Training WSCAD Engineer

Practice and Theory

SUITE X

WS CAD

Training – Day 1

- User interface
- Project Management – Part 1
- Symbol Manager
- Drawing
- Symbol-oriented work

WS CAD
ELECTRICAL ENGINEERING

Training – Day 2

- Preparations for the training project
- Part-oriented work
- Symbol Editor
- Using macros
- Terminal management
- Plug management

WS CAD
ELECTRICAL ENGINEERING

Training – Day 3

- Contactor management
- Creating macros
- Parent/Child Manager
- Cable management

Training – Day 4

- PLC management
- Cabinet
- Post-numbering

WS CAD
ELECTRICAL ENGINEERING

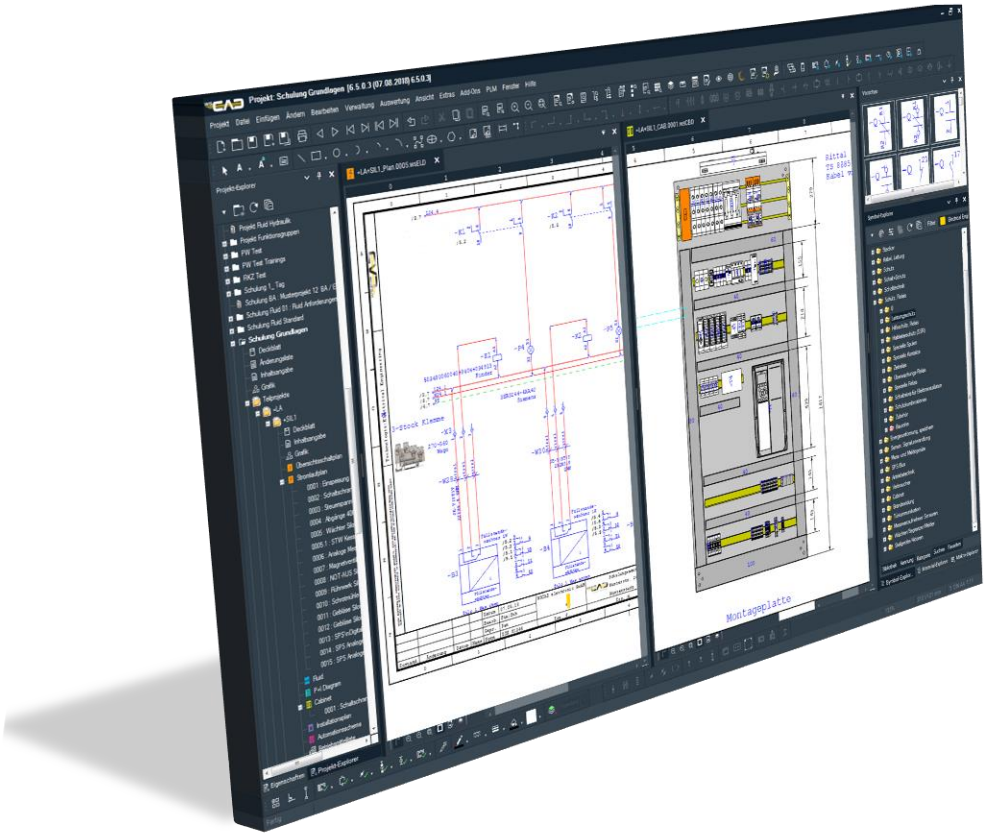
Training – Day 5

- Evaluations
- Documentation
- Creating new components
- Settings

Exercise 1 – User interface

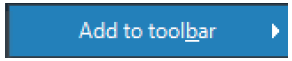
- Theoretical Section: **User interface**

- Please devote your attention to the instructor. Thank you!



Exercise 1 – User interface

- Call WSCAD.
- Customizing the user interface:
  1. Move the "Preview" window to the left and dock it there above the Project Explorer.
  2. Save this new work surface via the menu item "View | Work area | Save current state".
- Toolbars
  1. Create a new toolbar via the menu item "View | Toolbars | Customize" and by then clicking the button "New" on the right, and save it with a name.
  2. "Fill" this new toolbar with a menu command via "View | Toolbars | Customize | New toolbar" and by moving a menu command from right to the left, e.g., "Project | Export | PDF".
  3. "Fill" the new toolbar with an icon.

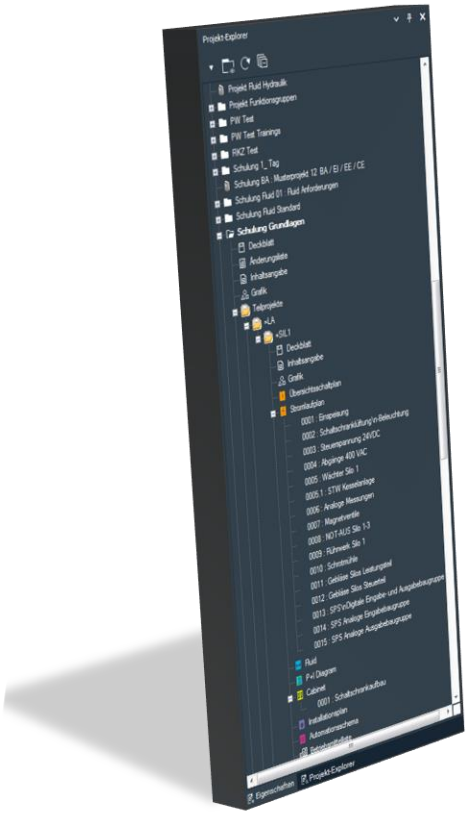
     To do this, go to the Symbol Explorer and select a single icon there, open the context menu with a right-click, go to Add to toolbar 

     and then select your new toolbar.

Exercise 2 – Project Management

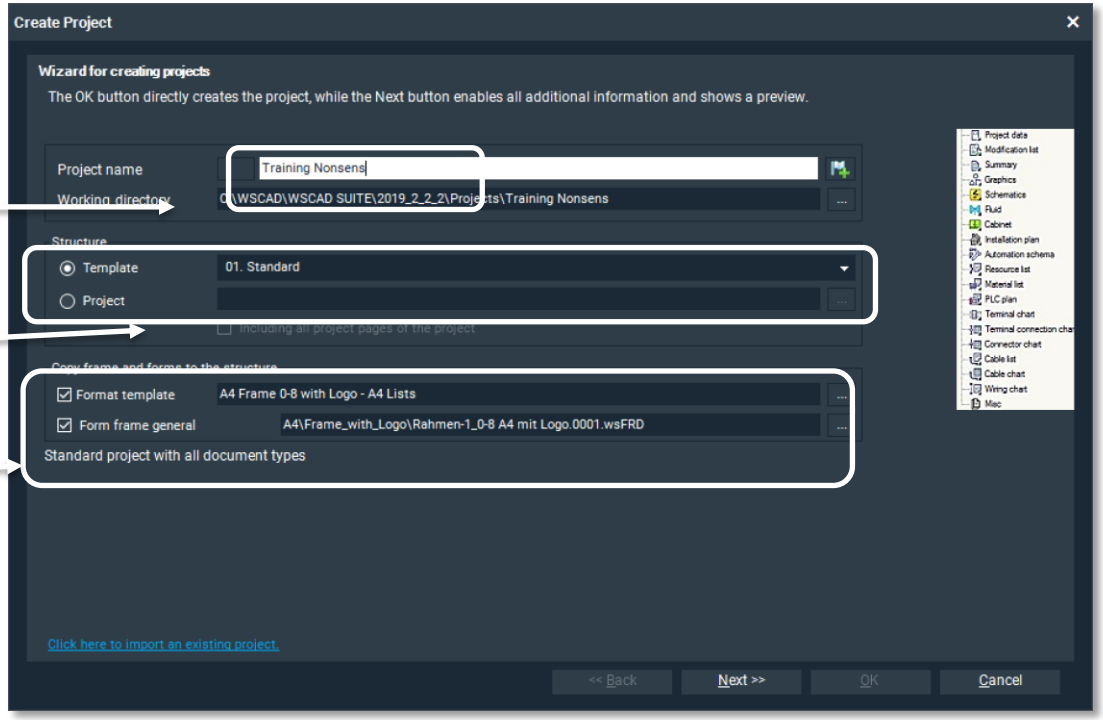
- Theoretical Section: **Project Management**

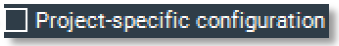
- Please devote your attention to the instructor. Thank you!

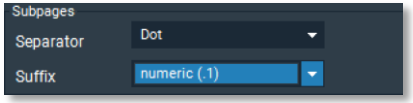


Exercise 2 – Project – New

- Create a new project
  - Menu item "Project | New"
  - Project name:
    - "Training nonsense"
  - Structure:
    - Select template
      - Template "01.Standard"
  - Frames and forms
    - Click "Next"
  - Next Frame
    - Click "Next"
  - Next Frame
    - Do not click
    - Set like this
    - Click "Next"
    - Click "Finish"

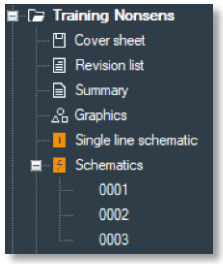




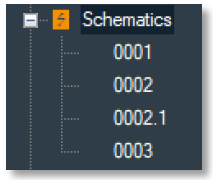


Exercise 2 – Project – New

- Create 3 pages under "Schematic"
  - Right-click on the document folder "Schematic"
    - New page
    - Enter "3" for the number of pages
    - => OK



- Add a subpage for page 2
  - Right-click on page 2
    - New subpage
      - Select "1" as the sub-level
      - OK

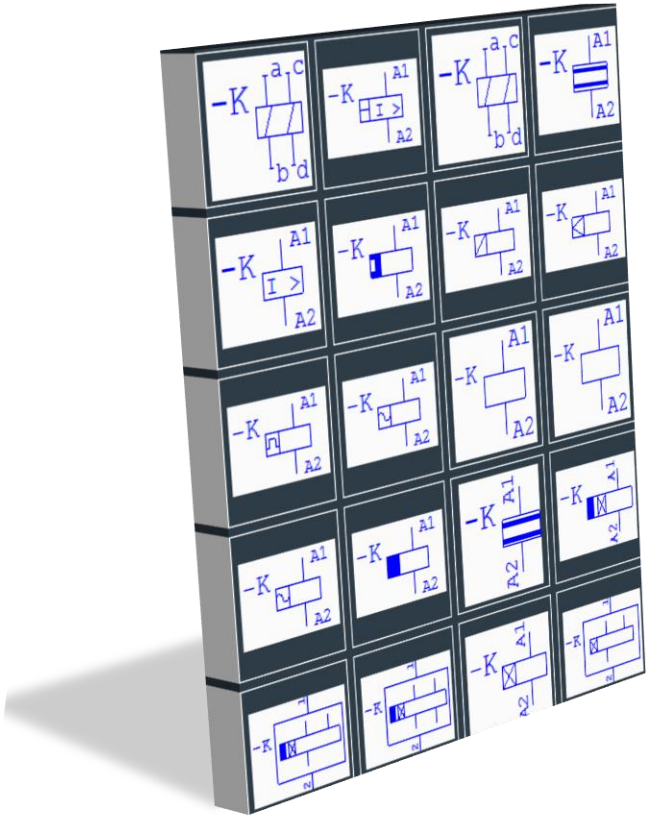


- Look at the structure of the schematic
- Delete the subpage "0002.1" and page 3

Exercise 3 – Symbol Manager

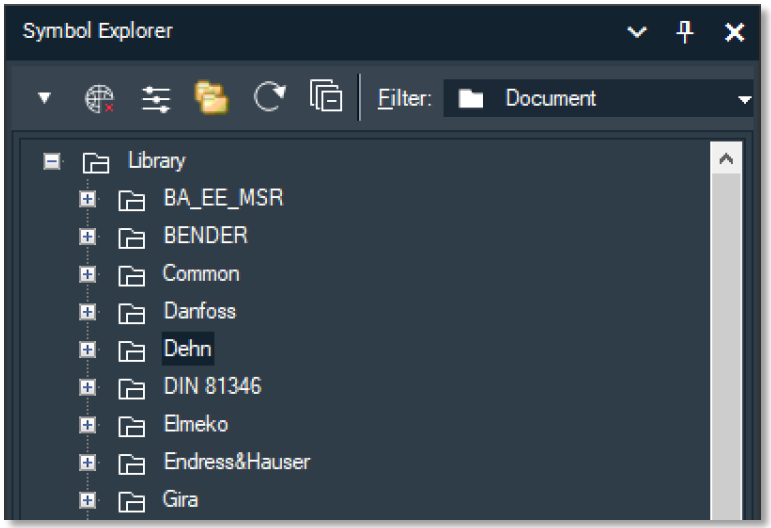
- Theoretical Section: **Symbol Manager**

- Please devote your attention to the instructor. Thank you!



Exercise 3 – Symbol Explorer

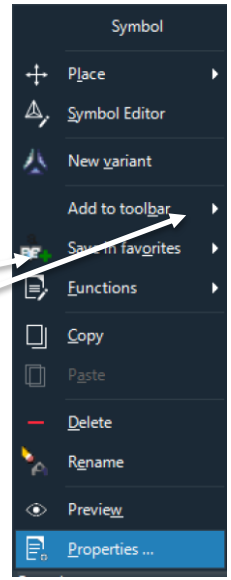
- Open the "Training Nonsense" project
- Open the first schematic page
- Filter settings in the Symbol Explorer:
  - Set various filter settings
    - Discipline
    - Document
    - Full / All folders
    - Norm filter
    - Online library
  - Set at the end:
    - Document
    - Full folders
    - Standard filter On



Exercise 3 – Symbol Explorer

- Take a look at the different views of the Symbol Explorer
  - Try out the different views
    - Library (default view)
    - ID
    - Category

  - Try the "Search" and observe the search result
    - Search criterion: "Fuse"
    - Search criterion: "Fuse*"
    - Search criterion: "*Fuse"
- Try the "Category" view
  - Select category
    - e.g.: Switches
  - Select subcategory
    - e.g.: Emergency switch
- Save favorites
  - Right-click on the emergency switch above and save this in the Favorites
  - Place this symbol into your new toolbar as well

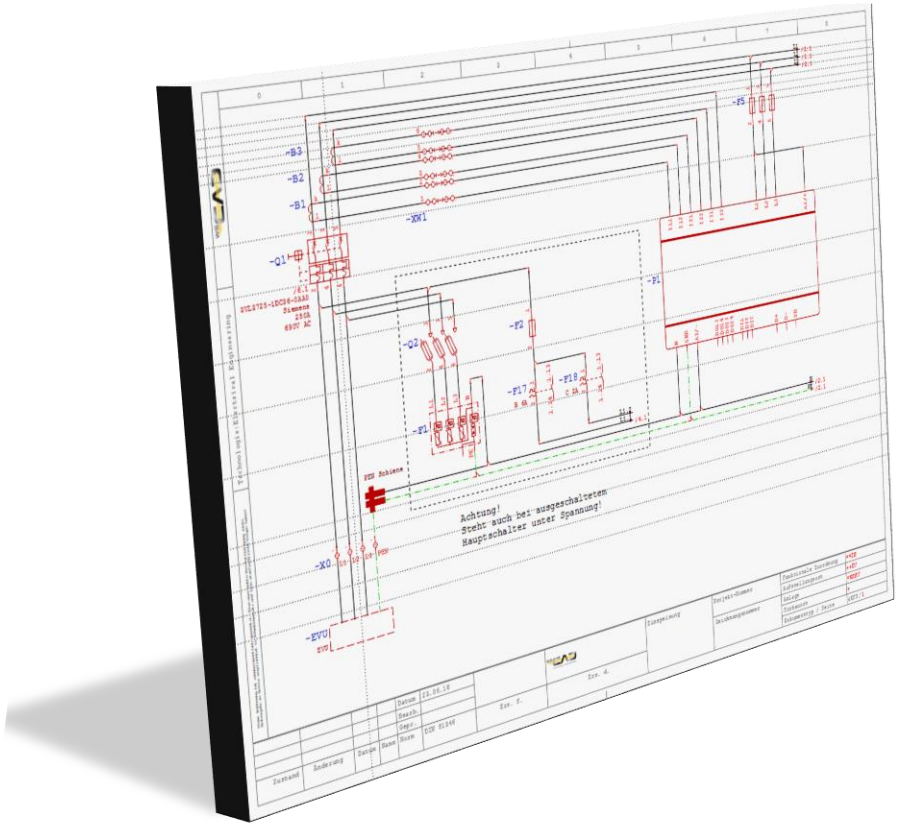


Exercise 4 – Drawing Sheet 1/2

- Theoretical Section: **Basic Settings Stick**

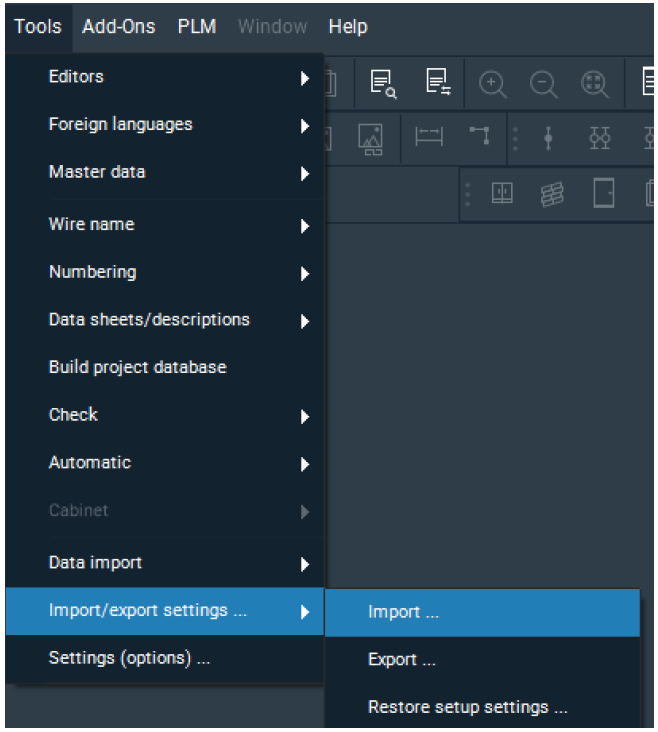
  Drawing

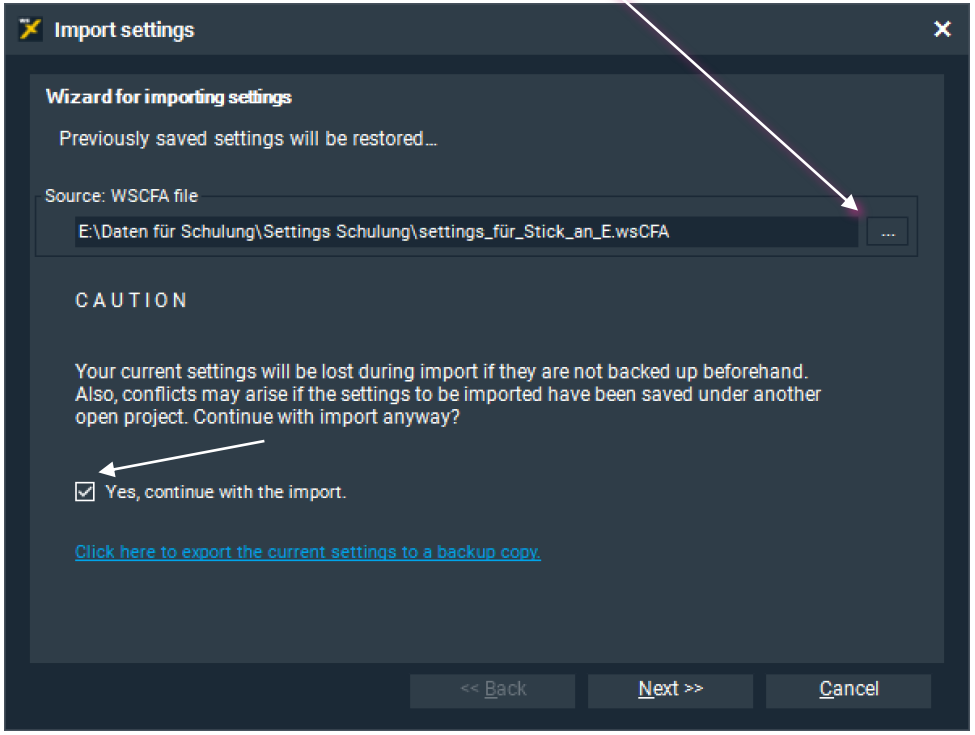
- Please devote your attention to the instructor. Thank you!



Basic Settings

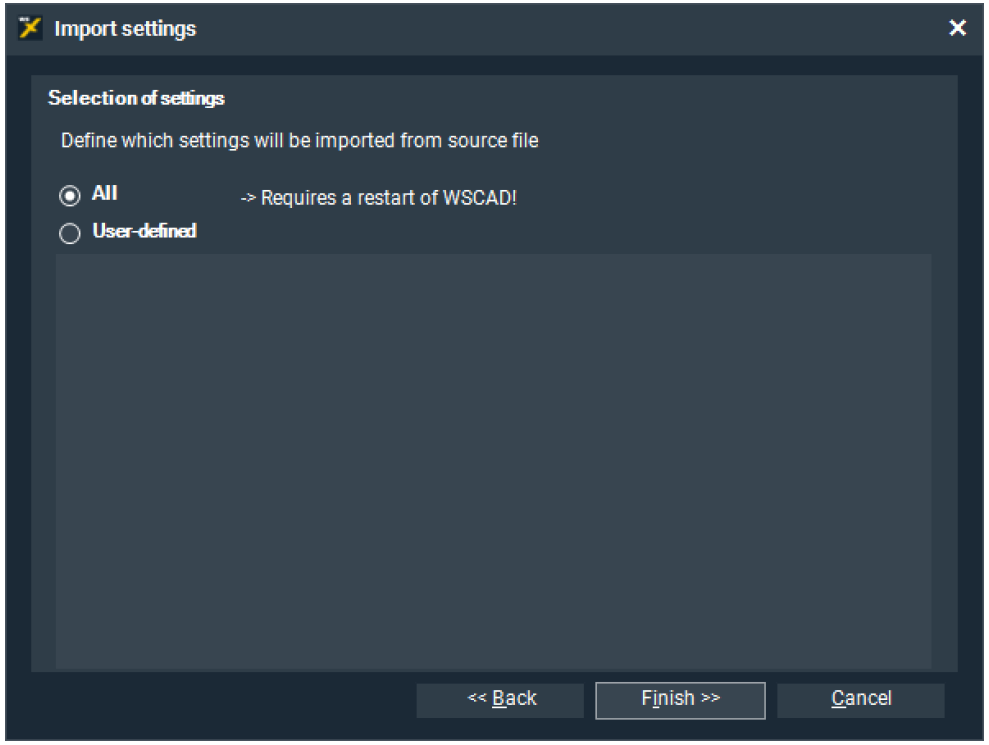
- For the training in the WSCAD training center, the basic settings should be centrally available.
- For all other training (on site), plug the WSCAD stick to your PC
  - Go under menu Tools on Import
  - Then select on the stick the settings file, according to the drive letter of the inserted stick





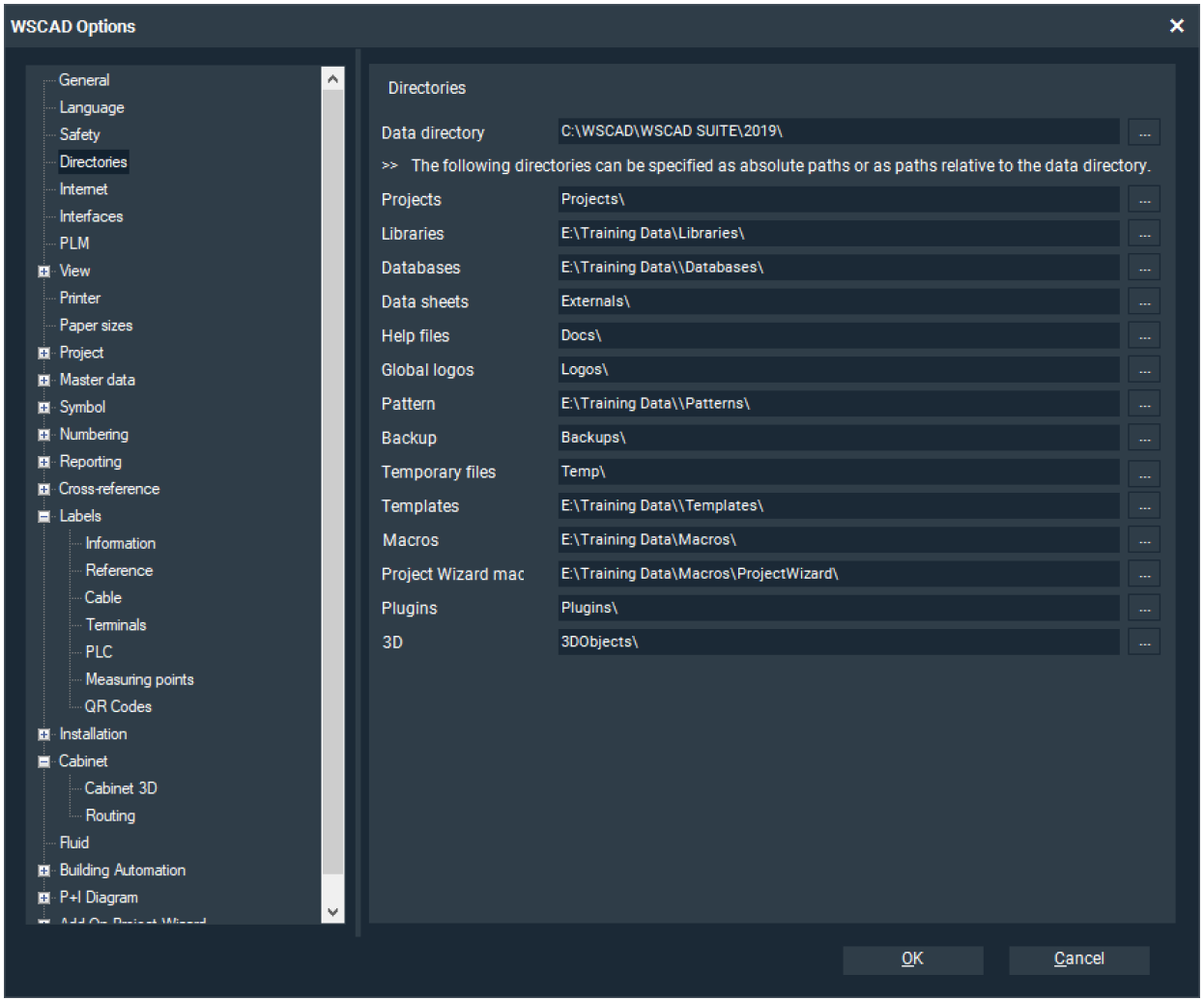
Basic Settings

- Select "All" and click on Finish



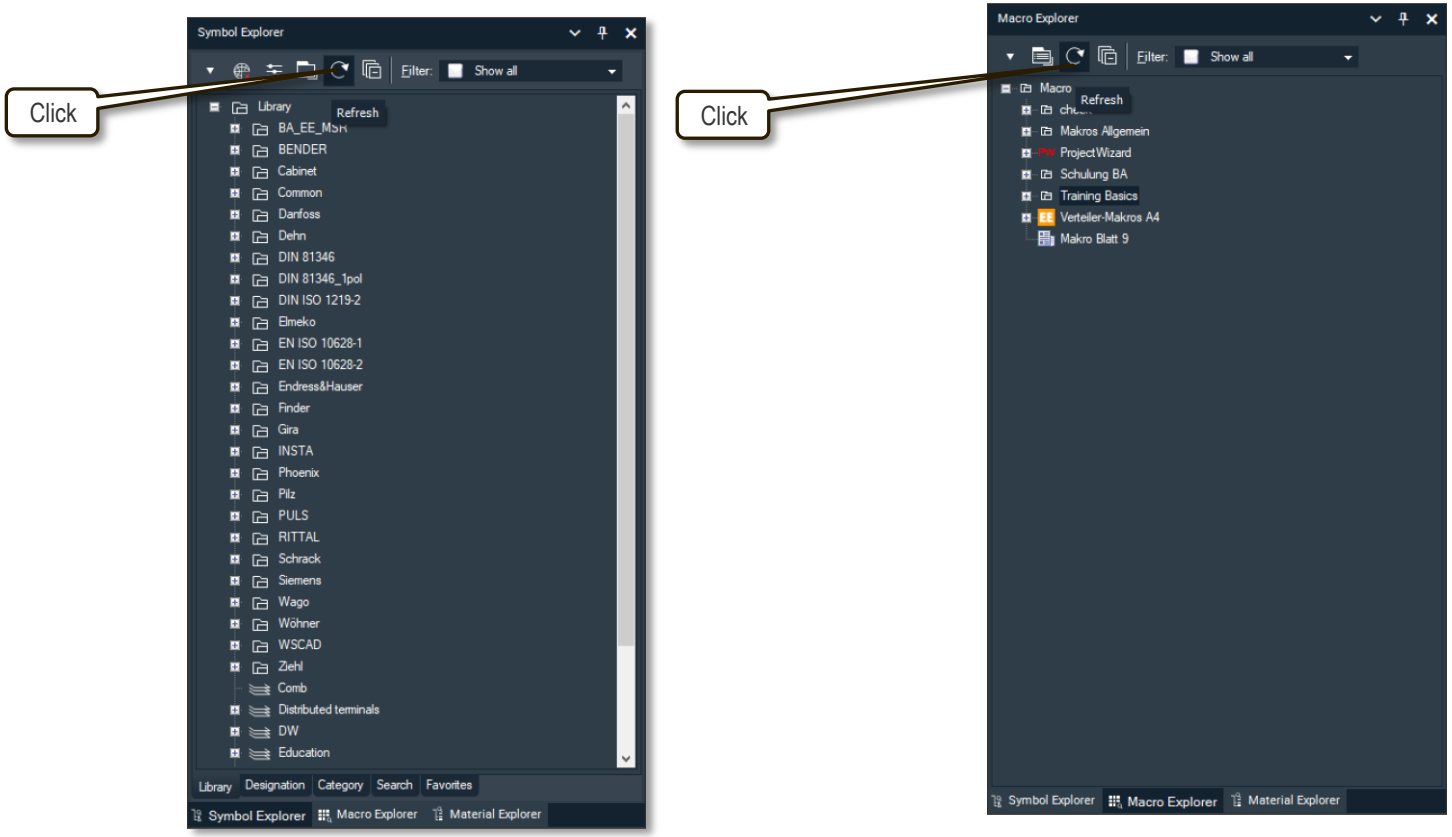
Basic Settings

- Control of the most important settings :
  - Under Menu Extras, go to Settings / Directories.



Basic Settings

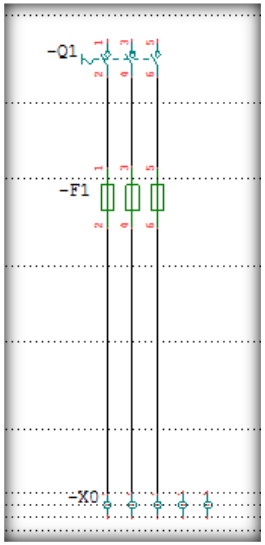
- Control of the most important settings :
  - Update the following windows :



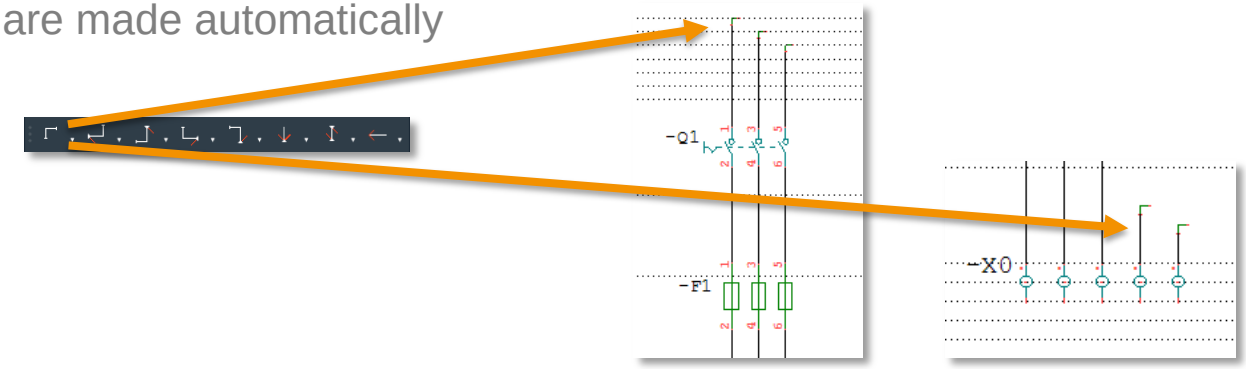
Exercise 4 – Drawing Sheet 1 Project Nonsense

- In the following, a drawing is to be created, as shown on slide 23
- Open the "Training Nonsense" project
  - Open the schematic page 0001
    - Zoom to full screen (press the F4 key)
    - Turn on Snap (F5)
    - Turn on Orthogonal mode
    - Turn on Auto-connections
    - Disable the Terminal Manager (red cross visible)

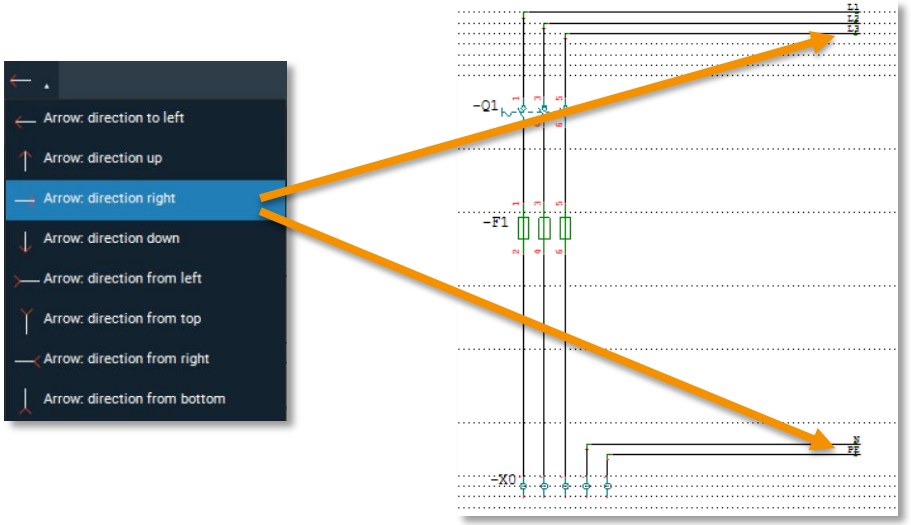
- Placing components from the toolbar
  - Insert symbol for component "Q1"
    - Place with the left mouse button
    - Simply confirm Properties dialog with OK
  - Insert symbol for component "F1"
  - Insert symbol for standard terminal "X0" (left)
    - "Green rubber thread" signals resulting connection
    - Simply confirm Properties dialog with OK
    - Connection to Q1 Pin2 is generated
    - Insert 4 more terminals
      - Assignment to X0 occurs automatically
      - Connections to Q1 are generated



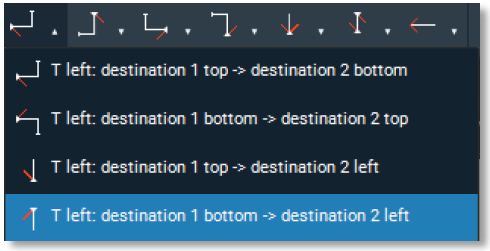
- Place the "corner symbols" above Q1 and below for N and PE
  - "Green rubber threads" signal the resulting connection
  - The connections are made automatically



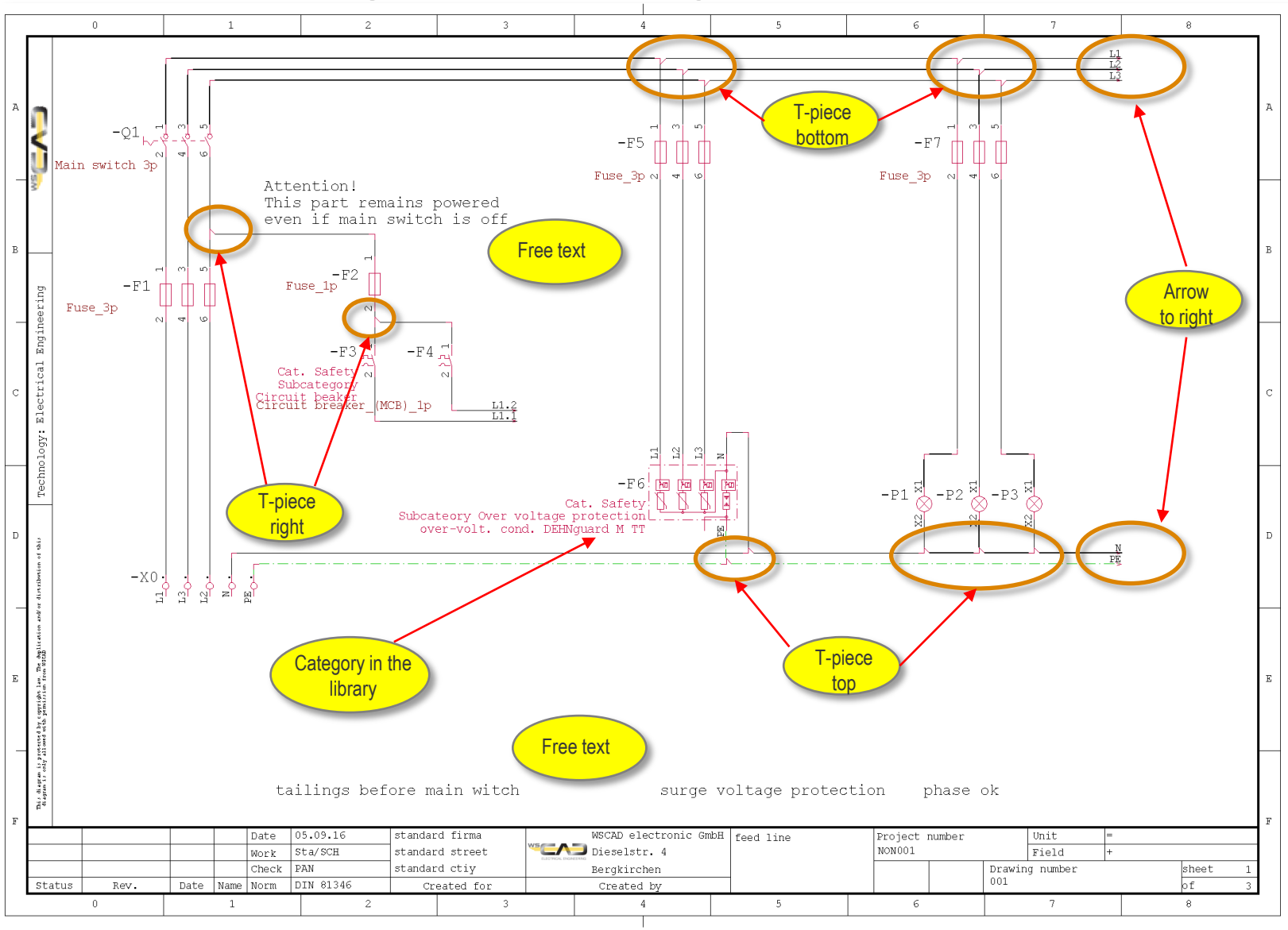
- Place the termination point symbol "Right arrow"
  - Destination wiring toolbar
    - Arrow: direction right
  - Enter the name "L1"
  - OK
  - Place "L2, L3, N, PE" accordingly

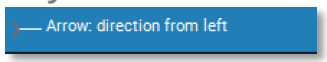


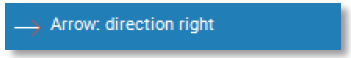
- Place "F2" using the Shift key
  - No connections to the potentials are generated
- Place "F3"
  - Locate the symbol in the symbol library
    - Use the details from the schematic template
    - Click the > symbol to display in the preview window
    - Place
      - Double-click on symbol name or
      - Right-click on symbol name > Place or
      - Click in the preview window
- Place the branches under "Q1" to "F2"
  - Pay attention to the correct symbol "T-piece right"
  - Common connection "Top"
  - Destination 1 = "Below"
  - Destination 2 = "Right"
- Complete the drawing according to the template in slide 23



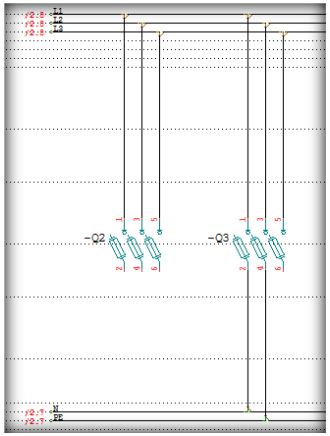
Exercise 4 – Drawing Sheet 1 Project Nonsense



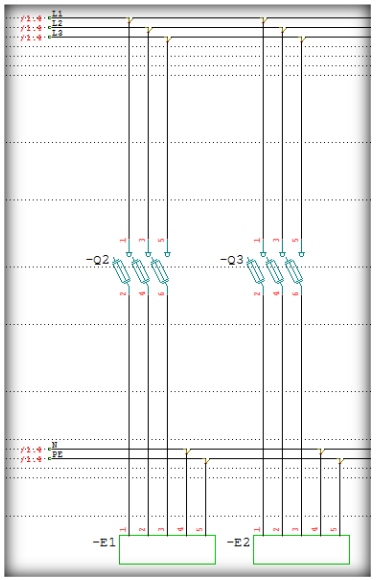
- In the following, sheet 2 of the project is drawn, as shown on slide 25.
- Place –Q2
- Draw the potentials L1, L2, L3, N, PE on the page
  - Place the left termination point symbols for direction from left
    - Arrow: direction from left 

  - Add the right termination point symbols on the page
    - Arrow: To the right 

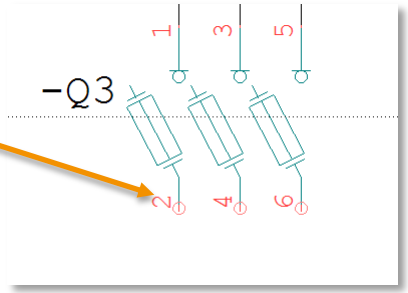
  - Q2 does not automatically connect to the potentials L1, L2, L3
- Place "T-pieces downwards" to connect to –Q2
- Tip: Always start with the potentials/signals to be inserted at the top margin of the sheet
- Place Q3
  - Q3 will automatically connect to L1, L2, L3
  - Unnecessary connections to N, PE are also generated – do not delete



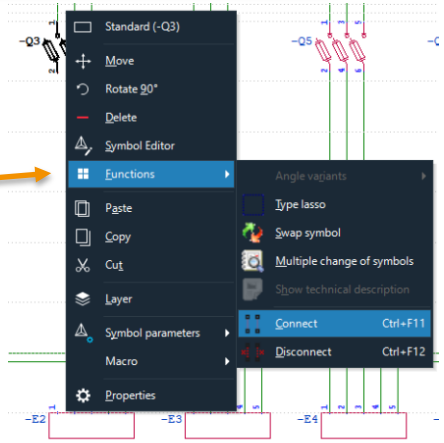
- Place E1
  - Connections to Q2 are automatically generated
  - Not N and PE
  - Connect E1 to N and PE using "T-pieces downwards"
- Place E2 as a copy of E1
  - Select E1
  - "Ctrl-C; Ctrl-V"
  - Move the cursor until the "rubber thread" appears on pins 3 to Q2
  - Place with a click
  - Now delete the destination wiring symbols of N and PE to Q2
    either by selecting the T-pieces individually
    and then deleting them or by drawing a window around both T-pieces
    ATTENTION: Always draw a window from left to right!
  - Connections Q2 to E1 are generated
  - Place missing T-pieces for N and PE downward



- Delete the connections from E2 to Q3
  - Select the connecting lines
  - Press the Delete key, or right-click on "Delete"
  - Note: The auto-connection function of the component pins is switched off
    - **"Red circle" at the pins**
    - **No connections to pins with the "circle" can be made**

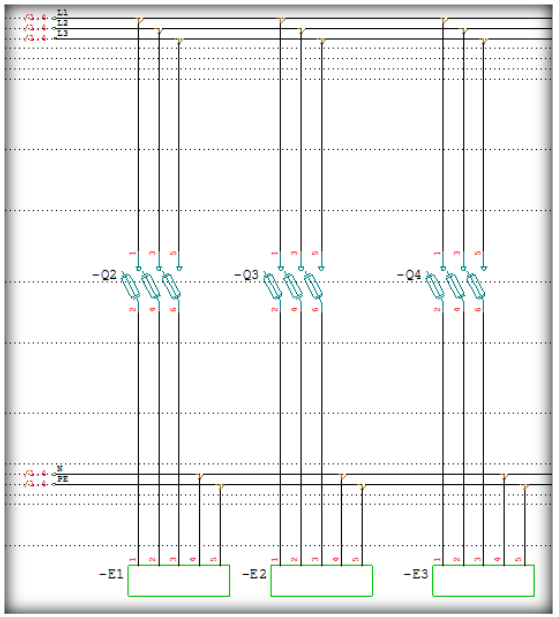


- Remove the "circle"
  - Right-click on –Q3 E2
    - Functions
      - Connect
  - Connections are restored

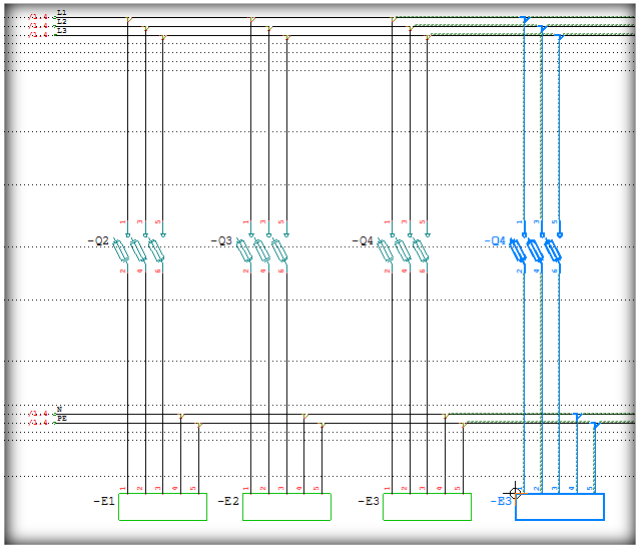


- Tip: Never delete the connection itself.
  - Instead, delete the destination wiring symbols or place such symbols in between.
  - Consequently, when drawing the window: always draw from left to right!

- Copy Q3 and E2
  - Select Q3 and, additionally with "Ctrl-click", E2
  - Copy the elements ("Ctrl+D")
  - Move the elements and place them at the position Q4/E3.
  - The connections between Q4 and E3 are generated
  - The connections to the potentials are not
  - Complete the connections
- Copy the 3 destination wiring symbols over Q3
  - Draw the area from the top left to the bottom right
  - Ctrl+D
  - Move until the corresponding rubber threads at Q4 are displayed and place
  - Proceed analogously with N and PE

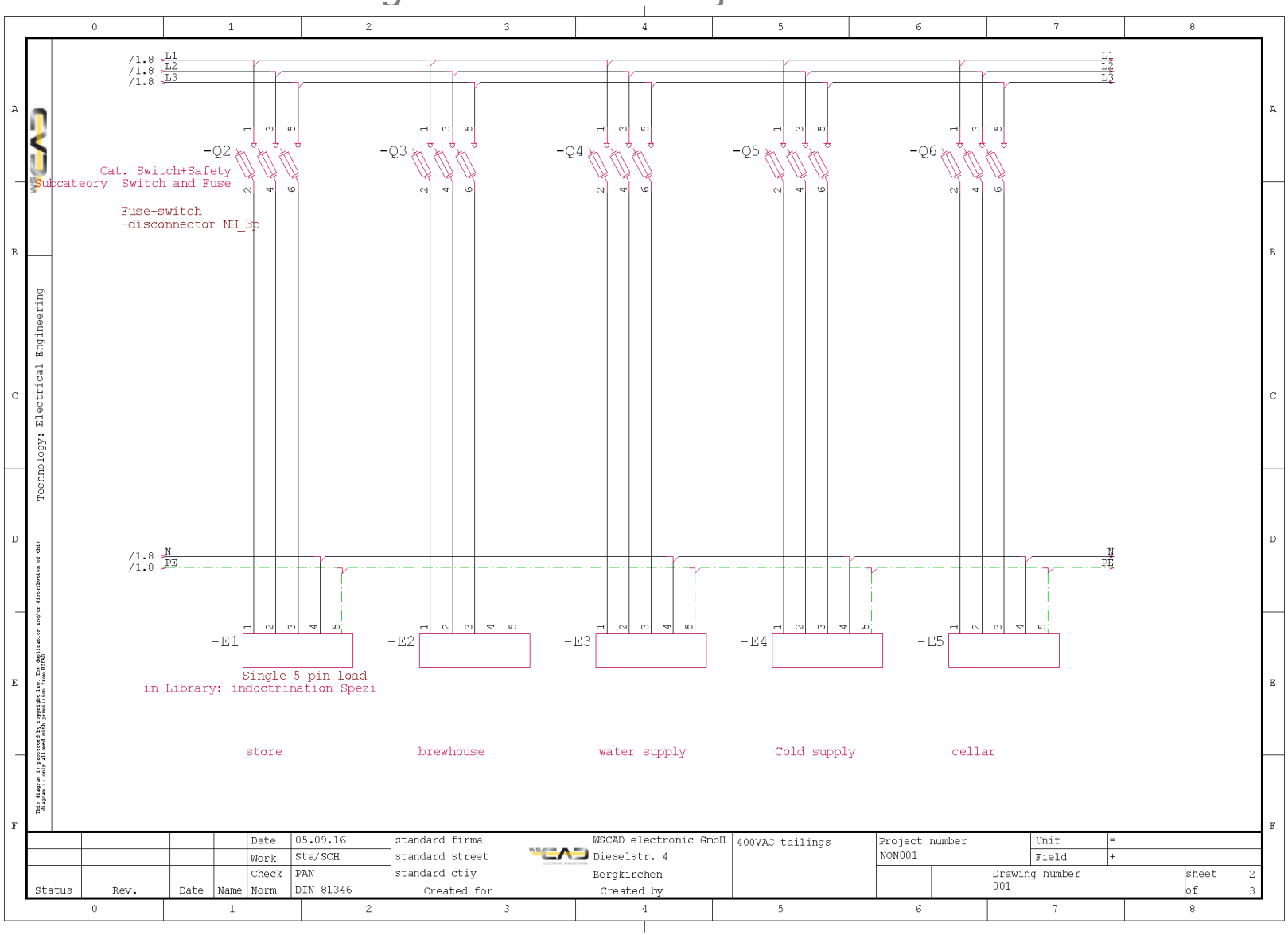


- Copy range (individual and multiple)
  - Select Q4 and E3, including the associated destination wiring symbols by using a window drawn from left to right
    - Draw area
    - Everything that appears in blue is selected
    - Too much / too little? -> Use the Cancel command and draw a new area
  - Press Ctrl+D and do not move the mouse
  - Use the arrow keys to move the copied area some grid points
  - Also try up / down / right / left
  - Press the "+" key 2x and the "-" key 1x
  - Slide the copied area using the arrow keys to the desired location
    - Q4 in the place of Q5
  - Press the Enter key



- Tip: Destination wiring symbols should generally be treated as symbols

Exercise 4 – Drawing Sheet 2 Project Nonsense

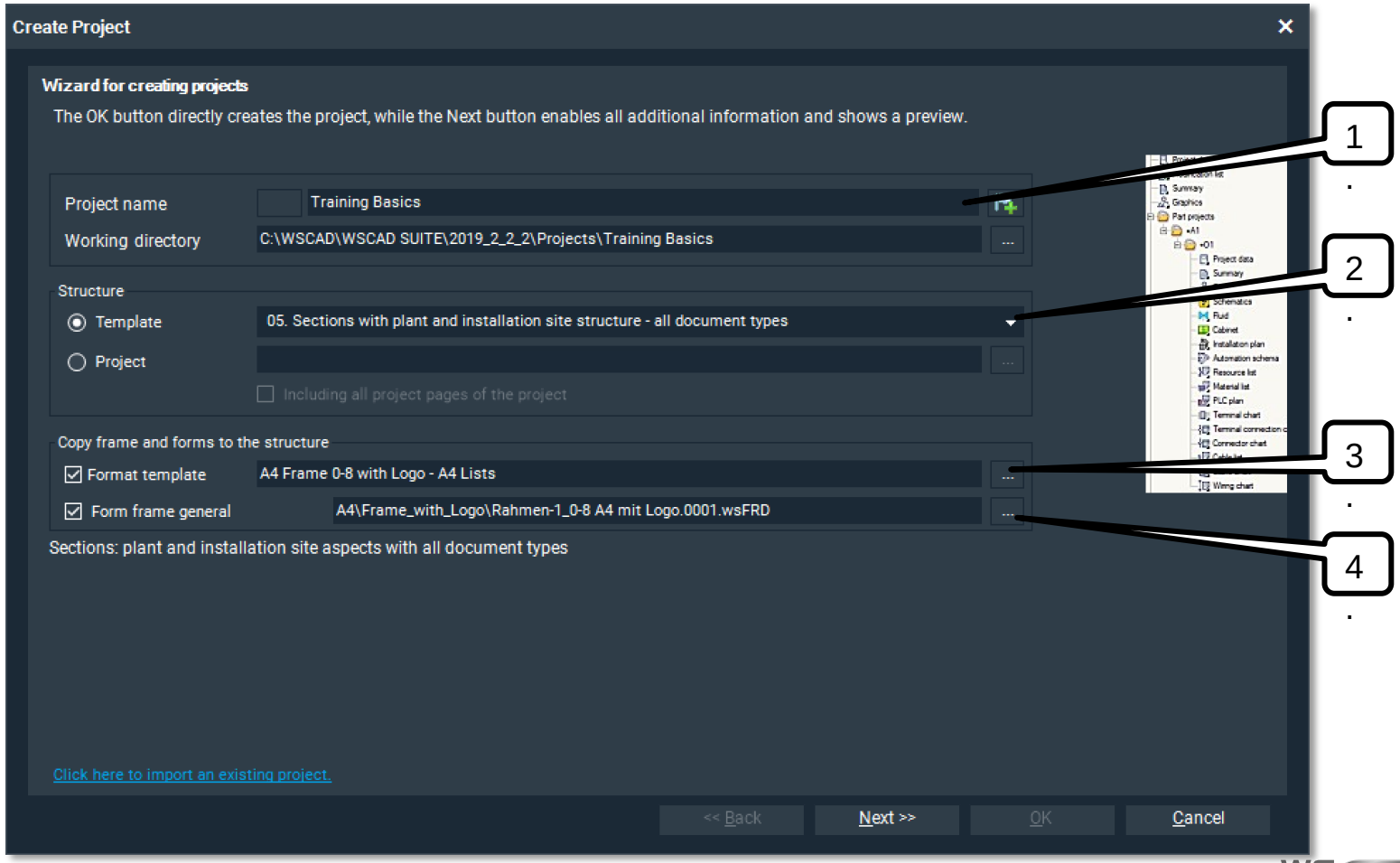


Exercise 5 – Training Project Preparations

- Theoretical Section: **Training Project Preparations**
- Please devote your attention to the instructor. Thank you!

Basics – Drawing – Creating a Second Project

- Creating a new project
  1. Create a new project called "Training Basics"
  2. Template: 05. Sections with plant and installation site structure
  3. Format template: as indicated below
  4. Form frame general : as indicated below, and press "Next"

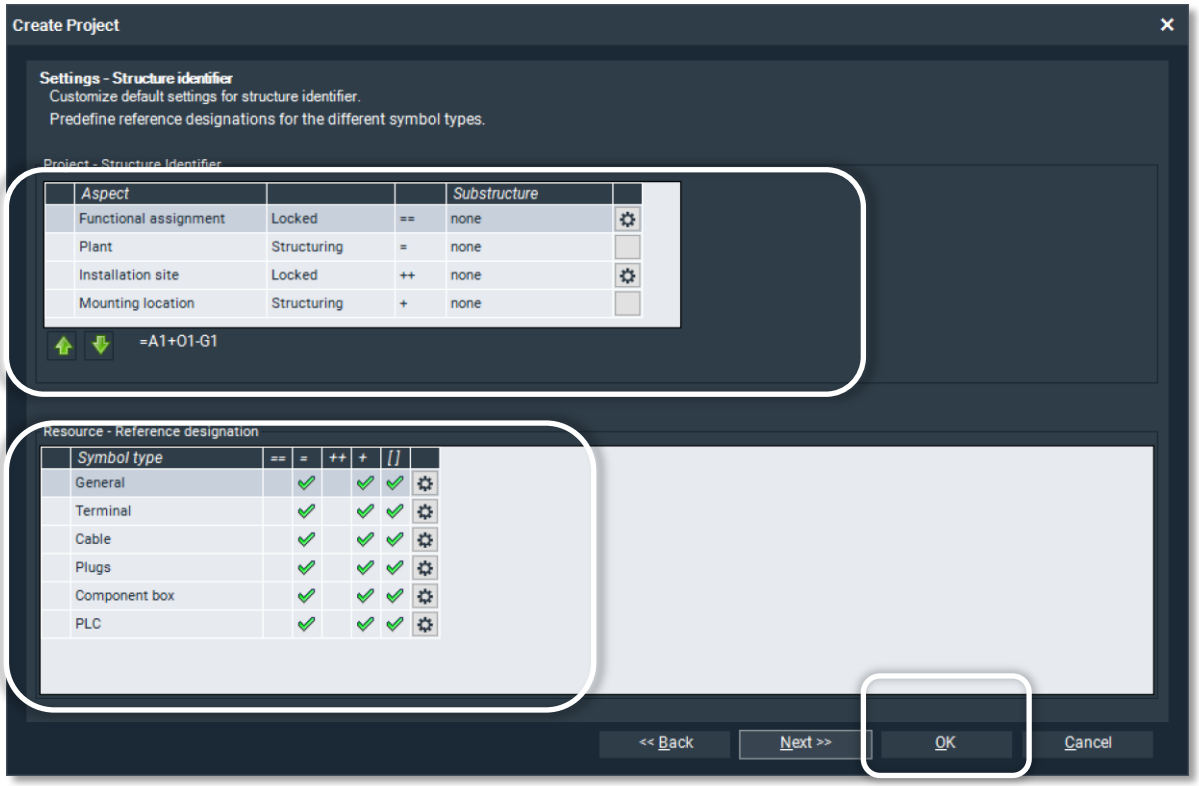


Basics – Drawing – Creating a Second Project

- Creating a new project

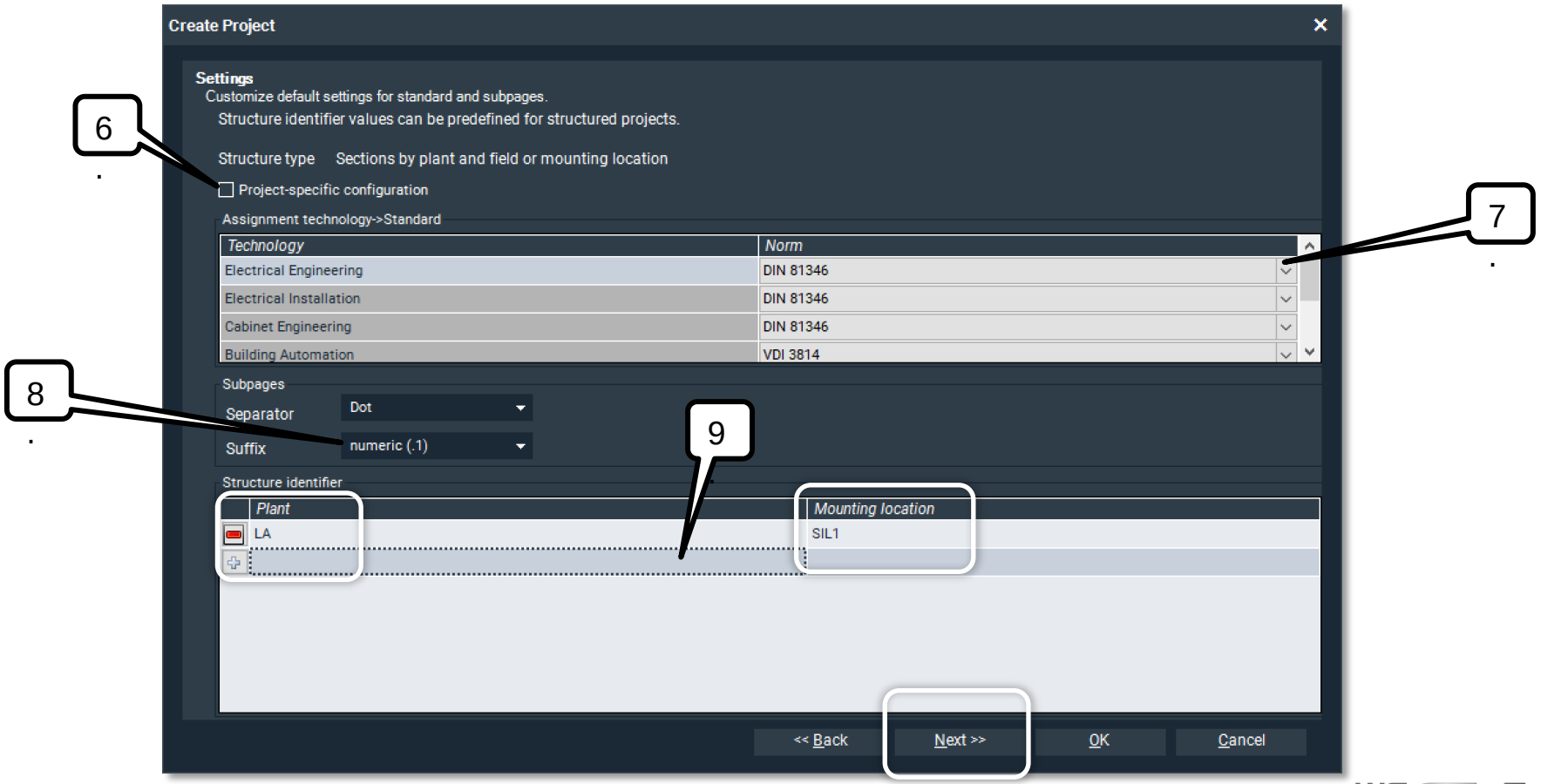
5. Settings as indicated below, and press "Next„

If it does not look like this form please ask



Basics – Drawing – Creating a Second Project

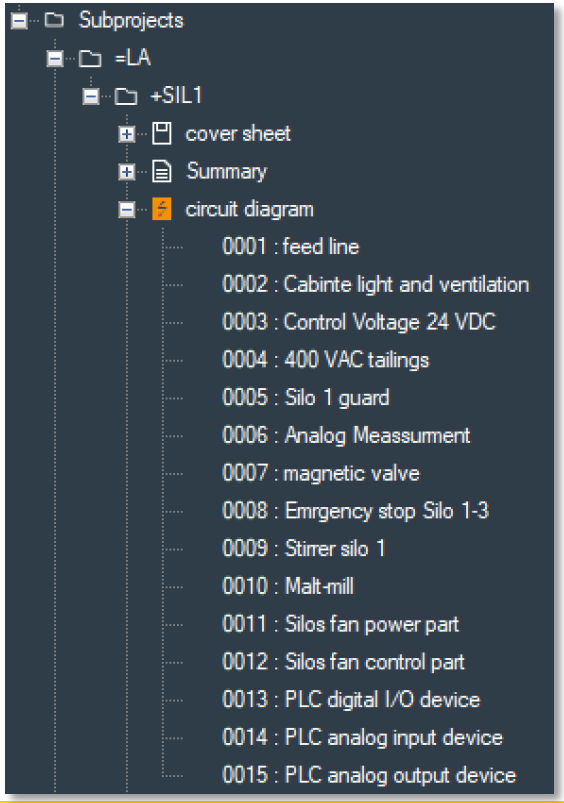
- Creating a new project
  6. Project-specific configuration: do not activate!
  7. Standards: as indicated below
  8. Subpages: as indicated below
  9. Enter structure identifiers as indicated below, and press „Enter„
  10. Press "Next„



Basics – Drawing – Creating a Second Project

- Creating a new project

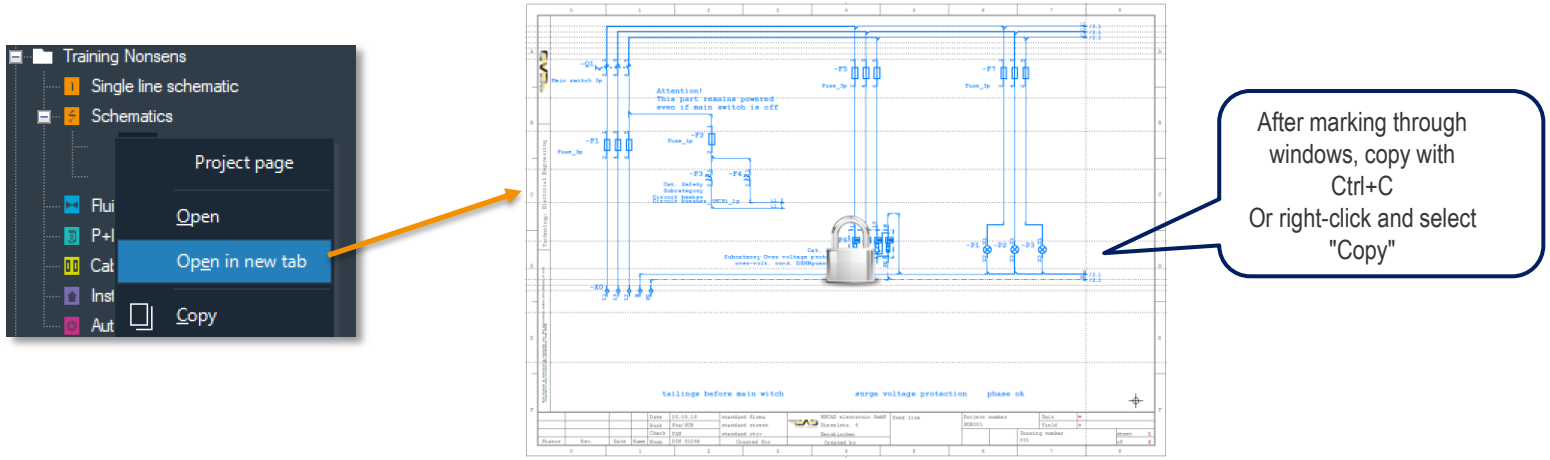
11. In the following step, you can enter global data from the project.
    Then press "Finish".
12. Create 15 pages in the new project in the document folder
    "Schematic" in =LA +SIL1.
13. Name the pages as indicated on the right by right-clicking on
    the schematic and then selecting "Edit project texts" under File contents.
    Your project structure should now look as shown on the right.
14. Open sheet 1, "Supply", by double-clicking.



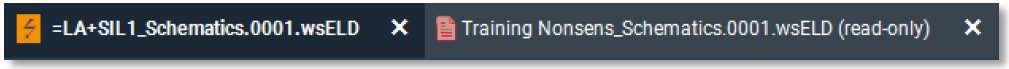
Basics – Drawing – Sheet 1

- Copying pages

1. Copy sheet 1 of the project "Nonsense" by opening it in the Project Explorer with a right-click and selecting "Open in new tab".



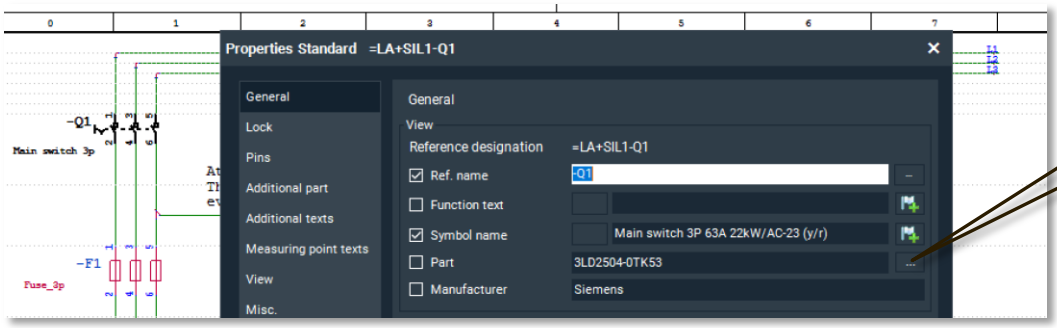
2. Open the active page (Sheet 1 "Supply") in the "Training Basics" project via the above tab.



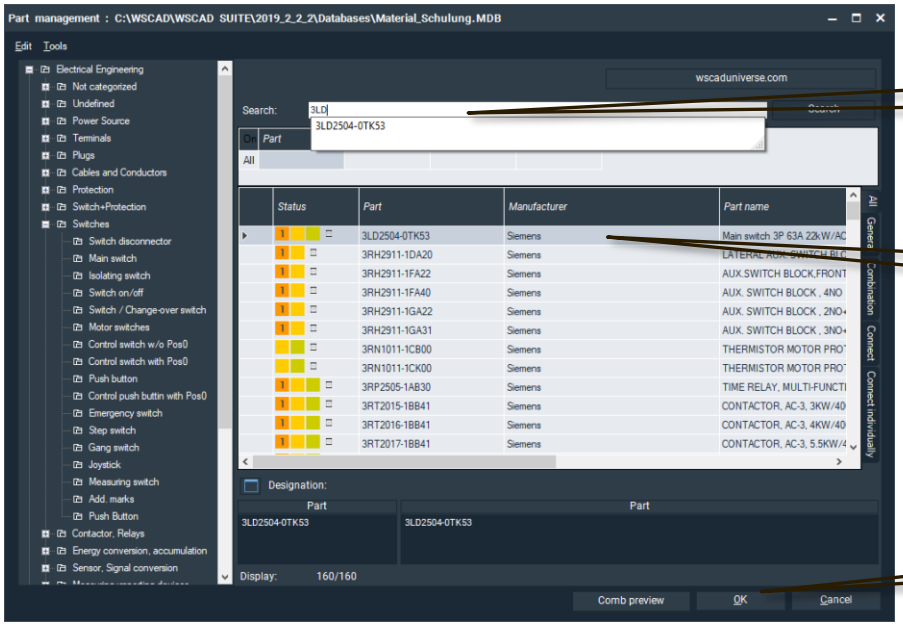
3. "Paste" into the blank sheet 1 using Ctrl+V or by right-clicking and selecting "Paste"

Basics – Drawing – Sheet 1

- Assign parts
  - Complete the drawing by entering the parts for the symbols as on slide 38 either via a) the properties of the symbol or b) by using the Quick Editor.
  - a) Right-click a symbol and then click "Properties"



Access to part database



Enter part number here

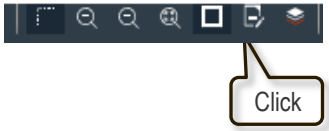
Choose

Confirm with OK

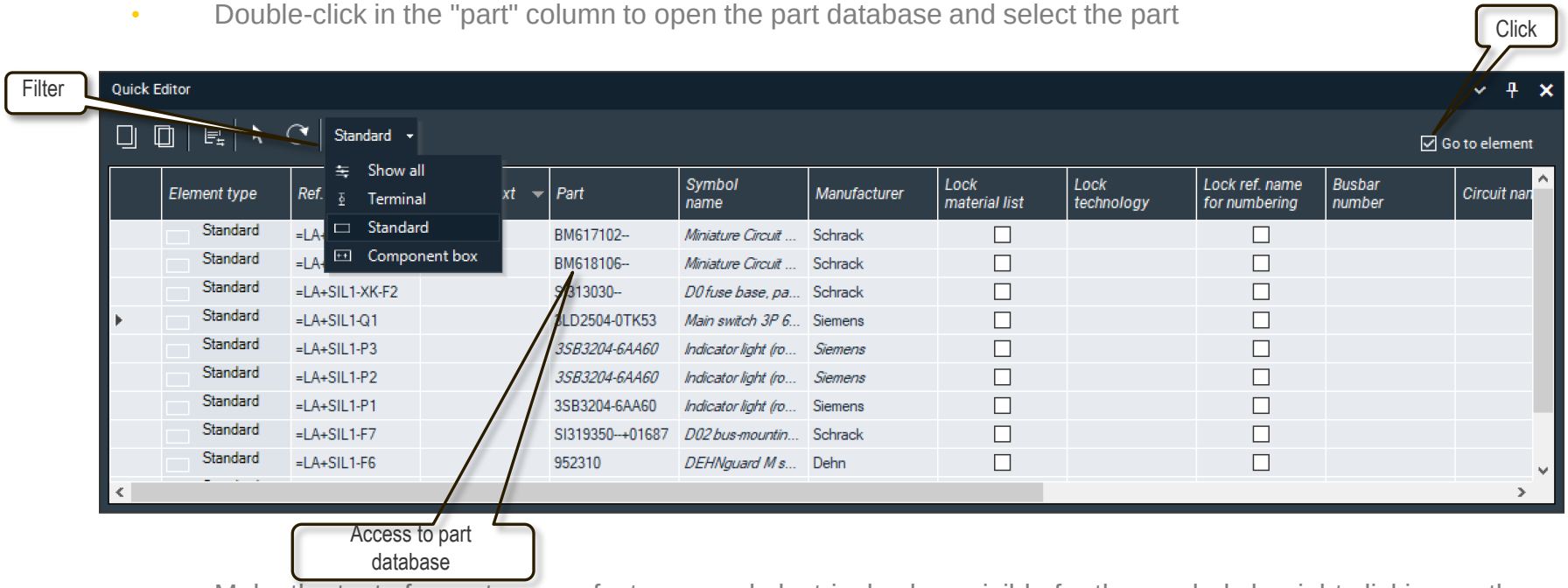
Exercise – Assign Part

Basics – Drawing – Sheet 1

- Assign parts
  - b) by using the Quick Editor
  - Call via:

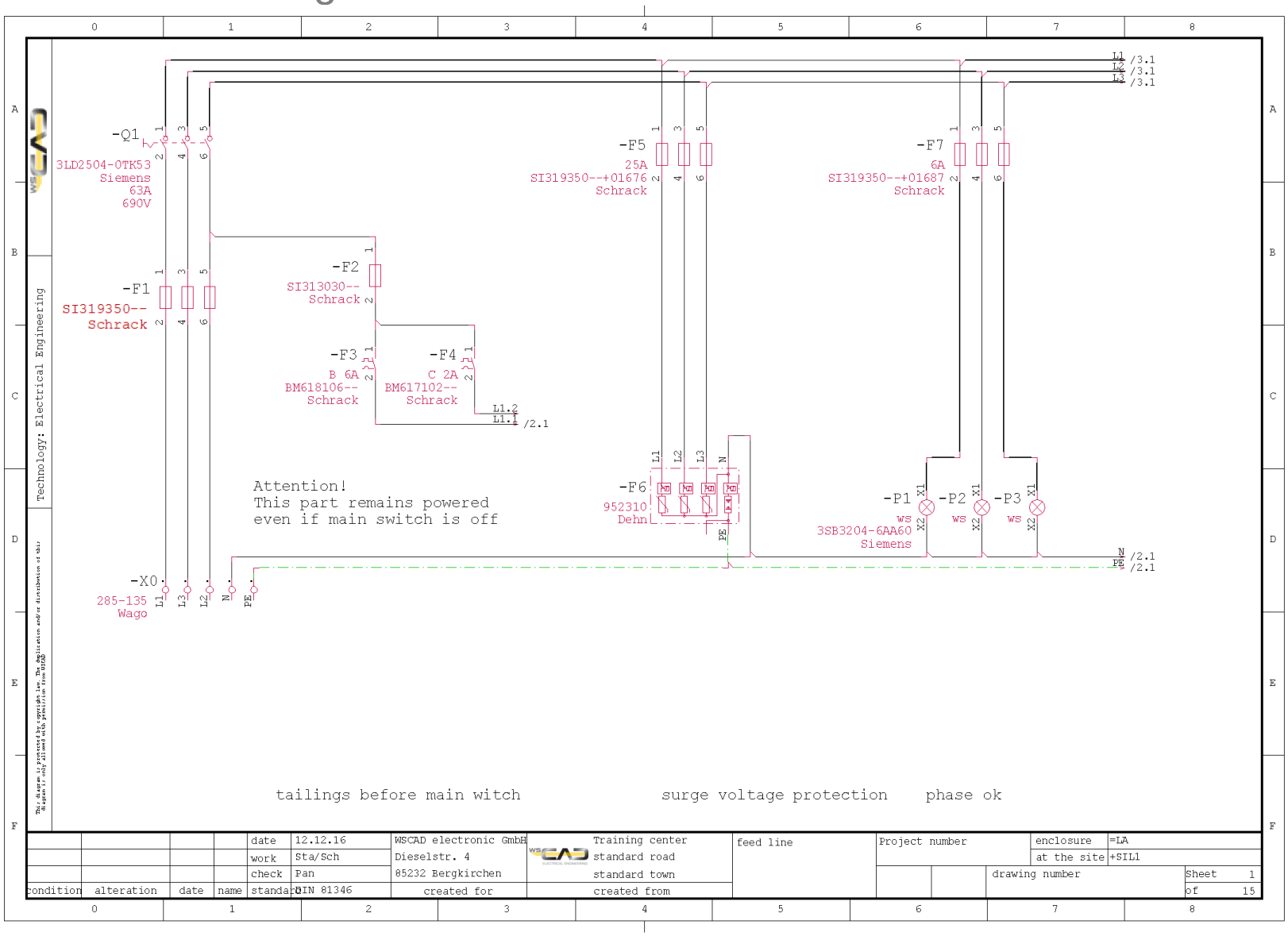


  - Select the type of symbol via the filter
  - By activating "Go to element", the selected element is marked with a pin needle
  - Double-click in the "part" column to open the part database and select the part



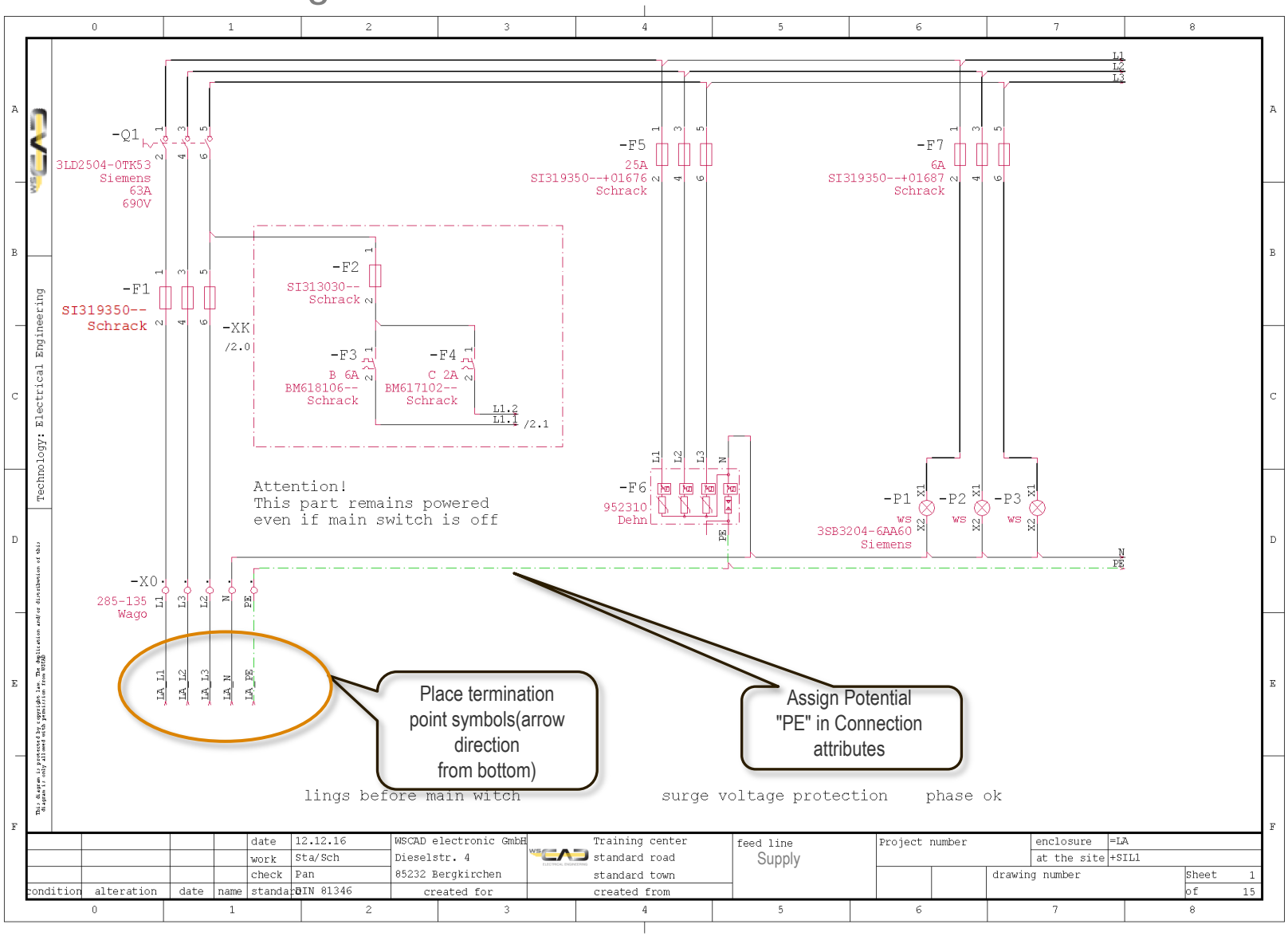
  - Make the texts for parts, manufacturers and electrical values visible for the symbols by right clicking on the texts and then selecting "visible" or at the symbol itself via "Properties | Additional texts".
  - You can display further columns by right-clicking on orange header bar

Basics – Drawing – Sheet 1



Exercise – Assign Part

Basics – Drawing – Sheet 1



Exercise 6 – Symbol Editor & Symbol/Macro Usage

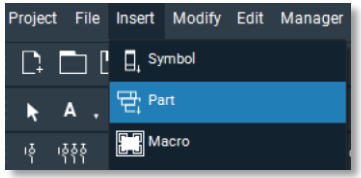
- Theoretical Section: **Symbol Editor**

  Symbol/Macro Usage

- Please devote your attention to the instructor. Thank you!

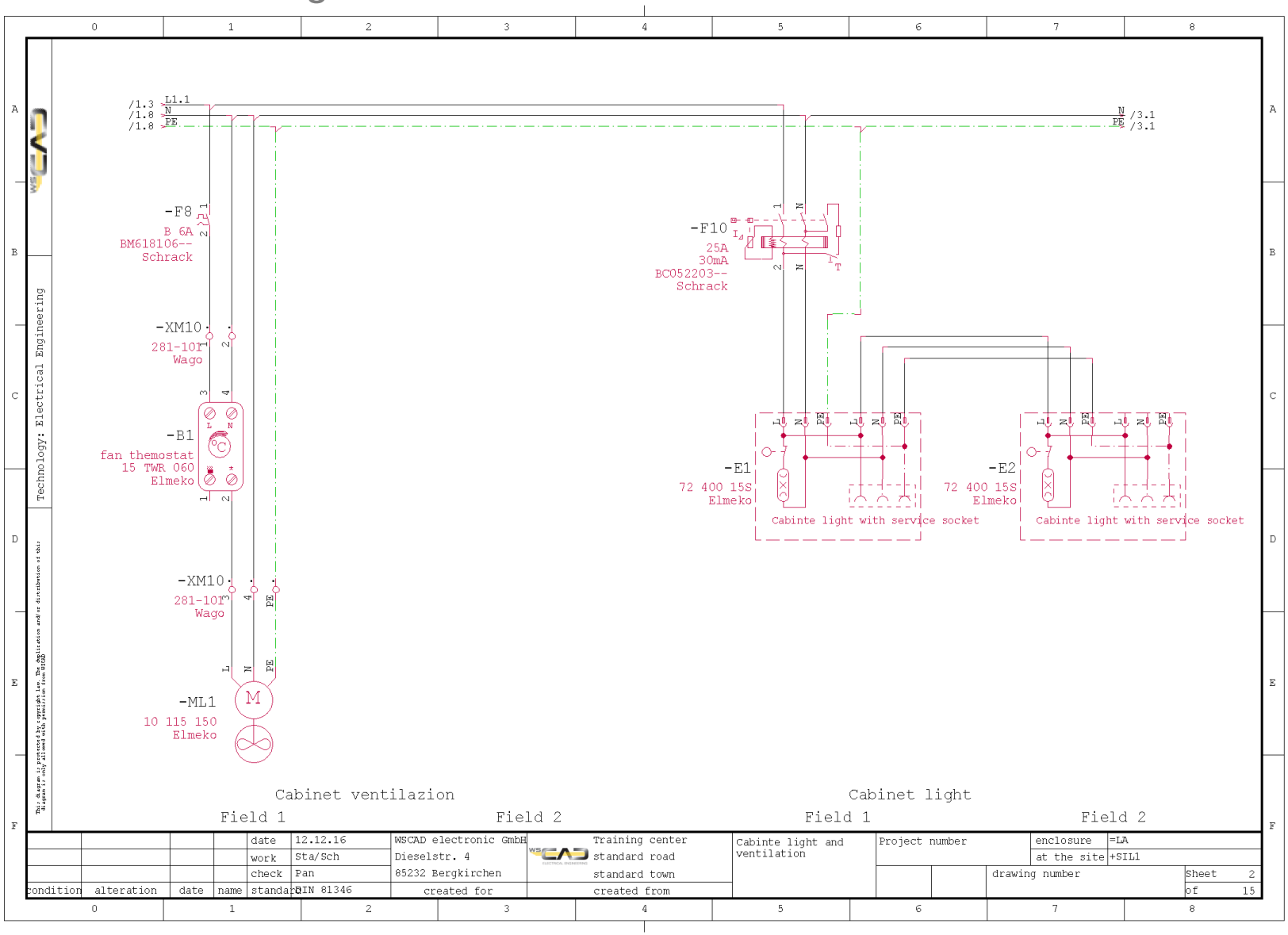
Basics – Drawing – Sheet 2

- Part-oriented placement of components
  1. Draw the plan template (sheet 2/1), slide 42
  2. Place the components via the part numbers from the part database

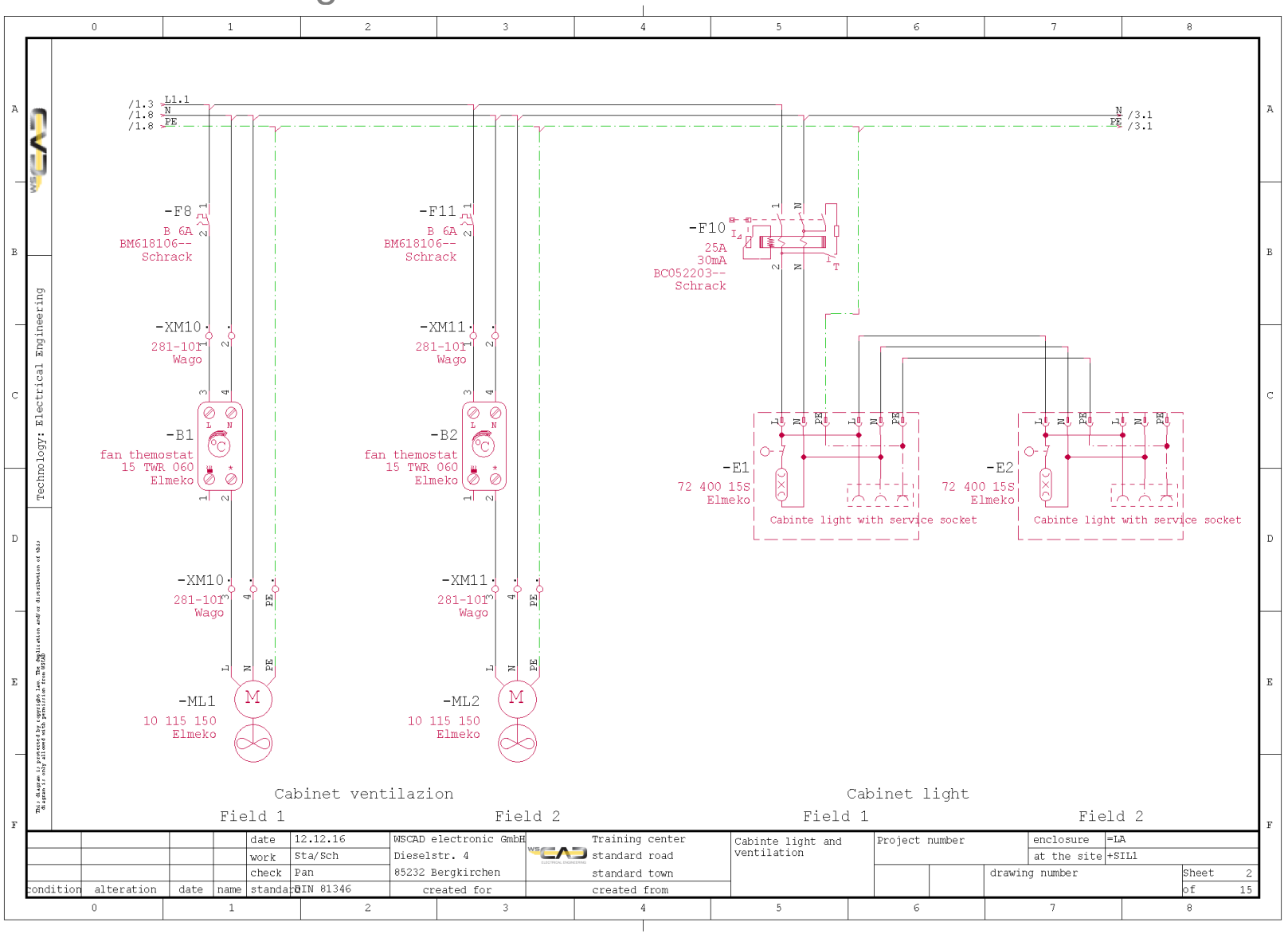


- Copy path 1 (ventilation), plan specification (sheet 2/2), slide 43
  3. Frame path 1 and copy it to path 3 (with Ctrl+d +Distance or context menu)
- Delete –B2 symbol
- Draw black box at this point as in the plan specification (sheet 2/3), slide 44
  4. Call black box and place
  5. Set and edit pins
- Symbol Editor
  6. Take black box over to Symbol Editor by right-clicking
  7. Edit symbol and save as a new symbol in the "Training_Specs" library
- Component box as in plan specification (sheet 2/4), slide 45
  8. Call component box and place
  9. Identify component box as "master"
- Jump to sheet 1 (sheet 1/2), slide 46
  10. Draw component box (slave)

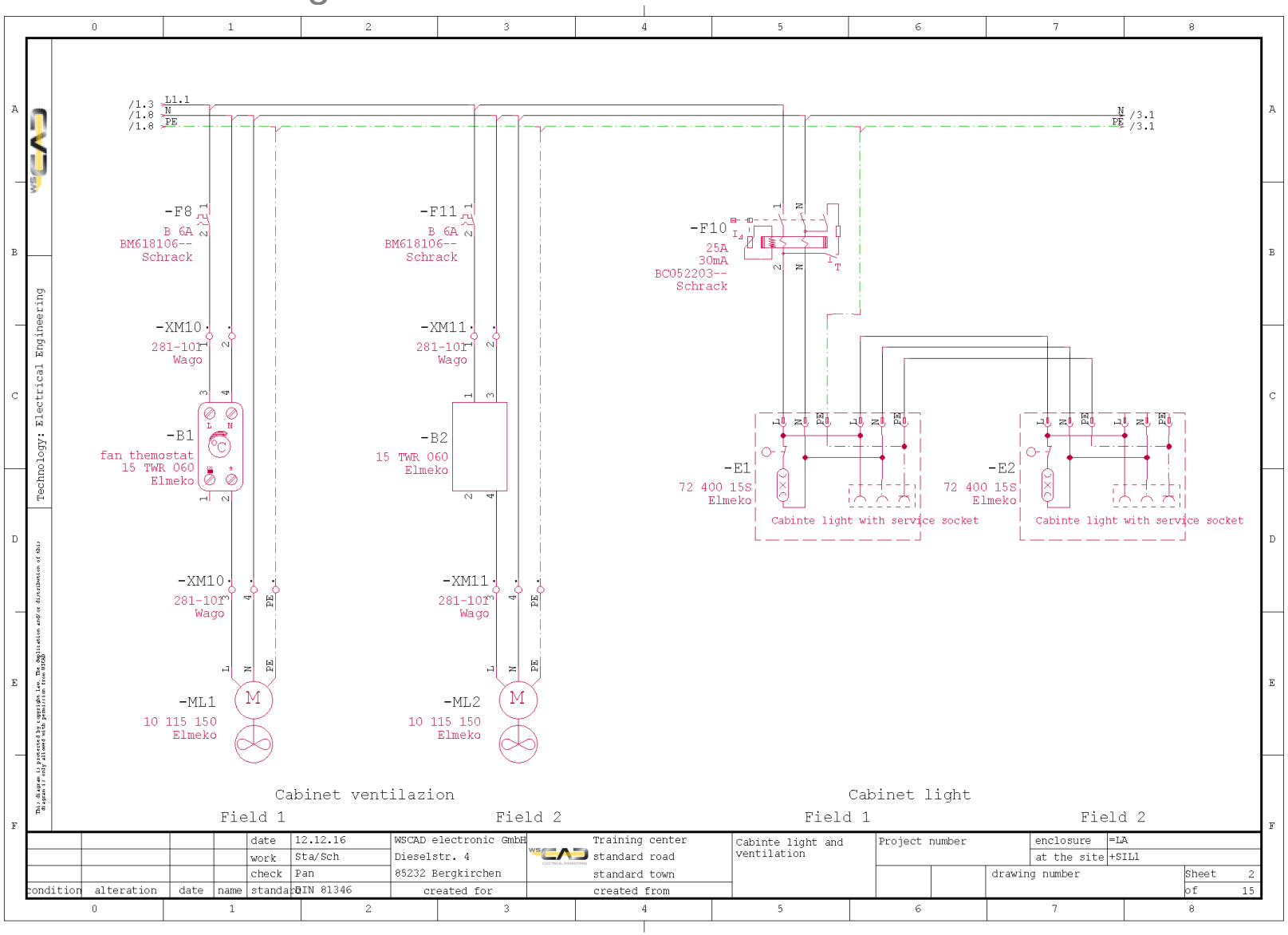
Basics – Drawing – Sheet 2/1



Basics – Drawing – Sheet 2/2

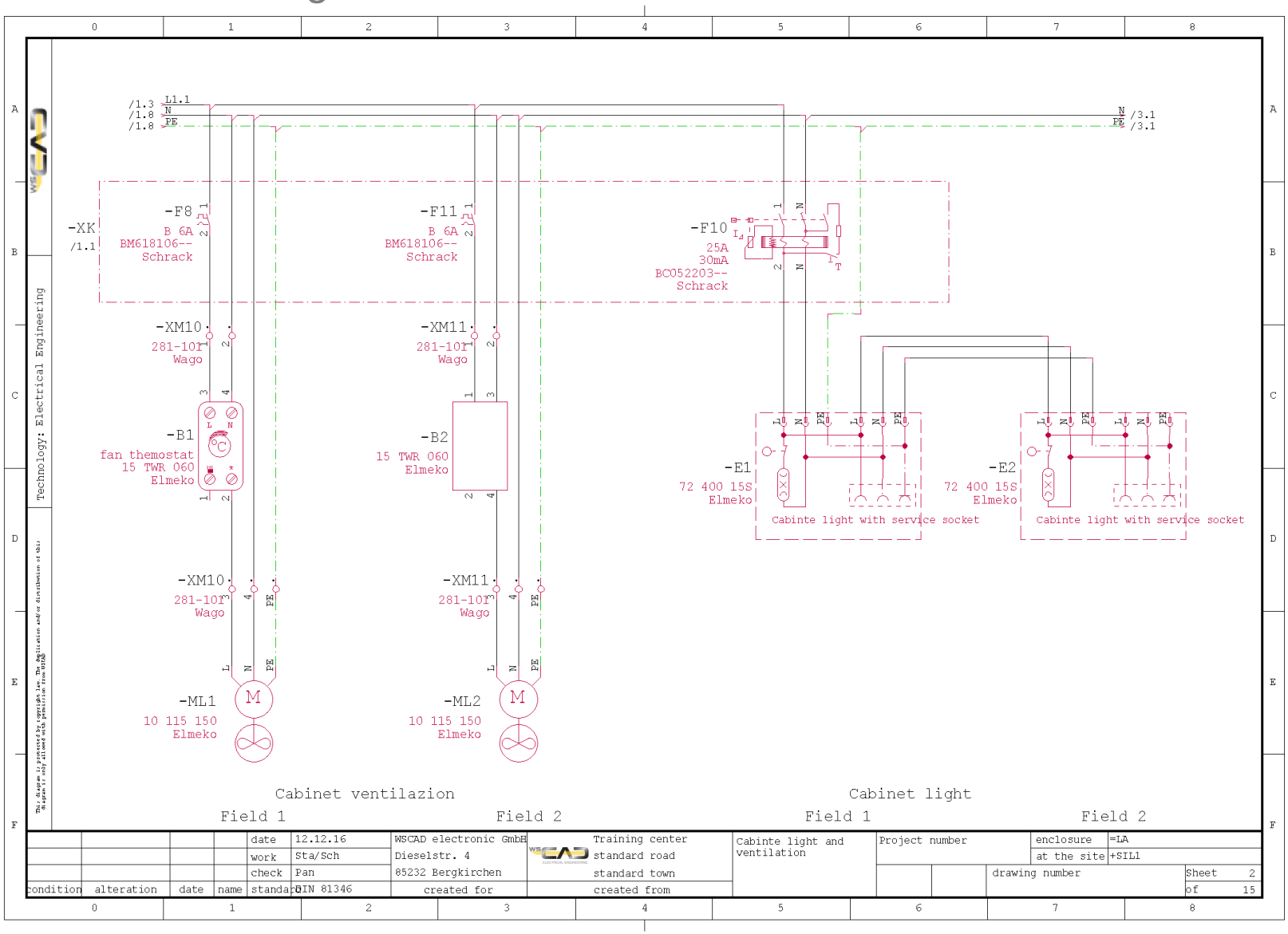


Basics – Drawing – Sheet 2/3

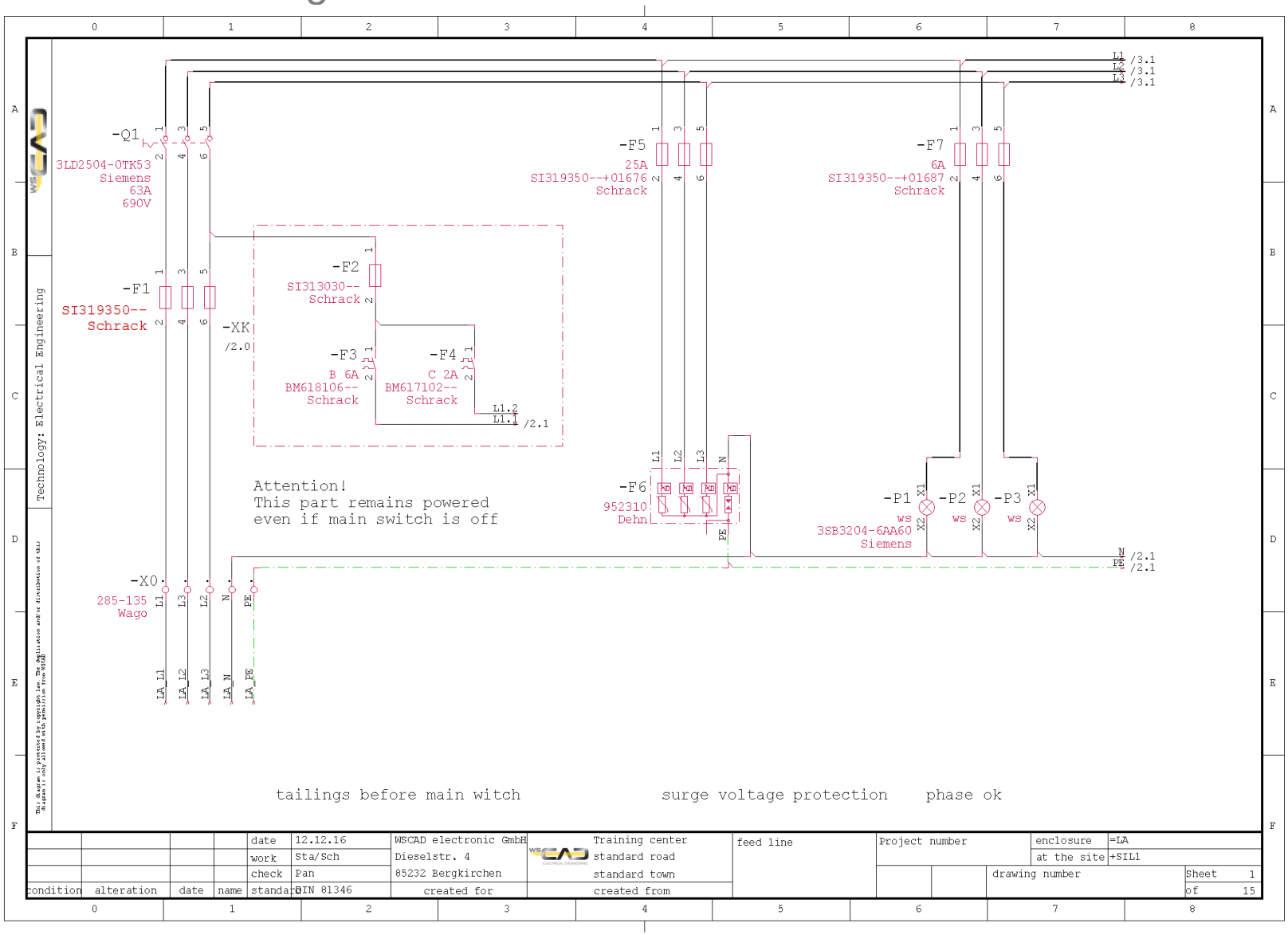


Exercise – Drawing Sheet 2

Basics – Drawing – Sheet 2/4

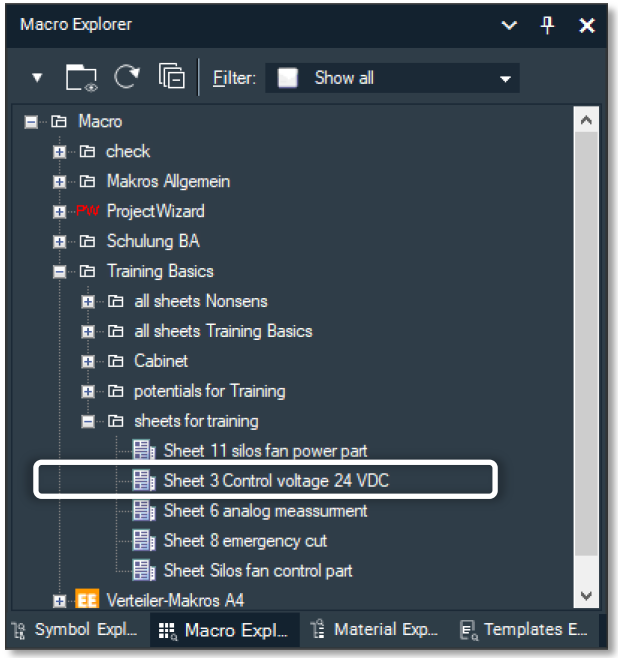


Basics – Drawing – Sheet 1

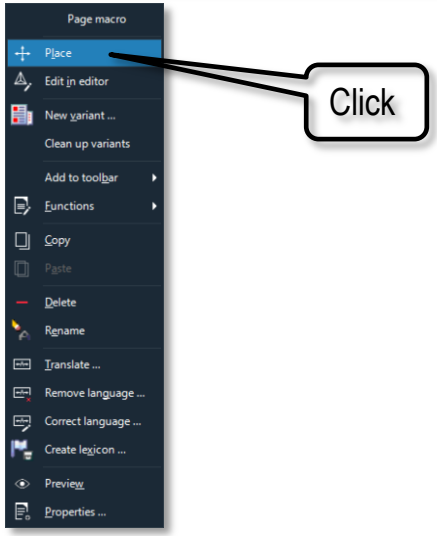


Basics – Drawing – Sheet 3

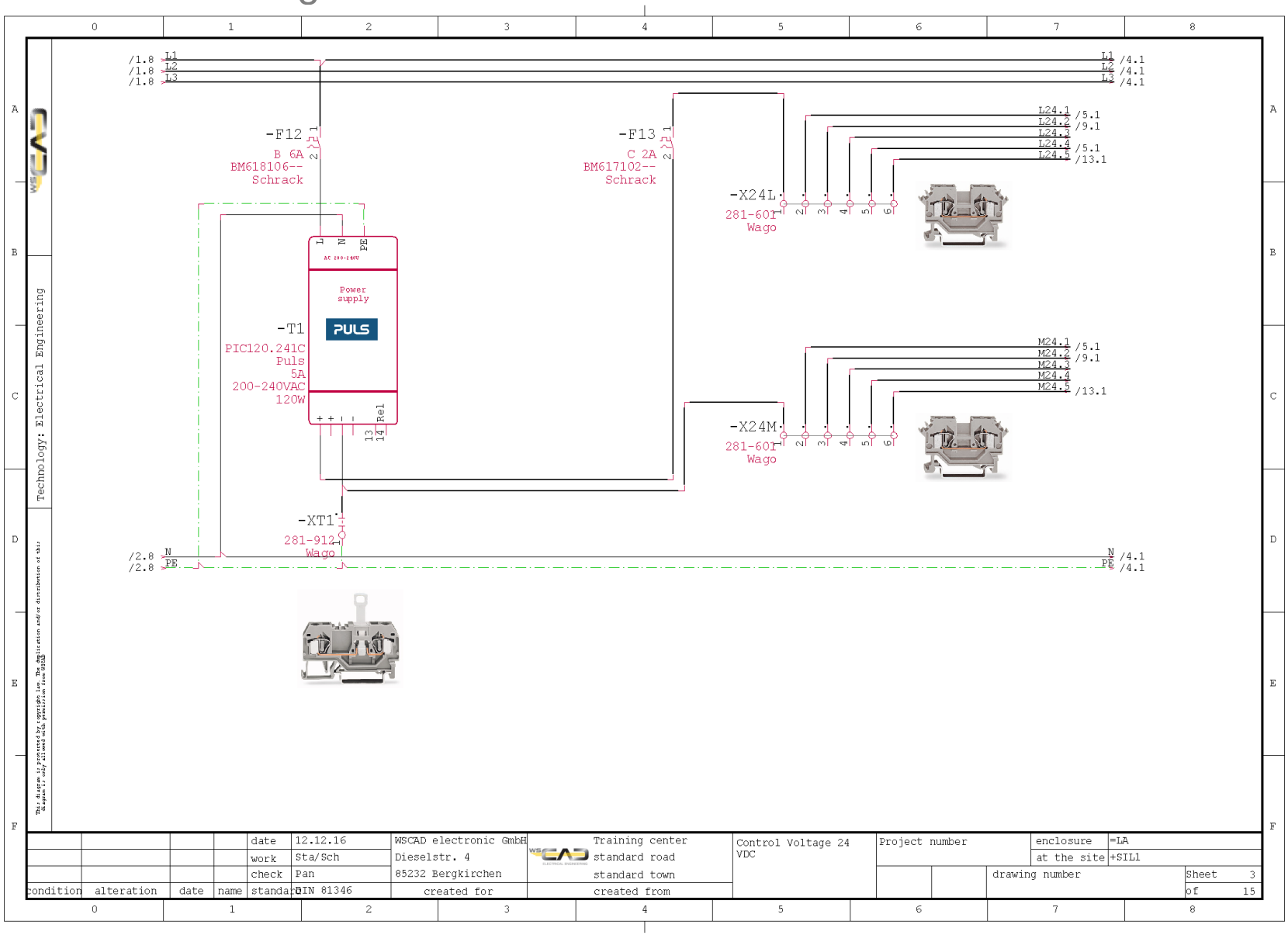
- Insert macro
  1. Open Sheet 3 "Control voltage 24 VDC"
  2. Go to the Drawing Macro Explorer
  3. Navigate to the macro
     "Sheet 3 Control voltage 24VDC"



4. Place the macro by right-clicking as shown on slide 48



Basics – Drawing – Sheet 3

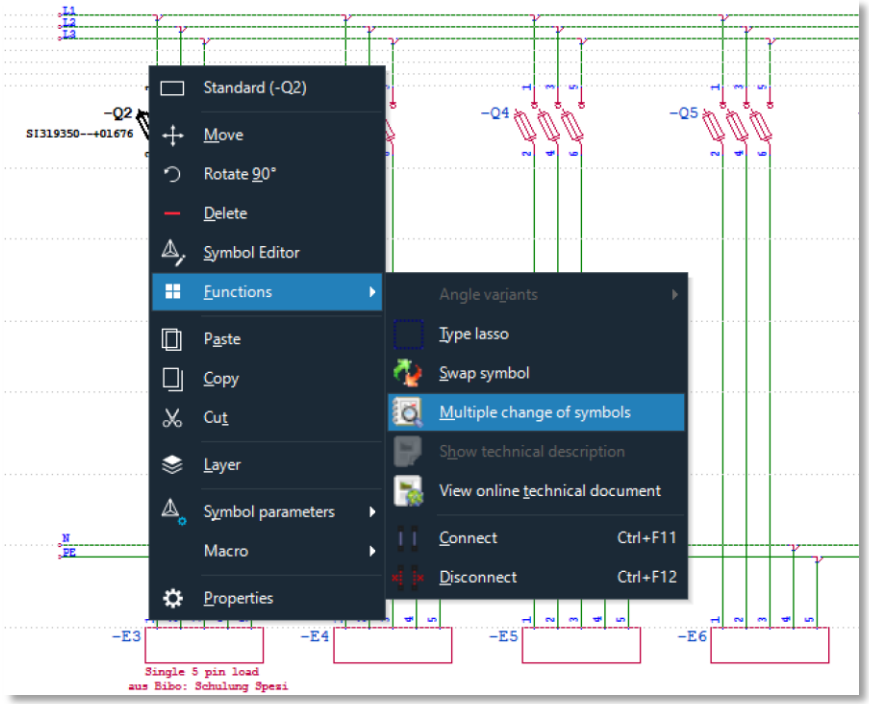


Basics – Replacing symbols – Part 4

- Copy page 2 of the project "Nonsense" plan,
  by opening the above page in the Project Explorer using "Open in new tab" and then selecting and copying the contents.
  Then open sheet 4 from the current project and paste the contents using Ctrl+V
  as shown on slide 51.

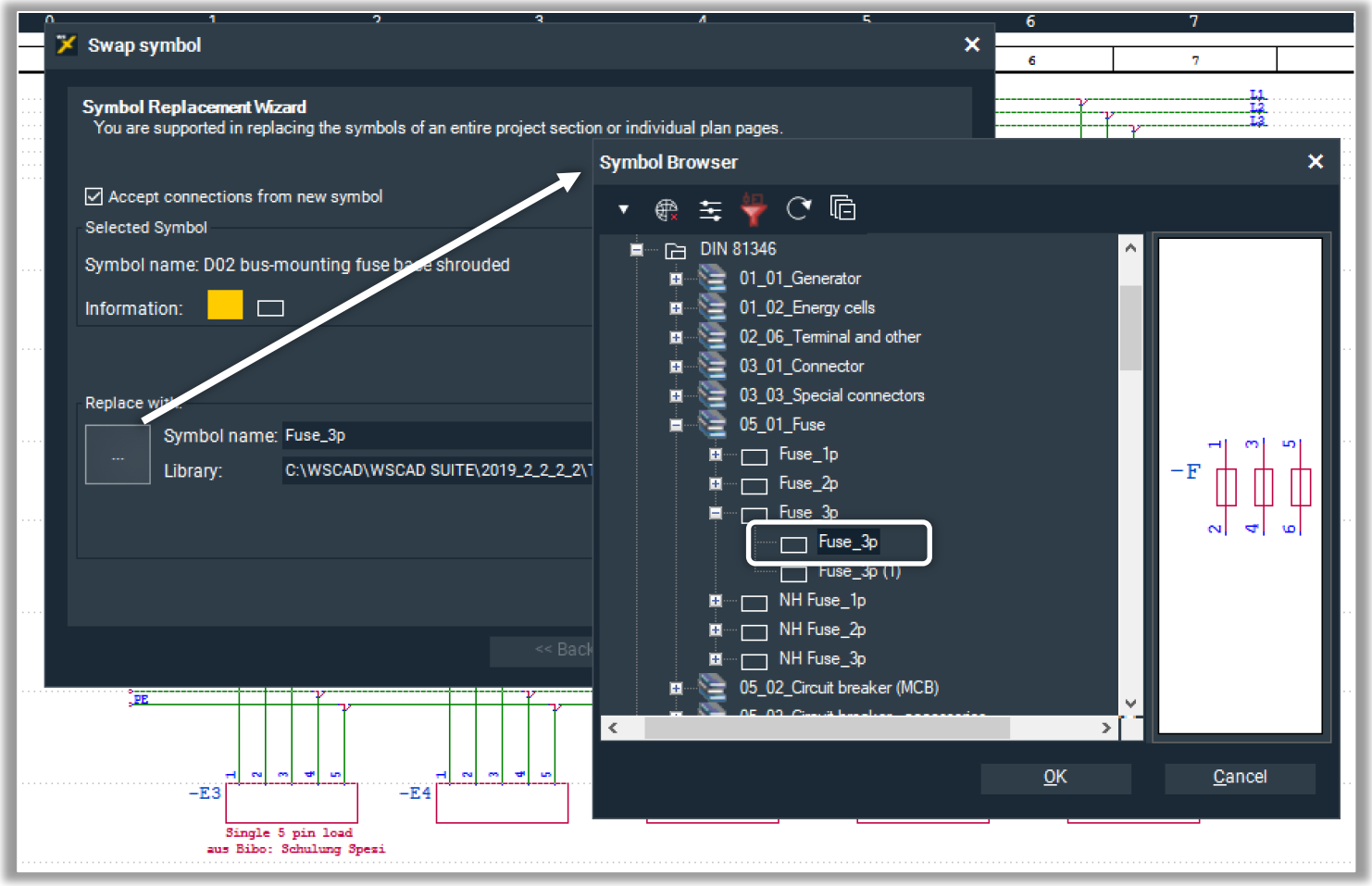
- Multiple change of symbols
  - Go to the context menu of -Q2 and replace multiple instances of the NH disconnector symbol with a fuse.

  As shown on slide 52



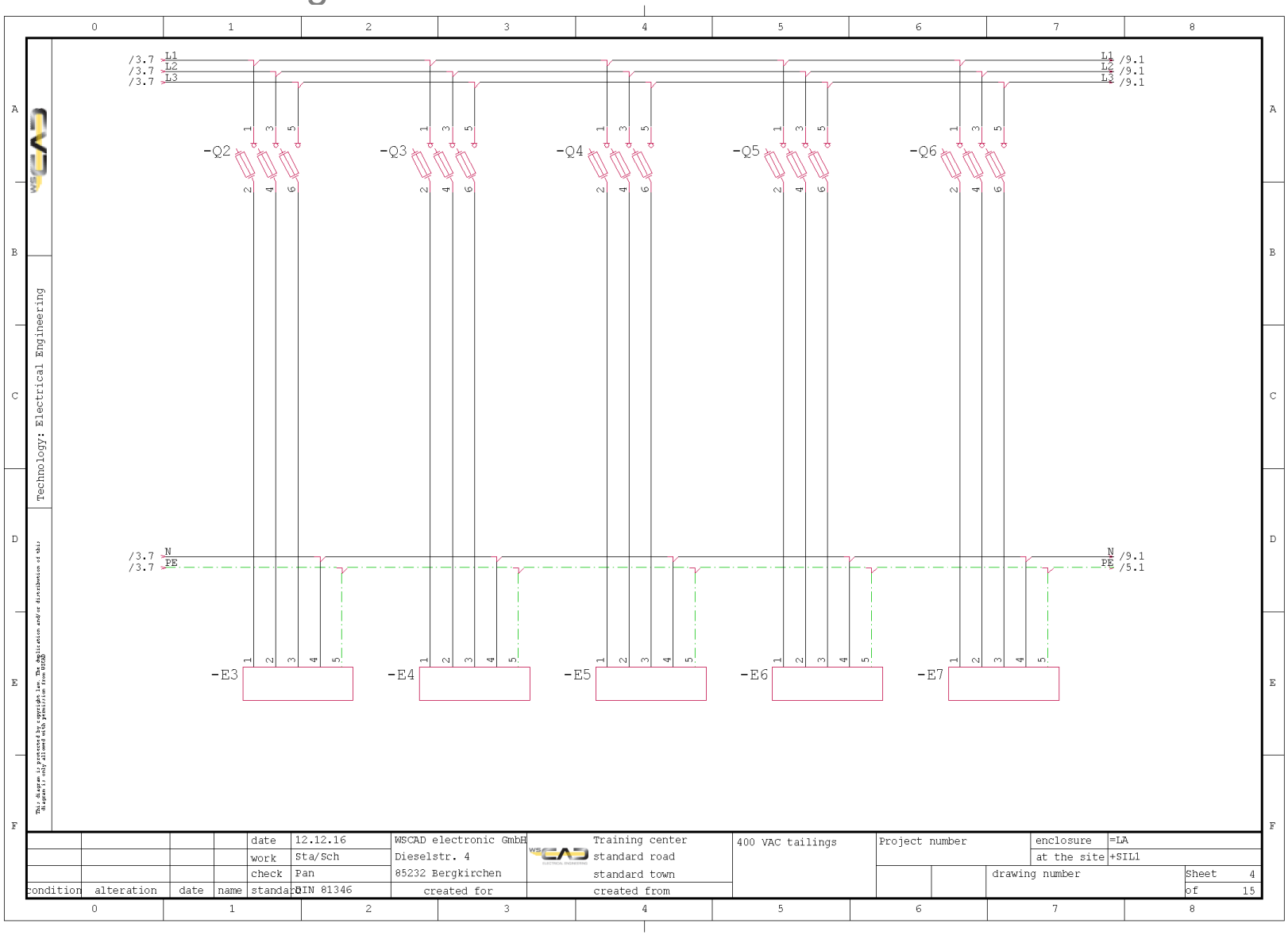
Basics – Replacing symbols – Part 4

- Multiple change of symbols
  - Within the Symbol Replacement Wizard choose the Symbol Fuse3p

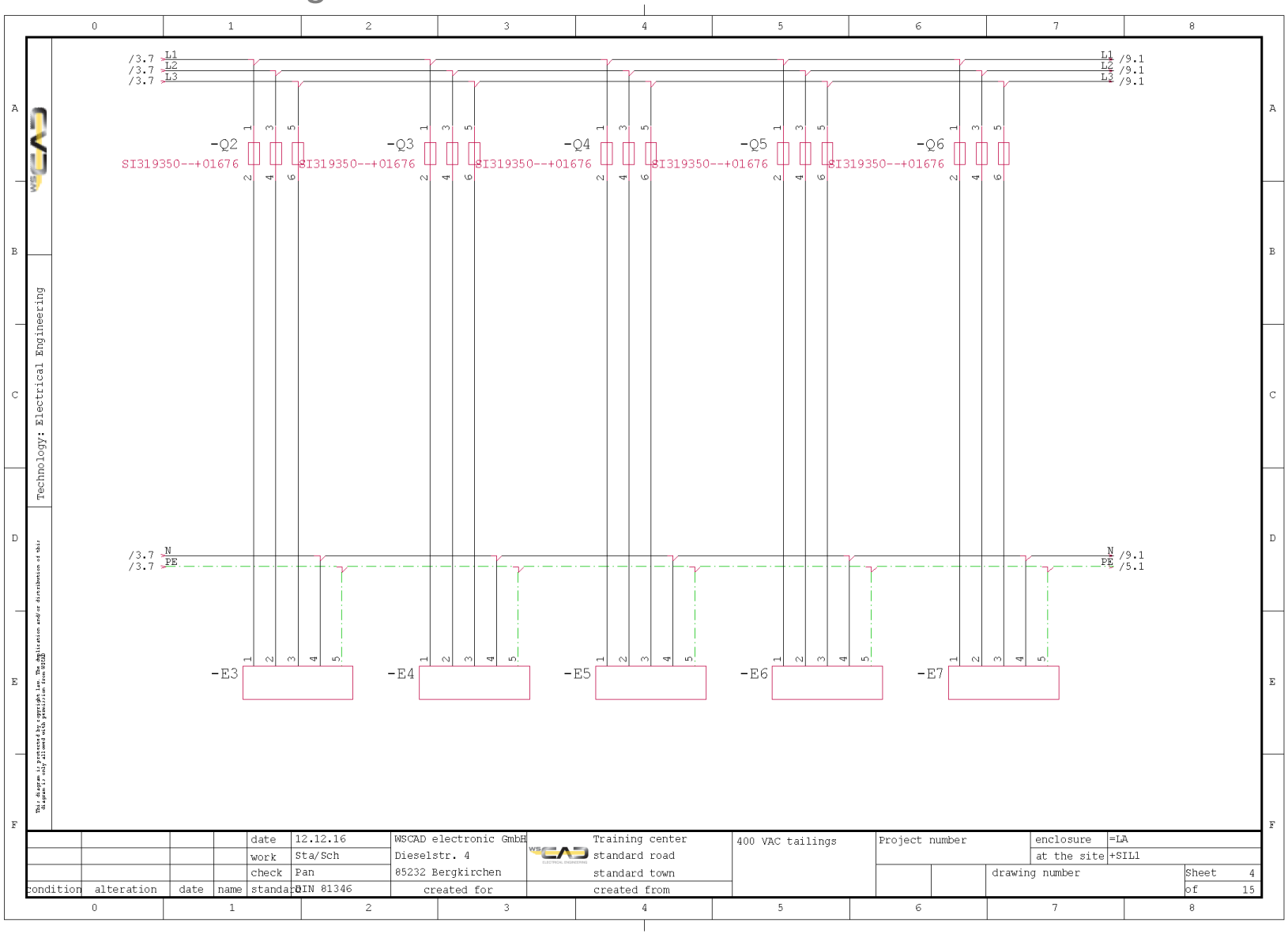


  - Assign the parts specified in plan specification sheet 4/2, slide 52

Basics – Drawing – Sheet 4/1



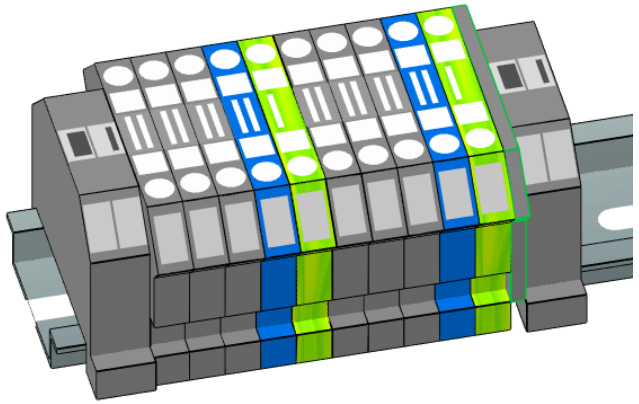
Basics – Drawing – Sheet 4/2



Exercise 7 – Terminals

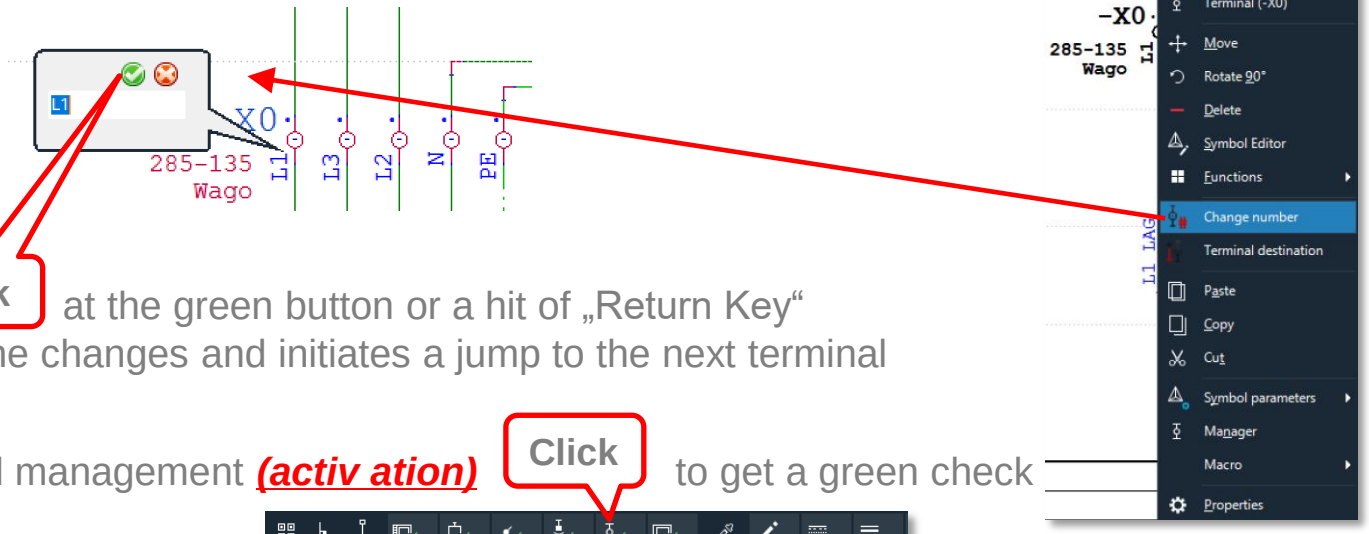
- Theoretical Section: **Terminals**

- Please devote your attention to the instructor. Thank you!



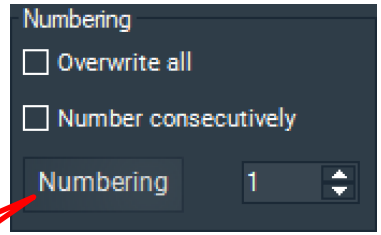
Basics – Terminals – Sheet 4

- Name terminals manually



- A **Click** at the green button or a hit of „Return Key“ saves the changes and initiates a jump to the next terminal

- Terminal management ***(activ ation)*** **Click** to get a green check

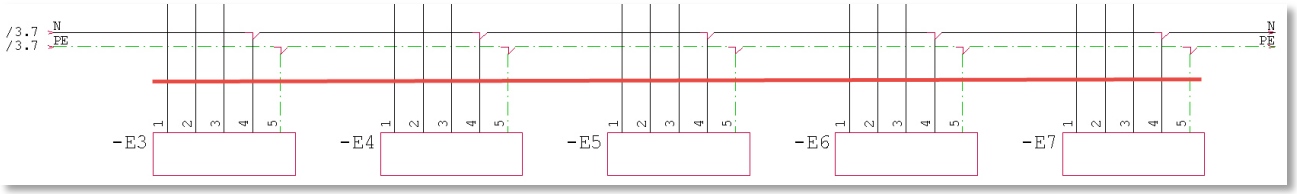
1. Activate the terminal management
2. Call the terminal management by using Menue „Manager | Terminals“.
3. Click „New“ and create a terminal strip with the name –X1 and define 25 terminals.
4. Number the terminals N and PE as shown at page 58.
5. Click  and leave the manager with OK button.

Übung Klemmen Blatt 4 ca. 20 min

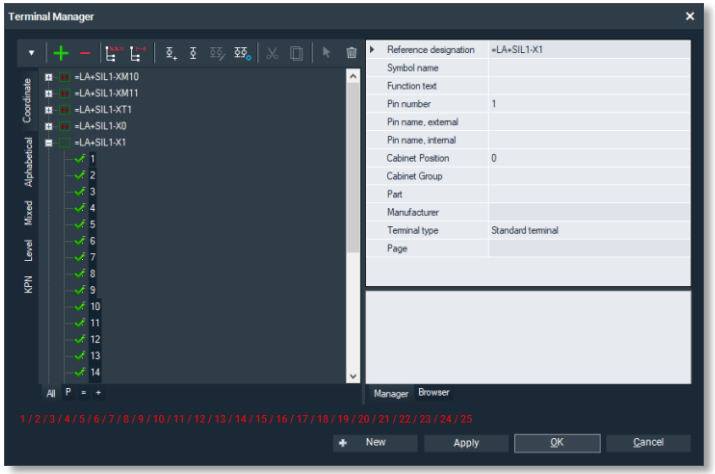
Basics – Terminals – Sheet 4

- Terminal management (active)

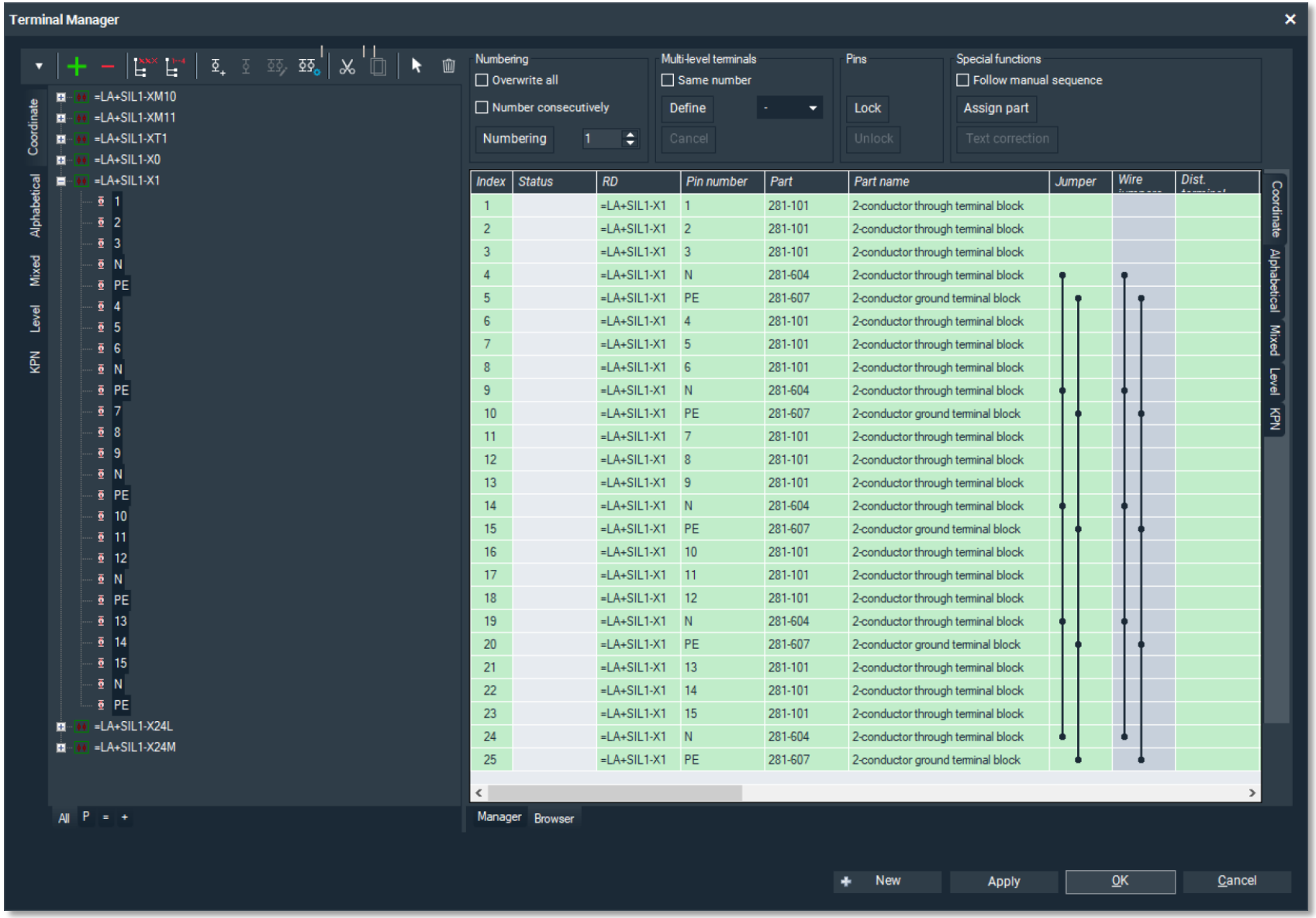
6. Then place the terminal strip in the plan
   via the command:
   using the line above the consumer symbols



7. Click in the manager on the terminal1 of =LA-X1,
   and then confirm with OK.
8. Check the result
   with the plan specification sheet 4/4, slide 57.
9. Alternatives for bridge lines
   and N/PE terminals can be seen
   on slides 58 and 59

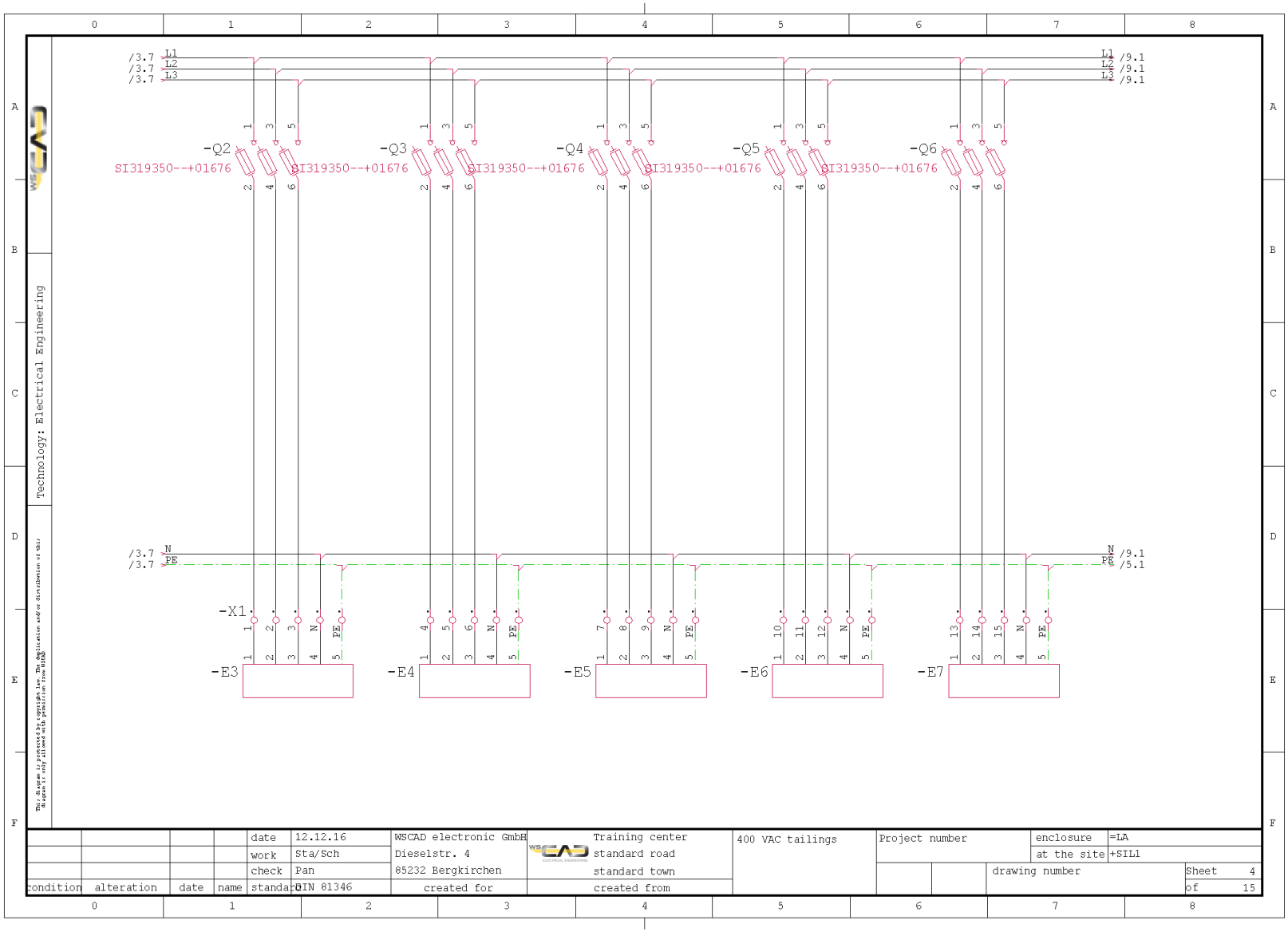


Basics – Terminals – Sheet 4/3

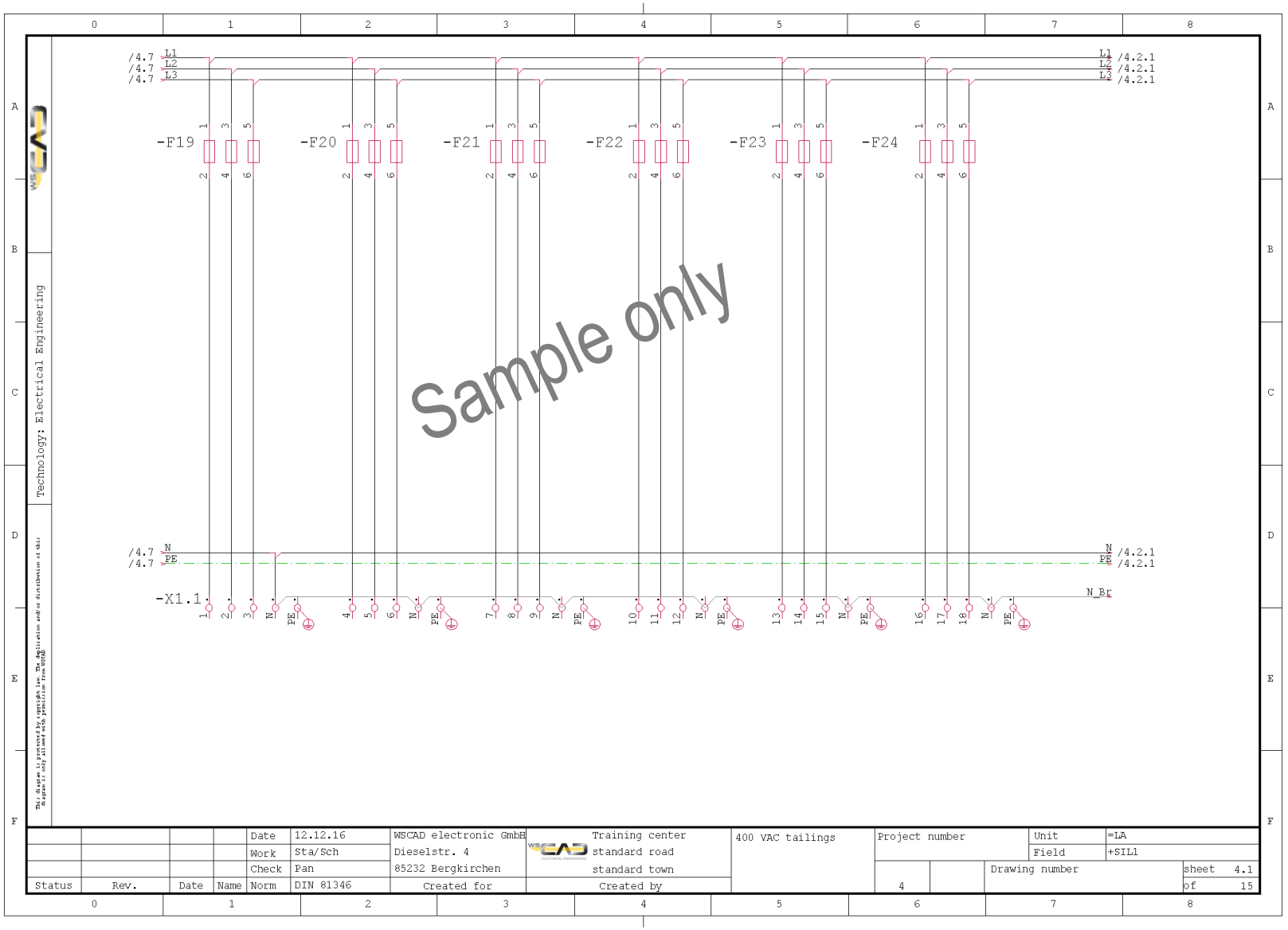


Index	Status	RD	Pin number	Part	Part name	Jumper	Wire	Dist.
1		=LA+SIL1-X1	1	281-101	2-conductor through terminal block			
2		=LA+SIL1-X1	2	281-101	2-conductor through terminal block			
3		=LA+SIL1-X1	3	281-101	2-conductor through terminal block			
4		=LA+SIL1-X1	N	281-604	2-conductor through terminal block			
5		=LA+SIL1-X1	PE	281-607	2-conductor ground terminal block			
6		=LA+SIL1-X1	4	281-101	2-conductor through terminal block			
7		=LA+SIL1-X1	5	281-101	2-conductor through terminal block			
8		=LA+SIL1-X1	6	281-101	2-conductor through terminal block			
9		=LA+SIL1-X1	N	281-604	2-conductor through terminal block			
10		=LA+SIL1-X1	PE	281-607	2-conductor ground terminal block			
11		=LA+SIL1-X1	7	281-101	2-conductor through terminal block			
12		=LA+SIL1-X1	8	281-101	2-conductor through terminal block			
13		=LA+SIL1-X1	9	281-101	2-conductor through terminal block			
14		=LA+SIL1-X1	N	281-604	2-conductor through terminal block			
15		=LA+SIL1-X1	PE	281-607	2-conductor ground terminal block			
16		=LA+SIL1-X1	10	281-101	2-conductor through terminal block			
17		=LA+SIL1-X1	11	281-101	2-conductor through terminal block			
18		=LA+SIL1-X1	12	281-101	2-conductor through terminal block			
19		=LA+SIL1-X1	N	281-604	2-conductor through terminal block			
20		=LA+SIL1-X1	PE	281-607	2-conductor ground terminal block			
21		=LA+SIL1-X1	13	281-101	2-conductor through terminal block			
22		=LA+SIL1-X1	14	281-101	2-conductor through terminal block			
23		=LA+SIL1-X1	15	281-101	2-conductor through terminal block			
24		=LA+SIL1-X1	N	281-604	2-conductor through terminal block			
25		=LA+SIL1-X1	PE	281-607	2-conductor ground terminal block			

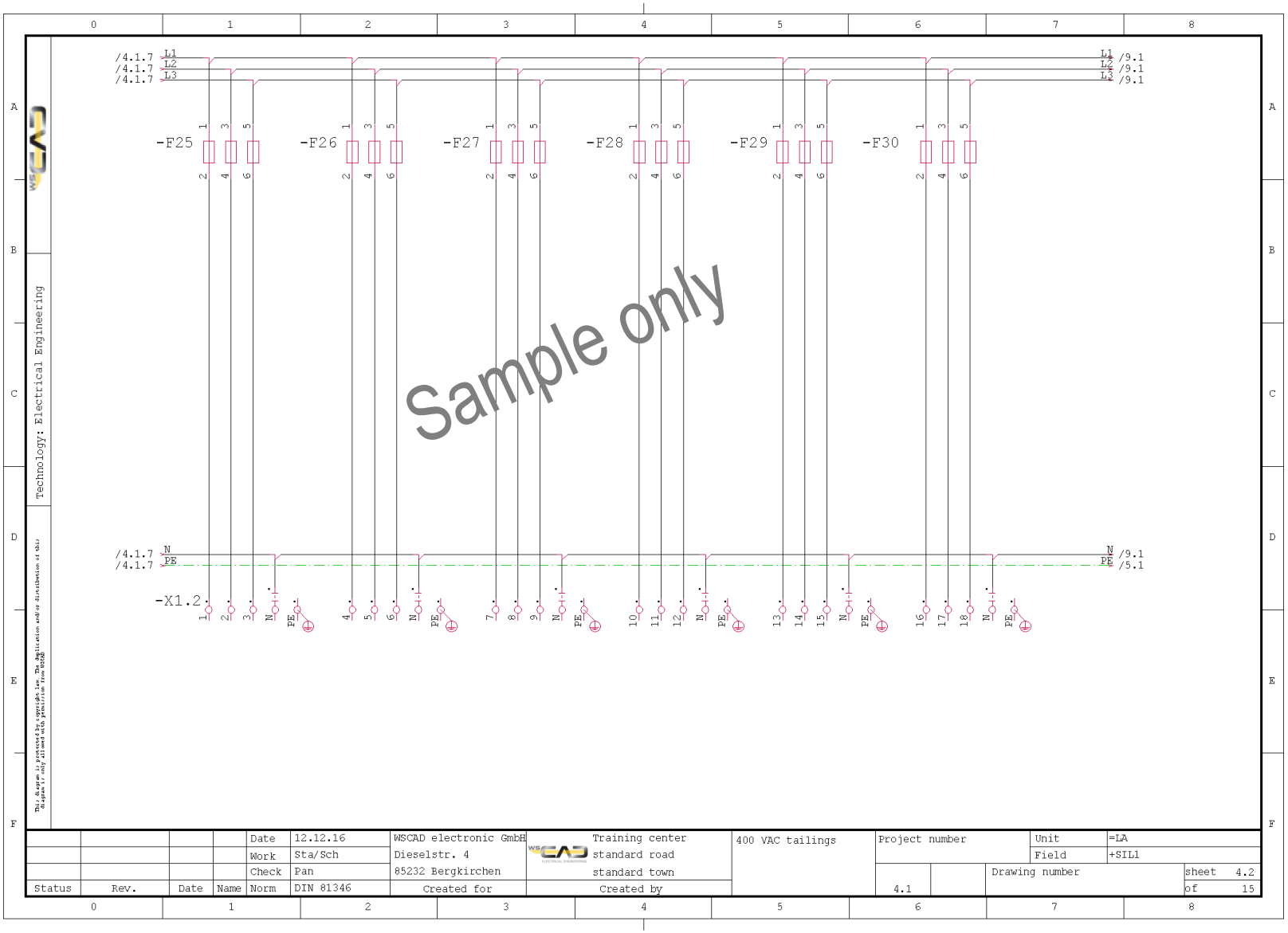
Basics – Terminals – Sheet 4/4



Basics – Terminals – Sheet 4/5



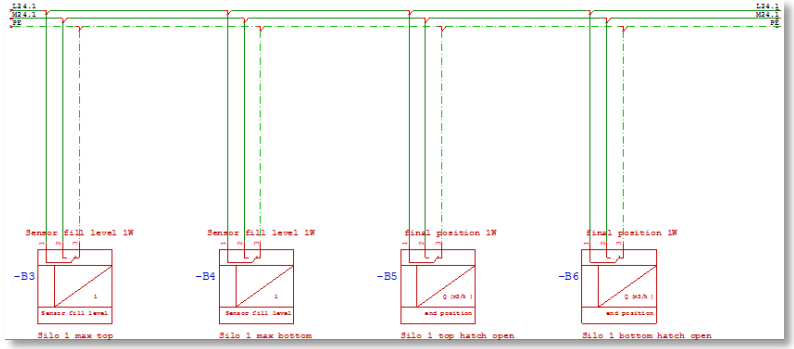
Basics – Terminals – Sheet 4/6



Basics – Multi-level terminals – Sheet 5

- Navigate to the macro "24V_PE_mittig" (24V_PE_centered) under "Potentials for Training" and place it.
- Draw the plan template sheet 5/1, slide 61.

1. The symbols for the monitors can be found in the library "BA_EE_MSR" under Wächter (monitors; leave a space between the terminal symbols of about 18 grids). The now automatically generated connections are wrong with respect to PE, but useful for the following placement. Later, you delete the T-pieces.

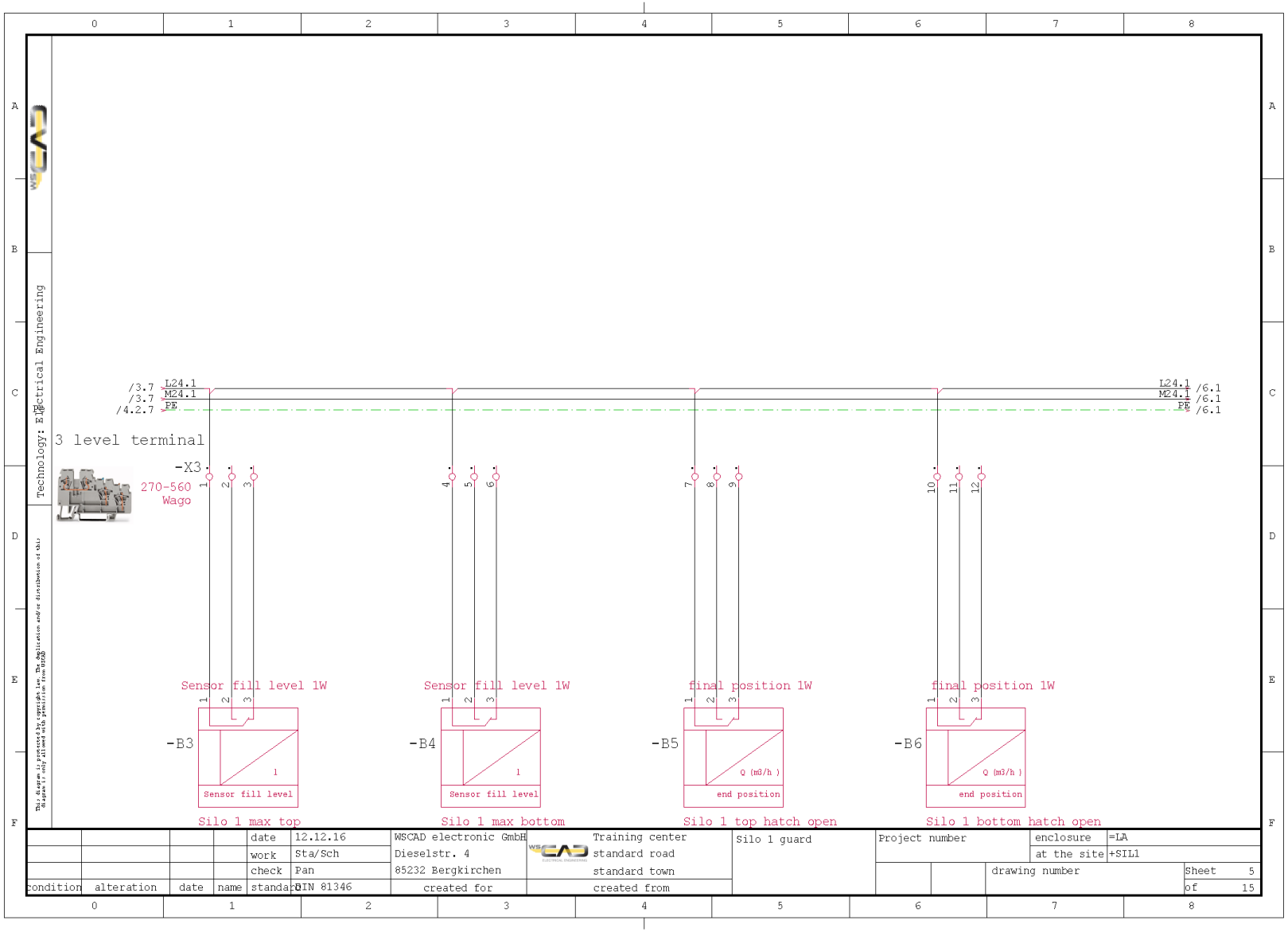


Symbols: 2x fill level sensors 1W
2x end positions 1W

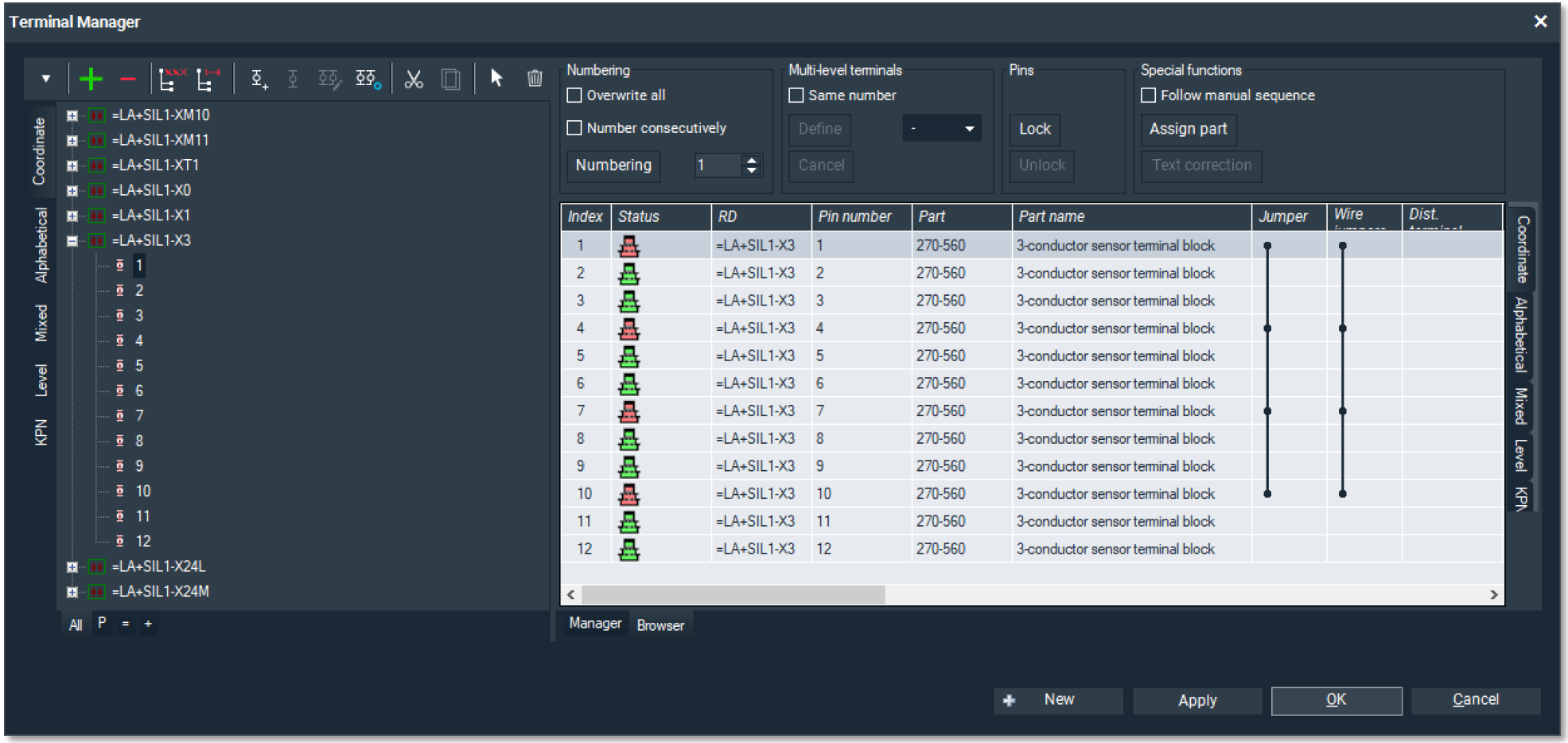
2. Place the terminals for the strip –X3 above the line:

3. In the Terminal Manager, press "New" for the terminal strip –X3 and define 12 terminals for this strip. Also place all other terminals via the "green" terminals in the Manager.
4. You then define all terminals as 3-level terminals in the Terminal Browser and assign the parts as indicated in, slide 62.

Basics – Multi-level terminals – Sheet 5/1



Basics – Multi-level terminals – Sheet 5/2



Basics – Multi-level terminals – Sheet 6

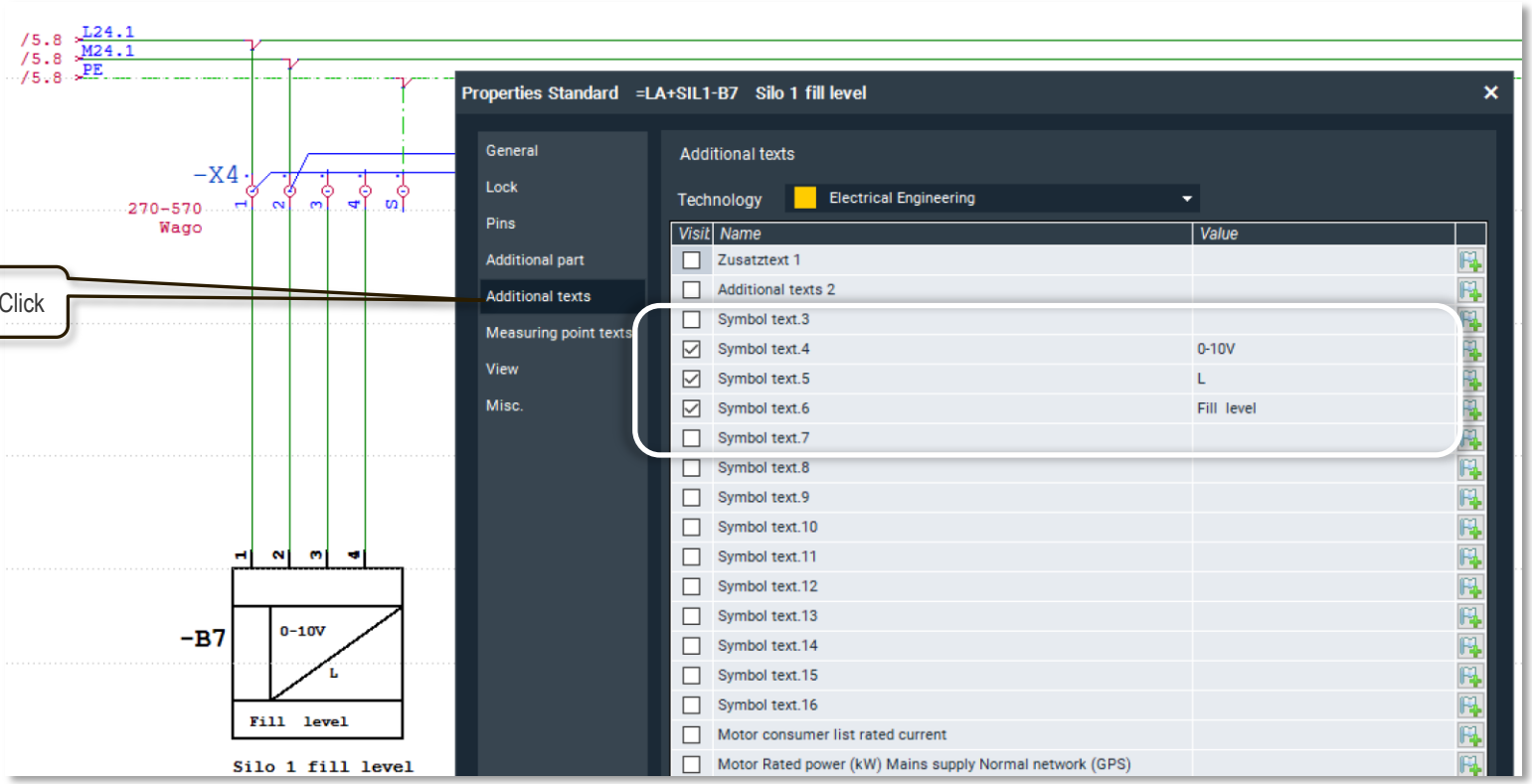
- Navigate to the macro "24V_PE_mittig" (24V_PE_centered) under "Potentials Training" and place it.
- Draw the plan template sheet 6/1, slide 65

1. The symbols for the sensors can be found in the library "BA_EE_MSR" under

   Sensoren (sensors). For –B7, use the symbol "Sensor_Transmitter allg. 4pol (sensor_Transmitter general 4-pin)".

   This symbol has been created with additional symbol texts. Then please edit this

   under the properties of the symbol as follows:



Basics – Multi-level terminals – Sheet 6

2. In the Terminal Manager,
   define the terminal strip –X4 with 10 terminals.

   Place the terminals for the strip –X4 individually.
3. You then define all terminals as 4-level terminals
   in the Terminal Browser

   and assign the parts and draw the bridges
   as indicated in sheet 6/2, slide 66.
4. Draw the bridge lines with command

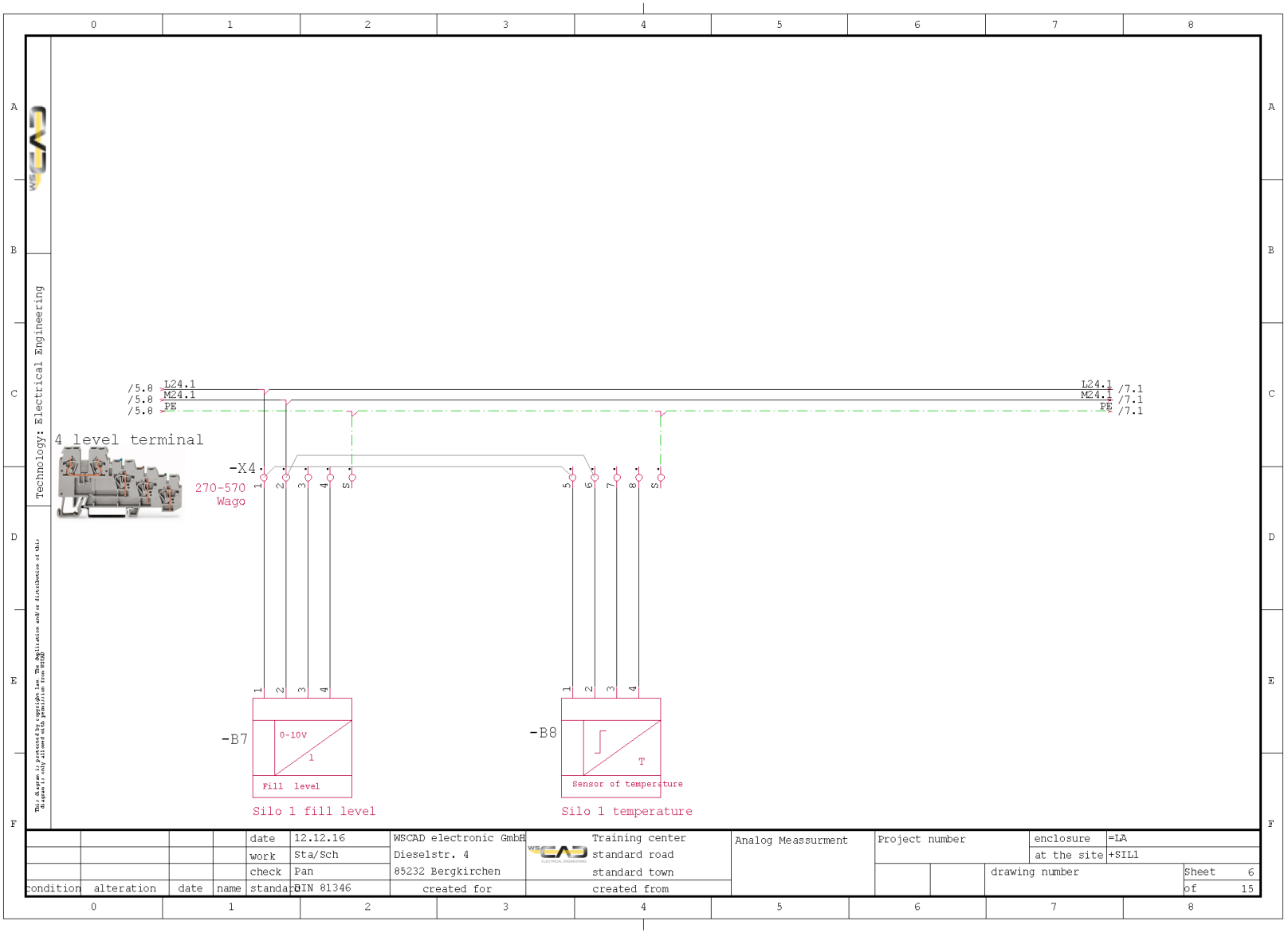
   

5. Turn off (F6) the orthogonal mode for this

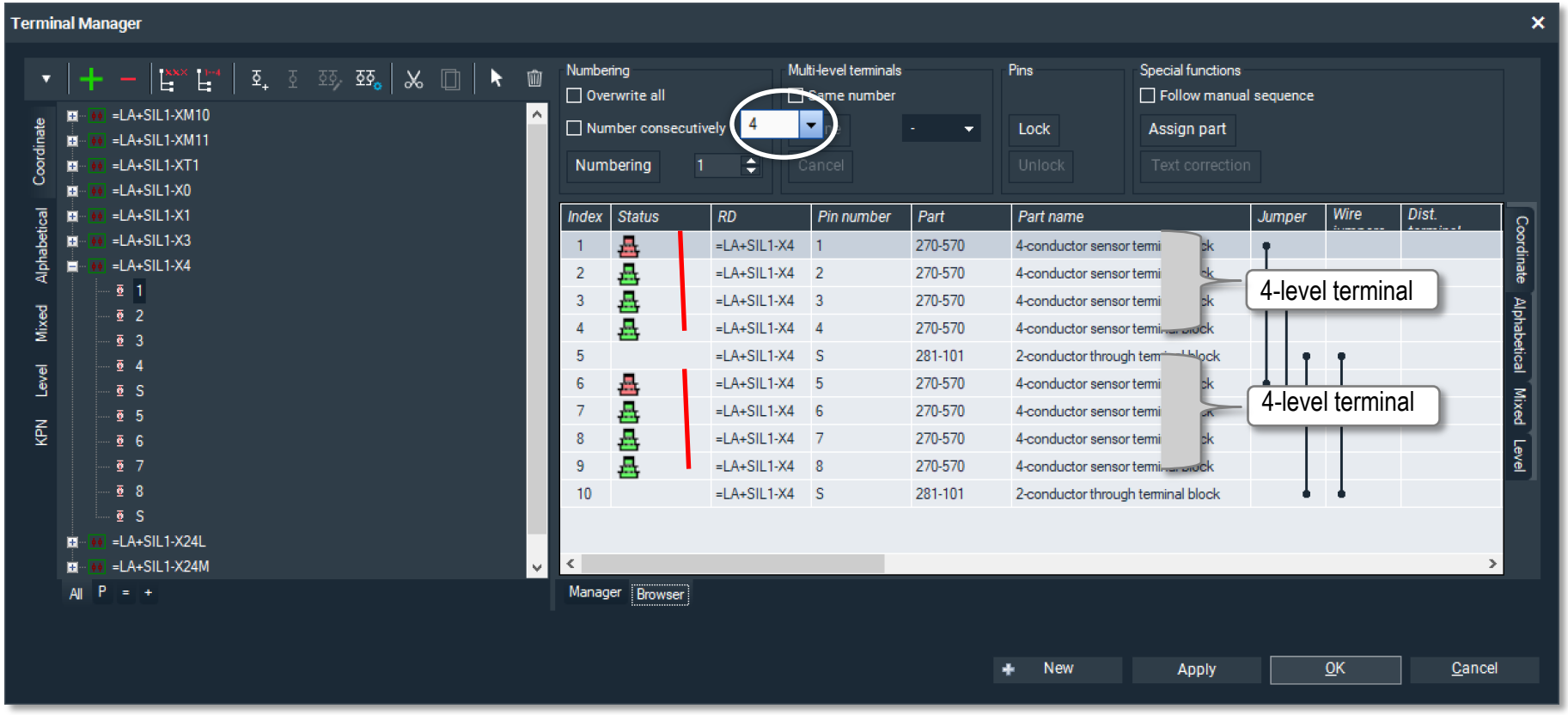
   

6. ***Don‘t forgett!*** Turn the orthogonal mode back to on (F6)

Basics – Multi-level terminals – Sheet 6/1

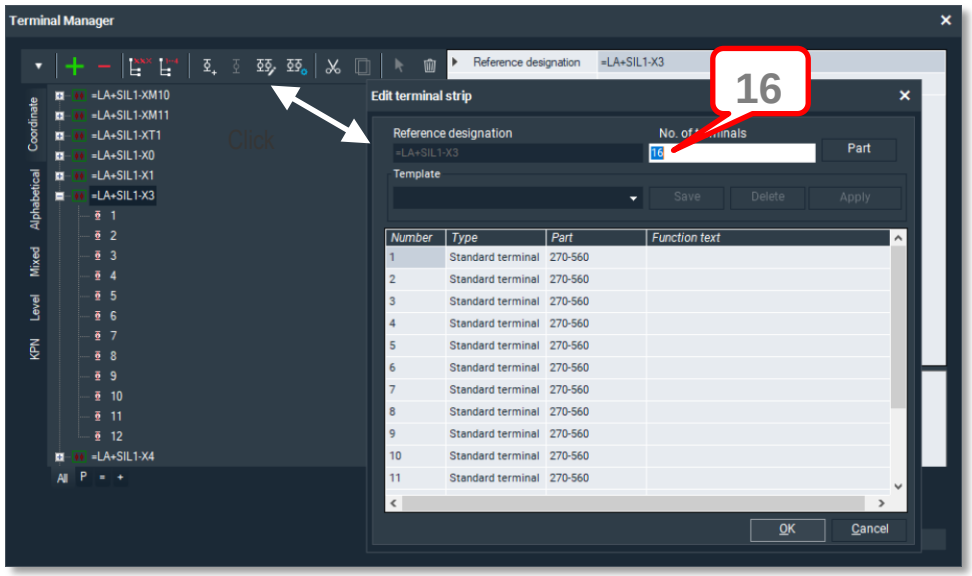


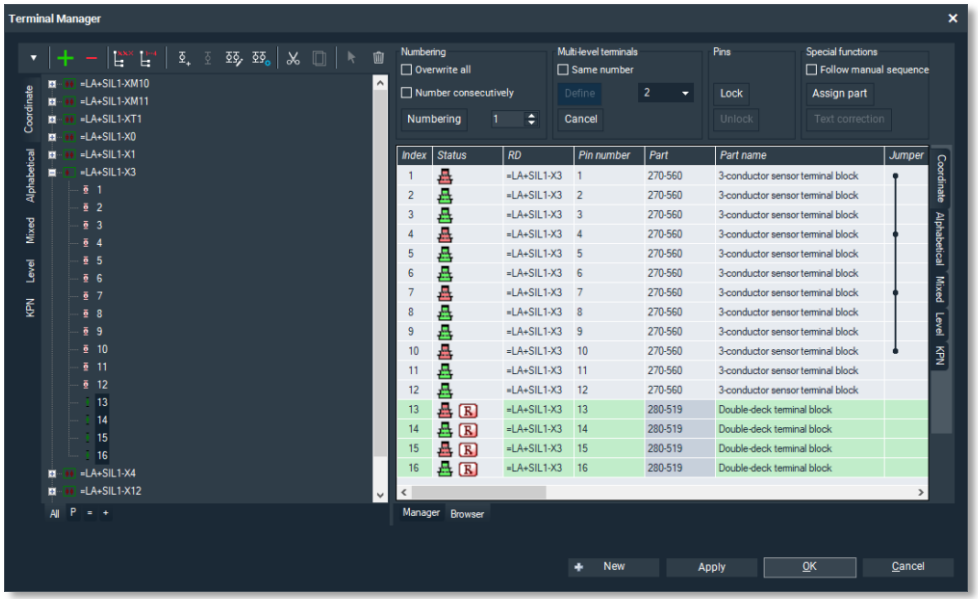
Basics – Multi-level terminals – Sheet 6/2



Basics – Terminal expansion – Sheet 7

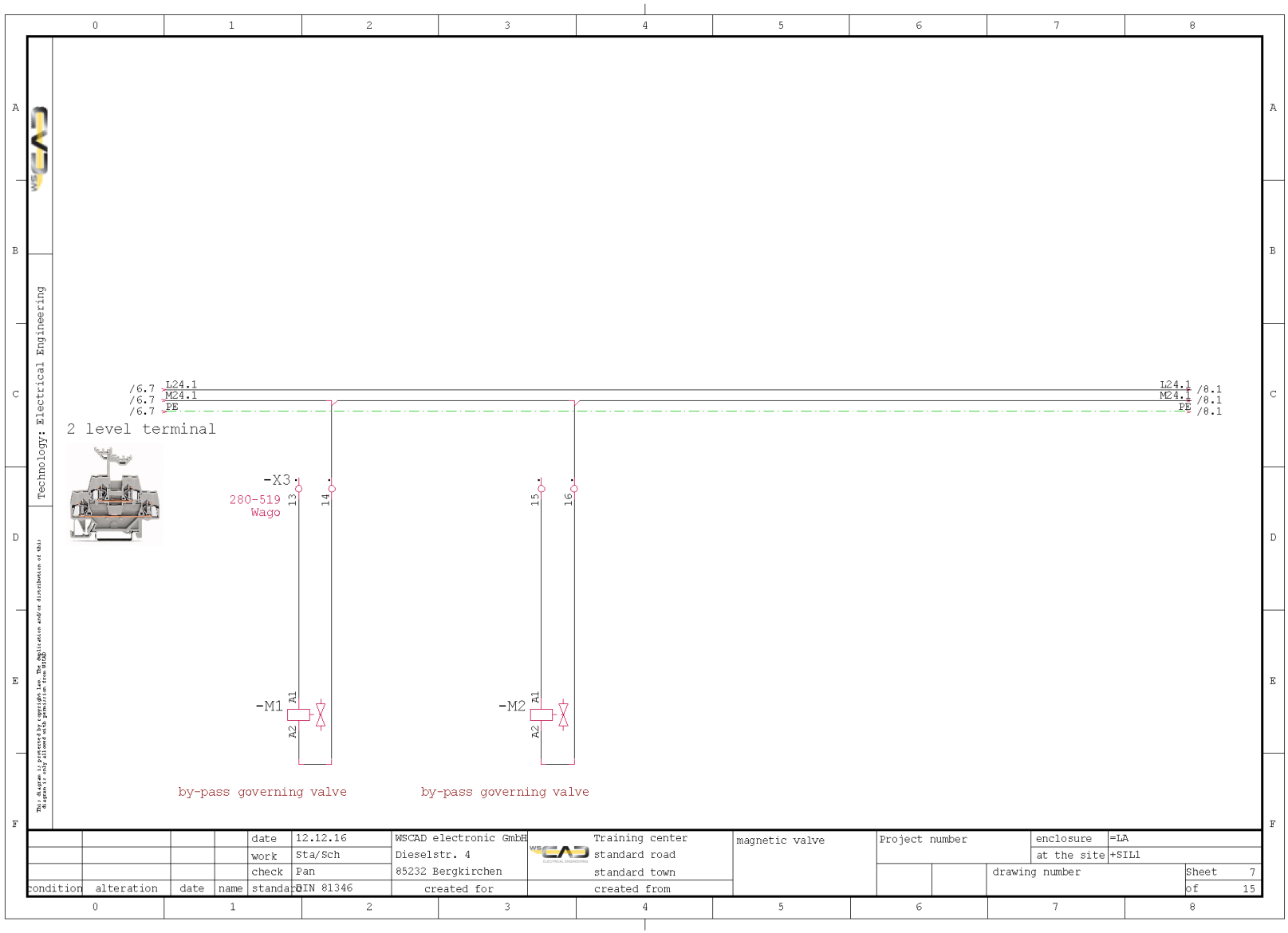
- Open sheet 7 from the project.
- Navigate to the macro "24V_PE_mittig" (24V_PE_centered) under "Potentials Training" and place it.
- Draw the plan template sheet 7/1, slide 68
  1. Open the Terminal Manager.
  2. In the Terminal Manager, expand terminal strip –X3 by 4 terminals by increasing the total number of terminals by 4 and assign the parts. Define the four new terminals as a two-level terminals.





3. Complete the drawing as in sheet 7/1, slide 68. The symbols for the control valves can be found by typing the term "control valve" in the Symbol Explorer under "Search".

Basics – Multi-level terminals – Sheet 7/1

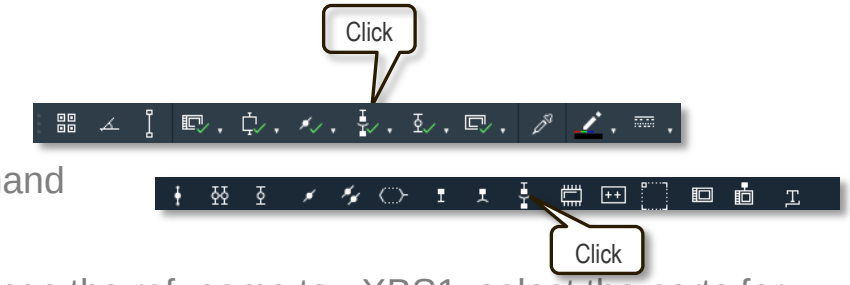


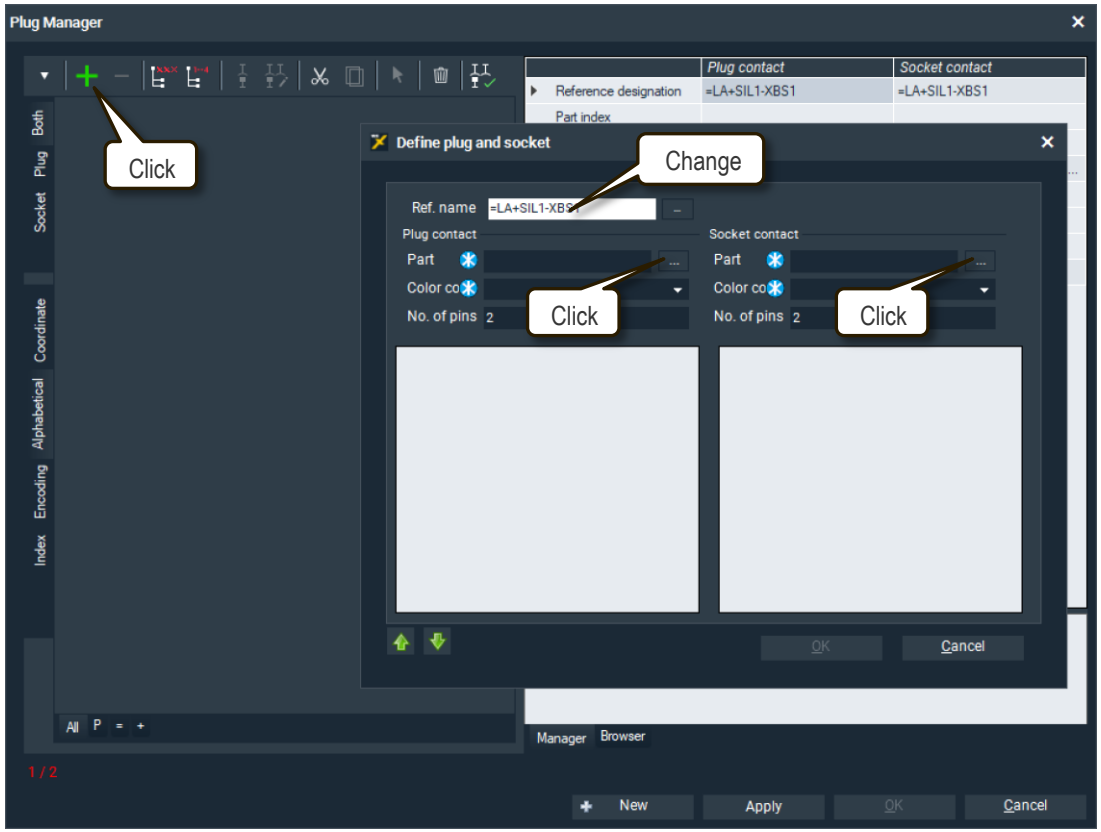
Exercise 8 – Plug Management

- Theoretical Section: **Plug management**
- Please devote your attention to the instructor. Thank you!

Basics – Plug management – Sheet 7

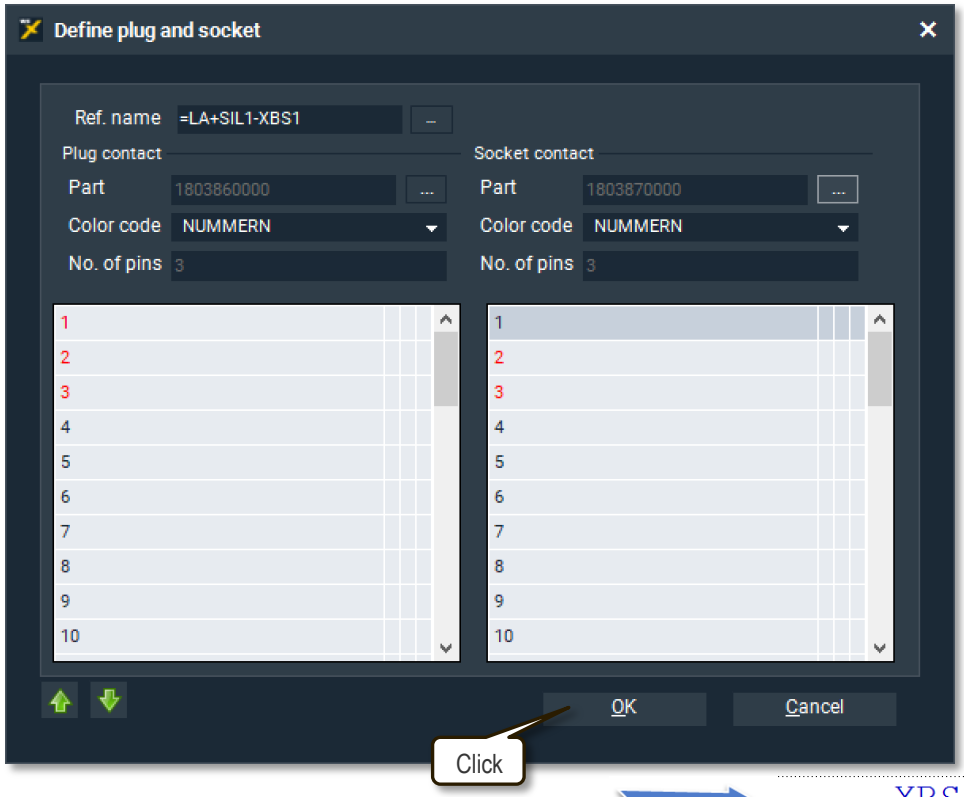
- Placing plug/socket connection above line
  1. Activate the terminal management
  2. Place a line over control valve 1 via the command over the control valves.
  3. In the Connection Manager, press "New", change the ref. name to –XBS1, select the parts for the plug contact and socket contact, and confirm.

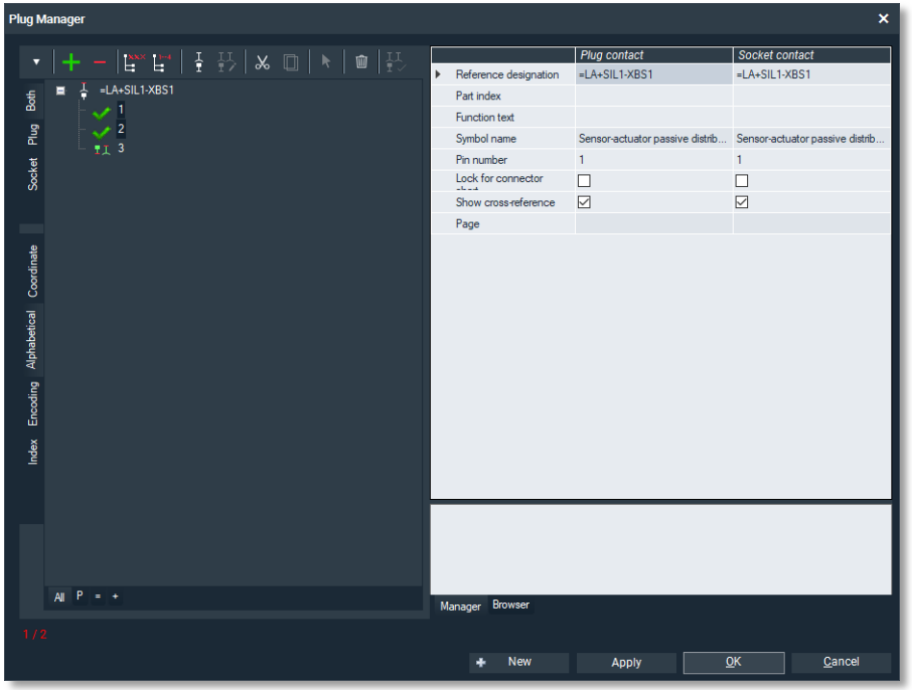




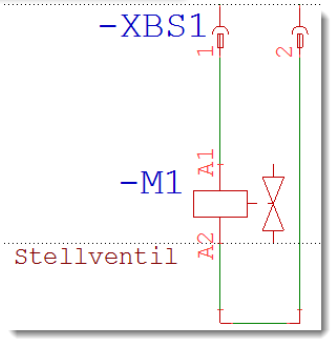
Basics – Plug management– Blatt 7

- Placing plug/socket connection above line





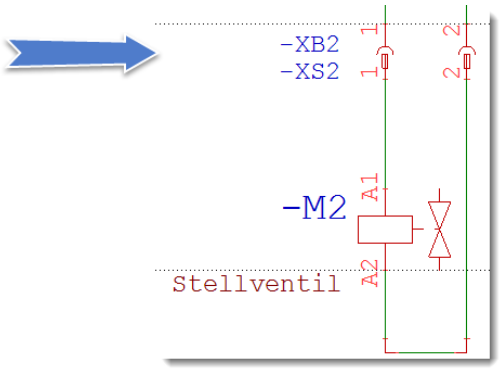
4. Ceck the result

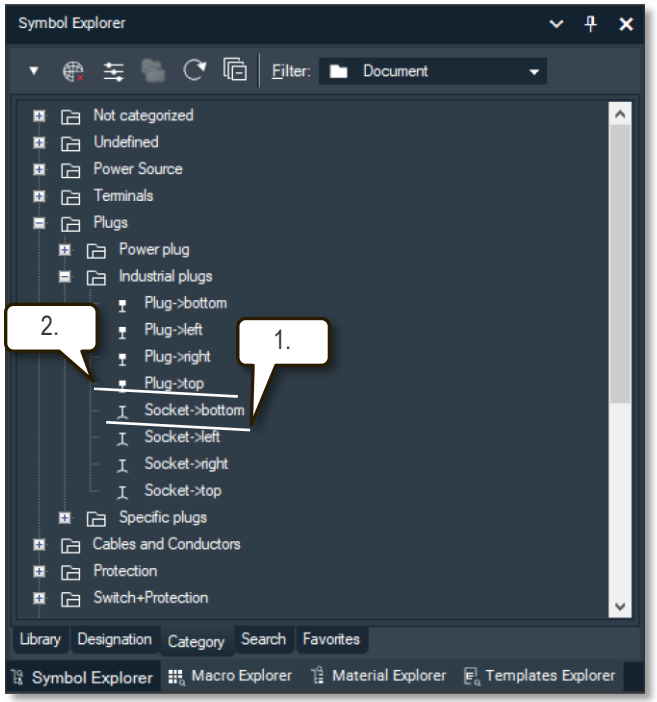


Übung Stecker Blatt 7 ca. 30 min

Basics – Plug management – Sheet 7

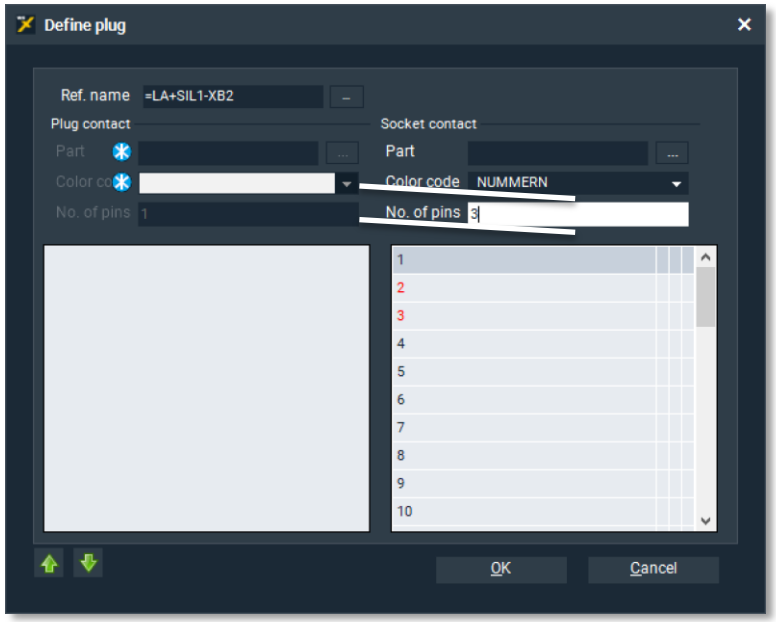
- Placement of plug/socket connection individually
  1. For the plug/contact connection above control valve 2, please use the placement of symbols via a selection from the Symbol Explorer by first placing the sockets (ref. name: –XB2) and then the plugs (ref. name: –XS2).
  2. In the Manager, press "New" in each case, select the parts and place them.
  3. Check the result

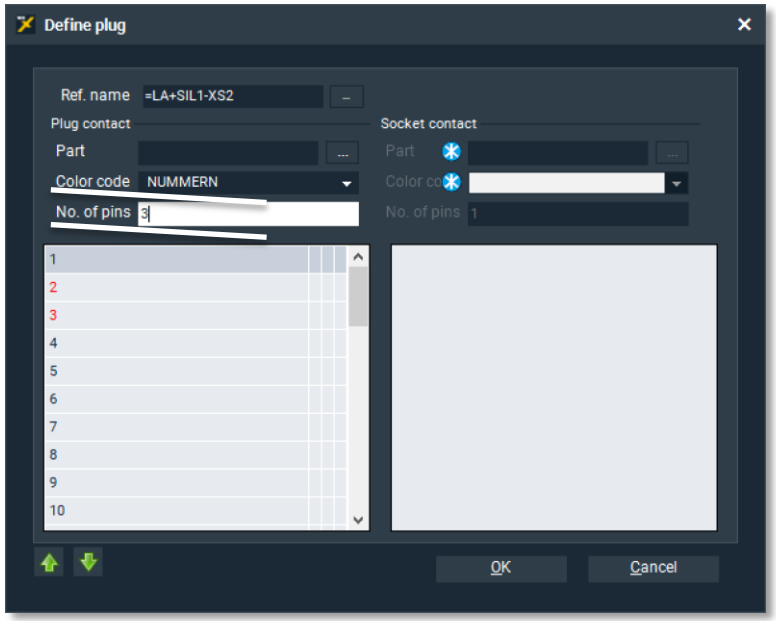




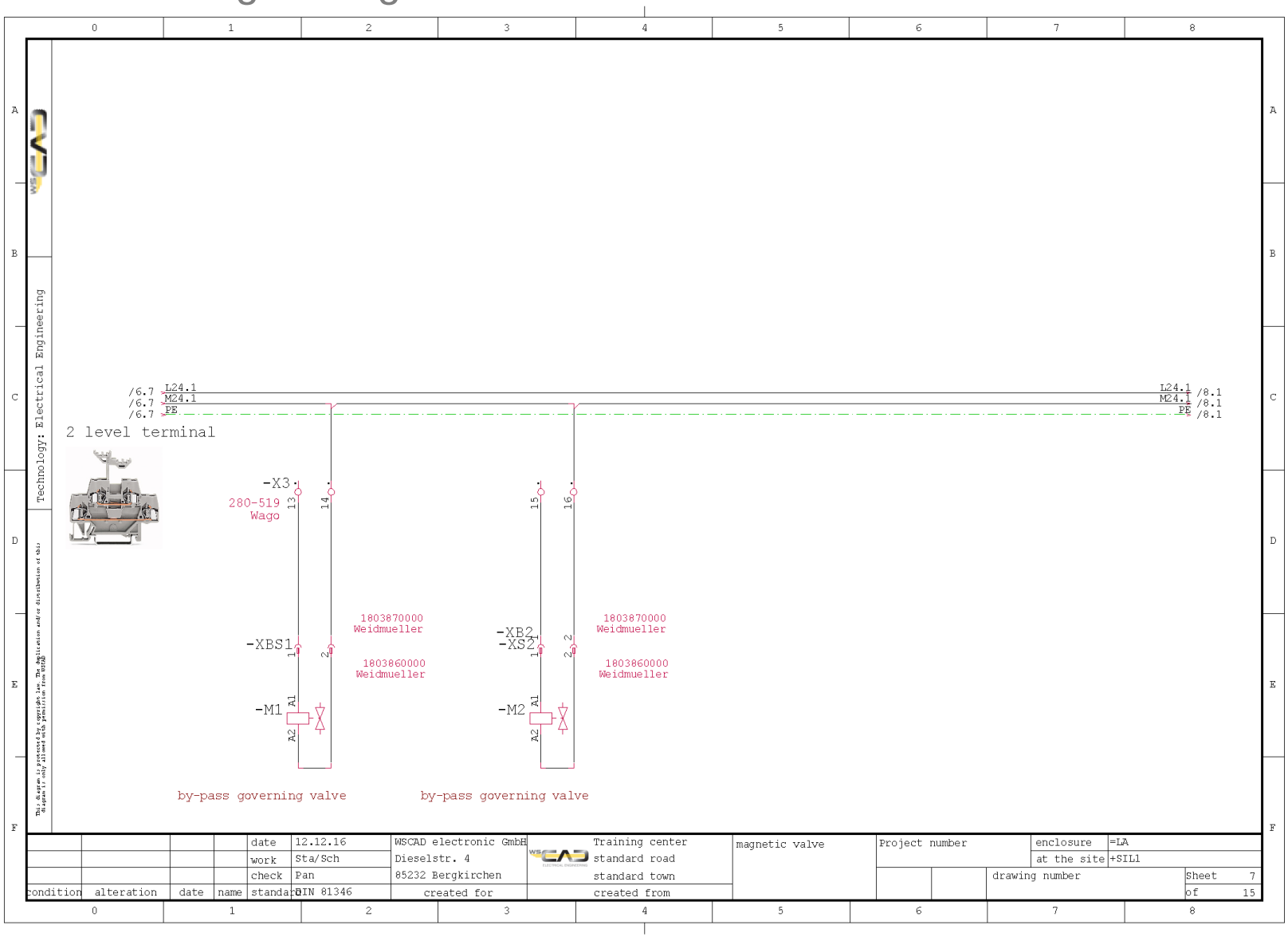
Basics – Plug management– Blatt 7

- Placement of plug/socket connection individually
  4. Open the Connection Manager and compare with slide 75.
  5. Delete –XB2 and –XS2 in the Manager and exit it.
  6. Repeat the placement of –XB2 and –XS2 awarded as described under 1., but do not assign any parts.
  7. Instead, choose a color code, and (important !!!) for the number of pins, enter as many pins as needed for your connection.

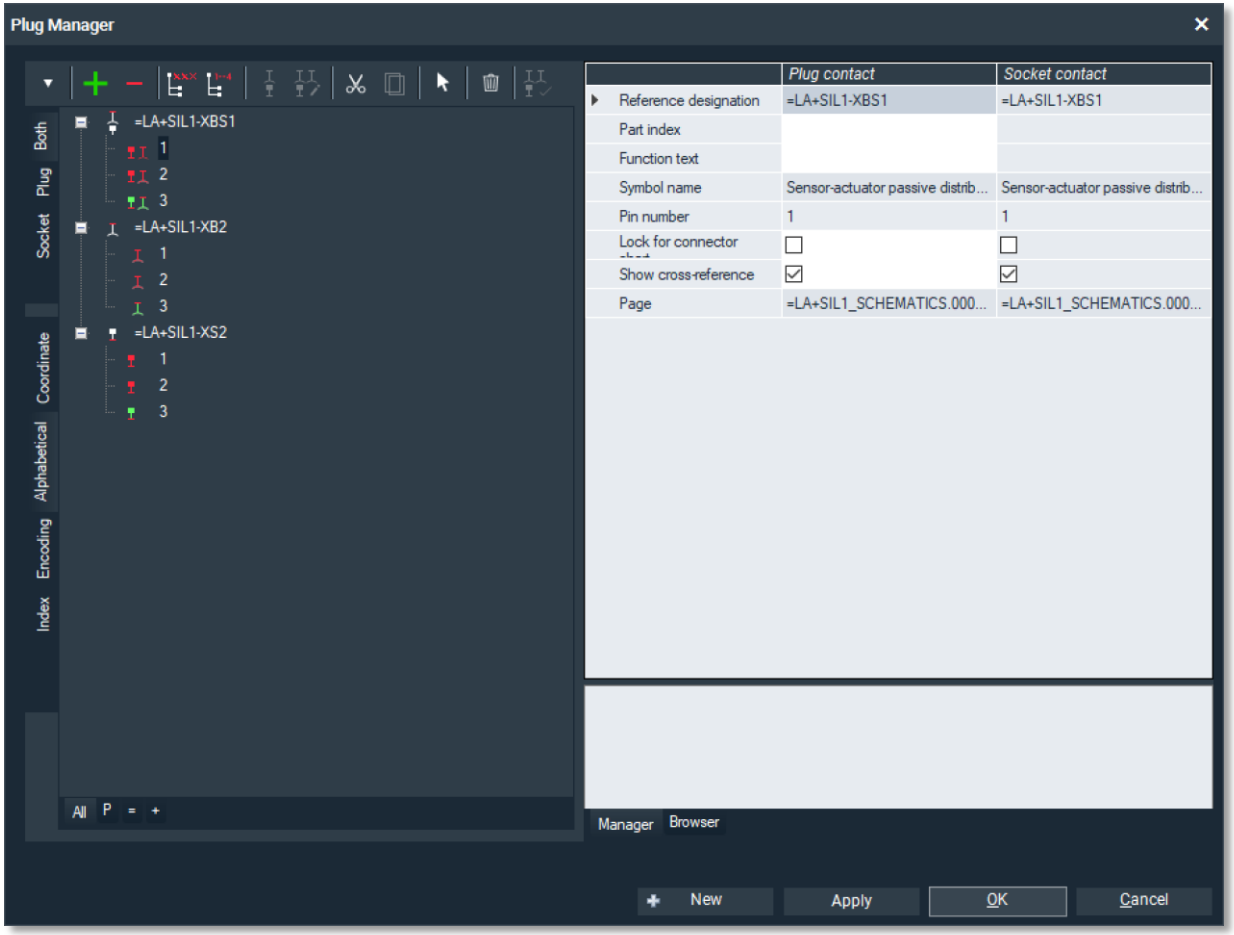




Basics – Plug management – Sheet 7/2



Basics – Plug management – Sheet 7/3



Exercise 9 – Contactor Manager

- Theoretical Section: **Contactor Management**

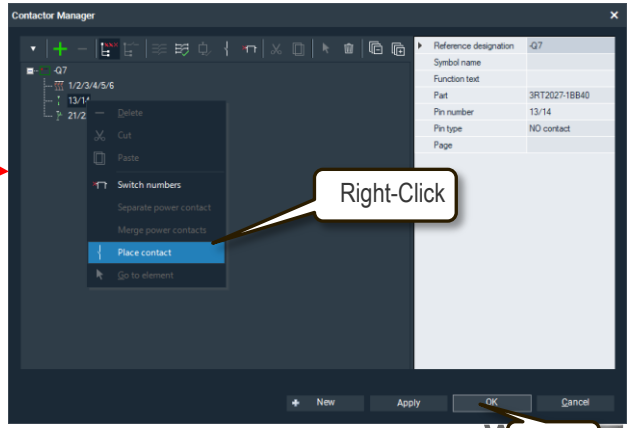
- Please devote your attention to the instructor. Thank you!

Basics – Contactor management

- Skip to project "Nonsense" (for training purposes)
  1. Open the "Nonsense" project and create a new sheet 3.
  2. Navigate to the macro "Sheet 3 motor only" under „all sheets Nonsens" and place it. See sheet Nonsense Sheet 3/1, Slide 78.

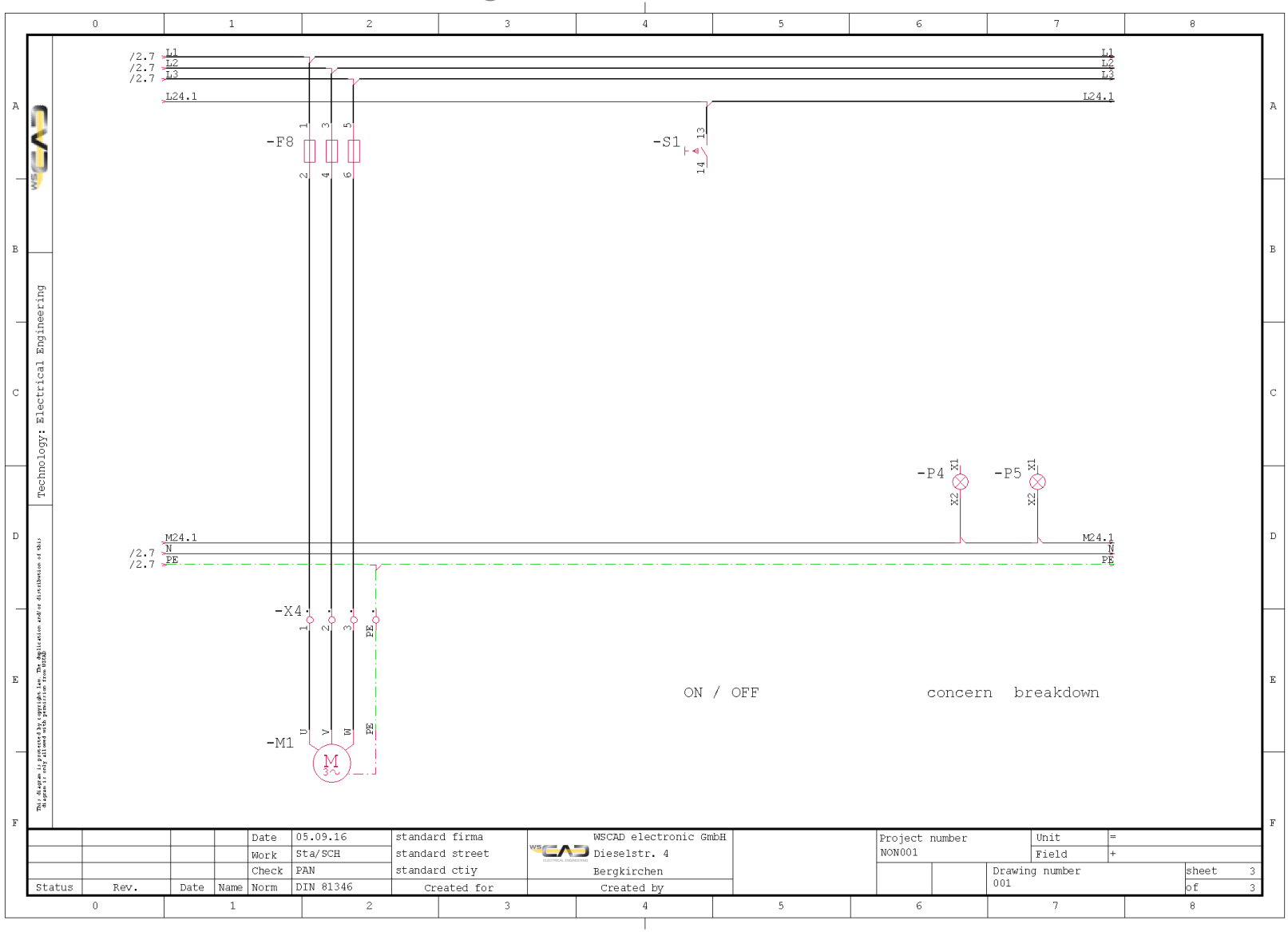
- Placement without parts
  1. Place a coil symbol. In the following Contactor Manager, click on "New", assign a function text in the properties and confirm 2x with "OK".
  2. Then place the contacts as in sheet Nonsense Sheet 3/2, Slide 79, and "marry" this via the Contactor Manager with the coil.
  3. Try the whole thing again by first placing a contact and then the coil. Please delete the spool with the contacts first.

- Placement with parts
  1. Delete the coil and contacts again.
  2. Place a coil symbol again and select a part (e.g., 3RT2027-1BB40) Place the N.o. contact (1S) through a placement from within the Contactor Manager.

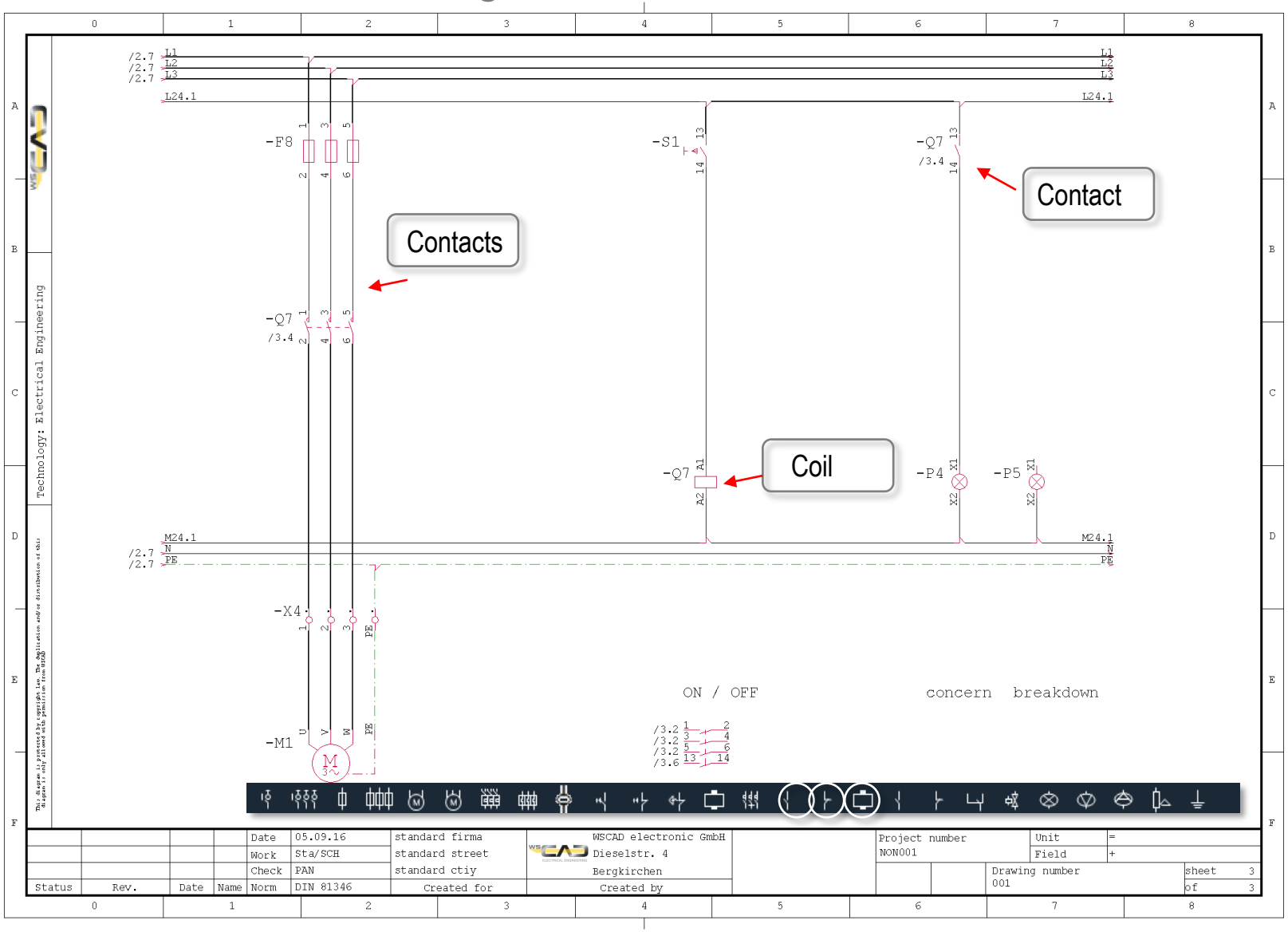


  3. Compare your result with Nonsense Sheet 3/3, Slide 80.

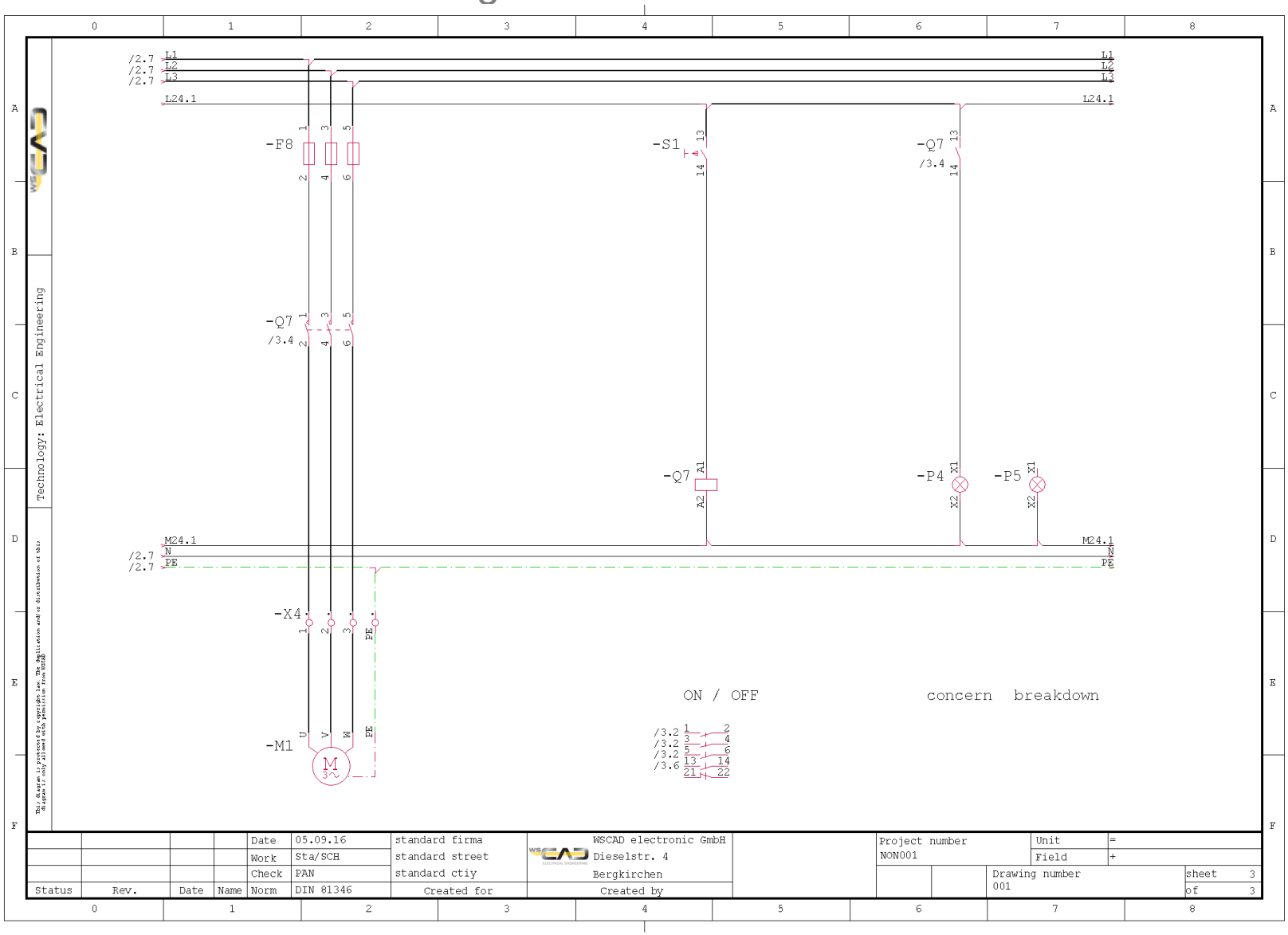
Basics – Contactor Manager – Nonsense Sheet 3/1



Basics – Contactor Manager – Nonsense Sheet 3/2



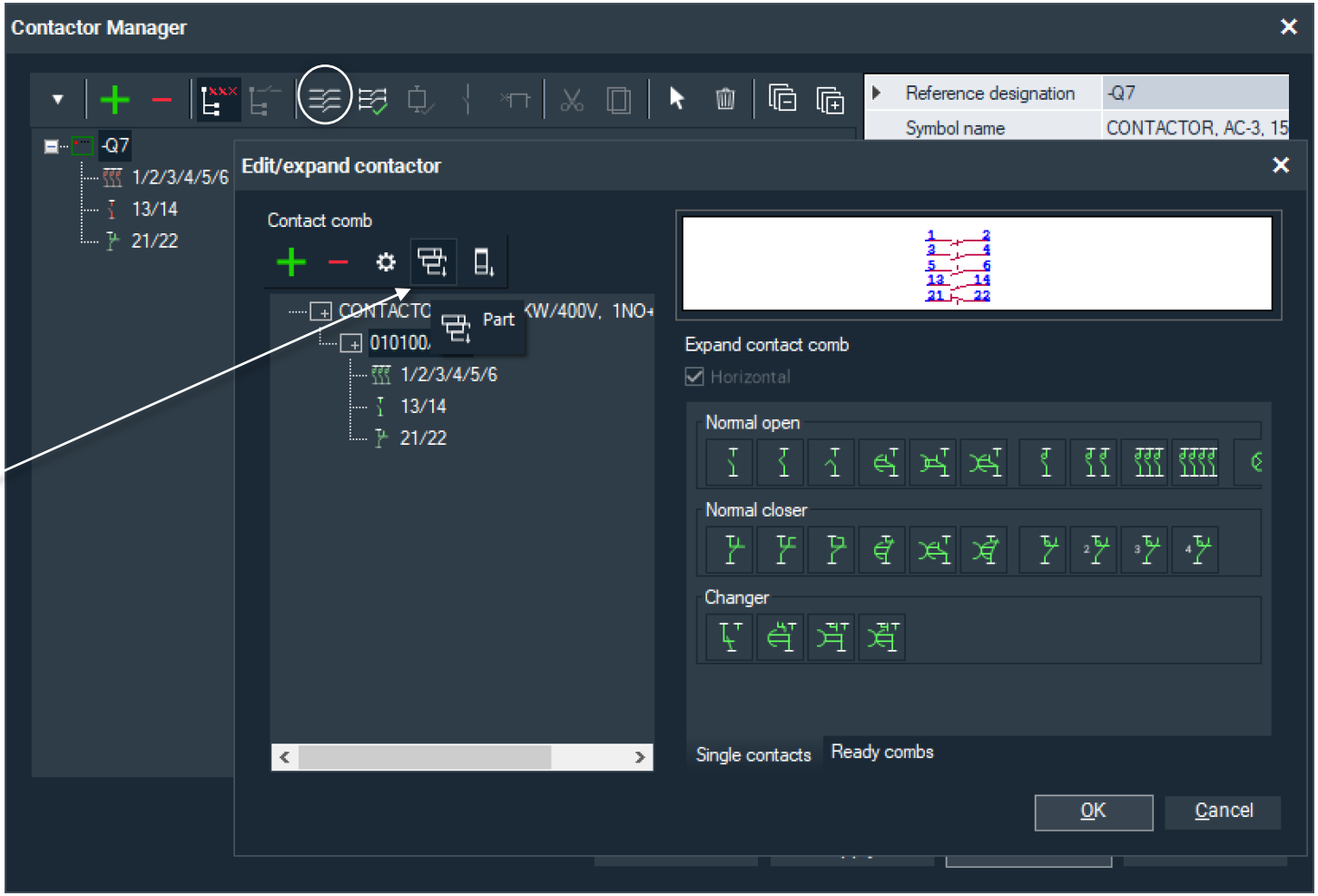
Basics – Contactor Manager – Nonsense Sheet 3/3



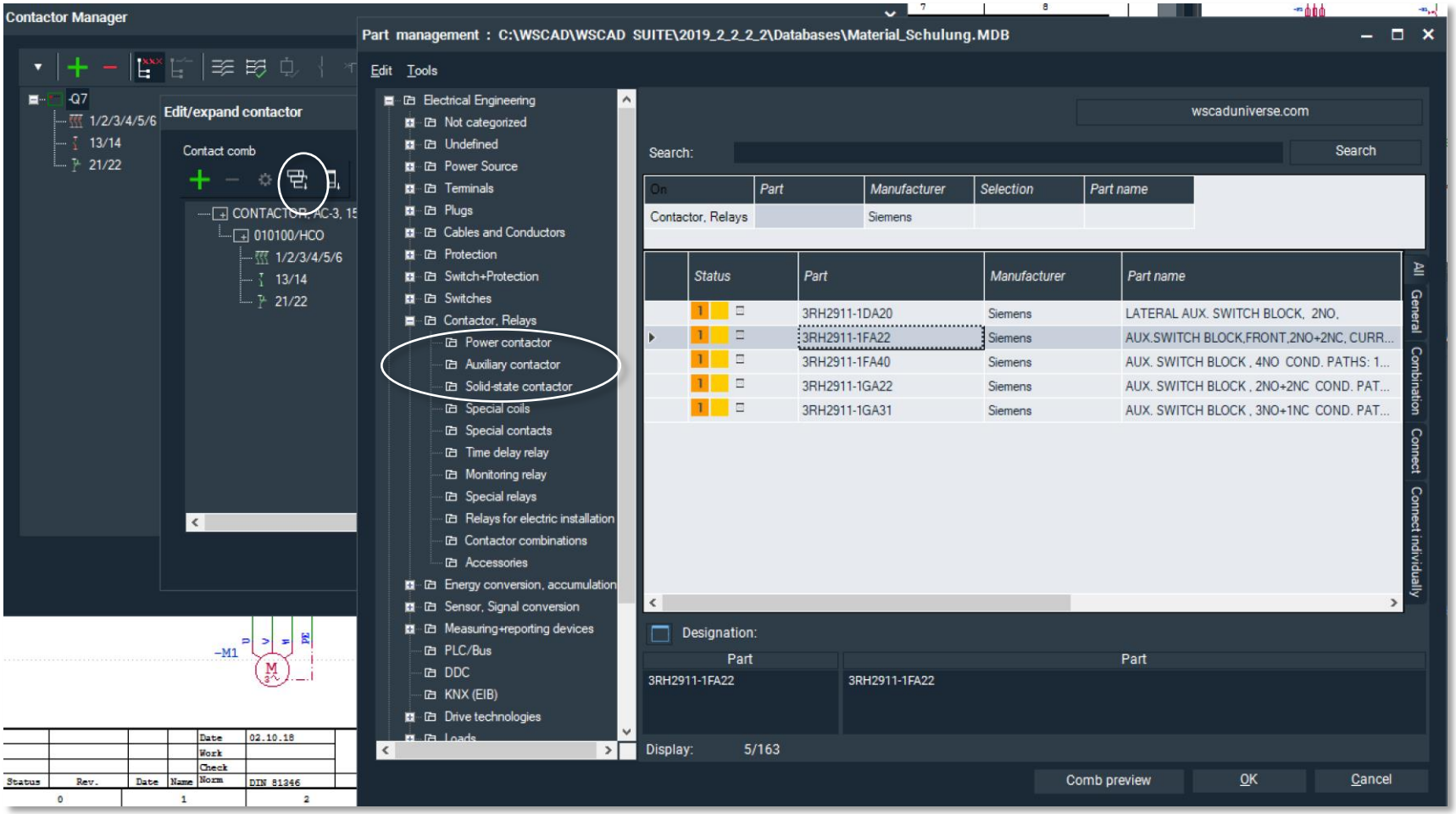
Basics – Contactor management - Contact extension

Open the Contactor Manager by right-clicking on the coil and expand the comb as indicated by using the "Edit comb„ command.

Call the article database



Basics – Contactor management - Contact extension



Switch to section

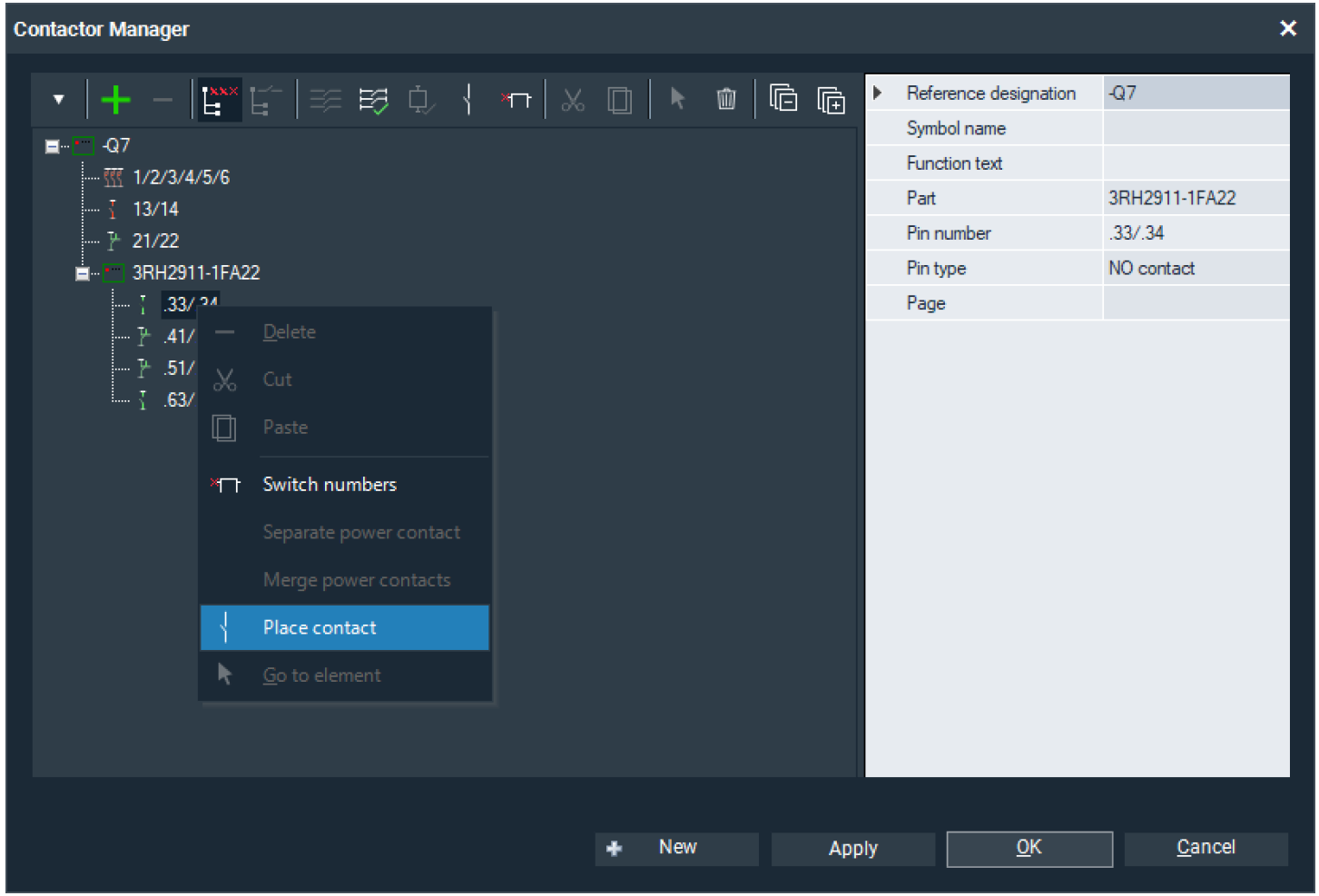
Auxilary contactor

Choose an Auxilary Switch Block

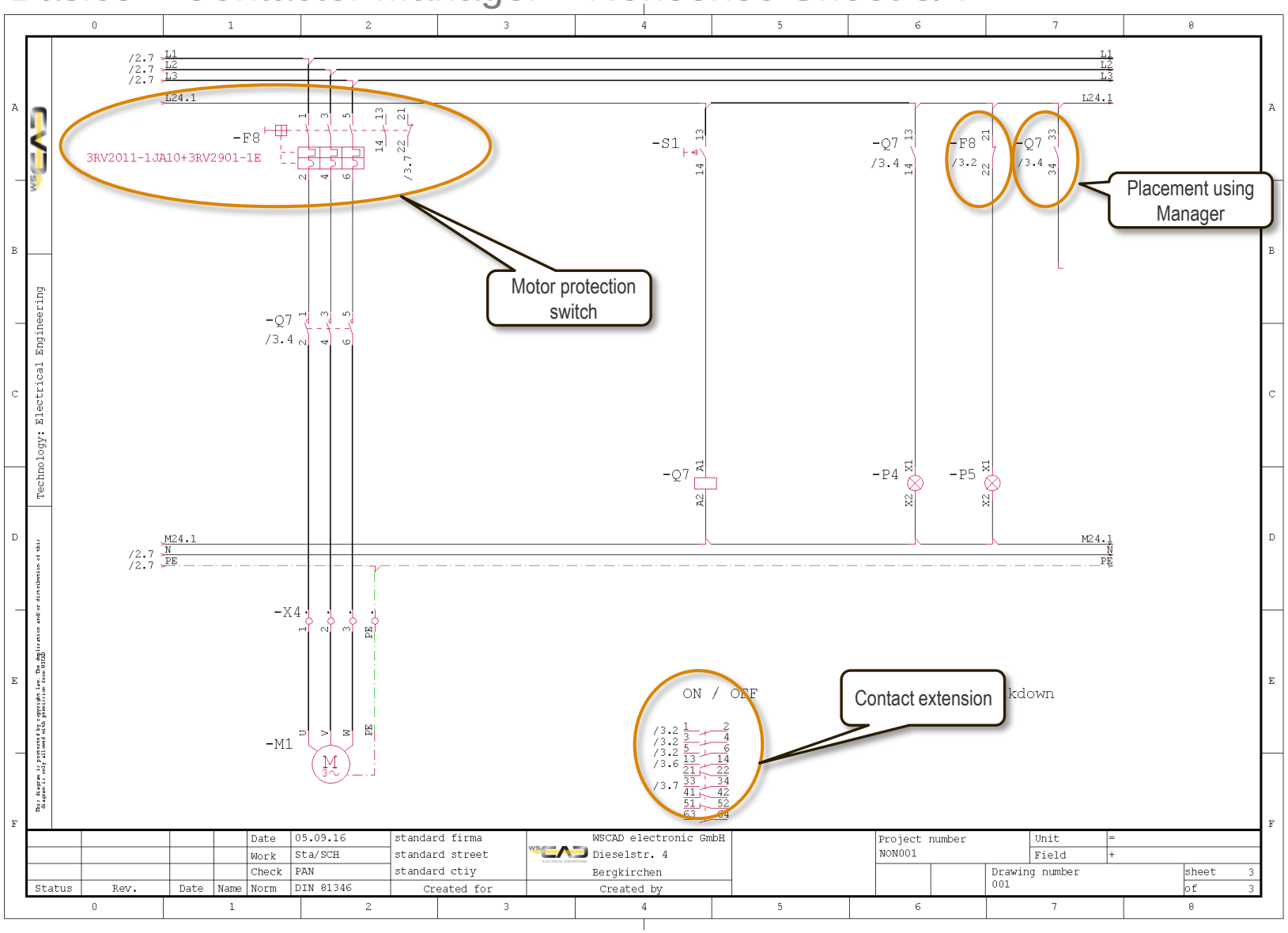
Basics – Contactor management - Contact extension

- Place another N.o. contact from the contactors as shown on slide 84.

- Delete the 3-pole fuse and instead place a motor protection switch above the part number as shown on slide 84 and place the N.c. contact.



Basics – Contactor Manager – Nonsense Sheet 3/4



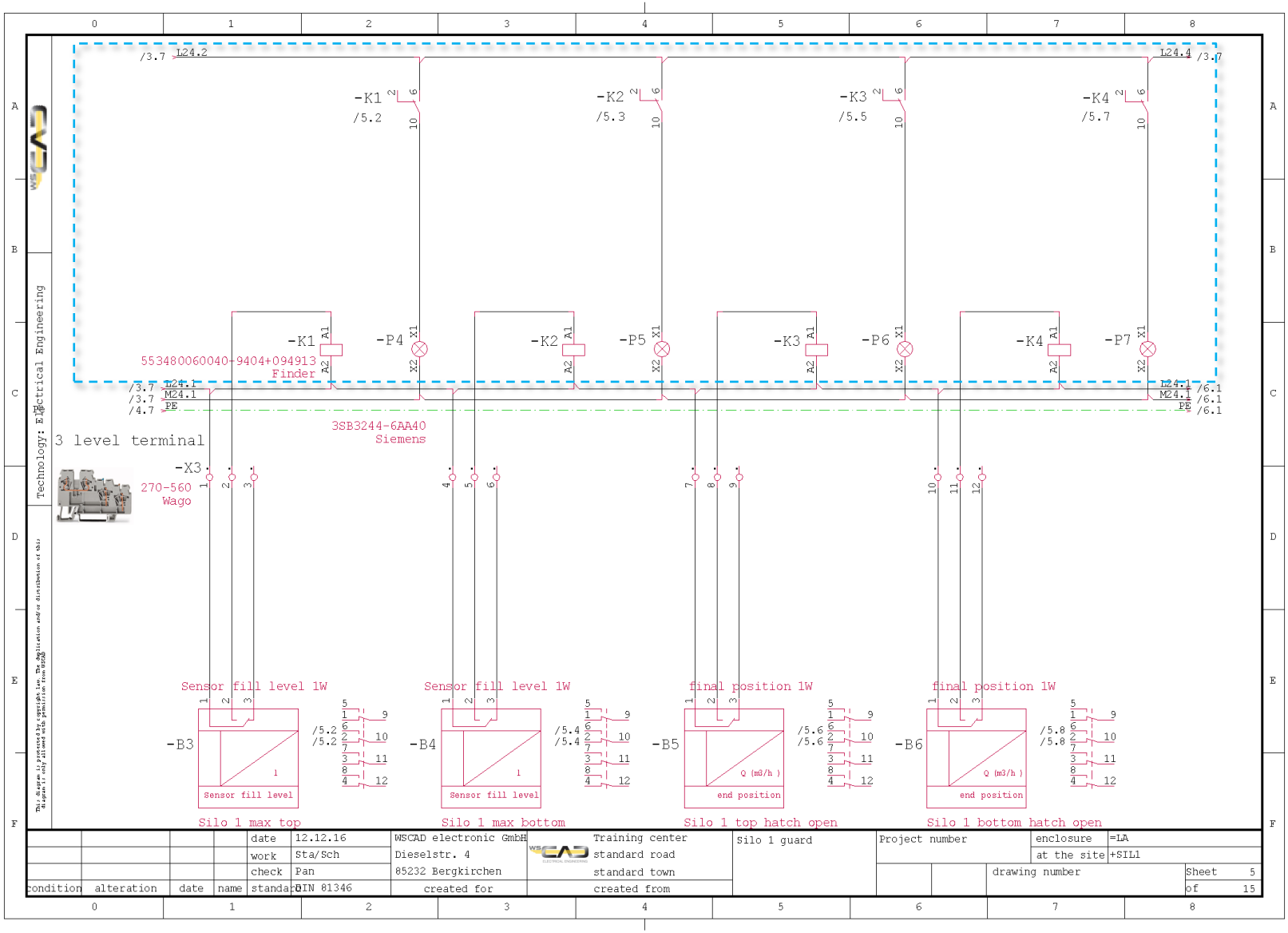
Basics – Additions

- Skip to "Training Basics" project
- Open *sheet 5* in the project here and add the relay, contacts and lamps to it as indicated in sheet 5/3, slide 86.

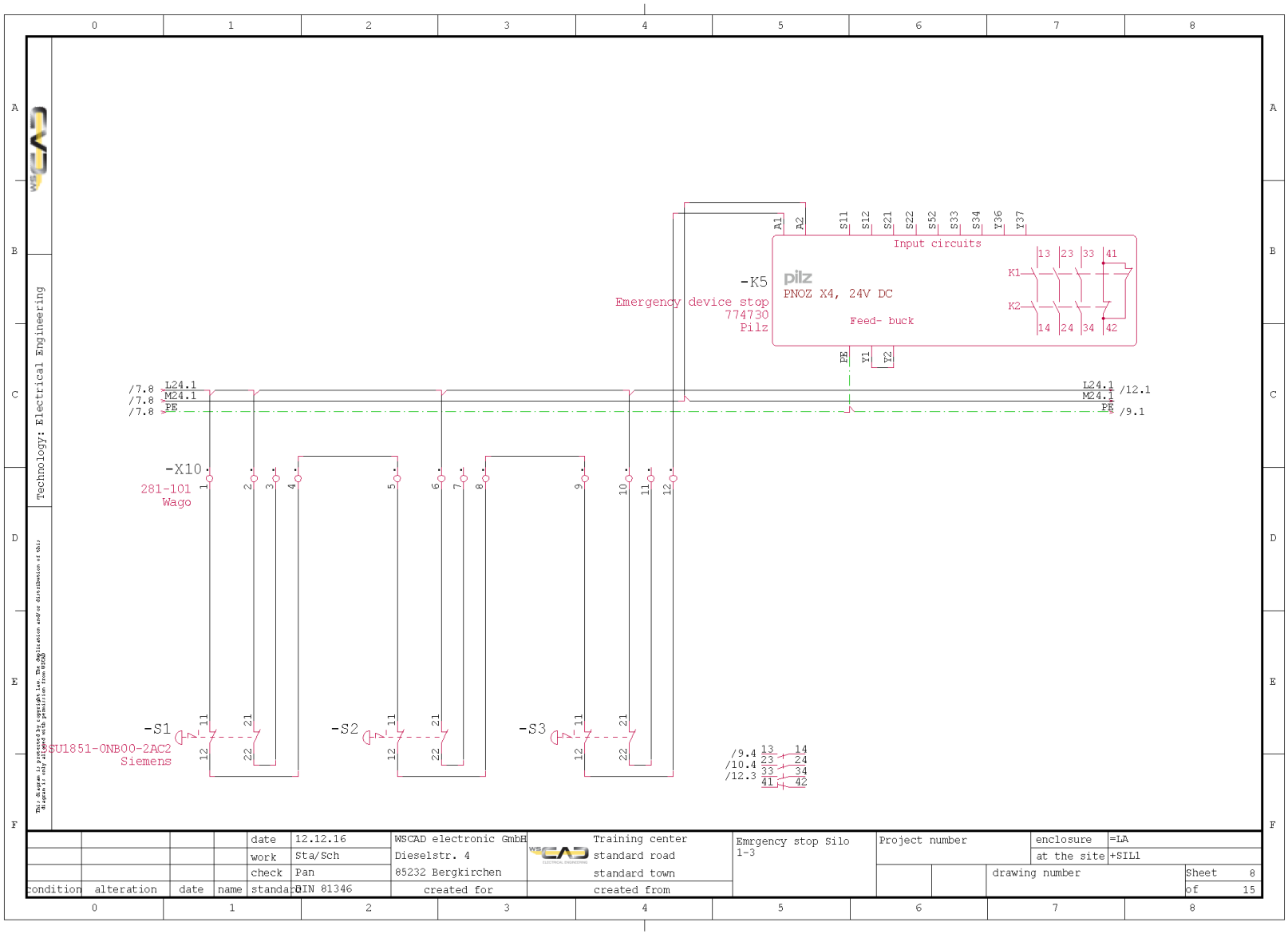
- Open *sheet 8* in the project and place the macro "„sheet 8 emergency switch" from the macro folder "Seiten für Schulung (pages for training)" as indicated in sheet 8/1, slide 87.

- Processing *sheet 9*
  1. Open sheet 9 in the project and place the macro "Phasen 3NPE_24V (phases 3NPE_24V)" from the macro folder "Potentials Training"
  2. Draw the plan template sheet 9/1, slide 88.

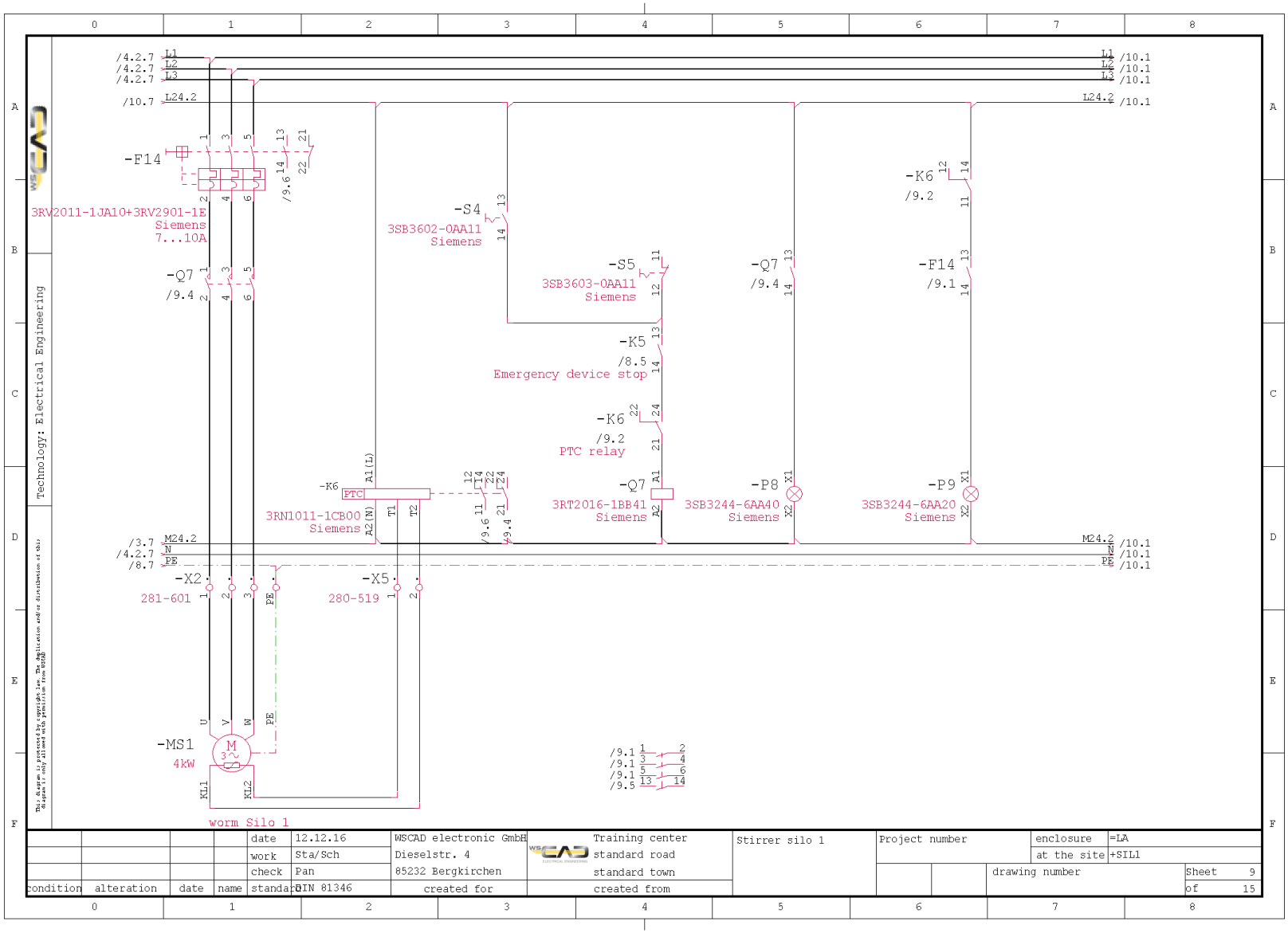
Basics – Additions – Sheet 5/3



Basics – Additions – Sheet 8/1



Basics – Additions – Sheet 9/1



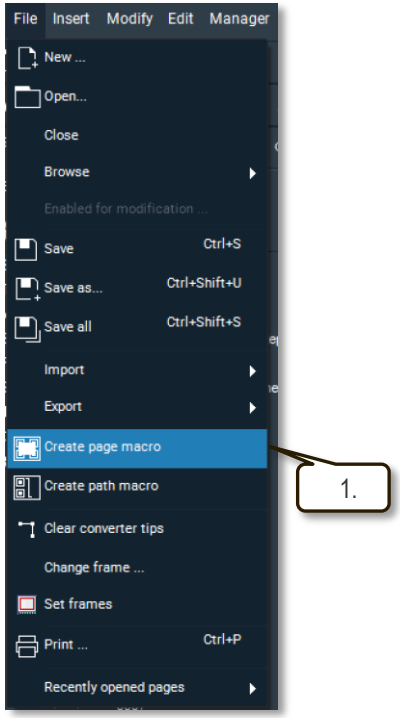
Exercise 10 – Macro Creation Part 1

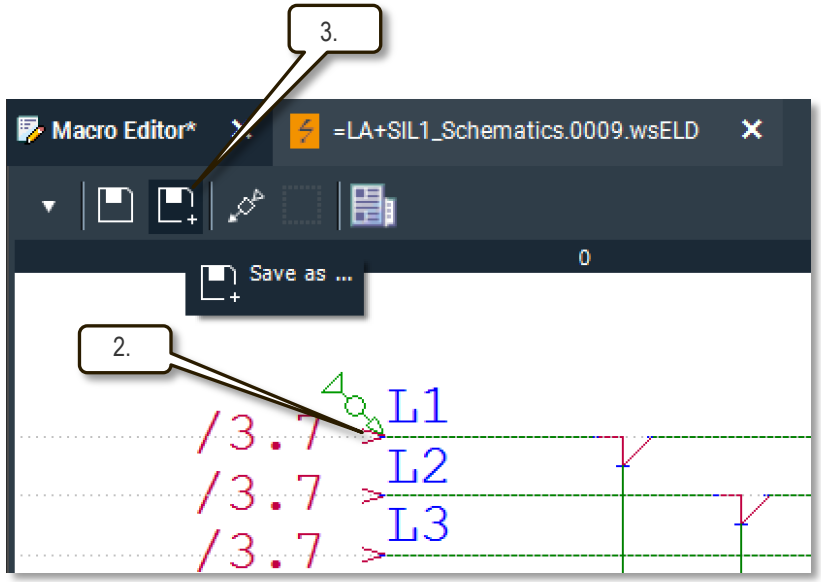
- Theoretical Section: **Macro Creation**

- Please devote your attention to the instructor. Thank you!

Basics – Additions

- Saving macros
  1. Save sheet 9 as a page macro by clicking on "Create page macro" in the menu bar.
  2. Set the insert point new on this place.
  3. In the following Macro Editor, save it as "macro sheet 9".



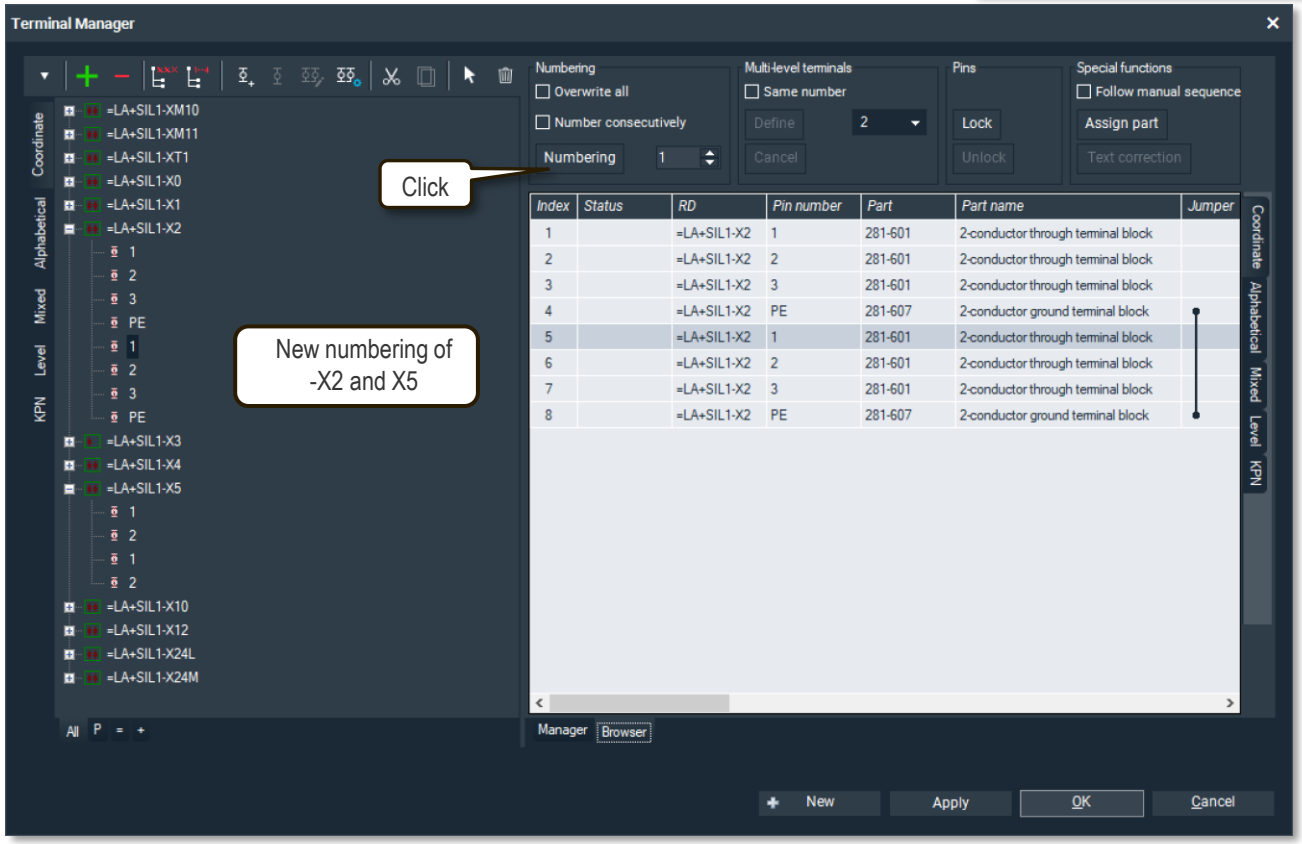


4. After that, the Macro Editor can be closed with the cross (x) button.

Basics – Inserting a complex macro

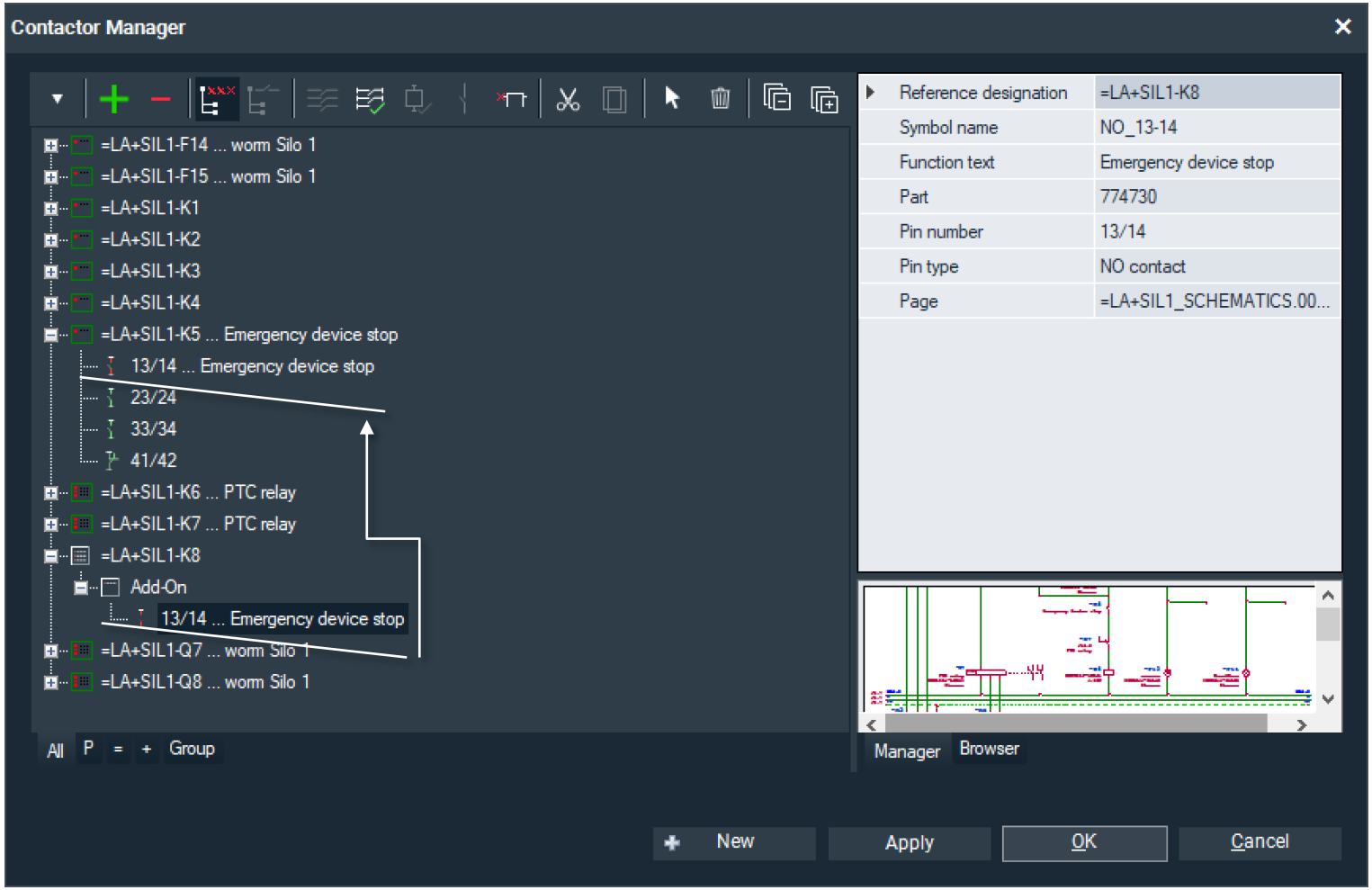
- Processing sheet 10

1. Open sheet 10 in the project and disable the terminal management.
2. *Attention: with an active terminal management new terminal strip will be crated*
3. Place the "macro sheet 9" macro saved above (as in sheet 10/1, slide 93).
4. Enable the terminal management and go to the Terminal Browser
5. Correct the terminal strips –X2 and –X5.

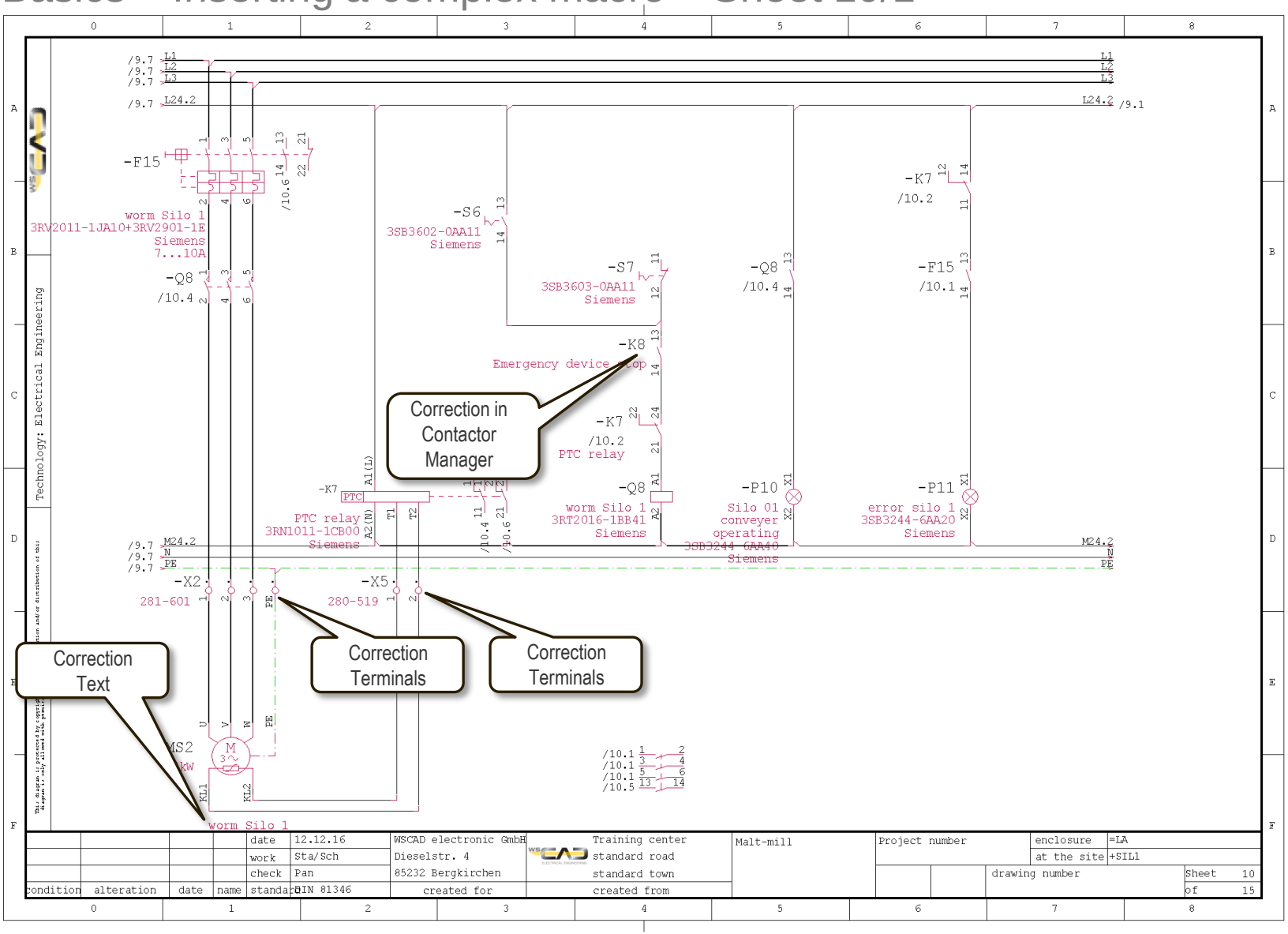


Basics – Inserting a complex macro

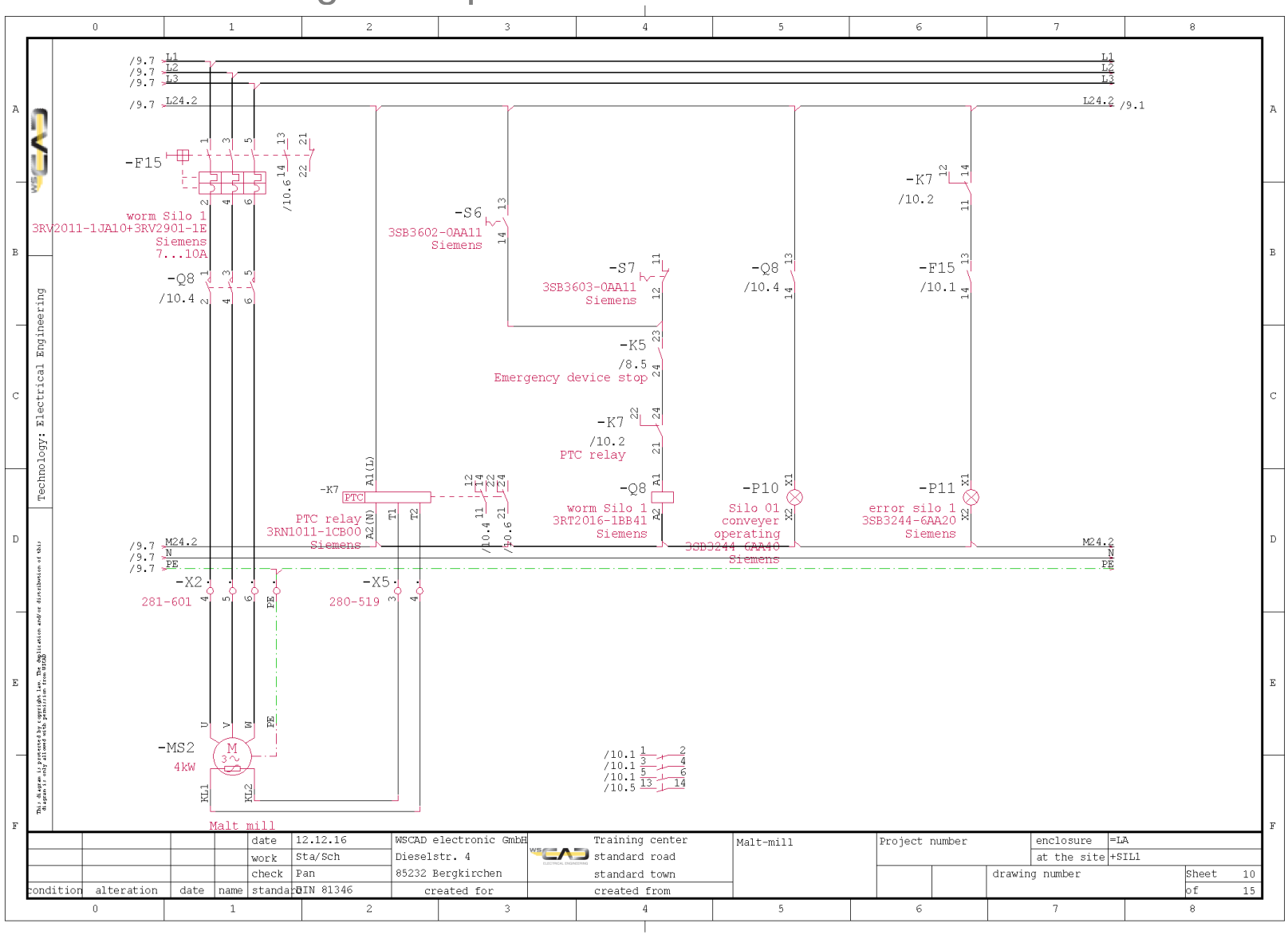
- Processing sheet 10
  5. Correct the contact –K8 and then check the result with sheet 10/2, slide 94.



Basics – Inserting a complex macro – Sheet 10/1



Basics – Inserting a complex macro – Sheet 10/2



Exercise 11 – Parent/Child Elements

- Theoretical Section: **Parent/Child elements**
- Please devote your attention to the instructor. Thank you!

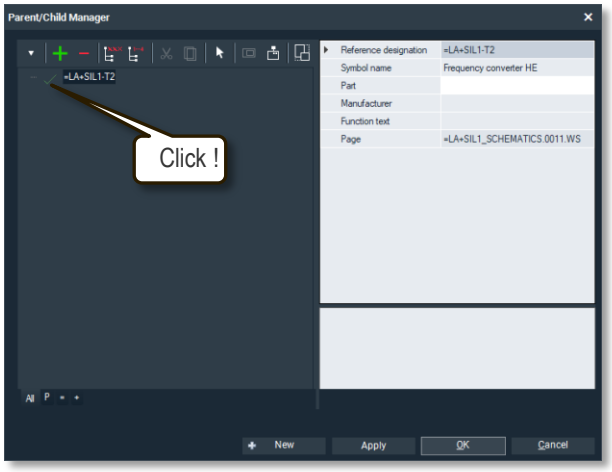
Basics – Parent/Child elements

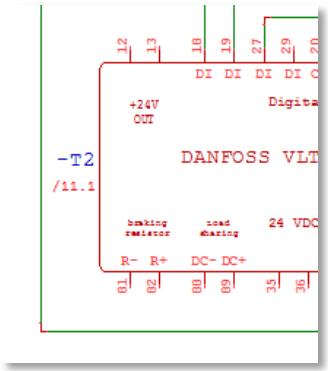
- Processing sheet 11
  1. Open sheet 11 in the project and place the macro „3 Phases+NPE“ from the macro folder "Potentials for Training".
  2. Draw the plan template (sheet 11/1, slide 97).

     Place the –T2 symbol as the parent element from the library "indoctrination Spezi", "Frequency Converter HE". In the Manager, press "New" and assign the part.

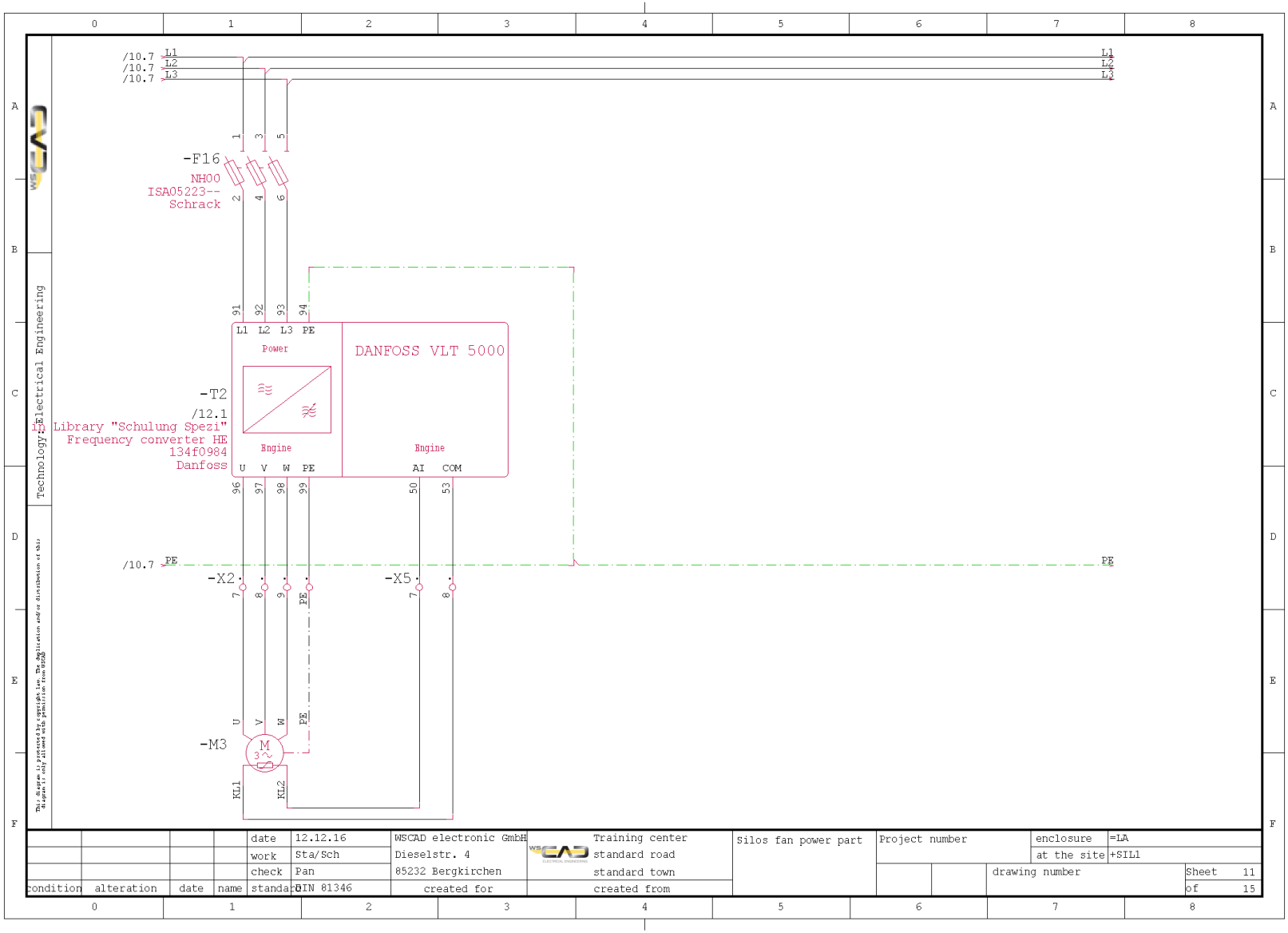
     Do not forget to expand the terminal strips –X2 and –X5 by 4 or 5 terminals.
- Processing sheet 12
  1. Open sheet 12 in the project and place the macro "24V oben_unten (24V top_bottom)" from the macro folder "Potentials Training".
  2. Draw the plan template (sheet 12/1, slide 98).

     Place the –T2 symbol as the child element from the library "indoctrination Spezi", "Frequency Converter HE". In the Manager, click on the parent element –T2 and "marry" the child element with the parent element. After confirming with "Ok", check the resulting cross-reference.

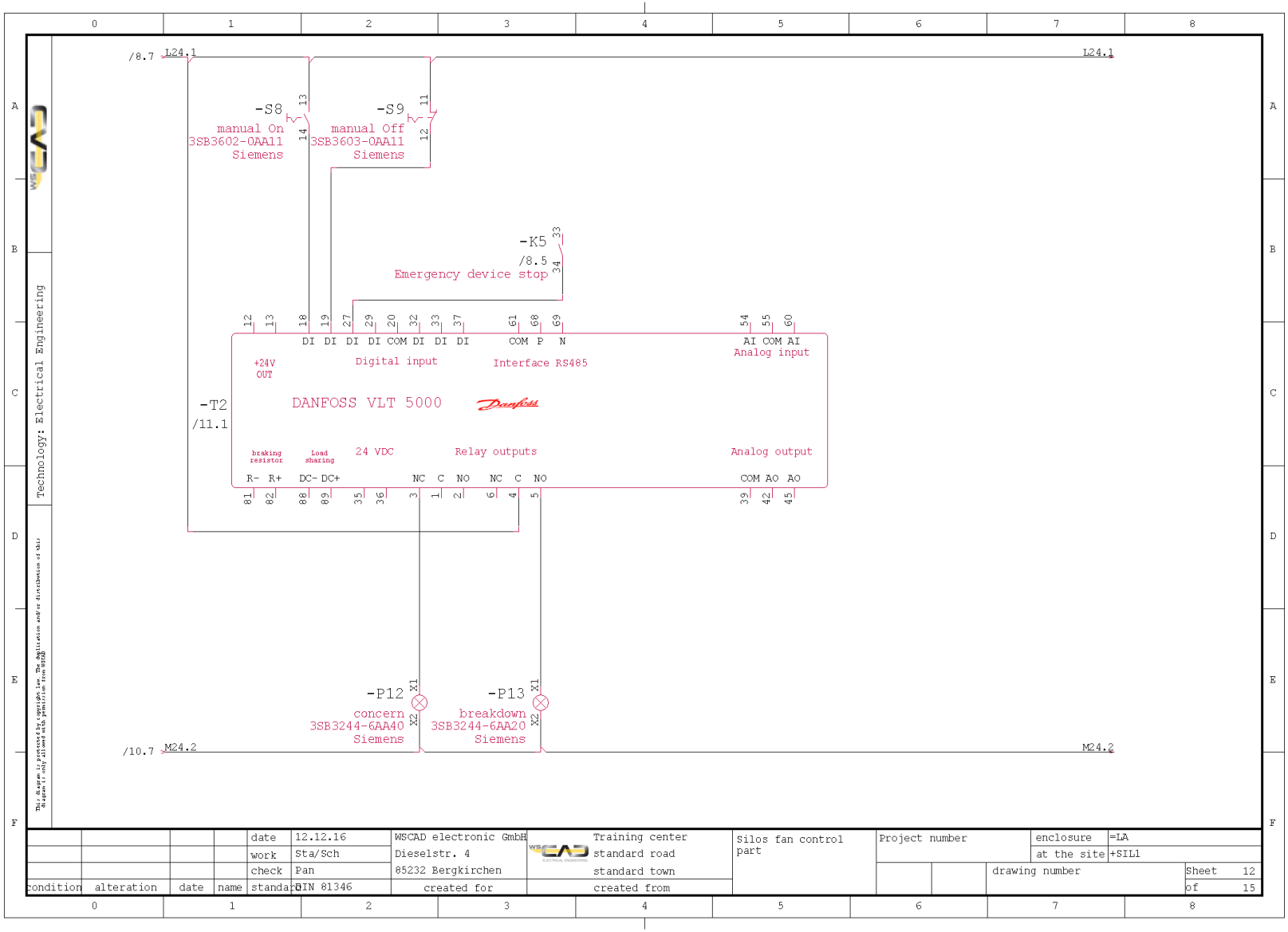




Basics – Parent/Child elements – Sheet 11/1



Basics – Parent/Child elements – Sheet 12/1



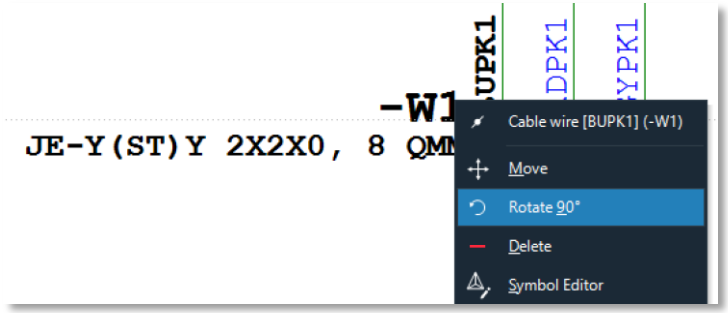
Exercise 12 – Cable Management

- Theoretical Section: **Cable management**
- Please devote your attention to the instructor. Thank you!

Basics – Cable Management

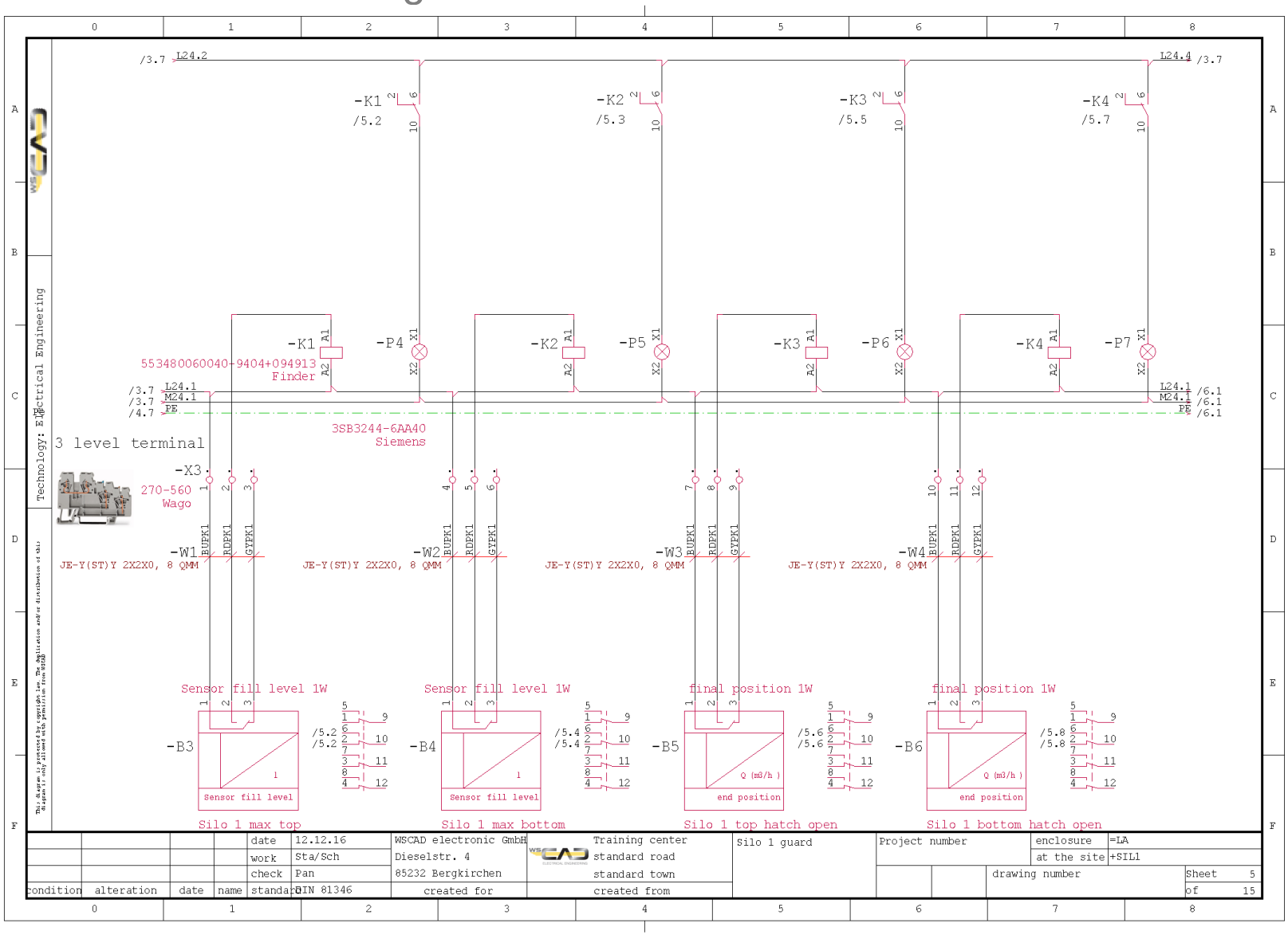
- Processing of sheets 5-11

1. Open sheet 5 in the project and place the cable as shown in sheet 5/4, slide 101.
   Rotate and move the cable data by 90° as shown.

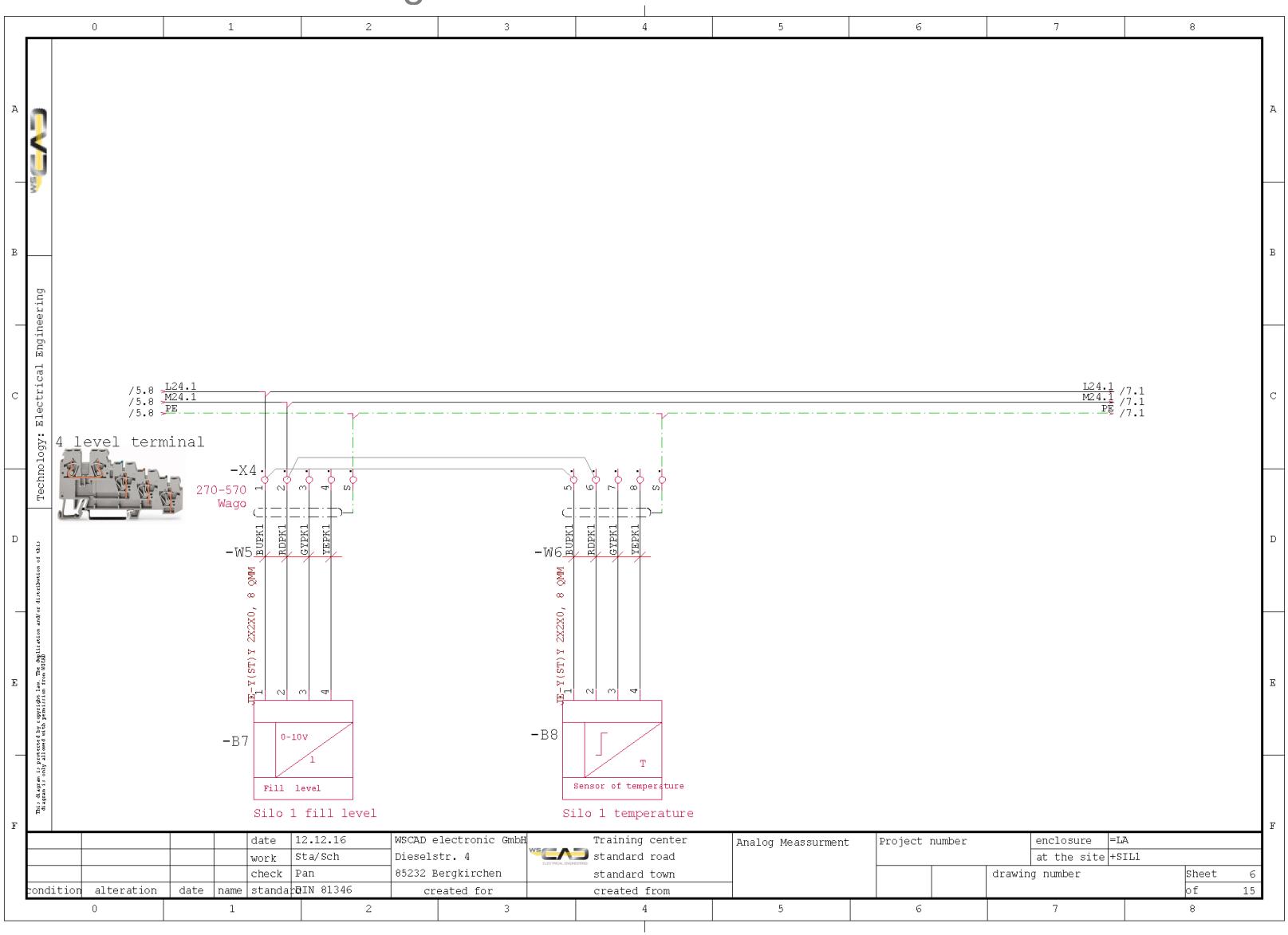


2. Open sheet 6 in the project and place the cable with the shielding as shown in sheet 6/2, slide 102.
3. Open sheet 7 in the project and place the cable as shown in sheet 7/3, slide 103.
4. Open sheet 8 in the project and place the cable as shown in sheet 8/2, slide 104.
5. Open sheet 9 in the project and place the cable as shown in sheet 9/2, slide 105.
   Note the wire assignment at the motor (wire color gnye) and possibly replace the wires.
6. Open sheet 10 in the project and place the cable as shown in sheet 10/2, slide 106.
   Note the wire assignment at the motor (wire color gnye) and possibly replace the wires.
7. Open sheet 11 in the project and place the cable with the shieldings as shown in sheet 11/2, slide 107.

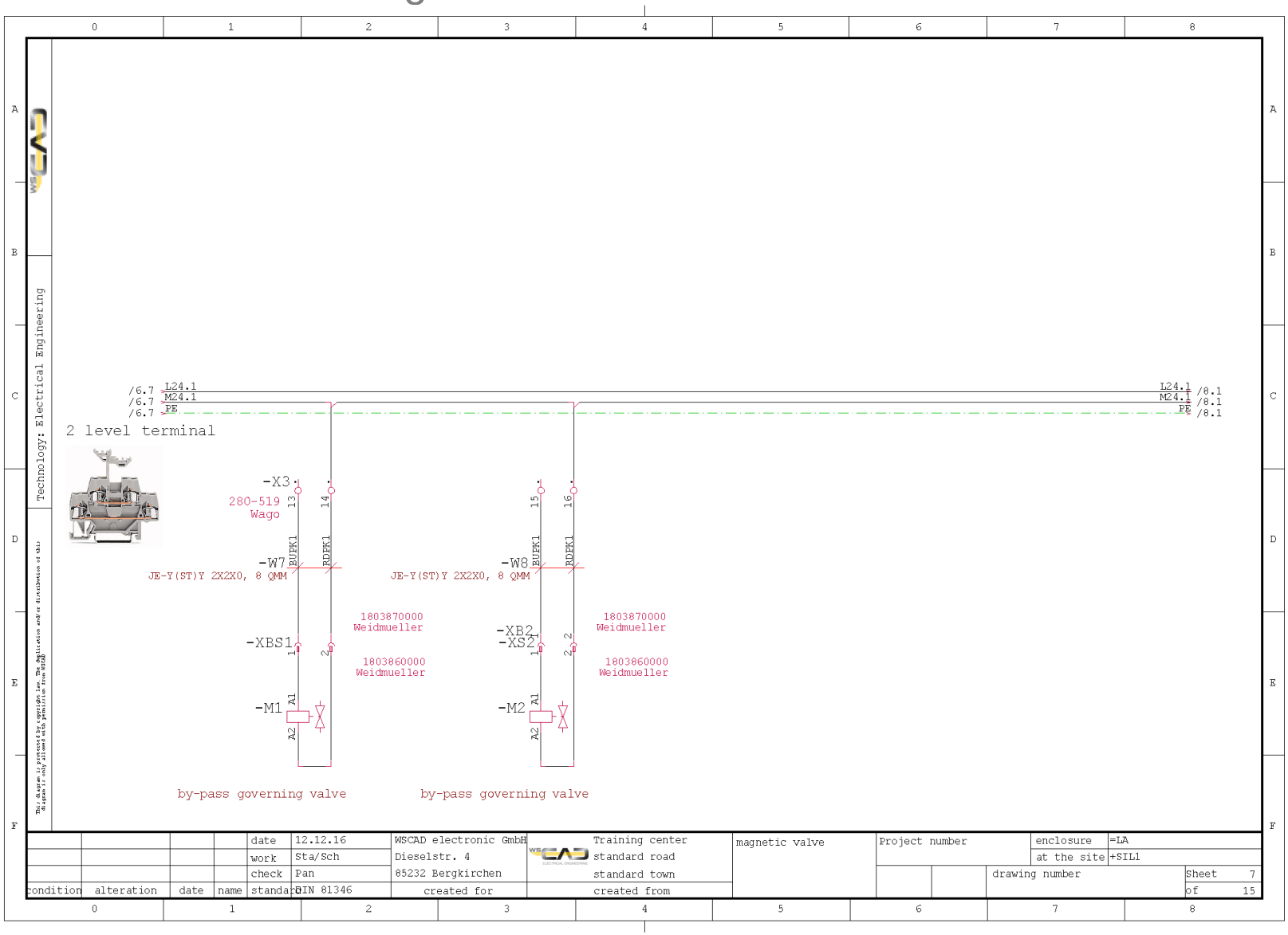
Basics – Cable Management – Sheet 5/4



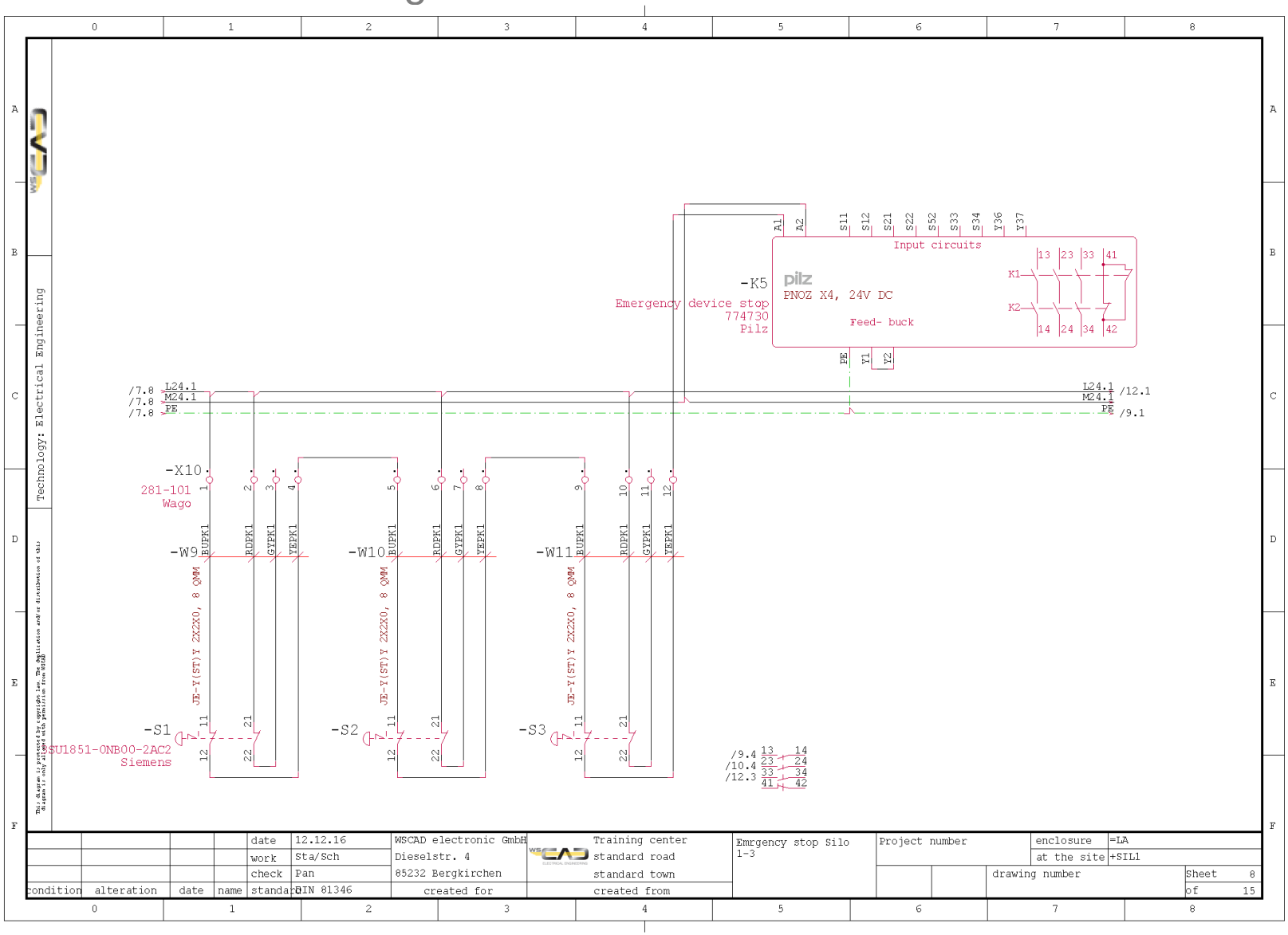
Basics – Cable Management – Sheet 6/2



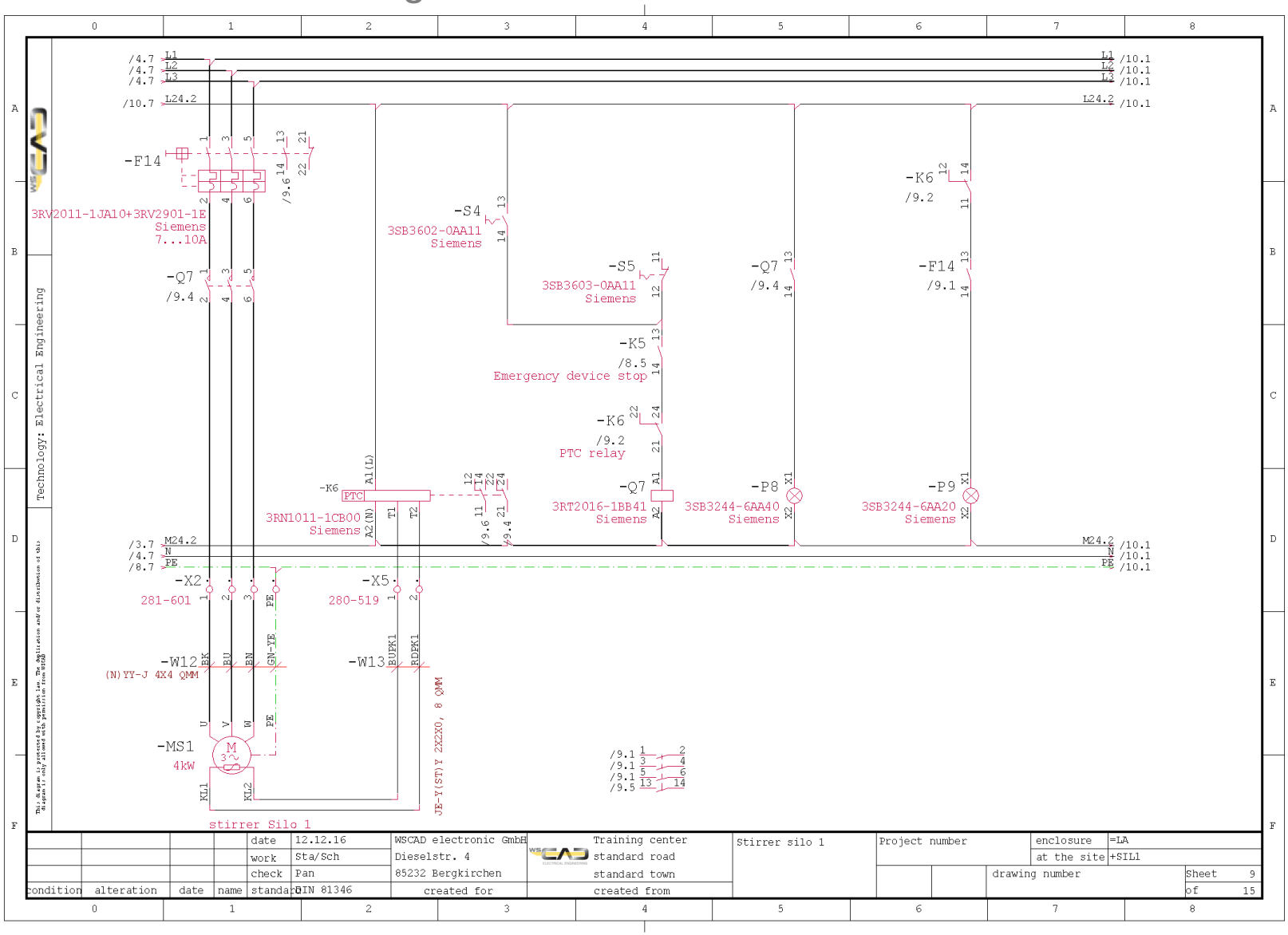
Basics – Cable Management – Sheet 7/3



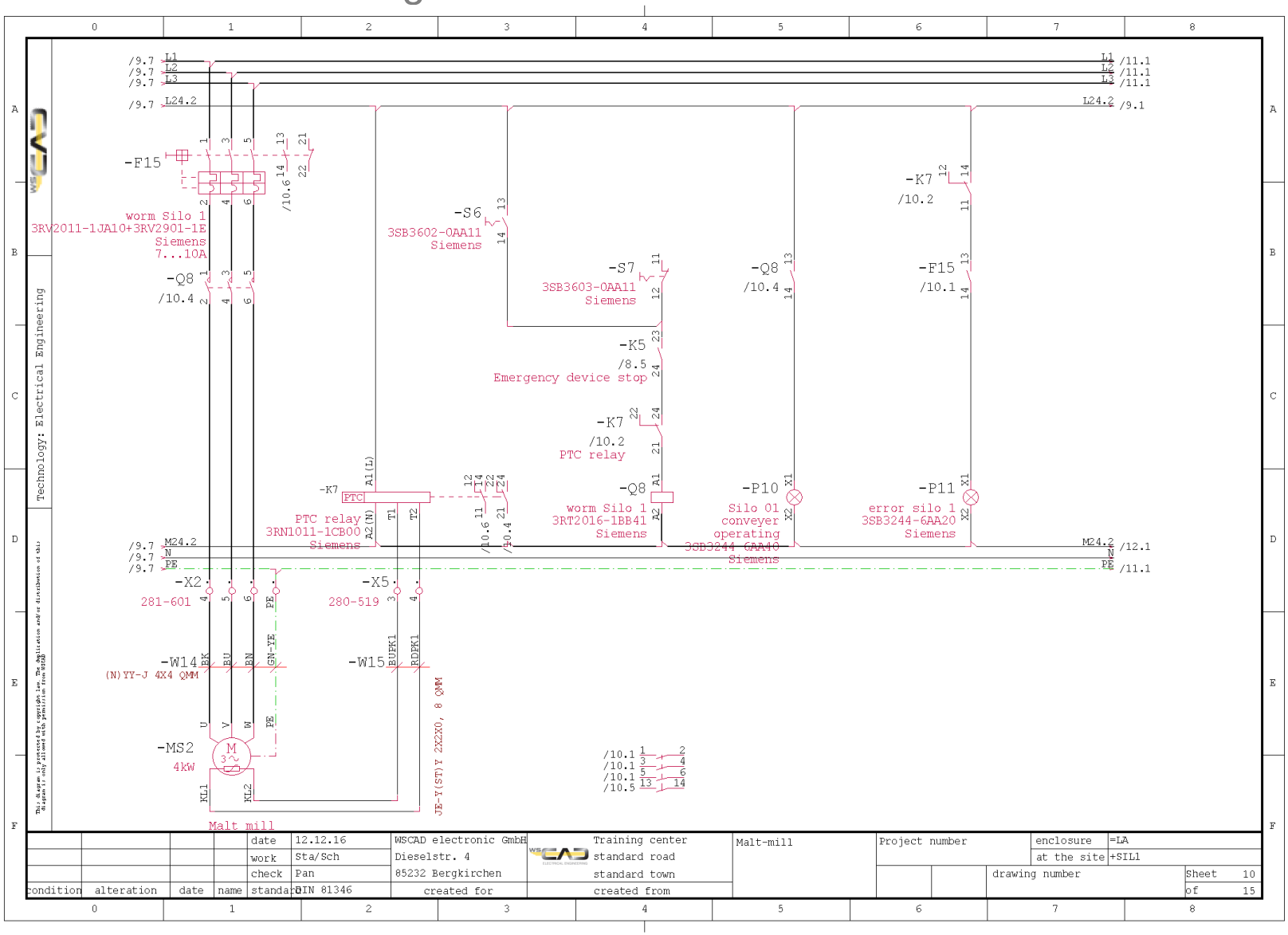
Basics – Cable Management – Sheet 8/2



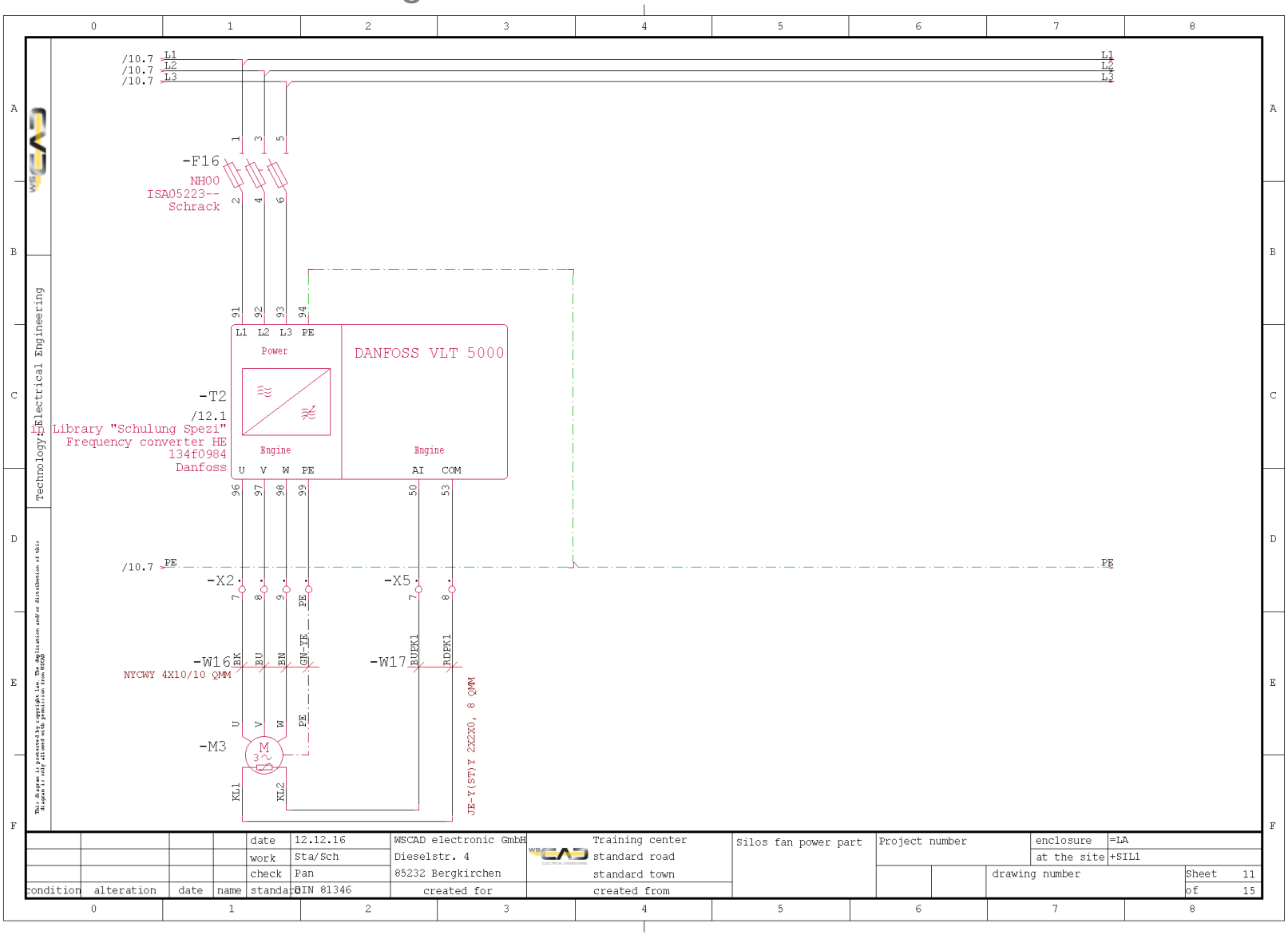
Basics – Cable Management – Sheet 9/2



Basics – Cable Management – Sheet 10/2



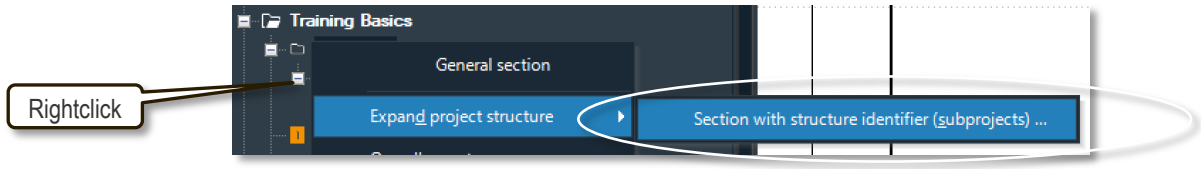
Basics – Cable Management – Sheet 11/2

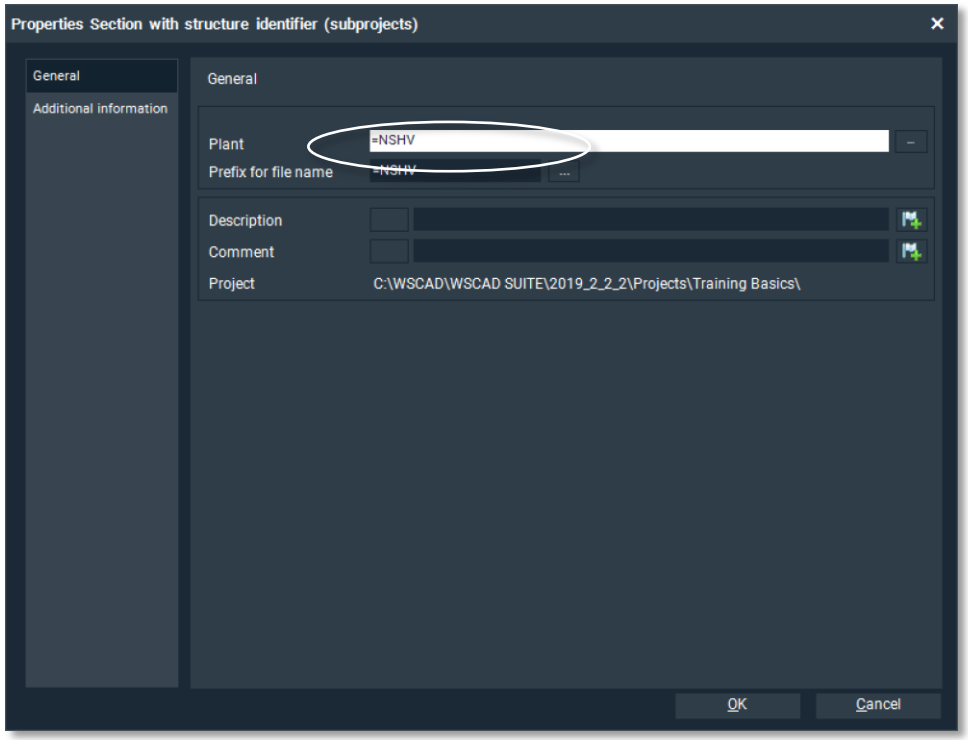


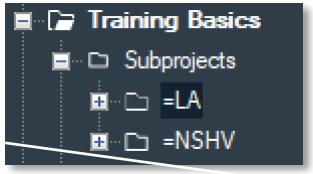
				date	12.12.16	WSCAD electronic GmbH	Training center	Silos fan power part	Project number	enclosure	=LA
				work	Sta/Sch	Dieselstr. 4	standard road			at the site	+SIL1
				check	Pan	85232 Bergkirchen	standard town				
condition	alteration	date	name	standard	DIN 81346	created for	created from		drawing number		Sheet 11
											of 15

Basics – Cable Management

- Cable duplicate (optional)
  1. Expand the project structure in the "Training Basics" project with the plant =NSHV by right-clicking on "Subprojects".

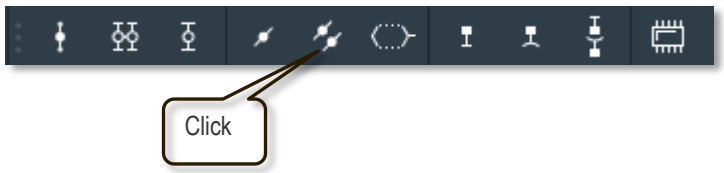


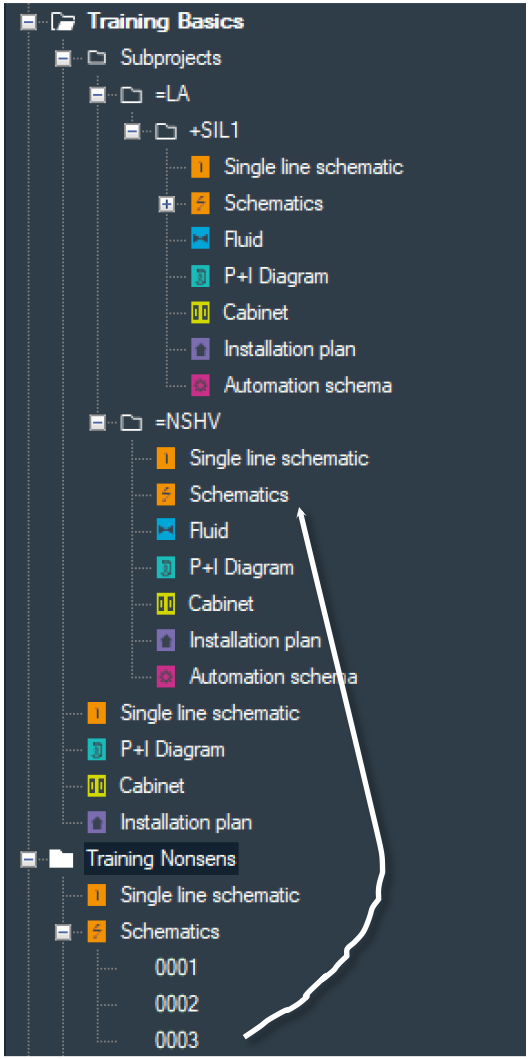




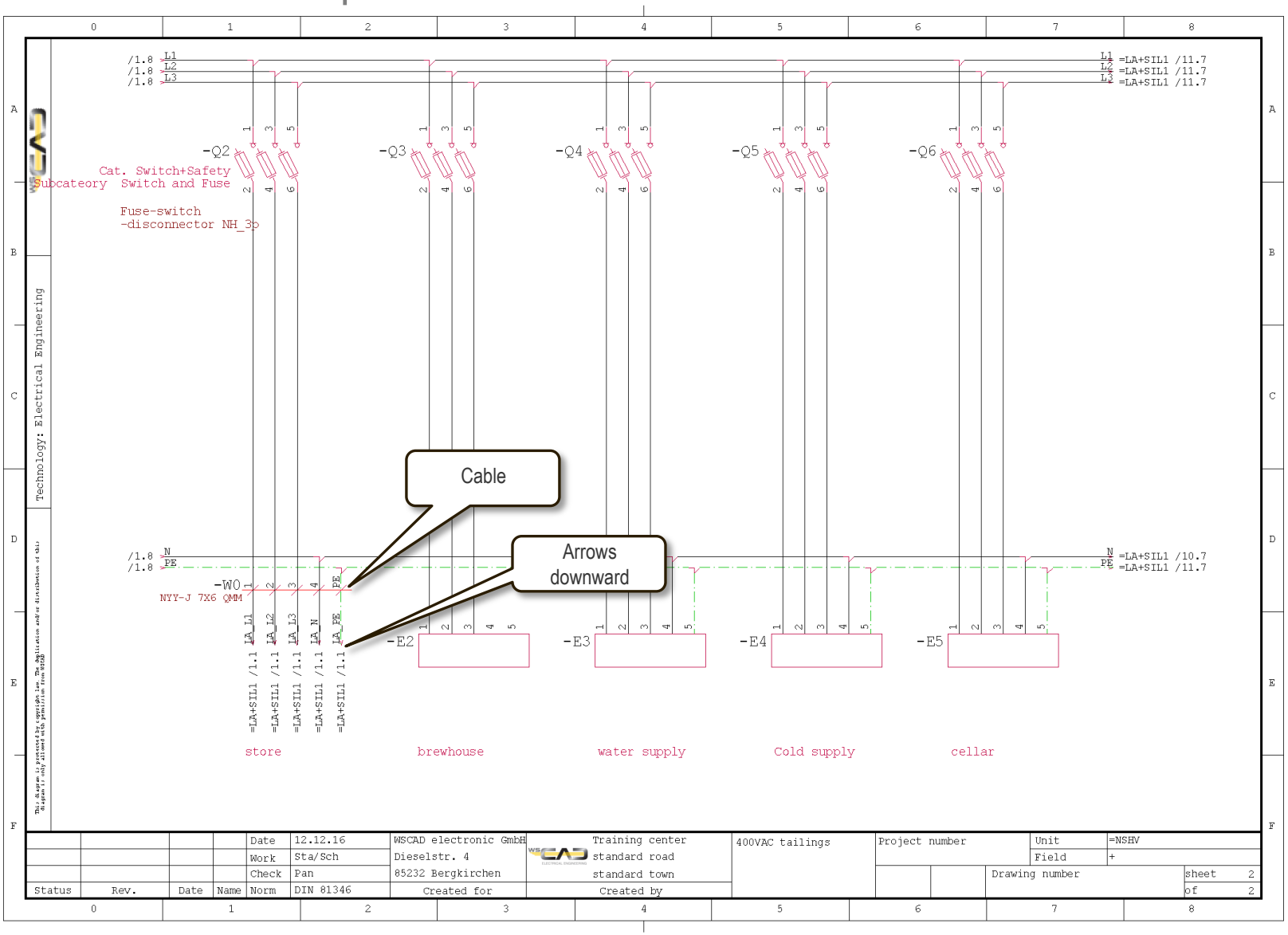
Basics – Cable Management

- Cable duplicate (optional)
  2. Copy sheets 1 and 2 from the project "Nonsense" in the Project Explorer.
  3. Add this to the active "Training Basics" project in the "Schematic" document folder.
  4. Open sheet 2 in =NSHV.
  5. Add the termination point symbols here (as feeder connection =NSHV to =LA) and place the cable –W0 as shown in slide 110.
  6. Open sheet 1 of =LA+SIL1 and place the cable duplicate of –W0 as shown in sheet =LA+SIL1, sheet 1, slide 111 via the command:

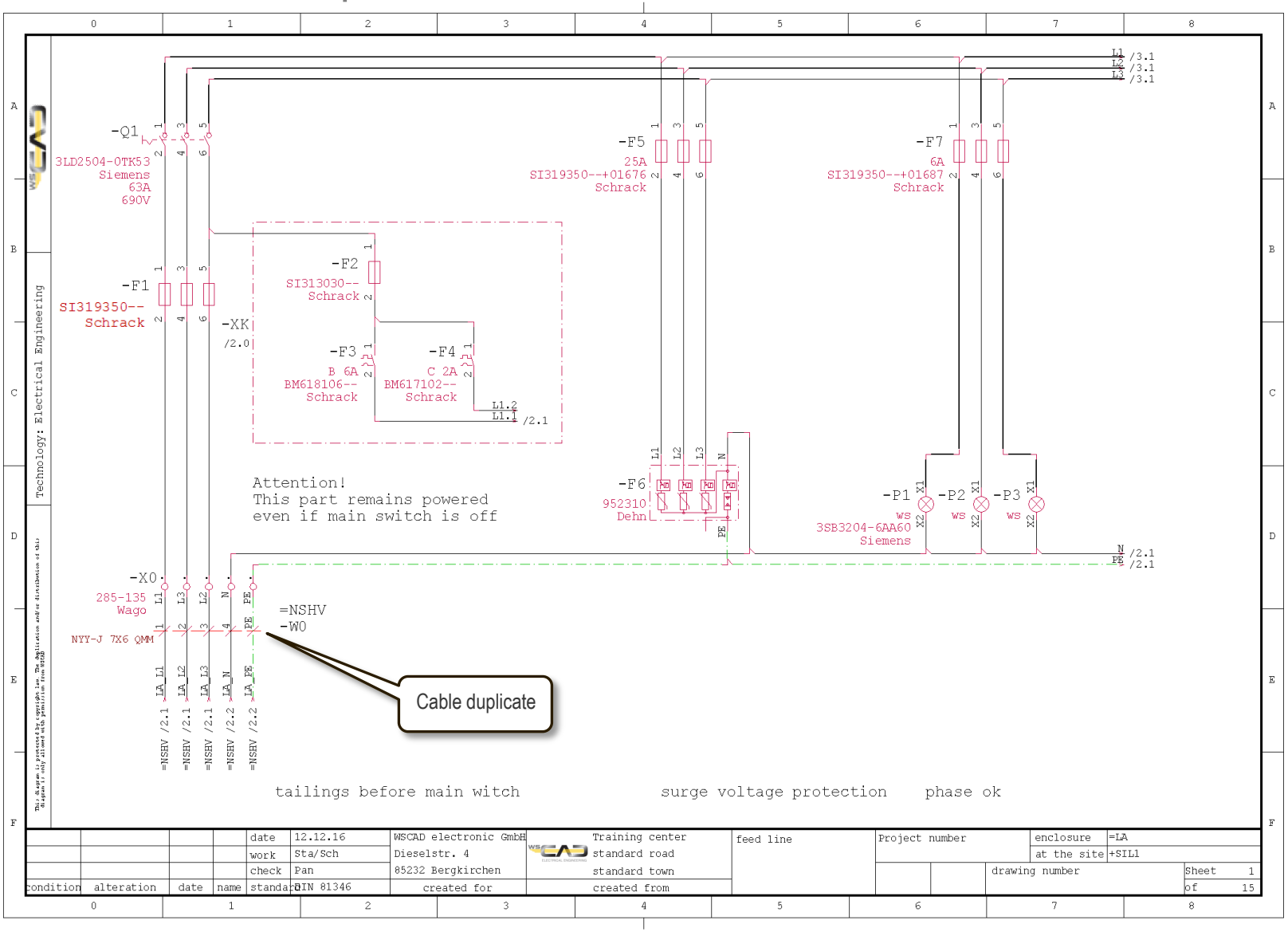




Basics – Cable duplicate – =NSHV Sheet 2

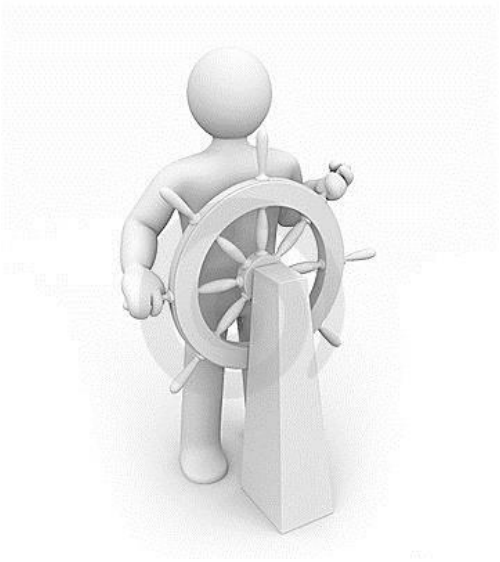


Basics – Cable duplicate – =LA +SIL1 Sheet 1



Exercise 13 – PLC Management

- Theoretical Section: **PLC management**
- Please devote your attention to the instructor. Thank you!

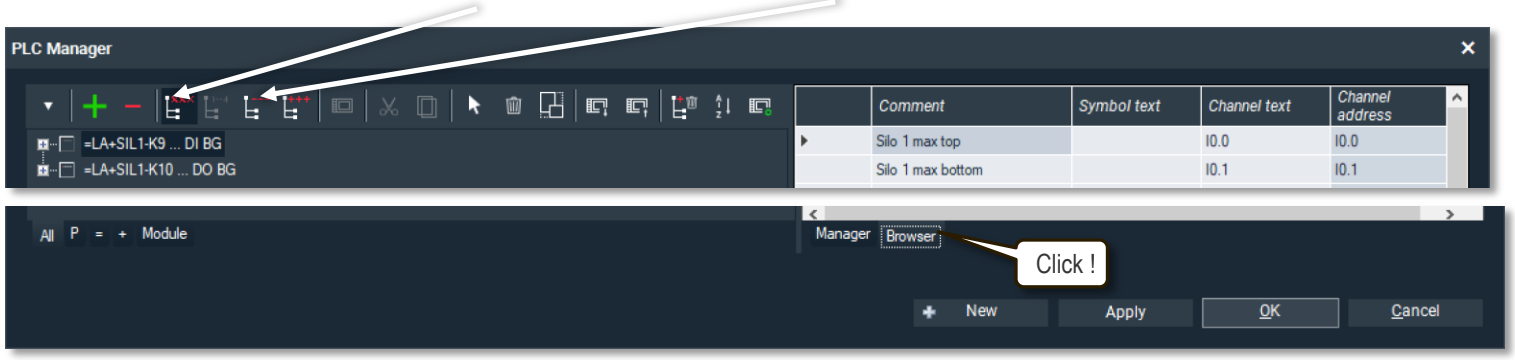


Basics – PLC management

- PLC parent elements

1. Open sheet 13 "PLC CPU / Digital input and output module" and place the macro "24V oben (24V top)".
2. Place the PLC modules as shown in sheet 13, slide 114.

   Note that the DI and DO PLC modules call the PLC Manager when placed. But not the CPU module, since this is a standard symbol.
3. Show texts here for functions texts and comment texts:

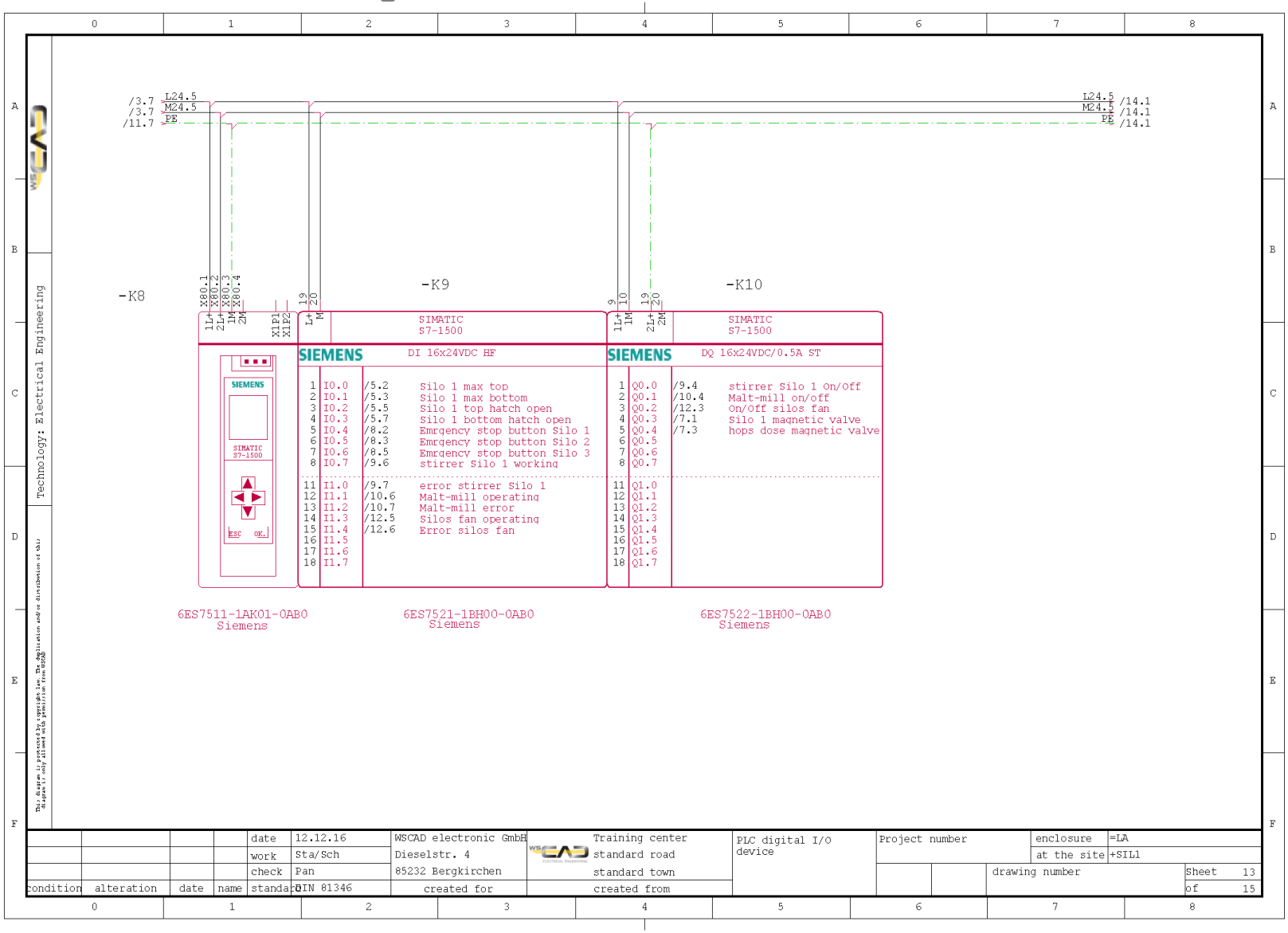


4. Specify the defaults for the functions texts and comment texts in the PLC Browser.

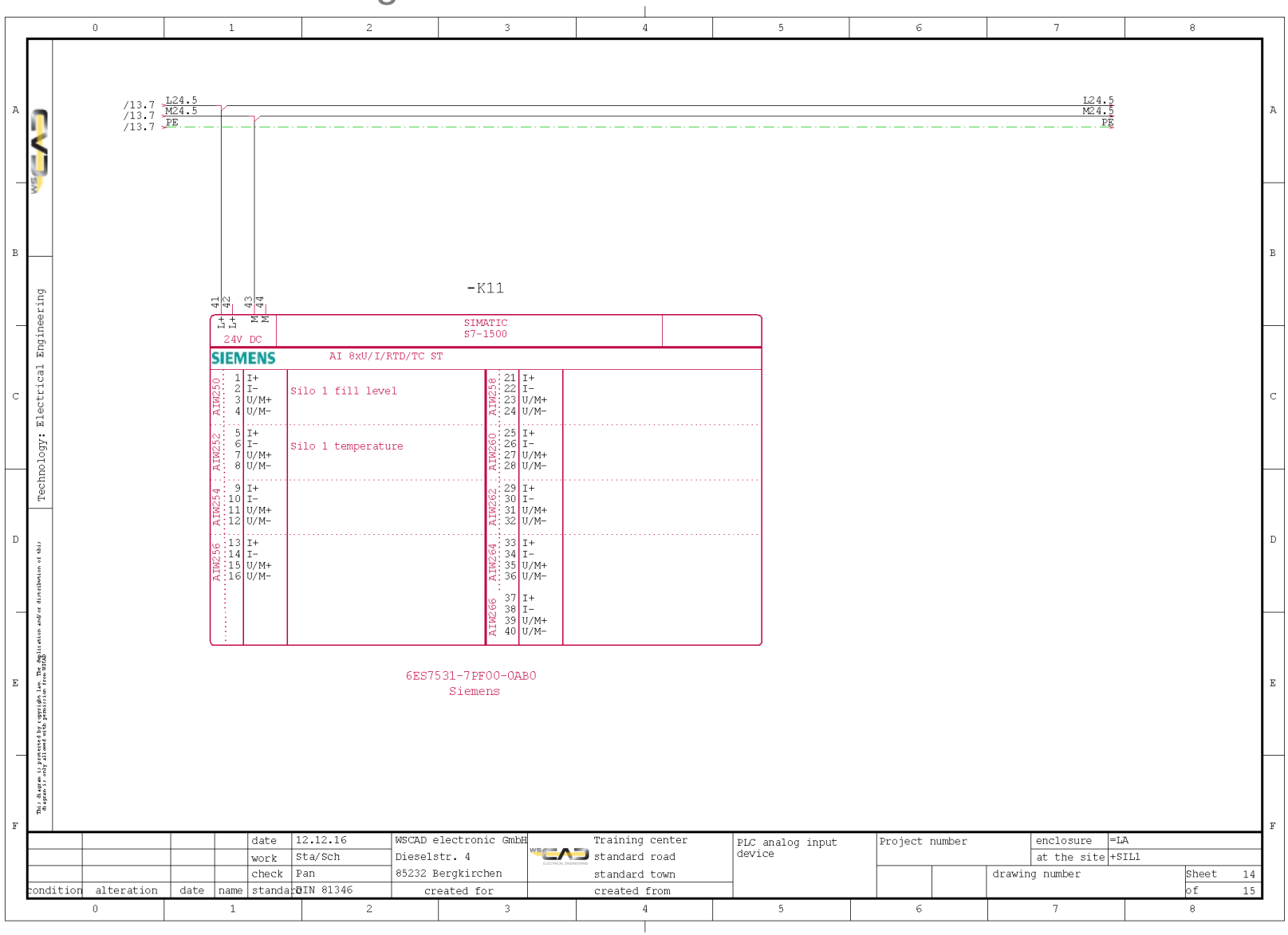
5. Repeat the above steps for sheet 14, "PLC analog input module" and sheet 115, "PLC analog output module" as shown in the sheets below in slides 116.

   The connection point patterns of the modules only serve as examples.

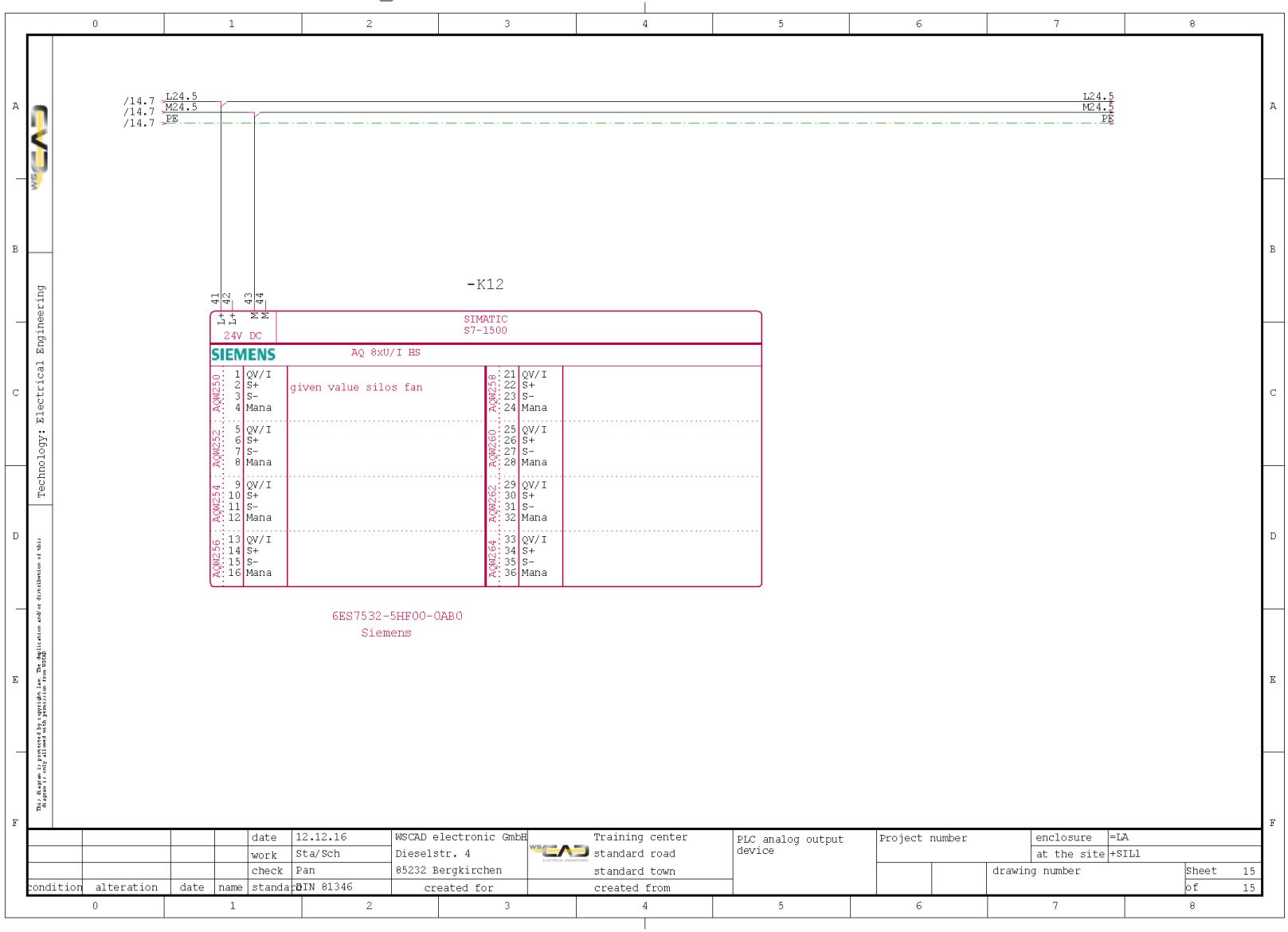
Basics – PLC management – Sheet 13



Basics – PLC management – Sheet 14



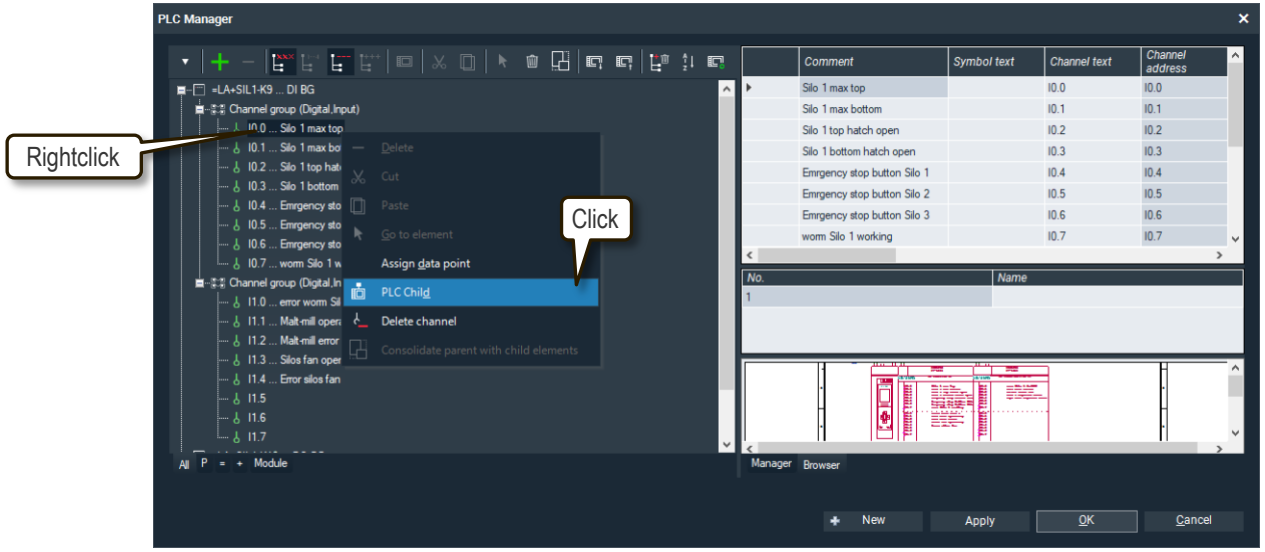
Basics – PLC management – Sheet 15

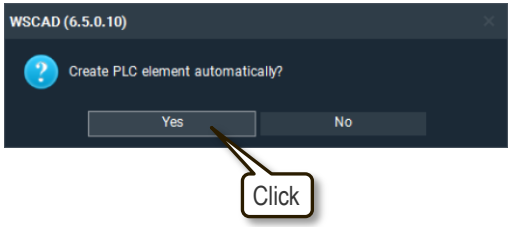


Basics – PLC management

- PLC child elements

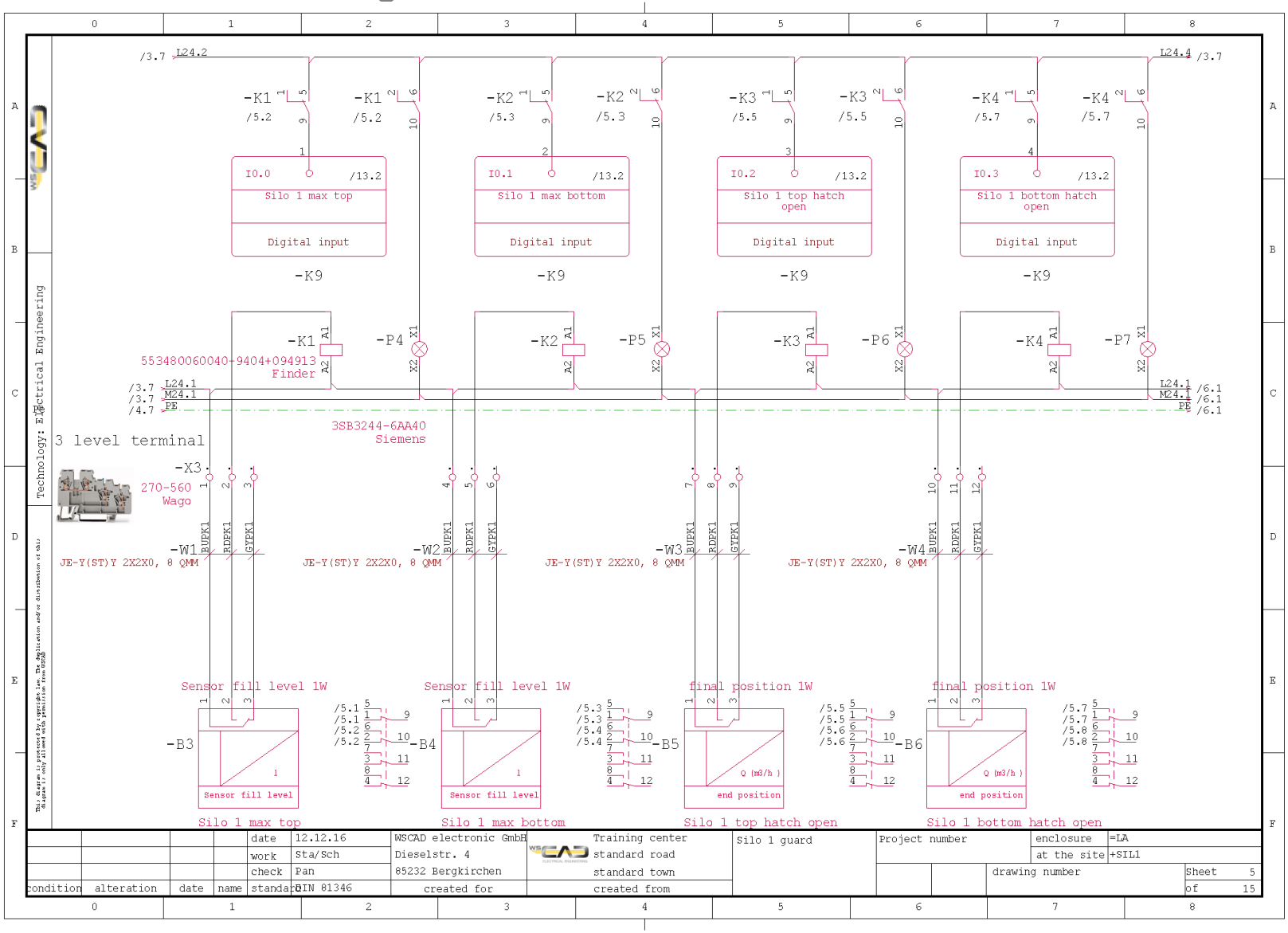
1. In the following, you will place the PLC child elements on the sheets specified below by using the "automatic placement" in the PLC Manager.



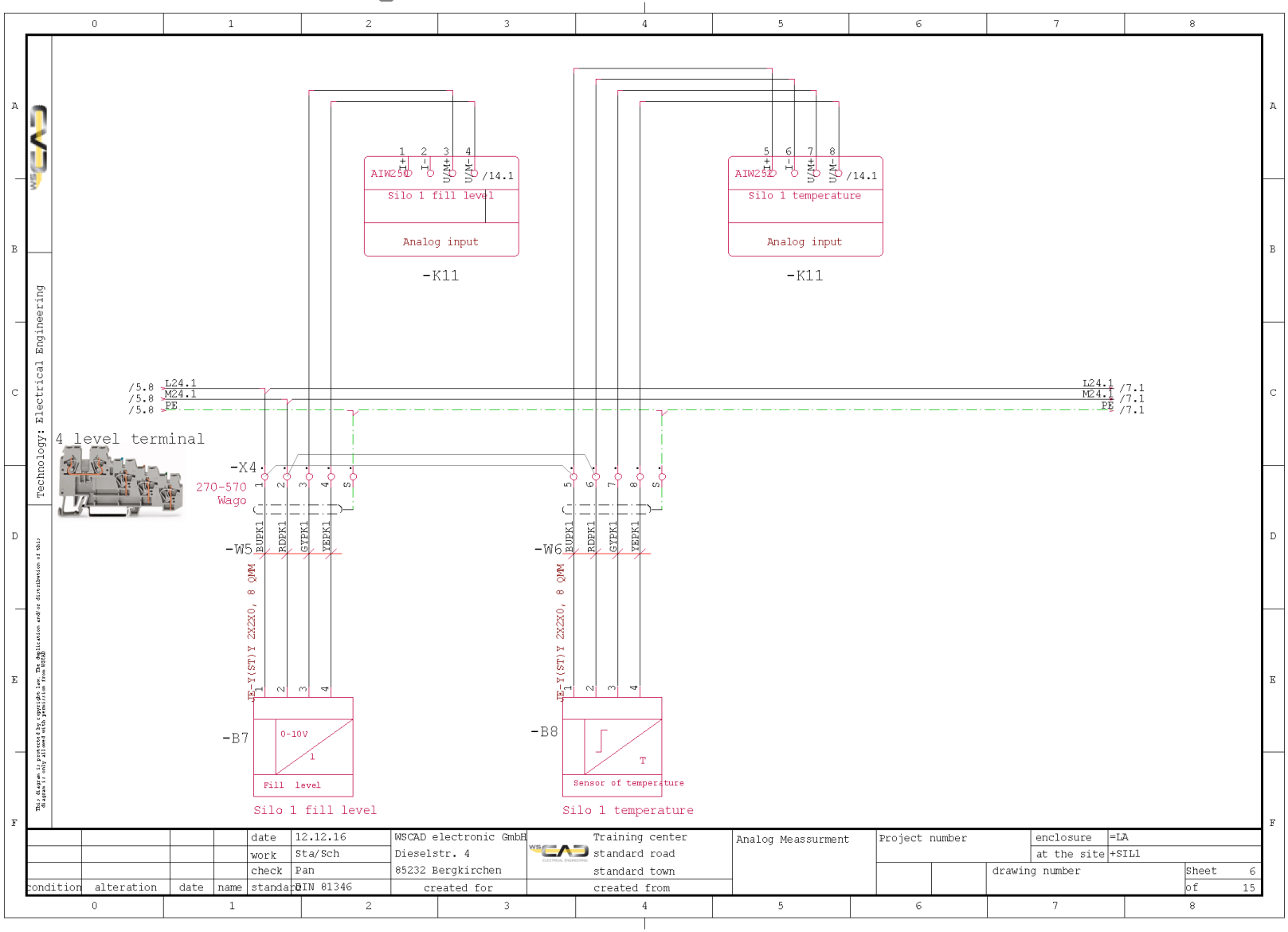


2. Open sheet 5, place the child elements and add the contacts as shown in sheet 5/5, slide 118.
3. Open sheets 6 – 12 in the project one at a time and place the PLC child elements as shown in sheet 6/3 (slide 119), sheet 7/4 (slide 120), sheet 8/3 (slide 121), sheet 9/3 (slide 122), sheet 10/3 (slide 123) and sheet 2/2 (slide 124).
4. Then check in the representations of the PLC modules on slides 125 – 127 whether the cross-references of the child elements are displayed.

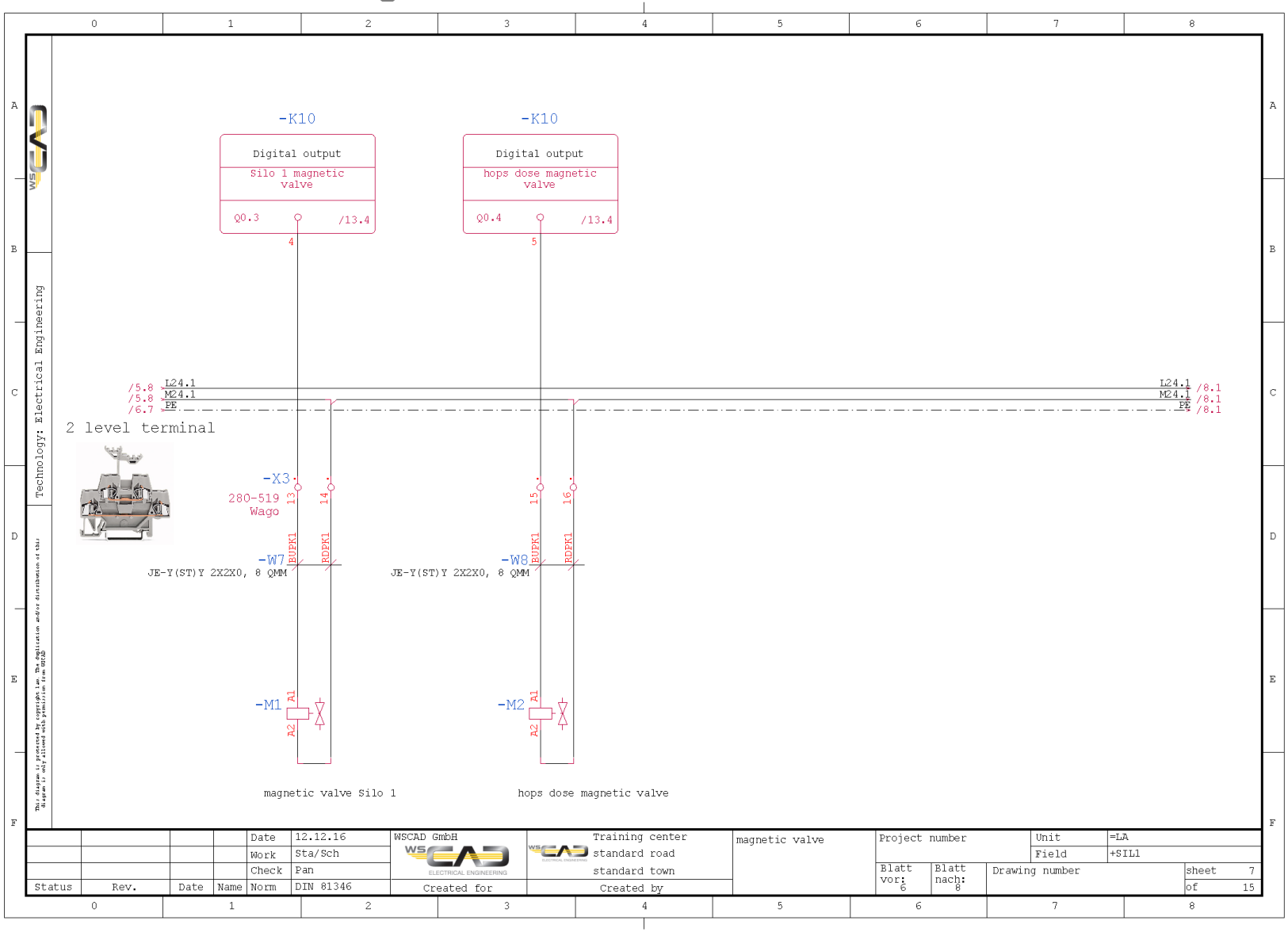
Basics – PLC management – Sheet 5/5



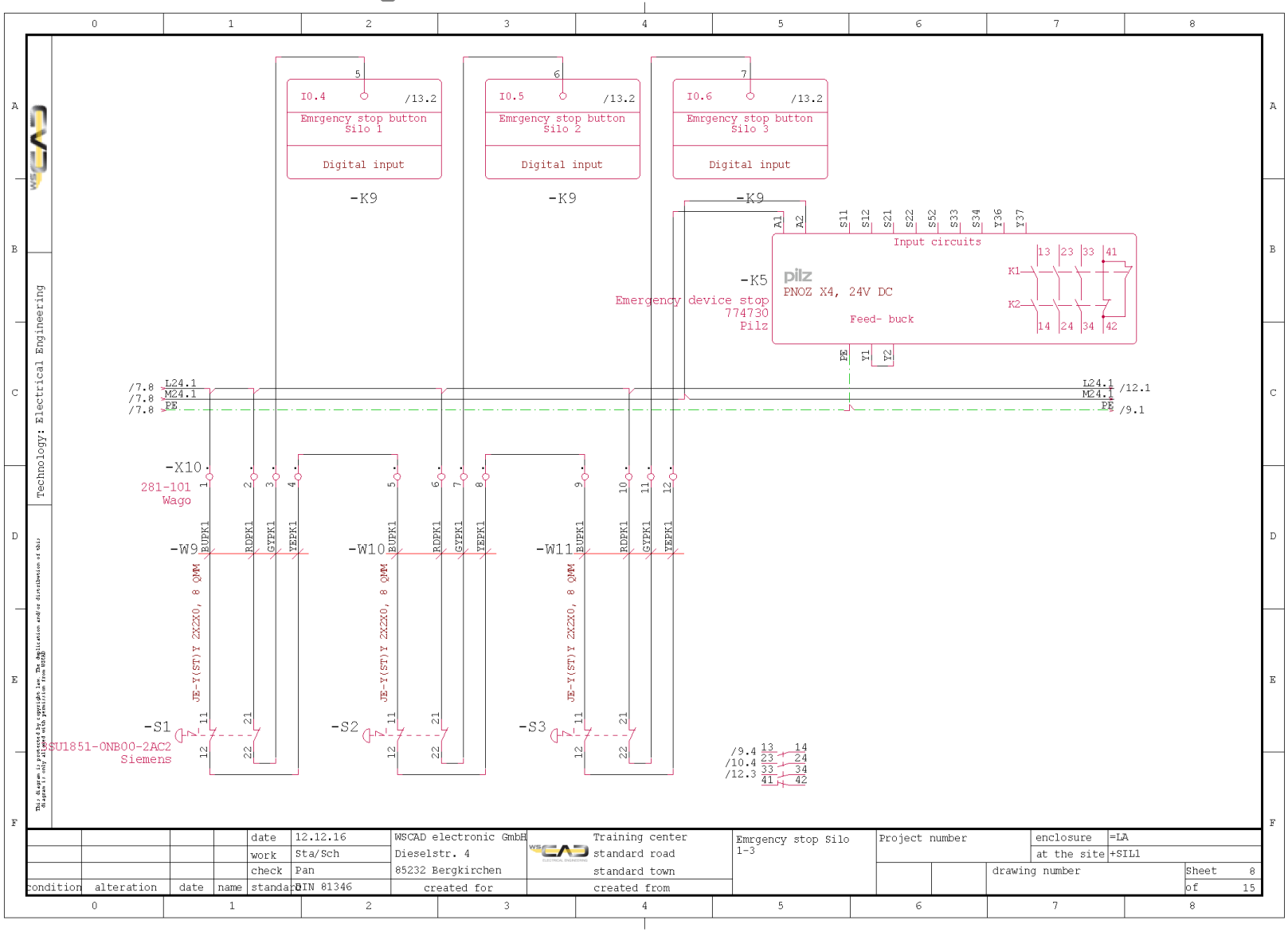
Basics – PLC management – Sheet 6/3



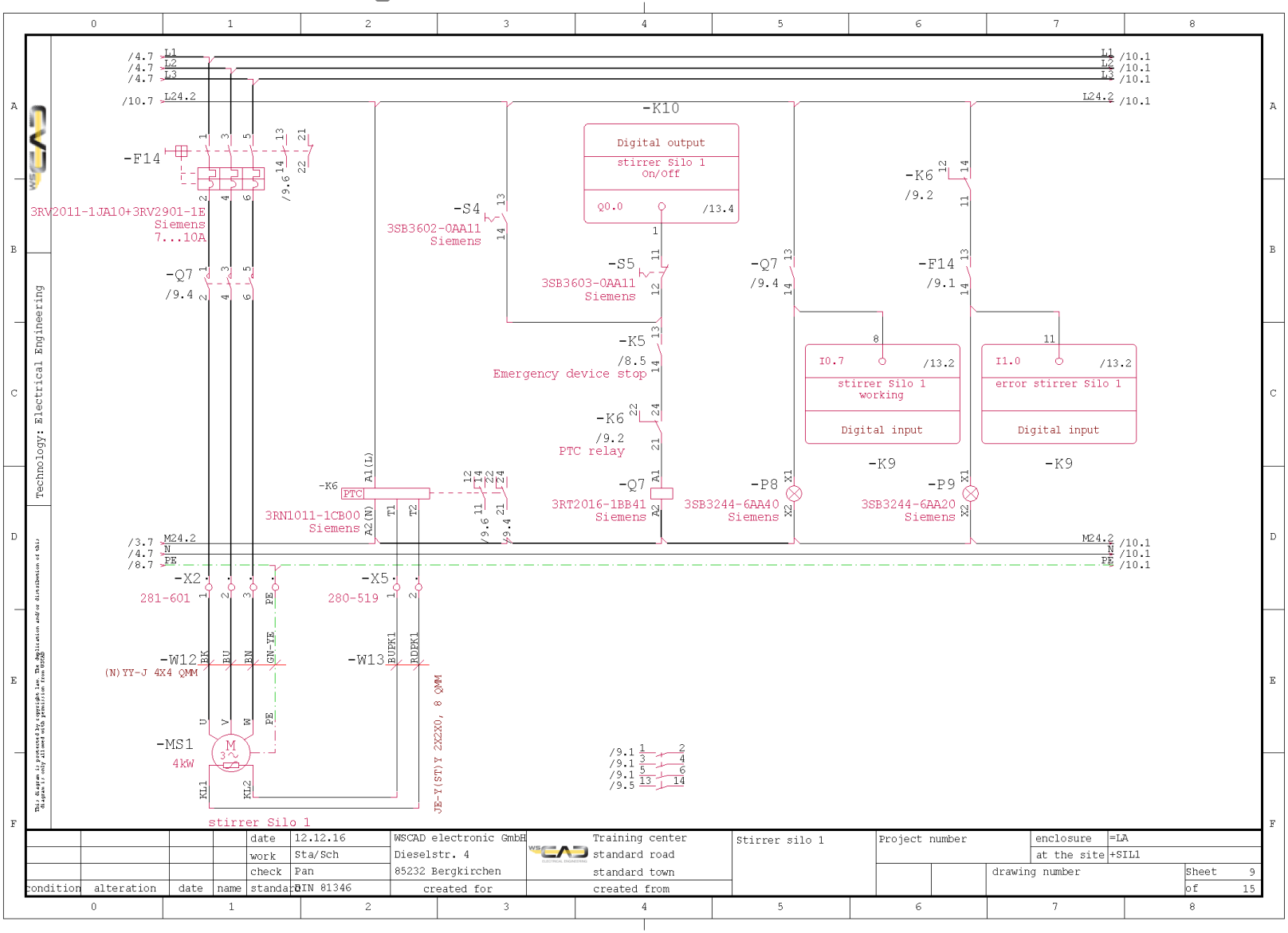
Basics – PLC management – Sheet 7/4



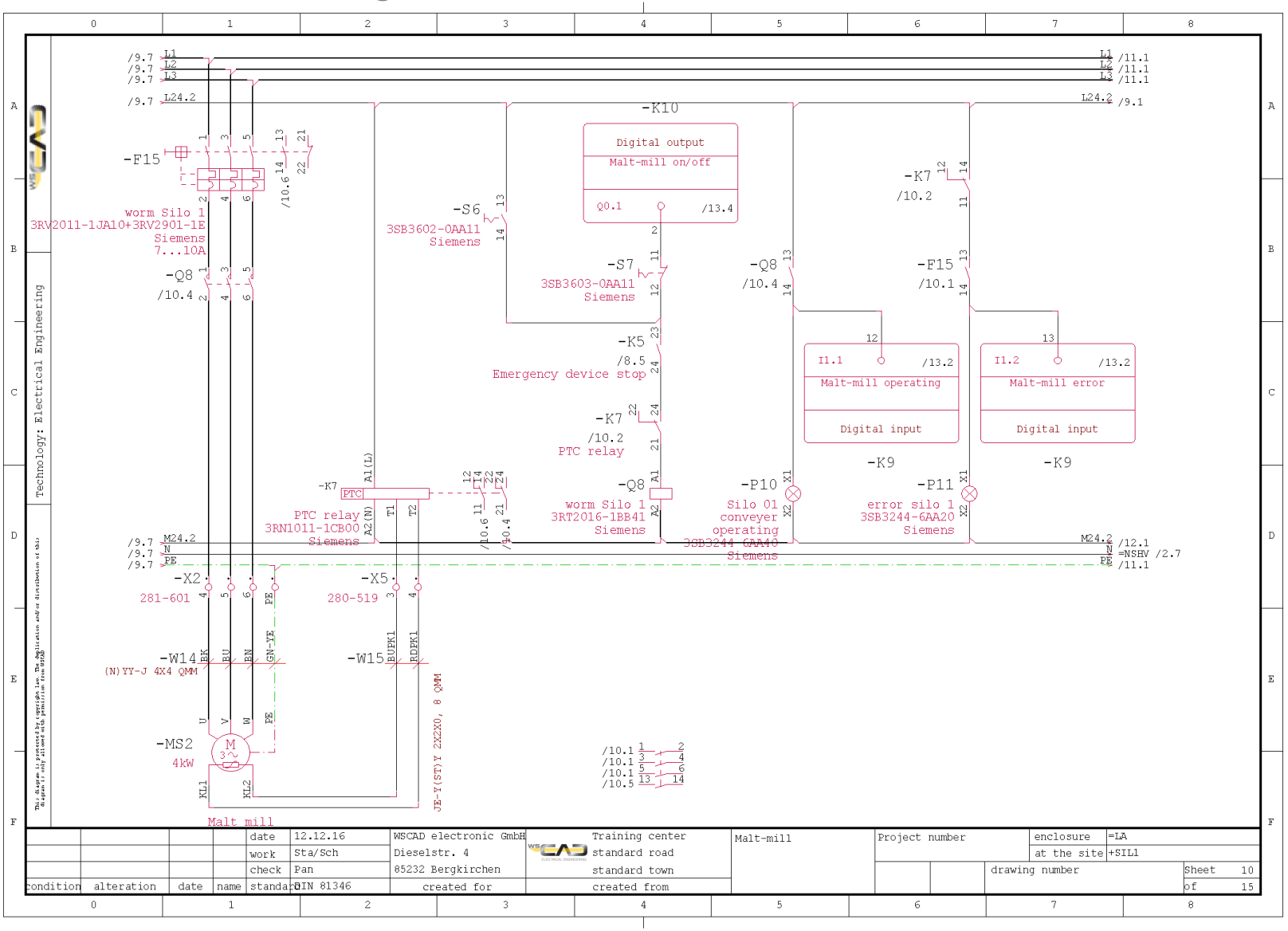
Basics – PLC management – Sheet 8/3



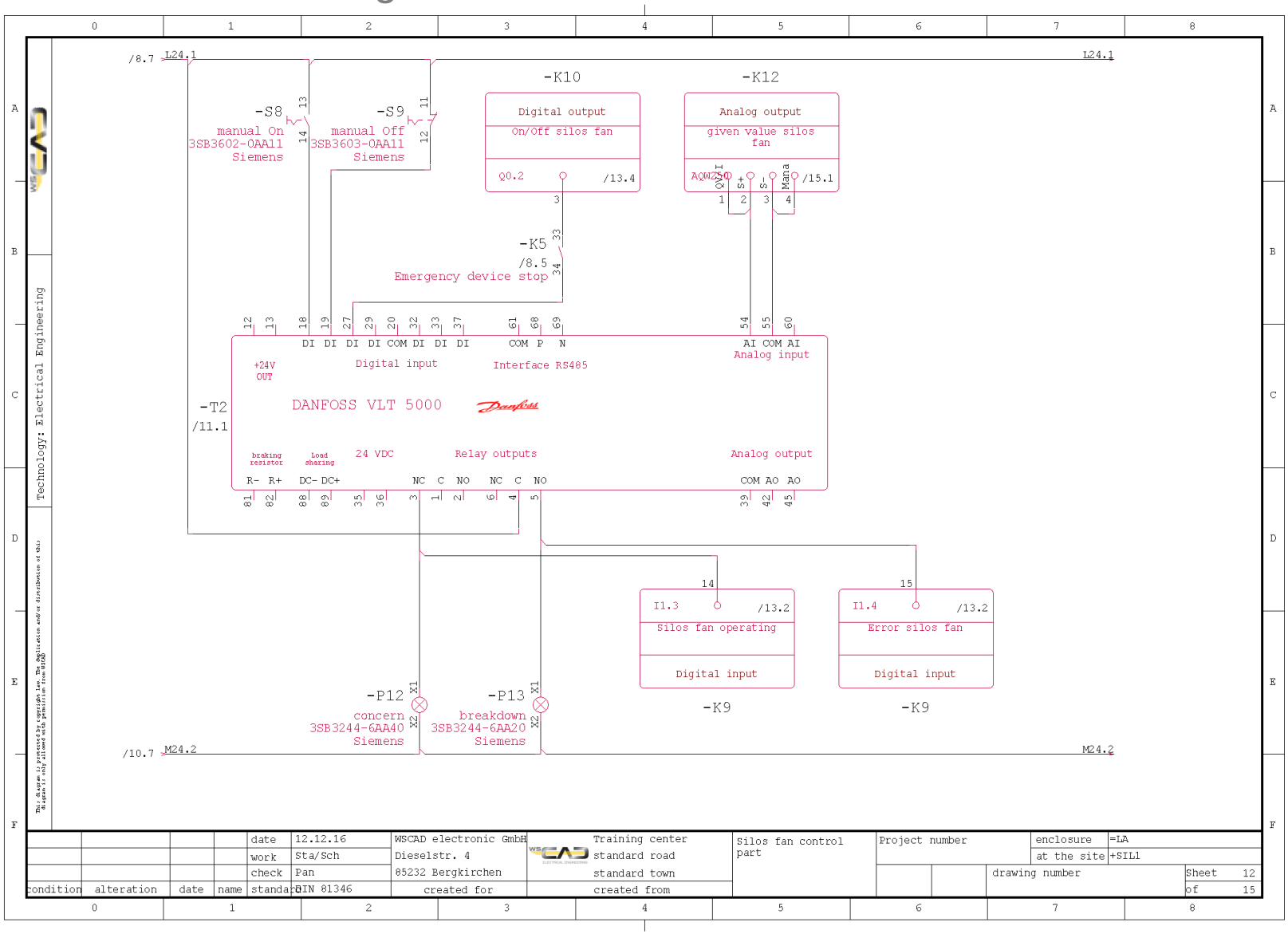
Basics – PLC management – Sheet 9/3



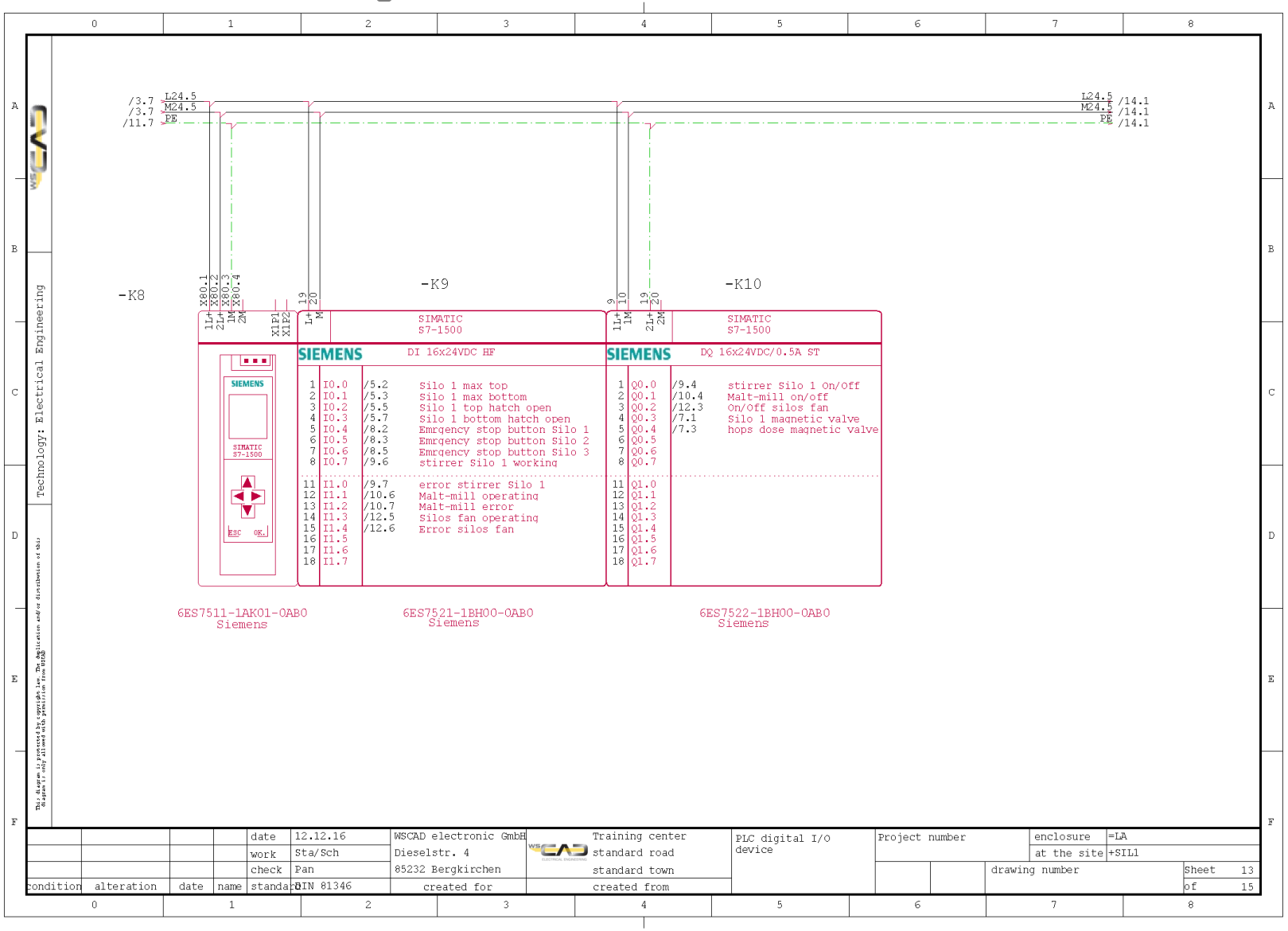
Basics – PLC management – Sheet 10/3



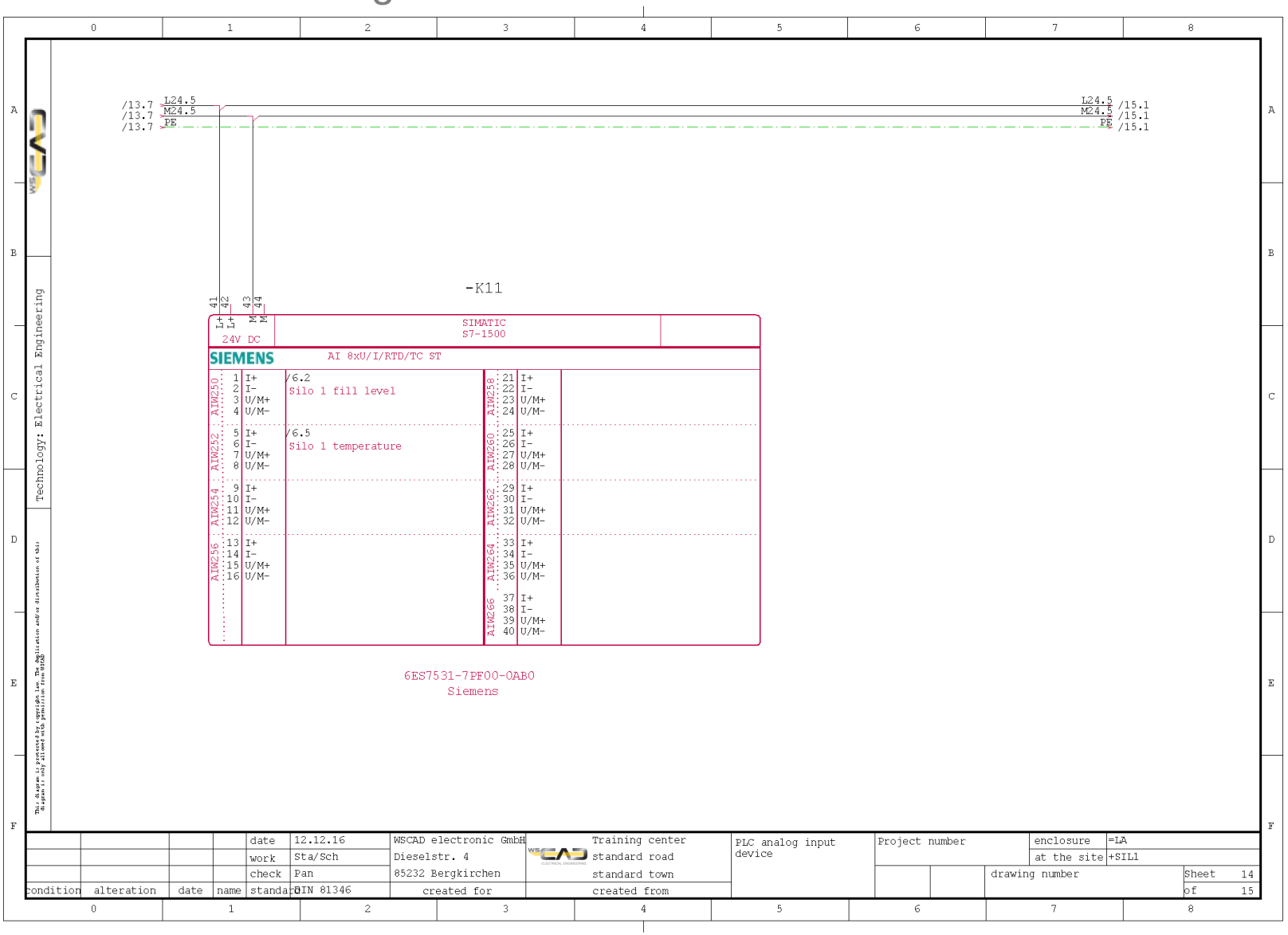
Basics – PLC management – Sheet 12/2



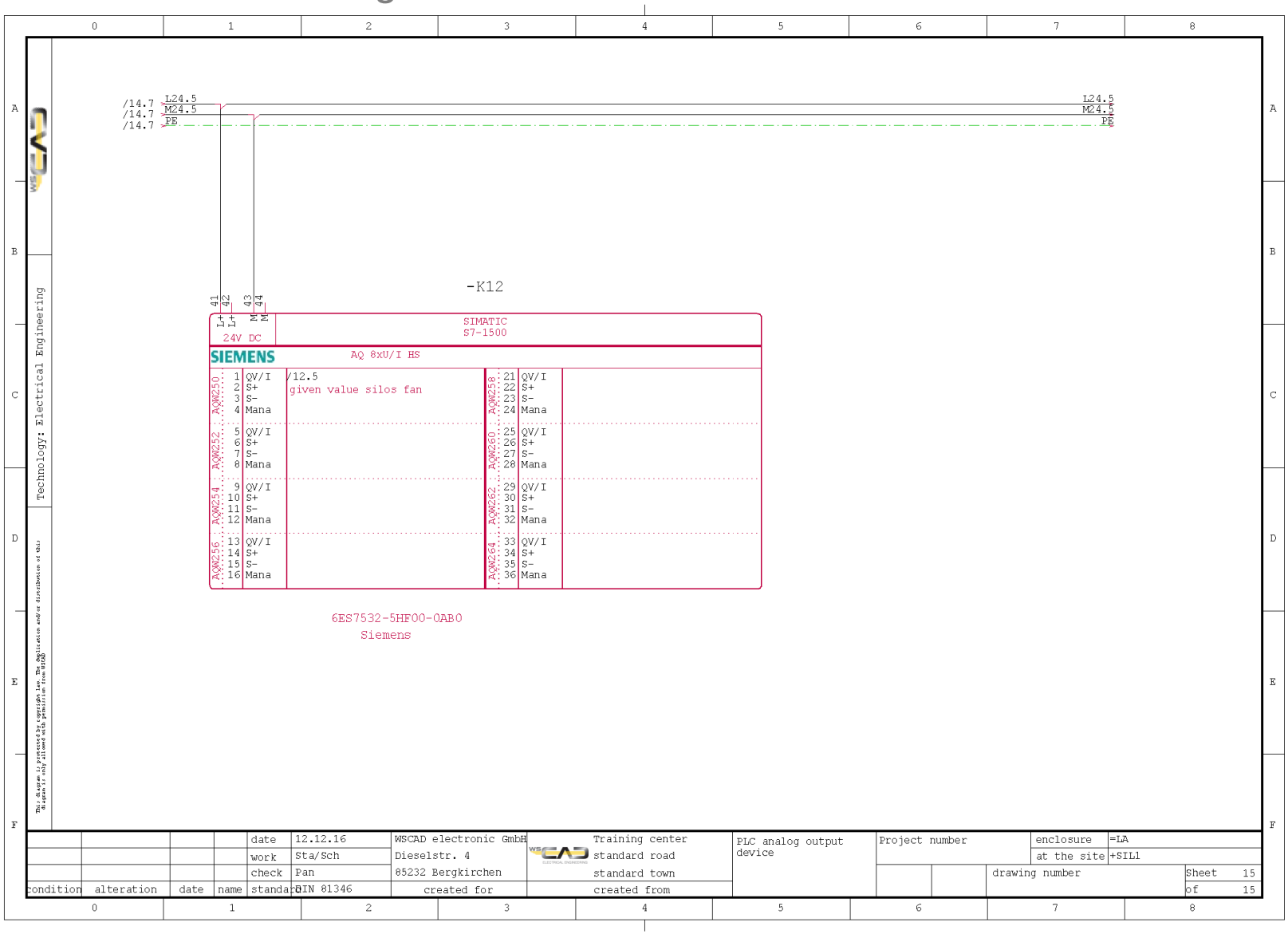
Basics – PLC management – Sheet 13



Basics – PLC management – Sheet 14

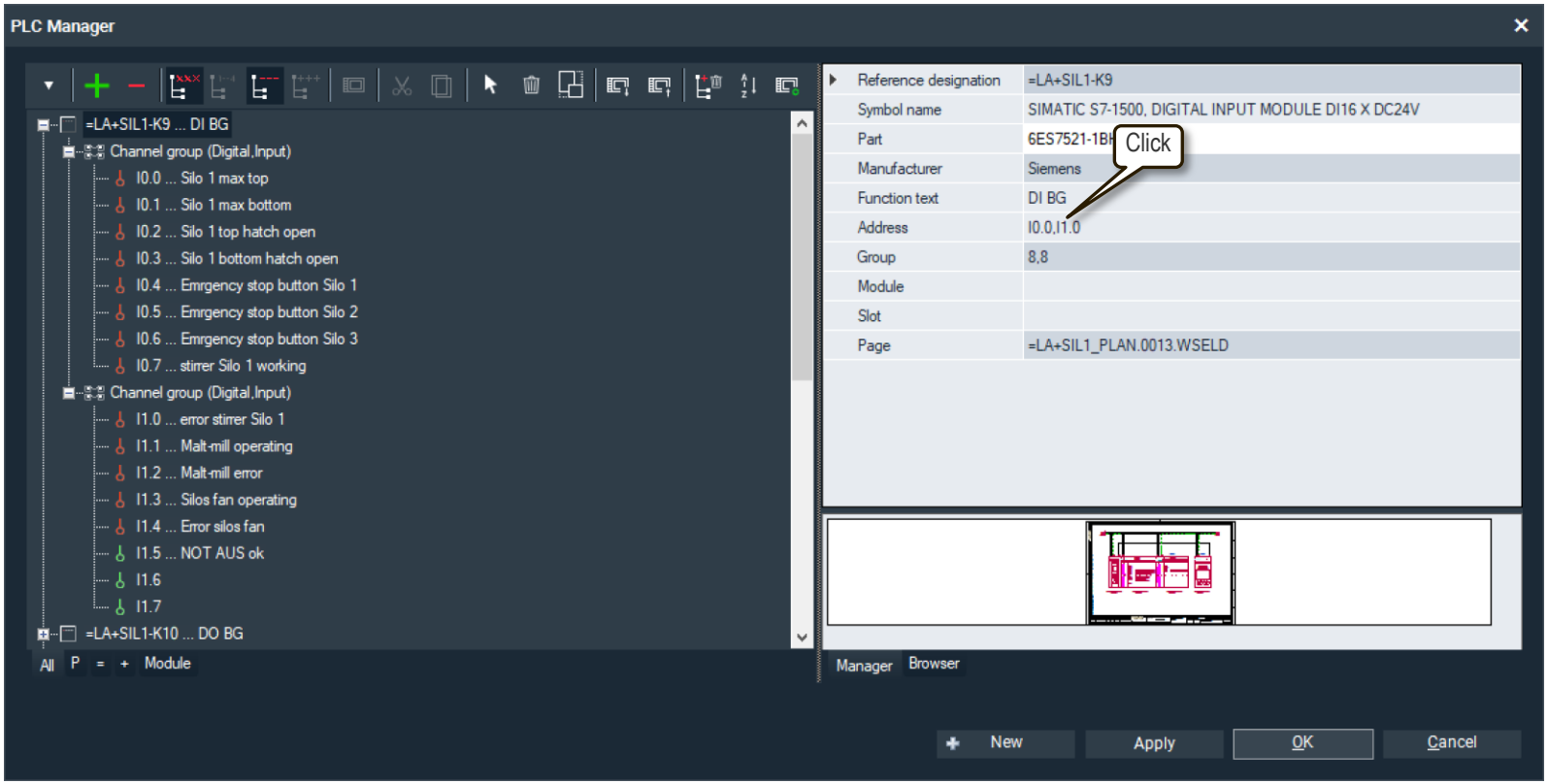


Basics – PLC management – Sheet 15



Basics – PLC management

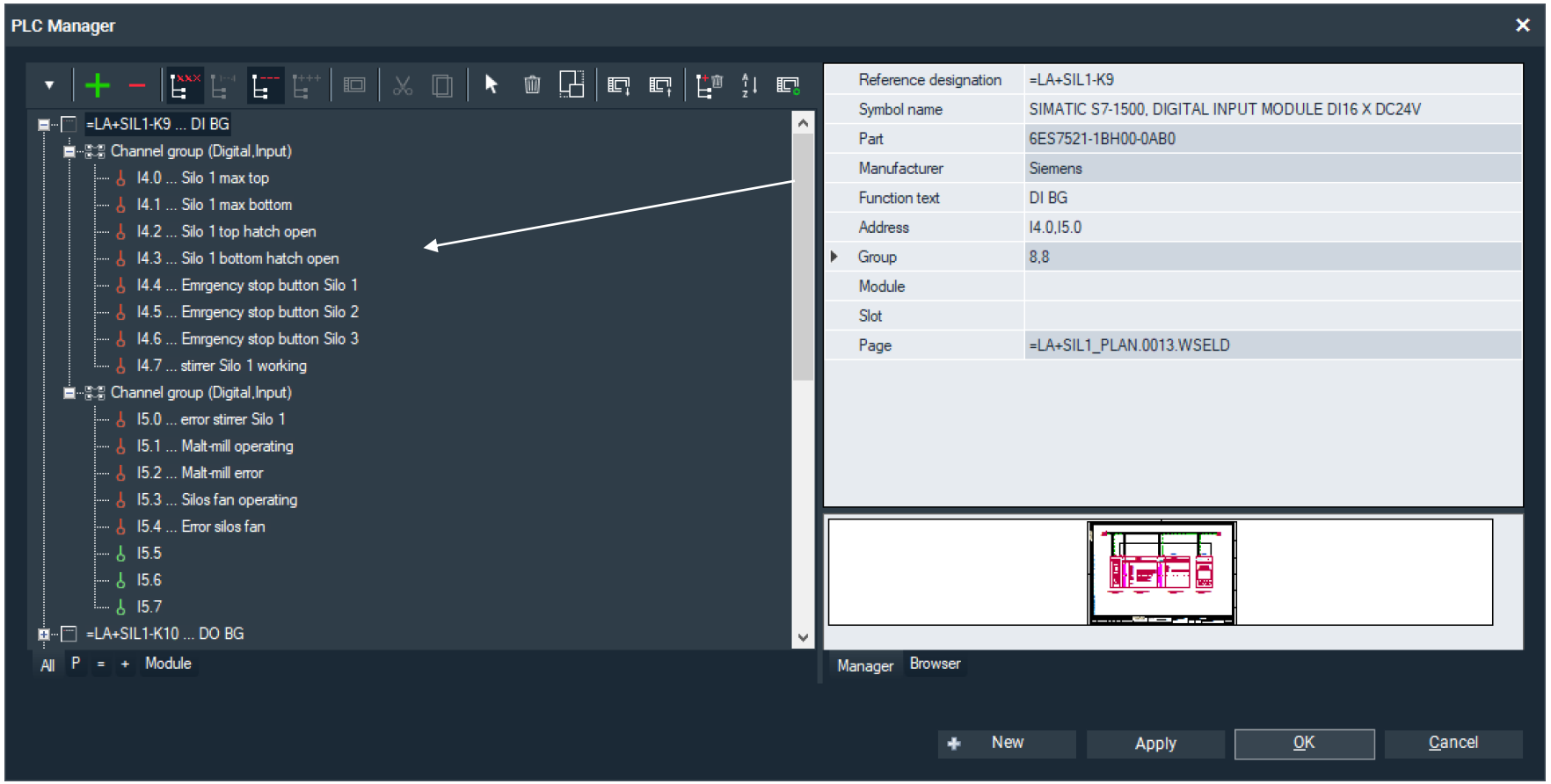
- PLC Manager Functions
  1. Call the PLC Manger.
  2. Change the addressing of the digital input module –K9 for both channel groups to E4.x and E5.x by clicking in the Manager in the "Address" line and making the changes there.



Input
I4.0,I5.0

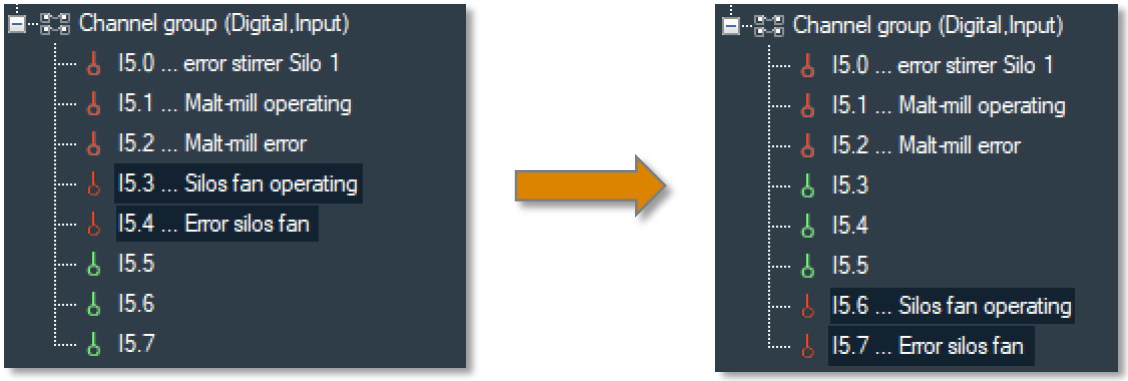
Basics – PLC management

- PLC Manager Functions



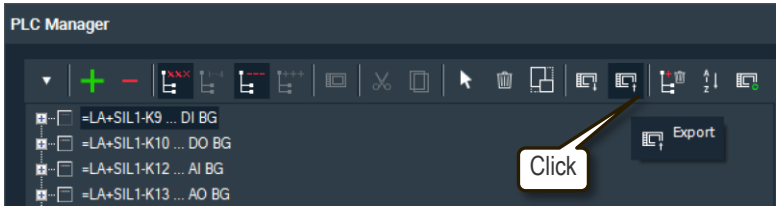
Basics – PLC management

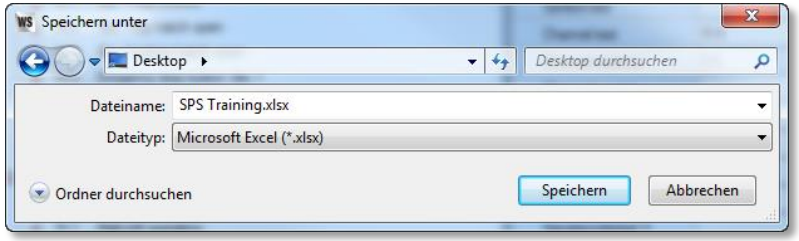
- PLC Manager Functions
  3. Move the data points E5.3 and E5.4 to E5.6 and E5.7 by marking them with the Ctrl key and dragging them down.
  4. Check the results in the schematic.



Basics – PLC management

- PLC Manager Functions
  1. Call the PLC Manger.
  2. Go to the "Export" command and use it to save the data point assignment of the PLC modules under the file name "PLC Training.xlsx" on your PC.

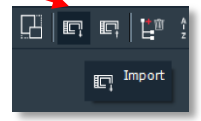


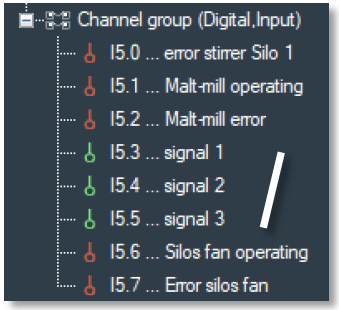


  3. Open this Excel file and enter some new texts in the "Comment" column and save this in the file.

	A	B	C	D
1	Pin	Comment	Symbol text	Reference
2	I4.0	Silo 1 max top		=LA+SIL1-K9
3	I4.1	Silo 1 max bottom		=LA+SIL1-K9
4	I4.2	Silo 1 top hatch open		=LA+SIL1-K9
5	I4.3	Silo 1 bottom hatch open		=LA+SIL1-K9
6	I4.4	Emrgency stop button Silo 1		=LA+SIL1-K9
7	I4.5	Emrgency stop button Silo 2		=LA+SIL1-K9
8	I4.6	Emrgency stop button Silo 3		=LA+SIL1-K9
9	I4.7	stirrer Silo 1 working		=LA+SIL1-K9
10	I5.0	error stirrer Silo 1		=LA+SIL1-K9
11	I5.1	Malt-mill operating		=LA+SIL1-K9
12	I5.2	Malt-mill error		=LA+SIL1-K9
13	I5.3	signal 1		=LA+SIL1-K9
14	I5.4	signal 2		=LA+SIL1-K9
15	I5.5	signal 3		=LA+SIL1-K9
16	I5.6	Silos fan operating		=LA+SIL1-K9
17	I5.7	Error silos fan		=LA+SIL1-K9

  4. Call the PLC Manager again in WSCAD and go to the "Import" command and import the above Excel file using the following dialog.
  5. Check the result in the Manager.





"If only what is finished were also always perfected."

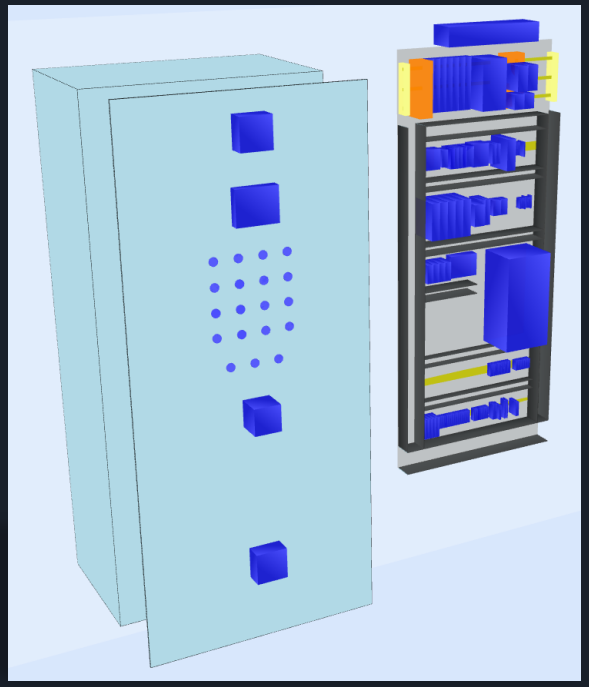
Peter Rosegger

(1843 – 1918)

WSCAD SUITE SUITE X

Training

Cabinet

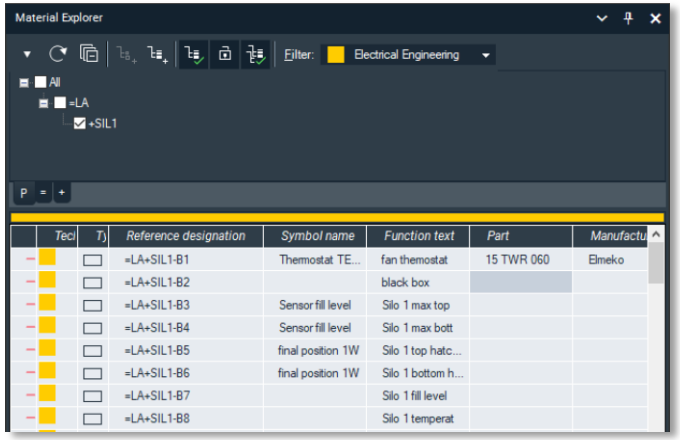


WS CAD

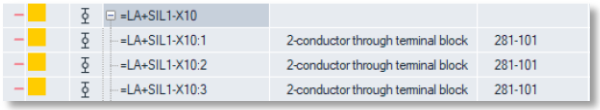
Cabinet – Prerequisites - Settings

- Prerequisites for Cabinet
  1. First create a page with the name "Cabinet layout" in the section =LA +SIL1 in the document folder "Cabinet".
  2. Check in the Material Explorer if you have assigned parts for all internal devices of the above-mentioned section.

Tip: If parts are missing:



- Possible for all components: double-click in the "Part" column
- or click on and then in the properties of the symbol
- for terminals: open the terminal strip



and click on and then call the Manager via the context menu and define the new part there.

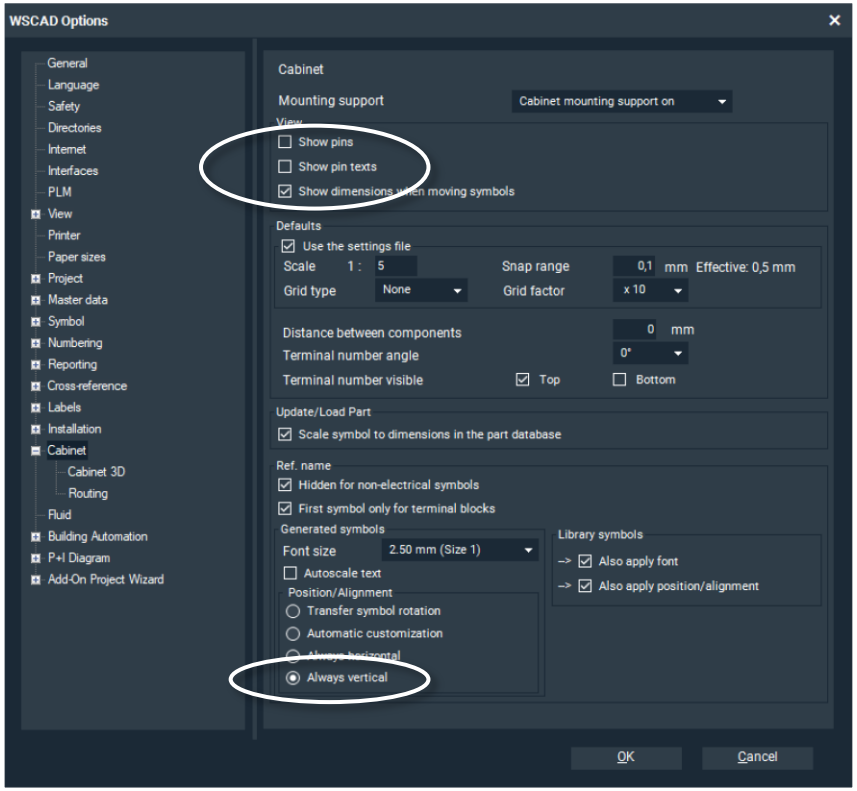
- or go directly to the Terminal Manager via "Manager".

- Settings for Cabinet
  3. Enable the Cabinet toolbar via "View | Toolbars".
  4. Check whether the command "Mounting support" is active.

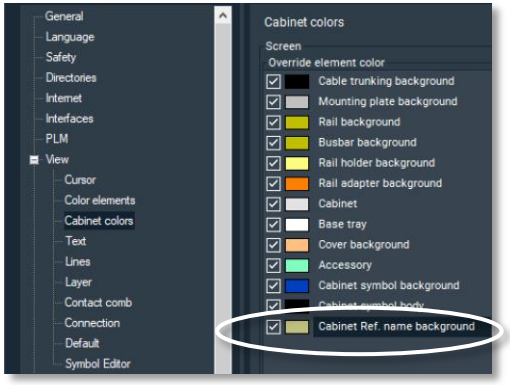
  5. Enable the "Alignment objects" toolbar.

Cabinet – Prerequisites - Settings

- Settings for Cabinet
  - Check the following settings under "Tools | Settings (options) | Cabinet":



  - And under Tools | Settings (options) | View | Cabinet colors:



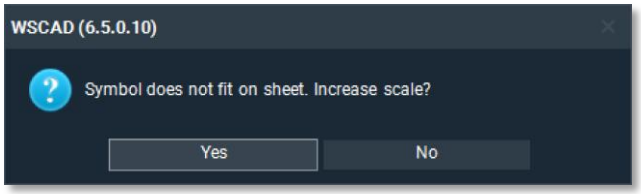
Cabinet – Mounting plate - Door

- Placement of mounting plate and door

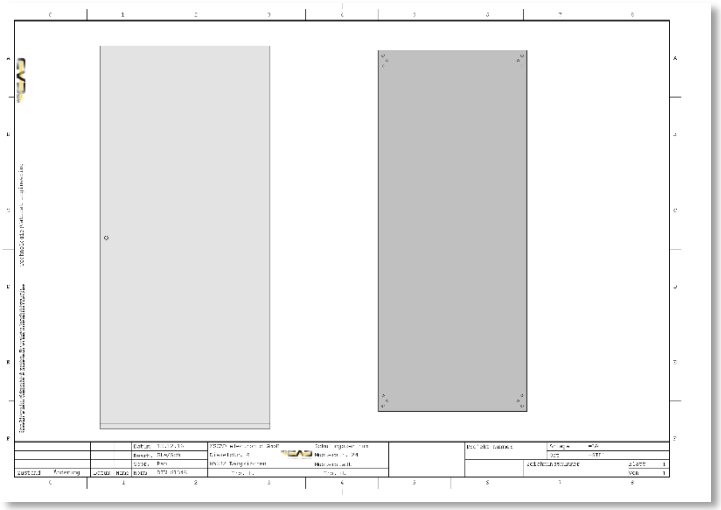
1. Set the filter in the Symbol Explorer to "Cabinet Engineering"
2. In the Library view, open the library Rittal, double-click on "PS4884_M" (for the mounting plate), and place it on sheet 1 in the Cabinet section.

   Even though it seemingly does not fit on the page, confirm the dialog to continue.

   In the following dialog, click "Yes":



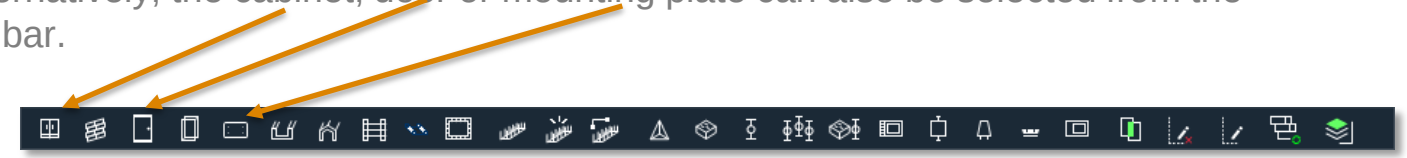
3. Now select the symbol "PS4884_S" icon (for the door) from the Symbol Explorer and place it next to the mounting plate on the left.



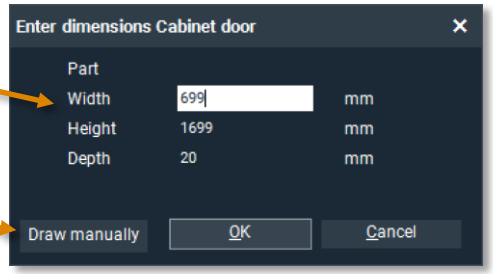
Cabinet – Mounting plate - Door

- Placement of mounting plate and door

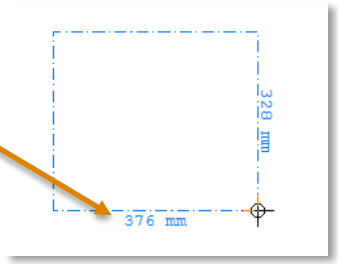
1. ***Tip:*** Alternatively, the cabinet, door or mounting plate can also be selected from the toolbar.



In the following dialog, the part database from which a part could be selected opens.
Do not look for any part
Click "Cancel"
and enter the dimensions manually
in the following window:



Or click here
to directly draw a
rectangle on the sheet.
The dimensions will be displayed.

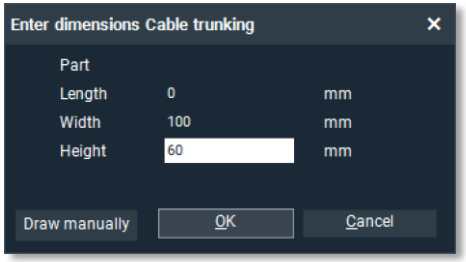


Cabinet – Cable ducts

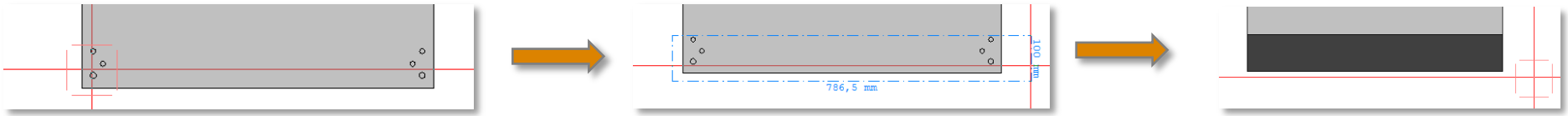
- Placing cable ducts

1. Click in the toolbar on cable duct.

2. In the part management, select nothing and click "Cancel".
3. In the following dialog, enter only the width of the duct (optionally also the height; useful for the 3D view).



4. After this, a square in the dimensions of the above duct width hangs on the cursor. Place this square slightly overlapping at the mounting plate with a left click and drag out the cable duct.



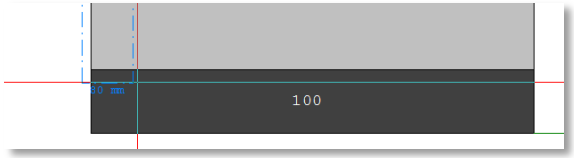
5. The duct is then cut off at the edges of the mounting plate and "extended" to them.

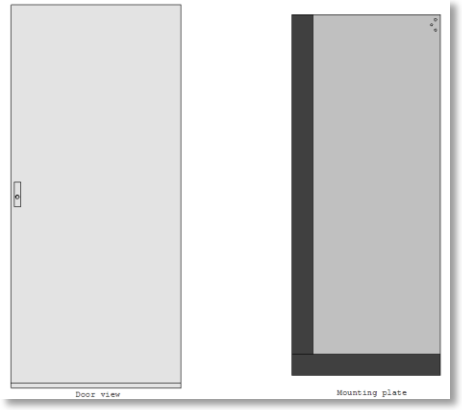
Tip: This automation feature only works when "Mounting support" is enabled!
If the duct should overlap the plate, then disable the mounting support.

Cabinet – Cable ducts

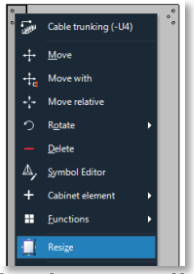
- Placing adjacent cable ducts
  1. Click again on  to place a new cable duct (width 80mm).
  2. Drag this duct vertically from the upper left corner of the mounting plate to the already placed horizontal duct.

     Let them overlap slightly.

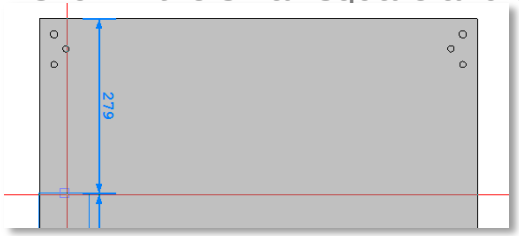


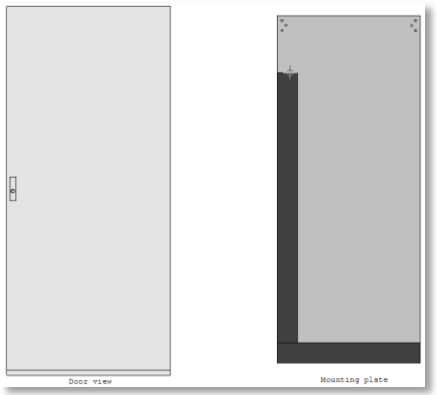


- Change size of cable ducts (Snap off )
  1. Right-click on the vertical duct and click "Resize".



  2. Click in the small square and drag it to 279 mm.

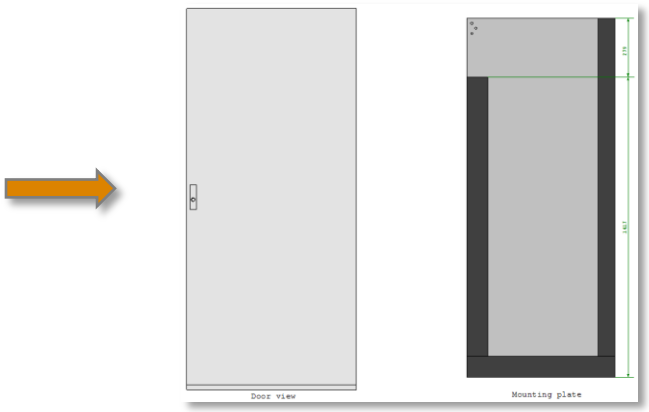




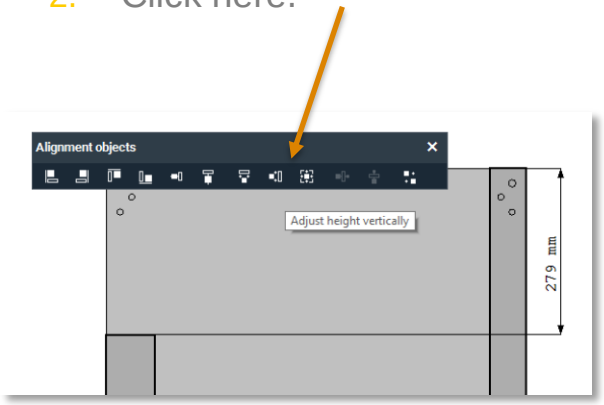
Cabinet – Cable ducts

- Placing additional cable ducts
  1. Click again on  to place a new cable duct (width 60mm).
  2. Drag this duct vertically from the upper right corner of the mounting plate to the already placed horizontal duct.

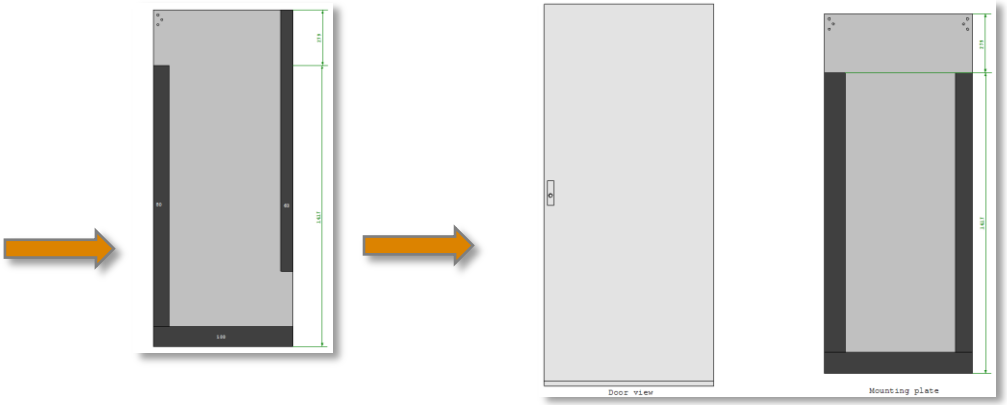
     Let them overlap slightly.



- Alignment function
  1. Use the Ctrl key and first mark the left and then the right vertical duct (ducts are then blue).
  2. Click here.

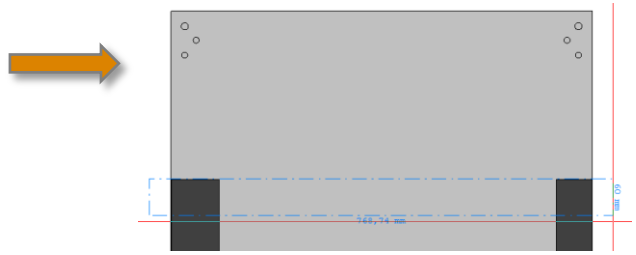


  3. Then slide the duct downwards.

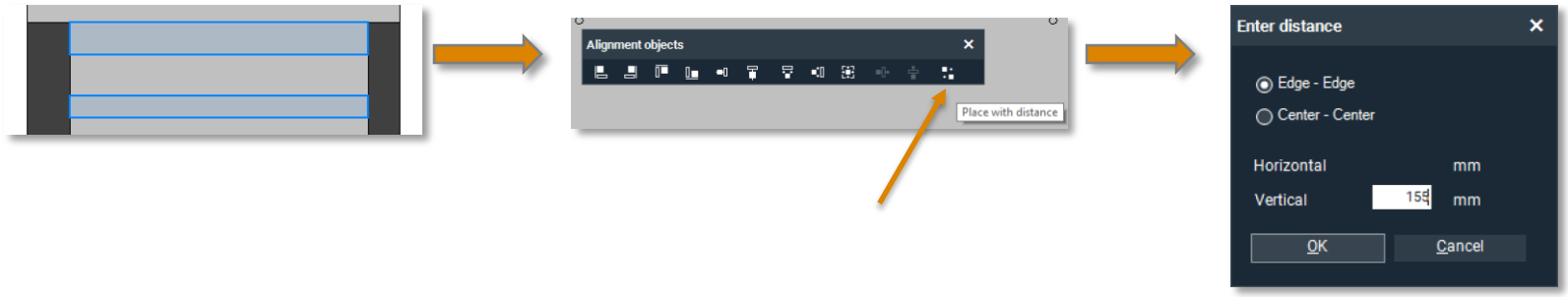


Cabinet – Cable ducts

- Complete the drawing as shown on slide 11.
- You can drag the horizontal ducts beyond the existing ducts, since they are cut off at the edges.



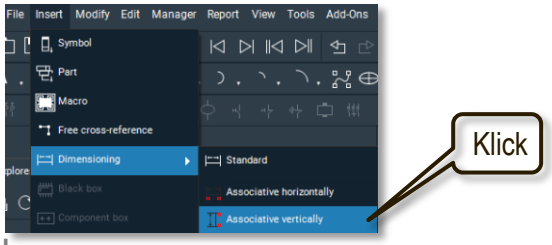
- You can also use multiple copying (Ctrl+D, distance, + or -).
- Place the horizontal cable ducts initially without any precise alignment.
- Dimensionally accurate alignment:
  1. Select the aligned ducts (the first marked one is the "master").
  2. Then click on the command "Place with distance".
  3. Enter the values in the following window and confirm with "OK".



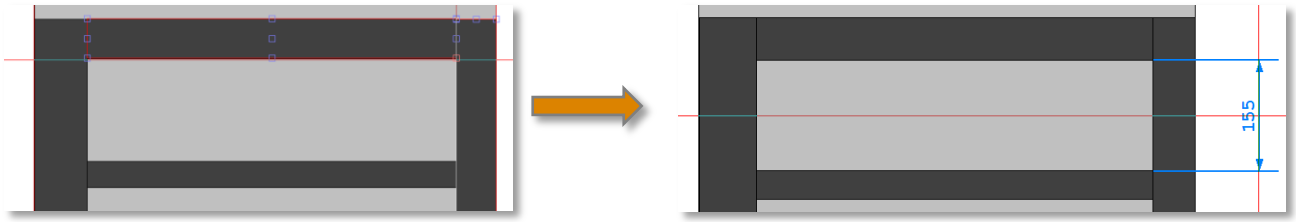
Cabinet – Cable ducts

- Associative dimensions (as shown on slide 11)
- This type of dimensioning is not only a very good method of dimensioning, but also a good way to control distances, sizes and orientations retrospectively.

1. Go to menu bar "Insert | Dimensions".

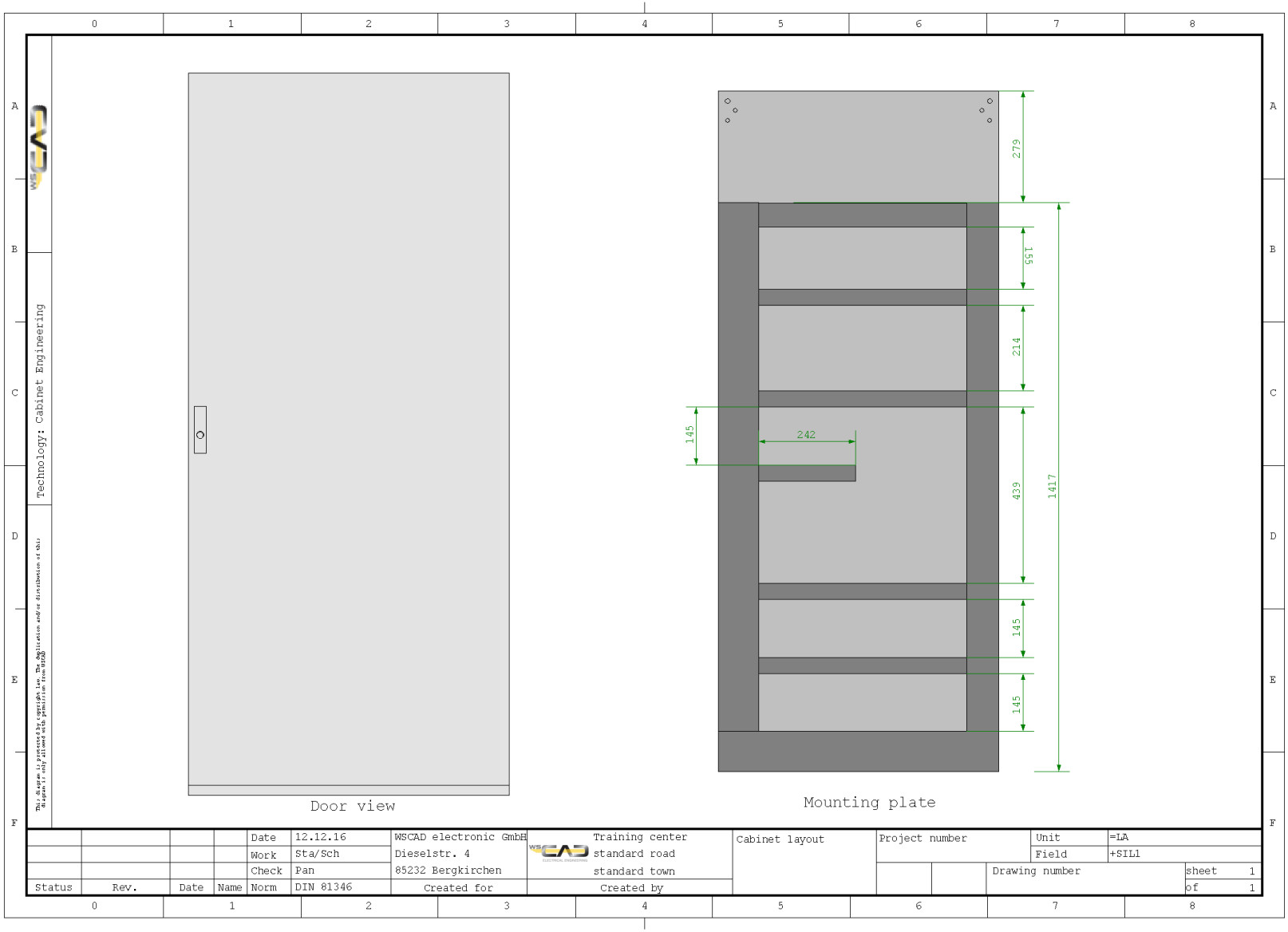


2. Form there, select "Associative vertically", for example.
3. If you now go over the drawing with the cursor cross, all the edge points of the objects are displayed. Choose the first and then the second edge point and drag out the dimensions.



4. When you move the dimensioned objects, the dimensions change "with constraints".

Cabinet – Cable ducts

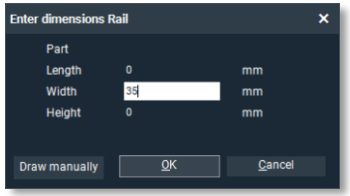


Cabinet – Mounting rails

- Placement of mounting rails (cap rails)

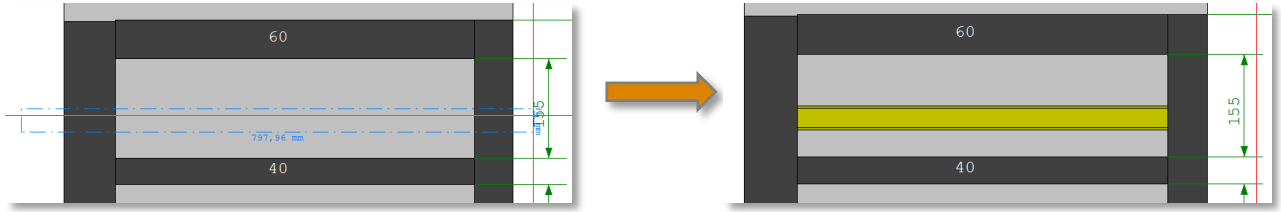
1. Click on the command "Cap rail":

2. Click in the "Part" dialog on "Cancel" and enter only the width of 35mm in the following dialog.

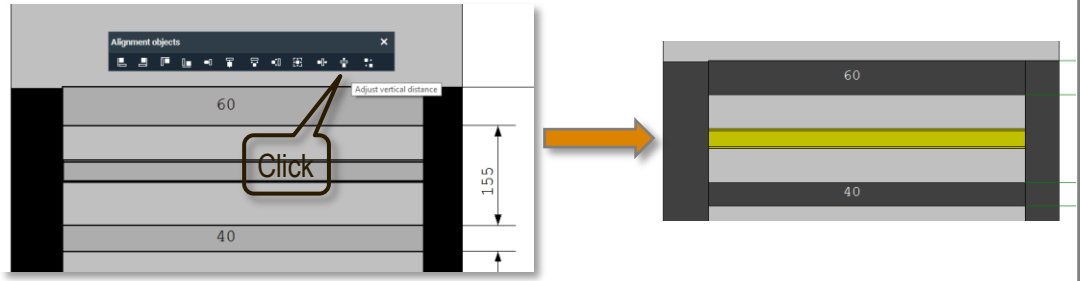


3. After that, you can place the cap rails in the same way as the cable ducts.

   Drag the cap rail to overlap the cable ducts; the rail is automatically clipped at the inner sides of the ducts.

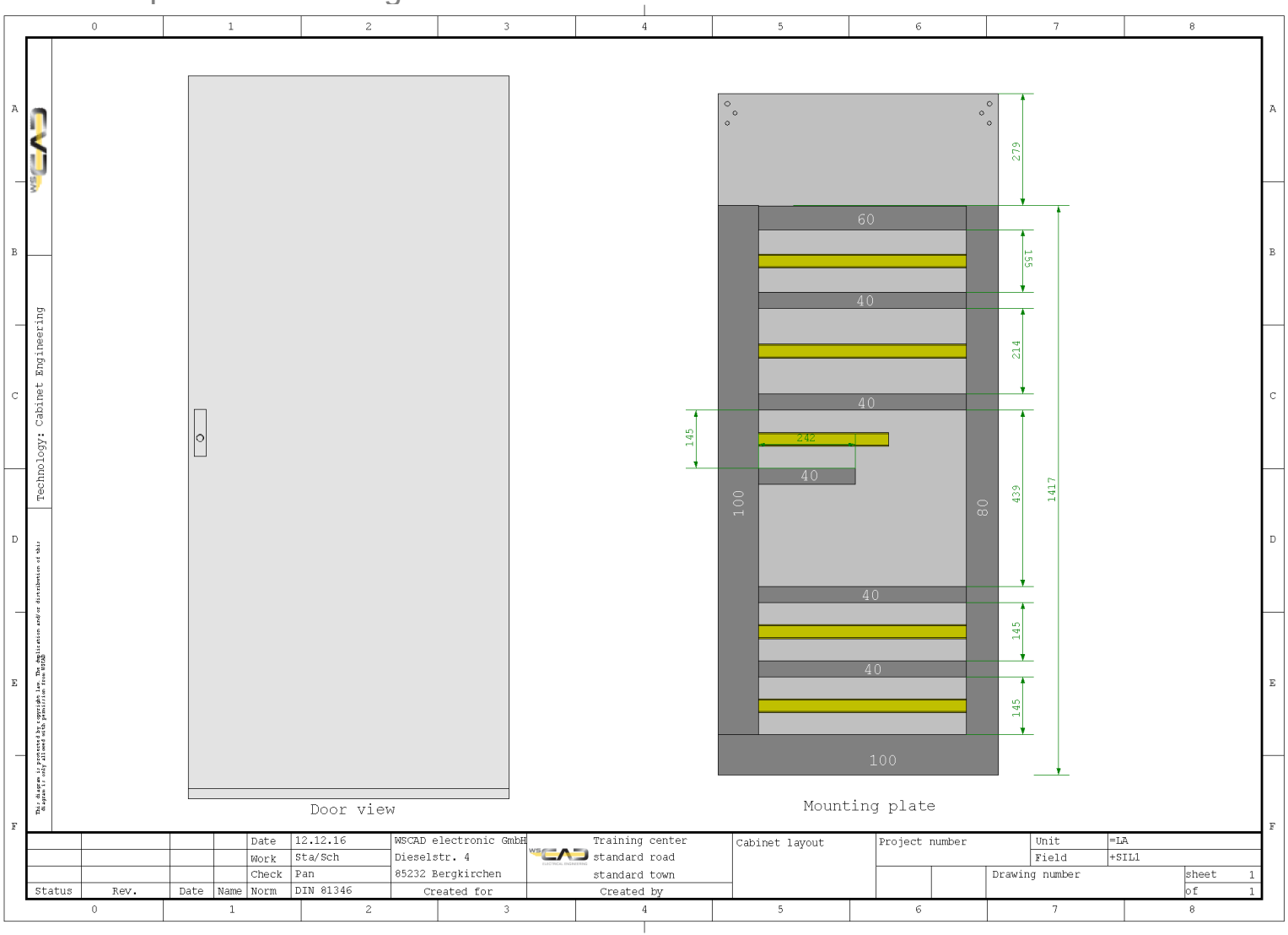


4. Center the rail between the ducts by marking the upper duct, cap rail and lower duct and then clicking the command:



Cabinet – Mounting rails

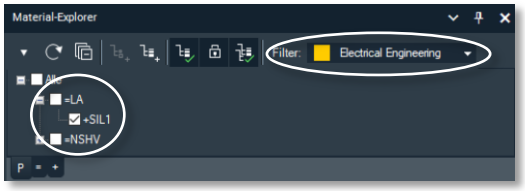
- Complete the drawing as shown here:



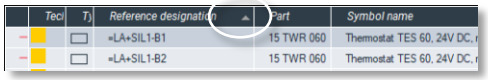
Cabinet - Placing components

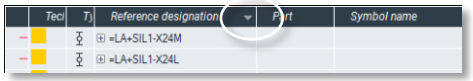
- Placing existing project components
- Components placed in the schematic

1. Open the Material Explorer and set the filter and section there:

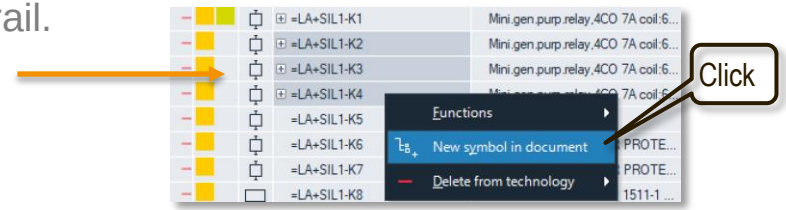


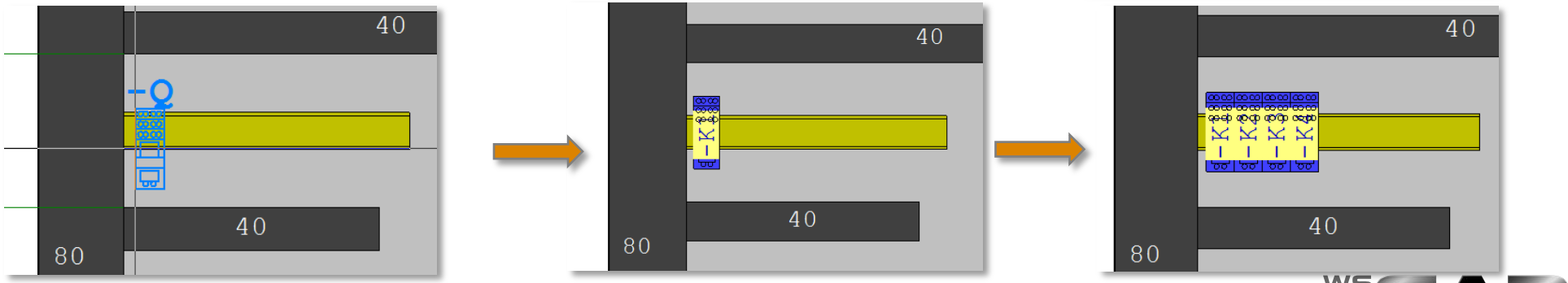
2. Sort the columns in descending or ascending order:





3. Double-click on the reference designation of the component (-K1) that you want to place and drag it (without holding down the mouse button) onto the drawing.
4. Click when you drag the component over a cap rail.
5. Highlight -K2 to -K4; right-click on it and multiple components can thus be placed.

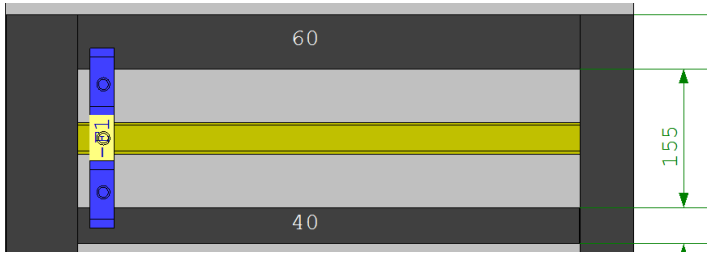




Cabinet - Placing components

- Changing parts of existing project components

1. Place the component -F1.



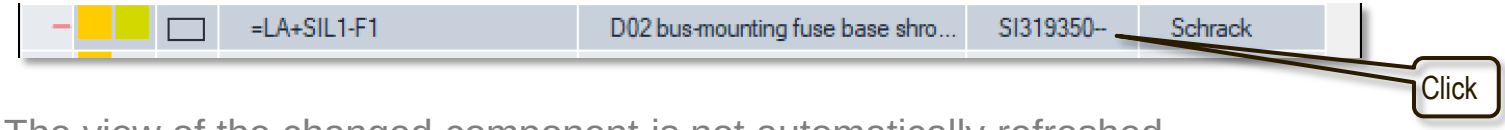
The part SI319350 (D02 bus-mounting fuse base) has been defined for -F1.

But this does not make sense for the setup.

It is therefore to be changed to a

fuse base 3-pole.

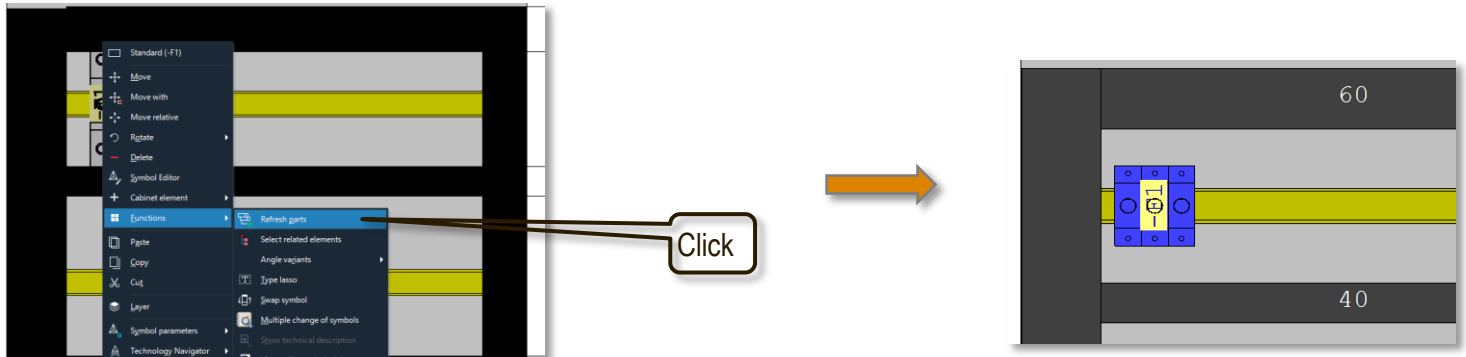
2. In the Material Explorer, double-click in the row of =LA+SIL1-F1 in the "Part" column and change part to SI312930– (D0 mounting fuse base 3 pole)



3. The view of the changed component is not automatically refreshed.

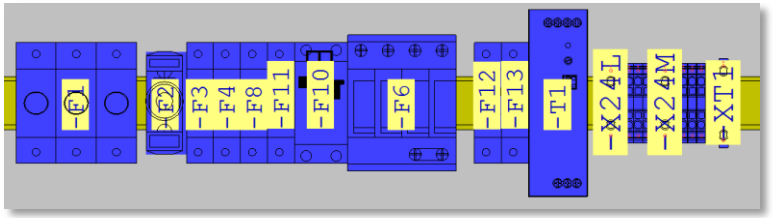
To do this, click in the drawing on -F1, then right-click and go to

"Function | Refresh parts".

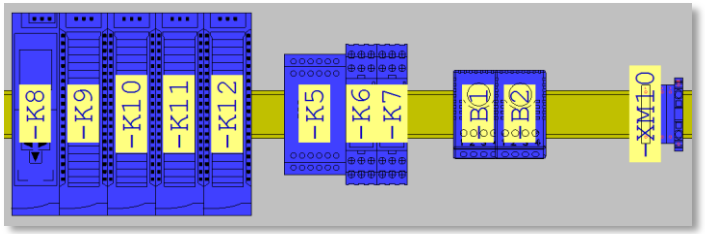


Cabinet - Placing components

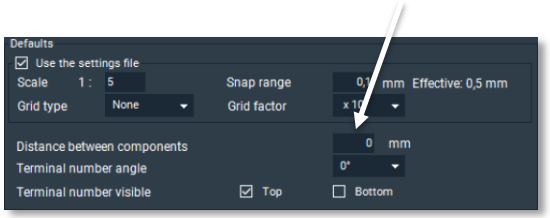
- Placement of components for cap rail mounting
- Place the following components as listed here and shown on slide 17.
  (Mark of the components in the specified order so that they are placed accordingly)
  - -F2, -F3, -F4, -F8, -F11, -F10, -F6
  - -F12, -F13, -T1



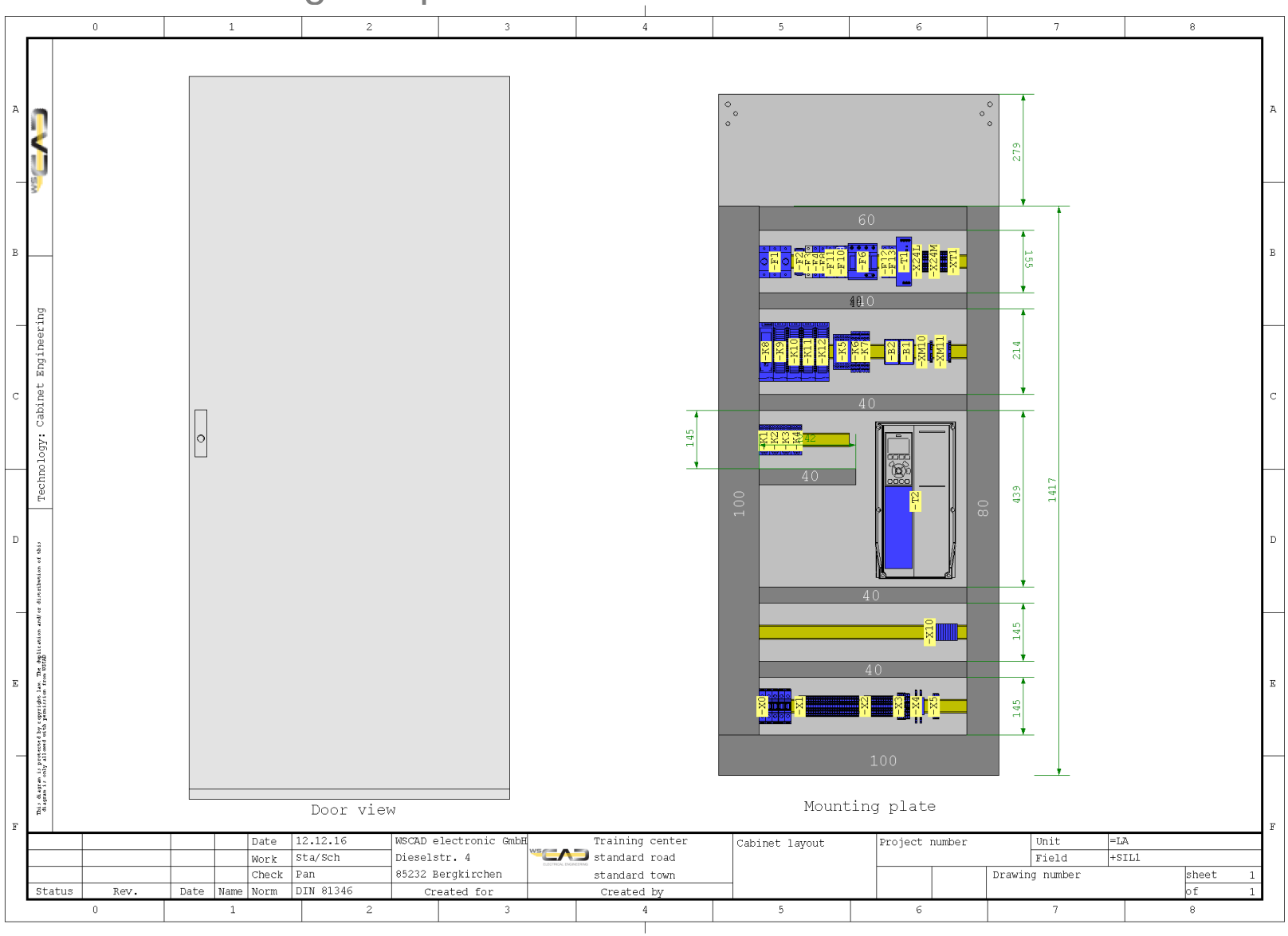
  - -K8, -K12
  - -K5, -K6, -K7
  - -B1, -B2



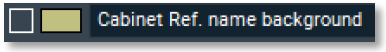
- Placement of parts without cap rail mounting
  - -T2, -E1
- Placement of terminal strips
  Tip: to place the following terminal strips together, the distance between them must not be "0" (leading terminal block or similar).
  Go to "Tools | Settings (options) | Cabinet "and change the value.
  - -X24L, -X24M, -XT1
  - -XM10, -XM11
  - -X0, -X1, -X2, -X3, -X4, -X5, -X10



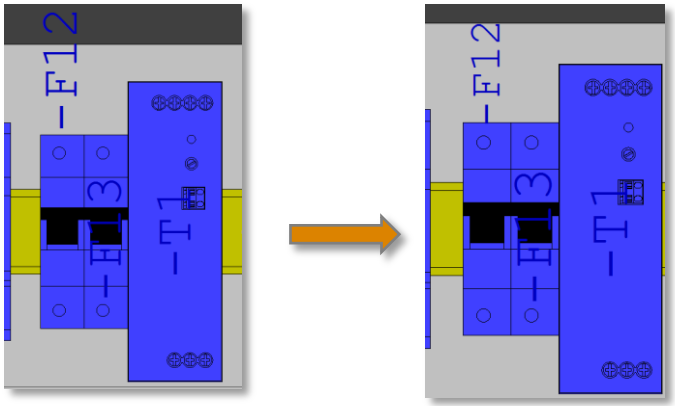
Cabinet - Placing components



Cabinet – Ref. name - Placement

- After components are placed, the ref. name is always set in the center. The direction, size and appearance depend on the settings.
  - Ref. name with/without background:
    1. Go to "Tools | Settings (options) | View | Cabinet colors".
    2. Activate the setting → 

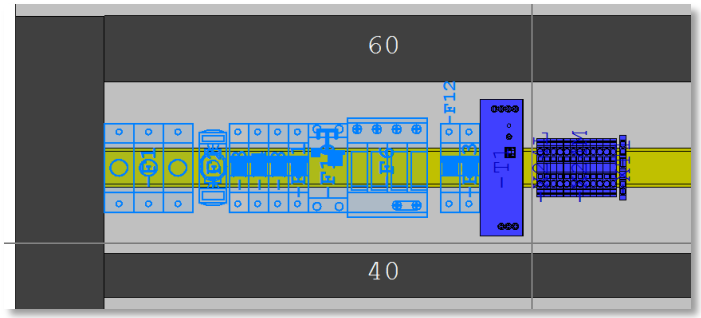
  - Move ref. name of **one** component:
    1. For example, click on -F12v and right-click on "Symbol parameters | Ref. name | Move".
    2. Use the mouse or the arrow keys to move the ref. name.
  - Edit ref. name properties:
    1. Click on -F12, then right-click on "Symbol parameters | Ref. name | Properties".
    2. You can change the text angle and text size as well as the orientation here.



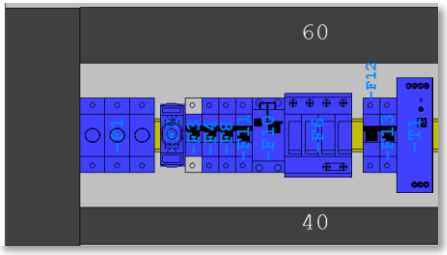
Cabinet – Ref. name - Placement

- Move ref. name of **multiple** components:

1. Mark all components for which you want to move the ref. name by using a window.

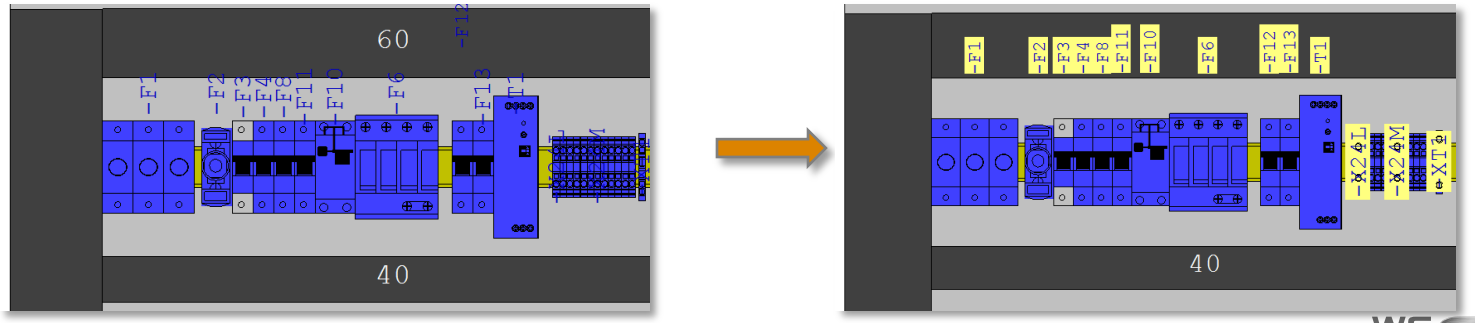


2. Then go the menu "Edit | Intelligent selection | Ref. name".
3. The ref. names are now light blue.



4. Right-click near the light blue ref. names, then click "Selection | Move".

   Now the insert points of all ref. names are visible.
5. Move all the ref. names, preferably by using the arrow keys, and confirm with Return.
6. If you go to "... Selection | Properties" as under 2., you can change the text features of all ref. names. In this case, go to "Alignment" and select "Centered - Left".

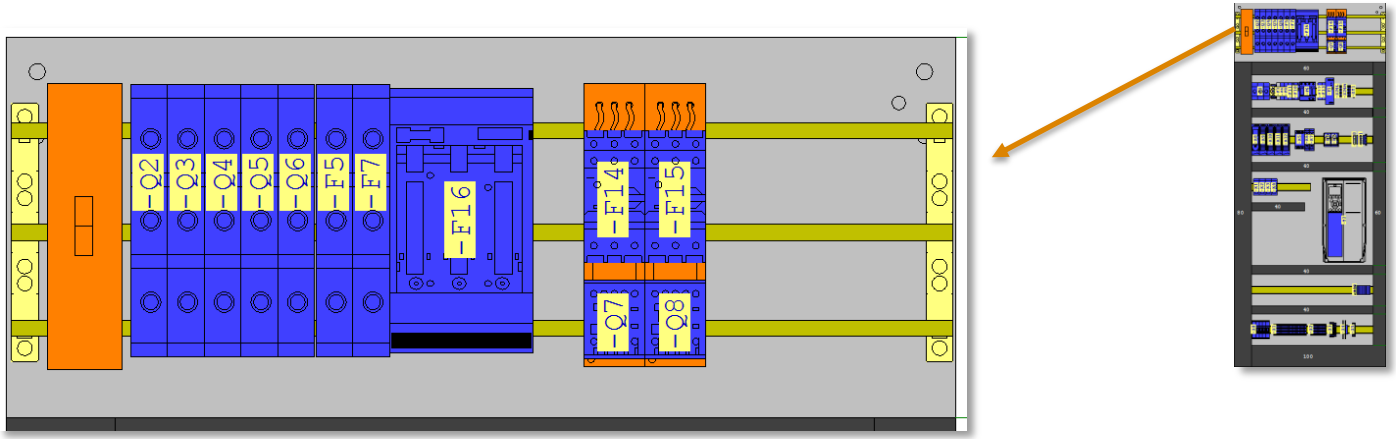


Cabinet – Placing components - Rail system

- Placement of components for rail system assembly:
  1. Click in the toolbar on Rail holder
  2. Place two rail brackets at the top of the mounting plate with the offered parts.

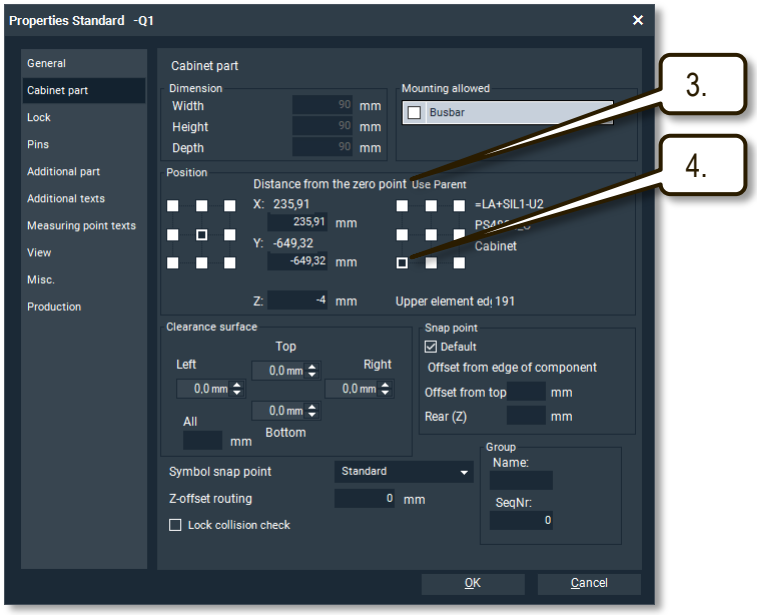
     Tip: Use the alignment commands for precise height alignment of the rail holder.
  3. Click in the toolbar on Busbar , select the offered parts, and place 3 busbars (length cut off accordingly here as well).
  4. Place a rail adapter (part no. 9342.220) on the left from the toolbar.
  5. Place -Q2 to -Q6, -F5, -F7, -F16 next to it on the right.
  6. For drive control of "Agitator Silo 1" and the "grist mill", the motor overload switch and contactors should also be placed on the rail system.

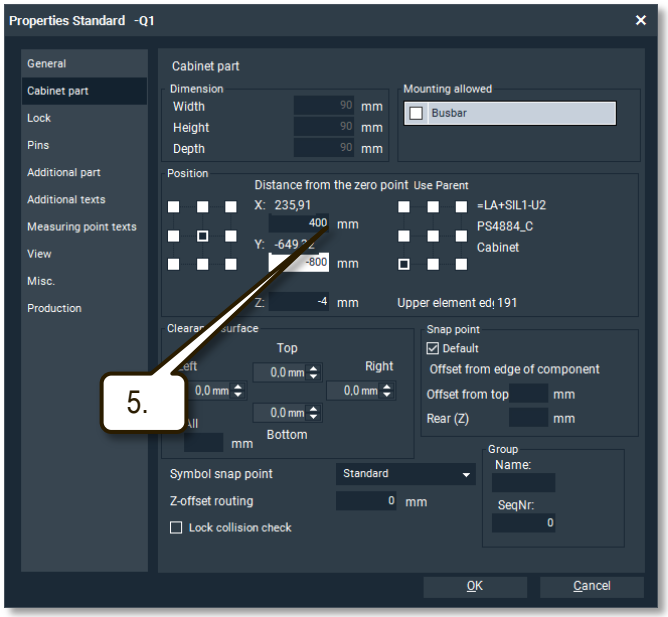
     To do this, first place 2 connection adapters (PartNo. 9340.370).
  7. Place -F14, -F15 above this and -Q7 and -Q8 below it.



Cabinet – Placing components - Door

- Precise placement of components on the door:
  1. Place the main switch -Q1 on the door; intentionally somewhat eccentric.
  2. Right click on -Q1, then click "Properties | Cabinet component ".
  3. Click Parent.
  4. Specify the Parent coordinates origin.
  5. Enter the new X/Y coordinates and confirm with "OK".

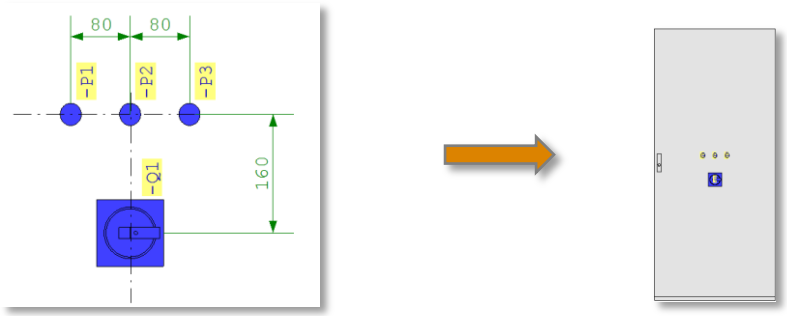




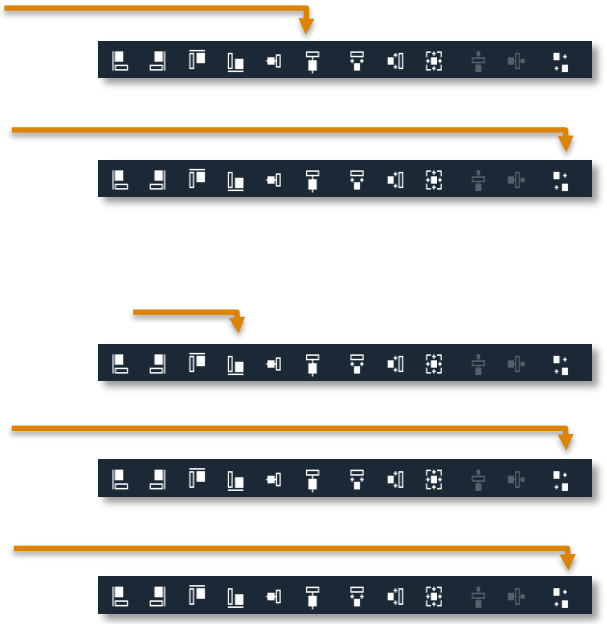
- Through the above dialog, components can be placed very precisely.

Cabinet – Placing components - Door

- Aligning multiple components (using the 3-phase control lamps as an example)
- Goal:



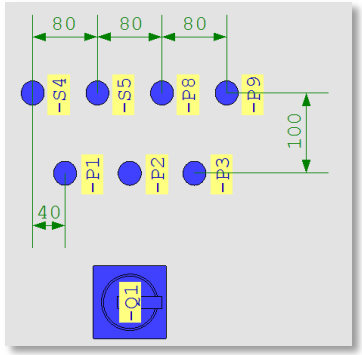
1. Place -P2 slightly above the main switch -Q1
2. First select -Q1, then -P2 and click on
   (Alignment of -P2 on the central axis of -Q1)
1. First select -Q1, then -P2 and click on
   (Click on "Center-Center", Vertical value: -160)
1. Place -P1 and -P3 slightly next to -P2
2. First select -P2, then -P1 and -P3, and click on
   (-P1 to -P3 are then aligned flush)
1. First select -P2, then -P1, and click on
   (Horizontal value: -80)
1. First select -P2, then -P3, and click on
   (Horizontal value: 80)



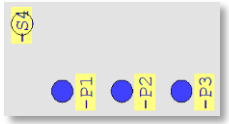
- Result: alignments of the components as shown above.

Cabinet – Placing components - Door

- Alignment of multiple components via a group (suing the elements from the "Agitator" drive as an example)
- Goal:

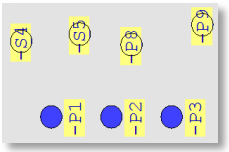


1. First place-S4 above -P1



2. First select -P1, then S4, and click on
   (Horizontal value: -40, Vertical value: -100)

3. Place -S5, -P8, -P9 "roughly" to the right of -S4



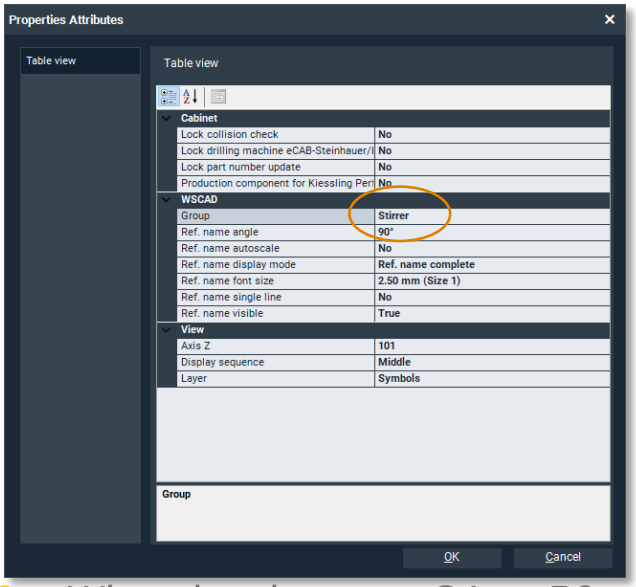
4. Mark-S4 to -P9 by means of a window and click on
   (Always draw a window from left to right!)
   (-S4, -S5, -P8, -P9 are then aligned flush)

Cabinet – Placing components - Door

5. Mark the above-mentioned elements again, then right-click and choose "Selection | Properties".

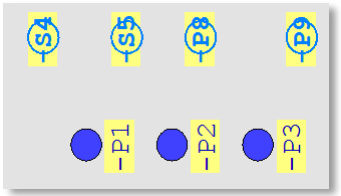
   This dialog has access to all the common properties of the selected elements.

   By assigning the "Group" attribute, these elements can be combined into a group.



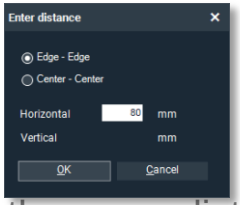
Under "Group", enter the name „Stirrer" and confirm with "OK".

When you select an element from -S4 to -P9, all elements are marked at the same time.



6. When the elements -S4 to -P9 are marked, click on

   (Horizontal value: -80, Vertical value: nothing)



   All elements of the „Stirrer" group are assigned the same distance from one another.

7. Check your result by dimensioning as on slide 23 above ("Goal").

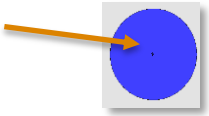
Cabinet – Placing components - Door

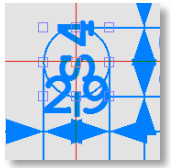
- Placement using the commands practiced above

1. If you want to continue the placement of the other elements (-S6, -S7, -P10, -P11) as shown above, and thus want to use -S4 as a "zero point element" (via the command "Place with distance"), you will need to first (!) ungroup -S4 to -P9. To do this, right-click on the group and delete the group name "Agitator" in the Properties dialog.

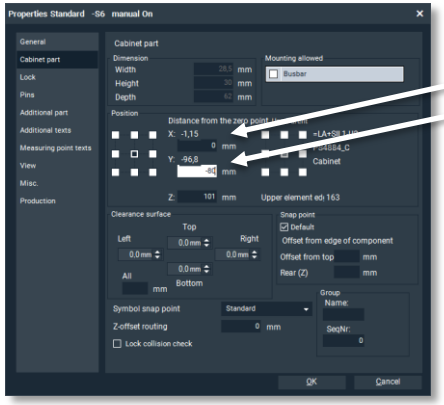
- **Or:** Placement via "Relative coordinates origin"

1. Click on -S4, then right-click, since grouped -> on -S4 /Move.
   The edge points and the central point of the symbol are recognizable.
2. Click the central point (you may need to turn off Snap using F5) and immediately press F9. At the center of the symbol, a new zero point can now be seen.



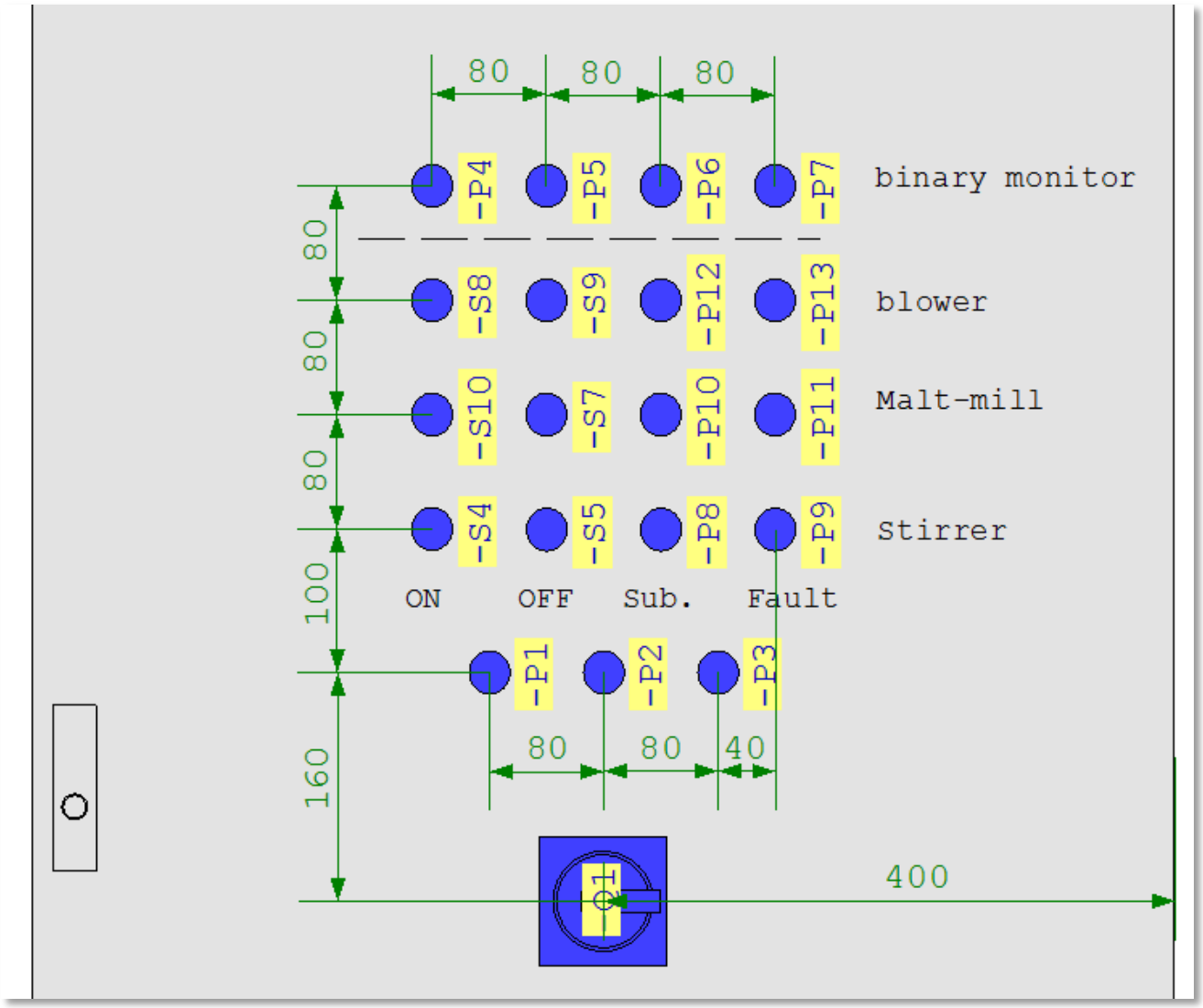


3. Place -S6 "roughly" above -S4.
4. Right-click on -S6, then open the dialog "Cabinet part" dialog via the properties.

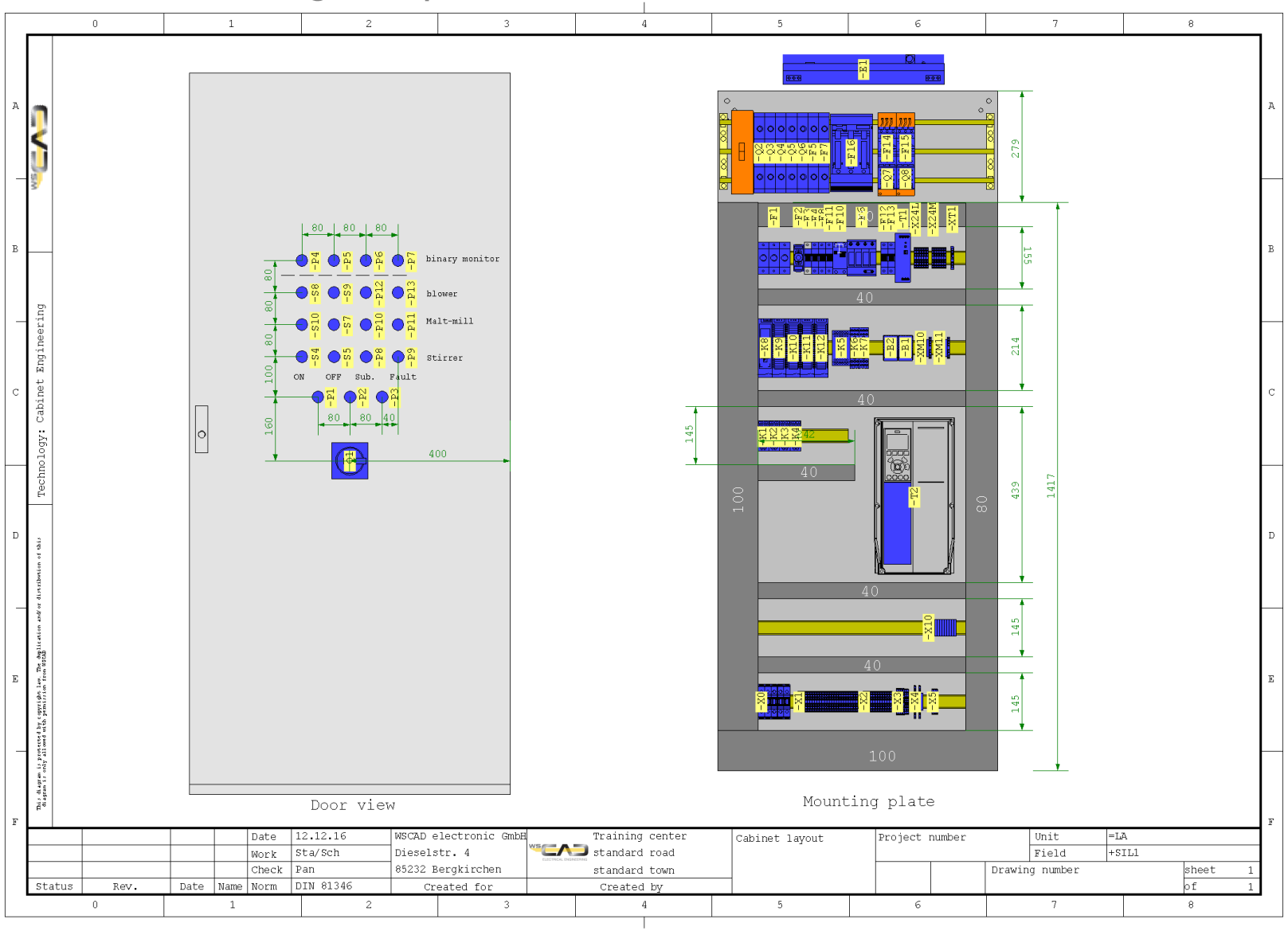


5. Now use the new zero point by entering the distances (0 and -80) under X and Y (no parent).
6. -S6 is thus now aligned. Place -S7, -P10, -P11 adjacently on the right and align them.
7. Complete your placement as shown on slide 26.

Cabinet – Placing components - Door



Cabinet - Placing components

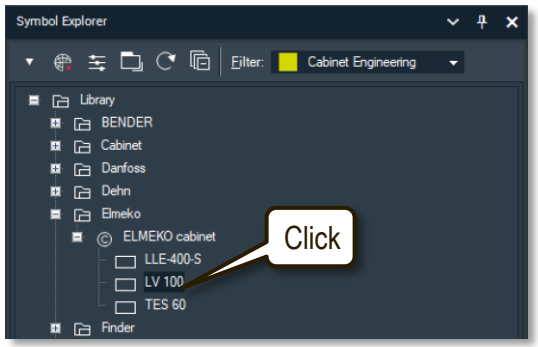


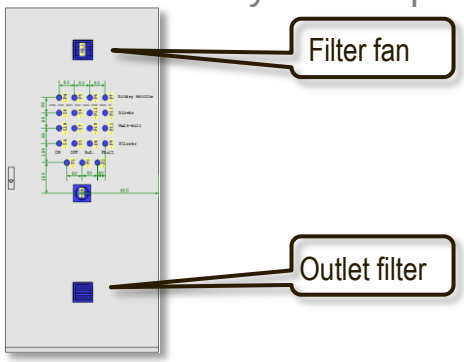
Cabinet – Placing additional components

- Placing non-existing project components
- Benefits: - Design cabinet without schematics
  - Part data created in the cabinet can be retrieved in the schematic
  - rapid assessment of space requirements
- Such components can be placed using
  - Symbol Explorer
  - "Insert | Part" command
  - Creation of virtual parts in the Managers
  - Commands 
  - Import function from the Material Explorer

Cabinet – Placing additional components

- Placing components using the Symbol Explorer:
  1. Place the still unplaced filter fan -ML1 on the door at the top
  2. For the still missing outlet filter, place the symbol LV 100 from the Symbol Explorer





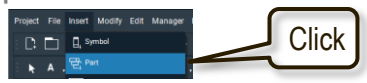
  3. In the properties of the symbol, the part is missing. Since this is not present in wscaduniverse.com, please create a new part for the outlet filter in the part database with the following data.
     Tip: First call up the part "10 115 150" as a "model".

     Part: GV100, Manufacturer: ELMEKO, Type designation: GV100, Part name: Outlet filter GV100.
  4. Assign this part to the component placed under 2.

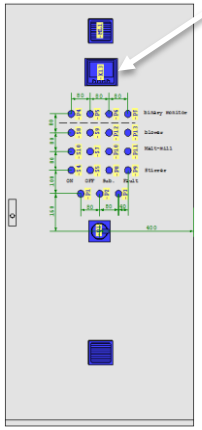
Cabinet – Placing additional components

- Placing components using the "Insert | Part" command

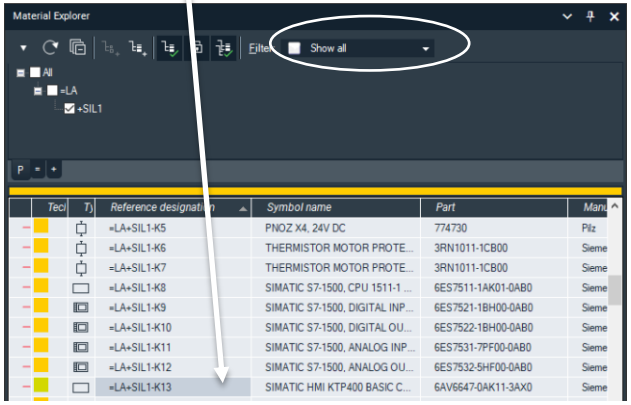
1. Go to the command



2. Bring up "wscaduniverse.com" in the part management and enter the following part number in the search field above: 6AV6647-0AK11-3AX0.
3. Then click "WSCAD Import" to import this part into your database.
4. Following a successful import, double-click on the new part number and place the panel above the controls.



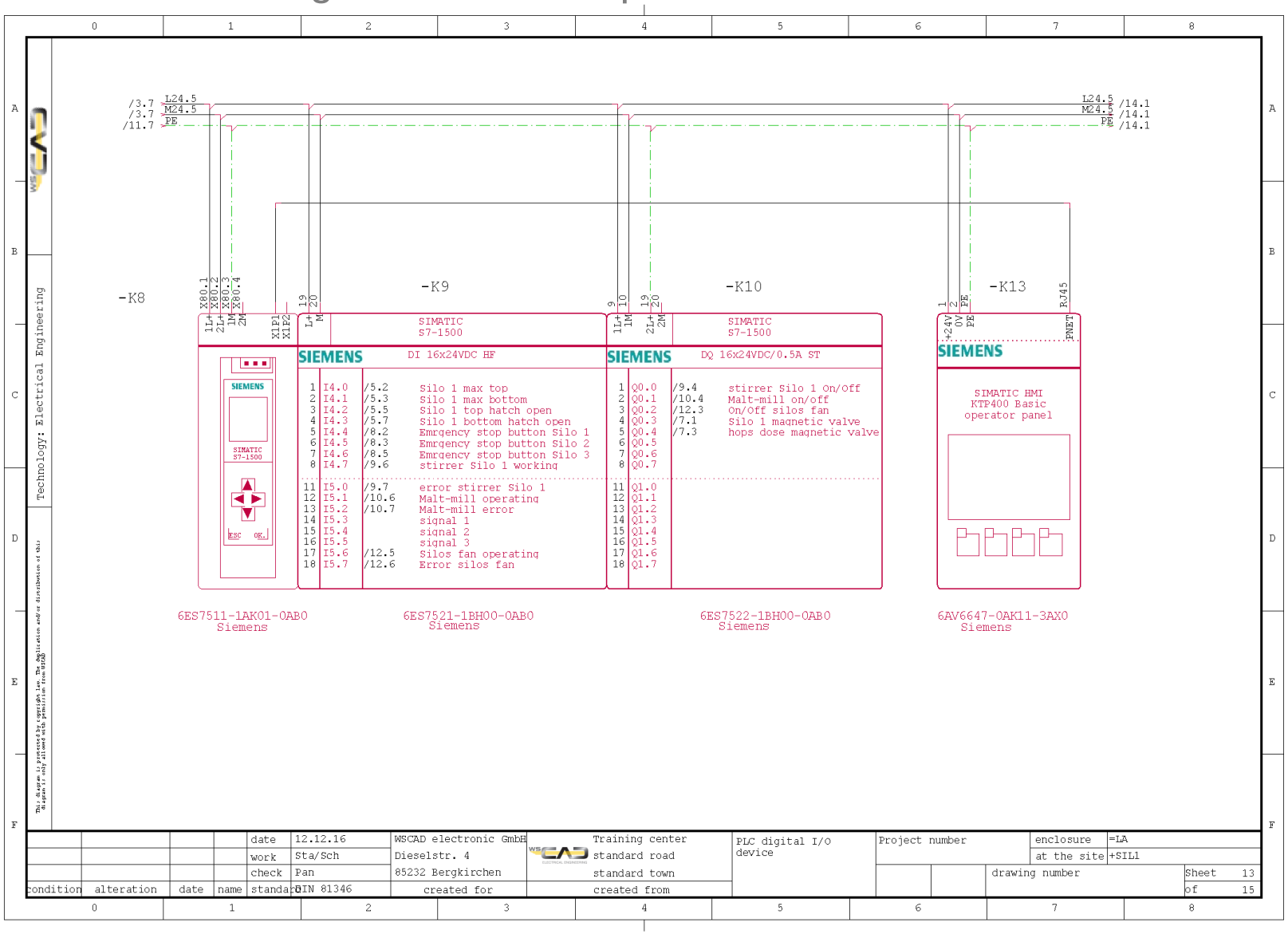
5. Open the Material Explorer. The new panel -K13 is then displayed there.



With the imported part data from -K13, a cabinet and EE symbol were simultaneously linked. This allows you to perform the placement in the schematic analogoussly to the placement in the cabiinet.

6. Open the schematic page 13, go to the Material Explorer and set the filter there to "Cabinet Engineering" or "All" and double-click on -K13, place the symbol for the panel and complete the drawing as shown on slide 31.

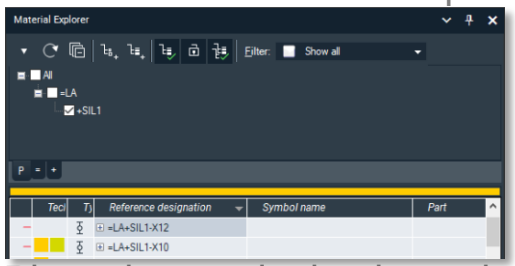
Cabinet – Placing additional components



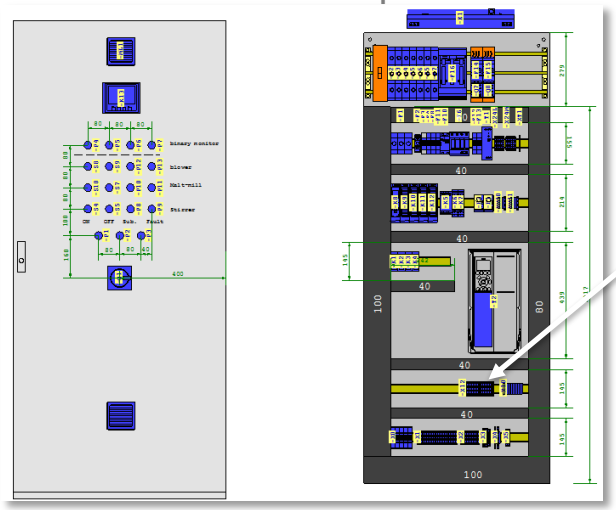
Cabinet – Placing additional components

- Placing components by creating virtual parts in the Managers.
- Let us assume, for example, that you need to know the space requirements for a terminal strip, but this terminal strip has not yet been configured in the schematic.

1. Go to the terminal management and create a new terminal strip –X12 with 20 Feed-through terminals, Part No: 281-601.
2. Open the Cabinet page in the project.
3. Find the new terminal strip –X12 in the Material Explorer (check the filter!).



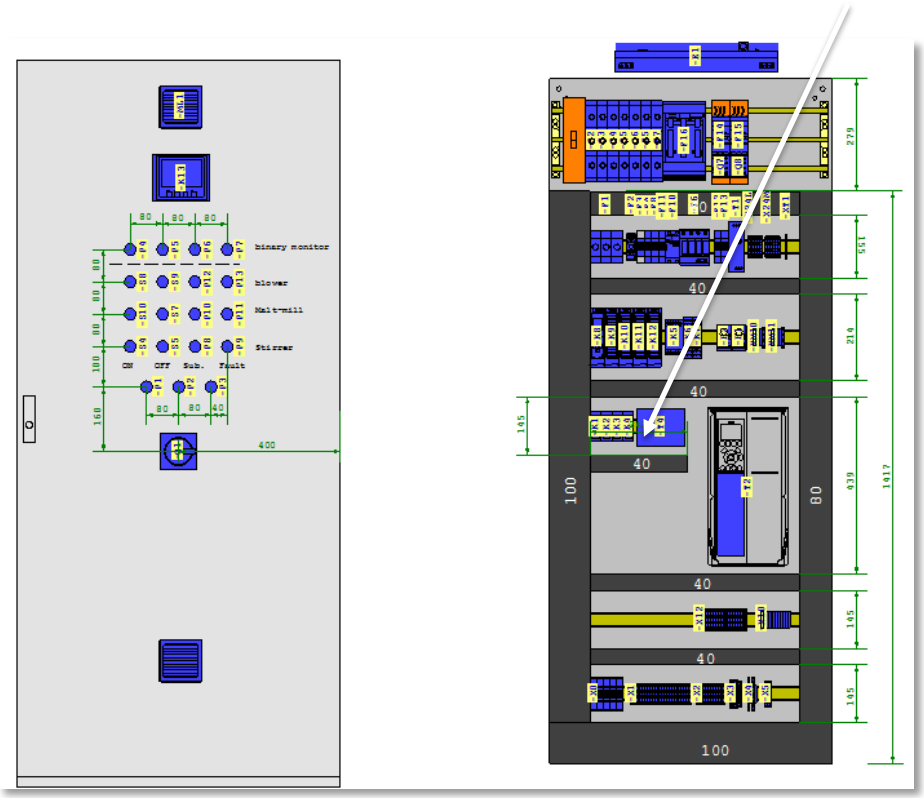
4. Place the terminal strip on the mounting plate by right-clicking.



Tip: The terminals created virtually under 1. can, of course, also be place in the schematic or booked out of the terminal management.

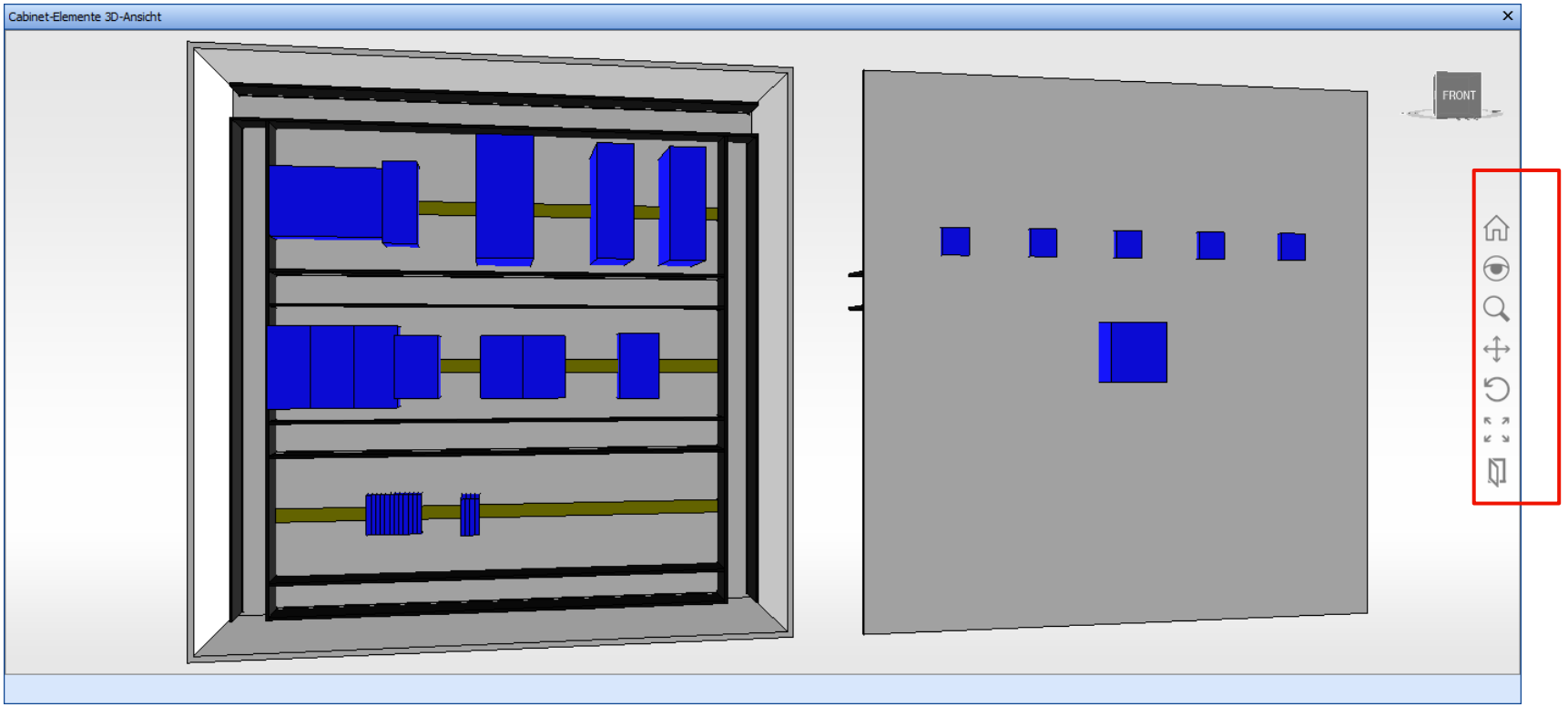
Cabinet – Placing additional components

- Placing components using the toolbar 
- This enables you to primarily place individual components in relation to the symbol type. With this command  , you can easily define or place standard symbols.
  1. Click on this command, then do not select any part and enter the dimensions for the new standard part (120/90/45).
  2. In the Properties dialog you can change the ref. name (-T4) and enter additional data.
  3. Place the component on a mounting rail.



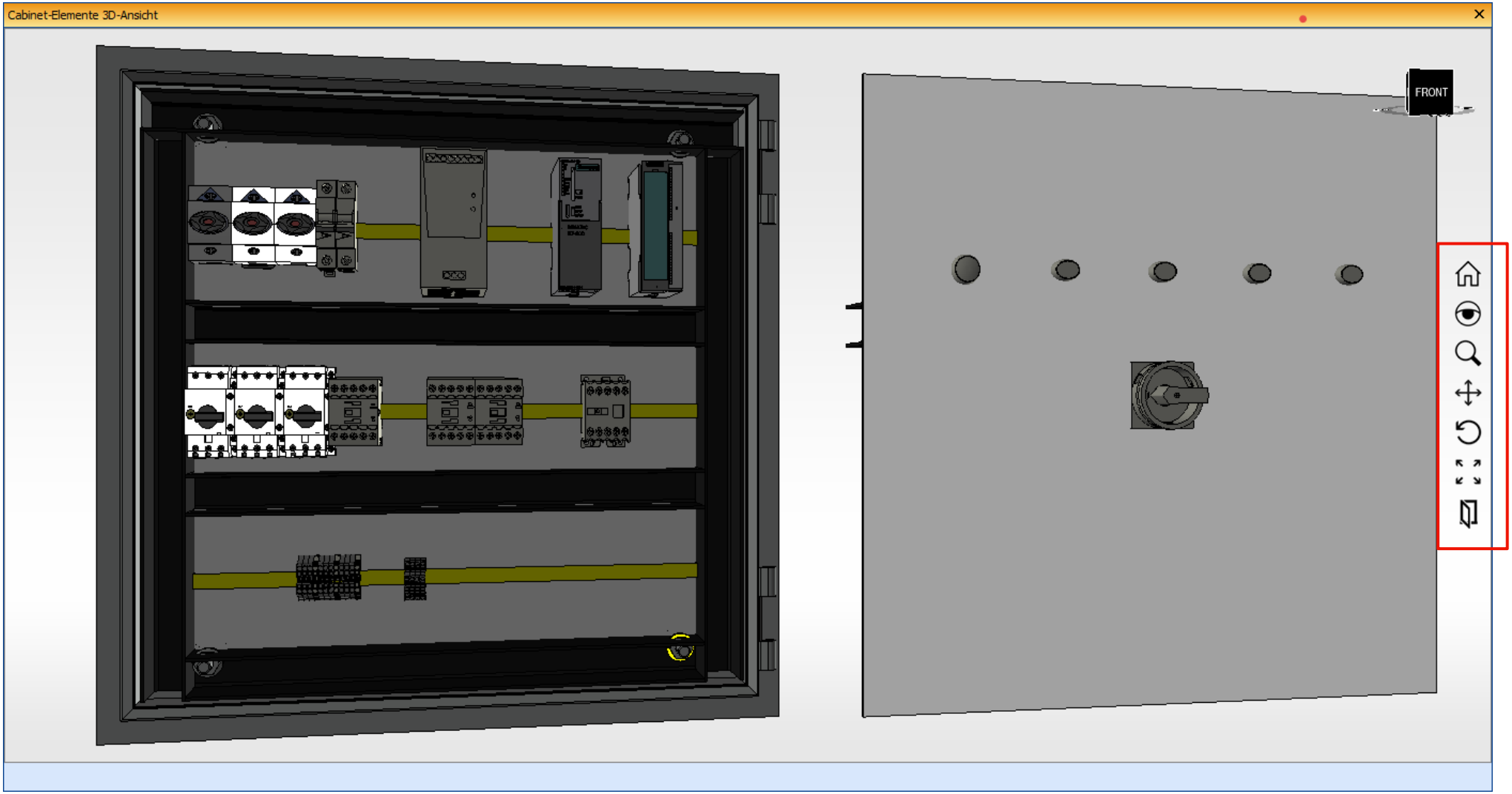
Cabinet – 3D-View (Expert License)

- Call of the 3D-View with CTRL-O will open a popup window
- Die 3D-view is used for:
  - Overwiew of reality
  - Collision check
- As a prerequisite you need the dimensoins for Z-axis of the symbols at the Article Database
- The system automatically calculates the Z-axis placement coordinates
- The view is controlled by the toolbar commands, and left mouse button



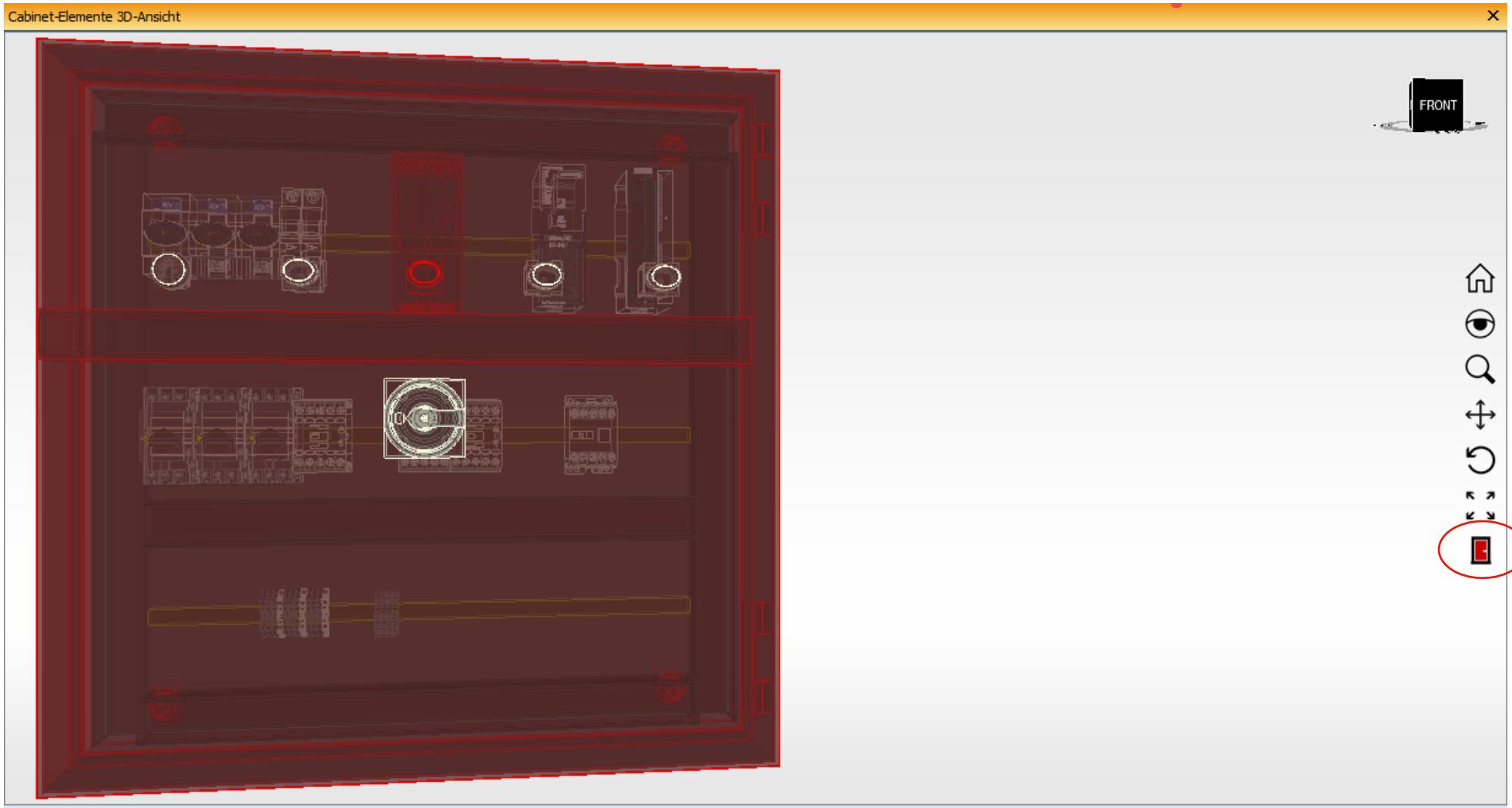
Cabinet – 3D-View (Expert License)

- 3D-Step data required



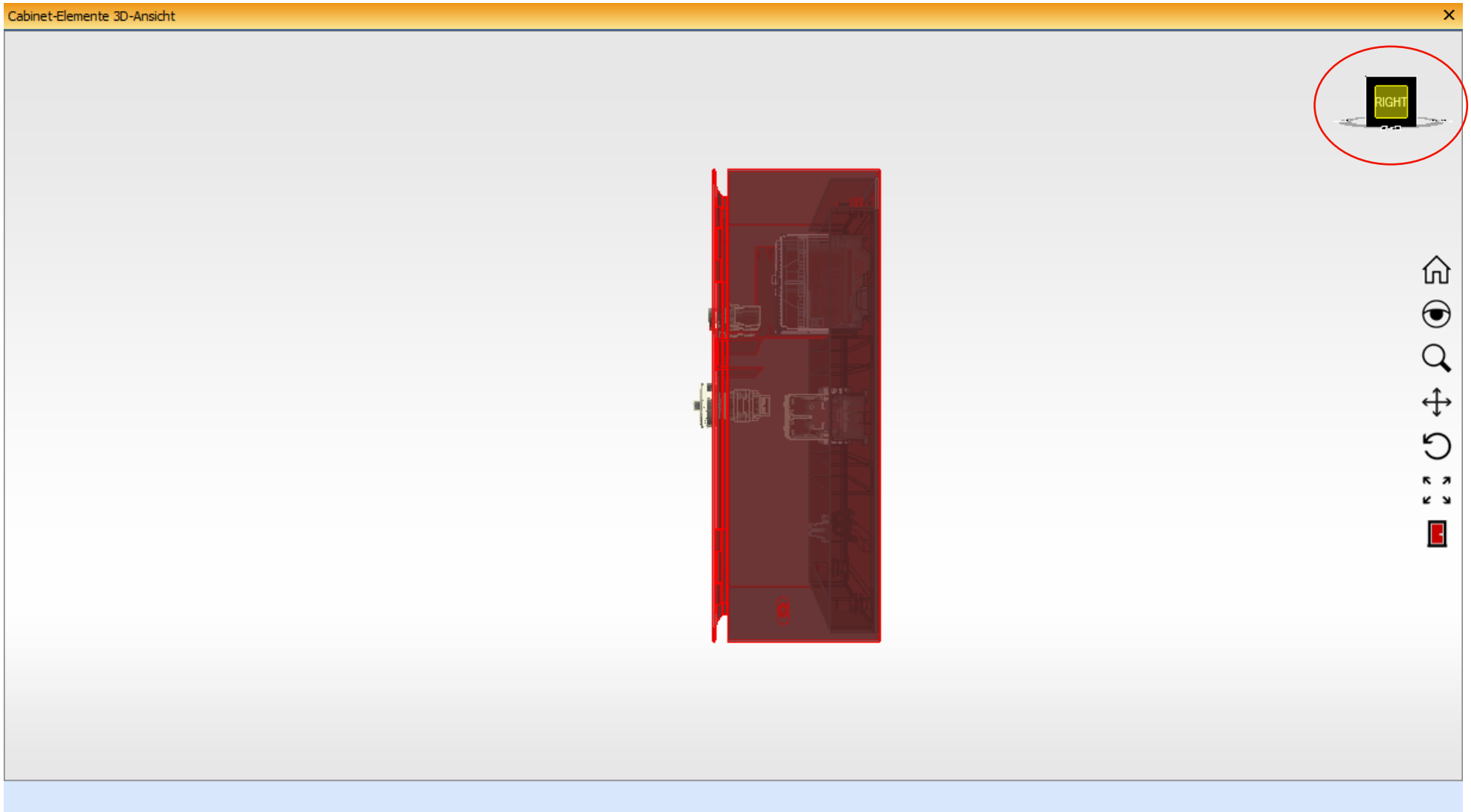
Cabinet – 3D-View (Expert License)

- Close the door to get a tramsparent view and to show collisions



Cabinet – 3D-View (Expert License)

- View from a side



WSCAD SUITE SUITE X

Training

Documentation

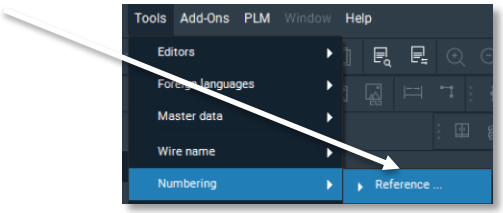
Documentation

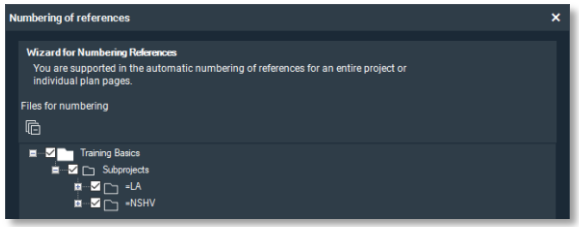
- Theoretical Section: **Documentation**
- Please devote your attention to the instructor. Thank you!

Numbering references

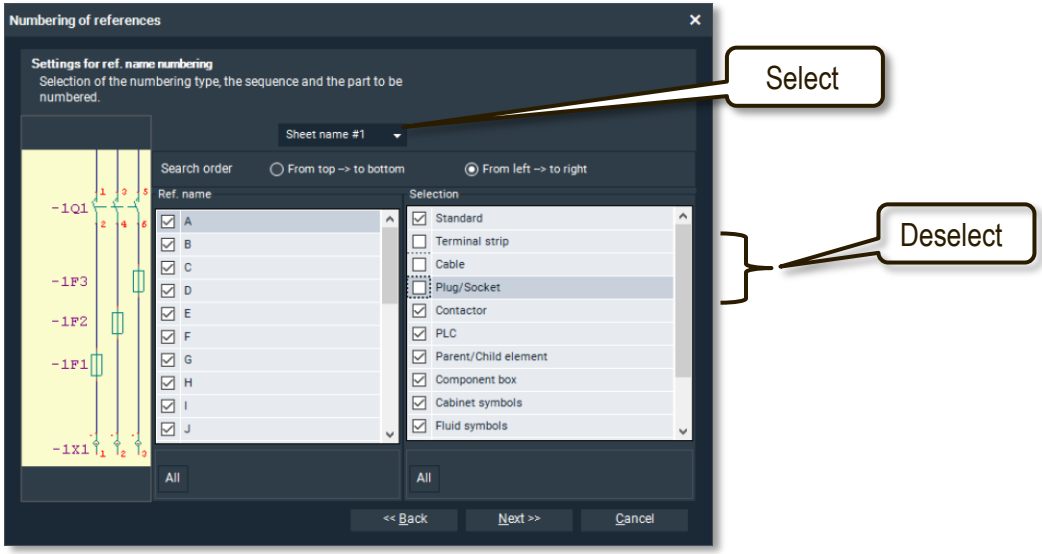
- Post-numbering of all components in the project
- Nomenclature of the components with sheet number - designation - count number (e.g., -10F2)

1. Go to the menu "Tools | Numbering | Reference".
2. Click "Yes" to create a project backup and confirm with "Done".
3. In the following wizard , select all sections in the project:



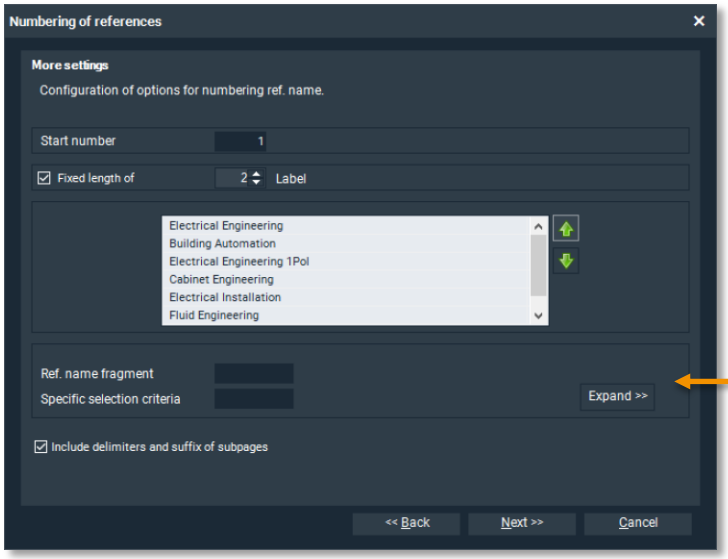


3. In the next step of the wizard, select the following options:

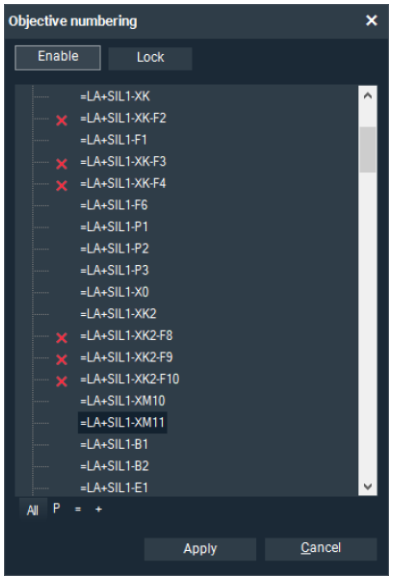


Numbering references

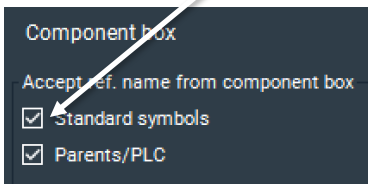
5. In the next step, you can define exceptions to the numbering. To do this, press "Expand".



6. Then select all components in the component box (that are not to be renumbered) and click on "Lock“ and "Apply“.



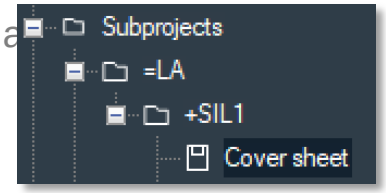
If these components are not to be shown as =LA-XK-..., then go to the settings and, under "Symbol Component box", clear the "Standard symbols" check box and confirm this,
and then enable and confirm it later.



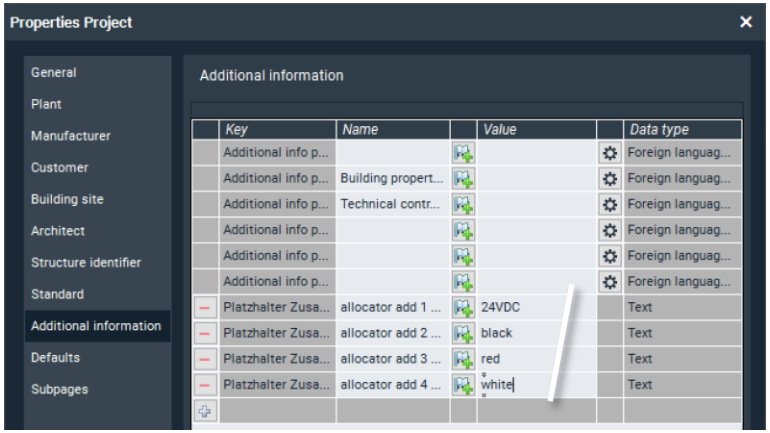
7. Click on "Next" and confirm in the nextstep
8. with "Finish" and then with "Done".
9. Spot-check the new numbering in the schematic and in the cabinet.
10. In sheet 4 in the schematic, the fuses are still mistakenly labeled with "-Q". change this manually to "-…F…".

Generate lists - Cover sheet

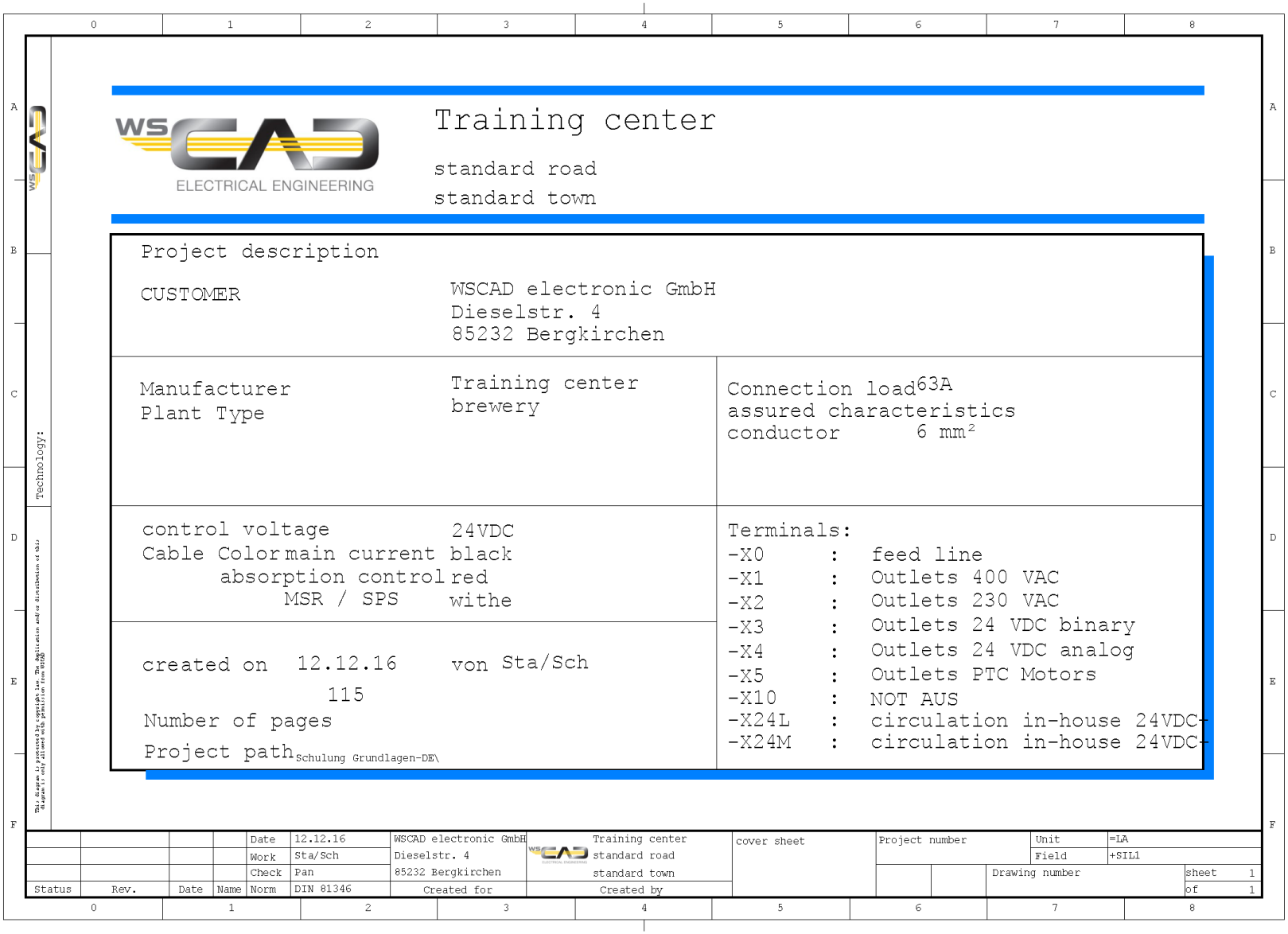
- The following lists are created for the section +SIL1
- Generate cover sheet
  1. Right-click in the Project Explorer on the document folder "Cover sheet" a
     select "Generate".
  2. Click "Finish" in step 2.
  3. View the cover sheet.



- Replace form for cover sheet
  1. Right-click on the document folder "Cover sheet" in the Project Explorer and now select "Cover sheet A4 12.0001" in the properties under Template. ...
  2. Generate the cover sheet as described above.
  3. Complete the following entries (these are immediately shown in the cover sheet):
     - Right-click on the project name / properties (at the very bottom):



Generate lists - Cover sheet



WSCAD ELECTRICAL ENGINEERING

Training center

standard road
standard town

Project description

CUSTOMER	WSCAD electronic GmbH Dieselstr. 4 85232 Bergkirchen	
Manufacturer Plant Type	Training center brewery	Connection load 63A assured characteristics conductor 6 mm²
control voltage Cable Color main current absorption control MSR / SPS	24VDC black red withe	Terminals: -X0 : feed line -X1 : Outlets 400 VAC -X2 : Outlets 230 VAC -X3 : Outlets 24 VDC binary -X4 : Outlets 24 VDC analog -X5 : Outlets PTC Motors -X10 : NOT AUS -X24L : circulation in-house 24VDC+ -X24M : circulation in-house 24VDC+
created on 12.12.16 115 Number of pages Project path Schulung Grundlagen-DE\	von Sta/Sch	

Technology:

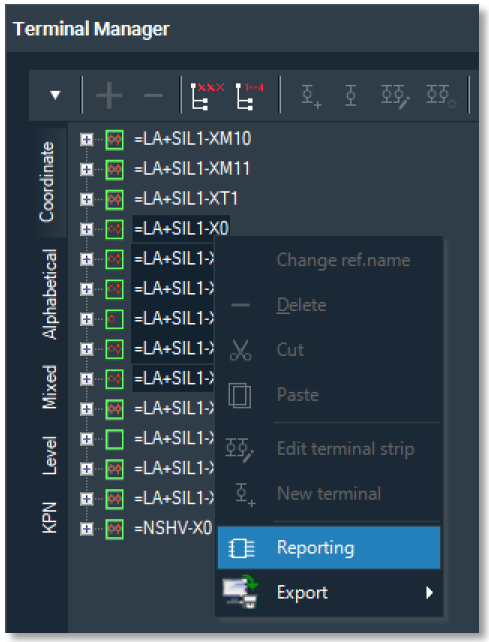
This diagram is protected by copyright law. The duplication and/or distribution of this diagram is only allowed with permission from WSCAD.

				Date	12.12.16	WSCAD electronic GmbH	Training center	cover sheet	Project number	Unit	=LA
				Work	Sta/Sch	Dieselstr. 4	standard road			Field	+SIL1
				Check	Pan	85232 Bergkirchen	standard town				
Status	Rev.	Date	Name	Norm	DIN 81346	Created for	Created by		Drawing number		sheet 1
											of 1

Exercise to generate cover sheet

Generate lists - Terminal chart

- Generate terminal chart
  1. Right-click on the "Terminal chart" document folder in the Project Explorer and select "Generate".
  2. Click "Finish" in step 2.
  3. The terminal charts of all terminal strips are generated.
  4. Replace the form "Terminal chart A3 01.0001" set under Template with "Terminal chart A4 04.0001" (the column for cable names is wider there).
- Generate terminal charts of selected terminal strips
  1. Open the Terminal Manager.
  2. Mark all terminal strips that you want to include in the terminal chart, then right-click on them and select "Report" and "Next".



After this, only the marked terminal charts will be generated as terminal charts.

Generate lists - Terminal chart

Terminal strip designation

=LA+SIL1-X3

Function text	=LA+SIL1-W8 JE-Y(ST)Y 2X2X0,8 QMM	=LA+SIL1-W7 JE-Y(ST)Y 2X2X0,8 QMM	=LA+SIL1-W4 JE-Y(ST)Y 2X2X0,8 QMM	=LA+SIL1-W3 JE-Y(ST)Y 2X2X0,8 QMM	=LA+SIL1-W2 JE-Y(ST)Y 2X2X0,8 QMM	=LA+SIL1-W1 JE-Y(ST)Y 2X2X0,8 QMM	Target designation external	Connection	Name	Jumpers	Target designation internal	Connection	Sheet/path
Silo 1 max top						BU	-5B1	1	1		-X24L	2	/5.1
Silo 1 max top						RD	-5B1	2	2		-5K1	A1	/5.1
Silo 1 max top						GY	-5B1	3	3				/5.1
Silo 1 max bottom					BU		-5B2	1	4		-5K2	A2	/5.3
Silo 1 max bottom					RD		-5B2	2	5		-5K2	A1	/5.3
Silo 1 max bottom					GY		-5B2	3	6				/5.3
Silo 1 top hatch open				BU			-5B3	1	7		-5K3	A2	/5.4
Silo 1 top hatch open				RD			-5B3	2	8		-5K3	A1	/5.5
Silo 1 top hatch open				GY			-5B3	3	9				/5.5
Silo 1 bottom hatch open			BU				-5B4	1	10		-5K4	A2	/5.6
Silo 1 bottom hatch open			RD				-5B4	2	11		-5K4	A1	/5.6
Silo 1 bottom hatch open			GY				-5B4	3	12				/5.6
		BU					-XBS1	1	13		-13K3	4	/7.1
		RD					-XBS1	2	14		-X4	2	/7.2
	BU						-XB2	1	15		-13K3	5	/7.3
	RD						-XB2	2	16		-8K1	A2	/7.3

Column headers: Cable des. / Cross-sec.

Technology:

Date	12.12.16	WSCAD electronic GmbH	Training center	terminal tie-up plan: =LA+SIL1-X3
Work	Sta/Sch	Dieselstr. 4	standard road	Project number 3
Check	Pan	85232 Bergkirchen	standard town	Unit =LA; Field +SIL1
Norm	DIN 81346	Created for	Created by	Drawing number; sheet 4 of 6

Status | Rev. | Date | Name

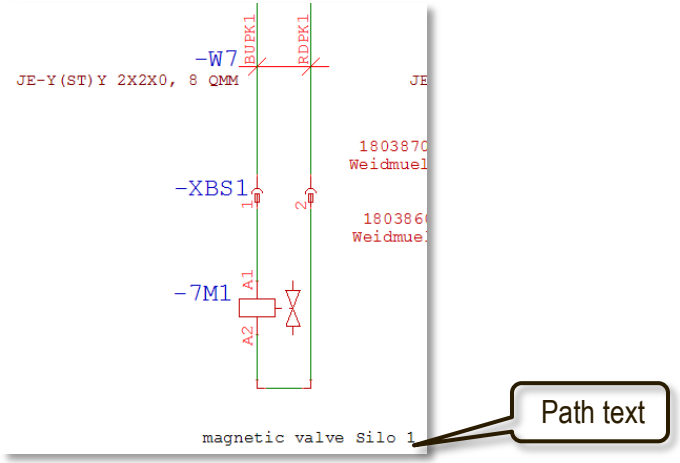
Exercise – Generate terminal chart

Generate lists - Cable list

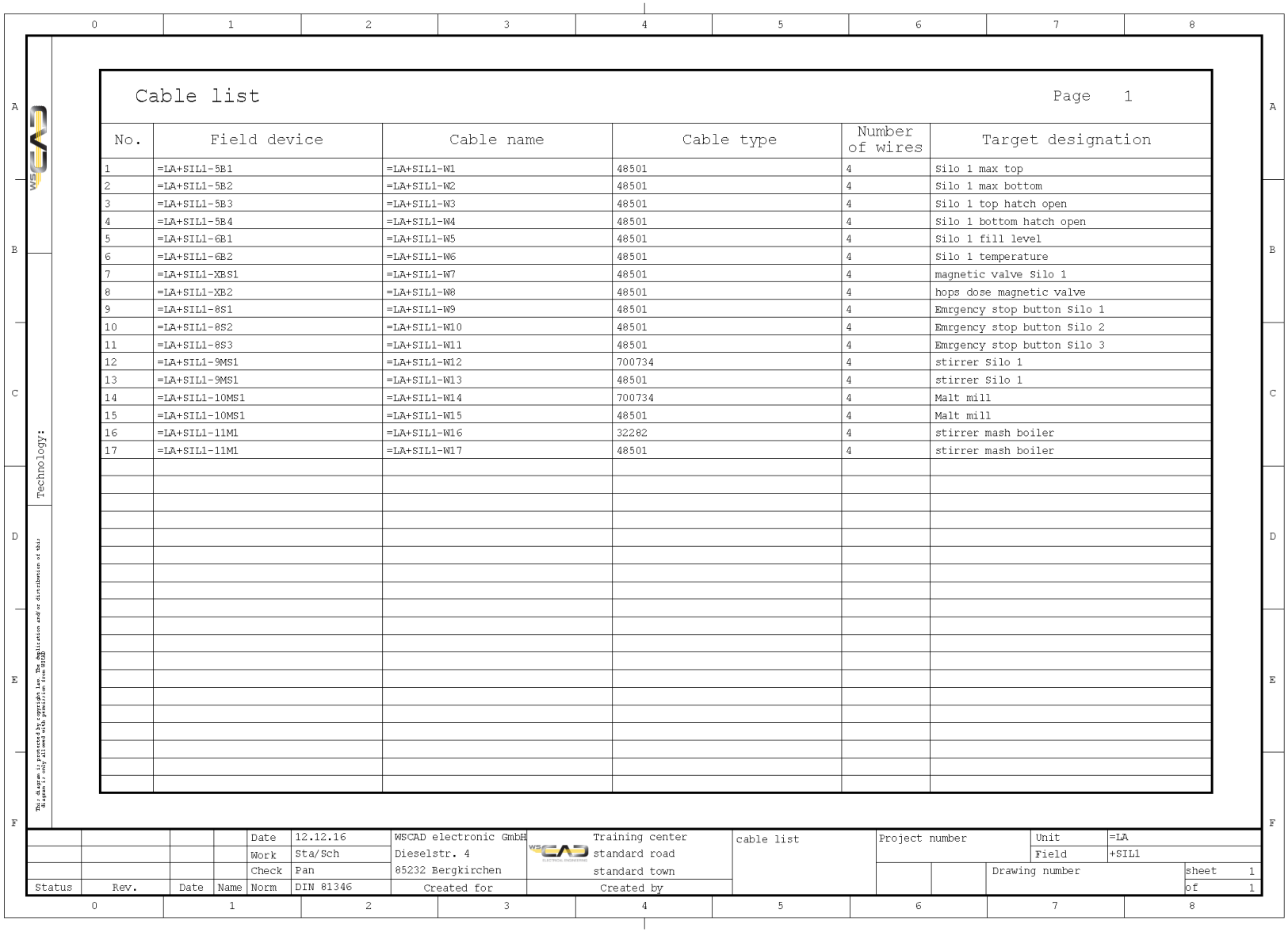
- Generate Cable List
  1. Right-click on the document folder "Cable list" in the Project Explorer.
  2. In step 2, set the sorting and click "Finish".
  3. The cable lists charts of all cables are generated.

  Tip: The "Target designation" column in the cable list includes a text allocator called "Function text device external". However, if there is a further device between external device and the cable as in sheet 7 (the solenoid valve), the function text is not taken over.
  Workaround: use the "path text" in such cases.
  4. Go to sheet 7 and then click on the command and enter "solenoid valve Silo 1" there, for example, and place this text below the valve.

When you now generate the cable list, this text will be transferred to the list.



Generate lists - Cable list



Cable list

Page 1

No.	Field device	Cable name	Cable type	Number of wires	Target designation
1	=LA+SIL1-5B1	=LA+SIL1-W1	48501	4	Silo 1 max top
2	=LA+SIL1-5B2	=LA+SIL1-W2	48501	4	Silo 1 max bottom
3	=LA+SIL1-5B3	=LA+SIL1-W3	48501	4	Silo 1 top hatch open
4	=LA+SIL1-5B4	=LA+SIL1-W4	48501	4	Silo 1 bottom hatch open
5	=LA+SIL1-6B1	=LA+SIL1-W5	48501	4	Silo 1 fill level
6	=LA+SIL1-6B2	=LA+SIL1-W6	48501	4	Silo 1 temperature
7	=LA+SIL1-XBS1	=LA+SIL1-W7	48501	4	magnetic valve Silo 1
8	=LA+SIL1-XB2	=LA+SIL1-W8	48501	4	hops dose magnetic valve
9	=LA+SIL1-8S1	=LA+SIL1-W9	48501	4	Emrgency stop button Silo 1
10	=LA+SIL1-8S2	=LA+SIL1-W10	48501	4	Emrgency stop button Silo 2
11	=LA+SIL1-8S3	=LA+SIL1-W11	48501	4	Emrgency stop button Silo 3
12	=LA+SIL1-9MS1	=LA+SIL1-W12	700734	4	stirrer Silo 1
13	=LA+SIL1-9MS1	=LA+SIL1-W13	48501	4	stirrer Silo 1
14	=LA+SIL1-10MS1	=LA+SIL1-W14	700734	4	Malt mill
15	=LA+SIL1-10MS1	=LA+SIL1-W15	48501	4	Malt mill
16	=LA+SIL1-11M1	=LA+SIL1-W16	32282	4	stirrer mash boiler
17	=LA+SIL1-11M1	=LA+SIL1-W17	48501	4	stirrer mash boiler

Technology:

Date 12.12.16
Work Sta/Sch
Check Pan
Norm DIN 81346

WSCAD electronic GmbH
Dieselstr. 4
85232 Bergkirchen
Created for

Training center
standard road
standard town
Created by

cable list

Project number

Unit =LA
Field +SIL1

Drawing number

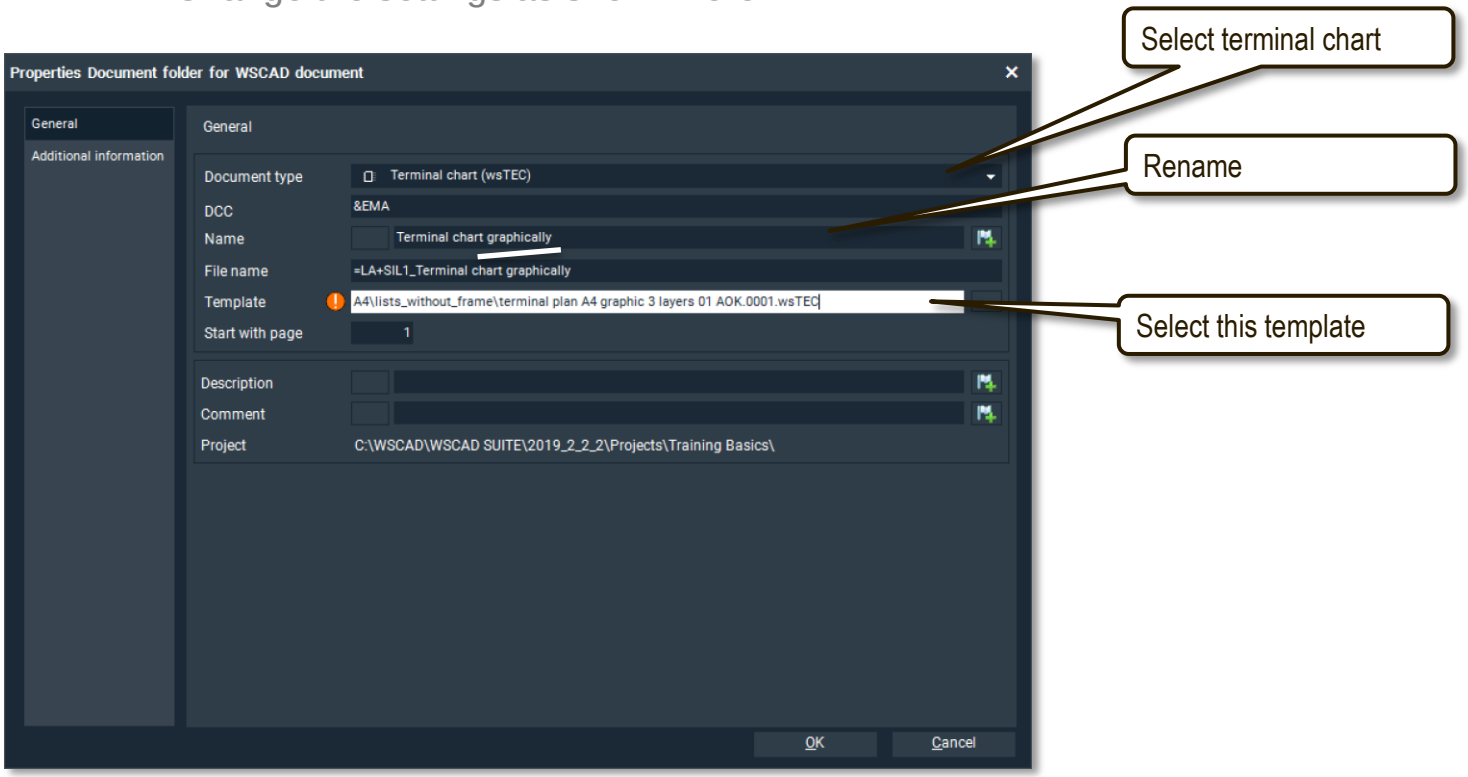
sheet 1
of 1

Status | Rev. | Date | Name

Generate lists – Graphical terminal chart

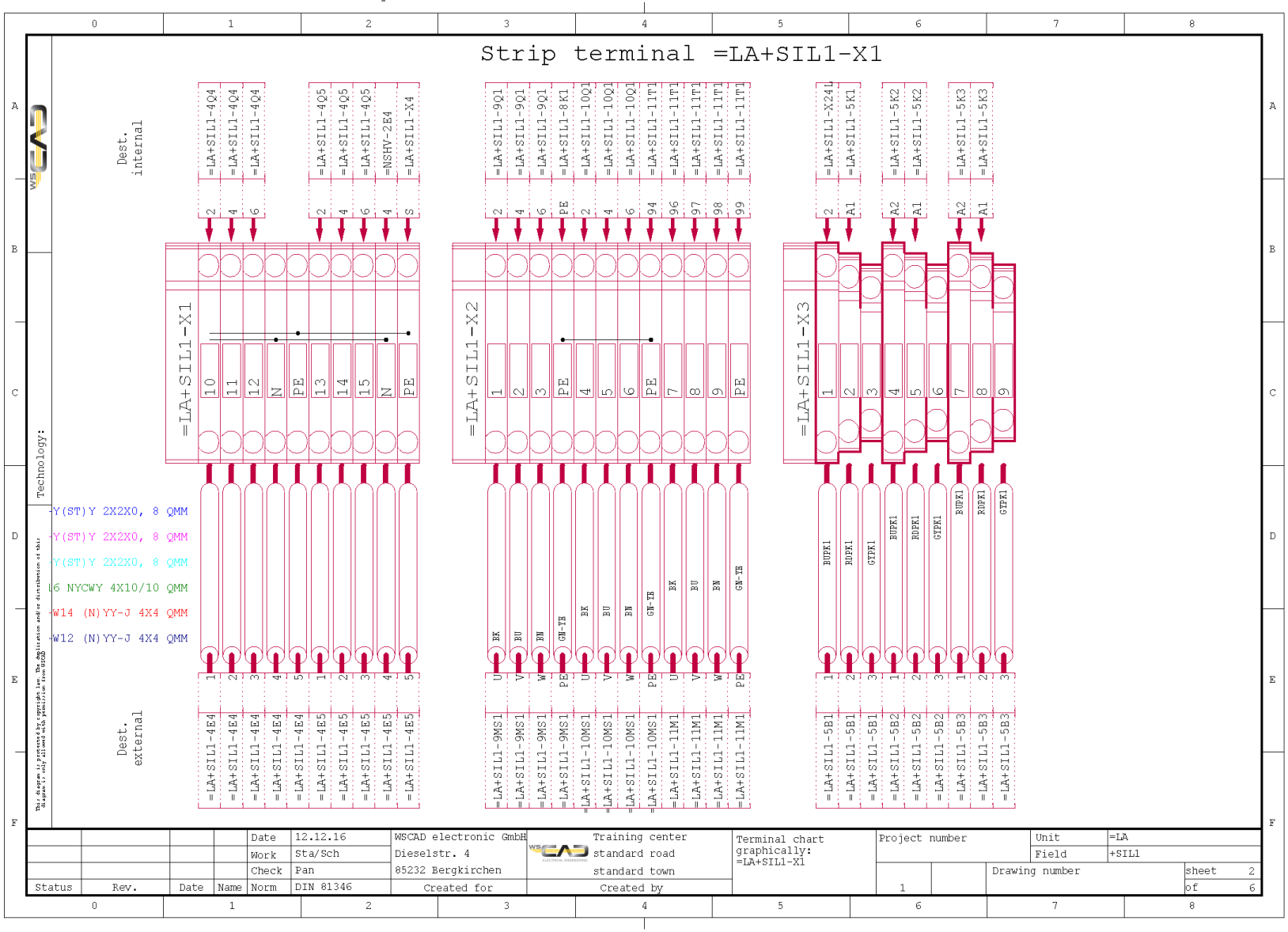
- Generate graphical terminal chart
  1. Create a new document folder in the +SIL1 section of the project.

  2. Change the settings as shown here:



  3. Generate the "Graphical terminal chart".

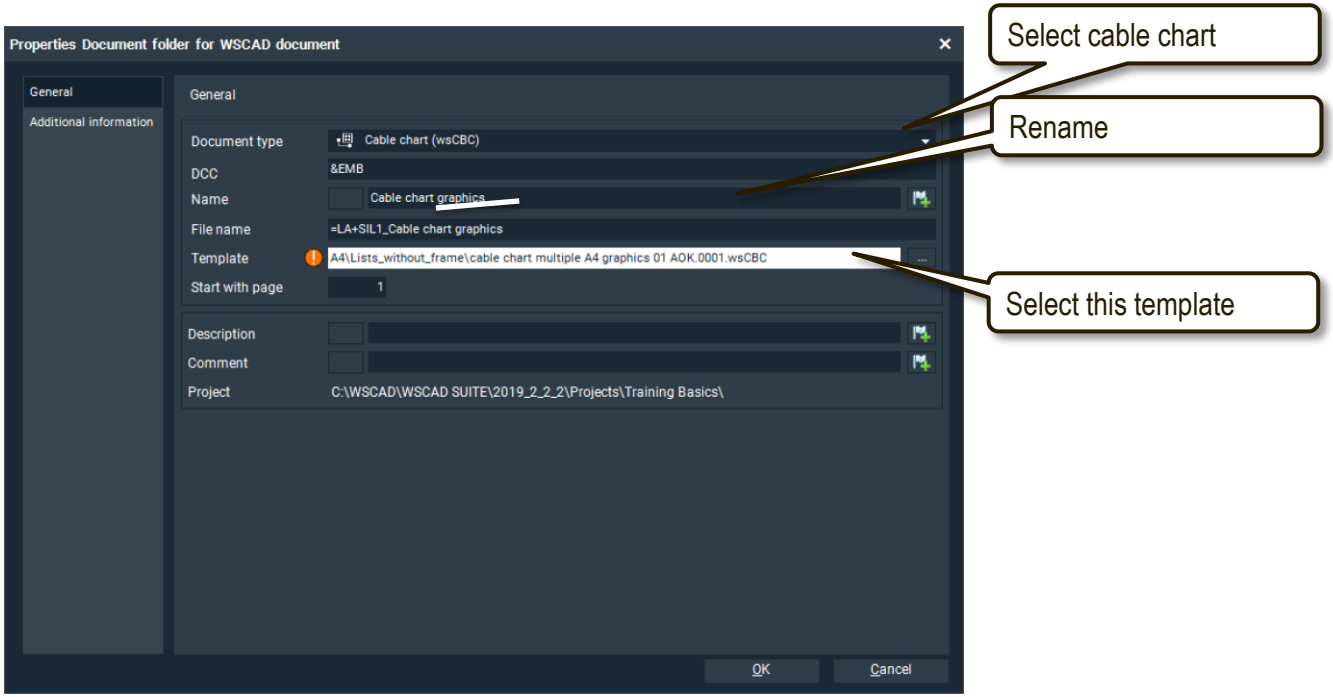
Generate lists – Graphical terminal chart



Generate lists - Graphical cable chart

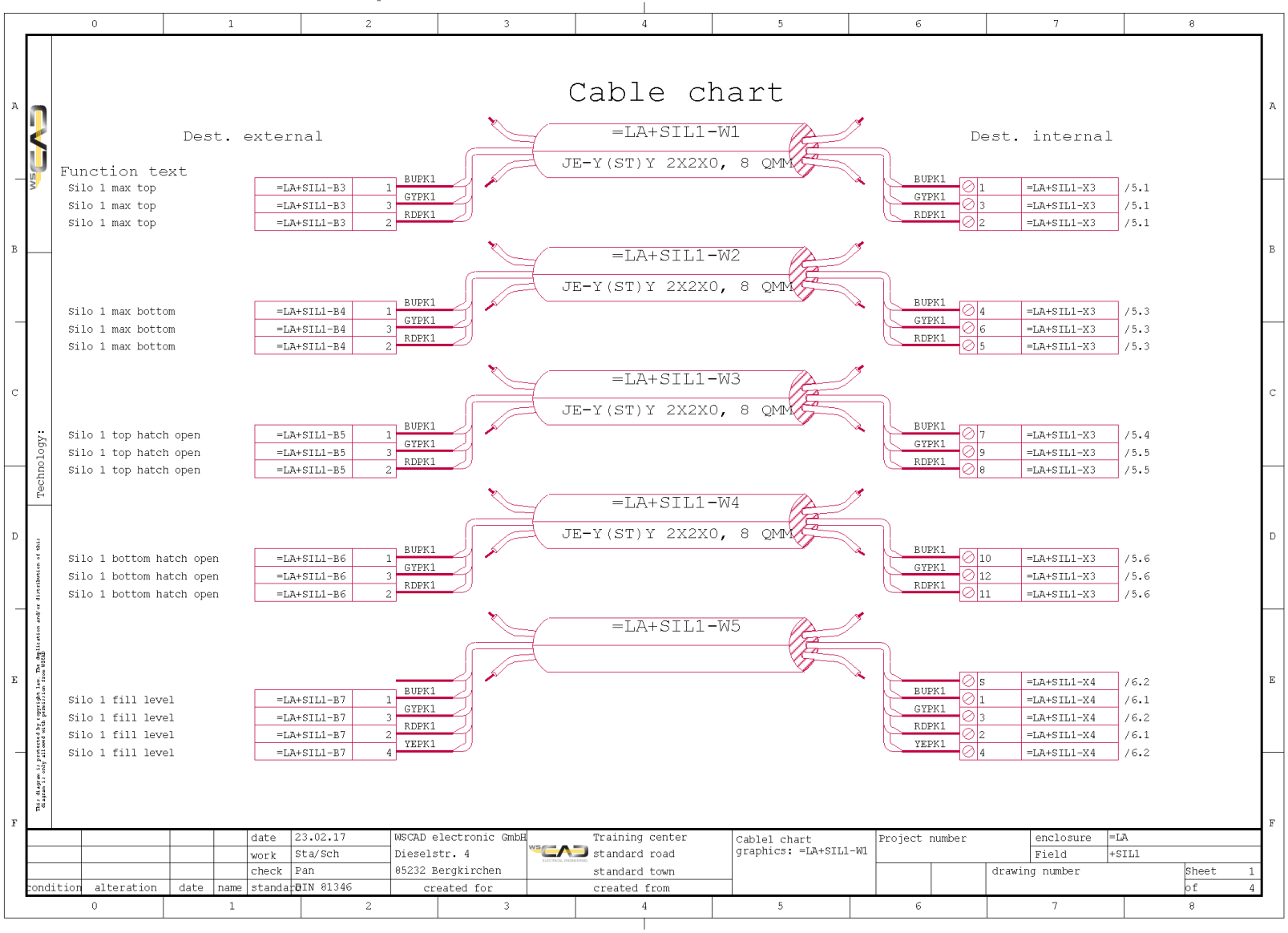
- Generate graphical cable chart
  1. Create a new document folder in the +SIL1 section of the project.

  2. Change the settings as shown here:



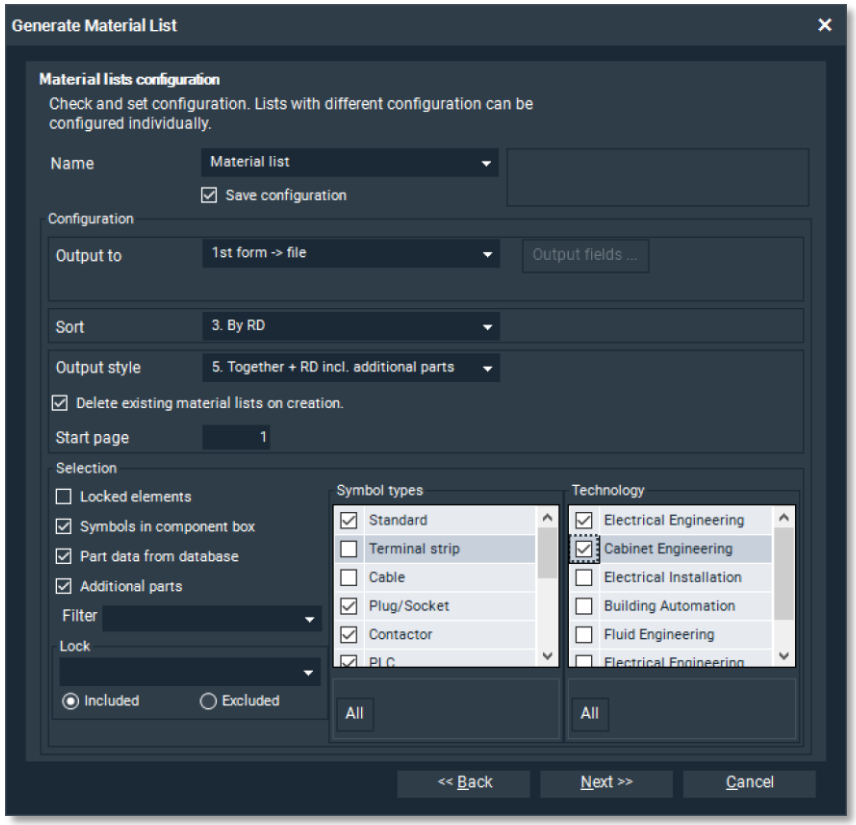
  3. Generate the "Graphical cable chart".

Generate lists - Graphical cable chart

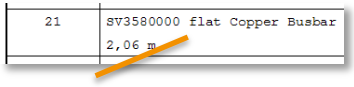


Generate lists - Material list

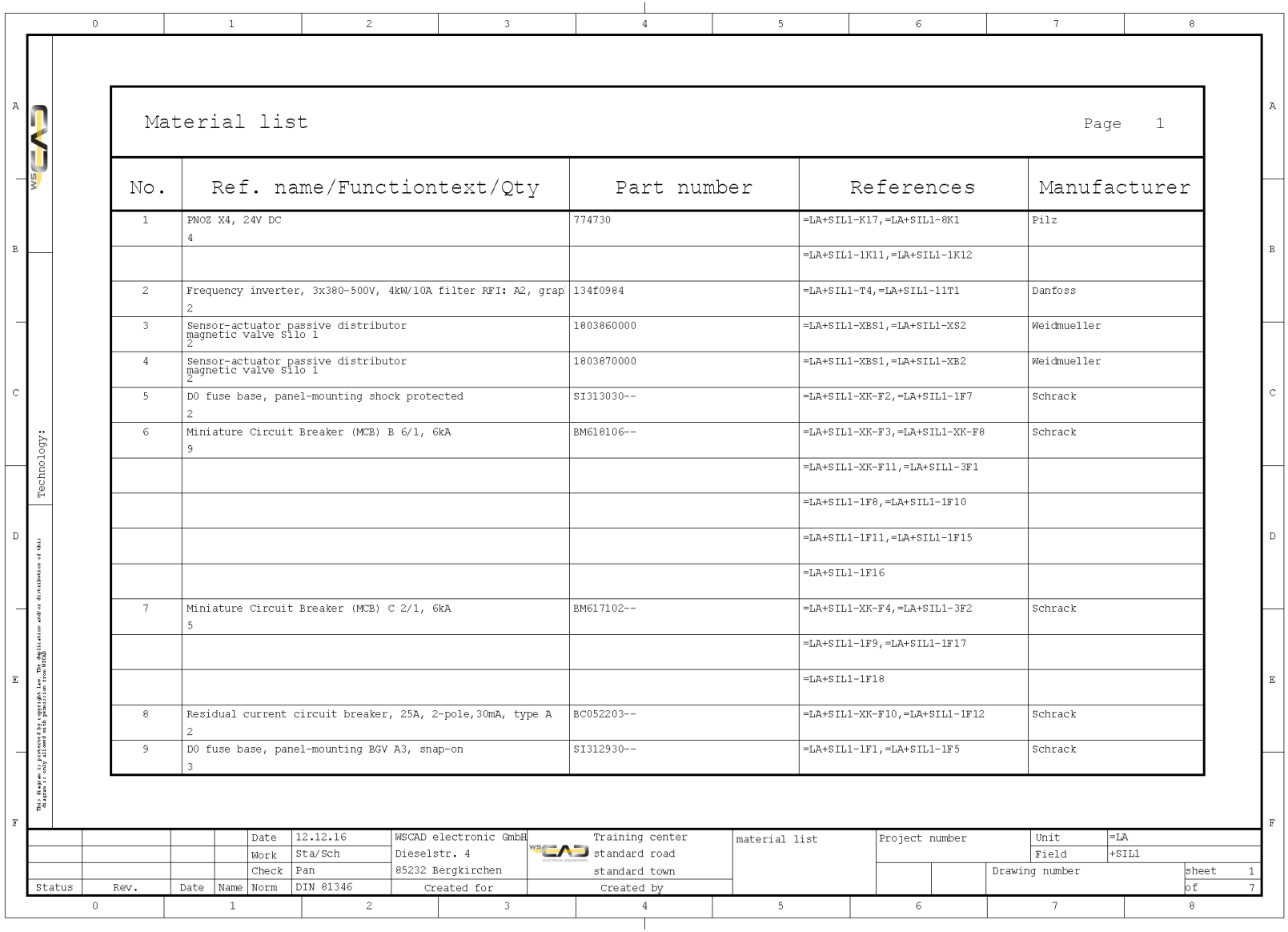
- Generate Material List
  1. Change the template for the material list to "Material list A4 02.0001".
  2. Generate the material list with the following settings:



  3. Check the result.
     It should not just list all the electrical components;
     even the total length of the copper busbar should have been determined.



Generate lists - Material list



Material list

Page 1

No.	Ref. name/Functiontext/Qty	Part number	References	Manufacturer
1	PNOZ X4, 24V DC 4	774730	=LA+SIL1-K17,=LA+SIL1-8K1	Pilz
			=LA+SIL1-1K11,=LA+SIL1-1K12	
2	Frequency inverter, 3x380-500V, 4kW/10A filter RFI: A2, grap 2	134f0984	=LA+SIL1-T4,=LA+SIL1-11T1	Danfoss
3	Sensor-actuator passive distributor magnetic valve silo 1 2	1803860000	=LA+SIL1-XBS1,=LA+SIL1-XS2	Weidmueller
4	Sensor-actuator passive distributor magnetic valve silo 1 2	1803870000	=LA+SIL1-XBS1,=LA+SIL1-XB2	Weidmueller
5	DO fuse base, panel-mounting shock protected 2	SI313030--	=LA+SIL1-XK-F2,=LA+SIL1-1F7	Schrack
6	Miniature Circuit Breaker (MCB) B 6/1, 6kA 9	BM618106--	=LA+SIL1-XK-F3,=LA+SIL1-XK-F8	Schrack
			=LA+SIL1-XK-F11,=LA+SIL1-3F1	
			=LA+SIL1-1F8,=LA+SIL1-1F10	
			=LA+SIL1-1F11,=LA+SIL1-1F15	
			=LA+SIL1-1F16	
7	Miniature Circuit Breaker (MCB) C 2/1, 6kA 5	BM617102--	=LA+SIL1-XK-F4,=LA+SIL1-3F2	Schrack
			=LA+SIL1-1F9,=LA+SIL1-1F17	
			=LA+SIL1-1F18	
8	Residual current circuit breaker, 25A, 2-pole,30mA, type A 2	BC052203--	=LA+SIL1-XK-F10,=LA+SIL1-1F12	Schrack
9	DO fuse base, panel-mounting BGV A3, snap-on 3	SI312930--	=LA+SIL1-1F1,=LA+SIL1-1F5	Schrack

Technology:

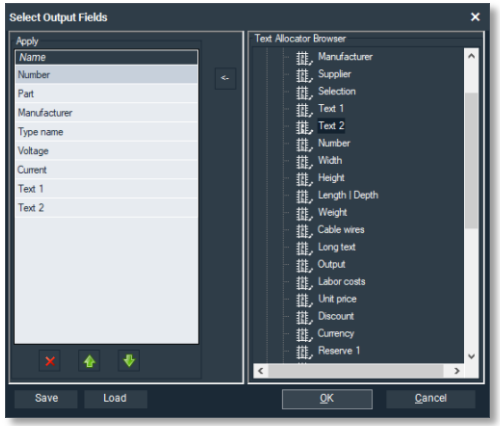
				Date	12.12.16	WSCAD electronic GmbH	Training center	material list	Project number	Unit	=LA
				Work	Sta/Sch	Dieselstr. 4	standard road			Field	+SIL1
				Check	Pan	85232 Bergkirchen	standard town				
Status	Rev.	Date	Name	Norm	DIN 81346	Created for	Created by			Drawing number	sheet 1 of 7

Generate lists - Material list

- Export material list
- The export to Excel, for example, allows you to transfer part data to other systems
  1. Go by right-clicking on the material list back into the configuration for the generation.
  2. Change "Output to" to Excel.
  3. Leave the other settings as is and click "Next".
  4. This opens an Excel list with the preset column attributes.

	A	B	C	D	E	F	G	H
1	idx	Symbol name	Number	Part	Ref name with stru	Manufacture	Function tex	Sequential n P
2	0	PNOZ X4, 24V DC	4	774730	=LA+SIL1-K17,=LA+	Pilz		1
3	1	Frequency inverter, 3x380-500V	2	134f0984	=LA+SIL1-T4,=LA+SI	Danfoss		2
4	2	Sensor-actuator passive distrib	2	1803860000	=LA+SIL1-XBS1,=LA	Weidmuelle	magnetic val	3
5	3	Sensor-actuator passive distrib	2	1803870000	=LA+SIL1-XBS1,=LA	Weidmuelle	magnetic val	4
6	4	D0 fuse base, panel-mounting	2	SI313030--	=LA+SIL1-XK-F2,=LA	Schrack		5
7	5	Miniature Circuit Breaker (MCB	9	BM618106--	=LA+SIL1-XK-F3,=LA	Schrack		6
8	6	Miniature Circuit Breaker (MCB	5	BM617102--	=LA+SIL1-XK-F4,=LA	Schrack		7
9	7	Residual current circuit breaker	2	BC052203--	=LA+SIL1-XK-F10,=L	Schrack		8

5. If you want to generate your own column attributes, you must set the output fields in the generation window of the material list.



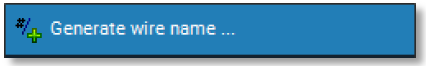
You do this by moving the attributes from left to right.

Tip: Relevant attributes can be found in the "Material list" and Material DB fields text allocator" folders.

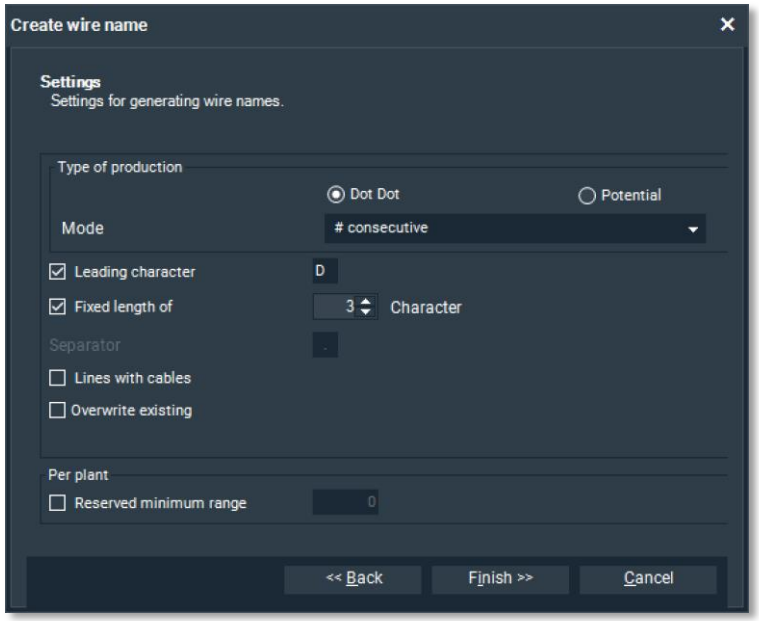
In the latter case, also under "Any custom field"

Generate lists - Wiring chart

- Generate wiring chart
- Before the generation, as much data (connection attributes) as possible should be contained in the individual connections (wires).

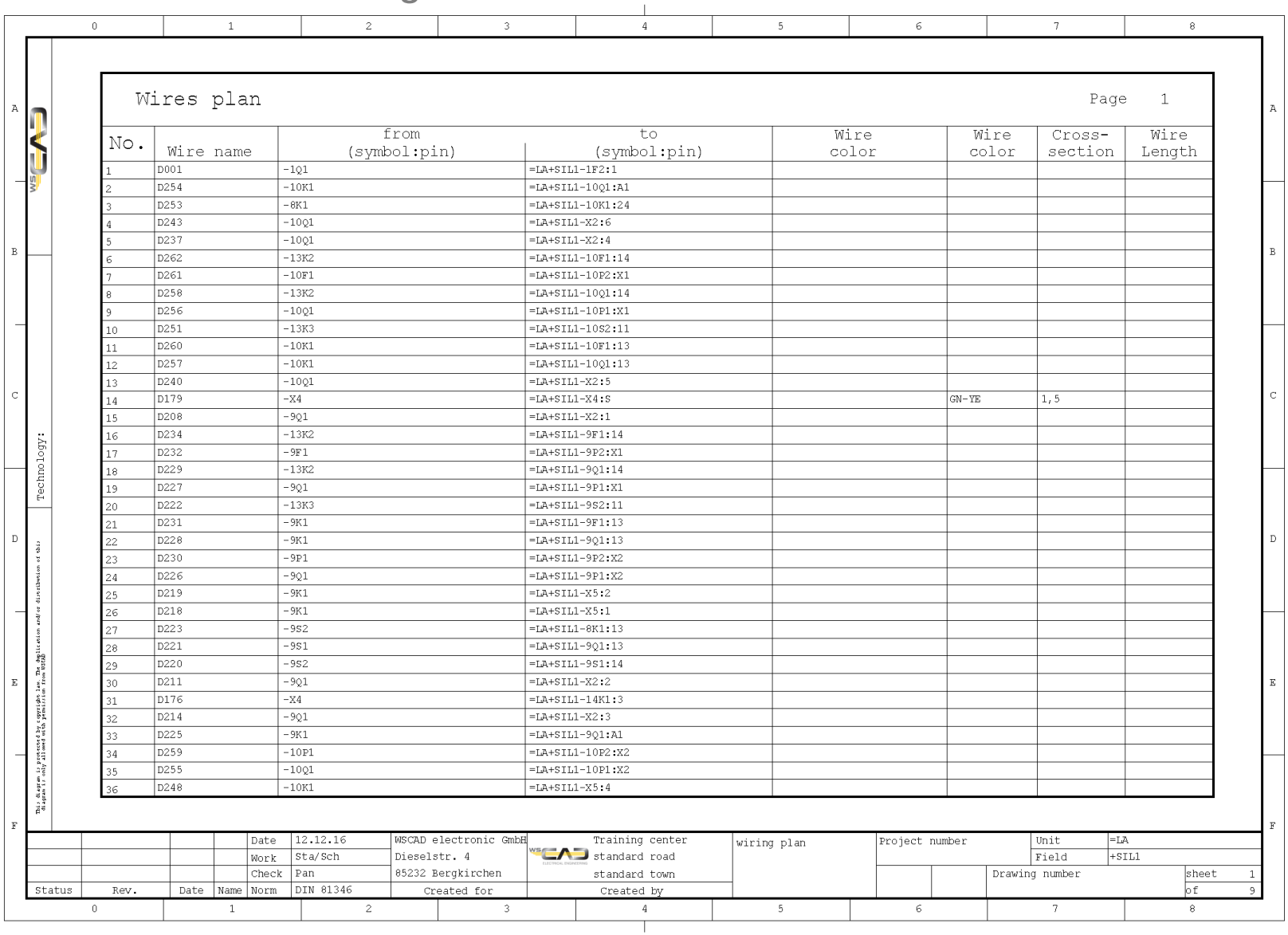


1. Go via the "Tools | Wire name" menu to
2. In the first frame, select the section +SIL1.
3. In the second frame, select the following settings and click "Finish".



4. Change the template for the wiring chart to "Wiring chart A4 03.0001".
5. Generate the wiring chart with the sorting set to "By wire name".

Generate lists - Wiring chart



Wires plan

Page 1

No.	Wire name	from (symbol:pin)	to (symbol:pin)	Wire color	Wire color	Cross-section	Wire Length
1	D001	-1Q1	=LA+SIL1-1F2:1				
2	D254	-10K1	=LA+SIL1-10Q1:A1				
3	D253	-8K1	=LA+SIL1-10K1:24				
4	D243	-10Q1	=LA+SIL1-X2:6				
5	D237	-10Q1	=LA+SIL1-X2:4				
6	D262	-13K2	=LA+SIL1-10F1:14				
7	D261	-10F1	=LA+SIL1-10P2:X1				
8	D258	-13K2	=LA+SIL1-10Q1:14				
9	D256	-10Q1	=LA+SIL1-10P1:X1				
10	D251	-13K3	=LA+SIL1-10S2:11				
11	D260	-10K1	=LA+SIL1-10F1:13				
12	D257	-10K1	=LA+SIL1-10Q1:13				
13	D240	-10Q1	=LA+SIL1-X2:5				
14	D179	-X4	=LA+SIL1-X4:S		GN-YE	1,5	
15	D208	-9Q1	=LA+SIL1-X2:1				
16	D234	-13K2	=LA+SIL1-9F1:14				
17	D232	-9F1	=LA+SIL1-9P2:X1				
18	D229	-13K2	=LA+SIL1-9Q1:14				
19	D227	-9Q1	=LA+SIL1-9P1:X1				
20	D222	-13K3	=LA+SIL1-9S2:11				
21	D231	-9K1	=LA+SIL1-9F1:13				
22	D228	-9K1	=LA+SIL1-9Q1:13				
23	D230	-9P1	=LA+SIL1-9P2:X2				
24	D226	-9Q1	=LA+SIL1-9P1:X2				
25	D219	-9K1	=LA+SIL1-X5:2				
26	D218	-9K1	=LA+SIL1-X5:1				
27	D223	-9S2	=LA+SIL1-8K1:13				
28	D221	-9S1	=LA+SIL1-9Q1:13				
29	D220	-9S2	=LA+SIL1-9S1:14				
30	D211	-9Q1	=LA+SIL1-X2:2				
31	D176	-X4	=LA+SIL1-14K1:3				
32	D214	-9Q1	=LA+SIL1-X2:3				
33	D225	-9K1	=LA+SIL1-9Q1:A1				
34	D259	-10P1	=LA+SIL1-10P2:X2				
35	D255	-10Q1	=LA+SIL1-10P1:X2				
36	D248	-10K1	=LA+SIL1-X5:4				

Technology:

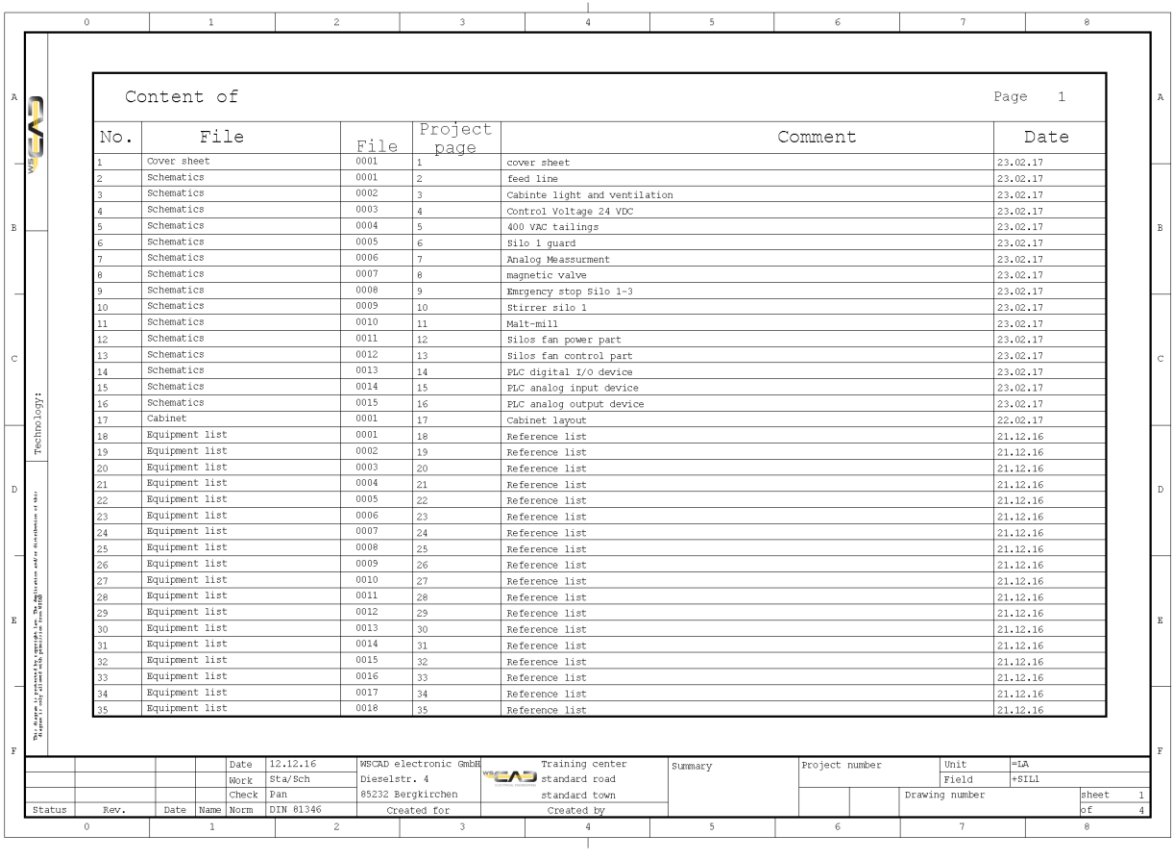
The drawing is protected by copyright law. The publication and/or distribution of this drawing is only allowed with permission from WSCAD.

				Date	12.12.16	WSCAD electronic GmbH	Training center	wiring plan	Project number	Unit	=LA
				Work	Sta/Sch	Dieselstr. 4	standard road			Field	+SIL1
				Check	Pan	85232 Bergkirchen	standard town				
Status	Rev.	Date	Name	Norm	DIN 81346	Created for	Created by			Drawing number	sheet 1 of 9

Exercise – Generate wiring chart

Generate lists - Summary

- Generate the table of contents
- The table of contents should only be created on completing of documentation.
  1. Go by right clicking to the summary and generate this for the current level in the project.
  2. Change the template for the summary to "Summary A4 04.0001"
     and generate this again.



Content of Page 1

No.	File	File	Project page	Comment	Date
1	Cover sheet	0001	1	cover sheet	23.02.17
2	Schematics	0001	2	feed line	23.02.17
3	Schematics	0002	3	Cabinte light and ventilation	23.02.17
4	Schematics	0003	4	Control Voltage 24 VDC	23.02.17
5	Schematics	0004	5	400 VAC tailings	23.02.17
6	Schematics	0005	6	Silo 1 guard	23.02.17
7	Schematics	0006	7	Analog Meassurment	23.02.17
8	Schematics	0007	8	magnetic valve	23.02.17
9	Schematics	0008	9	Emrgency stop Silo 1-3	23.02.17
10	Schematics	0009	10	Stirrer silo 1	23.02.17
11	Schematics	0010	11	Malt-mill	23.02.17
12	Schematics	0011	12	Silos fan power part	23.02.17
13	Schematics	0012	13	Silos fan control part	23.02.17
14	Schematics	0013	14	PLC digital I/O device	23.02.17
15	Schematics	0014	15	PLC analog input device	23.02.17
16	Schematics	0015	16	PLC analog output device	23.02.17
17	Cabinet	0001	17	Cabinet layout	22.02.17
18	Equipment list	0001	18	Reference list	21.12.16
19	Equipment list	0002	19	Reference list	21.12.16
20	Equipment list	0003	20	Reference list	21.12.16
21	Equipment list	0004	21	Reference list	21.12.16
22	Equipment list	0005	22	Reference list	21.12.16
23	Equipment list	0006	23	Reference list	21.12.16
24	Equipment list	0007	24	Reference list	21.12.16
25	Equipment list	0008	25	Reference list	21.12.16
26	Equipment list	0009	26	Reference list	21.12.16
27	Equipment list	0010	27	Reference list	21.12.16
28	Equipment list	0011	28	Reference list	21.12.16
29	Equipment list	0012	29	Reference list	21.12.16
30	Equipment list	0013	30	Reference list	21.12.16
31	Equipment list	0014	31	Reference list	21.12.16
32	Equipment list	0015	32	Reference list	21.12.16
33	Equipment list	0016	33	Reference list	21.12.16
34	Equipment list	0017	34	Reference list	21.12.16
35	Equipment list	0018	35	Reference list	21.12.16

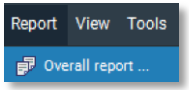
Technology:

Date 12.12.16 | Work Sta/Sch | Check Pan | Norm DIN 81346
WSCAD electronic GmbH, Dieselstr. 4, 85232 Bergkirchen
Training center, standard road, standard town
Summary
Project number
Unit =LA | Field +SIL1
Drawing number
sheet 1 of 4
Status | Rev. | Date | Name | Created for | Created by

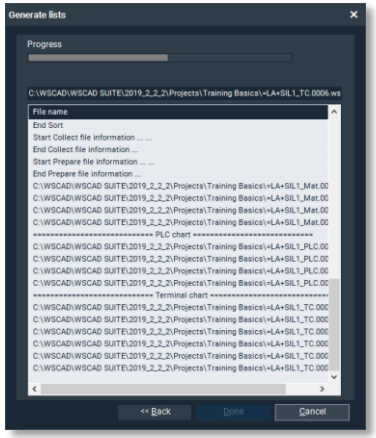
Generate lists - Overall report

- Generate lists via overall report
- Quick generation of all desired lists in one generation run
- Prerequisite: you have already set the configuration dialogs as indicated above or preconfigured them in the settings.

1. Go to the menu "Report | Overall report":



2. In step 1, select only the section +SIL1 (without measuring point list, if displayed).

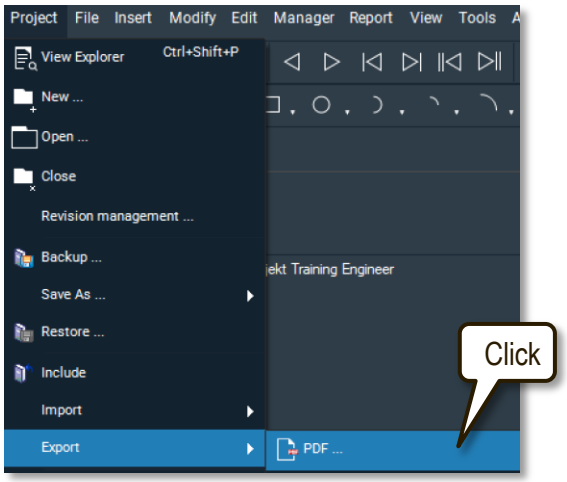


3. On clicking "Finish", the full generation run for the complete documentation occurs.

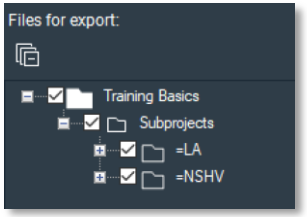
PDF output

- Project export as PDF document
- The result is an "intelligent" PDF file.
- This means that the project structure of WSCAD is mapped in the PDF file.
- For example, all cross-references in the schematics are referencing links.
- And with "mouseover", all entered symbol texts become visible (can be switched on/off).

1. Make sure your "Training Basics" project is open.
2. Go to the menu "Project | Export | PDF ":

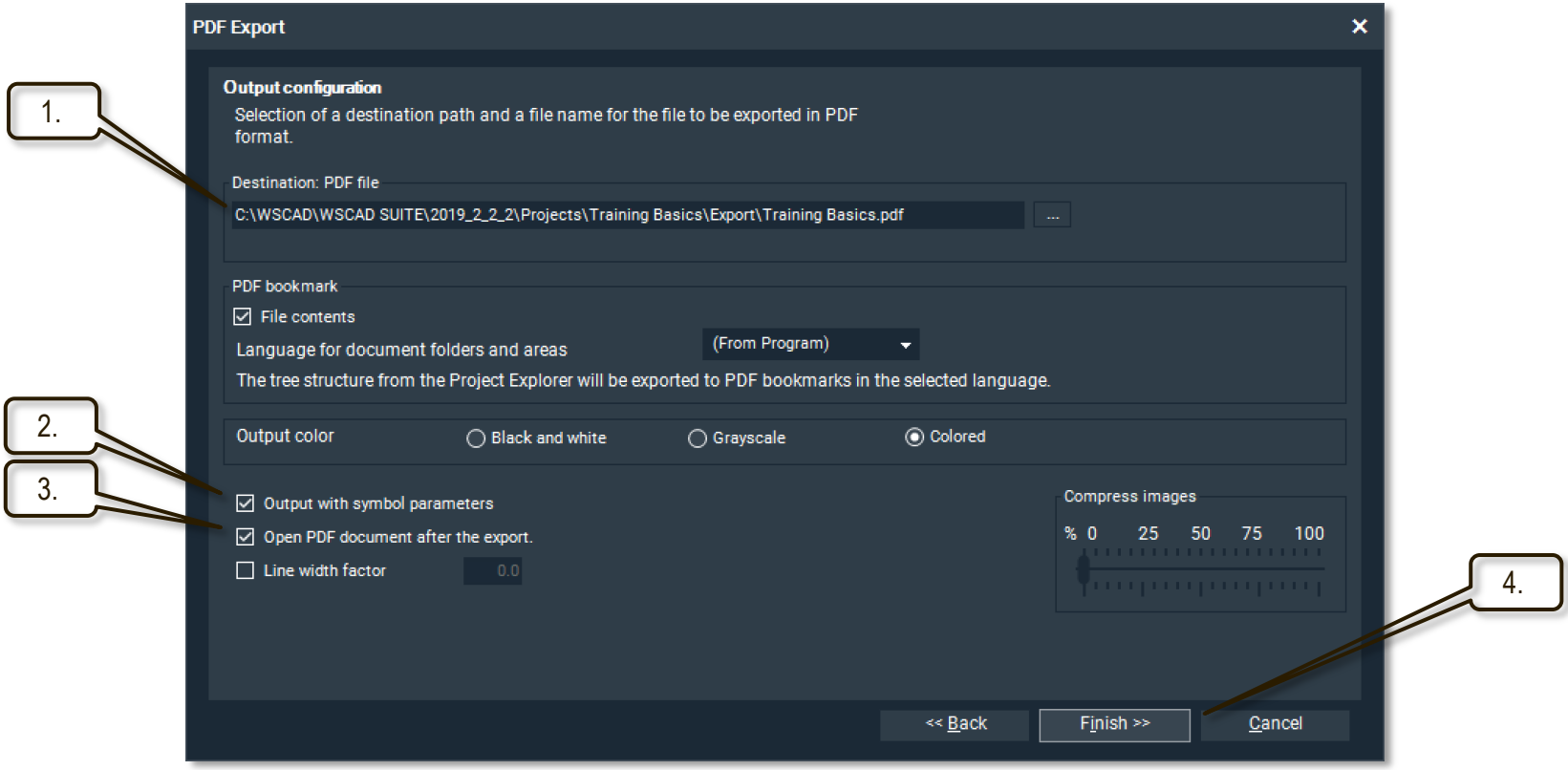


3. In the following dialog, "Step 1 of 3", define the scope of the output and go to "Next".

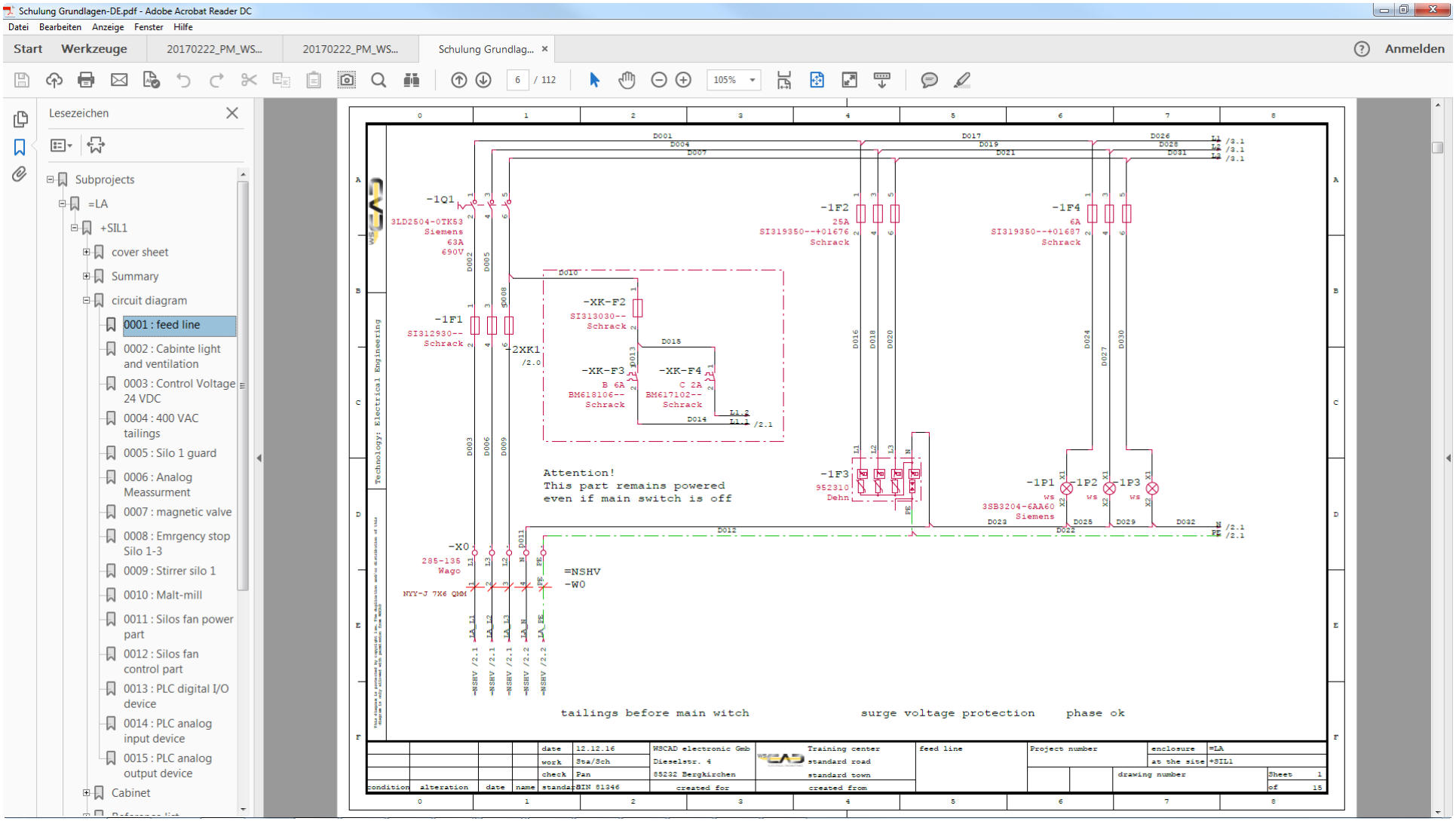


PDF output

- Project Export Dialog, "Step 2 of 3"
  1. Check here where the PDF is to be saved.
  2. Click to select if symbol texts are to be displayed in the PDF.
  3. Click to select if the PDF is to be opened immediately after exporting.
  4. Enter the value recommended here.
  5. "Finish". Then click "Done".



PDF output


Schulung Grundlagen-DE.pdf - Adobe Acrobat Reader DC
Datei Bearbeiten Anzeige Fenster Hilfe
Start Werkzeuge
Lesezeichen
Subprojects
=LA
+SIL1
cover sheet
Summary
circuit diagram
0001 : feed line
0002 : Cabinte light and ventilation
0003 : Control Voltage 24 VDC
0004 : 400 VAC tailings
0005 : Silo 1 guard
0006 : Analog Meassurment
0007 : magnetic valve
0008 : Emrgency stop Silo 1-3
0009 : Stirrer silo 1
0010 : Malt-mill
0011 : Silos fan power part
0012 : Silos fan control part
0013 : PLC digital I/O device
0014 : PLC analog input device
0015 : PLC analog output device
Cabinet
-1Q1
3LD2504-0TK53
Siemens
63A
690V
-1F1
SI312930--
Schrack
-2XK1
-XK-F2
SI313030--
Schrack
-XK-F3
B 6A
BM618106--
Schrack
-XK-F4
C 2A
BM617102--
Schrack
-1F2
25A
SI319350--+01676
Schrack
-1F4
6A
SI319350--+01687
Schrack
-1F3
952310
Dehn
-1P1
-1P2
-1P3
3SB3204-6AA60
Siemens
-X0
285-135
Wago
NYY-J 7X6 QM#
=NSHV
-W0
Attention!
This part remains powered
even if main switch is off
tailings before main witch
surge voltage protection
phase ok
Technology: Electrical Engineering
WSCAD electronic Gmb
Dieselstr. 4
85232 Bergkirchen
Training center
standard road
standard town
feed line
Project number
enclosure
=LA
at the site
+SIL1
drawing number
Sheet 1
of 15

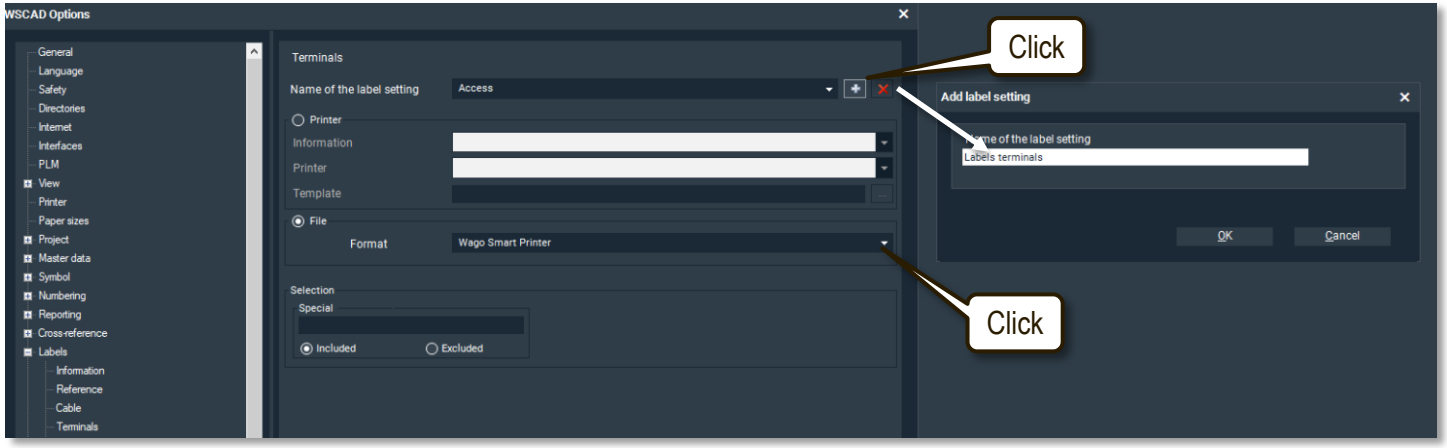
Labels - Output

- The output of labels can be achieved in three ways:
  - 1. with a special label printer
  - 2. with label sheets on a normal printer
  - 3. as a direct transfer to a labeling system

- Label printer or direct output

If such a printer exists, the output occurs as follows:

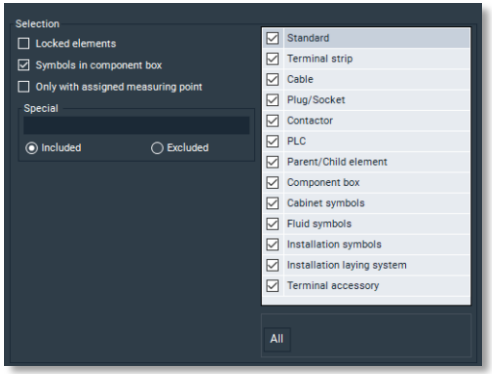
- Open the settings in WSCAD and go to Labels and then select *Reference* or *Cable* or*Terminals* or *PLC* or *Measuring points.*
- In each respective dialog, enter a new profile name for the output job and choose a format for the file format.



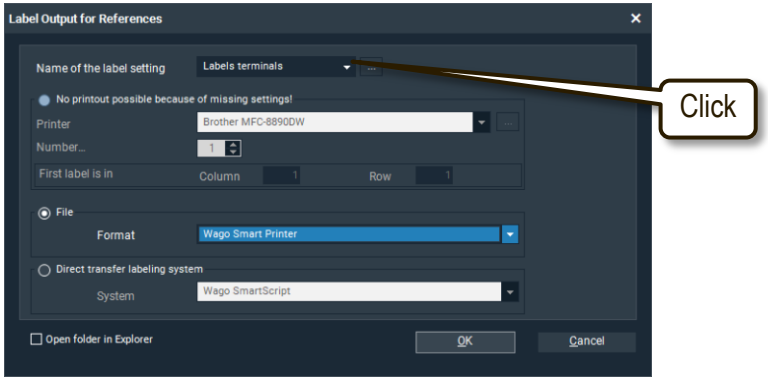
Labels - Output

- Label printer or direct output

- In the Reference category, you can also make additional selections



  - Confirm the settings with OK“.
- Go to the menu "Report | Labels" and then to Terminals, for example.
- Select the created profile.



- You can then produce a file that you can import into the software of the printer to print labels.

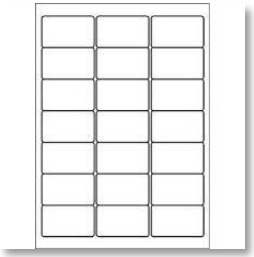
Labels - Output

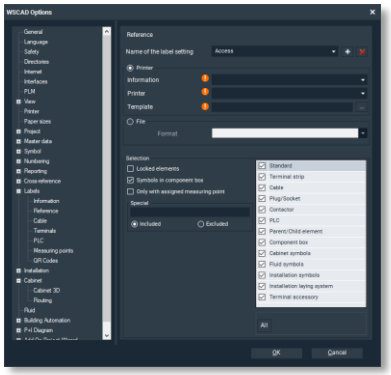
- With label sheets on a normal printer
- Principle of this label output:

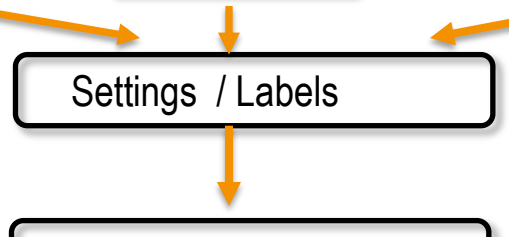
<u>Label Information</u>
(Number, size, distance label sheet)

<u>Labels template</u>
(Size, text specification for single label)

<u>Selection</u>
(Select reference, cables, terminals, PLC)

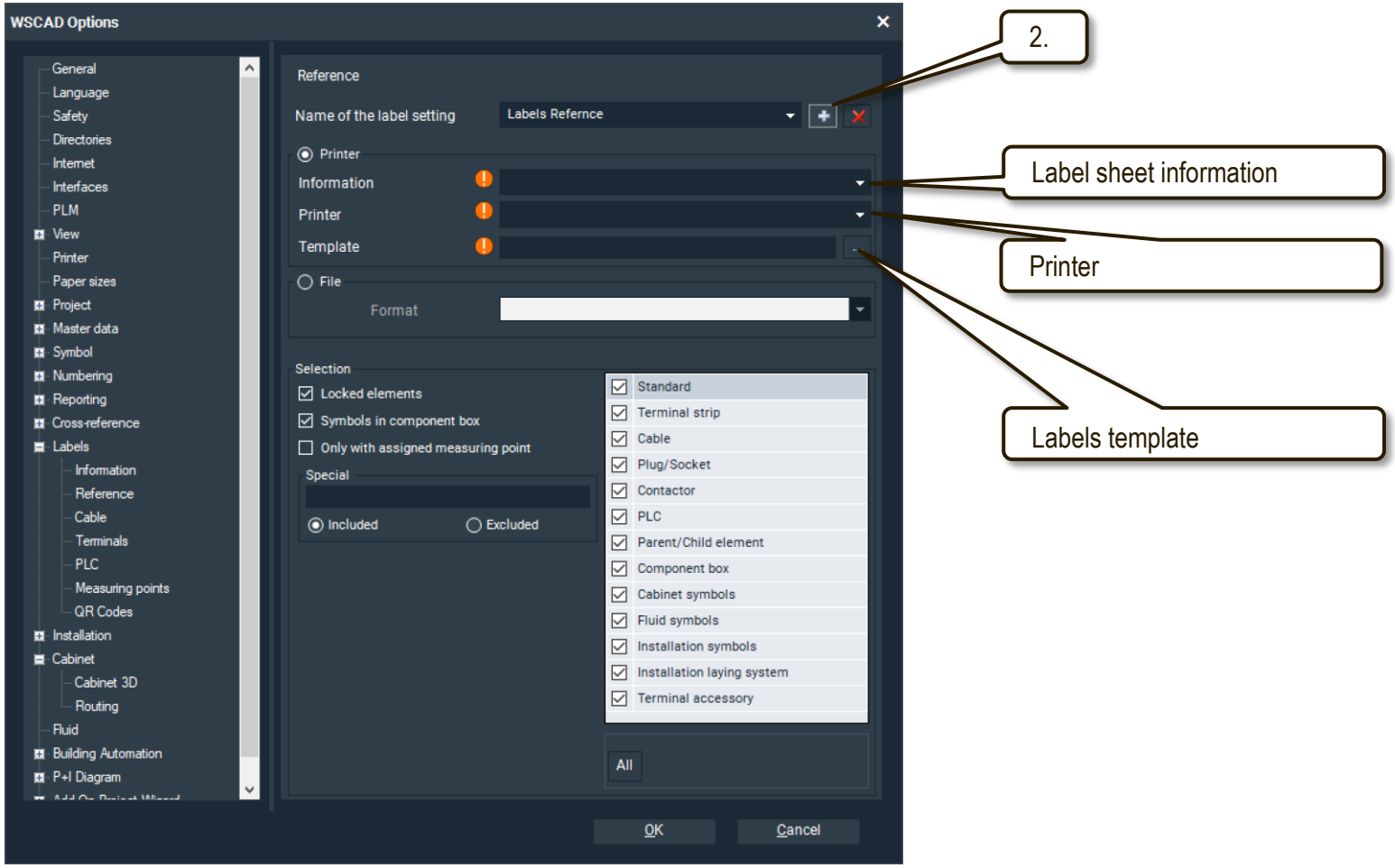






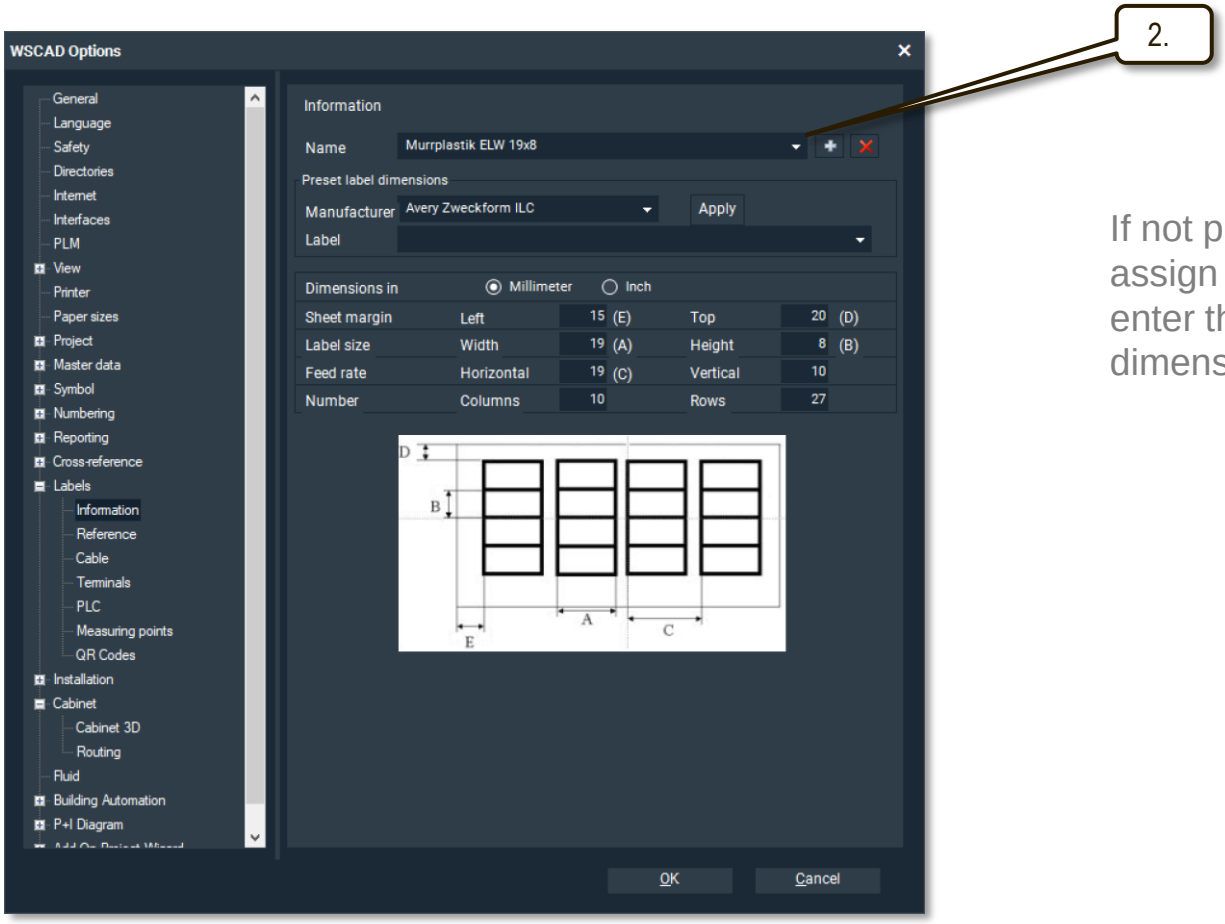
Labels - Output

- Settings for labels
  1. Go to "Settings | Labels | Reference".
  2. Enter a new profile name, "Component labels", here



Labels - Output

- Settings for labels
  1. Open "Settings | Labels | Information".
  2. Select the name of the information shown here.



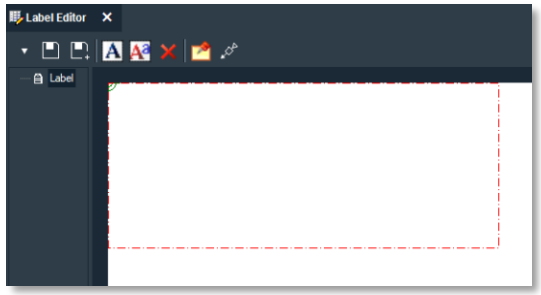
If not present, or for a new sheet, assign a new name via  and enter the displayed values for the dimensions.

3. Confirm with OK“.

Labels - Output

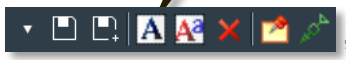
- Labels - Template
  1. Go to the "View | Additional windows" menu and select the Templates Explorer.
  2. Right-click on Template and create a new "Labels" folder.
  3. Right-click on the new folder and select "New label".
  4. Select the preset name of the label sheet.

  5. Confirm with "OK". The Label Editor opens.
     The red rectangle represents the working surface of the label defined above.
  6. Zoom in if necessary.

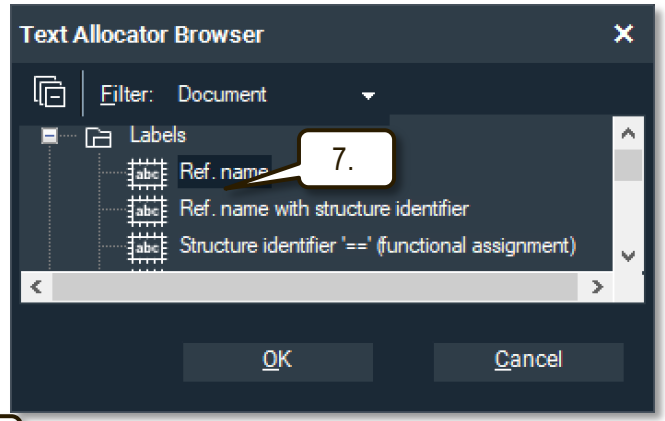


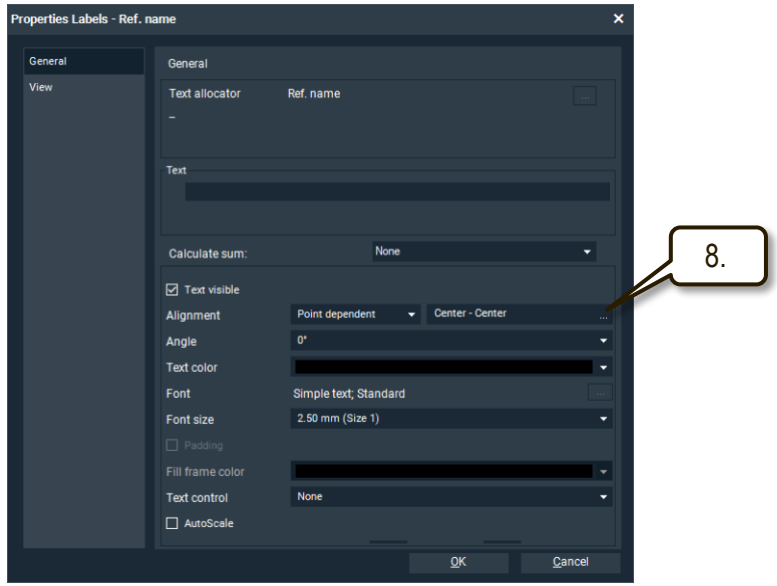
Labels - Output

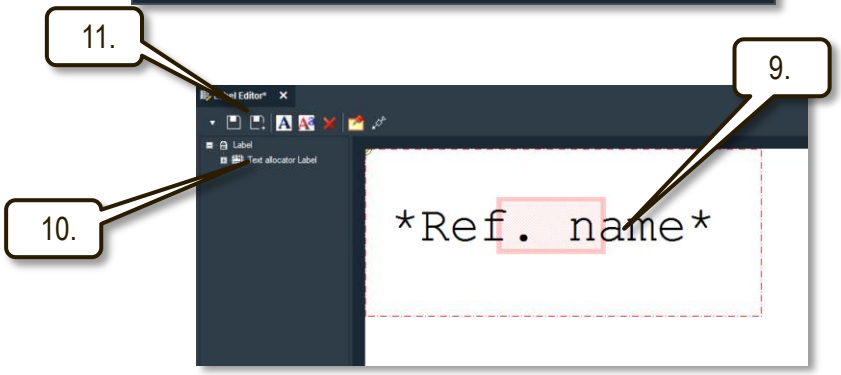
- Labels - Template

7. Using the command 

, select the text allocator "Ref. name" in the "Labels" folder with a double-click.
8. In the following window, correct the information.
9. Place this text allocator within the red workspace.
10. By right-clicking there you can access the text properties again.
11. Save this template with the name "Ref. name 19x8". Then close the editor.



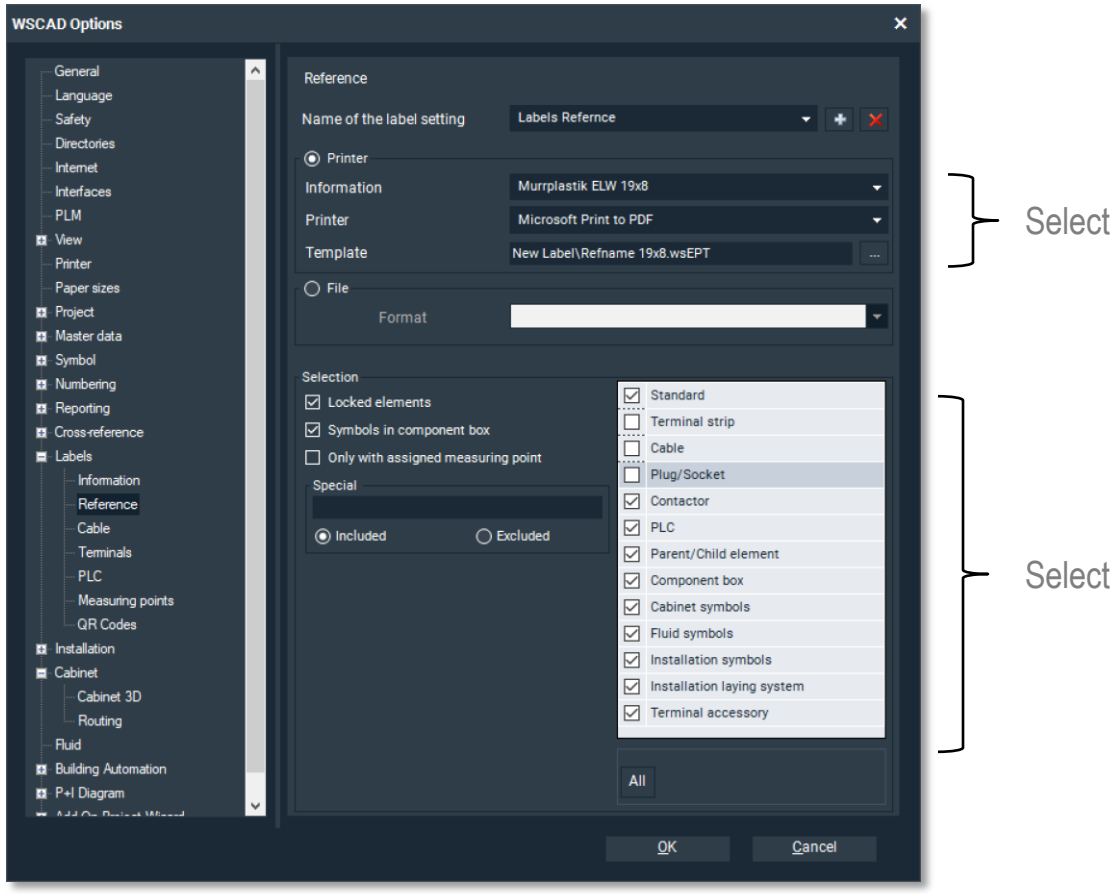




Labels - Output

- Settings for labels

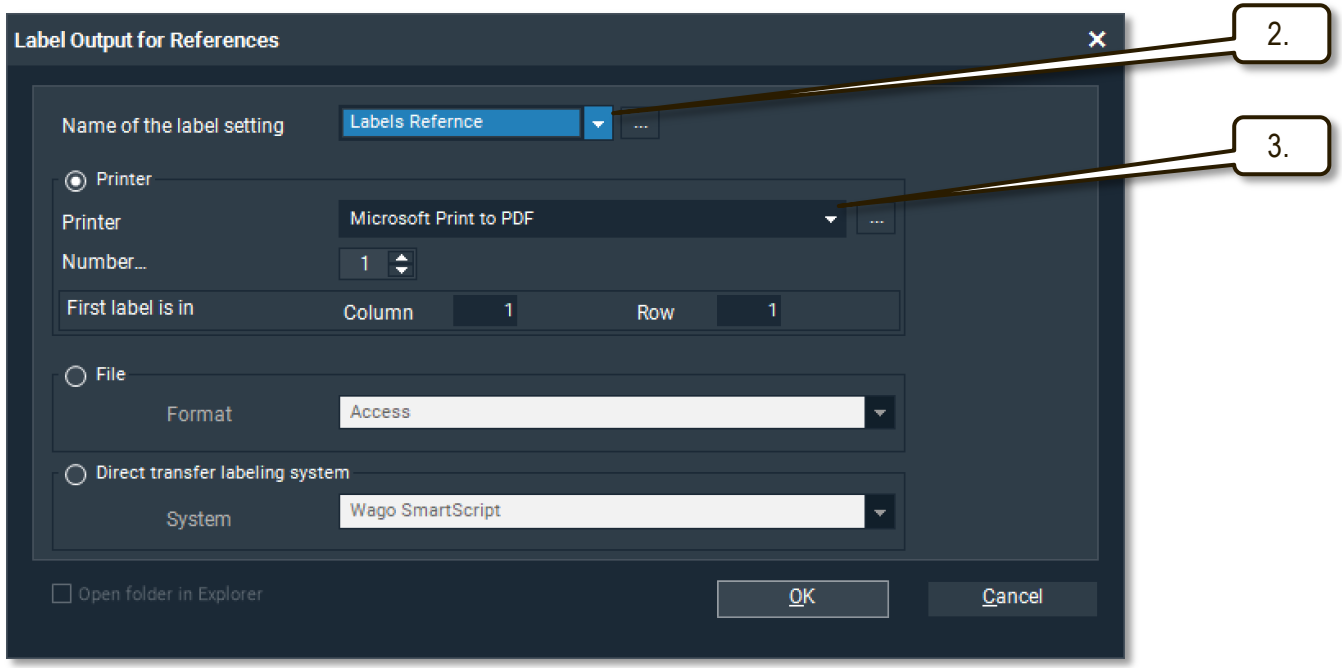
1. Reopen "Settings | Labels | Reference" and complete the information as shown:



3. Confirm the entries with "OK".

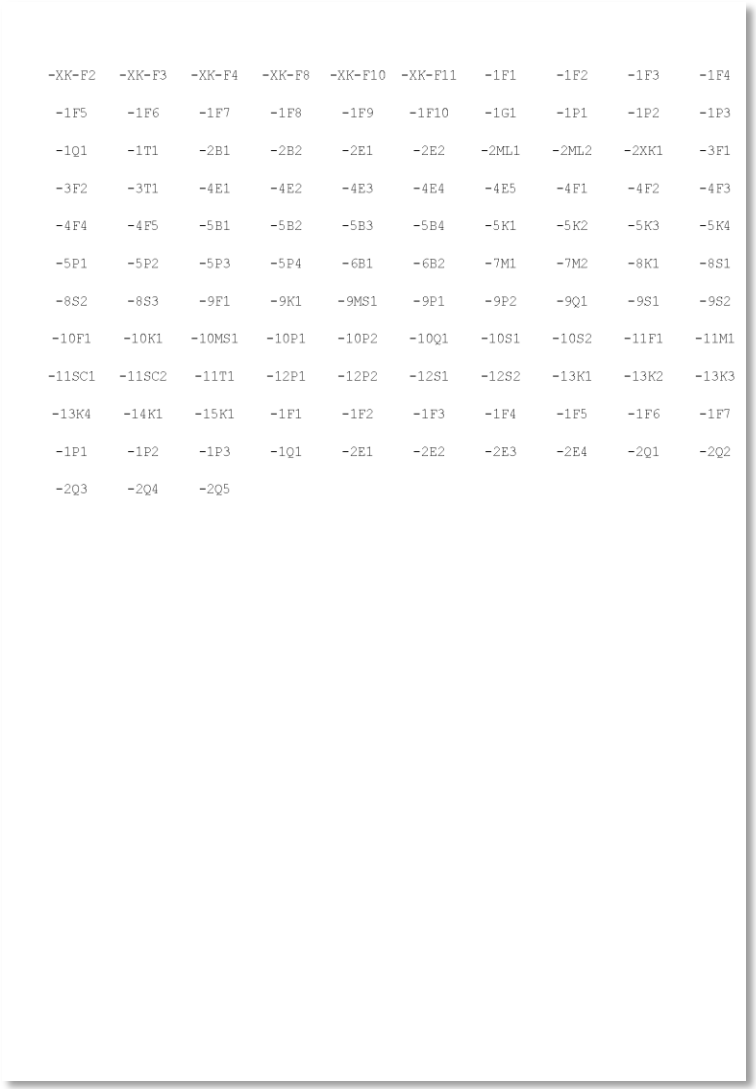
Labels - Output

- Reporting | Labels
  1. Open the "Report | Labels | Reference" menu.
  2. Verify that the correct profile name is set for the output.
  3. Set the output to printer and select a physical printer or the PDF output.
  4. Check the printed result.



Labels - Output

- Reporting | Labels



-XK-F2 -XK-F3 -XK-F4 -XK-F8 -XK-F10 -XK-F11 -1F1 -1F2 -1F3 -1F4
-1F5 -1F6 -1F7 -1F8 -1F9 -1F10 -1G1 -1P1 -1P2 -1P3
-1Q1 -1T1 -2B1 -2B2 -2E1 -2E2 -2ML1 -2ML2 -2XK1 -3F1
-3F2 -3T1 -4E1 -4E2 -4E3 -4E4 -4E5 -4F1 -4F2 -4F3
-4F4 -4F5 -5B1 -5B2 -5B3 -5B4 -5K1 -5K2 -5K3 -5K4
-5P1 -5P2 -5P3 -5P4 -6B1 -6B2 -7M1 -7M2 -8K1 -8S1
-8S2 -8S3 -9F1 -9K1 -9MS1 -9P1 -9P2 -9Q1 -9S1 -9S2
-10F1 -10K1 -10MS1 -10P1 -10P2 -10Q1 -10S1 -10S2 -11F1 -11M1
-11SC1 -11SC2 -11T1 -12P1 -12P2 -12S1 -12S2 -13K1 -13K2 -13K3
-13K4 -14K1 -15K1 -1F1 -1F2 -1F3 -1F4 -1F5 -1F6 -1F7
-1P1 -1P2 -1P3 -1Q1 -2E1 -2E2 -2E3 -2E4 -2Q1 -2Q2
-2Q3 -2Q4 -2Q5

WSCAD SUITE SUITE X

Training Expert

- Creating new components
- Creating part data
- Combi elements

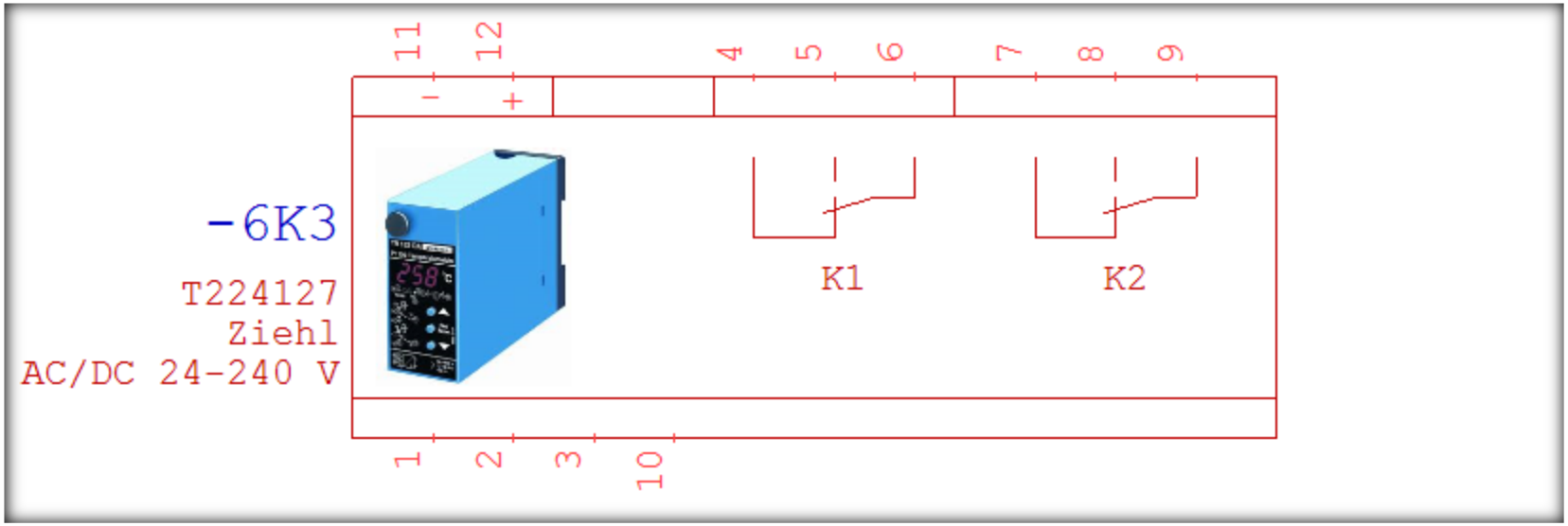
Project Preparations

- Variant a)
  - You are attending the "WSCAD Professional Engineer Training" (5 days)
    - Open the "Training Basics" project if not already done.

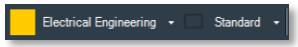
- Variant b)
  - You are attending the "WSCAD Expert Engineer Training" (3 days)
    - Plug the USB stick into your PC.
    - Start WSCAD.
    - Go to the menu "Project | Import | WSCAD SUITE".
    - Go to  and navigate on your stick to Project backup under Data for Training / Backup.
    - Import the "Training Basics" project“.

Create new component - New symbol - Standard

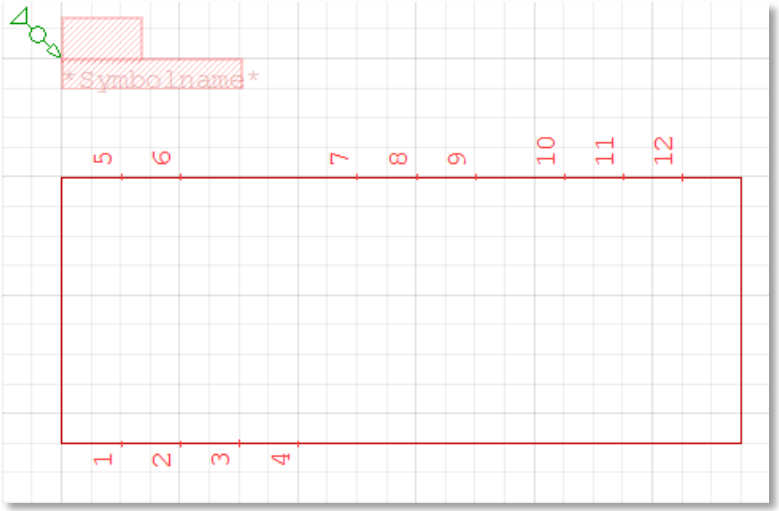
- In the following, a new component consisting of *symbol* and *part data* is to be created.
- It is to be initially implemented as a component without Manager functions, i.e., as a standard symbol. A Pt100 temperature relay from the company Ziel will be used as an example here.
- Goal of the exercise:



Create new component - New symbol - Standard

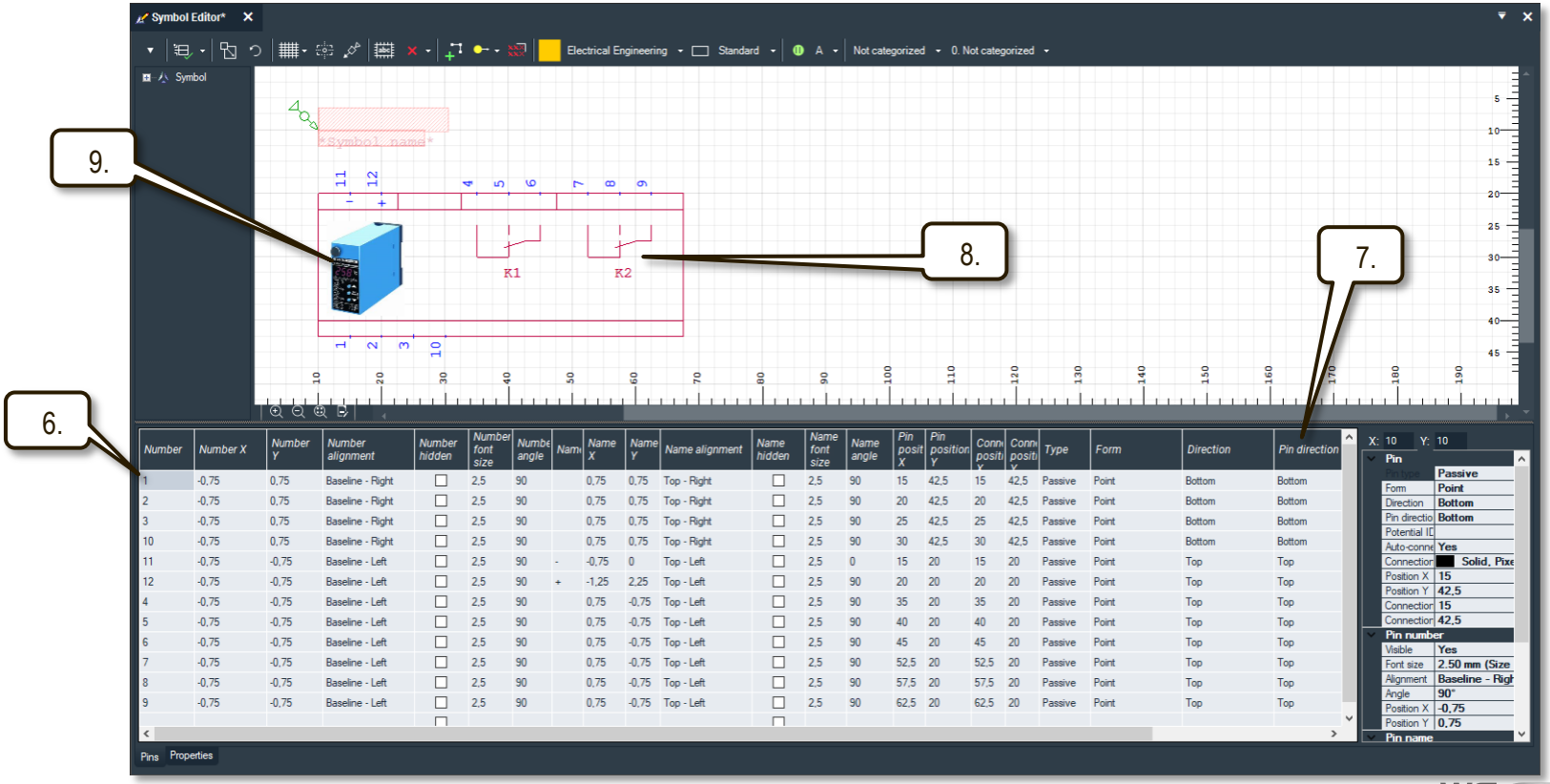
- Create symbol
  1. Locate the file "TR122DA_Data sheet.pdf" on the USB stick under /Data for Training Suite 2017/Documents and open this file.
  2. At the same time, also go to the WSCAD menu "Tools | Editors | Symbol" to open the "empty" Symbol Editor.
  3. Check the discipline and the symbol type In the toolbar 

  4. Draw a rectangle, height: 22.5; Length: 57.5.
  5. Place pins of the type Point  as depicted in the PDF or as shown here: (distance of 2 grids between the pins).



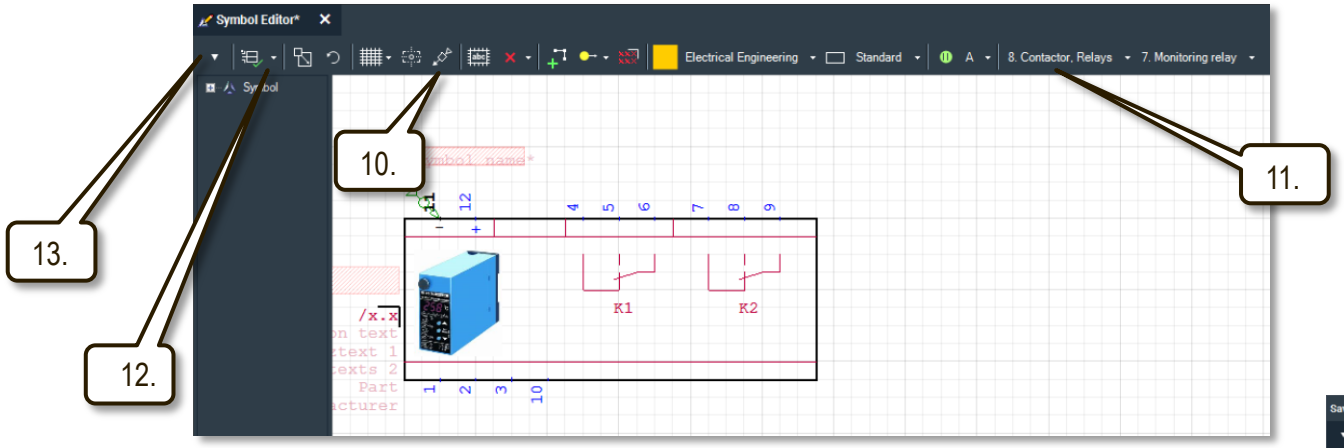
Create new component - New symbol - Standard

6. Edit the pin numbers as in the real device using the lower table
   (Drag and enlarge the window for the table).
   Enter additional pin names like "+, -" here and move them into the drawing.
7. Check the pin direction of the pins.
8. Add the graphical elements.
9. Place the photo by using the command  or via the "Insert Graphic | Image" menu
   (scaling: "Shift+S" -> smaller, "S" -> larger).

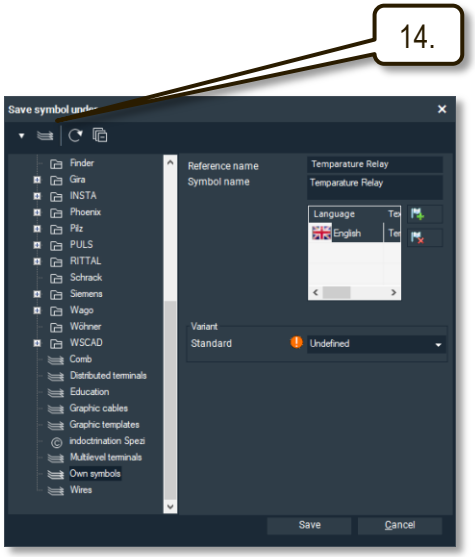


Create new component - New symbol - Standard

10. Place the insert point on the upper left pin (11).
11. Set a category and a subcategory.
12. In order to provide the symbol with all standard-compliant texts and parameters, click on the command "Adjust to norm" with the option "Adjust all to norm except pins".
13. Save the symbol with "Save As".

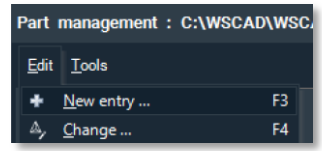


14. Create a new library using  and name this „Own Symbols".
15. Give the symbol a name and a standard and go to "Save".
16. Then close the Symbol Editor.



Create new component – New part - Standard

- Creating part data
  1. Open the Part management
  2. Go to "Edit | New entry..." on the top left.
  3. Click on the lower right on to delete existing values.
  4. Enter the following values and confirm with "OK".



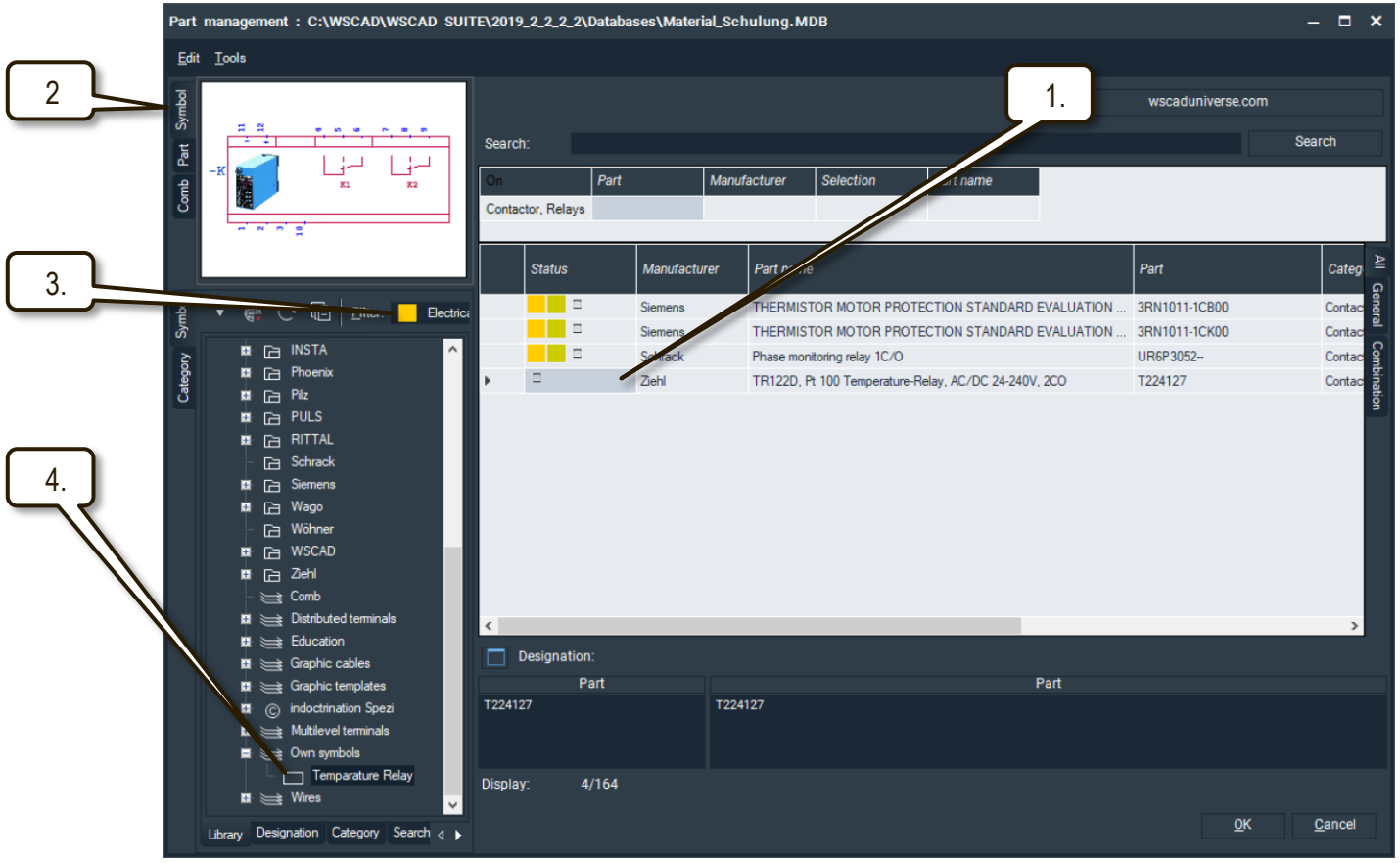
Change part

	WSCAD field	Database field	Field type	Value
	Part	artNumber	Text	T224127
€	Currency	artCurrency	Text	
	Manufacturer	artManufacturer	Long	Ziehl
	Selection	artSymbolSelection	Text	
	Comb	artKamm	Text	
	Order number	artNumberOrder	Text	
	Type name	artTypeName	Text	TR122D
	Name 1	artBEZEICHNUNG1	Text	TR122D, Pt 100 Temperature-Relay, AC/DC 24-240V
	Name 2	artBEZEICHNUNG2	Text	
	Category	artCategory	Long	8. Contactor, Relays
	Subcategory	artSubCategory	Long	7. Monitoring relay
	Width	cabWidth	Text	42
	Height	cabHeight	Text	82
	Length | Depth	cabLenght	Text	122
	Part name	Part name	Text	TR122D, Pt 100 Temperature-Relay, AC/DC 24-240V, 2CO
	Long text	Long text	Text	
	External documents	External documents	Text	E:\Daten für Schulung\Dokumente\TR122DA_Datenblatt.pdf
	Text 1	Text 1	Text	
	Text 2	Text 2	Text	
	Current	Current	Text	
	Voltage	Voltage	Text	AC/DC 24-240 V

Temporary link to the USB stick

Create new component – New part - Standard

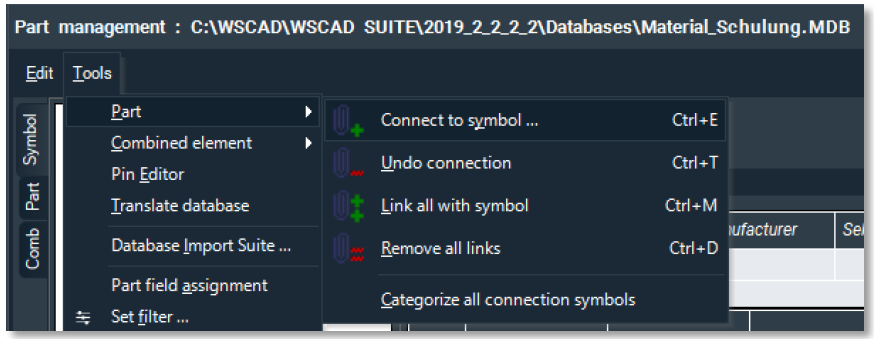
- Complete part data.
- In order to use the part-oriented placement method, the part still needs a symbol link:
  1. In the still open part management, the new part is marked.
  2. Switch the view at the top left to Symbol clicking the "Symbol" tab.
  3. Switch to Symbol Explorer below and set the filter to "Electrical Engineering".
  4. Go to the new library "Custom symbols" and click on the new symbol.



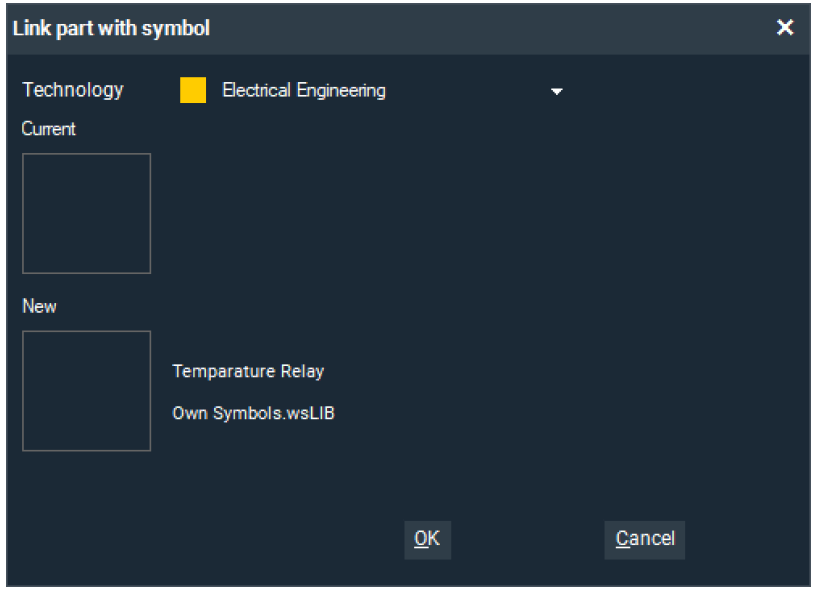
Create new component – New part - Standard

- Complete part data

1. Click at the top left on "Tools | Part | Connect to symbol ...".

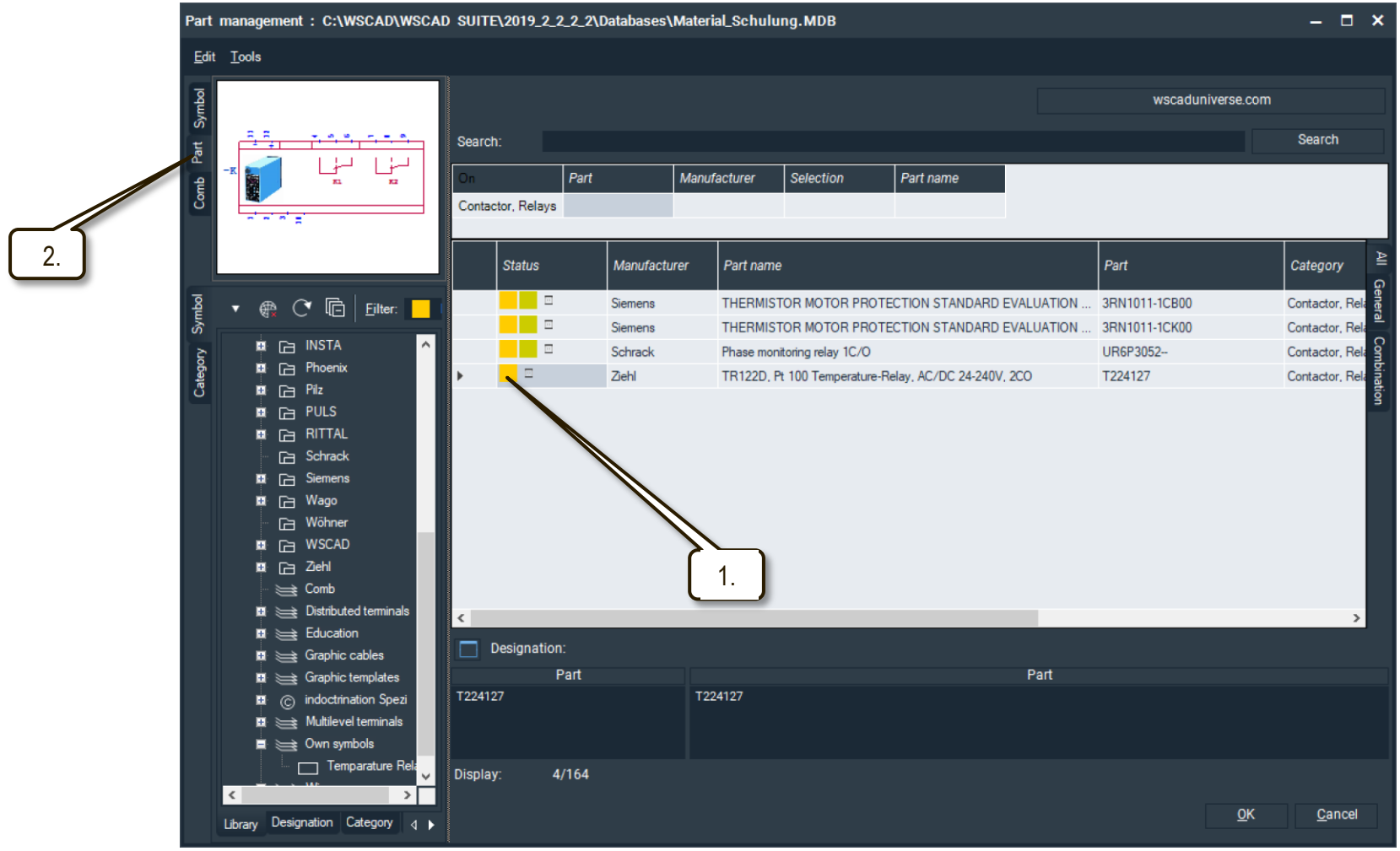


2. In the following dialog, check the discipline and confirm with "OK".

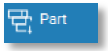


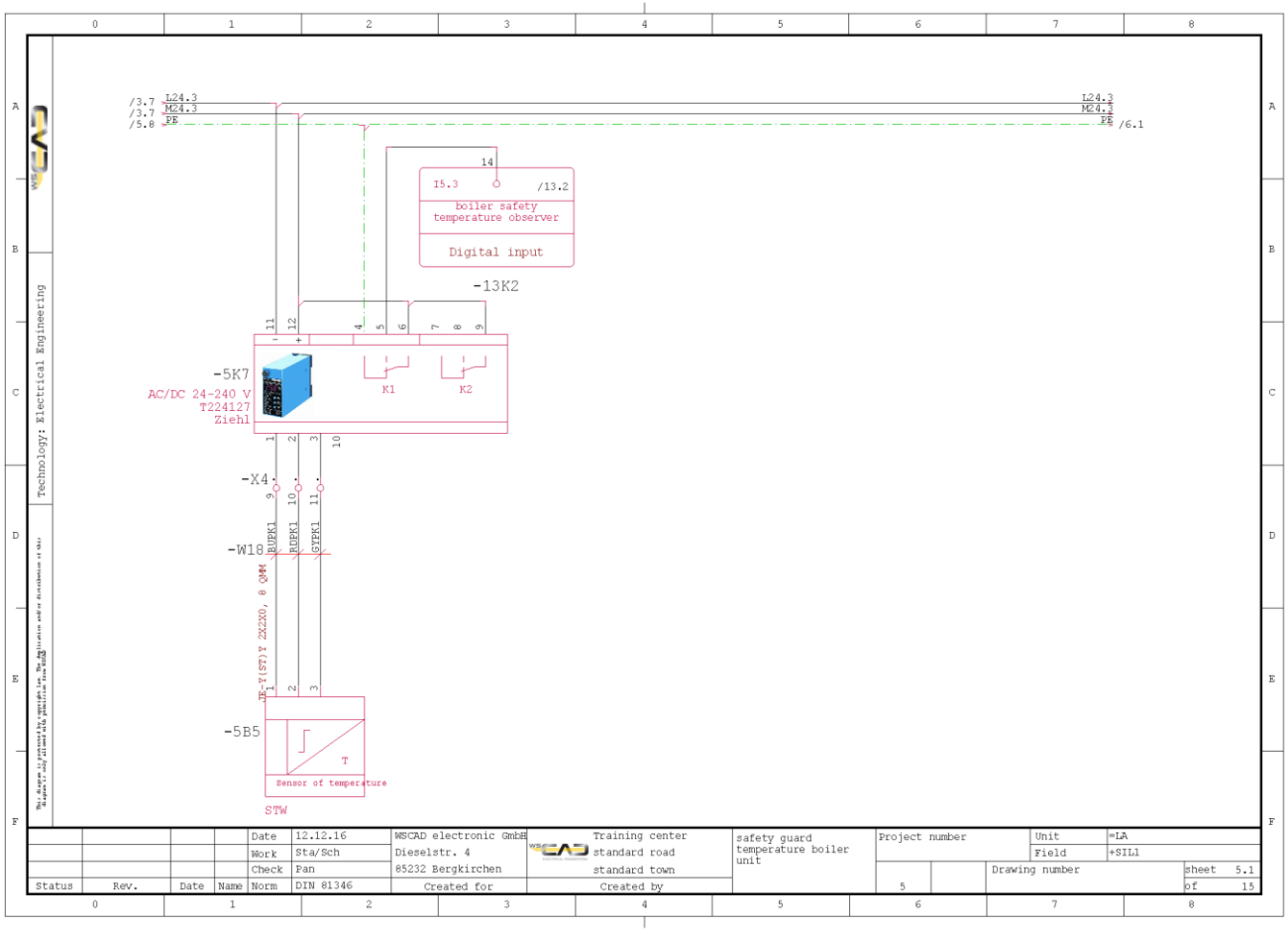
Create new component – New part - Standard

- Complete part data
  1. The new status of the part will then be displayed.
  2. If you click the "Part" tab, you will then see all the symbols connected to the part displayed.



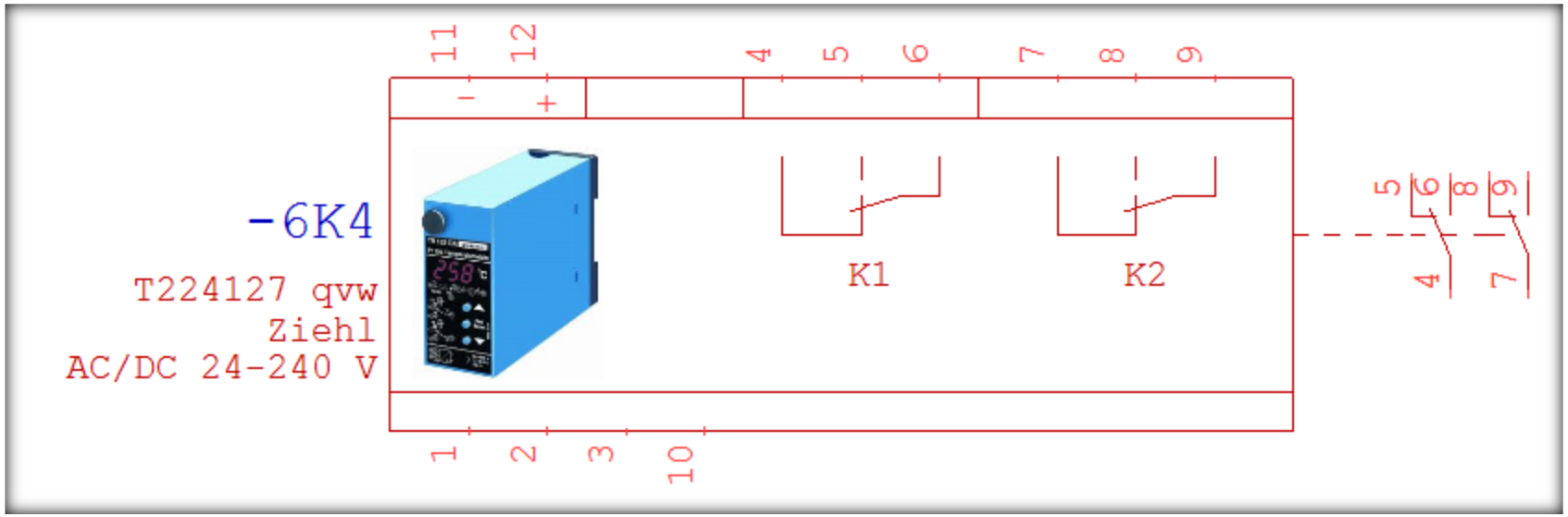
Create new component – Standard

- Place new component
  1. Create a subpage (5.1) in the project after sheet 5.
  2. Place the macro "24V oben" (24V top) from Potentials for training.
  3. Complete the drawing as shown here. Place the temperature relay with part orientation via the insert | Part„ menu 
  4. Change the PLC comment.



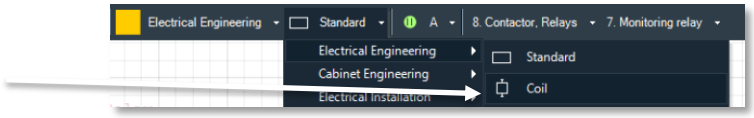
Create new component - New symbol - Coil

- A component with Manager functions is now to be implemented (as a coil symbol here). The newly created Pt100 temperature relay from the company Ziel will serve as an example.
- Goal of the exercise:

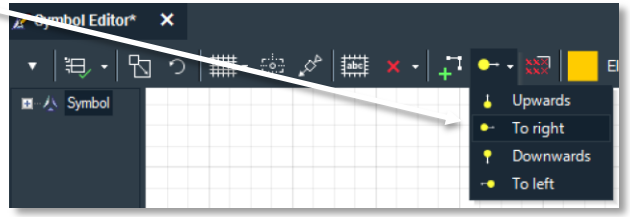


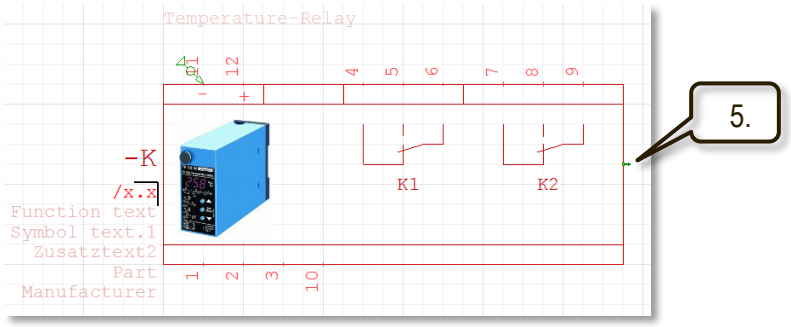
Create new component - New symbol - Coil

- Create coil symbol
  1. Go in the Symbol Explorer to the newly created symbol "Temperature Relay" in the "Custom Symbols" library.
  2. Right-click to open the Symbol Editor.
  3. Change the symbol type from Standard to Coil



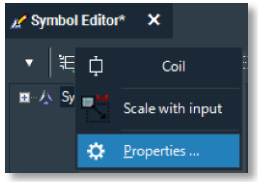
- *Note:* If the contact comb is to be displayed below the symbol, you can finish the editing in the Symbol Editor by saving the symbol. Otherwise, continue:
  4. Go to the command "Set docking point"
  5. and place the symbol "To the right"



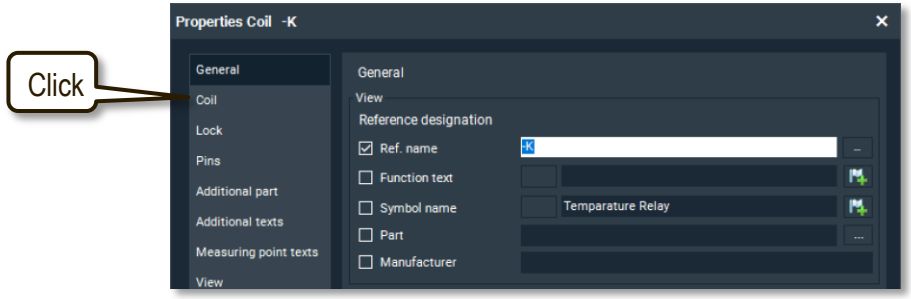


Create new component - New symbol - Coil

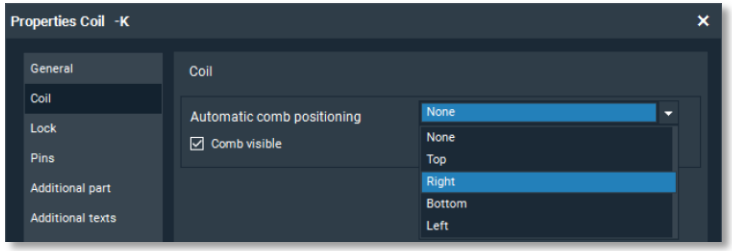
6. Right-click on "Symbol" (at the top left) and go to "Properties | Coil".



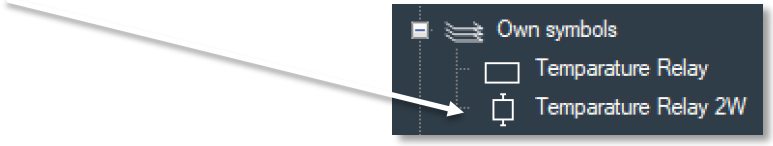
7. In the Properties dialog, click on "Automatic comb positioning".



8. Select the position "Right" here.

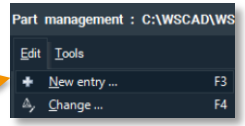


9. Save the symbol in the "Custom Symbols" library with the name "Temperature relay 2W qvw".

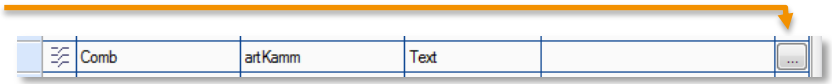


Create new component – New part - Coil

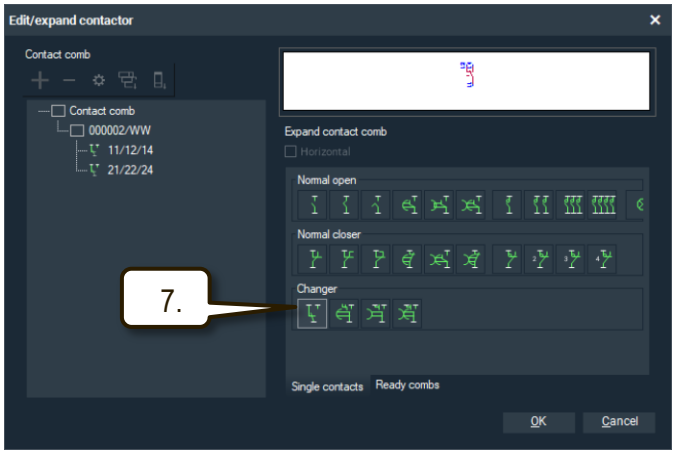
- Creating part data
  1. Open the Part management 
  2. Go to "Edit | New entry..." on the top left.

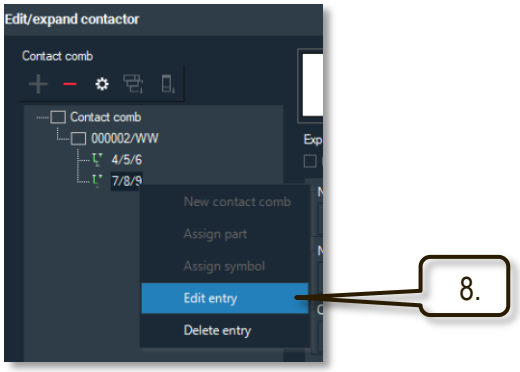


  3. The values from the last viewed record should be displayed here. In this case, the values of the TR122D.
  4. Enter a new part number "T224127 qvw". Leave all other values as is.
  5. Click on the "Comb" field at the back



  6. Click in the "Contact comb" dialog on 
  7. Click sequentially on the contacts that you want to define, here 2 change-over contacts.

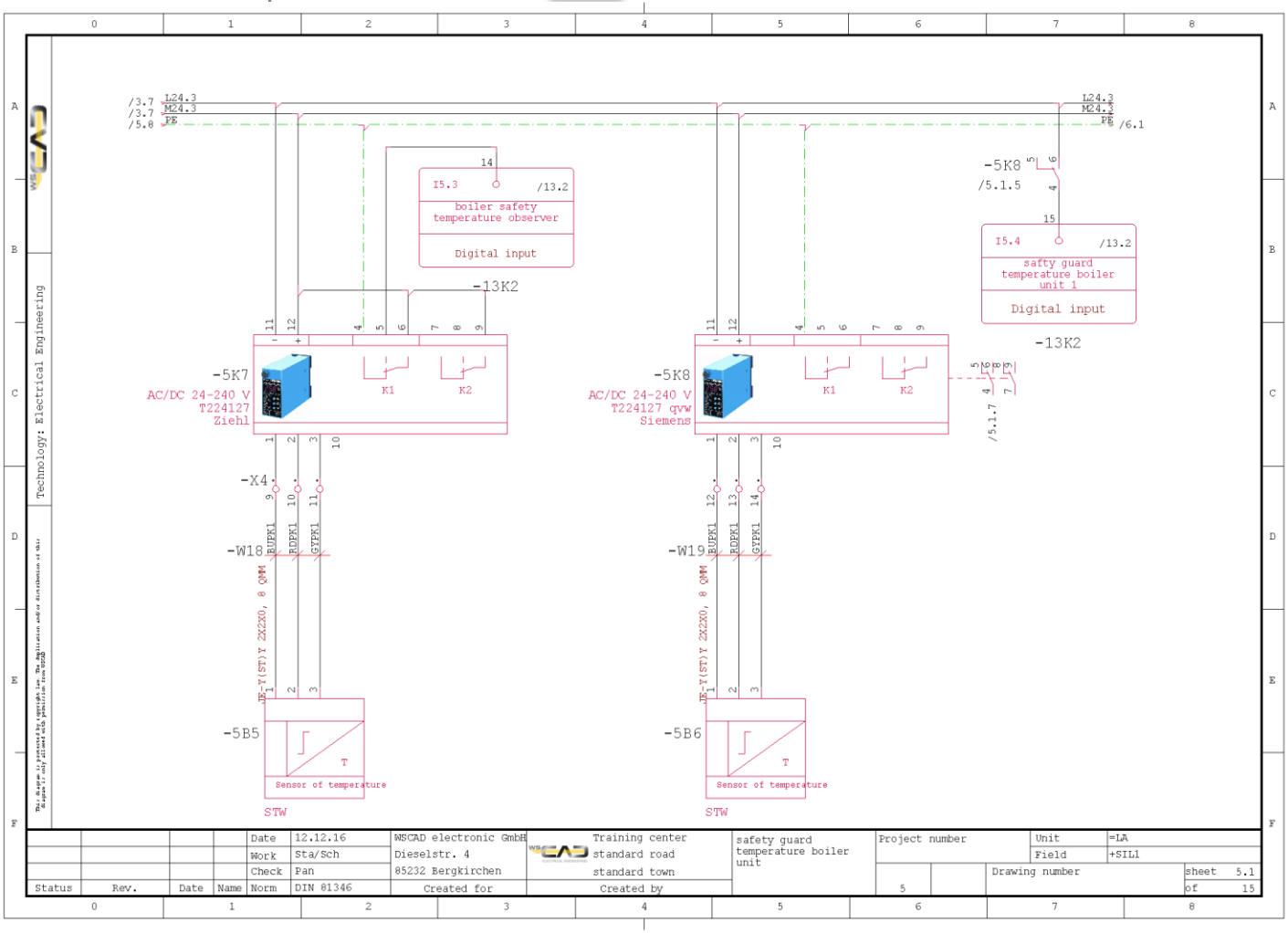




  8. To edit the contact numbers, click on the right side of the contact.
  9. Confirm this dialog and the part record with "OK".
  10. Connect the part with the new symbol "Temperature relay 2W qvw" and exit the part management with "OK".

Create new component – Coil

- Placing a new component
  1. Complete the drawing as shown here. Place the temperature relay with part orientation via the "Insert | Part" menu.



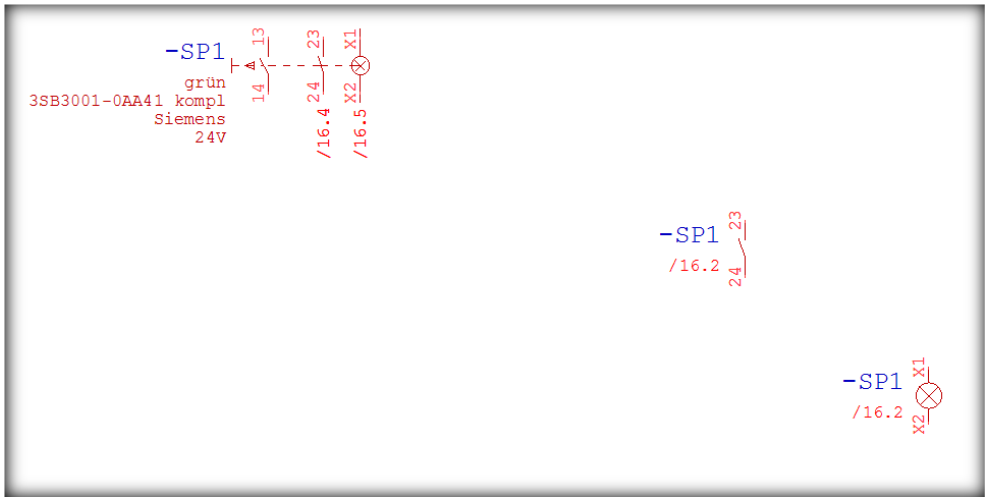
Create new component – Combined element - Coil

- In the following, a new component consisting of *symbol* and *multiple part data* is to be created.
- An *illuminated pushbutton* serves as an example here. It should be capable of being cross-referenced with respect to the lamp and contact element.

  It consists of 4 individual part components.

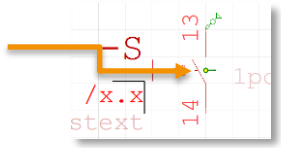
  - Button attachment
  - Two NO-contact switching elements
  - Lamp holder
  - LED lamp

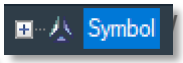
- Goal of the exercise:



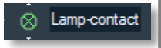
Create new component – Combined element - Coil

- Button - Create coil symbol
  1. Go to the Symbol Explorer and find the symbol "Pushbutton_1S"
  2. Go with a right-click to the Symbol Editor and change the symbol type from Standard to Coil.
  3. Place a docking point "to the right"

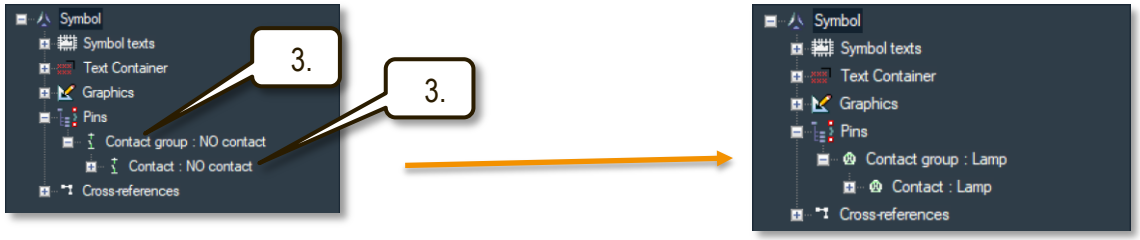


  4. Right-click on the symbol 

Properties / Coil and set the comb position to the right.
  5. Save this new symbol  as "Pushbutton_1S qvw" under the „Own Symbols" library.

- Modify lamp contact
  1. Go to the Symbol Explorer under DIN 81346 / 08_02 Auxiliary contactor to Contact/Lamp 

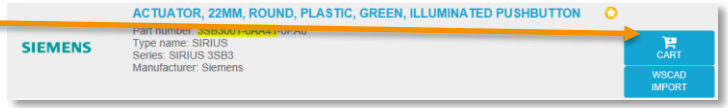
  2. Go to the Symbol Editor by right-clicking.
  3. Expand the symbol texts on the left and go to each contact group by right-clicking on it and then to Contact / Properties and change the group type to "Lamp".



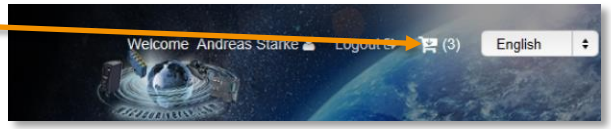
  4. Save this new symbol as "Contact/Lamp" under the „Own Symbols" library.

Create new component – Combined element - Coil

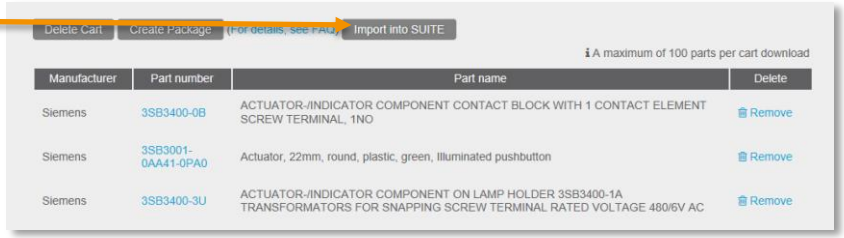
- Import part data
  1. Go with  to the part management and from there to wscaduniverse.com
  2. Enter the following part numbers sequentially in the upper search field and add them to the shopping cart:
     - 3SB3001-0AA41
     - 3SB3400-0B
     - 3SB3400-1A



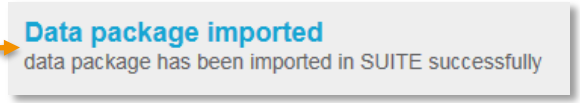
  3. Click on the shopping cart at the top



  3. Then click on



  4. In the following dialog "Overwrite symbols" click "No".
  6. Wait until the successful import is displayed.



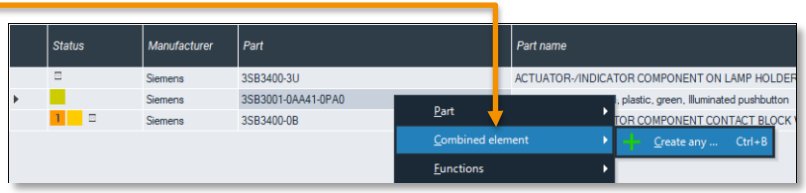
  7. Close the window.
  8. If you set the categories to „Electric Engineering“ in the part management, the imported parts are displayed.

Create new component – Combined element - Coil

- Create a combination element
- Goal: to combine multiple parts under a new "combined" part number

  In the material list, the components are listed individually again.

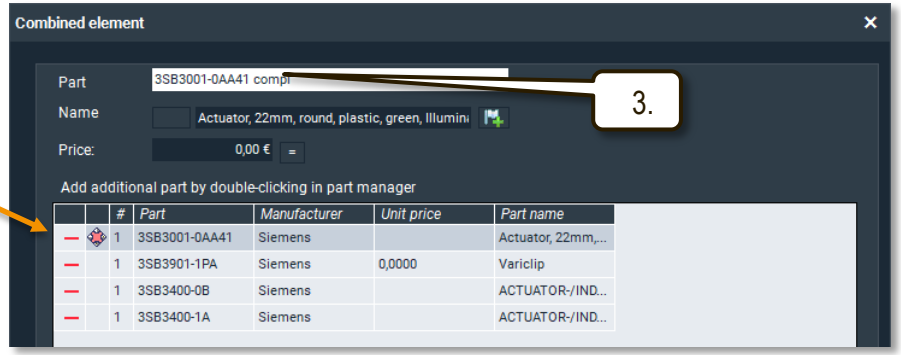
1. In the still open part management, right-click on "3SB3001-0AA41" and then on



2. The "Combined element" dialog is displayed.

   Find the following parts in the part management and double-click on them:

   - 3SB3400-0B (2x)
   - 3SB3400-1A
   - 3SB3901-1PA

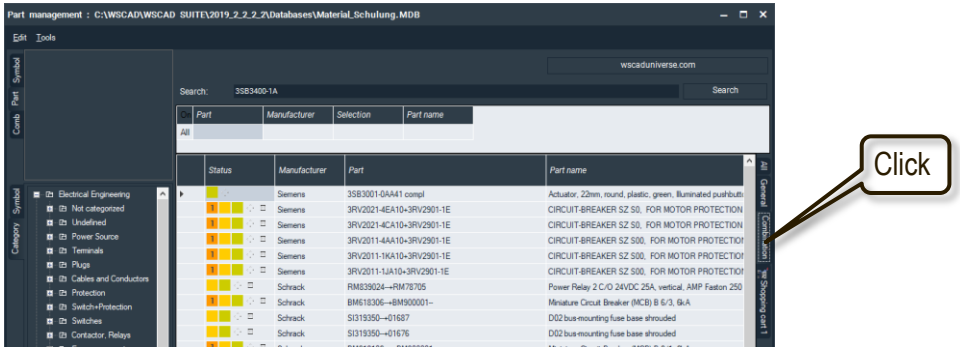


3. Change the part number of the combined element to "3SB3001-0AA21 compl" and confirm with "OK".

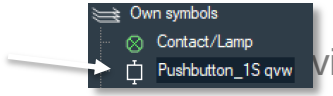
Create new component – Combined element - Coil

- Edit combined element
- Correct data and edit comb

1. Click on the "Combination" tab to quickly select all combined elements.



2. Right-click on this part and on "Part | Modify" (or press F4).
3. Go to the "Comb" field  in the "Edit contactor" dialog and click on 
4. Click on the right under NO on  and then on 
5. Correct the pin numbers by right-clicking and selecting "Edit entry".

6. Confirm the dialog with "OK".
7. Connect the part with the symbol 

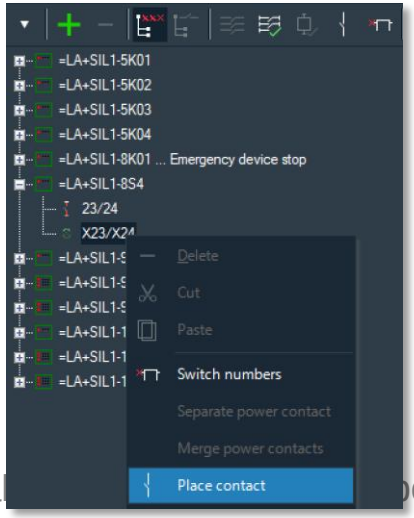
via "Tools | Part | Connect to symbol ...".
8. For example, correct the component name in the part record to "Full-Illuminated pushbutton complete"

   and end the processing in the management with "OK".

Create new component – Combined element - Coil

- Placing a new component
  1. Open Sheet 8 in the project and complete the drawing as shown on slide 23. Place the illuminated pushbutton with part orientation via the "Insert | Part" menu 

above the new combined part number "3SB3001-0AA41 compl" as a new coil element.
  2. The placement of the N.o. contact and lamp contact is best achieved by calling the contactor management and by right-clicking on the contact symbol and selecting "Place contact" there.



  2. Regenerate the material [illegible] poses.

124	Actuator, 22mm, round, plastic, green, Illuminated pushbutton 1	3SB3001-0AA41	=LA+SIL1-8S4	Siemens
	ACTUATOR-/INDICATOR COMPONENT CONTACT BLOCK WITH 1 CONTACT EI 2	3SB3400-0B		Siemens
	ACTUATOR-/INDICATOR COMPONENT LAMP HOLDER BA 9S WITHOUT LAMP 1	3SB3400-1A		Siemens
	Variclip 1	3SB3901-1PA		Siemens

Create new component – Combined element - Coil

- Placing a new component