ZUKEN®

E3. series®

First Steps

Getting Acquainted with E³.series

Technical Inquiries
Please contact our Support Department!
E-mail: e3-support@zuken.com

Note:
Zuken is not responsible for any errors, which may appear in this documentation. Liability, due to direct and indirect losses resulting from the delivery or use of this documentation, is excluded to the extent permitted by law. This documentation contains copyrighted information.
All rights, especially those pertaining to the duplication and distribution as well as the translation, are reserved. This documentation, whether wholly or in part, may not be reproduced in any form (photocopy, microfilm, etc.), or processed, duplicated or distributed using an electronic system without Zuken's prior written consent.

Contact
Zuken E3 GmbH
Laemmerweg 55
D-89079 Ulm/Einsingen

Phone : +49 7305 9309-0
Fax:+49 7305 9309-99
Web: http://www.zuken.com
E-mail: e3-info@de.zuken.com

© Copyright 2020, Zuken E3 GmbH

Contents

First Steps **1**

Getting Acquainted with the *E³.*series User Interface **1**

Starting *E³.*series 2

Opening a New or Existing Project 3

The *E³.*series User Interface 6

More Windows 7

The Settings Menu 8

Defining the Database Structure 9

Locating Components 10

Mouse Button Settings 11

Examples of Context Menus 12

Browsing in the Project 14

Overview of the Toolbars 17

Shortcut Keys 19

Predefined shortcut keys in *E³.*series 20

First Steps **22**

Getting Acquainted with *E³.*schematic for Electrical Engineering (in combination with *E³.*cable) **22**

Opening an Existing Project 23

Inserting a New Sheet in the Project 24

Entering Sheet Header Text 25

Displaying Placement Help Lines 28

Using Predefined Subcircuits 29

Assigning Symbols to Components 31

Creating Terminal Plans in *E³.*schematic 32

Editing Areas, Placing Devices and Laying Connections 34

Connecting the Motor using a Cable 38

Creating the Cable Drawing (*E³.*cable Functionality) 42

Placing Additional Device Elements with Online Cross-References 48

Changing Attributes 51

Modifications to Existing Drawings 53

Modifications to the Terminal Plan 56

Adding Graphic Text 59

Using Text Boxes 60

Defining Fields 61

Searching and Navigating within the Project 62

Switching between Display Modes 62

Displaying / Hiding Options 63

Lists and Output Files 65

Table of Contents 66

Embedding Additional Documents 67
Printing Sheets 68
Exporting as PDF 68
Saving a Project 68
First Steps **70**
Getting Acquainted with *E³*.panel for Panel Wiring and Construction **70**
Opening an Existing Project 71
Defining Mounting Rails and Cable Ducts 74
Placing Devices 75
Wiring in the Panel 77
3D Panel Display 80
Navigating within 3D View using Mouse 81
Dimensioning the Panel 84
Outputting a Wiring List 84
First Steps **86**
Getting Acquainted with *E³*.cable for Cabling Devices **86**
Opening an Existing Project 87
Changing the Sheet Format 88
Entering Sheet Header Information 88
Placing Blocks 89
Placing and Connecting Device Elements 90
Plugging (Unplugging) Plug Connectors 92
Modifying the Cable Structure 93
Modifying Existing Drawings 94
Adding Text 94
Searching and Navigating within the Project 95
Defining Cables 95
Assigning Conductors 96
Placing Different Connector Views 97
Switching between Display Modes 99
Lists and Output Files 99
Cover Sheet with Table of Contents 100
Embedding Additional Documents 101
Printing Sheets 102
First Steps **104**
Getting Acquainted with *E³*.cable for Cabling Vehicles **104**
Opening an Existing Project 105
Opening Sheet 1 of the Project 106
Entering Sheet Header Information 106
Drawing a Subcircuit 107
Plugging (Unplugging) Plug Connectors 109

Modifying the Cable's Structure 110
Assigning Conductors 111
Defining Attributes 111
Adding Text 112
Placing Different Connector Views 113
Switching between Display Modes 116
Searching and Navigating within the Project 116
Lists and Output Files 117
Cover Sheet with Table of Contents 118
Embedding Additional Documents 120
Printing Sheets 121
First Steps **123**
Creating Symbols and Components with the *E³*.series Database Editor **123**
What is a symbol? 124
What is a component? 124
Preparations for Creating Symbols 124
Creating a New Symbol 125
Creating a New Component 130
Setting Levels 133
Special Components 137
Creating a New Connector 138
Creating a New Cable 145

series

First Steps

Getting Acquainted with the *E³.*series User Interface

Welcome to Your First Tour of the E³.series User Interface

By following the steps described below, you can familiarize yourself with the ***E³.*series** User Interface. This tour points out how simple and easy it is to work with ***E³.*series**.

Please read the respective foreword to the individual chapters since they will help you perform the described actions. The subsequent chapters on ***E³.*schematic**, ***E³.*panel**, ***E³.*cable** and the *Database Editor* should be read in conjunction with this chapter. They were written as completely as possible, so information may be repeated, as required, in each of the chapters.

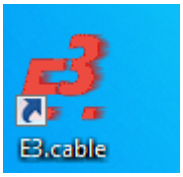
This Guide uses standard terms for defining the mouse button operations, e.g. **click** means press and release the left mouse button, **double-click** for two clicks of the left mouse button, **click and drag**, means click and hold the left mouse button while you move the pointer, and **right-click** for a click of the right mouse button.

Have fun with this introduction!

Starting *E³.*series

After installing ***E³.*series**, the ***E³.*series** icon will appear on your desktop.

Double-click on the icon to start the application.

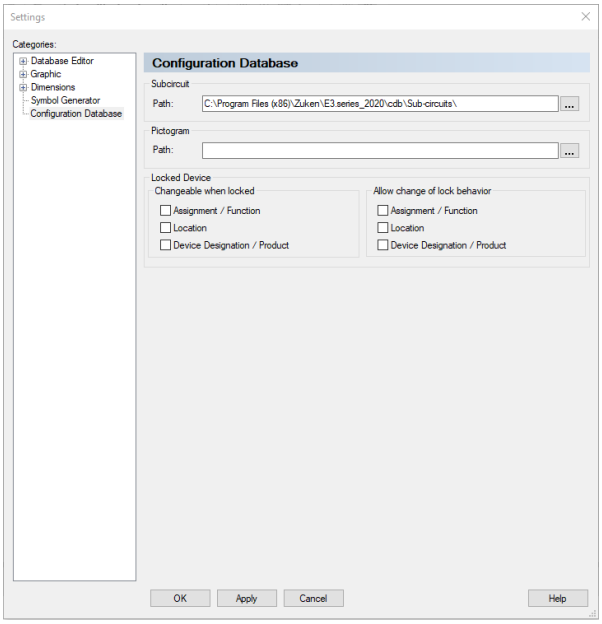


If the icon does not appear on your desktop, you can start ***E³.*series** with the command:
Start → Program Files → Zuken → E3.series 2020 → E3.cable.

The first time the program is started, the user interface will appear with the standard default settings.

NOTE: VALID ONLY IF US ENGLISH DEFINED AS INSTALLATION LANGUAGE!

To be able to access the subcircuits described in this introduction, the following must be configured:
Switch to Database Editor mode by selecting the **Tools → Start Database Editor Mode** command in the main menu bar.
Then specify the 'Subcircuit' path in the Settings (**Tools → Settings → Configuration Database**):



When everything has been defined, exit Database Editor Mode by selecting the **File → Exit** command in the main menu bar.

Opening a New or Existing Project

With ***E³*.series**, you always work within a project. A project can consist of any number of drawing sheets and additional documents, which are stored in a single file.

To create a new project, use the command

File → New or the corresponding icon

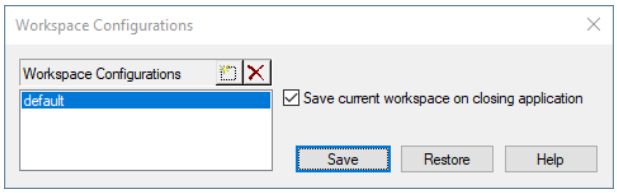
To open an existing project, use the command

File → Open or the corresponding icon and then select a project.

Note: The spelling of the main menu commands (e.g. ***FILE***) depends on the application look defined in ***E³***. (e.g. Microsoft Office 2013 – Gray). Depending on the style the menu commands are written in capital letters only or with capital initial letter (e.g. ***FILE*** and ***File***).

The user interface can be arranged or customized to fit your needs (i.e. to achieve the best possible working environment). User-defined settings regarding the user interface can automatically be loaded with every start of ***E³*.series**.

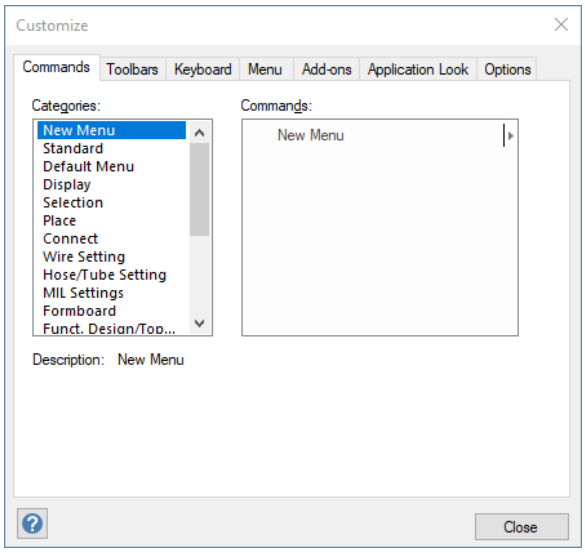
Check the **Save current workspace when closing application** option under **View -> Workspace Configurations…**. The next time you restart ***E³*.series**, the user interface automatically returns to the settings that were last saved. Using the **Restore** button it is possible to restore the workspace last saved.



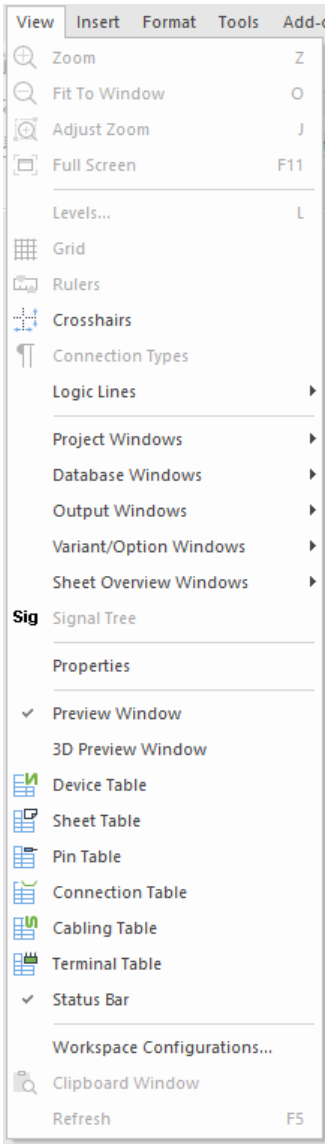
You can, for example, close the Output Window or reposition a toolbar. To close a window, click on the close icon in the window's upper right-hand corner.

To move a toolbar, simply click on the dotted vertical line at the beginning of the toolbar and drag the toolbar to the desired position.

To open or close a toolbar, create a user-specific toolbar or to shift, insert or delete icons within a toolbar, use the command **Tools → Customize...**

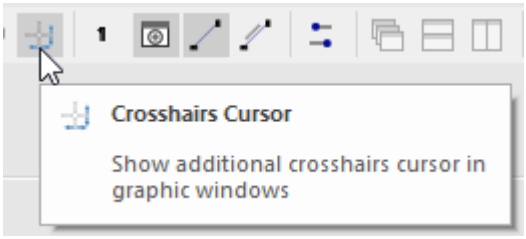


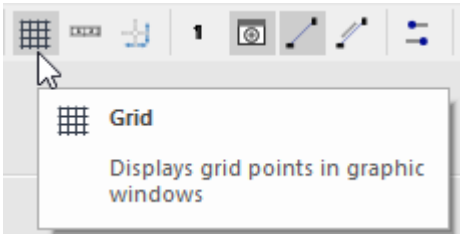
You can reopen a closed window in the **View** menu.



To make it easier to precisely place symbols and create connections, ***E³.*series** provides the option of displaying a crosshair cursor and placement grid as well as the normal cursor.

These can be activated with the icons and .



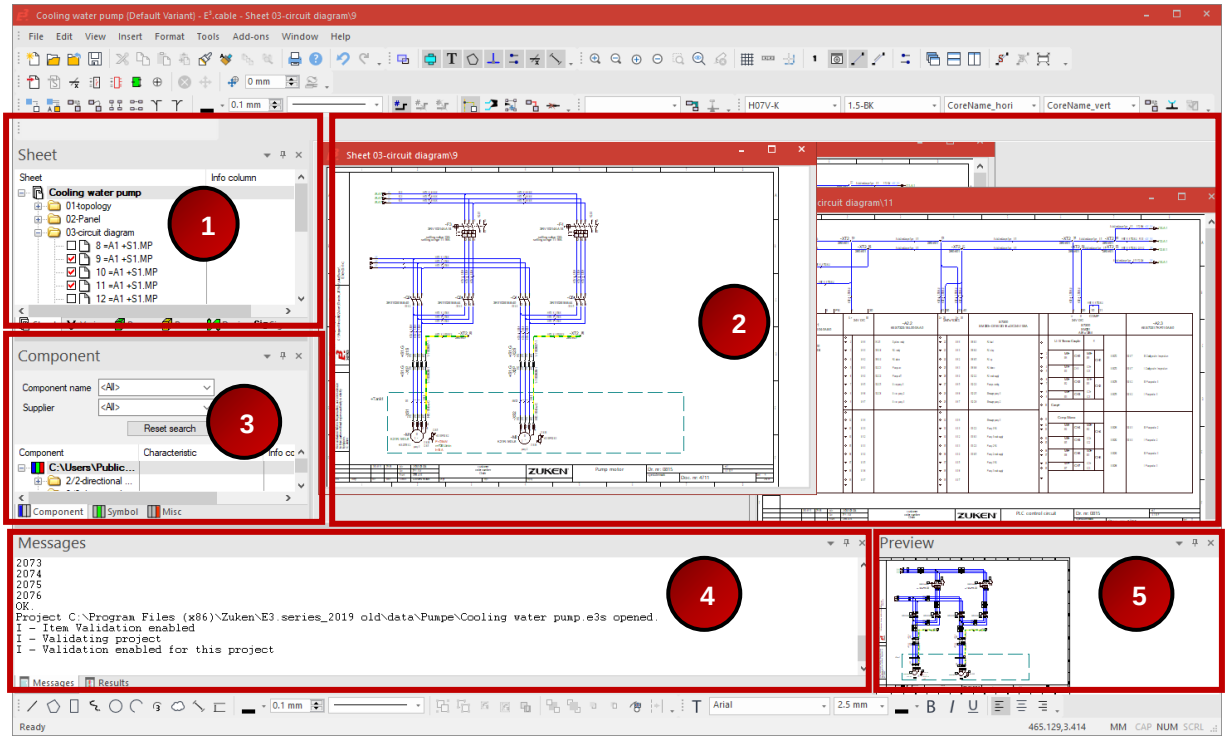


The Working Grid, Alternative Grid, as well as the View Grid can be defined independently from one another at any time in the **Settings** menu.

The *E³.*series User Interface

***E³.*series** is a pure Windows application that supports user functionality such as Drag & Drop and Copy & Paste. Many functions that you know about other Windows programs can be applied here. The complete documentation is available as online help and can be found by simply clicking on the Help icon or selecting **Help -> Contents** from the main menu bar.

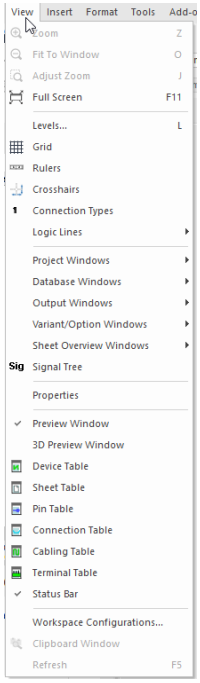
Let us begin with the program's user interface. It consists of various windows that show different views of the drawings and the internal, object-oriented database:



1. The **Project Window** displays all of the project's sheets and devices in a tree structure.
2. The **Workspace** displays the project's drawing sheets and other documents.
3. The **Database Window** shows the components and symbols of the database on which they are based.
4. The **Output Window** displays all messages and results.
5. The **Preview Window** allows you to graphically preview sheets and components/symbols.

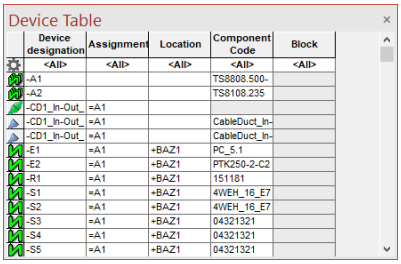
More Windows

Using the **View** menu in the main menu bar, it is possible to display the following table windows:



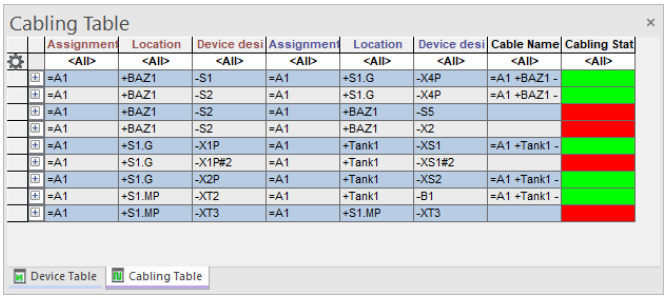
- **Device Table**

  Lists all devices used in the project.



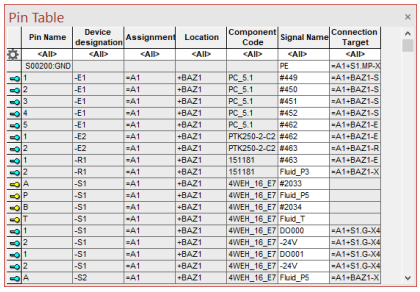
Cabling Table

Lists all graphically connected connections.



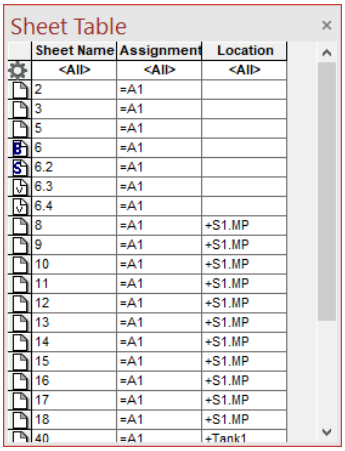
Pin Table

Lists all pins used in the project.



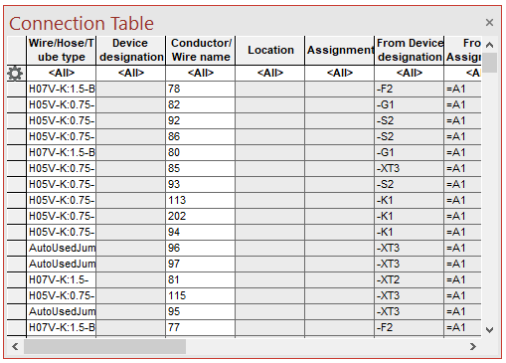
Sheet Table

Lists all sheets used in the project.



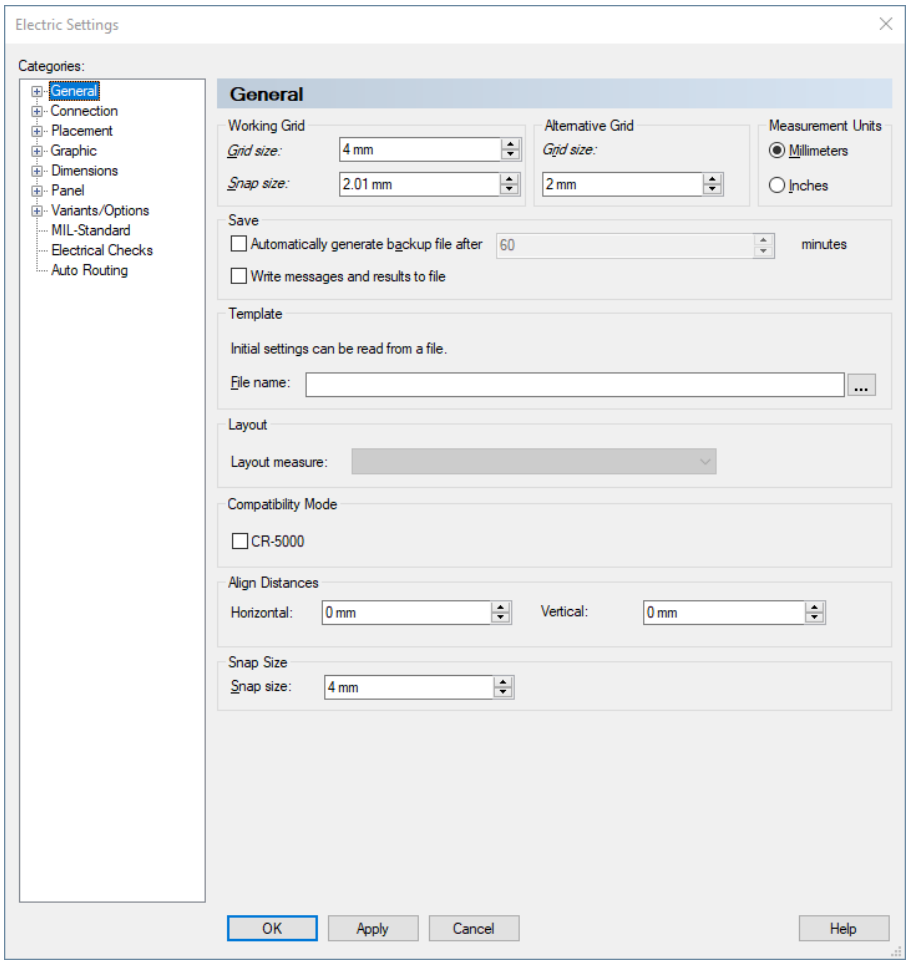
Connection Table

Lists all connections in the project.



The Settings Menu

The Settings menu is used to define various settings for ***E³*.series**.
The command **Tools → Settings** can be selected in the main menu bar and is only available when a project is opened.
Depending on the schematic type of the active sheet either the 'Electric' or 'Fluid' settings menu appears. Individual options for the two disciplines are handled separately and stored. These options are represented in the Settings menu by italic characters.



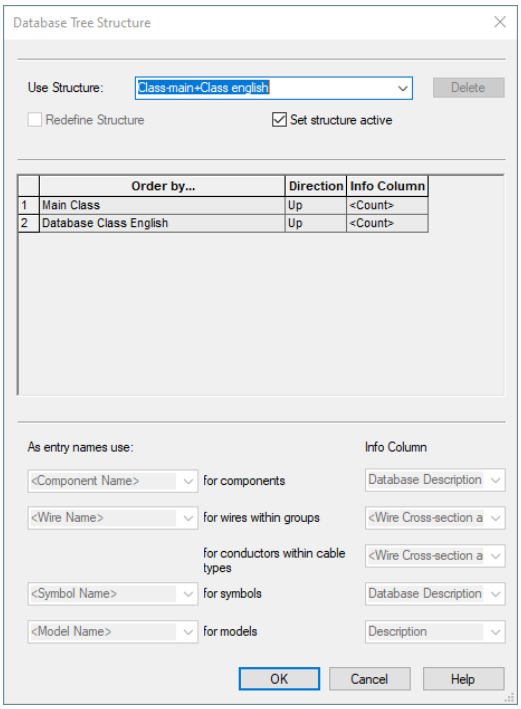
The settings for electric and fluid can be called up separately:
Tools → Electric Settings
Tools → Fluid Settings

Defining the Database Structure

Components, symbols and other objects used in ***E³.*series** are stored in a database.
The E3 database structure can be easily defined. Some common structures are already defined.
Right-click on the defined database name in the Database Window **Component** tab and select the **Tree Structure** command from the displayed context menu.

Select the structure template **Class-main+Class english** in the **Use Structure** section of the displayed **Database Tree Structure** dialog and confirm with **OK**.

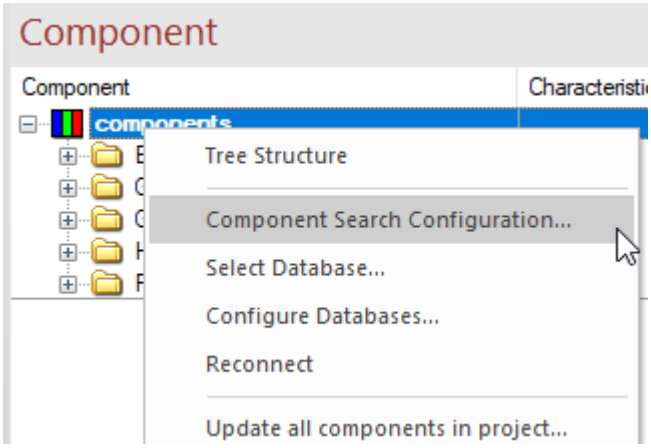
Your component database is now hierarchically structured. The first level corresponds with the main class and the next level contains the corresponding English classes.



Afterwards, switch to the Database Window's **Symbols** tab. Right-click on the defined database name and select the **Tree Structure** command from the displayed context menu. Select the structure template **Schematictype+Class_english**.

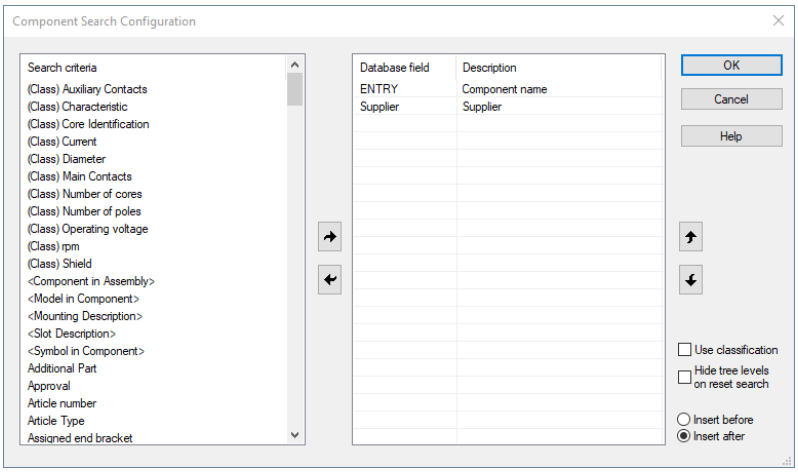
Locating Components

You can locate components either directly in the Component Database Tree or search for them by entering or selecting specific component attributes. To search by attributes, the search system must first be configured. To do this, right-click in the upper gray area of the Database Window and select the **Component Search Configuration...** command.



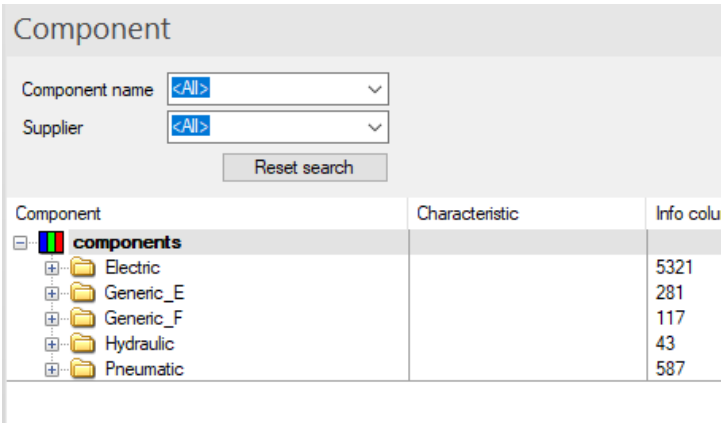
All available attributes in the component database will be listed on the left side and can be used as search criteria. The list on the right side shows which search criteria is used for the component search.

Select the attributes **COMPONENT NAME** and **SUPPLIER** from the left side of the menu and transfer them to the right side using the arrow button .
Confirm your configuration with **OK**. The **Component Search Configuration** dialog is closed.



The defined fields appear above the tree structure in the Database Window. From now on, the components will be displayed in the Component Tree based on the entries entered in the search fields.

Note: The attribute **Component name** defines the name of the component as it appears in the database.



Feel free to experiment with the functionality in ***E³*.series**. If you are not satisfied with the results, use the **UNDO** command (<**Ctrl+Z**> or the toolbar icon) to undo any actions.

If you wish, you can save the current drawing state as is (**File → Save or Save as...** command or the icon) and continue working on it at a later date.

Mouse Button Settings

When working with ***E³*.series**, we recommend using a three-button mouse. The mouse buttons are assigned the following functions:

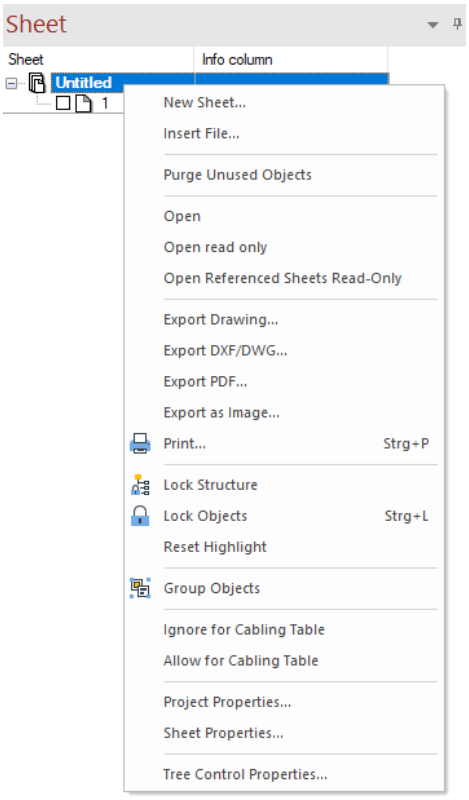
- **Left mouse button** *(select, move and modify)*
    - **Select objects:** Click on an element to select it. If you click on another element, that one becomes active and the previous one inactive. To select more than one element to be modified, hold down the **Shift** key and then left-click on each of the desired elements.
    - **Select objects within an area:** Hold down the left mouse button and drag the mouse to define a rectangular area. All objects within the area are selected when the mouse button is released.
    - **Move objects:** Left-click on an object and keep the mouse button pressed and move the object. Release the mouse button when the object appears at the desired position. If several objects need to be moved simultaneously, mark them as described above, click on one of the selected objects, and move all of them together.
    - **Enlarge and reduce the size of graphic objects:** Select the graphic and then click on one of the 'handles', hold down the mouse button and drag the 'handle' in the desired direction. Depending on the direction, the object is either enlarged or reduced in size. Additionally holding down the **Shift** key while dragging, maintains the aspect ratio.
- **Middle mouse button** *(zoom)*
  Press the middle mouse button and, drag the pointer diagonally to create a rectangle. The area selected will be enlarged to fill the window.
  The wheel on your mouse allows you to dynamically zoom on the sheet. Press **Ctrl** and scroll upwards to zoom in on the area. If you scroll back, the current area will be zoomed out.
- **Right mouse button** *(context menu)*
  Press the right mouse button to display a context menu. Apart from general commands, the bottom part of the menu contains the **Properties…** commands for the selected object. Context menus can also be displayed in the Project Window and the Database Window. Some examples can be found below.

Examples of Context Menus

After opening a project as described above, right-click on the project name in the Project Window to display a context menu that contains all the commands available.

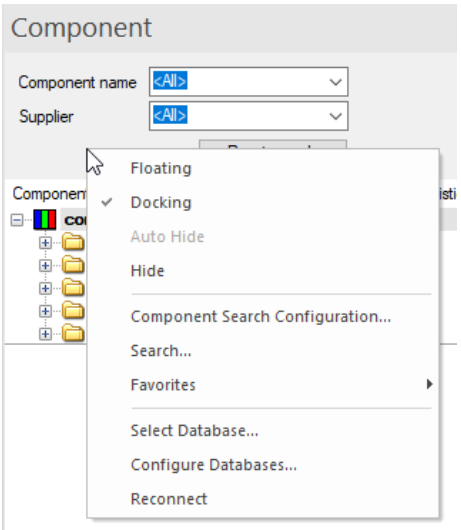
For example, in the case of a new project, you can create a **New Sheet...**.

In the displayed sub-menu, you can press the **OK** button directly.



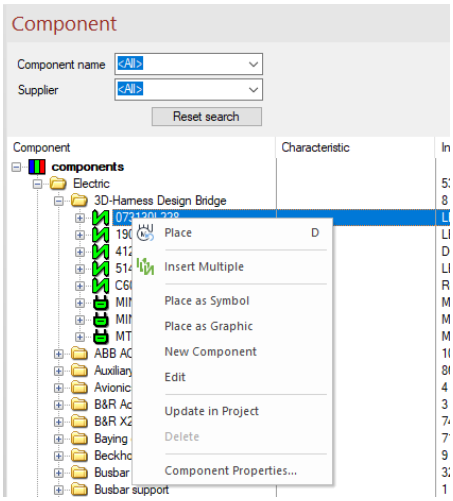
Note: Since the project is newly created, its project name is Untitled. ***E³.*series** creates a sheet in a new project depending on whether a corresponding file template is defined (**Tools -> Settings -> General**). Use this sheet, if available, or create a **New Sheet…** in the new project.

Click in the upper portion of the Database Window to display a context menu. The user can configure the database here or select a new symbol/component database.

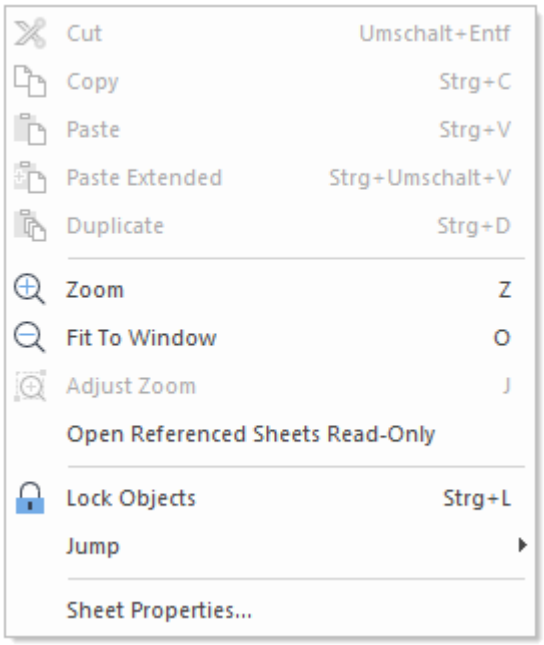


The following example shows the context menu for components and symbols from the database.
The functions **New Component** and **Edit** switch the user to the Database Editor mode. The Database Editor will be described later in a separate chapter.

The **Component Properties** command displays the selected component's properties as defined in the database.



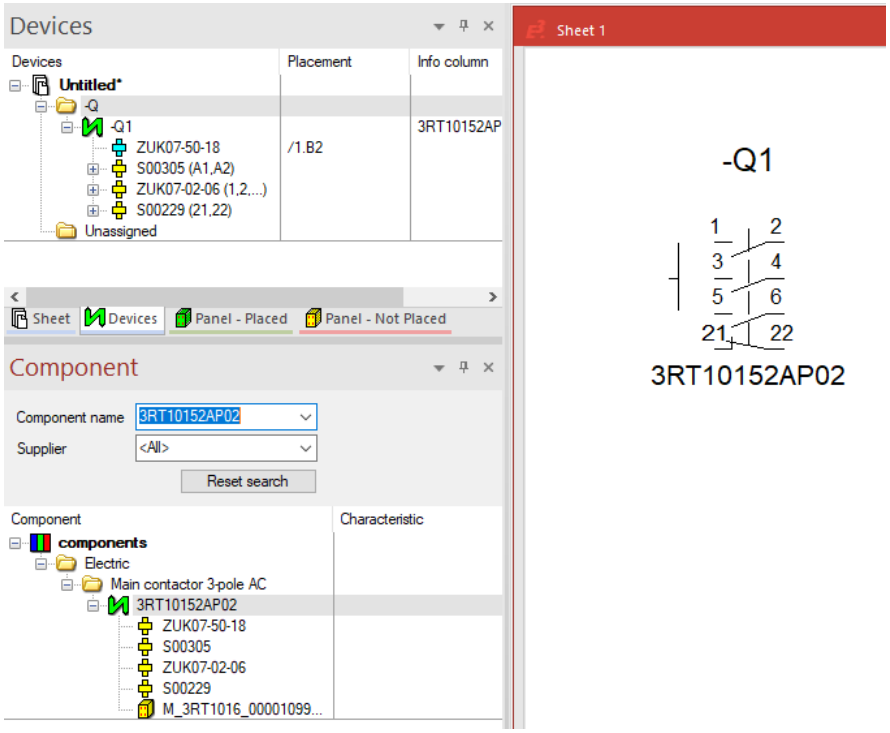
Context menus adapt themselves dynamically to the object selected. If nothing is selected, the context menu displays the standard commands along with the **Sheet Properties...** command.



Browsing in the Project

To browse through the project after opening a sheet, use the keys “page up” and “page down” to jump to the next sheet or the previous one.
The new sheet is displayed in the window of the previous sheet. The “**Pos1**” key jumps to the first sheet in the project and the “**End**” key to the last one.

Place the component **3RT10152AP02** from the class **Electric – Main contactor 3-pole, AC** on a sheet (see hint). Drag the component from the Database Window to the workspace or select the **Place** command from the component’s context menu. This contactor consists of four individual symbols and one model.
The component will be adopted from the database into the project automatically and assigned a device designation.
Next, place the remaining contactor symbols (coil**s** and contacts) on the sheet. To do so, open the Device Tree in the Project Window and drag the symbols from the device tree onto the sheet. Symbols, which have not been placed, are displayed in yellow in the device tree, those already placed displayed in blue.

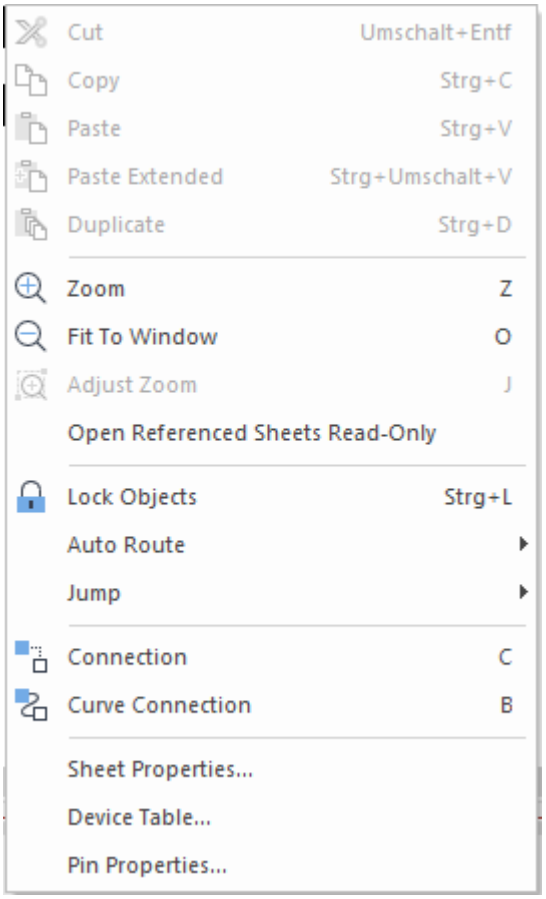


Right-click on one of the pins. A context menu containing additional menu items appears.

In addition to the **Sheet Properties…** command, the **Symbol Properties...** and **Device Properties…** commands are available for the object whose pin was selected.
The pin itself has **Pin Properties…**, which can then be accessed through this menu.

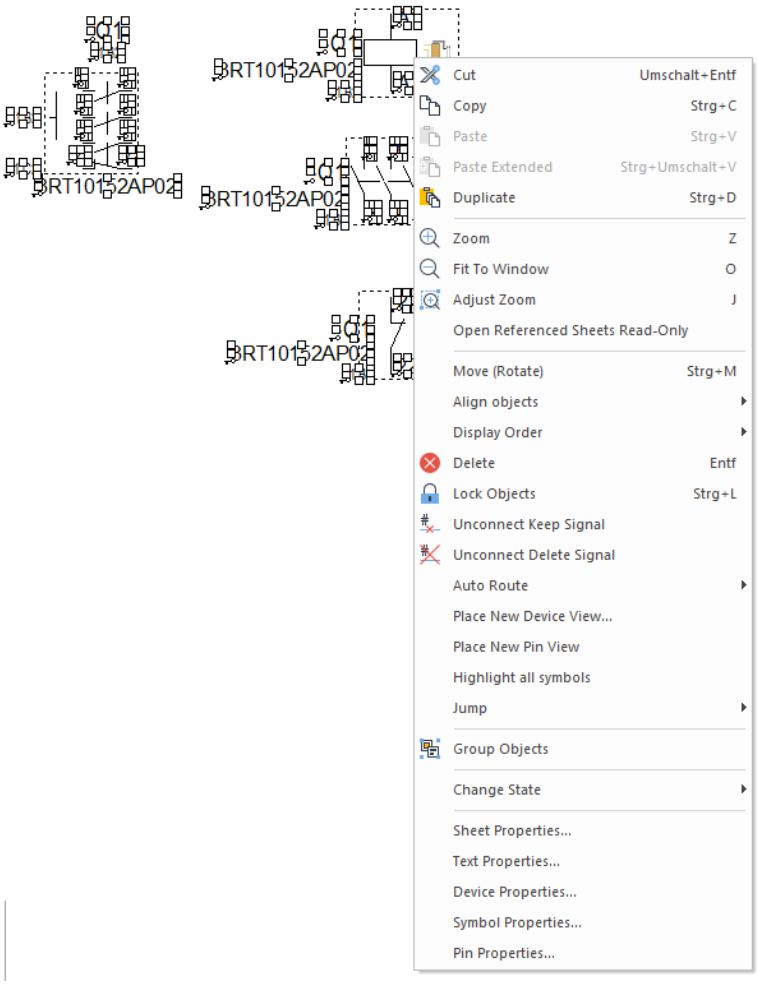
Besides these properties, additional commands have been added that refer specifically to the current object selected.
Context menus adapt themselves dynamically to all marked (selected) objects.

Note: The term "**device**" is used to signify a component that has already been placed in the project and can be found in the Project Window, whereas a "**component**" is selected from the Database Window.



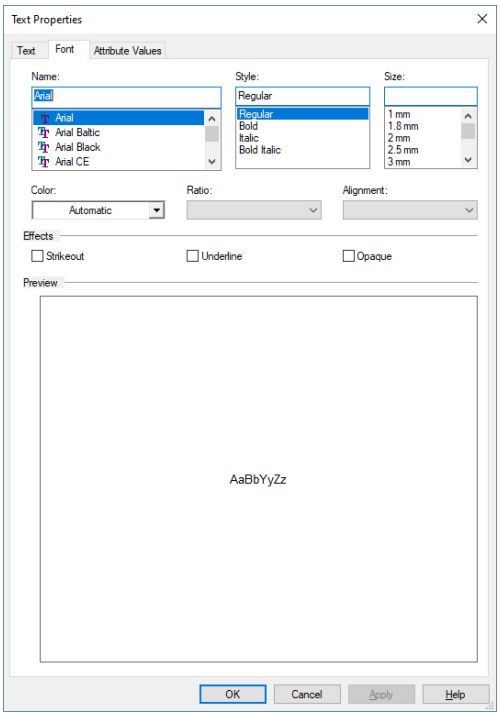
Select all the placed symbols by dragging a rectangle around them with the left mouse button. All of the elements inside the rectangle have now been selected.

Call up the context menu by right-clicking on any of the marked objects. Only those commands and properties pertaining to all of the objects selected are found in the context menu.



The property menus (e.g. Text Properties, Device Properties, etc.) display only entries or settings that the selected objects have in common. If a change is made, it applies to all the objects.

For example, all marked text can be assigned new properties using the **Text Properties** (*Font*, *Font Style*, *Size*, *Color* …).



Overview of the Toolbars

The ***E³*.series** user interface contains toolbars that offer quick access to several functions. To execute a command, simply click on the appropriate toolbar icon. The functions apply to selected objects or objects of a certain type.

The **Standard** toolbar contains the default functionality, such as *New* (new project), *Open*, *Close*, *Save*, *Cut*, *Copy*, *Paste*, *Paste extended*, *Graphic format painter*, *Attribute format painter*, *Duplicate*, *Show Clipboards*, *Print*, *Help*, *Undo* and *Redo*.

The **Text format** toolbar contains functionality used for texts.

New text can be inserted with the symbol . The text will receive the default values defined in the toolbar above, Font *Arial*, Size *2.5 mm*, Color *Automatic* and Font Style *Normal* (i.e. not bold, italic or underlined) and the alignment is left.

The **Graphic** toolbar contains functions that can be used to create and modify the various graphic elements.

The graphic elements are *Lines*, *Polygons, Rectangles, Curves, Circles, Arcs*, *Arcs (3 points)*, *Clouds*, *Dimensions* and *Running Dimensions*. In the example above, the object receives the predefined settings: color *Automatic*, line width *0.1 mm* and line style *solid*.

Additional properties for placed graphics, such as hatching, can be changed by right-clicking on the graphic and selecting the **Graphic Properties…** command in the context menu. Graphic objects can have predefined properties, which can be found in the **Tools → Settings → Graphic**.

The **Display** toolbar allows the user to navigate within the circuit diagram, to select components and symbols, remove selections; and for displaying the grid, ruler and crosshairs cursor, as well as connect points and open line ends.

The **Connect** toolbar is used to manipulate electrical connections. It contains commands for creating connection lines, and it helps modify (configure) the graphical connection lines.

Insert Connection: used to connect two pins.

Auto Route by selection: automatically connects two selected objects

Multi-Connection: creates multi-connection(s).

Insert Curve Connection: inserts curved connect lines.

Autoconnect Vertical: connects all of the pins vertically inside a rectangle.

Autoconnect Horizontal: connects all of the pins horizontally inside the rectangle.

Angle clockwise /Angle counterclockwise: Angles selected connection(s) clockwise or counterclockwise.

Generate Signals: The connection line is automatically assigned a system-generated signal when the button is pressed.

Use line properties of starting connect line: Adopts the parameters of an already existing graphic connect line when connecting to it.

The display mode for connection lines can be defined in the toolbar. In this example, the following values are defined: color *Automatic*, line width *0.1mm* and Line style *solid.*

The other icons in the toolbar are used for displaying open conductor connections and deleting connection lines.

Here is an overview of the commands used to place elements. The rest of the commands will be explained in the chapters regarding working with ***E³.*cable** and ***E³.*panel**.

Inserts a template symbol. The template symbol shows the contents of a connect line. The displayed contents depend on the attributes assigned to the connect line.

Inserts a field that can be assigned device designation, higher level assignment and location. Display of all device properties of devices within a field can be suppressed when identical with the properties of the field.

Places a dynamic symbol. A dynamic symbol is a box that can be assigned an item designation and pins.

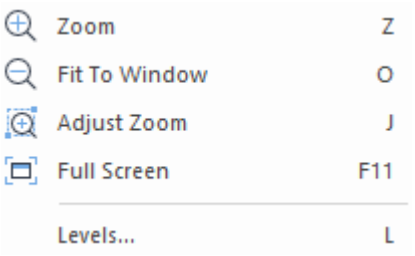
Places a block (***E³.*cable** only). A block represents either a functional unit that is not specified in detail or an electronic circuit's data.

Deletes symbols or components from the drawing sheet. While deleted from the drawing, they have not been deleted from the project and therefore, are designated as unplaced devices in the Project Window (indicated in yellow). To completely remove a symbol or component from the project, it must also be deleted from the Project Window.

All commands, which are contained in the toolbars, can also be found in the main menu bar. Toolbars can be customized to fit your needs by using the **Tools → Customize…** command.

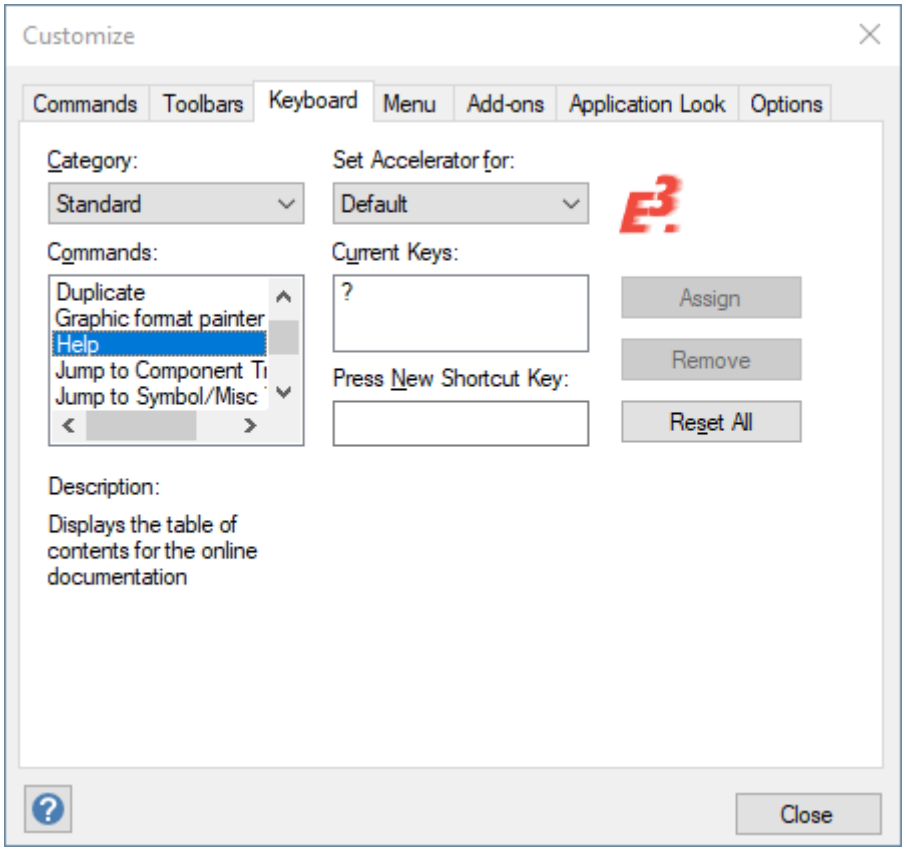
Shortcut Keys

Hotkeys are a series or combination of keyboard keys. Hotkeys allow quick execution of commands that are often used without navigating through menu trees. Commands, which can be activated by means of hotkeys, display the predefined hotkey definition to the right of the command name in their pull-down menus.
The **Settings** menu, for example, can be accessed by using hotkey **S**, the **Levels** dialog by using hotkey **L**.



Apart from the **ESC** key the configuration of existing shortcuts can be changed and new shortcuts can be defined for existing commands.

Using **Tools → Customize → Keyboard…** new or additional shortcuts can be created or already existing ones modified**.**



Predefined shortcut keys in E³.series

These shortcuts are defined in the E³ default application:

Menu	Command	Shortcut
File	New	CTRL+N
File	Open	CTRL+O
File	Save	CTRL+S
File	Save as	F12
File	Print	CTRL+P
File	Exit	Alt+F4
Edit	Undo	CTRL+Z / Alt+BACK
Edit	Redo	CTRL+Y
Edit	Cut	CTRL+X / SHIFT+DEL
Edit	Copy	CTRL+C / CTRL+INS
Edit	Paste	CTRL+V / SHIFT+INS
Edit	Paste extended	CTRL+SHIFT+V
Edit	Duplicate	CTRL+D
Edit	Move (Rotate)	CTRL+M
Edit	Delete	DEL
Edit	Change signal on selected pins	CTRL+W
Edit	Select all	CTRL+A
Edit	Search	F3
Edit	Reset Highlight	SHIFT+F3
View	Zoom	Z
View	Fit to Window	O
View	Adjust Zoom	J
View	Full Screen	F11
View	Levels	L
View	Refresh	F5
Insert	Last symbol	D
Insert	Dimension	M
Insert	Text	T
Insert	Connection	C
Insert	Curve Connection	B
Insert	Template Symbol	/
Tools	Settings	S
Misc	Input coordinates	P
Misc	Rotate	R
Misc	Rotate in any angle	CTRL+T
Misc	Mirror X	X
Misc	Mirror Y	Y
Misc	Jump	CTRL+J
Misc	Jump to tree	CTRL+left mouse button
Misc	Help	F1/SHIFT+?
Misc	Rename (not possible with all objects)	F2

You should now be familiar with the basic operating procedures in ***E³*.series**. The following chapters describe user examples to offer more information in detail.

Enjoy working with

series

First Steps

Getting Acquainted with *E³.*schematic for Electrical Engineering

(in combination with *E³.*cable)

Welcome to Your First Tour of E³.schematic - in Combination with E³.cable

In this chapter we will introduce you to the main functions of ***E³*** and show you how to create reliable circuit diagrams for electrical engineering projects.
Please read through the respective forewords of the individual chapters, which will then allow you to perform the described actions. Information, pertaining to basic operations and the User Interface, can be found in the chapter *The E³.series User Interface*.
At the end of the tour, you will have learned about the various functions of ***E³.*****schematic**, which can then be applied to your design needs.

Have fun with this introduction!

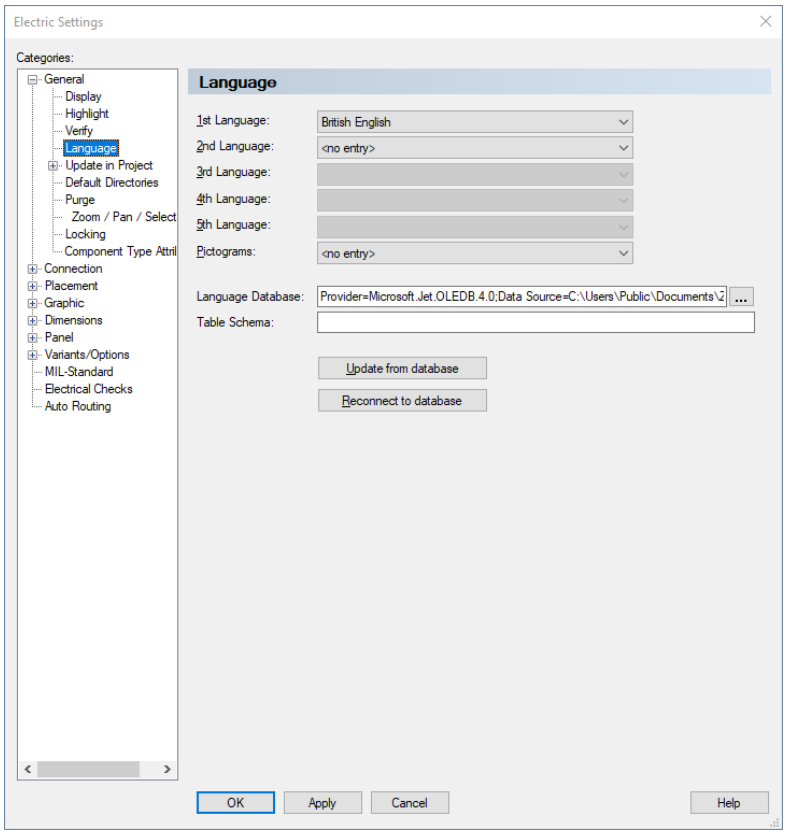
Opening an Existing Project

With ***E³.*****schematic**, you always work within a project. A project can consist of any number of drawing sheets and additional documents, which are stored in a single file.

To open the project, use the command

File → Open or the corresponding icon in the main menu bar and select the **Cooling water pump.e3s** project from the directory C:\Users\Public\Documents\Zuken\E3.series_2020\data\Pumpe\.
Note: The folders "***User***", "***Public***" and "***Documents***" are displayed in Windows Explorer with the language-specific names. The file path works with the English and user-defined language.

Now define the project language: British English.
Open the **Tools → Settings → General → Language** dialog and select **British English** as the first language.

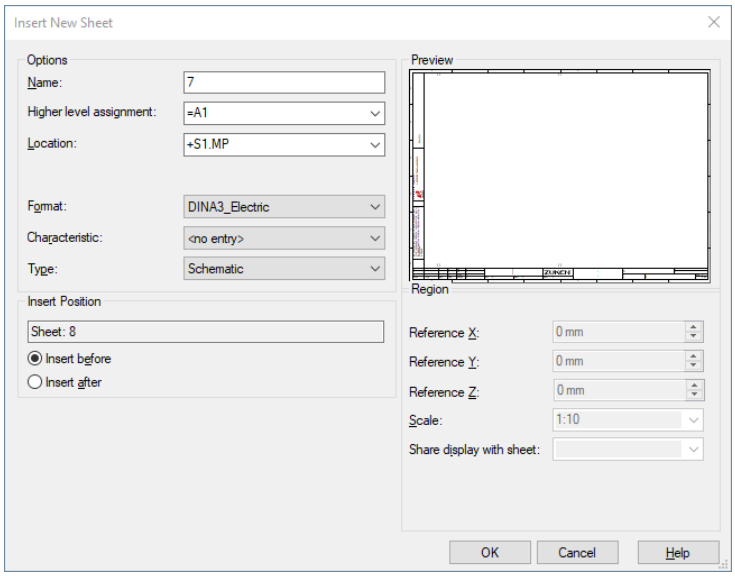


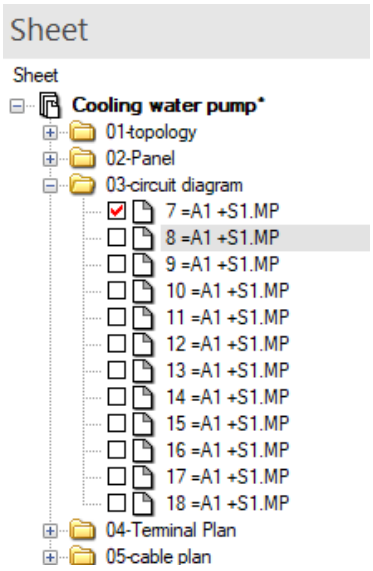
Next, you are going to add sheets, symbols, devices, subcircuits, texts and so on to the project.

Inserting a New Sheet in the Project

Insert a new sheet at the beginning of section **03-Circuit diagram**.

Open the folder **03-Circuit diagram** in the Project Window **Sheet** tab, right-click on sheet **8** and select the command **New Sheet** from the displayed context menu. The **Insert New Sheet** dialog appears.
Enter **7** as the sheet name, **=A1** as the higher level assignment, **+S1.MP** as the location and then select the sheet format **DINA3_Electric**.
Check to make sure that you have selected **Insert before** and press **OK**.
The new sheet **8=A1+S1.MP** will be inserted into the tree structure before sheet **8** with the desired format.





Different sheet formats (e.g. DIN, extra-wide, Ladder Diagram, etc.) with their unique size, layout and sheet header information can be stored in the database.

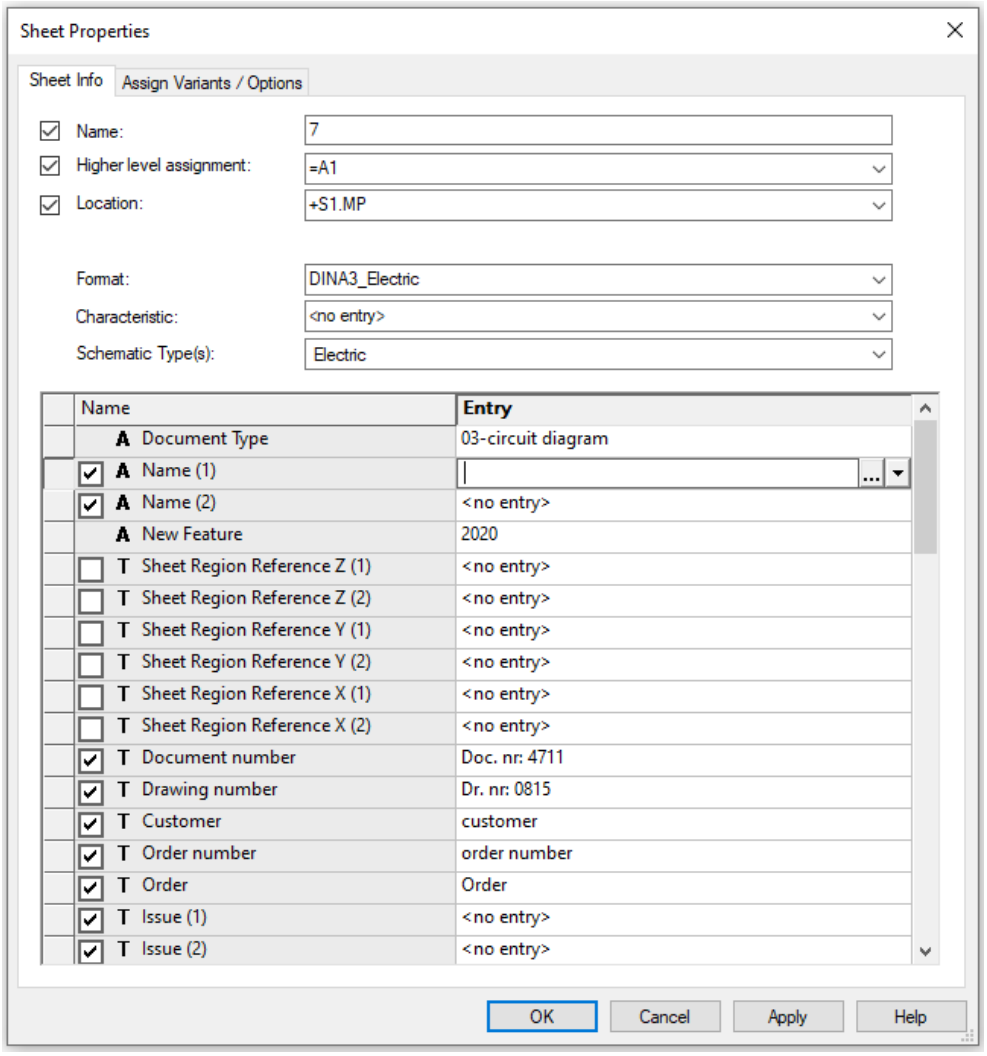
Entering Sheet Header Text

Sheet headers contain information typically found in the drawing sheet's margins, e.g. Drawing Title, Drawn by, Date, Revision, and so forth. The sheet headers are displayed in **the Sheet Properties…** menu, which can be called up with the context menu by right-clicking anywhere on the sheet.

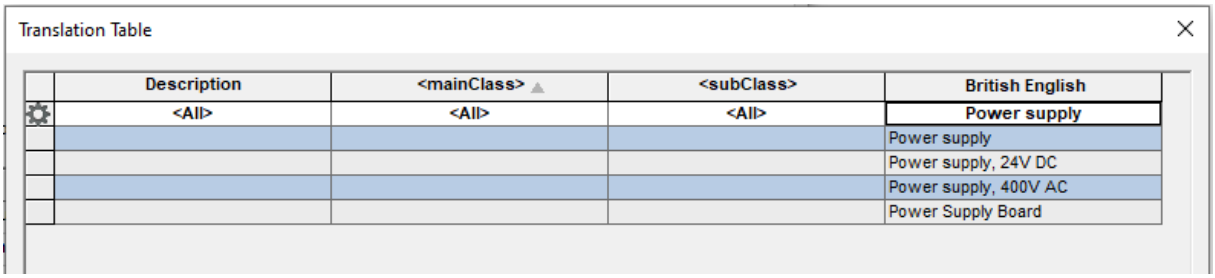
Modify the properties of the newly created sheet.
Overwrite the attributes ***Name (1)*** and ***Name (2)*** with the texts ***Power Supply*** and ***Main Switch*** from the text database.
The texts are entered at the predefined locations like defined in the sheet format.

First delete the entry from ***Name (1).***

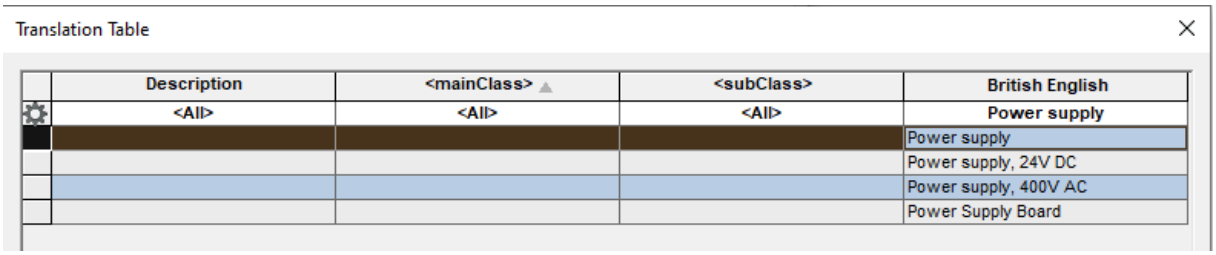
To use texts from the text database, double-click in the **Name (1)** entry field and press the ... button to open the **Translation Table** dialog.



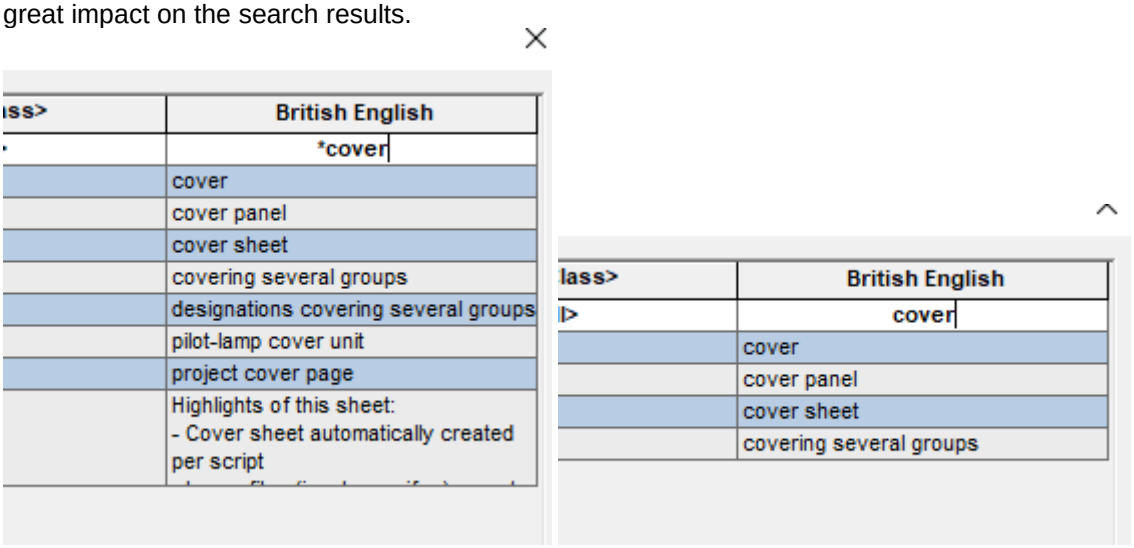
Enter the text “Power supply” in the *British English* column’s **Filter** field and then press RETURN.



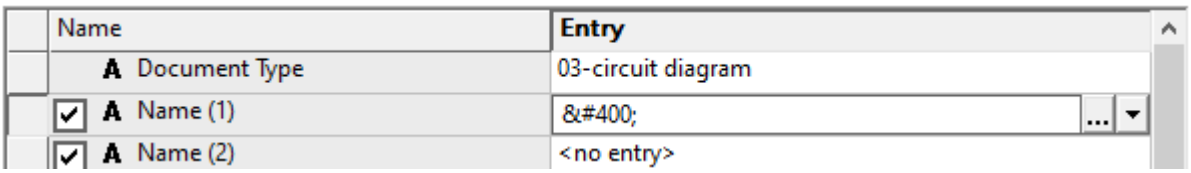
Select the text line from the filter results and press **OK** or double-click on the desired line.



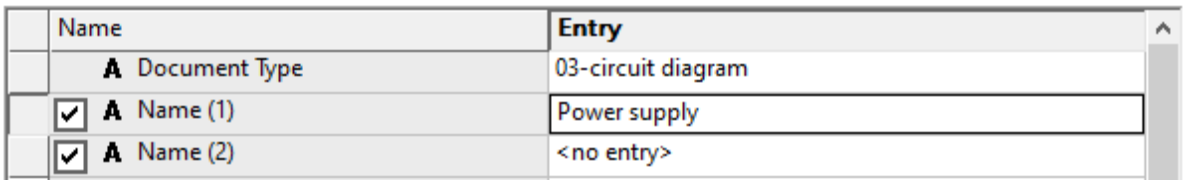
Note: The * character can be used as a wildcard.
The wildcard * represents any number of characters. Wildcards in search expressions thus have a great impact on the search results.



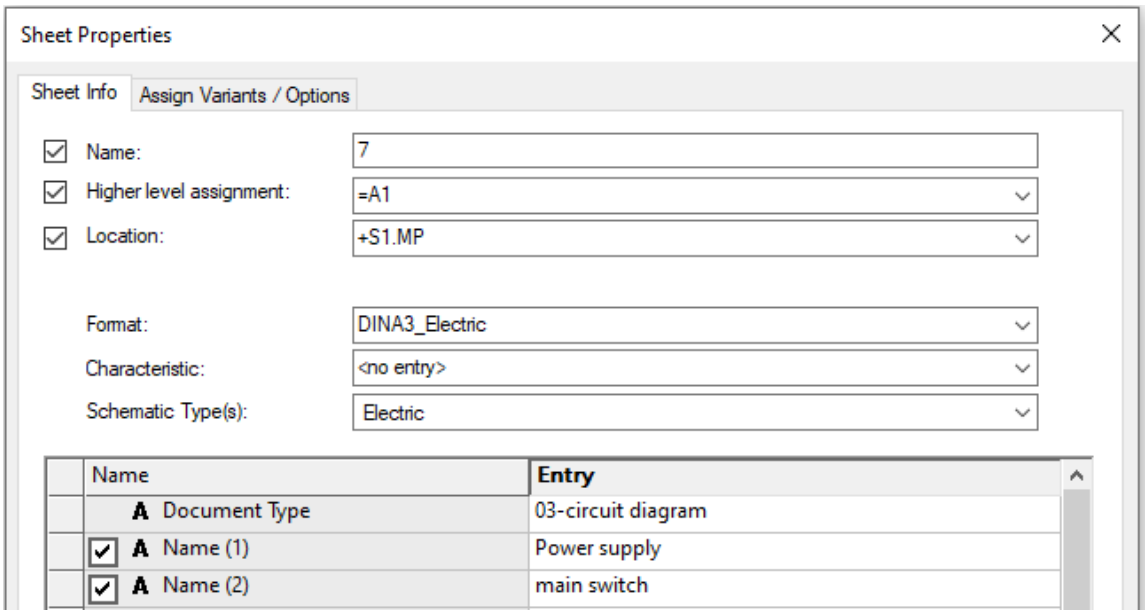
The text **Power supply** with its ID number 400 is transferred to the **Name (1)** attribute. The characters **&#** and **;** include the ID.



When leaving the entry field, the clear text ‘Power supply’ is displayed.



Now enter the text ‘main switch’ for the attribute **Name (2)** in the same manner.
Check the entries: **Higher level assignment** should be **=A1** and the **Location +S1.MP**.

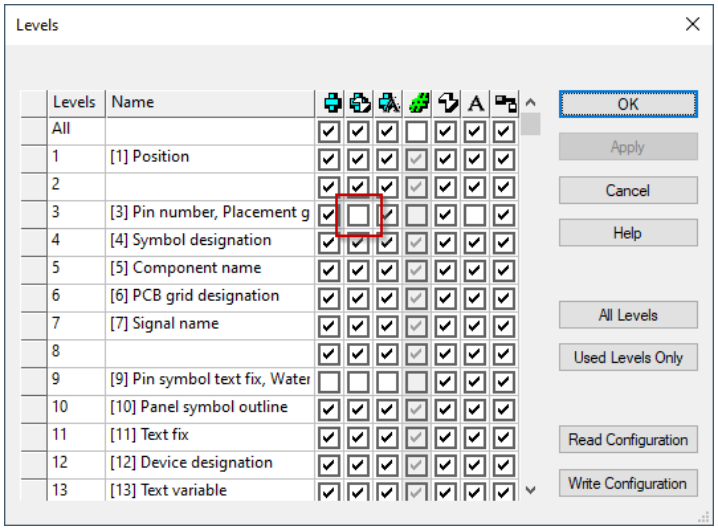


Displaying Placement Help Lines

It is possible to place subcircuits more easily using placement help lines, which can be switched “on” or “off” using the **Levels** dialog.
It is also possible to assign names to the individual levels. To open the dialog, use the **View --> Levels** command or simply press the key **L**.

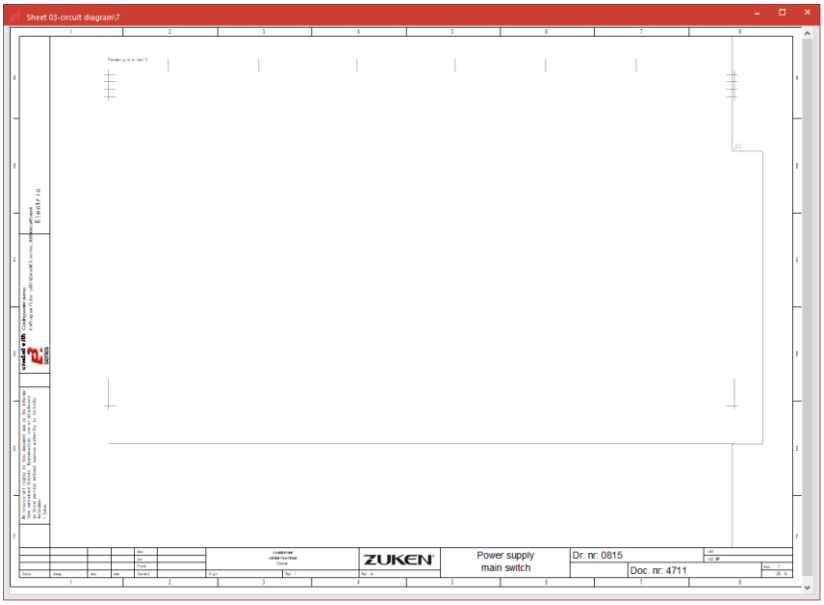
In the following Level Menu, level **3** should now be activated.

Possibility 1: Check the box (clicking on it)



Possibility 2: Import a predefined file containing the information required.
To do so, select the **Read Configuration** command and then the **Placement grid ON_e.vis** file from the directory `...\data\`.

Confirm your definitions by pressing **OK**. The **Levels** dialog is closed and help lines will appear on the sheet making it easier to place the subcircuits.

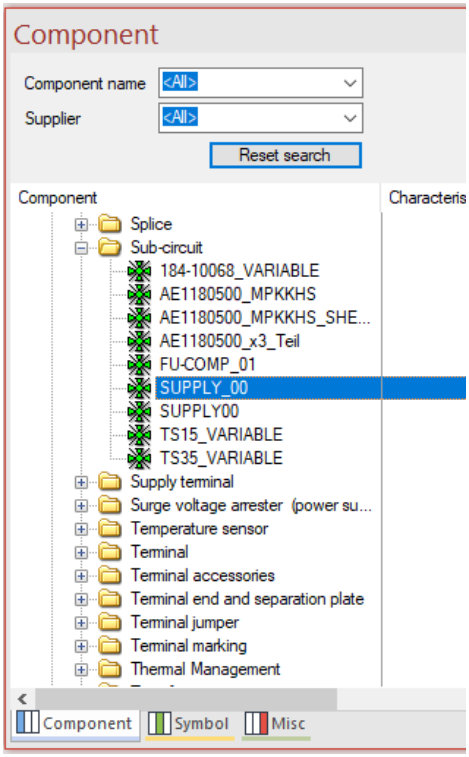


Using Predefined Subcircuits

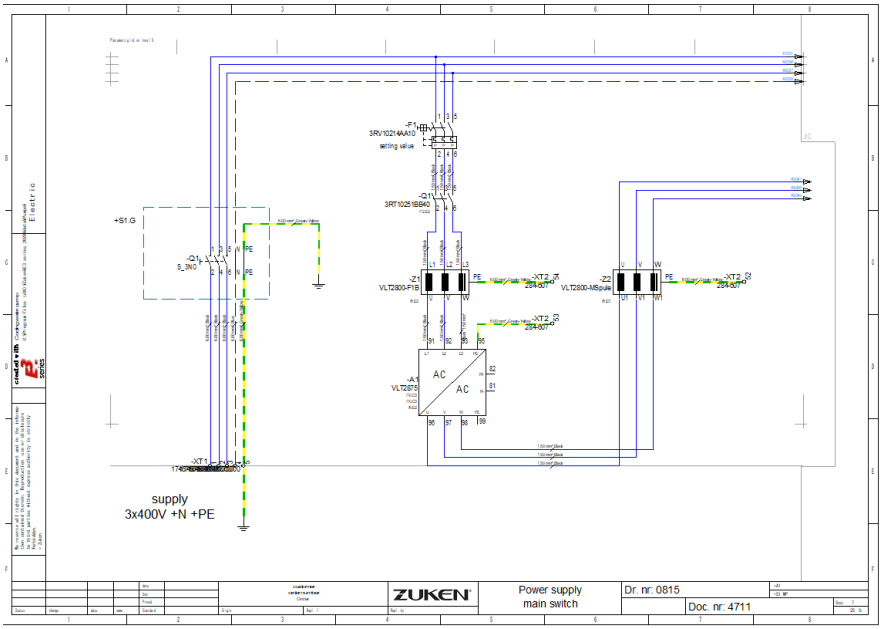
Predefined subcircuits are stored in the ***E³.*schematic** database. You can add these predefined subcircuits to your project.

These subcircuits are located in the **Electric– Sub-circuit** folder in the database window. Click on a subcircuit's name to see the subcircuit in the Preview window.

First, select the subcircuit **Supply_00** and drag it from the database window onto **Sheet 7**. The subcircuit is displayed at the cursor's position and will be placed on the sheet after releasing the mouse button. Place the subcircuit so that the signal references end up on the help lines in the upper right-hand corner.



Next, select the subcircuit **FU-COMP_01** and place it so that the open connection points fall on the 3 phase supply lines.



In ***E³.*schematic**, subcircuits are inserted intelligently into a drawing. This means that open connect points will be automatically connected to existing connection lines and the device designations, that are already contained in the subcircuit, will be checked and updated, as required, online – without delay!

The system automatically recognizes connection cross-references and enters them.

To select a complete subcircuit, left-click on a device of the subcircuit or right-click on a device to open the context menu. The subcircuit must not be selected. Then select the **Select Group** command from the displayed context menu.

With subcircuits, a new circuit diagram can be created, that is checked and based on approved standards.

Assigning Symbols to Components

***E³.*schematic** supports both symbol-oriented as well as component-oriented work.

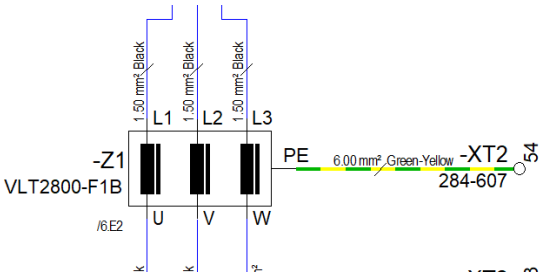
What is a symbol? A symbol is a circuit element that can be assigned to an existing component. Symbols can be used in any combination in a device. Symbols can be assigned any text the user chooses. Only when the device is assigned to a defined component, the symbols inside the component will be checked and the symbol text (e.g. pin names) automatically adapted.

What is a component? A component is simply a set of predefined symbols supplied with pin numbers. However, components can also be assigned part numbers, supplier names and so forth in the database. The advantage of using components is that the pin numbers are checked from the very beginning, which makes it impossible to overload a component (e.g. using more contacts than the relay has available) or to use incorrect symbols.

The components and symbols used in the subcircuit **Supply_00** are assigned corresponding component codes (article numbers). When you want to assign other component types, ***E³.*schematic** supports by only offering valid types from the database.

Zoom in on the area where the Coil **–Z1** is located.
Open the context menu by right-clicking on the symbol and select the **Device Properties…** command.

The **Component** section under the **Device** tab contains the component **VLT2800-F1B**. Click on the arrow in this field's right-hand margin and a list of all component types from the database will appear that are compatible for the symbol used in this device.



Select the component type ***VLT2800-MSpule*** and click **OK**. A message appears saying that some pins have new signals. The device **-Z1** is now assigned the component type ***VLT2800-MSpule***. If a parts list is created or a panel constructed, the correct component will be used automatically.

You can also click on the field

instead of the arrow button. The **Select Component** menu appears. A list of all components is displayed as structured as in the database window. It is possible to search for a component using the same filter methods as in the database. It can easily be decided, using the component description, whether it is the appropriate component.

The same procedure is carried out with the terminals of terminal strip **–XT1**.
Click on terminal **1** of **-XT1**. The complete device is selected. Click again on terminal **1** to remove the selection from the complete device and select terminal **1** only. Press the **Shift** key and left-click on terminals **2** and **3**. Open the **Device Properties** dialog and assign them the component type **1608620000**. Assign terminal **4** the type **1608630000** and terminal **5** the component type **1608670000**. Instead of selecting a component type from the list, the number can also be entered directly. This may be quicker than searching through the list.

Creating Terminal Plans in *E³.*schematic

***E³.*schematic** can create terminal plans that are always up to date according to the schematic drawing When working on your drawing, the terminal plan is updated in real time.

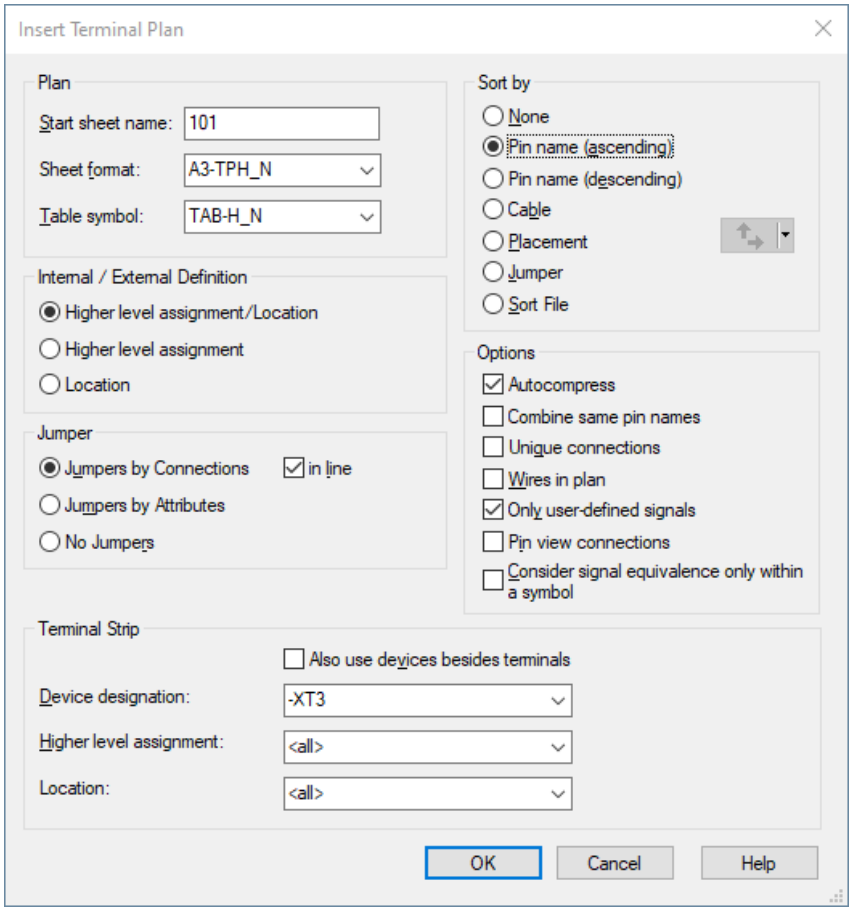
The terminal plan is inserted with the **Insert → Terminal Plan...** command from the main menu bar.

In the menu, you can define several properties for the terminal plan, such as the sheet format, display type, page number to start with, and the terminal strip, for which the terminal plan is being created.

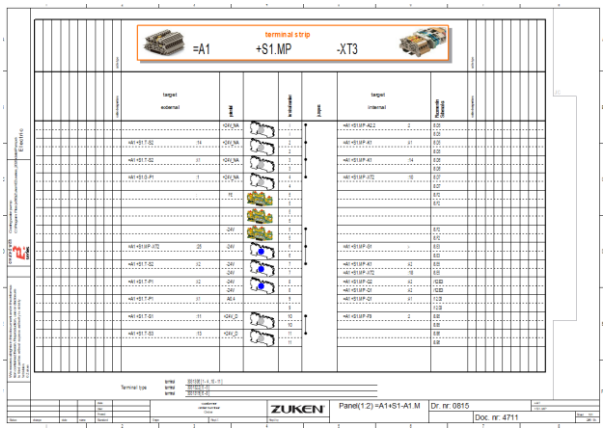
If no terminal strip is entered, all terminal strips contained in the circuit diagram are listed in the terminal plan.

It is also possible to define whether you want to reduce the number of terminals used by automatically placing the connections on both the internal and external sides (Autocompress), and what the criteria for generating the jumpers will be.

Enter the sheet name **101**, the sheet format **A3-TPH_N**, the display type (table symbol**) TAB-H_N** and the terminal strip **–XT3**. Also select the **Pin name (ascending)** option. Confirm your definitions with **OK**.

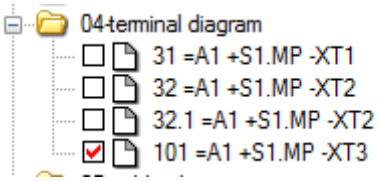


After entering the information in the Terminal Plan menu, the terminal plan for the terminal strip **–XT3** will be generated for the first time. From now on, all modifications concerning this terminal strip, regardless of which window you are working in, will be updated immediately in all displays (circuit diagram, terminal plan and project window) – without having to press another key and without any delay.



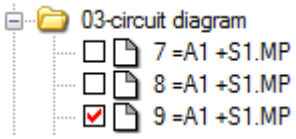
Right-click anywhere on the sheet **101** and select the **Sheet Properties...** command from the displayed context menu. Overwrite the **Name (1)** attribute with **Terminal Plan -XT3**. Select the term *Terminal Plan* from the Translation Table.

Then switch to the **Document Type** attribute and double-click in the field entry, click on the arrow and select **04-Terminal diagram** from the drop-down list of the **Entry** field. Confirm with **OK**.
The sheet is moved automatically to the *04-Terminal diagram* folder.



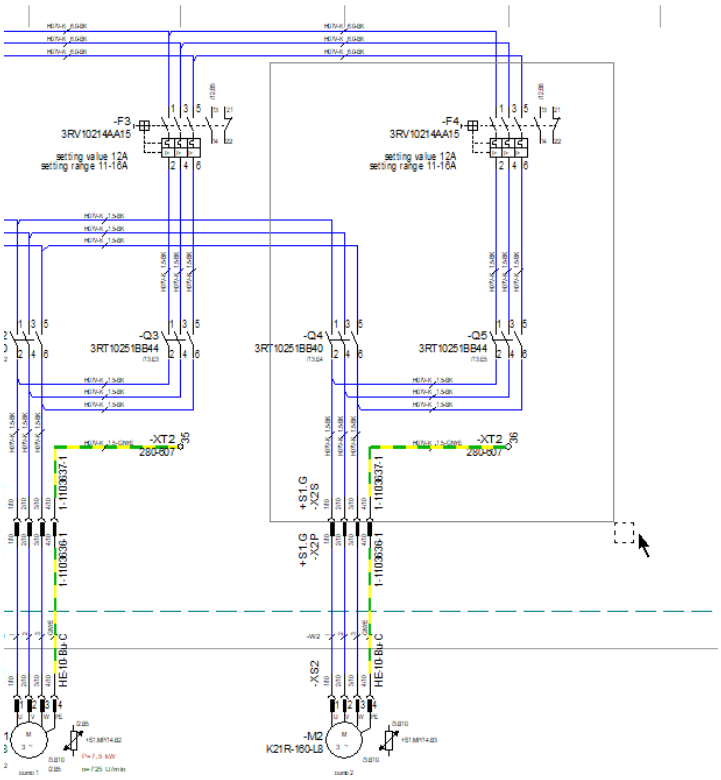
Editing Areas, Placing Devices and Laying Connections

Select the Project Window's Sheet tab and open sheet **9** by clicking on the box located in front of it.



Install an additional pump on this sheet. With regular working on a project, this can easily be accomplished by copying the second control pump and then connecting it. For training purposes, you will create one part by copying as well as placing and connecting the other parts manually.

Mark the area, as shown in the figure, by dragging a rectangle around it with the left mouse button.
When you have selected not all required or too many objects, you can adjust your selection by holding the **Shift** key and clicking on the respective objects that are either missing or selected by accident.



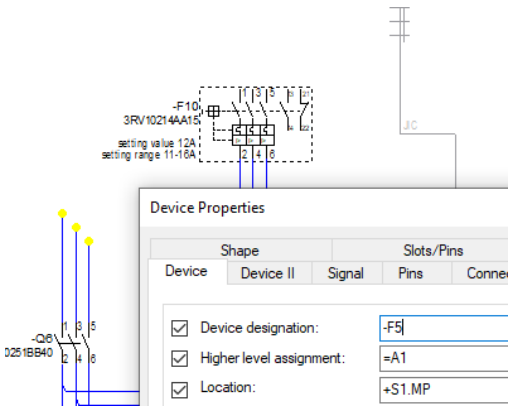
Make sure you include the bushings from connector **–X2S** but not connector **–X2P**. Also copy the outgoing conductors on pins **1**, **3** and **5** of the device **-F4**.

Use the *Copy* icon from the toolbar or the hotkey combination **CTRL-C** to copy the subcircuit. The selected elements are copied to the clipboard and can be pasted at another position.

Place the copied part with the *Paste* icon or **CTRL-V** onto the right side of the sheet. When copying as with pasting, ***E³.*** observes that no device designations are duplicated or that existing devices are crowded.

Activate the *Mark line end connect points* button in the toolbar to highlight open connect points in yellow.

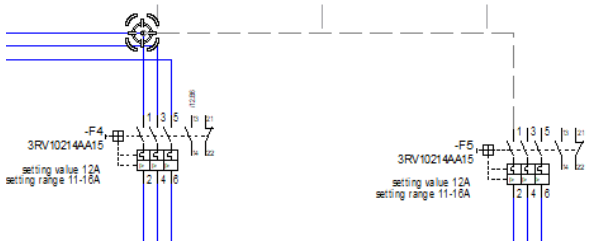
Rename the motor circuit switch **–F10** to **–F5** in the **Device designation** field of the **Device Properties….** dialog displayed by right-clicking on the respective switch.



The missing connection lines must now be drawn. Electrical connections are treated as “intelligent” connections because each of the connections possesses various attributes.
With manual connections, the corners are set automatically. Route connections (forced wiring connections), which are required for exact wiring representations can also be created just as quickly and easily. To see the different methods, create connections directly and others as “routes”.

To draw several connections, double-click on the *Insert Connection* icon . A single click on the icon activates the connection mode for one connection only. Notice how the cursor’s shape changes.

Position the cursor on the open line end of the motor circuit switch **–F5**. As soon as the cursor is positioned on a possible pin or connection line, its shape changes once again and the start point of the connection can be set. To do so, left click to start the connection. If corners are required in the connection, these can likewise be defined with a click of the mouse. However, ***E³.*** can automatically draw corners.

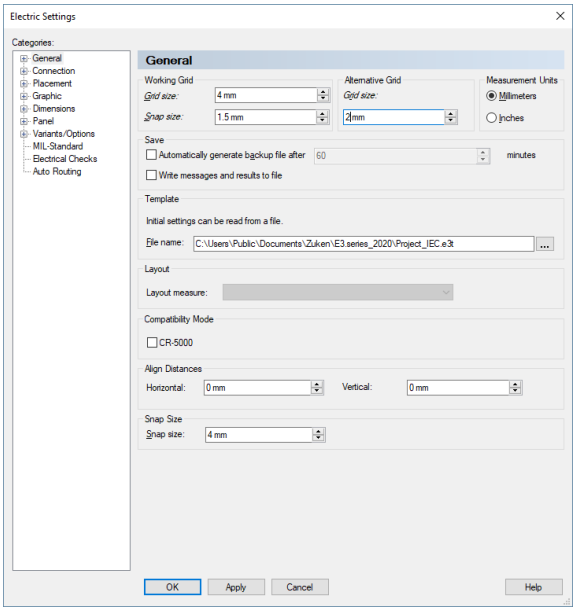


As you move the mouse, you will notice the connection being drawn dynamically. If the cursor is positioned over a possible pin, its shape changes again and by clicking the mouse button, the connection will be closed.

Now, move the cursor to the left to the existing corner point of the corresponding connection and close the connection by left-clicking.

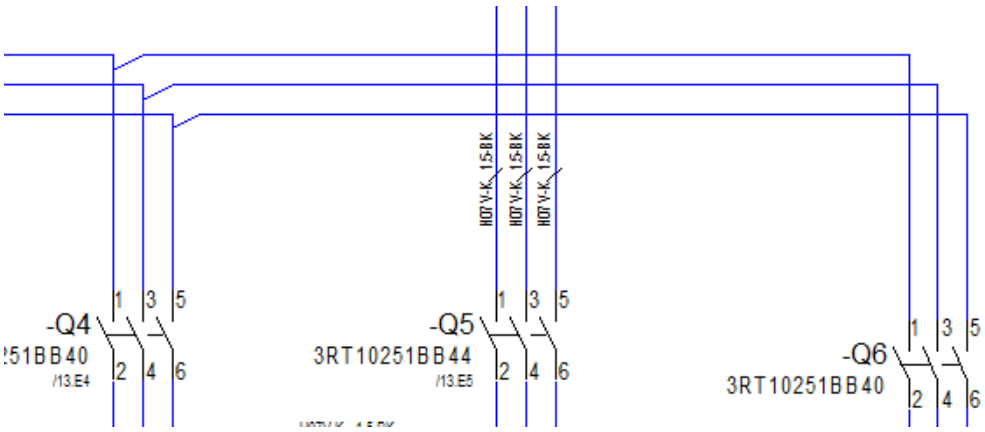
Because you are still in the *Insert Connection* mode, use the same procedure to draw the two remaining connection lines. Press the **Esc** key to terminate the connection mode.

The rest of the missing connections will be drawn as “Route” connections.
The module **E³.panel**, which is also used for the panel wiring, makes sure the wiring is defined correctly, even if the wiring sequence is not clear in the circuit diagram.
Since the angled (skewed) connections do not lay in the **4 mm** grid used so far, we must define the grid size as **2 mm** in the so-called **Alternative Grid (Tools --> Settings… --> General)**.



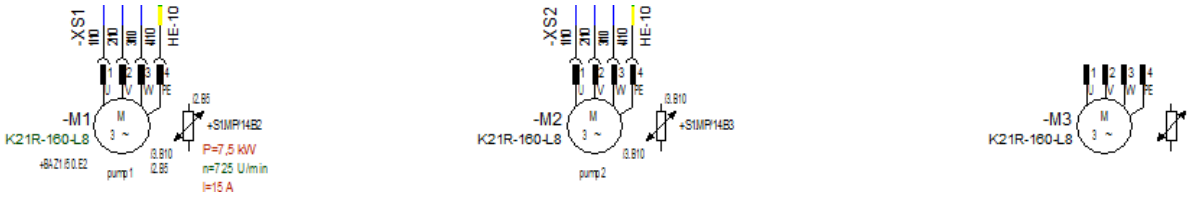
This “second grid” is activated using the **Ctrl** key.

By default, connection lines are drawn at right angles. Sometimes, however, tapered connections are required. These can be creating by keeping the **SHIFT** key pressed while connecting.
To draw these “route connections”, switch to the connection mode as described above. Click on the first open connection line of contactor **–Q6**, move the cursor to the left stopping shortly before you reach your objective **-Q4**, press the **Ctrl** key and click the left mouse button to mark the corner. While pressing the Ctrl key, also press the **SHIFT** key and position the cursor beneath the connection line. You will see the angled connection form, which can be closed by pressing the left mouse button once again. Follow the same procedure for the two remaining, open connections.

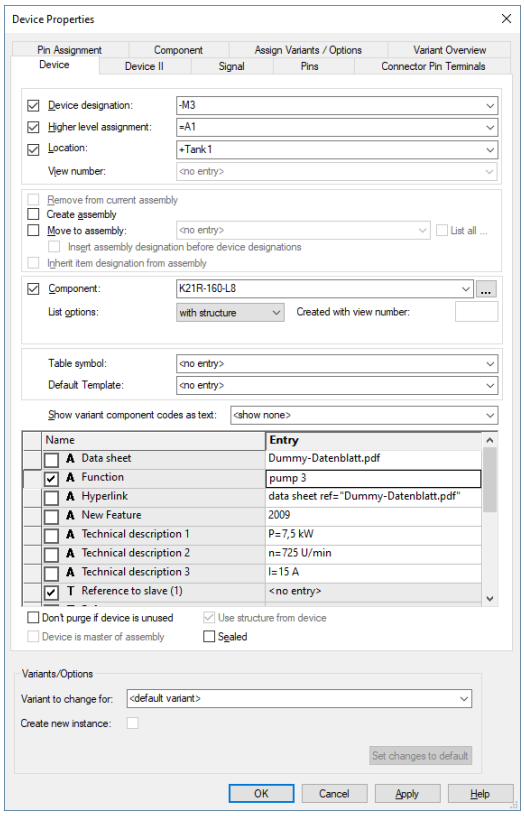


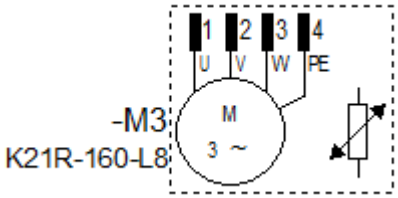
We now have to place the third pump that is missing in our circuit. Open the **Electric - Motor** folder in the Database Window’s **Component** tab and select the component **K21R-160-L8**. Place the motor (the first element of the component) by dragging it to the desired location on the sheet, next to **Pump 2**.
As soon as an element is clicked on in the Database Window, it will also be displayed in the Preview Window to see whether you have found the right symbol.

By placing any element of a component, the entire component will be loaded into the drawing and automatically receive a device designation (**-M3**).
The devices used in your drawing, are listed in the Project Window’s **Device** tab.



Using the **Device Properties…** menu, displayed by right-clicking on the motor, enter the text **Pump 3** for the attribute **Function**. This text will automatically appear underneath the motor symbol.


Device Properties
Pin Assignment
Component
Assign Variants / Options
Variant Overview
Device
Device II
Signal
Pins
Connector Pin Terminals
Device designation:
-M3
Higher level assignment:
=A1
Location:
+Tank1
View number:
<no entry>
Remove from current assembly
Create assembly
Move to assembly:
<no entry>
List all ...
Insert assembly designation before device designations
Inherit item designation from assembly
Component:
K21R-160-L8
List options:
with structure
Created with view number:
Table symbol:
<no entry>
Default Template:
<no entry>
Show variant component codes as text:
<show none>
Name
Entry
Data sheet
Dummy-Datenblatt.pdf
Function
pump 3
Hyperlink
data sheet ref="Dummy-Datenblatt.pdf"
New Feature
2009
Technical description 1
P=7,5 kW
Technical description 2
n=725 U/min
Technical description 3
I=15 A
Reference to slave (1)
<no entry>
Don't purge if device is unused
Use structure from device
Device is master of assembly
Sealed
Variants/Options
Variant to change for:
<default variant>
Create new instance:
Set changes to default
OK
Cancel
Apply
Help


1
2
3
4
U
V
W
PE
-M3
K21R-160-L8
M
3 ~

Connecting the Motor using a Cable

To connect the motor using a cable, several steps need to be executed. The connectors on the motor side as well as on the panel side need to be placed. The connect lines need to be drawn and the cable must be placed. Much of this work is done by ***E³.*schematic**.

Since the appropriate mating connectors have already been defined for the motor and connectors in the database, these will be inserted automatically. Also, the connect lines do not have to be individually placed. Autoconnect within an area can be used to complete these connections. This command is used to automatically connect all connect points within a defined area, which have not yet been connected with one another. If connections already exist in the area, these will also be edited.

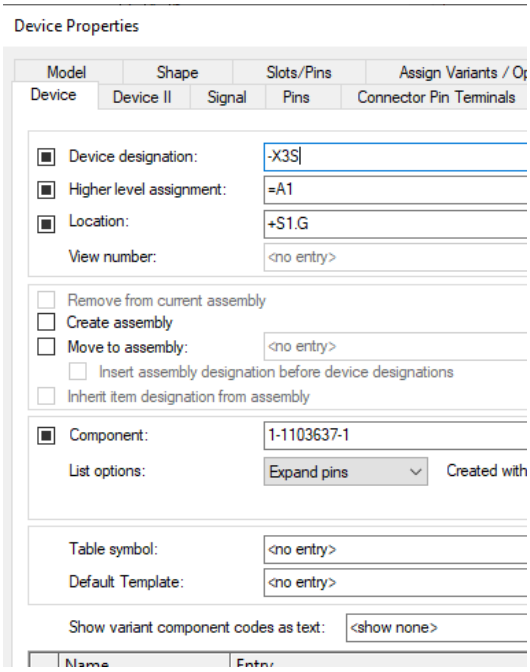
First, create the connection. This command can be activated with **Insert → Autoconnect → Inside Area Vertical** or using the *Autoconnect Vertical* icon .

Drag a rectangle around the connectors and the motor. After releasing the mouse button, the appropriate connectors and connection lines will be inserted.

Next, adjust names and visibility of the texts.
After all of the connectors have been inserted, select one of the contacts of connector **–X2S**, call up the **Device Properties…** from the context menu and change the **Device designation** of the connector **–X2S** to **–X3S**.

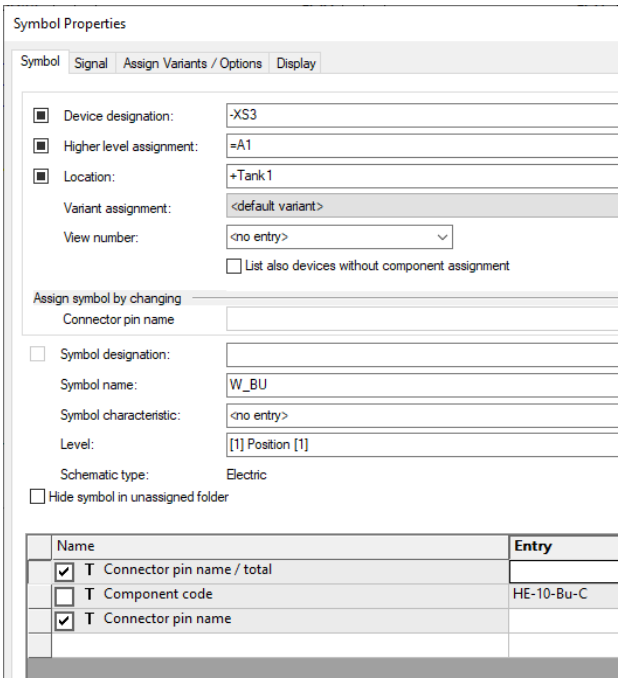
Repeat this procedure for connector **–X5** and change its **Device designation** to **–X3P**.

Lastly, change the **Device designation** from the motor's connector **–X4** to **–XS3**.

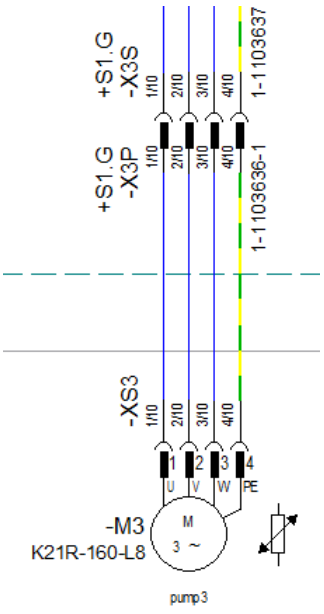


Superfluous information should be removed from the drawing.

Mark the last three contacts of connector **–XS3** (multi-select) and call up the **Symbol Properties…** menu, which displays the selected symbols' text and attributes. Uncheck the check box in front of **Device designation** so that this text is not displayed.
Additionally, deactivate the display of the text type **Component code** for the contacts **1-3**.

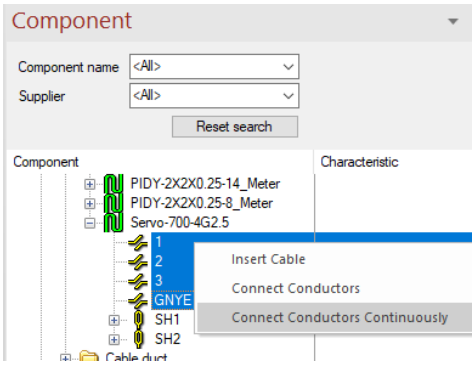


Compare your drawing with the example!

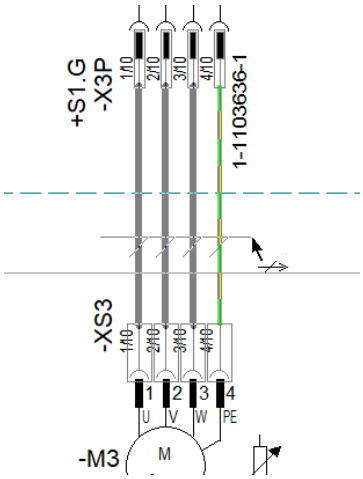


Now the conductors of an appropriate cable need to be assigned to the existing connection lines between the connectors.

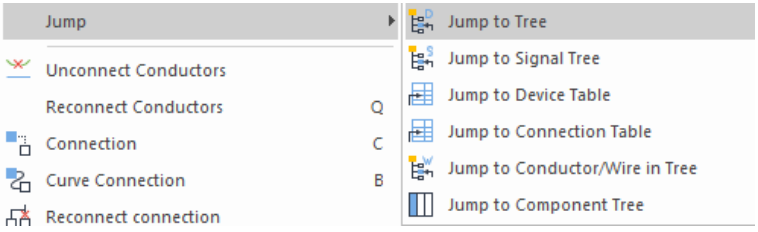
Search for the appropriate cable in the **Electric - Cable** folder in the Component Database (**Servo-700-4G2.5**), select the conductors **1…GNYE** and call up the **Connect Conductors Continuously** command in the cable's context menu.

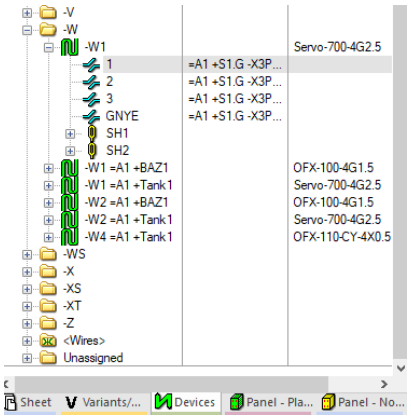


Then drag the selected conductors over the previously created connect lines. To do so, left-click on the right side next to the connection line, move the cursor over the conductors to the left side and "set" the conductors by left-clicking once again.

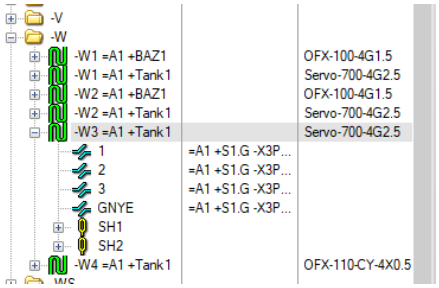


Next right-click on a connect line and select the **Jump to Tree** command from the displayed context menu to jump to the respective cable in the Project window's *Devices* tab.





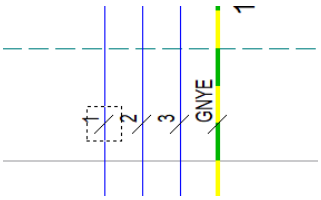
Select the cable **–W1** (green icon) and call up the **Device Properties** menu from the displayed context menu. Change the cable's **Device designation** to **–W3**, its **Higher level assignment** to **=A1** and its **Location** to **+Tank1**. The algebraic signs (-, =, +) do not have to be entered, they are entered automatically. The cable is displayed with its new device name.

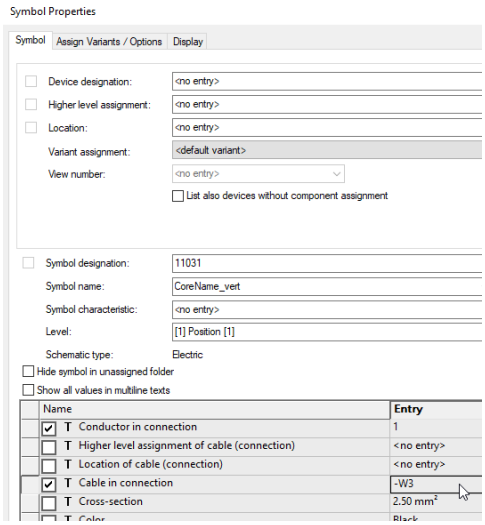


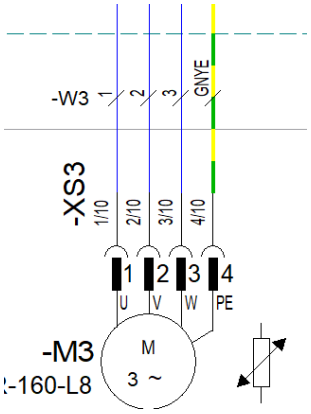
Because the cable's letter code as well as the conductor name itself need to appear on the first conductor, the **Symbol Properties** need to be adjusted.

Click on the first conductor's template symbol on the sheet and select the **Symbol Properties** command from the context menu.
Activate the corresponding checkbox and close the dialog.

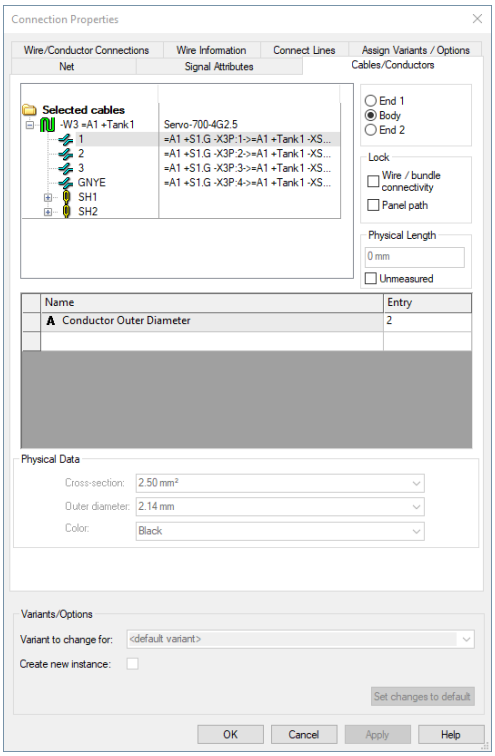
In addition to the conductor names, the cable (**-W3**) is now identified by its cable name in the drawing.







It is easy to control the connections. To do so, right-click on any of the connections to the motor and select the **Connection Properties** command in the context menu. In this menu, you can view or change the connection properties. Clicking on the **Cables/Conductors** tab gives you an overview of the cable. In the tree structure view, you can see which conductor is connected to which component/pin.



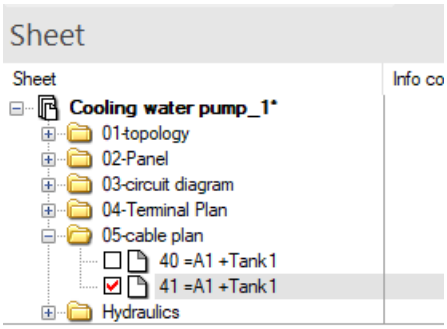
Creating the Cable Drawing (*E³.*cable Functionality)

This chapter offers a short look at cable documentation. The cable with its connected connectors is displayed in various views. ***E³.*cable** offers for example specialized functionality to create cable documentation. It is possible to create cable documentation for machinery or for cabling and displaying of harnesses.

While ***E³.*schematic** enables the user to display each cable as a report containing various information, ***E³.*cable** can also graphically display the cable and its connectors in several, different views.
***E³.*series**' object-oriented functionality is more advantageous to use since all modifications (as with the terminal plan) take effect immediately without any further updates.
Next, we will create a cable drawing.

Since the cable drawing requires its own sheet, we will have to create a new sheet. Open the **05-Cable plan** folder in the Project Window **Sheet** tab. Right-click on sheet **40** and select the **New Sheet** command. The new sheet will be named Sheet **41**.

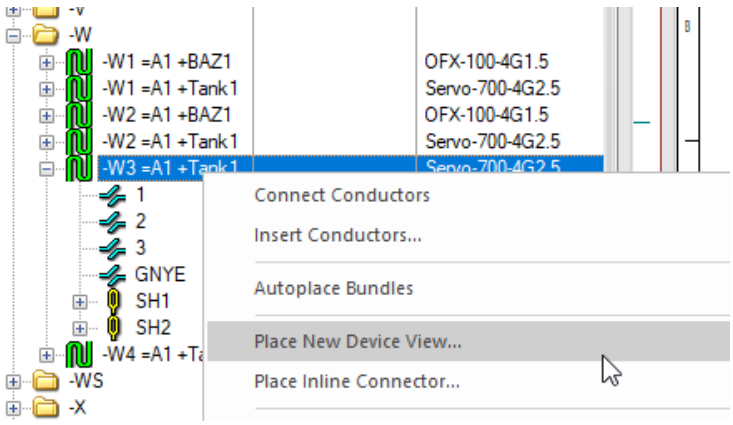
Right-click anywhere on the sheet and call up the **Sheet Properties...** command from the displayed context menu. Adjust the **Name (1)** attribute to **Cable plan –W3** and define **=A1** as **Higher level Assignment** and **+Tank1** as **Location**.



The help lines on the sheet, which were previously switched on (section: *Switching On the Placement Help Lines*), are no longer necessary. They can be deactivated by clicking on the Symbol Graphics box on level **3** in the **Levels** menu. The menu can also be opened by using the hotkey **L**.

We are going to use the cable **–W3=A1+Tank1** that was created earlier to create a cable plan.

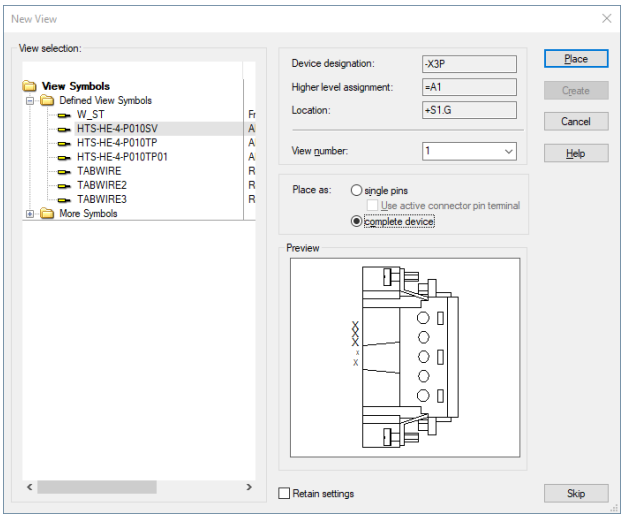
Switch to the Project Window **Devices** tab and search for the cable named **–W3=A1+Tank1**. Select the command **Place New Device View...** in the cable's context menu.



A menu allowing the user to select how the connector will be displayed in the new view now appears for each connector that is connected to the cable. In our example, there are two connectors but naturally, there can be several connectors connected to a cable/cable harness.

For the first view, we want to place the **complete connector**.
Select the **Place as: complete device** option. Select the **HTS-HE-4-P010SV** symbol from the **Defined View Symbols** and click on the **Place** button to close the menu.

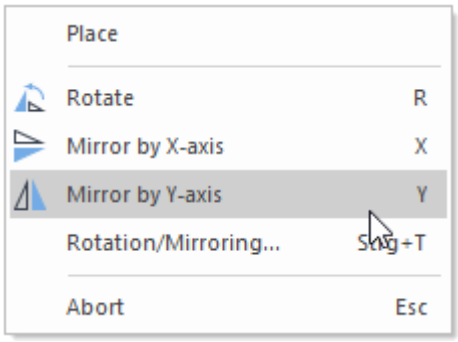
The selected symbol appears at the cursor position and can be placed on the sheet by moving the mouse to the desired position and clicking the left mouse button.
Place the view in the upper left hand corner of the sheet.

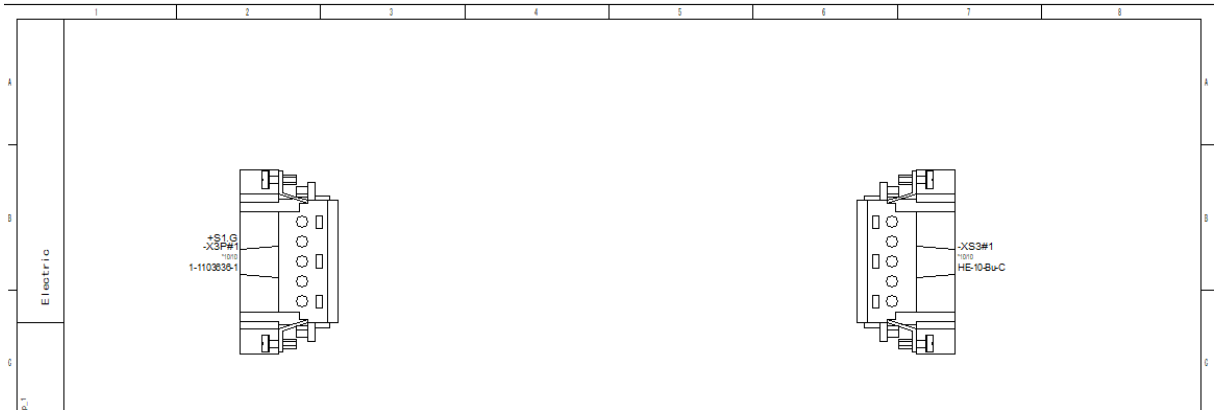


The menu for the cable's next connector appears automatically upon placing the first one.

Select the **HTS-HE-4-S010SV** symbol and close the menu with the **Place** command.

Before placing the symbol, it needs to be mirrored on the Y-axis. To do so, right-click and select the **Mirror by Y-axis** command in the context menu.
You will notice that the symbol is now mirrored and can be placed using the left mouse button.
Place the symbol in the upper right corner of the sheet.

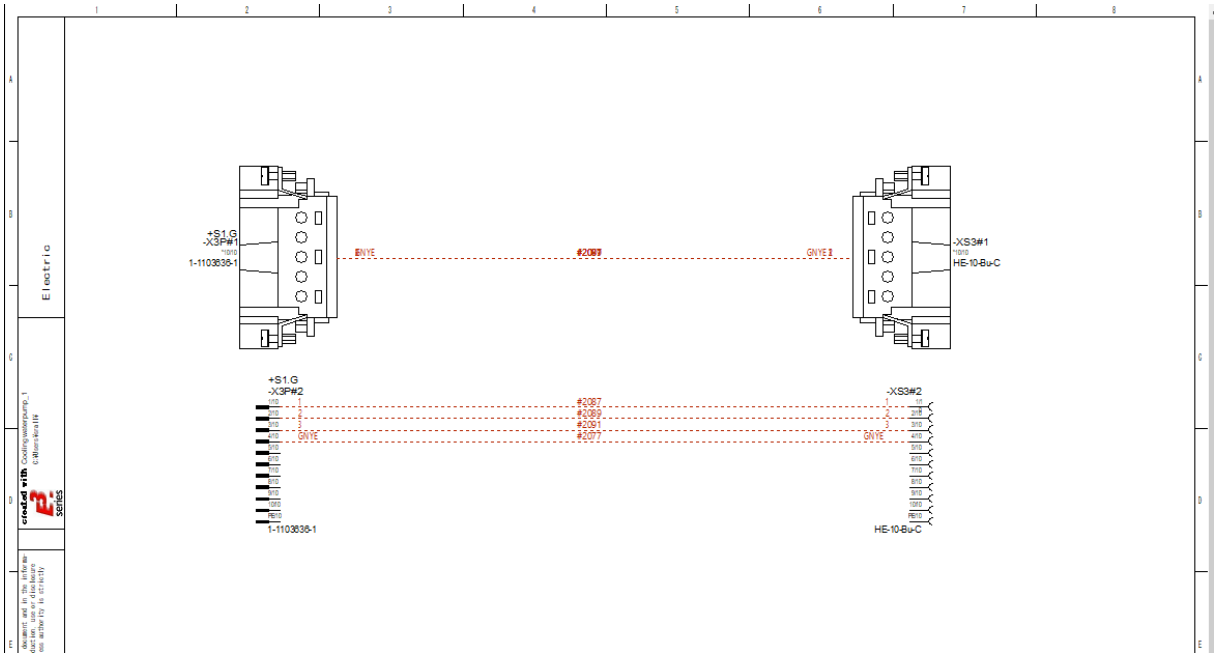




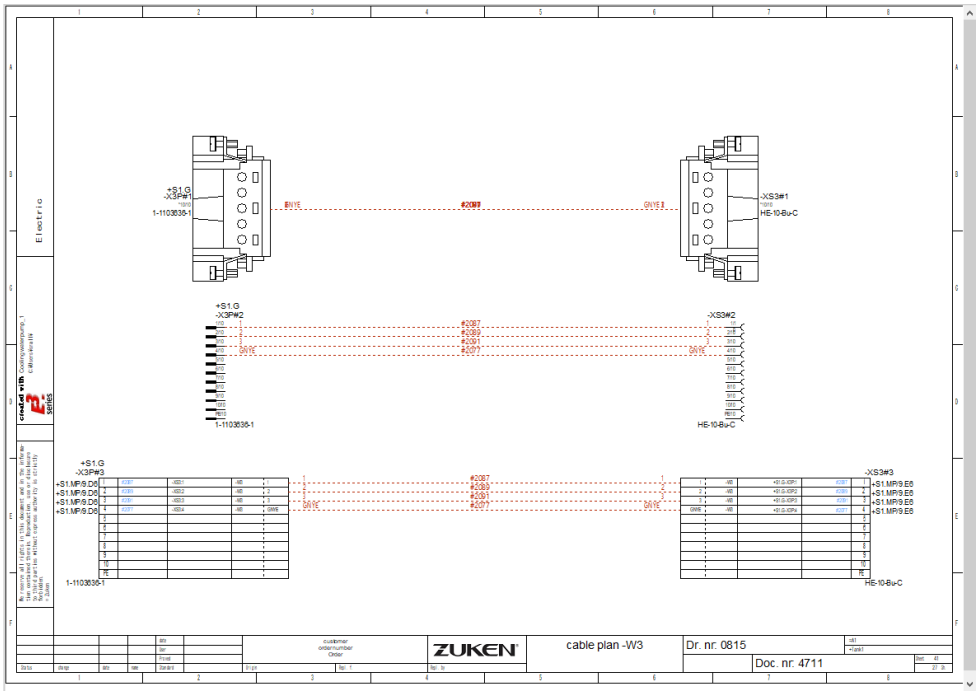
All symbols, which appear in the **Defined View Symbols** list in the menu, are already defined in the database for the connector so that no mistake occurs when selecting.

In addition to the complete connector view, we want to display the **individual pins** with their conductors. We are now going to call up a new view of the same cable **-W3**.
Return to the **Place New Device View...** menu, select **W_ST** as view symbol and select the **Place as: single pins** command.
Place the first connector view underneath the left connector. Then select the view symbol **W_BU** for the second connector, mirror the second connector on the Y-axis before placing it underneath the right connector on the sheet.

To see which cable conductors are connected to which connector pins, use the *Conductor Logic Lines* icon .



In the third view of the cable, we will display the connectors as “Tables”. Open the **Place New Device View...** menu of the cable **-W3**, select the symbol **TABWIRE3** for both connectors. Remember to mirror the second connector table on its Y-axis.



Different connector information is displayed automatically in the table. The output together with the symbol will be defined in the Symbol Database.

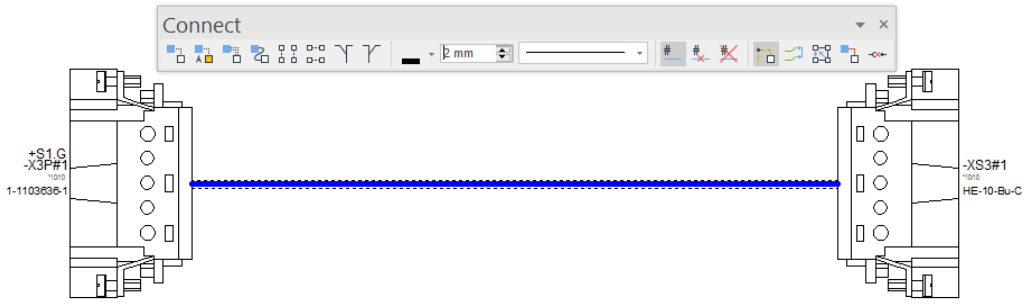
In our example, the following information is output:
the signal on the connector pin, the device’s pin name on the other side of the conductor, and the cable connected to the conductors with the cross-reference to the original connector pin.

+S1.G
-X3P#3

+S1.MP/9.D6	1	#2087	-XS3:1	-W3	1
+S1.MP/9.D6	2	#2089	-XS3:2	-W3	2
+S1.MP/9.D6	3	#2091	-XS3:3	-W3	3
+S1.MP/9.D6	4	#2077	-XS3:4	-W3	GNYE
	5				
	6				
	7				
	8				
	9				
	10				
	PE				

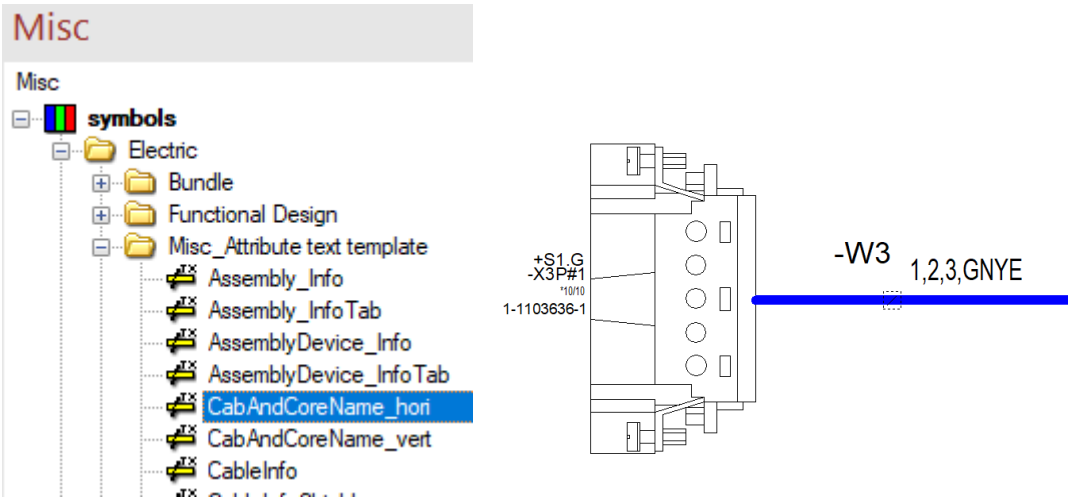
1-1103636-1

Draw a connect line between the complete connectors using the hotkey **C**. Since the connection line is going to display the entire cable, draw it somewhat thicker. Mark the line and change the line thickness to **2 mm** using the *Connect Line Width*.



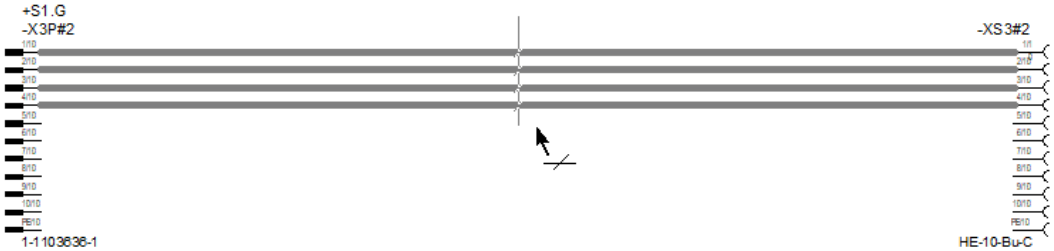
To identify which cable and conductors are used, place a template symbol on the connection line. The symbol itself defines the information that will be displayed.

Switch to the **Misc** tab in the Database Window and place the symbol **CabAndCoreName_hori** from the **Electric - Misc_Attribute text template** folder on the connection line. The text can now be moved or the text height modified.



We also want to connect the individual pins and display the conductors. Select the **Insert → Autoconnect → Inside Area Horizontal** command from the Main Menu bar or use the corresponding symbol from the toolbar , and drag a rectangle around the pins to connect.

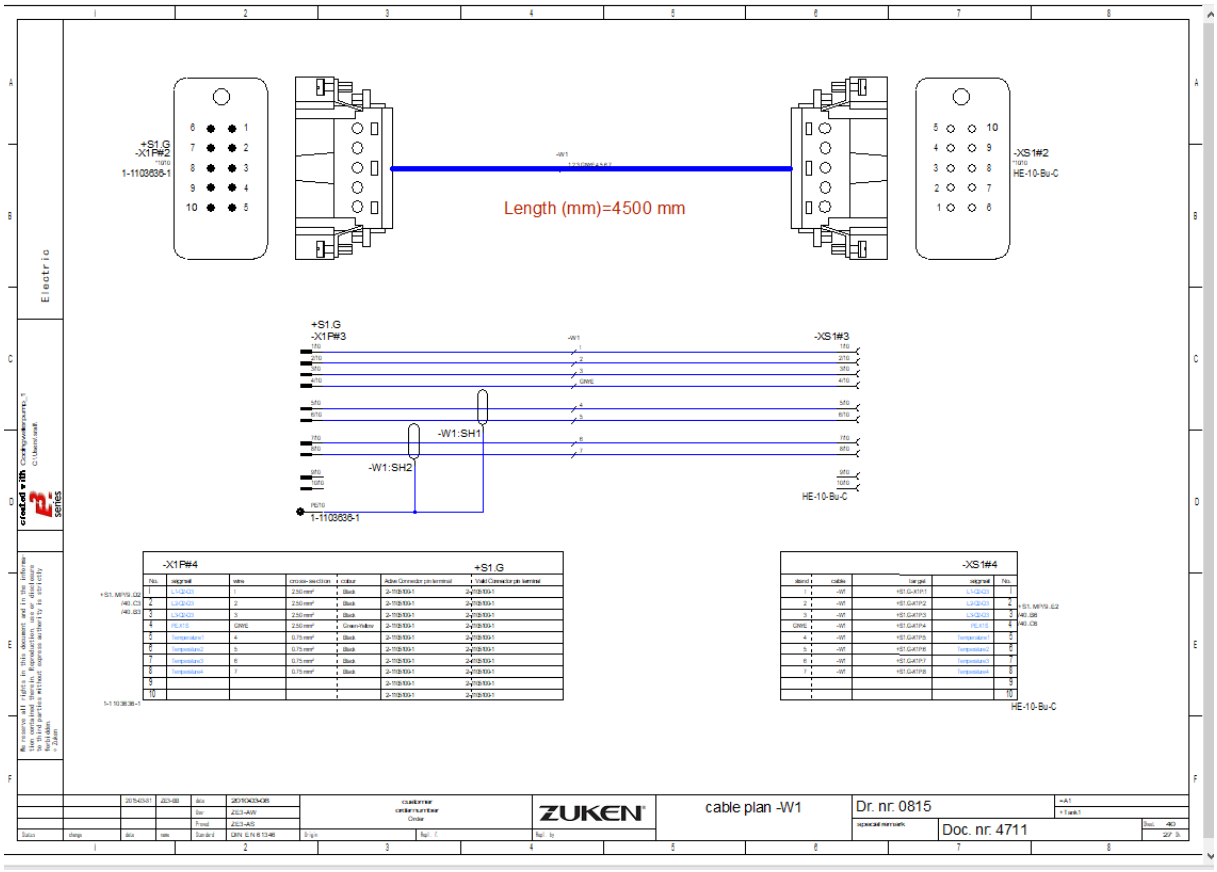
Then double-click on the *Insert Template Symbol* icon to activate the conductor indicator display mode. Place the conductors on the individual connections by dragging the cursor over the connect lines while pressing the left mouse button. To exit this mode, press the **Esc** key.



Before we continue, switch off the *Conductor Logic Lines* display by deactivating the corresponding button in the toolbar .

In this way, it is possible to create error-free cable drawings. The advantage here is, like already explained at the beginning of the chapter, that modifications, regardless of where they are carried out, are automatically updated in all views.
Information, which is required to produce a complete connector or cable harness, can be added in the project or to individual elements at any time, and used for reports, lists and interface files.

To see how the completed cable sheet should appear, open sheet **40** in our sample project. All cable wires are displayed along with their shielding, as well as any additional cable attributes that have been added.



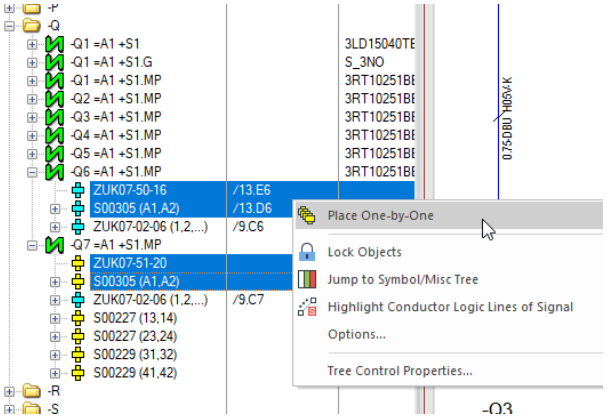
Placing Additional Device Elements with Online Cross-References

By inserting an additional pump and its controls on sheet **9**, we also loaded two more contactors into the project, in addition to having already used some of the contacts.
This specifically concerns the contactors **–Q6** and **–Q7**, which were inserted when copying the subcircuit. We still have to connect the contactors' coils. In this example, you will clearly see the online cross-references as well as a sampling of ***E³*.schematic**'s PLC functionality.

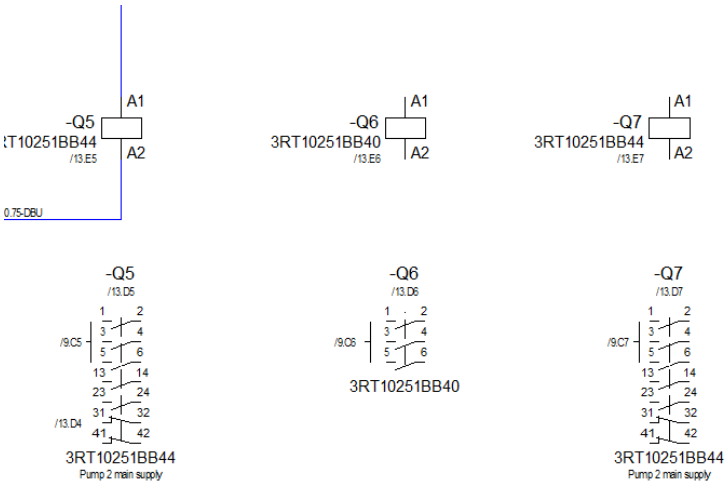
Open sheet **13** in the **03-Circuit diagram** folder in the Project Window *Sheet* tab by double-clicking on the sheet symbol or simply clicking on the box in front of the sheet symbol. The controls for the other "pump contactors" are located there. The new contactors need to be connected in the same way.

Click on the "**+**" signs in front of both devices, **–Q6** and **–Q7**, in the Project Window **Device** tab. You will see all of the symbols belonging to the respective devices. Those symbols, which have already been placed in the project, are displayed in blue and those that have not in yellow.

Select the first two symbols (contact arrangement and coil) under each device with multi-select (click on the symbols to be selected while pressing the **CTRL** key) and then with a right-click, call up the context menu and select the **Place One-by-One** command.



The symbols will appear one-by-one at the cursor position and can be placed as shown, next to the coil **-Q5**.

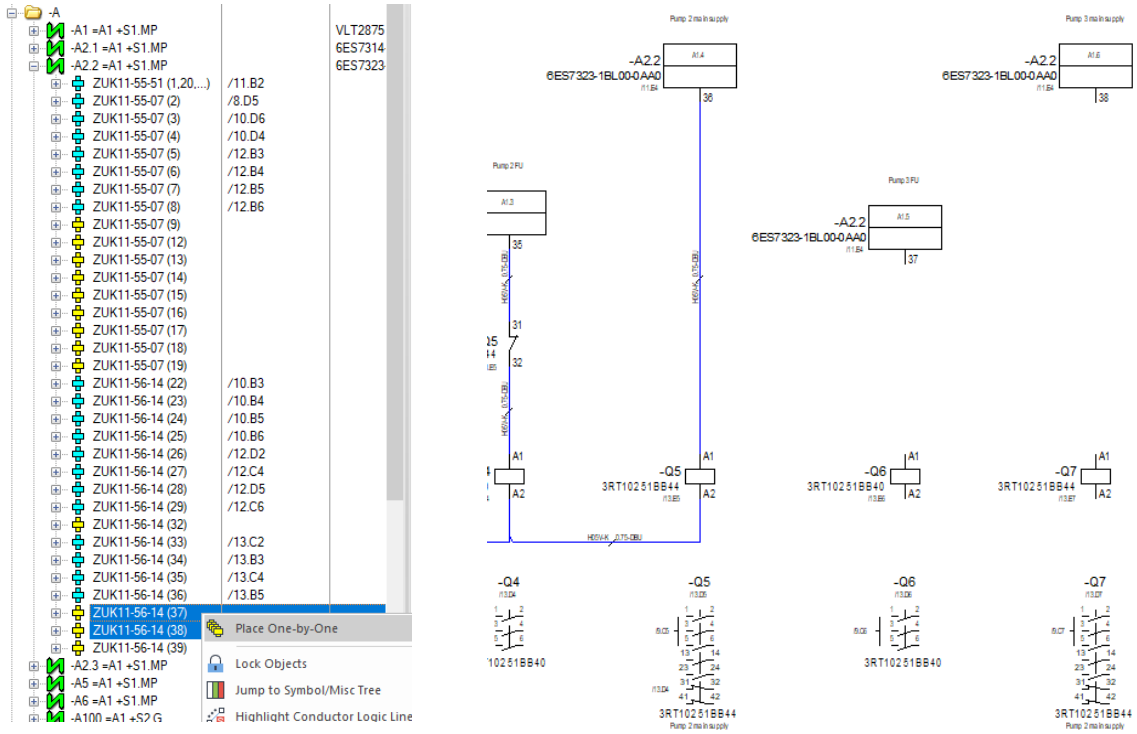


Since you can only select and place elements that have not yet been placed with the command **Place One-by-One**, this will prevent you from using an element more than once.

Coils and contact arrangements are defined as separate elements in the standard database. This allows them to be placed anywhere, either together or separately. You can, however, define the coil and contact arrangement as a single element in the database so that when you place one you also place the other.
A motor protection switch is a classic example of "combined symbols", in which the main contacts and cross-references to the auxiliary contacts are drawn as one symbol.

We are still missing both PLC outputs. These can be placed in the same way as the contactors **-Q6** and **-Q7**.

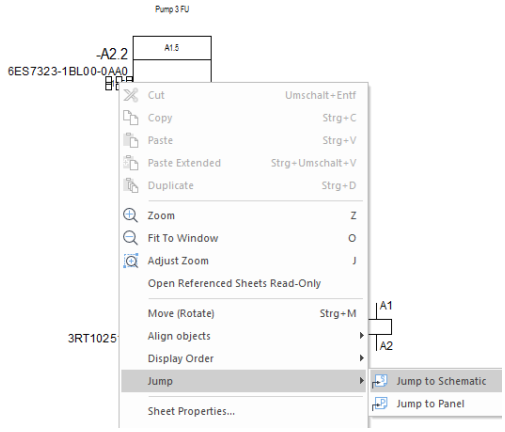
To find the appropriate symbol easily in the **Device** Tree View, it will be displayed in the Preview Window as soon as it is selected. Additionally the pin names are displayed in brackets after the symbols. Hence you have a good overview over which element of a device you have chosen, without looking at the graphic.
In our example the two PLC OUTPUT symbols of the device **-A2.2** with the pin names **37** and **38** are selected.
Select the two outputs, right-click and select the **Place One-by-One** command from the displayed context menu. Arrange the symbols according to the example above the coils.

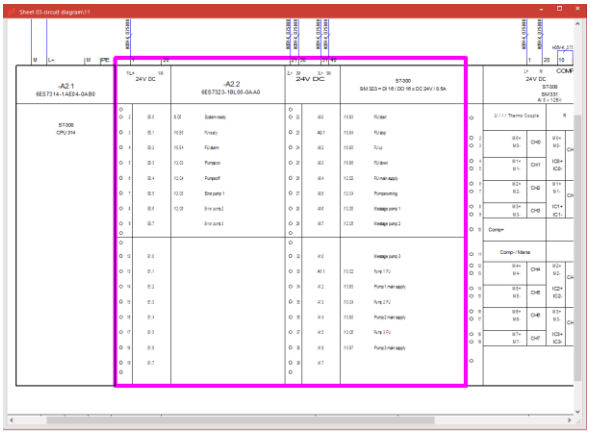


The addresses and functions are immediately entered for both outputs. The information contained in the PLC overview symbol is entered on sheet **11**. The PLC functionality of ***E³.*schematic** allows this information to be transferred immediately to the input/output symbols.
Changes to the PLC comments on the overview symbols are transferred online to the input or output symbols and vice versa.
The PLC information can also be transferred to PLC programming devices as an assignment list or the information from the assignment list will be entered automatically for the correct PLC symbol in the plan.

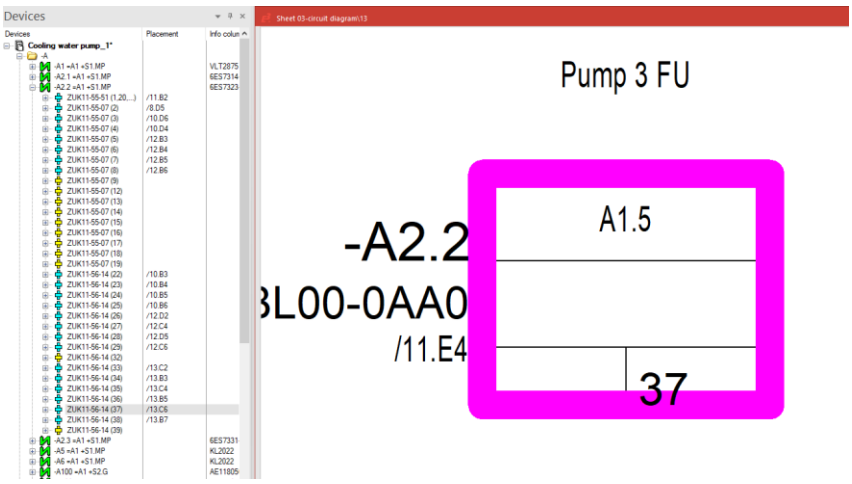
Cross-references between placed symbols are entered immediately in ***E³.*schematic** automatically. This applies to all symbols that are represented by several circuit diagram symbols, for example PLC symbols, individual symbols of a contactor or another device. The user can define how the cross-references are arranged and between which symbols they appear. You see immediately on the contact arrangements, which contacts have already been placed on the contactors **–Q6** and **–Q7**, and with the output symbols, where their overview symbols are located.

***E³.*schematic**'s special navigational features allow you to directly "jump" to any location. Right-click on any cross-reference text and select the **Jump to Schematic** or **Jump to Panel** command. The sheet, to which the cross-reference refers, will be opened automatically and the symbol highlighted. Remove the highlighting with **Shift**+**F3**.



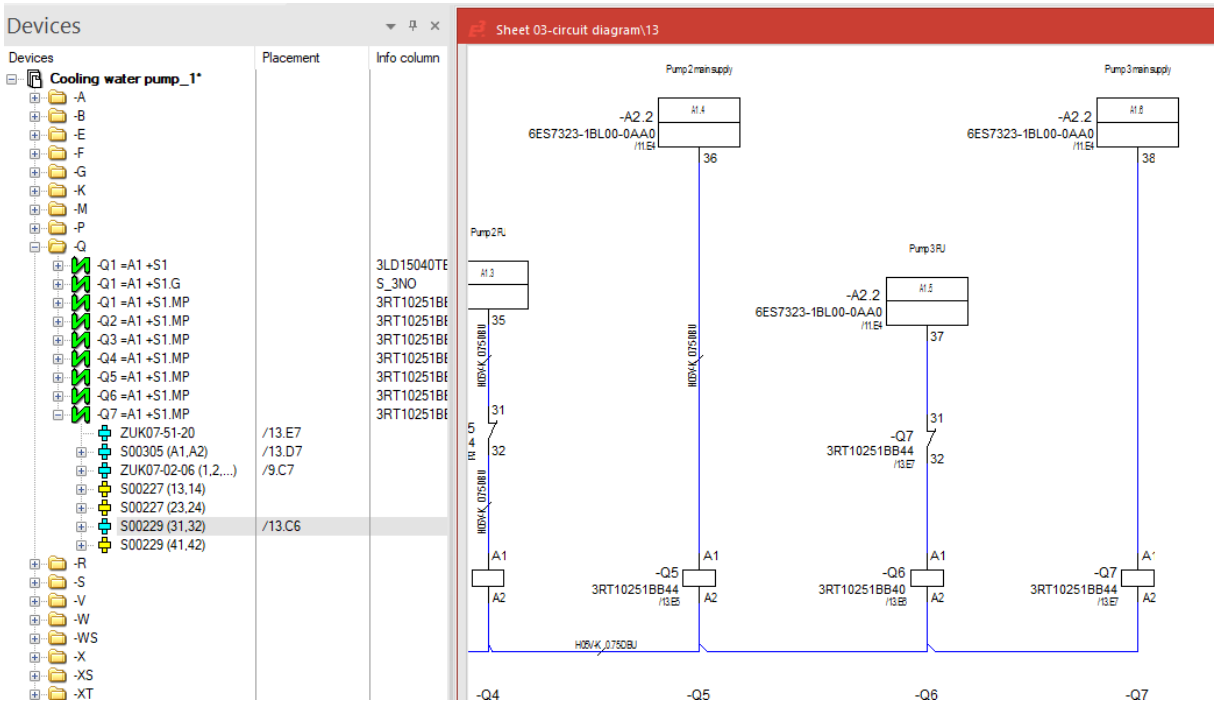


It is also possible to move about (navigate) using the "Online-Parts List" in the Project Window's **Device** tab. Double-click on any symbol in the tree structure, which has already been placed, and you will jump directly to that symbol.



To complete our subcircuit, we must add the missing connection lines. Connect the symbols using the command **Insert -> Autoconnect -> Inside Area**. The command is similar to **Autoconnect -> Inside Area Horizontal**, which we used in the previous section.
Note: Connections with corners cannot be created with **Autoconnect**. Thus, create the connection between **-Q6** and **-Q7** to **-Q5** manually. Consider route connections!

We must now add a contact, which is missing, to the contactor **–Q7**. Select the contact (pin numbers **31** and **32**) from the *Device Tree* and place it directly on the existing connection between the coil of **–Q6** and the PLC output symbol with Drag & Drop.

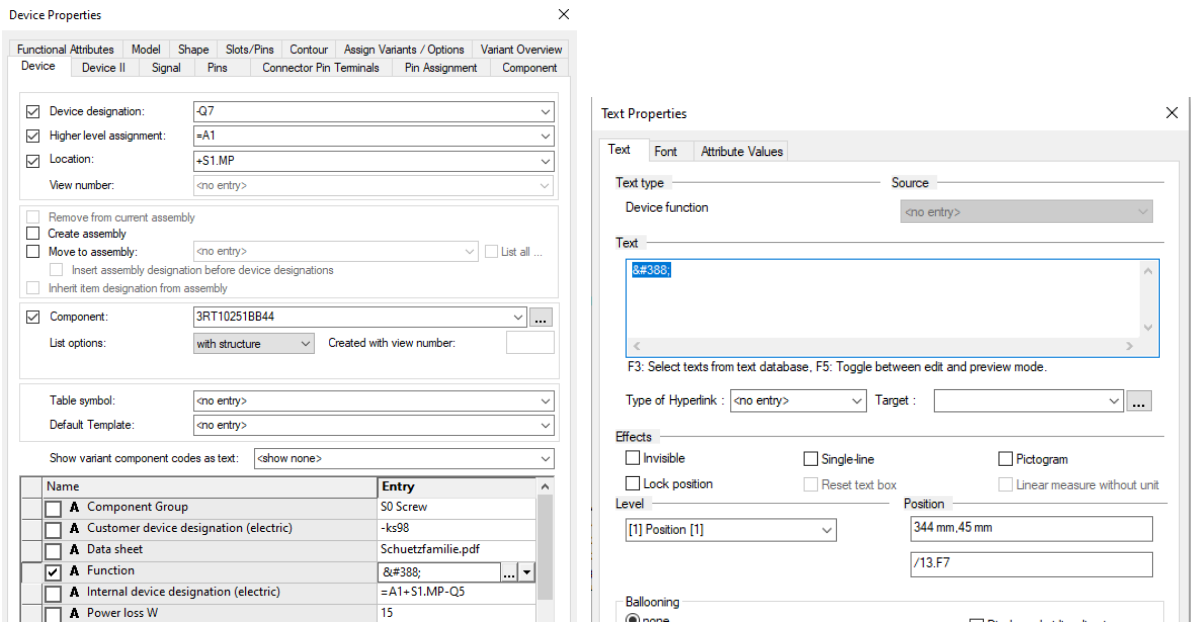


Changing Attributes

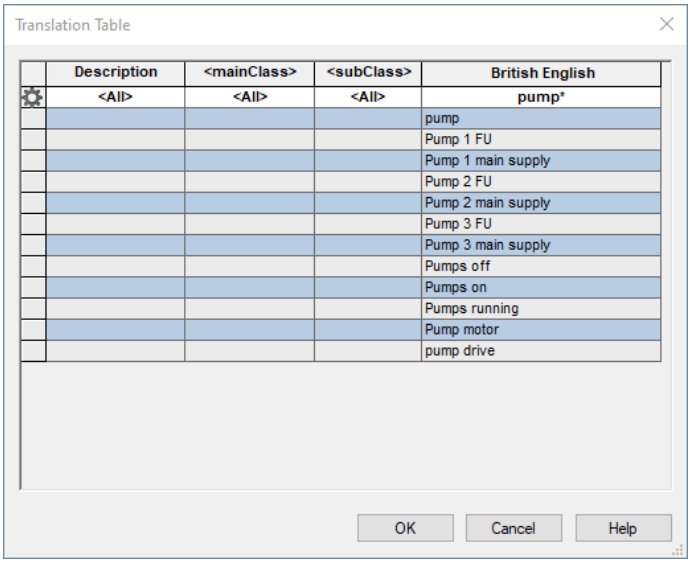
The contactors **–Q6** and **–Q7** were created by copying the devices **–Q4** and **–Q5** and as a result, also adopted their attributes. This is also true for their **Function** text. There are several, different ways to change the text.

The attribute can either be modified in the **Device** tab of the **Device Properties** called up from the device's context menu in the Device tree, or if attributes are displayed by texts in the drawing, these can easily be modified by using the **Text Properties**.

For example, right-click on the text underneath the contact arrangement **–Q7,** select the **Text Properties…** command and enter the text **Pump 3 main supply**. This results in the attribute being changed directly.



In both cases, it is not the actual function text that is displayed, but the text's wildcard, since this text is contained in the text database. **F3** opens the **Translation Table** dialog accessing the text database, from where the desired texts can be selected.



Texts placed through the text database are automatically translated when switching languages (**Tools → Settings → General → Language**). Up to five languages can be defined. Additionally, the "language" **Pictogram** can be selected.

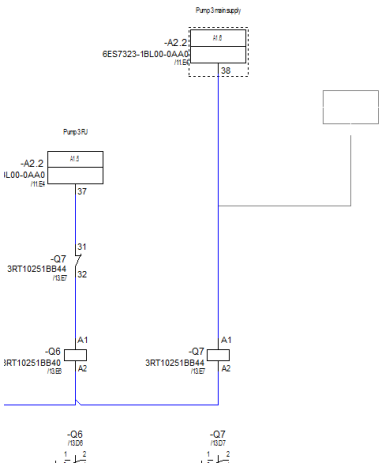
When all texts (also comments) are entered through the text database, the complete drawing can quickly be switched to another language (or several languages) by one click.

Modifications to Existing Drawings

It is crucial that the user is able to modify a drawing quickly, easily and reliably (proofed).
With ***E³.***, changes are displayed instantly in all representations of an object. Checks can also be done to the views that are practical for the workflow.
Connected components can be moved quickly and intelligently, also to other sheets.

An example for modifications is moving connected symbols.

Click on the right-hand PLC output symbol on sheet **13** and while pressing the left mouse button, move the symbol. You will notice that the connection line, connected to the symbol, moves automatically as well. This is also possible for symbols with several connections. When moving a symbol, ***E³.*** checks in real time to see whether the new position is allowed and if the connection line can be drawn without conflicting with any other elements.

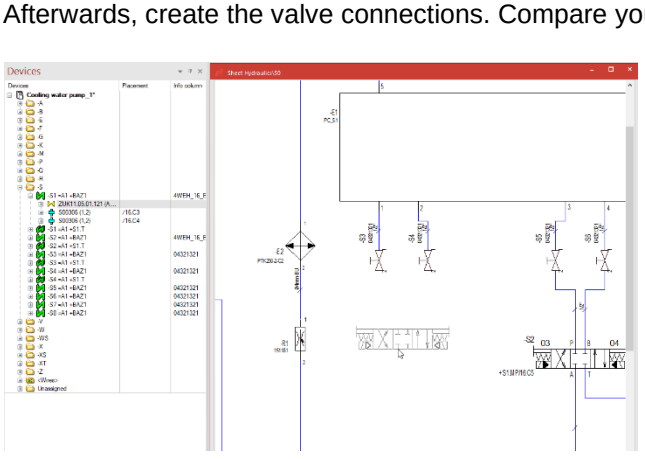


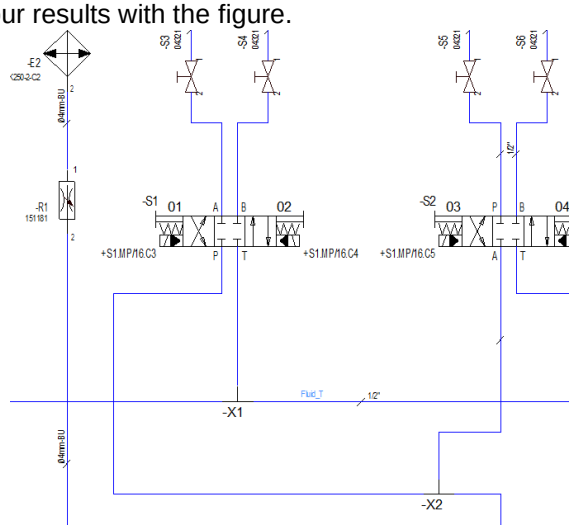
Note: The setting **Tools -> Settings -> Connection -> Reconnect after deleting symbols** defines what happens if a symbol is deleted from an existing connection. When the setting is active and the attributes assigned to both connections allow, the connection is automatically made. When the setting is inactive, the connections remain open.

Delete a contact or coil on sheet **13** by first selecting the appropriate symbol and then pressing the **Del** key. The symbol will be deleted and the connection closed. To undo this delete procedure, use the **UNDO** command with either **Ctrl+Z** or the corresponding icon .

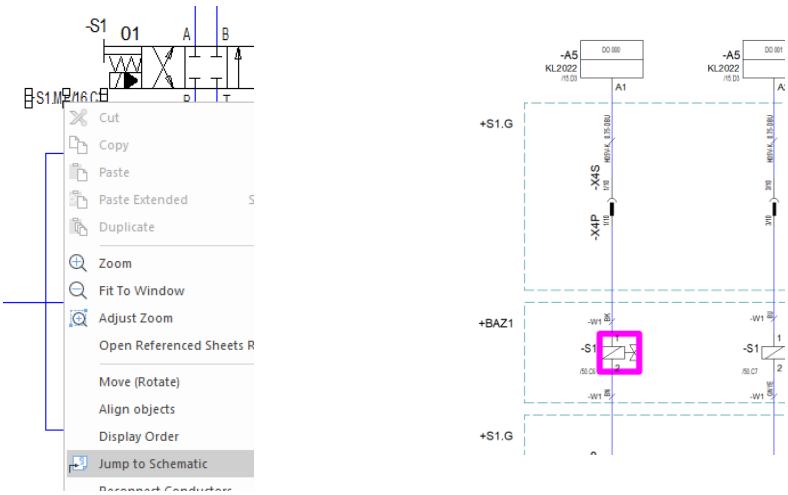
Open sheet **16** in the **03-Circuit diagram** folder to see the electrical drive for the directional gate valve **–S1**. Next, we want to place this valve in the hydraulics documentation on sheet **Hydraulics/50** and complete it with the hydraulic connections.

Open sheet **50**. Switch to the Project Window **Device** tab and place the hydraulic symbol of valve **–S1** from within the tree via **Drag & Drop** or by right-clicking and selecting the **Place** command.
Afterwards, create the valve connections. Compare your results with the figure.



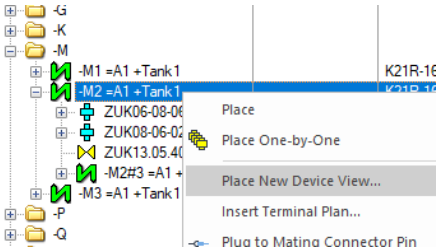


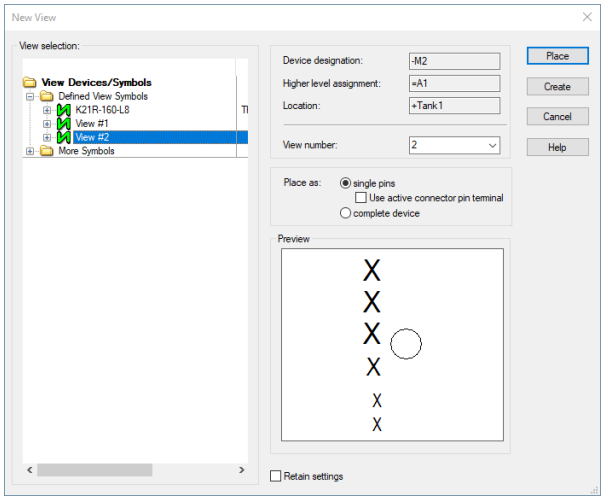
The hydraulic valve symbol **-S1=A1+BAZ1** displays 2 cross-references from which you can jump to the schematic documentation to the electrical drive for the valve coils.
To do so, right-click on the cross-reference text **+S1.MP/16.C3** and select the **Jump to Schematic** command from the displayed context menu.



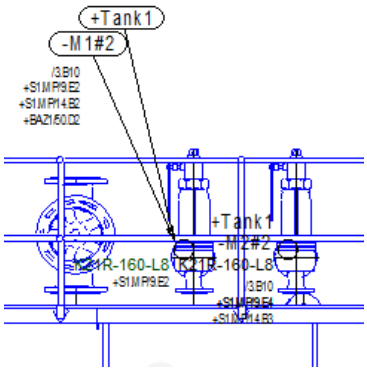
Now, we need to have the motor**–M2=A1+Tank1** displayed in the topology documentation with its insert position at the tank. ***E³.*cable** offers the possibility to place another view of the motor.

Open sheet **2** in the **01-Topology** folder, switch to the Project Window Device Tree, right-click on the motor**–M2=A1+Tank1** and select the **Place New Device View** command from the displayed context menu. Select **View #2** as the **Defined View Structure** in the **New View** dialog.

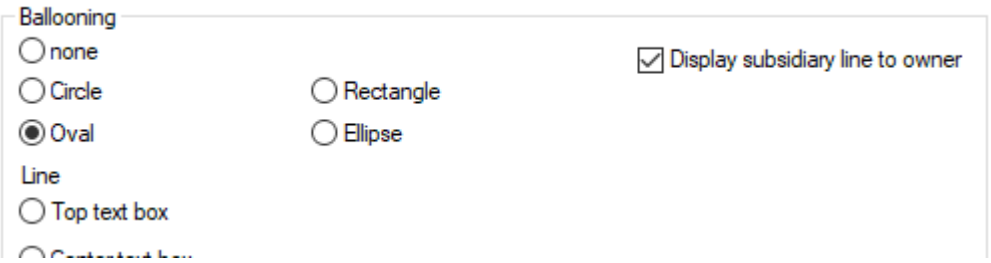




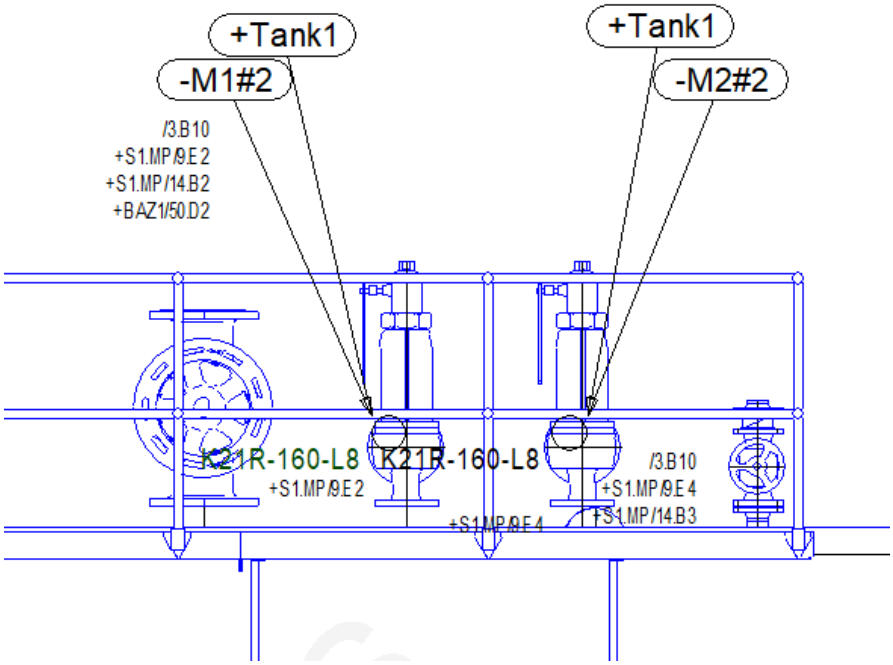
Place the view as shown:



The texts of the placed view can then be multi-selected and moved to the desired position. Right-click on the selected texts and select the **Text Properties** from the displayed context menu. Then select the **Oval** and **Display subsidiary line to owner** options in the **Ballooning** section.



Compare your results with those here:



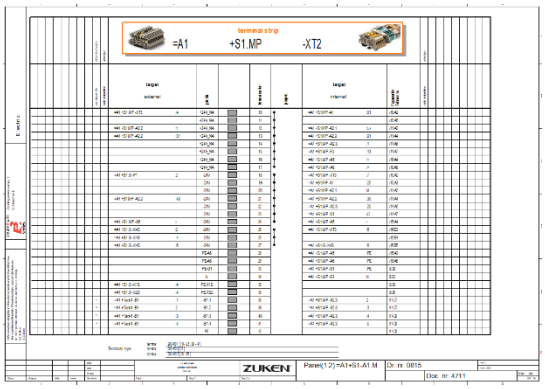
Modifications to the Terminal Plan

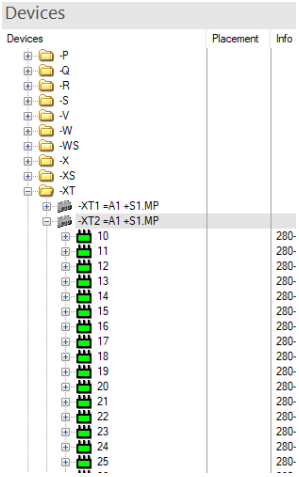
The terminal plan in ***E³.*schematic** is an on-line document. Changes to the circuit diagram are updated immediately and automatically in the terminal plan. Correspondingly, all changes made to the terminal plan are immediately updated in all other representations of an object (circuit diagram, panel plan, device list, …). This functionality is described in the following paragraphs.

Note: Online update of the terminal plan can be switched on and off with the **Online update** option in the **Terminal plan** section of **Tools -> Settings -> Placement -> Terminal**.

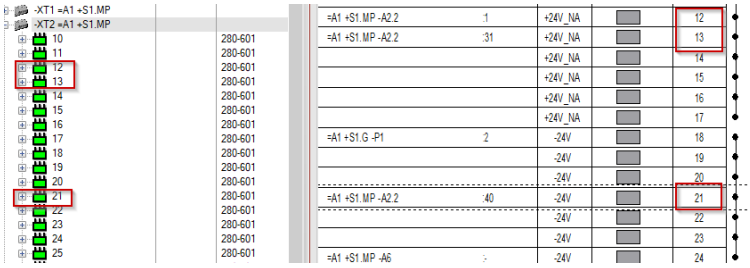
Open the terminal plan on sheet **32** in the **04-Terminal plan** folder. The format (table or graphic) can be defined when creating the terminal plan. The layout of the lines can also be user-defined. For this, corresponding template symbols exist in the symbol database.

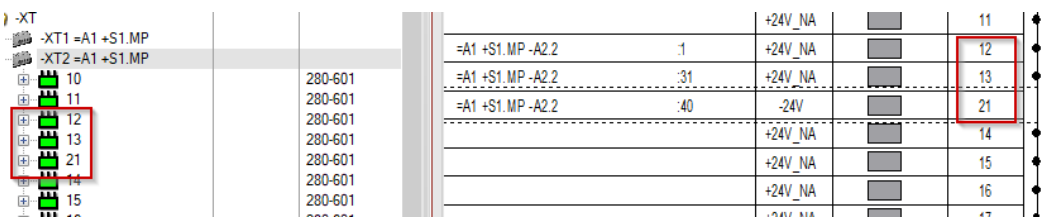
Enlarge the area at the top of the terminal plan so that you can read which components are connected to the terminal block. The sequence, in which the terminals appear in the terminal plan, is defined when first created. Otherwise, the sequence will be structured according to how the terminals are arranged in the Project Window's **Device** tab.
Also open the terminal block **-XT2=A1+S1.MP** in the **Device** tab.



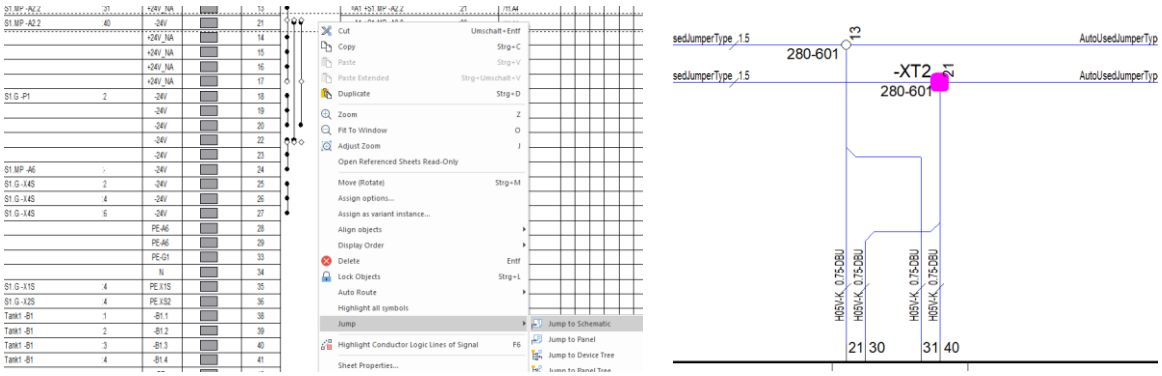


The first thing we want to do is to change the sequence of the terminals. As you can see in the terminal plan, the PLC **=A1+S1.MP-A2.2** is connected to the terminals **12, 13** and **21**. These terminals, which are connected to the PLC, can be easily placed side-by-side in the terminal plan.
To do so, click on the line containing terminal **21** and while pressing the left mouse button, drag it under terminal **13.** The terminal sequence is immediately changed. At the same moment, the sequence changes in the *Project Window*. The terminal block sequence can also be changed within the *Project Window*. These modifications are visible immediately in the terminal plan.

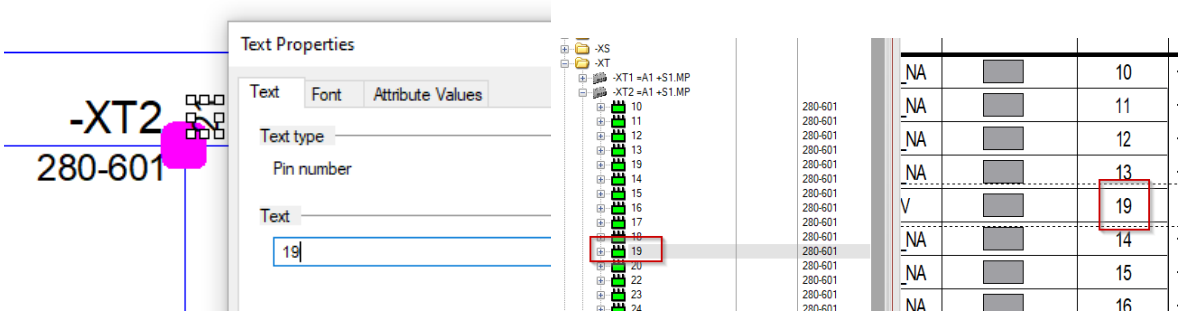




Up till now, these modifications have not yet taken effect in the circuit diagram. On the right-hand side of the line in the terminal plan, you can always see where the terminal is placed in the circuit diagram. Right-click on the text for terminal **21** to call up its context menu. Select the **Jump to Schematic** command, which automatically opens the corresponding sheet and marks the terminal in the circuit diagram.



Arrange the sheets in the window so that you can see terminal **21** in the circuit diagram and in the terminal plan. Change the terminal number in the circuit diagram from **21** to **19** in the **Text Properties**. Notice how this change takes effect immediately in the terminal plan. Change terminal number **19** in the terminal plan to the value **21** in the **Text Properties**. This modification is also visible immediately. These modifications will also be updated immediately in the *Project Window*.

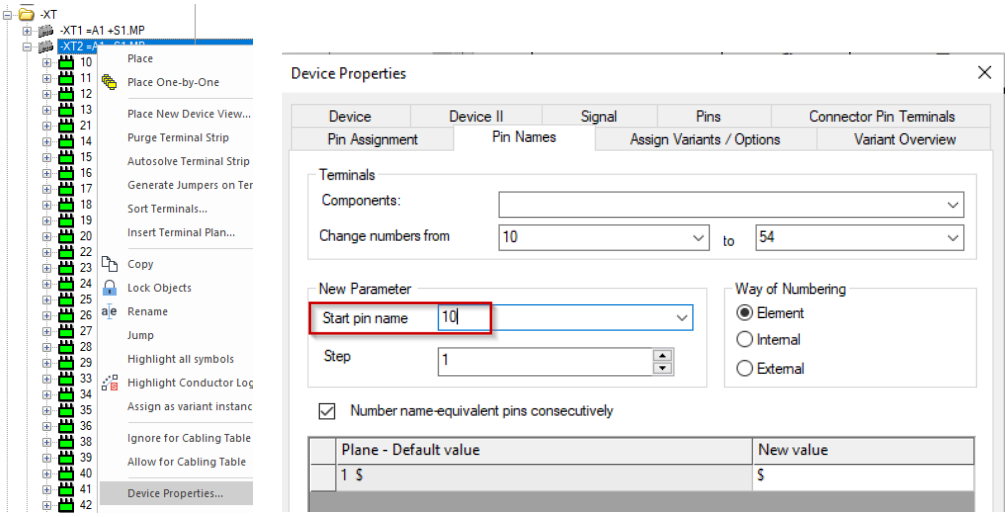


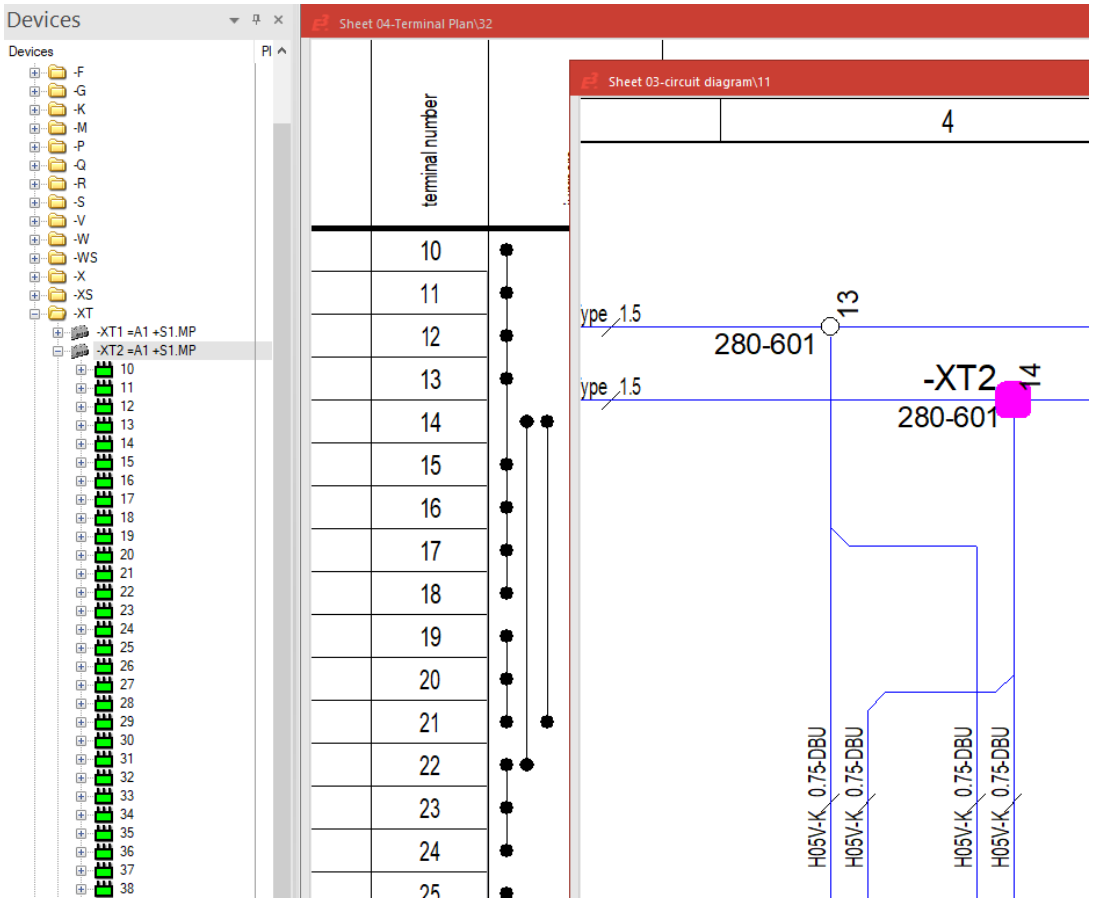
These modifications not only apply to the terminal numbers but also when new terminals are added to a terminal block. These are also updated immediately in the terminal plan. Any modifications to the device designations or the wiring and cabling used are also immediately visible.

Since the terminal plan with the activated online update option is always up-to-date, it is impossible for an outdated, “incorrect” version of the terminal plan to be displayed in your project. These modifications can be carried out wherever it is the easiest to do so.

Due to these modifications, we now may have an undesired sequence of terminal numbers being used on the terminal strip. Therefore, we have to renumber the complete terminal strip. For this ***E³.*schematic** offers the corresponding functionality.

Select the **Device Properties** command from the context menu of terminal strip **–XT2** in the *Project Window* and click on the **Pin Names** tab. A terminal strip can be completely or partially renumbered here according to different criteria. Enter the starting value as **10** in the line and close the menu with **OK**. You can now see that the terminal strip has been renumbered and the new terminal numbers are entered immediately in all views (project structure, circuit diagram, terminal plan).





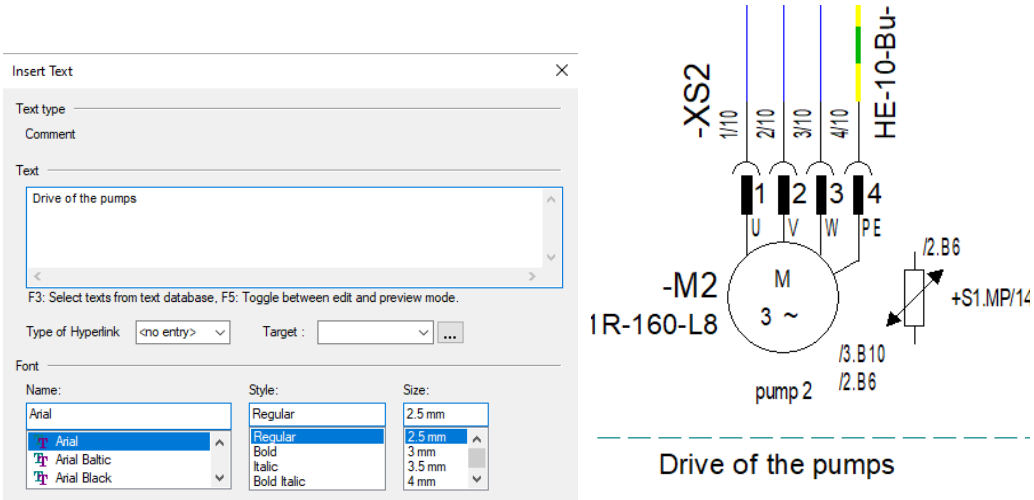
Adding Graphic Text

Circuit diagrams may contain free texts and graphics without electrical functionality. ***E³.*schematic** offers the user several possibilities for adding such elements to your drawings.

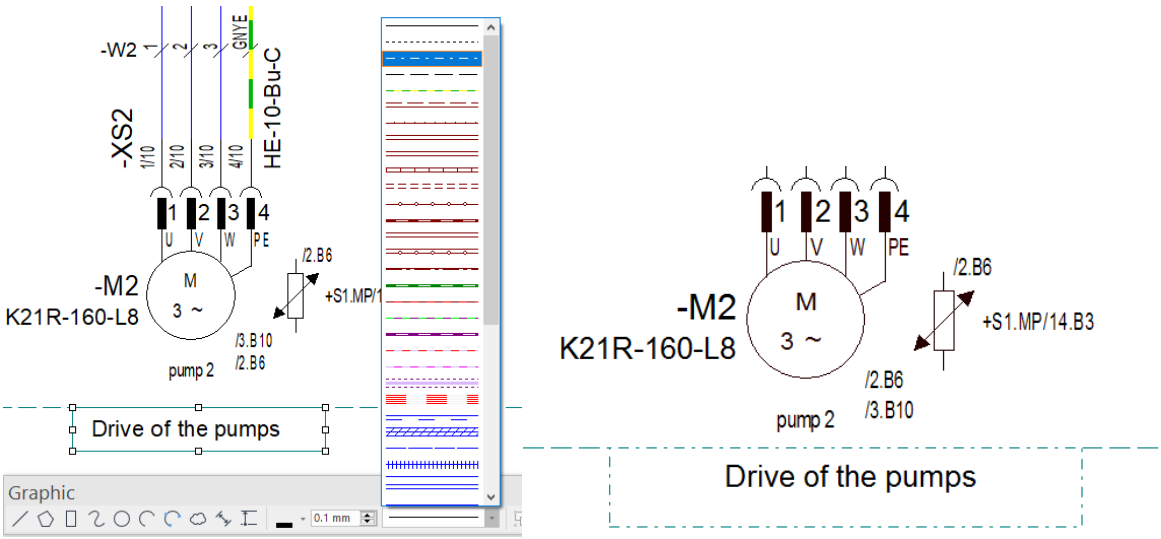
We must now complete the drawing by adding the necessary text. Open sheet **9** containing the pump motors and use the **Insert --> Text** command or the T icon in the text format toolbar.

Insert the text *Drive of the pumps* in the displayed **Insert Text** menu. The dialog allows you to type in the desired text (including multi-line text) and change the text parameters (e.g. font, text size, etc.). After clicking **OK**, the text will appear at the cursor position and can be placed anywhere on the sheet with a simple click of the mouse. In our example, place the text under the motor **-M2**.

The text size can be changed at any time through the **Text Properties** using the corresponding icon 2.5 mm or simply by changing the marked text directly.

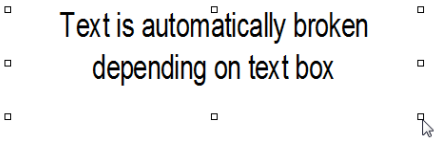


Then add a rectangle around the previously created text. To do this, select the icon ▭ in the graphic toolbar. The rectangle will be displayed at the cursor position and by clicking on one of the corner positions, drag a rectangle around the text.
To change the line type, click on the line type scrollbar and select the broken line type.

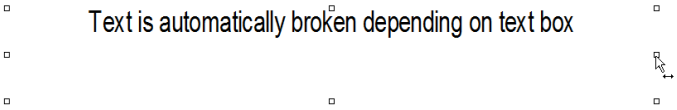


Using Text Boxes

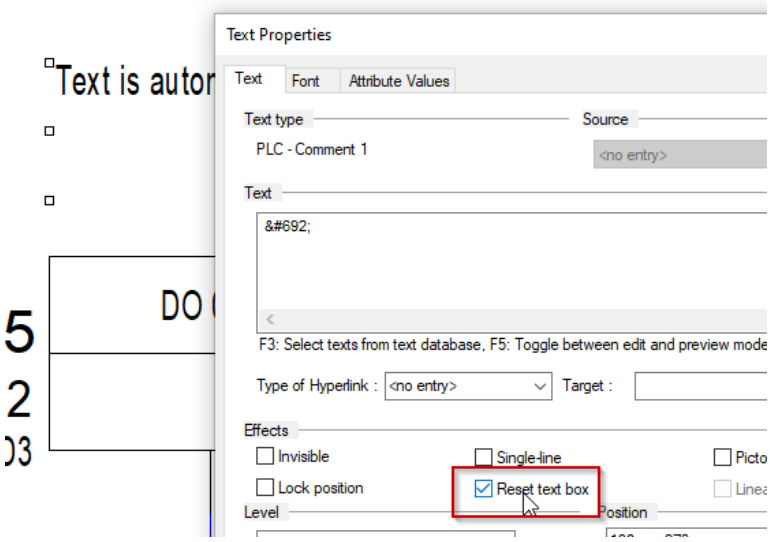
It is possible to define a text box specifying the maximum width and height of a text. This text box is displayed as rectangle in which the text is displayed. A line break is done automatically. This box is defined by modifying (e.g. enlarging, minimizing, etc.) the text handles.



We will now define such a text box. Open Sheet **16** in the **03-Circuit diagram** folder. The text for the PLC output **DO 000** is already defined as text box. Modify the text box by enlarging it horizontally. The line breaks will disappear.



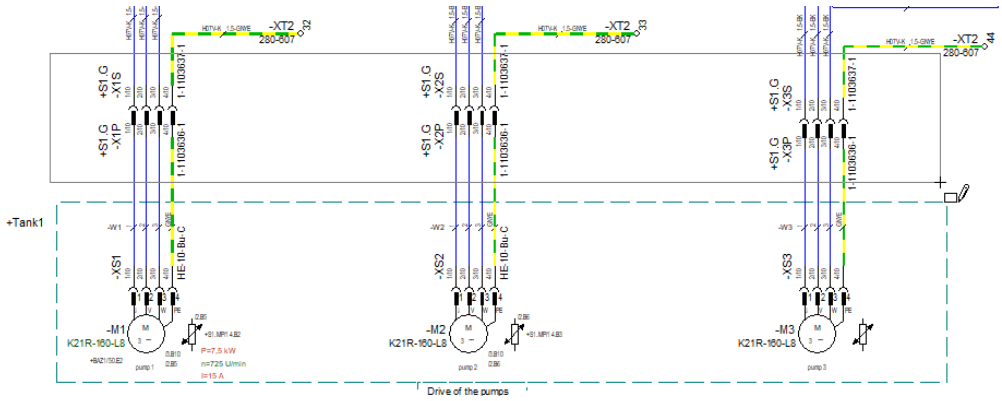
To delete such a box (but keep the text), the corresponding option must be unchecked in the **Text Properties**. The option is only active if a box is already defined.
Now right-click on the text and select the **Text Properties** command from the displayed context menu. By checking the **Reset text box** option, the text box can be removed.



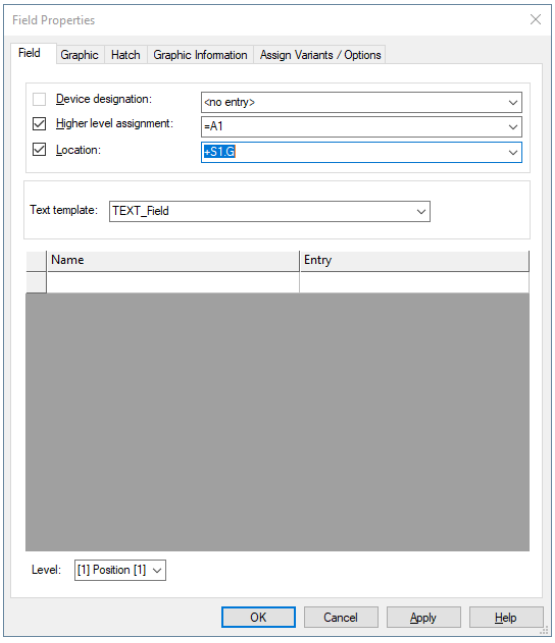
Defining Fields

To get a better overview of the plans, it is often useful to suppress common information, such as Device designation and Location, in an area. ***E³.*schematic** can use fields to accomplish this.

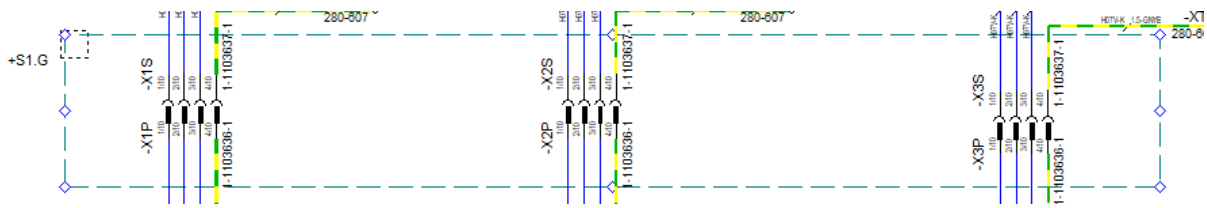
On sheet **9**, the connectors and the connected mating connectors have the same location designation. To suppress the display of the designations, use the **Insert --> Field...** command and while pressing the left mouse button, drag an appropriately sized rectangle around the connectors.



Afterwards, right-click on the dashed line of the field and select the **Field Properties…** command from the displayed context menu. Device designation and Location designation can be specified for this field in the **Field Properties** dialog.



Confirm the changes with **OK**. The values are suppressed for all symbols whose device designation and location designation correspond to those of the field. The suppressed values are not displayed.



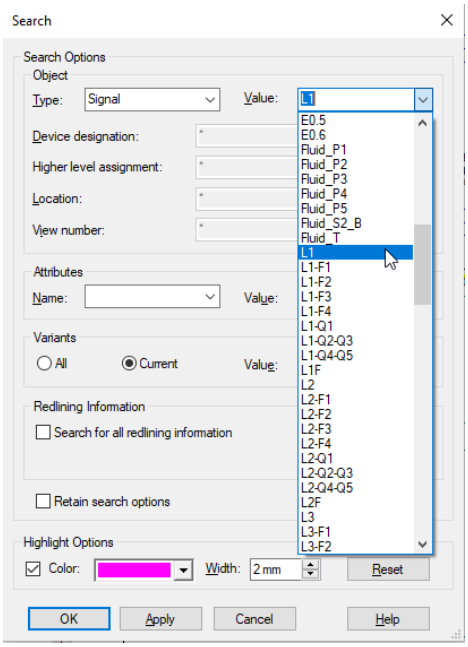
Searching and Navigating within the Project

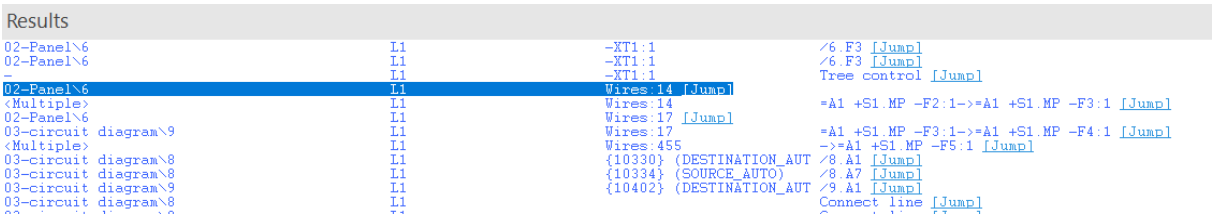
Trying to find a specific element in a large drawing can be very time-consuming. ***E³.*schematic**, however, makes this task quite simple. You define the information to be searched for and.at the push of a button, the corresponding sheet will be opened and the element highlighted in the drawing.

Select the **Edit → Search** command from the main menu bar or use the icon or the function key **F3**. The **Search** dialog appears. Enter the search criteria in the menu.

For example, enter the **Object: Type** as **Signal** and the **Value** of the signal to be searched as **L1**.
This results in a search being carried out for all pins and connection lines, which have the signal **L1**.

The search results will be displayed in the Output Window. By double-clicking on a line, you can jump directly to the sheet, on which the desired object is highlighted. This allows you to find information quickly, as well as keeping an overview of your drawing.





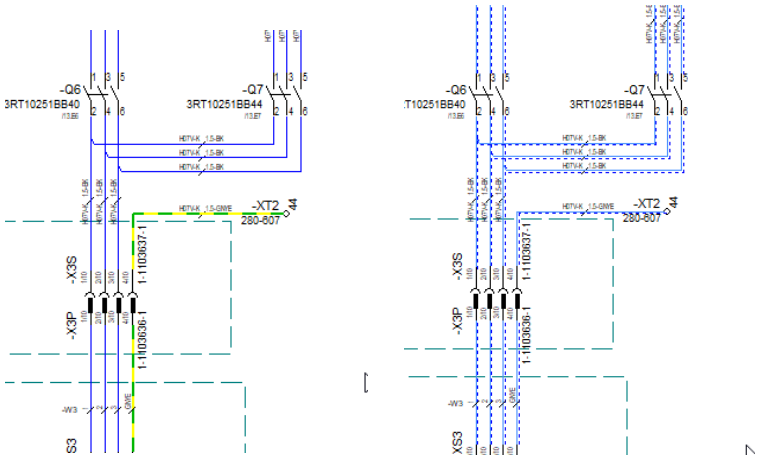
To delete the highlighting found on this object, use the **Edit → Reset Highlight** command or press the keys **SHIFT + F3**.

Switching between Display Modes

In addition to the pure, graphical information found in the drawing, you can also find “logic background information” (often displayed in the form of attributes). ***E³.*schematic** makes it possible to display parts of this logic information.

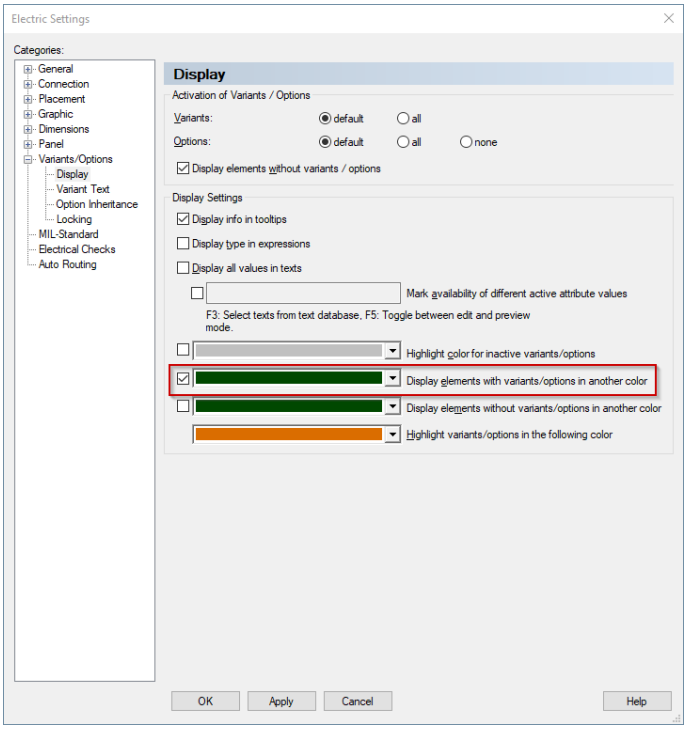
To switch between display modes, use the **View → Connection Types** command or the icon.

In the “normal” view, all connections are displayed as defined by their graphic attributes (color, line width, line type). Connection attributes are not shown graphically. This is the mode normally used to display the drawing’s printout.
Once the **Connection Types** mode is activated, all connections will be displayed as defined, depending on their attributes in different colors and line types. Thus, it is possible to show connections for example with their signals assigned or wires placed. In addition to the signal, the signal cross-references will be numbered. This gives you a quick, graphical overview of the connection information available in the project.



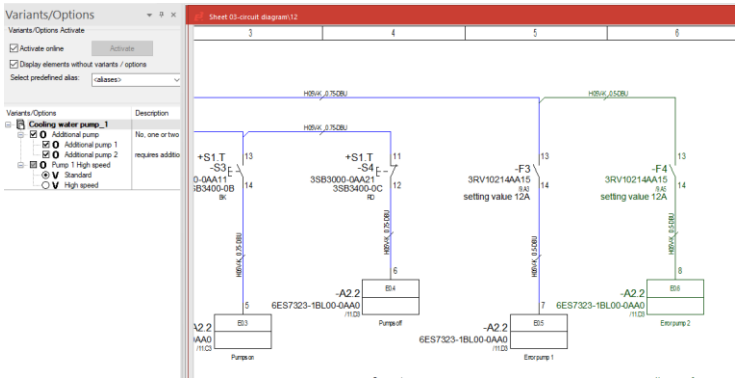
Displaying / Hiding Options

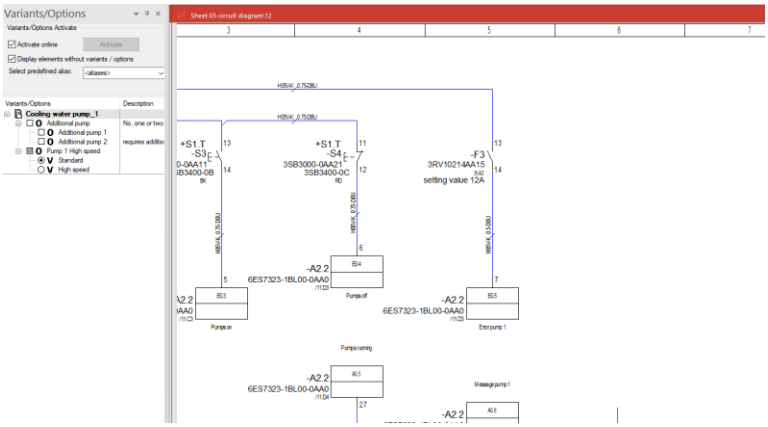
Options in projects are objects that can be switched on and off and have an optional character assigned. By comparison, variants are attributes or objects that have various characteristics.
The sample project contains two options: **Additional pump 1** and **Additional pump 2**. Open sheet **12** of the **03-Circuit diagram** folder.
Check the “Draw elements with variants / options in other color” option in the **Settings → Variants / Options → Display** dialog.



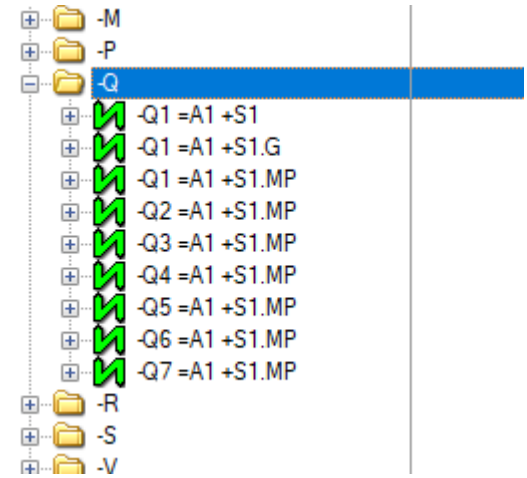
This option makes optional objects such as the devices **–F4**, **-A2.2:8**, **-A2.2:29** and **-P4**, and their connections displayed in green.

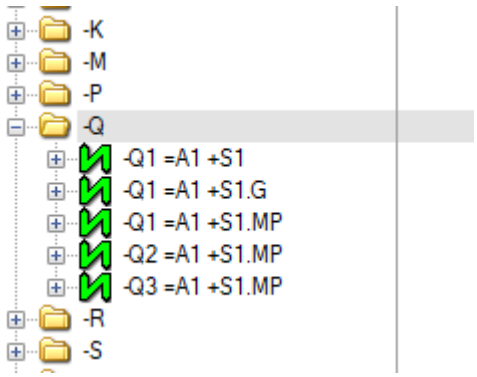
Now switch to the Project Window’s Variants/Options tab and deactivate the **Additional pump 1** option by unchecking the corresponding checkbox. You will see that objects assigned the option **Additional pump 1** are hidden.

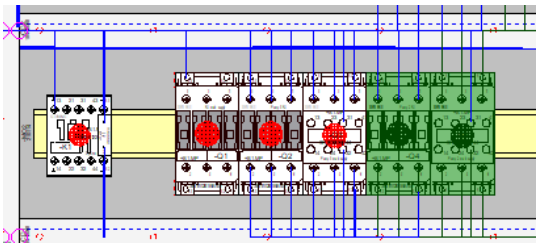


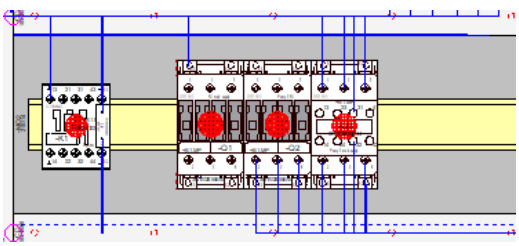


The corresponding objects are also “hidden” in the Device tree view and also on Panel sheet **6.2** in the **02-Panel** folder.









Lists and Output Files

Information contained in the project can be output to any document, list and/or file.

Thus, ***E³*** can be integrated in workflows in various ways. E³ offers for example predefined documents such as table of contents, bill of materials, connection lists, cable lists and terminal plans.

It is also possible to work with ***E³*** using the integrated programming interface (API). The API is based on Microsoft's COM technology (COM Interface) and offers the possibility to load or add data to the drawing using any programming language.

Several lists are predefined in ***E³*** and can be output directly in Microsoft Excel, Access, Word or ASCII format. Lists that are output in external programs may require the corresponding license. However, these outputs can also be embedded in the drawing as separate sheets.

To generate a cable list, use the **Tools → Reports → Excel → Cables** command. Excel starts automatically and generates a cable list, which contains all the cables used in the drawing along with any connected devices.

Cable List: Cooling water pump

				Cable	Type			Length (mm)
				=A1+BAZ1-W1	OFX-100-4G1.5			
Device Name	Pin	Connector Pin Ty	Wireseal	Core	Device Name	Pin	Connector Pin Ty	Wireseal	Length (mm)
=A1+BAZ1-S1	1			BK	=A1+S1.G-X4P	1	2-1105100-1		
=A1+BAZ1-S1	1			BU	=A1+S1.G-X4P	3	2-1105100-1		
=A1+BAZ1-S1	2			BN	=A1+S1.G-X4P	2	2-1105100-1		
=A1+BAZ1-S1	2			GNYE	=A1+S1.G-X4P	4	2-1105100-1		
				Cable	Type			Length (mm)
				=A1+BAZ1-W2	OFX-100-4G1.5			
Device Name	Pin	Connector Pin Ty	Wireseal	Core	Device Name	Pin	Connector Pin Ty	Wireseal	Length (mm)
=A1+BAZ1-S2	1			BK	=A1+S1.G-X4P	5	2-1105100-1		
=A1+BAZ1-S2	1			BU	=A1+S1.G-X4P	7	2-1105100-1		
=A1+BAZ1-S2	2			BN	=A1+S1.G-X4P	6	2-1105100-1		
=A1+BAZ1-S2	2			GNYE	=A1+S1.G-X4P	8	2-1105100-1		
				Cable	Type			Length (mm)
				=A1+Tank1-W1	Servo-700-4G2.5			4500
Device Name	Pin	Connector Pin Ty	Wireseal	Core	Device Name	Pin	Connector Pin Ty	Wireseal	Length (mm)
=A1+S1.G-X1P	1	2-1105100-1		1	=A1+Tank1-XS1	1	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	2	2-1105100-1		2	=A1+Tank1-XS1	2	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	3	2-1105100-1		3	=A1+Tank1-XS1	3	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	4	2-1105100-1		GNYE	=A1+Tank1-XS1	4	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	5	2-1105100-1		4	=A1+Tank1-XS1	5	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	6	2-1105100-1		5	=A1+Tank1-XS1	6	HE-C-Bu.AG-2.5		
=A1+S1.G-X1P	7	2-1105100-1		6	=A1+Tank1-XS1	7	HE-C-Bu.AG-2.5		

To generate a complete parts list, start the **Tools → Reports → Sheet → Bill of Material** command. The sheets containing the Bill of Material will be added automatically to the drawing.

bill of material

Higher Level Assignment: Location designation:

Pos.	No.	Article Number	Description	manufacturer	Device designation
1	3	1-1102283-2	HTS, HB Series, Housing, Size 4, Side Cl	AMP/Tyco	=A1+S1.G-X1SH, =A1+S1.G-X2SH, =A1+S1.G-X4SH
2	4	1-1103636-1	HTS, HE 10 Series, Insert, Size 4, Screw	AMP/Tyco	=A1+S1.G-X1P, =A1+S1.G-X2P, =A1+S1.G-X3P, =A1+S1.G-X4P
3	4	1-1103637-1	HTS, HE 10 Series, Insert, Size 4, Screw	AMP/Tyco	=A1+S1.G-X1S, =A1+S1.G-X2S, =A1+S1.G-X3S, =A1+S1.G-X4S
4	43	2-1105100-1	HTS, HE/HA Series, Contact, Pin, Male, 0	AMP/Tyco	=A1+S1.G-X1P(11), =A1+S1.G-X2P(11), =A1+S1.G-X3P(10), =A1+S1.G-X4P(11)
5	44	2-1105101-1	HTS, HE/HA Series, Contact, Socket, Fema	AMP/Tyco	=A1+S1.G-X1S(11), =A1+S1.G-X2S(11), =A1+S1.G-X3S(11), =A1+S1.G-X4S(11)
6	1	2CDS251001R0065	circuit breaker, 1 pole, 6A,T tipping cha	ABB	=A1+S1.MP-F9
7	1	3LD1504-0TB53	Emergency stop switch, 63 A, 22 kW, 3NO,	Siemens AG	=A1+S1-Q1
8	1	3RH1122-1BB40	Contactor relay, 3NO+0NC, DC 24 V, screw	Siemens AG	=A1+S1.MP-K1
9	4	3RT1025-1BB40	Contactor, AC-3 7.5 kW/400 V, DC 24 V, 3	Siemens AG	=A1+S1.MP-Q1, =A1+S1.MP-Q2, =A1+S1.MP-Q4, =A1+S1.MP-Q6
10	1	3RT1026-1BB44	Contactor, AC-3 7.5 kW/400 V, DC 24 V, 3	Siemens AG	=A1+S1.MP-Q3, =A1+S1.MP-Q5, =A1+S1.MP-Q7
11	1	3RT1916-1DG00	SUPPRESSION DIODE, DC 12...250 V, SURGE	Siemens AG	=A1+S1.MP-K1
12	1	3RV1021-0KA10	Circuit-breaker, size S0 A 0.9-1.25A,N 1	Siemens AG	=A1+S1.MP-F2
13	1	3RV1021-4AA10	Circuit-breaker, size S0 A 11-16A,N 192A	Siemens AG	=A1+S1.MP-F1
14	3	3RV1021-4AA15	Circuit-breaker, size S0 A 11-16A,N 192A	Siemens AG	=A1+S1.MP-F3, =A1+S1.MP-F4, =A1+S1.MP-F5
15	1	3SB3000-0AA11	22MM PLASTIC ROUND ACTUATOR: PUSHBUTTON	Siemens AG	=A1+S1.T-S3
16	1	3SB3000-0AA21	22MM PLASTIC ROUND ACTUATOR: PUSHBUTTON	Siemens AG	=A1+S1.T-S4
17	1	3SB3000-1HA20	22MM PLASTIC ROUND ACTUATOR: EMERGEN.-ST	Siemens AG	=A1+S1.T-S1
18	1	3SB3001-0AA61	22MM PLASTIC ROUND ACTUATOR: ILLUMINATED	Siemens AG	=A1+S1.T-S2
19	3	3SB3001-6AA20	22MM PLASTIC ROUND INDICATOR: INDICATOR	Siemens AG	=A1+S1.T-P4, =A1+S1.T-P1, =A1+S1.T-P5
20	1	3SB3001-6AA30	22MM PLASTIC ROUND INDICATOR: INDICATOR	Siemens AG	=A1+S1.T-P3
21	1	3SB3001-6AA40	22MM PLASTIC ROUND INDICATOR: INDICATOR	Siemens AG	=A1+S1.T-P2
22	2	3SB3400-0B	Actuator with one contact, 1NO, 1-pole,	Siemens AG	=A1+S1.T-S2, =A1+S1.T-S3
23	2	3SB3400-0C	Actuator with one contact, 1NC, 1-pole,	Siemens AG	=A1+S1.T-S1, =A1+S1.T-S4
24	5	3SB3400-1A	ACTUATOR/INDICATOR COMPONENT LAMP HOLDE	Siemens AG	=A1+S1.T-P1, =A1+S1.T-P2, =A1+S1.T-P3, =A1+S1.T-P4,

ZUKEN Bill of Material Dr. nr.: 0815 Doc. nr.: 4711

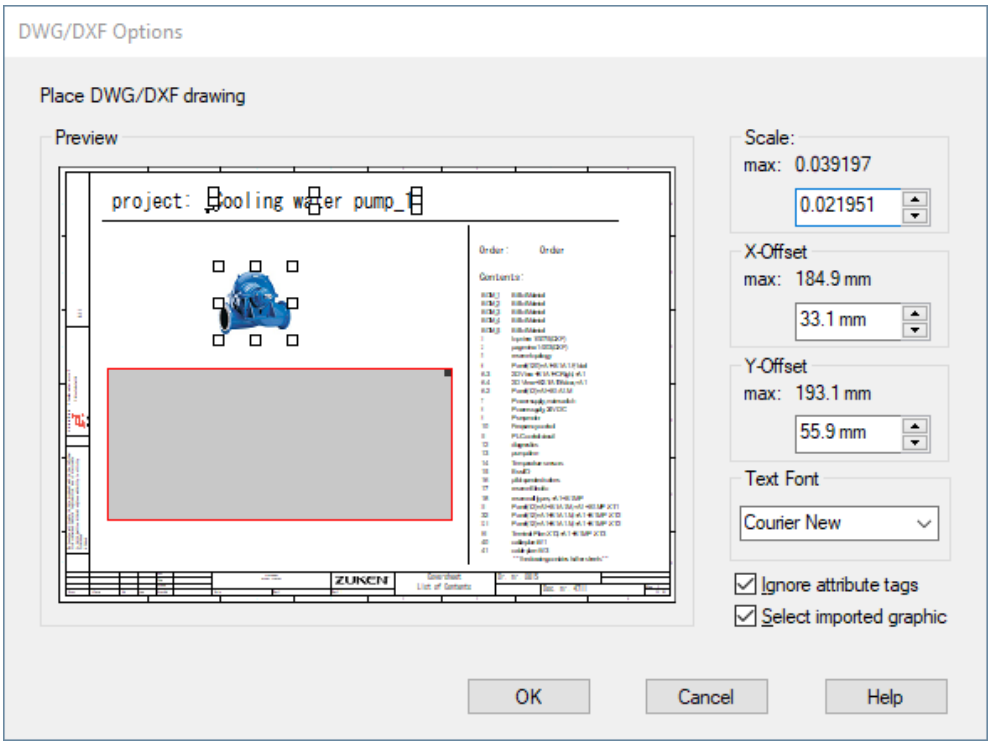
Table of Contents

Select the **Add-ons → E³.series Documents → Create Cover sheet** command to automatically create a cover sheet with a table of contents.

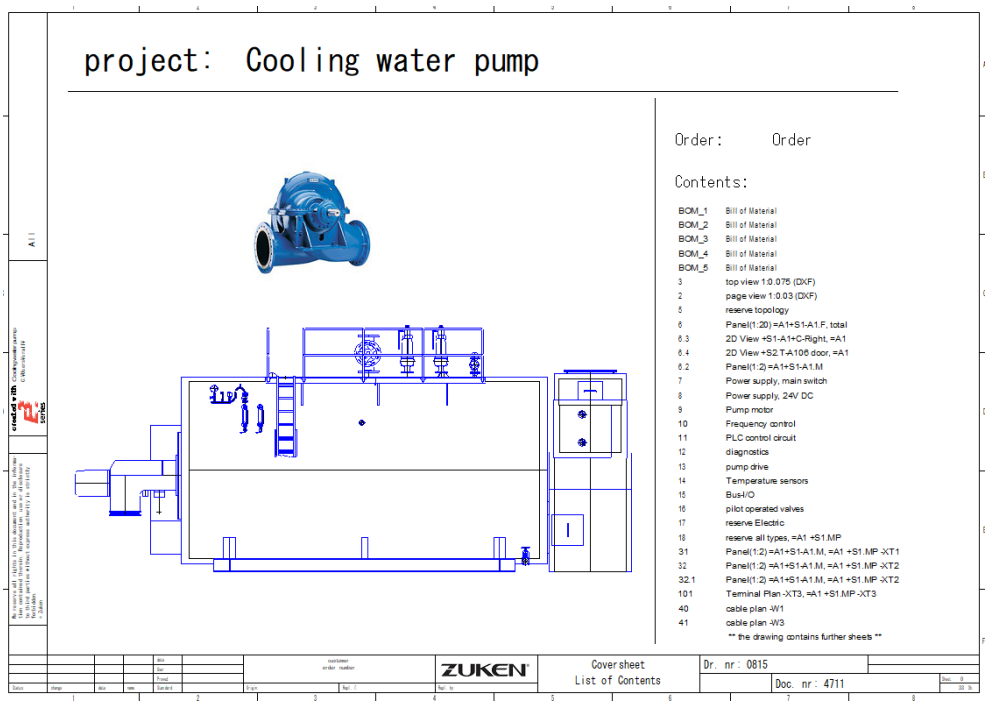
We are now going to add a graphic to our cover sheet, for example a picture of a pump or a logo. Select the command **File → Import → Image…** and then the **Pump-Z22.jpg** file from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\Pumpe* \. After placing the figure on the sheet, you can change its size.

Lastly, we need to add a mechanical drawing to our cover sheet. This can be done quickly and quite easily with the DXF/DWG import command.
Start the **File → Import → DXF/DWG…** command and select the **SideView.dxf** file from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\Pumpe*

The **DWG/DXF Options** menu appears, in which you can specify the size and position of the mechanical drawing. To change the size of the drawing, click on the "handle" in the upper right-hand corner of the gray rectangle and move it back and forth to adjust the size. The drawing can also be moved by simply clicking on the gray rectangle and dragging it to the desired position on the sheet. Should an exact scale and/or placement be desired, these values can be entered directly in the menu.



The cover sheet should look something like this:



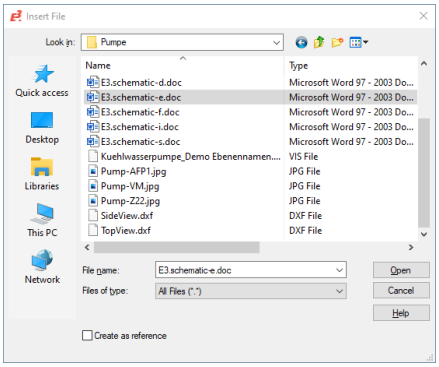
Embedding Additional Documents

Relevant information can be added to a project in ***E³.*schematic** if the user decides to do so. This can include documents from other applications, such as instruction manuals, buildup description, evaluation tables and so forth. ***E³.*schematic** allows you to integrate these documents into the project. A prerequisite is that the applications used to create these documents are compatible with Active-X (e.g., Microsoft Word, Excel).

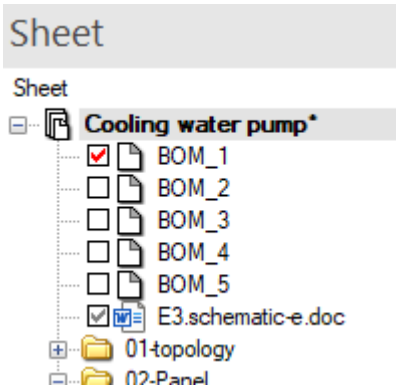
The following example will only work if you have Microsoft Word installed on your computer. Start the **Insert → File…** command. A menu appears, in which all of the Active-X compatible programs installed on your computer are listed.

Note: Files embedded in the project with **Insert -> Object…** can be placed on a sheet and depending on the file, be edited within the project. Files embedded in the project with **Insert -> File…** are inserted in the project structure. If the option **Create as reference** is activated, the file is linked in the ***E³*** project. If this option is inactive, a copy of the file is embedded in the structure.

Select the file **E³.schematic-e.doc** from the folder *C:\Users\Public\Documents\Zuken\E3.series_2020\data\Pumpe* \ and confirm with **OK**.



The document is embedded in the drawing of ***E³.*schematic**. As soon as you click on the document in the Project Window, the user interface changes to the Word interface so that you can edit the document. As soon as a sheet is selected from ***E³.*schematic**, you automatically return to the ***E³.*schematic** user interface.



Documents, which are embedded in your project, will also be saved. The file you selected, will remain in its original form and will not be changed by any actions within ***E³.*schematic**.

Printing Sheets

You can now print the sheets that you created or the entire project.

To do so, select the **File → Print…** command or the icon in the main menu toolbar. In the **Print** menu, you can specify for example which computer will be used to output the information, as well as the page format.

Exporting as PDF

The whole project can be exported as a PDF file. Thus, the project can be saved as a “neutral file” and can be displayed with any viewer that can read PDF files. To do so, select **File -> Export -> PDF…** from the main menu bar (a corresponding PDF license is required).

Besides the graphic a project structure of the sheets will be exported. Cross-references are saved as “links” and can thus be jumped to directly.

As in the Demo version the export functionality is not active, a PDF version of the project cannot be created. In the directories with the examples of the projects you will find, however, PDF files created from ***E³.*series**.

Saving a Project

After the entire project has been completed as well as the corresponding documentation created and printed, we must now save the project. Select the **File → Save** command or click on the icon. You can also save the drawing under another name with the **File → Save as…** command in case you want to return to the original state of the drawing.

You will need this completed project to continue working on the panel construction and wiring, which are described in the next chapter.

You should now be familiar with the basics of working with ***E³.*schematic**. Continue working with ***E³.*schematic** to become better acquainted with the wide range of functions the program offers, making the task of creating schematics much quicker and easier.

Enjoy working with

series

First Steps

Getting Acquainted with *E³.*panel for Panel Wiring and Construction

Welcome to Your First Tour through E³.panel

In this chapter you will be able to familiarize yourself with the main functions of this program. You will see how to construct and wire the panel using the circuit diagram's production data for manufacturing in combination with the circuit diagram.
Please read through the respective forewords of the individual chapters and perform the described actions. Information, pertaining to basic operations and the user interface, can be found in the chapter *The E³.series User Interface*. At the end of this tour, you will have learned about the various functions of ***E³.*panel**, which can then be applied to your design needs.

Have fun with this introduction!

Opening an Existing Project

***E³.*panel** is an additional tool for ***E³.*schematic** or ***E³.*cable**. To work with it, a special license for the corresponding module is required. ***E³.*panel** is completely integrated into the circuit diagram's creation and is based on the ***E³.*series** object-oriented data management system. This means that regardless of where the modifications are carried out, whether in the circuit diagram or in the panel wiring and construction, they will be updated immediately in all documents without the need for any additional updates. Thus, the panel construction offers another view of the components that have already been used and connected in the circuit diagram. Conversely, new components can be added to the panel at any time. They then exist as unplaced symbols in the circuit diagram, but as soon as the components are wired in the panel, the wiring information will also exist in the circuit diagram.

The following example uses the circuit diagram that was created in the previous chapter. Before beginning this chapter, make sure you have completed the schematic according to the examples described before.

If the program was ended and the drawing saved, start ***E³.*series** once again and open the project with the command **File → Open** or the corresponding icon from the toolbar. Select the project **Cooling water pump** from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\Pumpe*.

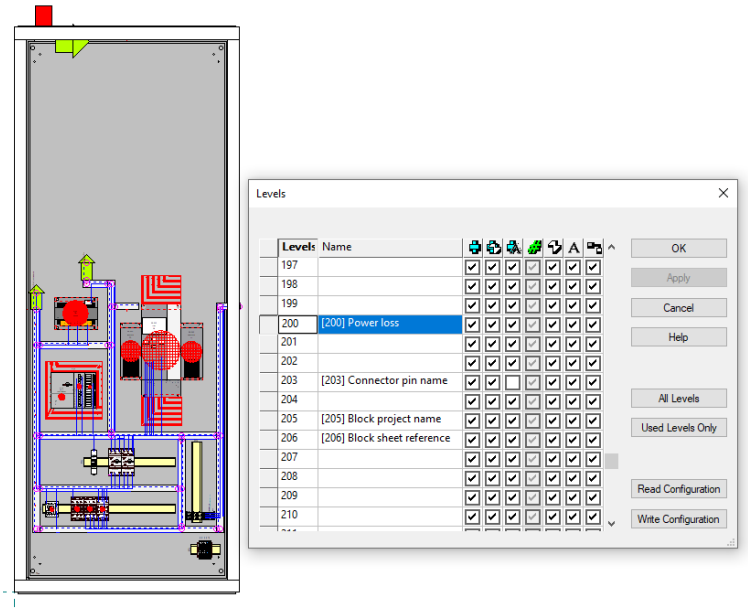
Part of the panel and wiring of the cooling water pump already exist for this project. In the following chapter, we will complete the panel as well as output the corresponding production records.

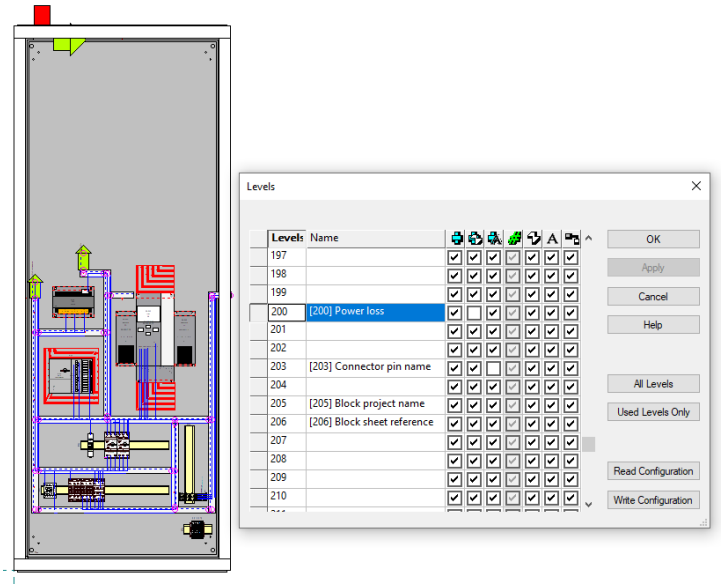
Open sheet **6** in the folder **02-Panel**. You can see the panel with its placed and wired devices.

E³ can display the panel in 2D as well as in 3D view. This means, the panel layout can be shown two-dimensionally as top view as well as three-dimensionally in a separate window.

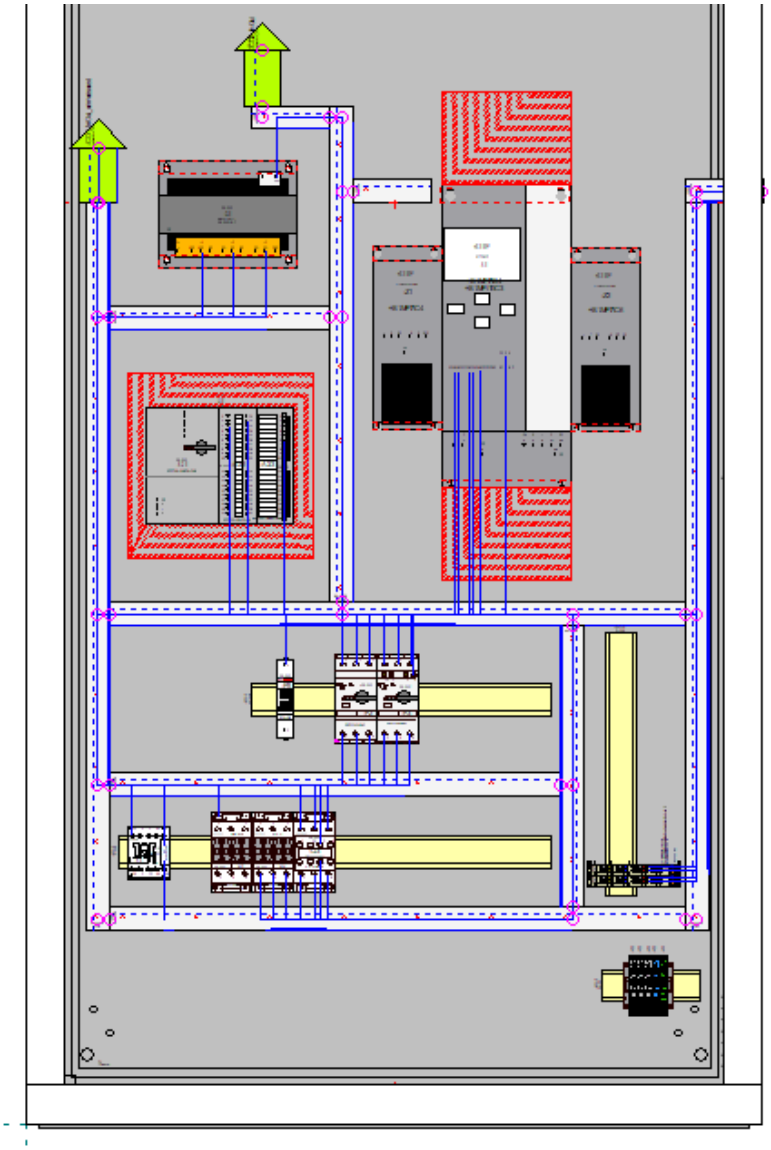
Naturally, all length specifications and checks in ***E³.*****panel** are calculated and proofed so that real results can be output three-dimensionally.

The areas in red show heat loss. Since this might not be very practical for drawing the plan, this level can be deactivated. Call up the **Levels** dialog using hotkey **L** or the **View → Levels...** command and deactivate the symbol graphic on level 200 (*Power loss*).





The panel view contains all objects relevant for the panel.
In addition to the devices, we can see the mounting rails (yellow), on which the devices are “snapped”, as well as the cable ducts (white), in which the wires are routed. Mounting rails and cable ducts exist as devices in the project but have no influence on the circuit diagram.



Defining Mounting Rails and Cable Ducts

Zoom in on the area to the left of the power switch **–F9** and the motor circuit switches. We are now going to insert an additional mounting rail for a terminal strip and a cable duct here.

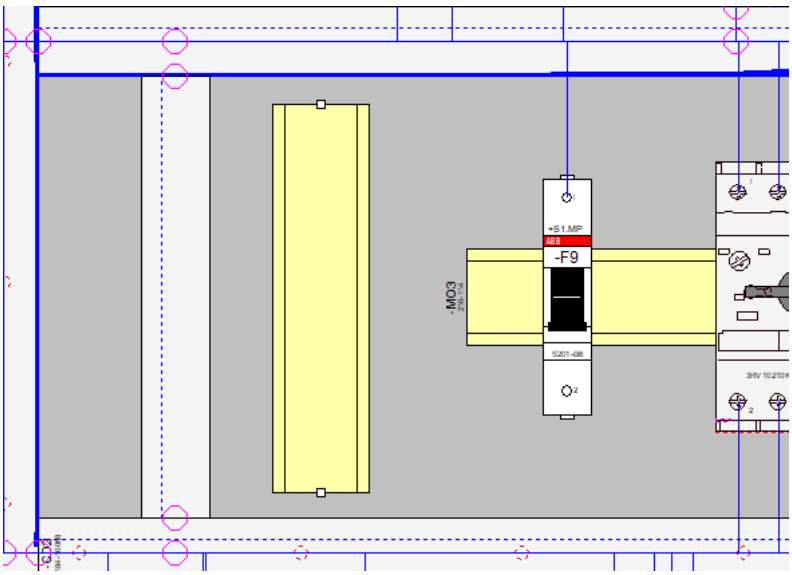
To insert a cable duct, call up the **Insert → Cable Duct...** command or use the icon from the toolbar. The duct's width and height (some also say its depth) can now be defined in the menu. Enter the **Width** as **25 mm**. It is also possible to specify an exact length for the cable duct. We will use a dynamic length for our duct. Do not enter a value for **Length** and close the menu with **OK**. Since a cable duct is treated as a device, it also receives a device designation.

The cable duct symbol will appear at the cursor position. Position the cursor on the upper cable duct, press the left mouse button, drag the mouse onto the lower cable duct and release the mouse button. This establishes the duct's starting point, length and direction, and then connects the two existing ducts.
You can also "overlap" the existing cable ducts. ***E³.*panel** automatically adjusts the ducts to the right length. Notice the small circles appearing on the upper and lower cable ducts where the new duct touches them. This indicates that the channels are logically connected and that these ducts can now be wired.

The mounting rail can be defined in the same way.

Use the **Insert → Mount** command or the icon from the toolbar. The **Insert Mount** dialog appears. Define the specifications for the mounting rail description. Enter the value **TS 35 mm** as the **Slot Description**. This is required because ***E³.*panel** uses this description to see if a device can be placed on a specific mounting rail. It compares the device's mounting rail description to see if it matches the mounting rail type, on which it will be placed. If it does, then it can be placed on this mounting rail, otherwise it rejects the placement. This prevents devices from being placed on a mounting rail, on which they do not fit. Insert the new mounting rail vertically in this area.

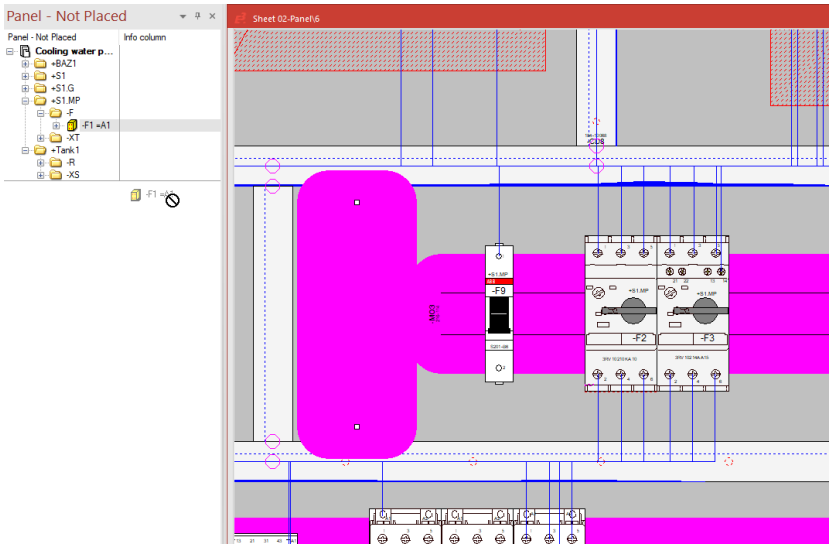
After the cable duct and mounting rail have been placed, they can be moved and their size changed at any time. Simply press the left mouse button and move them.



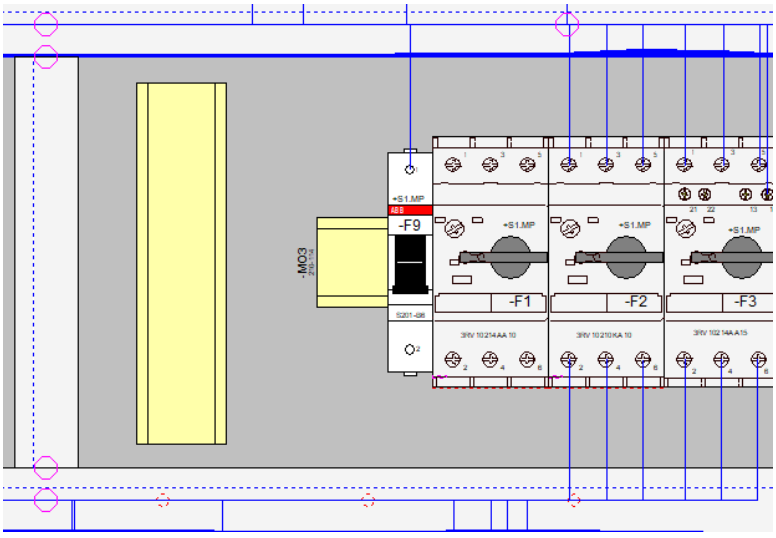
Placing Devices

Devices, which already exist in the circuit diagram, can now be placed and wired in the panel. Every device in the circuit diagram, which contains a corresponding panel description in the database, is listed in the **Not Placed** tab of the **Panel** Tree view in the Project Window.
This makes it easy to recognize, which devices can still be “placed“.
Devices are placed in the panel just like symbols in the circuit diagram using “Drag & Drop”. Each device is represented by a symbol, the device model ( ). This can be dragged from the *Project Window* onto the panel sheet.

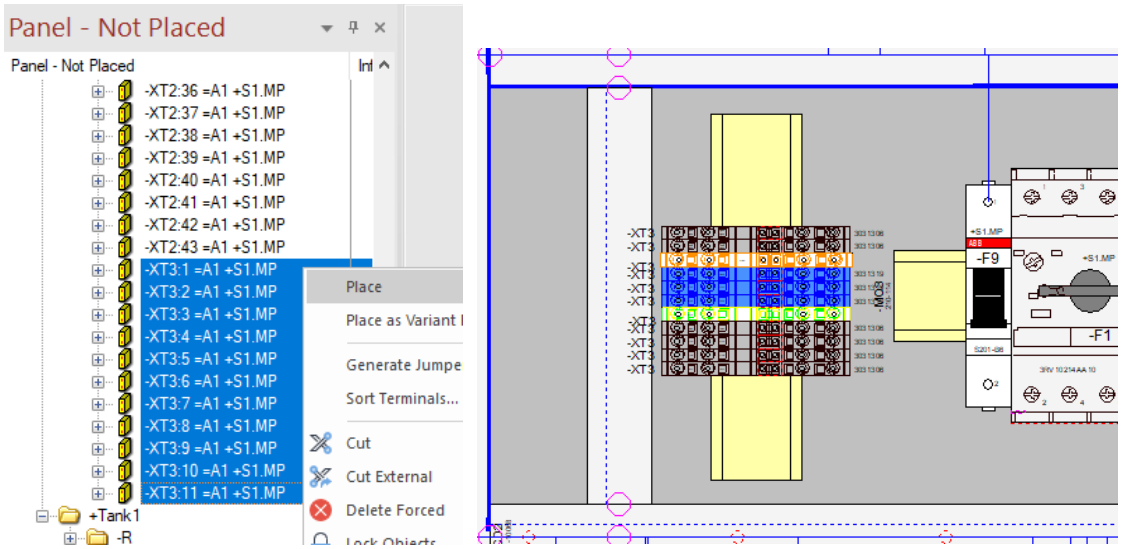
Open the **Panel - Not Placed** tab and there under the folder **+S1.MP** you will find the device **-F1**. This needs to be placed on the appropriate mounting rail in the panel.
Drag the device **–F1** from the *Project Window* and position it on the mounting rail between the devices **–F9** and **–F2**. As soon as a device is moved from the Project Window, all mounting rails, on which the device can be placed, are highlighted. When the device touches a highlighted mounting rail, it is snapped onto this rail.
Devices cannot be placed on one another, on cable ducts or on other invalid surfaces. ***E³.*panel** checks this constantly and only allows devices to be placed at valid positions.



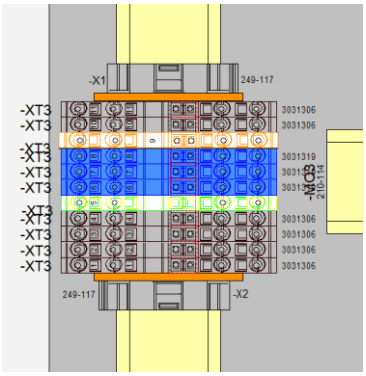
As soon as a device is placed, it is moved from the **Panel - Not Placed** tab to the **Panel – Placed** tab. In this tab, you will find all devices, mounting rails and cable ducts structured according to their placement. This makes it easy to recognize, which devices are placed on which mounting rails.



When placing a complete terminal block, the individual terminals are not placed one after the other but rather, they will be selected as an area from the *Project Window* **Panel – Not Placed** tab and then placed through the context menu. Switch to the Project Window's **Panel – Not Placed** tab, select the corresponding terminals of terminal strip **-XT3**. Call up the context menu and select the **Place** command to place all of the terminals at once.
Note: If the terminals cannot be placed, the distance between the terminal strip and the mounting rail is too short.



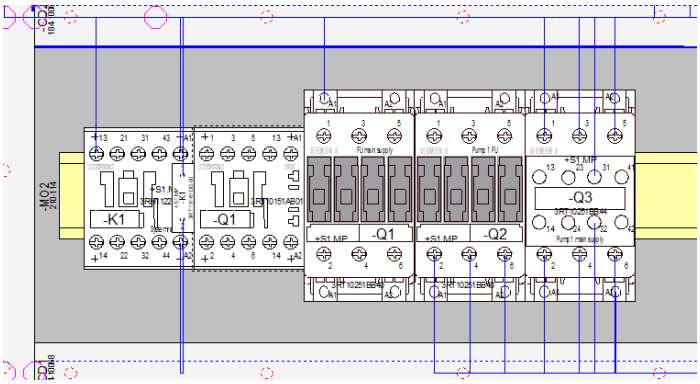
The terminal block will be completed with the appropriate end clamps (component: **249-117**) and end separation plates (component: **280-331**) from the database.



Not only can devices, which are already used in the circuit diagram, be placed, but components can also be taken directly from the component database and placed in the panel. Therefore, a component can be loaded directly into the project and then used later in the circuit diagram. Components are loaded from the *Database Window* onto the panel sheet using "Drag & Drop".

Open the folder **Electric – Main contactor 3-pole AC** in the *Database Window* **Component** tab. Search and select the component **3RT10151AB01**. You can see the model, which the component has been assigned in the Preview Window, provided that the panel sheet is active.

Place the component on the lower mounting rail directly next to the device **–K1**. By dragging the component onto the panel sheet, it is loaded into the project and simultaneously placed.
To select the device in the Project Window, call up the command **Jump to Tree** in the context menu. You can also see, on which mounting rail the component was placed as well as the new device designation it was assigned in the Project Window.



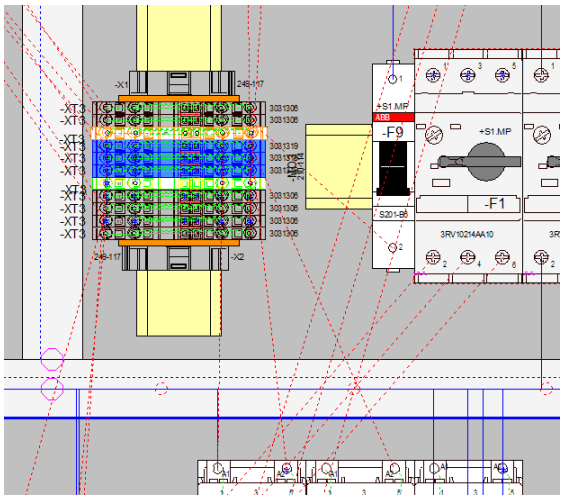
If you switch over to the **Device** tab in the *Project Window*, you will see all of the device's circuit diagram symbols there. They are not yet placed and can now be used in the circuit diagram. Modifications, which are made to the circuit diagram, will be updated immediately in the panel layout. You can, for example, change the device designation in the **Device** *Tree* from **–Q1** to **–Q100** using the **Rename** command in the context menu or the function key **F2**. As soon as the renaming has been completed, the new device designation will be entered automatically on the panel sheet.

Wiring in the Panel

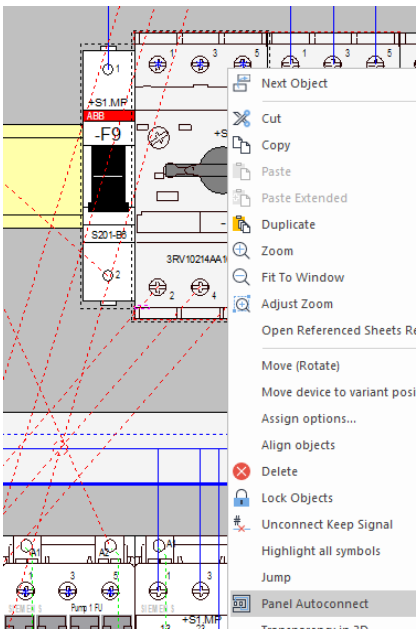
All electrical connections, which are defined in the circuit diagram, will be converted into "real" wires in the panel wiring. If wires or cables are already defined in the circuit diagram, they will also be used. Otherwise, the defined wire type will be used as the standard wire, provided that it fits.

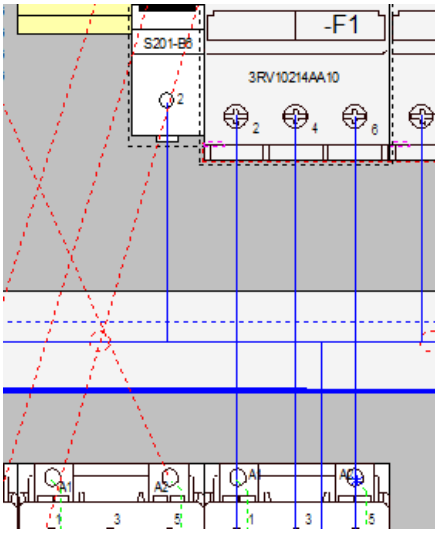
In order to see, which connections are already defined in the circuit diagram, switch on the signal logic display. This can be easily accomplished by pressing the icon from the toolbar.

You can see broken lines directly connecting some of the connect points. These show the wires, which must be laid. The connections show the shortest pathway and the defined connection sequence, depending on the number of wires, which can be assigned to a pin.



To save time when routing wires, ***E³.*panel** offers the possibility to create connections automatically.
Select the devices **–F9** and **-F1** on the panel sheet, right-click and select the **Panel Autoconnect** command from the context menu or from the toolbar. ***E³.*panel** now starts the automatic routing, in which it tries to find the nearest cable duct and the shortest connection within the cable duct net. When routing, boundary conditions, such as fill size and provisions for restricted areas, are considered.

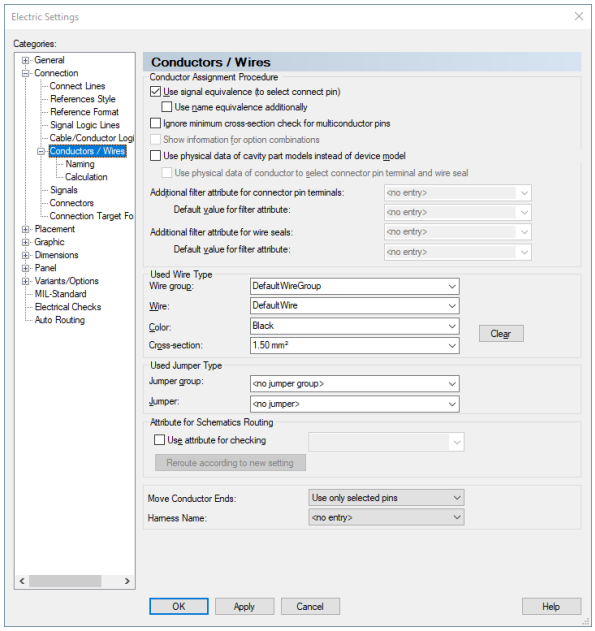




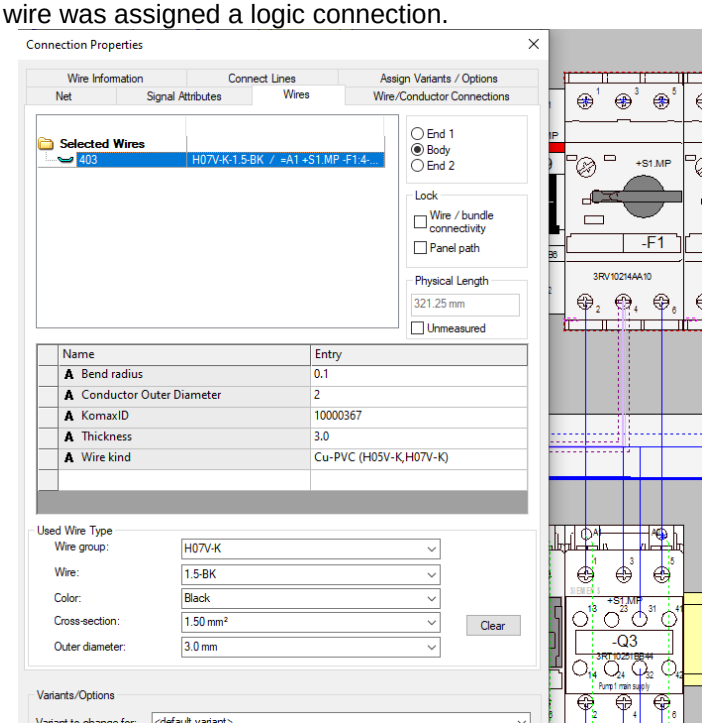
Now create the routing for the complete panel.
You can use the **Panel Autoconnect** command for each individual device or for an area, which contains several devices, or by using **Ctrl+A** to select all devices on the sheet and then execute the command.

To route only one wire to a pin, the command can also be started through the pin's context menu. In this way, it is also possible to route all wires in the panel.

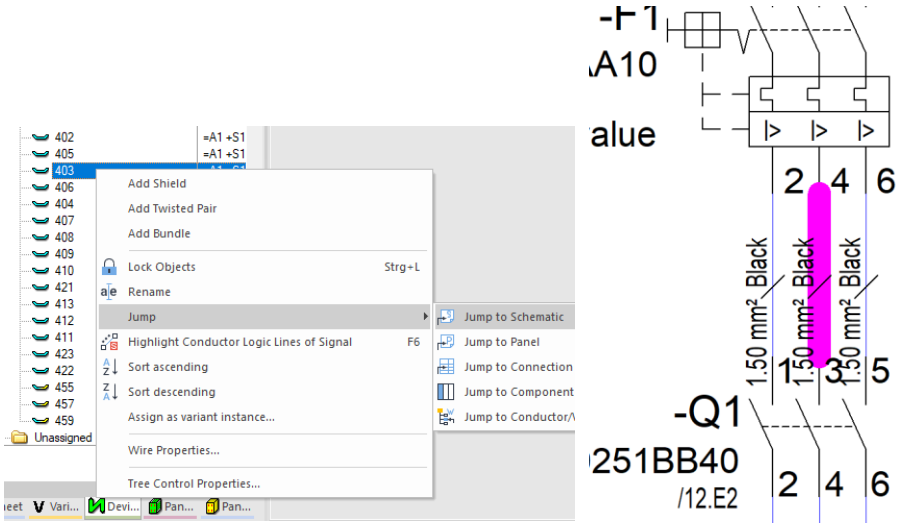
If it is not possible to route a wire (no cable duct pathway found, all ducts are full, too many wires to a pin, the defined wire cross-section does not fit the connection), this will be displayed in the Output Window's **Results** tab. You can then react in a corresponding manner (enlarge the cable ducts, define other pathways, change the wire type in the **Tools → Settings → Connection → Conductors/Wires** or using the toolbar).



Now we also want to see the results in the circuit diagram.
With the automatic panel connection, each wire was assigned a logic connection.
Click on the middle wire coming from the underside of device **–F1**. The wire will be highlighted and its routing can be easily viewed. Call up its **Connection Properties** from the context menu and click on the **Wires** tab. The wire will be displayed with its wire attributes, connected devices and wire number. This is automatically assigned as an ascending number, which can also be changed. Keep in mind the wire number (e.g. **403**).

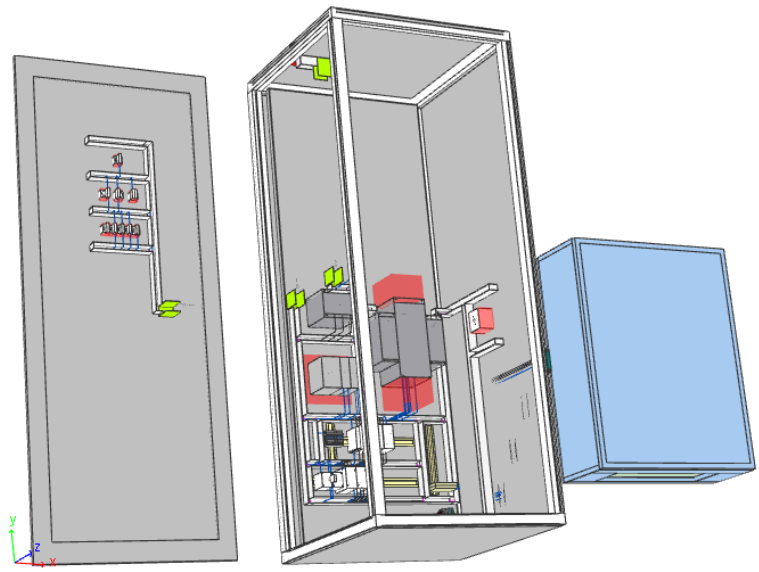


Close the panel sheet and switch to the Project Window's **Devices** tab. Locate the wire **403** in the **<Wires>** folder and select the command **Jump to Schematic** in the wire's context menu. This opens the corresponding sheet in the circuit diagram showing the connection on which this wire was placed by ***E³.*panel** and highlights the connection.



3D Panel Display

The 3D panel functionality in ***E³*.series** builds upon the familiar 2D functionality.
It is possible to dynamically switch between the 2D and 3D display on the **base sheet**. All commands, like Place, Move, Rotate, Route and so on are possible in both 2D and 3D views.
.



To toggle between the different display modi we will use the **Panel** toolbar. Right-click on any toolbar and select **Panel** from the displayed context menu.

Inserts a new mounting rail.

Inserts a new cable duct.

Inserts a new model contour (restricted areas, cutouts or drill-holes).

Automatically creates physical routing in panel.

Displays the wire parameters.

Displays the connection logic lines on pin.

Marks sheet comprehensive wires.

Switches between 2D and 3D view.

Displays models in wireframe or solid mode.

Displays perspective or orthographic (vertical) projection.

Activates/Deactivates lightning for various shading effects.

Displays model orientation in 2D.

Displays model orientation in 3D.

Displays 2D model graphic (top view) in 3D.

Displays STEP models in 3D.

Navigating within 3D View using Mouse

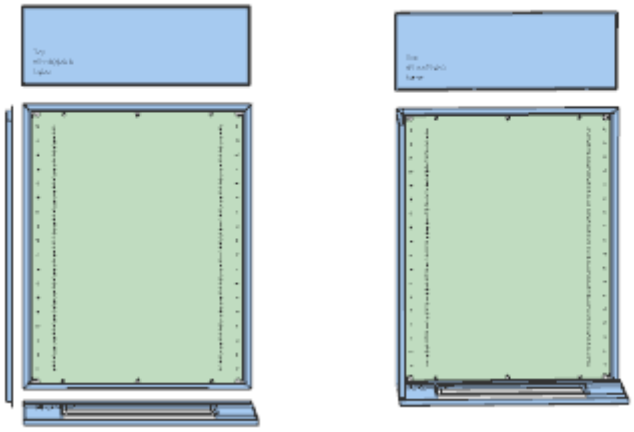
For navigating within the 3D view, the mouse keys are used as follows:

Mouse Assignments	Function
Click left mouse button	Selects object
Click and hold left mouse button	Move mouse horizontally to rotate 3D world on Z axis; move mouse vertically to rotate 3D world on X and Y axes.
Shift + click left mouse button	Add or remove an object
Ctrl + click left mouse button	Jumps to the tree view
Click right mouse button	Calls up object menu
Click and hold right mouse button	Moves the 3D world on the X and Y axes.
Scroll bar	Zoom A zoom rectangle can be created (like in 2D).
Click and hold right and left mouse buttons simultaneously	Rotates complete 3D world

Try the various 3D display and navigation possibilities.

We will now enhance the right panel on the panel base sheet **02-Panel\6** in 3D display mode. Activate the 3D display mode 3D , the 2D model graphic display and the orthographic (vertical) display using the Panel toolbar.

We will start by moving the left side wall and the floor onto the panel's back wall.

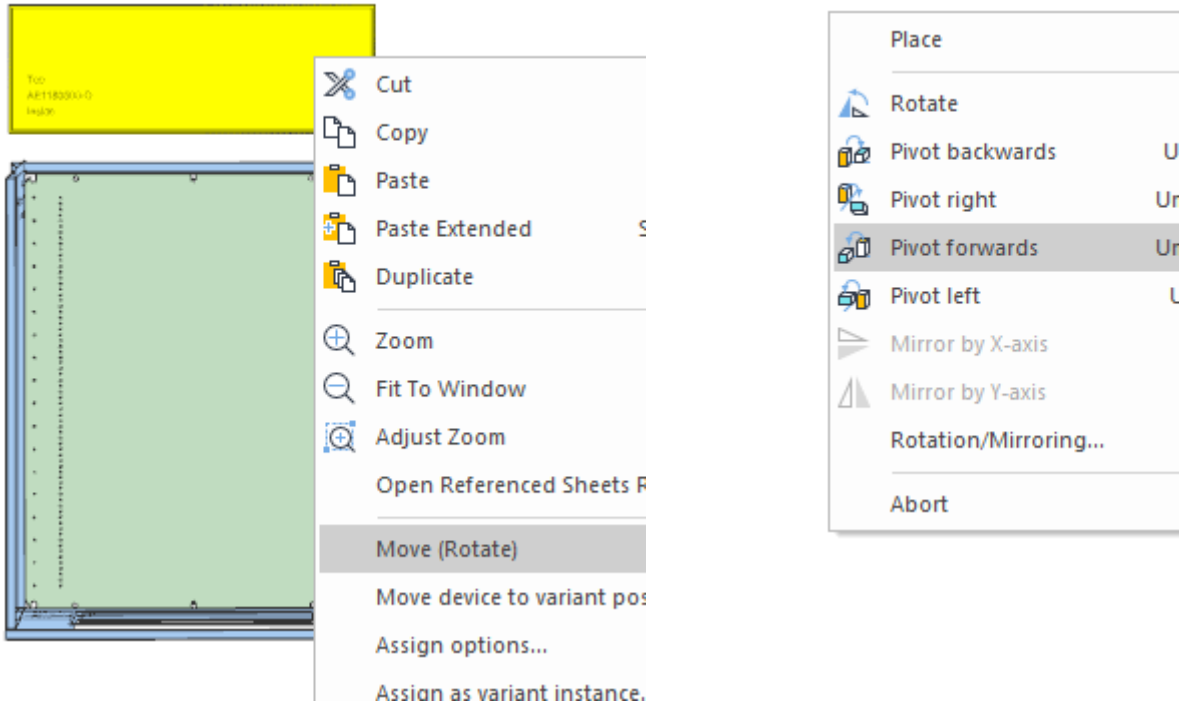


Before placing the top on the panel back plane, we need to add a signal lamp to the outside of the top.

To do so, right-click on the top and select the **Move (Rotate)** command from the displayed context menu, right-click again and select the **Pivot forwards** command from the displayed menu. Place the top. The top view corresponds to the top outside.

The following hotkeys are available for the “pivot” commands:

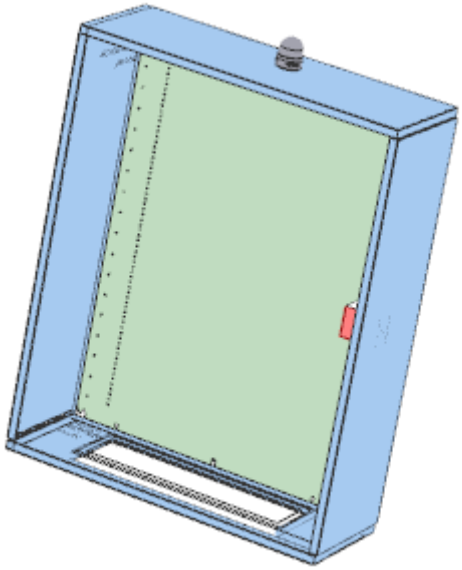
Pivot backwards: Shift + arrow key up
Pivot right: Shift + arrow key right
Pivot forwards: Shift + arrow key down
Pivot left: Shift + arrow key left



Now, select the indicating lamp **201.100.75** from the **Electric-→ Indicating element/lamp** folder and place it on the outside (upper side) of the top. Pivot the lamp forwards. The 3D view shows how the lamp goes through the top.

Next, we will mount the right side wall and a main switch to the back plane.
Select ***AE1180500-SW*** from the **Electric → Cabinet** folder of the database and place it to the right side of the panel.
Then switch to the Database window again to select the main switch **3LD11040TB51** from the ***Electric → Main switch*** folder and place it on the inside of the side wall.
Then pivot the side wall to place it onto the back plane. Right-click on the side wall, select the **Move (Rotate)** command from the context menu, then right-click again and select the **Pivot left** command. Afterwards, move the pivoted side wall to the right side of the panel back plane.

The completed panel should look as follows:



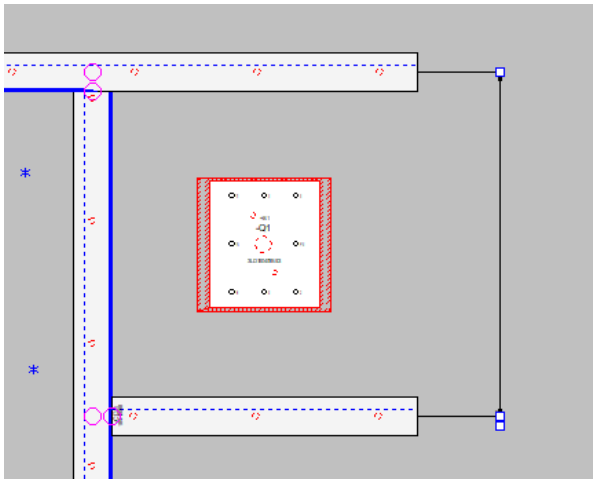
Dimensioning the Panel

We will conclude by adding dimensions to our panel. In this example, we will use the panel door of the complete cabinet. Open sheet **6.3** in the **02-Panel** folder.

The dimensions can be defined by using either the *Dimension* icon or the hotkey M

or the **Insert → Dimension** command in the Main Menu.

In the example shown, we will use the middle of the cable ducts. ***E³.*series** identifies potential dimension points by a cross. Click on a possible dimension point with the left mouse button. Find the second dimension point in the same way. Click on the left mouse button once again to define the dimension.



Outputting a Wiring List

***E³.*panel** can generate a wiring list, in which all of the wires that have been placed will be output along with their attributes (From, To, Wire number, Wire type, Color, Wire cross-section, length and routing). Execute this function with the **Add-ons → EXCEL Documents → Wiring List from Panel** command from the main menu bar.

Wiring list: Cooling water pump

From	To	Wire number	Wire type	Colour	Wire cross section	Length (mm)	Cable duct's routing path
=A1+S1.MP-Q2 :2	=A1+S1.G-X1S :1	53	H07V-K-1.5-BK	Black	1.50 mm²	2424	=A1+S1.MP-CD1; =A1+S1.MP-CD9;
=A1+S1.G-X1S :3	=A1+S1.MP-Q2 :6	55	H07V-K-1.5-BK	Black	1.50 mm²	2409	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X1S :5	=A1+S1.MP-A2.3 :6	189	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2257	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X1S :6	=A1+S1.MP-A2.3 :7	190	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2233	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X1S :7	=A1+S1.MP-A2.3 :8	191	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2234	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X1S :8	=A1+S1.MP-A2.3 :9	192	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2236	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.MP-Q2 :4	=A1+S1.G-X1S :2	54	H07V-K-1.5-BK	Black	1.50 mm²	2416	=A1+S1.MP-CD1; =A1+S1.MP-CD9;
=A1+S1.G-X2S :5	=A1+S1.MP-A2.3 :12	193	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2247	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X2S :6	=A1+S1.MP-A2.3 :13	194	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2223	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X2S :7	=A1+S1.MP-A2.3 :14	195	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2224	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.G-X2S :8	=A1+S1.MP-A2.3 :15	196	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2226	=A1+S1-CD1; =A1+S1-CD2; =A1+S1.M
=A1+S1.MP-A5 :A1	=A1+S1.G-X4S :1	152	H05V-K-0.75-DBU	Dark blue	0.75 mm²	1969	=A1+S1.MP-CD10; =A1+S1.MP-CD1
=A1+S1.MP-A5 :A2	=A1+S1.G-X4S :3	153	H05V-K-0.75-DBU	Dark blue	0.75 mm²	1979	=A1+S1.MP-CD10; =A1+S1.MP-CD1
=A1+S1.MP-A6 :A2	=A1+S1.G-X4S :7	155	H05V-K-0.75-DBU	Dark blue	0.75 mm²	1991	=A1+S1.MP-CD10; =A1+S1.MP-CD1
=A1+S1.MP-A6 :A1	=A1+S1.G-X4S :5	154	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2009	=A1+S1.MP-CD10; =A1+S1.MP-CD1
=A1+S1.MP-K1 :A1	=A1+S1.MP-XT3 :2	113	H05V-K-0.75-DBU	Dark blue	0.75 mm²	257	=A1+S1.MP-CD2; =A1-U1
=A1+S1.MP-K1 :1	=A1+S1.MP-K1 :A1	202	H05V-K-0.75-DBU	Dark blue	0.75 mm²	261	=A1+S1.MP-CD2
=A1+S1.MP-XT3 :7	=A1+S1.MP-K1 :A2	114	H05V-K-0.75-DBU	Dark blue	0.75 mm²	550	=A1-U1; =A1+S1.MP-CD2; =A1+S1.M
=A1+S1.MP-K1 :2	=A1+S1.MP-K1 :A2	201	H05V-K-0.75-DBU	Dark blue	0.75 mm²	257	=A1+S1.MP-CD1
=A1+S1.T-S2 :13	=A1+S1.MP-K1 :13	92	H05V-K-0.75-DBU	Dark blue	0.75 mm²	2502	=A1+S1.T-CD2; =A1+S1.T-CD5; =A1-

This was a short overview of the functionality of ***E³.*panel**. If you continue working with ***E³.*panel** you will get to know other functions, which will support your work and your preparation for production.

Enjoy working with

series

First Steps

Getting Acquainted with *E³.*cable for Cabling Devices

Welcome to Your First Tour of E³.cable for Cabling Devices

By following the steps described below, you will be able to familiarize yourself with the main functions of ***E³.*cable** for cabling devices. This tour points out how to create reliable circuit diagrams and cable drawings. Please read through the respective forewords of the individual chapters, which will then allow you to perform the described actions. Information, pertaining to basic operations and the user interface, can be found in the chapter The ***E³.*series** *User Interface*.
At the end of the tour, you will have learned about the various functions of ***E³.*cable**, which can then be applied to your design needs.

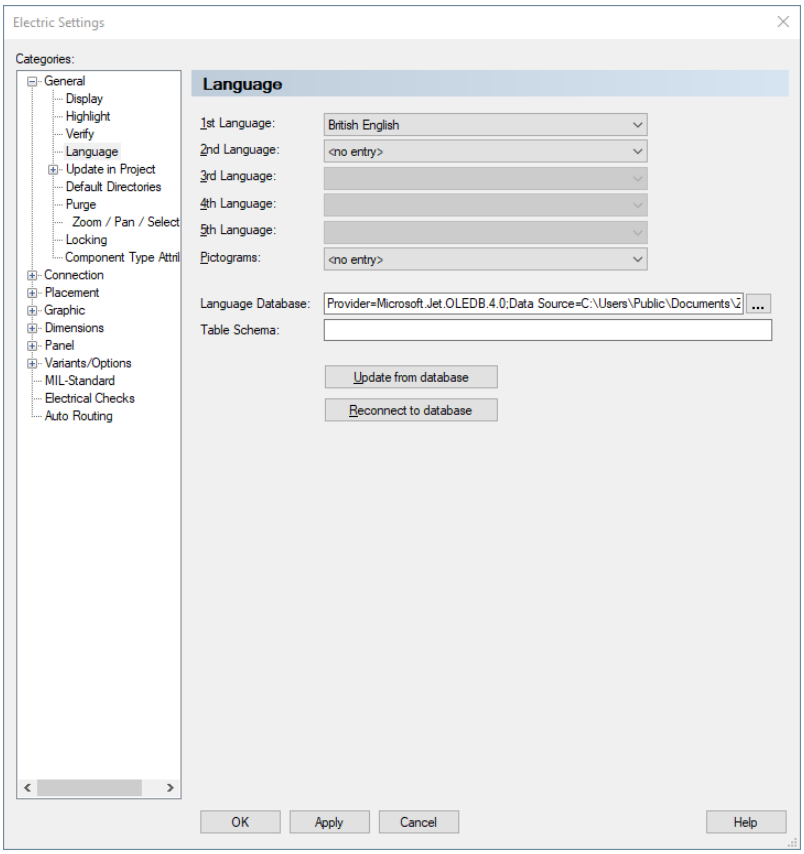
Have fun with this introduction!

Opening an Existing Project

With ***E³.*cable**, you always work within a project. A project may be comprised by any number of drawing sheets and additional documents, which are stored in a single file. Start the program.

To open a project, use the **File → Open** command or the corresponding icon in the standard toolbar. Select the project **CD-Player.e3s** from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\CD-Player\.*

First, the project language needs to be defined. Select the **Tools → Settings** command from the main menu bar to open the **Settings → General → Language** dialog. Define British English as the first language. Confirm with **OK**.



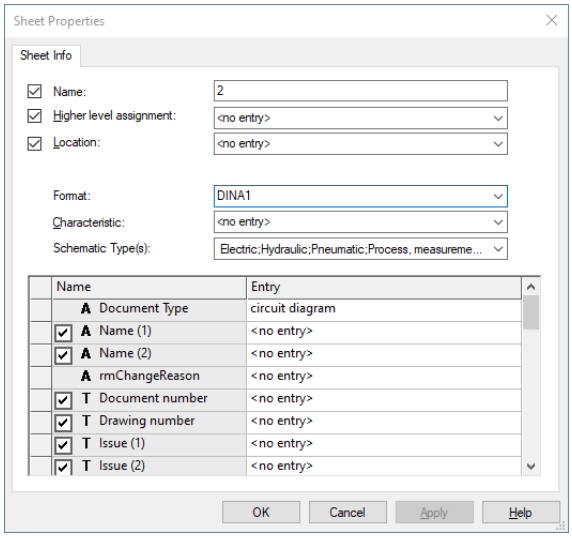
Changing the Sheet Format

Switch to the Project Window's **Sheet** tab. Open the *Circuit diagram* folder and select sheet **2** by double-clicking on the sheet itself or by simply clicking on the box in front of the sheet.

Different sheet formats (e.g. DIN, extra-wide, Ladder Diagram for the American market, etc.) with their unique size, layout and sheet header information are stored in the database.

The sheet format used in this example is **DINA1**. The sheet format can be changed at any time through the **Sheet Properties** menu. Right-click anywhere on the sheet to reveal the **Sheet Properties** menu and by clicking on the **Format** field, a drop-down list of all the sheet formats defined in the database will appear.

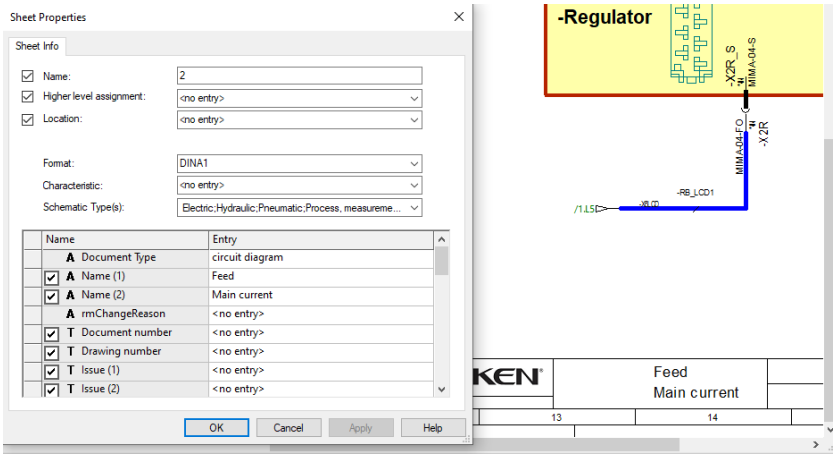
If a sheet format selected does not "fit" to the sheet and its information, an error message is displayed. If this is the case, make sure you select an appropriate sheet format.



Entering Sheet Header Information

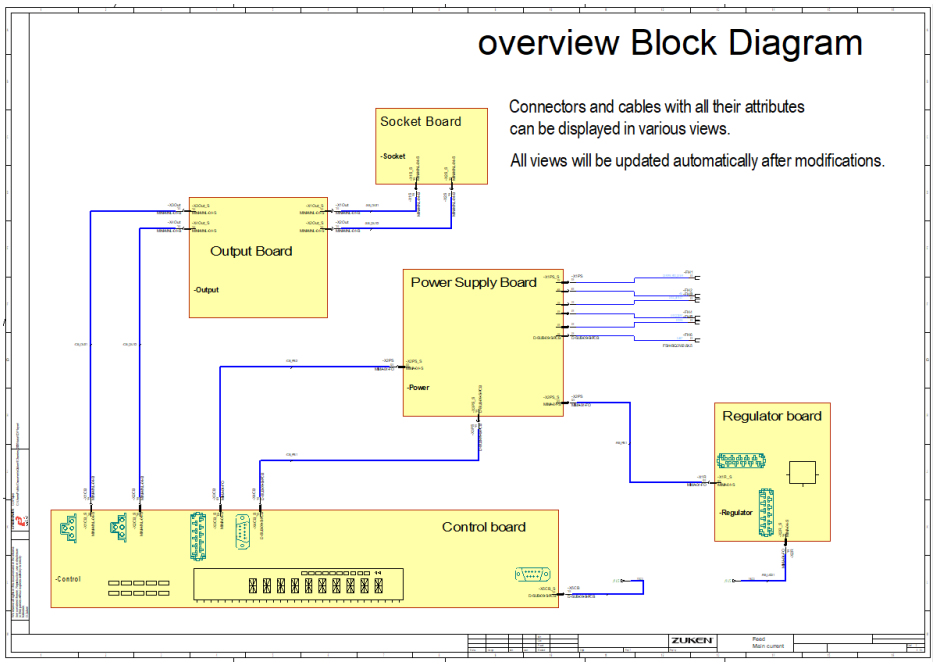
Sheet headers contain information typically found in the drawing sheet's margins, e.g. Drawing Title, Drawn by, Date, Revision, and so forth.
The sheet headers are displayed in the **Sheet Properties…** menu, which can be called up through the context menu by right-clicking on any empty area of the sheet or on the sheet in the Sheet tree view.

The standard text can now be entered in the sheet format.
Enter the text **Feed** for the **Name (1)** attribute, and **Main Current** for **Name (2)**.
Confirm with **OK**. The texts are displayed in the sheet's footer.

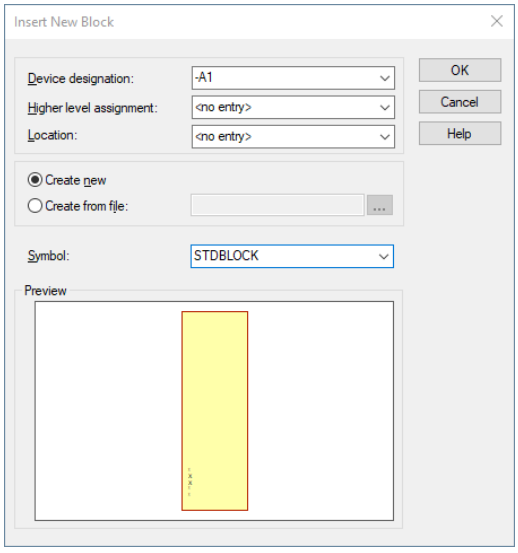


Placing Blocks

The example drawing contains several functional blocks. A functional block or simply block, contains either an electronic circuit (i.e. originating from an EDA system) or, as in the example, represents a functional unit (like a black box).

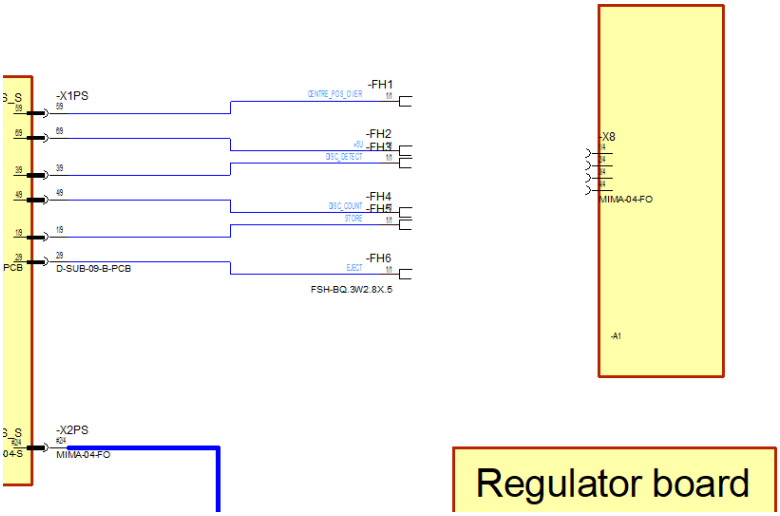


Now, place an empty block and label it. To place a block in the drawing, use the command **Insert → Block…** from the main menu bar. Select the standard block symbol **STDBLOCK** from the drop-down list in the **Insert New Block** dialog. The block symbol appears at the cursor position and can now be moved to the desired location. Left-click to place the block above the regulator board.



Next, select the connector **MIMA-04-FO** from the **Electric - Micro-MaTch** folder in the component database and place it on the block as shown in the image below.

The connector's contacts can be placed as a complete connector, as a pin group or as single pins. In our example, we will display them as single pins. Open the connector in the Database Window, select all of its pins simultaneously and place them onto the block's border using Drag & Drop or the **Place** command in the context menu. The connector pins snap on the border and can be moved along the block.



Placing and Connecting Device Elements

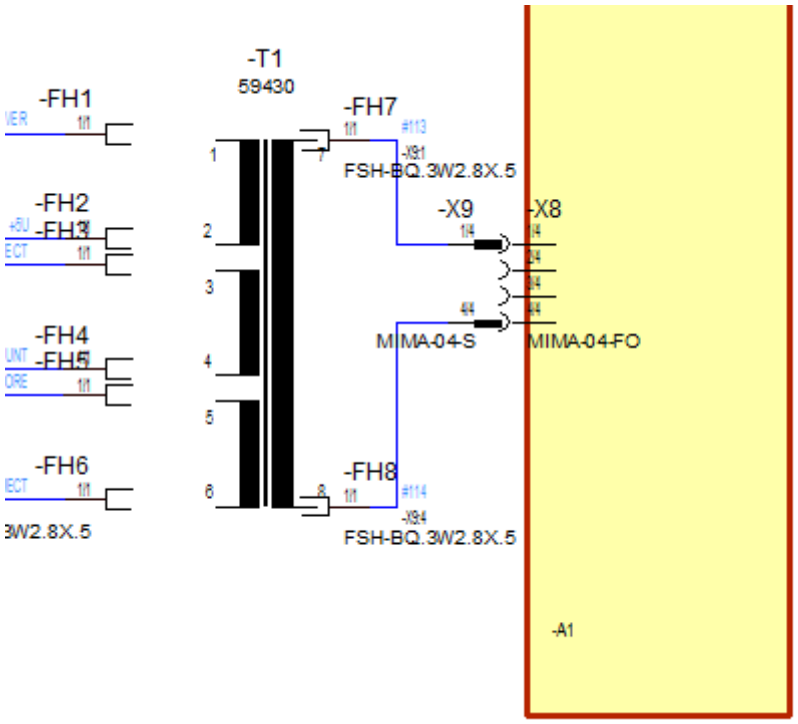
The circuit must now be completed by adding a transformer. Select the component **59430** from the **Generic_E - Transformer** folder in the Database Window and place it as shown in the figure.

Connect the transformer's primary connections **7** and **8** to the block connector pins. This can be accomplished by using the **Connection** command in the pin's context menu, the icon or the key **C**.

Once the command is activated, the cursor's shape changes. Position the cursor on one of the block's connector pins. Notice how its shape changes. Left-click on this pin to start drawing the connection. As you move the mouse, the connection is displayed as it would be drawn along with any corner points, which are automatically defined by the program.

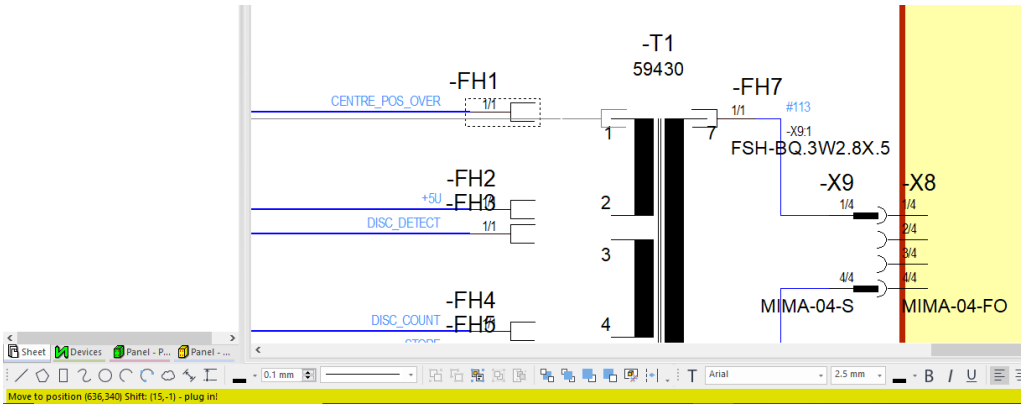
Corners can also be entered manually by clicking the left mouse button. When the mouse is moved across another possible connect point, the cursor again changes shape. Click on the mouse to complete the connection.

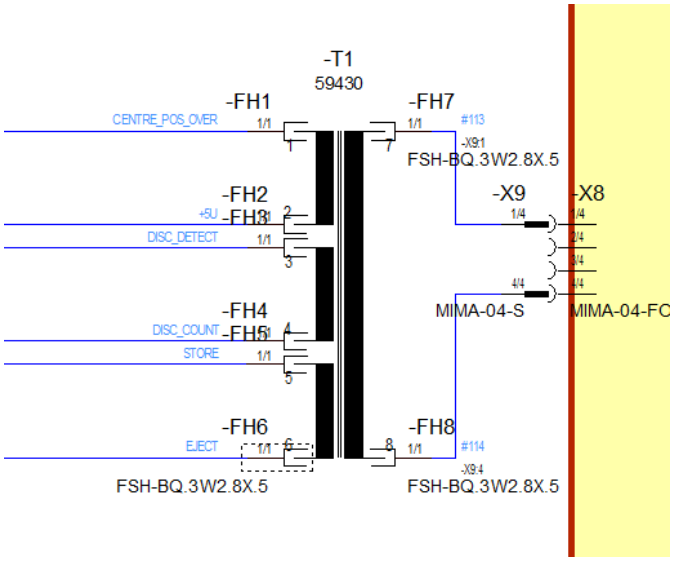
Once connected, the appropriate connector pins will be added automatically to each end of the connection. Since possible mating connectors and crimp parts for the connectors are already defined in the database, ***E³.*cable** will use the mating connector defined for the connector pin.


-T1
59430
-FH1
-FH2
-FH4
-FH6
-FH7
-FH8
FSH-BQ.3W2.8X.5
FSH-BQ.3W2.8X.5
-X9
-X8
MIMA-04-S
MIMA-04-FO
-A1

Plugging (Unplugging) Plug Connectors

We now have to connect the transformer's secondary pins with the predefined plug connectors. Select one or several plug connectors simultaneously and by dragging, connect them to the appropriate pin(s). The elements are physically connected when the status bar turns yellow and the cursor displays two connected connectors.

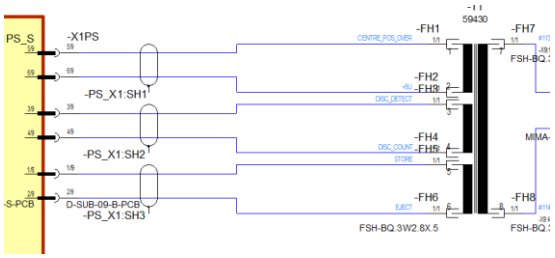




Modifying the Cable Structure

The six graphical connection lines have already been assigned conductors. Together they form the cable **–PS_X1** that is displayed the Project Window *Device* tab. We must now add shields to the conductors as shown in the example. To do so, enclose the single conductors in pairs by shield symbols.
In ***E³.*cable**, all conductors located under a shield symbol are considered to be shielded.

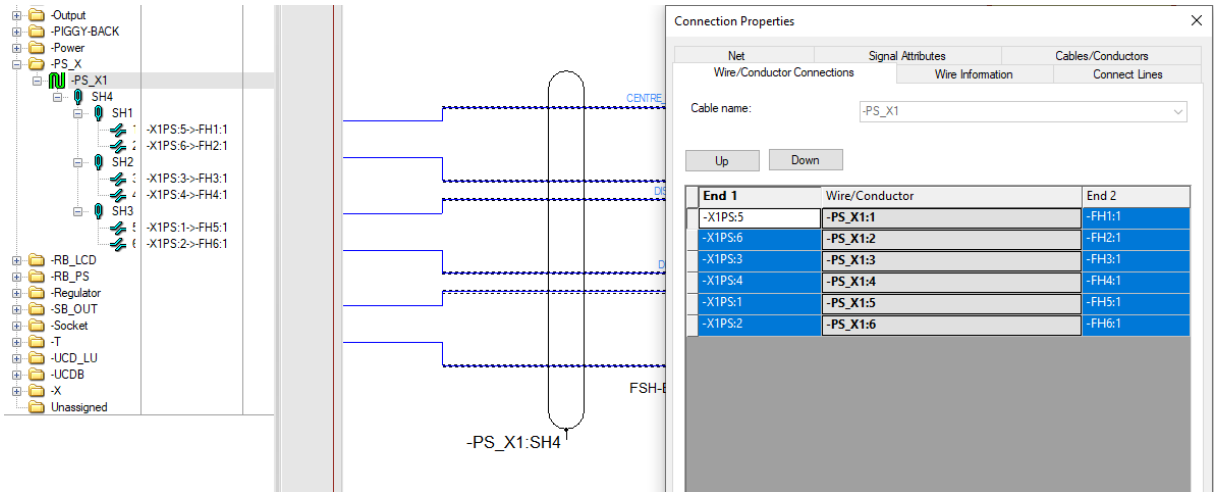
Drag the shield symbol **SHIELD3*2T** from the **Electric - Shield** folder in the *Misc* tab of the Database Window and place it so that it overlaps the top two connect lines. Repeat the same procedure with the other two pairs of connections.



Next, we have to place the shield symbol **SHIELD5*3T** over all six conductors. This symbol must be adjusted dynamically. To do so, select the shield symbol and adjust its height correspondingly. Drag the upper middle selection mark so that the shield overlaps all six connect lines.

The shields have been dynamically added to the cable **–PS_X1** in the Project Window *Device* tab and if any additional changes are made, they will be updated immediately.

Select all six conductors belonging to cable **–PS_X1** in the drawing sheet, right-click and select the **Connection Properties** command. All information, regarding the structure, signals, pins, line characteristics and attributes, will be displayed and can be modified as required.



Modifying Existing Drawings

It must be possible to modify an existing drawing quickly, easily and reliably (checked).

With ***E³.***, changes are displayed instantly in all representations of an object. Checks can also be executed for views that are appropriate to the workflow.
Connected components can be moved quickly and intelligently, also to other sheets.

When deleting a symbol from an existing connection, the connection closes automatically, provided that the attributes assigned to the connection, permit it. Otherwise, the connections remain 'open'.

Adding Text

In order to complete the sheet, we must add free texts, which do not belong to any component or symbol.

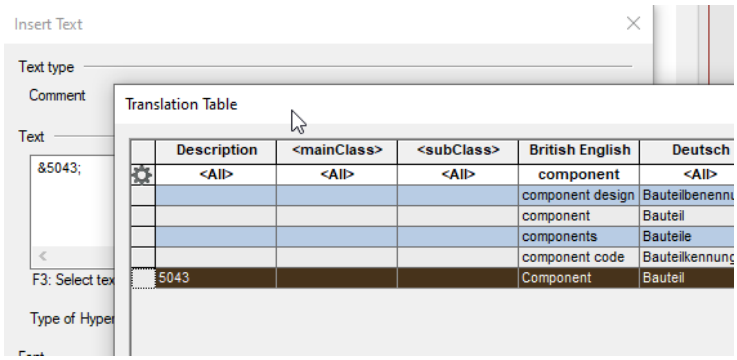
This can be accomplished by using the command **Insert → Text** or the T icon in the text toolbar. The **Insert Text** menu appears.

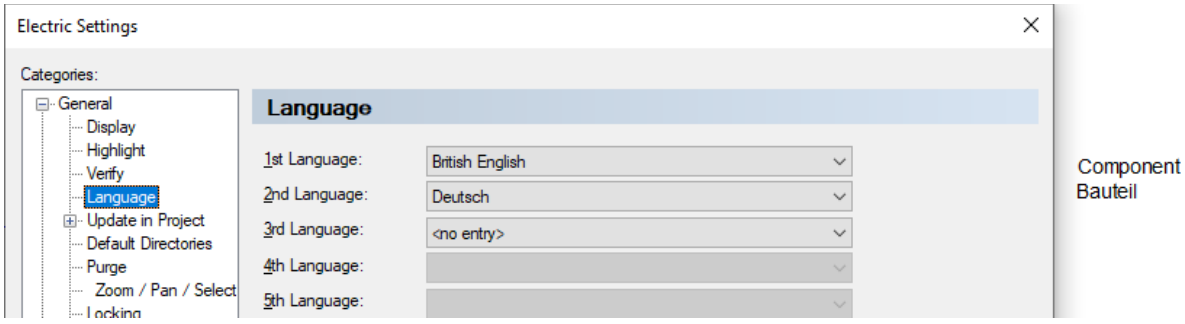
The **Text** menu allows you to type in the desired text (including multi-line text) and define the text's parameters (e.g. font, text size, etc.). After clicking **OK**, the text will appear at the cursor position and can be placed anywhere on the sheet with a simple click of the mouse.

Free texts can also be attached to an element. Select the desired element as well as the free text and "group" them together using the *Group Graphic* icon found in the graphic toolbar.

Text can also be accessed through the *Text Database* by pressing the **F3** key in the input menu.

Texts placed through the Translation Table will be translated online when changing the language and multiply displayed, if required. The project language can be defined in **Tools → Settings → General → Language**.





Searching and Navigating within the Project

Trying to find a specific element in a large drawing can be very time-consuming. ***E³.*cable**, however, makes this task quite simple. You define the information that will be used to search for the desired element in the **Search** menu. At the push of a button, the corresponding sheet will be opened and the element highlighted in the drawing.

The **Search** menu can be called up with the **Edit → Search** command, the icon or the function key **F3**.

Enter the object type and value in the menu that you want searched.

The search results will be displayed in the Output Window. By double-clicking on the desired line, you can jump directly to the sheet, on which the desired object is highlighted. This allows you to find information quickly, as well as keeping an overview of your drawing.

To delete the highlighting found on this object, use the **Edit → Reset Highlight** command or press the keys **SHIFT + F3**.

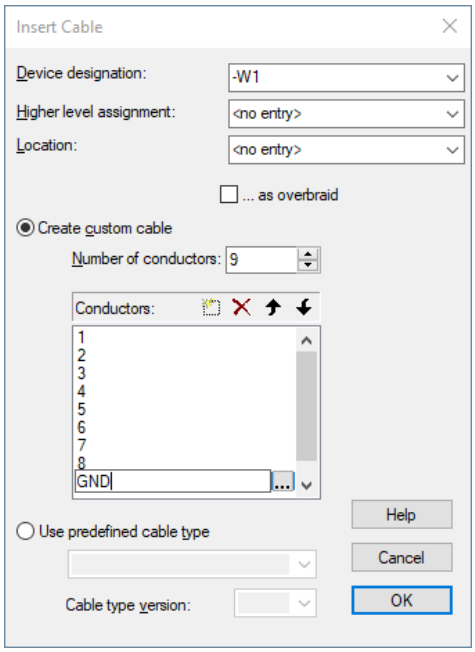
Defining Cables

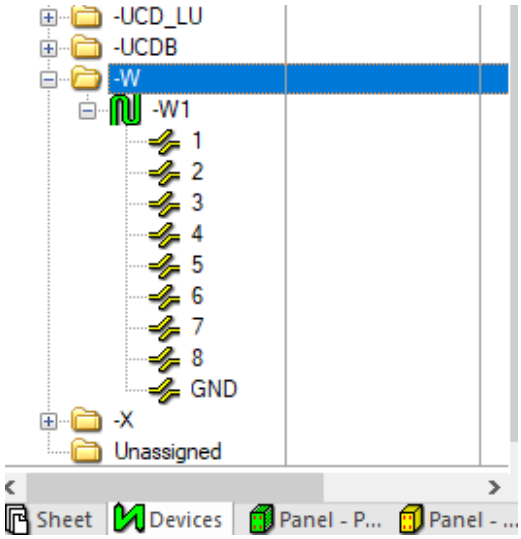
Connections are initially drawn as graphical connection lines but they can be completed with cable and/or conductor information. ***E³.*cable** makes it easy to assign individual conductors or any number of conductors to their corresponding graphical connection lines. No distinction is made between user-defined cables and predefined cables selected from the database.

Cables selected from the database, referred to as cable types, cannot be easily changed in the project. They retain their conductor arrangement, defined shielding and attributes, as long as the cable type is not modified.

We must now insert a user-defined cable. Select the **Insert → Cable** command, check the **Create custom cable** option and add **nine** conductors.
The individual conductors are numbered consecutively and displayed in the menu window. They can be renamed using **F2** or rearranged here as required. Use the arrow buttons to arrange the conductors in the desired sequence. Rename conductor **9** to **GND**.

Click **OK** and the newly created cable (**-W1**) is added to the **Device** Tree View in the Project Window. The conductors are displayed in yellow, which indicates they can still be used in the drawing.

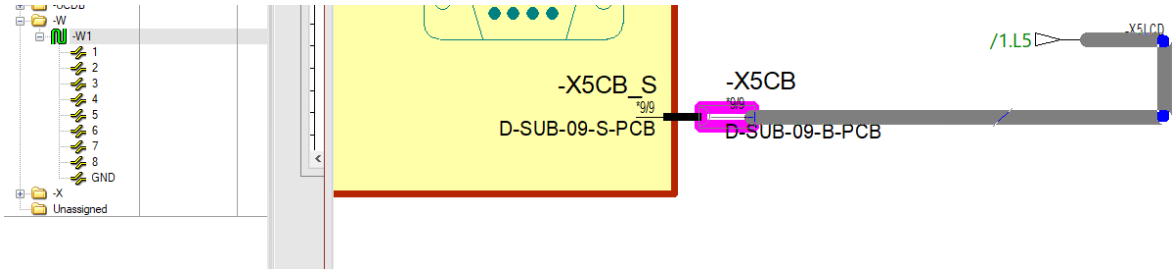




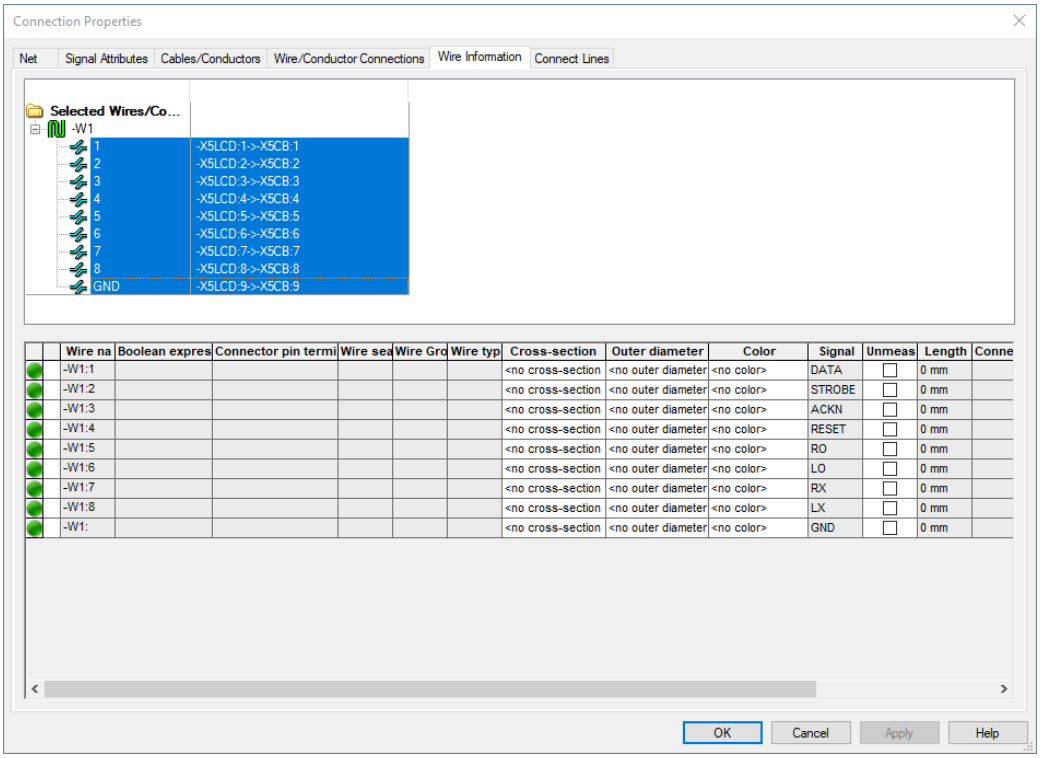
Assigning Conductors

The newly created cable (**-W1**) is to be assigned to an existing connection line and connected to the socket **–X5CB**. Locate the socket in the plan using the **Search** functionality. Zoom in on the area so that the socket and the connect line are clearly visible.

We must now assign the predefined cable **–W1** to the socket **–X5CB** on the connection line. Since a 1:1 connection is desired here, drag the cable **–W1** from the Project Window **Device** tab (the green symbol) onto the connection line. The connection line is highlighted as soon as it is recognized as such by ***E³.*cable**. After releasing the mouse button, the cable's nine conductors are assigned to the connect line.



Verify using the cable's **Connection Properties** that the conductors have been assigned 1:1 to each of the socket pins. Since the connector pins already have defined signals, these signals merely continue through the conductors.



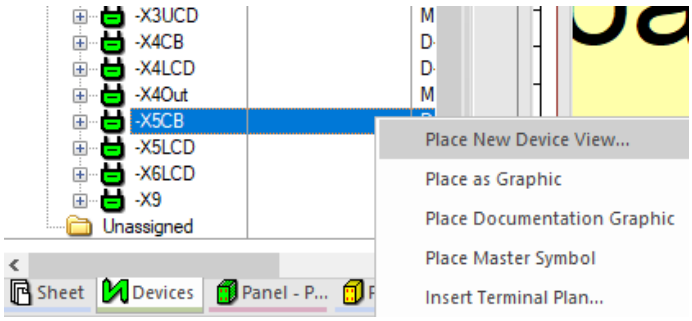
Placing Different Connector Views

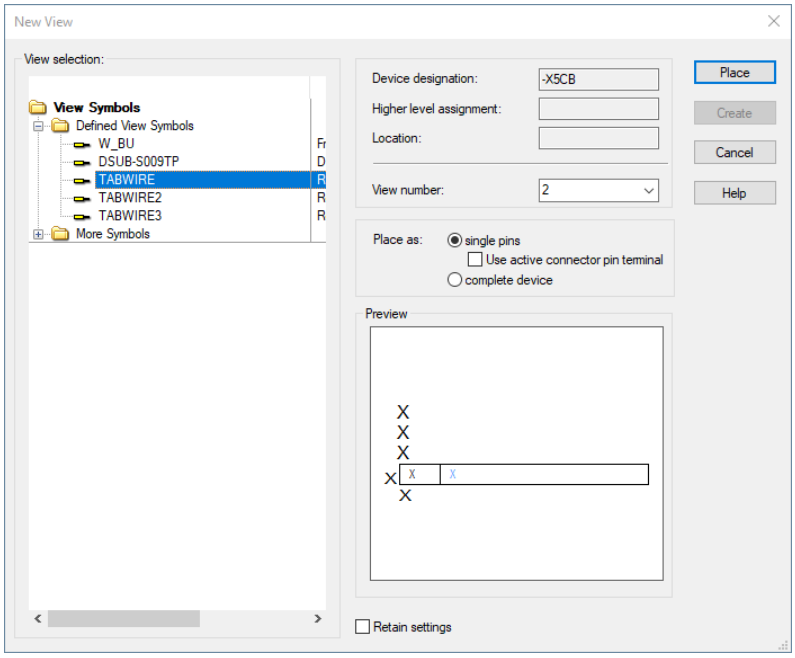
In order to meet the needs of the different documentation types, ***E³.*cable** supports the use of different connector views. Depending on the application, they can be depicted as overhead or side views, as single-pin or as complete connector representations.

A view is nothing more than a new graphical representation of a connector. It has the same characteristics as the original view. If information in one of the views is changed, it will also be changed in all other views of the same connector. This means that no matter where the change is made, all representations will be updated instantly. A connector view is another view of the same element. The different connector views are defined in the database.

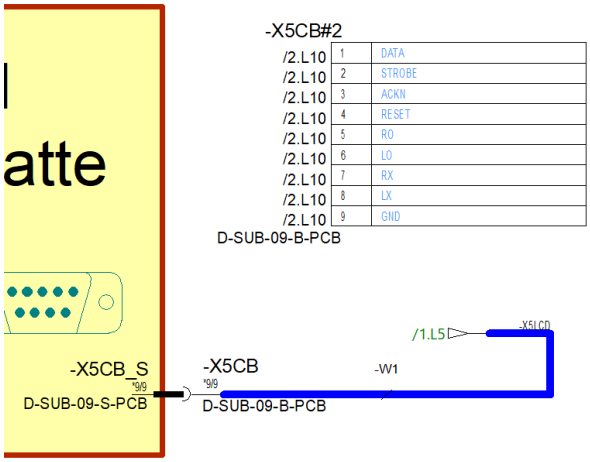
We must now create an additional view for the socket **–X5CB**. This special connector view is a dynamic signal table. To create the signal table, right-click on the socket **–X5CB** in the Project Window **Device** Tree and select the command **Place New Device View …** in the context menu.

Select **Place as: single pins** and the **TABWIRE** symbol name, which represents one line of the signal table. Click on **Place** and position the table next to the socket in the drawing on sheet **2**.



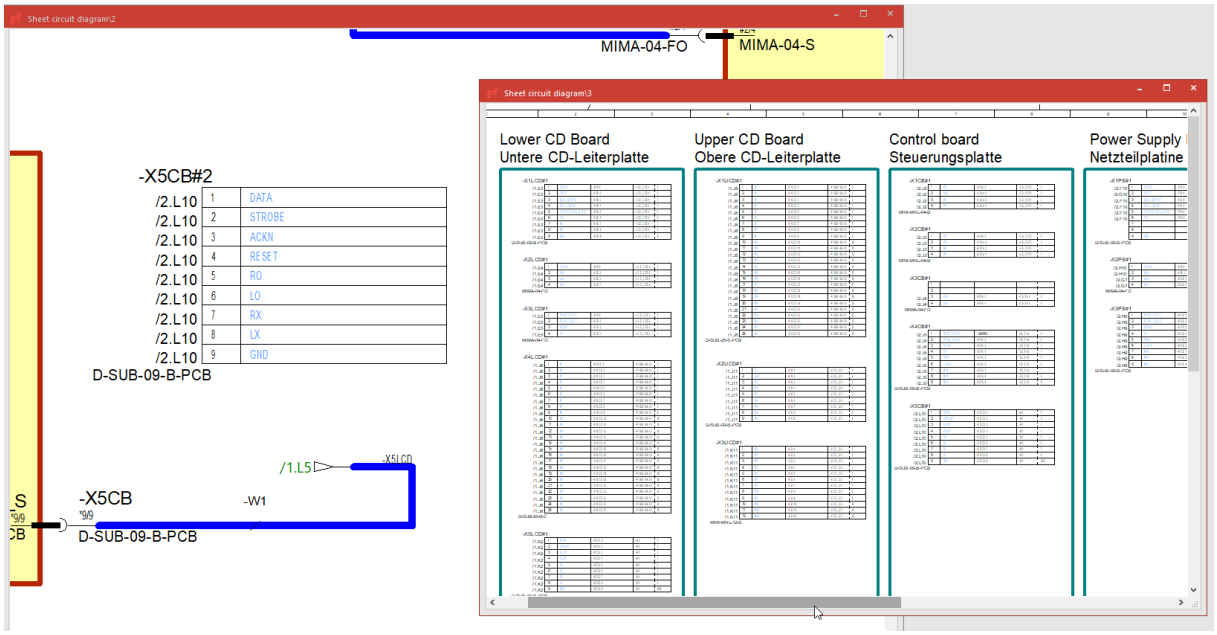


The signal table now lists the signals, which are defined for the individual connector pins of **–X5CB** and the cross-references to their original pins.



The signal definition can now be changed in the signal table as well as on the connector pins themselves. Both views of the socket are dynamically updated. This also applies when the views are placed on different sheets.
Any modifications, regardless of where they are carried out, will be updated immediately in the project.

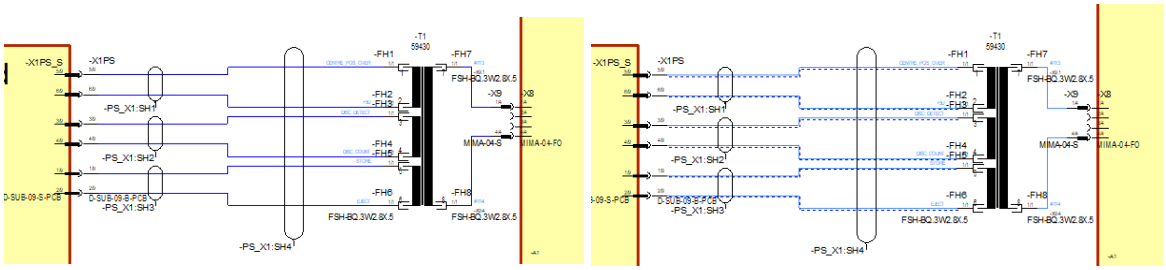
Open sheet **3** and review the signal tables for the connectors in the project.



Switching between Display Modes

In addition to the pure, graphical information found in drawings generated with ***E³***, logic background information is also contained in the plan (often displayed in the form of attributes). ***E³.*****cable** makes it possible to display this logic information.

To switch between display modes, use the command **View → Connection Types** or the ¶ icon.



In the “normal” view, all connections are displayed as defined by their graphic attributes (color, line width, line type). Connection attributes are not shown graphically in the “normal” view. This is the mode normally used to display the drawing’s output.

Once the Connection Types mode is activated, all connections will be displayed with different colors and line types, depending on their attributes (e.g., does the connection have a signal, and does it contain a wire). In addition to the signal, the signal cross-references will be numbered. This gives you a quick, graphical overview of the most important connection information available without having to continuously check the **Connection Properties** menu.

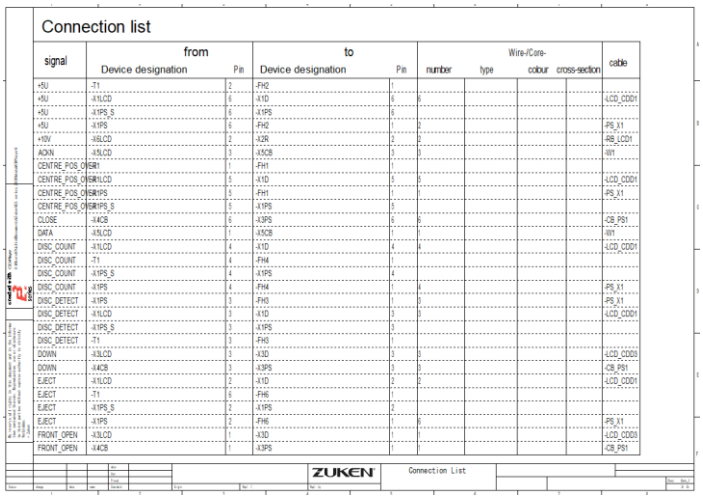
Lists and Output Files

Information contained in the project can be output to any document, list and/or file.

That is how ***E³*** can be integrated in various ways in the workflow.***E³*** offers for example predefined documents such as table of contents, bill of material, connection list, cable list and terminal plan.
It is also possible to work with the integrated Visual Basic programming interface (API). This API is based on Microsoft’s COM technology (COM interface) and offers the possibility to load or add data to the drawing using any programming language.
Several lists are predefined in ***E³*** and can be output directly in Microsoft Excel, Access, Word or ASCII Format, provided that they are installed on your PC. For example, you can create the cable list as an Excel file integrated in the drawing.

To do so, select the **Tools → Reports → Excel → Cables** command. Excel starts automatically and generates a cable list, which contains all of the cables as well as the devices they are connected to in the drawing.

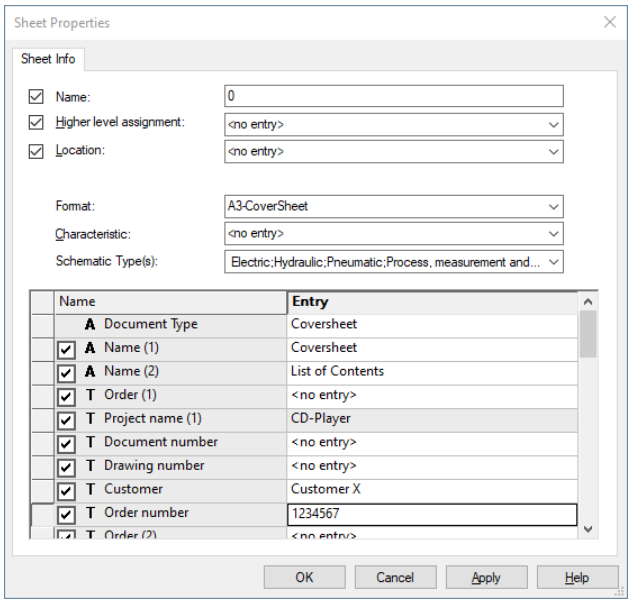
These reports can also be inserted at any time as separate sheets into the drawing. To generate a connection list, select the **Tools → Reports → Sheet → Connections** command. One or several sheets are created containing all connection details. The sheets are added to the Project Window’s **Sheet** tab.



Cover Sheet with Table of Contents

Start the **Add-ons → E³.series Documents → Create Cover sheet** command. *E³* automatically creates a cover sheet with a table of contents.

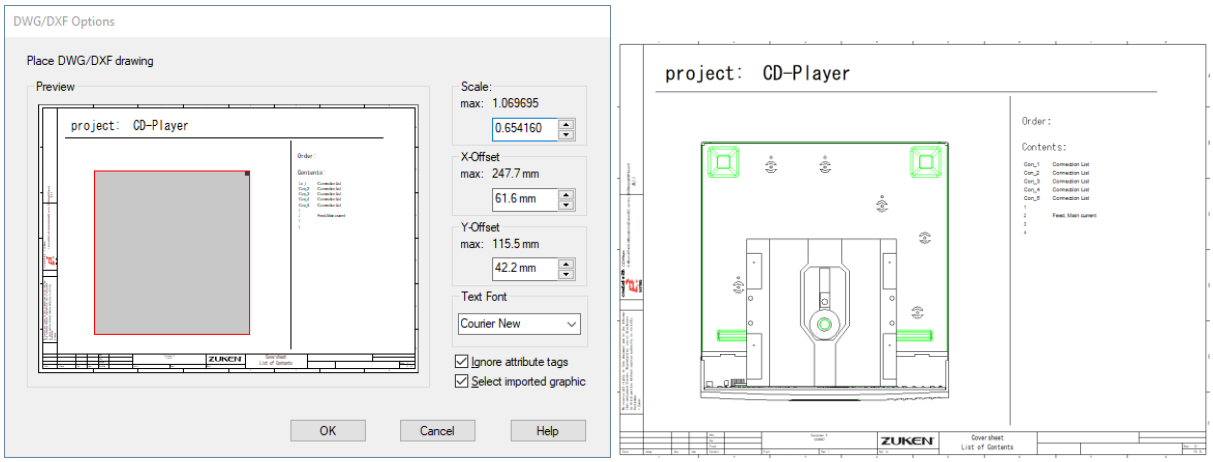
Fill the sheet texts as desired. In the example the sheet text **Customer** is defined as **Customer X** and **Order number** is **1234567**. To do so, click anywhere on the sheet and select the **Sheet Properties** command in the context menu. Enter the corresponding text in the appropriate fields. Press **OK** and the text is displayed on the sheet.



Next, add a mechanical drawing to the cover sheet. This can be done very quickly with the **DWG/DXF Import** command.

Start the **File → Import → DXF/DWG** command and select the **cd-top.dwg** file from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\CD-Player*. A menu appears, in which you can specify the mechanical drawing's size and position.

Change the gray rectangle's size by moving the "handle" in the upper right-hand corner back and forth, and then position it on the sheet accordingly. If an exact scale and placement position are desired, these values can be entered directly in the menu. Click **OK**.



We also want to add an image to our cover sheet, e.g. a company logo. Start the **File → Import → Image** command and select the file **E3-Logo.tif** from the directory …\data\.
Place the ***E³*.series** logo on the sheet and then adjust its size.

Embedding Additional Documents

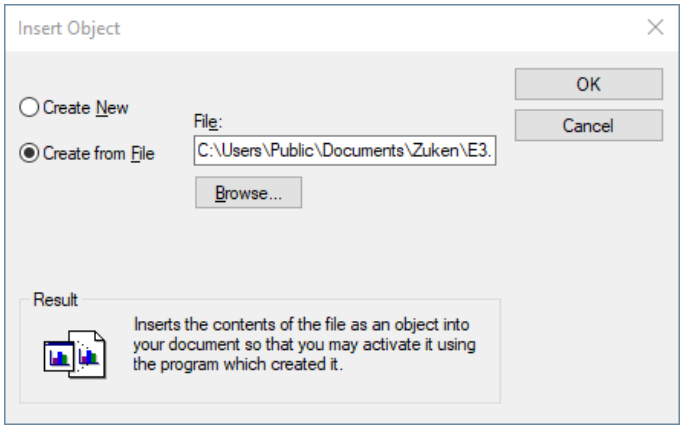
Relevant information can be added to a project in ***E³*.cable** if the user decides to do so. This may include documents from other applications, such as instruction manuals, design description, evaluation tables and so forth. ***E³*.cable** allows to integrate these documents into the project. A prerequisite is that the applications used to create these documents are compatible with Active-X (e.g., Microsoft Word, Excel).

The following example will only work if you have Microsoft Word installed on your computer.

Start the **Insert → Object…** command.

Note: Files embedded in the project with **Insert -> Object…** can be placed on a sheet and -depending on the file- edited in the project. Files embedded through **Insert -> File…** are integrated in the project structure. When the **Create as reference** option is activated, the file is linked in the ***E³*** project. When the option is inactive, a separate copy of the file is embedded in the structure.

The **Insert Object** menu appears displaying all programs installed on your computer that are compatible with Active-X.
The menu is always displayed in the language defined for the operating system.



To open an existing document, select the **Create from File** option and click on the **Browse…** button. Select the file *C:\Users\Public\Documents\Zuken\E3.series_2020\data\CD-Player***E3.cable-e.doc***.*

Microsoft Word with the document selected is now started within ***E³.*cable**. The ***E³*** user interface changes to the Word interface so that you can edit the document. As soon as a sheet is activated from ***E³.*cable**, you automatically return to the ***E³.*cable** user interface.

The Word document, which is embedded in your project, will be saved as copy together with the project. The file selected will remain in its original form and will not be changed by any actions within ***E³.*cable**.

Printing Sheets

The sheets that have been created or the entire project can be printed.

To do this, select the **File → Print…** command or the icon in the toolbar.
The **Print** menu opens. Specify the printer to be used to output the information, as well as the print properties such as page format. Define whether to print all of the pages or only specific pages. For example, enter **1;2** if sheets **1** and **2** are to be printed.

This was a short overview of the functionality of ***E³.*cable**. If you continue working with ***E³.*cable** you will get to know other functions, which will support your work.

Enjoy working with

series

First Steps

Getting Acquainted with *E³.*cable for Cabling Vehicles

Welcome to Your First Tour of E³.cable

By following the steps described below, you will be able to familiarize yourself with the main functions of ***E³.*cable** for cabling vehicles. This tour points out how to create reliable circuit diagrams and cable drawings. Please read through the respective forewords of the individual chapters, which will then allow you to perform the described actions. Information, pertaining to basic operations and the user interface, can be found in the chapter *The E³.series User Interface*. At the end of the tour, you will have learned about the various functions of ***E³.*cable**, which can then be applied to your design needs.

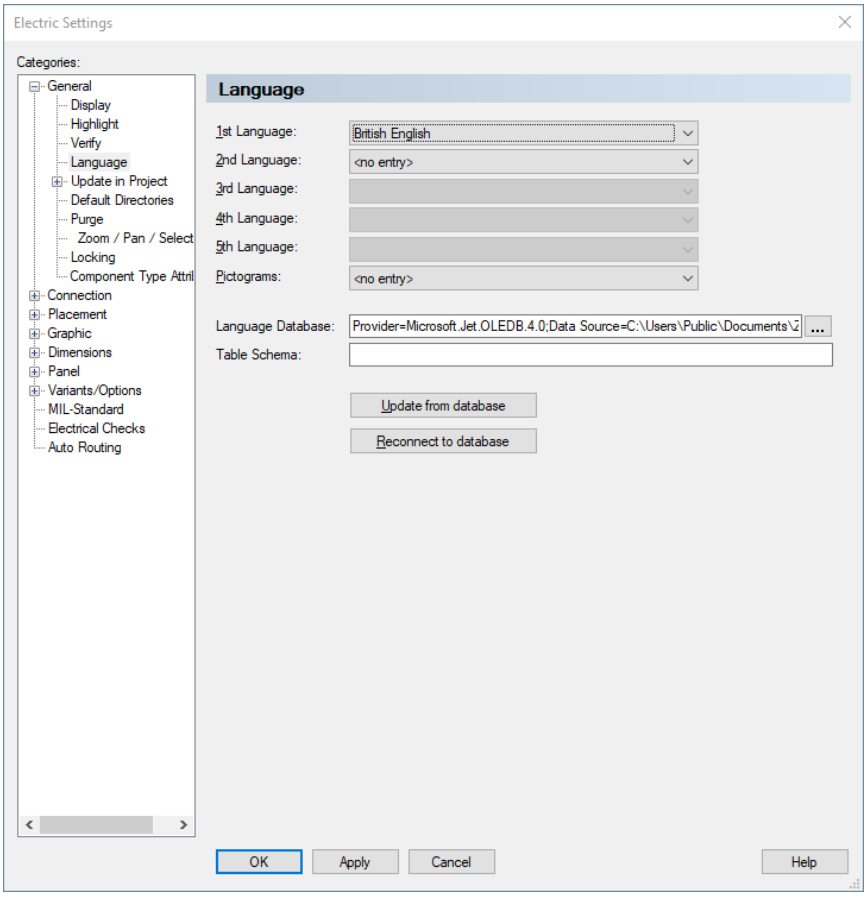
Have fun with this introduction!

Opening an Existing Project

With ***E³.*cable** you always work within a project. A project may be comprised by any number of drawing sheets and additional documents, which are stored in a single file. Start the program.

To open a project, use the command **File → Open** or the corresponding icon in the standard toolbar. Select the project **MX-30.e3s** from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\MX 30*.

First, the project language needs to be defined. Select the **Tools → Settings** command from the main menu bar to open the **Settings → General → Language** dialog. Define “British English” as the first language.



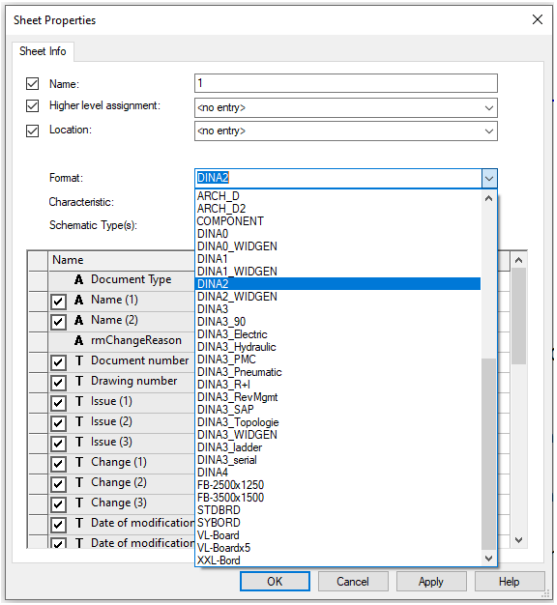
Opening Sheet 1 of the Project

Open the **Circuit diagram** folder in the **Sheet** Project Window. Select sheet **1** by double-clicking on the sheet itself or by simply clicking on the box in front of the sheet.

The different sheet formats (e.g. DIN, extra-wide, Ladder Diagram for the American market, etc.) with their unique size, layout and sheet header information are stored in the database.

The sheet format used for sheet **1** is **DINA2**. The sheet format can be changed at any time through the **Sheet Properties** menu. Right-click anywhere on the sheet to reveal the **Sheet Properties** menu and by clicking on the **Format** field, a drop-down list of all the sheet formats defined in the database will appear.

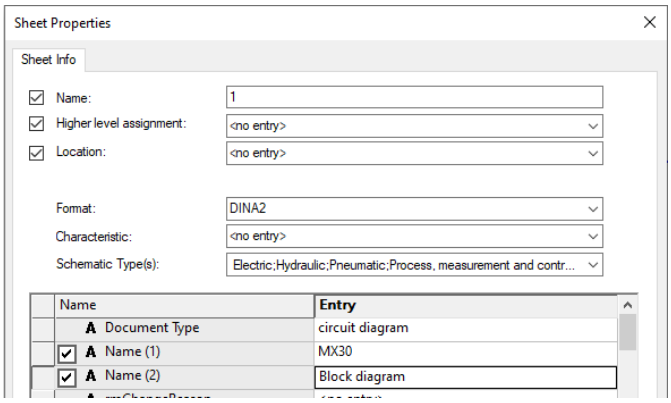
If a sheet format selected does not “fit” to the sheet and its information, an error message will be displayed. If this is the case, make sure you select an appropriate sheet format.



Entering Sheet Header Information

Sheet headers contain information typically found in the drawing sheet’s margins, e.g. Drawing Title, Drawn by, Date, Revision, and so forth. The sheet headers are displayed in the **Sheet Properties…** menu, which can be called up through the context menu by right-clicking on any empty area of the sheet.

Enter the text **Block diagram** for the **Name (2)** attribute.

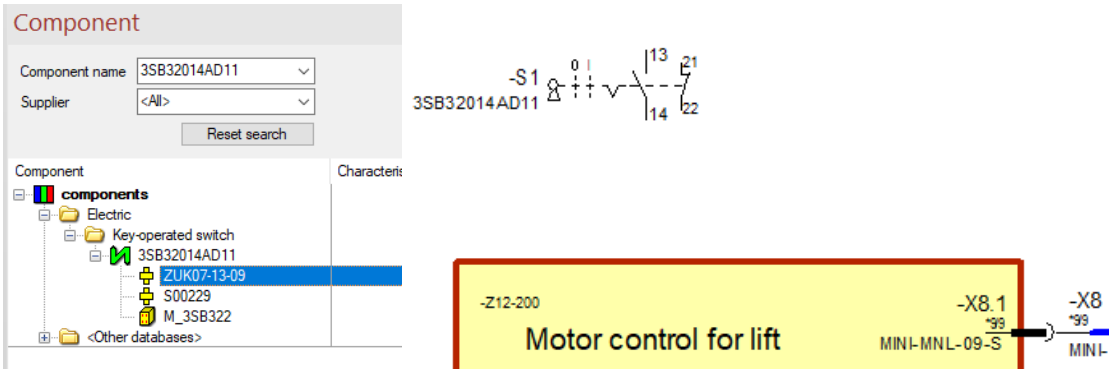


Drawing a Subcircuit

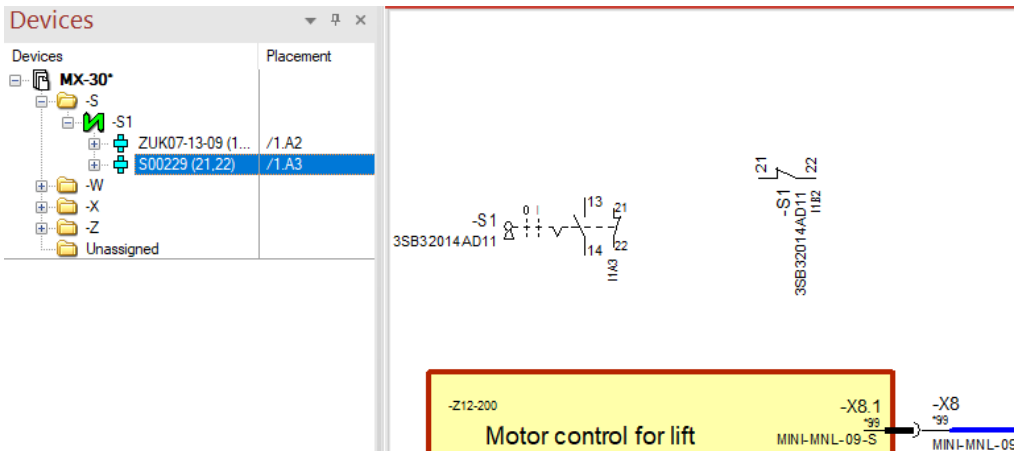
The existing block diagram must now be enhanced by adding an unload circuit. This unload circuit will then be connected to the other components using the cable set **–W14**.

Switch to the Database Window's **Component** tab.

Open the folder **Electric - Key-operated switch** and click on the ⊞ box of the key operated switch **3SB32014AD11** to display all of the symbols that are listed under it. Drag the first symbol onto the sheet and position it above the block titled **Motor control for lift**.



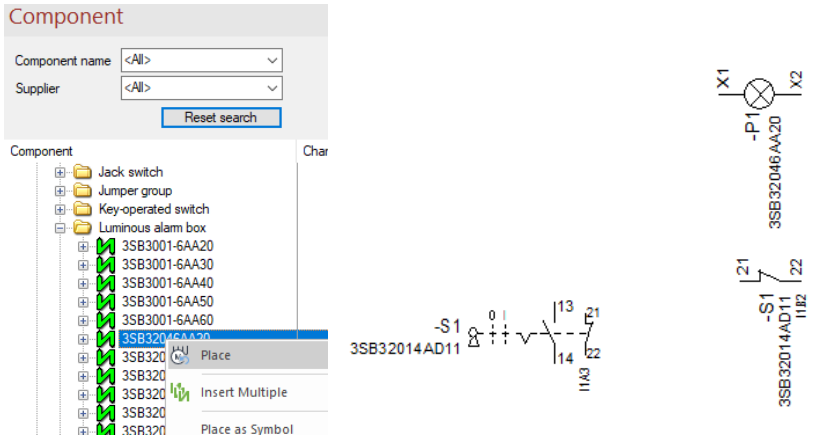
After that, drag the second, available symbol of the device **-S1** from the Project Window's **Device** tab and place it as shown in the figure using the **Place** command from the context menu. Before placing, right-click and select the command **Rotate** from the displayed context menu or use the key **R**.



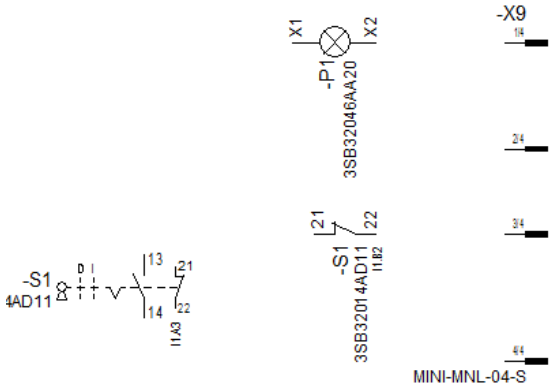
The key-operated switch's two symbols cross-reference each other and the cross-references are instantly updated in case of any modifications. If a device's elements are spaced relatively far apart or they are placed on different sheets, you can "jump" directly to a cross-referenced element by right-clicking on the referenced text and selecting the **Jump** command from the displayed context menu.

Next, place the indicator light **3SB32046AA20** above the key-operated switch's NC (normally closed) contact. It can be found in the **Electric - Luminous alarm box** folder in the **Component** Database Window.

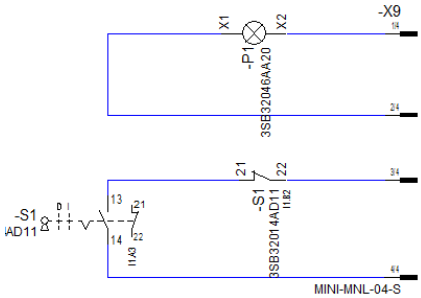
Right-click on the indicator light and select the **Place** command in the context menu. Drag the indicator light onto the sheet and press the right mouse button once again. Select the **Rotate** command in the context menu and then place it accordingly. Compare your results with the screen-shot.



We must now connect the subcircuit to the existing cable set by means of a connector.
Open the folder **Electric - Mini Universal MATE-N-LOK** in the **Component** Database Window and select the connector **MINI-MNL-04-S**. Place the pins as single pins. To do so, select the connector's 4 pins in the Database View, right-click and select the **Place** command in the context menu.
Drag the connector pins onto the sheet and, prior to placing them in the drawing, mirror them using the **Mirror by Y-axis** command. The individual pins can now be moved using the context menu's **Move (Rotate)** command or Drag & Drop as shown in the screen shot.



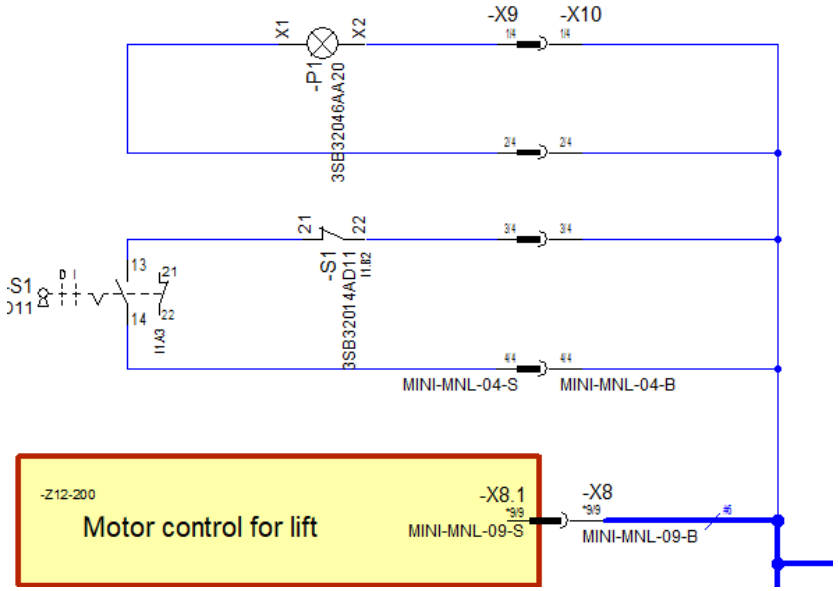
To connect the subcircuit, right-click on one of the pins and select the **Connection** command or simply press the **C** key. When creating connections, ***E³.*cable** defines the corners automatically and connects the desired pins.
Compare your results with those here:



Now connect the connector pins with the existing cable set **-W14**. Right-click on the uppermost pin and select the **Connection** command from the context menu, use key **C** or the icon.
Scroll to the cable set and search the connection point. When the connection can be created, the cursor's shape changes. Complete the connection by left-clicking on the pin.

Since all potential mating connectors and connector pin terminals are already defined in the database, ***E³.*cable** always selects the appropriate mating connector for each connector.

The three remaining connector pins can be connected to the cable set automatically. To do so, use the **Insert --> Autoconnect --> Inside Area Horizontal** command. After selecting the command, drag a rectangle around the three connector pins and the vertical connection line to **-W14** that was just created. The connections will be inserted automatically after releasing the left mouse button. This command also works in examples containing more device elements than in this example.

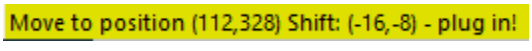


If a symbol is deleted from a connection, the connection will close automatically, provided that the attributes, which are assigned to both connections, permit it. Otherwise, the connections remain "open". Likewise, if you want to add a symbol to an existing connection, ***E³.*cable** will undo the connect line and insert the symbol into the connection.

***E³.*cable** also dynamically supports the movement of elements and maintains the connections while moving.

Plugging (Unplugging) Plug Connectors

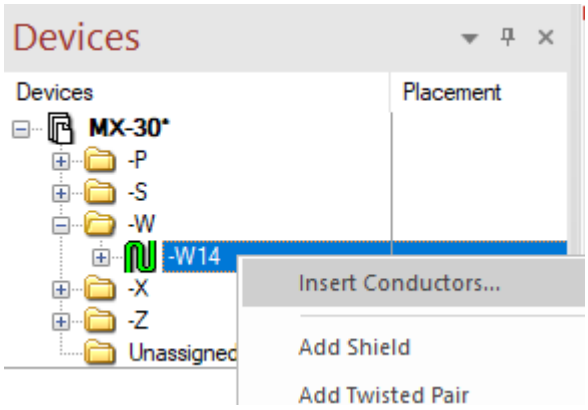
***E³.*cable** also supports the plugging and unplugging of connector elements.
To unplug a connector element, simply select it and drag it to a new position. After releasing the mouse button, the connection is broken. A connector element can also be plugged in by moving it in the direction of its mating connector.
When the display window at the bottom of the screen turns yellow as well as a plug and socket symbol appear next to the cursor, this indicates that a connection was made.

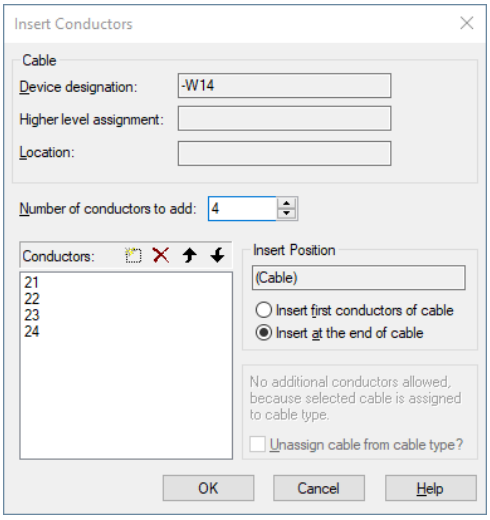


Modifying the Cable's Structure

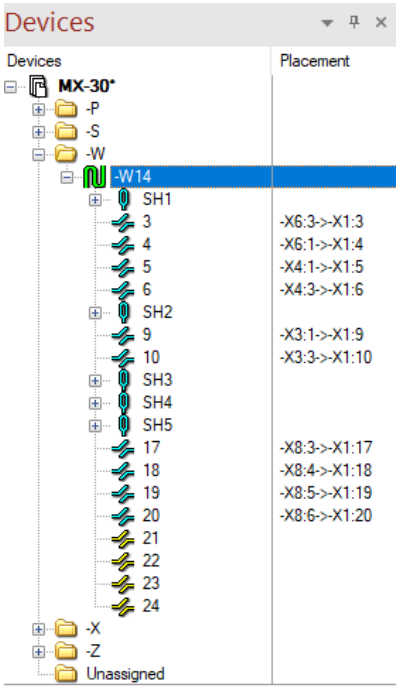
The example project already contains the cable set -W14 that needs to be enhanced by additional conductors. Right-click on the cable set **–W14** in the Device View of the Project Window and select the **Insert Conductors** command from the context menu.

Add four additional conductors to connect the unload circuit correctly. These conductors are automatically numbered **21–24**.



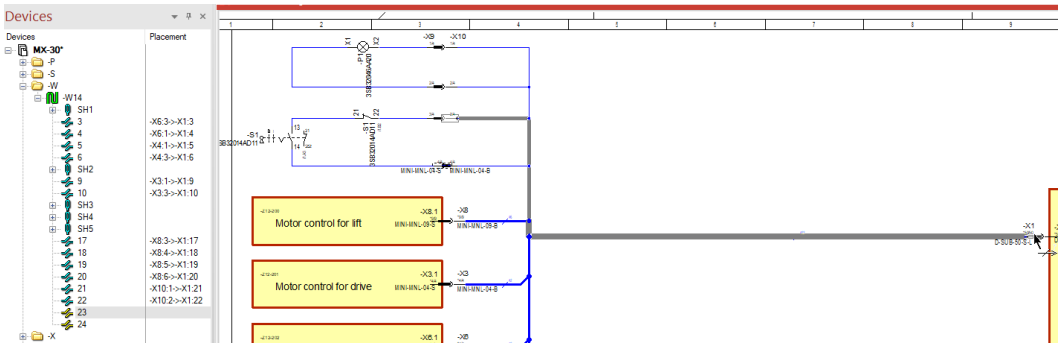


Confirm with **OK**.
These conductors are added to the cable set and can now be assigned. Conductors, which have not been placed yet, are displayed in yellow in the Device View.



Assigning Conductors

These new conductors can now be assigned to the connections between the unload circuit's individual pins and the main connector **–X1** on the block labeled **Main distribution**.
Drag the individual conductors from the Device View onto an individual pin of the respective unload circuit's socket connector (**-X10**). Release the left mouse button and position the cursor on the main connector **–X1**, which results in the pathway being highlighted as soon as ***E³.*cable** identifies the connector. By clicking on the pin **-X1**, the conductor will be assigned to this connection. Repeat this process for the other three pins.

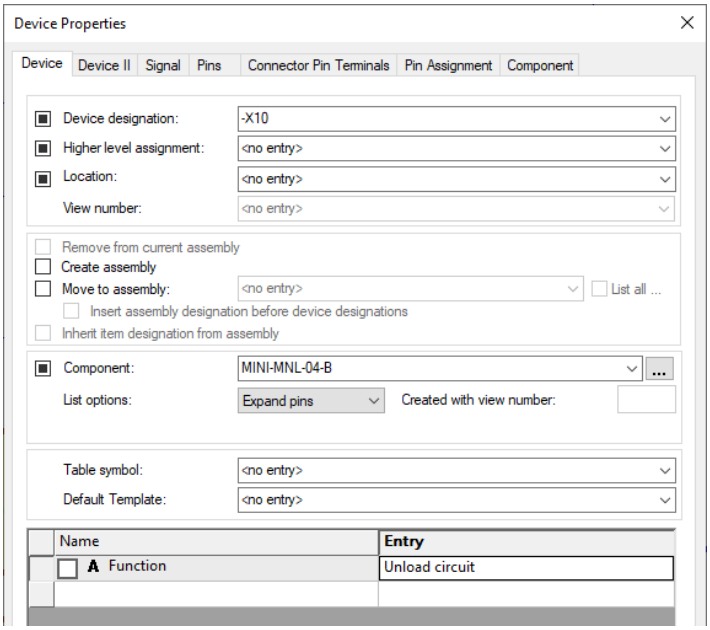


Now check to see whether you assigned the conductors correctly. Right-click on the connection line and select the **Connection Properties** command in the context menu. You can see the cable's structure in the **Cables/Conductors** tab and which pins the conductors are connected to in the tab **Wire/Conductor Connections**.

Defining Attributes

The socket's affiliation to the unload circuit must now be defined. Assign the socket **–X10** the attribute **Function**. Right-click on the socket **–X10** and select the **Device Properties** command from the context menu displayed.

You can select the desired attribute from the left-hand side of the attribute table and enter its value in the right-hand side. The check box, which is located in front of the attribute name, controls whether the attribute is displayed or not. Enter the value **Unload circuit** for the socket **–X10**.



Adding Text

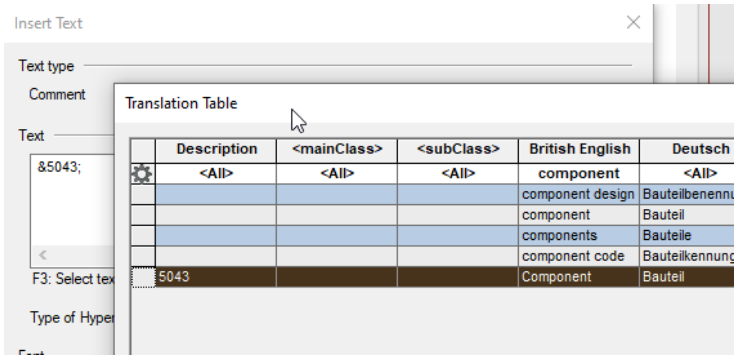
In order to complete the sheet, we must add free text, which does not belong to any component or symbol.

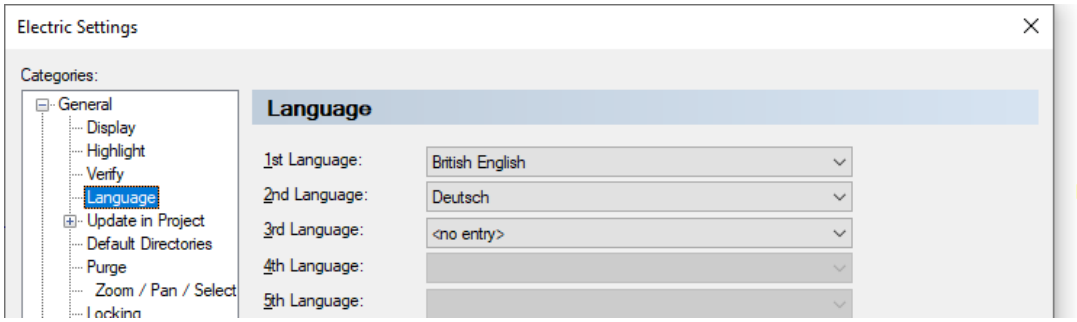
This can be accomplished by using the **Insert → Text** command or the T icon in the text toolbar.

The **Text** menu allows you to type in the desired text (including multi-line text) and change the text's parameters (e.g. font, text size, etc.). After clicking **OK**, the text will appear at the cursor position and can be placed anywhere on the sheet with a simple click of the mouse.

Free texts can also be attached to an element. Select an object as well as the free text and "group" them together using the *Group Graphic* icon found in the graphic toolbar.

Texts placed through the Translation Table will be translated online when changing the language and multiply displayed, if required. The project language can be defined in **Tools → Settings → General → Language**.



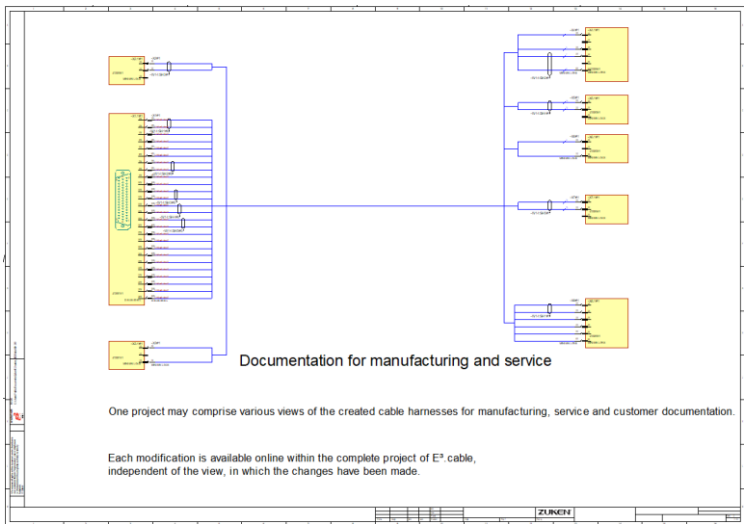


Component
Bauteil

Placing Different Connector Views

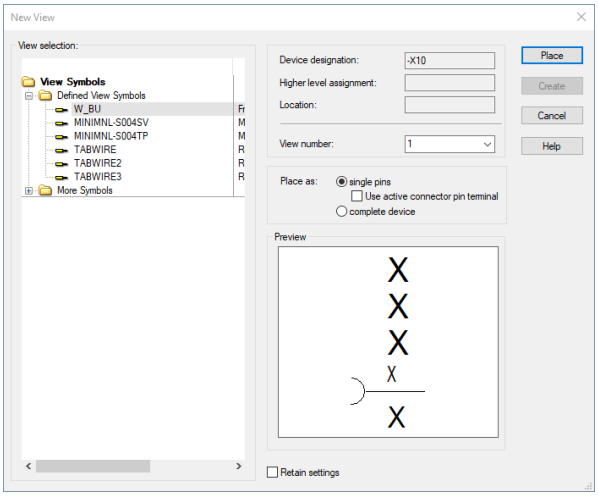
To meet the requirements of the different documentation types for electrical engineering, ***E³*.cable** supports the use of different views that can be selected for connectors. Different representations of the same cable set may often be required for assembly, service, manufacturing and customer documentation.

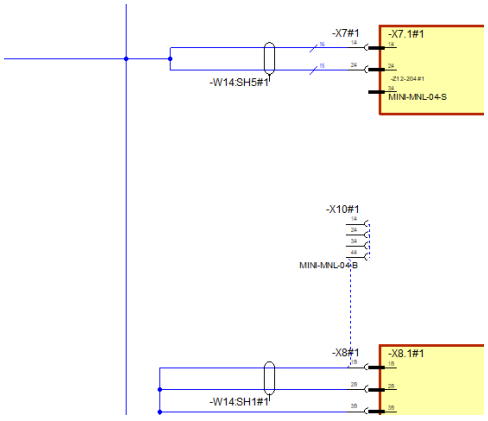
Open sheet **2** in our sample project. The cable set **–W14** for assembly and service is pictured at right. The elements displayed are nothing more than different views of the same connector elements shown in the block diagram on sheet **1**.



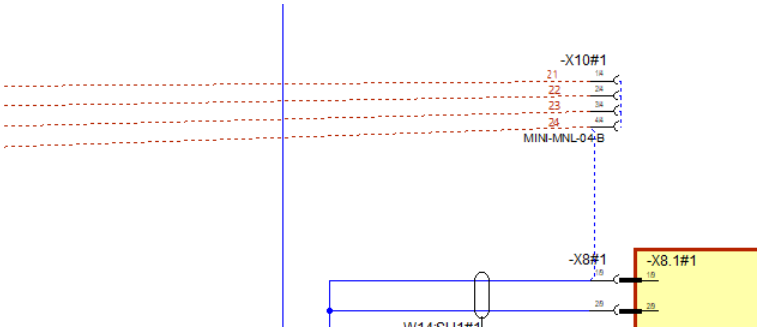
The newly created unload circuit is still missing a socket. To add this view, right-click on **–X10** in the Project Window **Device** View and select the command **Place New Device View…** in the context menu.

Select **single pins**, and the symbol **W_BU** from the **Defined View Symbols** list. Click **Place**. Next, mirror the pins on their **Y**-axis and place them above the connector view **–X8#1** on the sheet. Position the cursor on the sheet. By right-clicking, a context menu will appear which contains the **Mirror by Y-axis** command.



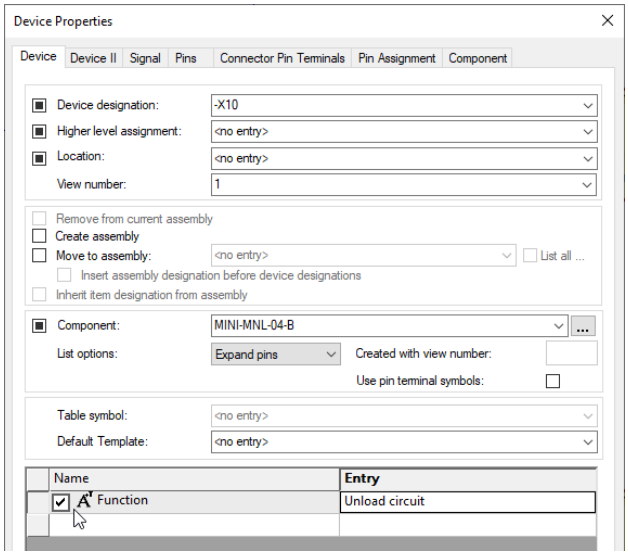


***E³.*cable** is now able to display the logical conductor connections to connector view **-X1#1**. If this is not the case, they can be displayed by clicking on the **Display unconnected conductors** checkbox using the **Tools → Settings → Connection → Cable/Conductor Logic Lines** command or use the icon in the toolbar.



Now connect the connector pins to the cable set. To automatically insert the connect lines and their conductor information, use the **Connect** or **Insert → Autoconnect → Inside Area Horizontal** command. After selecting this command, drag a rectangle around the cable set's connect line and the newly placed connector pins. Upon releasing the mouse button, the connections will be inserted.

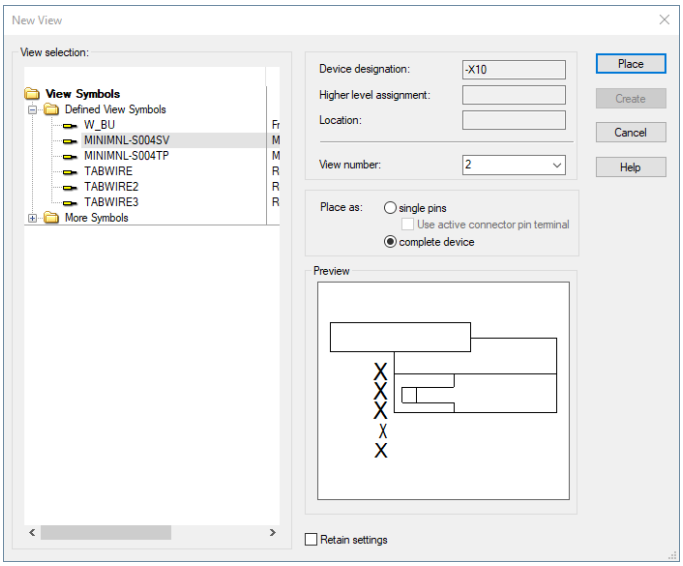
Next, display the attribute *Unload circuit* on the connector. To do so, use the **Device Properties…** command from the context menu of connector **–X10**, check the box in front of the **Function** attribute and move the text to the desired position.



Your results should look something like this:

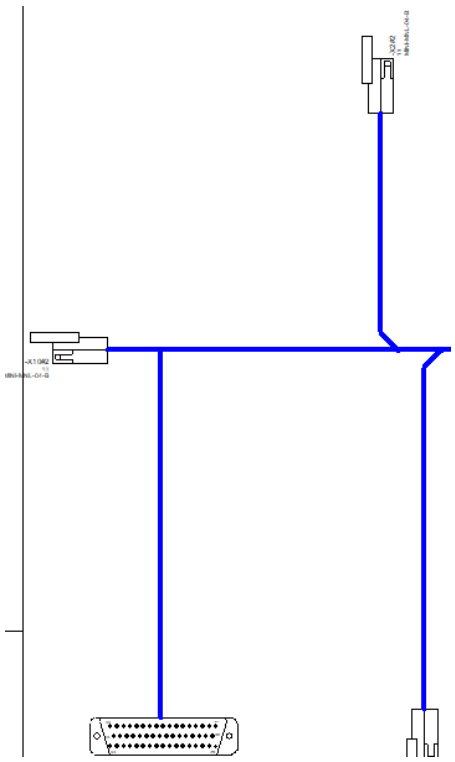
Another view of the same cable set **–W14** is displayed on sheet **3**. Once again, the unload circuit's **–X10** socket is missing from the drawing. We are also going to add another view of the socket on sheet **3**.

Right-click on the connector **-X10** in the **Device** Tree View and select the **Place New Device View** command. Select **View number 2** and the symbol **MINIMNL-S004SV**. Since we want to place the view as a **complete connector**, activate this button also.



Place the connector view on the left side of the sheet as shown in the example below. Connect the connector view with rest of the cable assembly.

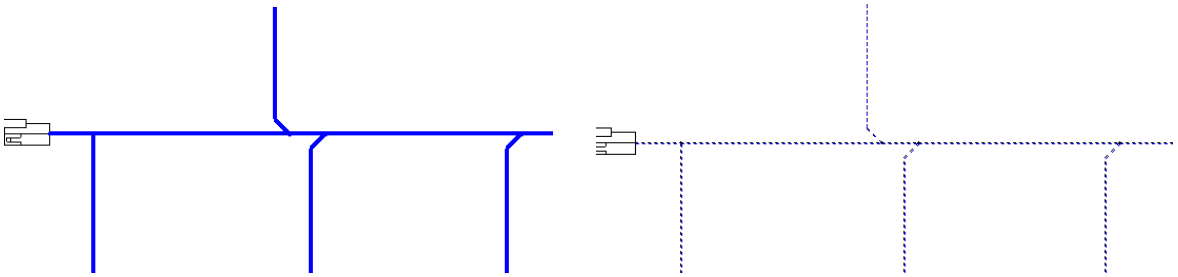
Your results should look something like this:



Switching between Display Modes

In addition to pure, graphical information found in drawings generated with ***E³***, logic background information is also contained in the plan (often displayed in the form of attributes). ***E³*.cable** makes it possible to display this logic information.

To switch between display modes, use the **View → Connection Types** command or the ¶ icon.



In the “normal” view, all connections are displayed as defined by their graphic attributes (color, line width, line type). Connection attributes are not shown graphically in the “normal” view. This is the mode normally used to display the drawing’s output.

Once the Connection Types mode is activated, all connections will be displayed as defined, depending on their attributes (e.g., does the connection have a signal, does it contain a wire). In addition to the signal, the signal cross-references will be numbered. This gives you a quick, graphical overview of the most important connection information available.

Searching and Navigating within the Project

Trying to find a specific element in a large drawing can be very time-consuming. ***E³*.cable**, however, makes this task quite simple. You define the information that will be used to search for the desired element in the **Search** menu. At the push of a button, the corresponding sheet will be opened and the element highlighted in the drawing.

The **Search** menu can be called up with the **Edit → Search** command, the icon or the function key **F3.**

Enter the object type and value in the menu that you want searched. The search results will be displayed in the Output Window. By double-clicking on the desired line, you can jump directly to the sheet, on which the desired object is highlighted. This allows you to find information quickly, as well as having an overview of your drawing.

To delete the highlighting found on this object, use the **Edit → Reset Highlight** command or press **SHIFT + F3**.

Lists and Output Files

Information contained in the project can be output to any document, list and/or file.

That is how ***E³*** can be integrated in various ways in the workflow.
E³ offers for example predefined documents such as table of contents, bill of material, connection list, cable list and terminal plan. It is also possible to work with the integrated Visual Basic programming interface (API). This API is based on Microsoft's COM technology (COM interface) and offers the possibility to load or add data to the drawing using any programming language.

Several lists are predefined in ***E³*** and can be output directly in Excel, Access, Word or ASCII Format, provided that they are installed on your PC. For example, you can create the cable list as an Excel file.

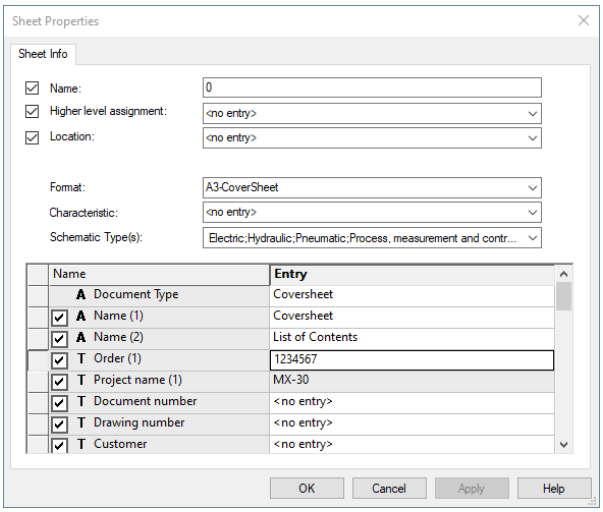
To generate a cable list, use the **Tools → Reports → Excel → Cables** command.
The cable list will be generated in Excel and displayed on the screen.

	A	B	C	D	E	F	G	H	I	
1										
2	Cable List:			MX-30						
3										
4					Cable	Type				Length (
5					-W14					
6	Device Name	Pin	Connector Pin T	Wireseal	Core	Device Name	Pin	Connector Pin T	Wireseal	Length (
7	-X8	1	MINI-S-0.5-1.5		1	-X1	1			
8	-X8	2	MINI-S-0.5-1.5		2	-X1	2			
9	-X6	3	MINI-S-0.5-1.5		3	-X1	3			
10	-X6	1	MINI-S-0.5-1.5		4	-X1	4			
11	-X4	1	MINI-S-0.5-1.5		5	-X1	5			
12	-X4	3	MINI-S-0.5-1.5		6	-X1	6			
13	-X4	4	MINI-S-0.5-1.5		7	-X1	7			
14	-X4	6	MINI-S-0.5-1.5		8	-X1	8			
15	-X3	1	MINI-S-0.5-1.5		9	-X1	9			
16	-X3	3	MINI-S-0.5-1.5		10	-X1	10			
17	-X2	1	MINI-S-0.5-1.5		11	-X1	11			
18	-X2	2	MINI-S-0.5-1.5		12	-X1	12			
19	-X5	1	MINI-S-0.5-1.5		13	-X1	13			
20	-X5	2	MINI-S-0.5-1.5		14	-X1	14			
21	-X7	1	MINI-S-0.5-1.5		16	-X1	15			
22	-X7	2	MINI-S-0.5-1.5		15	-X1	16			
23	-X8	3	MINI-S-0.5-1.5		17	-X1	17			
24	-X8	4	MINI-S-0.5-1.5		18	-X1	18			
25	-X8	5	MINI-S-0.5-1.5		19	-X1	19			

Cover Sheet with Table of Contents

Start the **Add-ons → E³.series Documents → Create Cover sheet** command to automatically create a cover sheet with a table of contents.

Fill in the required sheet texts in the **Sheet Properties**. In the example, define the text **Order (1)** with the value **1234567**. Confirm with **OK**.



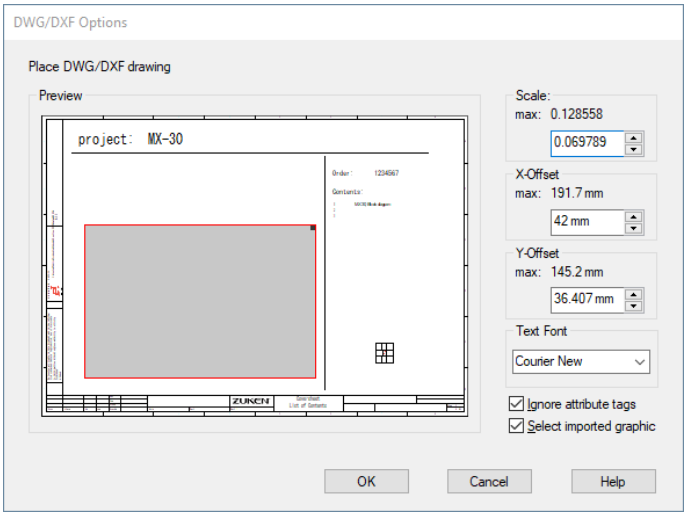
Next, we are going to add an image to our cover sheet, e.g. a company logo.
Select the **File → Import → Image** command and then the **E3-Logo.tif** file from the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data*\. Place the ***E³.*series** logo anywhere on the sheet and then adjust its size.

Lastly, add a mechanical drawing to the cover sheet. This can be done very quickly with the **DXF/DWG Import** command.

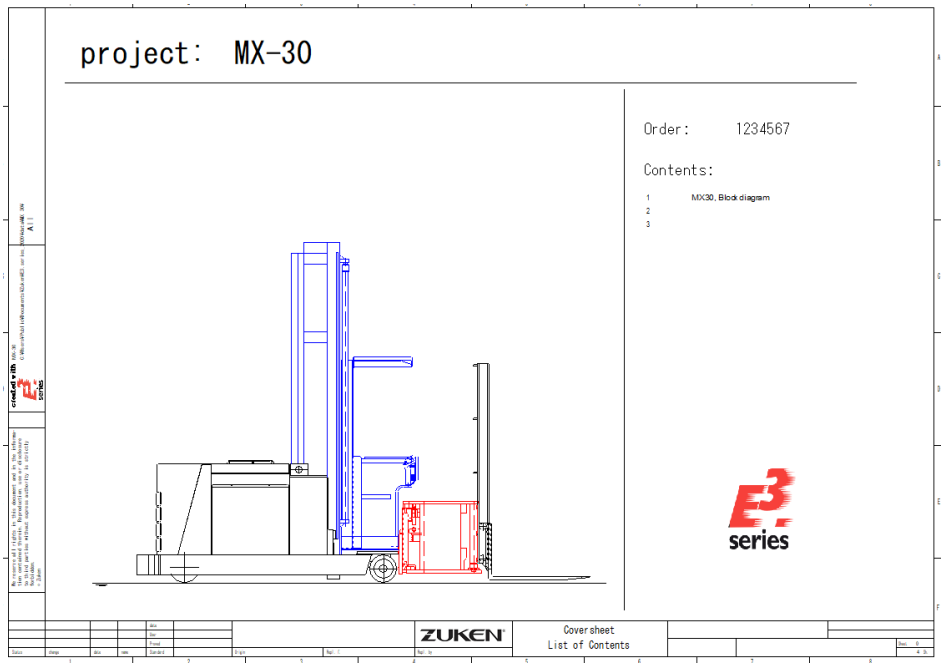
Use the **File → Import → DXF/DWG...** command and select the **mx30.dxf** file in the directory *C:\Users\Public\Documents\Zuken\E3.series_2020\data\MX 30*\.

A menu appears, in which you can specify the mechanical drawing's size and position.

Change the gray rectangle's size by moving the "handle" in the upper right-hand corner back and forth, and then position it on the sheet accordingly. If an exact scale and placement position are desired, these values can be entered directly in the menu. Click **OK**.



Your cover sheet should look like this:



Embedding Additional Documents

Relevant information can be added to a project in ***E³.*cable** if the user decides to do so. This can include documents from other applications, such as instruction manuals, design description, evaluation tables and so forth. ***E³.*cable** allows to integrate these documents into the project. A prerequisite is that the applications used to create these documents are compatible with Active-X (e.g., Microsoft Word, Excel).

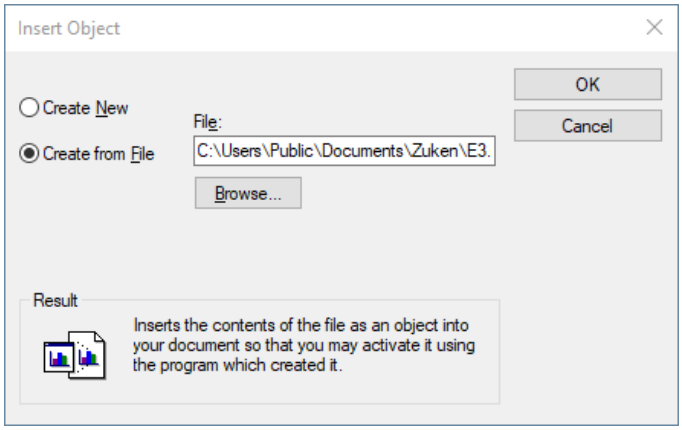
The following example will only work if you have Microsoft Word installed on your computer.

Start the **Insert → Object…** command.

Note: Files embedded in the project with **Insert -> Object…** can be placed on a sheet and -depending on the file- edited in the project. Files embedded with **Insert -> File…** are integrated in the project structure. When the **Create as reference** option is activated, the file is linked in the ***E³*** project. When the option is inactive, a separate copy of the file is embedded in the structure.

The **Insert Object** menu appears displaying all programs installed on your computer that are compatible with Active-X.
The menu is always displayed in the language defined for the operating system.

To open an existing document, select the **Create from File** option and click on the **Browse…** button. Select the file *C:\Users\Public\Documents\Zuken\E3.series_2020\data\MX 30***E3.cable-e.doc**.



Microsoft Word with the document selected is now started within ***E³.*cable**. The ***E³*** user interface changes to the Word interface so that you can edit the document. As soon as a sheet is activated from ***E³.*cable**, you automatically return to the ***E³.*cable** user interface.

The Word document, which is embedded in your project, will also be saved as copy together with the project. The file selected will remain in its original form and will not be changed by any actions within ***E³.*cable**.

Printing Sheets

You can now print individual sheets or the entire project.

To do so, select the **File → Print…** command or the icon in the main menu toolbar. In the **Print** menu, you can specify which computer will be used to output the information, as well as the page format.

Select the **Properties** button in the **Print** menu. You can choose whether to print all of the pages or only specific pages. Enter the page numbers **1-3** to print the sheets you just modified.

You now know how fast and easy it is to work in ***E³.*cable**. However, this was only a short overview of the functionality. If you continue working with ***E³.*cable**, you'll quickly get to know other useful functions that make work much easier.

Enjoy working with

series

First Steps

Creating Symbols and Components with the E³.series Database Editor

Welcome to Your First Tour of How to Create Symbols and Components with E³.series

Up till now, you have worked with components and symbols that already exist in the database.

The following chapter will show how to create own symbols and components with ***E³.*series** and then save them to the database.

What is a symbol?

A symbol is a circuit element that can be assigned to a component. Since many components use the same symbols, these symbols only have to be described once in the database. They can then be used in any combination to create different components.

What is a component?

Simply put, a component is a predefined set of symbols with pins, which possess pin numbers. A component is loaded with all of its symbols. All of its symbols automatically have the same item designation. Components can also be assigned additional information, e.g. part numbers, description, supplier, price and so forth.

Preparations for Creating Symbols

Symbols and components are created and modified in Database Editor Mode. This is a special mode within ***E³.*series** that allows you to create components and symbols as well as enter standard definitions for the attributes, text, colors etc. in the databases.

Since several users can access the databases simultaneously (configuration, components and symbols), all modifications are updated instantly for all users. If Database Editor Mode is started, a new instance of ***E³.*series** is started with specific database editor functions. To change to Project Editor Mode, for example, to use symbols from an existing drawing for a new symbol or to test created symbols or components, simply switch to the ***E³.*series** instance, in which a project is opened.

Creating a New Symbol

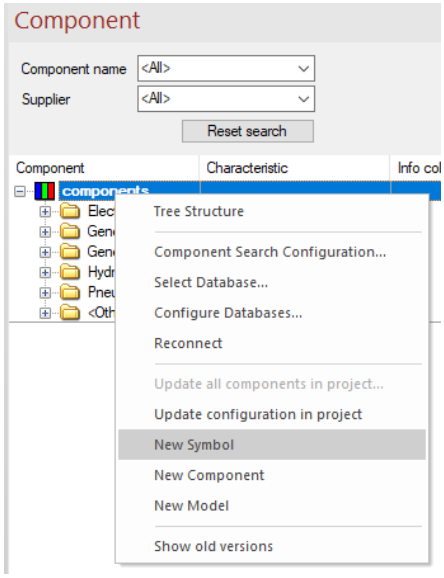
The fastest way to create a new symbol is to use a similarly existing symbol as a template, make the required changes and save it under a new name.
The following section, however, describes how to create an entirely new symbol without using a template.

A new symbol is created in a few steps:

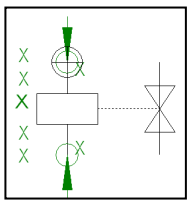
- Define the symbol type and name.
- Create the symbol's graphics.
- Define the pins.
- Define the text types used and their position on the symbol (Higher level assignment, Location, Device designation, Pin name, …).
- Define the origin.
- Define the space requirement.

We are going to create a new symbol for a valve coil.

Switch to Database Editor mode to create the symbol. To do so, select the **Tools -> Start Database Editor** command from the main menu bar or right-click on the uppermost level in the Database Window and select the **New Symbol** command from the context menu. This will start a new instance of ***E³*** in Database Editor mode. A drawing area for the new symbol is opened in Database Editor.

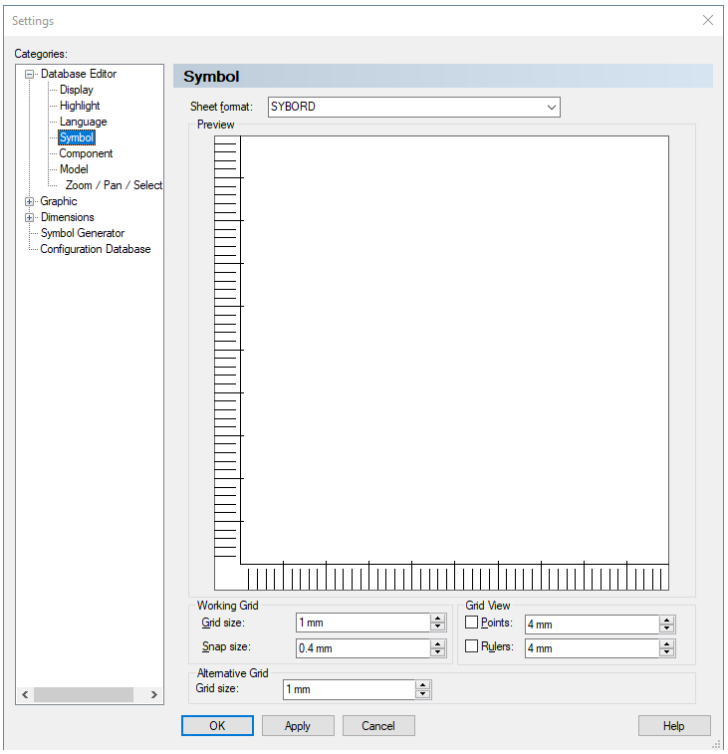


The new symbol should look like this when finished:



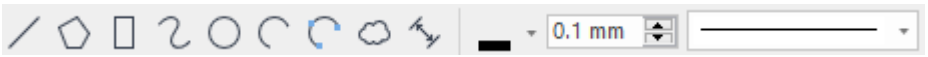
The cross in the middle of the drawing area represents the symbol's origin. It will also be used as the symbol's first pin.

Activate the display grid using the icon. The grid points will help you in drawing the symbol. More grid settings can be defined in the **Settings** (**Tools → Settings → Database Editor → Symbol**).



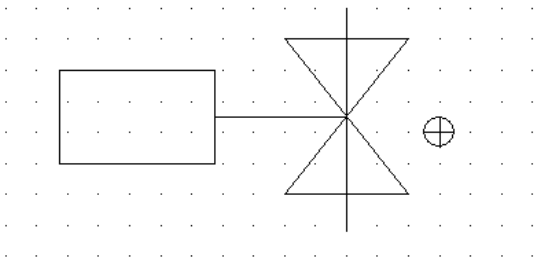
First, draw the graphic of the symbol.

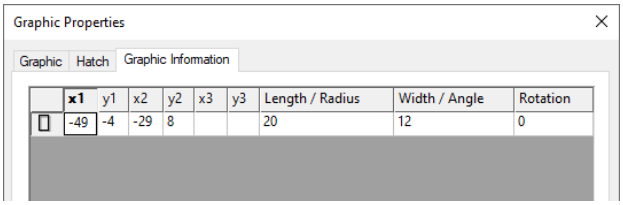
Zoom into the area around the origin and begin by drawing the rectangle and other lines as shown in the screenshot below. Use the Graphic toolbar to define and change the line width, color and type of a selected element.



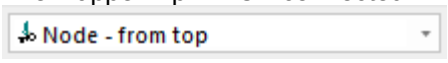
Remember to use the grid points to assist you when drawing the symbol. Activate the **Copilot** in **Tools → Settings → Database Editor → Display** to see the graphic's size definitions while drawing.

Using the **Graphic Properties – Graphic Information** tab it is possible to exactly define the graphic size.

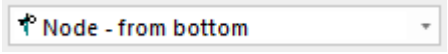




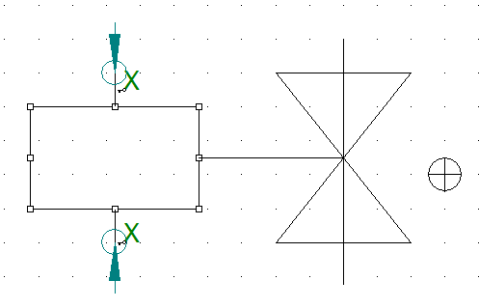
Next, we have to define the pins. ***E³.*series** defines electrical pins including their preferred direction of approach.

The upper pin is connected from top. Select the node symbol **Node - from top** 

from the drop-down list of the **Database Symbol Node** toolbar and place it using the icon.

Define the second pin in the same way but use the node symbol **Node - from bottom** 

.

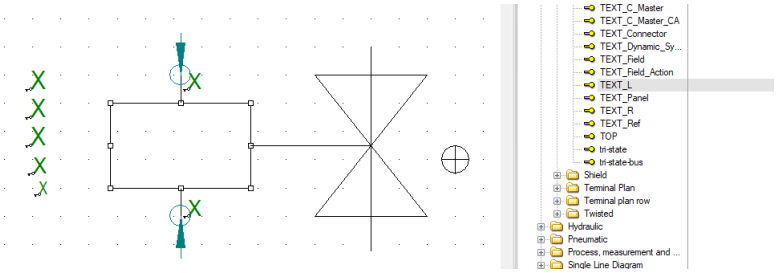


The symbol is assigned the pin names of the component within which it is used. Therefore, the pin sequence within a symbol is very important when assigning them.

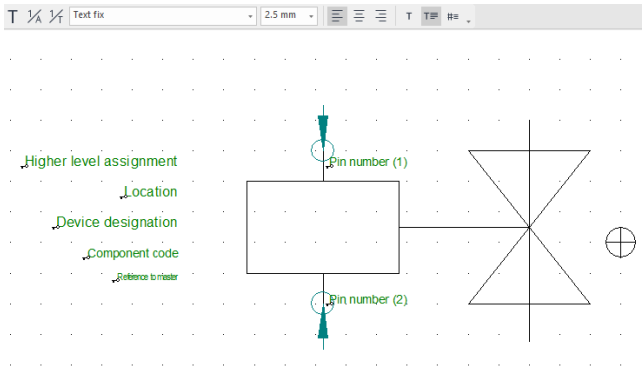
Lastly, assign the symbol text nodes for the higher level assignment, location and device designation.

Use the already defined template **TEXT_L** for the text positions. It can be found in the **Misc** view of the Database Window in the folder **Electric - Misc_Template**. The template **TEXT_L** can be placed like a symbol with Drag & Drop or by using the command **Place** in the context menu.

Notice how the text nodes are represented by a green X in the drawing. This means that the actual values with their defined text parameters have been entered at this position.
The graphic description is now complete.



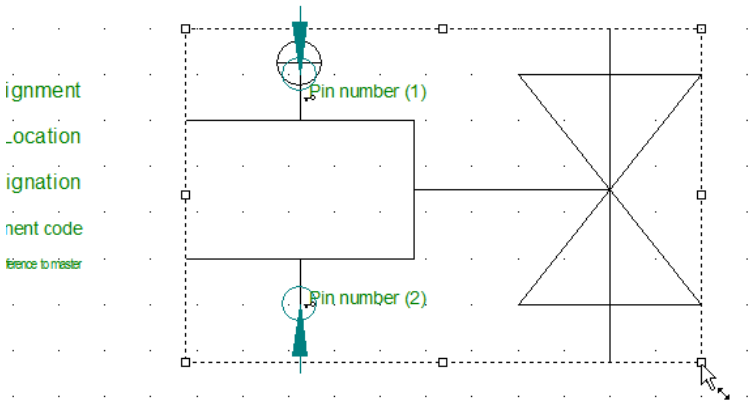
To see what text will be entered at the text nodes, change the display mode with the icon.



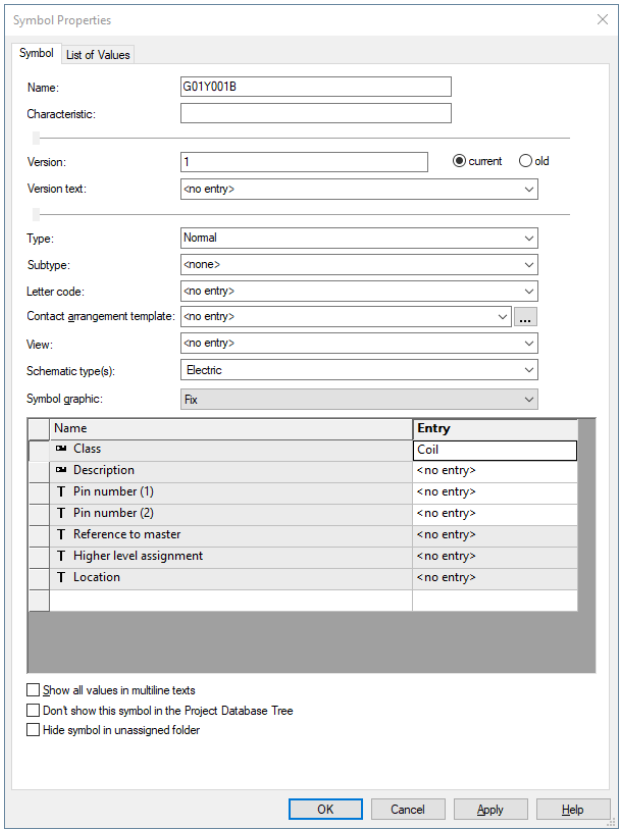
The next steps are to define the symbol's origin as well as its space requirement.

To define the origin click on the corresponding icon on the toolbar. Place the origin on the upper pin by left-clicking on the pin.

Next, we will define the symbol's space requirement. Click on the button on the toolbar. The system will define the space requirement based on the symbol's graphics. To change the space requirement manually, adjust the rectangle around the symbol by dragging the corresponding handles.



Finally, we have to define several symbol-specific properties such as the symbol's name under which it is stored in the database.
This information can be entered in the **Symbol Properties** dialog displayed when right-clicking on the drawing area and selecting the corresponding command from the displayed context menu: the name of the new symbol is **G01Y001B**, the symbol's class is **Coil** (this defines the folder, in which the symbol appears), the schematic type is **Electric**.

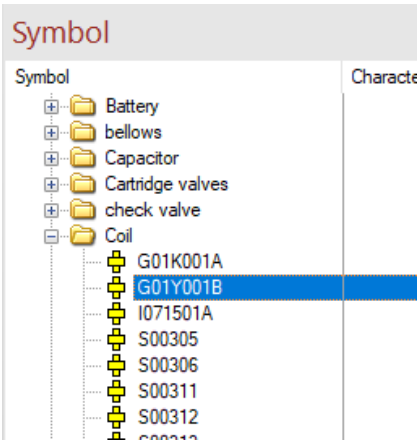


The symbol names used in the ***E³*** database are in accordance with the definitions in the *IEC 617* standard.

The symbol can be saved to the database using the **Save to Database** command in the symbol's context menu or by using **File → Save to Database** in the standard toolbar.

To exit Database Editor mode, use the command **File → Close**.

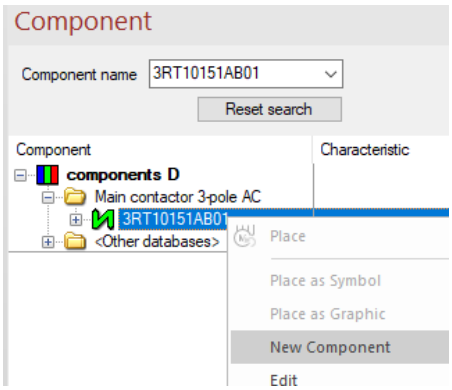
The new symbol can be used in the circuit diagram by all users accessing this database. It might be necessary to refresh the database by clicking on the database name and pressing the **F5** key.



Creating a New Component

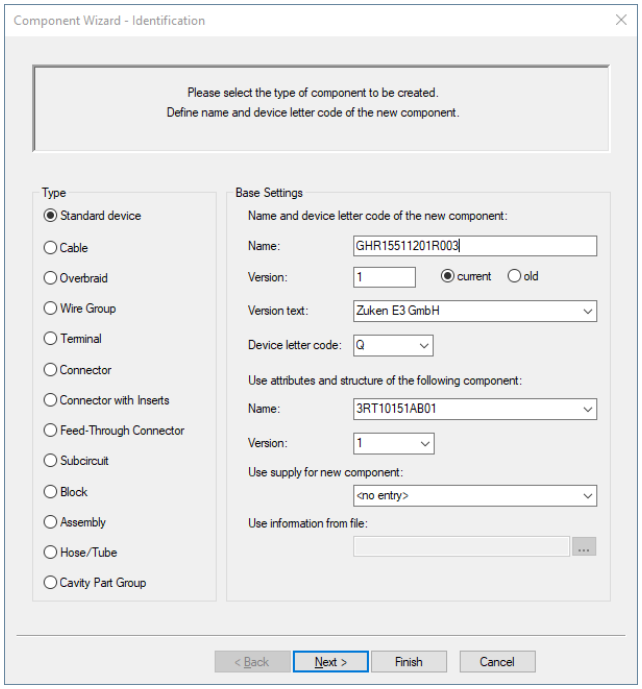
The following section describes how to create a new component. The new component is named **GHR15511201R003**. It is a loop contactor from the supplier ABB that has two NO contacts. To create this new component call up a similar one from the database, change it and save it to the database under a new name.

In this example, use the component **3RT10151AB01** found in the **Electric – Main contactor 3-pole AC** folder. Select the command **New Component…** from the component's context menu. The Component Wizard is started and the new component can be created.
Note: When this command is selected in Project mode, Database Editor mode is started.



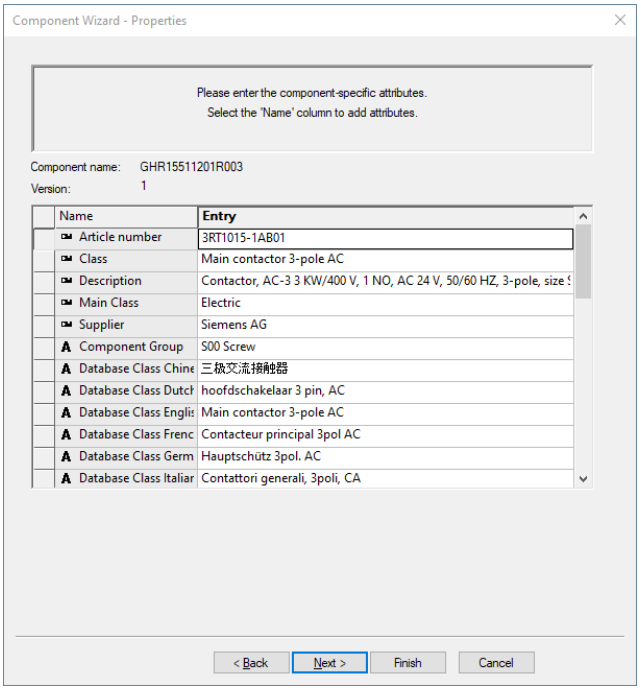
The Component Wizard guides you through the process of creating a component.

First, define the new component's type, name and other global parameters in the **Identification** dialog. In this example, we are going to create a standard device. Select **Standard device** and enter the name **GHR15511201R003**.



Click **Next**.

The next step is to enter the component-specific attributes in the **Properties** dialog. These values define the new component. The values of the component on which the new component is based, are already entered under the **Entry** column. Change only those entries that have differing text.



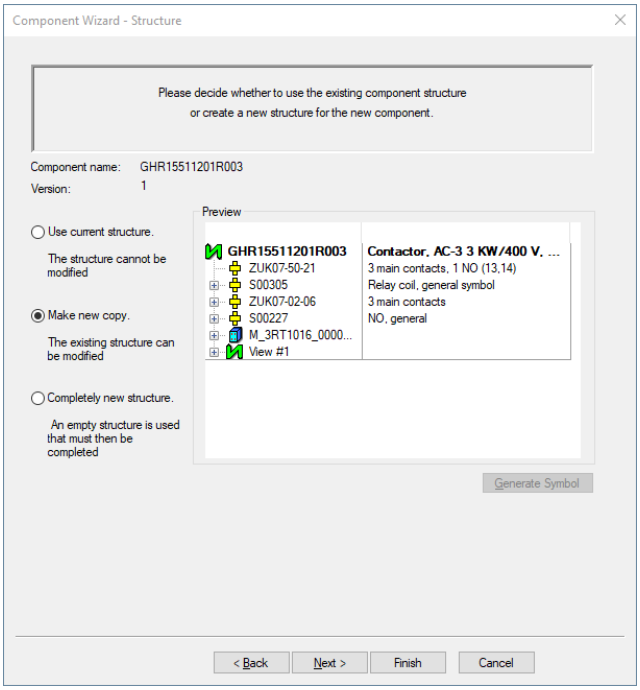
Note: Customer-specific properties can also be entered here. These must be previously defined in the **Format → Attribute Names** table.

Click **Next**.

Now define the component's structure as well as the symbols that will represent the component in the circuit diagram. This is done in the **Component Wizard - Structure** dialog.

If an existing structure is to be used, the component can be saved directly to the database without any additional steps.
Note: If the structure is to be changed, it is recommended to create a copy of the structure. Changes done in this menu to any of the original component's structure will apply to all components using this structure. This may cause undesired results in large projects.
If a **new copy** of the structure is created, it can be edited at any time without the modifications affecting other components.

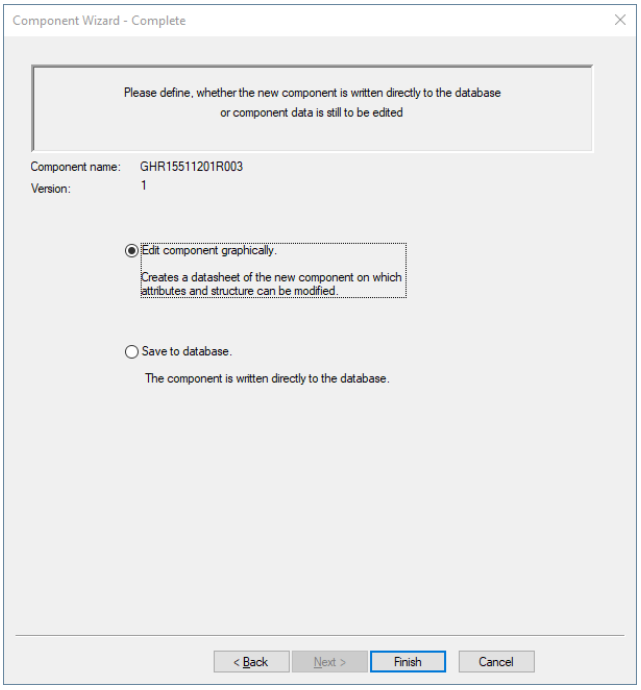
Activate the **Make new copy** option in the menu.

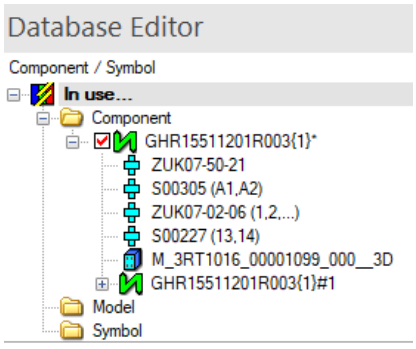


Click **Next**.

Finally, the Component Wizard allows you to decide whether to save the new component directly to the database or to continue editing the component.

Activate the **Edit component graphically** option and exit the Wizard by pressing **Finish**.
The component and its symbols are loaded to Database Editor for modification.

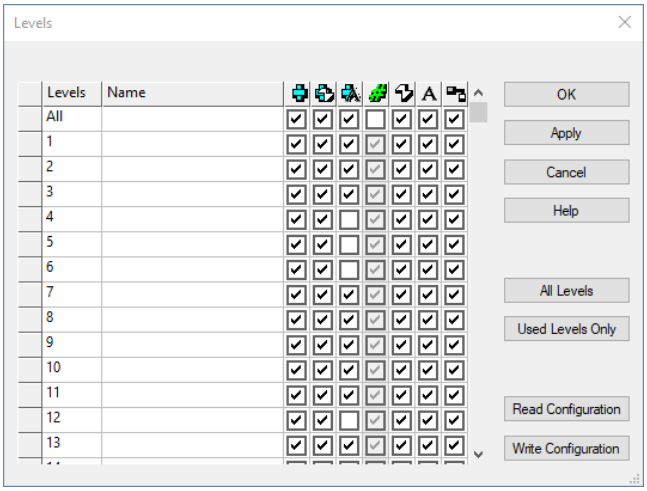




Setting Levels

***E³.*series** distributes information on various levels (or layers). Imagine these levels as transparent sheets, which can be displayed or not in the drawing. That is how it is possible to make the different information visible or invisible. Entire information types (e.g. empty text nodes) can also be switched on or off.
When editing a component, it is not necessary to use all of the available information so just switch off the superfluous information using the levels command.

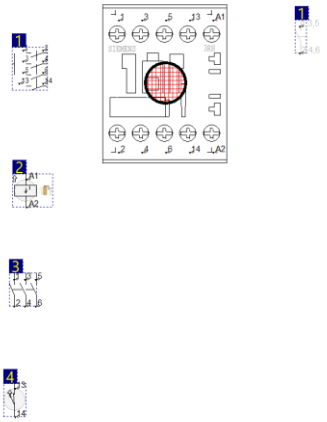
To do so, select the **View → Levels...** command or simply press the **L** key. A menu appears, in which you can switch on or off individual or all levels. The **Read Configuration** command is used for loading predefined level display configurations, the **Write Configuration** command is used to save existing configurations.



Select the **Read Configuration** command and **component-editor.vis** file from the *C:\Users\Public\Documents\Zuken\E3.series_2020\data* directory and close the menu. The configuration is loaded immediately. Close the menu with **OK**. The display of the symbols that belong to the component is reduced now and much clearer.

All of the component's symbols along with their pin names as well as the component's model (the component's representation in the panel) will now be displayed in the drawing area. Additional symbols can be added here by placing them from the Database Window or existing symbols can be deleted.

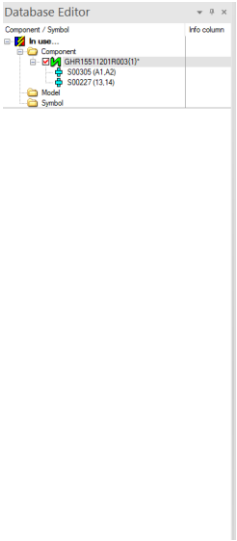
Show the symbol numbering by clicking on the "connection types" icon ¶.The drawing area will look as follows:



In the following example, delete the component's contact arrangement (marked as number **1**), the symbol for the main contacts (marked as number **3**) and the symbol for the display in another view (marked as view number **1**). Replace them with another contact arrangement and an additional NO contact. The model in the example is irrelevant and can be deleted.

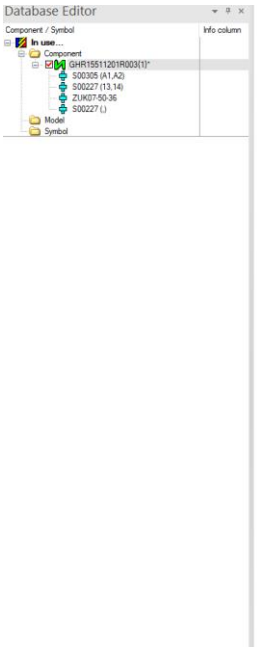
First, delete the symbols **1** and **3** that are no longer required from the drawing area. To do so, click on the symbols and press the **Del** key or select the **Delete** command from the context menu displayed by right-clicking on the symbols.
Note: The symbol numbering is dynamically changed when a symbol is added or deleted.

Activate the view symbol displayed in the Database Editor window by the name **ZUK07-02-01** by double-clicking on it. The view symbol is highlighted in the drawing area. Delete the view symbol as well as the model.
Double.click on the component name to reset the display on the drawing sheet.

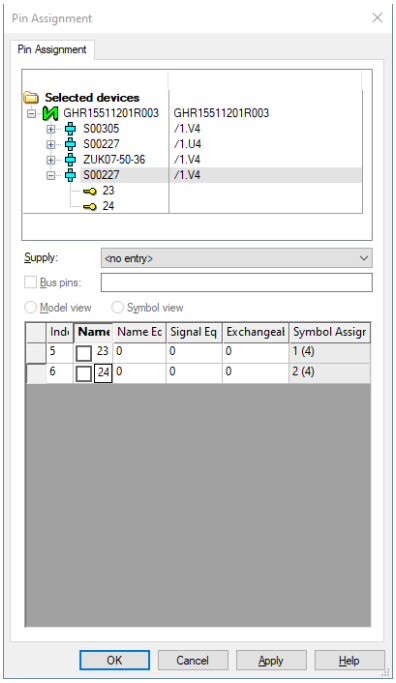


Switch to the **Symbol** tab of the Database window and place the symbol **ZUK07-50-36** from the **Electric - Contact Arrangement** folder. Drag it to the drawing area and arrange it above symbol **1**.

The second NO contact is stored in the symbol database in the **Electric - Contacts** folder. The symbol's name is **S00227**. Place the contact symbol between the symbols **1** and **2**.



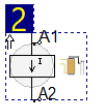
Pin names can also be defined and changed in the Database Editor.
Selecting the **Pin Assignment…** command from a symbol's context menu opens the **Pin Assignment** dialog in which the symbol's pin names are displayed. Pin names can be entered here. Activate this command on the NO contact symbol **S00227** that was added and enter both new pin names **24** and **23**. Confirm with **OK**.

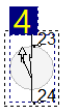


The next step in defining the component is to determine the order of the symbols within the component. This mode can be activated with the **Edit → Order Definition → Symbols in Component** command or the icon. The **Symbol Order Definition** dialog appears.

Click on the symbols in the order in which they are to appear in the component. This means that the contact arrangement should be first, followed by the coil, then the contact with pin numbers **13**, **14** and finally, the contact with the pin numbers **23**, **24**.

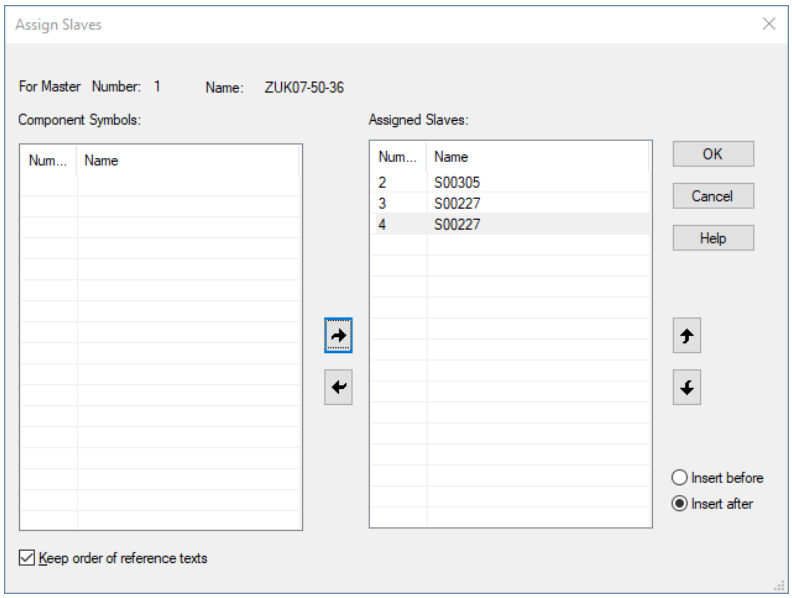
The new sequence will be displayed on the symbols in the blue text boxes. Exit the symbol order mode by pressing the <**ESC**> key.



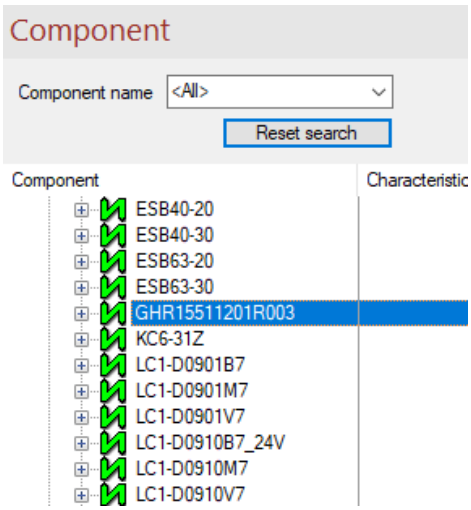


The last step in defining the component is to specify the relationship between the symbols within the component (in case component cross-references are desired).

To do so, right-click on the contact arrangement ***ZUK07-50-36*** and select the **Master/Slave Assignment…** command from the context menu. The **Assign Slaves** dialog appears. Transfer all symbols into the *Assigned Slaves* column. This produces an online cross-reference between the component symbols and the contact arrangement in project.



With this, the complete component is defined and can now be saved to the database. Right-click on the drawing area and select the **Save to Database** command from the displayed context menu or the **File -> Save to database** command from the main menu bar.



Special Components

In addition to standard symbols and components, ***E³.*series** can be used to create special components, especially for those used in ***E³.*cable** such as connectors and cables.

A **connector** can be depicted in different ways in a project, not only as a connector pin or a socket. They can be represented as:

- block connectors
- documentation graphics, which are displayed in the Preview Window and placed using the **Place as Graphic** command
- different views of a connector that can be selected by means of the **Place New Device View** command

A component can also be a **cable**, referred to as cable types. A cable type describes a special cable that is used in a defined form in the project. Its structure cannot be easily changed.

A cable type may contain the following information:

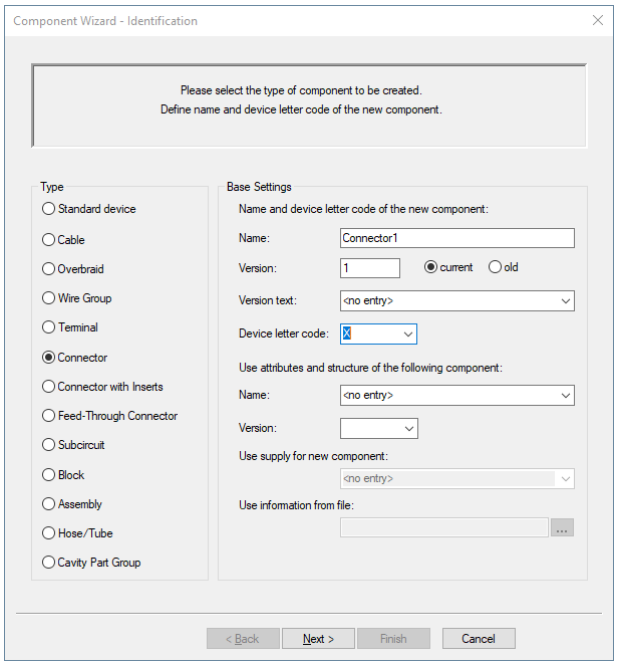
- component master data
- number of conductors with conductor attributes
- cable attributes
- shields
- twisted pairs
- bundles

Creating a New Connector

Start the Component Wizard by selecting the **New Component** command as described above. The Database Editor mode is activated and the **Component Wizard** started, displaying the **Identification** page.

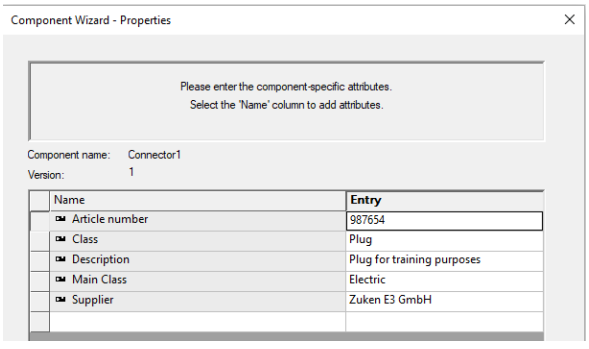
Select the component type **Connector**. Enter the new component's name as **Connector1** and its device letter code (here ***X***).

Since a completely new connector component is created, nothing is entered in the lower section of the dialog where a component can be defined whose structure is to be used.



Click **Next**.

The Component Wizard's **Properties** dialog is used to define the new component's master data. If the new component is based on an existing component, these fields will already contain entries. Since a completely new component is created in the example, these empty fields must be filled. Select any properties for the new connector.



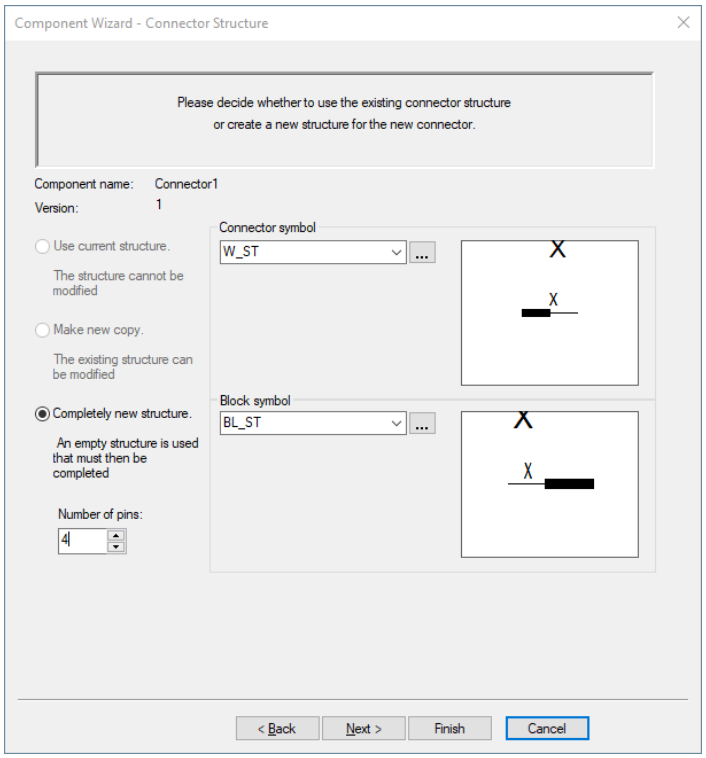
For component attributes that are not displayed in the list, they can be added in the last line by simply clicking on the line and selecting the desired attribute from the drop-down list. These user-specific attributes, however, must be previously defined in the attributes table.

This can be accomplished by using the **Format → Attribute Names** command in the Database Editor mode.

After defining the component's master data, click **Next**.

The Component Wizard's **Connector Structure** dialog appears next. The **Completely new structure** field is already active since a new structure for the connector is created.

Select a connector symbol to represent the connector in the circuit diagram. Select the socket symbol **W_ST**.
The connector display on a block is defined by the Block symbol. This definition is optional. In this example, however, a block symbol is also specified.
Select the block connector symbol **BL_ST**. The new connector **Connector1** can thus be placed on a sheet within the circuit diagram, as well as on a block. If the block symbol is not defined, the connector can only be placed in a circuit diagram
Define the **Number of pins** as **4**.

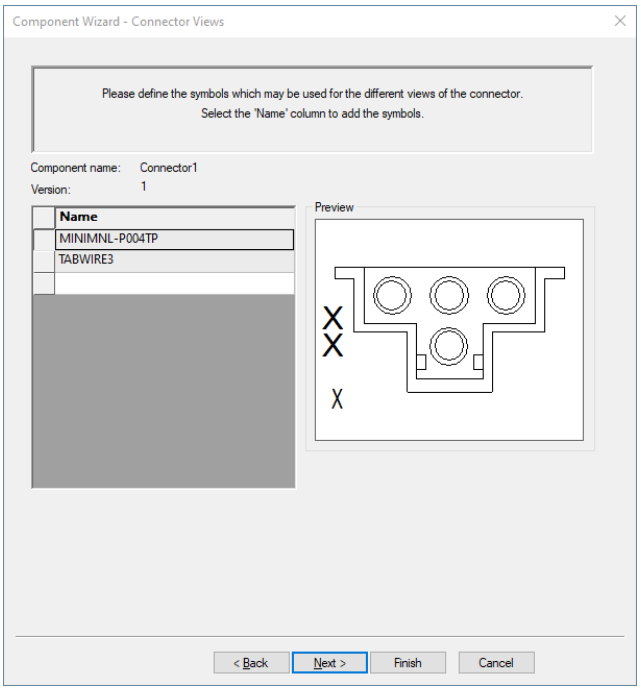


Click **Next**.

The next step is to define the different **Connector Views** in the Component Wizard's **Connector Views** dialog. A connector can be defined with any number of views. This definition is optional.

The drop-down list displays all connector views that are available as symbols in the database.
Select the symbol **MINIMNL-P004TP** as the side view and the symbol **TABWIRE3** as a second possible view. The **TABWIRE3** symbol is used to automatically create a table listing the pin designations, signals and conductor information. This symbol is useful for the cable documentation.

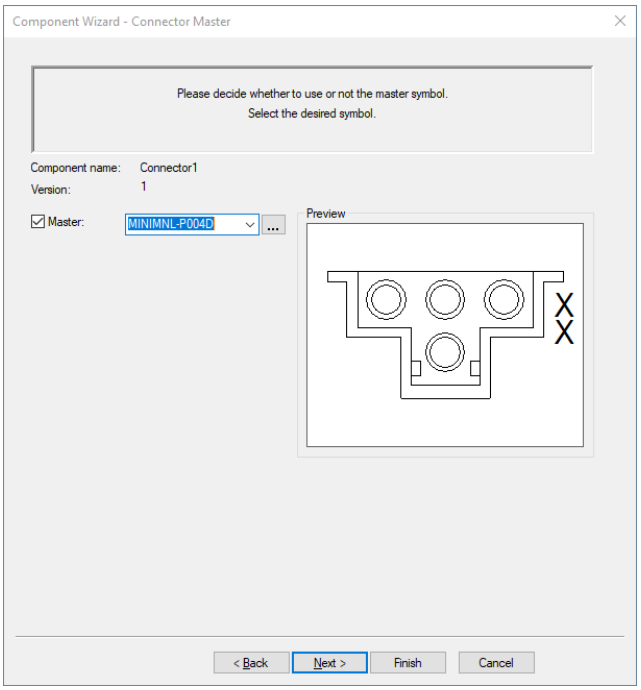
Note: To delete a symbol, right-click on the gray field next to the symbol name and select **Delete Symbol Name** from the displayed context menu.



If no alternative connector views are defined, the Component Wizard can be terminated by clicking on **Finish**. However, to complete the example, click **Next**.

The Component Wizard **Connector Master** allows you to define the connector's documentation graphics. This symbol already exists in the database. Select the **MINIMNL-P004D** symbol from the drop-down list.

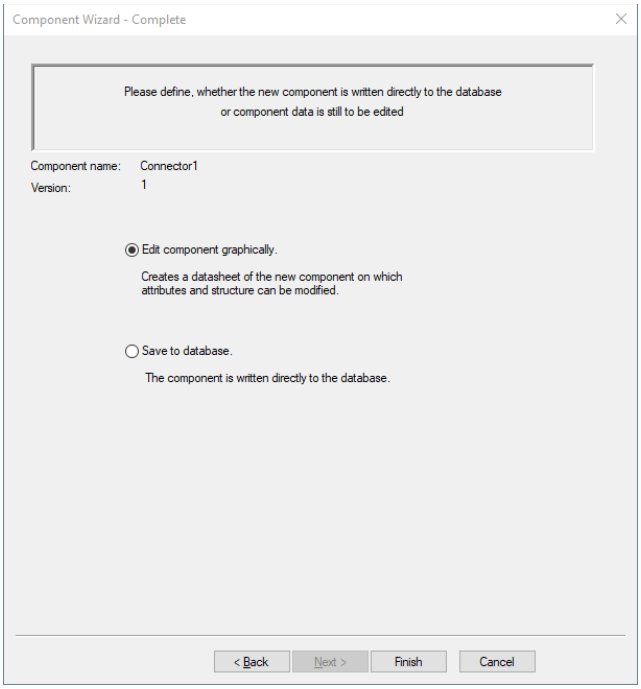
When selecting a component from the database, the master symbol will be displayed in the Preview Window. Master symbols can also be placed by means of the **Place as Graphic** command, and for documentation purposes, be attached to objects in the circuit diagram.



To continue, click **Next**.

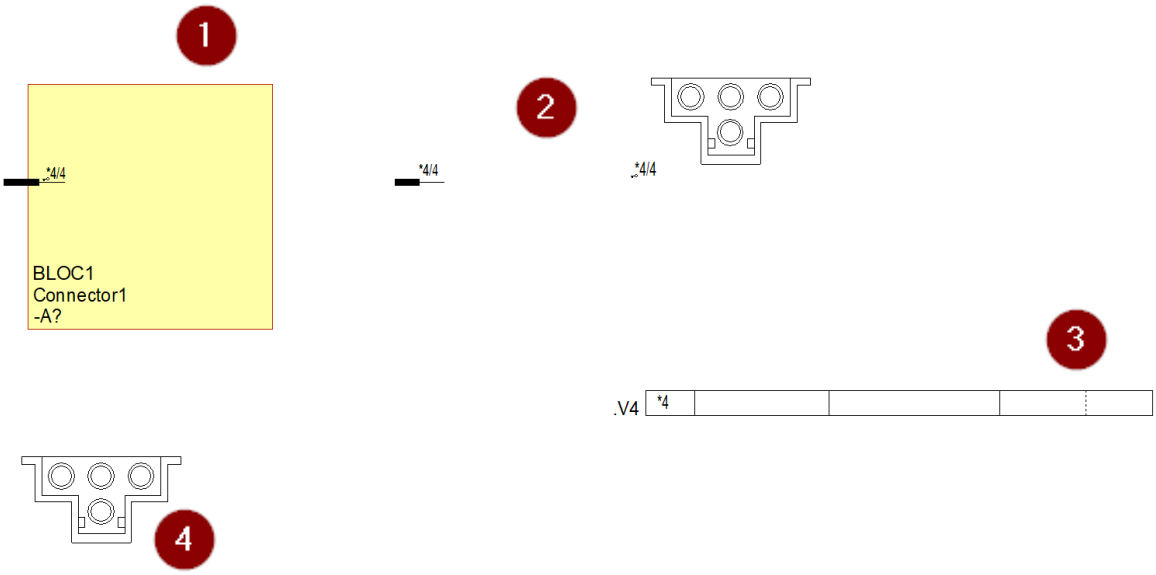
Finally, the Component Wizard allows you to save the newly created connector directly to the database in the **Complete** dialog.

Select the option **Edit component graphically**. The new connector will be graphically displayed and can then be modified.



Click **Finish** to exit the **Component Wizard**.

E³.cable opens a new sheet, in which the objects defined so far are arranged.



1 Display of connector pins on a block

2 Display of the connector pins outside of blocks

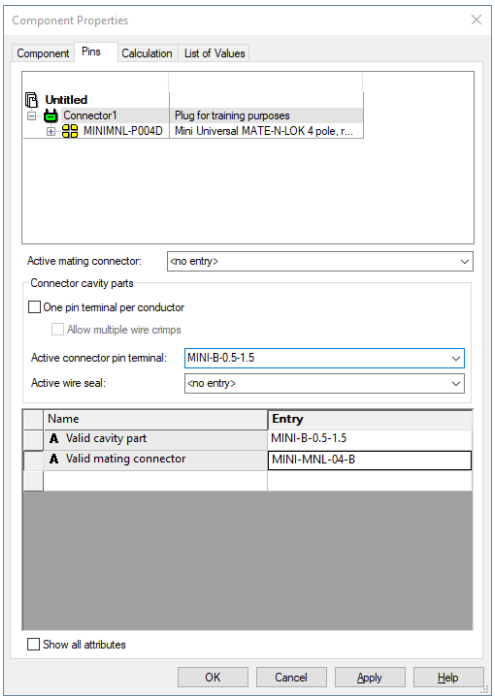
3 Other view(s) of the connector

4 Display of the connector in the Preview Window and when placed as graphic

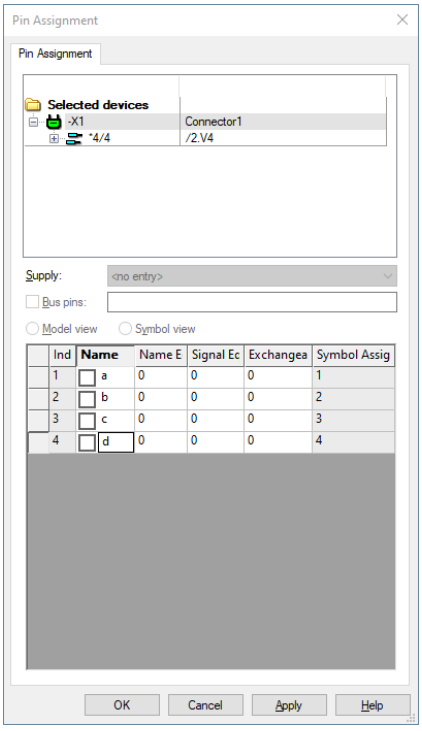
The **Component Properties** dialog box allows to define active **Mating Connectors** and **Connector Pin Terminals**.

In the example, the definitions for mating connector and connector pin terminal are valid for the whole connector and all pins. However, it is also possible to define valid connector pin terminals for each specific pin. The mating connector is automatically loaded from the database and placed upon connecting to the connector.

Right-click on the drawing sheet and select the **Component Properties…** command from the displayed context menu. Switch to the **Pins** tab and add the corresponding attributes **Valid mating connector** and **Valid cavity part**, then select values from the drop-down list. The valid cavity part is assigned the value **MINI-B-0.5-1.5** and the valid mating connector is assigned the value **MINI-MNL-04-B**. Select the cavity part as **Active connector pin terminal**.



Lastly, the pin names can be changed. Right-click on the drawing sheet and select the Pin Assignment… command from the displayed context menu.
If the pins are not defined, they will be numbered sequentially. In the example, the pin names will be changed (a, b, c, d).



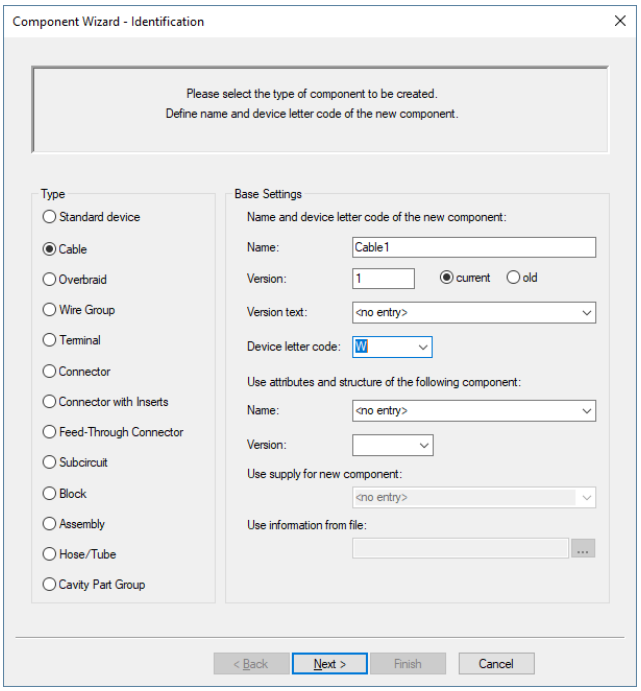
The new connector component, which has been completely described, can now be saved to the database. To do so, right-click in the drawing area and select the **Save to database** command from the displayed context menu or select **File -> Save to Database** from the main menu bar.

Creating a New Cable

Start the Component Wizard by selecting the **New Component** command as previously described. *E³* activates the Database Editor mode and the **Component Wizard** is started, displaying the **Identification** page.

Select the component type **Cable**. Enter the new component's name (here **Cable1**) and its device letter code (here **W**).

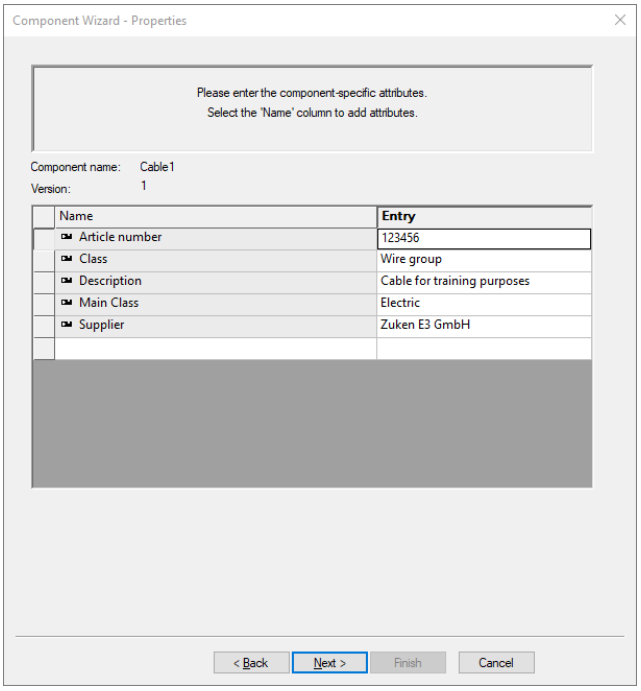
Since a completely new component is created, nothing is entered in the lower section of the dialog where a component can be defined whose structure is to be used.



Click **Next**.

The Component Wizard's **Properties** dialog is used to define the new component's master data. If the new component is based on an existing component, these fields will already contain entries. Since a completely new component is created in the example, these empty fields have to be filled. Select any properties for the new cable.

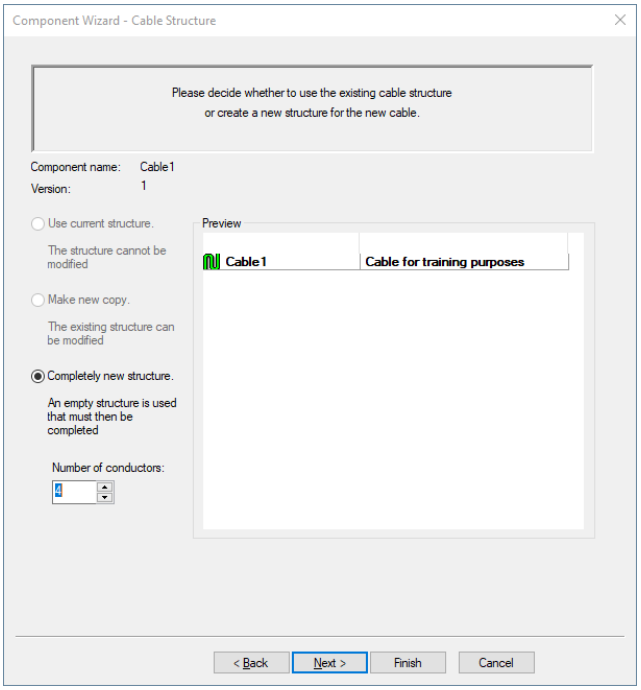
For component attributes that are not displayed in the list, they can be added in the last line by simply clicking on the line and selecting the desired attribute from the drop-down list. These user-specific attributes, however, must be previously defined in the attributes table.
This can be accomplished by using the **Format → Attribute Names** command in the Database Editor mode.



After defining the component's master data, click **Next**.

The Component Wizard's **Cable Structure** dialog appears next. The **Completely new structure** field is already active since a new structure for the cable is created. This definition cannot be changed.

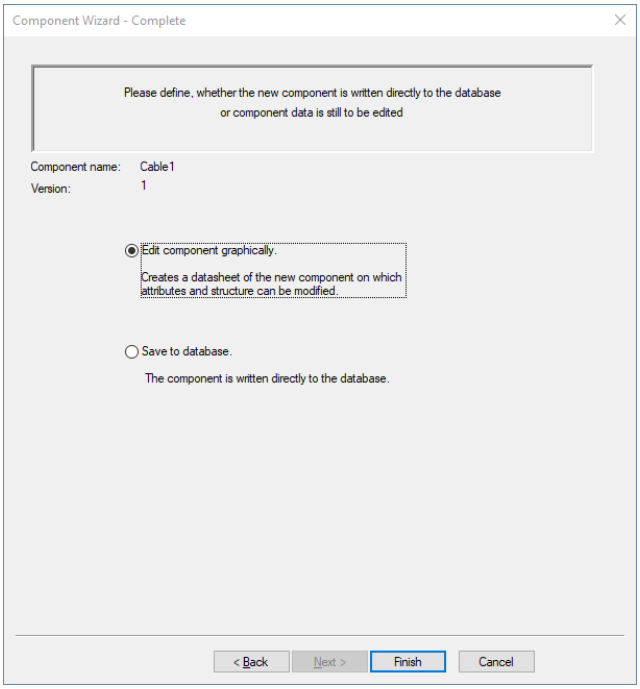
Enter the **number of conductors** as **4** in the example. If, at this point, the definitions are sufficient, exit the Component Wizard and the definitions will be written directly to the database.



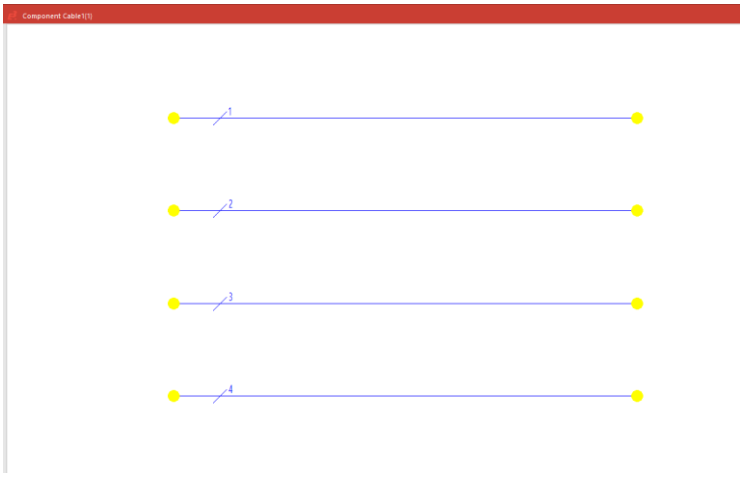
To complete the example, however, continue by clicking **Next**.

Finally, the Component Wizard allows you to save the new created cable directly to the database. The option **Edit component graphically** allows you to refine the cable definition even more.

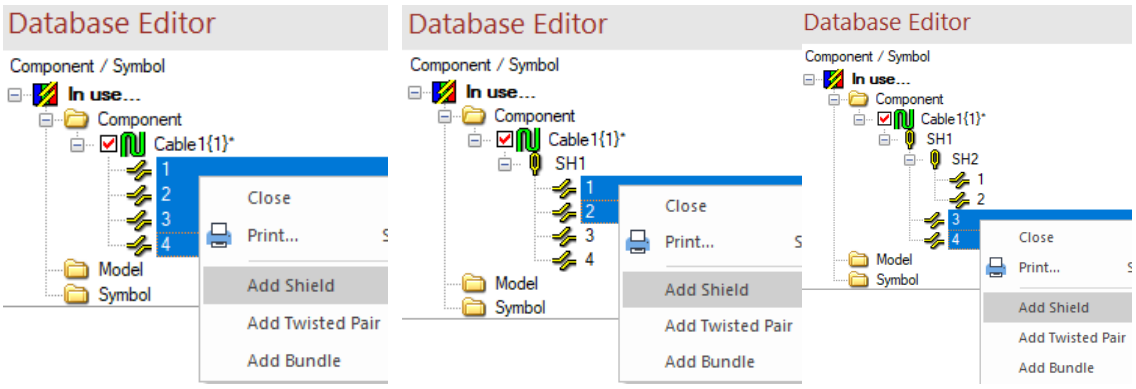
Select this option and click on **Finish**.



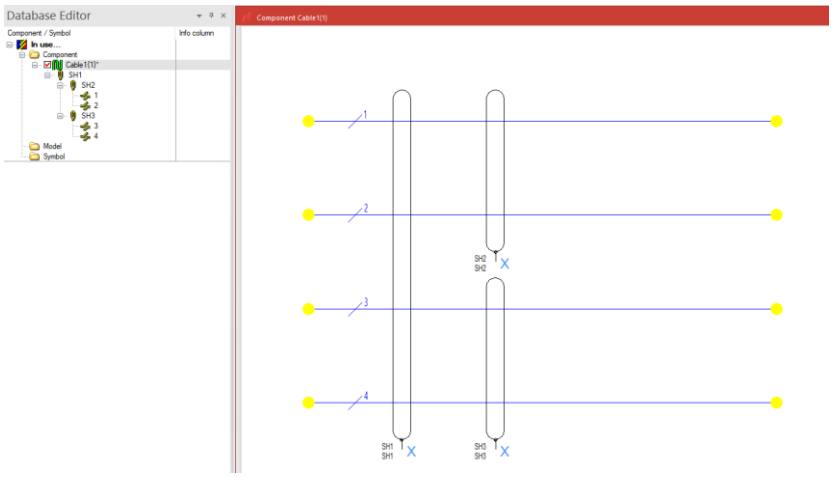
***E³.*cable** opens a sheet, on which the four conductors are displayed.



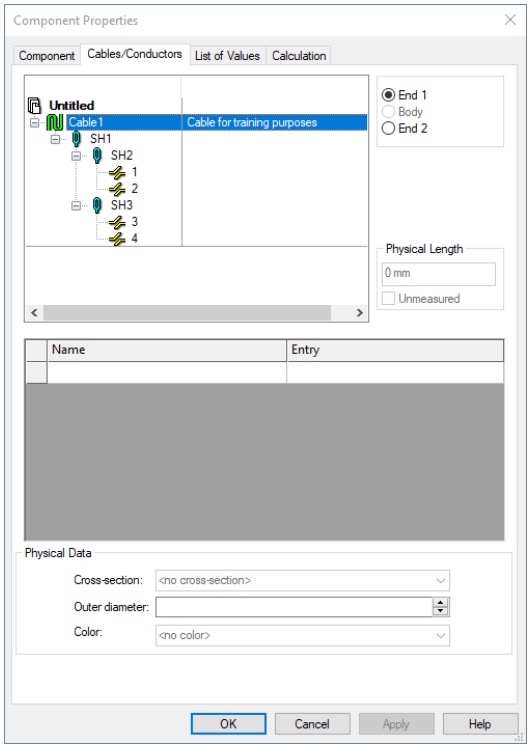
These conductors are completed by adding shielding. To do so, select the cable's conductors to be shielded in the Database Editor's Project Window and select the **Add Shield** command from the context menu displayed by right-clicking on the conductors. First, add a shield for conductors **1** to **4**. Next, add a shield to conductors **1** and **2**, and again to conductors **3** and **4**.



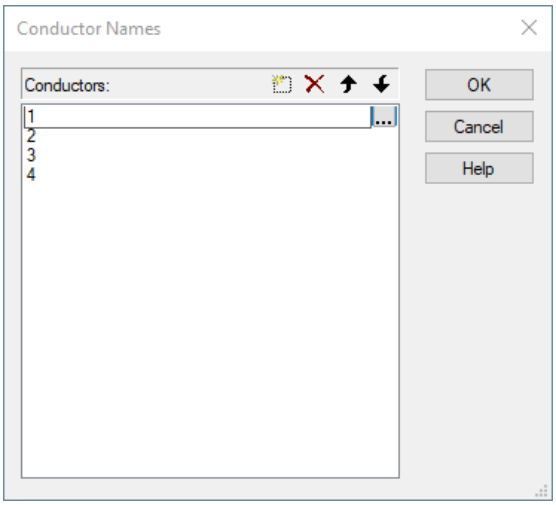
The results should look like the following:



Using the **Component Properties**, attributes can now be assigned to the component itself, the cable ends, the conductors and the conductor ends. To do so, right-click on a conductor or shield in the drawing area and select the **Component Properties…** command from the displayed context menu or right-click on the component in the Project Window and select the **Properties…** command from the displayed context menu.



Optionally, it is possible to modify the conductors' numbering and names. To do so, right-click in the drawing area and select the **Conductor Assignment…** command from the displayed context menu. Select a conductor and type in a new name or press **F2** to rename the conductor. To assign an entry from the Translation Table, press F2 and click on the ... button to select the translation table, for example to define language-dependent conductor colors.



The new cable component is complete and can now be saved to the database and is available to all users. Select the **File -> Save to Database** command from the main menu bar or **Save to Database** from the context menu displayed upon right-clicking in the drawing area.

You now have learned to create components, connectors and cables in the database using the ***E³.*series** *Database Editor*. The *Database Wizard* helps to create these additional elements in the database, which can then be used immediately in the project.

Enjoy working with

Americas

North America

Zuken USA Inc.
Westford, MA 01886, USA
Tel: +1 978 692 4900

Asia

Japan

Zuken (World Wide Head Office)
Yokohama, Kanagawa 224-8585, Japan
Tel: +81 45 942 1511

China

Zuken (Shanghai) Technical Center Co., Ltd.
Room 301, No.555 Nanjing West Road, Shanghai, 200041,
People's Republic of China
Tel: +86-21-3218-1784

Korea

Zuken Korea Inc.
Seoul 135-283, Korea
Tel: +82 2 5648031

Singapore

Zuken Singapore Pte Ltd.
#22-05 Gateway East, Singapore 189721
Tel: +65 6392 5855

Taiwan

Zuken Taiwan Inc.
Taipei 110, Taiwan
Tel: +886 2 7718 1116

Europe

Germany

Zuken GmbH (European HQ)
D-85399 Hallbergmoos, Germany
Tel: +49 89 7104059 00

Zuken E3 GmbH
D-89079 Ulm, Germany
Tel: +49 7305 9309 0

Zuken E3 GmbH
D-30659 Hannover, Germany
Tel: +49 511 8595 9489

Switzerland

Zuken E3 GmbH
CH-5504 Othmarsingen, Switzerland
Tel: +41 62 561 08 00

United Kingdom

Zuken UK Ltd.
Bristol, BS32 4RF, UK
Tel: +44 1454 207 801

France

Zuken S.A.
#91974 Les Ulis Cédex, France
Tel: +33 1 69 29 48 00

Italy

Zuken S.r.l.
20090 Milanofiori Assago, Milan, Italy
Tel: +39 02 575 921

Netherlands

Zuken GmbH
NL-6075 HA Herkenbosch, The Netherlands
Tel: +31 475 520 998

www.zuken.com/e3

All trademarks mentioned are the property of their respective owners, Copyright © Zuken E3 GmbH

ZUKEN®