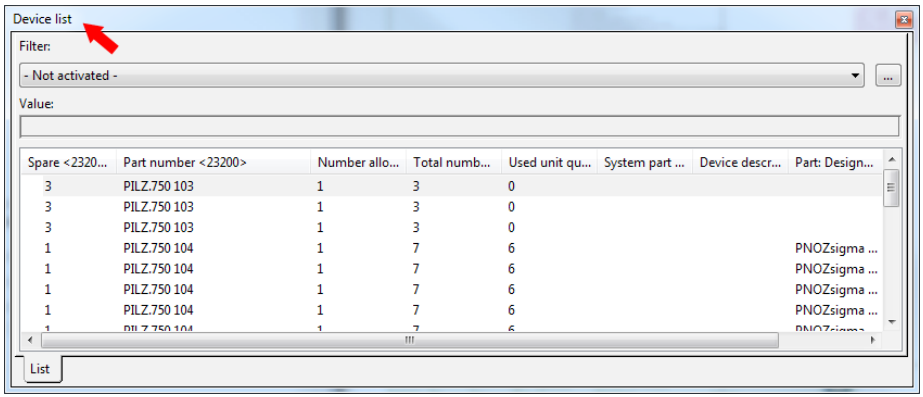


Fig. 12.21 Imported device list

12.2 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.

12.2.1 What are project options?

As the name indicates, project options are various options for (different design approaches) 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 easily 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.

12.2.2 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.

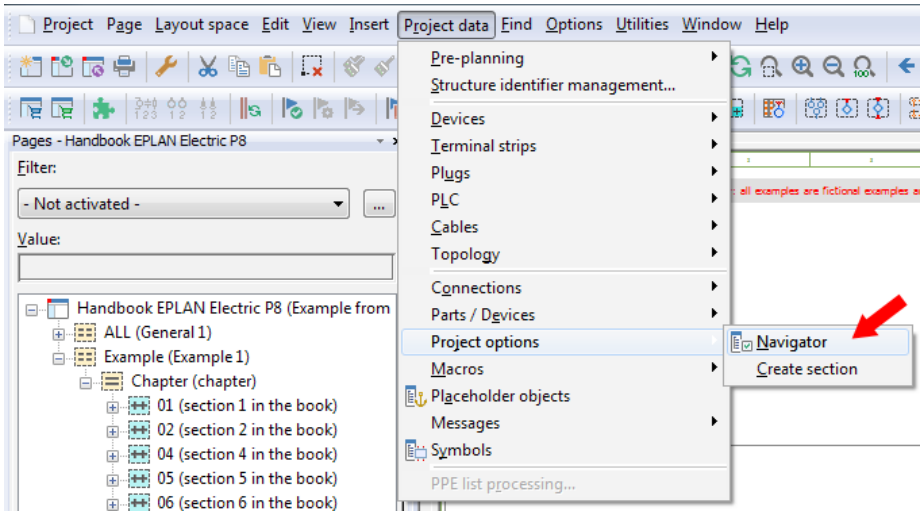


Fig. 12.22 Calling the project options navigator

- **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.

12.2.3 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 12.2.3.1 explains in just a few steps the most important points for creating these options and their associated parts.

12.2.3.1 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.

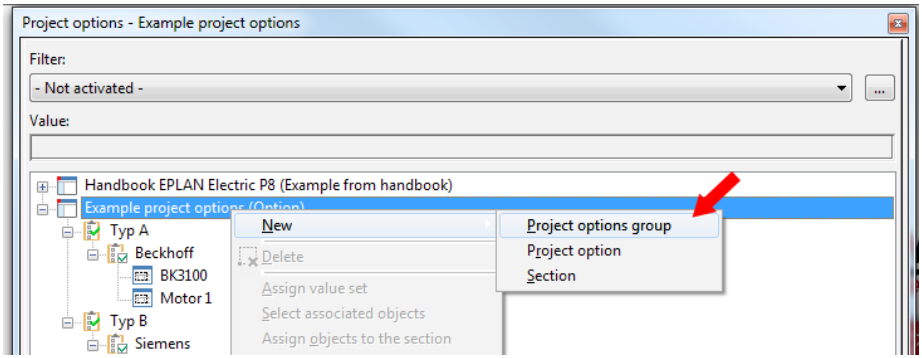


Fig. 12.23 Generating a project options group

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.

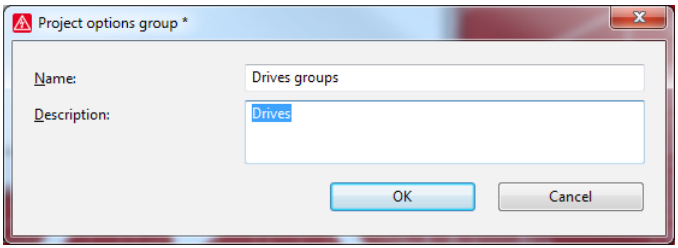


Fig. 12.24 Project options group dialog

You have to enter a **name** and **description**. When you click **OK**, the group is saved and displayed in the navigator.

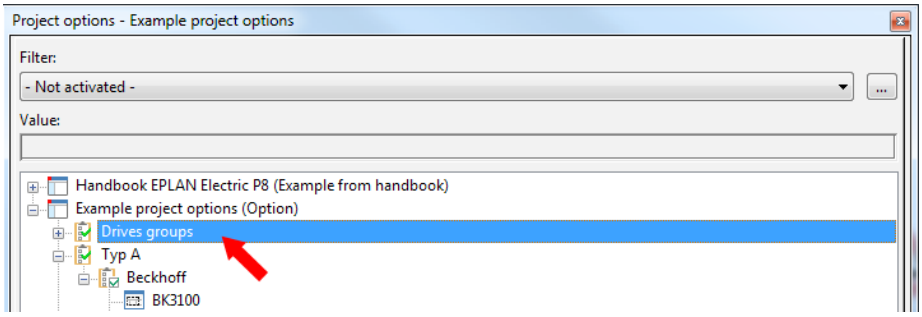


Fig. 12.25 View in the navigator

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.

12.2.3.2 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.

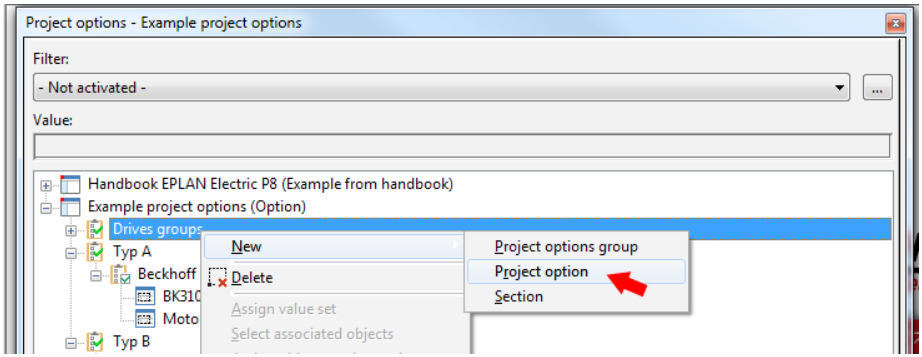


Fig. 12.26 Creating a new project option

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**.

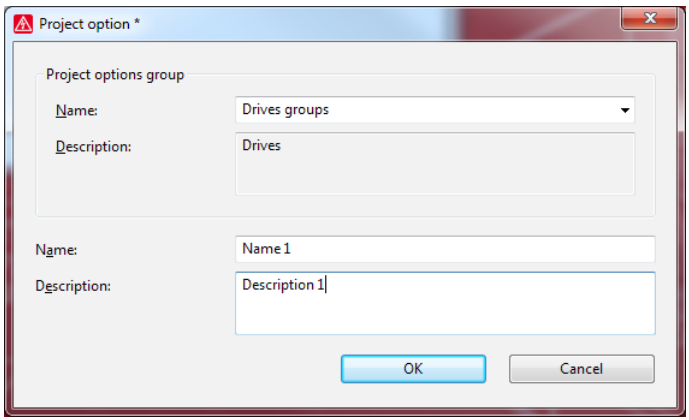


Fig. 12.27 Creating a new project option

When you click OK, the project option is saved and sorted under the selected project options group in the **project option navigator**.

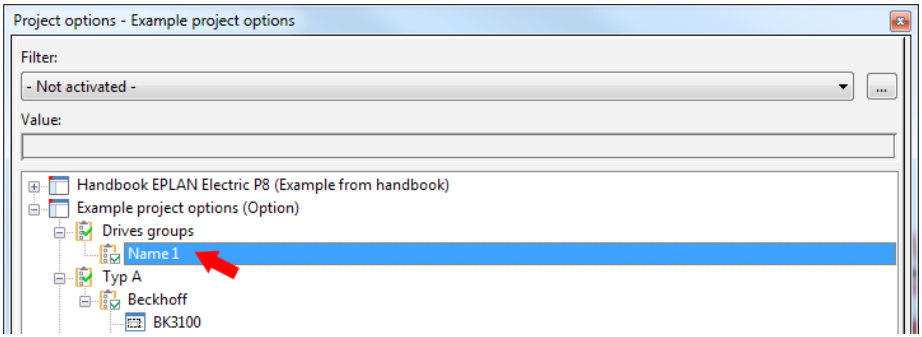


Fig. 12.28 Newly created project option

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.

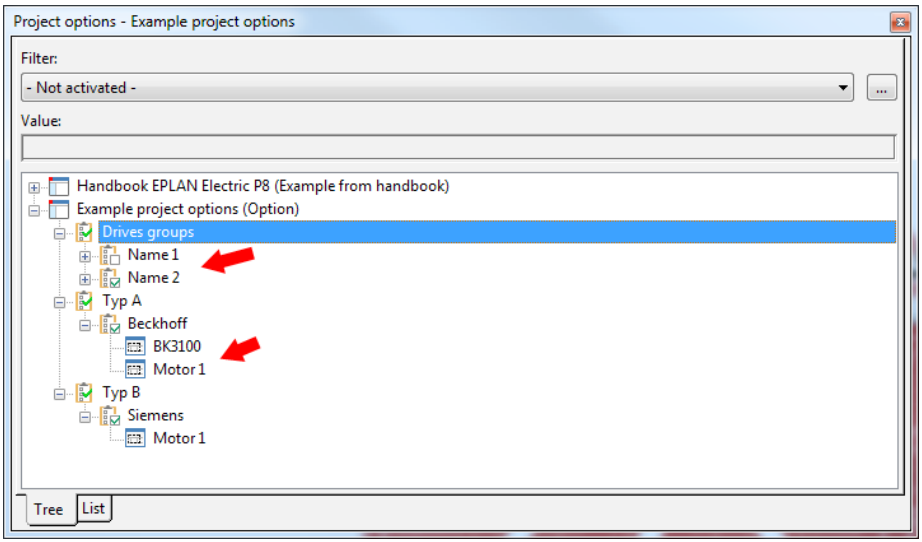


Fig. 12.29 Several project options

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.

12.2.3.3 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.

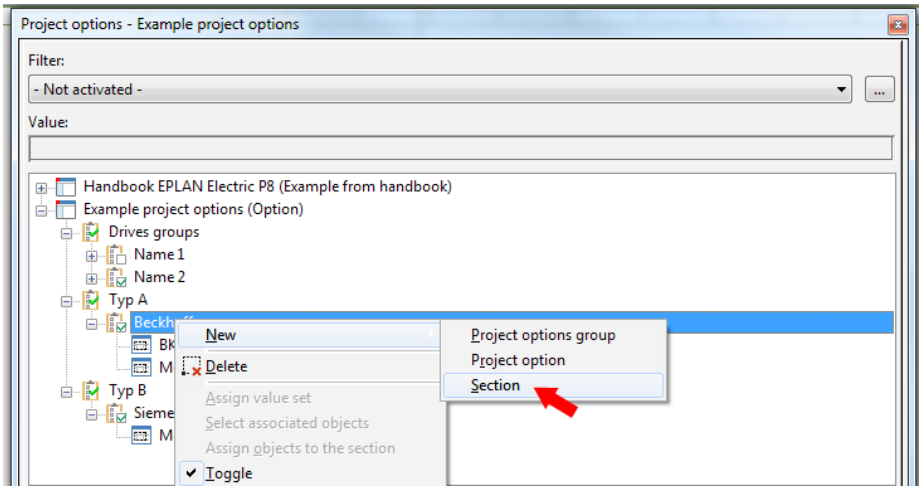


Fig. 12.30 Generating a new section

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.

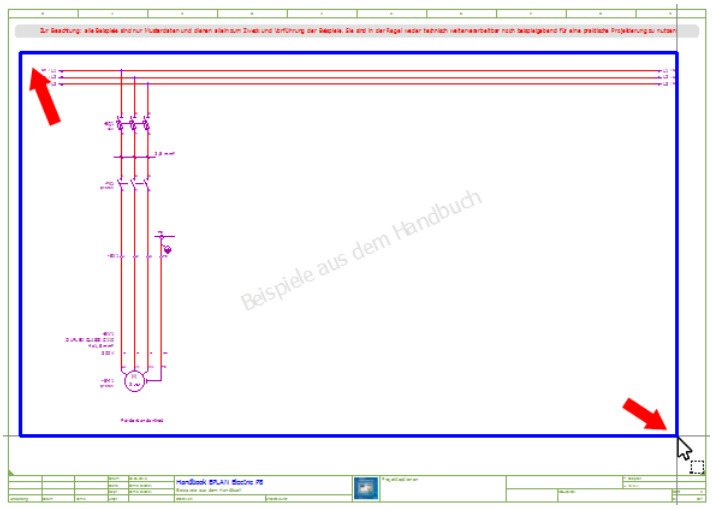


Fig. 12.31 Defining a section via a window

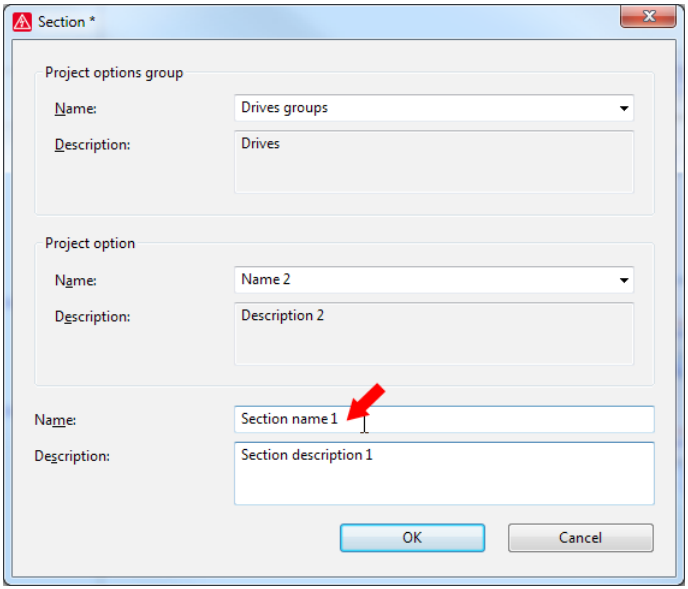


Fig. 12.32 Designating a new section

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**.

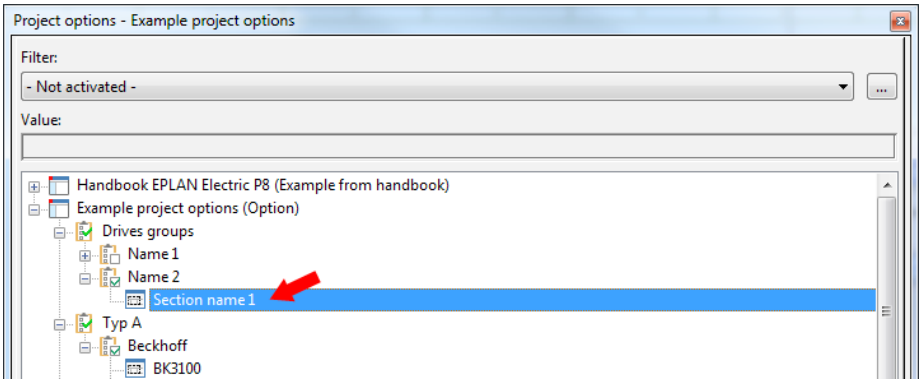


Fig. 12.33 Section in the navigator

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.

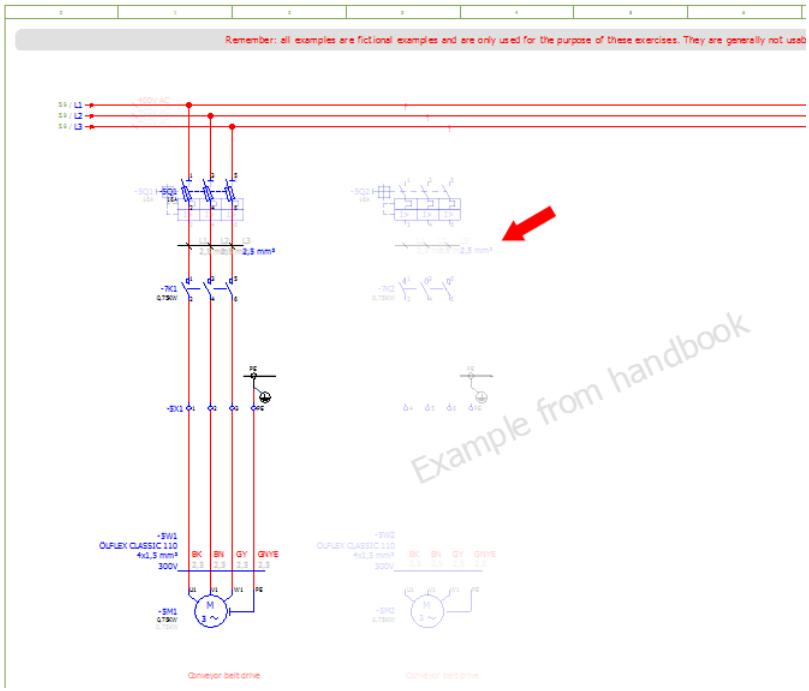


Fig. 12.34 Editing new circuit parts for a new section

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 before switching to transparent and place it over the now-enabled, transparent section. Now you can replace devices, add new devices, 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.

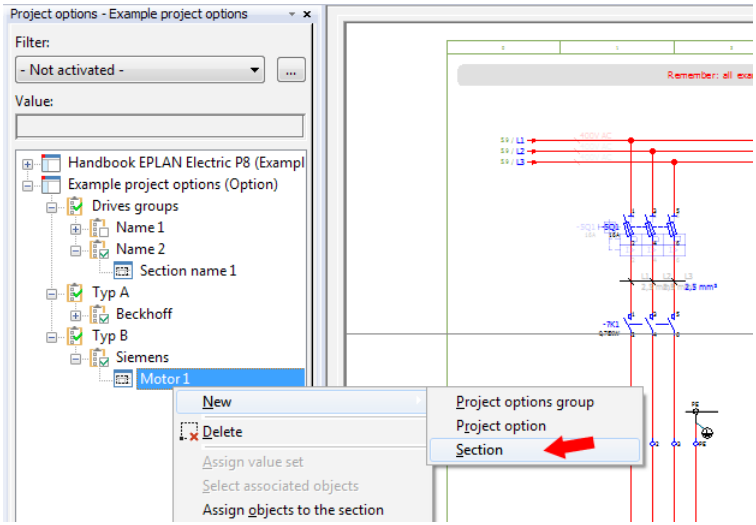


Fig. 12.35 Defining a new section

Once all changes for this section have been made, the section can be defined and saved. You again use windows to select the devices and save them with a click of the left mouse button. 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.

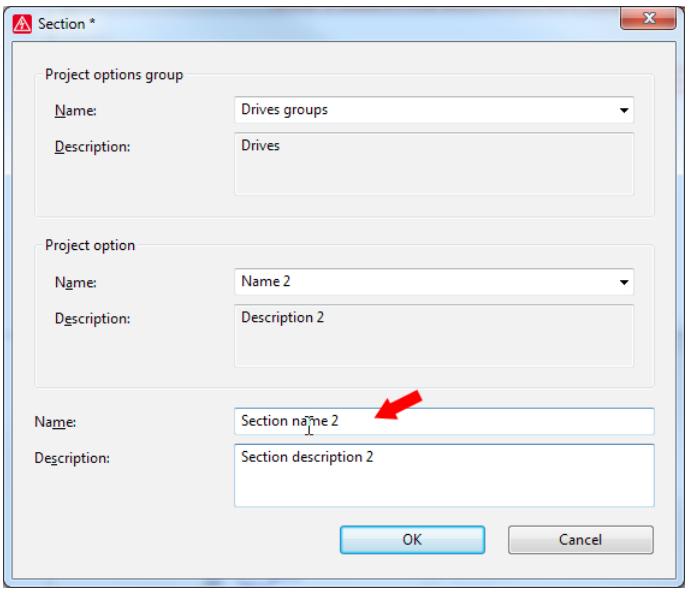


Fig. 12.36 Designating a new section

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.

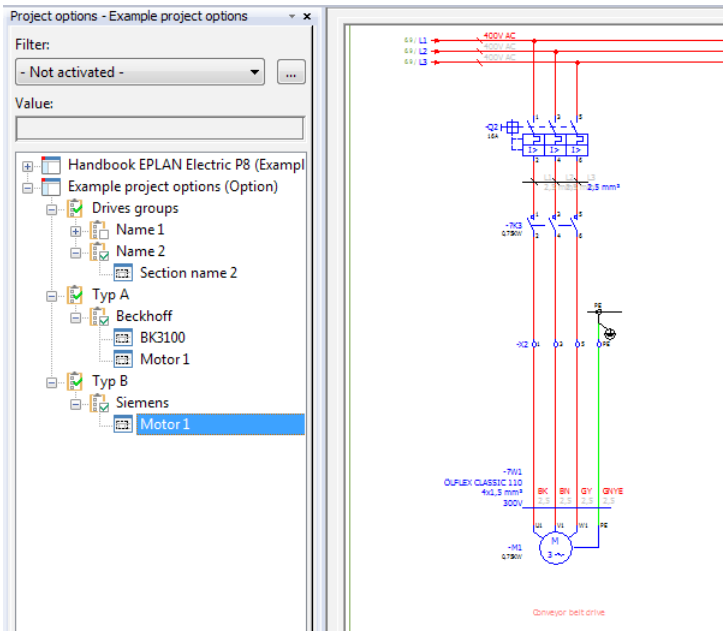


Fig. 12.37
Section with a motor

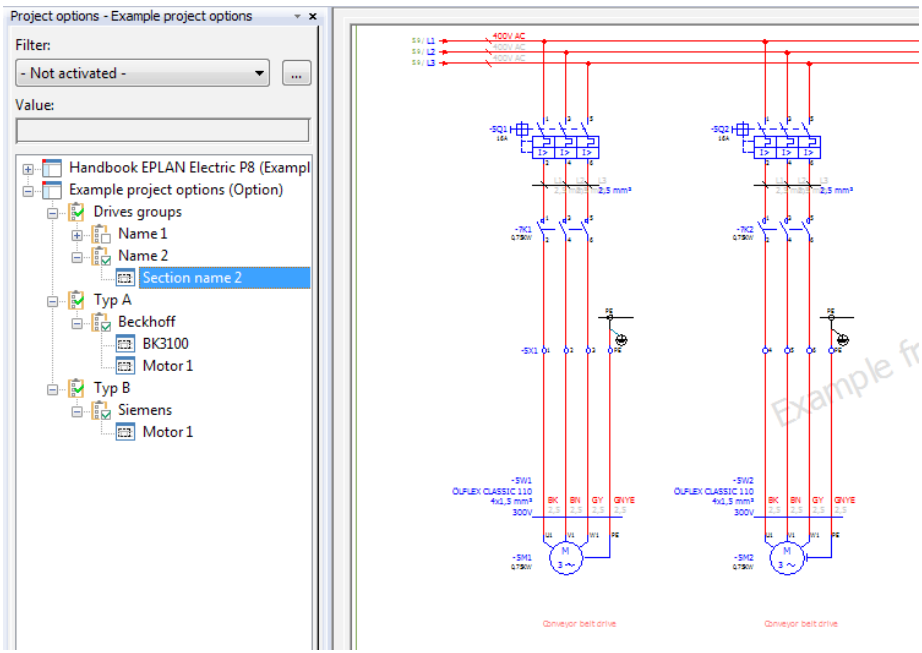


Fig. 12.38
The same page with another section

12.2.4 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. 12.39 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.

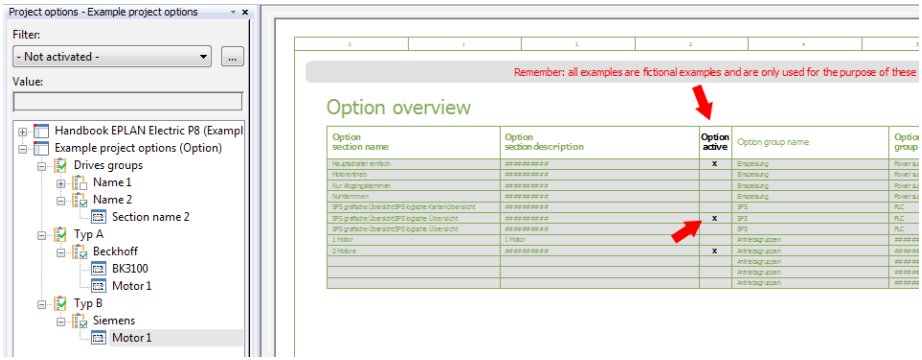


Fig. 12.39 Automatically generated options overview

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”.

I would also like to point out the EPLAN Solution Center. You will find a series of FAQs there, as well as have the chance to ask your own questions that will then be answered directly by EPLAN experts (the EPLAN Solution Center requires registration).

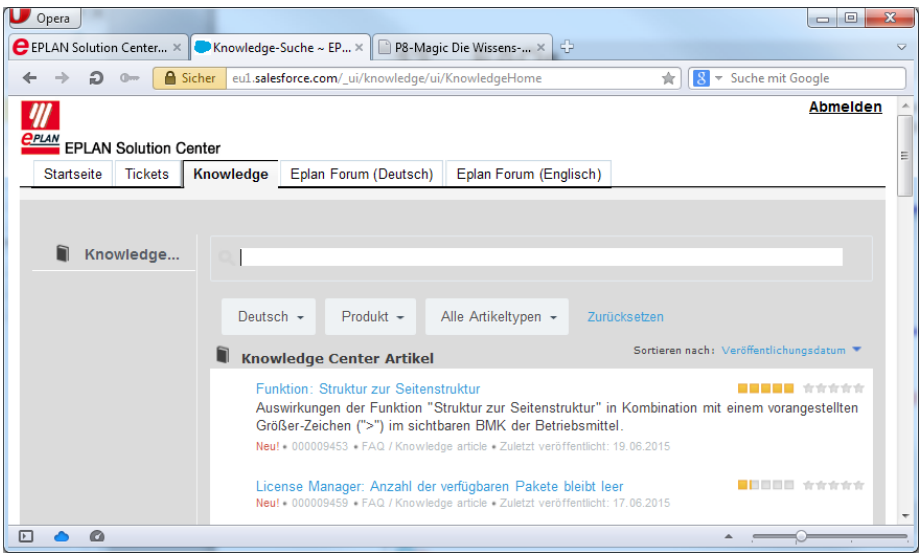


Fig. 13.1
EPLAN Solution Center on the Internet

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**. EPLAN then rotates the element by the entered angle.

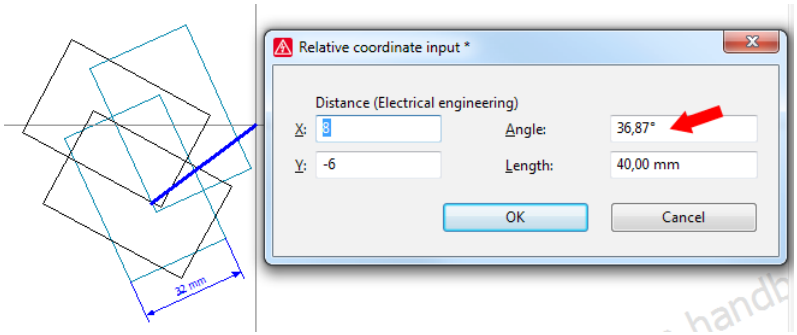


Fig. 13.2 Setting the angle in the dialog

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. Negative values are not permitted.

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 an object (such as a contactor) and its functions (here: contacts) that are distributed throughout the schematic. For example, you could change the DT throughout the schematic for all functions in one go.

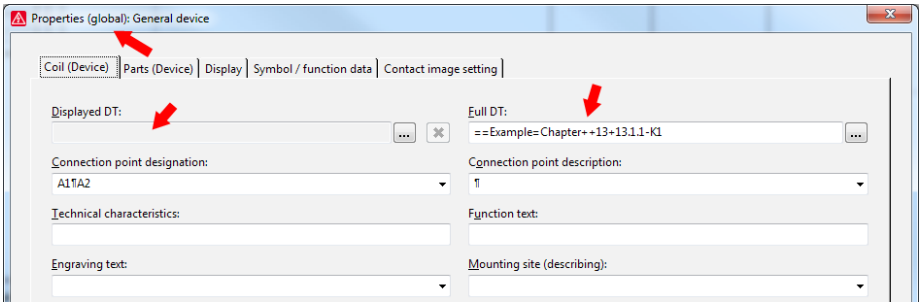


Fig. 13.3 Properties (global) option

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.

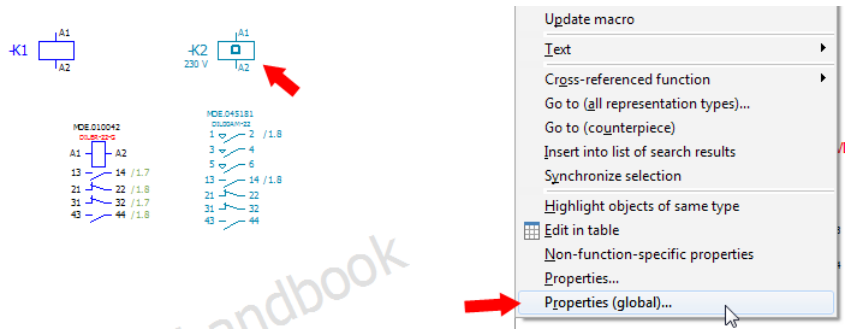


Fig. 13.4 Popup menu

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 any other function of the coil.

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.

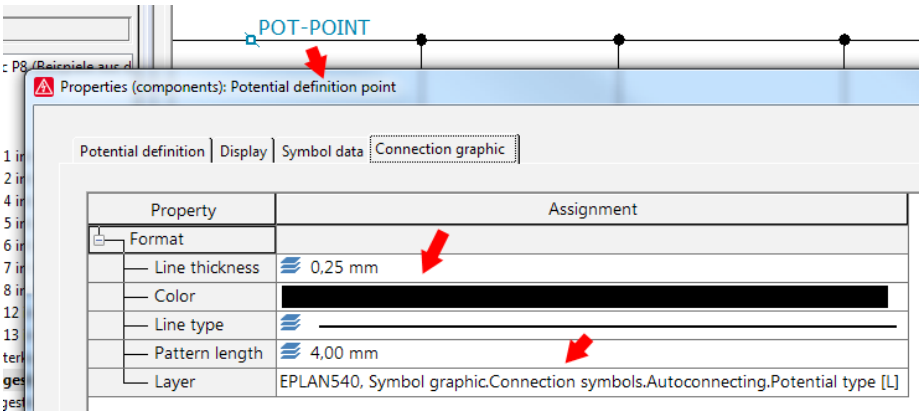


Fig. 13.5 Changing the formatting of autoconnecting lines globally

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.

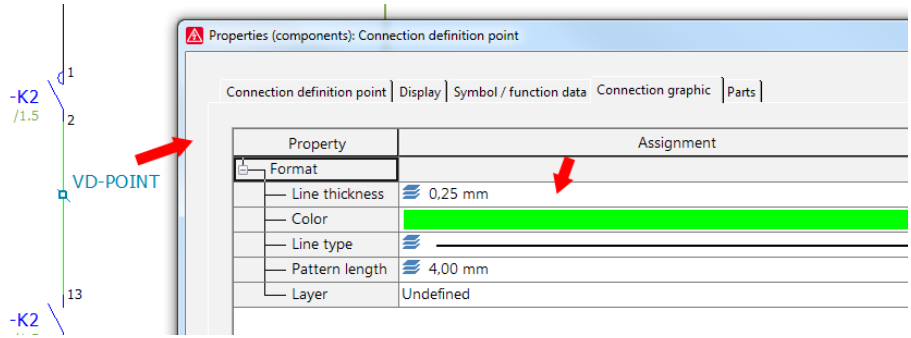


Fig. 13.6 Modifying the property of a partial connection

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 **ProjectOptions** 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.

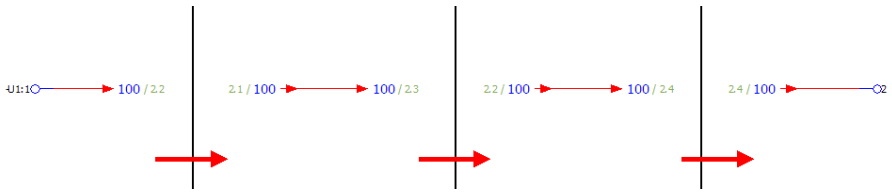


Fig. 13.7 Default sorting of interruption points

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.

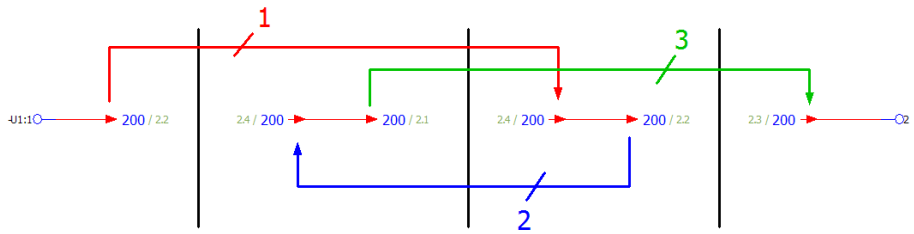


Fig. 13.8
Sorting interruption points against default

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.

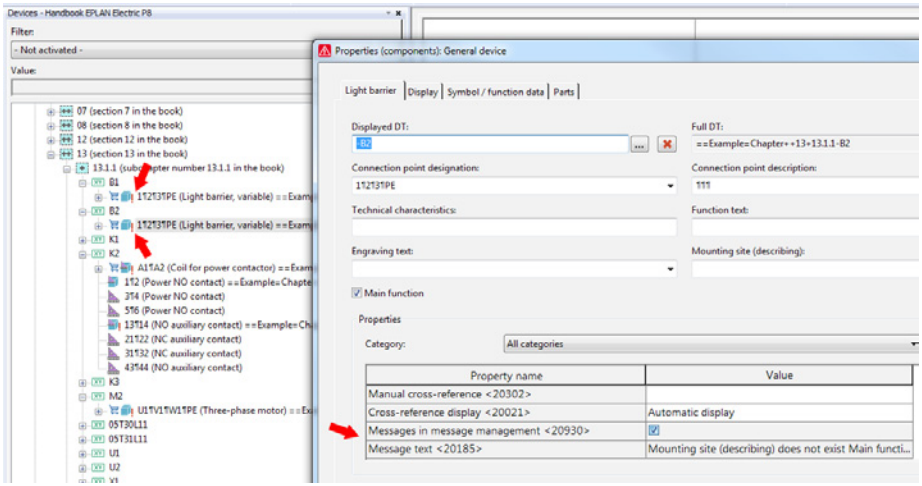


Fig. 13.9
Symbol properties

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.

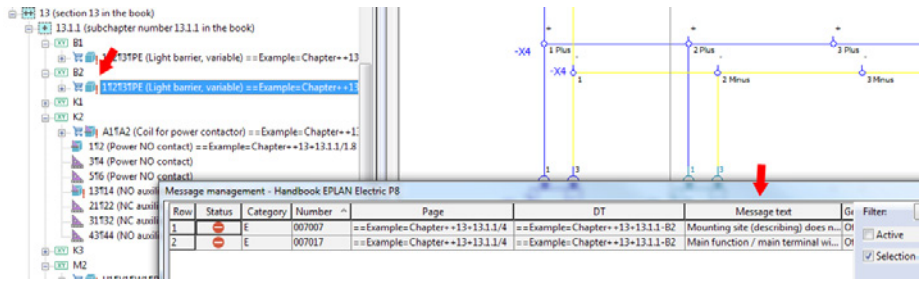


Fig. 13.10
Message in message navigator

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.

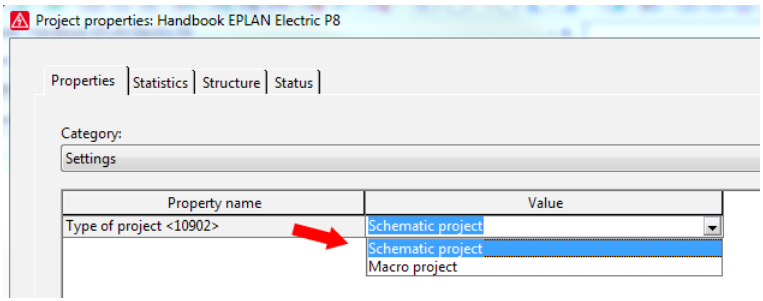


Fig. 13.11
Setting the project type

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.

TIP: Generally, you can cut out all pages in the project in question (or generate a page macro for all pages). Then, as there are no more pages and thus no page structure in the project, you can modify the page structure in the project properties. Afterwards, you can insert the pages back into the project (or import the page macro that you just created).

NOTE: Any subsequent change to the page structure must be considered carefully, because this may also change fundamental settings for objects. This can produce unwanted results – up to the point that the project becomes useless. You should therefore always test things out on a copy 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** does not 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. This should be disabled. The copy and paste functions will then again work as usual.

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.

NOTE: The *Delete* function is also available in various navigators. If you select functions (unplaced and also placed objects) in the navigator and then delete them, they will be deleted definitively – including the placed ones. You should therefore be very careful when deleting things in navigators.

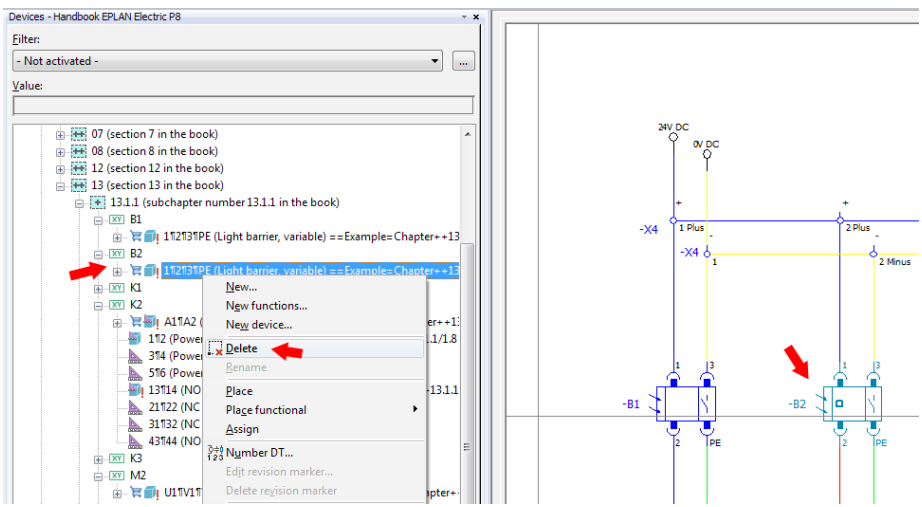


Fig. 13.12 Object selected to be deleted

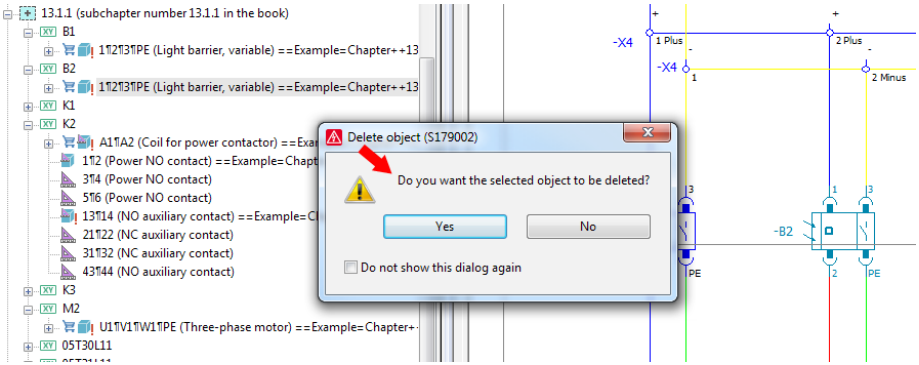


Fig. 13.13 Prompt before deleting the object

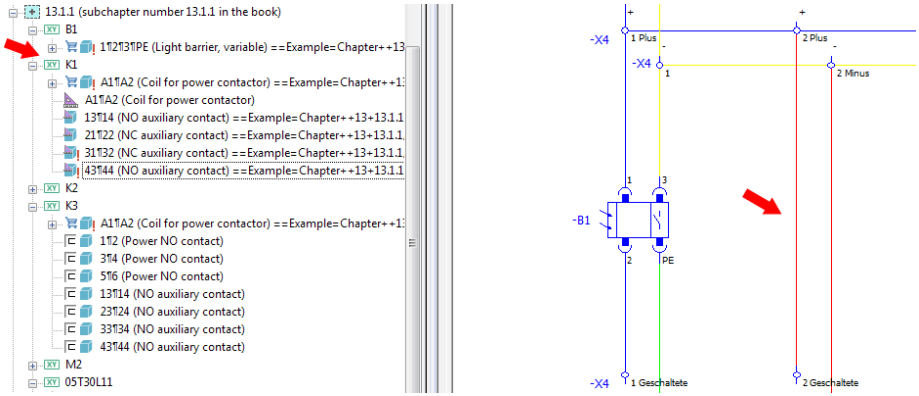


Fig. 13.14 The object has been deleted.

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 other EPLAN program packages, such as Fluid power or Preplanning, 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.

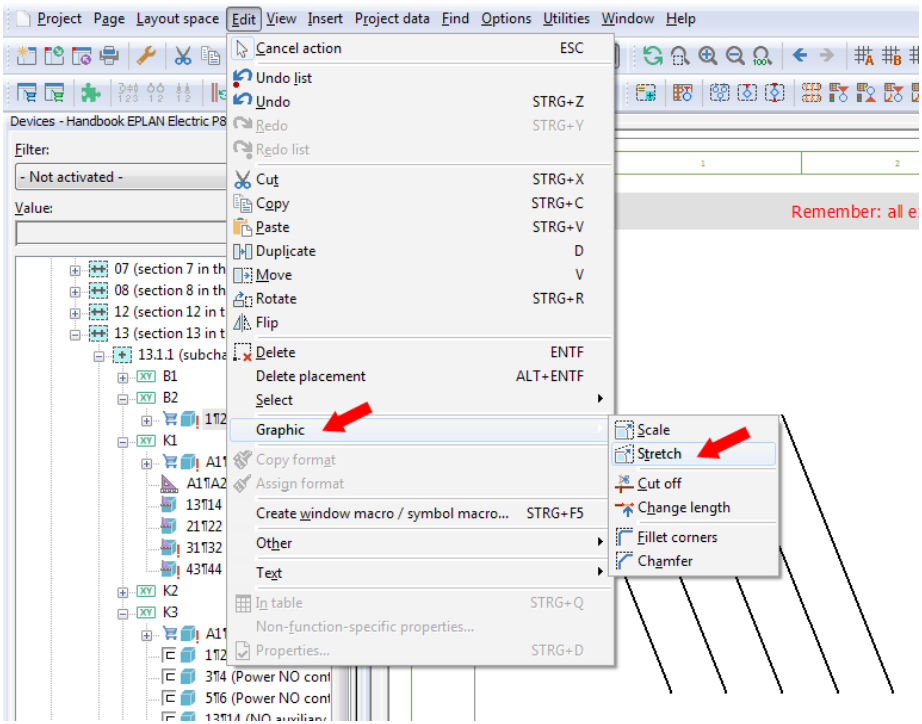


Fig. 13.15
Stretch function

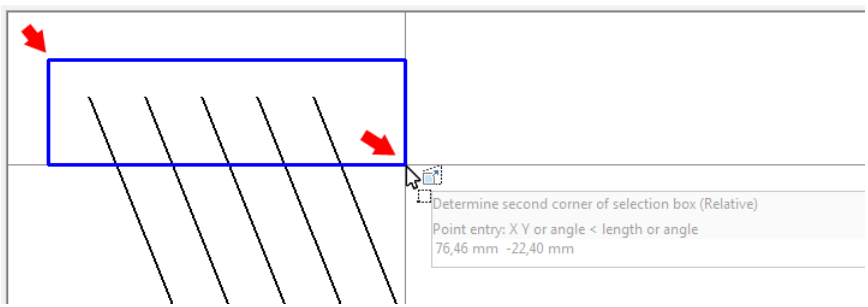


Fig. 13.16
Pulling open a window

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.

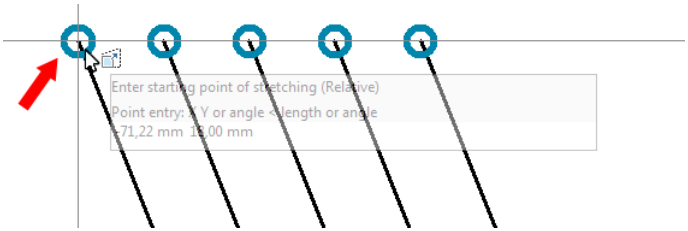


Fig. 13.17 Defining the starting point of the stretch

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).

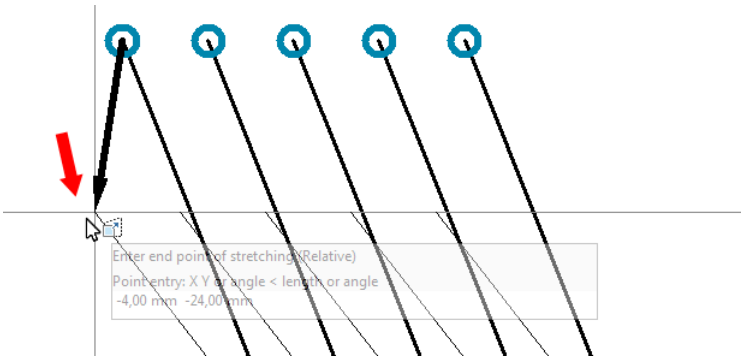


Fig. 13.18 Defining the end point of the stretch

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.

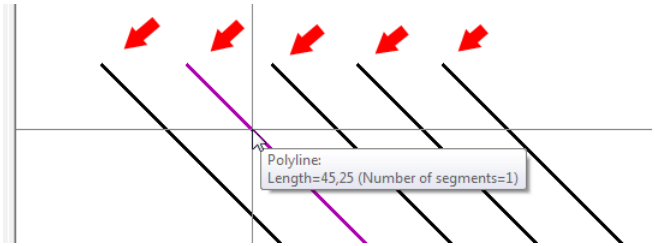


Fig. 13.19 All line data has been changed.

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.

NOTE: Without optional project management, the customer data must be entered in the project properties manually, unfortunately.

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.

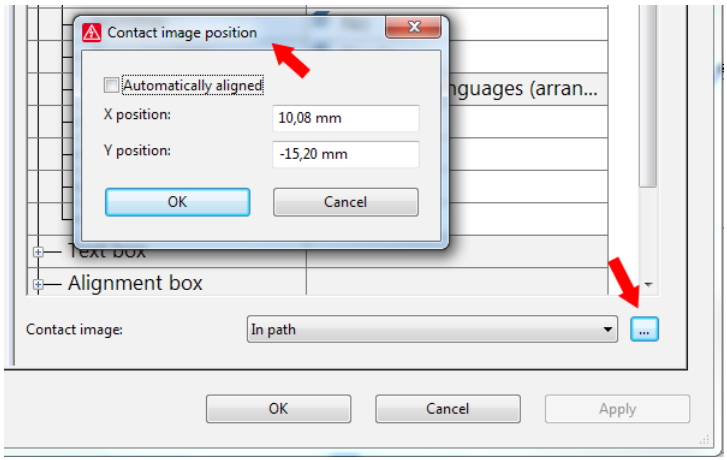


Fig. 13.20 Manually changing the position of the contact image

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.

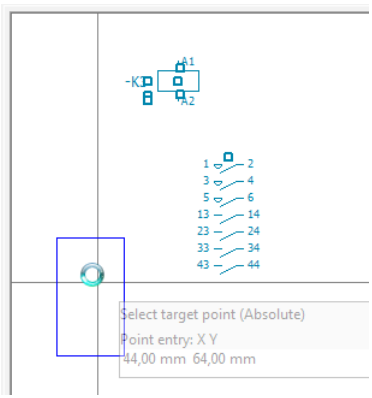


Fig. 13.21 Moving

Question: How do I find the previous "Superior" option in the project structure in case of different devices?

Answer: If the *Superior* option is no longer part of the project properties (*Structure* tab), you can reactivate it in the project's settings. You find the setting in the node **PROJECTS/[PROJECT NAME]/MANAGEMENT/COMPATIBILITY**.

But you should do without it, for reasons concerning norm-compliant representation. It is automatically activated in projects that have used this setting so far. But in new projects it is deactivated by default.

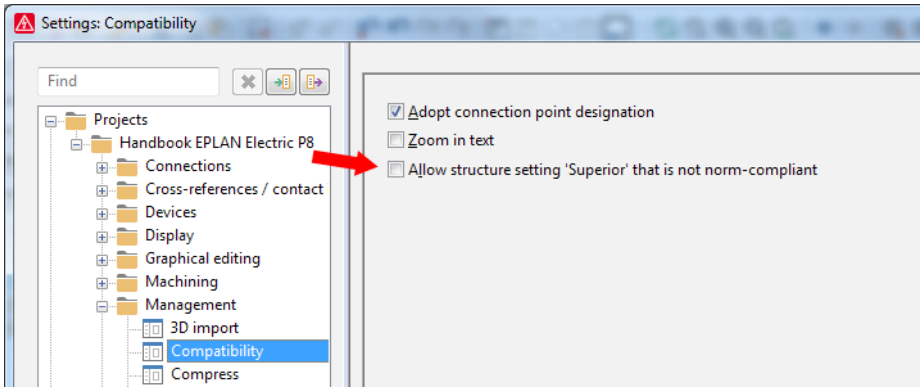


Fig. 13.22 Activating "Superior…" project setting

The setting does the following: 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.

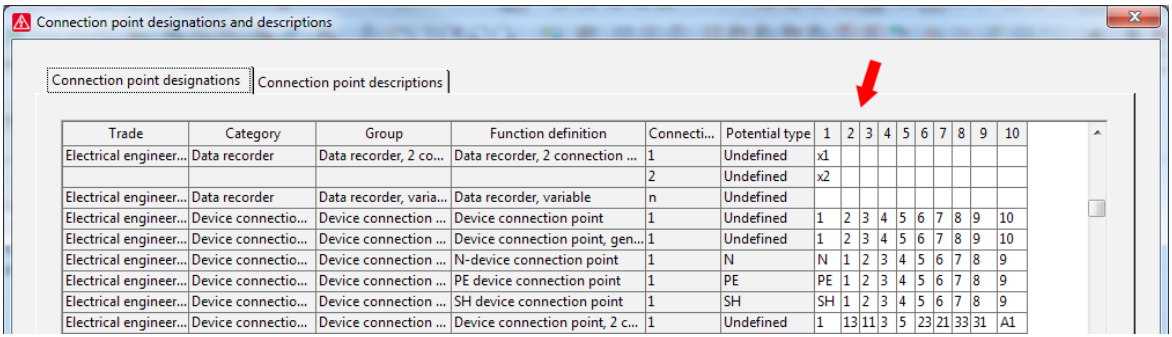
1. Launch **PROJECT DATA/MESSAGES/MANAGEMENT** message management with the proper check run.
2. In the message management window, activate the check box next to **SELECTION**.
3. Start the device navigator.
4. Click the desired (defective) item in the device navigator.
5. Now you will see only messages 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:

NOTE: Since this involves original EPLAN data records, these data records are reset to the EPLAN default in case of any version change. Keep this in mind when making changes.

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.



Trade	Category	Group	Function definition	Connecti...	Potential type	1	2	3	4	5	6	7	8	9	10
Electrical engineer...	Data recorder	Data recorder, 2 co...	Data recorder, 2 connection ...	1	Undefined	x1									
				2	Undefined	x2									
Electrical engineer...	Data recorder	Data recorder, varia...	Data recorder, variable	n	Undefined										
Electrical engineer...	Device connectio...	Device connection ...	Device connection point	1	Undefined	1	2	3	4	5	6	7	8	9	10
Electrical engineer...	Device connectio...	Device connection ...	Device connection point, gen...	1	Undefined	1	2	3	4	5	6	7	8	9	10
Electrical engineer...	Device connectio...	Device connection ...	N-device connection point	1	N	N	1	2	3	4	5	6	7	8	9
Electrical engineer...	Device connectio...	Device connection ...	PE device connection point	1	PE	PE	1	2	3	4	5	6	7	8	9
Electrical engineer...	Device connectio...	Device connection ...	SH device connection point	1	SH	SH	1	2	3	4	5	6	7	8	9
Electrical engineer...	Device connectio...	Device connection ...	Device connection point, 2 c...	1	Undefined	1	13	11	3	5	23	21	33	31	A1

Fig. 13.23 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

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.

project, but was not found, or not in full. 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.

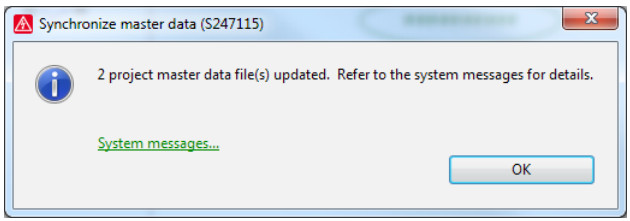


Fig. 13.24
Message regarding synchronization of master data

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: Why are blue boxes instead of graphical symbols shown in the reports?

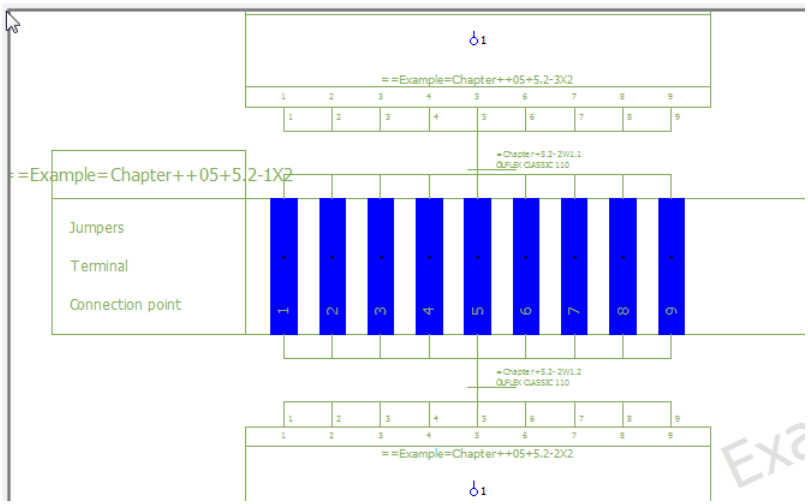


Fig. 13.25
Blue boxes

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.

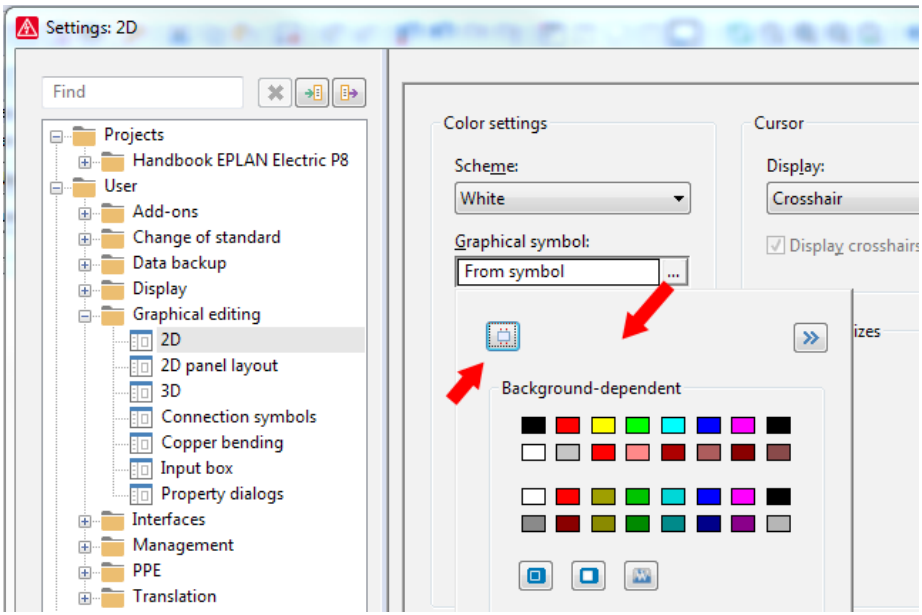


Fig. 13.26
Setting the color from the symbol

Now the graphical symbols will be displayed completely even in the reports.

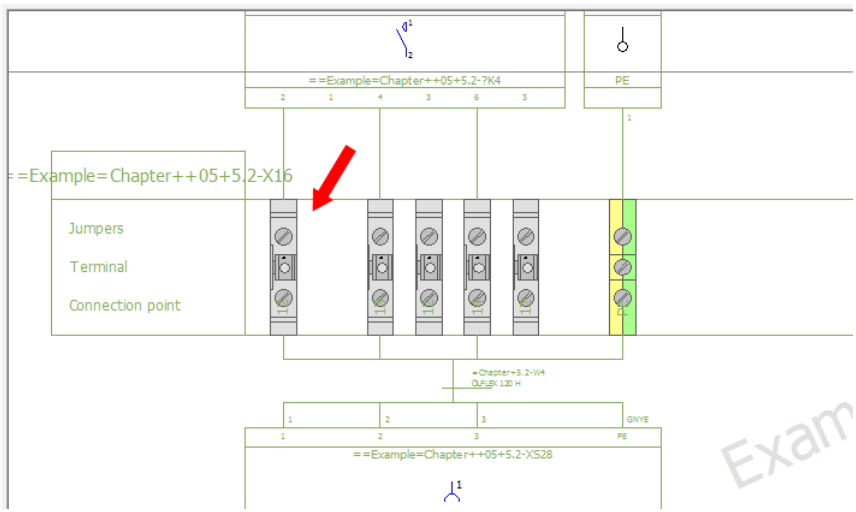


Fig. 13.27
Symbol color settings

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: 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.

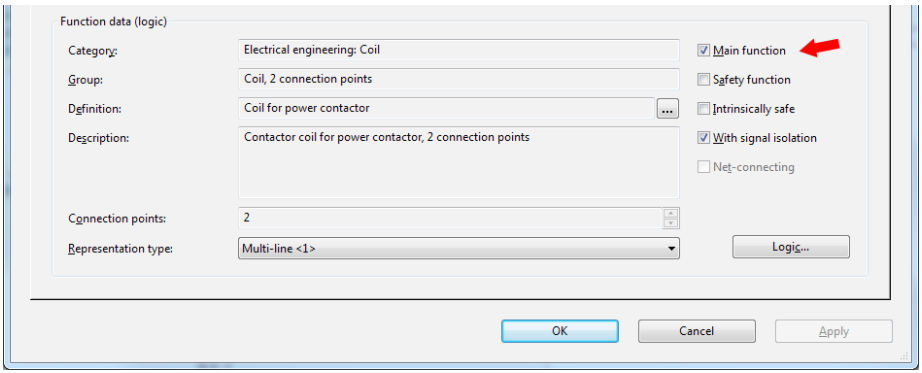


Fig. 13.28 Coil main function

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.

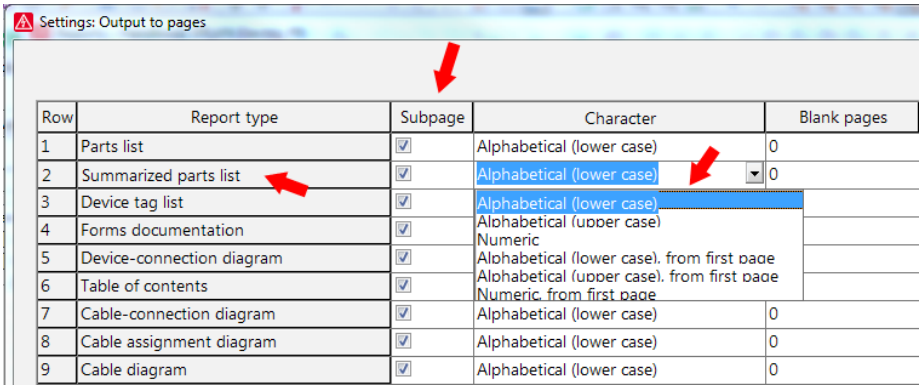


Fig. 13.29 Settings: Output to pages dialog

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.

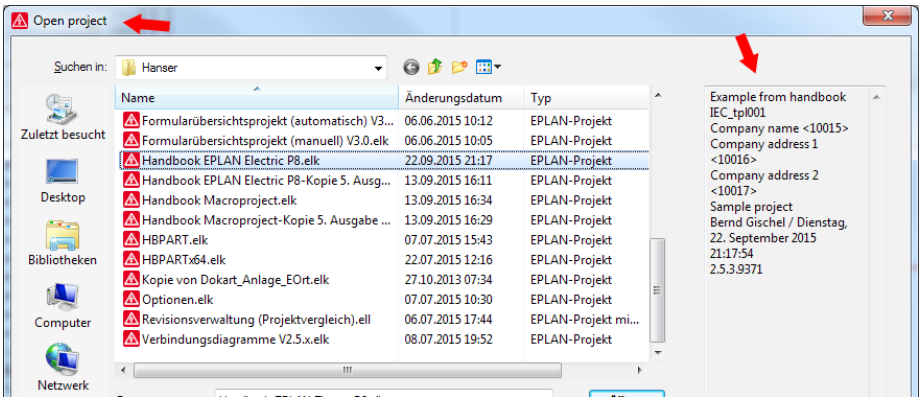


Fig. 13.30 Information in the Open project dialog

The following project properties are displayed (if they have been completed in the project properties):

Project description <10011>;
Project number <10013>;
Commission <10014>;
Company name <10015>;
Company address 1 <10016>;
Company address 2 <10017>;
Supplementary field [1] <10901 1> to Supplementary field [7] <10901 7>;

Last edited by: Logon name <10022>;
Modification date <10023>;
Last EPLAN version used <10043>;
Last EPLAN build number used <10044>

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\2.5\BIN\Eplan.exe" /setup*, where 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.

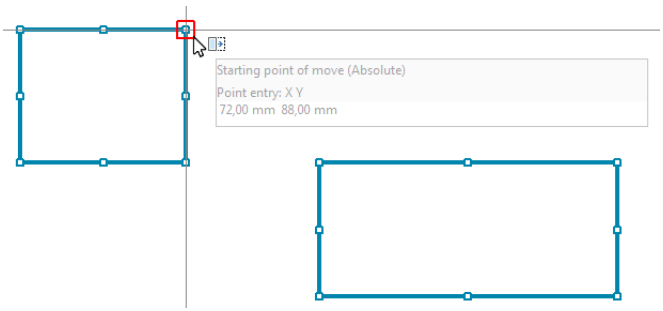


Fig. 13.31
Object snap on

Once object snap is activated, you will be able to capture objects (points) again.

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, you can change the sequence of the various iden-

tifiers used in the project (and also those not currently in use). The identifier is selected from the identifier list in the left area of the **Structure identifier management** dialog.

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.

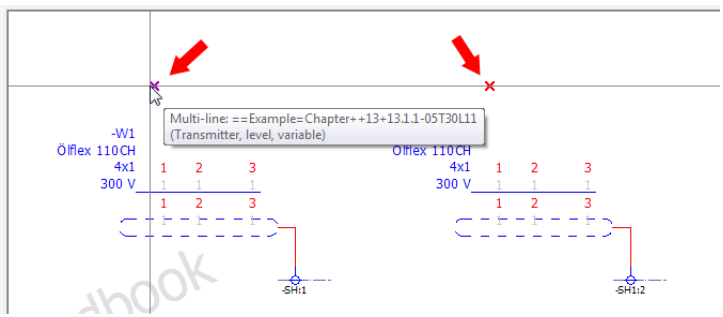


Fig. 13.32
Missing symbols

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).

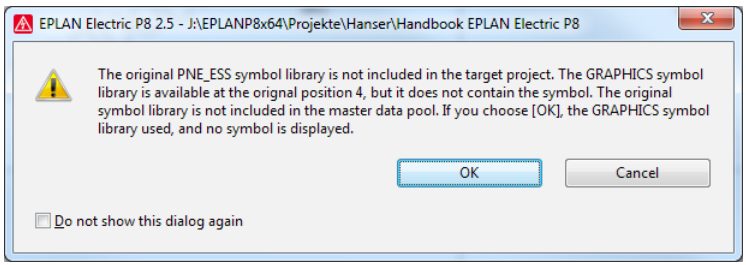


Fig. 13.33 EPLAN warning message when symbols are missing

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 EPLAN Electric 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 company 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: 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.

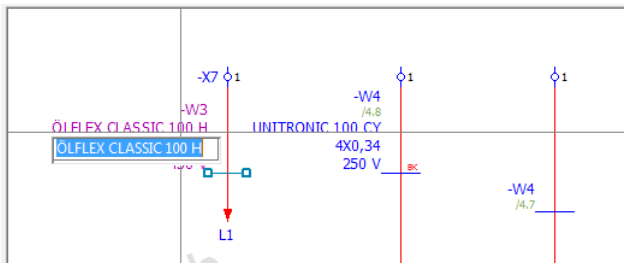


Fig. 13.34 Direct editing of properties

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, (mostly) filled 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/2D**, 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.

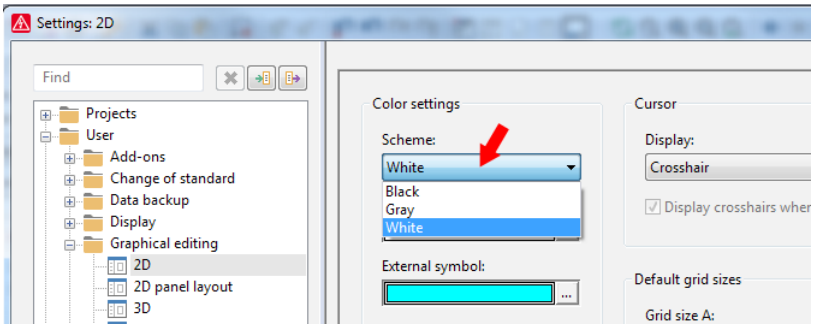


Fig. 13.35 Setting the background color of the graphical editor

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. The sequence of projects (position in the navigator) cannot be changed manually.

Question: Can I hide the description of structure identifiers in the page navigator?

Answer: Yes. This and other structural descriptions can be displayed and/or hidden via the popup menu using the *Configure representation* function.

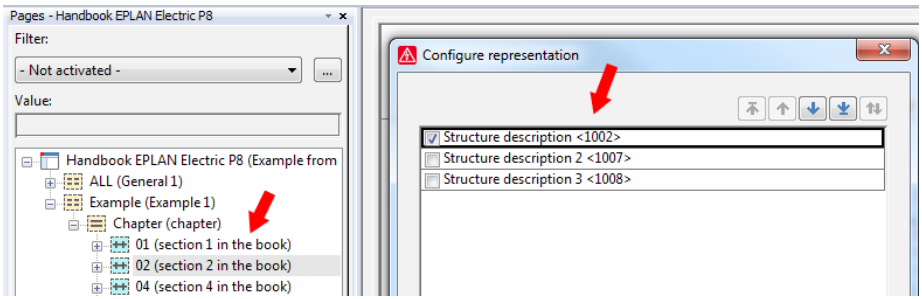


Fig. 13.36 Displaying and/or hiding structural descriptions in the page navigator

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.

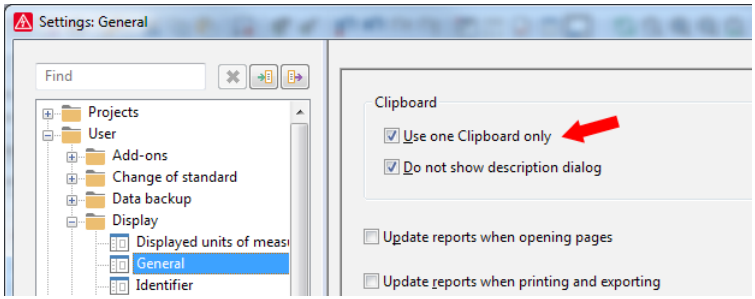


Fig. 13.37 Clipboard settings

NOTE: If the maximum clipboard limit has been reached, EPLAN will display a warning.

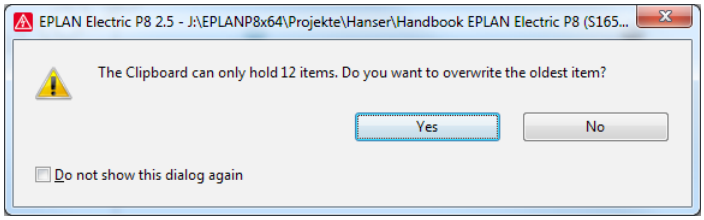


Fig. 13.38 Warning: Maximum clipboard limit exceeded.

Question: Can the *Update connections* function be assigned to a shortcut key?

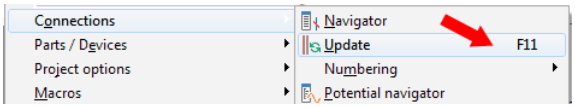


Fig. 13.39 User's own shortcut key F11

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.

You can create your own shortcut keys using the **OPTIONS/SETTINGS/USER/MANAGEMENT/SHORTCUT KEYS** menu. Then, you select the respective command and assign to it a suitable shortcut key.

Question: How can I suppress the check message "Coil without contacts"?

Answer: If the project uses contactors/relays, etc. as spares (without using contacts or parts), 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.

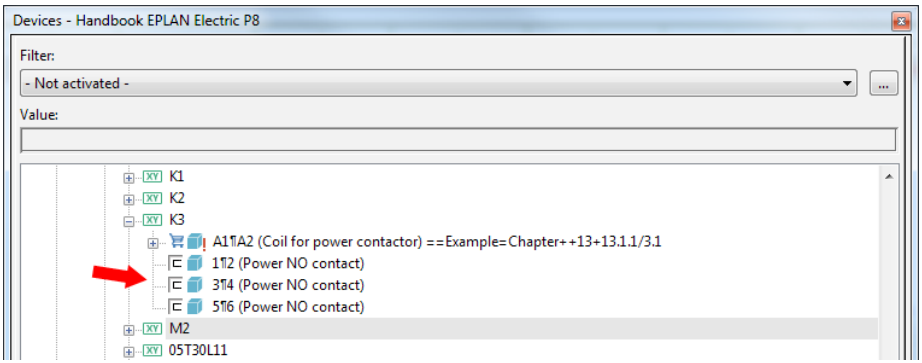


Fig. 13.40 Unplaced functions

Solution 2: If parts cannot be assigned, the missing functions, for example, can be generated automatically in the device navigator via the popup menu and the *New function* function.

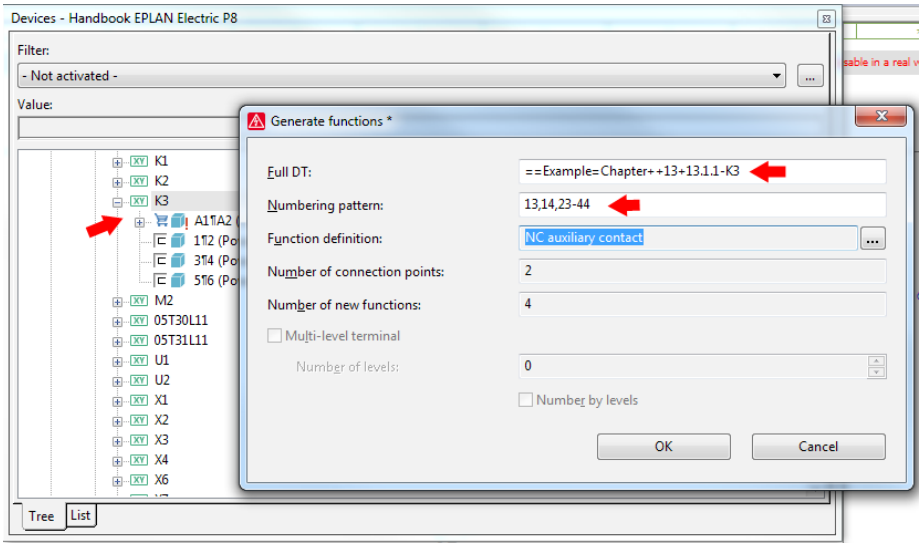


Fig. 13.41 Generating new functions with numbering pattern (here: 4 close contacts).

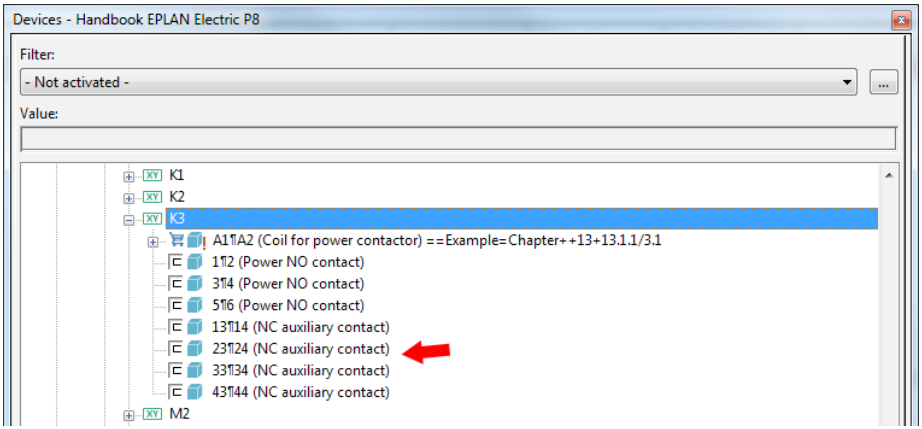


Fig. 13.42
Newly inserted functions

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. This opens the **Detailed selection**. dialog. In this dialog, a prefilter is created, which is then used to filter the project's structure identifiers on the basis of a set filter scheme.

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.

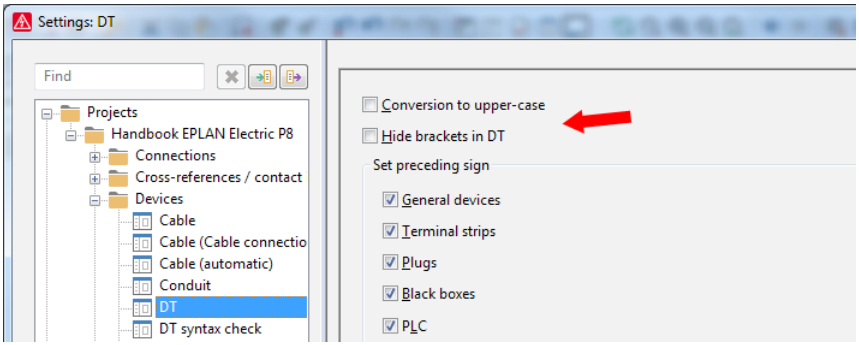


Fig. 13.43
Uppercase letters or lowercase letters

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?

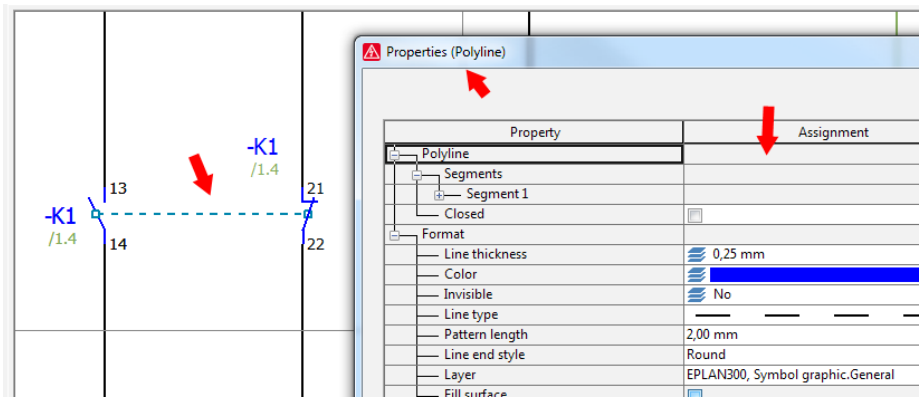


Fig. 13.44
Mechanical action line

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.

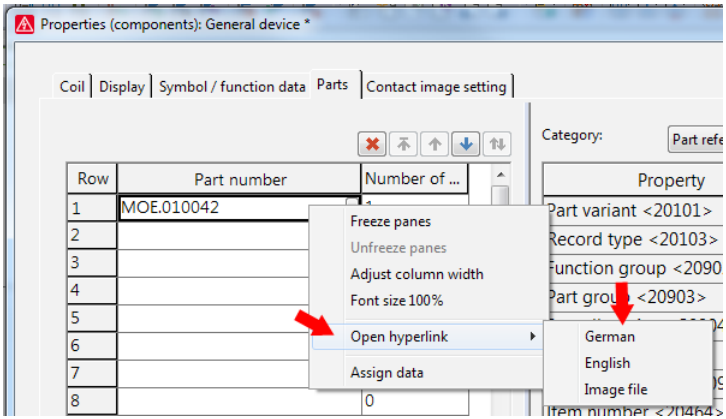


Fig. 13.45
Opening hyperlinks

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 on the *Documents* tab, 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?

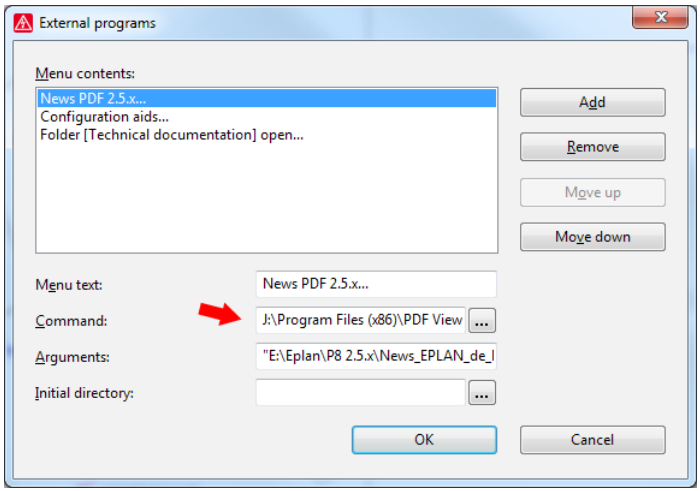


Fig. 13.46 Using external programs

Answer:

1. Open the external programs via the **OPTIONS/EXTERNAL PROGRAMS** menu.
2. Use the **ADD** button to generate a new menu item.
3. Enter a suitable (subsequently accessible via the **OPTIONS** menu) *menu text*, for example "Design aid…".
4. In the *Command* field, enter the path to the *Excel.exe* (for example: *J:\Program Files\Microsoft Office\Office14\EXCEL.EXE*).
5. In the *Arguments* field, enter the path to the Excel table: If there are blank spaces in the path name, then set it in quotes (for example: "F:\Excel\Color 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.

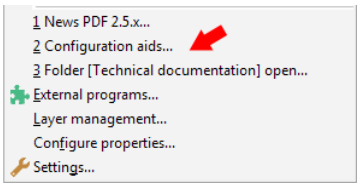


Fig. 13.47 New menu item in the Options menu

TIP: This menu item can also be assigned its own shortcut key.

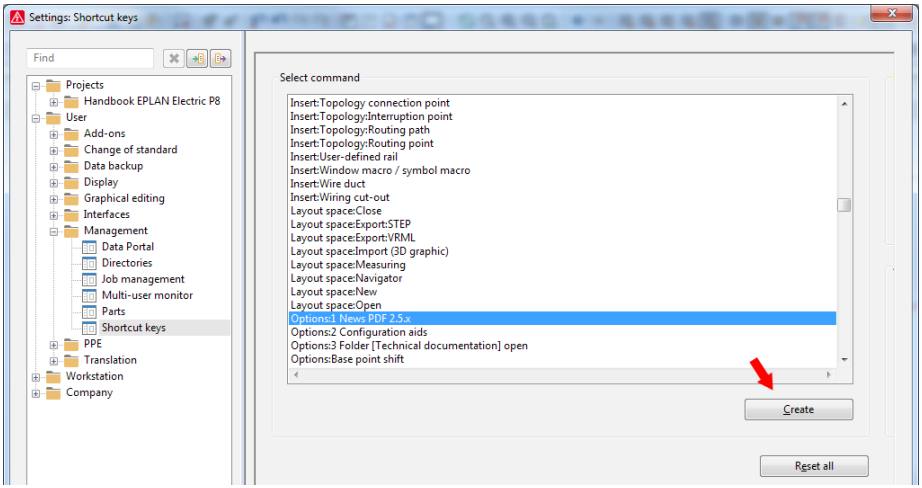


Fig. 13.48 Calling menu item with user-defined shortcut key

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?

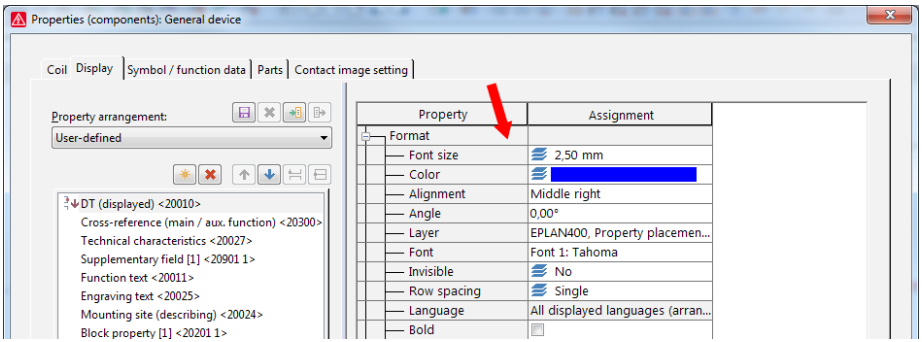


Fig. 13.49 Changing the row height – before

Answer: Yes, this is possible. 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.

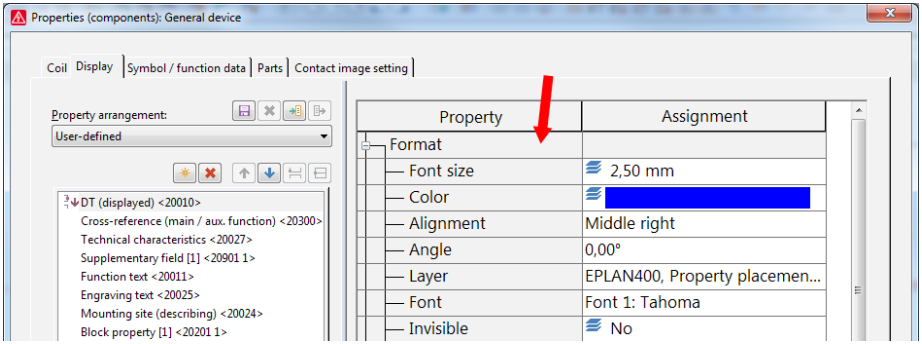


Fig. 13.50 Changing the row height – after

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?

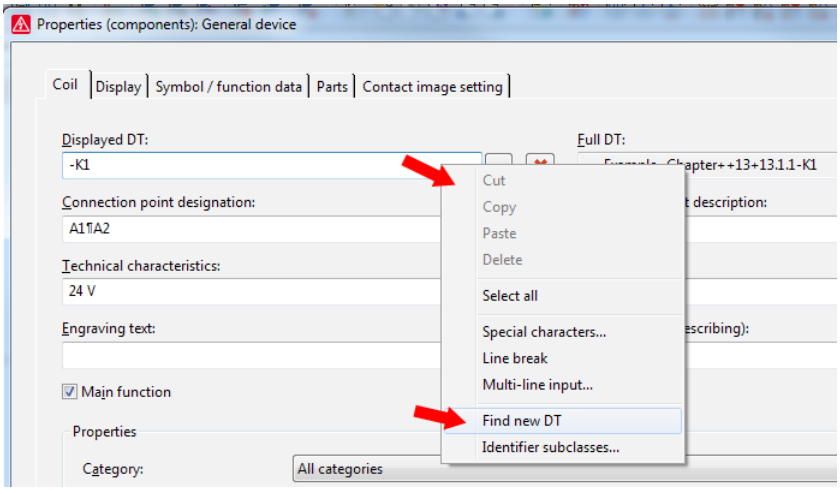


Fig. 13.51 Find new DT

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**.

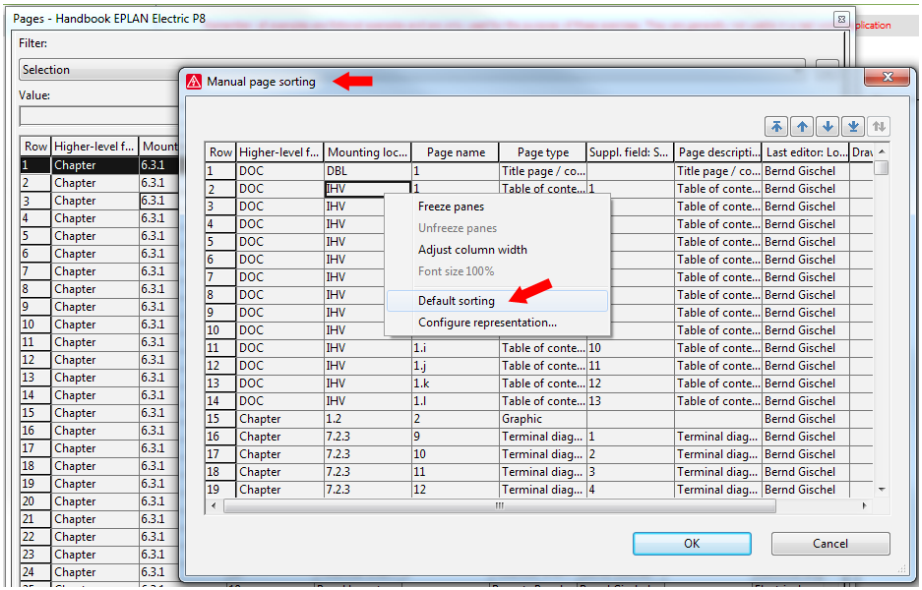


Fig. 13.52 Resetting manual page sorting

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. Then, you can call the part directly on the symbol and modify it.

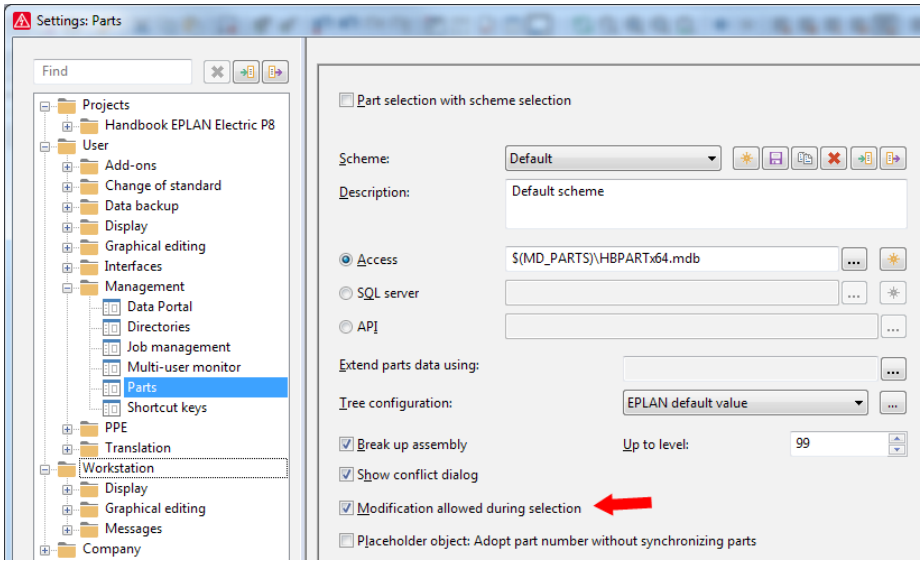


Fig. 13.53 Changing parts data setting during selection

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.

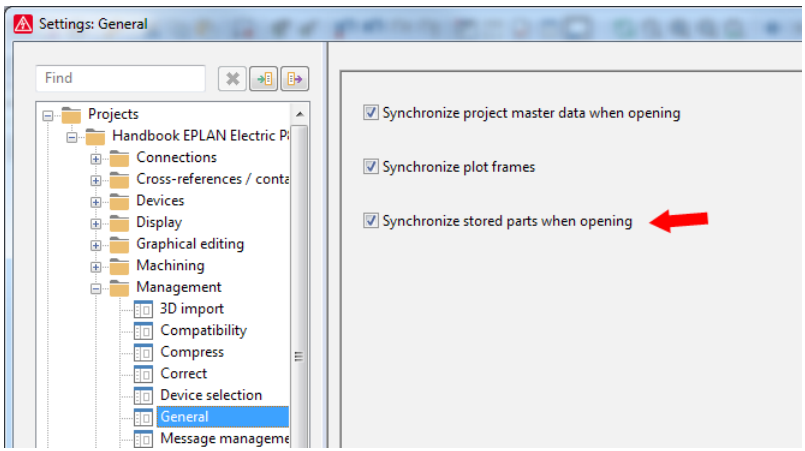


Fig. 13.54 Updating stored parts when opening the project

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.

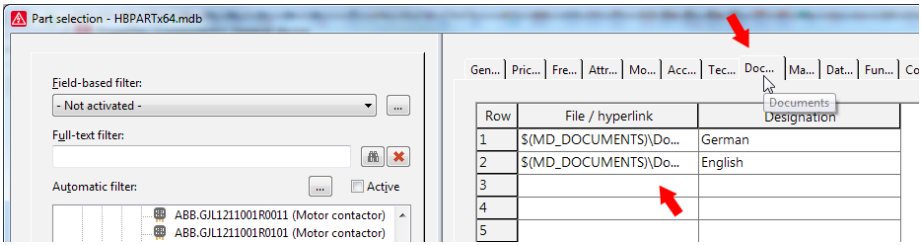


Fig. 13.55 Documents tab

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.

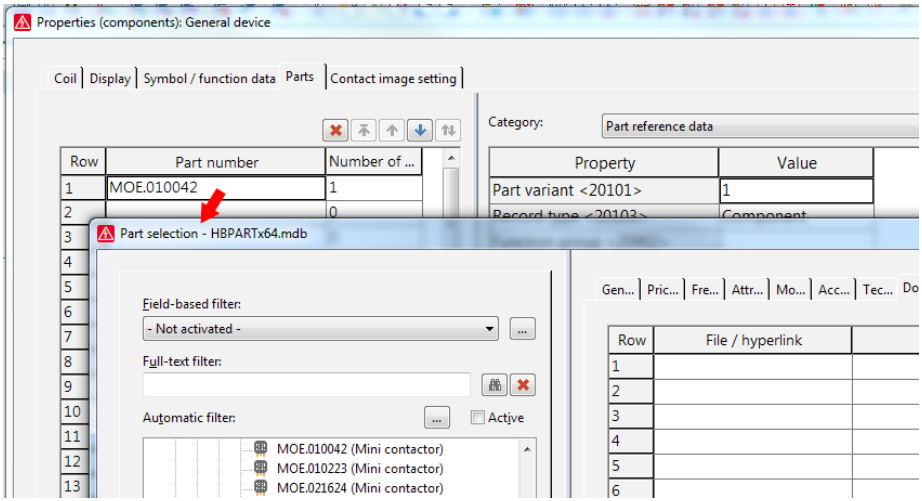


Fig. 13.56 Simple part selection

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.

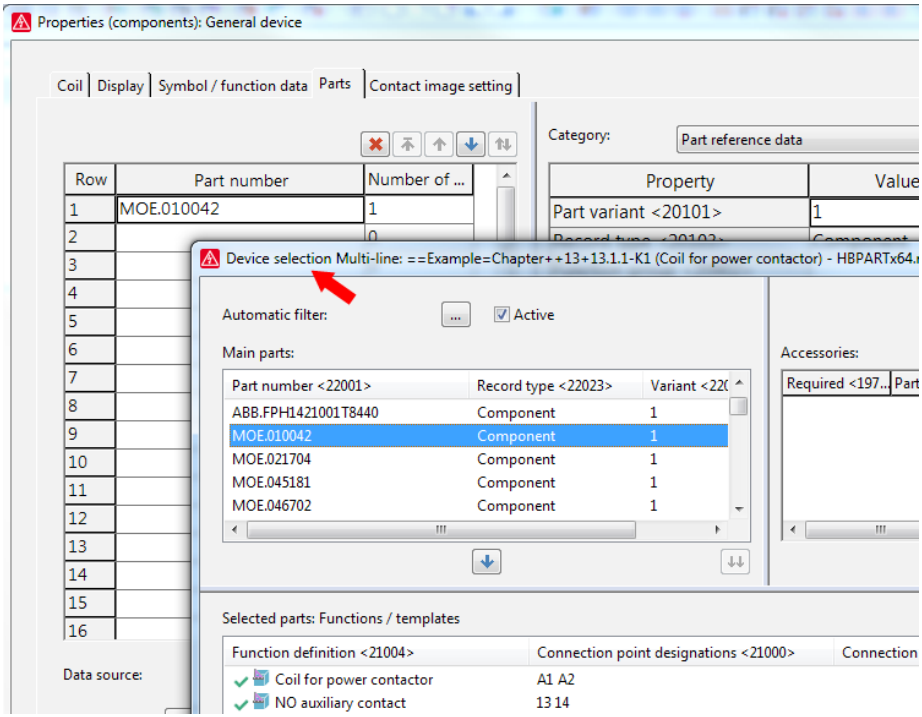


Fig. 13.57
Device selection

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**. Here, you select the **SETTINGS** function. EPLAN opens the **Settings: Part (User)** dialog. A new database is generated here via the *New* button.

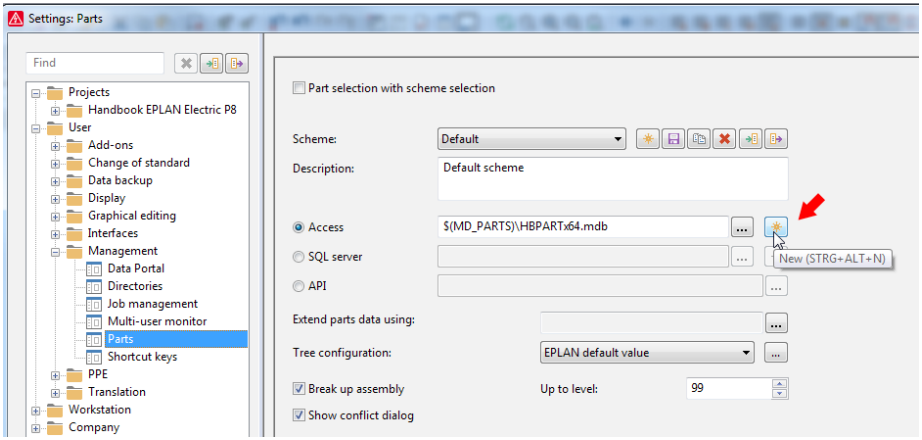


Fig. 13.58
Creating a new database

Click the *Open* button in the **New database** dialog, and EPLAN will apply the new database and enter it in the settings as the current database. Now leave the **Settings** dialog and parts management (by clicking **OK** and/or clicking the *Close* button in parts management).

In the **UTILITIES/PARTS** menu, select the **SYNCHRONIZE CURRENT PROJECT** function. EPLAN opens the **Synchronization of parts** dialog. The filter in the dialog is set to - *Not activated* -. EPLAN now lists all project parts. You can select them all, and use the button to move them to the new database on the right.

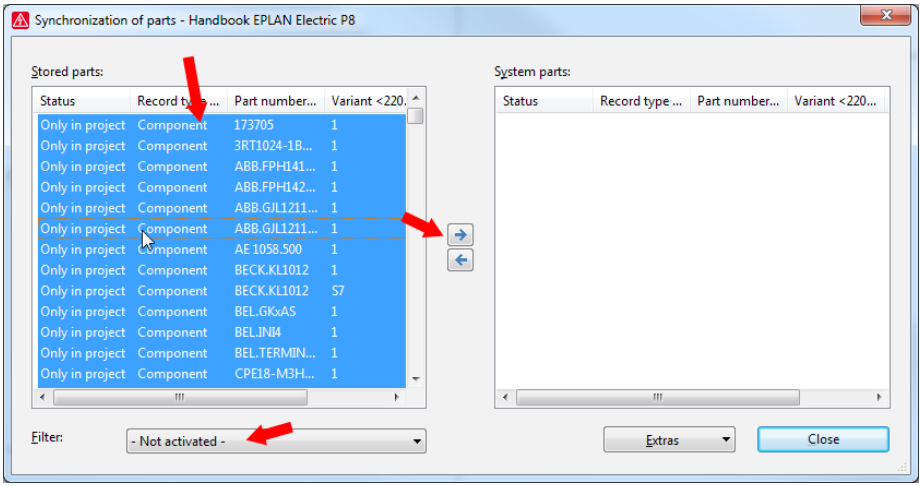


Fig. 13.59 Storing project parts in the new database

EPLAN starts synchronization without further prompts. 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: 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*.

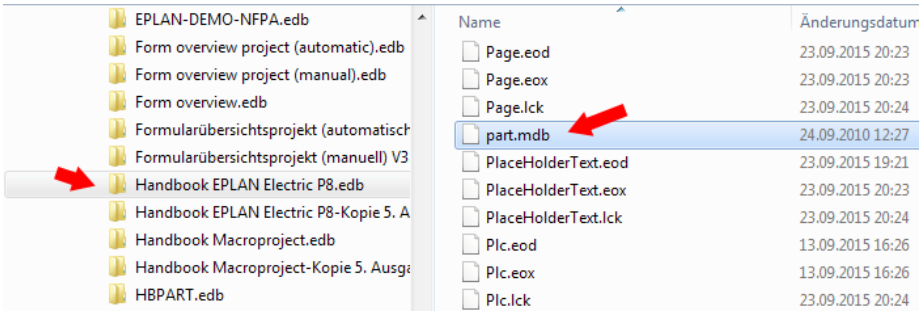


Fig. 13.60 Project-specific parts database

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: 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?

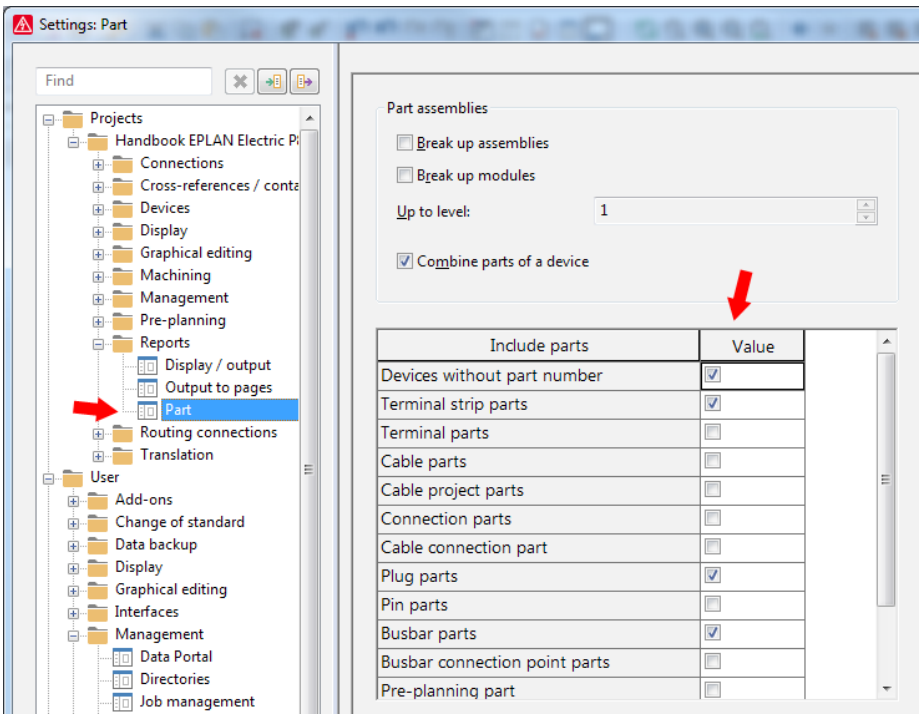


Fig. 13.61 Include parts

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.

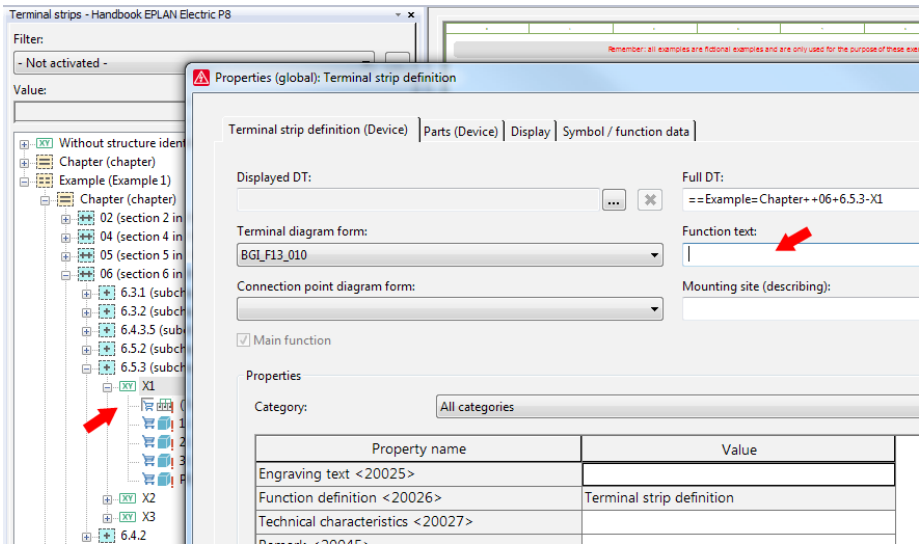


Fig. 13.62 Function text on terminal strip definition

If the terminal strip definitions do not exist, under **PROJECT/ORGANIZE/CORRECT** you can use an appropriate schema to have the terminal strip definitions be generated automatically by EPLAN.

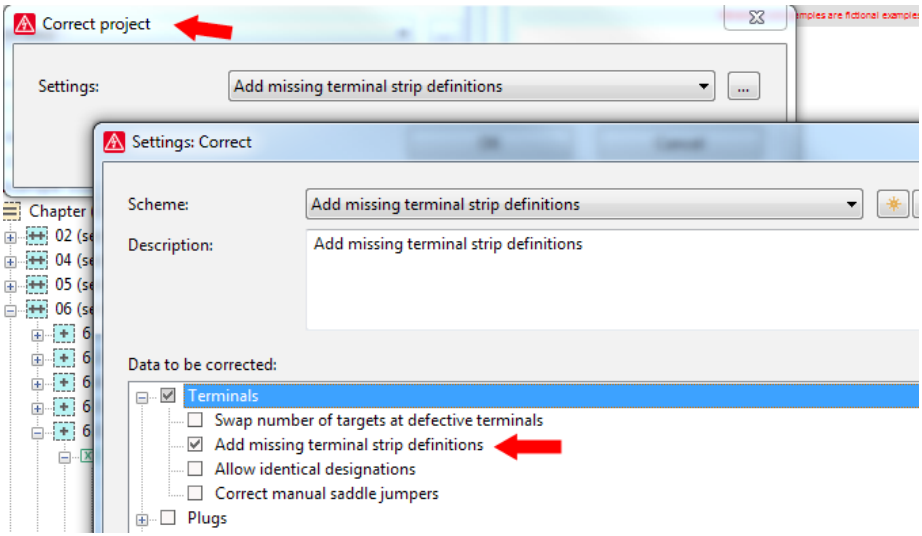


Fig. 13.63 Correcting terminal strip

NOTE: For the correction function to work, you do not have to select any terminal in the project. EPLAN automatically looks for the correct and missing objects, and then adds them all to the project. An individual selection is not possible.

Question: How can I generate plug definitions automatically?

Answer: To generate plug definitions automatically, go to the **PROJECT/ORGANIZE/CORRECT** menu to generate an appropriate scheme using the activated *Add missing plug definition* option.

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.

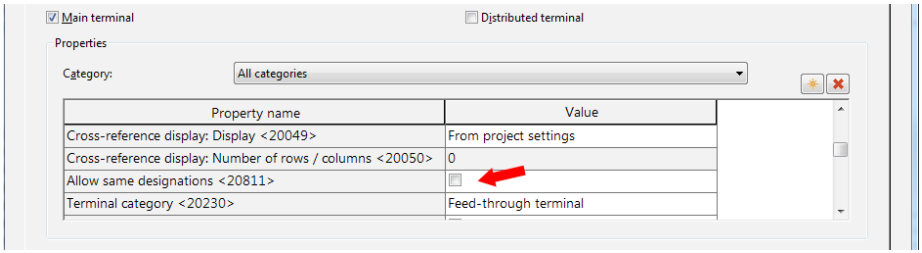


Fig. 13.64
Allow same terminal designations property

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.

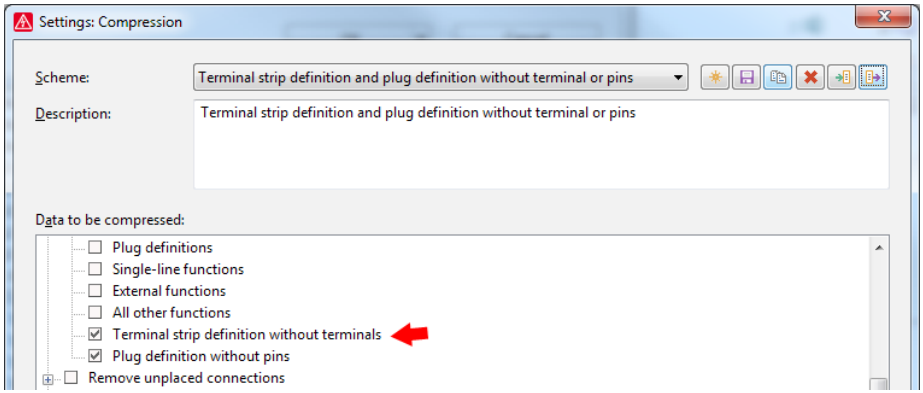


Fig. 13.65
Compression scheme

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.

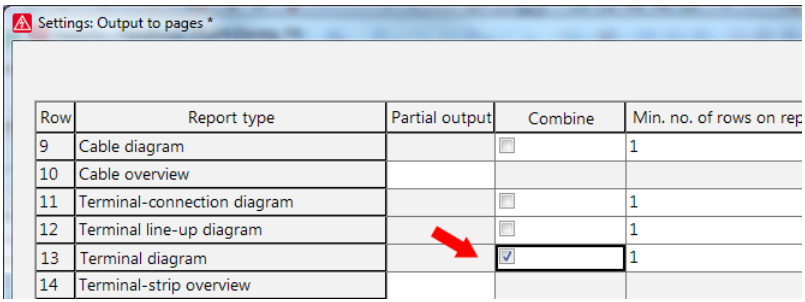


Fig. 13.66 Activate combination for a report

In addition to the *terminal diagram* report, the *Combine* option is also available for the following reports (sorted by the F number):

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;
Topology: Routing path diagram *.f35
Pre-planning: Structure segment plan *.f39
Pre-planning: Planning object plan *.f41
Pre-planning: Segment template plan *.f43
Assembly/Module overview *.f44
Distributed device list *.f45
Bundle/conduit plan *.f46

13.4 Cables

Question: Why can't EPLAN execute a correct device selection of cables (incl. assignment of conductor information) on open interruption points?

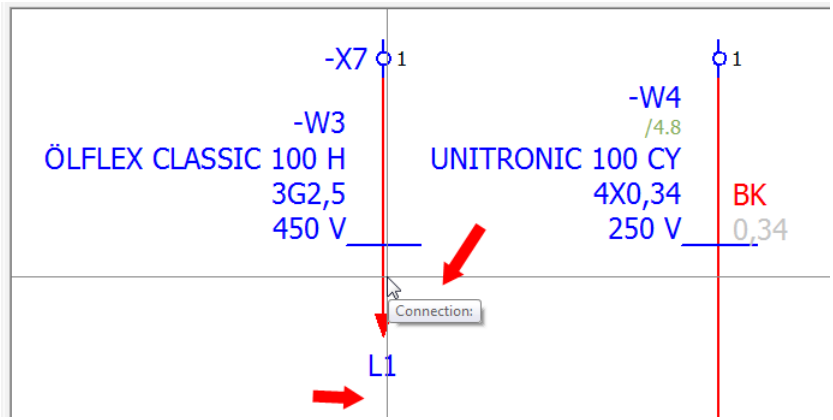


Fig. 13.67 Open connection

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.

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*.

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 most likely 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.

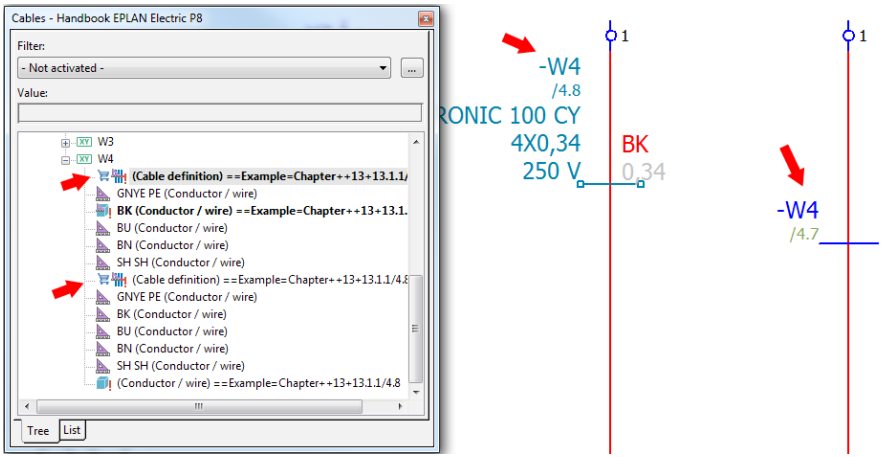


Fig. 13.68
Duplicate cable definition in the navigator

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 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 Properties, layers

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.

How is a property arrangement transferred? 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.

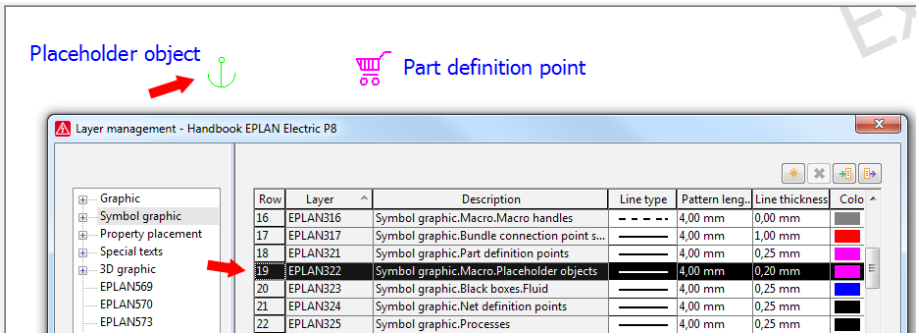


Fig. 13.69 Properties of the EPLAN322 layer

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*.

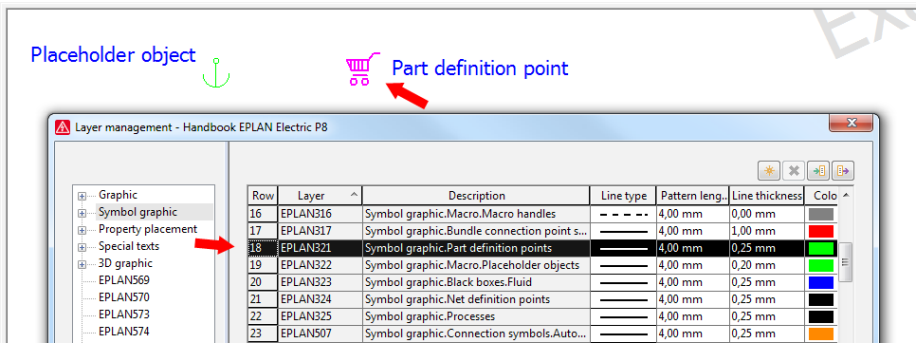


Fig. 13.70
Properties of the EPLAN321 layer

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.

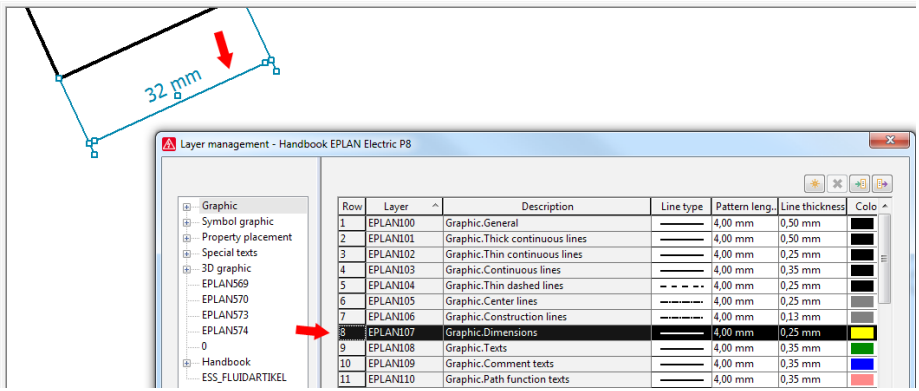


Fig. 13.71
Properties of the EPLAN107 layer

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.

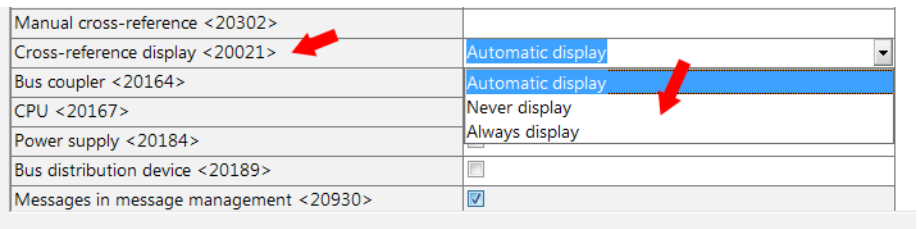


Fig. 13.72
Differences in cross-reference display <20021>

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).

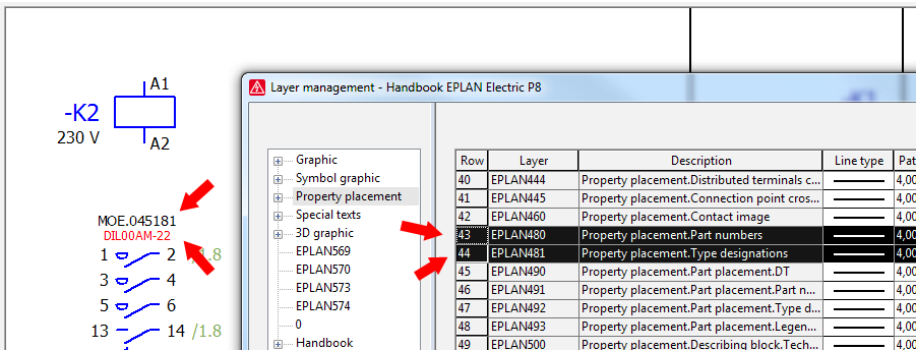


Fig. 13.73 Properties of the layers EPLAN480 and EPLAN481

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.

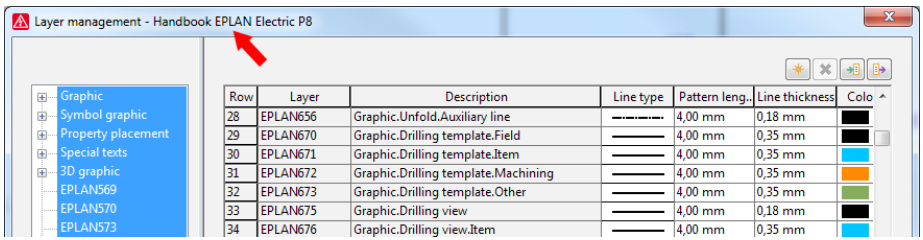


Fig. 13.74 Layer management and the project

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 *EPLAN108 Graphic.Texts* or *EPLAN110 Graphic.Path function texts*-layers.

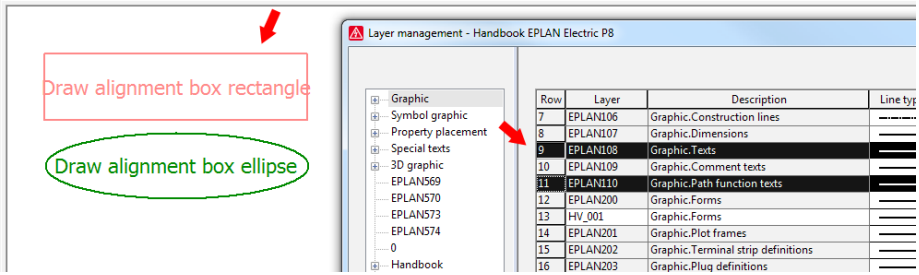


Fig. 13.75 Properties of the layers EPLAN108 and EPLAN110

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 possible, for example, 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>*

13.5.1 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).

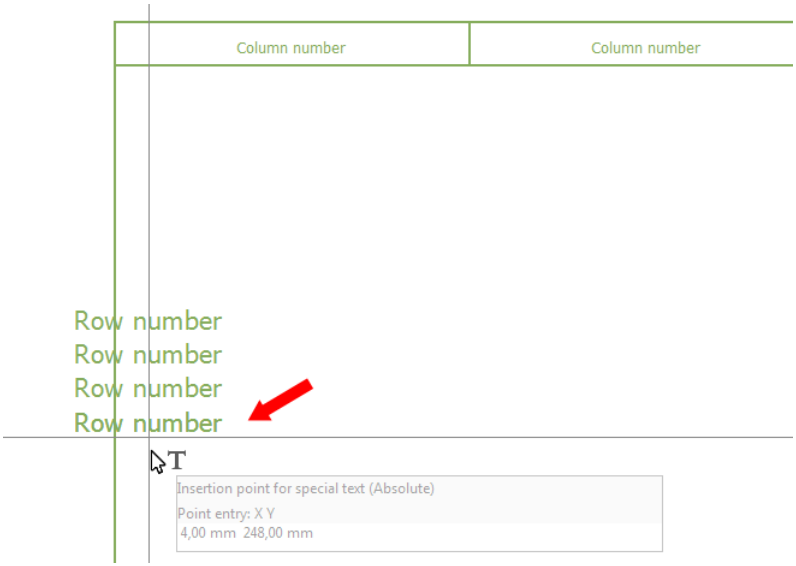


Fig. 13.76 Inserting row texts

- 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**.

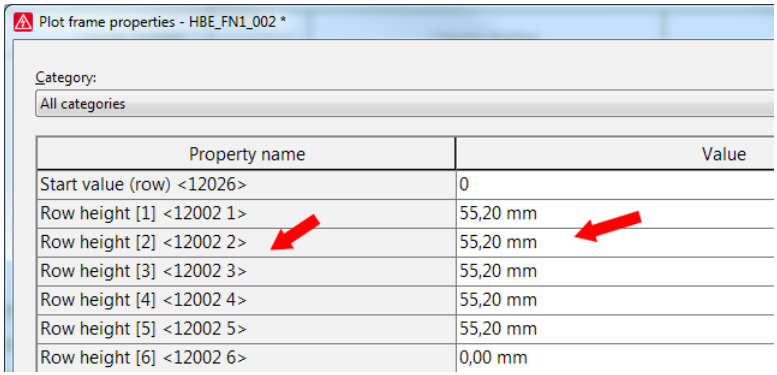


Fig. 13.77 Entering properties for the rows

- 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.

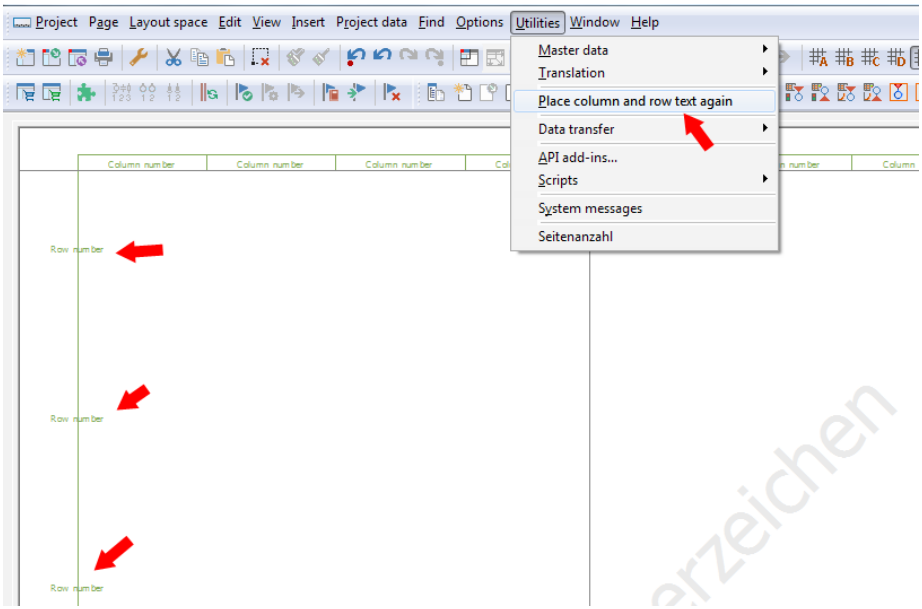


Fig. 13.78 Letting EPLAN determine the distances automatically

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.

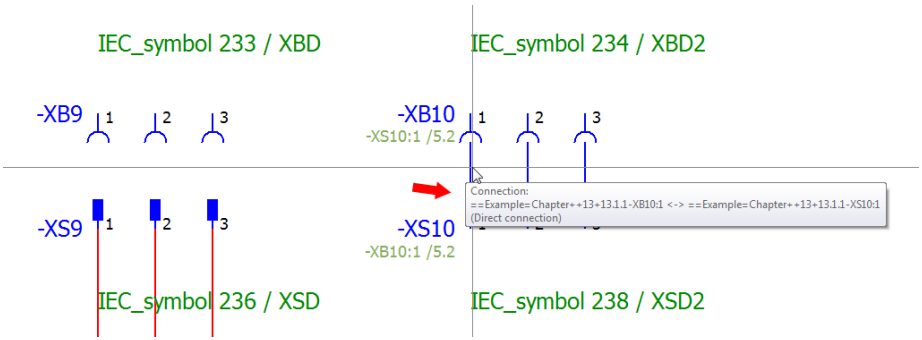


Fig. 13.79 Differences between XBD/XBD2 and XSD/XSD2 symbols

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).

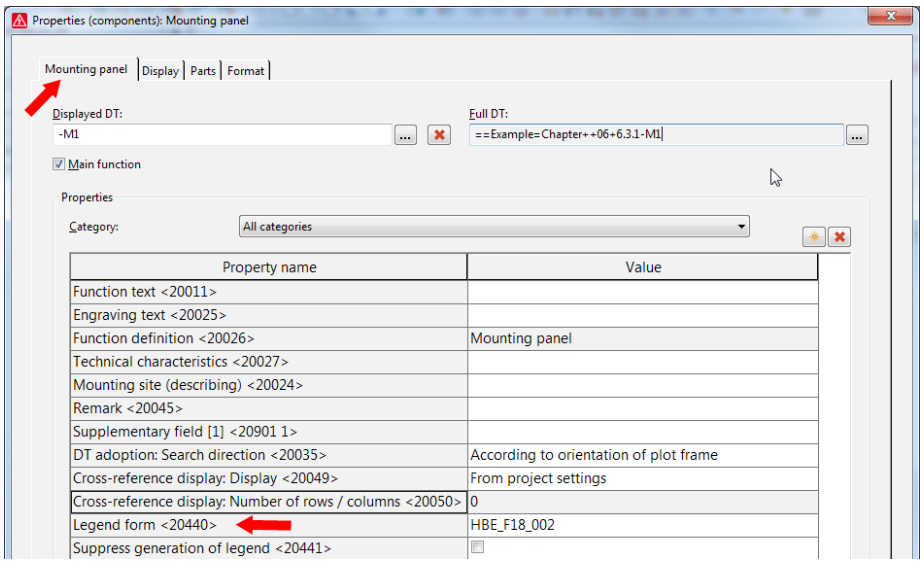


Fig. 13.80 Legend form entry

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).

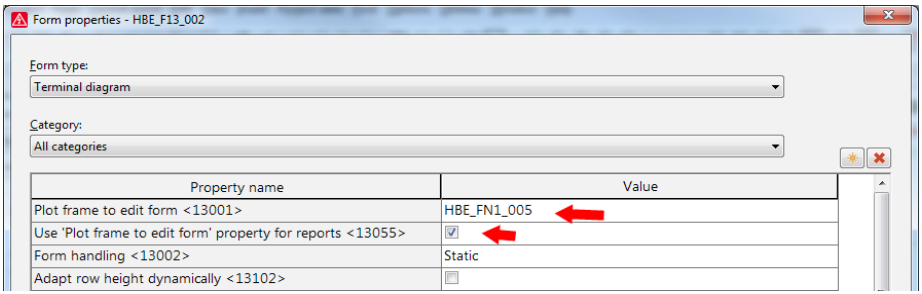


Fig. 13.81
Customizing form properties

- 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: (Format) Day.Month.Year
- Last edited by: Bernd Gischel
- Modification date: Format Day.Month.Year
- Form handling: Static form

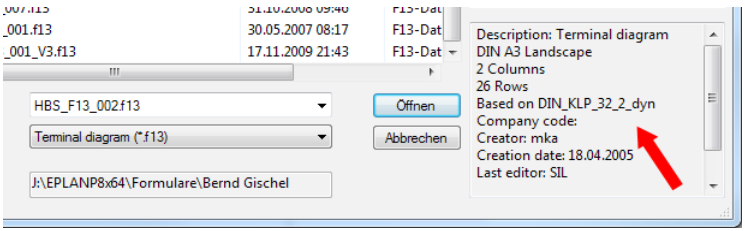


Fig. 13.82
Form properties in the Open form dialog

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: (Format) Day.Month.Year
- Last edited by: Bernd Gischel
- Modification date: (Format) Day.Month.Year

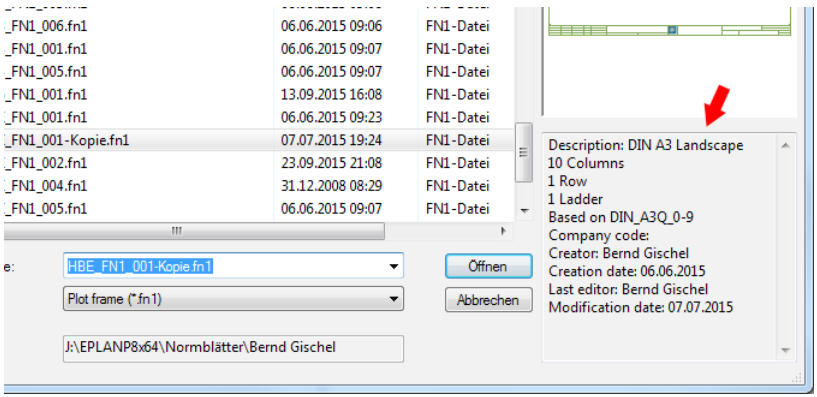


Fig. 13.83 Plot frame properties in the Open plot frame dialog

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).

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.

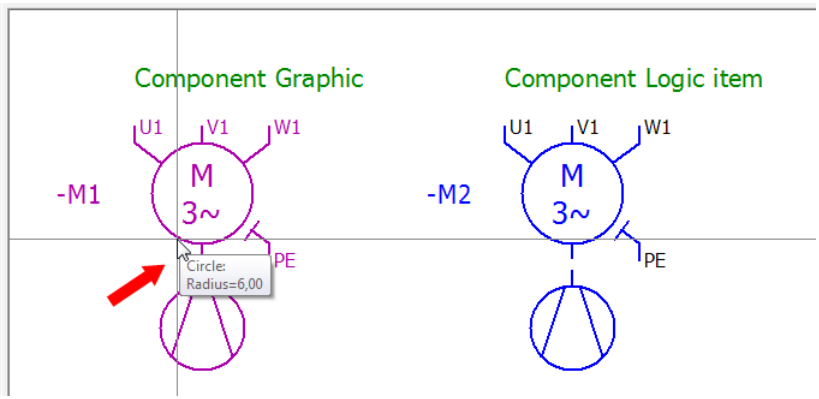


Fig. 13.84 Component converted into graphic

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.

13.6 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.

NOTE: EPLAN 21 projects can no longer be imported directly in EPLAN Electric P8 (version 2.4 and higher). Therefore, an older version (prior to version 2.4) must be used.

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.

13.7 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.

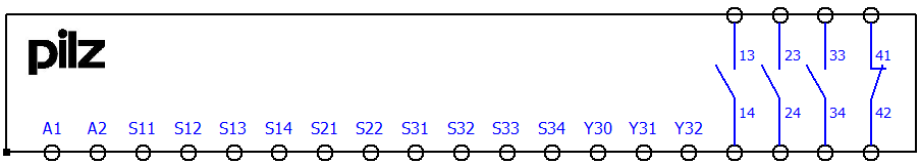


Fig. 13.85 Graphical overview

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.

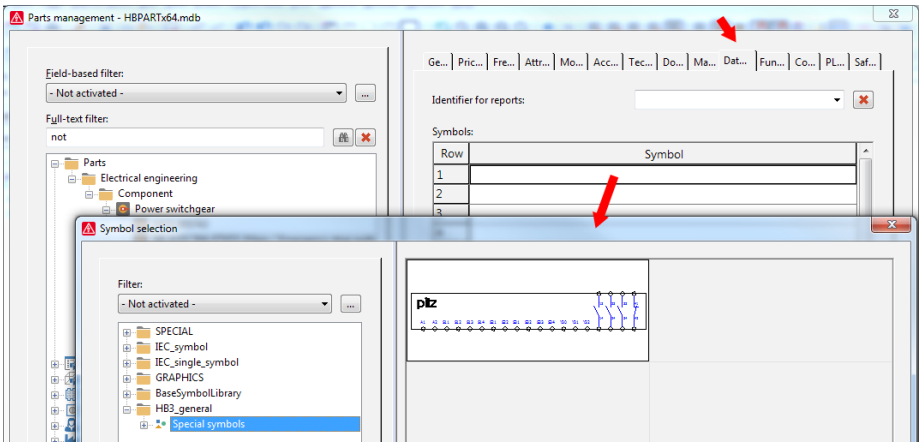


Fig. 13.86 Assign symbols to part

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.

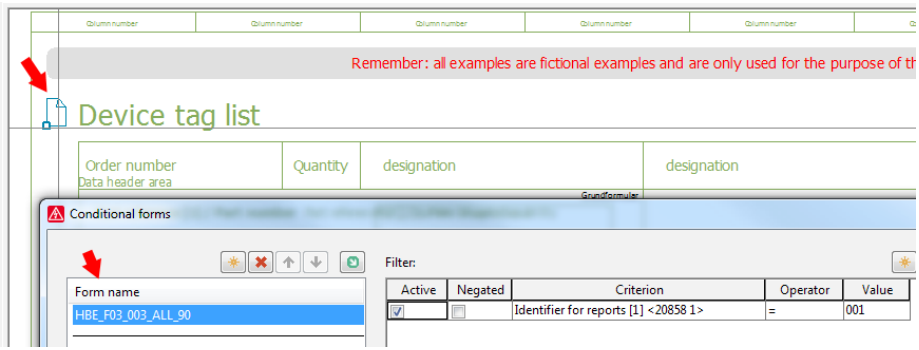


Fig. 13.87 Basic form editing

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.

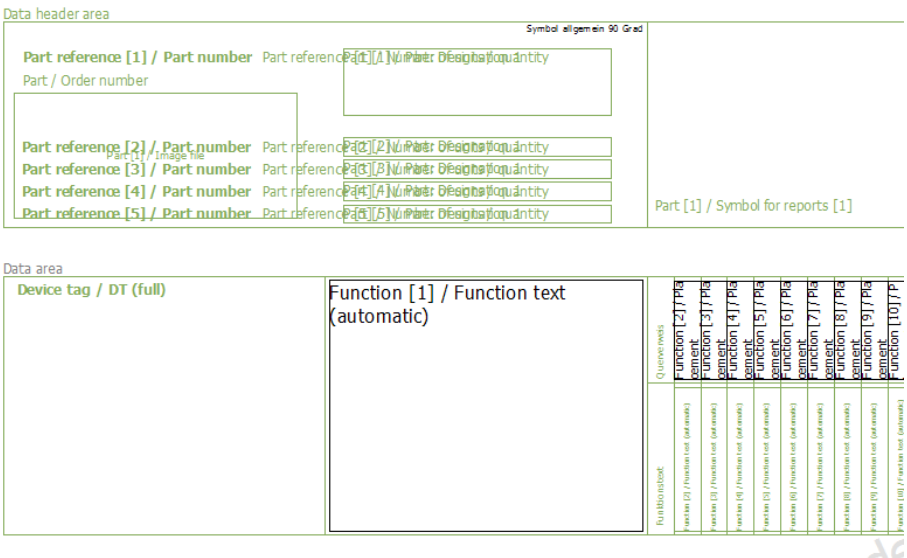


Fig. 13.88 Subform

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.

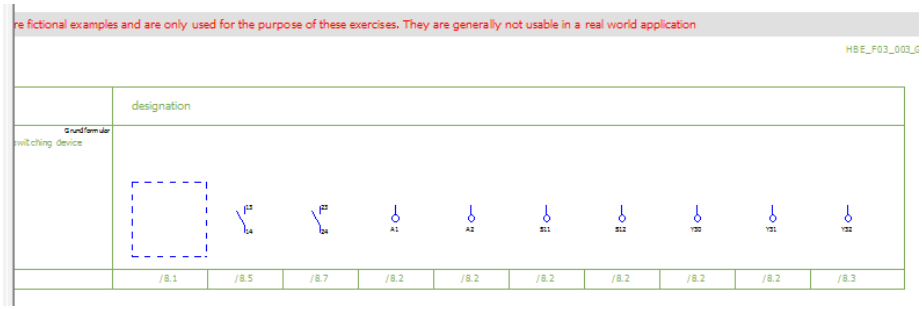


Fig. 13.89 Standard report without detailed representation

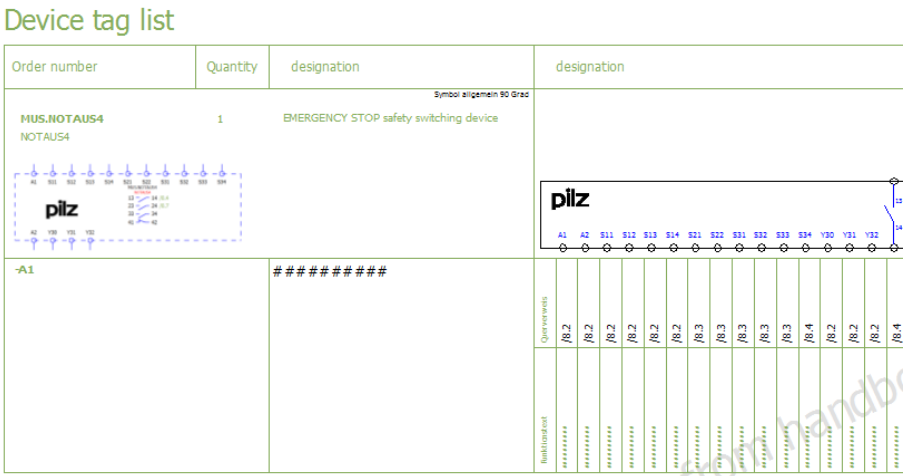


Fig. 13.90 Detailed report with symbol and special report

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.

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.

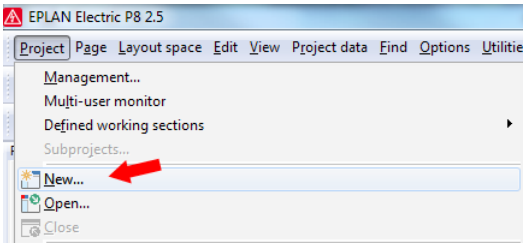


Fig. 14.1
Project/New menu

EPLAN opens the **CREATE PROJECT** dialog shown in Fig. 14.2 where settings need to be made.

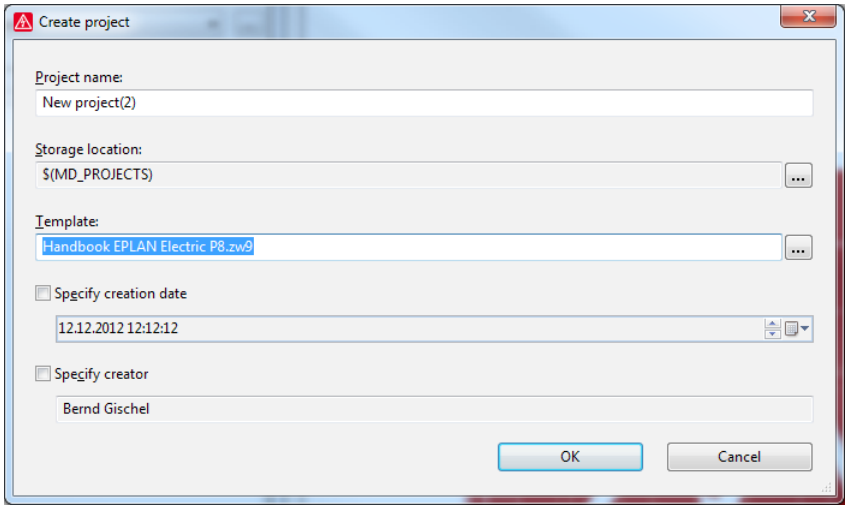


Fig. 14.2
Create project dialog

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_bas002.zw9
Creation date	Enter the desired date and time.	27.09.2015 xx:xx:xx
Creator	Enter the name of the person who is to be listed as the creator of the project.	Bernd Gischel

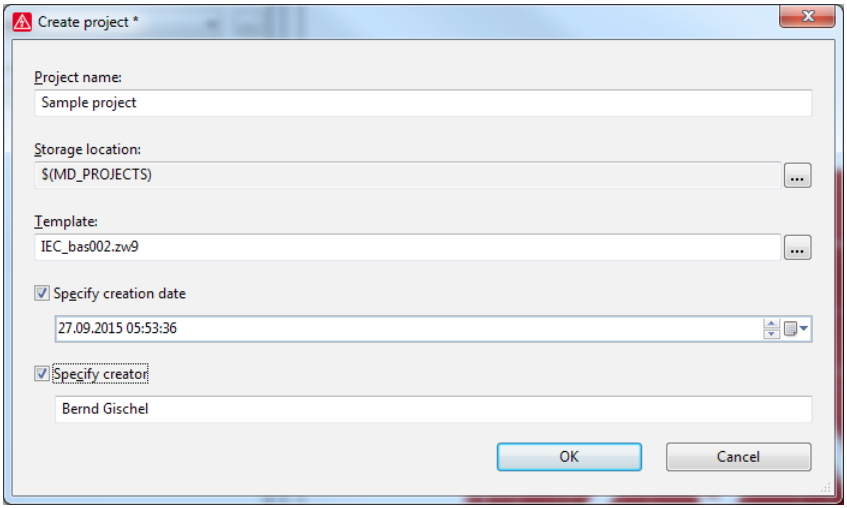


Fig. 14.3 Changed inputs in the Create project dialog

When you have entered all the information, you can close the dialog by clicking **OK**. EPLAN using the inputs to create a new project.

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.

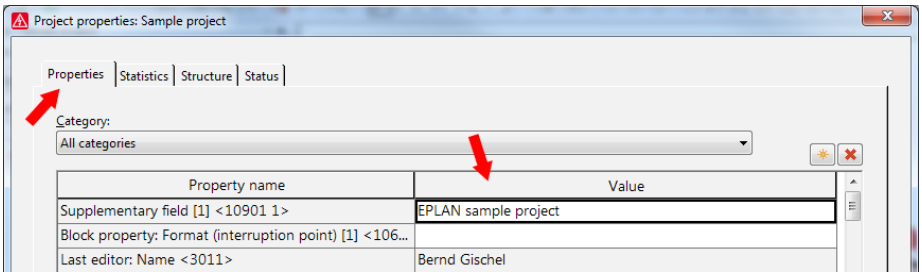


Fig. 14.4 Dialog for changing project properties

Now you can close the **PROJECT PROPERTIES** dialog. EPLAN then opens the new project in the **Page navigator**.

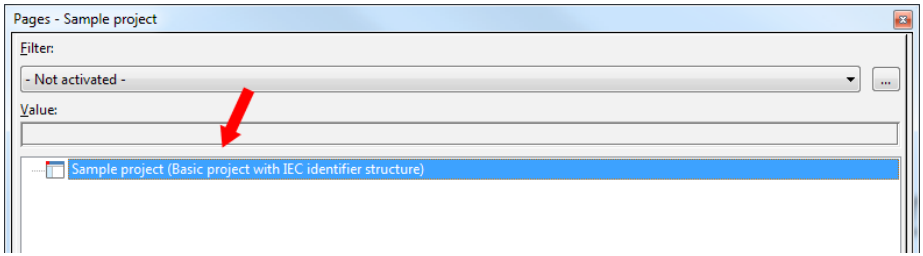


Fig. 14.5 Open project in the page navigator

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**.

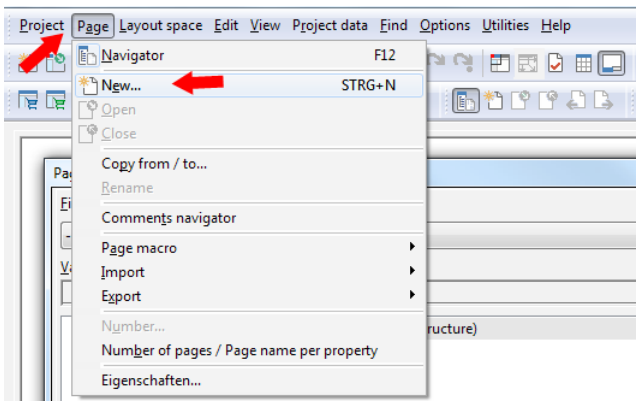


Fig. 14.6 Page menu

What are pages? Pages are needed in EPLAN, for instance, to create a graphical 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* (Electrical engineering trade) or *Fluid power schematic* (Fluid power trade). 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, as well as to filter them according to specific criteria.

Pages are created using the **PAGE/NEW** menu. Then you define the page properties (see Fig. 14.7 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.

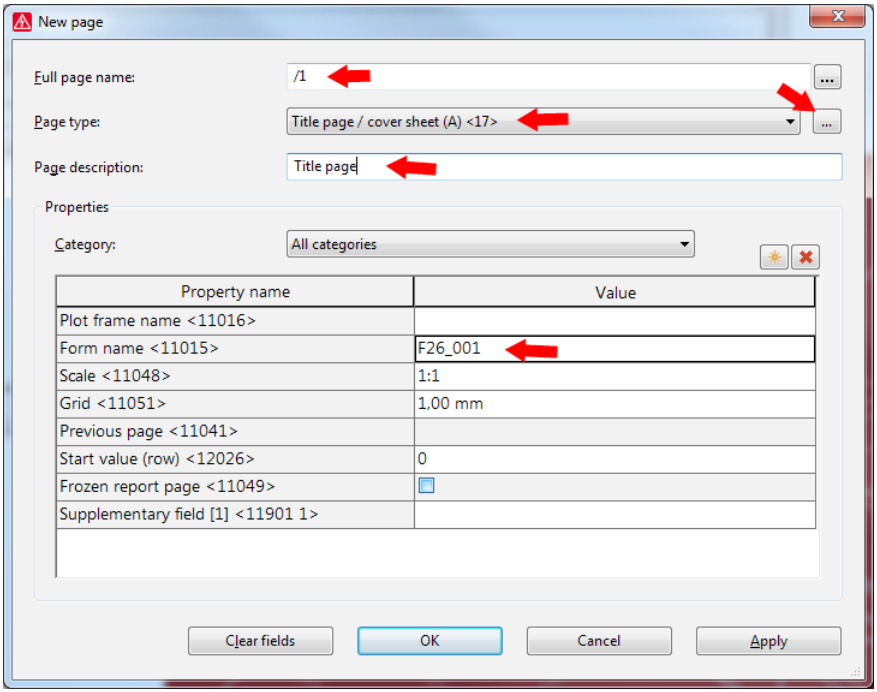


Fig. 14.7 Page properties dialog

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 the description of the page here.	"Cover sheet"
Optional; Properties	*Optional*; form name (Note: is usually automatically filled in by EPLAN)	*Optional*; F26_001 (here: special page type Cover sheet/Title page created manually)

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.

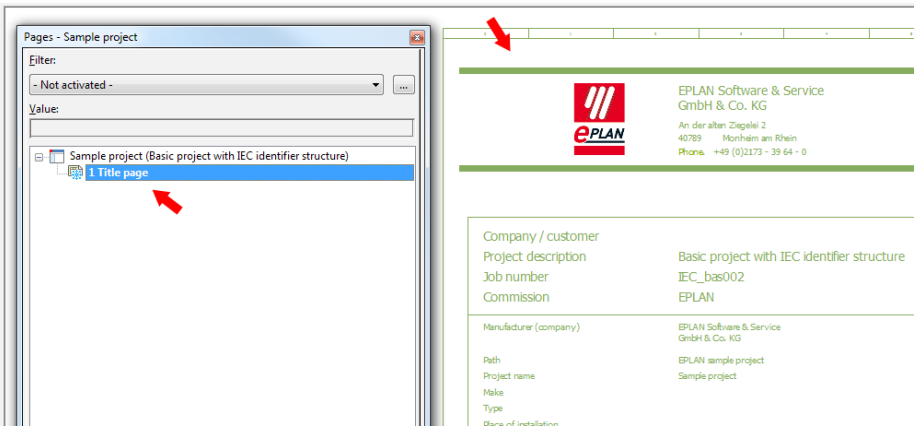


Fig. 14.8 First page – cover sheet

You use the same steps to create the next page.

- Call using 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.

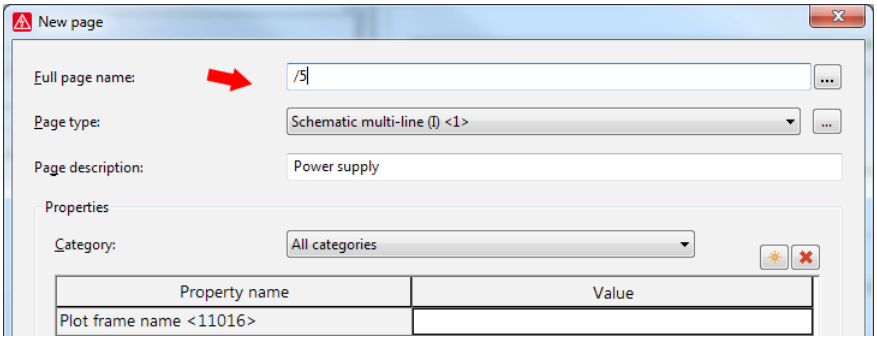


Fig. 14.9 New page with page type Schematic multi-line

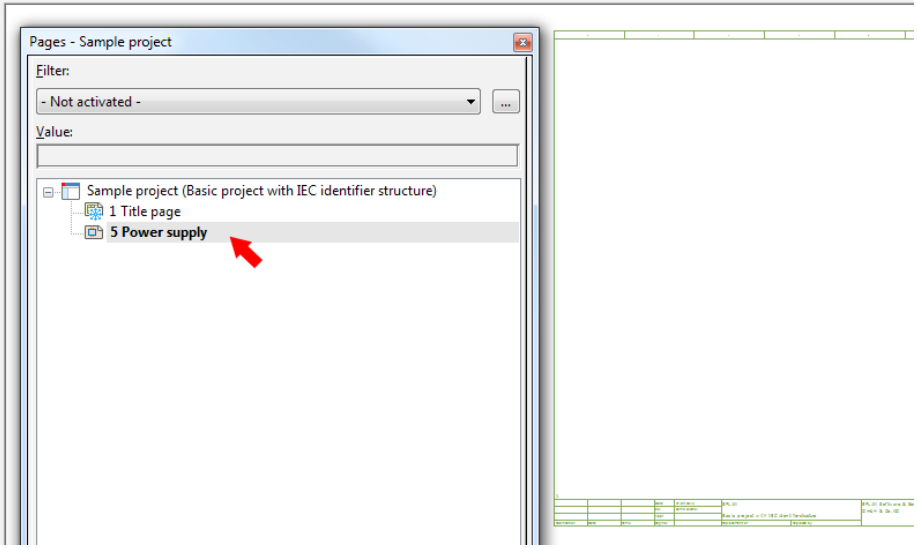


Fig. 14.10 Finished page in the page navigator

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.

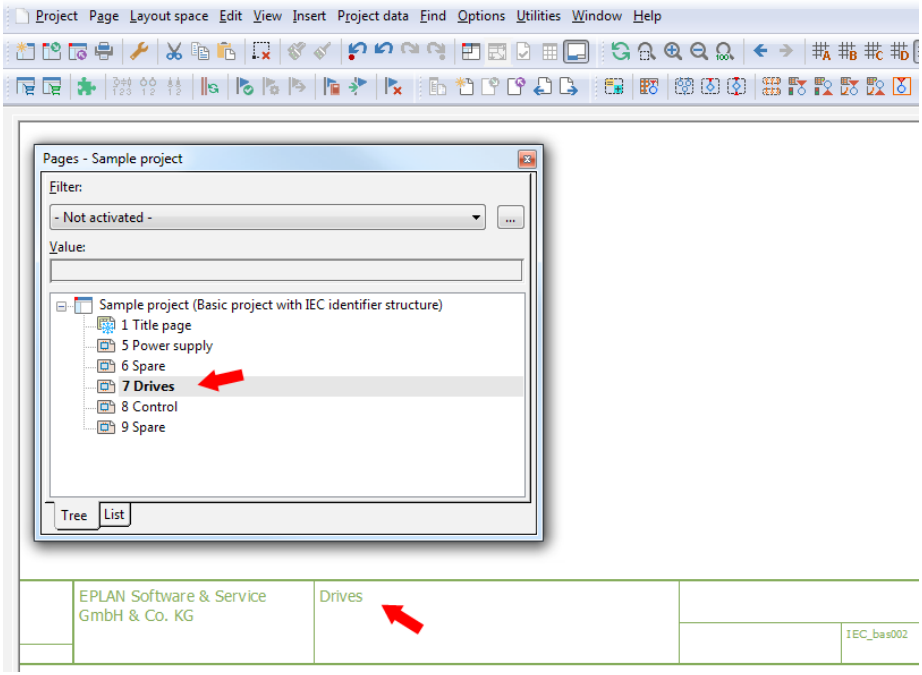


Fig. 14.11 Open page in the graphical editor

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.

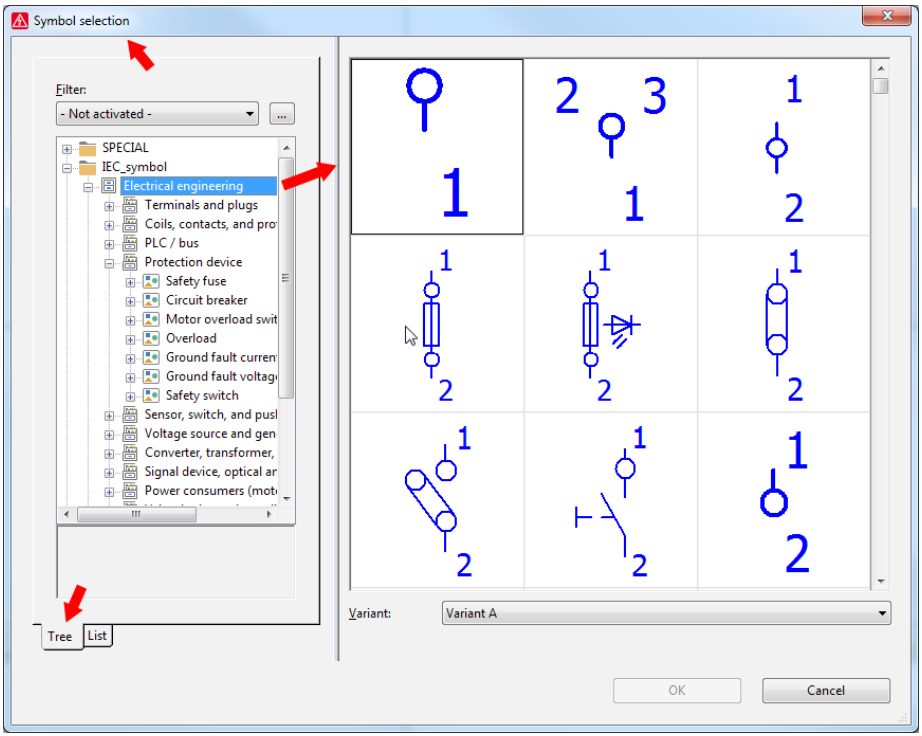


Fig. 14.12 Symbol selection dialog

In this dialog, you select the appropriate symbol and suitable variant (rotation angle) and click **OK** to apply it. EPLAN "hangs" the symbol on the cursor, and it can then be placed anywhere on the page. If the wrong variant was selected, the symbol can be rotated using the **TAB** key before being placed (the variant is exchanged).

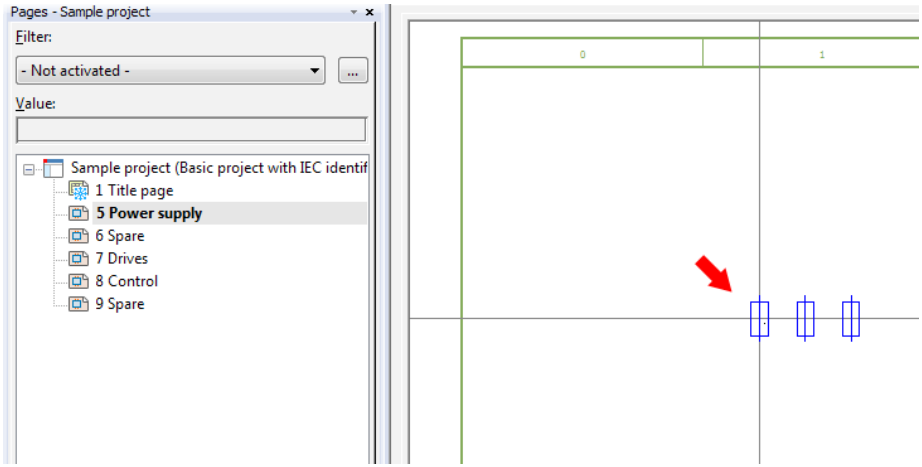


Fig. 14.13 Placing a symbol

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.

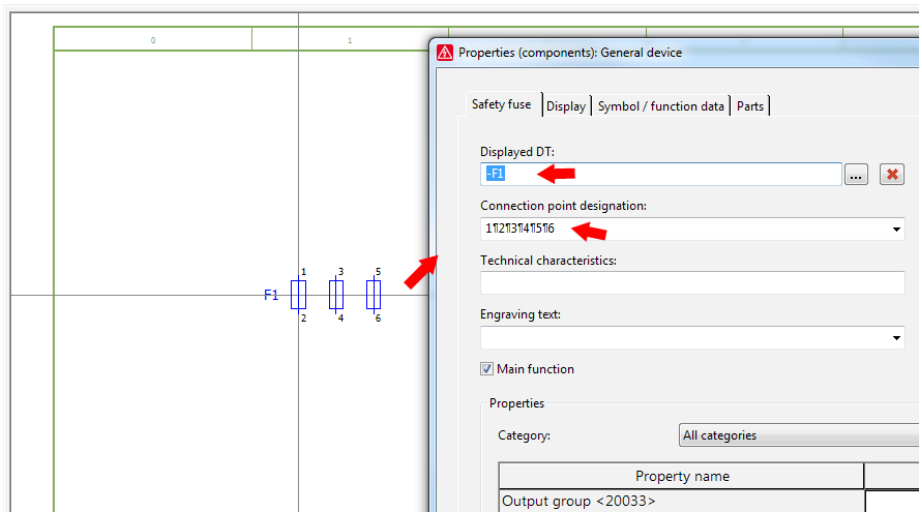


Fig. 14.14 Properties (component): General devices dialog

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 (grayed out) *Full DT* field 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**.

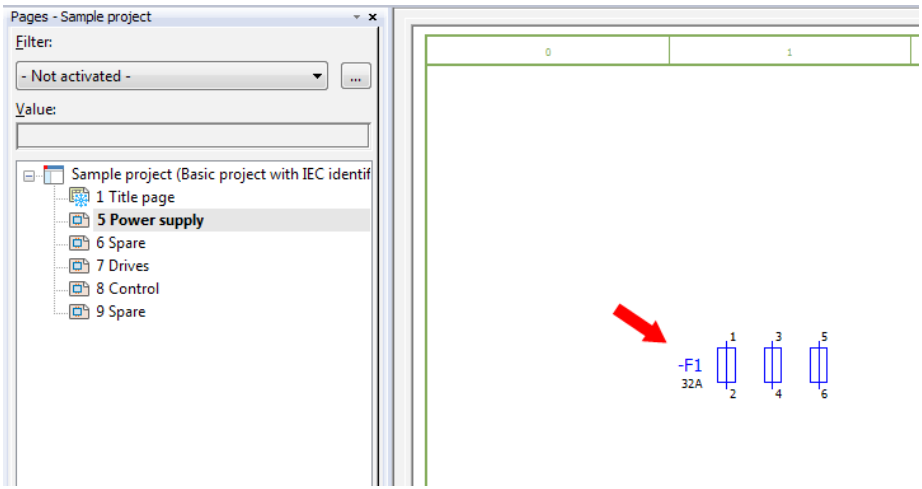


Fig. 14.15 Finished symbol with entered information

Now you can insert and place other symbols (terminals in the example). 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.

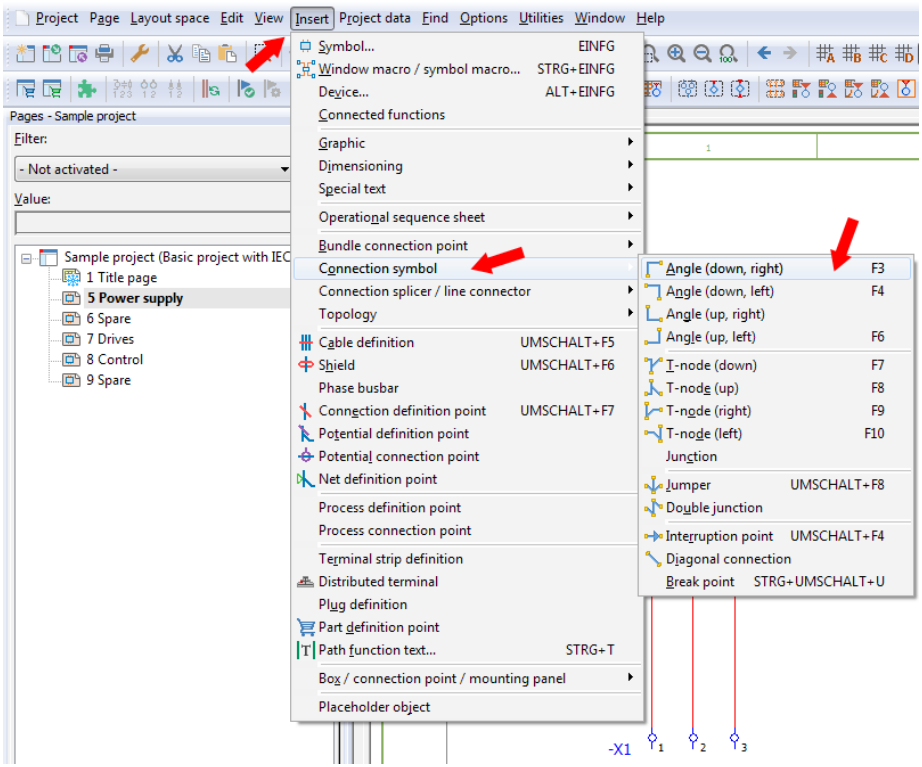


Fig. 14.16
Placing connection symbol

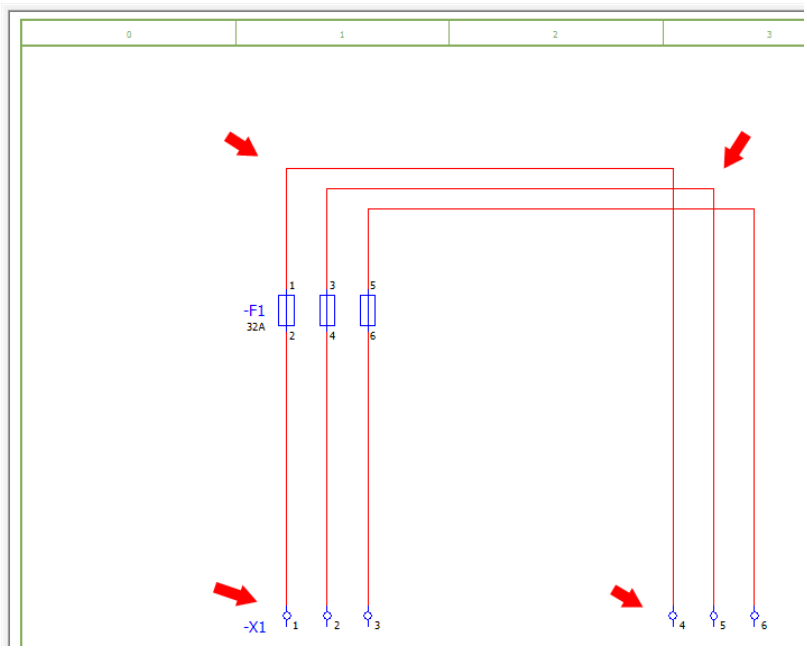


Fig. 14.17
Schematic with connection symbols

The terminals in the figures are inserting like normal devices using the **INSERT/SYMBOL SELECTION** menu or the **CTRL** key.

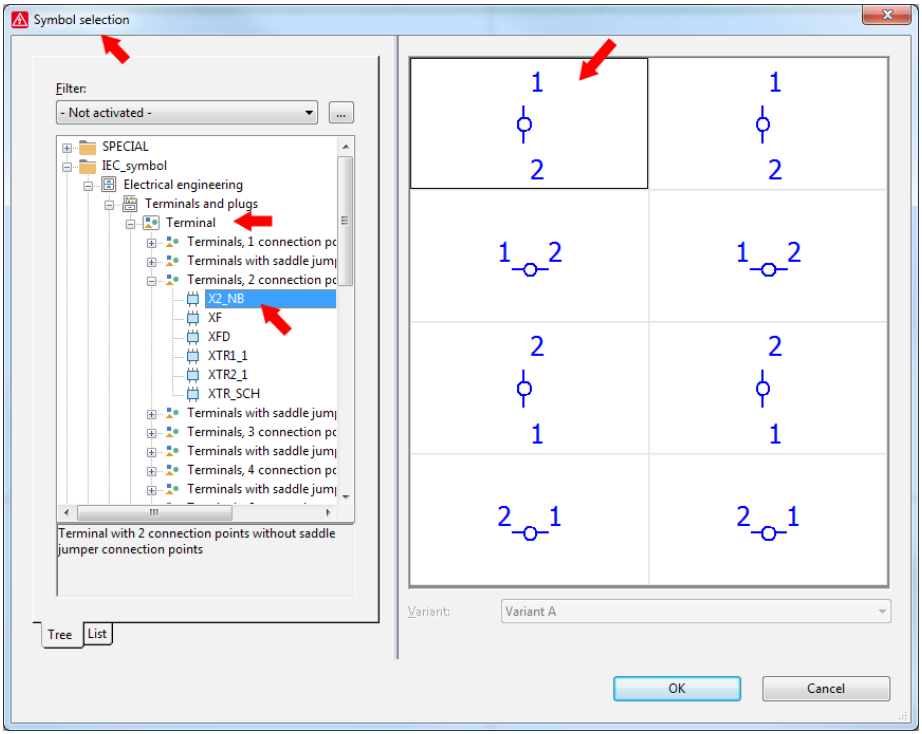


Fig. 14.18 Inserting terminals

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).

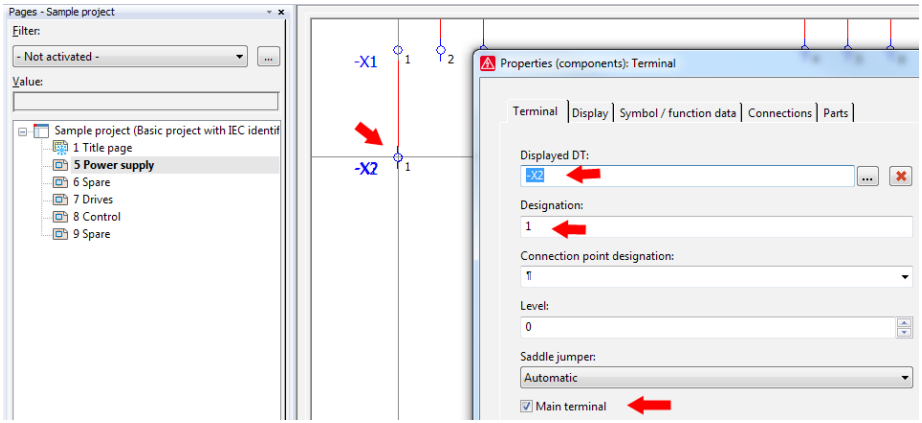


Fig. 14.19 Properties (component): Terminal dialog

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.

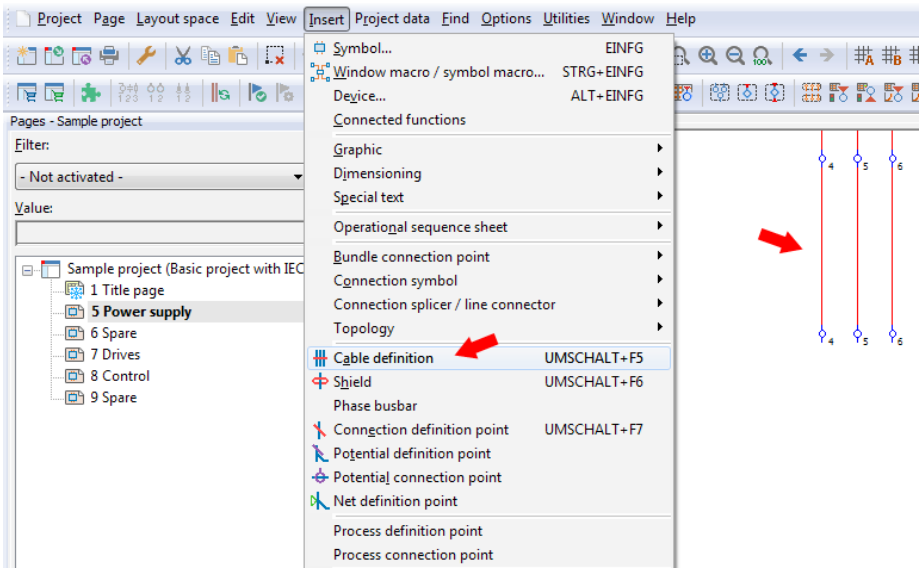


Fig. 14.20 Inserting cables

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).

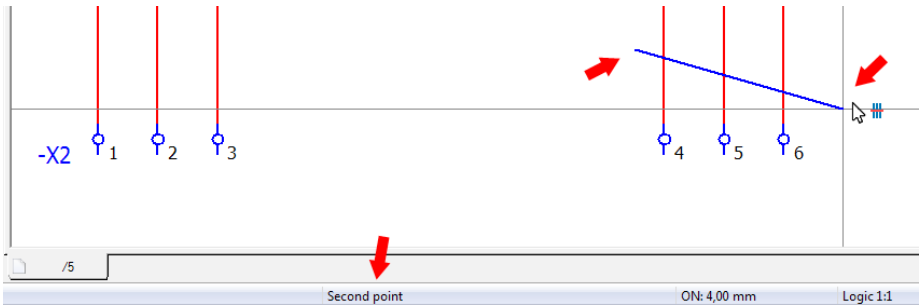


Fig. 14.21 Starting point of the cable definition

Now you drag the cable via the connection and left click to set the second point. EPLAN then opens the **PROPERTIES (COMPONENT): CABLE** dialog where you enter data such as the **Displayed DT**.

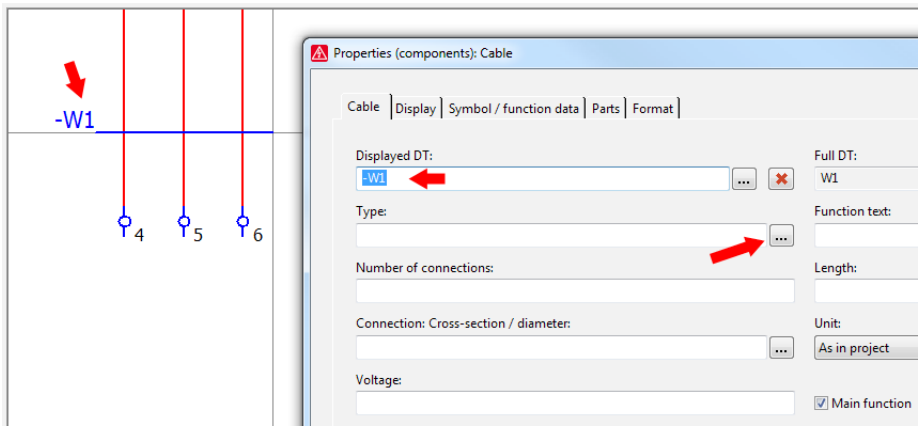


Fig. 14.22 Properties (component): Cable dialog

You can now add other cable data, such as **cable type** from **parts management**. The best way is to click the **[...]** button and select and apply the cable from parts management.

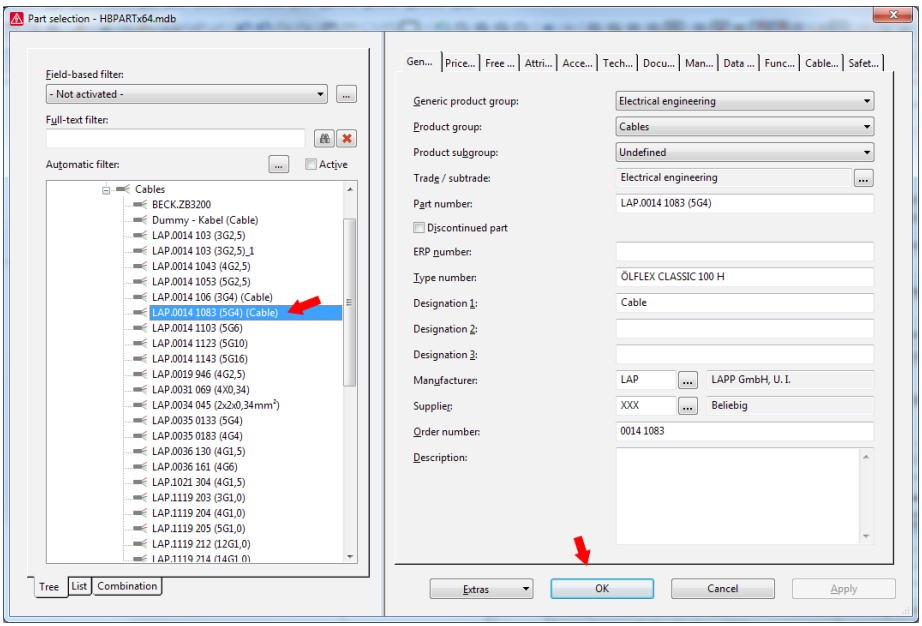


Fig. 14.23 Selecting a cable from parts management

When you click **OK**, parts management closes and EPLAN automatically adds the entered cable data such as cable type, number of conductors, etc.

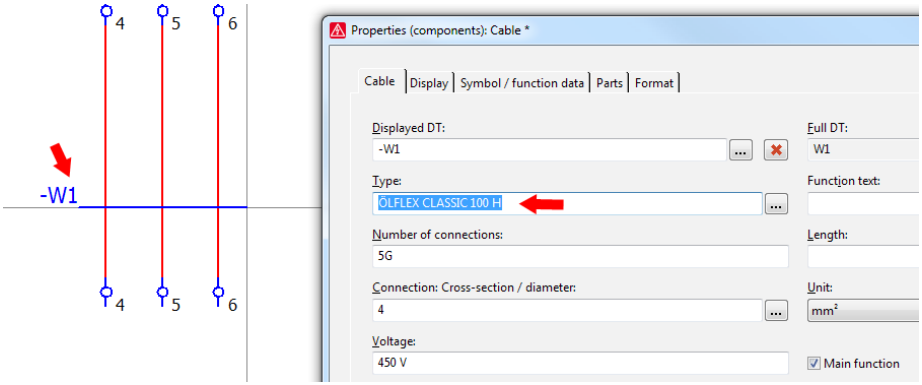


Fig. 14.24 Added cable data

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, without entering a part selection, 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.

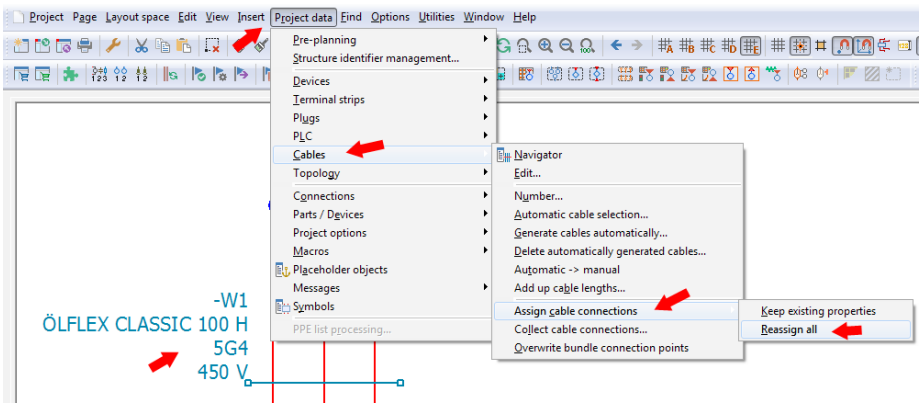


Fig. 14.25 Assigning conductors to the cable

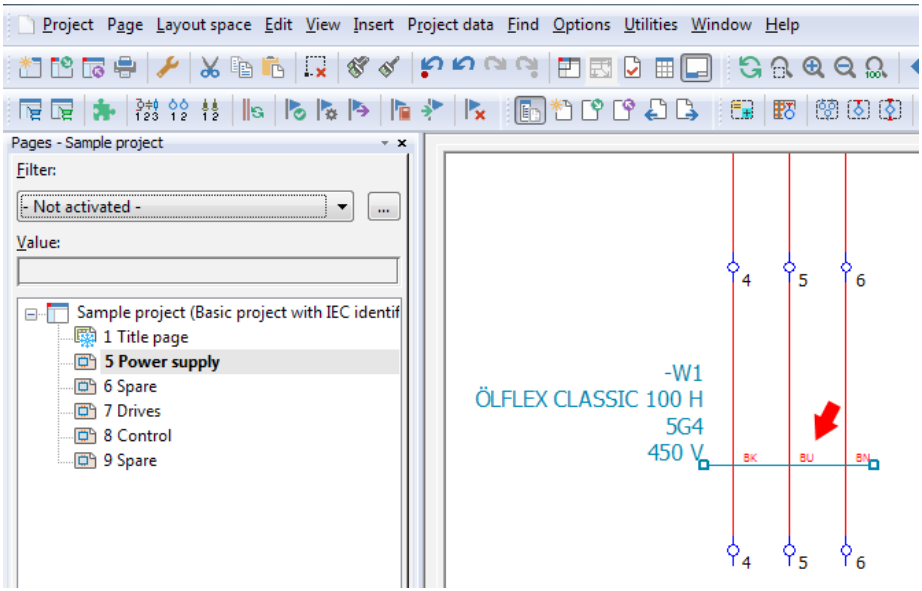


Fig. 14.26 Assigned conductors

14.3.3 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.

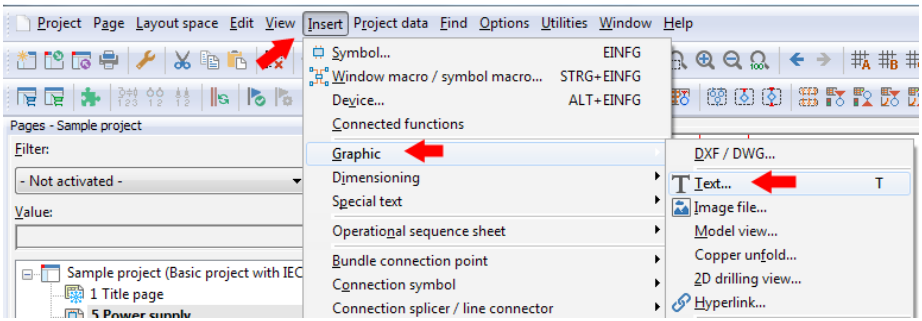


Fig. 14.27 Inserting free text

Path function text is inserted via the **INSERT/PATH FUNCTION TEXT** menu.

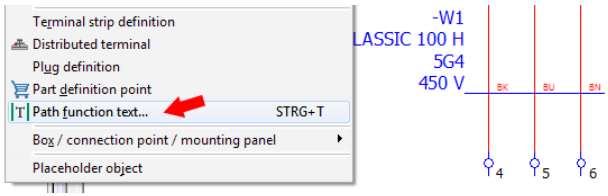


Fig. 14.28
Inserting path function text

After you have selected the type of text, EPLAN opens the **PROPERTIES/TEXT** dialog where you can enter the text and then place it.

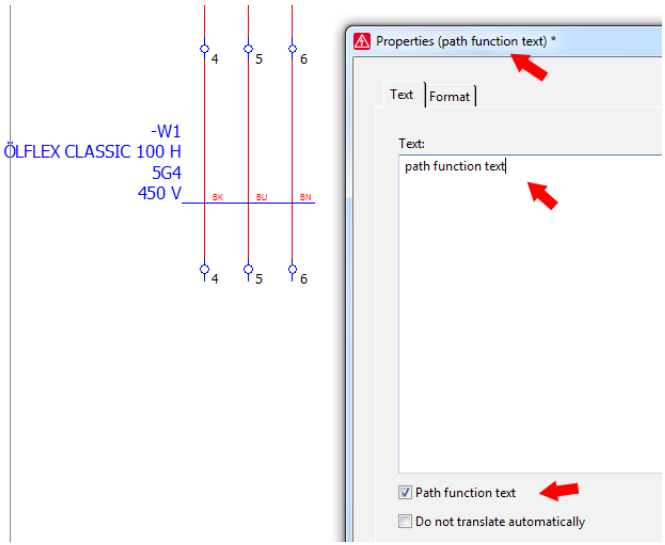


Fig. 14.29
Labeling with text

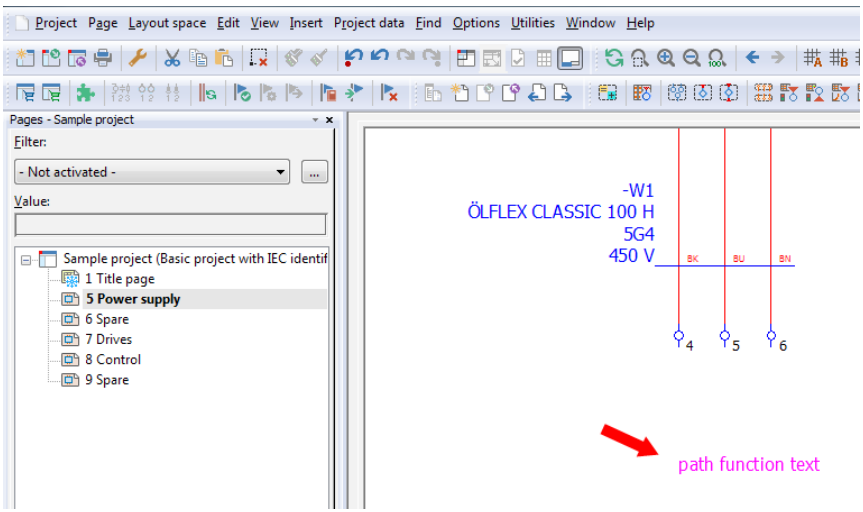


Fig. 14.30
Placed path function text

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.

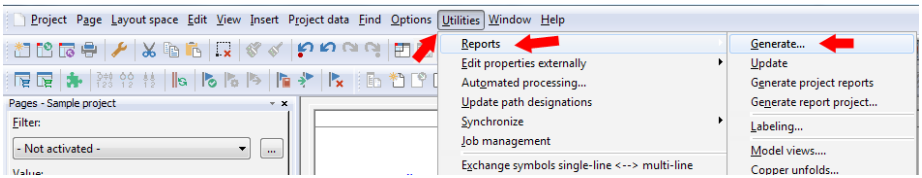


Fig. 14.31 Generate reports menu item

EPLAN opens the **REPORTS** dialog. Click the *Reports* tab.

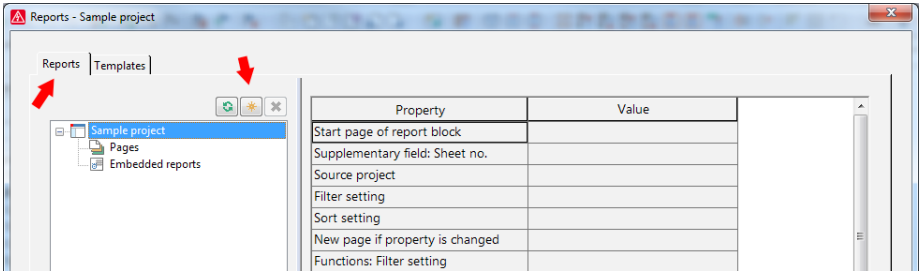


Fig. 14.32 Reports dialog

EPLAN opens the **SELECT REPORT** dialog after you click the button. Here you select the desired **Report type** and **Output format**.

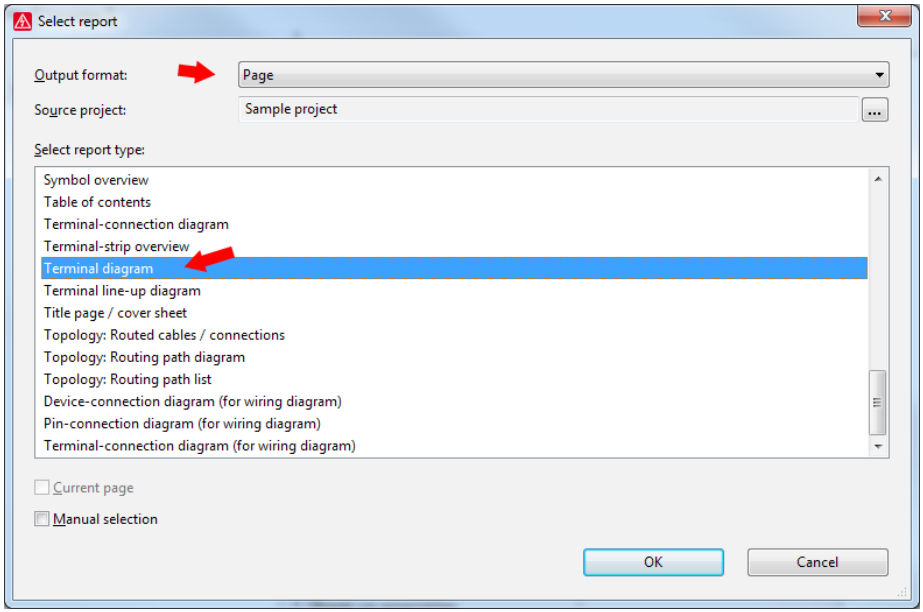


Fig. 14.33 Select report dialog

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.

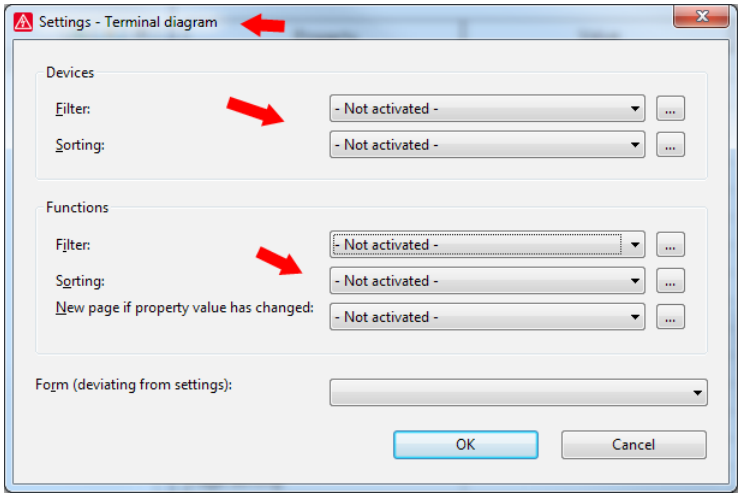


Fig. 14.34 Filter and sorting options

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.

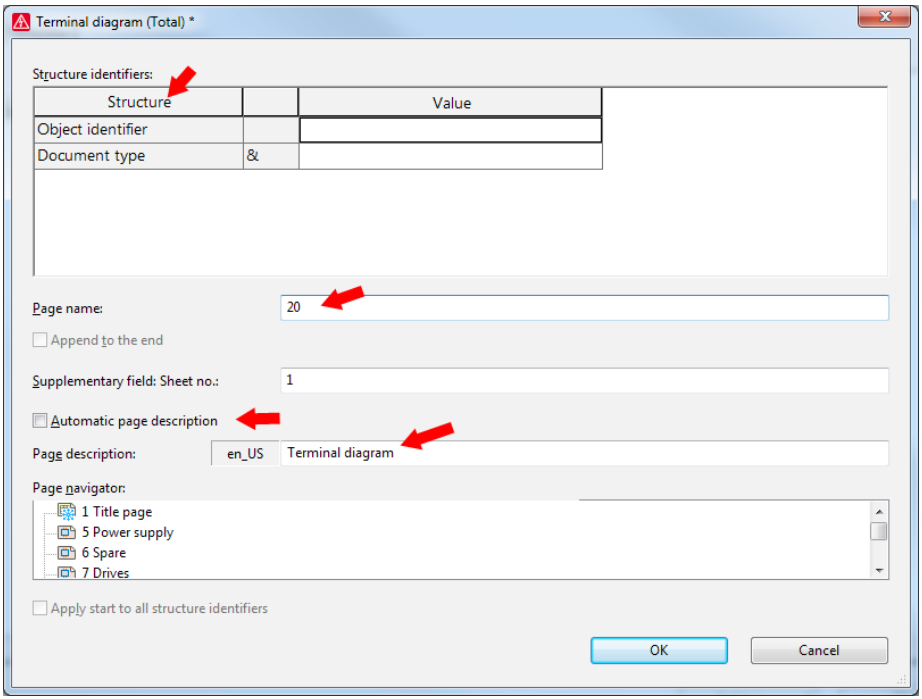


Fig. 14.35 Data for the terminal diagram page

After entering this data, click **OK** and EPLAN generates the diagram and automatically adds it as a new page or pages in the project.

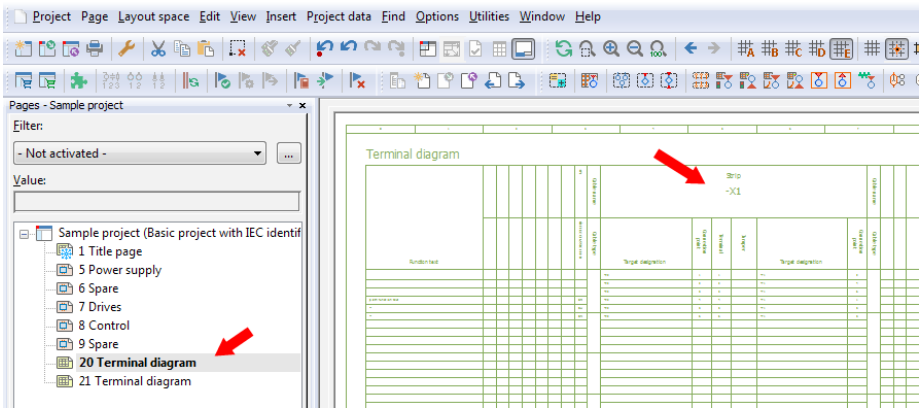


Fig. 14.36 Finished terminal diagram report

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. 14.37 shows a table of contents that was added to a project.

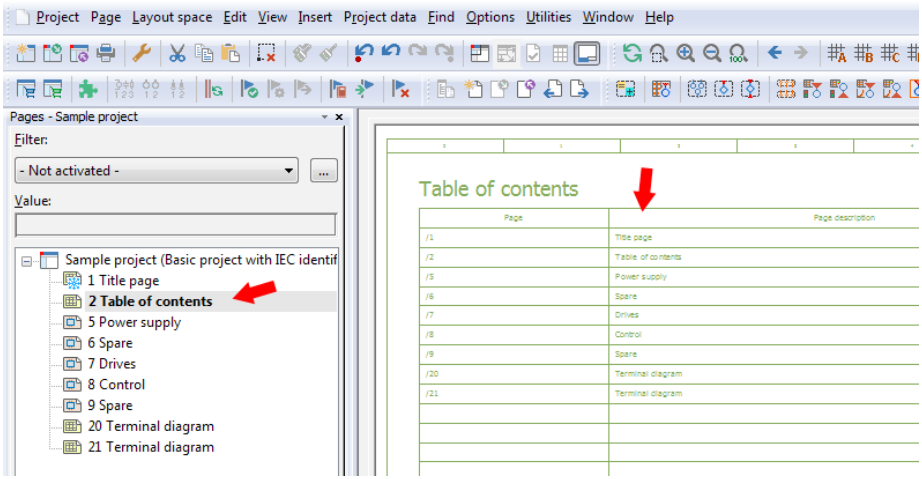


Fig. 14.37 Other reports

Index

Symbols

368
#H# *368*
#ID# *368*
#RO# *368*
#RW# *368*
<<...>> string *102 f.*

A

action line *593*
actions *439, 501*
adapt page structure *88*
add *147*
address *236, 239*
alignment *141*
alphanumeric *383*
angle variant *150*
API *39*
apply *386*
archive *491*
area of application *291*
arguments *595*
assign *176*
assign conductors *642*
assign main function *177*
assignment *218*
assignment lists *236*
assign pages *454*
asterisk *110*
AutoComplete *40*
autoconnecting *568*
AutoCorrect *40*
automated processing *439, 501*
automatic cable generation *223*
automatic cable selection *221*
automatic generation *223*
automatic page description *527*
auxiliary contact *154*
auxiliary function *176, 181*
available format elements *360*

B

backed up *485*
background *462*
background color *588*
backing up master data *505*
backup *440, 489*
back up data *502*
backup directory *496, 498*
backup drive *36*
back up entire project *493*
back up external documents *494*
backup file name *506*
backup function *489*
back up image files *494*
backup scope *489*
basic project *75, 567*
BIN *13*
black and white *458*
black triangle *59*
blank pages *329*
block editing *207, 436*
block properties *417*
bmp *460*
bold *438*
border *142, 149*
brackets *27*
break up assemblies *352*
browse *323*
by column or row *27*
by position *383*

C

cable assignment diagram *295*
cable conductor *217*
cable connection diagram *295*
cable data *225*
cable definition *228, 640*
cable definition line *20, 223*
cable diagram *297*
cable editing functions *221*
cable navigator *217*
cable numbering *21, 220*
cable overview *297*
cable parts *321*
cable properties *296*
cable selection *222*
cable type *223, 641*
cancel *220*
category *361, 391, 394*
CD/DVD burner *491*
center of rotation *566*

CFG *13*
changed pages only *464*
change of standard *37*
change positions *55*
characteristic *421, 547*
characters *329*
check form *526*
check routines *390*
check usage *384*
circle *117*
clipboard *34*
close *100, 438, 528, 534*
closed *100*
color codes *28*
color depth *458*
color designation *319*
color information *458*
color setting *37, 479, 581*
column *27, 533*
column configuration *66, 403*
combine *326*
COM interface *39*
comment function *592*
company fonts *42*
comparison criteria *421*
comparison project *71*
complete *440*
completed messages *396*
component data *421*
components *147, 634*
compress *30, 439, 499*
compression *458, 500*
compression function *258*
conductor *217*
conductor designations *226*
configure *47, 54, 402*
connection data *217, 223, 290, 317*
connection definition point *20, 568*
connection dialog *37*
connection junction *24, 37*
connection levels *294*
connection list *306*
connection point description *147*
connection point designation *147, 232, 421, 637*
connections *28, 110, 217, 367*
contact image *28, 154, 157*
contact image coil *272*
contact image margin *158*
contact image offset *576*
contact image position *577*
contactor coil *157*
contacts *157*
continued dimension *129*
conversion *537*
conversion to uppercase *20, 593*
coordinate input *111, 115*
coordinates *109, 114, 462*
coordinate system *110, 115*
copied *88, 520*
copy *90, 94, 128, 382, 439, 443, 461*
copy from/to *94*
copy to project *495*
correction *30*
create form *529*
create plot frame *535*
create report template *346*
creating *69*
creation date *628*
creator *628*
cross-reference *27*
cross-reference representation *27, 154*
cross-references *154*
currency *416*
cursor position *109*
cursor settings *37*
customer-supplied *605*

D

data area *527*
data backup *440, 485, 505*
database *385*
data entry fields *102*
data portal *541*
data records *318, 528*
data rows *528*
data to be compressed *500*
data type *235*
data volume *17, 500*
default cable type *223*
default directory *14*
default settings *40, 57, 321, 498, 500*
default values *158, 221*
definition *151*
delete *98, 128, 381 f., 439*
deleted *99*
delete report block *346*
description *407, 462, 490, 507*
description dialogs *34*
designations *414*
device connection diagram *293*
device cross-references *28*
device dialog *54*
device groups *436*
device navigator *174*
devices *361*
device selection *30, 226, 420*
device structure *69, 443*
device tag *174, 220*
device tag list *292*
device tag structure *24*
device types *166*
dialog elements *53*
dictionary *40, 383*
digital start address *241*
dimensioning *129*
dimension line termination *26*
direct entry *145*
directory *71, 369, 506*
directory structure *13, 34*
display *19, 22, 148*
displayed DT *147, 216, 637*
displayed languages *142*
distortion *462*
distributed view *232*
DOC *477*
document type *380*
dongle *8*
drawing directory *294*

DRX *116*
DRY *116*
DWG *446*
DXF *446*
DXF/DWG dialog *446*
DXF/DWG directory *452*
DXF/DWG files *38, 448*
dynamic *303, 527*
dynamic forms *328, 527*

E

edb *444*
edc *366*
edit *195, 217, 436*
edit externally *365*
editing in table *66*
editor *357*
edit properties *101*
edit terminal strips *197*
elk *71, 444*
ell *71*
elp *71*
elr *71*
els *71*
elt *71*
elx *71*
e-mail *440, 485, 491, 507*
e-mail message split size *36*
embedded reports *290*
enclosure legend *301*
end point *575*
engraving text *147*
enlarge *127*
enlarged *110*
enlargement *449*
entirely without report *443*
entirely with report *443*
entire project *222, 459, 464, 477*
epj *439, 467*
equals sign *318*
error *11, 394*
Excel *289, 356, 358, 362*
Excel template *367*
exchanging *366*
exclamation mark *570*
export *353, 445, 456, 466*
exported *373*
extension *142*
extent *202, 443*
external application *372*
external files *356*
external format *289*
external PPC system *39*
external project *97*

F

female pins *212, 214, 304*
filed-off project *71*
file off *491*
file type *366, 370, 449*
filter *59, 146, 292, 399, 441, 501*
filter dialogs *59*
filter setting *350*
find *385*
font *42, 140*
font size *140*
footer *359*
form *351*
format *58, 140, 182*
format elements *359, 361, 371*
format of the subpage *329*
formatting *58*
form editor *520, 523*
form errors *526*
form handling *527*
form name *523*
form overview project *514*
form project *70*
form properties *44, 291, 527, 530, 619*
forms *511, 519*
forms documentation *293*
form settings *317*
form structure *520*
found words *383*
free properties *416*
freeze report block *346*
full DT *147*
full name *415*
full page name *85, 632*
functional assignment *379*
function data *67*
function definition *30, 151, 155, 190, 205, 213, 241, 421*
function group *419*
functions *367*
function template *420*
function text *134, 147*
function text synchronization *179*

G

general *371*
generated *81*
generate file names *449*
generate functions *206, 242*
generate pin *215*
generate plugs *214*
generate project reports *355*
generate report *334*
global *566*
GRAFCET *20*
graphic *81, 455*
graphical approach *12*
graphical buttons *53*
graphical coordinate system *110*
graphical elements *127*
graphical functions *117, 523*
graphical output *289, 321, 326, 334*
graphical output pages *323, 328*
graphical pages *81*
graphical reporting *318*
graphical reports *289, 333*
graphics card *1*
grid *101, 109, 112*
grid sizes *37*
group *127*
group number *419*

H

handle *261, 269*
handle parts when outputting *320*
handles *58*
hardware *1*
hardware protection *8*
hash *110*
header *359*
higher-level function *379*
higher-level function number *380*
highlighted *221, 223, 233, 365, 520*

I

identical function text *318*
identifier *35, 368*
identifier block *380*
identifiers *21, 85, 145, 182, 375*
identifier set *21*
identifying property *232*
IEC identifier structure *74*
illogical values *368*
image files *456*
image formats *456*
IMAGES *461*
import *353, 372, 445, 459, 466f., 537*
import of parts *550*
increment *113*
indirectly *147*
information *109*
initial data *647*
insert *91, 128, 523*
inserted *459*
insertion mode *89*
insertion point *111*
installation *1*
installation site *379*
integer *329*
intelligence *134, 475*
interactive pages *81*
interfaces *38*
internal *39, 444*
internal output *289*
interruption points *569*
item numbers *383*

J

jpg *460*
jump functions *475*
jump point *389*

K

keep aspect ratio *462, 465*
keep source directory *461*
key combination *61*

L

label *359*
labeling *38, 289, 356*
labeling data *357*
labeling dialog *361*
language *142*
language display *149*
language output *357*
layer *140, 404*
layer configuration file *405*
layer designations *404*
layer export *405*
layer import *405*
layer management *404, 614*
layers *295, 302, 404*
layer settings *405*
layers of a macro *258*
legend form *618*
letters *31*
level *320*
license number *8*
line *117*
line thickness *30*
links *495*
lists *444*
list view *65, 79, 91, 146*
local *13*
logical coordinate system *111*
logical page *81*
logical PLC overview *233*
logic elements *113*
login data *542*
long-term archiving *456*
lost *100*

M

macro box *256*
macro name *260*
macro project *70, 255*
macros *255*
main form *325*
main function *176, 226, 637*
male pins *214*
management *434, 439*
managing *70*
manual *158, 225, 290, 349, 389, 404*
manual object assignment *261*
manual placement *301, 339, 341*
manual selection *341f.*
manufacturer *414*
manufacturer data *545*
master data *69, 505, 508*
master data directories *61*
maximum *527*
message class *393*
message description *389*
message management *11, 30, 388*
message number *394*
message priority *389*
message text *394*
message type *389*
method *490*
minimum number *328*
min. no. of report rows on report page *328*
motor overload switch contact image *154*
motor overload switches *28*
mounting data *418*
mounting location *324, 380*

mounting panel *264*
mounting site *147*
move *157, 218, 381, 439, 502*
move base point *115*
moved *148*
multi-line *27, 81*
multiple selection *508*
multiple starts *43*

N

name *35, 350*
naming conventions *69*
navigation elements *475*
navigator *165*
negative values *111*
network drive *34*
new *190, 205, 213, 228, 241, 346, 359, 529*
new device *193, 244*
new form *522*
new functions *206, 214, 242*
new layer *406*
new page *85*
new page if property is changed *350*
new project *74*
normal texts *133*
note *394*
no type of check *394*
numbering *21, 28, 179, 181, 195, 219, 454*
numbering format *182*
numbering function *181*
numbering (offline) *21*
numbering (online) *21*
numbering options *219*
numbering pattern *242*
number/name *150*
number of decimal places *35*
number (offline) *182*
number plugs *213*
number terminals *201*
numeric *202*
numeric keypad *435*

O

object-oriented approach *12*
ODBC *39*
offline *21, 181, 375*
offline check *390*
offline numbering *21, 181*
offline type of check *395*
one language (variable) *142*
online *12, 290, 317, 375, 389*
online assignment *21*
online check *390*
online/offline type of check *396*
online system *12*
only header data *444*
only logic pages *444*
open *87, 95, 323, 438, 460, 529*
open in new window *87*
operation *71*
options *357*
order number *414*
organize *439*
origin *110f.*
original project *491, 538*
output *19, 364, 450*
output directory *448, 458*
output file *357, 372*
output form *346*
output format *336*
output settings *458*
output sorting *463*
output to pages *321*
output type *357*
overview *27, 40, 516*
overview cross-references *27*
overwrite with prompt *461*
overwriting *72*
overwritten *521*
own layers *406*

P

pack *440*
packed *440, 485*
page *27, 84, 367, 445*
page break *326, 527*
page contents *88*
page description *101, 351, 632*
page editing *84*
page handling *31*
page macros *260, 270*
page name *27, 85, 585*
page navigator *79*
page number *89, 98, 329*
page numbering *568*
page properties *44, 91, 101, 305, 365, 523*
page property texts *136*
page range *464*
page scale *270*
page sorting *323, 325, 351, 598*
page structure *72, 77, 86, 97, 323, 443, 454, 514*
page type *27, 33, 53, 81, 632*
pair index *610*
pairs *569*
part *153*
part assemblies *320*
partial output *325, 351*
part number *153, 414*
part quantity list *292*
parts data *413*
parts database *30*
part selection *30, 39, 153*
parts list *291*
parts management *172, 244, 411*
parts master data *468*
path areas *53*
path function text *236*
path function texts *134*
path numbering *31*
PDF export *476*
PDF file *477*
PDF format *475*
percent *462*
performance *35*
PE terminals *202*
phone *35*
P_ID *13*
pin connection diagram *302*

PIN number *232*
pins *212*
placed *290, 462*
placeholder object *273*
placeholder text *523, 533*
placing *326*
plain text *133*
PLC address *233, 236*
PLC assignment lists *22*
PLC box *241*
PLC card *166, 232, 243*
PLC card overview *302*
PLC components *302*
PLC data *245*
PLC diagram *301*
PLC navigator *231*
PLC numbering *21*
PLC overview *232, 234*
PLC-specific settings *241*
PLC terminal *232, 235*
plot frame *31, 533*
plot frame documentation *300*
plot frame editor *531*
plot frame properties *44, 158, 533*
plot frames *511, 531*
plug diagram *303*
plug navigator *212*
plug overview *304*
plugs *212, 214, 304*
plug/socket strips *214*
png *460*
point wiring *24*
polyline *117*
positions *158*
post-processing *516*
potential overview *300*
prefix *20*
preselection *30*
preview *94, 452*
preview dialog *179*
preview of result *183, 220*
previous versions *537*
prices/other *415*
print *463*
print command *463*
printer *463*
print margins *465*
print position *465*
print preview *465*
print settings *464*
print size *464*
print to scale *464*
profile *440*
program directory *13*
program start *15*
project *69*
project cables *295*
project checks *30, 390*
project copy *443*
project data *165, 212, 290, 318, 321, 328, 341, 363, 519, 523*
project database *42, 435, 440*
project directories *435, 440*
project documentation *289, 294, 519*
project editing *30, 71, 74*
project file *486*
project information *440*
ProjectInfo.xml *486*
project languages *142, 357*
project management *15, 434*
project master data *18, 30, 38, 61, 70, 323, 332*
project messages *375*
project name *74, 100, 496*
project options *553*
project options group *553*
project options overview *563*
project path *496*
project properties *18, 44, 77, 486*
project property texts *136*
project selection *95*
project setting *332*
project-specific *404*
project structures *22*
project template *72*
project tree *435*
project type *70f.*
properties *43, 77, 101, 194, 350*
property *148*
property arrangement *56*
property arrangements *611*
property dialog *111*
property fields *437*
property name *43, 147*
property number *36, 43*
property selection *45, 53, 101, 148*
purchase price *415*

Q

quantity unit *415*

R

range *33*
read back *372*
read-only *368*
read write *368*
rectangle *117*
reduce *127*
reduction *449*
relative coordinate input *115*
remove project data *500*
rename *439*
renaming *98, 366*
reorganize *439*
repetition *318, 357*
replace *382, 385*
report pages *290, 329*
report runs *317*
reports *19, 35, 53, 134, 289, 334, 443*
report type *53, 291, 322, 336, 347*
representation type *151, 270*
restore *495f.*
restore back *495*
restored *485*
restore master data *507*
restrictions *346*
review *390*
revised project *71*

revision control *30*
revision overview *300*
revision tracking *71*
round *329*
row texts *533*
RX *109*
RY *109*

S

save *437, 478*
save additionally *491*
saved *373*
scale *101, 127, 566*
scaling factor *449*
schematic project *70, 627*
scheme *181, 220, 241, 358 f., 371, 389, 391, 447, 453*
script name *502*
scrolling *37*
sections *553*
select *12, 98, 180, 435*
select associated objects *263*
selected *342, 365, 371, 450*
selected format elements *360*
select form *323*
selection *88, 464*
selection list *147*
select plot frame *534*
separators *27, 38*
sequence *53, 202, 403, 585*
sequence chains *20*
sequence sheets *20*
sequential numbering structure *74*
service time *419*
set data types *235*
settings *17, 164, 358*
settings – company *41*
settings – project *18*
settings – station *41*
settings – user *34*
shopping cart *551*
shortcut keys *40*
single-line *27, 81*
single-screen solution *2*
size *462*
snap *584*
sort *196, 383*
sort code *569*
sorting *81, 384*
sorting into *454*
sorting sequence *86*
sort setting *350*
source *217, 229, 446, 477*
special characters *20, 586*
special text *45, 48, 136, 523*
split size *491*
standard base point *115*
standard layers *405*
start application *362, 372*
starting point *575*
start page *350*
static *527*
static coordinate input *115*
static forms *328, 527*
status *502*
status bar *109*
storage location *69, 75, 435*
storage medium *491, 507*
stored *31, 43, 61, 70, 323, 499, 529*
storing *17*
stretch *574 f.*
string of characters <<...>> *437*
structure *77, 146*
structure identifier management *81, 375*
structure identifier overview *304*
structure identifiers *20, 22, 31, 35, 85, 348, 375*
student version *573*
style version *448*
sub-entries *404*
subpages *31, 329*
subsequent editing *366*
summarized *324*
summarized parts list *292*
superior *577*
supplementary field *350*
supplier *414*
switched on *113*
symbol *27, 145, 634*
symbol data *150*
symbol/function data *150*
symbol library *19, 31, 146*
symbol list *145*
symbol names *145*
symbol overview *305*
symbol preview *145*
symbol project *70*
symbol properties *27, 44*
symbol representation type *146*
symbol selection *145, 147, 171, 635*
symbol variants *145*
synchronization *70, 512*
synchronize *29, 179*
synchronize function texts *180*
synchronize master data *291, 522, 580*
synchronize parts *468*
system master data *18, 30, 60, 70, 332, 511 f.*
system messages *35*

T

tab *366*
tabbed page list *88*
table of contents *294*
target *217, 229*
target and point wiring *24*
target designations *454*
target directory *496*
target file *362*
target project *76*
target system *448*
target wiring *24*
technical characteristic *147*
technical data *419*
template project *567*
templates *75, 333 f., 346, 363, 628*
temporary *70, 512, 532, 535*
temporary page *523, 529*

temporary project *71*
terminal connection diagram *297*
terminal diagram *299*
terminal editing *166*
terminal line-up diagram *298*
terminal number *232*
terminal parts *321*
terminal schemes *203*
terminal strip definitions *608*
terminal strip navigator *195*
terminal strip overview *299*
text dialog *58*
text expansion *143*
text file *358*
text input field *139*
texts *132*
title page/cover sheet *305*
T-node *25, 37*
total *324*
total + higher-level function *325*
transferred *180*
translation *31, 133, 139, 143, 383*
translation run *33*
tree structure *80*
tree view *64, 91, 146*
two-screen solution *2*
types of check *389*

U

UNDO command *100*
UNDO function *366, 437, 531*
unfold *28*
ungroup *128*
units *28, 35*
unpack *440*
unpacked *485*
unplaced *217, 242, 281, 292, 444, 499*
update *346*
update pages *355*
upper-case letters *20*
use available function data *164*
used *217*
use existing *167*
use graphical coordinates *114*
use original DT *225*
user *440*
user-defined *57, 380*
user interface *34, 63, 108*
user settings *36*
user supplementary field *437*
using the mouse *63*
utilities *289*

V

value *147*
value sets *272, 569*
variable name *277*
variant *150, 270*
viewer *622*
viewpoints *165*

W

warning *394*
watermark *30*
wearing part *419*
which cables? *222*
window legend *301*
window macros *70, 268*
wire *217*
workbook *88*
working area *108*
workspace *34 f.*
workstation *435*
write back *233, 290, 367*
write protection *491*
writing back *220*

X

XML file *359, 366, 468*
X,Y *110*

Y

Y positions *115*

Z

zipped project *71*
zw1 *505*
zw2 *505*
zw3 *505*
zw4 *505*
zw5 *505*
zw6 *505*
zw7 *505*
zw8 *505*