

# **Getting Started**

06 / 2006



Copyright © 2006 by EPLAN Software & Service GmbH & Co. KG

EPLAN Software & Service GmbH & Co. KG assumes no liability for either technical or printing errors or for deficiencies in this technical information and cannot be held liable for damages that may directly or indirectly result from delivery, performance, and use of this material.

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

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

EPLAN Electric P8<sup>®</sup>, EPLAN Fluid<sup>®</sup>, EPLAN PPE<sup>®</sup> and LOGOCAD TRIGA<sup>®</sup> are registered trademarks of EPLAN Software & Service GmbH & Co. KG.

 $\rm MS\text{-}DOS^{\it @}$  / Windows  $\rm NT^{\it @}$  / Windows  $\rm 2000^{\it @}$  / Windows  $\rm XP^{\it @}$  / Microsoft Windows  $^{\it @}$  are registered trademarks of the Microsoft Corporation.

Microsoft<sup>®</sup> Excel<sup>®</sup> and Microsoft<sup>®</sup> Access<sup>®</sup> are registered trademarks of the Microsoft Corporation.

RITTAL® is a registered trademark of the Rittal GmbH & Co. KG.

CLIP PROJECT® is a registered trademark of the Phoenix Contact GmbH & Co.

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

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

AutoCAD<sup>®</sup>, Autodesk Inventor<sup>®</sup> and Volo<sup>™</sup>View Express are registered trademarks of Autodesk, Inc.

SIMATIC HW Konfig<sup>®</sup> is a registered trademark of the Siemens AG.

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

ObjectStore<sup>®</sup> is a registered trademark of eXelon Corporation.

D-ISAM<sup>®</sup> is a registered trademark of Informix Software Inc.

Hardlock E-Y-E<sup>®</sup> is a registered trademark of Aladdin Knowledge Systems, Ltd Int.

InstallShield® is a registered trademark of InstallShield, Inc.

PMS<sup>®</sup> is a registered trademark of PMS Compelec GmbH.

MicroStation<sup>®</sup> is a registered trademark of Bentley Systems.

Acrobat<sup>®</sup> is a registered trademark of Adobe Systems Inc.

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

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

# **Table of Contents**

| Introduction                                | 3  |
|---|----|
| User's Comments                             | 3  |
| Remarks on the Practice Project             | 4  |
| Starting EPLAN                              | 6  |
| Getting to Know the EPLAN User Interface    | 8  |
| What You Need to Know in Advance            | 9  |
| Interface Elements                          | 9  |
| Changing the User Interface                 | 13 |
| Creating a Project                          | 19 |
| What you Need to Know in Advance            | 19 |
| Creating a Project Using the Project Wizard | 20 |
| Opening and Editing Pages                   | 25 |
| What You need to Know in Advance            | 26 |
| Opening Pages                               | 26 |
| Editing Page Properties                     | 28 |
| Creating a Page                             | 31 |
| Opening and Closing a Project               | 34 |
| Opening the Project                         | 34 |
| Paging Through the Project Pages            | 36 |
| Creating a View                             | 37 |
| Closing the Project                         | 42 |
| Creating a Schematic                        | 44 |
| What you Need to Know in Advance            | 44 |
| Inserting "Angle" Connection Symbols        | 44 |
| Inserting Potential Connection Points       |    |
| Inserting "T-node" Connection Symbols       |    |
| Inserting Graphical Elements                | 51 |
| Inserting General Devices                   | 54 |
| Inserting Terminal Strips and Terminals     | 62 |

| Drawing Cable Definition Lines                                   | 66  |
|--|-----|
| Updating Connections   | 68  |
| Inserting Path Function Texts                                    | 68  |
| Inserting "Interruption Point" Connection Symbols                | 70  |
| Generating Cross-references                                      | 73  |
| What you Need to Know in Advance                                 | 73  |
| Automatically Generating Interruption Point Cross-reference      | s74 |
| Inserting More Connection Symbols and General Devices            | 76  |
| Inserting Contactor Coils and Contact Images                     | 79  |
| Inserting Contacts and Contact Image for Motor Overload Switches | 83  |
| Selecting Parts  | 87  |
| What you Need to Know in Advance                                 | 87  |
| Assigning Parts  | 87  |
| Generating Reports   | 96  |
| Generating a Terminal Diagram                                    | 96  |
| Performing Settings for Parts Lists                              | 102 |
| Printing Other Report Pages                                      | 103 |
| Creating and Inserting Macros                                    | 108 |
| What you Need to Know in Advance                                 | 108 |
| Creating a Window Macro  | 108 |
| Inserting a Window Macro   | 110 |
| Finding and Replacing Identifiers                                | 114 |
| Finding and Replacing Device Tags                                | 114 |
| Finding Text and Editing Search Entries                          | 119 |
| Completing and Updating Reports                                  | 122 |
| Updating Reports   | 122 |
| Creating Report Templates  | 123 |
| Generating Project Reports                                       | 124 |
| Printing a Project   | 127 |
|  |     |

# Introduction

Dear users,

The team at EPLAN Software & Service GmbH & Co. KG are pleased to welcome you as a new user.

This guide is for users who have never used EPLAN. Using a practice project, we will familiarize you with the basic functions of EPLAN.

Since this documentation is designed as an introduction to EPLAN, many of the more complicated functions and relationships will be ignored. These details are covered in our seminars, where you can also find help with your own individual and company-specific tasks. You should definitely take part in our seminar program in order to make the best use of the functionality and possibilities of the system.

We provide you with an extensive online helpsystem in the program itself. In addition to conceptual information and dialog descriptions, here you will also find "step by step" instructions for most program functions. The "Proceed as follows" section of the operating information gives you step by step instructions for using the program. Once you have started the program and require information on a particular dialog, you can then call up the appropriate help by simply pressing the [F1] key.

### **User's Comments**

Before you begin reading, please note the following symbols and styles used in this handbook:



Text preceded by this image contains extra Notes.



#### Tip:

After this image, useful tips to facilitate your interaction with the program are presented.



#### **Example:**

Examples are highlighted by this graphical symbol.

• User interface elements are marked in **bold** so they can immediately be located in the text.

- *Italic* text provides particularly important information that you should definitely pay attention to.
- Directory names, and direct input (among other things) are displayed in a non-proportional font.
- Function keys, keyboard keys, and buttons within the program are shown in brackets (e.g., **[F1]** for the "F1" function key).
- To improve the flow of the text, we often use "menu paths" in this
  document (for example, Help > Contents). In order to find a particular
  program function, the menus and options shown in a menu path must
  be selected in the sequence shown. For example, the menu path
  mentioned above calls up the EPLAN help system.
- The operating information provides step-by-step instructions for reaching specific targets, and in many places, after the right-arrow symbol ⇒ you'll also find the program's reaction to your action.
- In combination with settings or fields (e.g., check boxes) which can only be switched on or off, in this document we often use the term "activate" (after which the setting is active ☑) and "deactivate" (after which the setting is inactive □).

# **Remarks on the Practice Project**

In the following chapters, you will gradually create and evaluate a small practice project. So that you can successfully follow the instructions, we assume that you have already installed EPLAN and have the necessary software protection and license required.

While writing this getting started guide, we used the fully licensed program with its entire scope of functionality. If you have not licensed all the modules, this documentation (or some figures in this documentation) may possibly go beyond the scope of functionality of the software you have purchased, that is, functions are also described which are not available to you.

All settings can be made in a central location in the program (in the **Options** menu under the menu item **Settings**). For the practice project, we used the standard settings of the project template used. Don't change these settings while working with the practice project; otherwise you may see different results.

You can control EPLAN using both the mouse and the keyboard. To quickly call up program functions, you often also have additional buttons on the toolbars and/or key combinations (e.g., **[Ctrl] + [C]** to copy elements). Since a list of the different call-up options would make this guide much larger, we usually use the mouse controls in the action instructions.

# Starting EPLAN

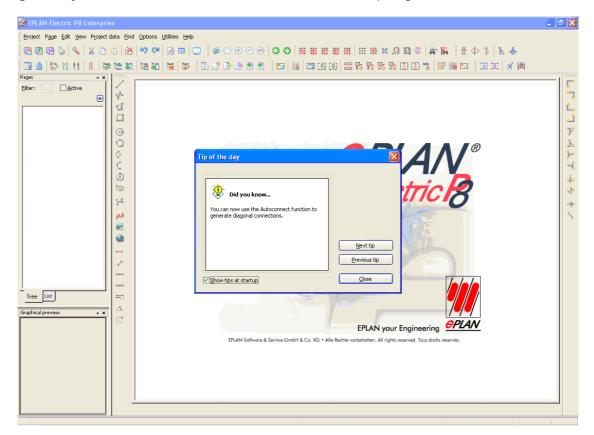
In this chapter, you will learn how to start EPLAN easily.

During program installation, a program icon is created on the Windows Desktop. You can use this icon to start EPLAN immediately.

Double-click the EPLAN icon 1.



⇒ EPLAN starts. Beside the EPLAN main window, the dialog **Tip of** the day is also shown on the screen by default. In this informational dialog, which is automatically shown every time you start the program, you are shown useful information on program functions.



- Deselect the Show tips at startup check box if you do not wish this dialog to appear every time the program is started.
- Click on [Close] to close the Tip of the day.

Before you create a project, you will first see a little more information about the user interface in the next chapter.

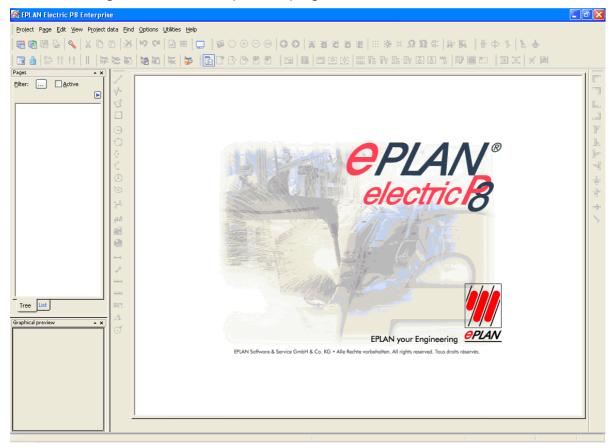
#### ■ Note:

You can leave EPLAN at any time by first closing all open dialogs with **[Esc]**, **[Cancel]**, or by clicking the mouse on **⋈** in the title bar, and then selecting the menu item **Project > Exit**.

# Getting to Know the EPLAN User Interface

In this chapter, you will learn about the most important elements of the program interface. You will also learn how to change the interface easily.

When first started, EPLAN begins with a pre-configured interface. Besides various other interface elements, you see the **Page navigator** on the left side of the main window. On the first start, this window is empty. The area on the right side with the background image will be used later as a working area for the opened pages.



#### What You Need to Know in Advance

#### What is the EPLAN main window?

The main window is the entire working area of the program; it's size and position can be changed. For many other interface elements (e.g., menu bars, toolbars, "dockable" windows like the Page navigator) the main window is more or less the "frame". You can attach ("dock") these elements to the window edges inside the main window.

### What is the Page navigator?

The **Page navigator** is a window in which EPLAN shows the pages of all open projects. You can choose between two view types. In the tree view, the pages are displayed hierarchically by page type and identifier (such as higher-level function, mounting location, etc.). In the list view, this information is shown arranged in a table. You can switch between the two views by clicking on the corresponding tab. In the Page navigator, you can edit the pages of one project, for instance, copy pages, delete them, or change page properties. Multiple pages from different projects can not be edited at one time.



#### giT 🕛

To show and hide the Page navigator quickly, you can use the [F12] key.

#### Interface Elements

#### Title bar

The horizontal bar at the top edge of a Windows application is the title bar. It contains the program icon with the system menu, the program name, and the buttons controlling the size setting of the window.





If you have a project open, then the name of the current project will also appear in the title bar, along with the name of the page opened, if applicable.



#### **Example:**

This is how the title bar looks if you have opened the page =ANL+SCP/1 in the project EPLAN start, which you will create later.

EPLAN Electric P8 Enterprise - EPLAN\_start - [=ANL+SCP/1]



#### Menu bar

The menu bar is located below the title bar. It contains the most important commands and dialog calls.



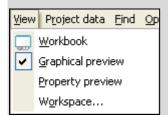
In order to show all the menu items of a menu, left-click on the corresponding menu. Until you have opened a project and a page, you will not be able to select many of the menu items. These menu items are shown in a grayed-out view.

Some menu items work like a switch, that is, they can be turned on or off. This applies particularly to the View menu for the Graphical preview menu item. If the option is switched on, then this is indicated with a preceding check mark .



#### **Example:**

The following example shows the "switched-on" menu item Graphical preview in the View menu.



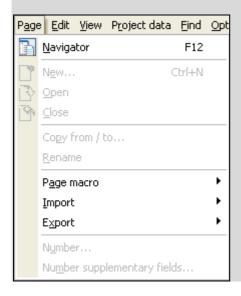


In the Graphical preview, the pages you have selected in the Page navigator are shown in a reduced view. Using this window, you can, for instance, quickly search through all the pages of a project.

For menu items to which an icon is assigned, the corresponding symbol is highlighted with a frame in the switched-on state. This is the case, for instance, for the Page navigator, which you can call in the Page menu using the **Navigator** menu item (the icon in the switched-on-state: 1).

#### 😿 Example:

The following illustration clarifies the example just explained using the "switched-on" Navigator menu item.



#### Note:

If you can call a menu item using a key combination, that combination is shown next to the menu item. You have the option of changing the keyboard layout assigned by EPLAN and assigning your own key combinations to commands. The corresponding dialog can be found under the menu item Options > Keyboard shortcuts.

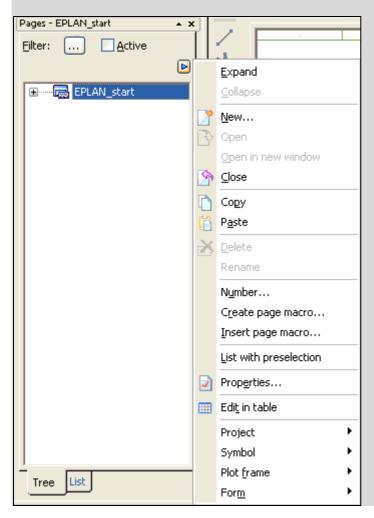
#### Popup menu

In many places in the program (in fields of windows or dialogs, in an opened page) you can open up a popup menu by right-clicking. The popup menu shows the most commonly used menu items for the object in question.

If menu items are present in the popup menu of a field which go beyond the standard functions, you can see that from a "popup menu button" . This button is located above the corresponding field.

#### 🗑 Example:

The figure shows the popup menu for the Page navigator, opened using the popup menu button.



#### **Toolbars**

As standard, the toolbars are located below the menu bar and consist of buttons allowing you to directly call up the most important EPLAN functions. Here, too, you do not initially have all the buttons available.



#### 🙀 Example:

The following figure shows you the **Standard** toolbar for an opened project.



#### Status bar

The status bar is located at the lower edge of the window. When you point the mouse at a menu item or a button in a toolbar, a brief informative text relating to the action called by this command is displayed here.

Toggle display of page navigator

If you have the cursor in an opened page, the status bar shows you information about the cursor position, grid, and logical status, along with data about the currently selected element, in some cases.



#### 🗑 Example:

In the practice project you will create later, for instance, we have opened the page =ANL+SCP/1 and positioned the cursor on the coordinates X:176 / Y:216. On: 4.00 mm means that the Grid snap option is enabled and that the grid resolution for this page is 4 mm. This page is a Logic page (page type: Schematic multi-line) on a 1:1 scale.

X: 176.00 mm Y: 216.00 mm

ON: 4.00 mm Logic 1:1

### Window size adjustment

The ability to modify the size of the EPLAN main window (and also other dialogs) is indicated by three diagonal lines in the lower right-hand corner of the window. In the maximized view of the main window, the diagonal lines are not shown. To show them, the window must be reduced in size, for instance using the **button**.



You can change the window size by positioning the mouse pointer in the frame at the edge of the window, and then pulling the window frame left / right or up / down while holding down the left mouse button.

# **Changing the User Interface**

You can change the view of the user interface to fit your own work habits. Many dialogs, e.g., Page navigator, which you often need for editing your data, can be permanently positioned on your screen while working with EPLAN and undocked from the EPLAN main window in the same manner as independent windows.

You can position any of these "Windows" (in an undocked state), and every other dialog, by pulling the title bar of the dialog to the desired position while holding down the left mouse button.

#### Docking and undocking control elements

Some user interface control elements, such as the menu bar, the toolbars or various windows like the **Page Navigator**, can be positioned inside or outside of the EPLAN main window.

- 1. In the title bar of the EPLAN main window, click the minimize button <u>a</u>l.
  - ⇒ The EPLAN main window will be minimized.
- 2. Next click the title bar of the **Page navigator** with the left mouse button and keep the button pressed.



- 3. Press [Ctrl] to avoid docking, and pull the Page navigator to a free position on the Windows Desktop.
  - ⇒ If the borders of the respective control element are displayed with a *thick* line, EPLAN positions the control element as an independent, undocked element on the Windows desktop.
- 4. Repeat this procedure for additional control elements. To do this for a toolbar or the menu bar, click on a "free" spot within the bar.

Practice docking control elements, too.

- 5. To do this, click the title bar of the desired undocked control element (e.g., the **Page navigator**) with the left mouse button and keep the button pressed.
- 6. Pull the element to the desired position in the EPLAN main window.

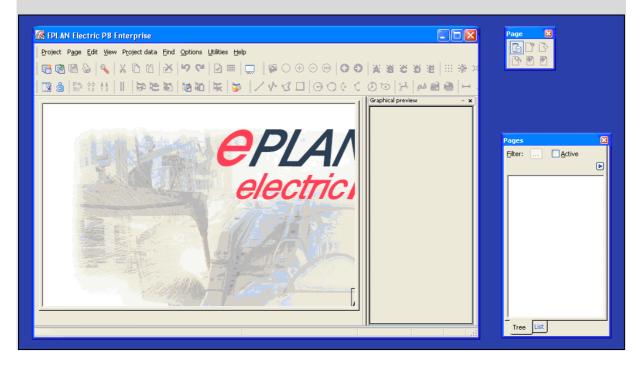
⇒ To dock the undocked element to the border of the EPLAN main window (or the border of another dockable element), you must move the mouse pointer exactly to the border of the EPLAN main window. You can recognize this by the border of the element being displayed as a *thin* line. This will be a horizontal or vertical line, depending on whether you wish to position the control element at the top, bottom, left, or right. You can recognize the orientation from the thinly drawn preview rectangle.

When the program is exited, EPLAN notes the last settings of these elements (docked, undocked, size, position) and restores them the next time the program is started.

### <del>\</del>

#### **Example:**

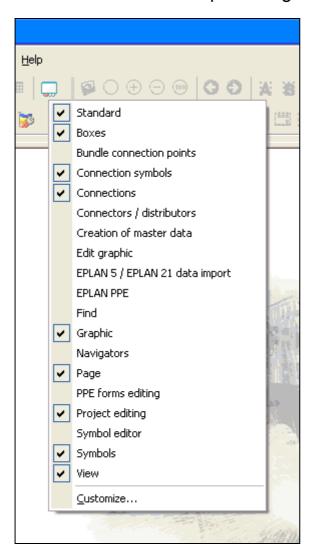
The following illustration shows the EPLAN user interface reconfigured by docking and undocking control elements. Moreover, another toolbar ("Symbols") has been hidden and two more moved around inside the main window ("Graphics" and "Connection symbols").



#### Hiding and displaying toolbars

In EPLAN, there are a number of predefined toolbars for the different program areas. To avoid unnecessarily reducing the size of your EPLAN user interface, you have the option of hiding the toolbars you don't need.

- 1. Click the right mouse button on a free area in the menu bar or toolbar.
  - $\Rightarrow$  The popup menu shows all the available toolbars. Displayed toolbars are marked with a preceding checkmark  $\checkmark$ .



- 2. Select the **Standard** toolbar, for instance, and click the check box in front of the toolbar name.
  - ⇒ EPLAN closes the popup menu and hides the toolbar.
- 3. Repeat the procedure and deactivate some more toolbars.

Practice unhiding toolbars, too. To do this, call up the popup menu again and click on the name of a hidden toolbar (e.g., Standard).



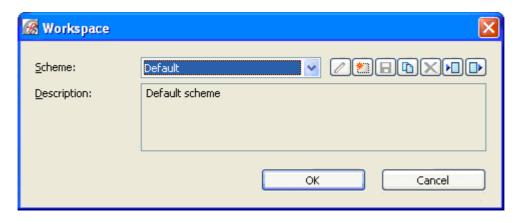
For the hiding and displaying of toolbars, you also have the **Customize** dialog available. Call this dialog, for instance, using the Customize menu item on the popup menu, or using the menu items Options > Toolbars > Customize. You can also use this dialog to customize and extend the predefined toolbars and create your own toolbars.

#### **Selecting workspaces**

Once you have changed the EPLAN user interface to suit your needs, you can save this arrangement as a "workspace." In an EPLAN workspace, the size and position of the EPLAN main window are determined, as well as the position, size, and settings of dockable dialogs, menus, and toolbars.

Using the workspace, you can quickly adapt the EPLAN interface to different tasks. We provide you with a few predefined workspaces which you can use as templates for your own workspaces.

- Select the menu item View > Workspace.
  - ⇒ The Workspace dialog opens.



- 2. From the Scheme drop-down list, select the entry Connections as an example of a workspace.
- 3. Click [OK].

The interface changes you made will disappear. Besides the Page navigator, the main window now also contains the two windows Potentials and Connections. In these special navigators, all the potentials and connections of a project are shown.

#### Restoring the original view

- 1. Select the menu item **View > Workspace**.
  - ⇒ The **Workspace** dialog opens.
- 2. From the Scheme drop-down list, select the entry Default.
- 3. Click [OK].

EPLAN now shows the view displayed when it first started up. Using this action, you can reset an EPLAN user interface that you may have changed back to the original state.

You can find more detailed information about the user interface – for instance, how you can define your own workspaces – in the EPLAN online help.

# **Creating a Project**

Now that you have learned something about the operation of the user interface in the last chapter, you will learn how to create a project quickly and easily using the **Project Wizard**.

# What you Need to Know in Advance

### What is a project?

In EPLAN, schematics and attached documents such as lists and overviews are created as pages within projects. A project consists of a collection of different types of documents. Projects are stored and organized in a special database – the "project management".

### What is a project structure?

In EPLAN, "Project structure" means the combination of all identifier structures used in the project for objects, pages, devices, and functions. All objects in a project (pages, devices, and functions) must be identified and placed in a hierarchical structure within the project. In a hierarchically arranged project structure, for instance, you can store and find pages and devices more easily. The identifiers for project structure are called "structure identifiers".

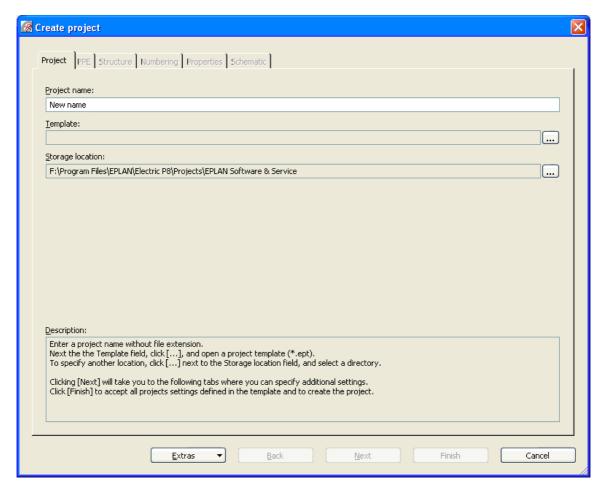
### What is a project template?

In order to create a new project, you always need a template. When using the project template, you create a project in which some settings are already predefined. For instance, the structure of page identifiers and device tags are stored in a project template. The file extension for project templates is  $\star .ept$ .

# Creating a Project Using the Project Wizard

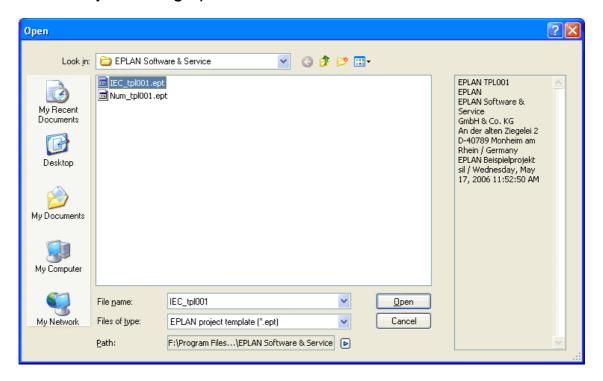
EPLAN provides several options for creating a project. Using the **Project wizard**, even as a beginner you can very quickly and simply create a project containing a cover sheet and a schematic page. In the dialog of the wizard – the **Create project** dialog – you can enter all the principal data for a new project step by step.

- Select the menu item Project > New (Wizard).
  - ⇒ The Create project dialog opens.

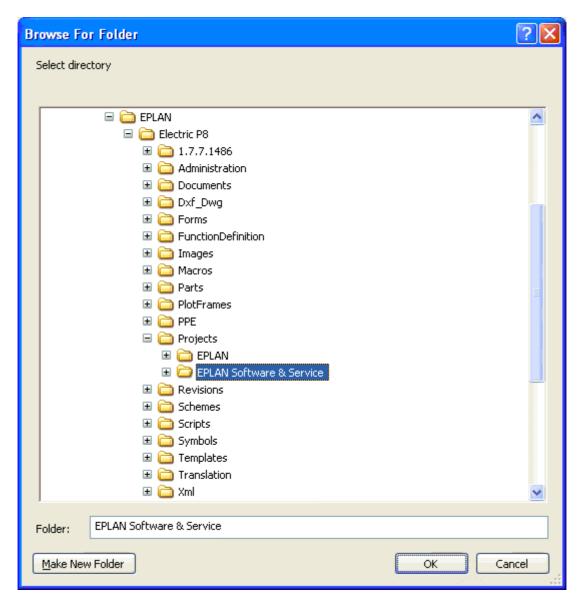


- 2. In the **Project name** field on the first tab, enter the name for your first project. Name the project EPLAN start.
- 3. Select a **Template** for the project. To do this, click the **[...]** button next to the field of the same name. If you don't select a project template here, you will not be able to perform any more steps in the Project wizard.

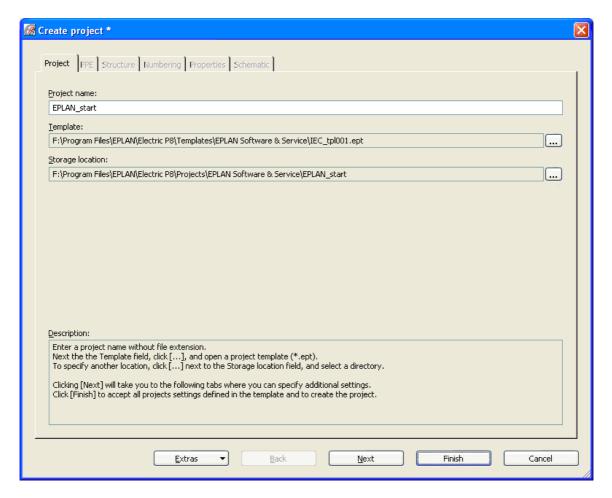
 $\Rightarrow$  The **Open** dialog opens.



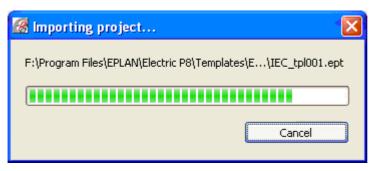
- 4. In the Open dialog, select the project template IEC\_tpl001.ept.
- 5. Click [Open].
  - ⇒ The **Open** dialog is closed and the name for the project template is accepted.
- 6. Select a **storage location** for the project. To do this, click the [...] button next to the field of the same name.
  - ⇒ The **Browse For Folder** dialog opens.



- 7. By default, your projects are managed in the **Projects** directory in a "company folder" created during installation (**Company code**); in the illustration, this is the EPLAN Software & Service folder. Select this "company folder" and click on [Make New Folder].
  - ⇒ A new folder is created below the currently selected "company folder", whose temporary name New Folder is highlighted with an edit frame.
- 8. Here, enter the name EPLAN\_start and click [OK].
  - ⇒ The **Browse For Folder** dialog is closed and you are returned to the Project wizard.



9. Click [Finish].



- The **Importing project** dialog opens. The Project wizard copies the template into the new project. This procedure may take some time. Then, the **Create project** dialog closes and the project EPLAN\_start is displayed in the **Page navigator**. The projects form the first level in the tree.
- 10. Click on the plus sign 

  in front of the project name in the tree view.
  - ⇒ The tree is opened. In the level below the EPLAN\_start project, the pages are now shown.



You have just created a project using the **Project wizard**, which automatically has two pages.

The selected project template enables hierarchically arranged structures for all objects in the project. It has a project structure already determined. Structure identifiers for the "higher-level function" and the "mounting location" are used to identify pages and devices.

In the next chapter, you will learn how to assign structure identifiers to the automatically generated pages. There, you will also read how to open and create pages.

#### ■ Note:

Once you are more familiar with EPLAN, you can use the different tabs of the **Project wizard** to change the predefined settings from the project template regarding structure, numbering, and properties. Using the **Schematic** tab, you can also determine structure identifiers for the first schematic page.

# **Opening and Editing Pages**

In the previous chapter, you used the Project wizard to create a project with two pages. In this chapter, you will learn how to open pages. You will also edit the page properties of the existing pages and create a new page.

In EPLAN, every page is assigned to a particular *page type*. A project generally consists of pages of different types. The page types help structure a project. As you can already see from the different icons in the **Page navigator**, the two pages of your project EPLAN\_start have different page types. Page 1 is a Title page / cover sheet page type, and page 2 is a Schematic multi-line page type. The page type is generally assigned when the page is created, but it can also be changed later.



#### What You need to Know in Advance

#### What is a title page or cover sheet?

A page with the "Title page / cover sheet" page type contains general information on the project, such as the project name, a description of the project, the drawing number, revision information, and so on. The title page or cover sheet is usually the first page of a project.

### What is a multi-line schematic page?

A multi-line schematic page is a page for which the schematic documentation has multi-line symbols. It has special editing capabilities for the creation and analysis of schematic diagrams. For instance, symbols inserted are automatically connected with one another.

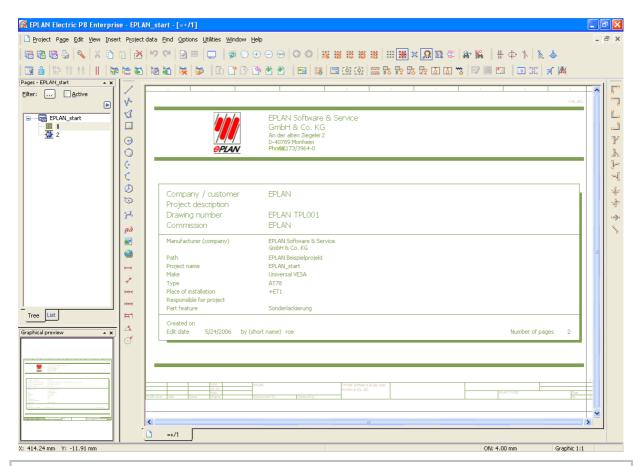
#### What is a graphical editor?

A graphical editor is the part of the program you use to create and edit schematics and mechanical drawings. It is started when you open a page. Besides the window in which the page is displayed, the menu bar also has numerous program functions (such as the **Insert** menu) available for graphical editing.

The window is dockable and has its own standard Windows buttons. In a "maximized" view, the window of the editor fits into the main window in such a way that the title bar is integrated into the menu bar of the main window (see also the next illustration). In the "minimized" view, the window has its own title bar.

# **Opening Pages**

- 1. In the tree view of the **Page navigator**, select page 1.
- 2. Select the menu item Page > Open.
  - ⇒ EPLAN opens the page in the **graphical editor**. The cover sheet of the project is shown. The page name is displayed not only in the title bar of the main window, but also as a tab under the graphic. In the **Page navigator**, you can recognize the opened page from the fact that the page name 1 is now in bold.



# 🕕 Tip:

To open a page selected in the **Page navigator**, you can also double-click on the page or press the **[Enter]** key.

Open the page 2 in the same way. This will automatically close the first page and display the empty schematic in the **graphical editor**.

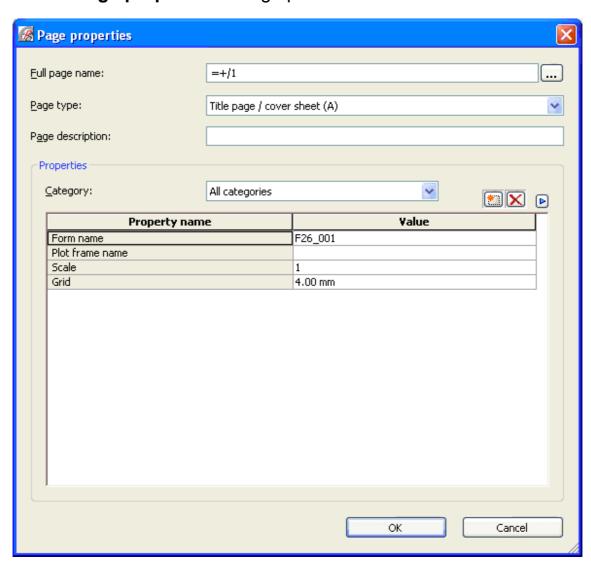
## Tip:

In EPLAN, you also have the option of viewing multiple pages on the screen at one time. For instance, select page 1 again, then from the popup menu of the **Page navigator** select the menu item **Open in new window**. EPLAN opens the page in another window of the **graphical editor**. For every opened page, a tab is shown below the graphical editor window. You can quickly switch between pages by clicking these tabs.

# **Editing Page Properties**

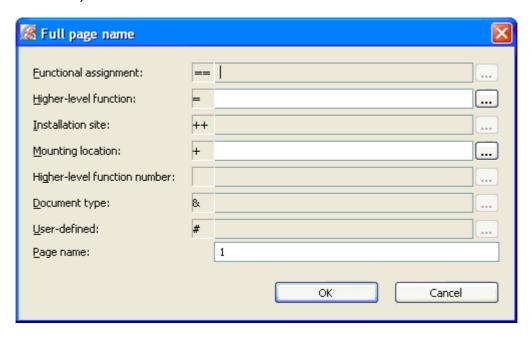
For the practice project, you selected a project template with predefined identifier structures. The pages created, however, don't yet have structure identifiers. In the following section, we describe how you can assign the identifier by editing the page properties.

- 1. In the tree view of the **Page navigator**, select the cover sheet (page 1).
- Then select the menu item Edit > Properties.
  - ⇒ The Page properties dialog opens.

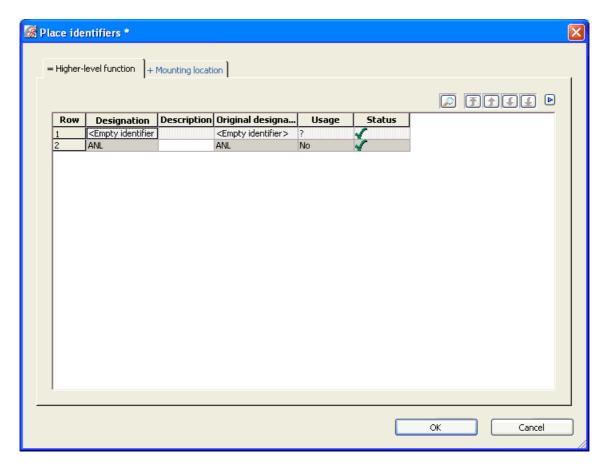


3. Change the value in the **Full page name** field. To do this, click the [...] button next to the field of the same name.

⇒ EPLAN opens the **Full page name** dialog. Here, you can enter the structure identifier and the name of the page (e.g., the page number).



- 4. Enter the value ANL into the **Higher-level function** field.
- 5. Enter the value DBT into the **Mounting location** field.
- 6. Click **[OK]**.
  - ⇒ The **Full page name** dialog is closed. The structure identifiers are accepted for the field of the same name in the **Page properties** dialog.
- 7. In the Page description field, enter the text Cover sheet.
- 8. Click [OK].
  - ⇒ The **Place identifiers** dialog opens. Using this dialog, you can determine the order of newly created structure identifiers within a project. At this point, however, you do not need to worry about this dialog.



#### 9. Click [OK].

⇒ The **Place identifiers** dialog is closed. In the **Page navigator**, the cover sheet now shows the structure identifier entered (=ANL+DBT/1).

Repeat this action for the schematic page (page 2). In the **Full page name** dialog, as the structure identifier for the **Higher-level function**, enter the designation ANL again. However, as the **Mounting location** for the schematic page, enter the designation SCP and change the entry in the **Page name** field from 2 to 1.

For the **Page description** for this page, the **Page properties** must still have the text Main circuit entered, and finally the position of the new structure identifier SCP must be confirmed with **[OK]**.

Now all pages in the project have structure identifiers. To display all pages in the **Page navigator**, click the plus sign ℍ in the tree in front of the identifiers ANL, DBT, and SCP. You may first have to enlarge the **Page navigator** window a little before you can read all the page descriptions.



#### **I**S Note:

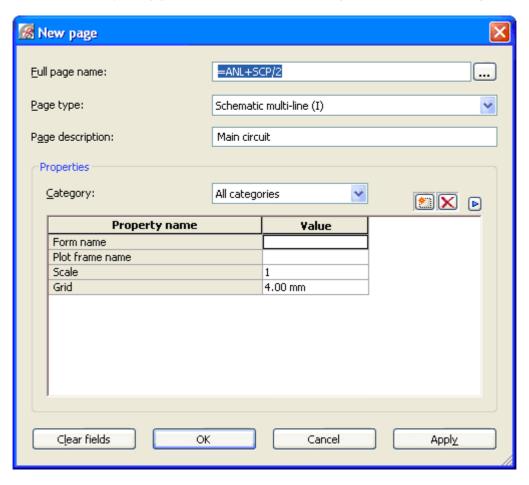
If you have made a change during page editing (for instance, changed page properties, inserted graphical elements, etc.), then EPLAN **saves** these change immediately. There is no separate save necessary, and so there is no corresponding menu item in the program!

# **Creating a Page**

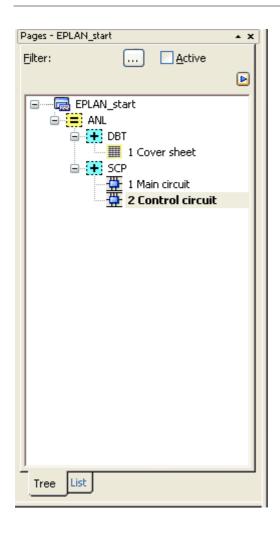
For the schematic editing in a later chapter, you will need another page of the Schematic multi-line type.

- 1. In the tree view of the **Page navigator**, select the schematic page =ANL+SCP/1.
- 2. Select the menu item **Page > New**.

⇒ EPLAN opens the **New page** dialog. Most property fields already have values. EPLAN accepts the properties of the selected page and automatically suggests the next free page for the **full page name**.



- 3. Click on [Clear fields].
  - ⇒ The contents of most property fields are deleted. The automatically suggested **Page name** and the prefilled **Page type** are kept.
- 4. In the Page description field, enter the text Control circuit.
- 5. Click [OK].
  - ⇒ The new page is shown in the tree view of the **Page navigator** and opened in the **graphical editor**.



# Closing a page

Since you don't immediately need the second schematic page, close it again.

- 1. Select the menu item **Page > Close**.
  - ⇒ The page =ANL+SCP/2 is closed.

### Tip:

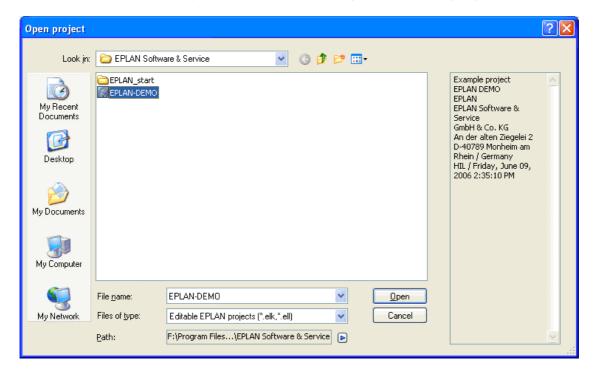
If you have the cursor in an opened page, then you can also close the page using the key combination **[Ctrl] + [F4]**.

# **Opening and Closing a Project**

Before you start with the actual creation of a schematic, you will first look at a predefined demonstration project in this chapter. You will also learn how to open and close a project, and you will learn a few basic commands for viewing and editing a project.

# **Opening the Project**

- 1. Select the menu item **Project > Open**.
  - ⇒ The **Open project** dialog opens. Here, the projects are shown which are located in your predefined project directory by default.

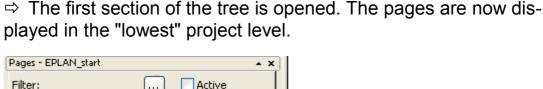


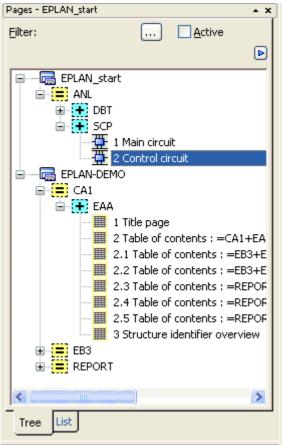
- 2. Select the project EPLAN-DEMO.elk.
- 3. Click [Open].
  - ⇒ The page description is displayed in the Page navigator.
- 4. Click on the plus sign 

  in front of the project name EPLAN-DEMO in the tree view. Then repeat this for the plus sign 

  in front of the structure level CA1, then for the 

  next to the structure level EAA.





- 5. Now open the first page (=CA1+EAA/1) of the demo project, for instance by selecting the tree view and then double-clicking.
  - ⇒ The cover sheet of the project EPLAN-DEMO is shown in the graphical editor.

First, page through the pages of this extensive project.

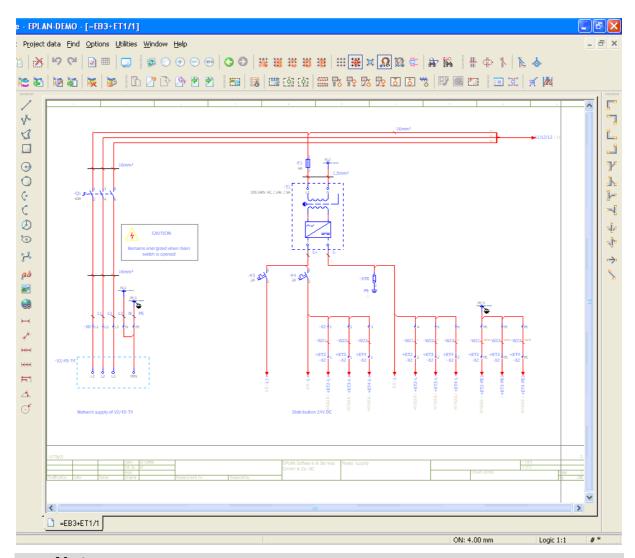


To open and close structure levels with subordinate levels, from the **Page navigator** you can also use the two menu items **Open** and **Close** from the popup menu. For instance, if you want to show all the pages of a project in the tree at once, then select the (closed) level with the project name and select the menu item **Popup menu > Open** (or press the [\*] key on the numeric block of your keyboard).

# **Paging Through the Project Pages**

Using the appropriate program functions, you can look at the pages of a project one after the other in the **graphical editor**.

- 1. Select the menu item **Page > Next page** to page to the next page of the EPLAN-DEMO project.
  - ⇒ EPLAN shows you the next page in the window of the **graphical** editor.
- 2. Repeat this action a few times and look at the contents of the pages in the **graphical editor**.
  - ⇒ The pages are shown one after another in the **graphical editor**. If the tree structure in the **Page navigator** was not yet completely opened, this will be done while paging through the pages. The page name and the description of the currently opened page are highlighted in bold in the **Page navigator**.
- 3. Page back a few pages, too. To do this, select the menu item **Page > Previous page**.
- 4. Finally, finish paging and select the first multi-line schematic page (=EB3+ET1/1 with the description Power supply) of the demo project.



#### ■ Note:

To page through the project pages, you can also use the keys **[PgDn]** / **[PgUp]** or the buttons in the **Page** toolbar. However, if your cursor is located in the **Page navigator**, the keys will not page through the pages but will cause the cursor to jump to the bottom or top page shown in the tree view.

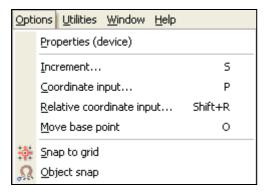
# **Creating a View**

Using the page =EB3+ET1/1 now opened in the demo project, we would like to introduce you to a few functions which will be useful later when you create schematics.

#### Turning on the grid display

A glance at the two menu items **View** and **Options** will show you that in EPLAN there are numerous program functions which make it easier for you to draw schematics.

Both for the demo project and for your practice project, the **Options** menu has the menu items **Snap to grid** and **Object snap** enabled by default.



These two options make it possible to orient elements directly to the grid points or element points when drawing. Please leave these options activated while creating the following schematic. The size of the grid used is stored as a page property and shown in the status bar.

1. Select the menu item **View > Grid**, to switch the grid display on.

The use of grid snap is independent of the grid display options. If this view bothers you later when creating the schematic, then turn the grid display off in the same way.

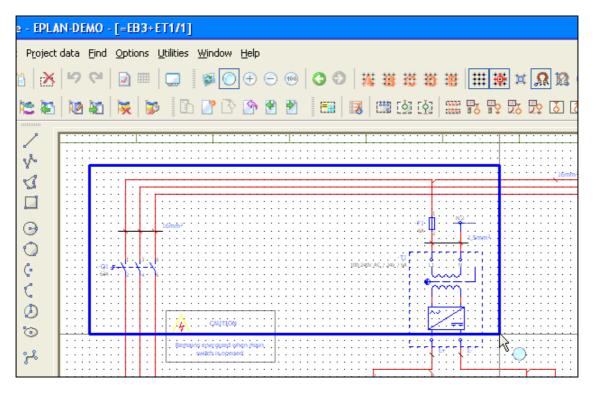
#### **Using window zoom**

Normally, the entire page is shown in a window of the **graphical editor**. For schematic editing, it is often a good idea to enlarge a certain screen segment in order to be able to see details better. For instance, you can use the "window zoom" in order to show a certain window segment in an enlarged view.

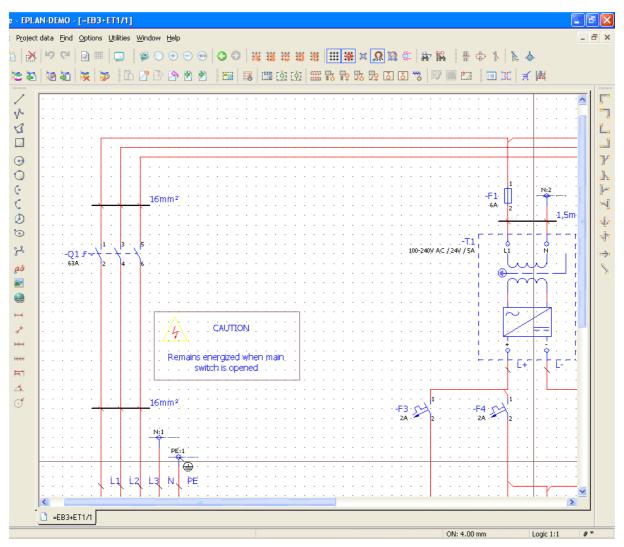
- 1. Select the menu item **View > Zoom > Window**, or press the **[Z]** key.
  - ⇒ In addition to the mouse pointer, a magnifying glass will also be displayed.



- 2. Left-click the mouse in the upper left corner of the schematic, and use the mouse to open an area which includes about a quarter of the schematic page.
  - ⇒ The area to be enlarged is delineated with a thick frame while it is being selected.



- 3. Click the left mouse button again.
  - ⇒ The selected area is displayed at maximum size.



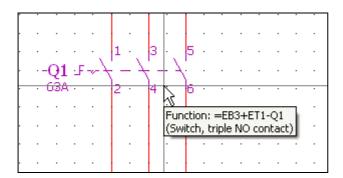
If the entire page should be shown, simply press the key [3] (menu path: View > Zoom > 100%).

# Tip:

You can also use an optional mouse wheel to zoom in EPLAN. By default, during scrolling the mouse wheel acts in the same way you are familiar with from other CAD programs. If you scroll up / down with the mouse wheel, the page is enlarged / reduced. In the settings, you also have the option of changing the scrolling behavior so that it matches the behavior of text editing programs. In this case, when you scroll with the mouse wheel, the visible section of the page is moved upwards or downwards.

#### **Deleting and restoring elements**

- 1. Move the mouse pointer onto the element with the designation -Q1.
  - ⇒ The element is displayed in a different color. Below the symbol, a small text window (**Tooltip**) is displayed with information about the element.



- 2. Select the element by clicking on it.
  - ⇒ The color of the element changes again.
- 3. Select the menu item **Edit > Delete** (or press **[Delete]**).

The element -Q1 is completely deleted. You can, however, use the **Undo** command to reinsert the element.

4. Then select the menu item **Edit > Undo**.

#### ■ Note:

When editing elements in the **graphical editor**, besides the usual Windows operating methods (first select elements, then call command), in most cases the reverse order of operation (first call the command, then select elements) is also supported.

Practice this once for the element -Q1 you just inserted.

- 1. Select the menu item **Edit > Delete**.
  - $\Rightarrow$  In addition to the mouse pointer, two other symbols are also displayed. The box with the red X is the symbol assigned to the delete function, and the box with the dashed line stands for the area to be selected



- 2. Left-click the mouse button on the upper left of the element, and use the mouse to drag a frame around -Q1.
  - ⇒ The element -Q1 is removed.
- 3. This time, press the key combination [Ctrl] + [Z] to undo the **Delete** command again and to reinsert the element.

# **Closing the Project**

- 1. Select the menu item **Project > Close**.
  - ⇒ The EPLAN-DEMO project is closed.

If your cursor is in the **Page navigator**, then the project for which you have selected one or more pages is closed.

#### ■ Note:

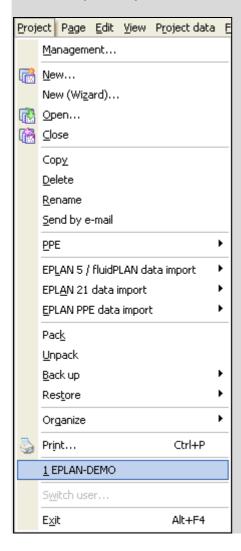
Please note that the menu item **Close** in the popup menu of the page navigator only closes the page currently located in the foreground. If you want to close a project using the popup menu of the page navigator, then you should select the menu item **Popup menu > Project > Close**.

# 🕕 Tip:

In the **Project** menu, beneath the menu item **Print**, a list of the last projects opened is displayed. You can use this list to open a corresponding project without needing to use the menu item **Open**.

# 💡 Example:

You have just closed the demo project. Now click on the **Project** menu and select the EPLAN-DEMO entry. EPLAN opens the demo project again. After it opens, practice closing it again.



Now that you have looked at the complicated demo project in this chapter, in the next two main sections you will create a much "simpler" schematic.

# **Creating a Schematic**

In this chapter, you will begin with the creation of a simple schematic. To do this, first open – if necessary – the schematic page =ANL+SCP/1 in your practice project, with the description **Main circuit**.

# What you Need to Know in Advance

### What is a symbol?

A symbol is a graphic which is used in EPLAN to show functions. It does not contain any logical data. These are stored in the function.

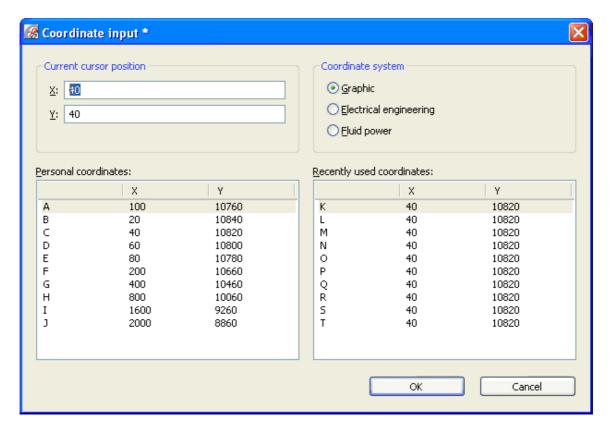
## What is a component?

A component is a graphical element for the representation of a function. It consists of a function and a symbol. The function contains the logical data and the symbol contains the graphical data. A component has a device tag, connection point designations, and so on.

# **Inserting "Angle" Connection Symbols**

As the first elements, insert a few angles into your schematic. Angles and other elements (T-nodes, interruption points, etc.) are considered in EPLAN to be connection symbols, used in the schematic to show the paths of connections.

- Select the menu item Insert > Connection symbol > Angle (down, right).
  - ⇒ The angle hangs on the cursor.
- 2. Select **Options > Coordinate input**.
  - ⇒ The Coordinate input dialog opens.



- 3. In the Current cursor position group box, for X: enter 40 and for Y: enter 248.
- 4. Click [OK].
  - ⇒ The **Coordinate input** dialog is closed. Along with the cursor, EPLAN places the angle at the given position. The current cursor position is also shown in the bottom left corner of the status bar.
- 5. You can immediately insert more angles. To do this, call **Coordinate** input again, as shown above, and place more **Angles** (down, right) at the positions X:48 / Y:244 and X:56 / Y:240.
- 6. Then select Popup menu > Cancel action to terminate the action.

#### ■ Note:

In the entire practice project, we place the different schematic elements into precisely defined positions to make it possible to compare the results. To do this – as described above – you can use Coordinate input or position the elements at the given coordinates using the mouse or with the cursor keys. If you use the Coordinate input dialog to do so, each element is directly placed after confirming the coordinates with [OK]. If you position the element using the mouse or the keyboard, then after positioning you must still left-click the mouse or press [Enter] in order to place the element on the page. This kind of determination of insertion coordinates is not absolutely necessary when editing your own schematics.



#### Tip:

To call the Coordinate input quickly, you can also press the [P] key.

## Generating connection lines automatically

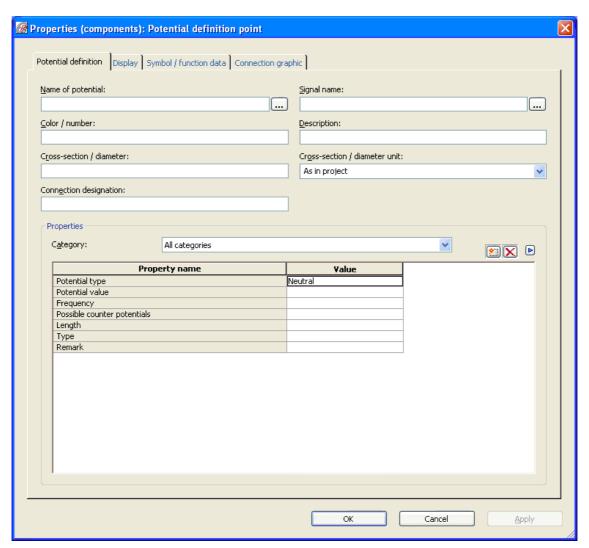
Repeat the action "Insert angle" for the symbol Angle down, right (menu path: Insert > Connection symbol > Angle (down, right)) and insert three more angles at the coordinates X:336 / Y:248, X:344 / Y:244, and X:352/Y:240.

Once the connection points of two angle symbols are directly horizontally (or vertically) aligned in the schematic, they are automatically connected with a connection line. This is called *autoconnecting*, and the connection lines created this way are called autoconnect lines. Only these autoconnect lines are recognized and reported as electrical connections between symbols. Autoconnecting is always performed if the connection points of two symbols are aligned horizontally or vertically.

# **Inserting Potential Connection Points**

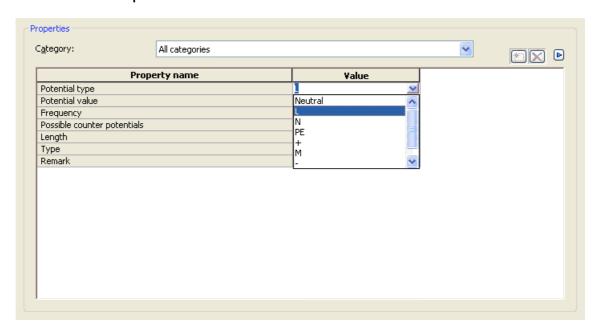
The potentials in the schematic can be defined by potential connection points.

- Select the menu item Insert > Potential connection point.
  - ⇒ The symbol for the potential connection point hangs on the mouse pointer.
- 2. Position the first potential connection point at the coordinates X:40 / Y:96 and click the left mouse button to place it.
  - ⇒ The Properties (components): Potential definition point dialog opens.



3. On the **Potential definition** tab in the properties dialog, for the **Name of potential**, enter the value L1.

4. In the **Properties** group box for the **Potential type**, select the value L from the dropdown list.

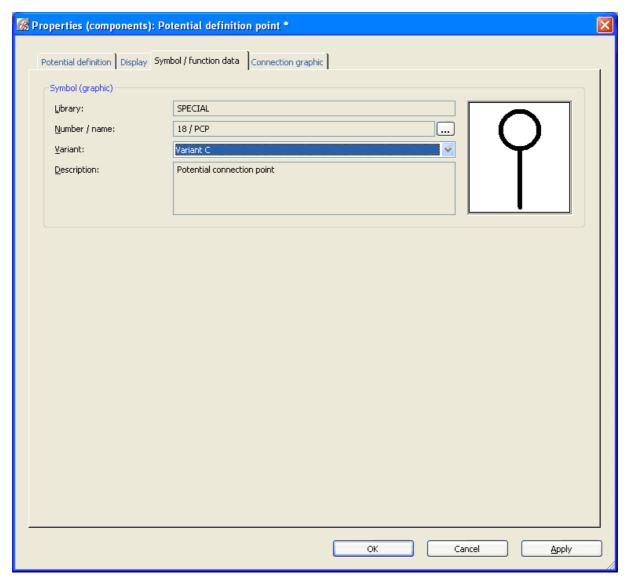


#### 5. Click [OK].

⇒ The properties dialog is then closed.

You can immediately insert more potential connection points. First position the connection points L2, L3 (both with potential type L) at the coordinates X:48/Y:96 and X:56/Y:96 and the connection point PE/N (with potential type Neutral) at the coordinates X:64/Y:96.

Then place two more potential connection points PE and N (with potential types of PE and N) in the schematic at the coordinates X:64/Y:148 and X:76/Y:136. After the symbols are placed, a different symbol variant must be selected in the properties dialog. To do this, click on the **Symbol / function data** tab in the properties dialog and, in the **Variants** field, select variant C.



Finish this action by selecting the menu item **Cancel action** from the popup menu or pressing the **[Esc]** key.

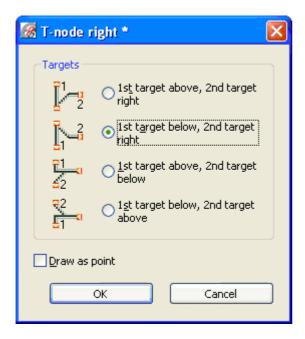
# 🕕 Tip:

If there are different variants of a symbol (such as for a potential connection point), then you already have several possibilities during positioning to select between the different variants. For instance, if the symbol is hanging on the mouse pointer, you can press the **[Ctrl]** key and move the cursor in a circle. The various variants are displayed. Once you have decided on a variant, first release the **[Ctrl]** key, then click the left mouse button to place the symbol. A different way to page between the variants available consists of pressing the **[Tab]** key while inserting.

# **Inserting "T-node" Connection Symbols**

There are T-nodes for four different directions. For each direction, there are then four variants.

- Select the menu item Insert > Connection symbol > T-node (right).
  - ⇒ The T-node hangs on the mouse pointer
- 2. Position the T-node at the coordinates X:64 / Y:120 and click the left mouse button to place it.
- 3. End the action using **Popup menu > Cancel action**.
- 4. Then select the menu item **View > Insertion points**.
  - ⇒ The insertion points of all placed schematic elements are shown in the schematic as "black squares".
- Double-click on the insertion point of the T-node.
  - ⇒ The T-node right dialog opens.
- 6. Here, select the option 1st target below, 2nd target right.



7. Click [OK].

Repeat this action for the connection symbol **T-node down** (menu path: Insert > Connection symbol > T-node (down)), and insert two more T-nodes at the coordinates X:176 / Y:248 and X:192 / Y:244. No variant selection is necessary.

Insert another Angle up, left as well (menu path: Insert > Connection **symbol > Angle (up, left))**, and place it at the coordinates X:76 / Y:120.

If you do not want the insertion points to be shown, then select **View > Insertion points** again.



## Tip:

For fast display and hiding of the insertion points, you can also use the [ i ] key.

# **Inserting Graphical Elements**

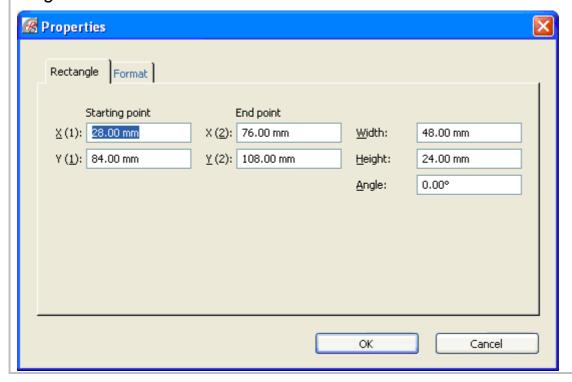
Graphical elements are elements like lines, circles, rectangles, and so on, which contain no information relevant to the schematic. To draw these elements, you have several commands available in the Insert > Graphic menu.

#### **Drawing rectangles**

- 1. Select the menu item Insert > Graphic > Rectangle.
  - ⇒ The rectangle hangs on the mouse pointer.
- 2. Position the cursor at the coordinates x:28 / y:84.
- Click the left mouse button to determine the starting point of the rec-3. tangle.
- Then move the cursor to the coordinates X:76 / Y:108 and click the left mouse button to place the end point of the rectangle.
- 5. Then select **Popup menu > Cancel action** to terminate the action.



You can also determine the exact data for the rectangle after drawing it. To do this, double-click on a line of the rectangle. In the properties dialog which appears, enter the **Starting point** and the **End point** of the rectangle into the fields of the same name.

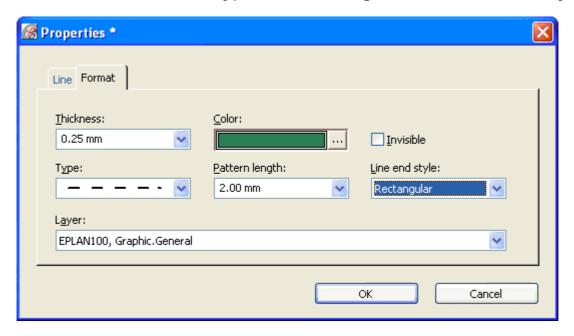


## **Drawing lines**

- 1. Select the menu item Insert > Graphic > Line.
  - ⇒ The symbol for the line hangs on the mouse pointer.
- 2. Position the cursor at the coordinates X:60 / Y:148.
- 3. Click the left mouse button to determine the starting point of the *first line* of the rectangle.
- 4. Then move the cursor to the coordinates X:72 / Y:148 and click the left mouse button to place the end point of the first line.
- 5. Repeat this action and position the starting point of the second line at the coordinates X:72 / Y:136. Place the end point of the second line at the coordinates X:84 / Y:136.
- 6. Then select **Popup menu > Cancel action** to terminate the action.

To "visually" highlight the action line, change the formatting properties of the lines inserted.

- 7. Double-click, for instance, on the "PE" line (the first line).
  - ⇒ The **Properties** dialog opens.
- 8. In the **Properties** dialog, switch to the **Format** tab.
- 9. Change the settings here. For instance, select a different Line thickness, Color, Line type, Pattern length, and/or Line end style.



## 10. Click [OK].

Repeat this action for the second action line. To differentiate the lines, select different formatting properties (e.g., a different color or a different line type).

# **Inserting General Devices**

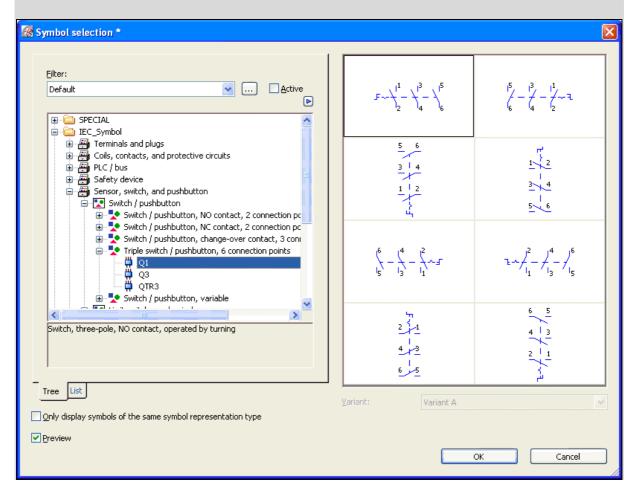
Unlike other symbols (such as the connection symbols), general devices can only be selected via symbol selection.

#### Selecting a symbol

In **Symbol selection**, you can select symbols from a tree or list view. In the **tree view**, symbols are divided neatly into different groups based on their function definitions. Here, you can page through the different groups until you find the symbol you want.

# 💡 Example:

The following illustration shows symbol selection through the tree view for the first device to be inserted.



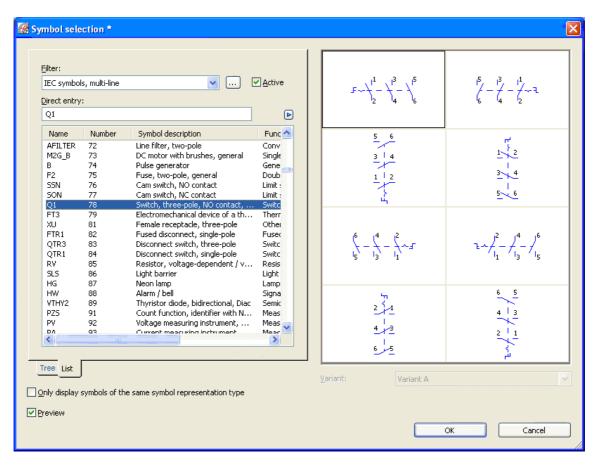
If you select a symbol in the tree, below the tree a **symbol description** of that symbol is displayed. On the right side in the dialog, the **symbol selection** presents you with a **preview**. Once you have selected the desired symbol in the **preview**, you can then accept it with a double-click.

For the practice project, we are using a symbol selection from the **list view**. You can also select a symbol very quickly using **direct entry** in the list view.

#### ■ Note:

The symbol selection via **direct entry** described below works only if you leave the columns of the list configured so that the **name** of the symbol is in the first column.

- Select the menu item Insert > Symbol.
  - ⇒ The **Symbol selection** dialog is opened.
- 2. In the **Symbol selection** dialog, select the **List** tab.
- 3. To reduce the number of symbols shown, select a filter now. To do this, from the Filter drop-down list, apply the IEC symbols, multiline entry.
- 4. Select the **Active** check box to activate the filter settings.
  - ⇒ The various symbols are managed in "symbol libraries". Now the list only shows the multi-line symbols from the IEC\_Symbol symbol library.
- 5. Enter Q1 into the **Direct entry** field.
  - After you enter the first character, the cursor jumps to the first symbol whose **name** begins with that character, and selects it. The selected symbol is displayed in the preview window on the right side, with all existing variants. The first variant of each is selected in the preview window.



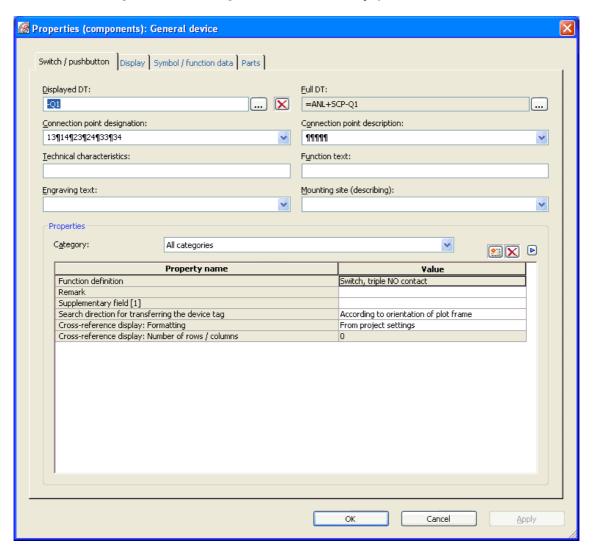
- In the list, select the three-pole switch Q1 and click [OK].
  - ⇒ The symbol hangs on the mouse pointer and can be placed on the current page in the **graphical editor**.
- 7. Position the symbol at the coordinates X:40 / Y:196 in the schematic and then click the left mouse button.

#### **Determining properties**

After placing the switch on the page, the **Properties (components)** dialog opens. Some fields are already prefilled with entries. In EPLAN, devices are automatically numbered by default when inserted. This behavior is called *online numbering*. In the settings, you can deactivate online numbering and also change the predefined numbering format.

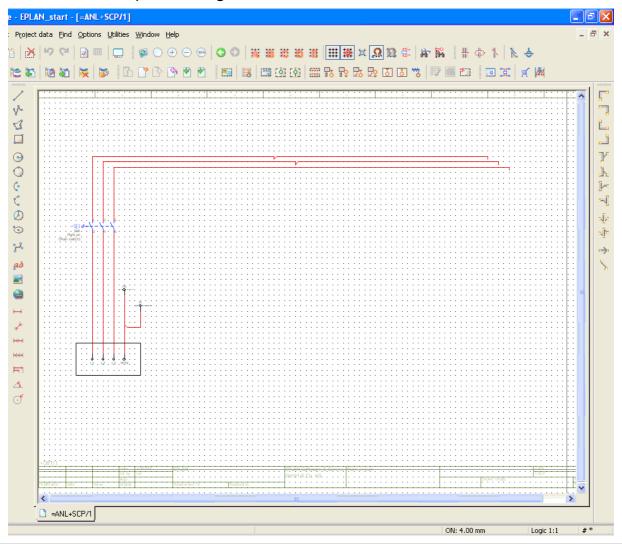
1. In the **Properties (components): General device** dialog, switch to the **Switch / pushbutton** tab.

⇒ The fields **Displayed DT**, **Connection point designation**, and **Connection point description** are already prefilled with entries.



- 2. As the new Connection point designation, enter 192934956.
  - ⇒ To insert the character ¶, which is used here to separate the individual connection points, press the key combination [Ctrl] + [Enter]. You can also select the menu item Line break from the popup menu.
- 3. Enter the value 16A into the **Technical characteristics** field.
- 4. In the Function text field, enter the text Plant on \( \text{(Main switch)} \).
- 5. Click **[OK]**.
- 6. Then select **Popup menu > Cancel action** to terminate the action.

You have now inserted a switch with three NO contacts into the schematic. -Q1 is shown as the displayed DT. In the schematic, you also see the connection point designations entered and the function text.



# Tip:

To call **symbol selection** quickly, you can also use the **[INSERT]** key or the **[III]** button in the **Symbols** toolbar.

## Inserting other general devices

Proceed in the same fashion as above for the components listed below. In the list view, select **symbol selection** using **direct entry**. To do this, enter the symbol name given below for each one, and leave the filter on multi-line IEC symbols activated. Symbols of the same type can be inserted immediately one after the other. Before you can then select a different symbol, you must finish the action and then call **symbol selection** again.

Position the symbols on the given coordinates and fill in the properties of the components inserted, if necessary. The entries of the prefilled properties are given in parentheses in the following listing.

| No. | Symbol description                                    | Entries           |
|-----|---|-------------------|
| 1.  | Fuse, single-pole, general                            |                   |
|     | Symbol name for direct entry:                         | F1                |
|     | Coordinates:  | X:176/Y:220       |
|     | Displayed DT:   | (-F1)             |
|     | Connection point designation:                         | (1¶2)             |
|     | Technical characteristics:                            | 6A                |
| 2.  | Fuse, single-pole, general                            |                   |
|     | Symbol name for direct entry:                         | F1                |
|     | Coordinates:  | X:192/Y:220       |
|     | Displayed DT:   | (-F2)             |
|     | Connection point designation:                         | (1¶2)             |
|     | Technical characteristics:                            | 6A                |
| 3.  | Single-phase transformer with two windings and shield |                   |
|     | Symbol name for direct entry:                         | T11               |
|     | Coordinates:  | X:176/Y:196       |
|     | Displayed DT:   | (-T1)             |
|     | Connection point designation:                         | 400V¶24V¶0V¶0V¶PE |

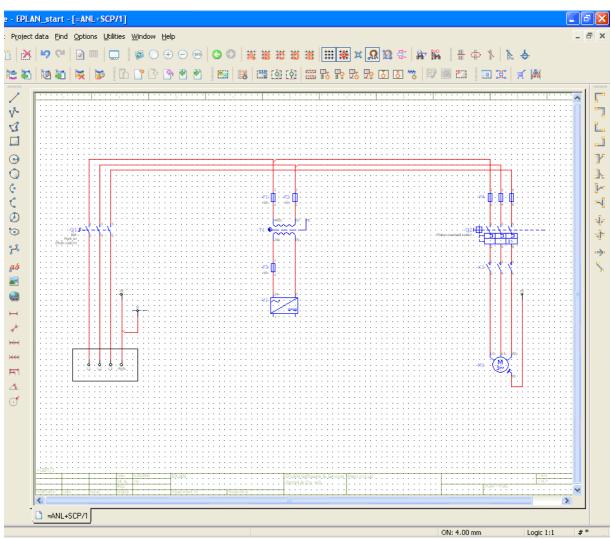
| No. | Symbol description   | Entries                   |
|-----|--|---------------------------|
| 4.  | Fuse, single-pole, general   |                           |
|     | Symbol name for direct entry:  | F1                        |
|     | Coordinates:   | X:176/Y:168               |
|     | Displayed DT:  | (-F3)                     |
|     | Connection point designation:  | (1¶2)                     |
|     | Technical characteristics:   | 6A                        |
| 5.  | Rectifier, two-phase bridge, secondary 2 connection points                           |                           |
|     | Symbol name for direct entry:  | G22                       |
|     | Coordinates:   | X:176/Y:140               |
|     | Displayed DT:  | (-V1)                     |
|     | Connection point designation:  | 24¶+¶0¶-                  |
| 6.  | Fuse, three-pole, general  |                           |
|     | Symbol name for direct entry:  | F3                        |
|     | Coordinates:   | X:336/Y:220               |
|     | Displayed DT:  | (-F4)                     |
|     | Connection point designation:  | $(1 \ 2 \ 3 \ 4 \ 5 \ 6)$ |
| 7.  | Power circuit breaker / motor over-<br>load switch with switch mechanism<br>and line |                           |
|     | Symbol name for direct entry:  | QL3_1                     |
|     | Coordinates:   | X:336/Y:196               |
|     | Displayed DT:  | (-Q2)                     |
|     | Connection point designation:  | (1¶2¶3¶4¶5¶6)             |
|     | Function text:   | Motor overload switch     |
| 8.  | Power NO contact of a contactor  |                           |
|     | Symbol name for direct entry:  | SL                        |
|     | Coordinates:   | X:336/Y:168               |
|     | Displayed DT:  | -K1                       |
|     | Connection point designation:  | (1¶2)                     |
|     |  |                           |

For the other NO power contacts you have to insert, remove the suggested **displayed DT** -?K1, and don't enter any device tag there. For projects compliant with IEC standards, the DT is taken from the left. The behavior that occurs when components which have no DT of their own displayed and take their DT from a different component is called *DT* adoption in EPLAN.

| No. | Symbol description                                 | Entries       |
|-----|--|---------------|
| 9.  | Power NO contact of a contactor                    |               |
|     | Symbol name for direct entry:                      | SL            |
|     | Coordinates:                                       | X:344/Y:168   |
|     | Displayed DT:                                      | -             |
|     | Connection point designation:                      | 3¶4           |
| 10. | Power NO contact of a contactor                    |               |
|     | Symbol name for direct entry:                      | SL            |
|     | Coordinates:                                       | X:352/Y:168   |
|     | Displayed DT:                                      | -             |
|     | Connection point designation:                      | 5¶6           |
| 11. | Three-phase asynchronous motor, one rotation speed |               |
|     | Name in the symbol file:                           | M3            |
|     | Coordinates:                                       | X:336/Y:96    |
|     | Displayed DT:                                      | (-M1)         |
|     | Connection point designation:                      | (U1¶V1¶W1¶PE) |

Assign the motor a **PE potential connection point** as well. As already described in section "Inserting Potential Connection Points" on page 47, select the menu item **Insert > Potential connection point** and insert **variant C** of the symbol at the coordinates X:360/Y:148. As **Name of potential** and **Potential type**, you must enter PE.

Then insert another Angle up, right (menu path: Insert > Connection symbol > Angle...) at the coordinates x:352/Y:80 and an Angle up, left at the coordinates x:360/Y:80.

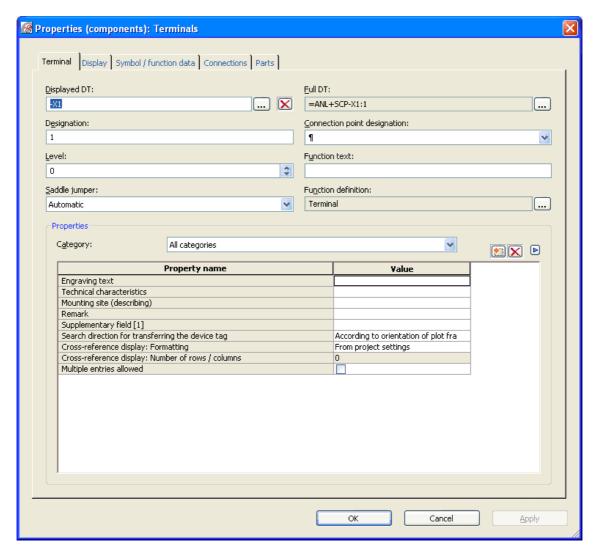


# **Inserting Terminal Strips and Terminals**

Terminals are a kind of device and, like general devices, can only be inserted into your schematic using **symbol selection**.

- Select the menu item Insert > Symbol.
  - ⇒ The **Symbol selection** dialog is opened.
- 2. In the **Symbol selection** dialog, select the **List** tab.
- 3. If you haven't yet, activate the Filter IEC symbols, multi-line.
- 4. Enter x into the **Direct entry** field.

- 5. In the list, select the three-line switch x and click **[OK]**.
  - ⇒ The terminal symbol hangs on the mouse pointer and can be placed on the current page in the **graphical editor**.
- 6. Position the first terminal at the coordinates X:336 / Y:140 in the schematic and then click the left mouse button.
  - ⇒ The **Properties (components): Terminals** dialog opens. Here is where you determine the properties of the terminals. Terminals are also automatically numbered by default. Accordingly, the **Displayed DT** and **Designation** on the **Terminal** tab are already filled with the entries –x1 and 1.



#### 7. Click [OK].

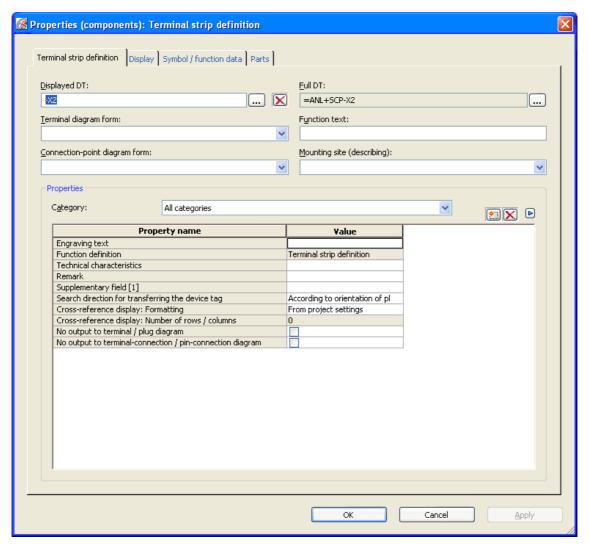
⇒ The terminal with the displayed DT -x1 is placed in the schematic.

- 8. Position three more terminals at the coordinates x:344/y:140, x:352/y:140, and x:360/y:140. For the first two terminals, you can accept the predefined terminal designations 2 and 3, but for the last terminal change the designation to PE.
- 9. Finish the action by selecting the menu item **Cancel action** from the popup menu.

## Inserting a terminal strip definition

In EPLAN, terminal strips are managed using terminal strips definitions. The terminal strip definition identifies the terminal strip and contains all the relevant data for the strip as well as the terminal strip part information.

- 1. Select the menu item **Insert > Terminal strip definition**.
  - ⇒ The symbol for the terminal strip definition hangs on the mouse pointer.
- 2. Position the symbol at the coordinates X:12 / Y:36 and click the left mouse button to place the terminal strip definition.
  - ⇒ The **Properties (components): Terminal strip definition** dialog opens. Due to the automatic numbering, the field **Displayed DT** is already prefilled with the entry -x2.

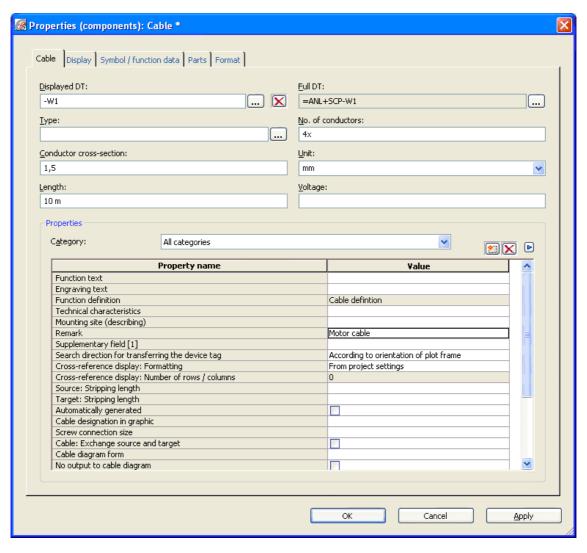


- 3. In the **Displayed DT** field on the **Terminal strip definition** tab, enter the strip designation -x1 already assigned in the circuit diagram.
- 4. In the Function text field, enter the text Terminal strip motor.
- 5. Click [OK].
  - ⇒ The terminal strip definition is inserted at the given coordinates. As terminal strip definition, the text -X1=Terminal strip motor is displayed.
- 6. Then select **Popup menu > Cancel action** to terminate the action.

# **Drawing Cable Definition Lines**

Cables can be displayed graphically with cable definition lines (or shields). The cable definition line must cross the corresponding connections.

- 1. Select the menu item Insert > Cable definition.
  - ⇒ The symbol for the cable definition line hangs on the mouse pointer.
- 2. Position the cursor at the coordinates x:324 / y:116.
- 3. Click with the left mouse button to establish the starting point of the line.
- 4. Then move the cursor to the coordinates X:364 / Y:116 and click the left mouse button to place the end point of the line.
  - $\Rightarrow$  The **Properties (components): Cables** dialog opens. Due to the automatic numbering, the field **Displayed DT** is already prefilled with the entry -W1.
- 5. In the fields **No. of conductors, Conductor cross-section, Unit,** and **Length**, enter the following entries: 4x, 1.5, mm, 10 m.
- 6. In the Remark field, enter the text Motor cable.



#### 7. Click [OK].

- ⇒ In the schematic, the cable definition line with the displayed DT
   –w1 is shown along with the data entered.
- 8. Then select **Popup menu > Cancel action** to terminate the action.

When drawing the cable definition line, a connection definition point is automatically set at each intersection of connections with the cable definition line. The cable conductors are determined via the connection definition points. You will only see these automatically generated connection definition points if you have activated the view of insertion points.

# **Updating Connections**

Connection lines generated by autoconnecting are at first simply graphical lines and not connections with data. For certain actions (e.g., closing a page, generating reports, etc.) the connections are first automatically updated. In doing this, new connections are generated based on the available information or existing connections are updated. The individual connection can get its data from project settings, potentials, or from a connection definition point.

Since the connection data is not automatically updated for each action, we recommend manually updating before certain actions (such as the execution of check runs, or working in navigator dialogs).

- 1. Select the menu item **Project data > Connections > Update**.
  - ⇒ EPLAN regenerates all connections in the project.

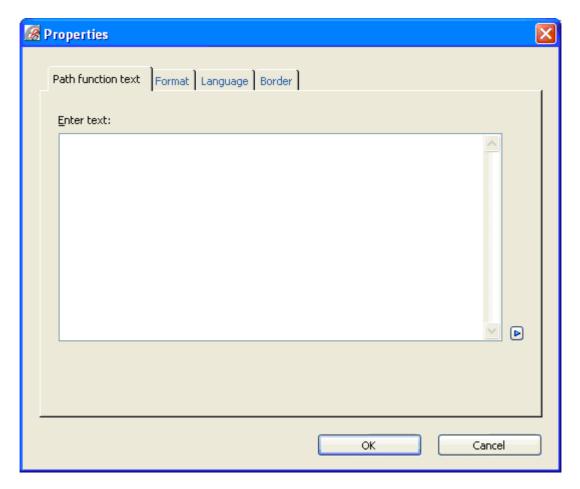
#### ■ Note:

You can tell if connections are up-to-date by looking at the right edge of the **status bar**: If the "#" sign is displayed there, the project has out-of-date connections. If the "\*" sign is displayed, the open page has out-of-date connections.

# **Inserting Path Function Texts**

Path function texts ease documentation because you don't have to enter a function text for each component. If no other function text is entered for a component, the function text from the path will be used for reports (e.g., in terminal diagrams).

- 1. Select the menu item Insert > Path function texts.
  - ⇒ The properties dialog is opened.



- 2. On the Path function text tab, enter the text Power supply 3ph / 400V.
- 3. Click **[OK]**.
  - ⇒ The path function text hangs on the mouse pointer.
- 4. Position the cursor at the coordinates X:40 / Y:64.
- 5. Click the left mouse button to place the text in the desired path.
  - ⇒ The path function text is placed at the given coordinates.
- 6. Then select **Popup menu > Cancel action** to terminate the action.

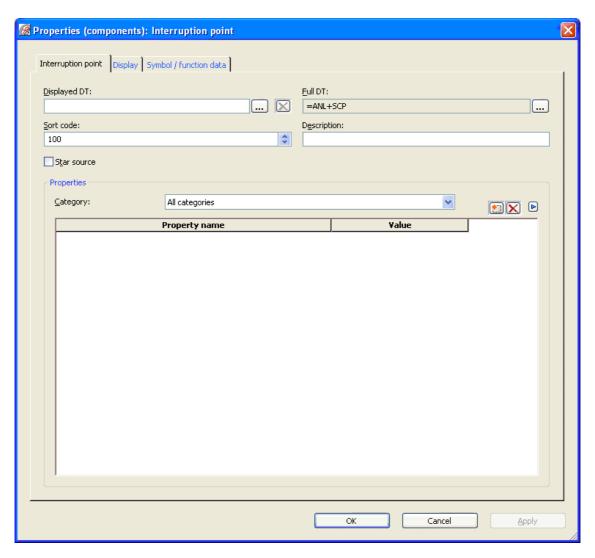
Repeat this action to insert two more path function texts (Distribution 24 V DC at the coordinates X:176 / Y:64 and Conveyor belt at X:336 / Y:64).

#### ■ Note:

Be sure when inserting path function texts that the insertion point of the text is aligned with the insertion point of the component which should receive the path function text.

# **Inserting "Interruption Point" Connection Symbols**

- 1. Select the menu item Insert > Connection symbol > Interruption point.
  - ⇒ The interruption point symbol hangs on the mouse pointer.
- 2. Hold [Ctrl] and then move the cursor in a circle.
  - ⇒ The different variants are displayed.
- 3. Select the variant with the arrow pointing downwards (variant D) and release the [Ctrl] key.
- 4. Position the interruption point at the coordinates X:176 / Y:96 and click the left mouse button.
  - ⇒ The Properties (components): Interruption point dialog opens.

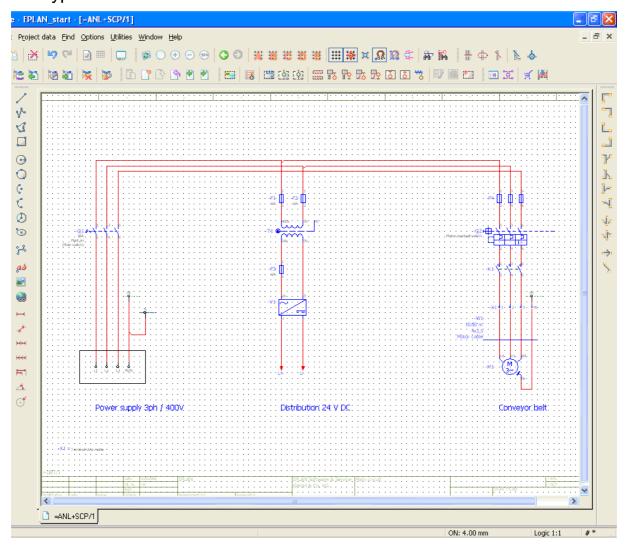


- 5. In the **Displayed DT** field, enter the device tag L+.
- 6. Click **[OK]**.
  - ⇒ The interruption point is inserted at the given coordinates.
- 7. Position another interruption point with the designation L- at the coordinates X:192/Y:96.
- 8. Then select **Popup menu > Cancel action** to terminate the action.

### Inserting more action lines

Your first schematic page is now almost complete. Now you must insert two more action lines. To do this, follow the description "Drawing Lines" in the section "Inserting Graphical Elements" (starting on page 51) and draw the first line from the coordinates x:356/y:148 to the end point x:368/y:148. Assign this "PE" line the same formatting properties you assigned to the first "PE" line.

Position the action line for the NC power contact -K1 at coordinates X:336/Y:168 and X:352/Y:168. Change the formatting properties for this line, too. For instance, select a different line thickness and a different line type.



### **Generating Cross-references**

In the previous chapter, you generated a schematic on the first schematic page of your practice project. In this chapter, you will complete the schematic on the second page, thus generating cross-references.

EPLAN differentiates between different types of cross-reference. The program automatically inserts most of these cross-references automatically (online). Based on the practice project, you will get to know some of the most common cross-reference types. More detailed information on the different types of cross-reference can be found in the EPLAN online help.

### What you Need to Know in Advance

#### What are cross-references?

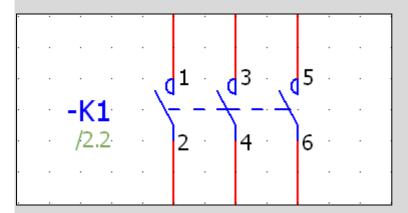
Devices can consist of different elements and be distributed across multiple pages of schematics. It can also be necessary to depict a device multiple times. In such cases, cross-references identify the fact that individual components belong together. A cross-reference shows you where the other part of a device can be found in the schematic. Using cross-references, you can reliably find a component or associated parts on a series of pages.

### How are cross-references displayed in the schematic?

The cross-references are shown in the order [Separator] Page (name) [Separator] Column by default. The separator used in front of the page is "/", and the separator used between the page and column is ".".

### 🗑 Example:

To do this, we will use an example (still to be created) from your practice project. The NC power contact -K1 on the first schematic page will later be cross-referenced to a coil which (then) is located on page 2 in column 2. In our example, the resulting cross-reference /2.2 will be shown under the displayed DT -K1.



By default, the cross-references receive a different color than the other elements displayed.

### **Automatically Generating Interruption Point Cross-references**

First, insert two interruption points on the second schematic page. Interruption points are used to represent connections across more than one page. The cross-references between the interruption points are automatically generated by EPLAN.

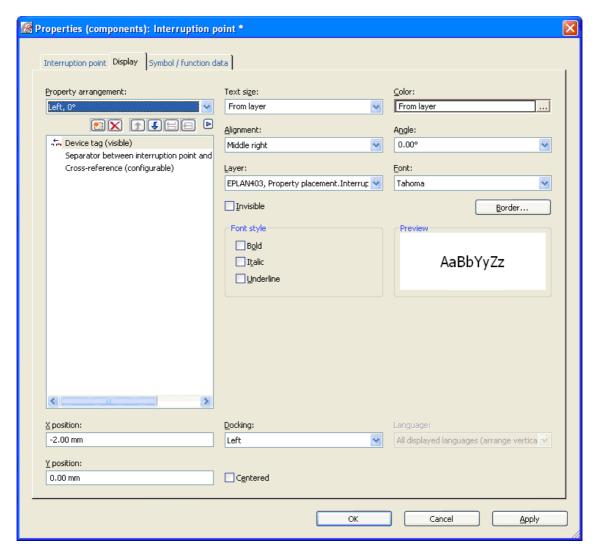
First open the page =ANL+SCP/2 with the description Control circuit. To do this, select them in the **Page navigator** and double-click on the page. EPLAN opens the page in the graphical editor.

On this page, for instance, using the key combination [Shift] + [F4], insert an interruption point L+ at coordinates X:36 / Y:260. See also the description in the section "Inserting 'Interruption Point' Connection Symbols" on page 70.

### Changing display settings for the interruption points

To prevent the device tag from crossing future autoconnect lines, additional settings must be made in the appropriate property dialogs.

- In the Properties (components): Interruption point dialog, switch to the Display tab.
- 2. From the dropdown list **Property arrangement**, select the entry Left, 0°.
  - ⇒ The entries will change the **Alignment**, **X position**, and **Docking** fields.

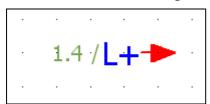


### 3. Click [OK].

 $\Rightarrow$  The interruption point L+ is inserted. The displayed properties are shown to the left of the insertion point.

- 4. Position a second interruption point L- at coordinates x:36/y:100 and change the **Property arrangement** on the **Display** tab to Left,  $0^{\circ}$ .
  - $\Rightarrow$  The interruption point L- is inserted.
- 5. Then select **Popup menu > Cancel action** to terminate the action.

The cross-references are displayed at both inserted interruption points beside the device tag.



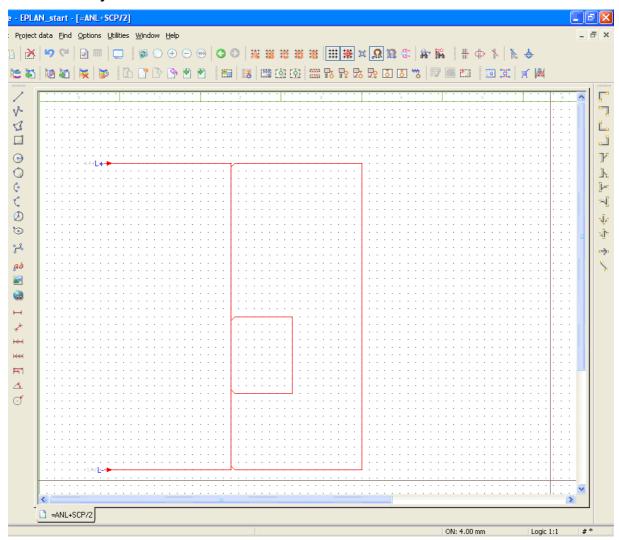
The cross-references refer to the interruption points with the same displayed DT located on the first schematic page. On this page, too, corresponding cross-references are displayed at the interruption points.

## Inserting More Connection Symbols and General Devices

Insert the connection symbols listed below into the second schematic page. To do this, for instance, select the menu item **Insert > Connection symbol > ...** and proceed according to the descriptions in the chapter "Creating a Schematic" (starting on page 44).

| Symbol description   | Coordinates |
|----------------------|-------------|
| 2 x Angle down, left | X:168/Y:260 |
|                      | X:132/Y:180 |
| 2 x Angle up, left   | X:168/Y:100 |
|                      | X:132/Y:140 |
| T-node, down         | X:100/Y:260 |
| T-node, up           | X:100/Y:100 |
| 2 x T-node, right    | X:100/Y:180 |
|                      | X:100/Y:140 |
|                      |             |

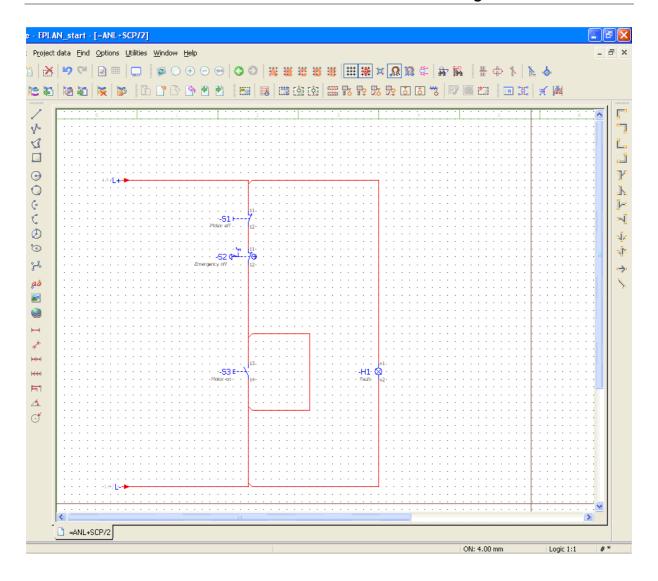
For the second **T-node, right**, which you place at coordinates x:100 / y:140, you must use the variant **1st target below, 2nd target right**. To do this, after inserting, click on the insertion point of the T-node and then select the corresponding option in the **T-node right** dialog (see section "Inserting 'T-node' Connection Symbols" on page 50.). Or you can use the **[Ctrl]** key to rotate the symbol on the cursor and select the right variant that way.



Finally complete your second schematic page by placing the general devices. To do so, call up **symbol selection** (for instance, using **Insert > Symbol**) and then follow the description "Inserting General Devices" in the chapter "Creating Schematics".

The following table lists the symbols to be inserted from the symbol library **IEC\_Symbol**. Entries predefined by EPLAN are shown in parentheses.

| No. | Symbol description  | Entries           |
|-----|---|-------------------|
| 1.  | Pushbutton, NC contact, general   |                   |
|     | Symbol name for direct entry:   | SOA               |
|     | Coordinates:  | X:100/Y:240       |
|     | Displayed DT:   | (-S1)             |
|     | Connection point designation:   | (11¶12)           |
|     | Function text:  | Motor off         |
| 2.  | Emergency stop switch / Emergency stop pushbutton, NC contact, with turn-to-reset |                   |
|     | Symbol name for direct entry:   | SONOT2            |
|     | Coordinates:  | X:100/Y:220       |
|     | Displayed DT:   | (-S2)             |
|     | Connection point designation:   | (11¶12)           |
|     | Function text:  | Emergency shutoff |
| 3.  | Pushbutton, NO contact, operated by pushing                                       |                   |
|     | Symbol name for direct entry:   | SSD               |
|     | Coordinates:  | X:100/Y:160       |
|     | Displayed DT:   | (-S3)             |
|     | Connection point designation:   | (13¶14)           |
|     | Function text:  | Motor on          |
| 4.  | Lamp / indicator light, general   |                   |
|     | Symbol name for direct entry:   | Н                 |
|     | Coordinates:  | X:168/Y:160       |
|     | Displayed DT:   | (-H1)             |
|     | Connection point designation:   | (x1¶ $x2)$        |
|     | Function text:  | Fault             |

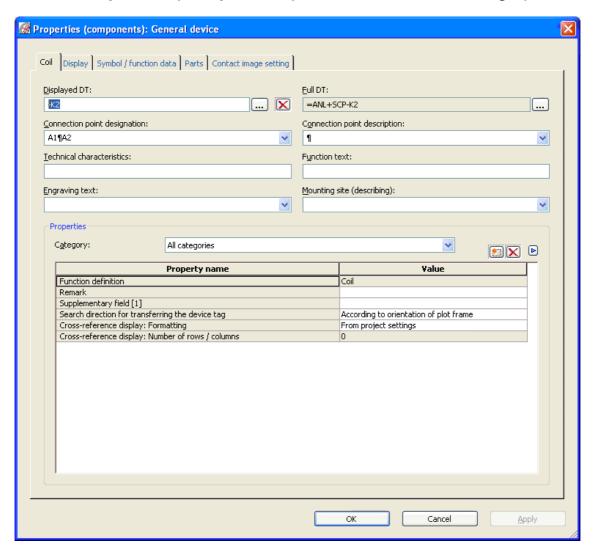


### **Inserting Contactor Coils and Contact Images**

All the contacts of a device are shown in a contact image. This form of cross-reference display is often used for contactor coils or motor overload switches.

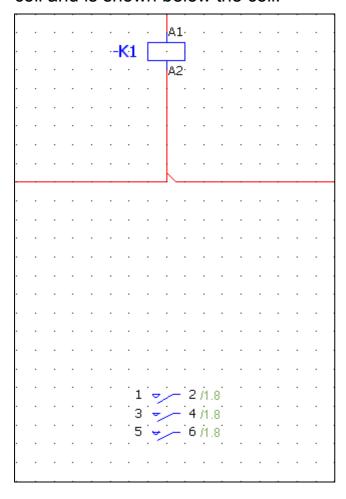
- Select the menu item Insert > Symbol.
  - ⇒ The Symbol selection dialog is opened.
- 2. In the **Symbol selection** dialog, select the **List** tab.
- 3. If you haven't yet, activate the Filter IEC symbols, multi-line.
- 4. Enter K into the **Direct entry** field.
- 5. In the list, select the coil **K** and click **[OK]**.

- ⇒ The symbol hangs on the mouse pointer and can be placed on the current page in the **graphical editor**.
- 6. Position the coil at the coordinates X:100 / Y:120 in the schematic and then click the left mouse button.
  - ⇒ The Properties (components): General devices dialog opens.



- 7. In the properties dialog, switch to the **Coil** tab.
- 8. Enter the value -K1 into the Displayed DT field.
- 9. Apply the predefined Connection point designation A1 ¶A2.
- 10. Click [OK].
- 11. Then select **Popup menu > Cancel action** to terminate the action.

Since we are using the same displayed DT for the coil as for the power contacts on the first schematic page, EPLAN automatically generates a contact image. This includes the connection point designations, the symbols, and the schematic positions of the contacts cross-referenced to the coil and is shown below the coil.



### Inserting another contact for the coil

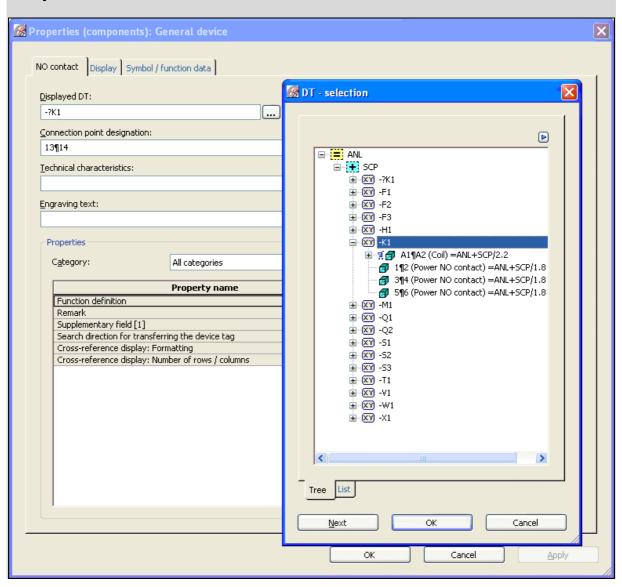
Now use **symbol selection** to insert another contact for the coil. For searching via **direct entry** enter s and then select the NO contact with symbol number 1. Place the NO contact at coordinates x:132/y:160 and enter -K1 here, too, as the **Displayed DT**.



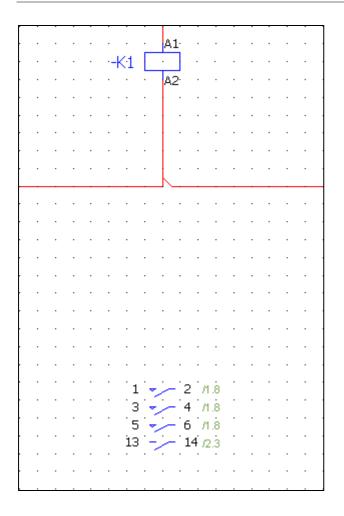
You can also apply the **Displayed DT** using a DT selection. To do this, click the [...] button located next to the **Displayed DT** field. The **DT - selection** dialog which then appears shows all the DTs present in the project which match the given function. Here you can apply an already existing DT with **[OK]** after selecting it, or use **[Next]** to generate a new DT.

### Example:

The illustration shows you an example of **DT - selection** for the NO contact just inserted.



The cross-reference to the coil on the same page is displayed in the schematic immediately. For the contact to be shown in the contact image as well, you may need to update the view (menu path: **View > Redraw**).



# **Inserting Contacts and Contact Image for Motor Overload Switches**

Finally, insert two more contacts on this page for the motor overload switch -Q2 on the first schematic page.

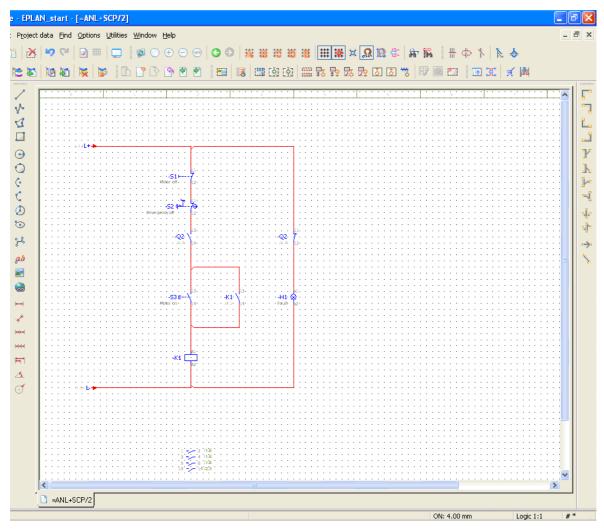
- 1. Select the menu item **Insert > Symbol**.
  - ⇒ The **Symbol selection** dialog is opened.
- 2. In the **Symbol selection** dialog, select the **List** tab.
- 3. If you haven't yet, activate the Filter IEC symbols, multi-line.
- 4. Enter o into the **Direct entry** field.
- 5. In the list, select the NC contact **O** and click **[OK]**.
  - ⇒ The symbol hangs on the mouse pointer and can be placed on the current page in the **graphical editor**.

- 6. Position the NC contact at the coordinates X:168 / Y:200 in the schematic and then click the left mouse button.
  - ⇒ The Properties (components): General devices dialog opens.
- 7. In the properties dialog, switch to the **NC contact** tab.
- 8. Enter the value -Q2 into the **Displayed DT** field.
  Or apply the displayed DT using DT selection. To do so, click the [...] button next to the **Displayed DT** field, then select the entry -Q2 in the **DT selection** dialog which then appears.
- 9. Click [OK].
- 10. Then select **Popup menu > Cancel action** to terminate the action.

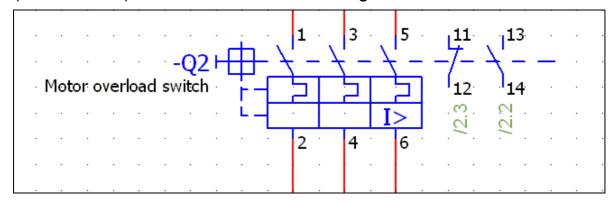
Now use **symbol selection** to insert another NO contact (symbol name for **Direct entry**: s; symbol number: 1). Place the NO contact at coordinates x:100/Y:200 and enter -02 here, too, as the **Displayed DT**.

Once the contact is placed, a corresponding cross-reference for the motor overload switch is shown on the first schematic page.

You have now completed the creation of the schematic on the second page.



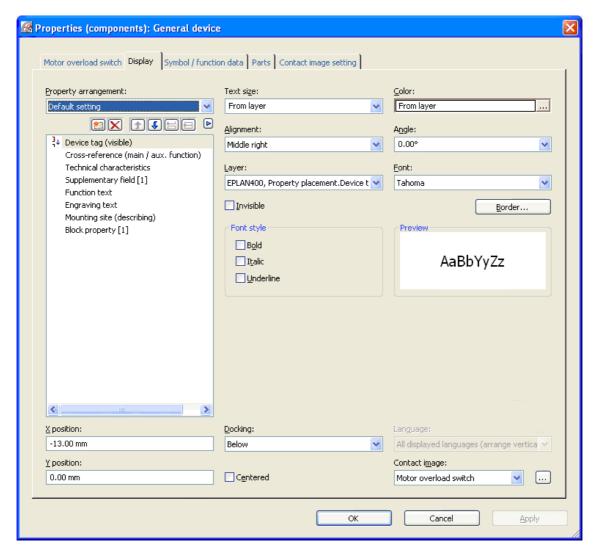
On the motor overload switch itself, you see the contacts just inserted as a contact image. The cross-referenced contacts are shown as symbols on the action line of the motor overload switch. In addition, the connection point designations and the schematic positions of the contacts can be seen in the contact image. Open the first schematic page (=ANL+SCP/1) and look at the contact image.



For contacts to be shown in a contact image, appropriate display settings must be made for the contact image in the properties dialog of the motor overload switch.

Look at these settings in the motor overload switch.

- 1. Double-click in the schematic on motor overload switch -Q2.
  - ⇒ The Properties (components): General devices dialog opens.
- 2. In the properties dialog, switch to the **Display** tab.
  - ⇒ The **Contact image** dropdown list shows the correct setting **Motor overload switch** for the contact image. This setting is preset by default during insertion of motor overload switches.



### 3. Click [OK].

Now you are finished with both of the schematic pages of your project. In the next chapter you will learn how to assign parts to the devices inserted.

### **Selecting Parts**

In the previous two chapters, you created a simple schematic. Before you start with reports, for instance, you can automatically generate a parts list, you must enter parts for the devices inserted.

The **Parts** tab is available in the property dialog of the components to do this. Here you can manually enter the parts or select them using **part selection**.

### What you Need to Know in Advance

### What is part selection?

**Part selection** is a dialog you can use to select parts and their data from a data source. Normally, parts stored in EPLAN's internal **parts management** are available for **part selection**.

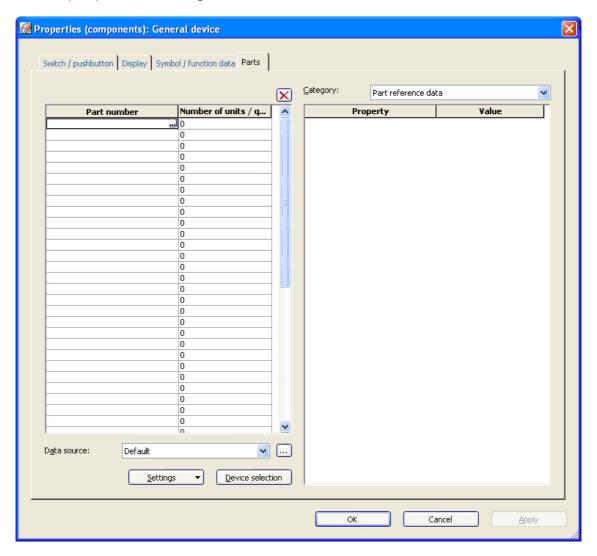
### What is parts management?

**Parts management** is the part of the program in which you can manage information specific to parts and people; items such as part number, customer, manufacturer, or supplier data. The data is stored in an internal EPLAN database. You can access a parts database already filled with sample data or create your own database which you must then fill with data. To call up **parts management**, select the menu item **Utilities > Parts > Management**.

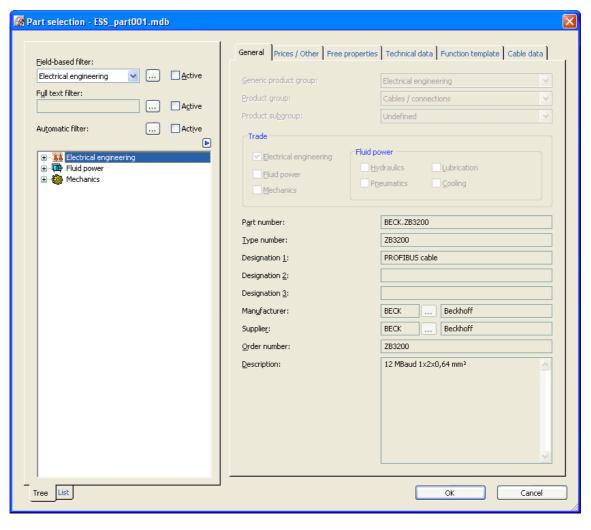
### **Assigning Parts**

- 1. First, if necessary, open the first schematic page (=ANL+SCP/1) of your practice project.
- Use View > Insertion points to show the insertion points of the elements inserted.
- 3. Double-click on the insertion point of switch -Q1.
  - ⇒ The Properties (components): General devices dialog opens.

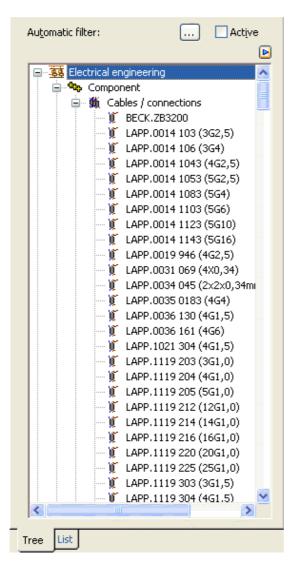
4. In the properties dialog, switch to the **Part** tab.



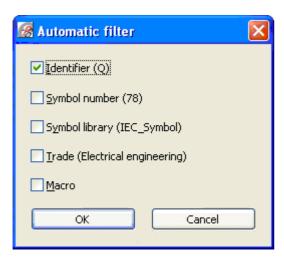
- 5. Click [...] in the **Part number** field. The button appears once you click on the corresponding table cell in the **Part number** column.
  - ⇒ The **Part selection** dialog opens. Structurally, this dialog is exactly like **Parts management**. In the left half, the parts are shown in a tree or a list, where the tree structure is subdivided into multiple product supergroups (electrical, fluid, mechanical). In the right half of the dialog, you see the data belonging to the element or elements selected in the left window.



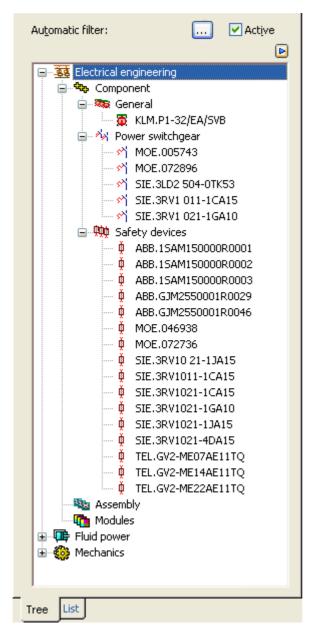
- 6. Select the entry Electrical engineering in the tree structure.
- 7. Click on the popup menu button and select the menu item **Expand** from the popup menu.
  - ⇒ The structure of the **Electrical engineering** product group will be expanded. The parts numbers are shown on the lowest level in the tree.



- 8. To reduce the amount of data shown, select a filter now. To do this, click the [...] button next to the **Automatic filter** field.
  - ⇒ The **Automatic filter** dialog opens. The check box **Identifier (Q)** is already prefilled.



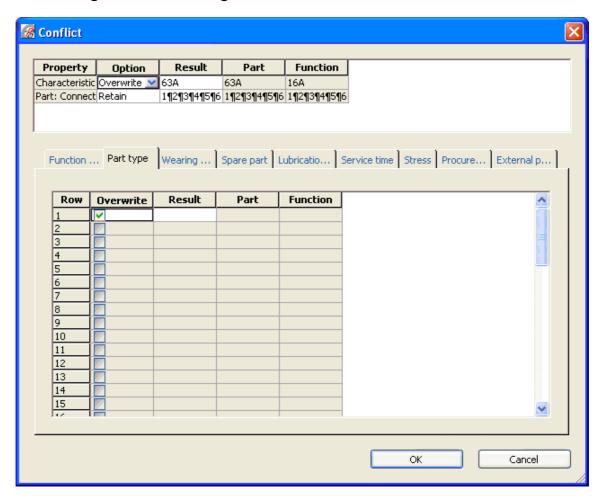
- 9. Click **[OK]** to confirm the setting.
  - ⇒ The Automatic filter dialog is closed.
- 10. In Part selection, select the corresponding Active check box for the Automatic filter so that the filter settings will be used.
  - ⇒ Now the parts matching the filter settings are shown in the tree structure.



- 11. In the tree from product group **Power switchgear**, select the part SIE.3LD2 504-0TK53.
  - ⇒ On the right side of **Part selection**, the parts data of the selected parts are now shown in the data fields.

#### 12. Click [OK].

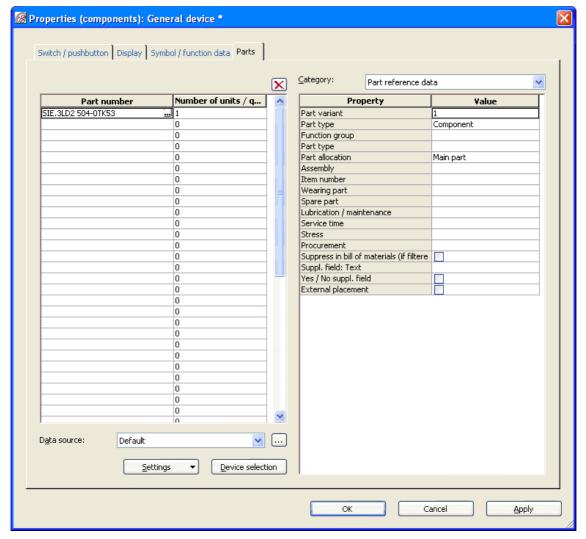
⇒ EPLAN opens the **Conflict** dialog. Whenever there is a difference between the data stored for the part and the data in the component, this dialog is called during **Part selection**.



13. For the property **Characteristic**, click the **Option** field and select the entry Retain from the drop-down list. This allows the characteristic 16A stored in the component (= **Function**) to be retained.

### 14. Click [OK].

⇒ The conflict dialog is then closed. The parts data selected is applied to the corresponding fields of the **Parts** tab, and the **Number of units / quantity** field is preset to 1.



- ⇒ Which data is shown to you on the right side of the tab depends on the setting in the dropdown list **Category**. If the entry Parts data is selected, then you will see data here such as the **Part number**, which is taken from **Parts management** and cannot be changed. If you selected **Part reference data**, then data is shown here, such as **Service time** which you can also store for a part in **Parts management**. In contrast to the "pure" parts data, however, you have the option of changing this data in the properties dialog and adapting it to the corresponding device in your own project.
- 15. Close the properties dialog with **[OK]**.
  - ⇒ The new parts data is written to the component.

#### ■ Note:

If you mistakenly enter or select the wrong part number in the **Part** tab, you can delete it by selecting the corresponding table row and clicking on the  $\times$  (Delete) button.

Repeat this action for the devices listed below. Assign the parts to each device based on the **Part number** listed. If the conflict dialog appears, then the properties stored in the component (= **Function**) will be kept. To do this, you must always select the **Retain** option for each property.

| Devices                       | Displayed<br>DT | Part number  | Number of units / quantity |
|-------------------------------|-----------------|--|----------------------------|
| Page =ANL+SCP/1               |                 |  |                            |
| Safety fuse                   | -F1             | PHO.0913032  | 1x                         |
| Safety fuse                   | -F2             | PHO.0913032  | 1x                         |
| Transformer                   | -T1             | SIE.4AV2400-2EB00-0A                                     | 1x                         |
| Safety fuse                   | -F3             | PHO.0913032  | 1x                         |
| Safety fuse                   | -F4             | SIE.5SG5700<br>SIE.5SE2310<br>SIE.5SH5010<br>SIE.5SH4362 | 1x<br>3x<br>3x<br>3x       |
| Motor overload switch         | -Q2             | MOE.046938   | 1x                         |
| Cable                         | -W1             | LAPP.0014 1043 (4G2,5)                                   | 1x                         |
| Motor                         | -M1             | SIE.1LA7070-4AB10-<br>ZA11                               | 1x                         |
| Terminal strip-<br>definition | -X1             | PHO.3004524<br>PHO.0442079                               | 3x<br>1x                   |

| Devices                                 | Displayed<br>DT | Part number                | Number of units / quantity |
|---|-----------------|----------------------------|----------------------------|
| Page =ANL+SCP/2                         |                 |                            |                            |
| Switch / pushbutton (NC)                | -S1             | SIE.3SB3201-0AA21          | 1x                         |
| Switch / pushbutton (emergency shutoff) | -S2             | PILZ.400410                | 1x                         |
| Switch / pushbutton (NO)                | -S3             | SIE.3SB3201-0AA41          | 1x                         |
| Coil                                    | -K1             | SIE.3RT1024-1BB44-<br>3MA0 | 1x                         |
| Signal lamp                             | -H1             | SIE.3SB3217-6AA20          | 1x                         |

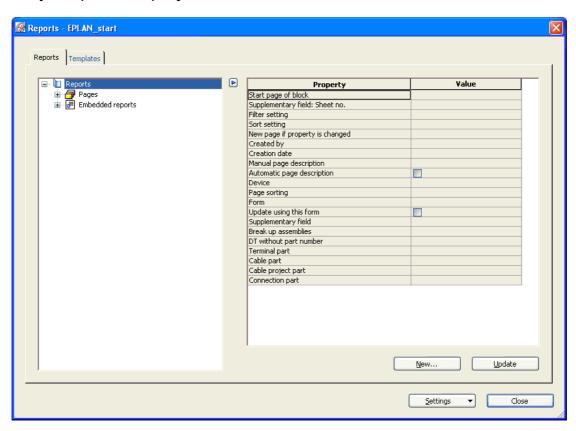
After you have assigned parts to some devices in this section, you can start on reports in the next chapter.

## **Generating Reports**

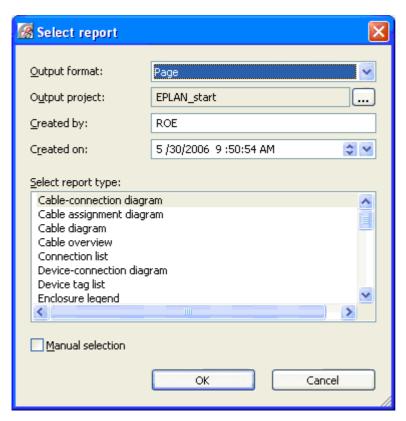
Now that you have created a schematic in the previous chapters and assigned parts to the devices inserted, you can have EPLAN generate reports on the information in your schematic and automatically output different report pages.

### **Generating a Terminal Diagram**

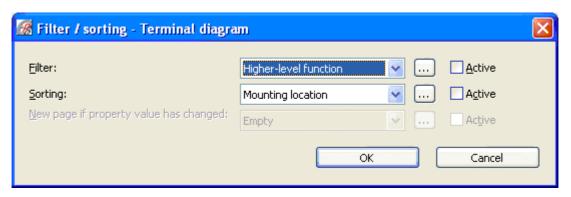
- Select the menu item Utilities > Reports > Generate.
  - ⇒ The **Reports EPLAN\_start** dialog opens. You can use this dialog to create and manage project reports.
- 2. Change to the **Reports** tab.
  - ⇒ In the tree on the left side, reports already generated for a project are shown. If you expand the tree (e.g., using **Popup menu > Expand**), then only the two sublevels **Pages** and **Embedded reports** are displayed. Up until now, of course, no reports have been created for your practice project!



- 3. Click the [New] button.
  - ⇒ EPLAN opens the **Select report** dialog.

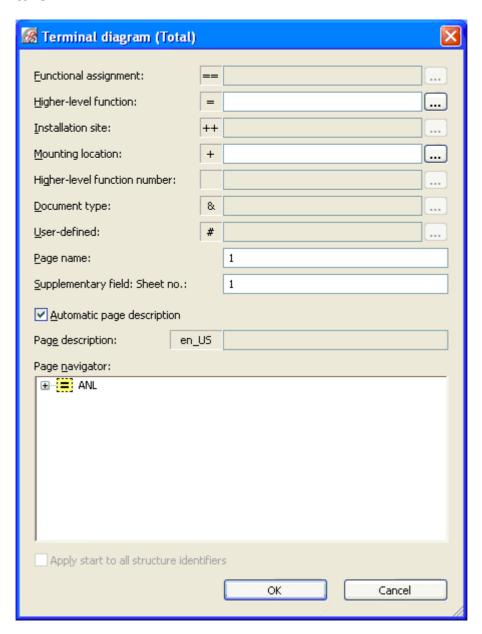


- 4. In the **Select report type** field, select the entry Terminal diagram. For this entry to be displayed, you must either enlarge the dialog or scroll through the entries using the scrollbar.
- 5. Click [OK].
  - ⇒ The **Filter / sorting Terminal diagram** dialog opens. Using this dialog, you can determine filter and sorting settings for the pages to be output. For your project, however, this is not required.

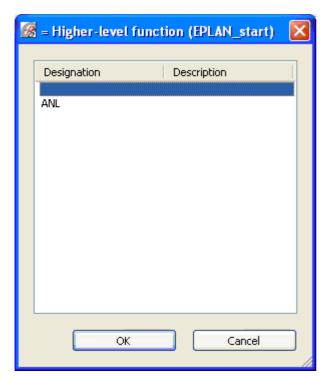


6. Click **[OK]** to confirm the predefined settings.

⇒ EPLAN opens the **Terminal diagram (Total)** dialog. Here you can specify how report pages will be sorted into the existing page structure.

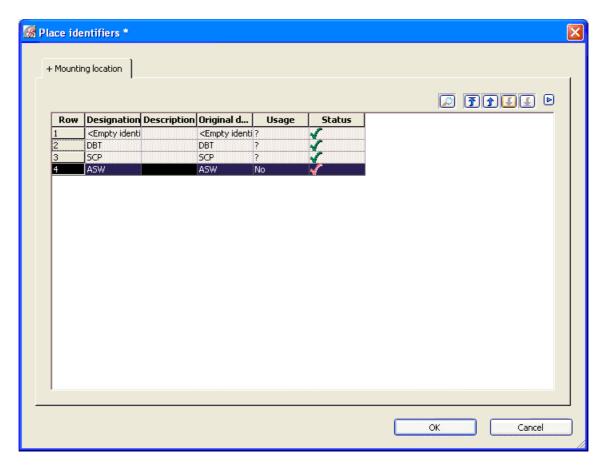


7. Enter the value ANL into the **Higher-level function** field. Since the identifier already exists in your project, however, you can select it using a selection dialog. To do this, click the [...] button next to the **Higher-level function** field.



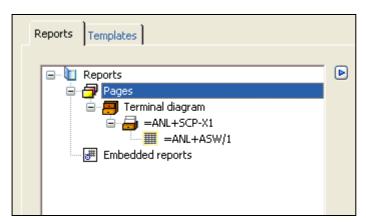
- 8. In the dialog =Higher-level function (EPLAN\_start) which appears, select the designation ANL and click on [OK] to return to the Terminal diagram (Total) dialog.
- 9. Enter the value ASW into the **Mounting location** field.
- 10. Click **[OK]**.
  - ⇒ The **Place identifiers** dialog opens.
- 11. Here, select the row with the designation ASW and move the row all the way to the bottom by clicking the 

  the order of identifiers created in the project and makes it possible for pages with the new identifier to be sorted in "after" the schematic pages.



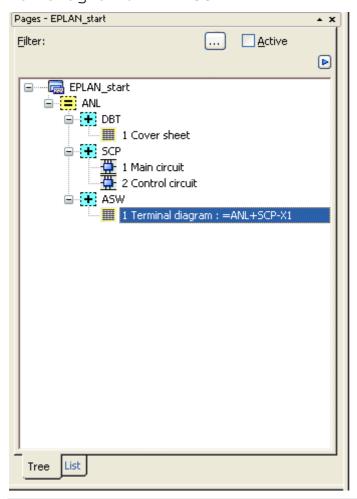
#### 12. Click [OK].

- ⇒ EPLAN generates the page with the terminal diagram. All open dialogs are closed, and you return to the **Reports EPLAN\_start** dialog.
- 13. Now expand the tree on the left side. Select the **Pages** level and select the menu item **Expand** from the popup menu.



- ⇒ Under the report **Terminal diagram**, the report page =ANL+ASW/1 has been created for the terminal strip =ANL+SCP-X1.
- 14. Click [Close].

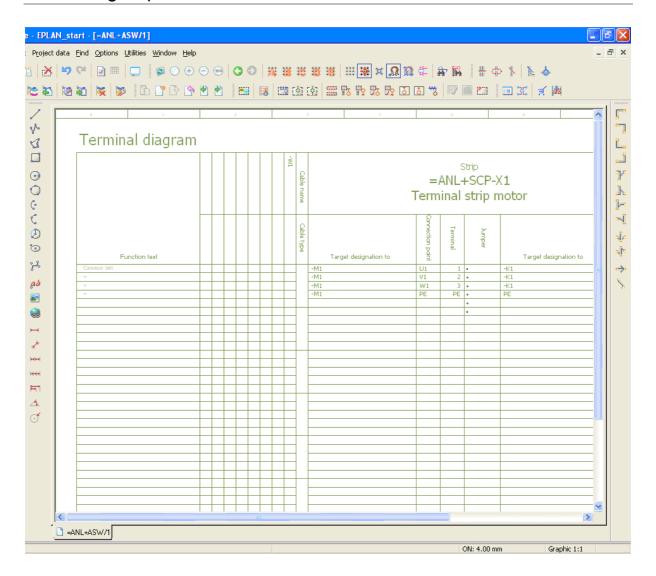
In the **Page navigator**, you now see the new page. You may first have to expand the tree and enlarge the window of the **Page navigator** before being able to see the new page =ANL+ASW/1 with the description Terminal diagram : =ANL+SCP-X1.



#### Note:

A Page description like Terminal diagram: =ANL+SCP-X1 is created if, for instance, the check box Automatic page description is checked in the Terminal diagram (Total) dialog. (See also the illustration of the dialog in the action just completed.) By deactivating the Automatic page description check box, you also have the option of entering your own description for each report page.

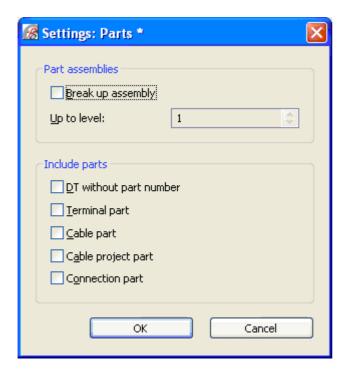
Open the page by double-clicking on it, and look at the terminal diagram in the **graphical editor**.



### **Performing Settings for Parts Lists**

Before generating the parts lists, you must first set a special setting. Otherwise, parts entered on the cable definition line will not be taken into account during output of the parts list.

- To do this, select the menu item Utilities > Reports > Generate.
  - ⇒ The Reports EPLAN\_start dialog opens.
- 2. Click the **Settings** button.
- Select the Parts list menu item.
  - ⇒ The **Settings: Parts** dialog opens.



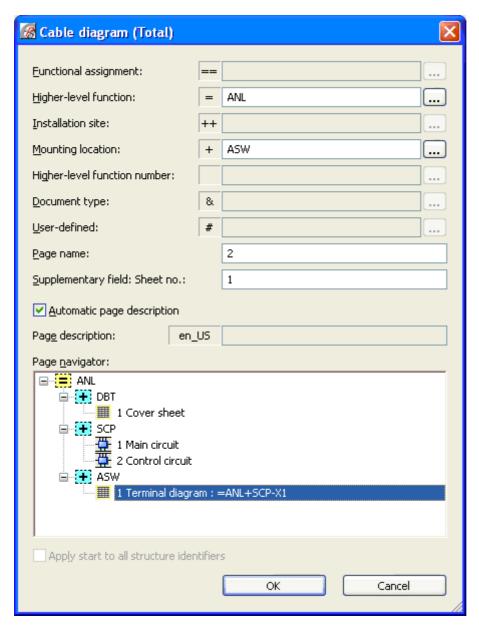
- 4. Select the Cable part check box.
- 5. Click [OK].

Now continue with the output of the other report pages.

### **Printing Other Report Pages**

Repeat the action described in the first section of this chapter for the reports **Cable diagram**, **Parts list**, and **Table of contents**. Proceed in the order listed here, and generate the reports one after the other in the **Reports - EPLAN\_start** dialog using the **[New]** button. On each report, select the corresponding report type from the **Select report** dialog.

For the other report pages, no new identifiers need to be assigned. For instance, you do not need to enter new identifiers in the **Cable diagram** (**Total**) dialog, but rather can apply an existing structure from the **Page navigator** field. To do this, expand the tree structure in this field and select the report page =ANL+ASW/1 (terminal diagram).

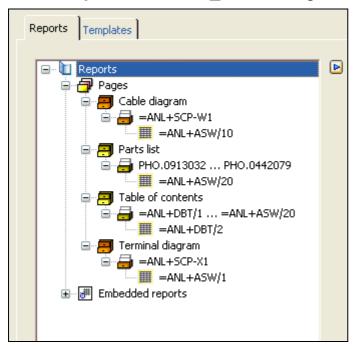


The identifiers of this page are applied to the **Higher-level function** and **Mounting location** fields, and for the **Page name**, the next free page is suggested. Instead of "2", enter the page number "10" into the **Page name** field.

For the subsequent output of the **Parts list**, select the corresponding page =ANL+ASW/10 (cable diagram) in the tree. This time, enter the page number "20" into the **Page name** field.

Then sort the **Table of contents** after the cover sheet by selecting the page =ANL+DBT/1.

After you have created the table of contents, the expanded tree structure in the **Reports - EPLAN\_start** dialog looks like this:



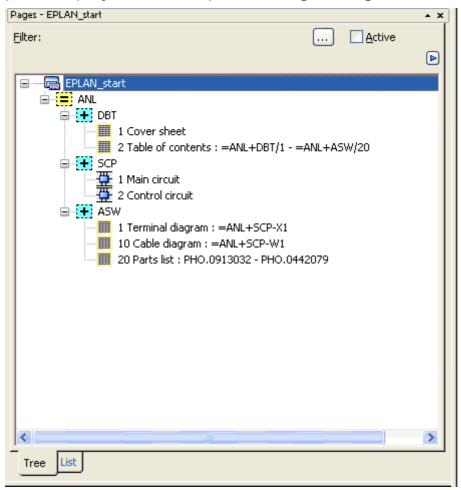
The different reports are sorted in alphabetically ascending order (**Cable diagram**, **Parts list**, **Table of contents**, **Terminal diagram**). The yellow icon **3** is used to mark all report types which represent overviews (e.g., parts lists, tables of contents, terminal diagram overview, etc.) The orange icon **3** characterizes function-relevant report types (e.g., cable diagram, terminal diagram, terminal connection diagram, etc.).

The level below that, marked in the tree with a yellow or orange icon with an opened drawer ( or or or orange icon with a report pages belonging to a particular element (e.g., a cable or a terminal strip) make up a single block.

In your current practice project, there is only one report block for each report. If, on the other hand, there were two terminal strips -x1 and -x2 in the schematic, then, for instance, the terminal diagram pages for terminal strip -x1 would make up their own block, just as the terminal diagram pages for terminal strip -x2.

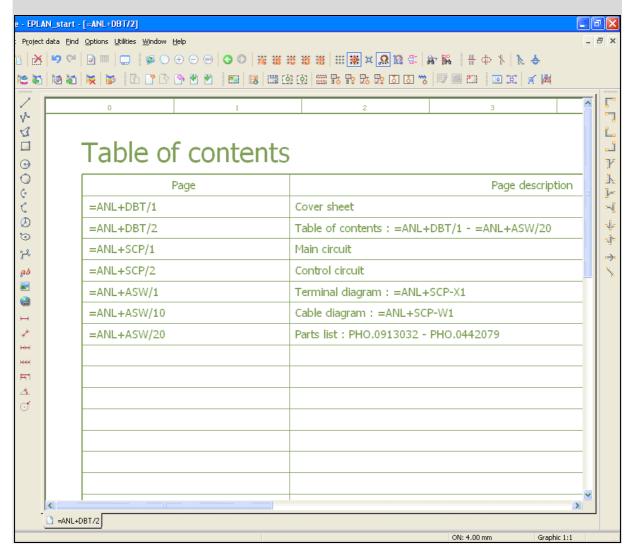
The pages produced are shown on the lowest level in the tree. They have the same icon  $\blacksquare$  as in the **Page navigator**.

Quit the dialog using [Close], and look at the changed structure of your practice project in the expanded Page navigator.



Then open the newly generated pages and look at these pages in the **graphical editor**.

For instance, the enlarged view of the table of contents (page =ANL+DBT/2) would look like this in the **graphical editor**.



With the generation of report pages, the creation and editing of your practice project is finished for now. In the following chapters, we will cover some additional topics, such as creating and inserting macros and searching project data. Then you will update the reports in your project using report templates.

If you like, you can now print your project pages out. To do this, read the chapter "Printing a Project" on page 127. Naturally, you can also wait until later to print the project pages, for instance at the end of the sequence of chapters.

# **Creating and Inserting Macros**

In EPLAN you have the option of using different types of macros. In the following sections, for instance, you will create a *window macro* and then insert it. Creating and inserting macros of other types, such as *page* and *symbol macros*, is done in a similar manner. You can find more information on these other macro types in the EPLAN online help.

### What you Need to Know in Advance

#### What are macros?

Macros in EPLAN are cut-outs of pages or projects that are saved for further use. Creating macros is helpful because you do not have recreate cut-outs every time a routine operation comes up.

#### What is a window macro?

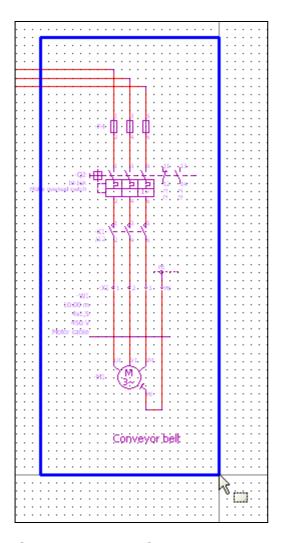
A window macro is an arbitrary area of a page. All objects whose insertion points are inside this area are stored in the window macro.

## **Creating a Window Macro**

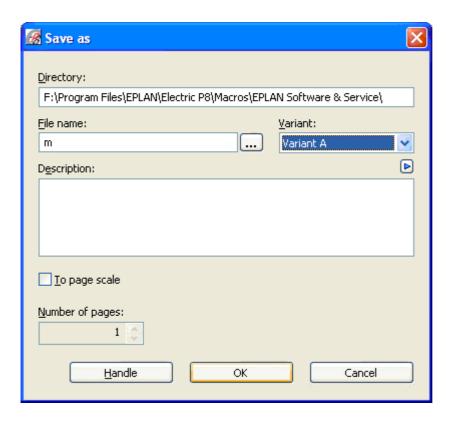
- 1. First, if necessary, open the first schematic page (=ANL+SCP/1) of your practice project.
- 2. Select the menu item Edit > Create window macro.
  - ⇒ In addition to the mouse pointer, a box with a dotted line is also displayed.



- 3. Position the cursor in the upper right corner of the schematic (e.g., at coordinates x:300 / Y:264), click the left mouse button, and drag the mouse to open an area enclosing the *entire* schematic section with the **Path function text** Conveyor belt.
  - ⇒ The area to be selected is outlined with a thick frame while it is being selected.



- 4. Click with the left mouse button again (if the cursor is at the coordinates x:388/y:48, for instance).
  - ⇒ All objects in the area are selected and EPLAN opens the **Save as** dialog. In the **Directory** field, the preset default directory for macros is displayed.

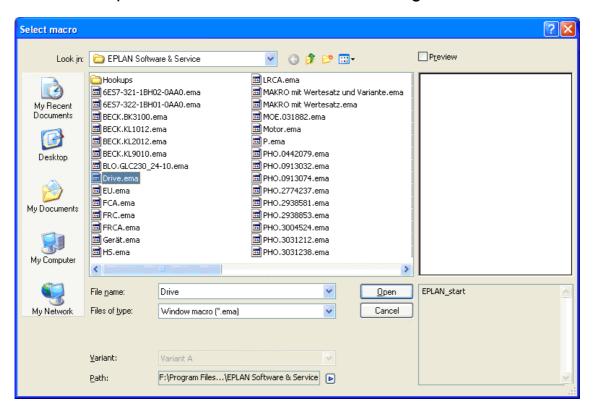


- 5. In the **File name** field, enter Drive as the name of the macro.
- 6. In the **Description** field, enter Conveyor belt drive as the description for the macro. The text entered here is displayed in a comments field when inserting macros and simplifies the selection for you.
- 7. Click [OK].
  - ⇒ The **Save as** dialog is closed. The macro is saved in the specified directory under the name Drive.ema.

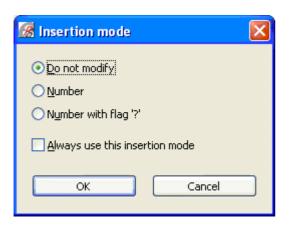
### **Inserting a Window Macro**

Before you insert the macro that you just saved again, first create another schematic page of the type <code>Schematic multi-line</code>. To do this, in the tree view of the <code>Page navigator</code>, select your second schematic page (<code>=ANL+SCP/2</code>) and use the key combination [Ctrl] + [N], for instance, to create a new page with the <code>page description Macro page</code>. Insert the macro on this sample page opened in the <code>graphical editor</code>.

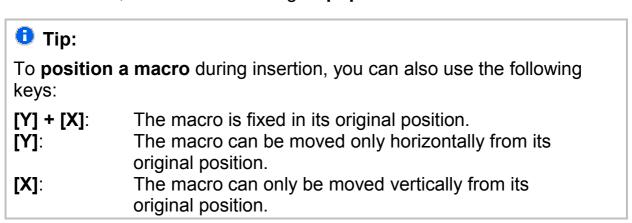
- Select the menu item Insert > Window macro.
  - ⇒ EPLAN opens the **Macro file selection** dialog.



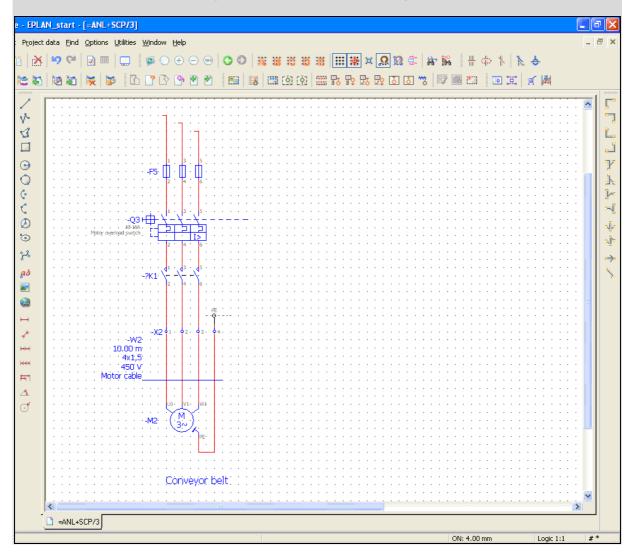
- 2. Select the **Preview** check box.
  - ⇒ A graphical preview of the currently selected macro is displayed in the right side of the dialog.
- 3. Select the macro Drive.ema in the specified macro directory.
  - ⇒ The source project in which the macro originated is shown along with the description of the macro in a comment field below the preview.
- 4. Click [Open].
  - ⇒ The macro hangs on the mouse pointer.
- 5. Position the macro at an arbitrary position in the schematic and click the left mouse button to place it.
  - ⇒ The **Insertion mode** dialog opens. In this dialog you define the manner in which devices are to be renumbered on insertion.



- 6. Select the **Number** option. This causes the devices inserted by the macro to be numbered automatically (online). For the counter of a DT, this takes the next free counter for each identifier into account.
- 7. Click [OK].
  - ⇒ The **Insertion mode** dialog is closed and the macro is placed. Since the macro is still hanging on the mouse pointer, you could insert the macro again.
- 8. However, end the action using **Popup menu > Cancel action**.



For instance, on the example page (=ANL+SCP/3), the window macro Drive inserted there might look like this in the **graphical editor**.



### Tip:

Using the menu path Page > Page macro > Insert, you can insert either a window macro or a page macro. In the Select macro dialog which appears next, you must then select the entry Window macro (\*.ema) as the File type. As usual when inserting page macros, this creates a new project page.

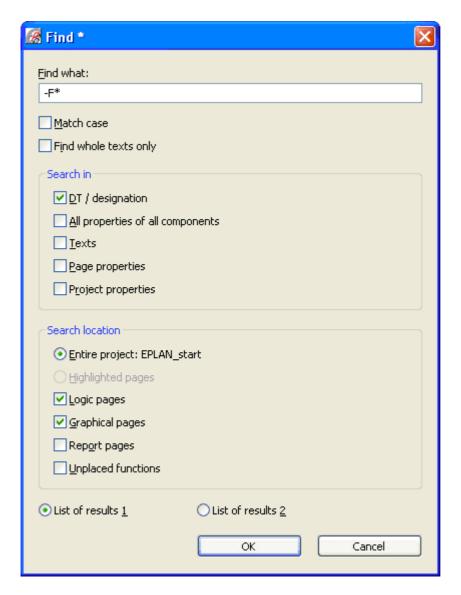
# Finding and Replacing Identifiers

While editing a project, it is often necessary to search the project for particular project data in order to change it. In the following chapter, we will show you how to search your practice project for a certain DT and a certain text and how you can then edit these objects.

## **Finding and Replacing Device Tags**

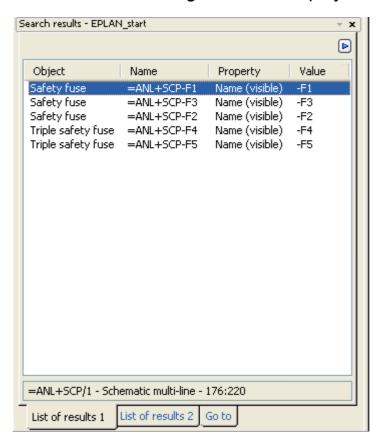
All the fuses in the project with the displayed DT -F should receive a different identifier.

- 1. Select the menu item **Find > Find**.
  - ⇒ The Find dialog opens.
- 2. In the **Find what** field here, enter the search term -F\*. You can also use placeholders (such as \* or ?) in the search.
  - ⇒ The settings necessary for your search are already prefilled by default. If the settings for your dialog don't match those of the figure shown below, you still need to perform steps 3 through 6. Otherwise, you can click **[OK]** immediately.



- 3. In the **Search in** group box you can specify the objects in which you would like to search for the term. Check the **DT** / **designation** check box here.
- 4. In the Search location group box, you can exclude certain page types from the search. Here, check the Entire project: EPLAN\_start option along with the Logic pages and Graphical pages check boxes.
- 5. You also have the option in this dialog of entering the results of a search into one of two possible **results lists**. Select the **List of results 1** option.
- 6. Click [OK].

⇒ The **Find** dialog is closed. EPLAN begins the search, opens the **Search results** dialog, and then displays the entries found.



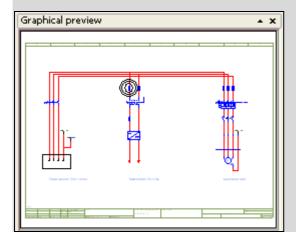
All fuses in the project are listed in the dialog in tabular form. If you cannot see all the entries, drag the dialog to enlarge it and adjust the width of the individual columns if necessary.

Below the listed search results, EPLAN shows you a text field with the page on which each selected object is located.



In the **Search results** dialog, if you select an object located on a project page, the **Graphical preview** shows the corresponding page. The position of the object is highlighted in the preview with concentric circles.

The following illustration shows the **Graphical preview** after the search result with the value -F1 has been selected in the results list.

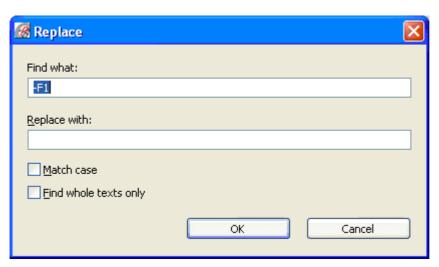


#### Tip:

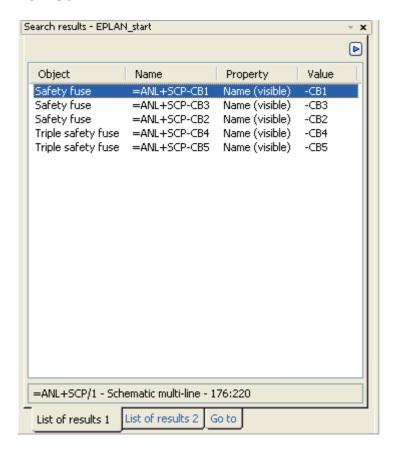
To call the **Find** dialog quickly, you can also use the key combination **[Ctrl] + [F]** or the **b**utton in the **Find** toolbar.

#### Replacing device tags

- 1. Select all entries in the Search results dialog.
- 2. Select **Popup menu > Replace**.
  - $\Rightarrow$  In the **Replace** dialog which appears, the value -F1 of the first selected entry is shown.

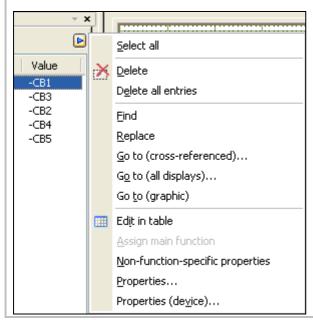


- 3. Remove the 1 in the **Find what** field and enter the value -CB into the **Replace with** field.
- 4. Click [OK].
  - ⇒ In the **Search results** dialog, the identifier -F is replaced by the new value -CB. In the schematic, too, all selected devices are renamed.



### Tip:

The popup menu of the **Search results** dialog includes additional very useful program functions for the finding and editing of project data. For instance, you can use the menu item **Go to (graphic)** to jump from a selected object in the results list to the corresponding position in a project page. Using the menu item **Edit in table**, multiple selected objects can be edited together in a subsequent dialog. In this way, for instance, you could rename the DTs of the selected devices.



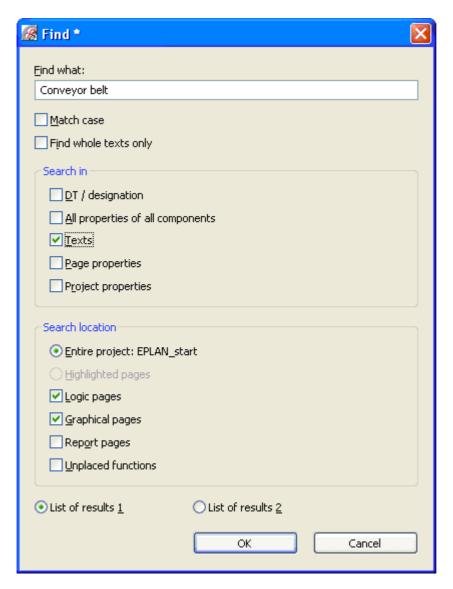
## **Finding Text and Editing Search Entries**

Before starting a new search, first remove all the previous entries. Otherwise the search results will remain in this result list.

- 1. Select all entries in the Search results dialog.
- 2. Select Popup menu > Delete all entries.

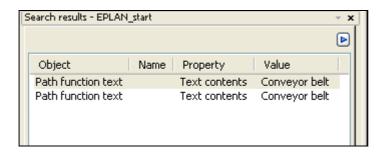
The **Search results** dialog is now empty again. Now start a new search, by pressing the key combination **[Ctrl] + [F]**.

- 3. In the Find what field here, enter the search term Conveyor belt.
- 4. In the **Search in** group box, uncheck the **DT / designation** check box and check the **Texts** checkbox instead.



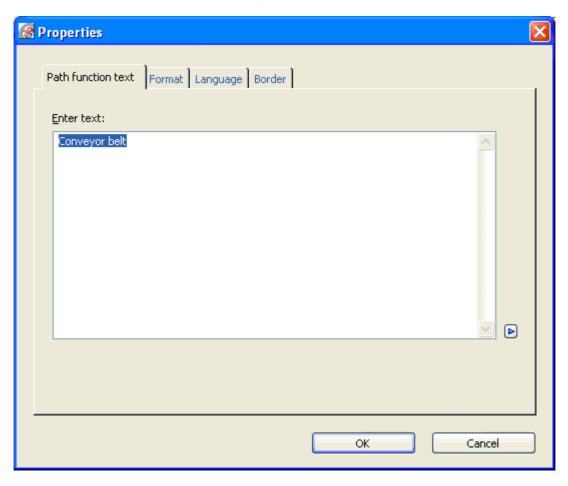
#### 5. Click [OK].

⇒ The **Search results** dialog shows you two entries for the search term **Conveyor belt**.



- 6. Select the second entry in the list.
  - ⇒ If you have opened the **graphical preview**, you will see that the second entry is a path function text which you inserted with the window macro on the page =ANL+SCP/3.

- 7. In the popup menu, select the menu item **Properties** (or double-click).
  - ⇒ The Properties dialog opens. On the Path function text tab, the text Conveyor belt is already selected.



- 8. Enter the new text New drive here directly.
- 9. Click **[OK]**.
  - $\Rightarrow$  In the results list and in the schematic on the page =ANL+SCP/3, the new path function text now appears.

Before you print out your whole project, you should first complete and update your reports. How you do this is described in the next chapter.

# **Completing and Updating Reports**

In the last two chapters, you completed and changed the data in your practice project. You created a new page and added additional devices to the project using the inserted window macro. For these changes to take effect in the project, you still have to complete and update the reports.

## **Updating Reports**

- 1. In the tree view of the **Page navigator**, select the structure identifier level =ANL (or select the name of your practice project directly).
- 2. Select the menu item **Utilities > Reports > Update**.
  - ⇒ All report pages below the structure identifier level =ANL are updated.

If you then look at the pages of your practice project, you will see that the overviews in the project (**Table of contents**, **Parts list**) have been updated. If you had made changes in the project to terminal strip -x1 or cable -w1 (for instance, specifying a different function text, a new target, etc.) then these changes would have been included in the update.

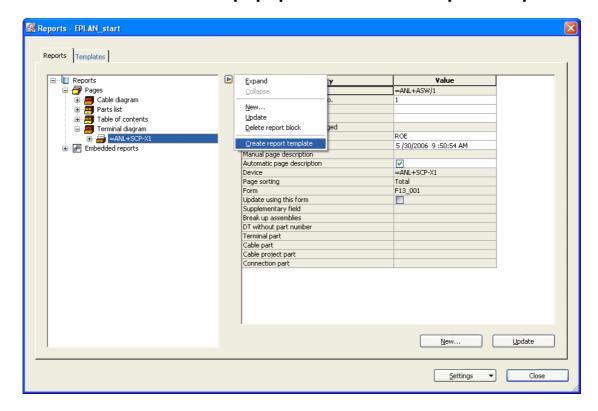
For the two new devices – terminal strip  $-x_2$  and cable  $-w_2$  – an update of the previously generated report pages is *not* sufficient. Instead, you must generate a terminal diagram for  $-x_2$  and a corresponding cable diagram for  $-w_2$ .

You could proceed as described in the chapter "Generating Reports" on page 96, and output new report pages in that way. But EPLAN also provides you with the possibility of using existing reports to generate **report templates**. Then you can report on the entire project based on those templates.

### **Creating Report Templates**

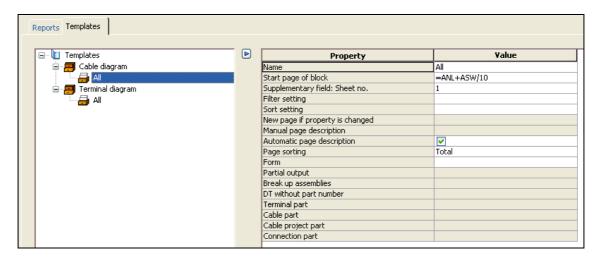
Report templates can be created from scratch in the **Reports - EPLAN\_start** dialog using the **Templates** tab, or based on an existing report. Since you already have reports with a terminal and cable diagram, use the latter procedure.

- Select the menu item Utilities > Reports > Generate.
  - ⇒ The Reports EPLAN start dialog opens.
- 2. Change to the **Reports** tab.
- 3. For the report **Terminal diagram** report, select the report block =ANL+SCP-X1 and select **popup menu > Create report template**.



- Repeat this procedure for the report Cable diagram. To do this, select the report block =ANL+SCP-W1 and again select Popup menu > Create report template.
- 5. Change to the **Template** tab.
- 6. Now expand the tree on the left side. Select the **Templates** level and select the menu item **Expand** from the popup menu.
  - ⇒ In the tree structure, the templates created are shown.

7. Now, if you select the template **All** under **Cable diagram**, for instance, the table on the right side will show the specified properties of this template. Here you might change the **Start page** for your cable diagram pages.



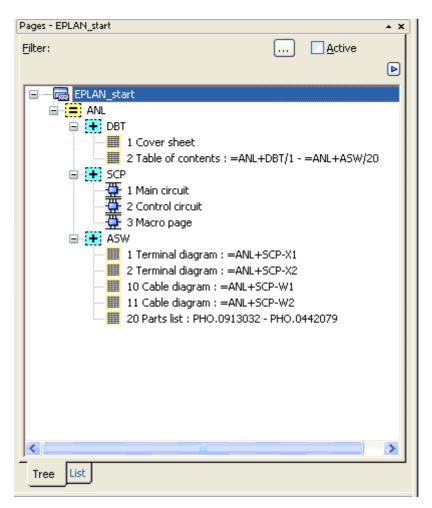
8. Click [Close].

## **Generating Project Reports**

If you now report on the entire project, then reports are first generated from all existing report templates. Then EPLAN updates all already existing reports.

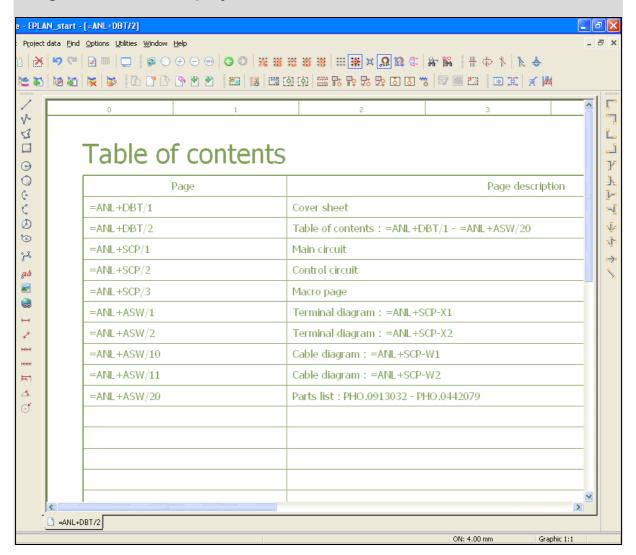
- Select the menu item Utilities > Reports > Generate project reports.
  - ⇒ Reports are generated for the project. Depending on the size of the project, this process can take some time.

After you have completed and updated the reports in this way, expand the tree view of your practice project in the **Page navigator** and look at the changed structure.



Then page through your project and look at the new and changed pages in the **graphical editor**. In the following chapter, you will conclude by learning how to print the pages of your projects.

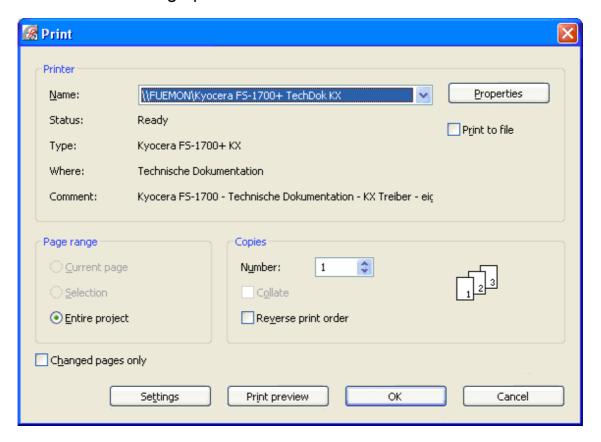
For instance, the enlarged view of the table of contents (page =ANL+DBT/2) would look like this in the **graphical editor** after the reports are generated for the project.



# **Printing a Project**

To conclude this getting started guide, print all the pages of your project. In EPLAN, you naturally also have the option of printing a single page or any arbitrarily selected pages of a project. The print can be in black and white or in color, and can either be sent to a printer or to a file.

- 1. In the tree view of the **Page navigator**, select your project EPLAN start.
- 2. Select the menu item **Project > Print**.
  - ⇒ The **Print** dialog opens.



- 3. In the **Printer** box, select the desired printer from the drop-down list.
- 4. If you have accidentally selected only one page in the **Page navigator** or if the cursor is located on an opened page, then by default only the current page will be printed. You can tell this by the fact that in the **Page range** group box, the **Entire project** option is not activated. In this case, activate this option.
- 5. Click **[OK]**.

⇒ All pages of the projects will be printed.

#### ■ Note:

Using the **Print preview**, which you can call from the print dialog using the button of the same name, you have the option of checking the pages to be printed before they are actually printed. If the pages shown don't match your expectations (if a print margin is missing, for instance), you can close the preview window and change the print settings if necessary in the **Print** dialog using the **[Settings]** button.