



# CADMATIC

## CADMATIC Electrical

Working with a project database

Copyright © 2025 by Cadmatic

All rights reserved. No part of this publication may be reproduced or transmitted in any form or by any means without the express written permission of the publisher.

CADMATIC is a registered trademark of Cadmatic Oy. All other marks are property of their respective owners.

Cadmatic

Linnankatu 52

20100 Turku

Finland

Tel. +358 2 412 4500

[www.cadmatic.com](http://www.cadmatic.com)

# Contents

1. Introduction .....	5
2. Project tree .....	6
2.1. Manage Electrical windows .....	6
2.2. View device occurrences .....	6
2.3. Edit devices .....	8
3. Project information and settings .....	10
3.1. Edit project's target information .....	11
3.2. Edit project settings .....	12
4. Database tool .....	13
4.1. Edit device .....	13
4.2. View and edit with the selection list .....	14
4.3. Create, edit and remove documents .....	17
4.3.1. Create a document .....	17
4.3.2. Edit a document .....	18
4.3.3. Remove a document .....	18
4.4. Mass edit drawing frame/label information .....	19
4.5. Create new objects in the database .....	20
4.6. Define and insert objects created in the database .....	23
4.7. Mass edit in Excel .....	28
4.8. Mass edit in the grid .....	30
4.9. Copy from a clipboard project .....	31
5. Management tools .....	34
5.1. Cable types .....	34
5.2. Cable packages .....	35
5.3. Product information .....	36
5.4. Additional information .....	39
5.5. Plate definitions .....	41
5.6. Product models .....	43
5.6.1. Create a product model .....	43
5.6.2. Edit the product model .....	45
6. Database reporting tools .....	46
6.1. Define report settings .....	46
6.2. Create a new report .....	47
6.3. Update report .....	50

6.4. Edit report template ..... 51

# 1. Introduction

This guide introduces you to CADMATIC Electrical, focusing on the database functions for the Schematics toolset. You will learn to add a drawing to a database, and to add and manage existing data in a database. Before starting, it is recommended that you have completed the basic training for Schematics.

---

**Note:** This guide is based on CADMATIC Electrical version 2025H1. In other versions, some of the functions might work differently.

---

In this guide, the example project included in the installation is used. Copying the example project to another directory is recommended because new installations can affect the default location project, erasing experimenting done in that location.

**To create a copy of the example project, do the following:**

1. Select **File > Open > Exercise drawings directory**.
2. Open the *Electrical* folder.
3. Copy the *Electrical Example Project* folder to the desired directory.


## 2. Project tree

You can use the project tree to view device occurrences and to edit devices, for example.

You can view the project tree in the **Electrical** and **Electrical 2** windows. By default, these windows open to the left side of the screen.

### 2.1. Manage Electrical windows

By default, the **Electrical 1** and **Electrical 2** windows are docked to the left but you can drag them to the desired locations in the application window. For example, leave **Electrical 1** to the left side of the screen and drag **Electrical 2** to the right side and dock it using the arrow button on the right.

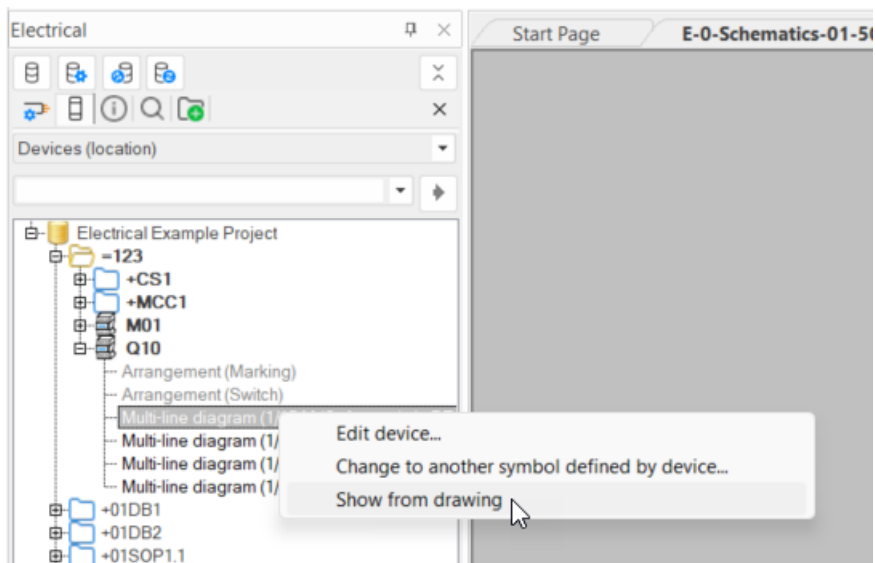
You can add new tabs to the **Electrical** windows with the  (Add a new tab) button. In addition, you can change the order of the tabs as desired.

The settings are saved when you close the application.

### 2.2. View device occurrences

Do the following:

1. Open the *E-0-Schematics-01-501.drw* file included in the example project.
2. In the **Electrical** window, select **Devices (location)** from the uppermost drop-down menu. By default, the **Electrical** and **Electrical 2** windows open to the left side of the screen.
3. Open node **=123**.
4. Open the **Q10** node to see a list of all occurrences in all project drawings.  
Bold text means the item is in the current drawing, while gray means the item is in another drawing.
5. View device occurrences by right-clicking them, and selecting **Show from drawing**.



The first occurrence is shown. You can view the next device occurrence by pressing Enter.

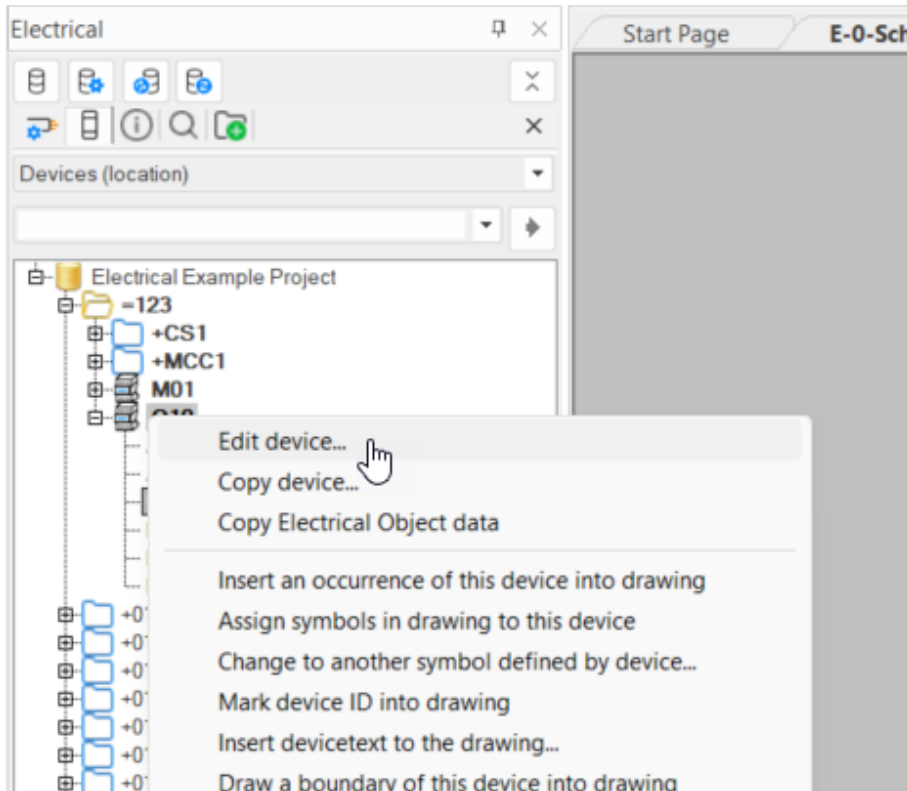
Stop viewing occurrences by pressing Esc.

## 2.3. Edit devices

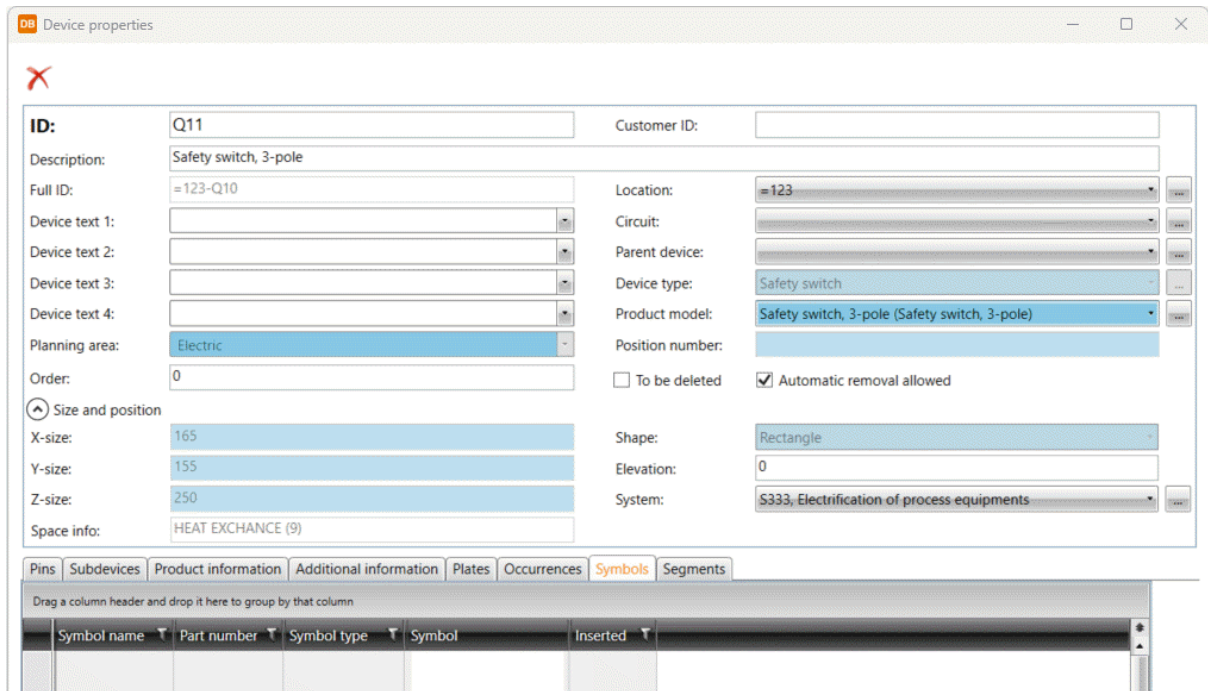
You can easily edit devices via the project tree.

**Do the following:**

1. In the project tree, right-click the safety switch **Q10** and select **Edit device**:



- In the **Device properties** dialog, change the ID from **Q10** to **Q11**.

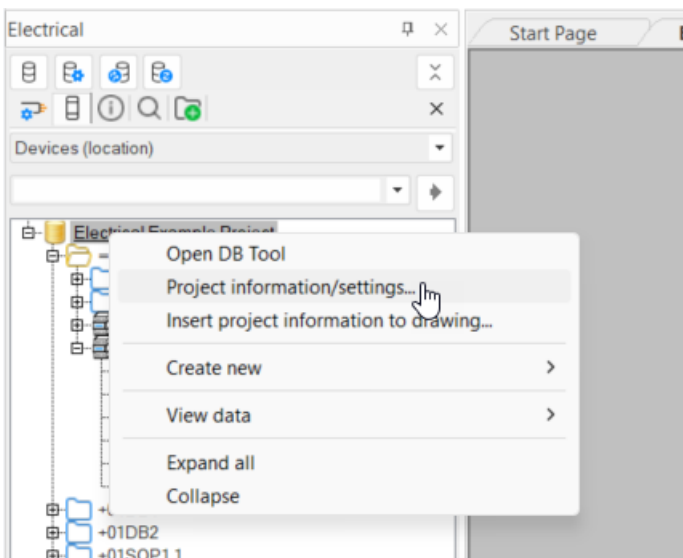


- Click **OK**. The device ID updates in the project tree and in all the drawings automatically. [View occurrences](#) in the project to see the changed ID.


## 3. Project information and settings

In **Project information/settings**, you can edit and check project's default settings, current settings, project information and database information (such as product models).

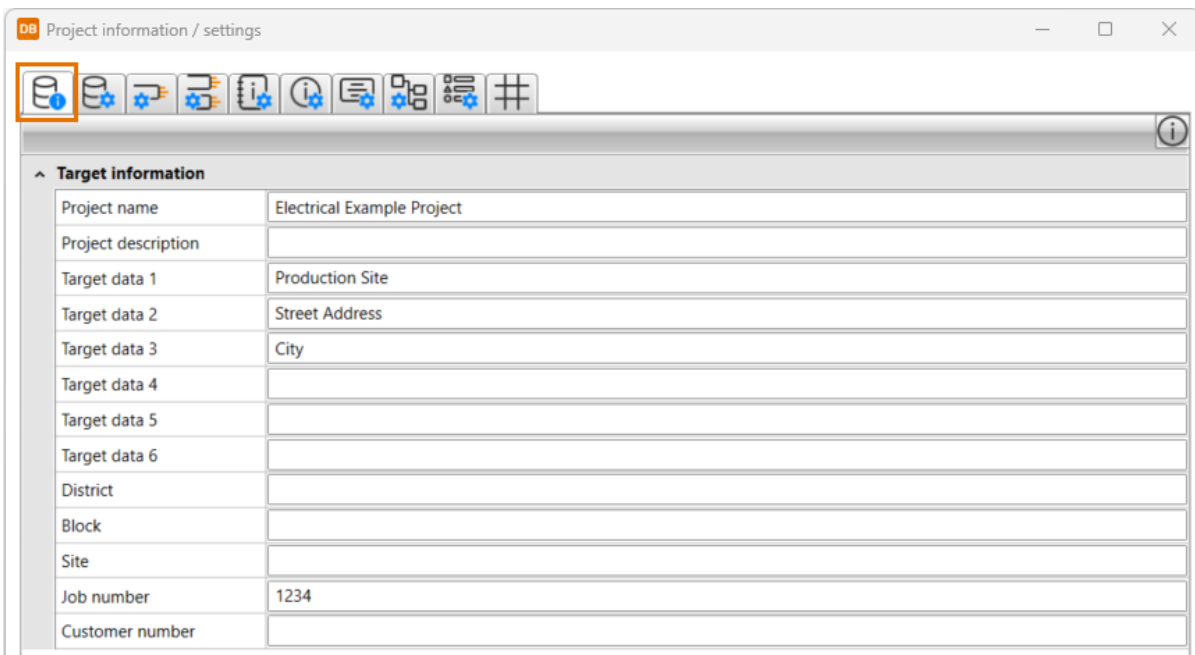
Open project information and settings by selecting **Electrical** tab > **Projects** group > **Settings**.  
Alternatively, right-click the project tree root node and select **Project information/settings**:



### 3.1. Edit project's target information


After opening the project information/settings, click the  (**General information**) tab from the top left corner of the dialog if the project information is not already open. Fill in and edit the information.

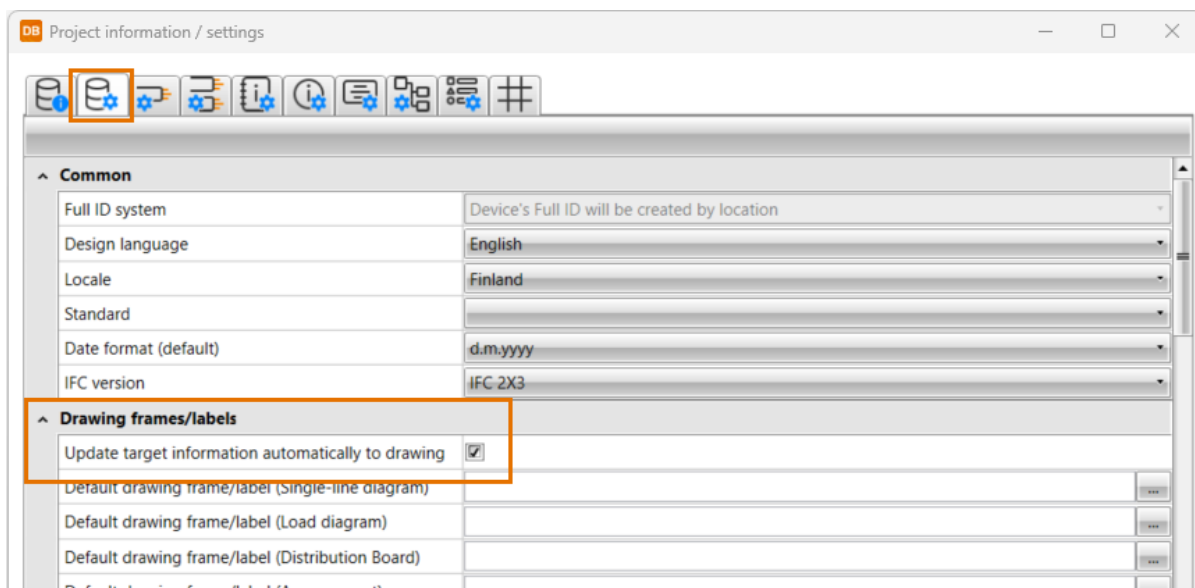
Do not click **OK** yet since we will next edit some project settings.



## 3.2. Edit project settings

Do the following:


1. In the **Project information / settings** dialog, go to the project settings by selecting the  (Project settings) tab.
2. In the **Drawing frames/labels** section, click the check box for **Update target information automatically to drawing**.



When this option is enabled, project information is automatically updated to drawing frames. Target information is shown at the bottom part of the frame.

3. In the **Cross-references** section, disable the **Include row information** setting. Later, when we close the dialog with **OK**, the auxiliary contact packs lose the row letter from the references.
4. Try changing the other settings to see how they affect the drawing.


After finding the optimal settings for your needs, it is recommended that you edit the default settings accordingly. You can access the default settings from the bottom left corner of the **Project information / settings** dialog. After changing them and saving, they will be applied to all new projects by default. The current project will not be affected.

Learn more about the various project settings by clicking the  button at the bottom of the dialog.

## 4. Database tool

Open the database tool by selecting **Electrical** tab > **Projects** group > **Open**. A separate dialog opens where you can see the database view of the project and edit the project.

You can keep the database dialog open all the time i.e. you can make changes both in the drawing and in the database without closing neither.

**Note:** The drawing must be saved to update the data to the database. If you have the database tool already open, you need to reload it after drawing side changes from the top left by selecting **Home** > **Refresh**. If database changes are done and you want to apply them to the drawings, click the  (**Sync changes from database to drawing**) button in the **Electrical** window.

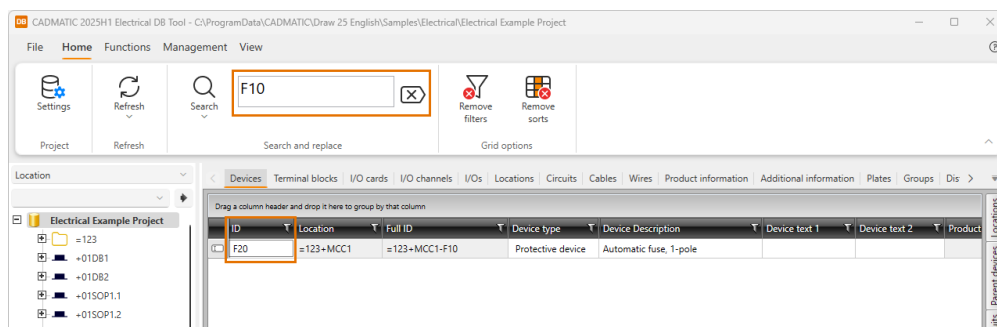
### 4.1. Edit device


Let's edit the device ID.

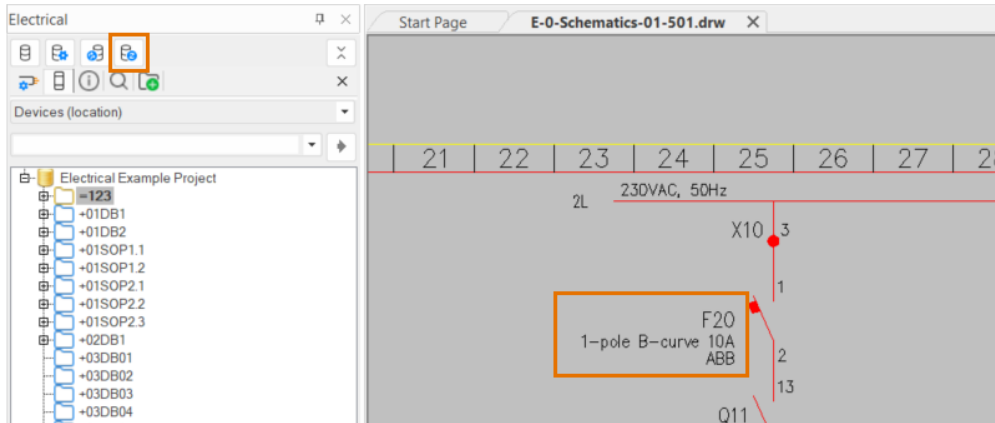
**Note:** Avoid editing the symbol ID attribute in the property dialog. It only changes the current symbol ID and not the other occurrences.

**Do the following:**

1. Select **Home**.
2. Use the search function: type *F10* to find one device from +MCC1.
3. Double-click the ID field, edit the ID in the grid from **F10** to **F20** and press Enter.



- Go back to the drawing side, and click the  (Sync changes from database to drawing) button in the **Electrical** window. The device ID is changed to **F20**.

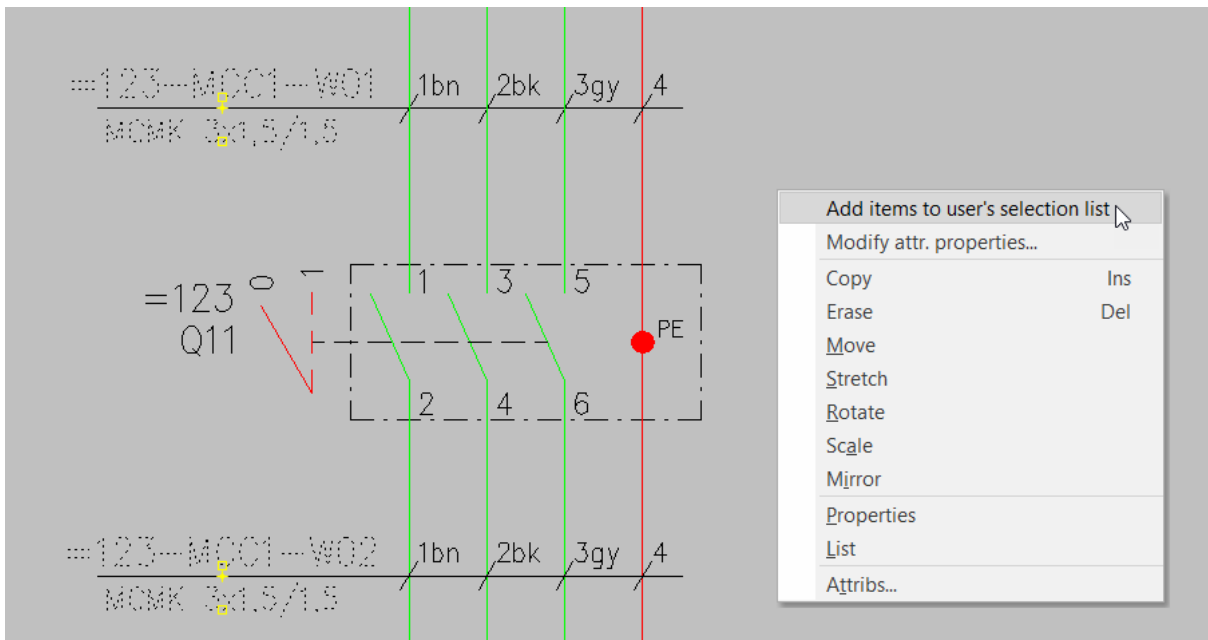


This is similar to [what we did earlier](#) when editing **Q10** via the project tree on the drawing side.

## 4.2. View and edit with the selection list

Do the following:

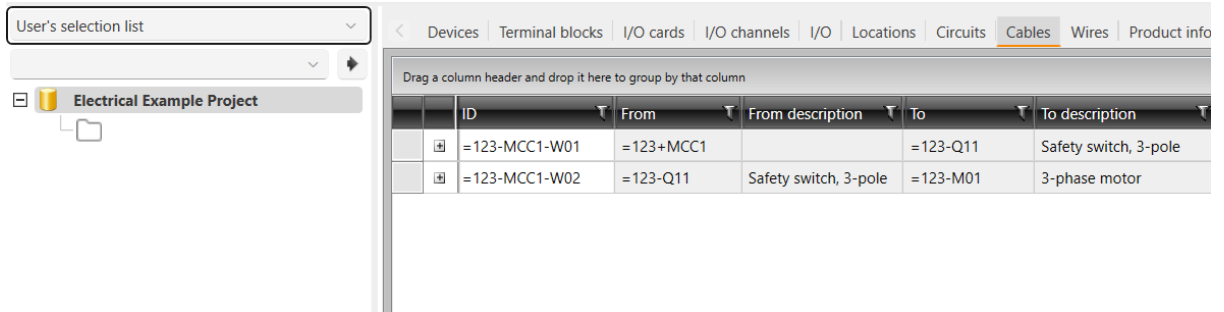
- In the drawing, select the cable markings **=123-MCC1-W01** and **=123-MCC1-W02** by clicking the cable marking symbols.
- Right-click, and select **Add items to user's selection list**.



- Go back to the database tool, and select **Home > Refresh**.

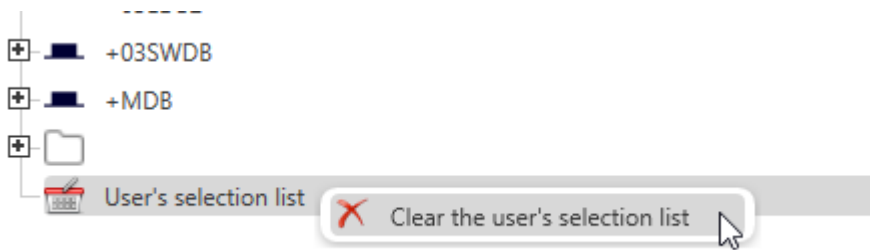
4. Select the **Cables** tab.
5. From the drop-down menu, select **User's selection list**. Alternatively, find the selection list at the end of the project tree and click it.

The grid shows the two cable items added to the selection list.

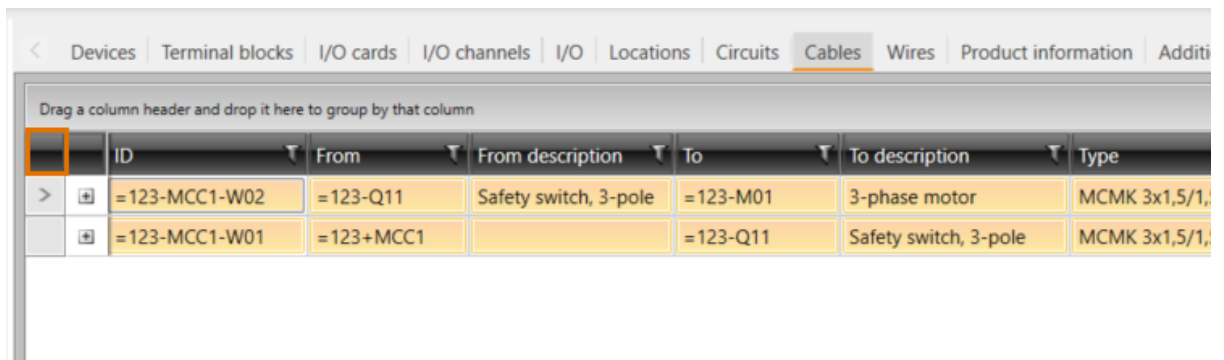


Now you can, for example, view and mass edit these cables.

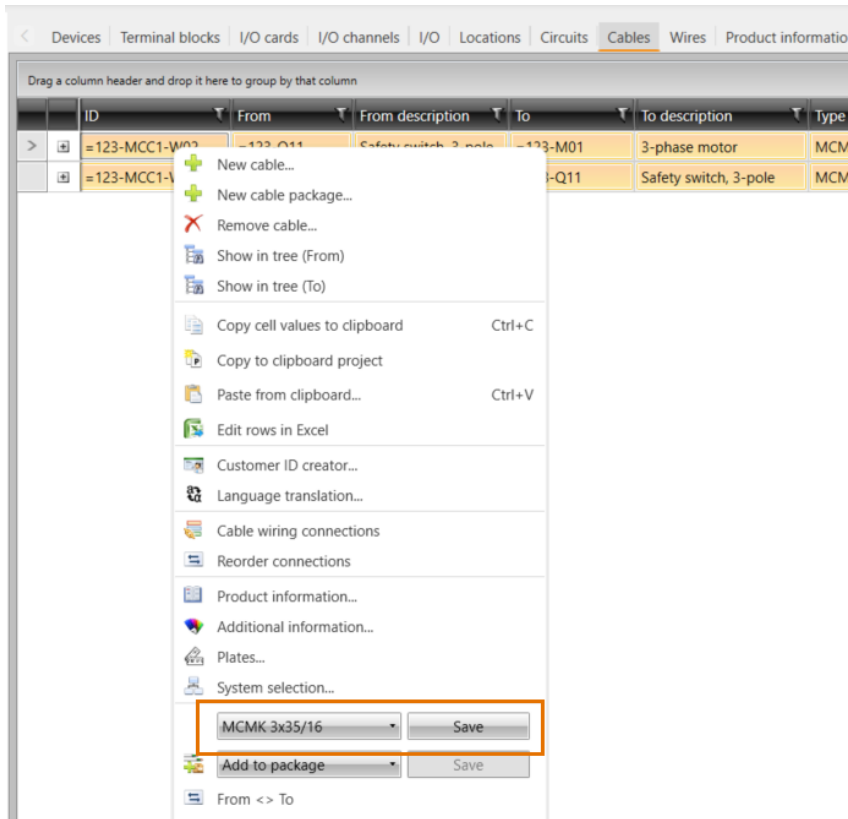
You can easily clear the whole selection list by right-clicking it in the project tree view and selecting **Clear the user's selection list**.




6. Select both of the cables in the grid with the Ctrl or Shift key, or click the left-most black header row cell to select all items in the grid:

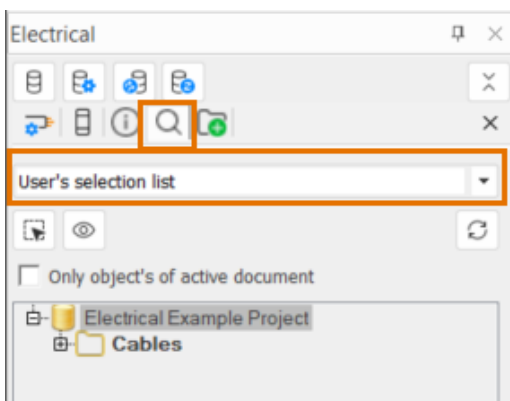


- Right-click, and select some other cable type for these cables (similar cable recommended). In this example, we select **MCMK 3x35/16** because it has the same number of wires. You can also add more suitable cables to your project.



- Go back to the drawing side, and synchronize it with the  (**Sync changes from database to drawing**) button. Cable types are changed, and wiring markings are updated.

You can add different types of items to the user's selection list anytime you want. You can also view the user's selection list via the **Electrical** window:



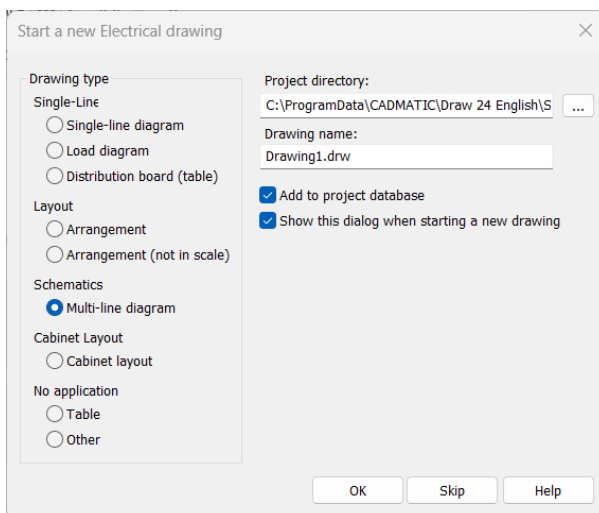
## 4.3. Create, edit and remove documents


**Important:** Never rename project drawings in Windows' File Explorer as it must be done in document management. The same applies to deleting documents from the project.

### 4.3.1. Create a document

Do the following:

1. On the drawing side, create a new document by selecting **Electrical** tab > **Documents** group > **New**.
2. Use the following definitions:
  - **Drawing type** – Multi-line diagram
  - **Project directory** – The same as where your project is currently
  - **Add to project database** – Always select this to link the document to the database.




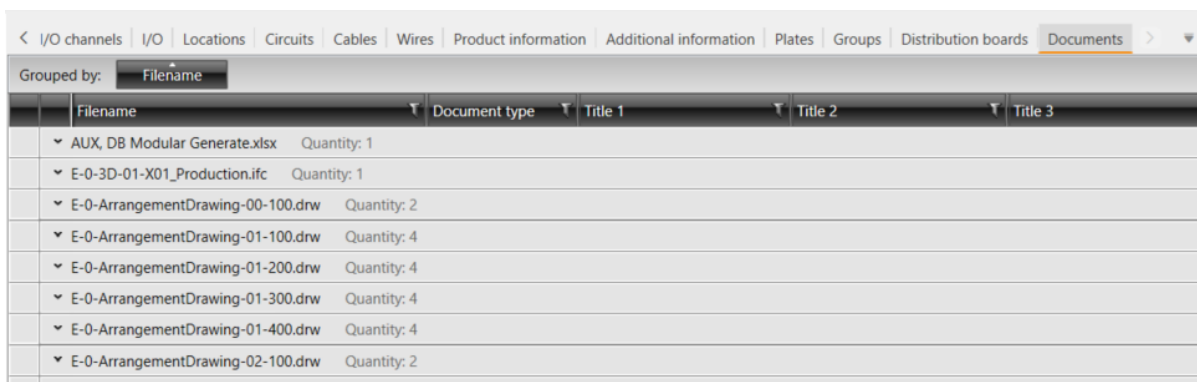
**Tip:** If you forget to add the document to the database when creating a new drawing, you can add it later by clicking the  (**Add document to project**) button in the **Electrical** window.

3. Click **OK**. The default frame is automatically added.  
You have now created the new drawing, and added it to the project. You can now start designing.
4. Save and close the document.

## 4.3.2. Edit a document

Do the following:

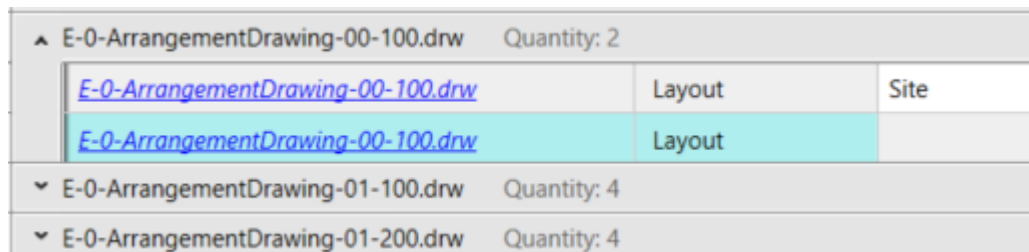
1. In the **Electrical** window, click the  (**Open DB Tool**) button to open the database tool.
2. Select the **Documents** tab.
3. Drag and drop the **Filename** column header to the grouping row if it is not there already.



Filename	Document type	Title 1	Title 2	Title 3
▼ AUX, DB Modular Generate.xlsx	Quantity: 1			
▼ E-0-3D-01-X01_Production.ifc	Quantity: 1			
▼ E-0-ArrangementDrawing-00-100.drw	Quantity: 2			
▼ E-0-ArrangementDrawing-01-100.drw	Quantity: 4			
▼ E-0-ArrangementDrawing-01-200.drw	Quantity: 4			
▼ E-0-ArrangementDrawing-01-300.drw	Quantity: 4			
▼ E-0-ArrangementDrawing-01-400.drw	Quantity: 4			
▼ E-0-ArrangementDrawing-02-100.drw	Quantity: 2			

Files and their frames/labels are arranged together.

- The blue row represents the whole document.
- The gray row represents document's frame/label.



▲ E-0-ArrangementDrawing-00-100.drw	Quantity: 2		
E-0-ArrangementDrawing-00-100.drw	Layout	Site	
E-0-ArrangementDrawing-00-100.drw	Layout		
▼ E-0-ArrangementDrawing-01-100.drw	Quantity: 4		
▼ E-0-ArrangementDrawing-01-200.drw	Quantity: 4		

4. Right-click the blue row, and select **Rename**.
5. Enter a new name, and click **OK**.

## 4.3.3. Remove a document

Do the following:

1. On the **Documents** tab, right-click a blue row and select **Remove document**. The **Remove document from project** dialog opens.
2. Select from the following options:

- Delete the drawing from the project, and clean objects from the database.
- Delete the drawing from the project and the project directory. Additionally, clean object data from the database. This option is recommended.

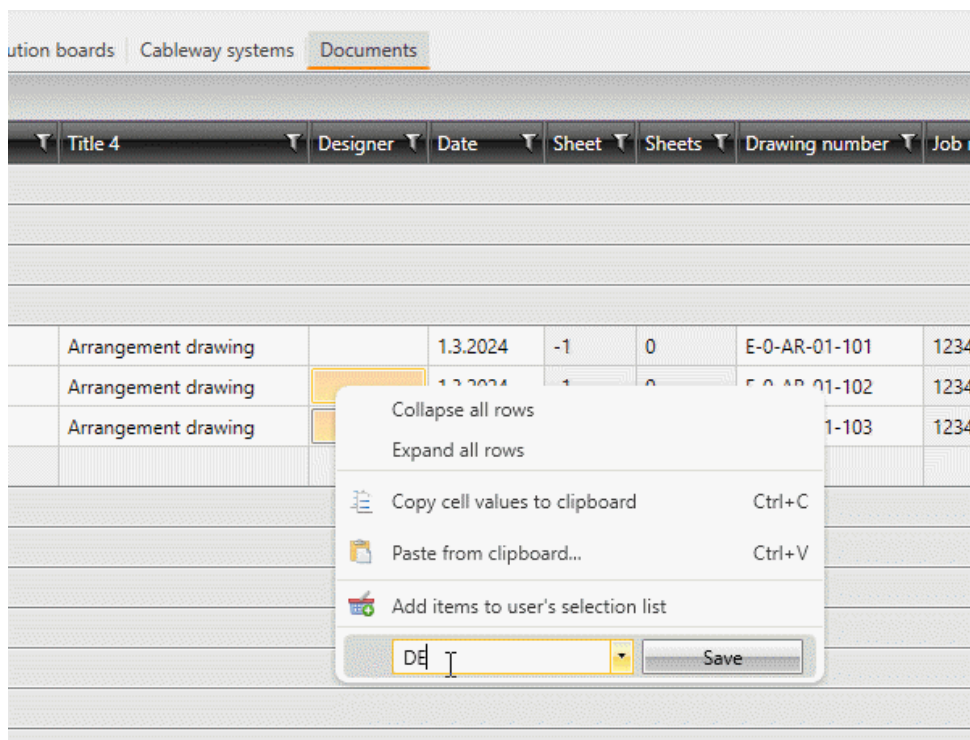
3. Click **Remove**.


## 4.4. Mass edit drawing frame/label information

On the **Documents** tab, you can also mass edit drawing frames and labels. You can select several cells with Ctrl or Shift. For example, you can add the designer to multiple rows at the same time.

**Do the following:**

1. In the **Designer** column, select multiple cells with Ctrl or Shift.
2. Right-click on the selected cells.
3. Enter the value, and click **Save**.

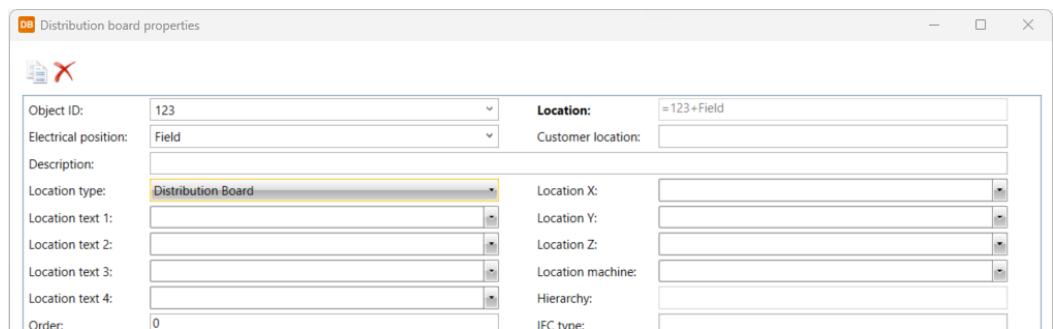


After this you can update the drawing with the  (**Sync changes from database to drawing**) button to update values to frames/labels.

## 4.5. Create new objects in the database

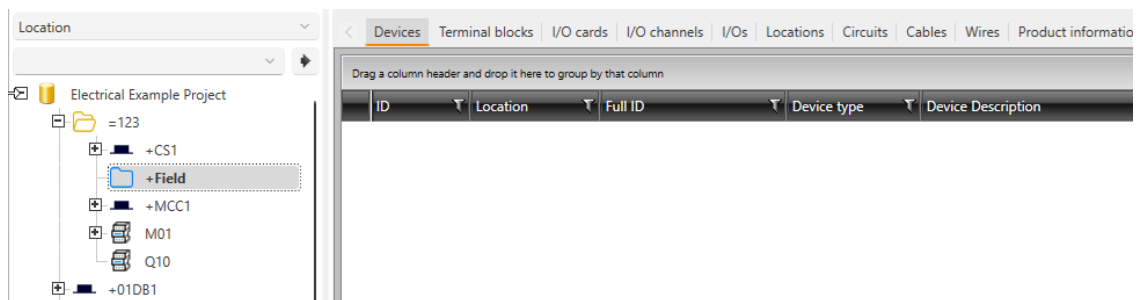
Do the following:

1. Create a new location:
  - a. Select the **Locations** tab in the database tool.
  - b. Right-click on the grid, and select **New location**. The **New location** dialog opens.
  - c. Define the following information in this order:
    - **Electrical position** – Field
    - **Object ID** – 123
    - **Location type** – Distribution Board



2. Click **OK**. The new location is now shown in the **Locations** grid.
3. Create a new device:
  - a. Select the **Devices** tab.
  - b. Find the new location from the project tree, and click it.

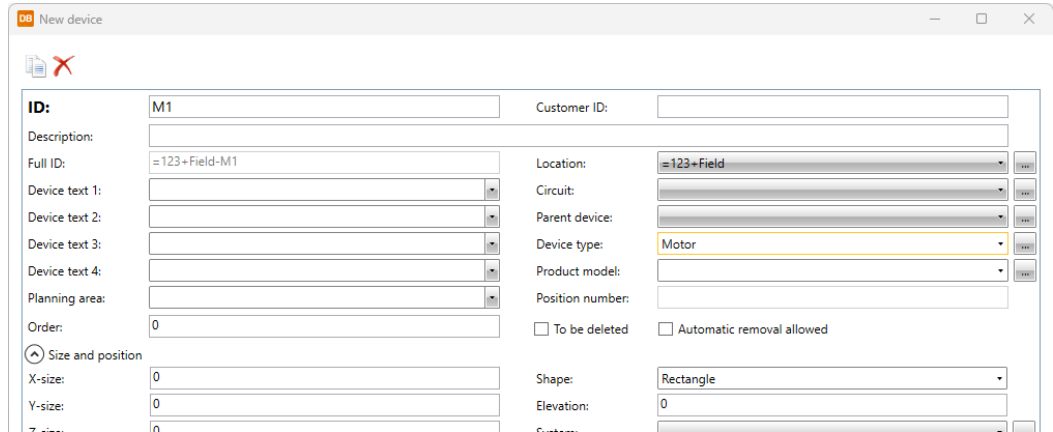
There are no devices in this location since it was newly created.



- c. Add a new device to this location by right-clicking the empty grid, and selecting **New device**.

d. Define the following information:

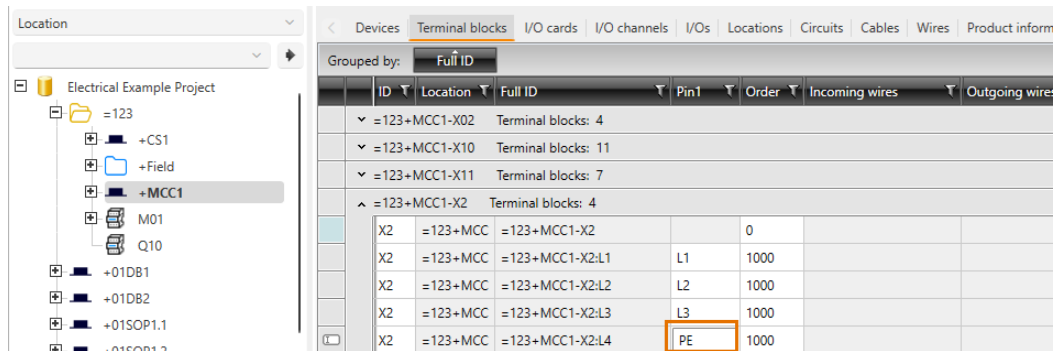
- **ID** – M1
- **Location** – Already correct because we have the =123+Field selected in the tree
- **Device type** (optional) – Motor





e. Click **OK**. The new device is now shown in the device grid.

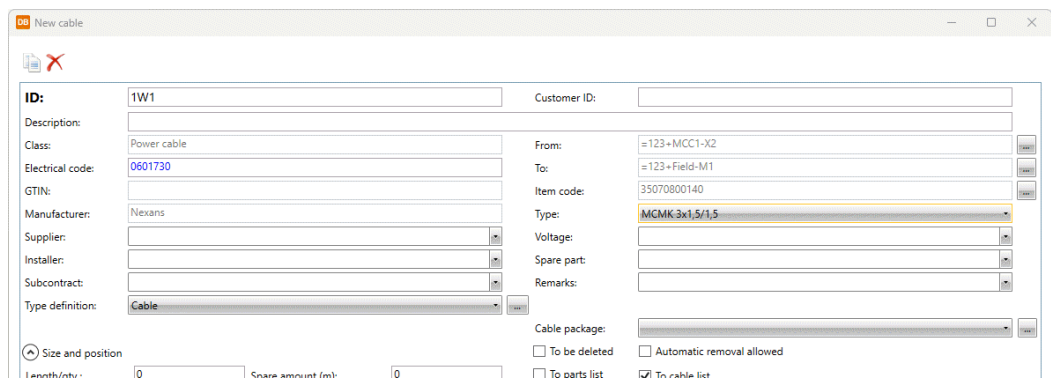
4. Create a new terminal strip:


- a. Select the **Terminal blocks** tab.
- b. Select **=123+MCC1** from the project tree.
- c. Right-click on the grid, and select **New terminal strip**.
- d. Define **X2** as the ID.
- e. Click **OK**.
- f. Add pins to the terminal strip:
  - i. Select and right-click the newly created terminal strip, and select **Generate multiple terminal blocks**.
  - ii. Define **L1** as the starting number and **4** as the quantity, and click **OK**.
  - iii. After generating the blocks, rename the 4<sup>th</sup> block to **PE** in the grid.



5. Select the **Cables** tab.
6. To clear filtering based on the selected location, select the root of the project tree.
7. Create a new cable:
  - a. Right-click on the grid, and select **New cable**. The **New cable** dialog opens.
  - b. Define the following information:

- **ID** – 1W1
- **From** (optional) – =123+MCC1-X2; select by clicking the  button
- **To** (optional) – =123+Field-M1; select by clicking the  button
- **Type** – MCMK 3x1,5/1,5; select from the drop-down menu

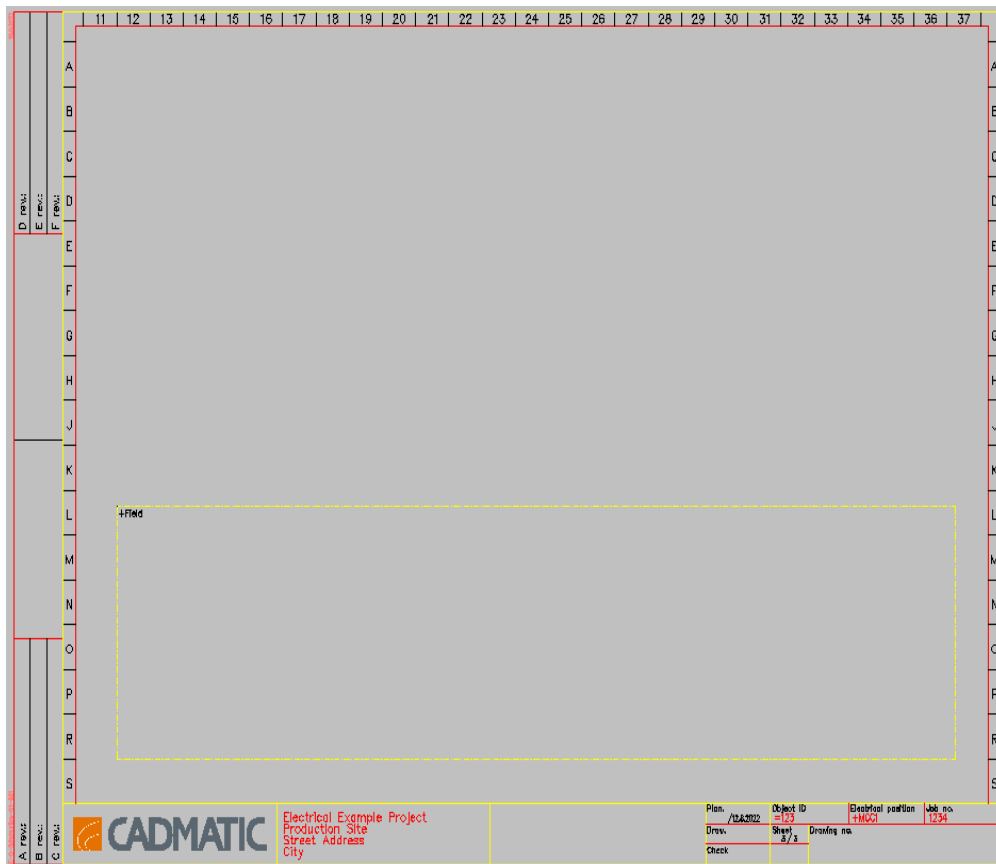


- c. Click **OK**. The new cable is now shown in the cable grid.
8. Move back to the drawing side, and synchronize it with the  (**Sync changes from database to drawing**) button. The objects created are now visible in the project tree.

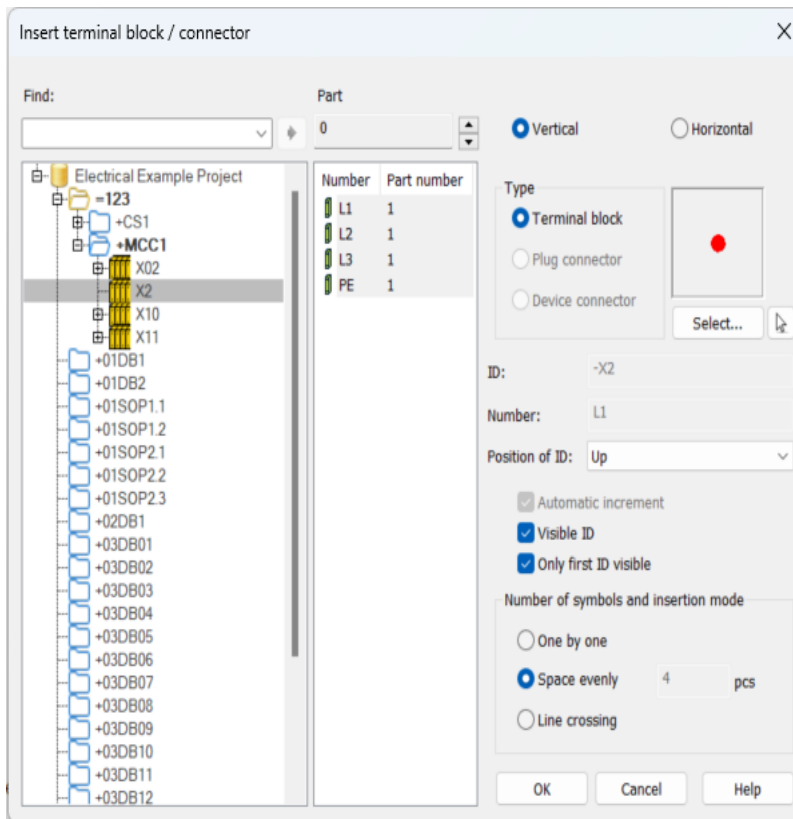
## 4.6. Define and insert objects created in the database

Do the following:

1. Create a new sheet in the *E-0-Schematics-01-501.drw* drawing by going, for example, to the second sheet and pressing Ctrl + PgUp.
2. Define a location to the frame:
  - a. In the **Electrical** window, select **Devices (location)** from the uppermost drop-down menu.
  - b. In the project tree, right-click **+MCC1** and select **Set data of this location to the drawing frame, location boundary or marking**.
  - c. Select the drawing frame, and press Enter. The frame's location information is automatically updated to **=123+MCC1**.
3. Insert a location boundary to the drawing:
  - a. In the project tree, right-click **+Field** and select **Draw a location boundary into drawing**.
  - b. Draw a large boundary to the bottom part of the drawing.



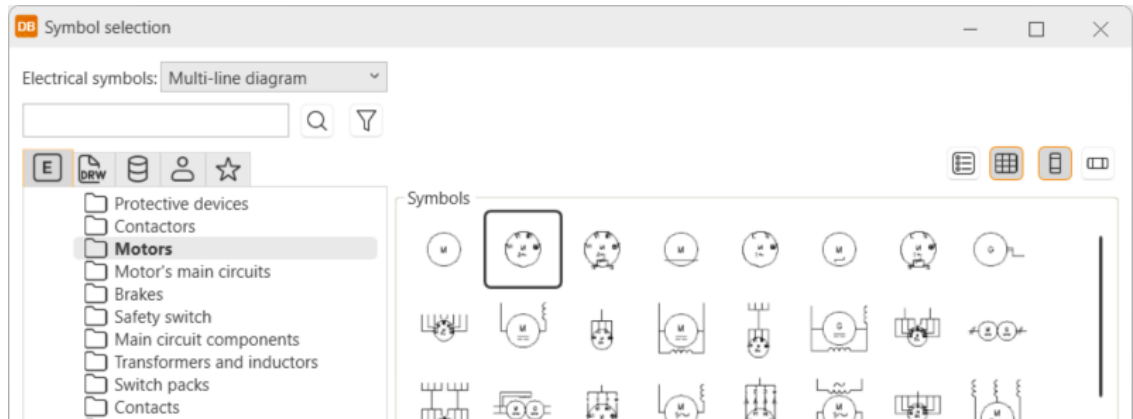
4. Insert terminal blocks to the drawing:
  - a. In the **Devices (location)** project tree, search for the **X2** terminal strip either below location **=123+MCC1** or with the search function.
  - b. Right-click it, and select **Insert terminal blocks of this strip in drawing**.
  - c. Define the settings as follows:
    - Select the desired pins from the middle section with Shift or Ctrl.
    - Select **Vertical**.
    - **Type** – Terminal block
    - **Position of ID** – Up
    - Select **Visible ID**.
    - Select **Only first ID visible**.
    - **Number of symbols and insertion mode** – Space evenly, 4



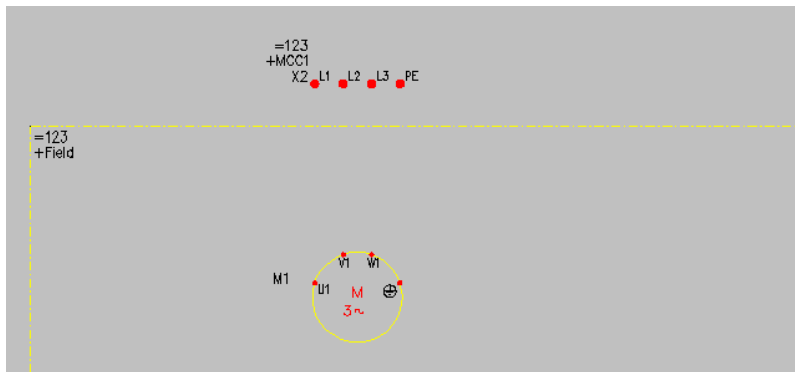
- d. Click **OK**.
- e. Insert the terminal blocks to the drawing above the location boundary. Leave the distance between terminal blocks as 7 (raster jump with 3.5 x 2).

5. Insert a device:

- a. In the project tree, search for the device **M1** below **+Field**.
- b. Right-click, and select **Insert an occurrence of this device into drawing**. The **Insert symbol** dialog opens. The device does not have a symbol yet.
- c. Click **Other symbol**. The **Symbol selection** dialog opens.
- d. Select **Motors** and then the 3-phase motor:



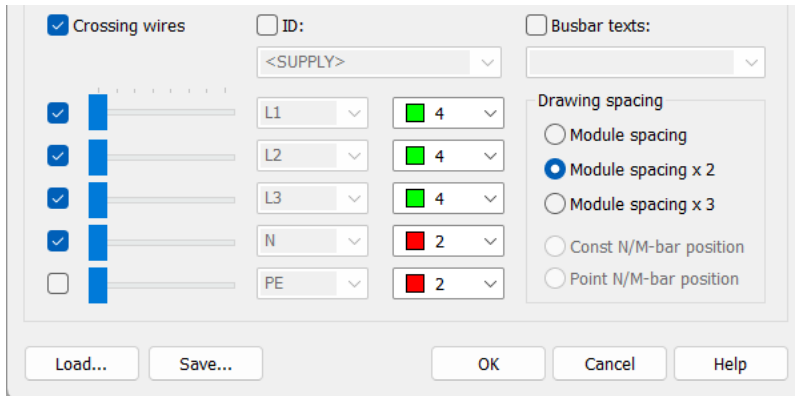
- e. Click **OK**.
- f. Insert the motor symbol inside the **+Field** location boundary, right below the terminal blocks so that the terminal blocks and motor terminals align.



The **Symbol attributes** dialog opens.

- g. Click **OK**.
6. Draw a busbar between the terminal strip and the motor:
- a. Select **Schematics** tab > **Wiring** group > **Draw busbars**.

b. Define the settings as follows:



c. Click **OK**.

d. Draw the busbar by selecting **L1** terminal as the first point, and go down to connect to motor **M1** as the second point.

e. Delete excess wires below **L2** and **L3**.

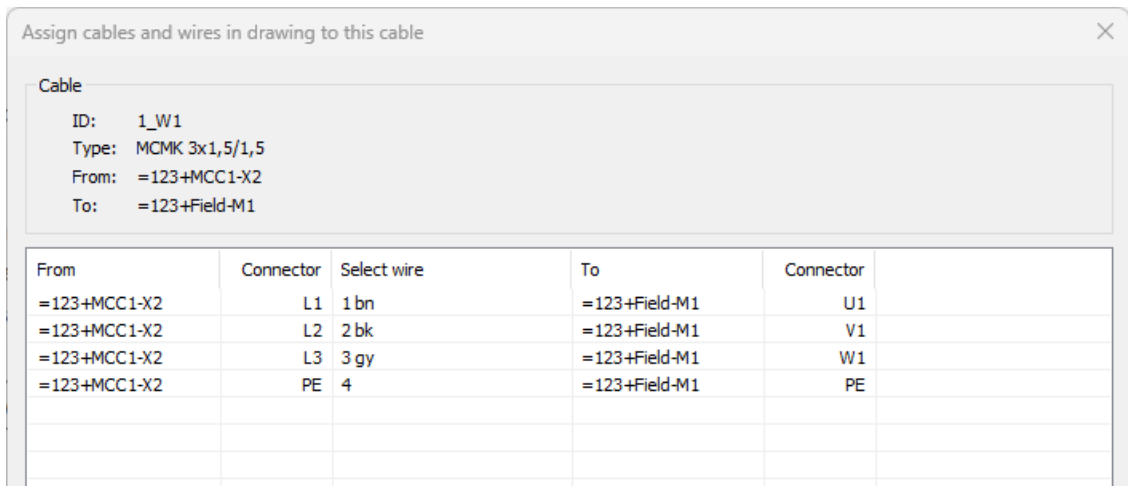
7. Define the cable and its wires in the drawing:

a. In the **Electrical** window, select **Wirings** from the uppermost drop-down menu.

b. Find **1W1**, right-click it and select **Connect and mark cable and its wires**.

c. Select the first point to be to the right of the busbar, then draw the line across to the left side.

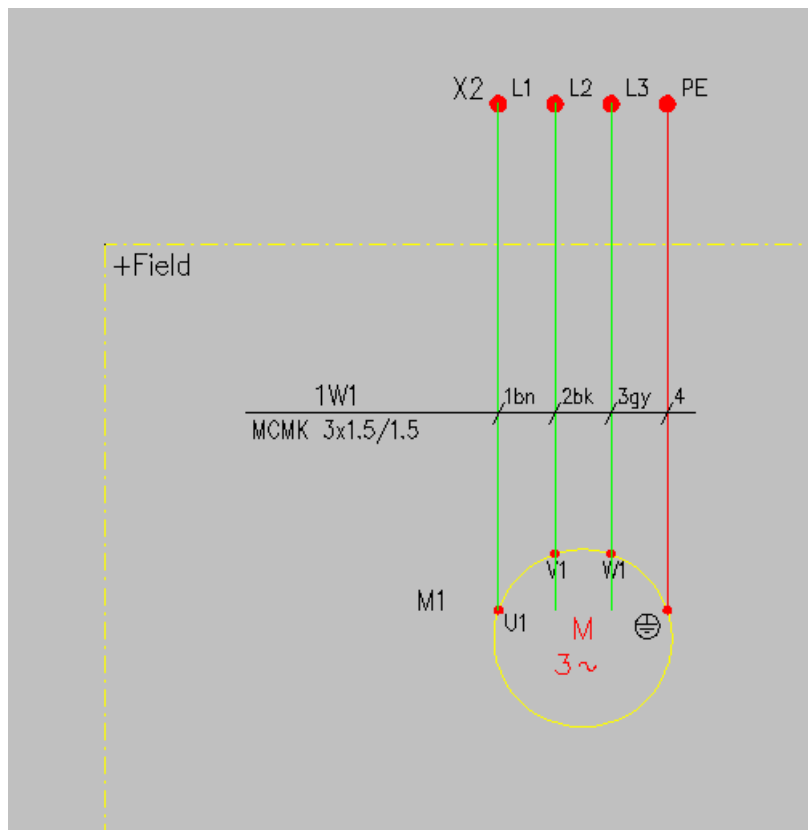
d. Accept the wire connections in the connection dialog.



**Note:** Wire connections are done automatically. If there are terminal blocks with pin number PE or N, they are automatically connected to the PE and N wires. Wire

connections can be changed in this dialog if the suggestions are not desirable.

---



## 4.7. Mass edit in Excel

Do the following:

1. In the database tool, select the **Cables** tab.
2. From the project tree, select location =123+MCC1. Only the cables leaving from the distribution board +MCC1 are shown in the grid.
3. Click the left-most black cell on the header row to select all items in the grid.

ID	From	From description	To	To description	Type
=123-MCC1-W26	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5
=123-MCC1-W25	=123+MCC1		=123+CS1	CONTROL SYSTEM	JAMAK 4x(2+1)x0,5
=123-MCC1-W10	=123+MCC1		=123-Q11	Safety switch, 3-pole	MMO 7x1,5
=123-MCC1-W11	=123+MCC1		=123-M01	3-phase motor	MMO 7x1,5
=123-MCC1-W01	=123+MCC1		=123-Q11	Safety switch, 3-pole	MCMK 3x35/16
=123-MCC1-W22	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5
=123-MCC1-W21	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 8x2x0,5+0,5
#1_W1	=123+MCC1-X2		=123+Field-M1		MCMK 3x1,5/1,5

Alternatively, select the cables with Ctrl or Shift.

4. Right-click on the grid, and select **Edit rows in Excel**. Excel opens.
  - Grey color means the value cannot be edited; usually editing is then done in the grid with the right-click menu options.
  - White color means the value can be edited.
5. In the first **ID** cell, enter the value *MCC1-W1*, and then from the cell's bottom right corner drag the green square to the last ID in the **ID** column. Excel changes the IDs as follows:

ID	From	From description	To	To description	Type	Supplier	Installer	Remarks	Product info
MCC1-W1	=123+MCC1		=123-Q10	Safety switch, 3-pole	MCMK 3x1,5/1,5				
MCC1-W2	=123+MCC1		=123-Q10	Safety switch, 3-pole	MMO 7x1,5				
MCC1-W3	=123+MCC1		=123-M01	3-phase motor	MMO 7x1,5				
MCC1-W4	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 8x2x0,5+0,5				
MCC1-W5	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5				
MCC1-W6	=123+MCC1		=123+CS1	CONTROL SYSTEM	JAMAK 4x(2+1)x0,5				
MCC1-W7	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5				
MCC1-W8	=123+MCC1-X2		=123+Field-M1		MCMK 3x1,5/1,5				

6. Minimize Excel without closing it, and go back to the database tool.
7. Right-click on the grid, and select **Import rows from Excel** to import back the edited data. The changed IDs are updated in the grid:

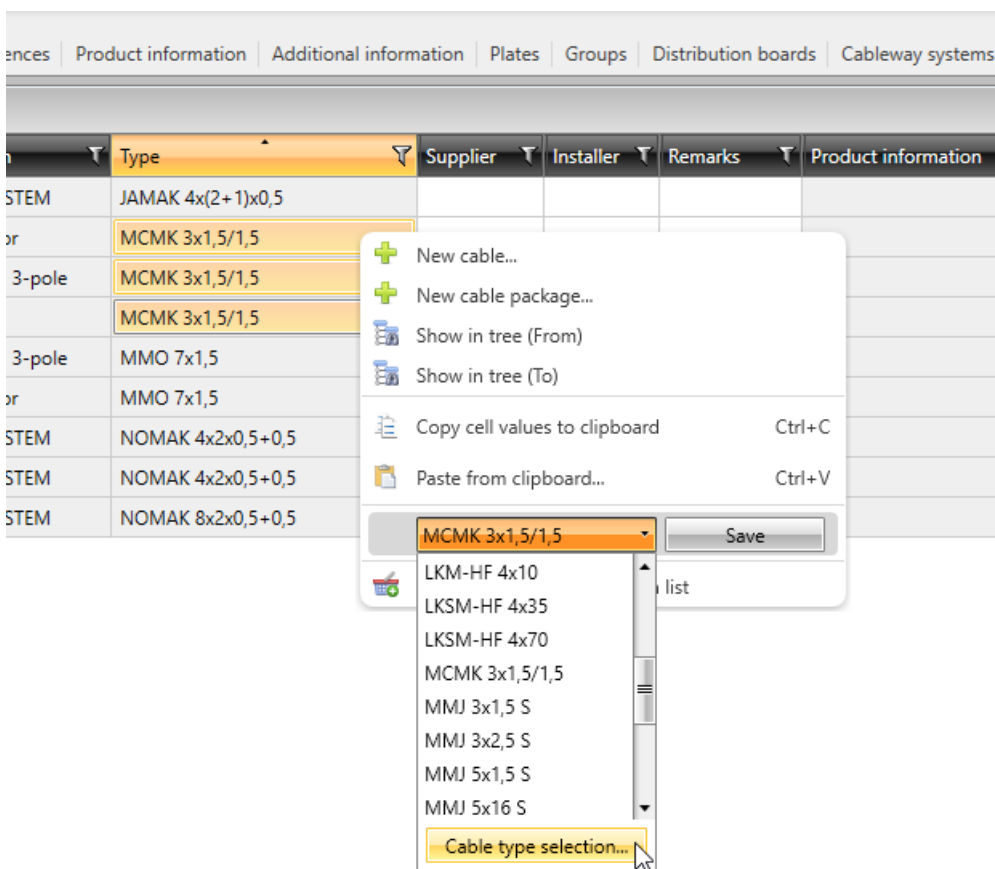
Drag a column header and drop it here to group by that column						
	ID	From	From description	To	To description	Type
>	MCC1-W1	=123+MCC1		=123-Q10	Safety switch, 3-pole	MCMK 3x1,5/1,5
	MCC1-W2	=123+MCC1		=123-Q10	Safety switch, 3-pole	MMO 7x1,5
	MCC1-W3	=123+MCC1		=123-M01	3-phase motor	MMO 7x1,5
	MCC1-W4	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 8x2x0,5+0,5
	MCC1-W5	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5
	MCC1-W6	=123+MCC1		=123+CS1	CONTROL SYSTEM	JAMAK 4x(2+1)x0,5
	MCC1-W7	=123+MCC1		=123+CS1	CONTROL SYSTEM	NOMAK 4x2x0,5+0,5
	MCC1-W8	=123+MCC1-X2		=123+Field-M1		MCMK 3x1,5/1,5


8. Open the *E-0-Schematics-01-501.drw* drawing, and synchronize it with the **Sync changes from database to drawing** button to see the changed IDs.

## 4.8. Mass edit in the grid

Do the following:

1. In the database tool, select the **Cables** tab.
2. In the project tree, select =123.
3. Find the **Type** column.
4. Select all **MCMK** cable cells with Ctrl one by one.
5. Right-click and select **Cable type selection** from the drop-down menu:



6. Select the **Standard databases** tab.
7. Use the global filter in the top right corner to find **MCMK 3x2,5/2,5**.
8. Double-click **MCMK 3x2,5/2,5** with item code **0600123**. The cable types change in the grid.
9. Synchronize the drawing with the  (**Sync changes from database to drawing**) button, and view the changed cable markings.

## 4.9. Copy from a clipboard project

With a temporary clipboard project, you can easily copy items (such as drawing files, sheets, devices, circuits, and locations) in your project or from one project to another. You can define rules for copying objects and generating new IDs.

### Do the following:

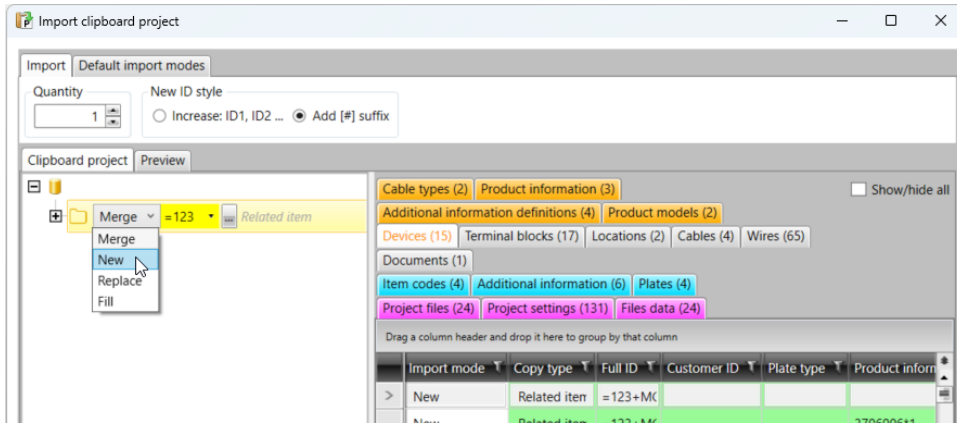
1. Copy the first sheet of the *E-0-Schematics-01-501.drw* drawing:
  - a. In the database tool, select the **Documents** tab.
  - b. Drag-and-drop the **Filename** column header to the grouping row.
  - c. Find and select the 1<sup>st</sup> sheet row of *E-0-Schematics-01-501.drw*.

Filename	Document type	Title 1	Title 2	Title 3	Title 4	Designer	Date	Sheet	Sheets	Drawing
▼ AUX_DB Modular Generate.xlsx	Quantity: 1									
▼ Drawing11_11_edited.drw	Quantity: 2									
▼ E-0-3D-01-X01_Production.ufc	Quantity: 1									
▼ E-0-ArrangementDrawing-00-100.drw	Quantity: 2									
▼ E-0-ArrangementDrawing-01-100.drw	Quantity: 4									
▼ E-0-ArrangementDrawing-01-200.drw	Quantity: 4									
▼ E-0-ArrangementDrawing-01-300.drw	Quantity: 4									
▼ E-0-ArrangementDrawing-01-400.drw	Quantity: 4									
▼ E-0-ArrangementDrawing-02-100.drw	Quantity: 2									
▼ E-0-CabinetLayout-01-101.drw	Quantity: 2									
▼ E-0-DistributionBoardTable-01-005.drw	Quantity: 4									
▼ E-0-DistributionBoardTable-01-006.drw	Quantity: 3									
▼ E-0-LoadDiagram-01-01.drw	Quantity: 2									
▼ E-0-Report-00-001_Document list.xlsx	Quantity: 2									
▼ E-0-Report-00-002_Lighting fixtures list.xlsx	Quantity: 2									
▼ E-0-Report-00-003_Cables list.xlsx	Quantity: 2									
▼ E-0-Report-00-004_Cable connection list.xlsx	Quantity: 2									
▼ E-0-Report-00-005_Cable Name Plates.xlsx	Quantity: 2									
▼ E-0-Schematics-01-501.drw	Quantity: 3									
E-0-Schematics-01-501.drw	Schema							0	0	
E-0-Schematics-01-501.drw	Schema	Motor-operated valve	Incoming water	Wiring- and circuit diagram			13.2024	1	2	E-0-SC-0
E-0-Schematics-01-501.drw	Schema	Motor-operated valve	Incoming water	Wiring- and circuit diagram			13.2024	2	2	E-0-SC-0
▼ E-0-SingleLine-00-001.drw	Quantity: 2									

- d. Right-click the row, and select **Copy to clipboard project**. The **Copy to clipboard project** dialog opens. The **Export** tab shows all the selected documents and their information. If necessary, you can change the export settings on the **Settings** tab.
  - e. Click **Copy** to copy the information to the clipboard project. The dialog closes when copying is ready.
2. Select **Functions** tab > **Import** menu > **Import clipboard project**.  
 The **Import clipboard project** dialog opens. The dialog shows the objects you are about to import to your project.
  3. Define **2** as the quantity.
  4. As the new ID style, select **Add [#] suffix**.

5. Select the import mode:

- **New** – New objects are created from the clipboard objects. Identical IDs are handled according to the **New ID style** option.
- **Merge** – Objects with identical IDs are merged. For example, **MCC1** stays as **MCC1**.



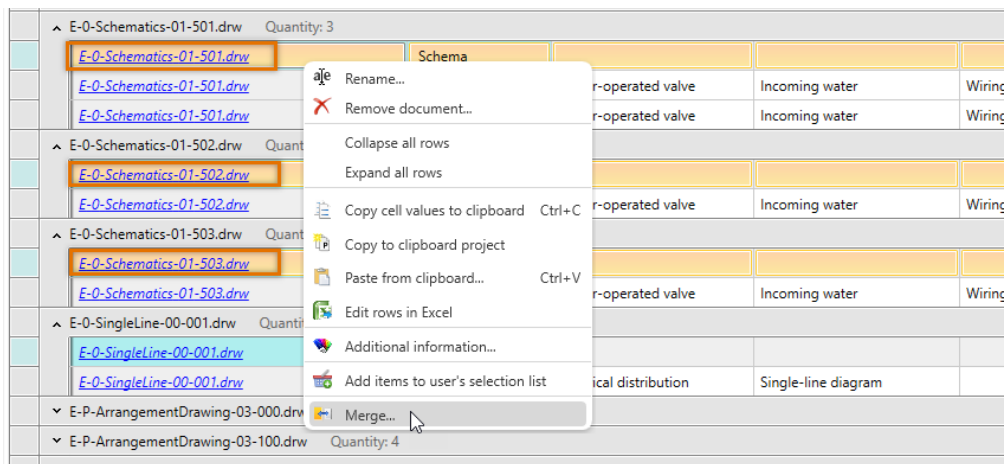
For more information on the other modes (rarely used in this case), see the online help.

6. Click **Continue**.

7. Preview the changes, and then click **Import**. New documents *E-0-Schematics-01-502* and *E-0-Schematics-01-503* are added to the project.

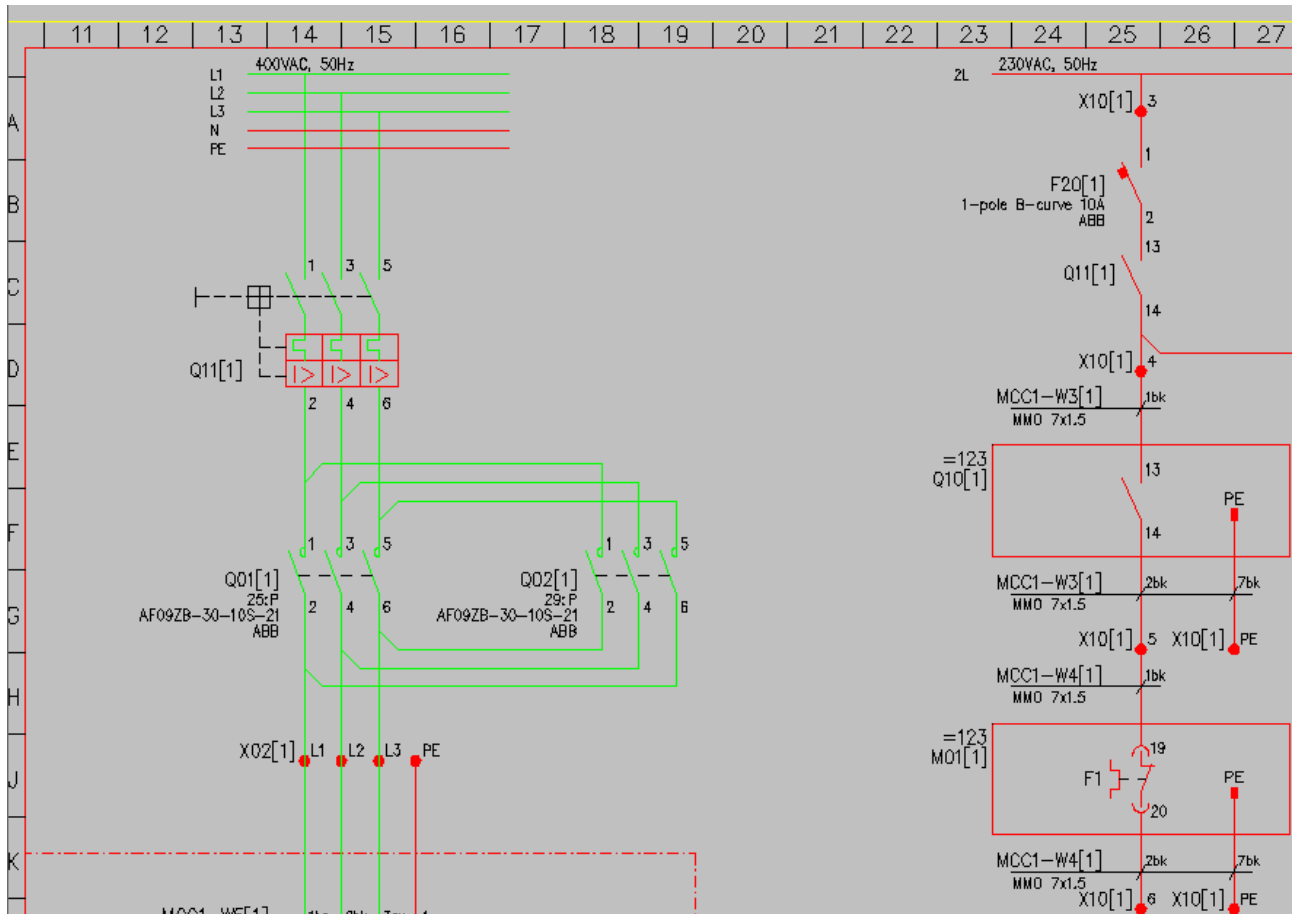
8. Merge the documents:

- If the *E-0-Schematics-01-501.drw* file is open on the drawing side, close it.
- Select the *E-0-Schematics-01-502.drw* file (blue row, first copy).
- Press Ctrl, and select the *E-0-Schematics-01-503.drw* file (blue row, second copy).
- With Ctrl still down, select the *E-0-Schematics-01-501.drw* file (blue row, original).
- Right-click, and select **Merge**.



- f. In the **Merge objects** dialog, click **Merge**.
- g. When the application asks about deleting the 0 layer, click **Yes**. The files are now merged, and you can close the dialog with **Close**.
- h. Open the *E-0-Schematics-01-501.drw* file.

Here is an example of sheet 4 (IDs have [1] at the end because this is the first copy):



## 5. Management tools

Via the **Management** tab, you can manage cable types, cable packages, product information, additional information, plate definitions, product models, and type definitions.

### 5.1. Cable types

The cable type database includes all the available cable types. Cables also have wires defined so they can be used in wiring diagrams. When the cable type is changed, for example, the possible wire markings will also be updated.

Let's create a new cable type to the shared database. Typically, it is much easier to start with an existing cable type so we will first copy an existing one.

**Do the following:**

1. Select **Management** tab > **Cable types**.
2. Select the **Standard databases** tab.
3. Find cable type **MCMK 3x1,5/1,5** with item code **35070800140**.
4. Right-click it, and select **Copy**.
5. Edit the information as follows:

The screenshot shows the 'Cable type properties' dialog box with the following fields and values:

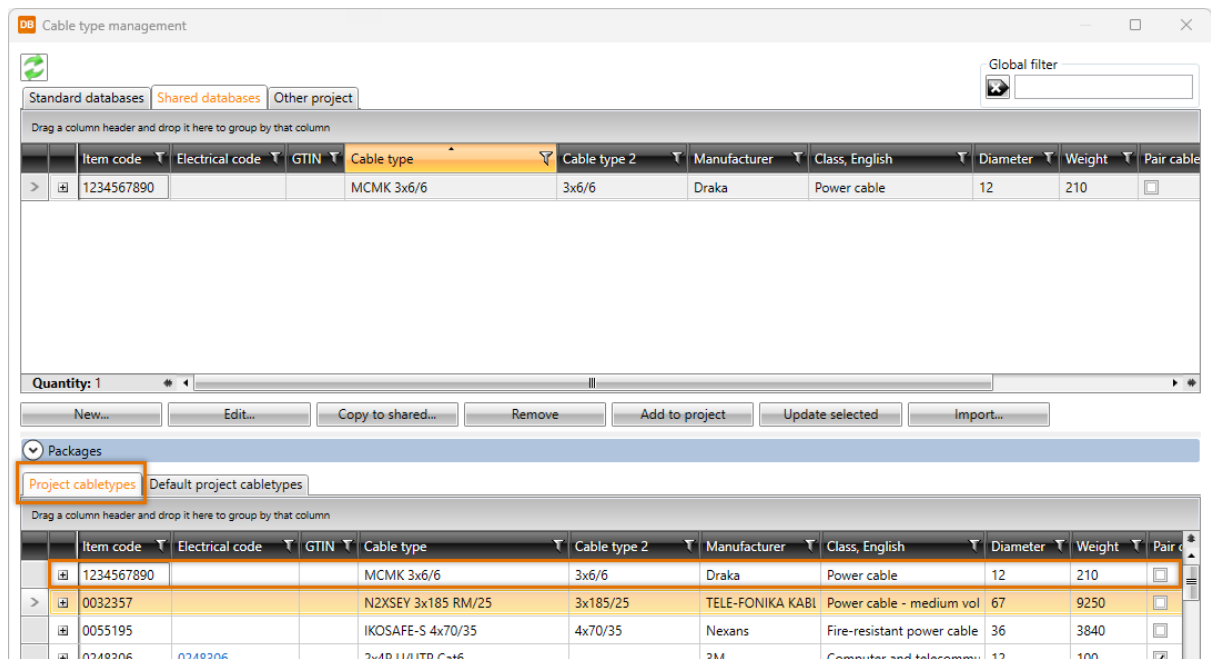
- Item code: 1234567890
- Electrical code: (empty)
- GTIN: (empty)
- ETIM Class: (empty)
- Cable type: MCMK 3x6/6
- Cable type 2: 3x6/6
- Replacing product: (empty)
- Manufacturer: Draka
- Class: Power cable
- Diameter: 12
- Weight: 210
- Specifications 1: 0,6/1 kV
- Specifications 2: -15...+70°C
- Specifications 3: (empty)
- Specifications 4: (empty)
- Plan unit: (empty)
- Position number: (empty)
- Position symbol: (empty)
- Options:  Pair cable  Wire set  Dual-cable

The 'Wires' table is as follows:

Item code	Cable wire index	Wire number	Cross section	Material	Signal type	Color index	Color, English	Product item code	Product reference
1234567890	1	1	6	Cu	SIG	Brown	bn		
1234567890	2	2	6	Cu	SIG	Black	bk		
1234567890	3	3	6	Cu	SIG	Grey	gy		
1234567890	4	4	6	Cu	PE	No color			

6. Click **OK**. The cable type is added to the shared database.

7. Drag and drop the cable type from the shared database to the project database.



The cable type is now available in wiring/cabling functions.

## 5.2. Cable packages

In cable package management, you can create sets including the desired cable types. They can then be used in arrangement or schema drawings via the wiring/cabling functions. The application does not include any default cable packages.

Let's create a cable package to the shared database.

**Do the following:**

1. Select **Management** tab > **Cable packages**.
2. Select the **Shared databases** tab.
3. Right-click the empty space, and select **New**.
4. From the bottom left corner, click **Select/Edit**.
5. Find **AMCMK 3x25/16 AN** and **AMCMK 3x35/16 AN**, add them to the selected types, and click **OK**.
6. Add the cable package to the project. After this its available in wiring/cabling functions.

## 5.3. Product information

In product information management, you can select pre-existing product information or create your own to the database.

Product information management differs from the other management tools a little. The main differences are:

- In **Standard databases**, you can download the desired manufacturer’s databases from the **Product database** drop-down menu.
- In **Shared databases**, you can use different databases to save your own product information, and import data from Excel, for example.

Let's add product information to the shared database, and assign it to devices.

### Do the following:

1. Select **Management** tab > **Product information**.
2. Select the **Shared databases** tab.
3. From the **Product database** drop-down menu, select **User products 1**.
4. Right-click on the grid, and select **New**.
5. Fill in the following information:

The screenshot shows a window titled "DB New product information". It contains the following fields:

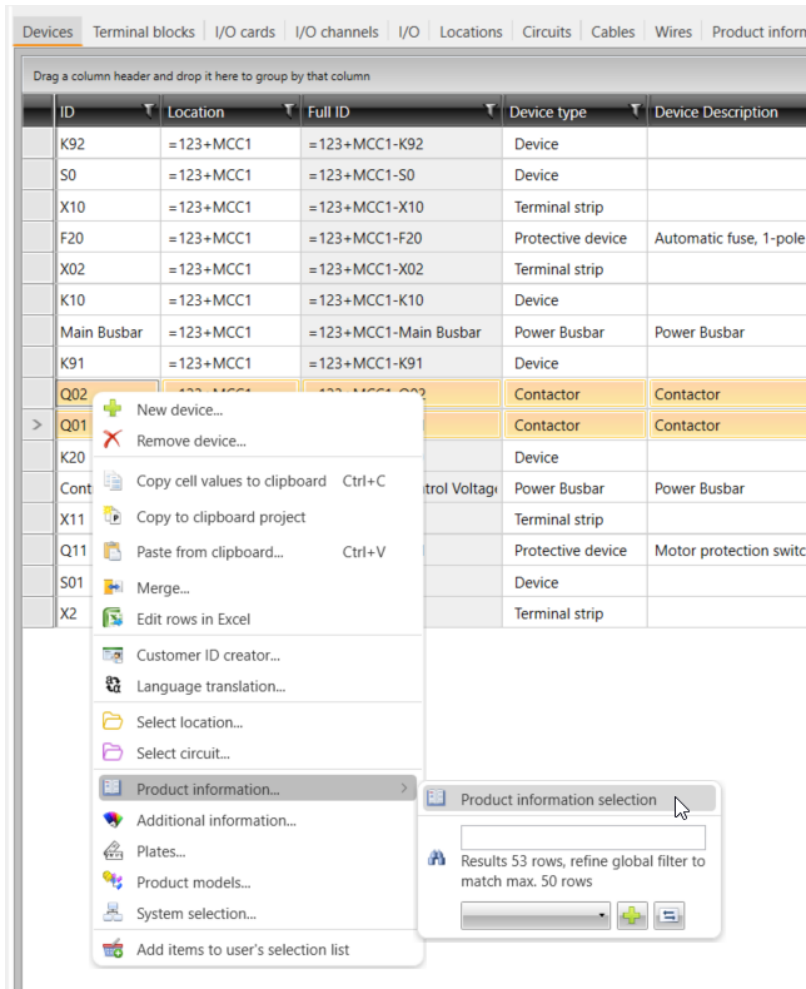
- Product database: User products 1
- Manufacturer: ABB
- Type: AF09ZB-3D-10S-21
- Plan unit: (empty)
- Item code: 1SBL136059R2110
- Name: Contactor 3NO, 25A, 5.5kW
- Power: 5.5kW
- Installation: (empty)
- Product group: Contactor
- Name 2: (empty)
- Elec 1: (empty)
- Product info type: Device

Below the fields is a table with two columns: "Information" and "Value".

Information	Value
Item code	1SBL136059R2110
Electrical code	
Type	AF09ZB-3D-10S-21
Manufacturer	ABB
Power	5.5kW
X-Size	0
Y-Size	0
Z-Size	0
Shape	Rectangle
GTIN	
Specifications 1	
Specifications 2	
Specifications 3	
Specifications 4	

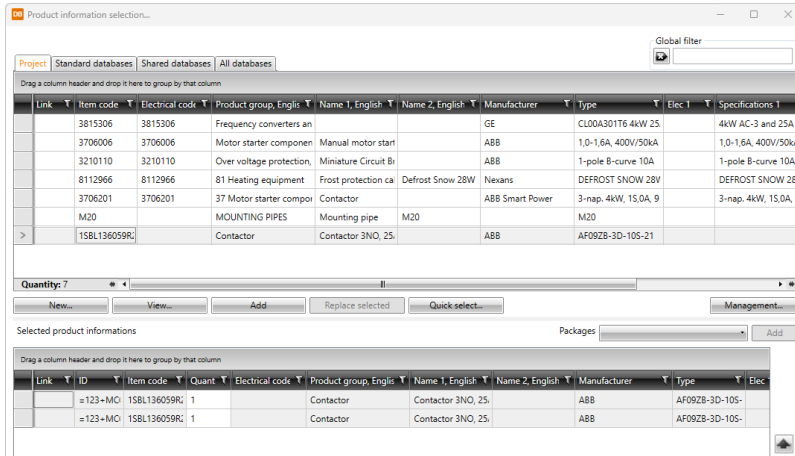
6. Click **OK**.


7. Add the product information to the project, and close the dialog. Now you can assign the product information to devices.
8. Select the **Devices** tab.
9. Filter the grid with location =123+MCC1.
10. Select **Q01** and **Q02**, right-click and select **Product information > Product information selection**.



11. Remove old product information by selecting them from the grid and clicking **Remove**. Confirm the removal with **OK**.

- Assign the newly created product information to the devices by dragging and dropping.




- Click **OK** to close the dialog.
- Open the *E-0-Schematics-01-501.drw* drawing, and synchronize it with the  (Sync changes from database to drawing) button.
- Check that the information for **Q01** and **Q02** is up-to-date.

## 5.4. Additional information

In additional information management, you can add/define data fields for objects, project, labels, etc. You can freely add and mass edit the information in the database tool or on the drawing side.

Let's create a new piece of additional information to the shared database.

### Do the following:

1. Select **Management** tab > **Additional information**.
2. Select the **Shared databases** tab.
3. Right-click on the grid, and select **New**.
4. Select **Value**.
5. Fill in the following information:
  - **Name** – Additional information name
  - **Required value** (optional) – Shown in a separate field as a reference value
  - **Attribute** (recommended) – Name of the attribute where this additional information is shown, for example, in symbols
  - **Value** (optional) – Default value when added to an object
  - **Type** – Used for filtering in grids, for example
  - **Translate** – If the translation tool is used, whether the value is translated or not
  - **Attribute adds additional information** – If the symbol has this additional information's attribute and it is given a value, whether it creates additional information automatically or not
  - Click the  button to select objects for which this additional information is used
  - **Do not add additional info to the object** – When the additional information is given to a product model or product information and this is enabled, you cannot edit the value for

individual objects with selected product model / product information (constant value).

6. Click **OK**.
7. Add additional information just created to the project.

You can edit the values directly in the grid. You can also add and edit additional information by right-clicking and in the object edit dialog.

## 5.5. Plate definitions

In plate definition management, you can define plate definitions that can be assigned for devices, locations, cables, etc. You can then generate a plate report that can be used to print the plates.

Let's create a new plate definition to the shared database.

### Do the following:

1. Select **Management** tab > **Plate definitions**.
2. Select the **Shared databases** tab.
3. Right-click on the grid, and select **New**.
4. Fill in the following information:
  - **Parent type** – Device
  - **Plate type** – Device plate (switch)
  - **X-size** (as mm) – 64
  - **Y-size** (as mm) – 20
  - **Fixing method** – Sticker
  - **Text size** – 3.5
5. From the **Table** drop-down menu, select **Devices**.
6. From the **Field** section, drag and drop **ID** to the first row in the **Plate texts** section.
7. From the **Table** drop-down menu, select **Additional information**.
8. From the **Field** section, drag and drop **Position text 1**, **Position text 2** and **Position text 3** to the second row.
9. Add hyphens (-) between the position texts, as shown in the image:

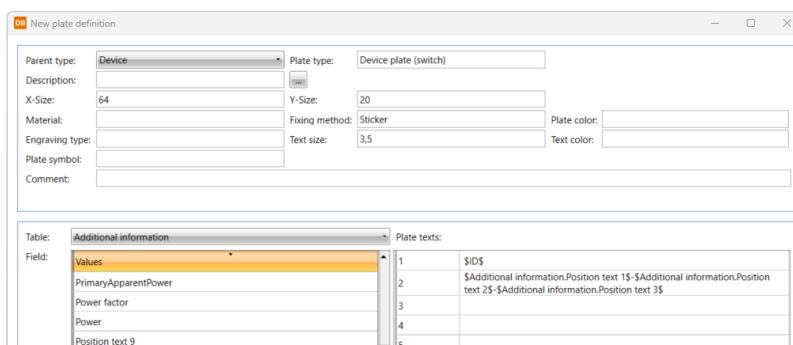
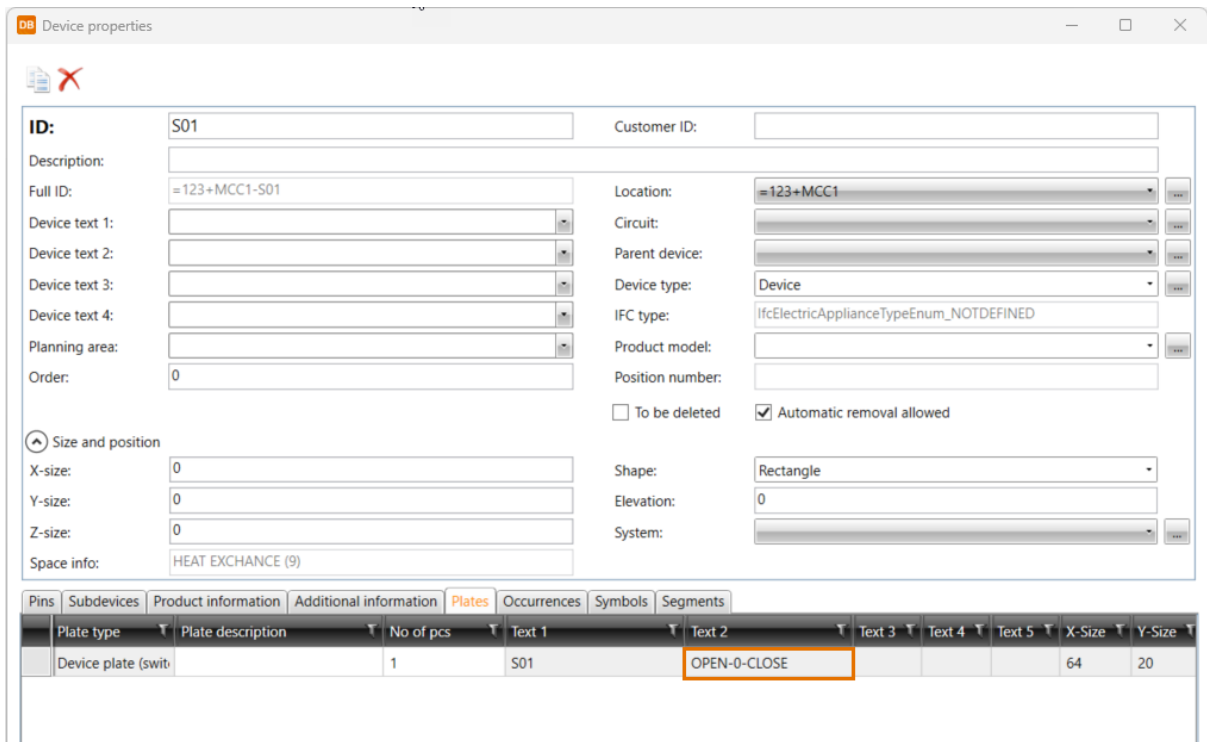


Table:	Field:	Plate texts:
Additional information	Values	1 ID
	PrimaryApparentPower	2 \$Additional information.Position text 1-\$Additional information.Position text 25-\$Additional information.Position text 35
	Power factor	3 ;
	Power	4 ;
	Position text 9	5 ;

10. Click **OK**.

11. Add plate definition to the device:
  - a. Select location **=123+MCC1**.
  - b. Select the **S01** device, right-click it and select **Plates**.
  - c. In Shared databases, add the plate definition by double-clicking.
  - d. Click **OK**.
12. View plate information by editing the device and selecting the **Plates** tab. The device's position text 1 is **OPEN**, position text 2 is **0** and position text 3 is **CLOSE** so the result is **OPEN-0-CLOSE**.



## 5.6. Product models

A product model includes a significant amount of information about a device, such as 2D and 3D symbols, size information, positions, etc. A product model can also be created based on an existing device.

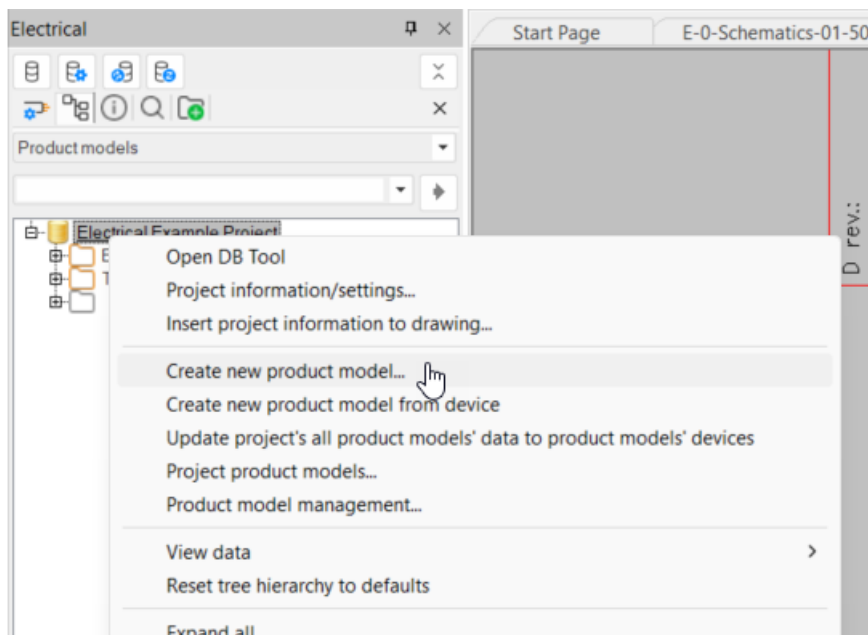
Project product models are available in the **Product models** project tree. In addition, you can open product model management by selecting **Electrical** tab > **Management** group > **Product models**.

### 5.6.1. Create a product model

Let's create a new product model in the *E-0-Schematics-01-501.drw* drawing.

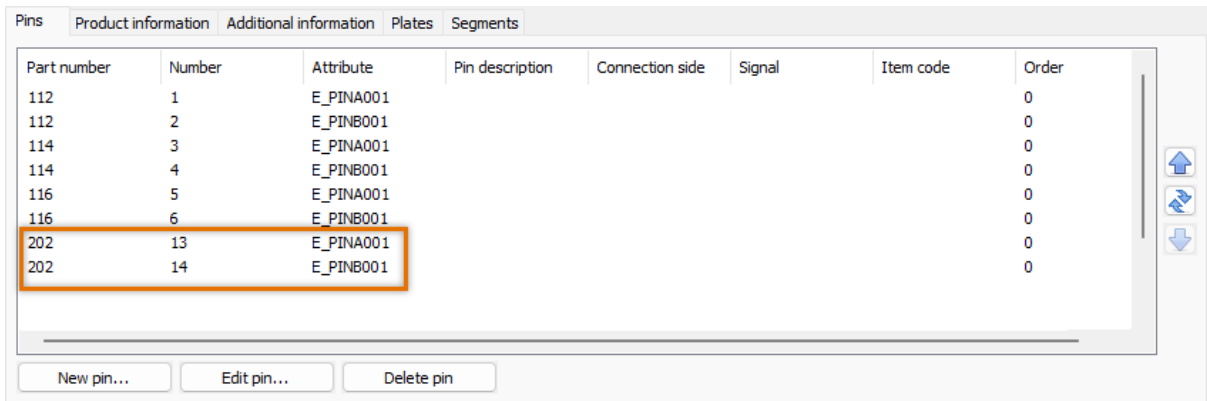
**Do the following:**

1. Select **Product models** from the project tree drop-down menu.
2. Start creating a product model by right-clicking in the tree and selecting **Create new product model**.

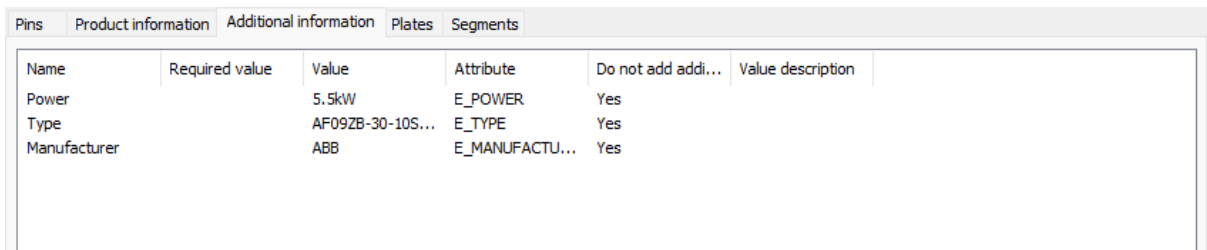


3. In the **Product model** dialog, define the following information:
  - **Name** – Contactor 5.5kW
  - **Device type** – Contactor
  - **X-size** – 45
  - **Y-size** – 91.5

4. Add a symbol:
  - a. Click **Add symbol**, and select **New Multi-line diagram symbol > Vertical** and then **Contactors**.
  - b. Select a 3-phase contactor symbol from the menu.
  - c. Click **Add symbol**, and select **New Multi-line diagram symbol > Vertical** and then **Contacts**.
  - d. Select a normally-open contact.
5. On the **Pins** tab, add pin numbers 13 and 14.



6. On the **Additional information** tab, click **Additional info selection** and add information as in the following image:



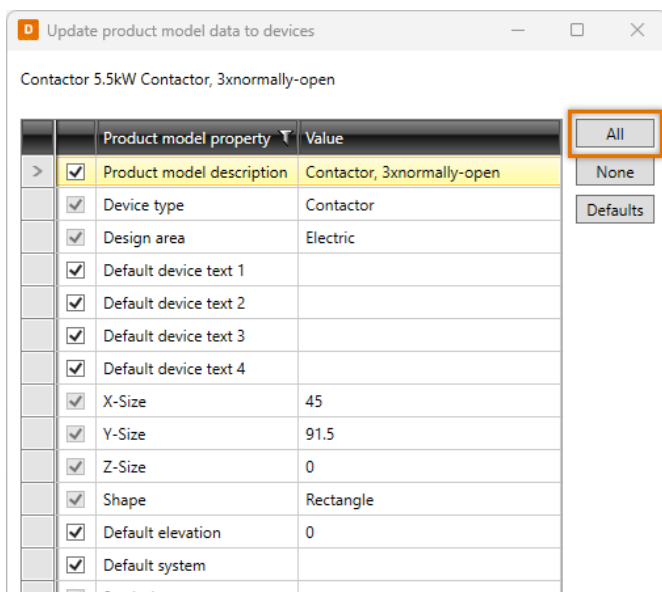
7. Click **OK**. The product model is now available in the product model tree:
8. Go to sheet 3.
9. In the **Product models** project tree, right-click the product model and select **Insert one by one**. The **Insert symbol** dialog opens.
10. Select **Continue inserting same device's symbols** and click **OK**.
11. Insert the first symbol somewhere on the sheet, enter *K01* as the ID, click **OK** and insert the second symbol.

After inserting the symbols, the number of product models update to 1 in the project tree. The project tree is also automatically updated when inserting, editing or deleting product models.

## 5.6.2. Edit the product model

Do the following:

1. Right-click the **Contactor 5.5kW** product model, and select **Edit**.
2. Change the additional information **Type** from **AF09ZB-30-10S-21** to **AF09ZB-3**.
3. Make sure that **Update devices automatically** is set to **Yes**.
4. Click **Close**. The **K01** contactor updates automatically.
5. Insert a new contactor: select **Schematics** tab > **Devices** group > **Symbols** > **Contactors** and then insert the 3-phase contactor by double-clicking.
6. As the ID, enter **K02**.
7. Click **OK**.
8. In the **Product models** tree, right-click **Contactor 5.5kW** and select **Assign devices**.
9. Click **All**, then click **OK**.



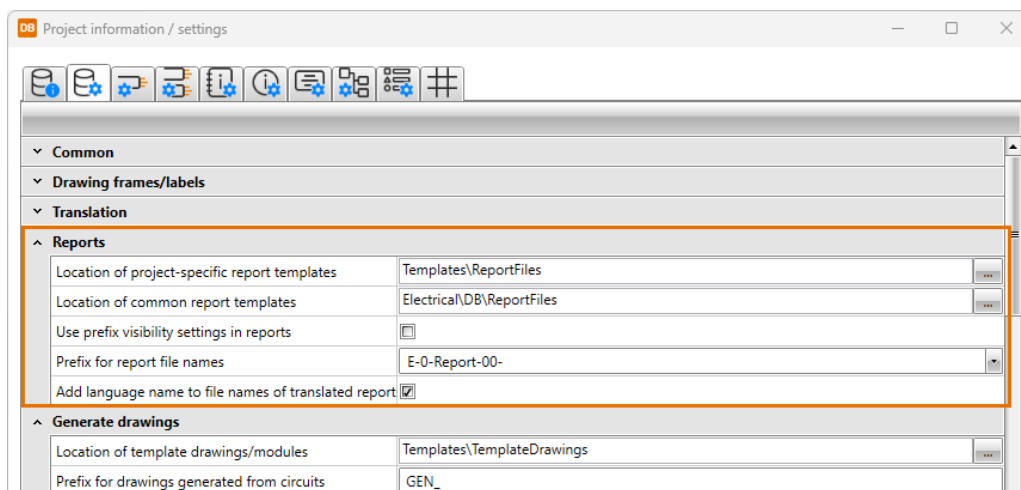
10. Select the **K02** contactor, and press Enter. **K02** has now been assigned to the product model, and the number in the project tree has been updated to 2.
11. Save the drawing.
12. Go to product model management.
13. Move **Contactor 5.5kW** from the project database to the shared database. The model is now available in all other projects and for all users, if required.

## 6. Database reporting tools

The reporting tools allow you to create user-specific reports in Excel or PDF. The reports are easy to create, update and share with other users. The application provides basic templates for Excel reporting, and you can also use them to create your own customized reports.

### 6.1. Define report settings

In the **Project information / settings** dialog, you can find report settings in the **Reports** section on the **Project settings** tab:



- **Location of project-specific report templates** – Define the path for project-specific templates. You can either use the full path, or a path relative to project location. In the example project, templates are searched from the project folder *Templates\Reportfiles*.
- **Location of common report templates** – By default, the Electrical standard template path is used. However, you can create your own common report location.
- **Use prefix visibility settings in reports** – Select whether location = and + and device - prefixes are used in reports.
- **Prefix for report file names** – Define a constant prefix for the current project templates. You can use links to project information.
- **Add language name to file names of translated reports** – In general, you should disable this. Otherwise, the report language name will be added at the end of the file name (usually not desired).

## 6.2. Create a new report

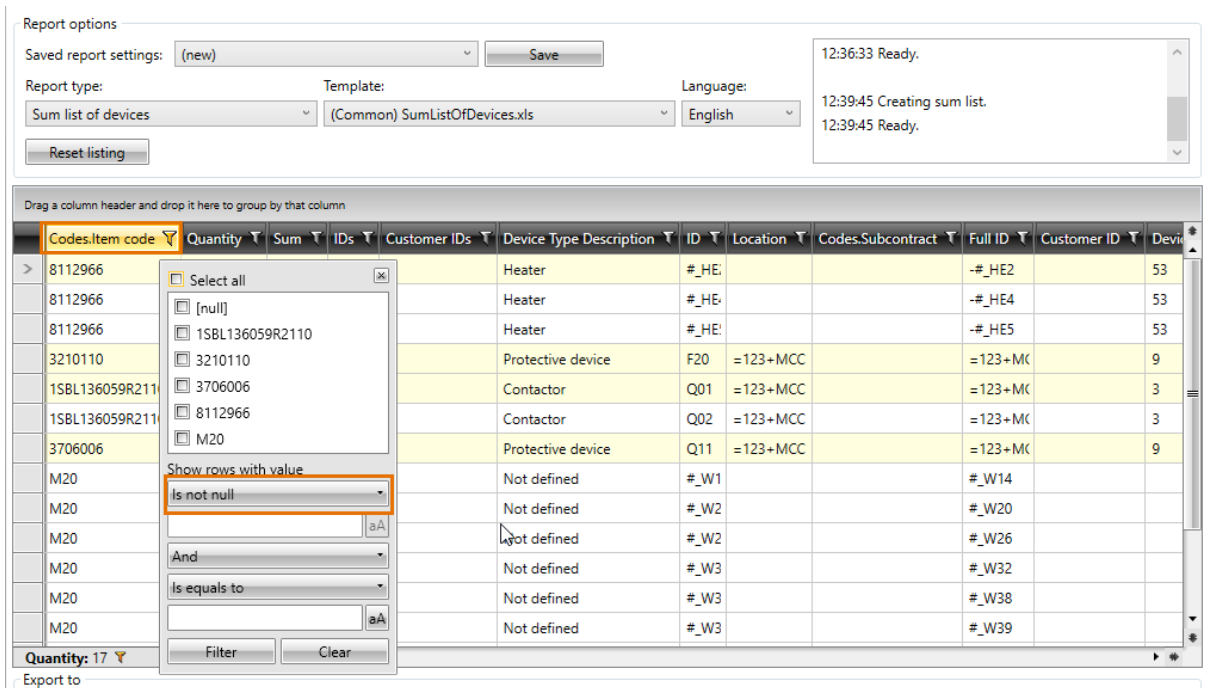
To start creating a report of your own, select **Electrical** tab > **Projects** group > **Reports** menu > **Reports**.

Explanations for the settings:

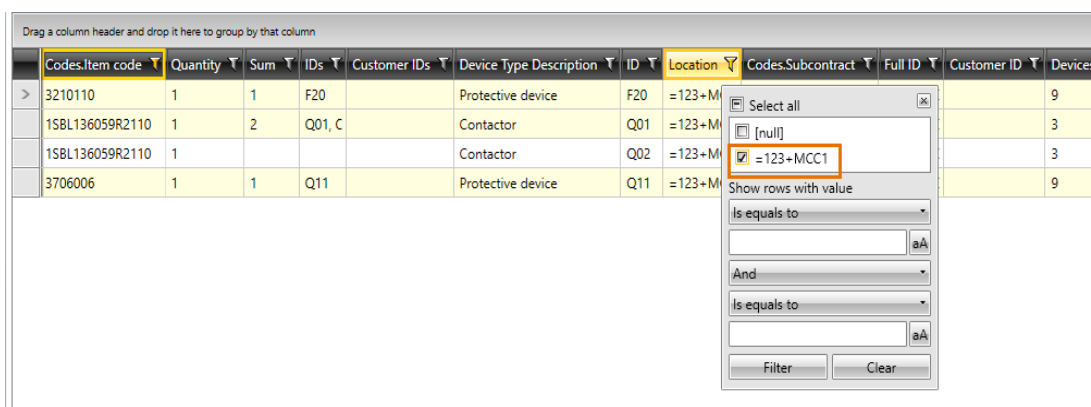
- **Saved report settings** – Save current report settings for future use.
- **Report type** – Select the report type you want to use.
- **Template** – Select Excel template.
- **Language** – Select language. The fields are translated according to the Excel file project settings and the fields defined to be translated.
- **Export to** – Select format of the file/files.
- **Folder** – Select where the report is generated to. If the report is added as a project document, the default should not be changed.
- **Prefix** – The first section comes from the project settings. You can define the second section freely.
- **Add to project as a project document** – The document is added to project documents.
- **Create separate reports from groups** – If information has been grouped, separate Excel files are created.
- **Create separate sheets from groups** – If information has been grouped, reports are generated to different sheets in the same Excel file.

**Do the following:**


1. Define the following settings (the rest can be left as they are by default):
  - **Report type** – Sum list of devices
  - **Template** – (Common) SumListOfDevices.xls
  - **Export to** – Excel file
  - Select **Add to project as a project document**
2. Remove items without product information by excluding the empty **Codes.Item code** with a filter:

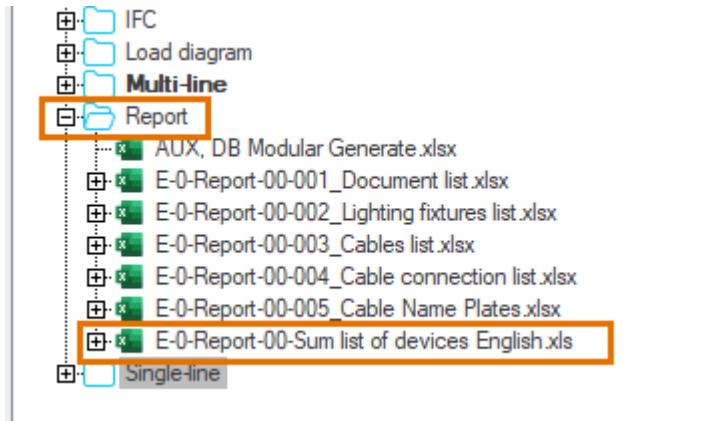


3. Only include **=123+MCC1** distribution board equipment by using a filter:



The yellow rows represent the total amount while white represent the individual devices.

4. Click **Next**.
5. If necessary, rename the report in the grid. The program suggests the name based on the prefix and the list type.
6. Click **OK**.
7. Synchronize the drawing with the  (**Sync changes from database to drawing**) button.
8. In the **Electrical** window, open the **Documents** tree. The report is below the **Report** node:



You can open the report by right-clicking and selecting **Open document**.

## 6.3. Update report

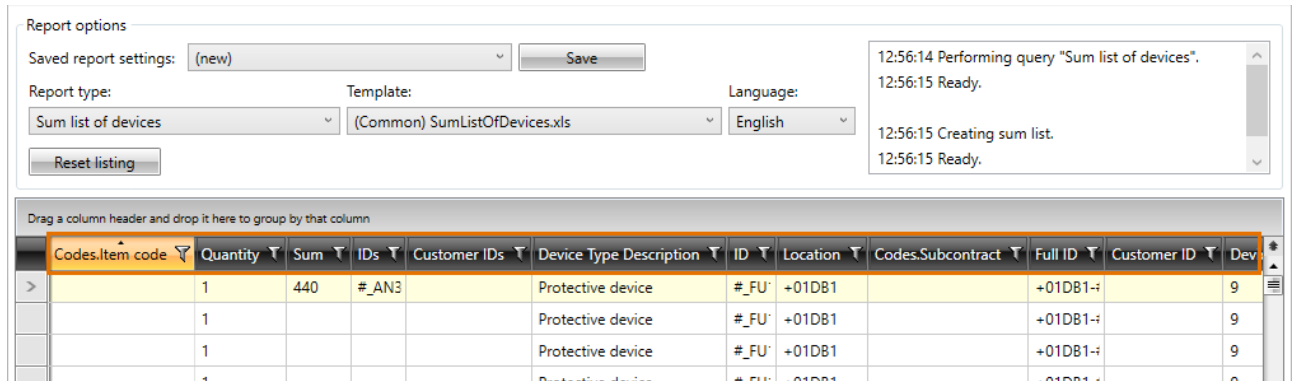
Do the following:

1. Go to the first sheet of the multi-line diagram drawing.
2. Edit **Q01** and then **Q02** product information:
  - a. Right-click the device, and select **Edit device**.
  - b. Select the **Product information** tab.
  - c. From the bottom left corner, click **Select/Edit**.
  - d. Select, for example, the **ABB** product database.
  - e. Select a suitable product, such as **Item code: 102617**.
3. After you have updated both devices save the drawing.
4. Update the report from the project tree by right-clicking and selecting **Update report**. Wait a little for the report to update, and then open the report again. It is now updated:

	A	B	C	D	E	F	G
1	<b>Device list by products</b>						
2							
3							
4							
5							
6	<b>Item</b>	<b>Device</b>	<b>Specifications</b>	<b>Manufacturer</b>	<b>Type</b>	<b>Unit price</b>	<b>Amount</b>
7	3210110		1-pole B-curve 10A 6kA,	ABB	1-pole B-curve 10A	0.00	3
8	102617	MC1A310AT1 CONT. 24V50-60		ABB	MC1 1/0 4kW/AC-3 20A/AC-	0.00	2
9	1SBL136059			ABB	AF09ZB-30-10S-21	0.00	4
10	3706006		1,0-1,6A, 400V/50kA	ABB	1,0-1,6A, 400V/50kA	0.00	3
11							

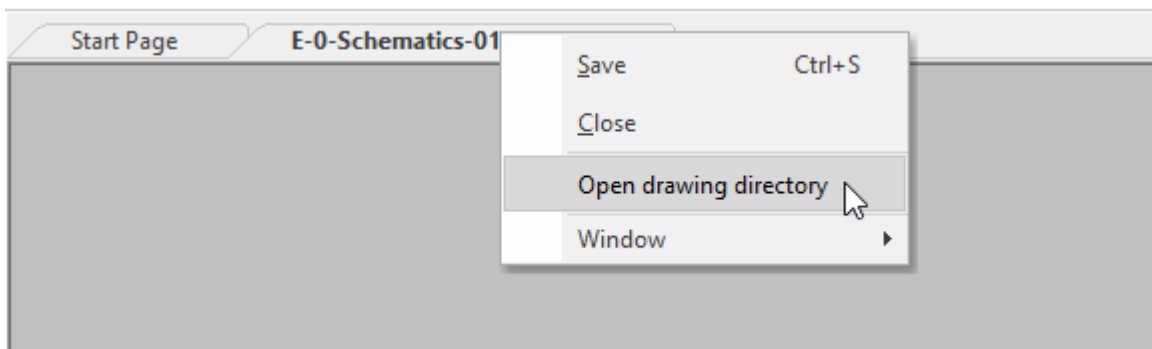
## 6.4. Edit report template

Let's edit the template used. You can easily copy the linkable data from the reporting tool grid headers:



Do the following:

1. Go to *C:\ProgramData\CADMATIC\Draw [version number] English\Electrical\DB\ReportFiles* where standard templates are located in.
2. Copy the template *SumListOfDevices.xls* into the drawing directory *...\Templates\ReportFiles*. You can open the project directory by right-clicking the drawing tab:



3. Open the template after copying it.

	A	B	C	D	E	F	G
1	<b>Device list by products</b>						
2							
3							
4							
5							
6	<b>Item</b>	<b>Device</b>	<b>Specifications</b>	<b>Manufacturer</b>	<b>Type</b>	<b>Unit price</b>	<b>Amount</b>
7	\$Codes.Cod	\$Name 2, English\$	\$Specifications 1\$	\$Manufacturer\$	\$Type\$	\$Unit	\$Sum\$
8							

To construct templates:

- You can use free text or pictures above the links.
- You can add project data to the header:
  - The **\$P:xxx\$** format searches for project information. The available values are:
    - **Project name** – \$P:Name\$
    - **Project description** – \$P:Project description\$
    - **Target data 1-6** – \$P:Target [n]\$ ([n]=number of target data)
    - **Job number** – \$P:Job Number\$
    - **Customer number** – \$P:Customer Number\$
    - **District** – \$P:District\$
    - **Block** – \$P:Block\$
    - **Site** – \$P:Site\$
  - The **\$H:xxx\$** format searches for the first value from the grid. For example, with device list with only =123+MCC1 location filtered \$H:Location\$ will print value =123+MCC1 ones.
  - The **\$xxx\$** format finds the selected value for each row. For example, **\$Manufacturer\$** will print device manufacturer information to each row.
- Column headers are just basic text. Links are given below them.

4. Construct the template:

a. Add **Project name**, **Project description** and **Job number** to the header.

ID	Location	Devices.Device
#_HE!		53
#_HE!		53
F20	=123+M	1-F2 9
F20[1	=123+M	Location
F20[2	=123+M	\$Location\$
Q01[	=123+MCC	

b. Add **Location** information to the header.

c. Remove header **Device** and its link **\$Name 2, English\$**. Instead, add **Device description**

and link under it **\$Device description\$**.

Device list by products						
1						
2	Project name:	\$P:Name\$				
3	Project desc:	\$P:Project description\$				
4	Project number:	\$P:Job Number\$				
5	Distribution board:	\$H:Location\$				
6						
7	<b>Item</b>	<b>Device description</b>	<b>Specifications</b>	<b>Manufacturer</b>	<b>Type</b>	<b>Unit price</b> <b>Amount</b>
8	\$Codes.Code\$	\$Device description\$	\$Specifications 1\$	\$Manufacturer\$	\$Type\$	\$Unit \$Sum\$
9						
10						
14						

5. Save the template and rename it as *Product calculations.xls*.
6. Rerun the report tool with the same settings as previously but use this new template instead. The new template is found with a name *(Project) Product calculations.xls* at the top of the list.