WS CAD

SUITE X

EE CE PI FE BA EI

Electrical Engineering, Cabinet Engineering, P&ID, Fluid Engineering, Building Automation, Electrical Installation

GETTING STARTED

Table of Contents

Introduction

General

This "Getting Started" guide is designed to introduce you to the handling of the WSCAD SUITE as quickly as possible with the help of a practice example. Consequently, many functions of the program are not described here. The intent and purpose here is to enable you to independently create simple schematics with the associated report lists.

The exercise examples here refer to the project **SUITE DEMO**. To use the examples, you will need either the WSCAD SUITE X Demo Version or a licensed software package from the WSCAD SUITE.

The scope of the WSCAD SUITE X Demo Version corresponds to the **Expert** version. All automatic functions such as contactor management, PLC Manager, part management, cross-references, numbering, etc., are available. However, the projects created in the Demo Version are not usable in a licensed version. Furthermore, there is a transparent Demo logo on all drawing sheets.

In the Demo Version, only the symbols and parts required for the exercises and some general symbols are included. You can, however, access the symbols and parts provided on the Internet (at www.wscaduniverse.com) in the Demo Version as well.

In addition, we can offer you the following options to further familiarize yourself with the WSCAD SUITE.

- Retracing of the **SUITE DEMO** project in a new project
- Invocation of the online help with the complete description of WSCAD SUITE (via the menu option **Help** | **Overview** or as context-sensitive help via the F1 key).
- Participation in WSCAD seminars
- Beginner online training via the Internet at your workplace

Contact for questions

If you have any further questions about the WSCAD SUITE or want to request an online training, please contact our technical sales department.

WSCAD GmbH

Technical Sales

Dieselstraße 4

D-85232 Bergkirchen, Germany

Tel: +49 (0) 8131 / 3627 - 98

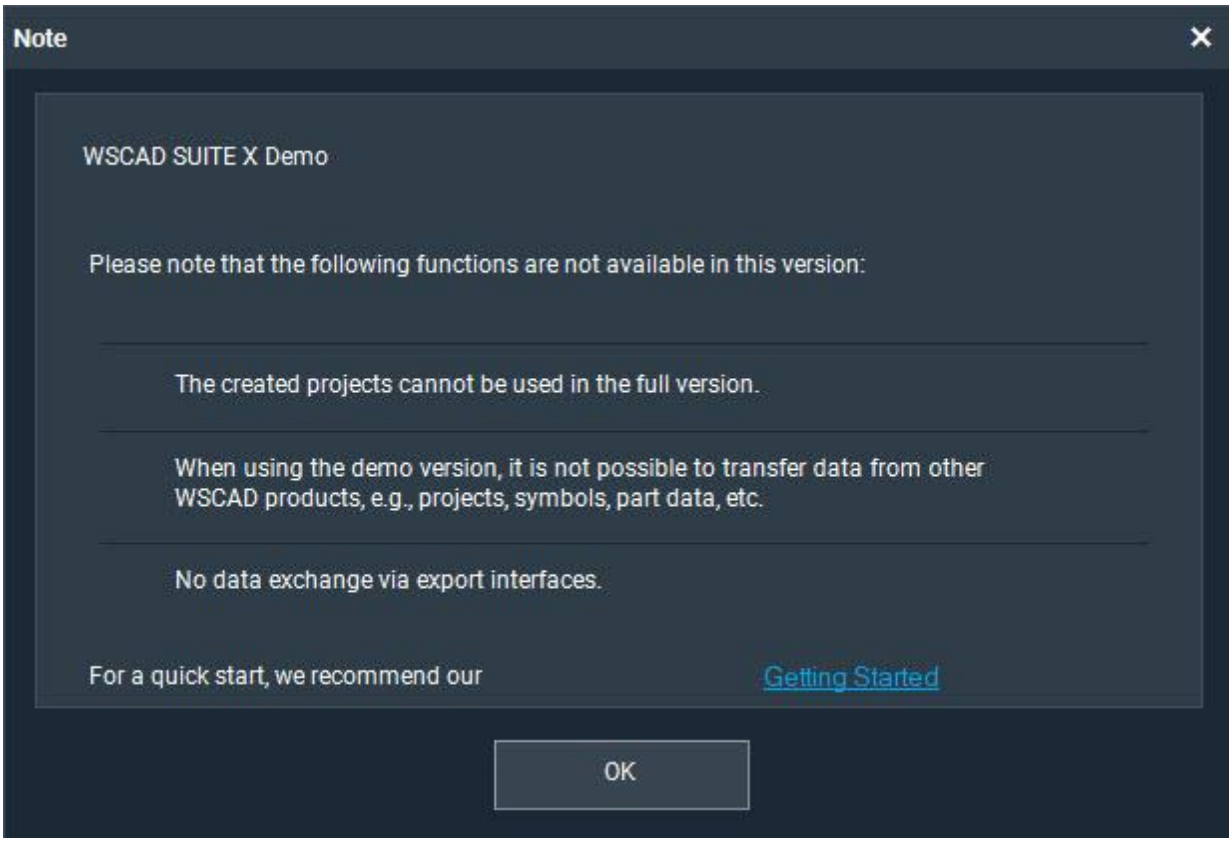
Fax: +49 (0) 8131 / 3627 - 52

sales@wscad.com

Starting the demo version

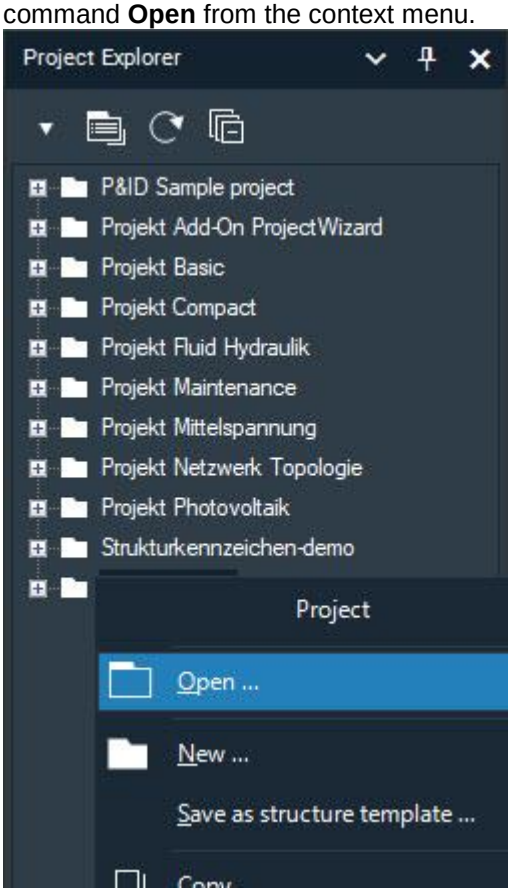
After installing the software (see the WSCAD SUITE X Installation and Licensing Guide), you will find a WSCAD link on your desktop.

1. Double-click on the WSCAD link.
2. Confirm the note on the limitations of the demo version with **OK**.



3. Enable the check box **I agree to the collection and use of my personal data** and click **OK**.

4. Right-click in the **Project Explorer** of the WSCAD SUITE on the project **SUITE DEMO** and select the command **Open** from the context menu.



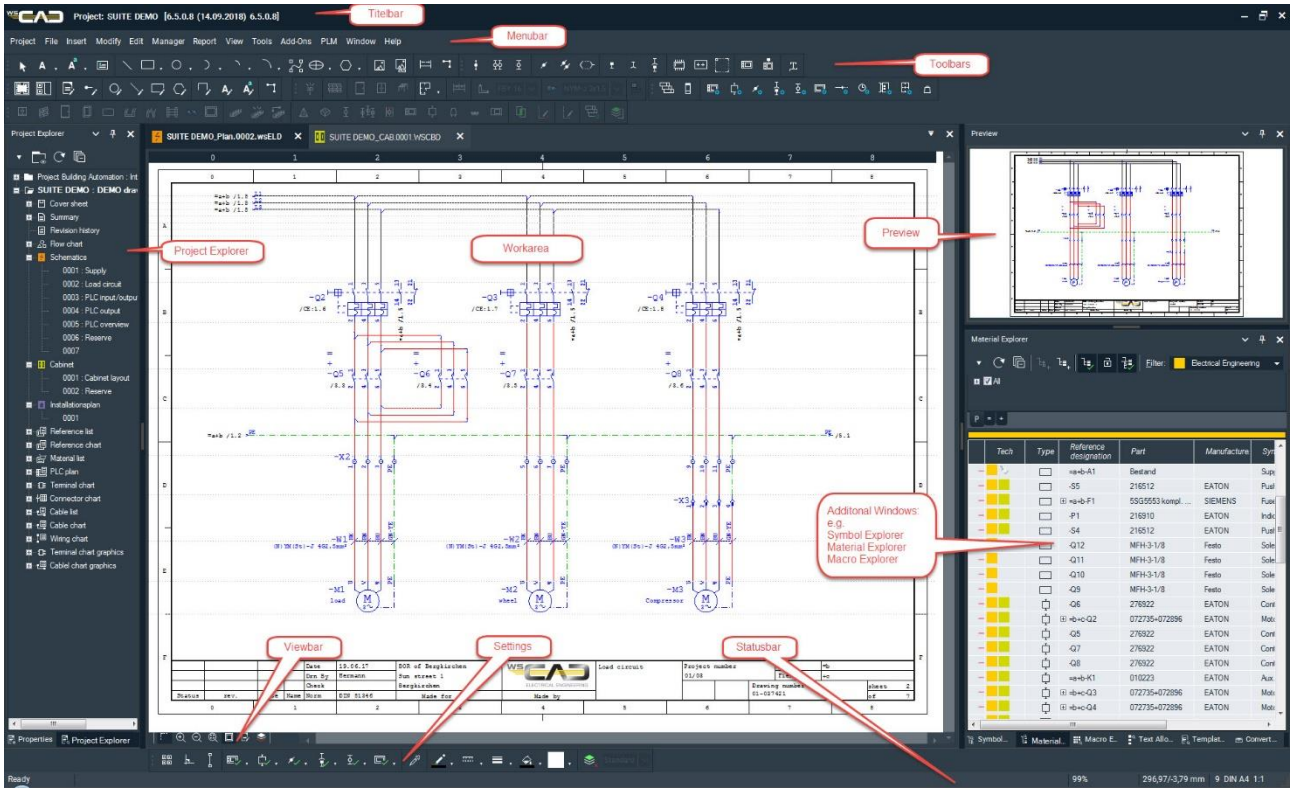
The **SUITE DEMO** prepared is a project that was prepared explicitly for this "Getting Started" Guide.

User interface

Work area

The work area can be divided and customized to specifically suit the way you work. By default, the work area is composed of the following components:

- Title bar
- Menu bar
- Toolbars
- Left additional window (with Project Explorer)
- Drawing area
- Preview window
- Right additional window (with Symbol Explorer)
- View bar
- Status bar



Set up the work area so that only the windows and toolbars that you need for your project are displayed. You can select a predefined work area for each technology via **View** | **Work areas** in the menu bar and then customize that work area to suit your needs. The additional windows on the left and right of the drawing area (Project Explorer, Symbol Explorer, ...) can be arranged freely and can also be placed as tabs on the side of the work area in the background if required. You can save your own work area for subsequent reuse.

In addition, you can invoke and close dialogs that are only used temporarily in the project (such as editors, for example) at any time. The invocation occurs via menu commands, context menus, keyboard functions or command bars.

Title bar

The title bar displays the open project.

Menu bar

The menu bar contains all the available menus and their menu commands.

Toolbars

The toolbars are composed of individual toolbars that can be displayed or hidden as desired.

Left additional window

By default, the left additional window contains the **Project Explorer** and the **Properties** of the open drawing sheet (project page). You can switch between the functions via tabs.

Drawing area

The drawing area contains one or more open drawing sheets (plans). The selection of the desired drawing sheet occurs via tabs. The contents of the drawing sheet can be moved vertically and horizontally with the scroll bars.

Preview window

The Preview window displays a preview of the element that is selected in the currently open Explorer (e.g., a symbol from the Symbol Explorer).

Right additional window

The additional window on the right by default contains the **Symbol Explorer**, the **Drawing Macro Explorer**, the **Material Explorer** and the **Templates Explorer**. You can switch between the functions via tabs.

View bar

The View bar is located below the drawing sheet. The toolbar buttons shown on the View bar depend on the open drawing sheet.

The View bar can contain the following toolbar buttons (from left to right):

- Show/hide path
- Zoom in on drawing sheet
- Zoom out of drawing sheet
- Display complete drawing sheet
- Show or hide drawing sheet frames
- Show or hide Quick Editor
- Background layer on/off
  This function prevents the background layer from being edited unintentionally.

Status bar

At the bottom of the work area is the status bar.

The status bar displays the following information (from left to right):

- Information about the current work step or path of the open project
- Zoom factor as a percentage
- Distance of the mouse pointer from the top left margin of the drawing sheet
- Path, sheet size, and scale

Additional windows

You can arrange additional windows (such as the Project Explorer, Symbol Explorer, etc.) freely in your work area. The additional windows are displayed via the menu command **View** | **Additional windows**. You can also display the additional windows using keyboard shortcuts (see the section **User interface** | **Keyboard shortcuts**) or via the **Standard** toolbar.

If required, the additional windows can be placed as tabs on the side of the work area in the background via the menu command **View** | **Additional windows** | **All automatically to background**.

You can reset your work area to the default layout via **View** | **Additional windows** | **Reset window layout**. However, all changes to the layout will be lost as a result.

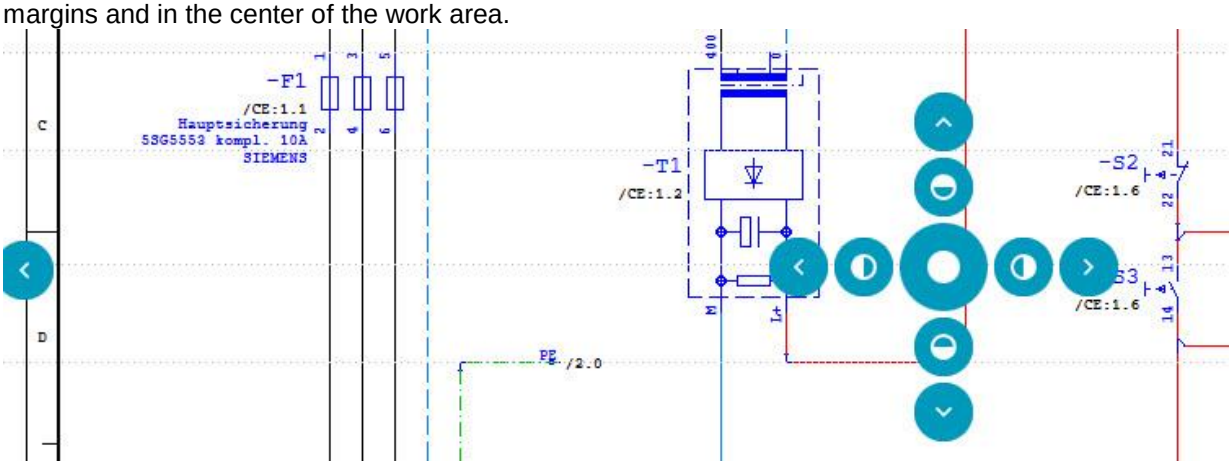
The title bar displays the name of the additional window. Clicking with the right mouse button in the title bar of the additional window opens a context menu with the following functions:

- **Restore**
  The additional window is restored to its previous state.
- **Maximize**
  The additional window is maximized and takes up the entire desktop.
- **Hide**
  The additional window is closed.
- **Floating**
  The additional window floats freely on the desktop.
- **Dock**
  If the additional window is floating on the desktop, it can be docked back at its old location.
- **Auto hide**
  The additional window is placed as a tab in the background and can be displayed again via the tab.
  This is symbolized by the horizontal pin  at the top right edge of the additional window.
  Clicking on the horizontal pin docks the additional window back to its old location. This is symbolized by the vertical pin  at the top right edge of the additional window.

You can change the height and width of both floating and docked additional windows as usual under the Windows operating systems.

Positioning additional windows

1. Click with the left mouse button on the title bar of the additional window and hold down the button.
2. Drag the additional window away from its current position. Guide symbols (arrows) appear at the margins and in the center of the work area.



3. Drag the mouse pointer to the desired guide symbol. A blue preview field shows how the additional window would be integrated into the work area.
4. Release the mouse button at the desired position. The window is docked.

If the Project Explorer is docked at the left edge of the work area, for example, and you want to dock it at the right edge instead, drag the Project Explorer window into the center of the work area and then release the mouse button on the right arrow of the guide symbol.

Note

Additional windows can also be merged with the guide symbol in the center. They are then treated as one window. The individual additional windows are accessible via tabs. They can be ungrouped again with the context item **Floating**.

Placing additional windows in the background

1. Click on the vertical pin symbol in the title bar of the additional window. The additional window is placed as a tab in the background at the margin of the work area.
2. To display the additional window, position the mouse pointer over the tab (horizontal pin symbol). The content can then be used.
3. The additional window is hidden again when you click anywhere else in the work area.
4. To dock the additional window again, click on the horizontal pin symbol in the title bar of the window.

Save Work area

1. Set up the additional windows according to your wishes.
2. In the menu bar, click on **View** | **Work area** | **Save current state**.
3. Assign a name for your new work area in the **Export layout** dialog and click **OK**. Your new work area will subsequently be displayed under **View** | **Work area**.

Note

You can save different work areas for different projects.

Toolbars

The toolbars (also called command bars) contain symbols via which important functions can be quickly and conveniently performed. Symbols that are grayed out are not currently available.

The toolbars can be shown or hidden as desired. Depending on which version and which add-ons are installed, different toolbars are available.

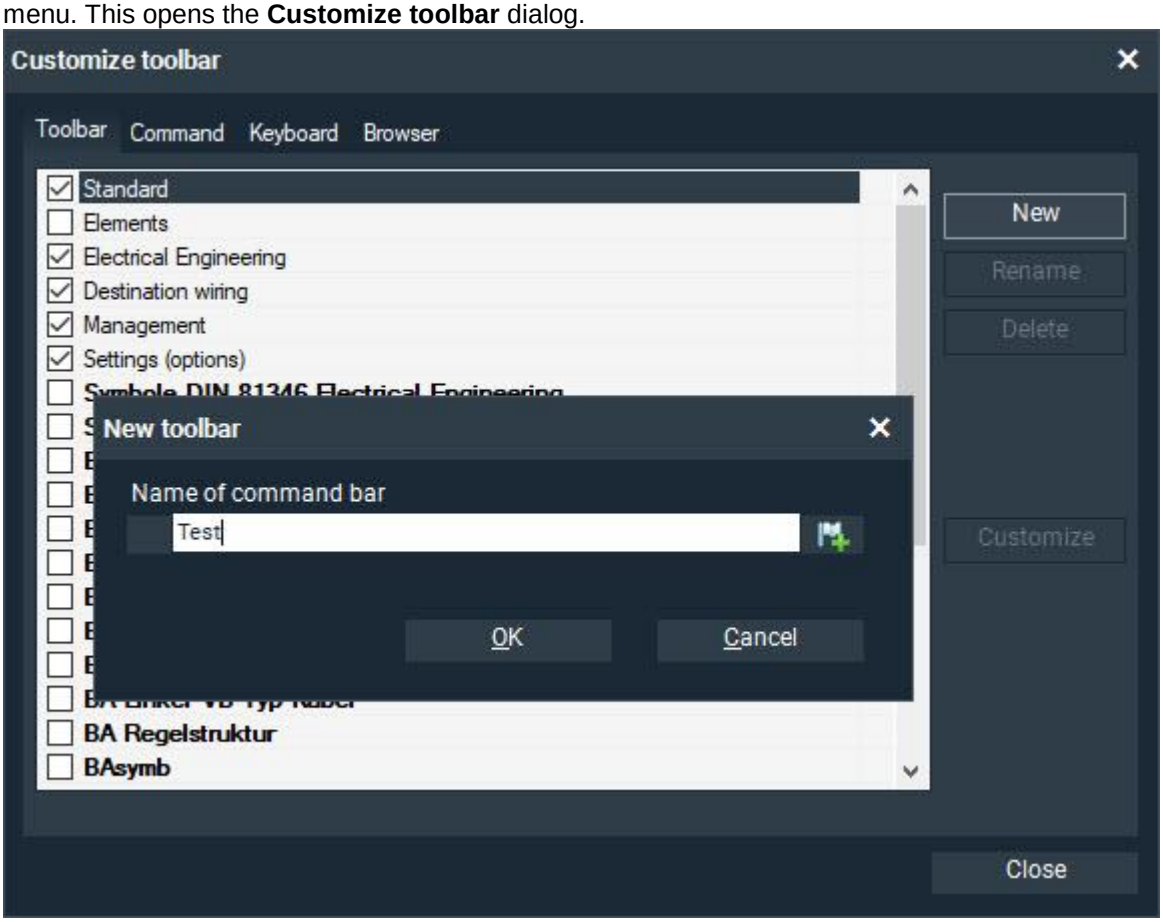
You can customize the existing toolbars or create new ones. Some predefined toolbars cannot be changed.

Show/hide toolbars

1. Click with the right mouse button on a toolbar and select the bar to be hidden or shown from the context menu. The displayed bars are indicated with a check mark.

Creating or editing a toolbar

1. Click with the right mouse button on a toolbar and select the command **Customize** from the context menu. This opens the **Customize toolbar** dialog.



2. If you want to create a new toolbar, click on **New** and assign a name for the new toolbar. Click **OK**. The new toolbar appears in the list.
3. Select the toolbar and click **Customize**.
4. Select a **Category** and take over the desired commands from the right window to the left window by double-clicking. You can add menu commands, symbols and macros to the toolbar.
   Note: Newly created toolbars or changes to existing toolbars are immediately visible in the work area.
5. Finally, click **Close**.

Moving toolbars

A toolbar can be moved within the toolbars. It can, however, also be floated on the desktop or arranged on the side or below the drawing area.

1. Click with the left mouse button on the dotted line to the left of the toolbar and hold down the mouse button.
2. Drag the toolbar to the desired location and release the left mouse button.

Transferring symbols directly

You can transfer symbols from the Symbol Explorer directly into the toolbars.

1. Click on the symbol with the right mouse button in the Symbol Explorer.
2. Select the command **Add to toolbar** from the context menu and then select the desired toolbar.

Mouse functions

In a graphical user interface, you control the cursor with the mouse (e.g., when inserting symbols) and perform functions with the mouse buttons. The left and the right mouse buttons have different functions.

Left mouse button (LM)

Function	Description
Single click in the menu bar	Open menu or execute menu command
Single click in the toolbar	Execute symbol function
Single click in the drawing area on element	Select element
Single click + Ctrl key in the drawing area on elements	Select multiple elements
Single click in the context menu	Execute the menu command
Double-click in the Project Explorer	Open tree structure or open project page in the drawing area
Double-click in the drawing area on element	Open Properties dialog

Right mouse button (RM)

Function	Description
Single click in menu bar or toolbar	Show/hide toolbars
Single click in the drawing area on element	Open the context menu of the element
Single click in the drawing area on background	Open the context menu of the drawing sheet

Mouse wheel

In the drawing area, the content can be enlarged or reduced with the mouse wheel (zoom function). This function can be enabled (default) or disabled via **Tools** | **Settings (options)** | **General** | **Zoom via mouse wheel**.

If the content of the drawing area is zoomed, the content can be moved by holding down the mouse wheel button (panning function). The wheel must not be turned when doing this.

Cursor

The cursor is controlled with the mouse. The appearance of the cursor can be changed via **Tools** | **Settings (options)** | **View** | **Cursor**. By default, the lines view is set, since the snap functionality is also available in this case.

If the default windows cursor is set, the cursor itself does not move in the snap grid, but the individual elements do.

Multiple selection

If you draw a frame over multiple elements in the drawing area by holding down the left mouse button, on releasing the mouse button, all elements in the frame will be marked in blue. The whole selection can now be collectively edited (e.g., via the context menu or the toolbars).

Important!

- When drawing a frame from top left to the bottom right, only the framed elements are marked.
- When drawing a frame from bottom right to the top left, in addition all connection lines of the framed elements are marked too.

Keyboard shortcuts

Keyboard shortcuts provide quick access to frequently used functions.

With the keys of your keyboard, you can:

- invoke important functions using the function keys F1-F12 (snap, zoom, online help, ...)
- invoke important functions using keyboard shortcuts (Ctrl+key, Alt+key, ...)
- expand main menus using the Alt key plus the underlined letter. Subsequent navigation is achieved via the arrow keys.
- control the cursor using the arrow keys.

You can use the F1 key to invoke the integrated help. You will find an overview of all the keyboard shortcuts there in the table of contents under **User interface** | **Keyboard shortcuts**.

Arrow keys

The four arrow keys (up, left, down and right) can be used to position the cursor horizontally or vertically. When snap is activated, the cursor moves within the set snap grid. When you additionally press the CTRL key, the cursor increment is increased considerably in both cases.

Overview of the main keyboard shortcuts

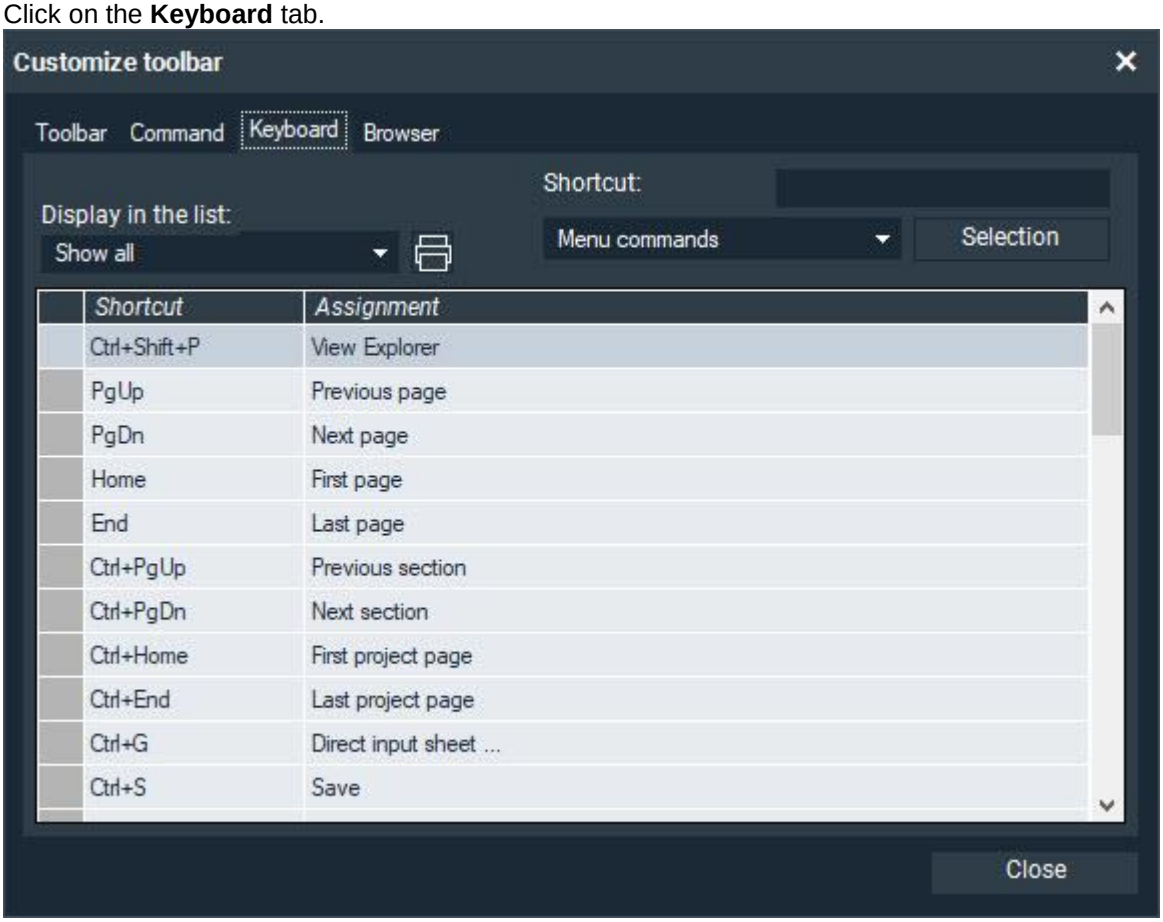
Key	Function
F1	Call the online help.
F2	Zoom in.
F3	Zoom out.
F4	Adjust drawing sheet to window.
F5	Snap on/off.
Shift + F5	Disables the snap area completely, which means you can move the cursor between pixels (the smallest unit). This only makes sense with a high zoom setting. The grid continues to be displayed. You can undo this function only by pressing Shift+F5 again.
F6	Orthogonal drawing mode on/off.
F7	Grid on/off.
F9	Absolute/relative cursor zero point (with graphic symbol)
F10	Auto connect on/off: On: Connections are set when symbols are positioned. Off: No connections are set. Indicated with red circles via View/Lock on Symbol/Line pins.
F11	Rebuild image (rebuilds without saving).
F12	Show/hide frames.
Alt + F4	Exit WSCAD SUITE.
Alt + F6	Next additional window.
Shift + Alt + F6	Previous additional window.
Shift + Esc	Close the open additional window.
Alt+Return	Open the Properties dialog box.

Key	Function
Alt + menu letter (hotkey)	Select a menu.
Return	Close an entry.
Esc	Cancel an action.
Del	Delete.
Home	Jump to first position in active window (project, page, text, etc.).
End	Jump to end position in active window (project, page, text, etc.).
Page Up ↑ / Page Down ↓	Scroll through the drawing record.
Ctrl + left mouse button	When holding down the Ctrl key - with every left-click, additional elements can be selected.
Ctrl + Cursor	If you hold down the Ctrl key and move the cursor over a symbol, the stored properties (e.g., part, manufacturer, etc.) are displayed.
Ctrl + Page Up↑	Jump to the previous section.
Ctrl + Page Down ↓	Jump to the next section.
Ctrl + Home	Jump to the first page of the open project.
Ctrl + End	Jump to the last page of the open project.
Ctrl + minus key	1. Insert horizontal auxiliary line in drawing sheet at cursor position (not possible via the number pad). 2. Delete horizontal auxiliary line in drawing sheet at cursor position (not possible via the number pad).
Ctrl + <	1. Insert vertical auxiliary line in drawing sheet at cursor position. 2. Delete vertical auxiliary line in drawing sheet at cursor position.
Ctrl+F4	Close the active window.
Ctrl+F6	Switch to the next window.
Ctrl+Tab	Display all open windows.
Ctrl + Space	Invoke the last menu or command again.
Ctrl + c	Copy a selected area to the clipboard.
Ctrl + d	The selected elements are copied directly and can be repositioned.
Ctrl + v	Paste the content from the clipboard.
Ctrl + x	Cut a selected area to the clipboard.
Ctrl + s	Save file.
Ctrl + Shift + u	Save file as.
Ctrl + Shift + s	Save all.
Ctrl + y	Redo last command.
Ctrl + z	Undo last command.

Defining keyboard shortcuts

You can place menu commands, individual symbols or macros (combination of multiple elements) as well as the execution of external files on keyboard shortcuts.

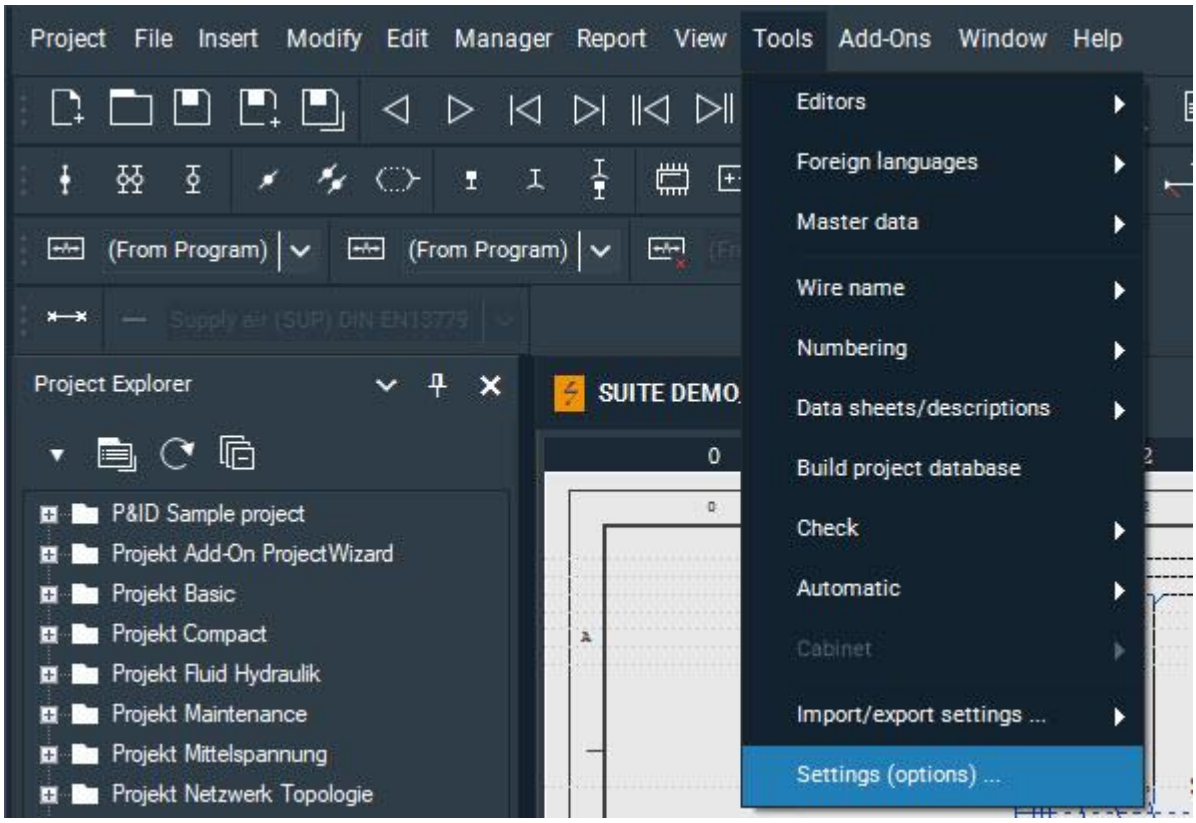
1. Click with the right mouse button on a toolbar and select the command **Customize** from the context menu.
   The **Customize toolbar** dialog opens.
2. Click on the **Keyboard** tab.



3. Click in the **Shortcut** field and enter the desired key combination via the keyboard.
4. Select the function category (e.g., menu commands or load symbol) in the drop-down list on the right and click on **Selection**. If the key combination is already assigned, you will be notified accordingly.
   The dialog of the selected category opens
5. Select the function and confirm with **OK**. The keyboard shortcut is displayed as the last entry in the list. This requires the list to be set to **Show all**.
6. Finally, click **Close**.

Settings

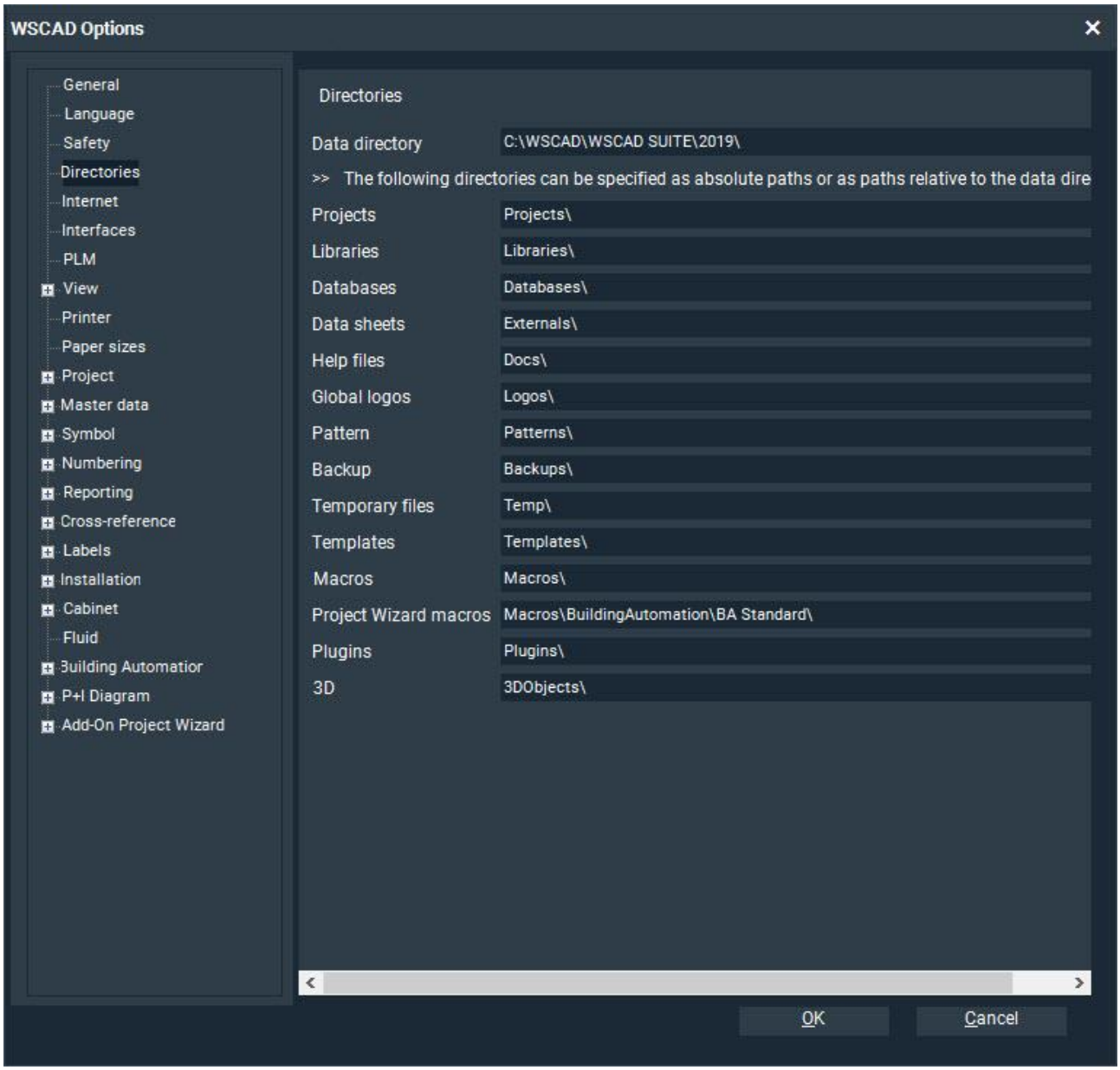
You can configure the WSCAD SUITE according to your requirements via the menu item **Extras** | **Settings (options)**.



In the **WSCAD options** dialog, a number of different settings are offered via a menu tree. The most important settings are explained briefly below. A detailed description can be found in the online help in the topic **Settings (options)**.

Directories

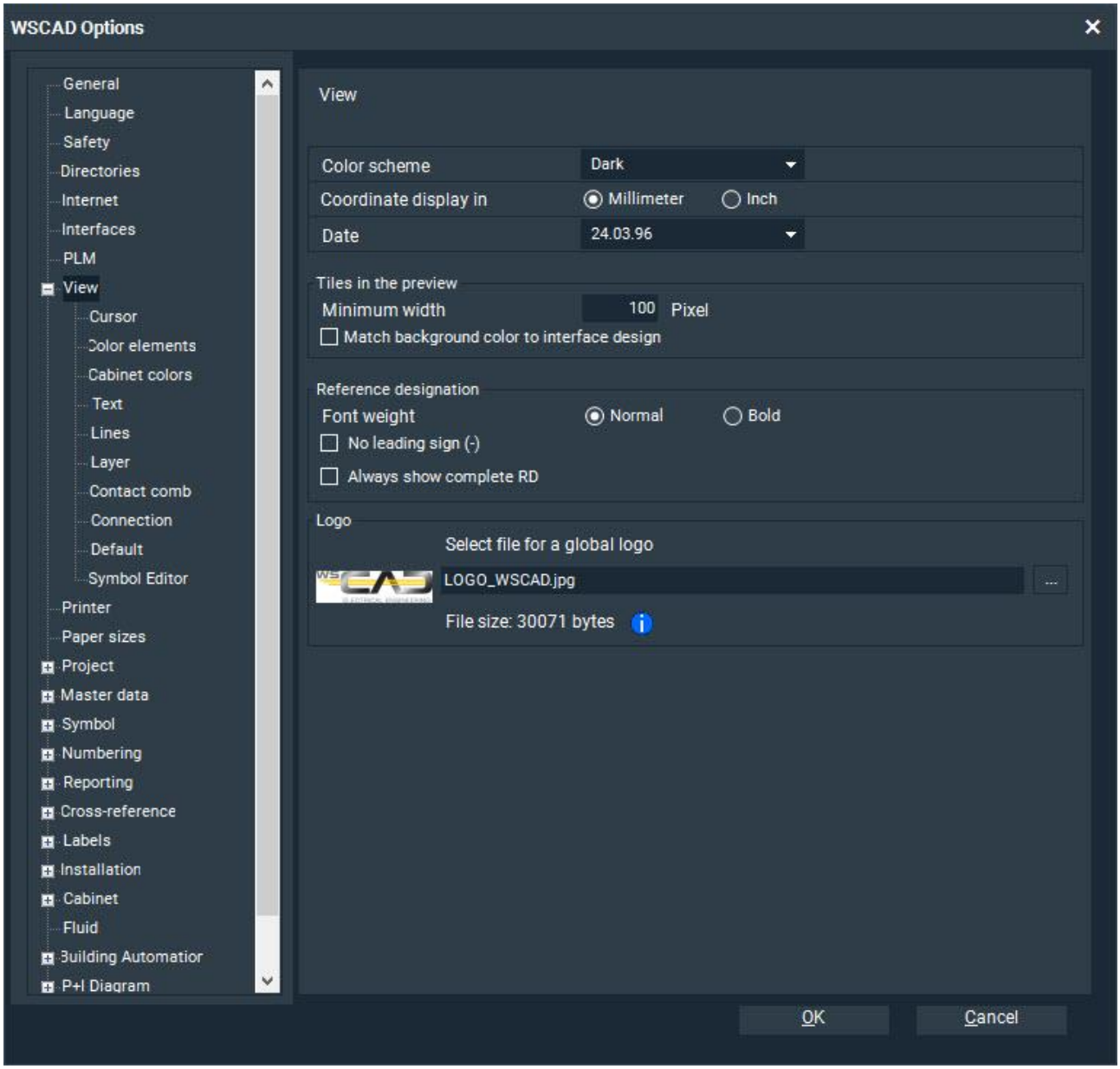
In the **Directories** menu, you can specify the directories for different WSCAD program files. By default, the directories are created in the installation path (e.g., `C:\\WSCAD\\WSCADSUITE\\2019`). You can, however, specify other directories relative to the installation directory or even as absolute paths if you want parts or libraries to be stored at a central location on the network (server PC) so that they can be accessed from any PC.



View

In the **View** menu, you can define settings for the appearance of the work area.

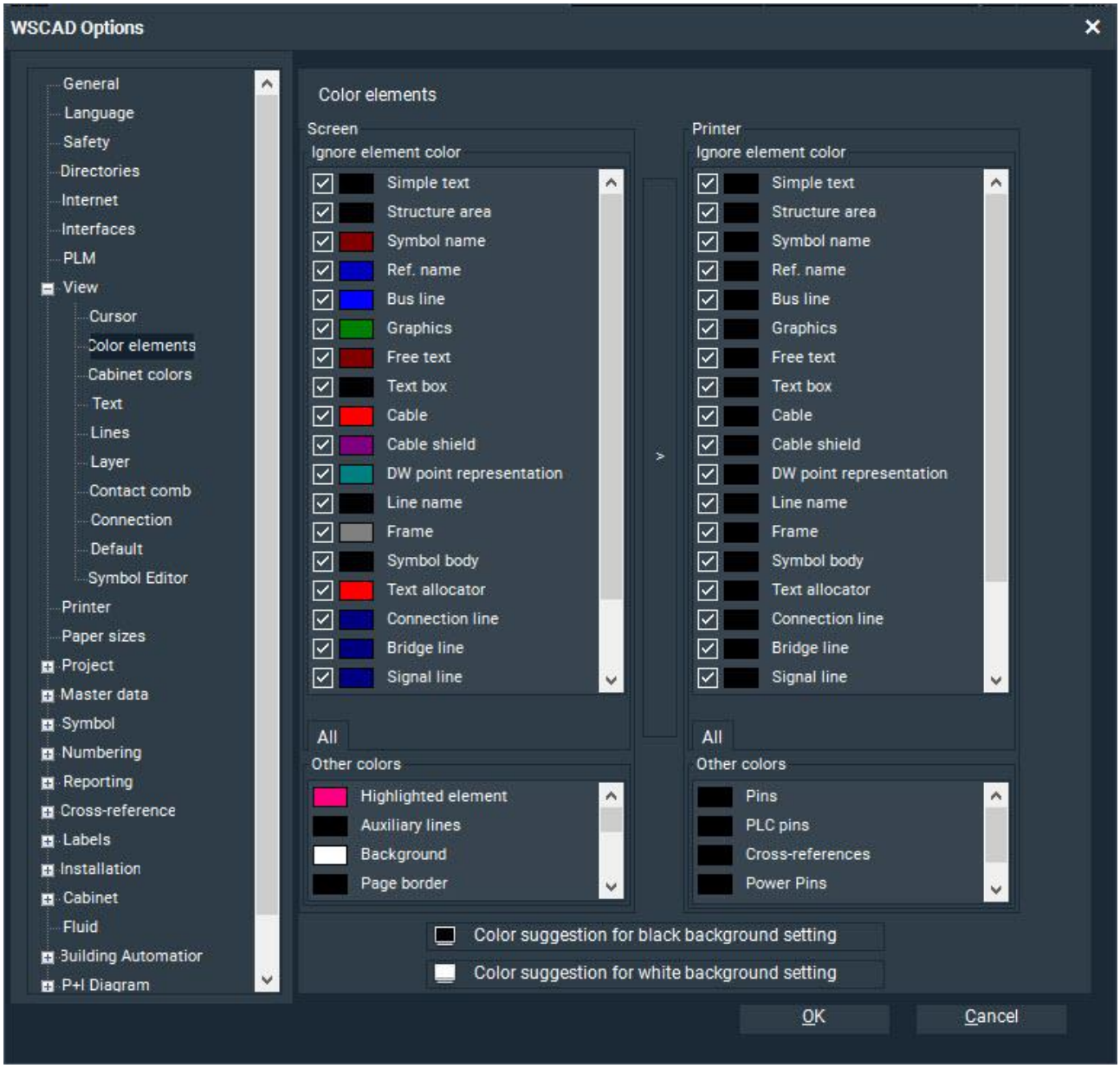
- Settings made via the **Interface design** drop-down list affect the complete look and feel of the entire application.
- Under **Select file for a global logo**, you can replace the standard WSCAD logo with your own company logo. This logo will be displayed in the drawing frames globally for all projects. If another logo is chosen for a single project, this logo will be overwritten for that project. The logo should not be larger than about 30kB in order to not reduce the speed of the page setup.



Color elements

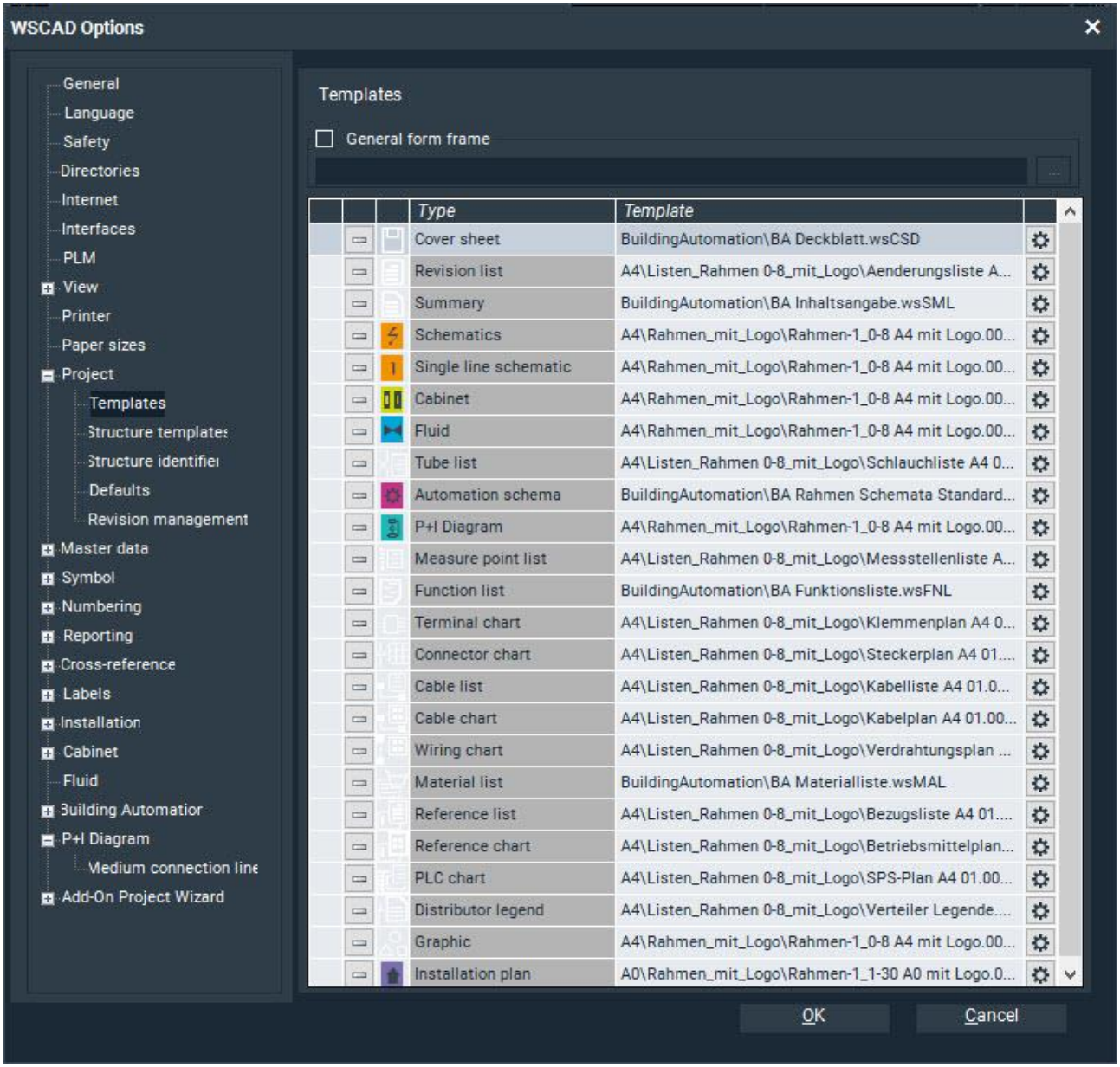
In the **View** | **Color elements** menu, you can set the colors of all elements (symbols, text, lines, ...) within the drawing sheet.

If a check mark is placed before an element, the element will be displayed with the color selected here. If no check mark is set, the element will be displayed with the color in which it was originally drawn.



Project

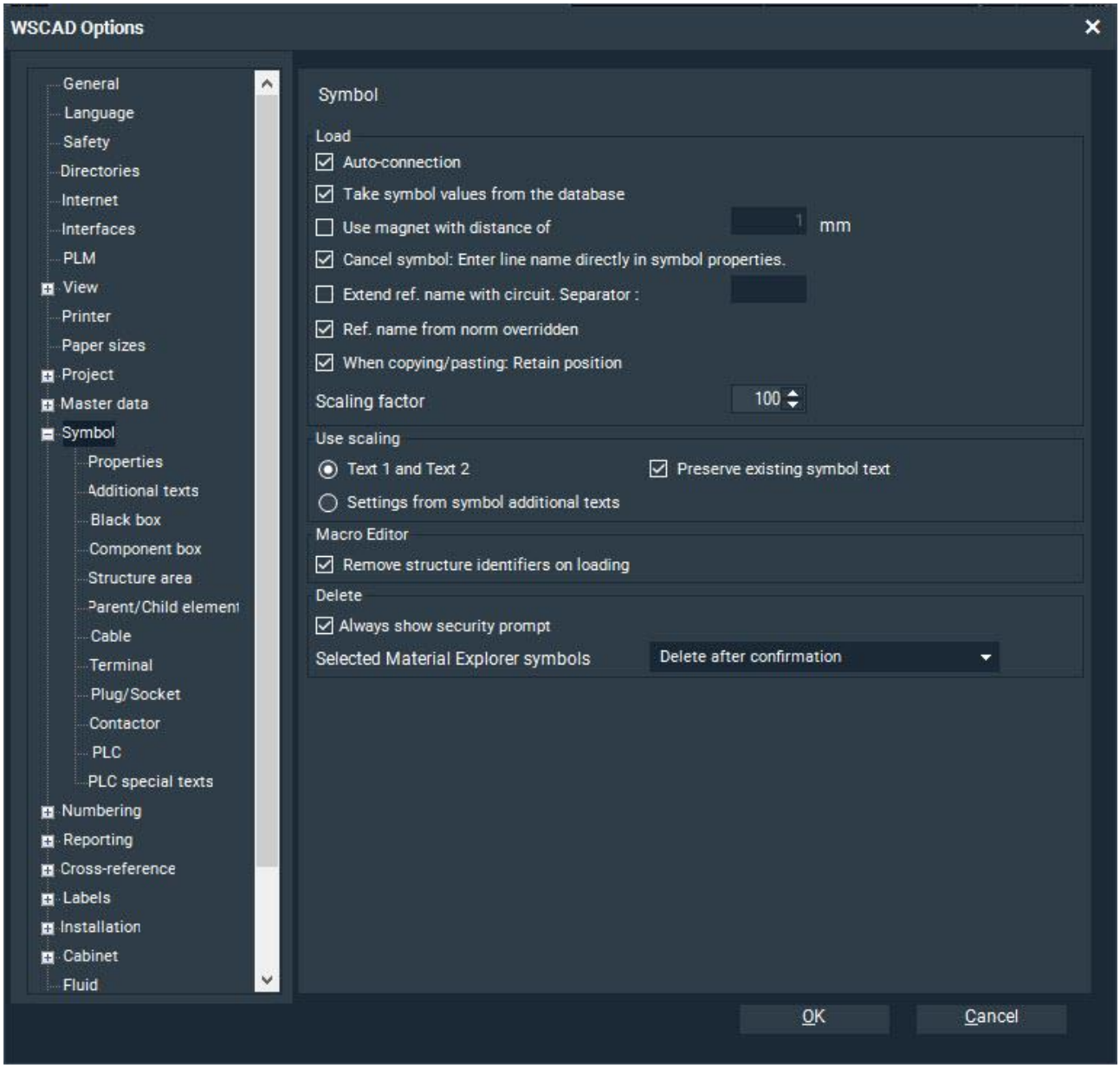
In the **Project** | **Templates** menu, you can choose which forms and drawing frames are to be used for drawing sheets and lists.



Symbol

In the **Symbol** menu, you can determine the behavior of symbols when they are inserted into drawing sheets.

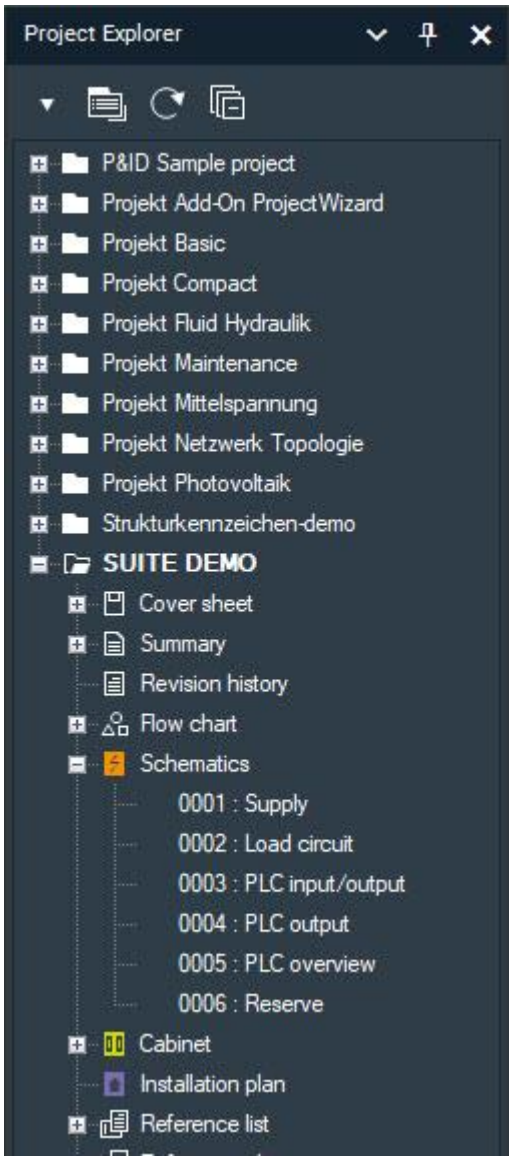
- It is best to leave the **Auto-connection** check box enabled so that a connection line is automatically drawn between two connection points (pins).
- The **Use magnet with distance of** check box is helpful when positioning symbols on connecting lines.



Project management

Project Explorer

With the **Project Explorer**, you can manage your projects. It provides all the important functions to create, edit, back up and restore projects. All menu commands for project management can be accessed via the down-arrow symbol button in the toolbar of the Project Explorer. Within the Project Explorer, you can open context menus with the right mouse button. The menu commands offered on the context menu depend on whether you have selected a project, a document folder, or a drawing sheet.



The Project Explorer shows you all projects. The opened project is highlighted in bold. You can view the document folders of a project by expanding the project with the plus symbol. You can likewise also expand or collapse the individual document folders via the plus or the minus symbol.

With the Project Explorer, you can, among other things:

- Open and close projects
- Create new projects
- Import projects
- Back up and restore projects
- Create, copy, and delete new pages and subpages within the document folders

Using the **Properties** tab in the Project Explorer, you can have the most important properties of the object selected in the Project Explorer displayed. The entries that appear in bold can be edited. All properties of the selected object can be accessed via the **Properties** command in the context menu.

In the Project Explorer, you will find the project SUITE DEMO, which serves as a model project. This project was prepared for this "Getting started" Guide. It includes a default database, as well as gaps in the schematic, which will be filled in the course of the tasks described here.

Note

Please bear in mind that the focus when creating schematics is on "How do I do something?" and does not correspond to the usual procedures followed during project planning!

Open project

1. Click with the right mouse button in the Project Explorer on the desired project and select the command **Open** from the context menu.

Close project

1. Click with the right mouse button in the Project Explorer on the open project and select the command **Close** from the context menu.

Inserting standard symbols

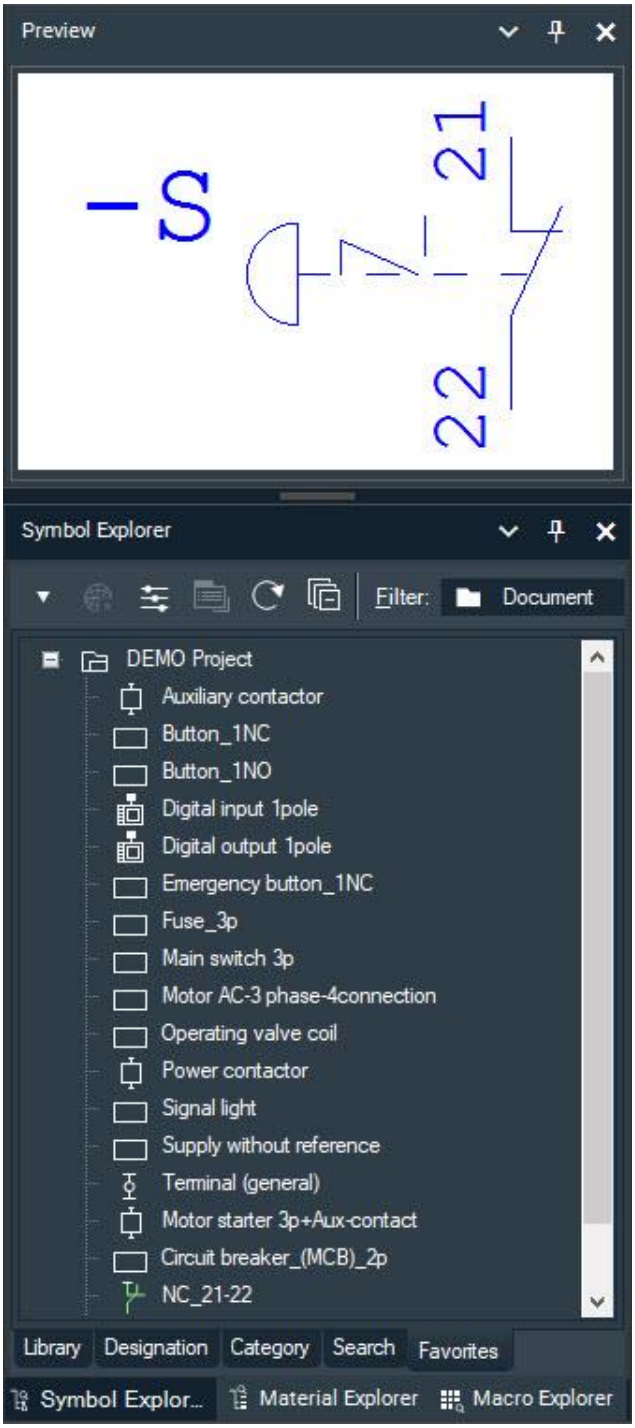
Push-button

In this section you learn how to insert the push-button – S1 (standard symbol) between the symbols – Q4 and – S2 in the "Supply" schematic.

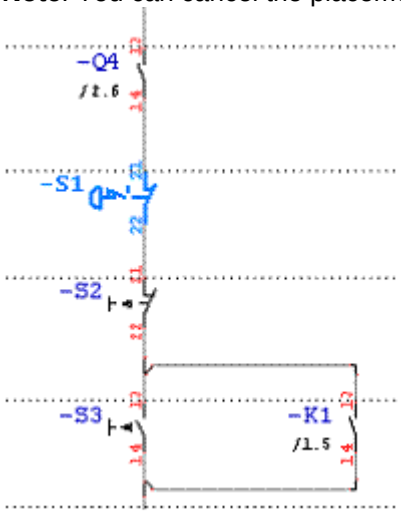
Inserting a push-button

Prerequisite: The project **SUITE DEMO** is open.

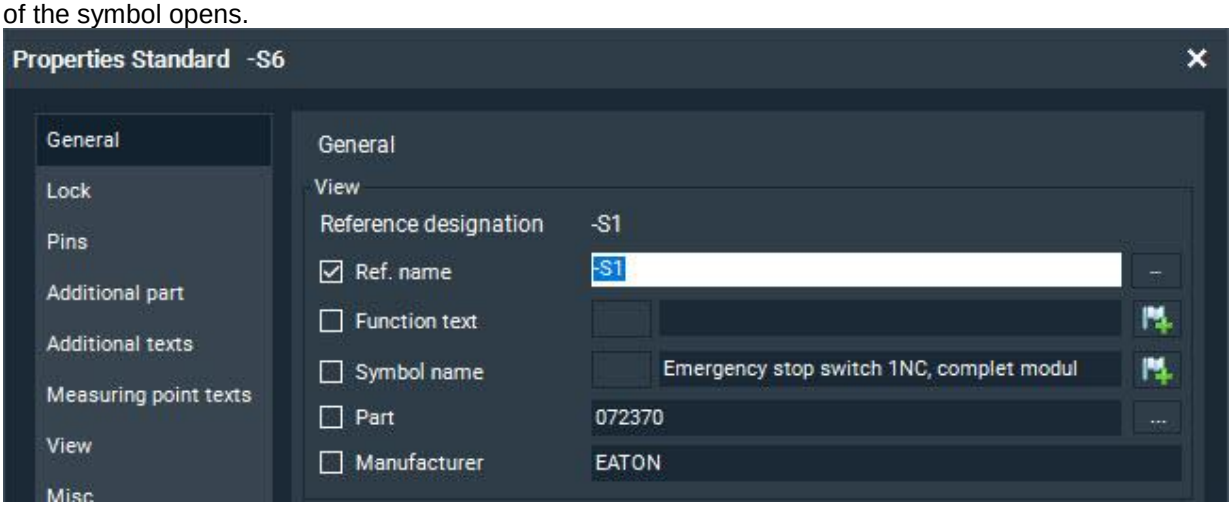
1. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0001: Supply**.
2. Click in the Symbol Explorer on the **Favorites** tab and open the folder **DEMO Project**. Click on the entry **Emergency button_1NC**. In the preview window, you will see the symbol of the emergency off button.



3. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
4. Move the mouse pointer to the desired position in the drawing sheet - in our case to the connecting line between the symbols `-Q4` and `-S2`
   Note. You can cancel the placement with the right mouse button.



5. Place the symbol by clicking the left mouse button or by pressing the Return key. The Properties dialog of the symbol opens.



6. Enter the reference name **-**`S1` for the symbol in the **Ref. name** field.
7. Click on the Customize button in the **Part** line. The Part Management opens.
8. In the part management, double-click in the **Part** column for the part `072370`. The part, symbol name and manufacturer are taken over to the Properties dialog of the symbol.
9. Confirm the Properties dialog with **OK**. The emergency off button is placed.
10. The symbol for the emergency off button still hangs on the mouse pointer in case a further emergency off button needs to be inserted. Click the right mouse button to cancel the insertion of a further symbol.

Note

You can also insert a symbol by clicking in the menu bar on **Insert | Symbol**, browsing in the libraries displayed in the Symbol Browser and then double-clicking on the desired symbol to transfer it to the drawing sheet.

Inserting terminals and terminal strips

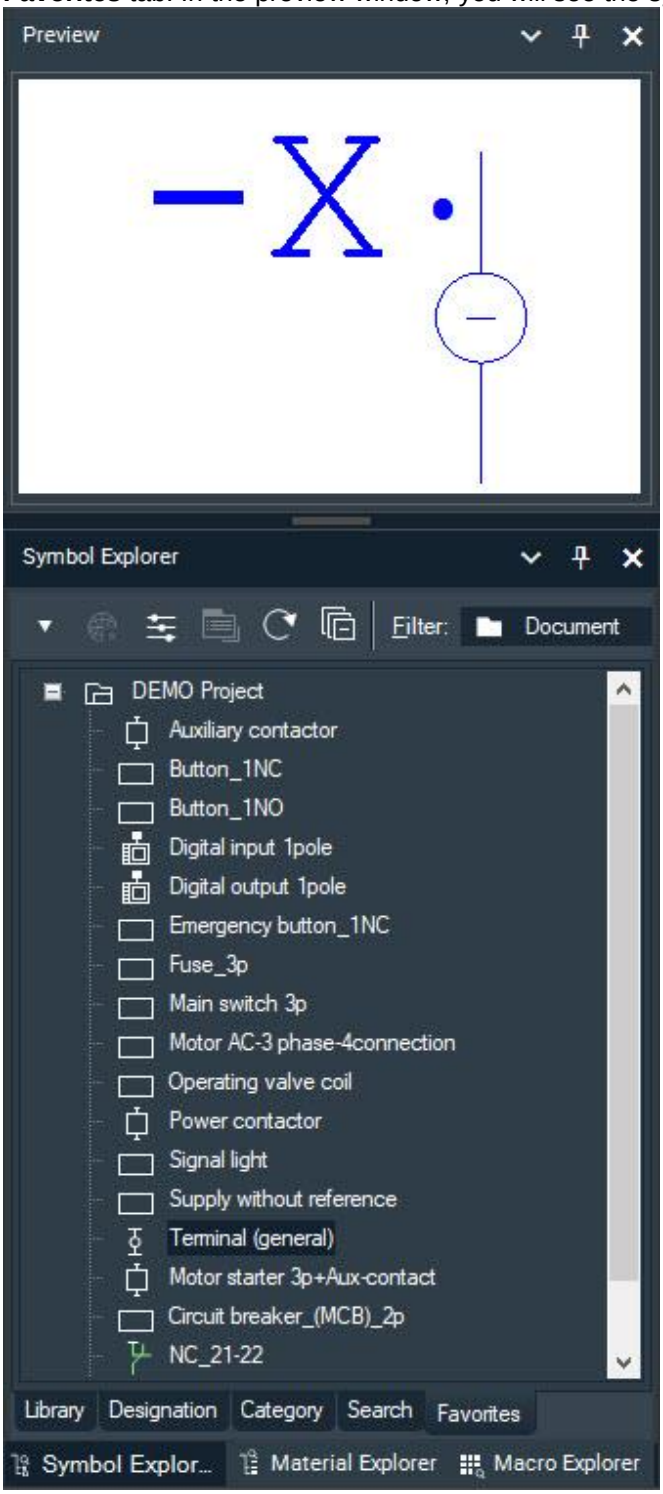
Terminals, individual

In this section you will learn how to insert individual `-X1` terminals above the `-A1` symbol in the "Supply" schematic.

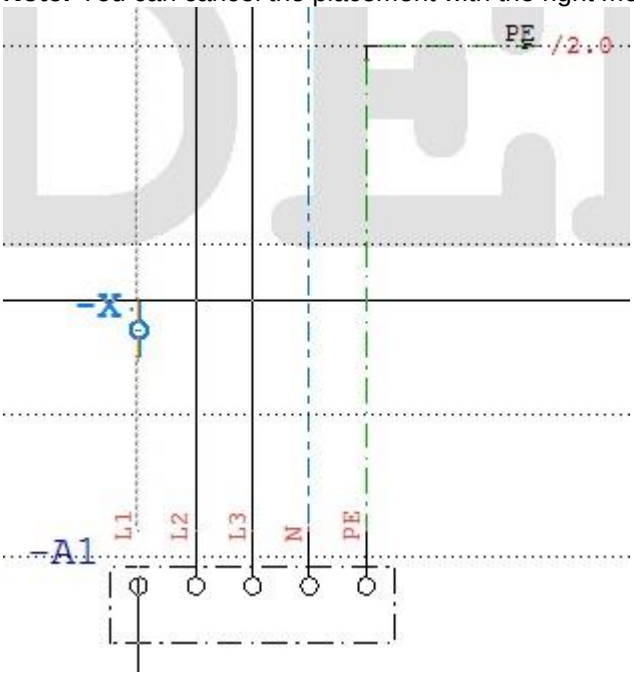
Inserting terminals individually

Prerequisite: The drawing sheet **Schematic** | **0001: Supply** is open.

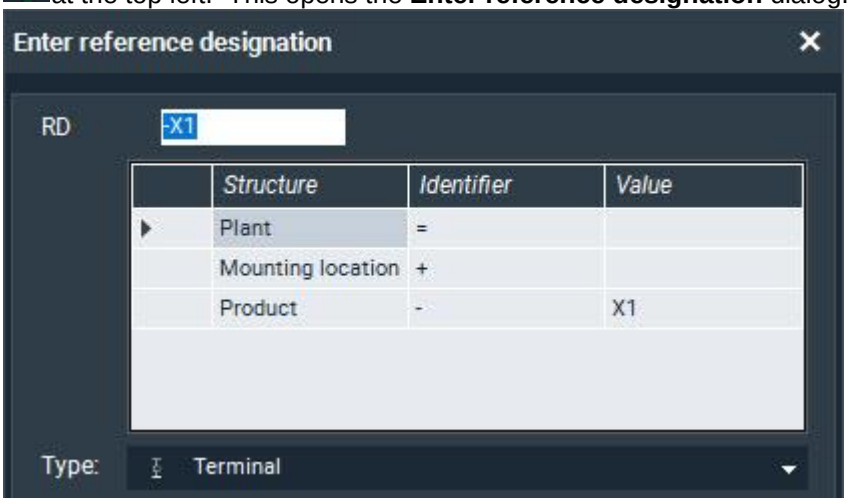
1. Click in the Symbol Explorer on the entry **Terminal (general)** in the folder **DEMO Project** on the **Favorites** tab. In the preview window, you will see the symbol of the terminal.



2. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
3. Move the mouse pointer to the desired position in the drawing sheet - in our case to the left line `L1` above the symbol `-A1`.
   Note: You can cancel the placement with the right mouse button.

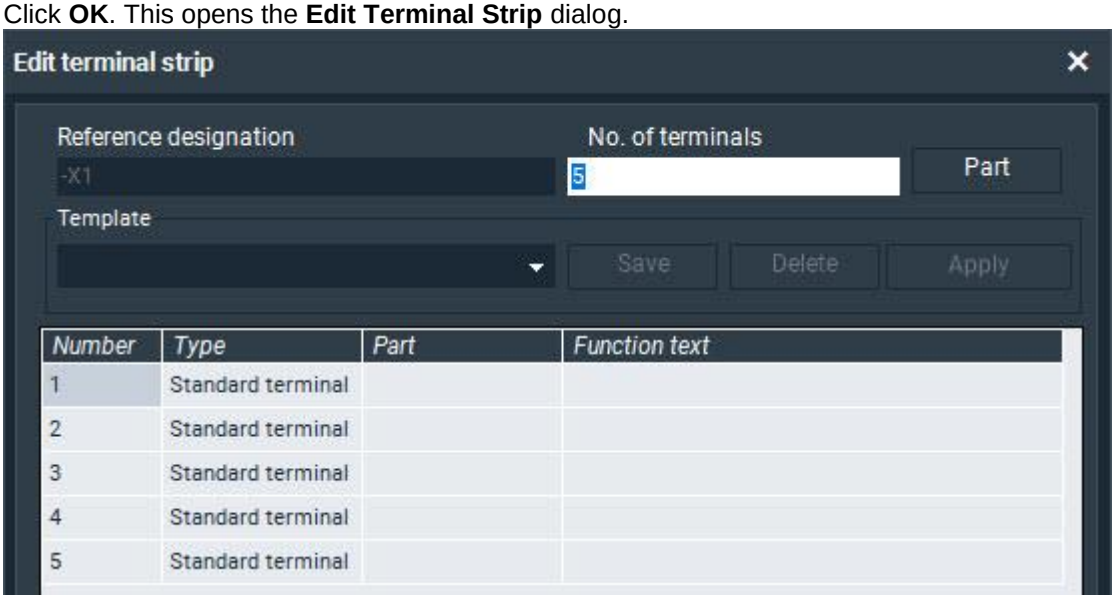


4. Place the symbol by clicking the left mouse button or by pressing the Return key. The Terminal Manager opens.
5. To define a new terminal strip, click in the toolbar of the Terminal Manager on the symbol button **New**  at the top left. This opens the **Enter reference designation** dialog.

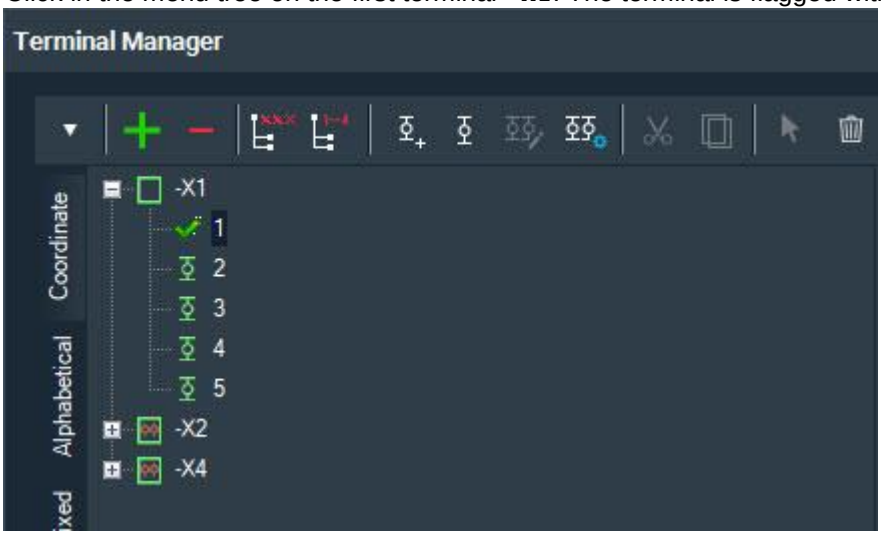


6. Enter the reference name `-X1` for the terminal strip in the **RD** field.

7. Click **OK**. This opens the **Edit Terminal Strip** dialog.

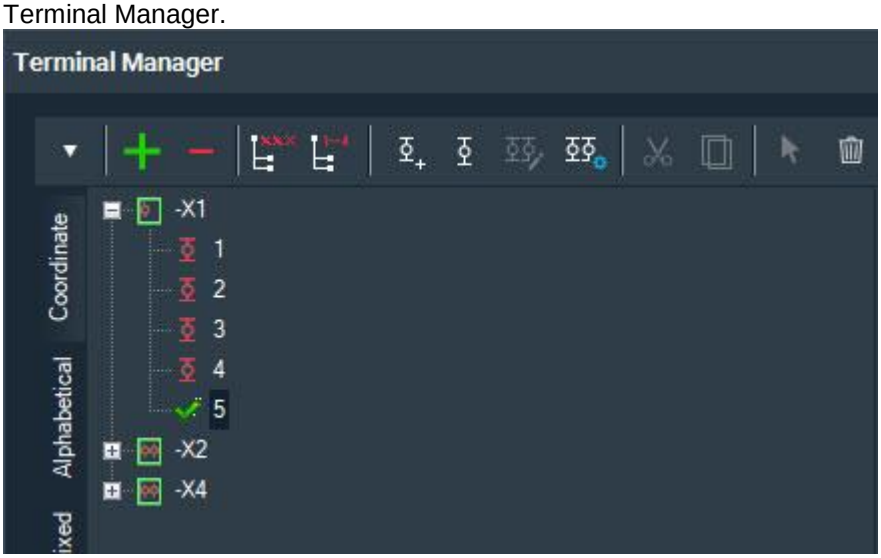


8. Enter the value `5` in the **No. of terminals** field. This creates 5 terminals for the terminal strip.
   Note: Assigning a part is not relevant to this exercise, but can be done via the **Part** button or by double-clicking in the **Part** column.
9. Click **OK**. The terminal strip `-X1` with the five terminals is now placed in the menu tree of the Terminal Manager.
10. Click in the menu tree on the first terminal `-X1`. The terminal is flagged with a green check mark.

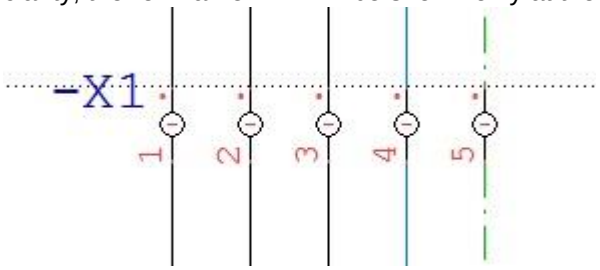


11. Click **OK**. The terminal is placed on the selected line.

12. Place the remaining four terminals by placing them via the Symbol Explorer and marking them in the Terminal Manager.



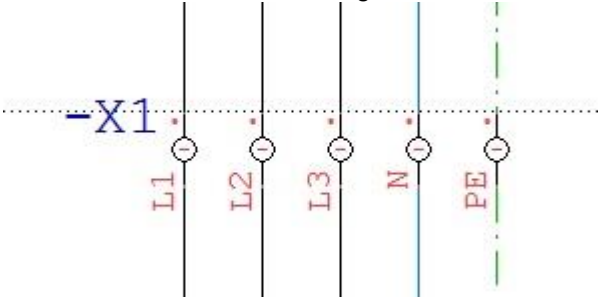
13. Finally, the terminal strip with the terminals 1 – 5 are placed on the connection lines. For the sake of clarity, the ref. name `-X1` will be shown only at the first terminal.



14. Start the Terminal Manager again. To do this, you have the following options:
    - Click in the menu bar on **Manager | Terminals.**
    - Click with the right mouse button on one of the five terminals and select the command **Manager** from the context menu.
15. Click in the menu tree of the Terminal Manager on the terminal strip `-X1` and then click on the **Browser** tab.
16. Double-click in the fields of the **Pin number** column and enter the designation of the connection lines (`L1`, `L2`, `L3`, `N` and `PE`) there. Changed fields are highlighted in green.

Index	Status	RD	Pin number	Part	Sheet/path	Part name	Functio
1		-X1	L1		/1.1	Terminal (general)	
2		-X1	L2		/1.1	Terminal (general)	
3		-X1	L3		/1.1	Terminal (general)	
4		-X1	N		/1.1	Terminal (general)	
5		-X1	PE		/1.1	Terminal (general)	

17. Click **OK**. The Terminal Manager is closed. The terminal strip should look like the following:



Terminal strip

In this section you will learn how to insert the terminal strip −X1 above the −A1 symbol in the "Supply" schematic.

First, please first delete the just created terminal strip − X1.

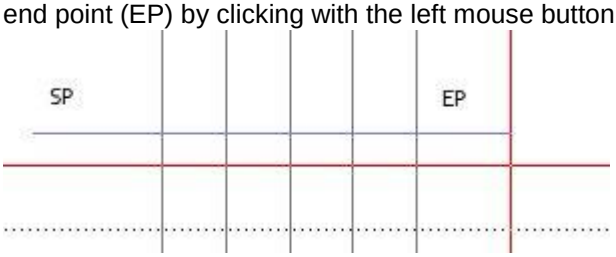
Deleting a terminal strip

Prerequisite: The drawing sheet **Schematic** | **0001: Supply** is open.

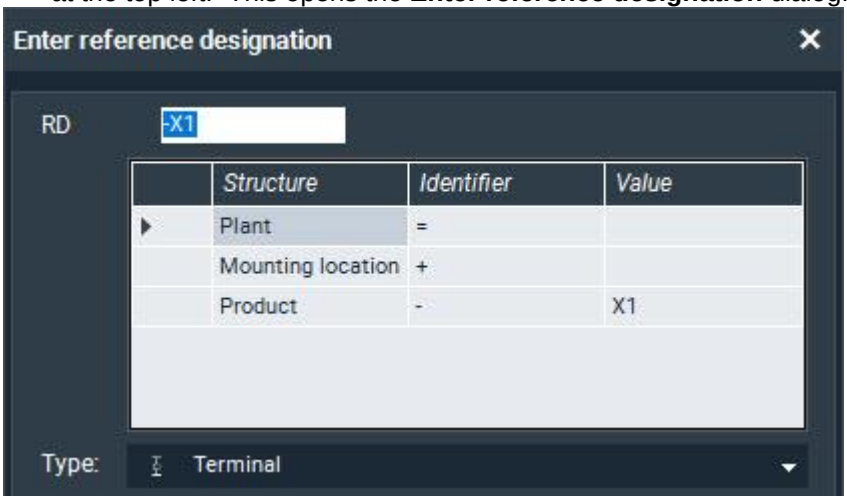
1. To delete the terminal strip, you have the following options:
   - Open the Terminal Manager via the main menu **Manager** | **Terminals**. Click on the terminal strip −X1 and then on the symbol button **Delete**  .
   - Draw a frame from top left to the bottom right over the terminal strip in the schematic using the left mouse button and release the mouse button. The terminal strip is marked in blue. Click with the right mouse button on the marked terminal strip and select the command **Selection |** Delete from the context menu. Confirm with **Yes**.
2. The message "Delete terminals! Keep vacant terminals?" appears. Click **No**.

Inserting a terminal strip

1. To insert a terminal strip, you have the following options:
   - Click on **Insert** | **Electrical Engineering** | **Terminal strip** in the menu bar. The cursor color changes to red.
   - In the Symbol Explorer, on the **Favorites** tab, in the **DEMO project** folder, click with the right mouse button on the entry **Terminal (general)** and select the command **Place** | **To connections** from the context menu. The cursor color changes to red.
2. Set a start point (SP) by clicking with the left mouse button on the left next to the line L1 and above the symbol −A1. Draw a horizontal line across the five existing connections (L1, L2, L3, N and PE). Set an end point (EP) by clicking with the left mouse button again on the right next to the line PE.

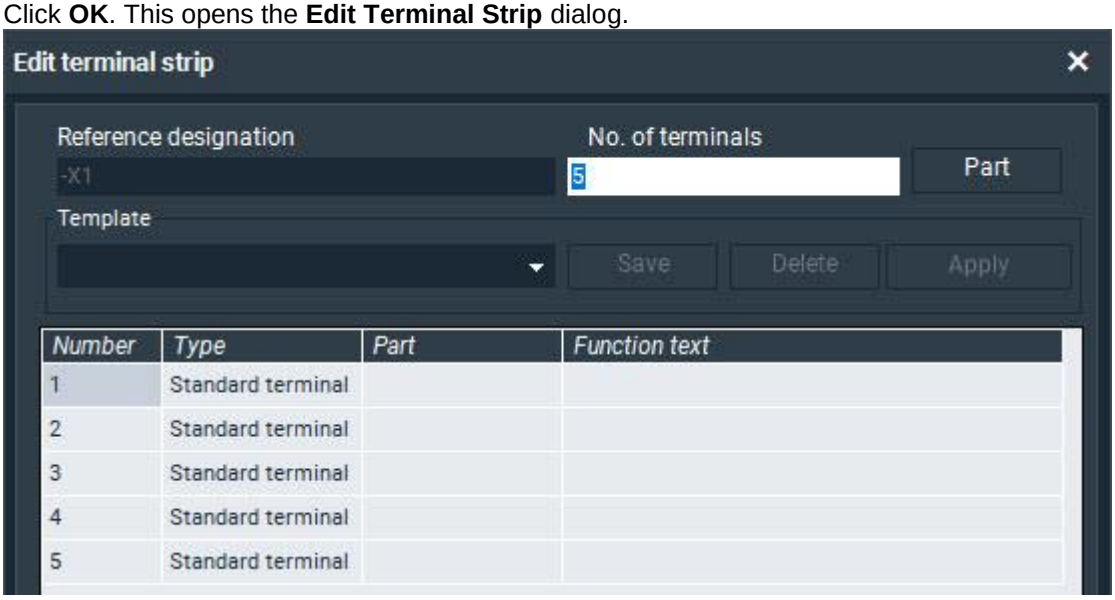


3. The Terminal Manager opens.
4. To define a new terminal strip, click in the toolbar of the Terminal Manager on the symbol button **New**  at the top left. This opens the **Enter reference designation** dialog.

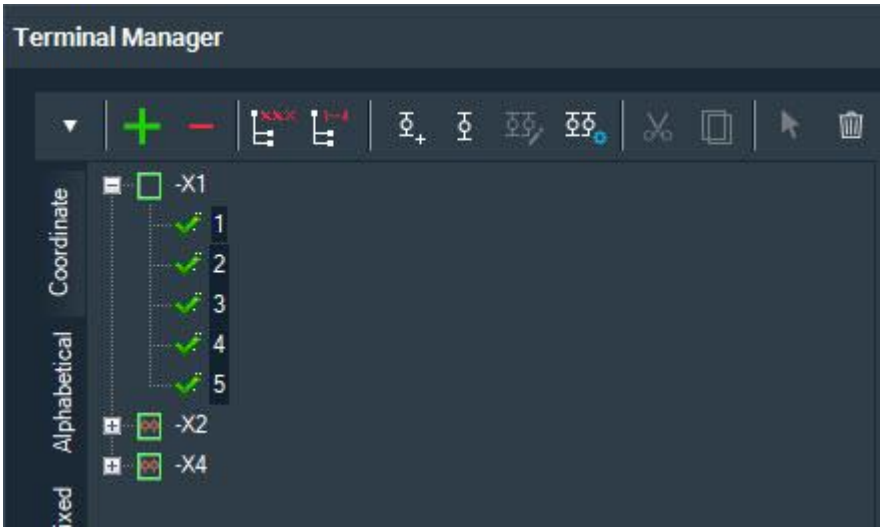


5. Enter the reference name −X1 for the terminal strip in the **RD** field.

6. Click **OK**. This opens the **Edit Terminal Strip** dialog.



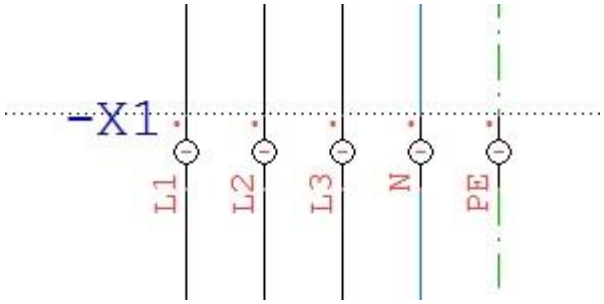
7. Verify that the value `5` is entered in the **No. of terminals** field. This creates 5 terminals for the terminal strip.
   Note: Assigning a part is not relevant to this exercise, but can be done via the **Part** button or by double-clicking in the **Part** column.
8. Click **OK**. The terminal strip `-X1` with the five terminals is now placed in the menu tree of the Terminal Manager.
9. Click in the menu tree on the first terminal `-X1`. All five terminals are marked with a green check mark because five connections were crossed.



10. Click in the Terminal Manager on the **Browser** tab.
11. Double-click in the fields of the **Pin number** column and enter the designation of the connection lines (`L1`, `L2`, `L3`, `N` and `PE`) there. Changed fields are highlighted in green.

Index	Status	RD	Pin number	Part	Sheet/path	Part name	Functio
1		-X1	L1		/1.1	Terminal (general)	
2		-X1	L2		/1.1	Terminal (general)	
3		-X1	L3		/1.1	Terminal (general)	
4		-X1	N		/1.1	Terminal (general)	
5		-X1	PE		/1.1	Terminal (general)	

12. Click **OK**. The Terminal Manager is closed. All terminals are placed at once. The terminal strip `-X1` should look like this:



Inserting motors

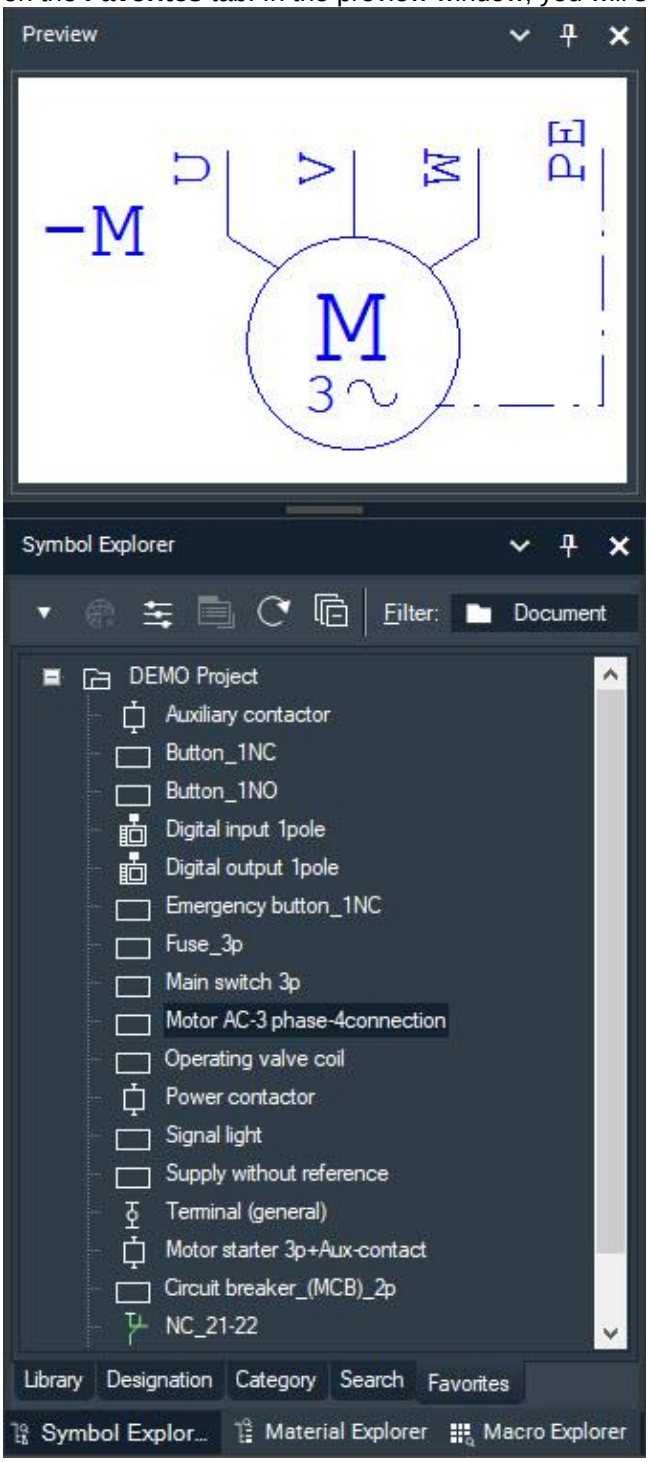
Motor, individual

In this section, you will learn how to insert the motor `-M2` between the motors `-M1` and `-M3` in the "Load circuit" schematic.

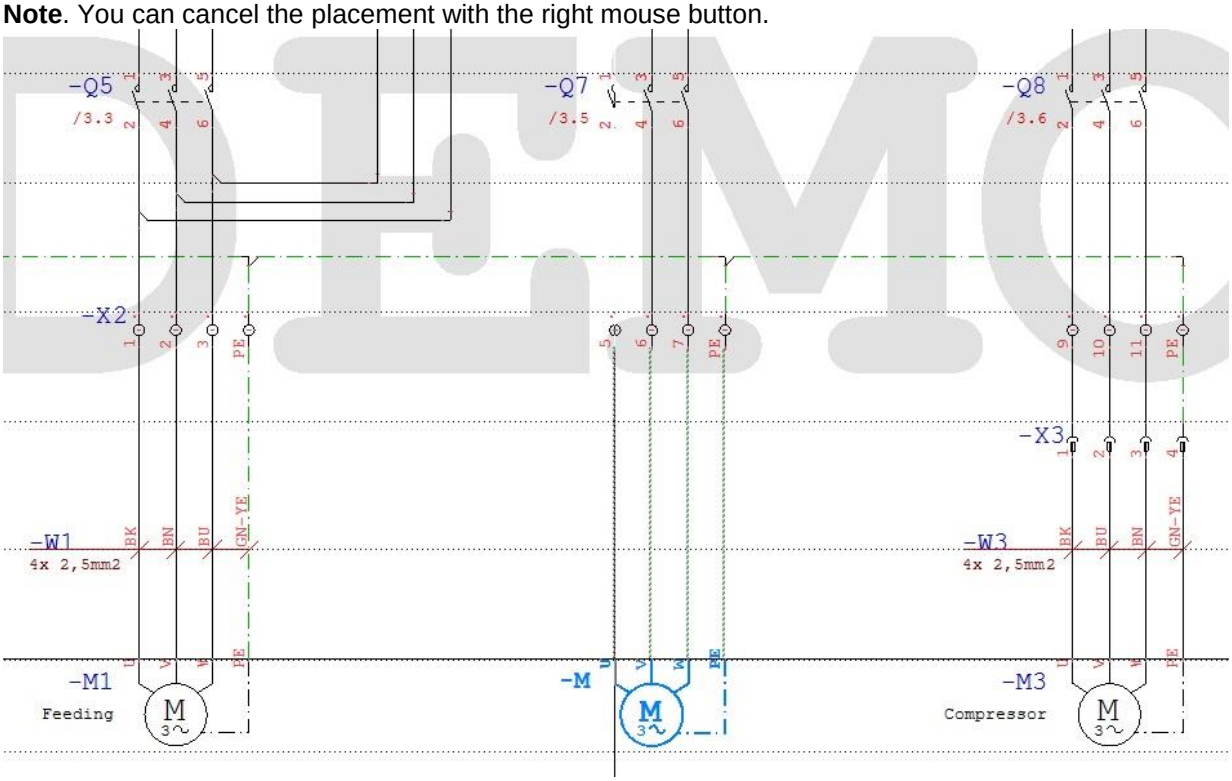
Inserting a motor individually

Prerequisite: The project **SUITE DEMO** is open.

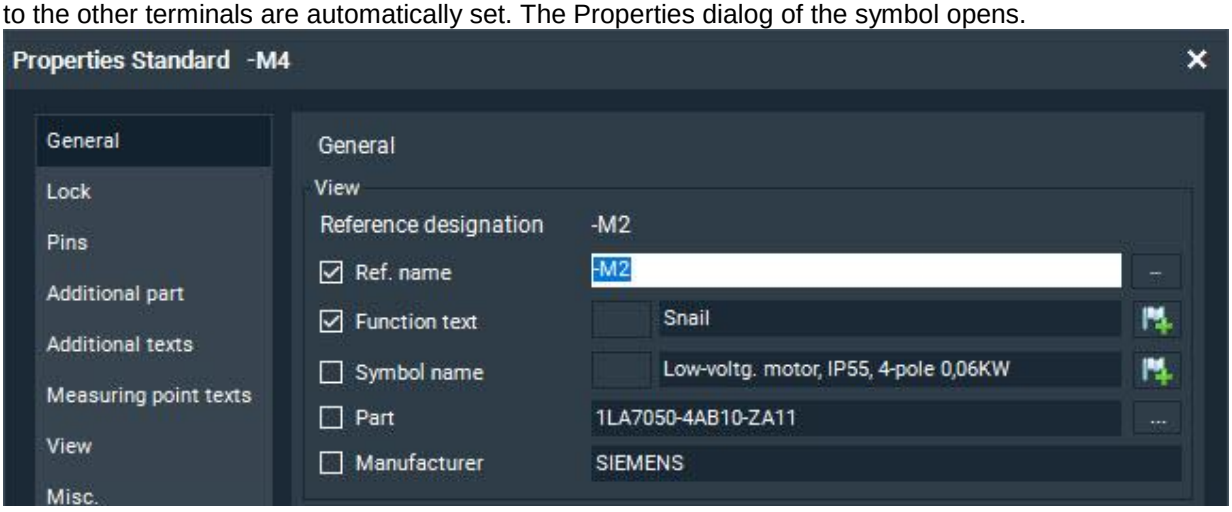
1. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0002: Load circuit**.
2. Click in the Symbol Explorer on the entry **Motor AC-3 phase-4 connection** in the folder **DEMO Project** on the **Favorites** tab. In the preview window, you will see the symbol of the motor.



3. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
4. Move the mouse pointer to the desired position in the drawing sheet - in our case under the symbol `-Q7` on the left line `5`. horizontally aligned on both the `-M1` and `-M3` motors.
   Note. You can cancel the placement with the right mouse button.

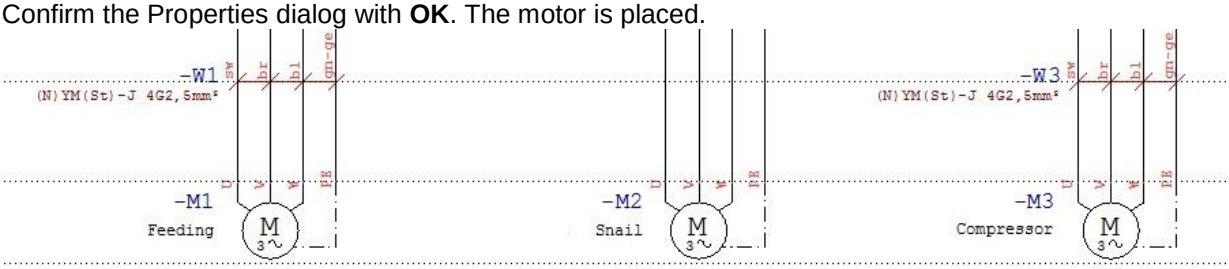


5. Place the symbol by clicking the left mouse button or by pressing the Return key. The connection lines to the other terminals are automatically set. The Properties dialog of the symbol opens.



6. Enter the reference name `-M2` for the symbol in the **Ref. name** field.
7. Click on the Customize button in the **Part** line. The Part Management opens.
8. In the part management, double-click in the **Part** column for the part `1LA7050-4AB10-ZA11`. The part, symbol name and manufacturer are taken over to the Properties dialog of the symbol.
9. Click in the **Function text** field and specify the function of the motor, e.g., `Snail`.
10. Select the check box before the **Function text**, field so that you can see the function in the drawing sheet.

11. Confirm the Properties dialog with **OK**. The motor is placed.



12. The symbol still hangs on the mouse pointer in case a further motor needs to be inserted. Click the right mouse button to cancel the insertion of a further symbol.

Inserting a motor individually using copy & paste

In this section, you will learn how to insert the motor `-M2` between the motors `-M1` and `-M3` in the "Load circuit" schematic using copy & paste. The procedure is also possible for other symbols.

First, please delete the `-M2` motor that was just created.

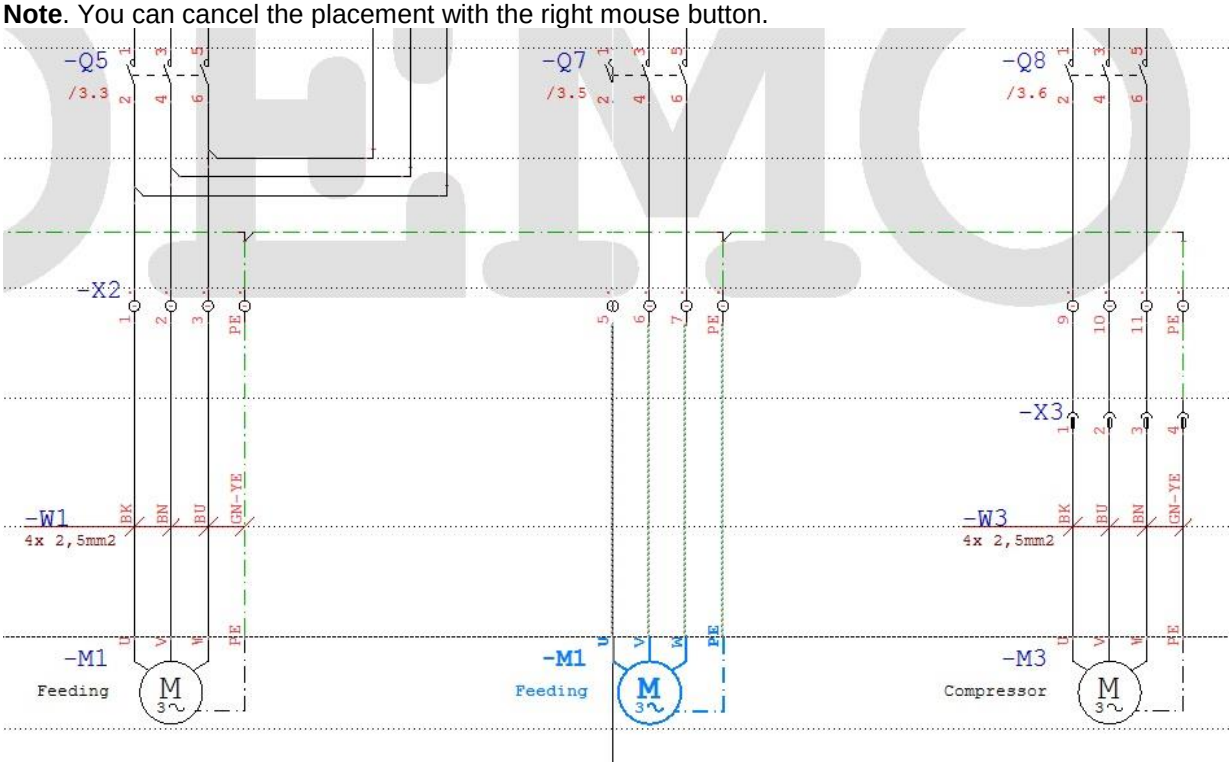
Deleting a motor

Prerequisite: The drawing sheet **Schematic** | **0002: Load circuit** is open.

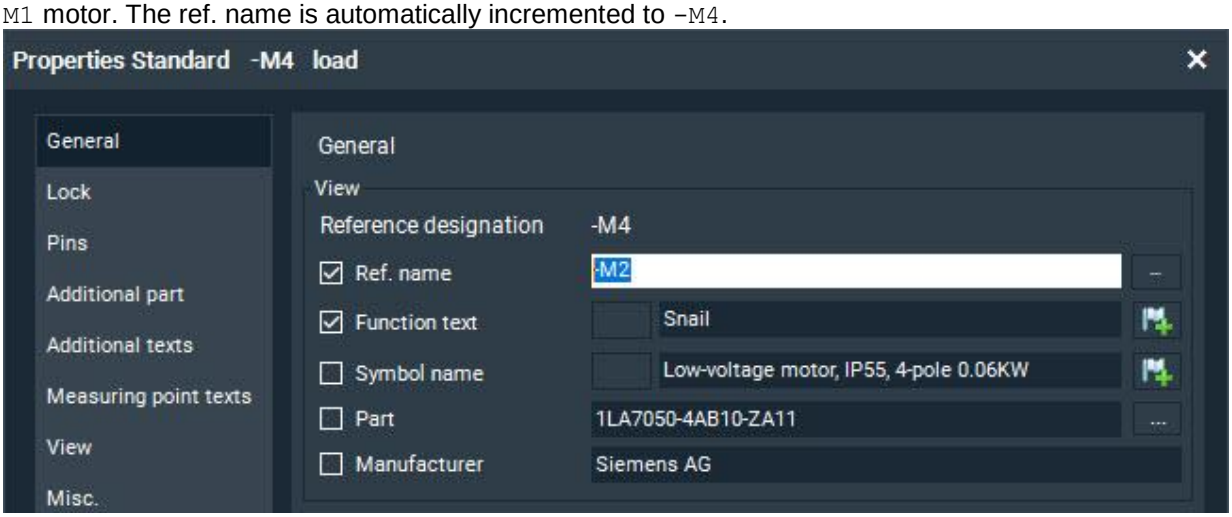
1. Click with the right mouse button on the motor `-M2` and select the command **Delete** from the context menu.
2. Select the check box **Also delete references from the Material Explorer** and confirm the prompt with **Yes**.

Inserting a motor using copy & paste

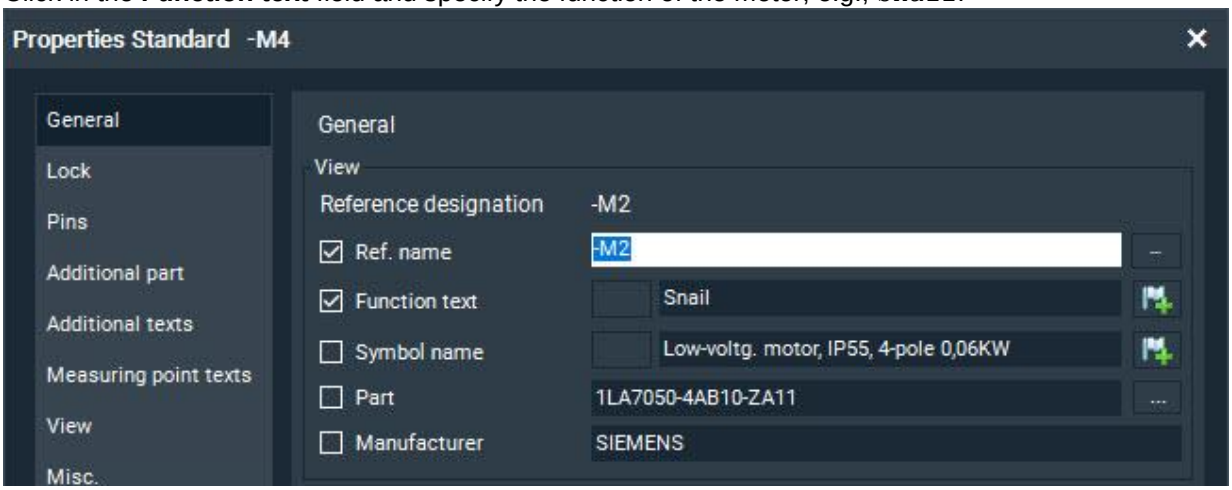
1. Click with the right mouse button on the motor `-M1` and select the command **Copy** from the context menu.
2. Click with the right mouse button on a free location in the drawing sheet and select the command **Paste** from the context menu. The motor hangs on the mouse pointer.
3. Move the mouse pointer to the desired position in the drawing sheet - in our case under the symbol `-Q7` on the left line `5`. horizontally aligned on both the `-M1` and `-M3` motors.
   Note. You can cancel the placement with the right mouse button.



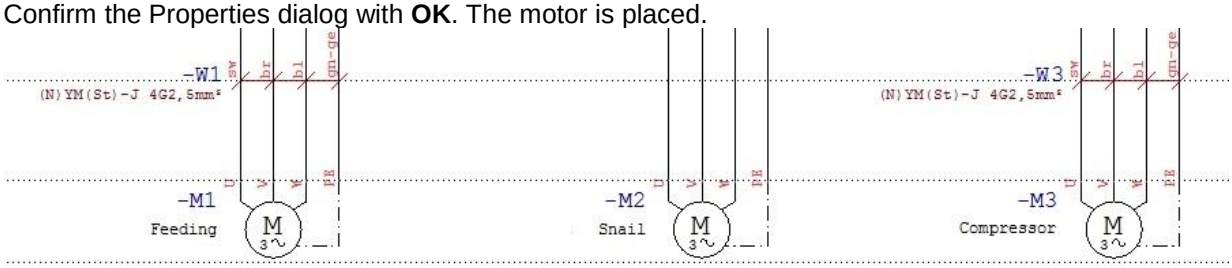
4. Place the symbol by clicking the left mouse button or by pressing the Return key. The connection lines to the other terminals are set. The Properties dialog of the symbol opens with the data of the copied `-M1` motor. The ref. name is automatically incremented to `-M4`.



5. Enter the reference name `-M2` for the symbol in the **Ref. name** field.
6. Click in the **Function text** field and specify the function of the motor, e.g., `Snail`.



7. Confirm the Properties dialog with **OK**. The motor is placed.



Motor, multiple

In this section you will learn how to insert the two motors `-M2` and `-M3` in the "Load circuit" schematic using the multiple function. The procedure is also possible for other symbols.

First, please delete the `-M2` motor that was just created as well as the `-M3` motor.

Deleting motors

Prerequisite: The drawing sheet **Schematic** | **0002: Load circuit** is open.

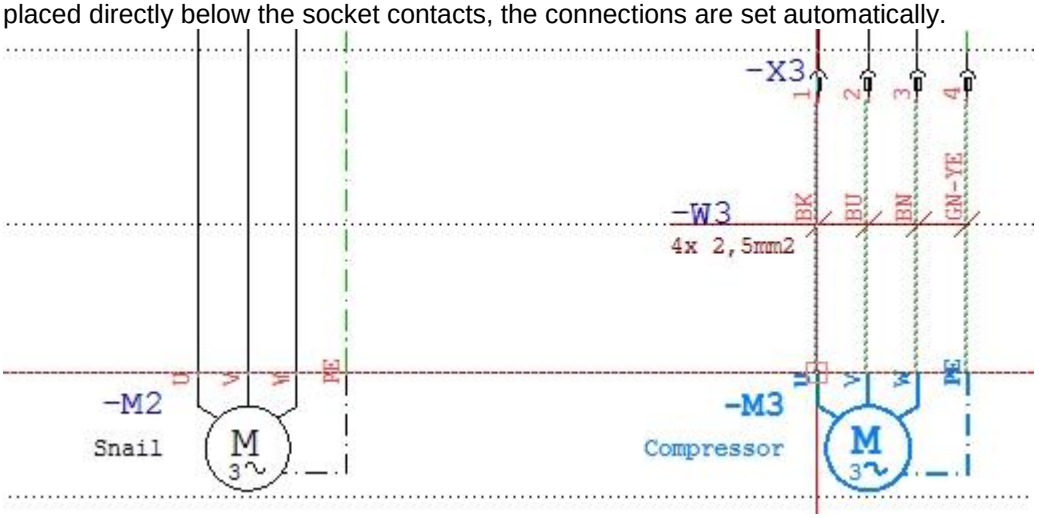
1. Click on the motor `-M2`, hold down the Ctrl key and then click on the motor `-M3`. Both motors are marked in blue. Click with the right mouse button on one of the two motors and select the command **Selection** | **Delete** from the context menu.
2. Select the check box **Also delete references from the Material Explorer** and confirm the prompt with **Yes**.

Inserting multiple motors

1. Click with the right mouse button on the motor `-M1` and select the command **Copy** from the context menu.
2. Click with the right mouse button on a free location in the drawing sheet and select the command **Paste** from the context menu. The motor hangs on the mouse pointer.
3. Press the plus key on the keyboard. Another copy of the motor is created. You can add further motors by pressing the plus key again and remove them with the minus key. The distance between the original and copy determines the distance between the additional copies. The copy axis can be horizontal, vertical, or even diagonal. Diagonal is only possible if the orthogonal drawing mode is turned off (F6 key).
4. Place the two motors below `-Q7` and `-Q8`. Horizontally aligned on the motor `-M1`. The ref. names of the motors are automatically numbered with `-M2` and `-M3` after the placement.
5. Open the **Quick Editor**. To do this, you have the following options:
   - In the menu bar, click on **View** | **Additional windows** | **Quick Editor**.
   - Click in the view bar below the drawing area on the **Quick Editor** symbol.
6. In the Quick Editor, click in the column header **Ref. name**. The list is sorted by the ref. names.
7. Scroll to the ref. names `-M1`, `-M2` and `-M3`.
8. Change the function text in the **Function text** column for the `-M2` motor to `Snail` and for the `-M3` motor to `Compressor`. The changes are immediately applied in the schematic.

	Element type	KNX	Ref. name	Symbol name	Function text	Part	Manufacturer	Lock material list
	Contact comb					072735+072896		☐
	Contact comb					072735+072896		☐
	Contact comb					072735+072896		☐
	Standard		-M1	Low-voltage mot...	Feeding	1LA7050-4AB10-...	Siemens AG	☑
	Standard		-M2	Low-voltage mot...	Snail	1LA7050-4AB10-...	Siemens AG	☑
▸	Standard		-M3	Low-voltage mot...	Compressor	1LA7050-4AB10-...	Siemens AG	☑
	Coil		-Q2	Motor protector 3...	motor protection /	072735+072896	EATON	☐
	Coil		-Q3	Motor protector 3...	motor protection ...	072735+072896	EATON	☐
	Coil		-Q4	Motor protector 3...	motor protection ...	072735+072896	EATON	☐

9. The –M3 motor has not yet been connected to the socket contacts –X3. The connections can be set automatically, but this requires the motor to be moved.
10. Click with the right mouse button on the motor –M3 and select the command **Move** from the context menu. The crosshair is located in a red frame.
11. Move the symbol with the mouse pointer or with the arrow keys and the Return key. When the motor is placed directly below the socket contacts, the connections are set automatically.



Inserting cables

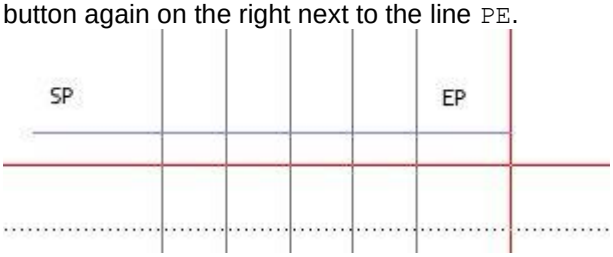
Cable

In this section, you will learn how to insert the cable `-W2` between the terminal strip `-X2` and the motor `-M2` in the "Load circuit" schematic.

Inserting a cable

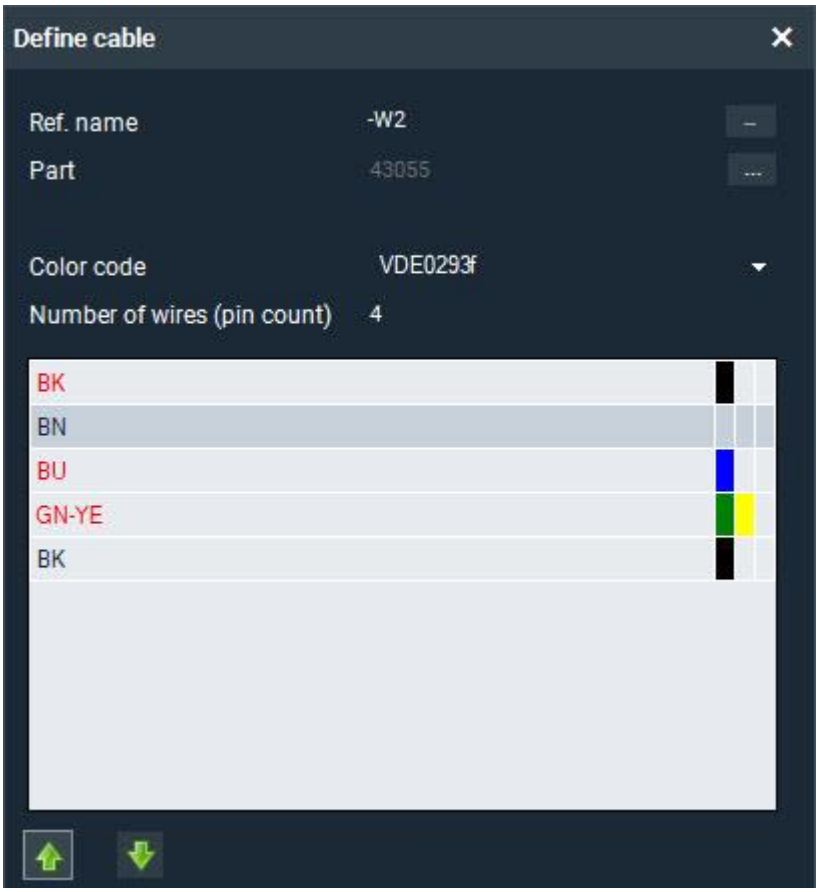
Prerequisite: The drawing sheet **Schematic** | **0002: Load circuit** is open.

1. To insert a cable, you have the following options:
   - Click on **Insert** | **Electrical Engineering** | **Cable** in the menu bar.
   - Click with the right mouse button in a toolbar and enable the toolbar **Electrical Engineering**. Click in the bar on the symbol button **Cable**  .
2. The crosshair turns red. Set a start point (SP) by clicking with the left mouse button on the left of line `5` and above the symbol `-M2` on the height of `-W1` and `-W3`. Draw a horizontal line (marked blue) across the four existing connections (`5`, `6`, `7` and `PE`). Set an end point (EP) by clicking with the left mouse button again on the right next to the line `PE`.

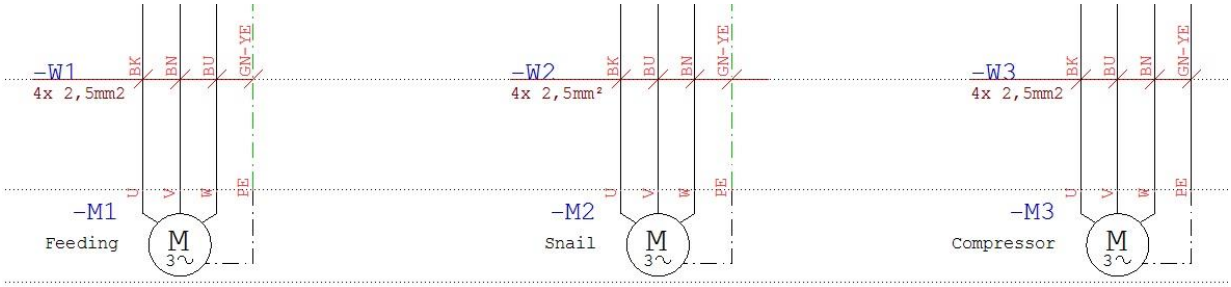


3. The Cable Manager opens.
4. To define a new cable, click in the toolbar of the Cable Manager on the symbol button **New**  at the top left. The **Define cable** dialog opens. The number of wires is already filled, since it corresponds to the number of crossed connections.

5. Enter the reference name `-W2` for the cable in the **Ref. name** field.



6. Click on the Browse button in the **Part** line. The Part Management opens.
7. In the part management, double-click in the **Part name** column for the part with the part name `(N)YM(St)-J 4G2,5mm²`. The color code for the four wires is accepted and displayed.
8. Sort the color code with the arrow symbol buttons in the following order: BK, BN, BU, GN-YE, BK
9. Confirm with **OK**. The cable `-W2` with the four wires is now created in the menu tree of the Cable Manager.
10. Click **OK**. The Cable Manager is closed. All cables are placed at once. The cable `-W2` should look like this:



Inserting PLC parent/child elements

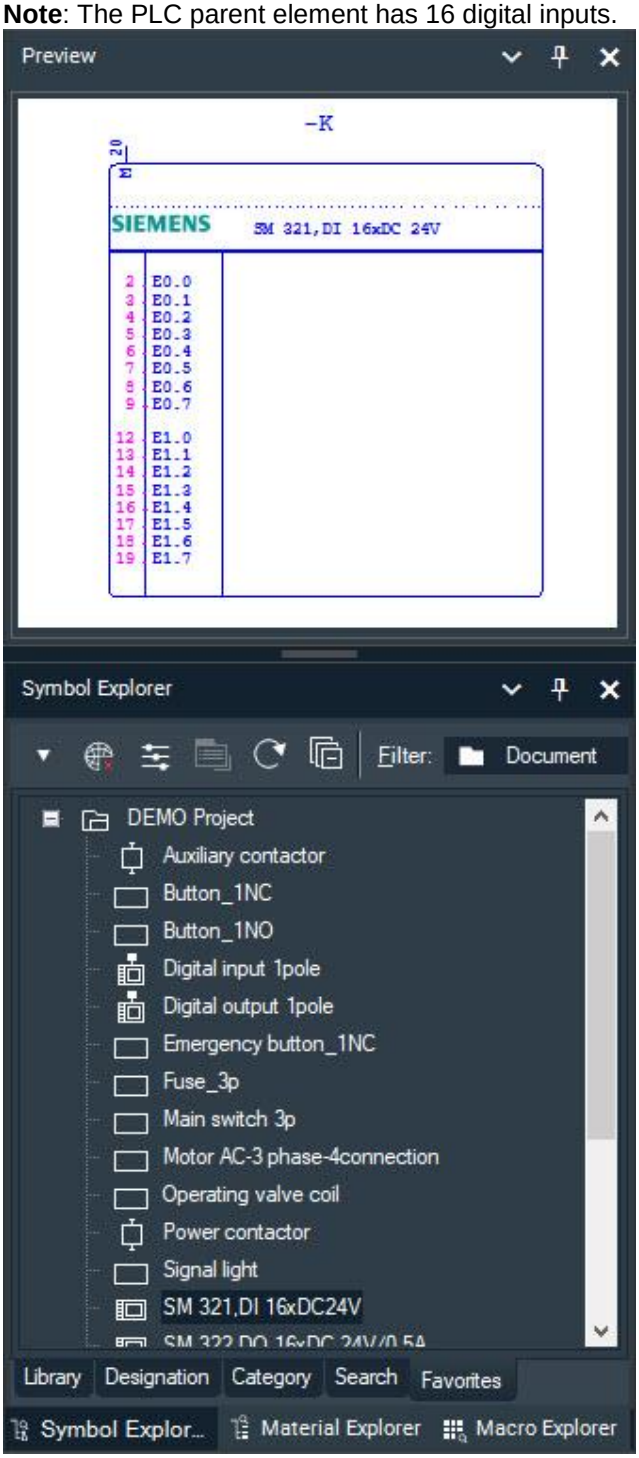
PLC parent element

In this section, you will learn how to insert the PLC parent element `-K3` between the PLC parent elements `-K2` and `-K4` in the "PLC chart" schematic.

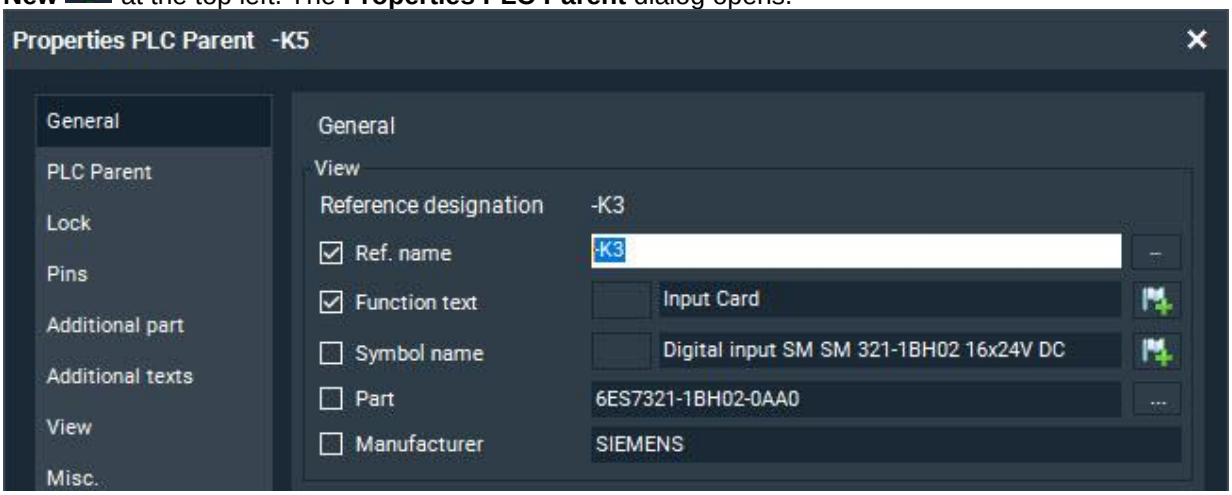
Inserting a PLC parent element

Prerequisite: The project **SUITE DEMO** is open.

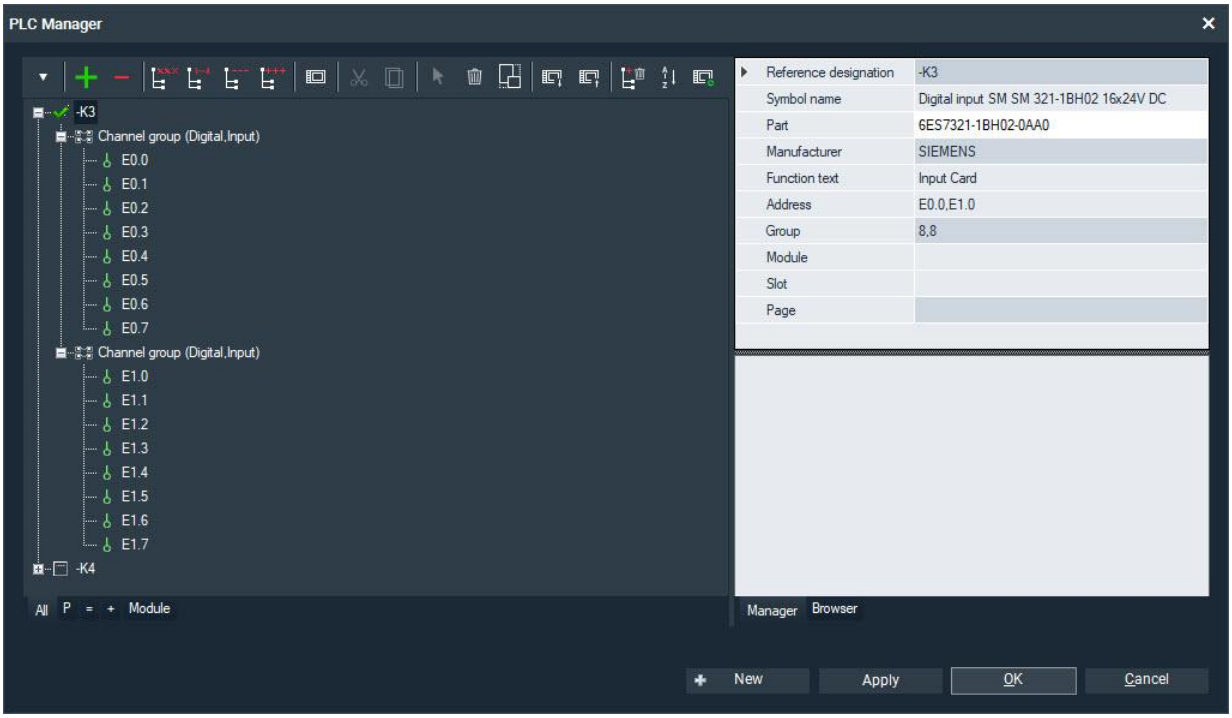
1. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0005: PLC chart**.
2. Click in the Symbol Explorer on the **Favorites** tab, in the **DEMO Project** folder, on the entry **SM 321,DI 16xDC24V**. In the preview window, you will see the symbol of the PLC parent element.
   Note: The PLC parent element has 16 digital inputs.



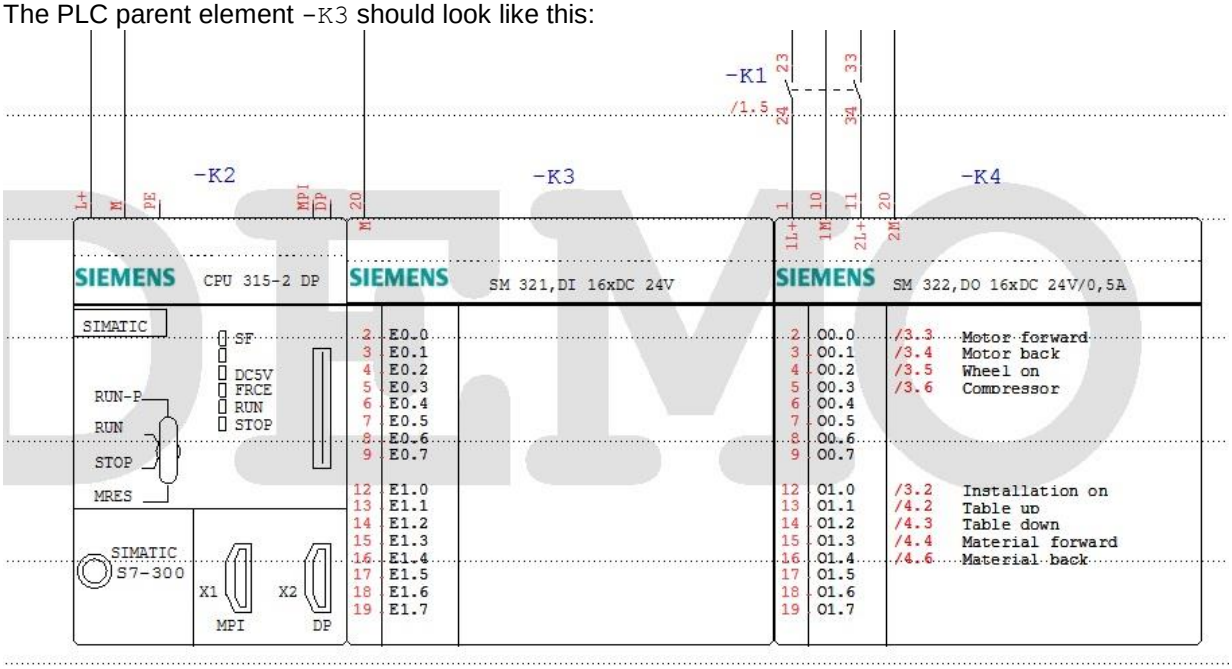
3. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
4. Move the mouse pointer to the desired position in the drawing sheet - in our case between the symbols `-K2` and `-K4`.
   Note. You can cancel the placement with the right mouse button.
5. Place the symbol by clicking the left mouse button or by pressing the Return key. The PLC Manager opens.
   To define a new PLC parent element, click in the toolbar of the PLC Manager on the symbol button **New**  at the top left. The **Properties PLC Parent** dialog opens.



6. Enter the reference name `-K3` for the PLC parent element in the **Ref. name** field.
7. Click on the Customize button in the **Part** line. The Part Management opens.
8. In the part management, double-click in the **Part** column for the part `6ES7321- 1BH02-0AA0`.
9. Enter the function `Input card` in the **Function text** field.
10. Click **OK**. The PLC element `-K3` was placed in the menu tree of the PLC Manager. All channels are still free.



11. Click **OK**. The PLC Manager is closed.
12. The symbol still hangs on the mouse pointer in case a further PLC parent element needs to be inserted. Click the right mouse button to cancel the insertion of a further symbol.
    The PLC parent element `-K3` should look like this:



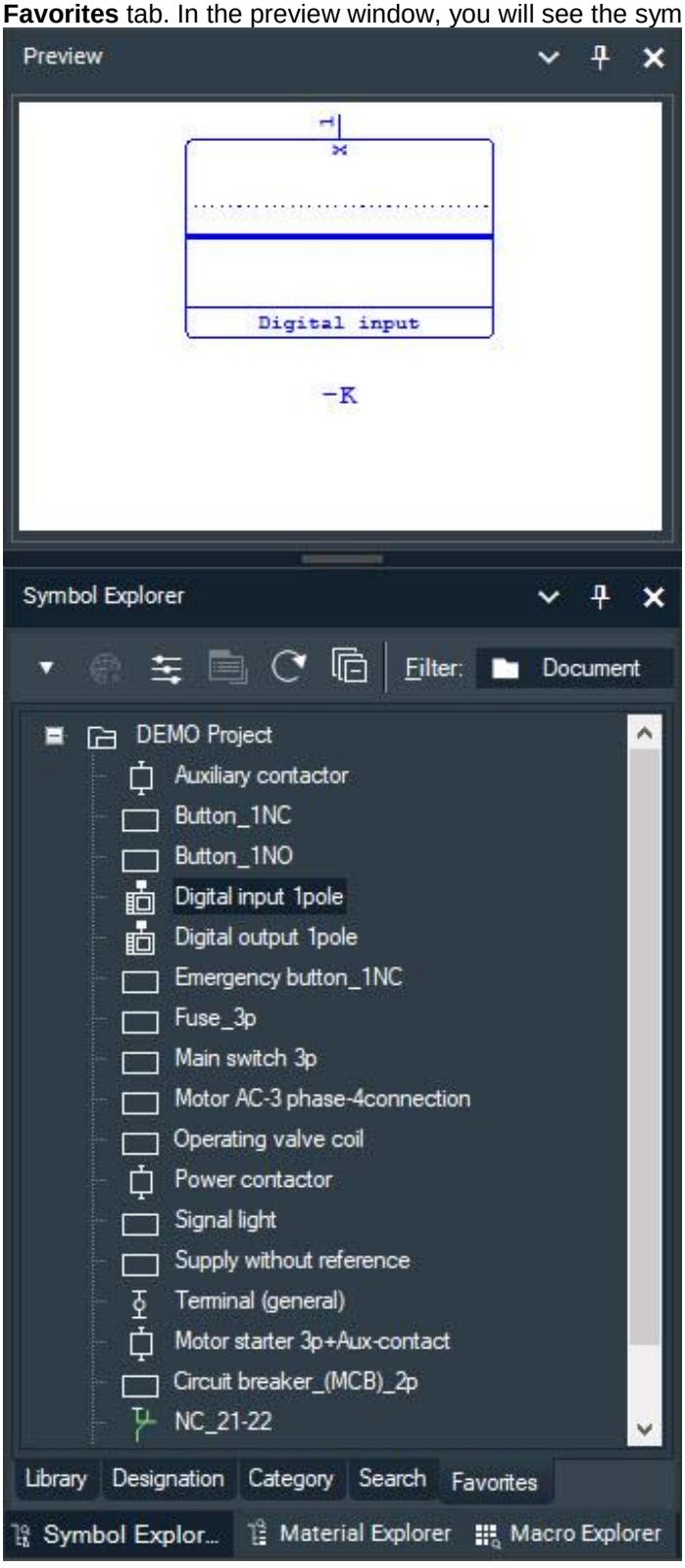
PLC child element

In this section, you will learn how to insert two PLC child elements `-K3` below the `-S4` and `-S5` switches in the "PLC input/output" schematic.

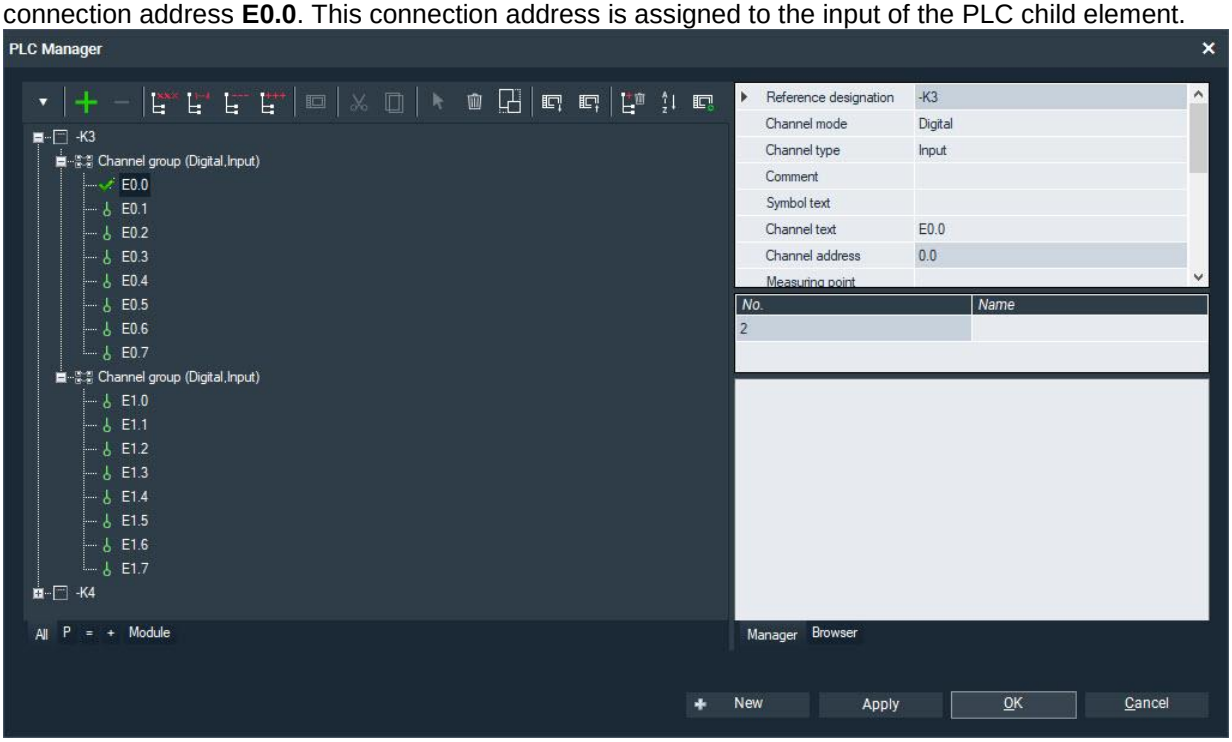
Inserting a PLC child element

Prerequisite: The project **SUITE DEMO** is open.

1. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0003: PLC input/output**.
2. Click in the Symbol Explorer on the entry **Digital input 1Pol** in the folder **DEMO Project** on the **Favorites** tab. In the preview window, you will see the symbol of the PLC child element.

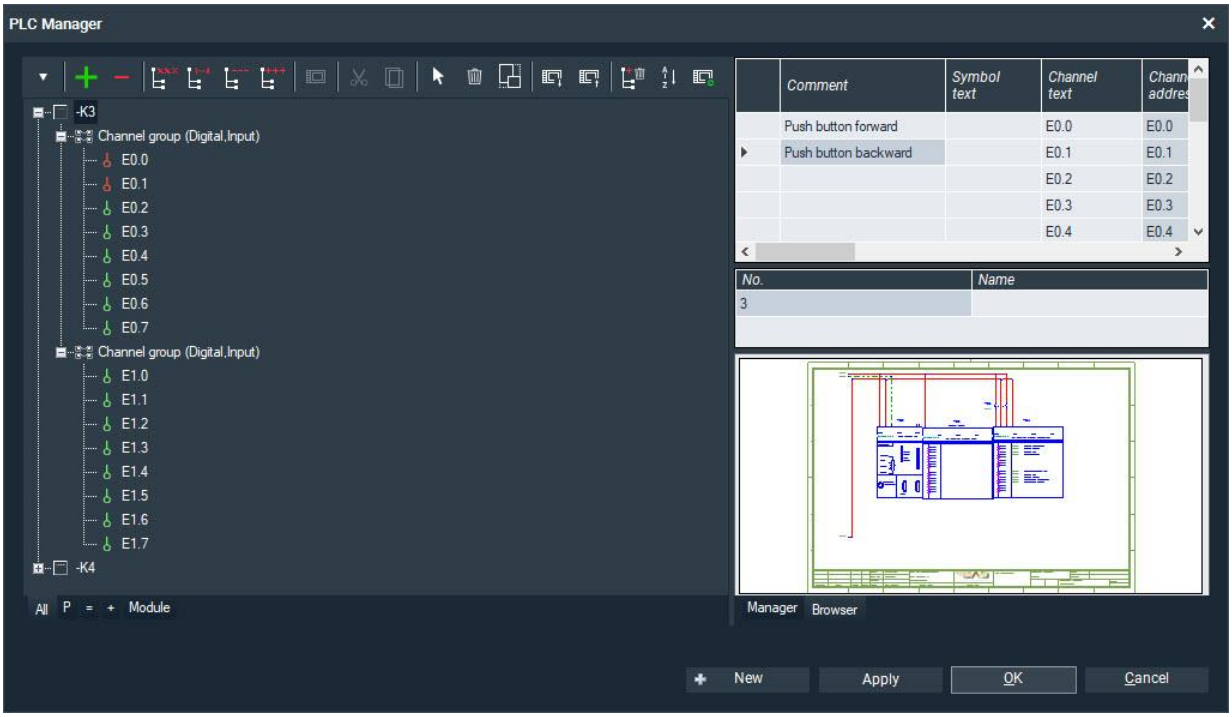


3. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
4. Move the mouse pointer to the desired position in the drawing page - in our case, below the switch `-S4`. **Note**. You can cancel the placement with the right mouse button.
5. Place the symbol by clicking the left mouse button or by pressing the Return key. The PLC Manager opens.
6. Click in the menu tree at the PLC parent element **-K3** in the **Channel group (Digital, Input)** on connection address **E0.0**. This connection address is assigned to the input of the PLC child element.

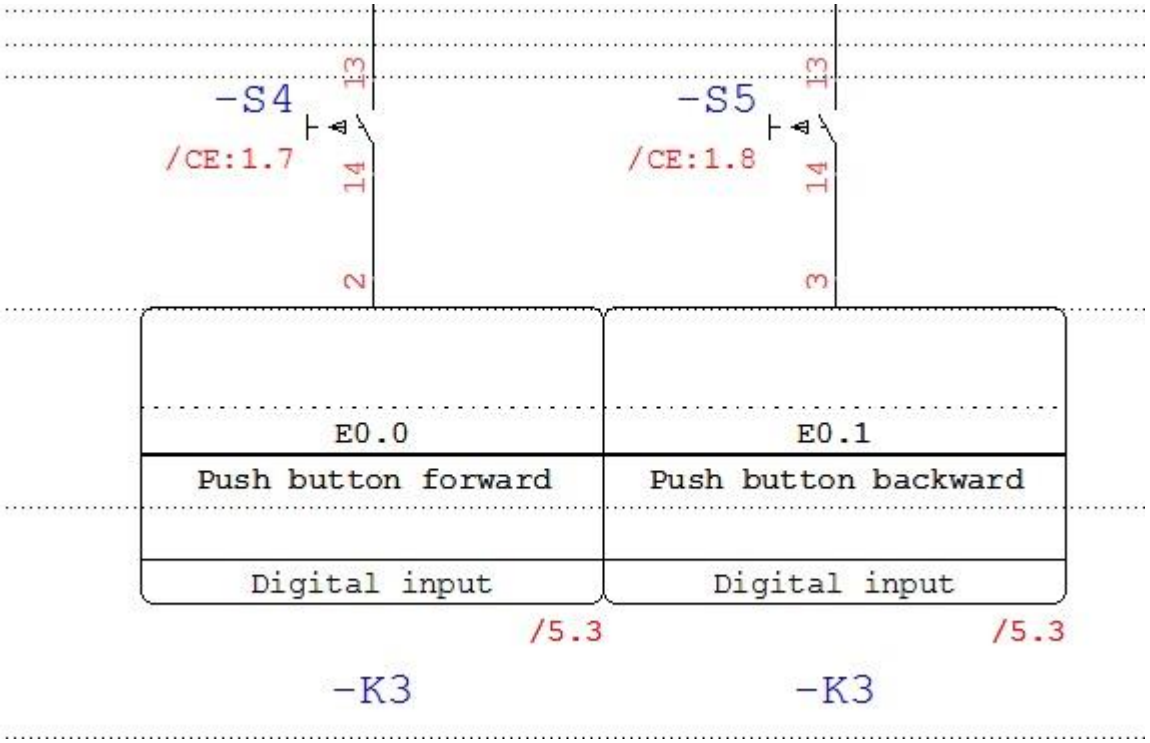


7. Click **OK**.
8. Place the next PLC child element below the switch `-S5`.
9. The PLC Manager is opened again.
10. Now assign the channel **E0.1** to the input of the second PLC child element and confirm this with **OK**.
11. The symbol still hangs on the mouse pointer in case a further PLC child element needs to be inserted. Click the right mouse button to cancel the insertion of a further symbol.
12. Call up the PLC Manager again. To do this, you have the following options:
    - Click in the menu bar on **Manager | PLC**.
    - Click with the right mouse button on a child element and select the command **Manager** from the context menu.
13. Click in the menu tree on the PLC parent element **-K3**. Busy channels are marked in red, free channels in green.
14. Click on the **Browser** tab.

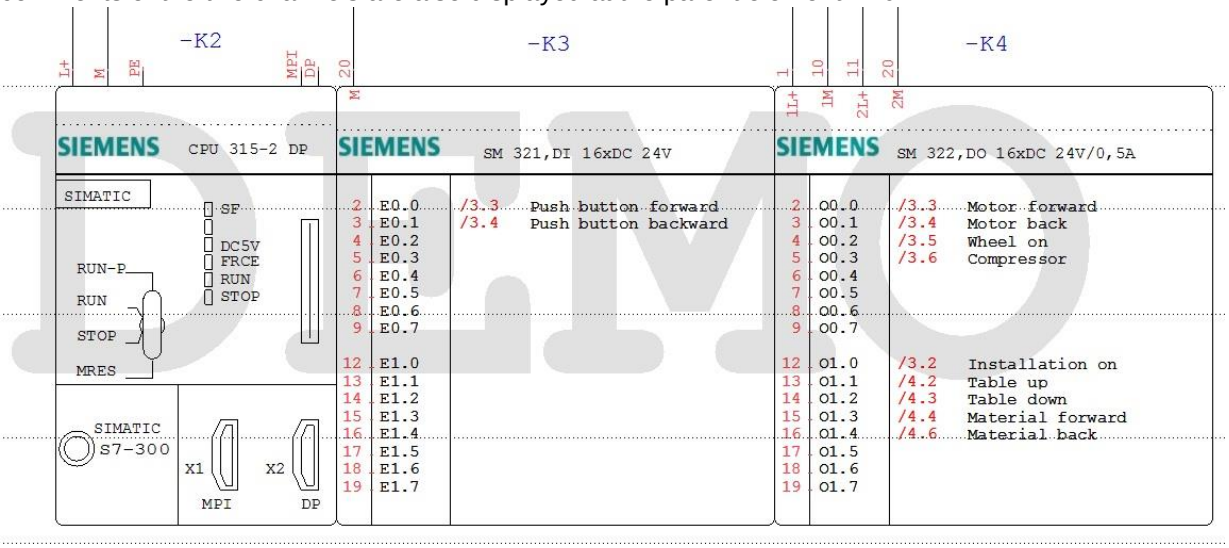
15. Enter the function `Push button forward` for the channel **E0.0** and the function `Push button backward` for the channel **E0.1** in the **Comment** field.



16. Click **OK**. The comments are entered in the two child elements.



17. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0005: PLC chart**. The comments of the two channels are also displayed at the parent element `-K3`.



18. The cross references `/3.3` and `/3.4` are also displayed at the two channels. The cross reference is composed of the page and the path on which the child element is placed. A right click on the cross reference takes you to the child element.

Inserting contactors

Power contactor

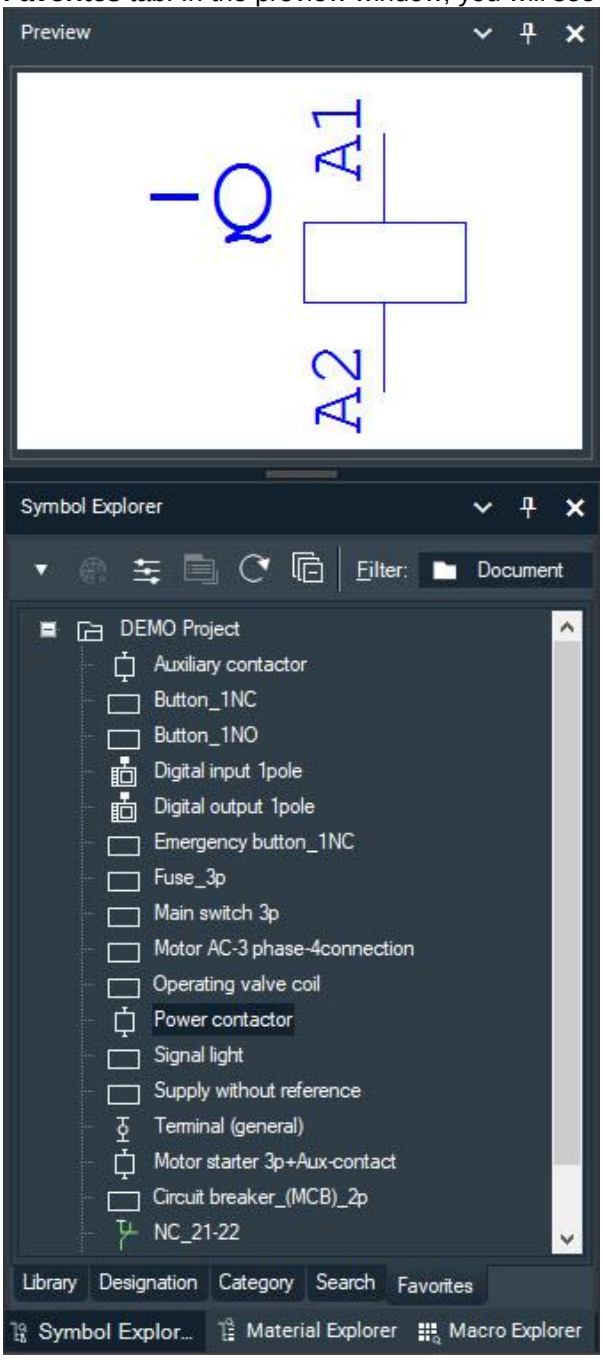
In this section you will learn how to insert the power contactor -Q6 with coil, main contact and auxiliary contact.

The coil is placed together with its comb below the contact -Q5 in the "PLC input/output" schematic. The main contact is placed on the "Load circuit" schematic, and the auxiliary contact is placed on the "PLC input/output" schematic.

Inserting the coil of the contactor

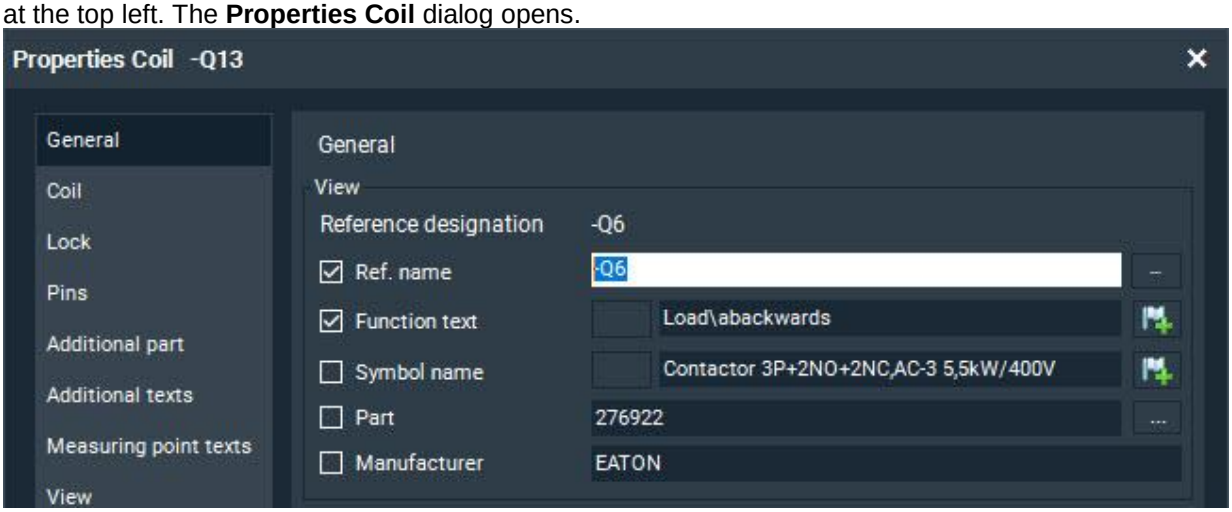
Prerequisite: The project **SUITE DEMO** is open.

1. Double-click in the Project Explorer on the drawing sheet **Schematics** | **0003: PLC input/output**.
2. Click in the Symbol Explorer on the entry **Power contactor** in the folder **DEMO Project** on the **Favorites** tab. In the preview window, you will see the symbol of the contactor.

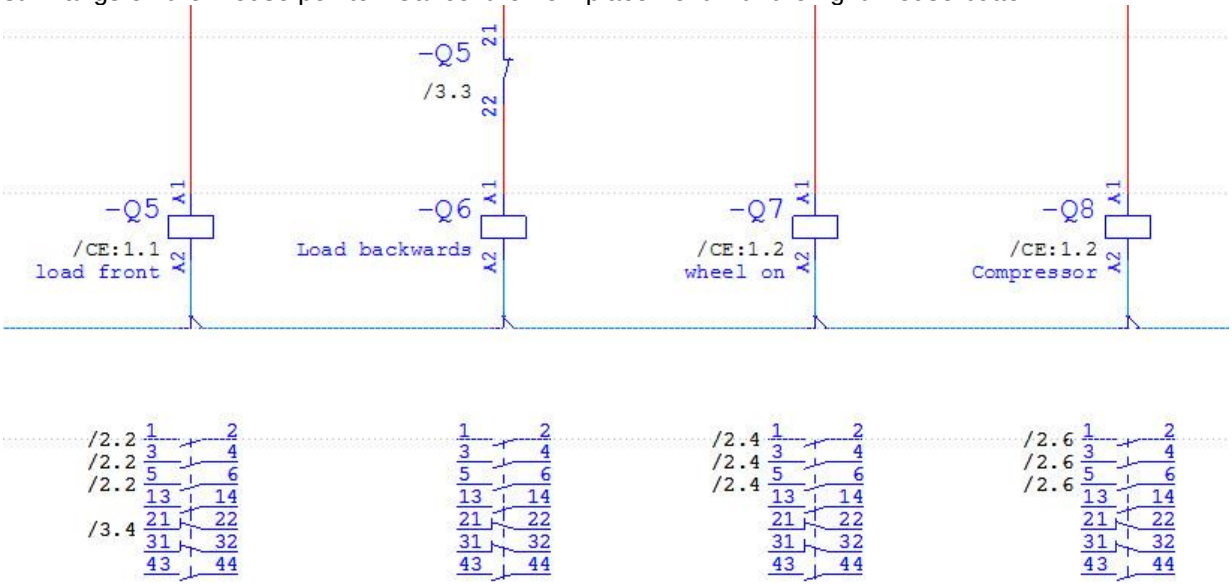


3. To take over the symbol in the drawing sheet, you have the following options:
   - Double-click on the entry in the Symbol Explorer. The symbol "hangs" on the mouse pointer.
   - Click in the Preview window. The symbol "hangs" on the mouse pointer.
   - Click with the right mouse button on the entry in the Symbol Explorer and select the command **Place | Single** from the context menu. The symbol "hangs" on the mouse pointer.
4. Move the mouse pointer to the desired position in the drawing sheet - in our case below of the auxiliary contact `-Q5` on the line. horizontally aligned on both the `-Q5` and `-Q7` coils.
   Note. You can cancel the placement with the right mouse button.
5. Place the symbol by clicking the left mouse button or by pressing the Return key. The Contactor Manager opens.

   To define a new contactor, click in the toolbar of the Contactor Manager on the symbol button **New** at the top left. The **Properties Coil** dialog opens.

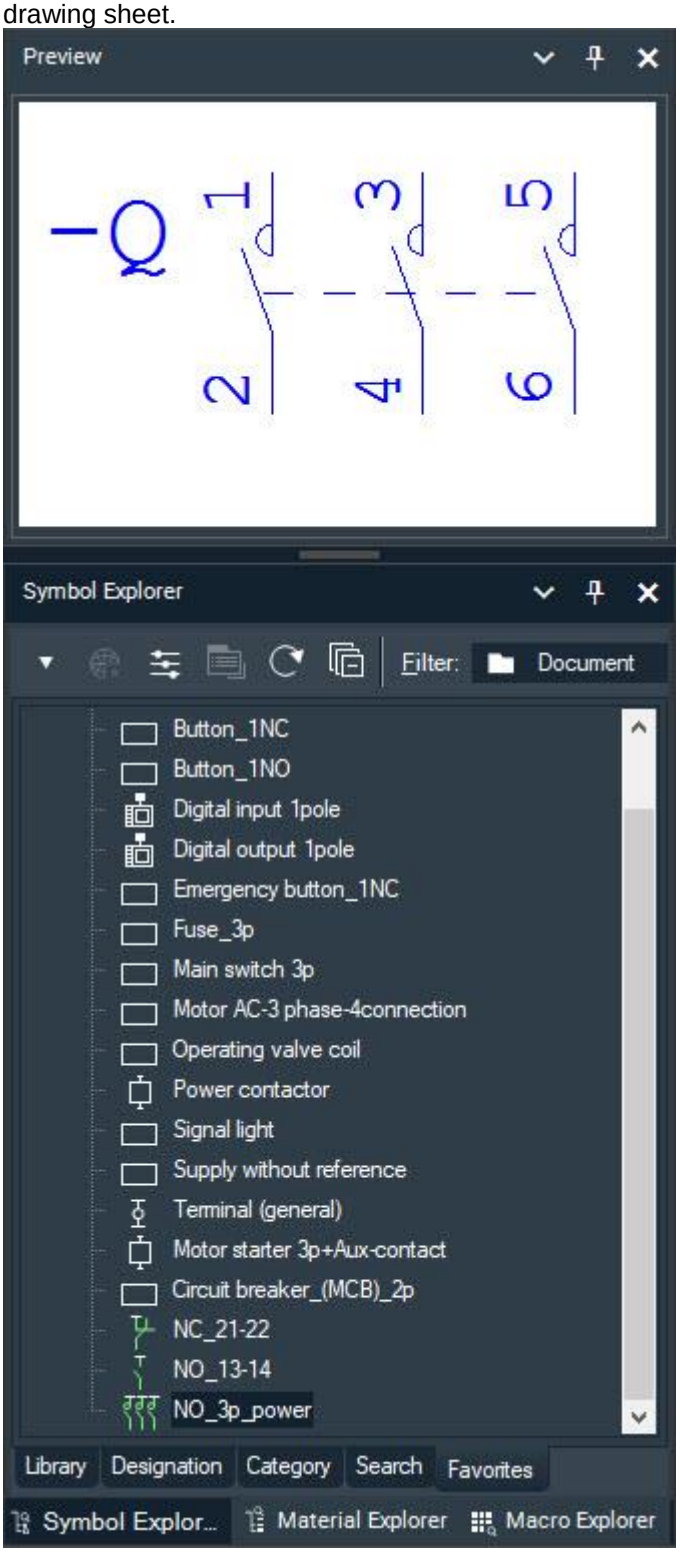


6. Enter the reference name `-Q6` for the contactor in the **Ref. name** field.
7. Click on the Customize button in the **Part** line. The Part Management opens.
8. In the part management, double-click in the **Part** column for the part `276922`. The part is taken over.
9. Enter the function `Load backwards` in the **Function text** field and select the check box so that the text in the schematic is visible.
   Note: `\a` wraps the text. For example, if you enter `Load\abackwards`, a new line will be created after `Load`.
10. Click **OK**. The contactor `-K6` was placed in the menu tree of the PLC Manager.
11. Click **OK**. The Contactor Manager is closed, and the coil is placed together with its comb. The symbol still hangs on the mouse pointer. Cancel the new placement with the right mouse button.

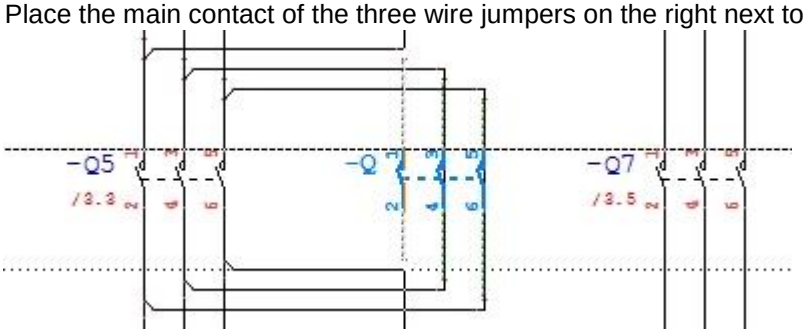


Inserting the main contact of the contactor

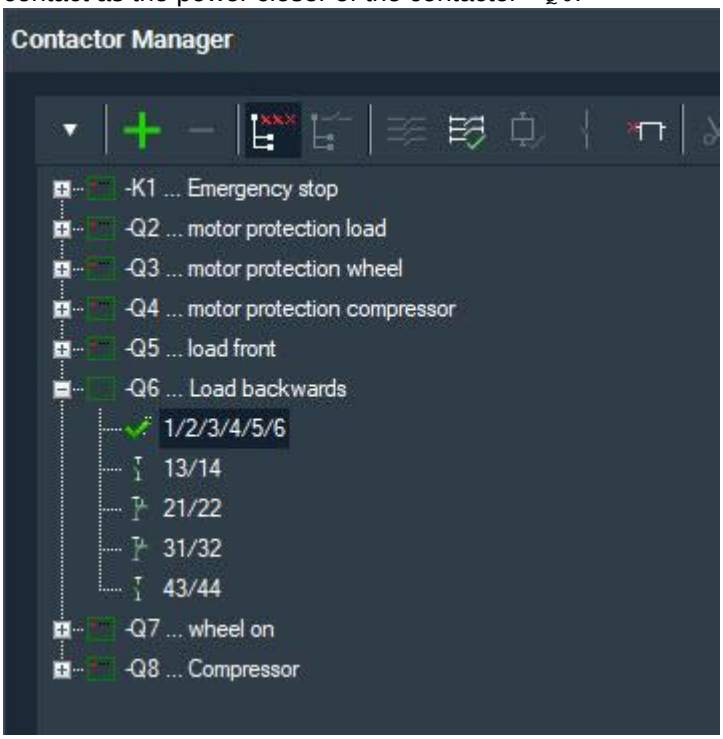
12. Now insert the symbol **NO_3p_power** via the Symbol Explorer in the **Schematics** | **0002: load circuit** drawing sheet.



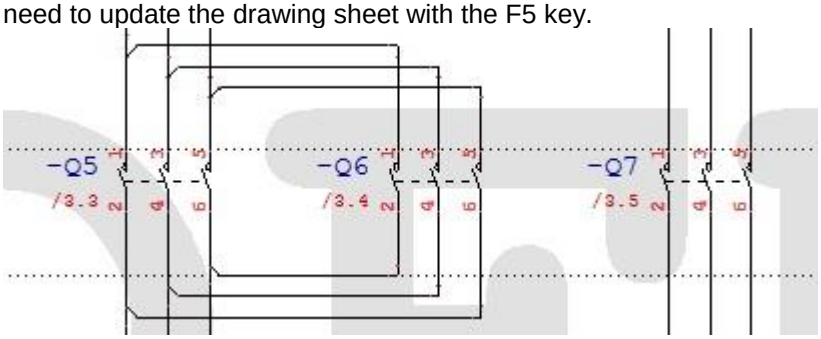
13. Place the main contact of the three wire jumpers on the right next to `-Q5`.



14. The Contactor Manager opens. Click on the first sub-item of the contactor `-Q6`. This defines the main contact as the power closer of the contactor `-Q6`.



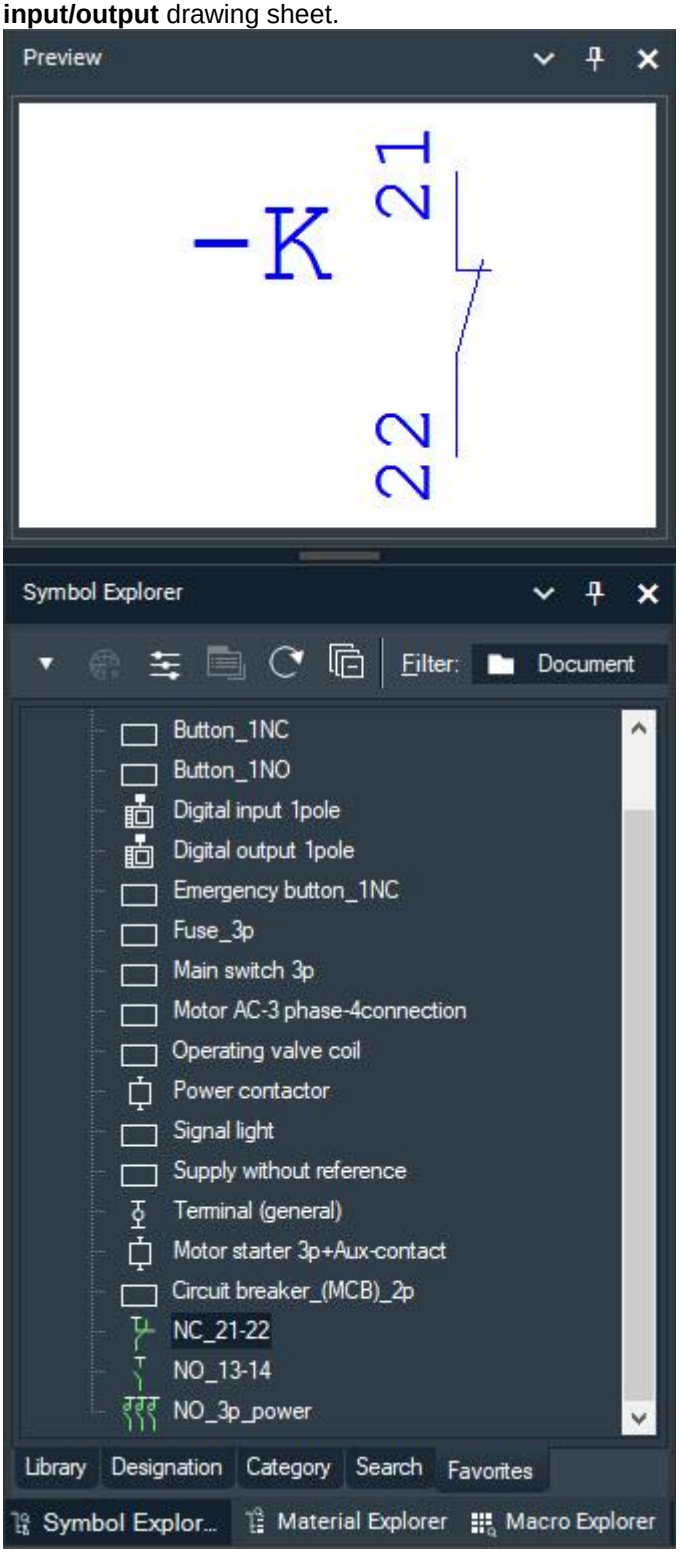
15. Click **OK**. The Contactor Manager is closed. The symbol still hangs on the mouse pointer. Cancel the new placement with the right mouse button.
The main contact `-Q6` shows a cross reference to the coil (`/3.4`, drawing sheet 3, path 4). You may need to update the drawing sheet with the F5 key.



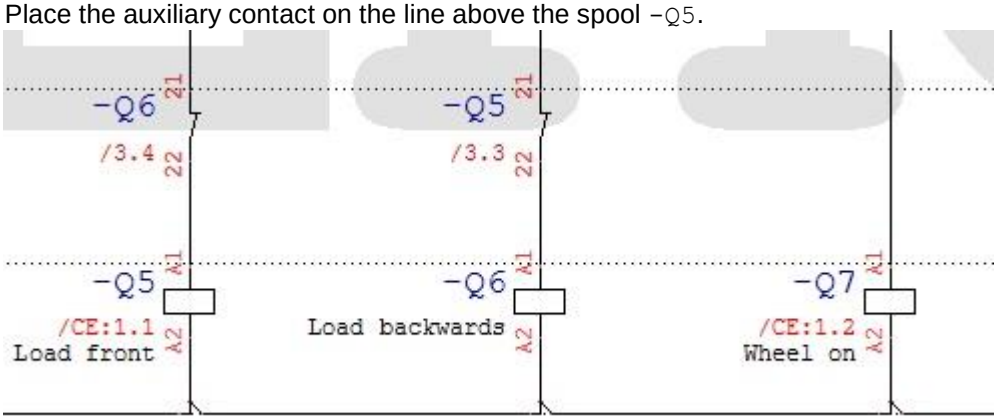
16. Click with the right mouse button on the cross-reference of the main contact `-Q6`. This takes you to the drawing sheet **0003: PLC input/output**. The crosshair is at the comb of the coil. There are cross references to the main contact (`/2.3`, drawing sheet 2, path 3) there as well.

Inserting the auxiliary contact of a contactor

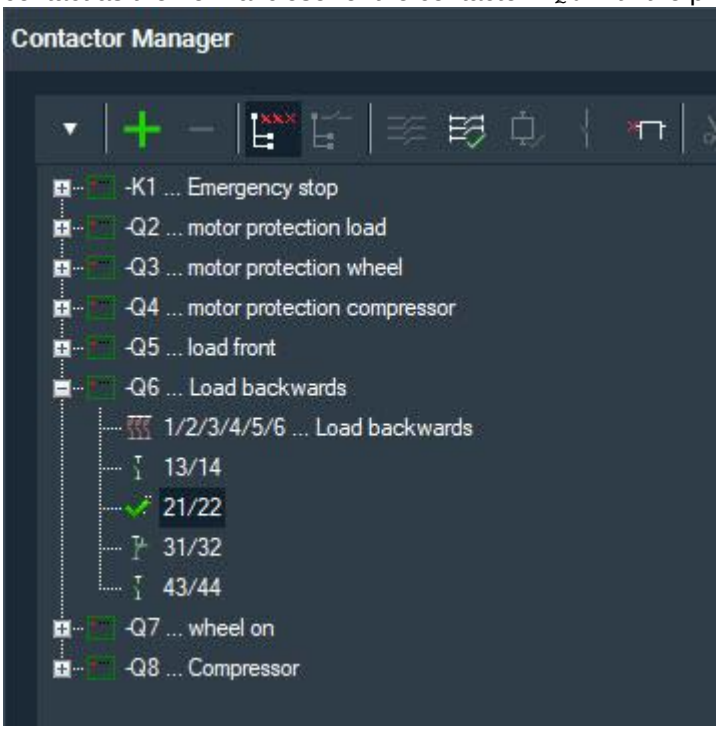
17. Now insert the symbol **NO_21-22** via the Symbol Explorer in the **Schematics** | **0003: PLC input/output** drawing sheet.



18. Place the auxiliary contact on the line above the spool `-Q5`.

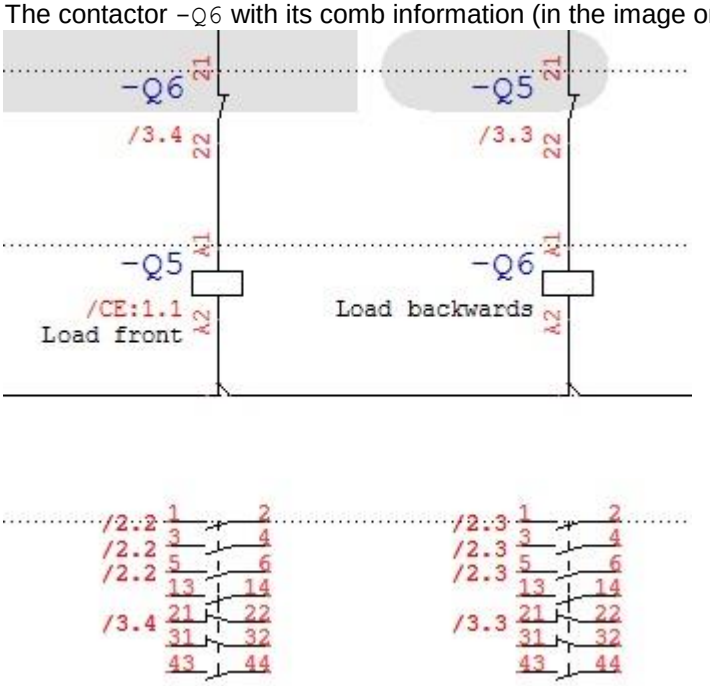


19. The Contactor Manager opens. Click the third sub-item of the contactor `-Q6`. This defines the auxiliary contact as the normal closer of the contactor `-Q6` with the pin number 21/22.



20. Click **OK**. The Contactor Manager is closed. The symbol hangs on the mouse pointer. Cancel the new placement with the right mouse button.
The auxiliary contact shows a cross reference to the coil `/3.4`, drawing sheet 3, path 4).

21. The contactor -Q6 with its comb information (in the image on the right below) should look like this:



The cross-references to the contacts of the contactor are shown on the left side of the comb.

Reporting

Overall report

You can create multiple lists and plans at once without having to configure them individually in advance.

Starting an overall report

1. Click in the menu bar on **Report | Overall report.**
2. Mark all lists and plans.

3. Click **Finish**. The marked list and plans are created. Existing list and plans are overwritten.
4. Finally, click **Finish**.
5. Open one of the lists in the Project Explorer.

Individual reports

You can generate a wide variety of lists and plans individually. Each list has different configuration options available. The lists and plans are based on forms that can be individually customized.

Creating a list or plan

1. To generate a list or a plan, you have the following options:
   - Click with the right mouse button in the Project Explorer in the open project on the document folder of the list that you want to generate and select the command **Generate** from the context menu.
   - Click on **Report** in the menu bar and select the desired list.
2. This opens a wizard, with which you can configure and then create the list. The explanation of the configuration options can be found in the help (F1 key).
3. Click **Finish**. The list is created. Depending on the setting, an existing list can be overwritten or the new list can be appended to the existing list.
4. Finally, click **Finish**.

Overview of the various lists and plans

Cover sheet

The cover sheet contains information on the project, the unit and the customer. Most of this information can be found in the project properties.

WSCAD
Dieselstr. 4
85232 Bergkirchen

Customer:	Borough of Bergkirchen Sonnenstr. 1 Bergkirchen
Drawing number	01-03-7421
Order no.:	01037421-1

Project name: Suite Demo first steps	Nominal value: 10kW
Drawn by: harner	Unit type: IP65
Number of pages: 1	Location: Bergkirchen
Protection type: IP65	Date: 12.06.13
Norm: DIN 81346	

Summary

The summary is a list of all plans and lists contained in the project.

Content of DEMO drawn with Suite Ultimate

No.	File	Project page Current	Project page Total	Comment	Comment 1
1	WSCAD Demo_Deck.0005.wsCSD	1	49	Cover page	
2	WSCAD Demo_I.0001.wsSML	2	49	Summary	
3	WSCAD Demo_I.0002.wsSML	3	49	Summary	
4	SUITE DEMO_Stand.0001.wsREV	4	49	Revision history	
5	WSCAD Demo_fG.0001.wsGRD	5	49	Flow chart	
6	WSCAD Demo_Plan.0001.wsELD	6	49	Input	
7	WSCAD Demo_Plan.0002.wsELD	7	49	load circuit	
8	WSCAD Demo_Plan.0003.wsELD	8	49	PLC input/output	

Revision list

With WSCAD SUITE, it is possible to create projects with revision statuses. The revision list contains information about the changes, e.g., the name, user, date, description, etc.

Reference list

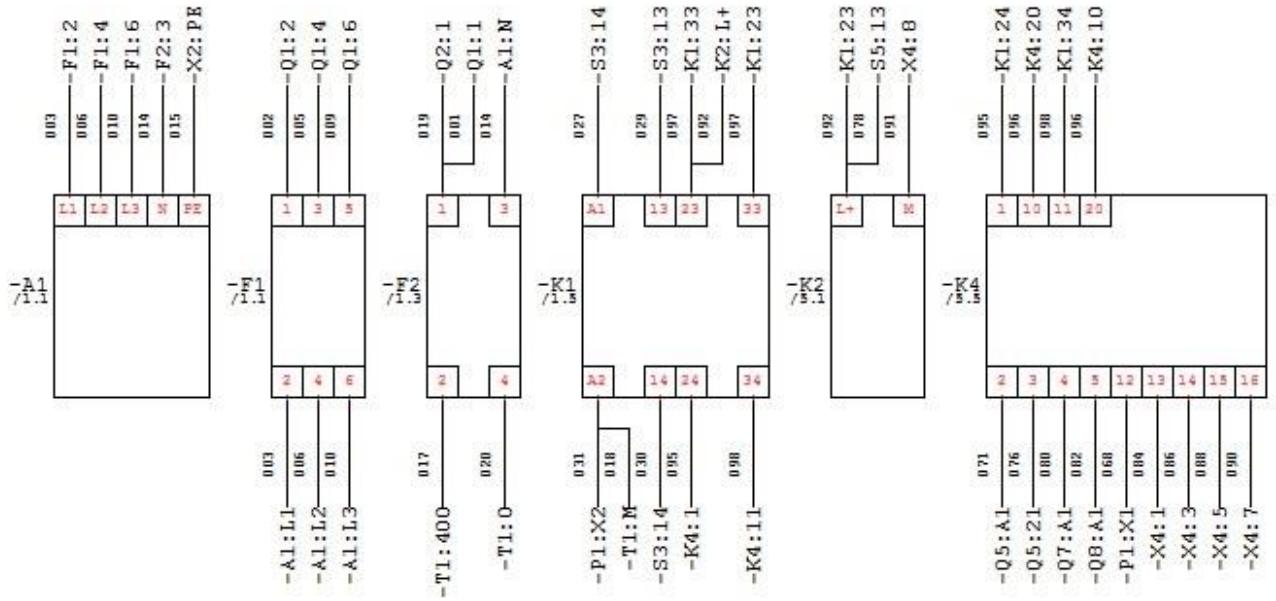
In the reference list, all pins of the individual components are listed, including the connected components.

reference list

Resource Name	Resource Connection	Resource Function text	Target Name	Target Connection	Target Function text
-K4	1,1L+	Output card	-K1	24,!b	
-K4	10,1M	Output card	-K4	20,2M	Output card
-K4	10,1M	Output card	-K2	M	supply PLC
-K4	11,2L+	Output card	-K1	34,!b	
-K4	20,2M	Output card	-K4	10,1M	Output card
-K4	20,2M	Output card	-X4	105	Terminal FACP
-K4	2	Output card			
-K4	3	Output card			

Reference chart

The reference chart presents a symbol with its connections and connection information in a very simple graphical display.



Material list

In the material list, all materials of the project are listed.

Material list

No.	Symbol name Function text QTY	Part number	References
1	Socket->bottom 1		-X3
2	Digital output SM 322-1BH00 16x24V DC Output card 1	6ES7322-1BH00-0AA0	-K4
3	Double level terminal block terminal valves 5	870-501	-X4
4	ugh terminal, 2, lat.label 3	870-901	-X1
5	ugh terminal, 2, lat.label terminal motor 9	870-901	-X2

PLC plan

The PLC plan is a list of all PLC-relevant information such as the ref. name, channel, PLC comment texts, SPS symbol texts, etc.

I/O PLC -K4

Address	Symboltext	Comment
D0.0		motor forward
D0.1		motor back
D0.2		wheel on
D0.3		Compressor
D0.4		
D0.5		
D0.6		
D0.7		
D1.0		installation on
D1.1		Table up
D1.2		Table down
D1.3		Material forward
D1.4		Material back

Terminal chart

The terminal chart lists all existing terminals in the schematics of the project.

Terminal chart	Cable name, external									Function text	Dest. external		Terminal number
								-W207 3x1,5mm²	-W7 4x1,5mm²		Name	Connection	
-X4									BU	Material back	-Q12	A2	104
-X4								2		shut-off fire	-RLT01-1S1	0V	105
-X5													1
-X5													2
-X5													3

Connector chart

The connector chart lists all plugs and sockets of the schematics of the project with information about the connected components.

Connector Chart -X3

Plug HDC HE-6 Female (screw) std. housing 1xPG16								Socket
	Target			Cable				
Sheet/path	Function text	Connection	Name	Wire	Name	Plug Connection	Function text Socket	Socket Connection
/2.6	Compressor	U	-M3	BK	-W3	1		1
/2.6	Compressor	V	-M3	BU	-W3	2		2
/2.6	Compressor	W	-M3	BN	-W3	3		3
/2.6	Compressor	PE	-M3	GNYE	-W3	4		4

Cable list

The cable list lists all existing cables in the schematics of the project with the cable name, cable type, cable destination, etc.

cable list

No.	field device	Cable name	Cable type	number of
1	-M1	-W1	NYM-J 4X2,5 NCC	4
2	-M3	-W3	NYM-J 4X2,5 NCC	4
3	-Q9	-W4	NYM-J 4X1,5 NCC	2
4	-Q10	-W5	NYM-J 4X1,5 NCC	2
5	-Q11	-W6	NYM-J 4X1,5 NCC	2
6	-Q12	-W7	NYM-J 4X1,5 NCC	2
7	-RLT01-1S1	-W207	NYM-J 3X1,5 NCC	2
8	-RLT01-B2	-RLT01-W12	LiYCY (TP) 2x2x0,5	2

Cable chart

The cable chart is a list of all individual wires with information about the connected components.

Cable -W1 Type: 4x2,5mm² part numbe NYM-J

Dest. external						
Name	Connection	Function text	Function text	Cable type	cable number	
-M1	U	load	Cable load	4x2,5mm²	BK	-X2
-M1	V	load	Cable load	4x2,5mm²	BN	-X2
-M1	W	load	Cable load	4x2,5mm²	BU	-X2
-M1	PE	load	Cable load	4x2,5mm²	GNYE	-X2

Wiring chart

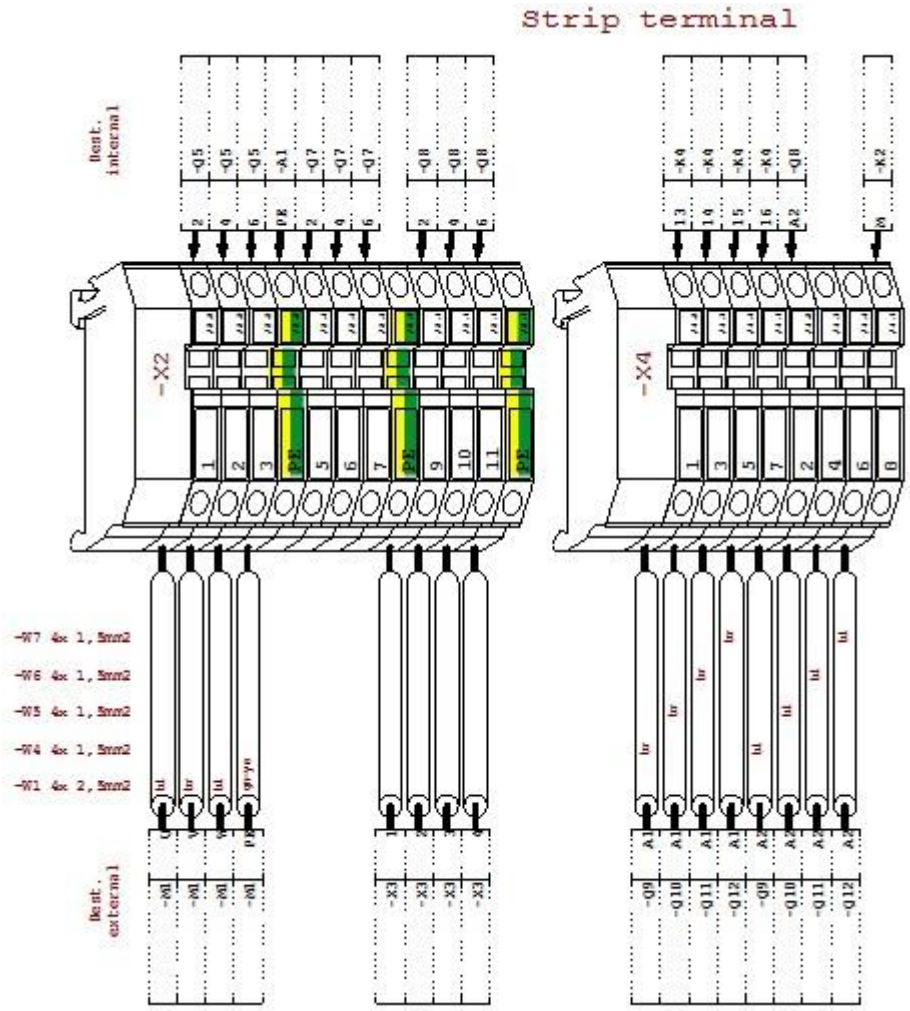
The wiring chart lists all individual wires in the schematics of the project. Each individual connection is listed with details on the first and second destination, wire name, cross-section, wire color and wire length.

Wires plan — Page 1

No.	Wire name	from (symbol:pin)	to (symbol:pin)	Wire color	Cross-section	Wire Length
1		-PE:1	-Q1:1			
2		-F1:1	-Q1:2			
3		-A1:L1	-F1:2	BK	1,5 mm²	
4		-F1:3	-Q1:4			
5		-A1:L2	-F1:4	BK	1,5 mm²	
6	L2	-Q2:3	-Q1:3			
7		-F1:5	-Q1:6			
8		-A1:L3	-F1:6	BK	1,5 mm²	
9	L3	-Q2:5	-Q1:5			
10		-PE:3	-A1:N	BU	1,5 mm²	
11	PE	-X2:PE	-A1:PE			
12		-T1:400	-PE:2			
13		-K1:A2	-T1:M	BU	1,0 mm²	
14	L1	-Q2:1	-PE:1			
15		-T1:0	-PE:4			

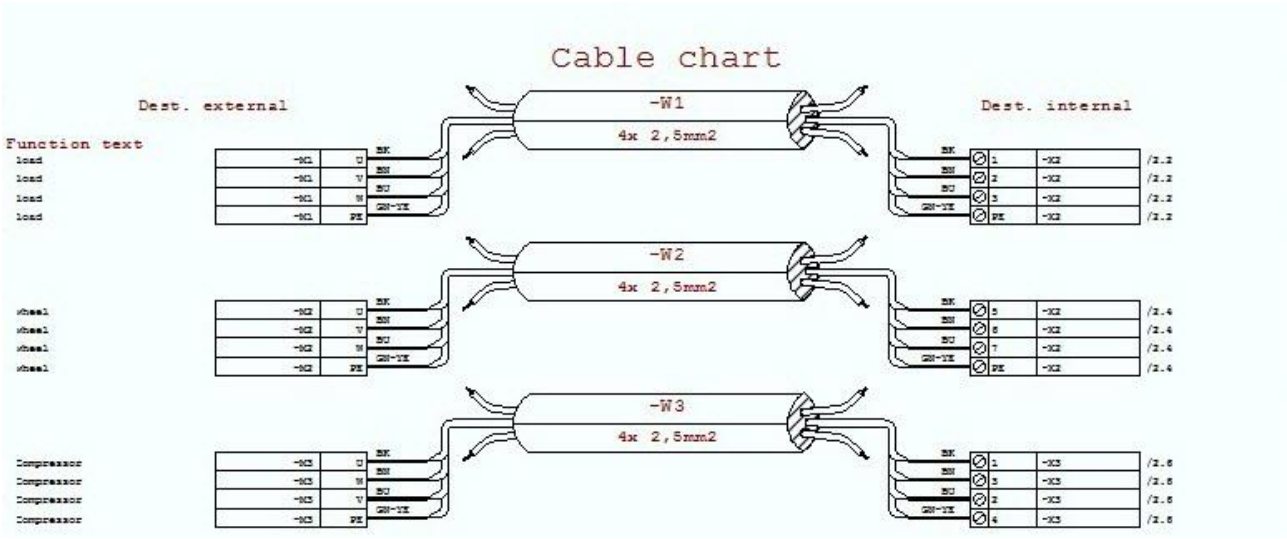
Graphical terminal chart

The graphical terminal chart shows all terminal strips and their terminals as well as the elements connected to the terminals.



Graphical cable chart

The graphical cable chart shows all the cables and their cable wires as well as the elements connected to the wires.



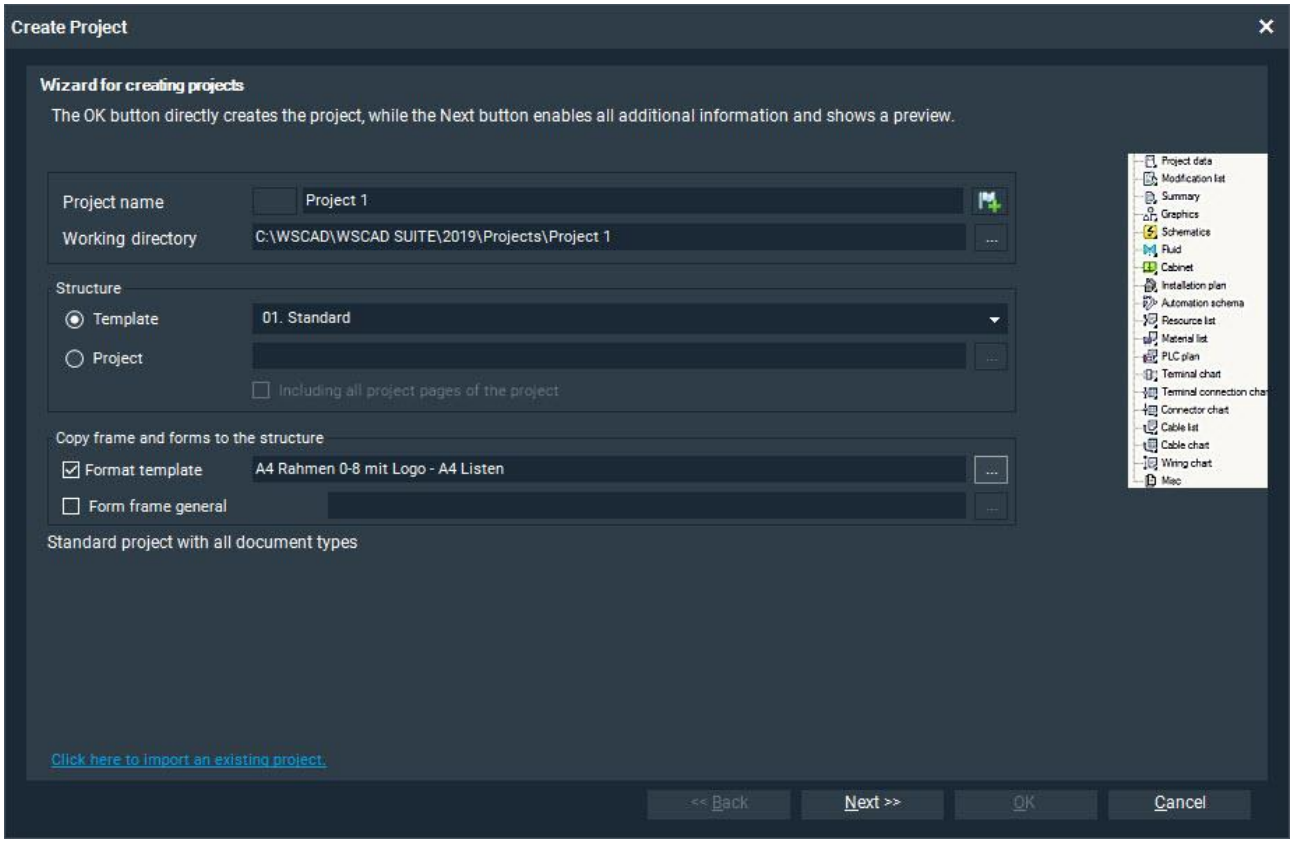
More functions

New project

To create new projects, you can use the **Create Project** wizard. You can create your project from a standard template or from custom templates or use an existing project as a template. In addition, you can optionally incorporate frames and forms in the structure and save it for further use.

Wizard Step 1: Project Name and Templates

In Step 1 of the wizard, you can specify the name of the project and define the templates.



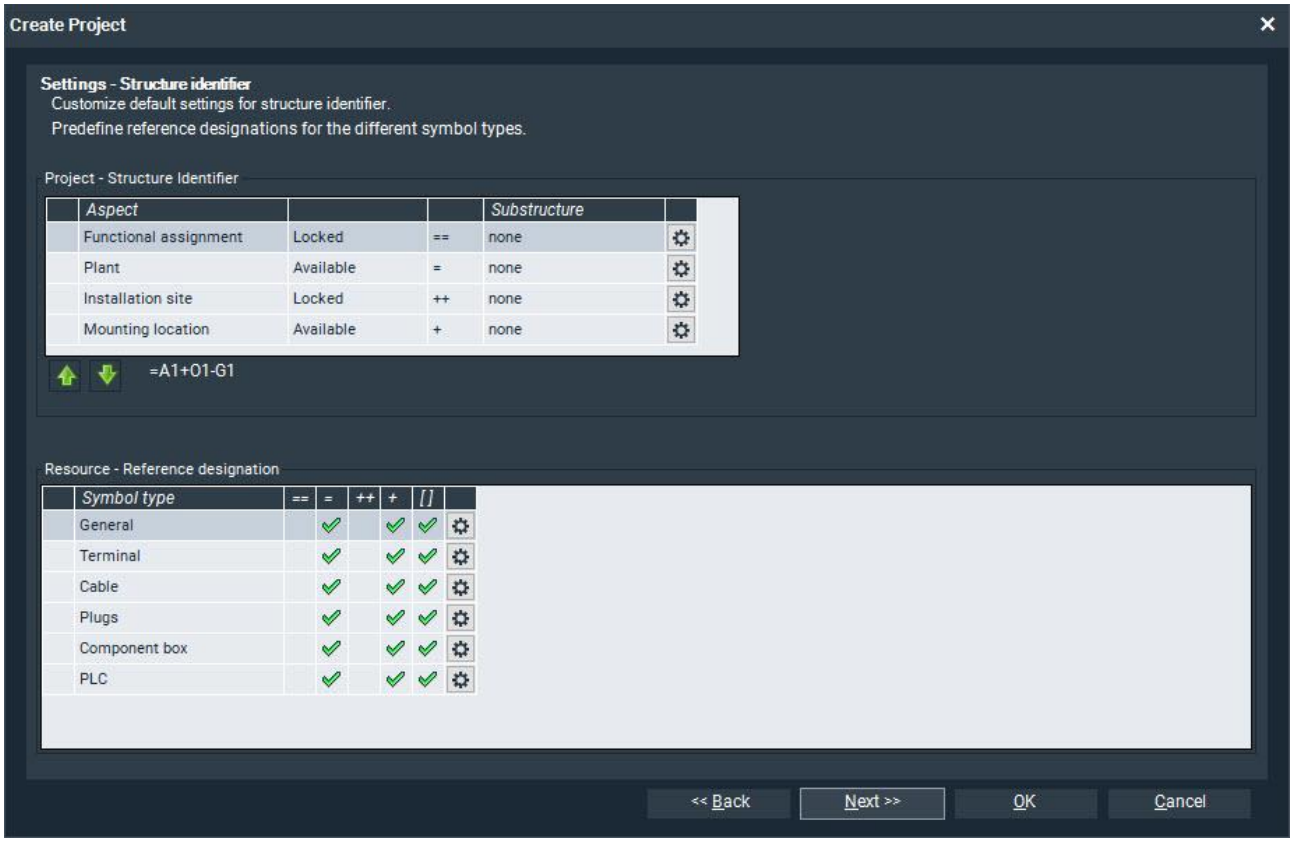
1. To create a new project, you have the following options:
   - Click on **Project** | **New** in the menu. The **Create Project** wizard (step 1 of 5) starts.
   - Click in the toolbar of the Project Explorer on the down arrow symbol button and select the menu item **New Project** from the context menu. The **Create Project** wizard is started.
2. First enter a name for your project in the **Project Name** field.
3. If you want to create your project in multiple languages, click on the flag button on the right of the input field. This opens the **Multilanguage edit** dialog. Click on the flag button and select the desired language. Enter the name of the project in the selected language in the input field at the top and click **OK**.
4. Use the preset default directory as the **Working directory** for your project or browse to the directory of your choice.
   Note: You can change the global storage location under **Tools** | **Settings (options)** | **Directories**.

5. Define the structure for your project in the **Structure** section:
   - If you want to use the default template or a custom template, select the radio button **Template** and choose a template from the drop-down list. The templates determine the layout and the display of the Project Explorer menus and the output lists contained therein. The structure of the template appears on the right.
   - If you want to use an existing project as a template, select the radio button **Project** and navigate to the location of the desired project. If you want the project pages of the existing project to also be taken over in addition to the structure, select the check box **Including all project pages of the project**.
     Note: This is useful if you are planning series productions and your projects differ only marginally from one another.
6. If required, you can select a format template in the **Copy frame and forms to the structure** area. To do this, select the check box **Format template** and click on the corresponding Browse button. The **Select format template** window opens. Click on the Browse button, select a template and then click on **Open**.
   Note: You can also edit the existing format templates and save them as your own format templates. For each listed file type, you can select alternative forms/frames via the Edit symbol button. Using the plus button, save the modified format template under a new name in a format template directory of your choice.
7. If required, you can select a special form frame only for the lists in the **Copy frame and forms to the structure** area. The format templates for the plans are not changed as a result. To do this, select the check box **Form frame general** and click on the corresponding Browse button. This opens the **Templates browser** window. Navigate to the form frame of your choice and click **OK**,
   Note: The form frame can be subsequently managed via the project properties (**Properties** menu item in the project context menu).
8. If you want to make additional settings, click **Next.** You are taken to step 2 of 5 (see Wizard Step 2: Aspects).
9. If you want to create the project without further settings, click on **OK** and finally on **Finish**.

Note: If you want to import an existing project, click on the link **Click here to import an existing project**. The Import Project **wizard** is called. You can now select an existing project and import it under a new name.

Wizard Step 2: Structure identifiers

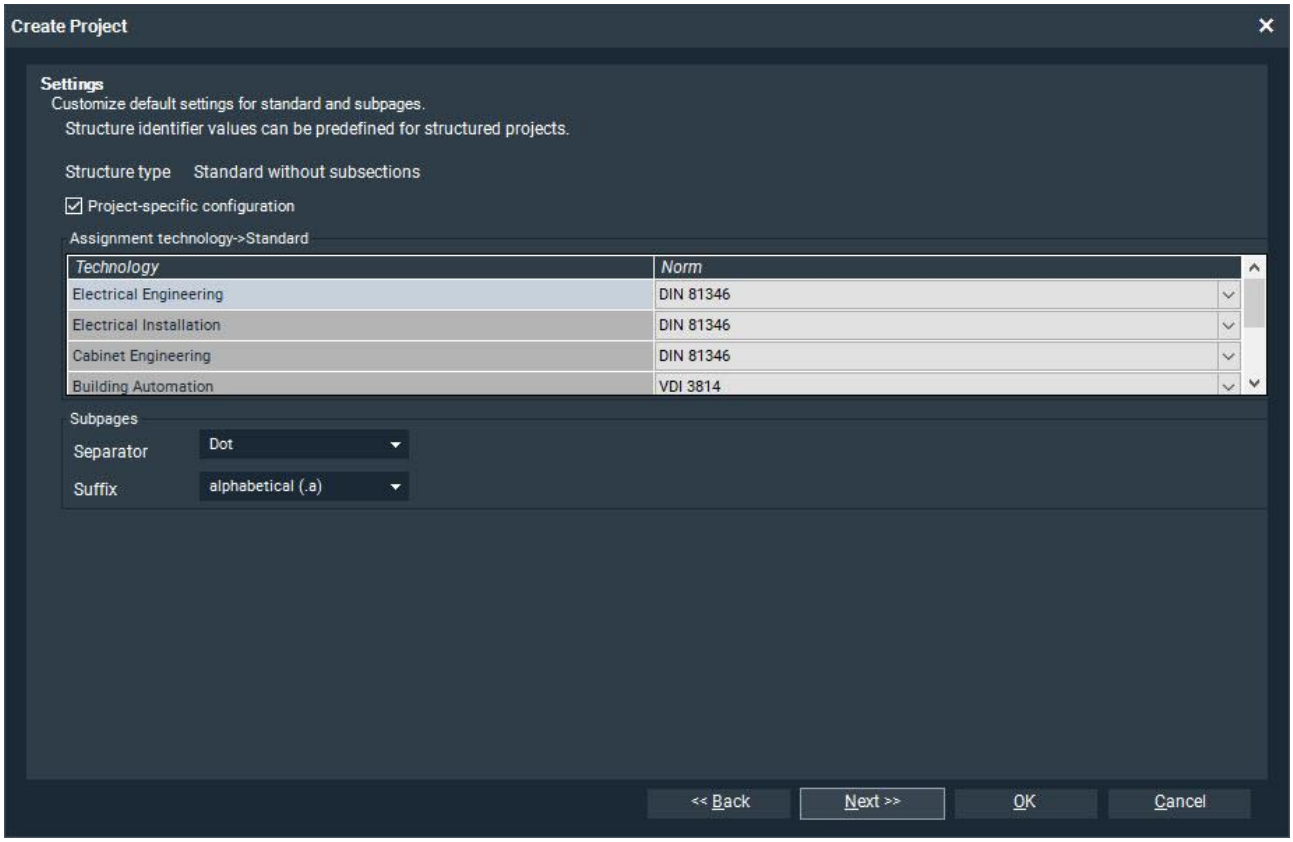
In Step 2 of the wizard, you can specify the structure IDs for the project and the resources.



1. In the **Project - Structure Identifier** section, you can configure which aspects of the project should be locked, available or structuring.
   - Click on the Edit symbol button for the desired aspect.
   - Select the desired aspect setting.
   - If you want to create a substructure, select the desired separator.
   - Finally, click **OK**.
2. Use the arrow symbol buttons to reorder the aspects as desired (e.g., ==, =, ++, + or ==, ++, =, +).
3. In the **Resource - Reference designation** section, you can specify for which resources the aspects are referencing and for which ones they are descriptive, and whether or not they should be inheritable in each case.
   - Click on the Edit symbol button for the desired resource.
   - Select the desired setting for each aspect.
   - Use the **Structure area** drop-down list to specify whether the aspects of the area should be inherited to the ref. name within a structure area.
   - Finally, click **OK**.
4. If you want to make additional settings, click **Next.** You are taken to step 3 of 5 (see Wizard Step 3: Settings).
5. If you want to create the project without further settings, click on **OK** and finally on **Finish**.

Wizard Step 3: Settings

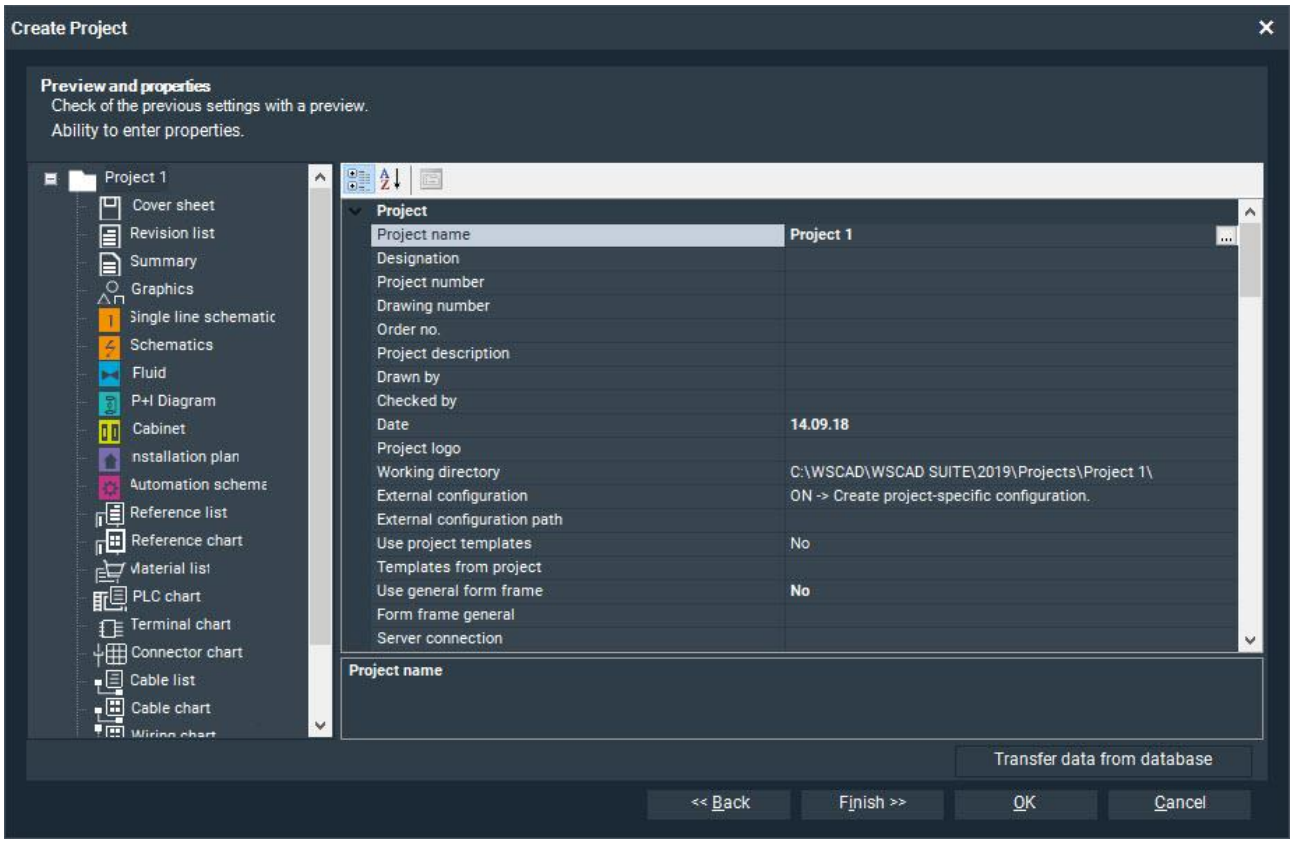
In Step 3 of the wizard, you can specify additional settings for the project.



1. If you want to create your new project on a project-specific basis and not globally, select the check box **Project-specific configuration** (Standard). The project-specific parameters are created.
2. Only for custom project template: If the parameter values defined in the project settings (e.g., under **Tools** | **Settings (options)** | **Symbol**) are to be transferred from the custom project template as well, select the check box **from selected project template**.
3. Only for custom project template:If the list configurations defined in the project settings (e.g., under **Tools** | **Settings (options)** | **Reporting** | **Material list**) are to be transferred from the custom project template as well, select the check box **List configuration**.
4. If necessary, assign the required standard to each discipline (technology) in the **Assignment technology -> standard** area.
5. In the **Subpages** area, use the drop-down lists to specify the **Separator** and the **Suffix** for the subpages. Subpages are usually required in existing projects where the page numbering may no longer be changed.
6. If required, click on the plus symbol button to create the structure IDs for the project in the **Structure ID** area depending on the project structure.
7. If you want to make additional settings, click **Next.** You are taken to step 4 of 5 (see Wizard Step 4: Preview and Properties).
8. If you want to create the project without further settings, click on **OK** and finally on **Finish**.

Wizard Step 4 & 5: Preview and Properties

In Step 4 and Step 4 of the wizard, you can view and edit the properties of the project and the document folders.



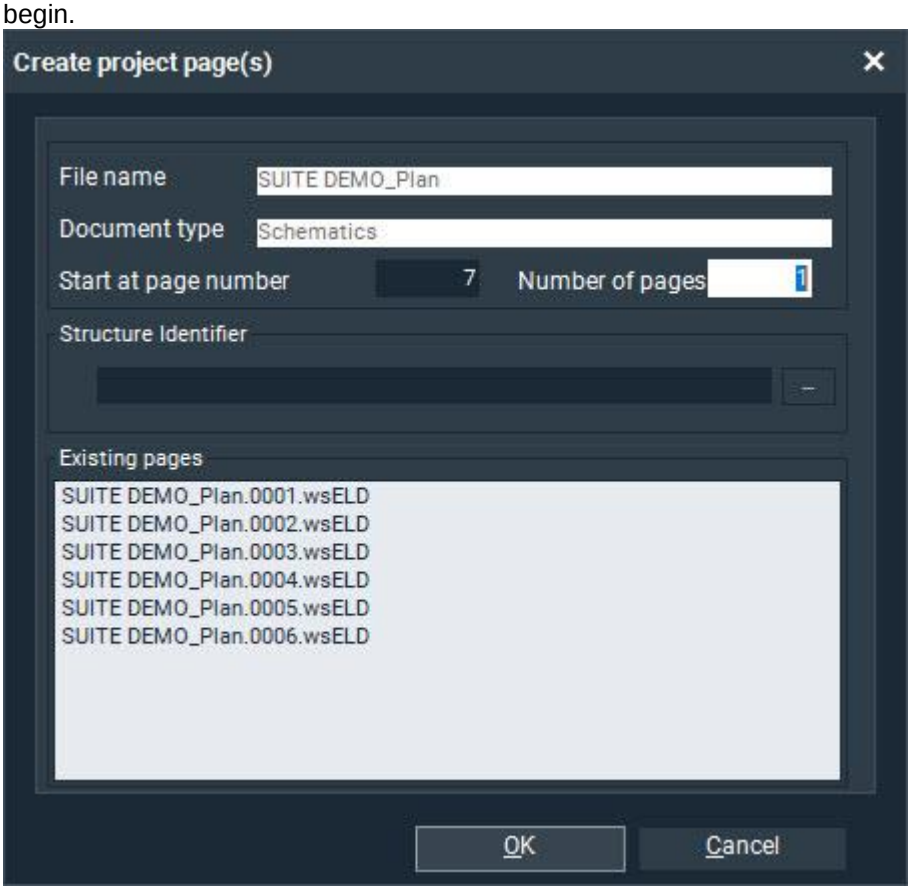
1. Navigate in the left pane in the project structure to the desired object. The properties of the object are displayed in the right pane. Properties marked in bold can be changed.
2. If you want to change the properties of an object, click in the right pane on an entry marked in bold, edit the preset text or path and then press the Return key.
3. Repeat steps 1 to 2 for further objects.
4. Click **Finish**. You are taken to step 5. The project is created.
5. Finally, click **Finish**.

New drawing sheets

A project may contain different document folders, e.g., the Schematics folder for the EE discipline, the Cabinet folder for the CE discipline and the Installation Plan folder for the EI discipline. For each document folder, new drawing sheets can be created, for example, multiple schematic pages.

Creating a new schematic page

1. Click with the right mouse button in the Project Explorer on the document folder **Schematics** of the open project and select the command **New page** from the context menu.
2. Specify how many schematic pages are to be created and with which number the new pages should begin.



3. Click **OK**. The new pages are created.
4. Click in the Project Explorer with the right mouse button on the first of the newly created schematic pages and select the command **Properties** from the context menu.
5. In the **Properties Schematics** dialog, enter the name of the schematic page in the **File contents** field.
6. Click **OK**. Repeat steps 4 and 5 for the remaining schematic pages.

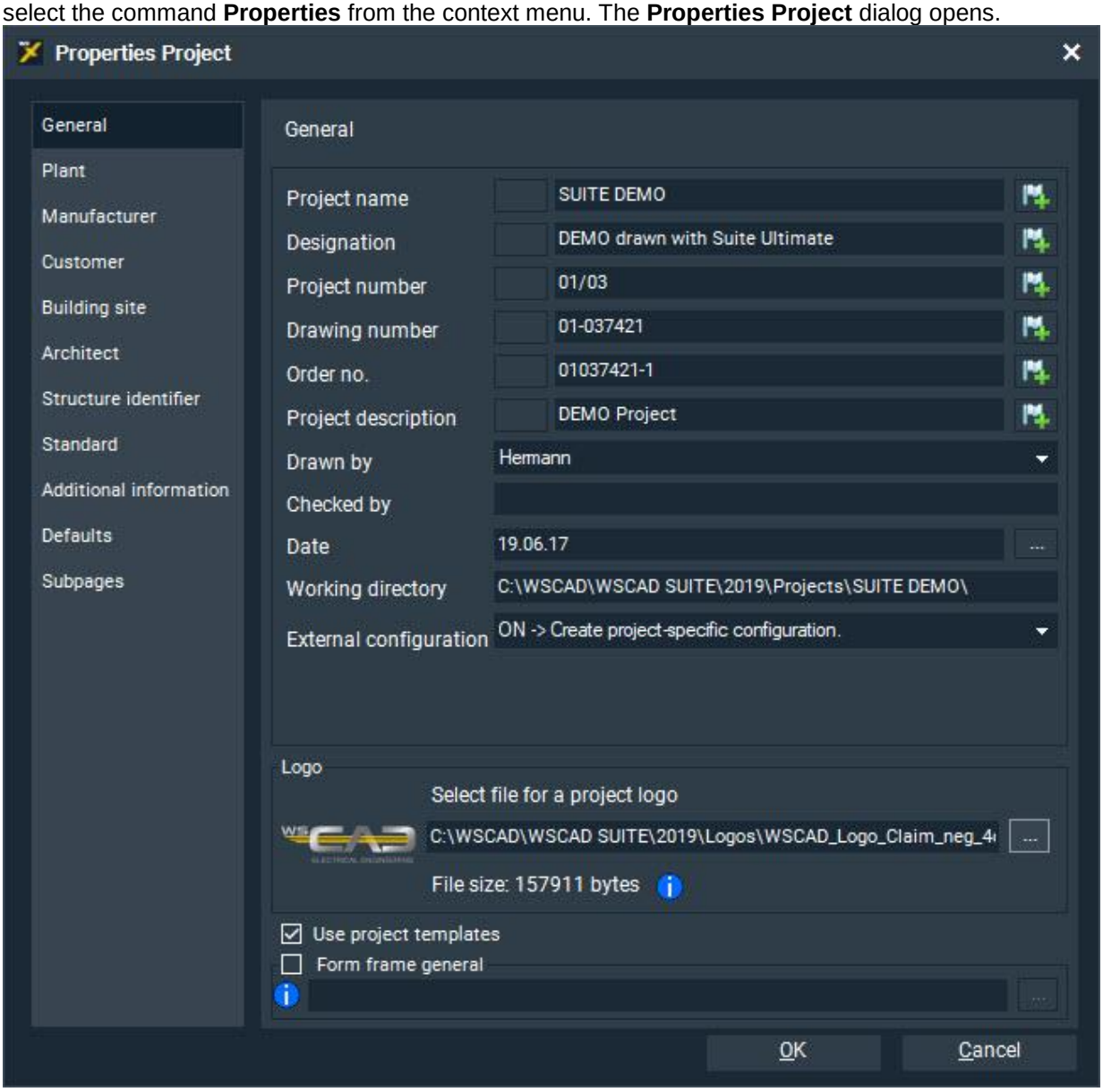
Company logo

You can specify a company logo for the project that will appear within the frame on all pages of the project.

Under **Extras** | **Settings (options)** | **View**, you can specify a global logo that applies to all projects. If a project logo is specified, the global logo will be overwritten.

Specifying a project logo

1. Click in the Project Explorer with the right mouse button on the project name of the open project and select the command **Properties** from the context menu. The **Properties Project** dialog opens.



2. Click in the **Logo** area on the Browse button and select the company logo. The selected logo will be displayed as a preview graphic in the Properties dialog of the project.
3. Click **OK**.

Changing the displayed logo

If the project logo is not displayed in the frame, it is possible that the placeholder for the logo in the frame is set to display the global logo instead of the project logo. This setting must therefore be changed.

1. Open the drawing sheet and select the frame.
2. Click with the right mouse button on the frame and select the command **Edit in editor** from the context menu. This opens the Frame editor.
3. Right-click in the drawing area of the Frame editor on the text allocator for the logo and select command **Properties** from the context menu.
4. Select the radio button **Project logo**.

5. Click **OK**.

Potentials

You can change the line names and the connection names of potential lines. The line name denotes the potential of the line, e.g., L1, L2, L3 or PE. The connection name indicates the connection of the potential line from one page to another page.

Prerequisite: A schematic is open.

1. Click with the right mouse button on a directional arrow of the potential line and select the command **Properties** from the context menu.
2. Edit the line name in the **Properties Termination Point Symbol** dialog by entering a new name or by selecting an existing name from the drop-down list.
3. Change the connection name if required. By default, a connection from page 1 to page 2 is preset with `1`, a connection from page 2 to page 3 with `2`, and so on.
4. Click **OK**.

Macros

You can insert macros into a drawing sheet. Macros consist of multiple symbols and the related connections, including tags and component properties. Frequently used circuit parts or even entire circuit pages can be saved as macros in the macro library.

Prerequisite: A drawing sheet is open.

1. To take over a macro in the drawing sheet, you have the following options:
   - Click on **Insert** | **Drawing macro** in the menu bar. The Drawing Macro Browser opens.
   - Click on the **Drawing Macro Explorer** tab in the right additional window.
2. Double-click In the menu tree of the macro library on the desired macro, e.g., on the macro **Supply 5p** in the **Supply and Phase** folder. The symbol "hangs" on the mouse pointer.
3. Move the mouse pointer to the desired position in the drawing sheet.
   Note. You can cancel the placement with the right mouse button.
4. Place the macro by clicking the left mouse button or by pressing the Return key.

Destination wiring

The wiring of the schematic is done through destination wiring. On placing the elements, their pins are automatically connected to the lines, and destination wiring elements are inserted. In order to use this function optimally, snap (F5 key) and orthogonal drawing mode (F6 key) should be turned on so that the elements are placed within the grid.

The following destination wiring elements are available:

- Angle
- T- pieces
- Jumper
- Cross
- Arrows (termination point symbols)

Destination wiring elements can be moved, deleted and changed. You can insert new destination wiring elements into the drawing sheet via the **Destination wiring** toolbar.

Moving a destination wiring element

Prerequisite: A schematic is open.

1. Click with the right mouse button on the destination wiring element.
   Note: if snap (F5 key) is turned off, the destination wiring element can be grasped more easily.
2. Select the command **Move** from the context menu. The crosshair will change to red and be located within a blue square.
3. Leave the crosshair within the square and click the left mouse button. As long as you hold down the mouse button, you can move the destination wiring element.
4. Release the mouse button. The destination wiring element is placed.

Changing a destination wiring element

Prerequisite: A schematic is open.

1. Click with the right mouse button on the destination wiring element.
   Note: if snap (F5 key) is turned off, the destination wiring element can be grasped more easily.
2. Select the command **Destination wiring** from the context menu and then select the desired destination wiring element from the list.
3. The old element is replaced by the newly selected element.

Placing a new destination wiring element

Prerequisite: A schematic is open.

1. Click in the **Destination wiring** toolbar on the desired destination wiring element.
   Note: You can choose other variants of the element via the arrow symbol button on the right of the destination wiring element.
2. Move the mouse pointer to the desired position in the drawing sheet. Place the element by clicking the left mouse button or by pressing the Return key.
3. If you want to place more elements, repeat steps 1 and 2. The symbol will still be hanging on the mouse pointer.
4. If no further elements are to be placed, cancel the action with the right mouse button.

Note

After placing a termination point symbol (arrow), the properties dialog of the symbol opens. You can define the line name (e.g., L1, L2, PE,...) there. The designations are necessary to enable the potentials to be properly wired and their cross-references to be correctly assigned.