CADMATIC Electrical

Drawing a schema

www.cadmatic.com | support@cadmatic.com | sales@cadmatic.com

Cadmatic
Linnankatu 52
20100 Turku
Finland
Tel. +358 2 412 4500
www.cadmatic.com

Contents

1. Introduction

This guide introduces you to the CADMATIC Electrical, Schematics application. You will learn to create wiring diagrams, as well as draw cables and distribution board's internal wires with the database functions. The database functions excluded, this exercise is suitable for both CADMATIC Electrical Premium and Basic users.

In this exercise, we create a wiring diagram for a motorized valve. You will find the example drawing at the end of this guide.

Note: This exercise is based on CADMATIC Electrical version 2024T2. In other versions, some of the functions may work differently.

2. Start a drawing

With the drawing creation function, you can start a new electrical drawing and be sure that all the necessary settings are defined.

Do the following:

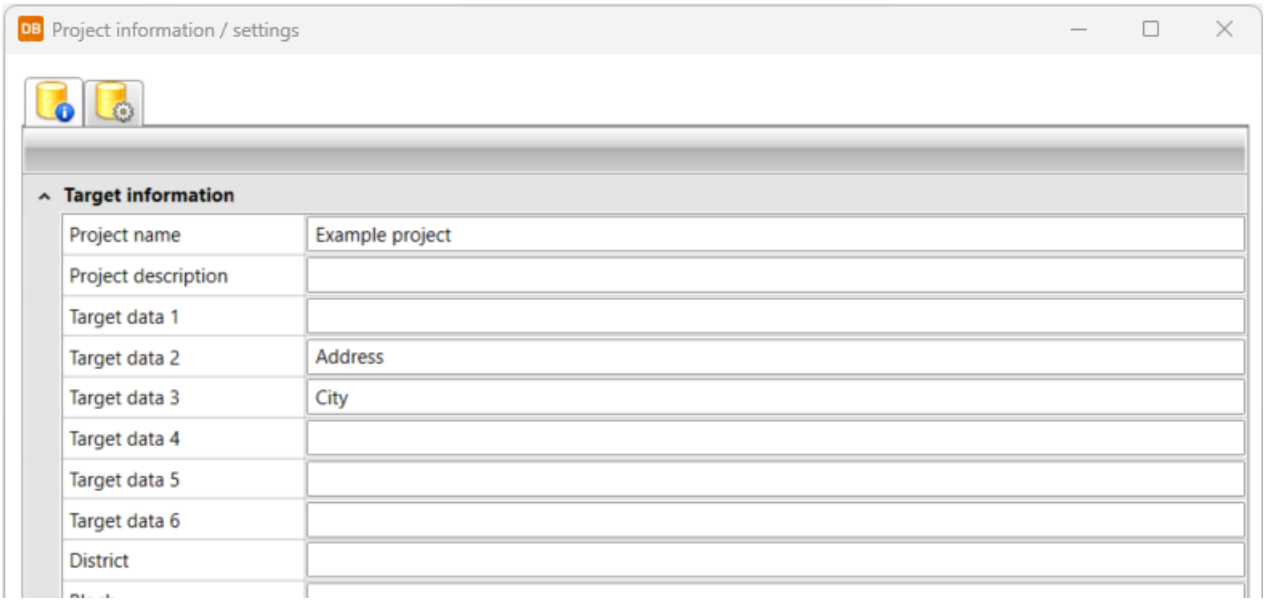
1. Select **Electrical** tab > **Documents** group > **New**.
   The **Start a new Electrical drawing** dialog opens.
2. Define the following:
   - **Drawing type** – **Multi-line diagram**
   - **Project directory** – *Draw 24 English\Samples\Electrical\Schema*
   - **Drawing name** – *Multiline_exercise.drw*

   If you are using the Premium level, you can add the drawing file to the project. This guide, however, will focus on drawing without the database.
3. Click **OK**.

Next, we will define the general project information that can be updated to the drawing frame.

In the **Electrical** window, open the settings by right-clicking *Multiline_exercise.drw* and selecting **Project information/settings**.

The **Project information/settings** dialog opens. Fill in the information related to the project:



Click **Save**, and go to the drawing file to create the first sheet, SLEHTI1, by clicking the  (**Create new sheet**) button in the **Frames and sheets** group on the **Schematics** tab.

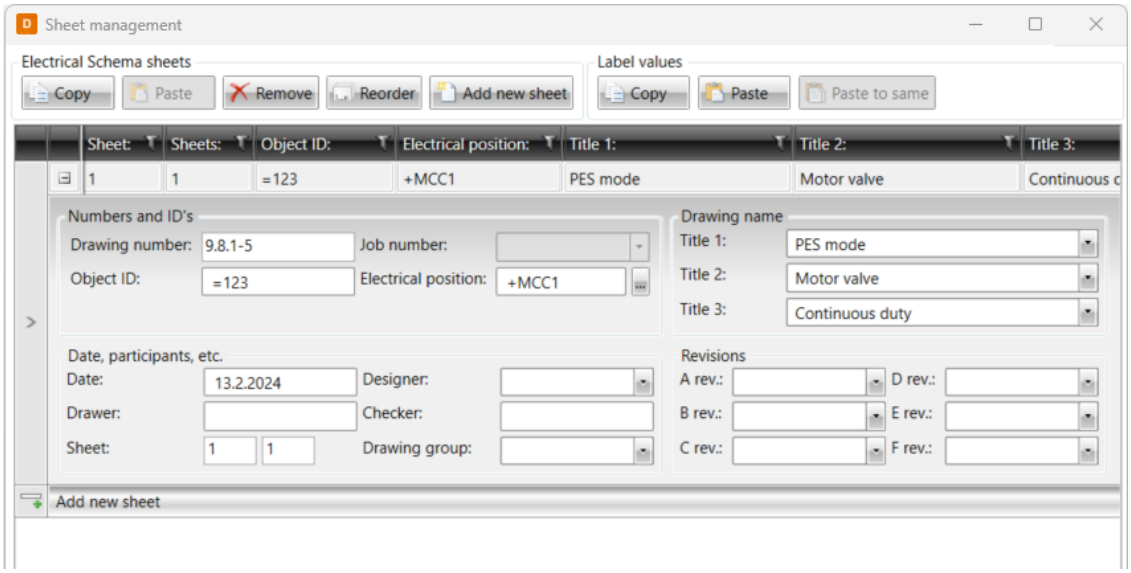
The program also changes this sheet as the active sheet (layer SLEHTI1, SLEHTI2, etc.). The default drawing frame is also inserted on this sheet automatically.

3. Fill in drawing frame values

The drawing frame includes attributes that you can fill in with the filling function.

Do the following:

1. On the **Schematics** tab, in the **Frames and sheets** group, click the (**Edit frame/label information**) button.
   The **Sheet management** dialog opens.
2. Define the following:
   - **Drawing number** – *9.8.1-5*
   - **Object ID** – *=123*
   - **Electrical position** – *+MCC1*
   - **Title 1** – *PES mode*
   - **Title 2** – *Motor valve*
   - **Title 3** – *Continuous duty*



3. Click **OK**. The values are updated to the drawing frame:

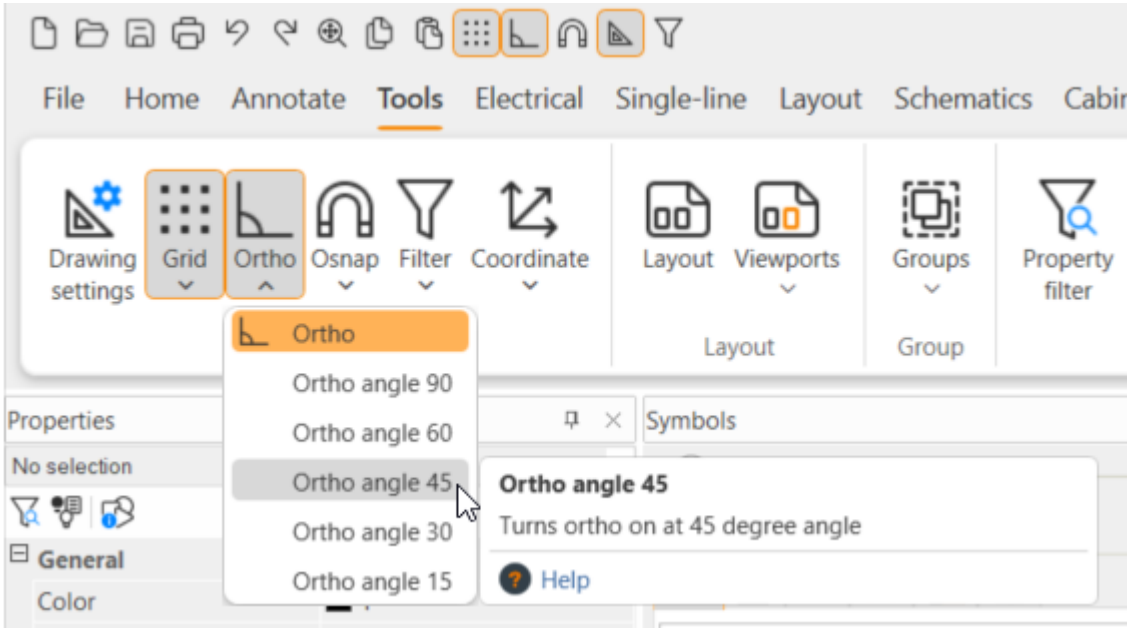
4. Save the drawing by clicking the button on the quick access toolbar.

4. Draw main circuit

4.1. Set ortho on

First, let's set ortho on, as it makes wiring and other designing easier.

On the **Tools** tab, in the **Drawing settings** group click the **Ortho** menu and select **Ortho angle 45**.



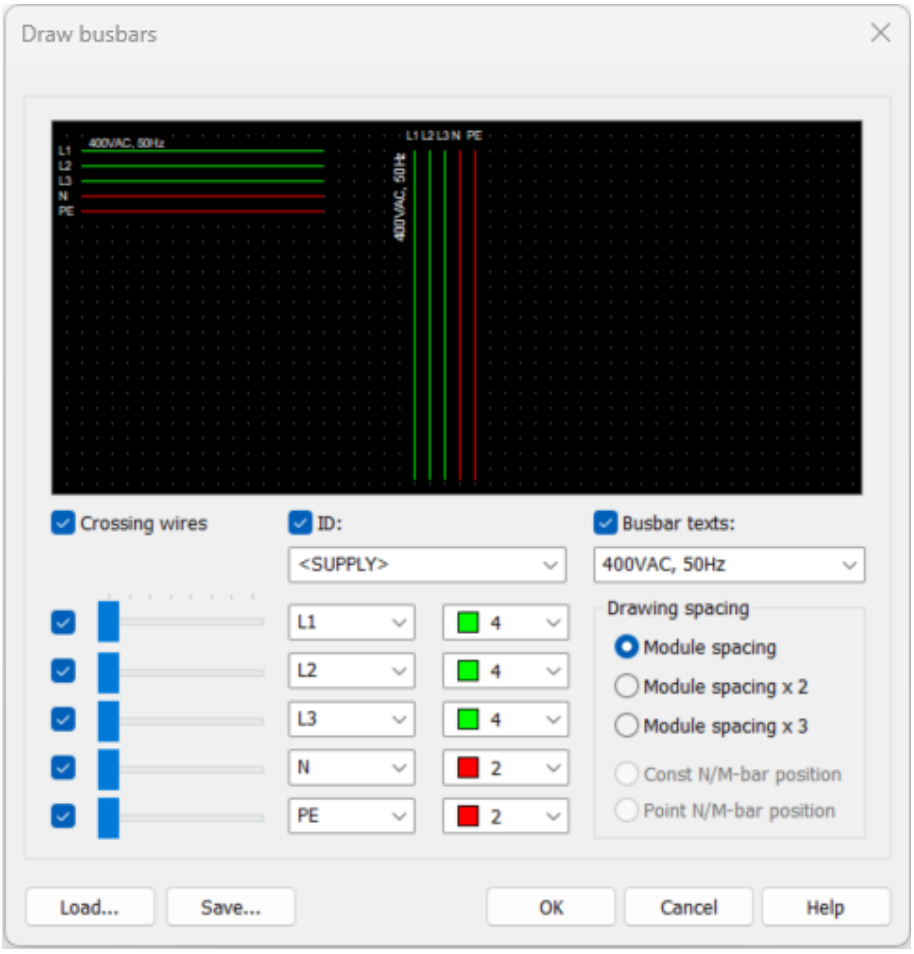
This will set the ortho on with an angle of 45.

4.2. Draw a 3-phase voltage supply bar

Let's first add a 3-phase voltage supply bar in the drawing.

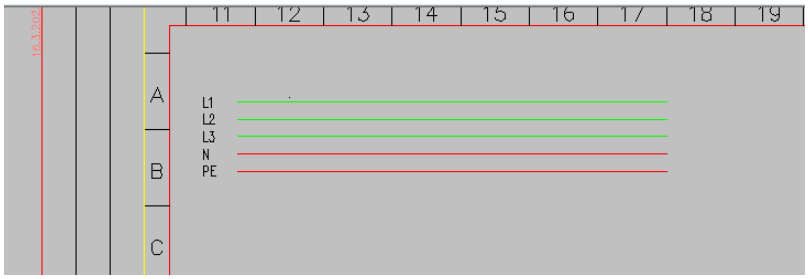
Do the following:

1. Select **Schematics** tab > **Wiring** group > **Draw busbars**. The **Draw busbars** dialog opens.
2. Define the settings as in the following image:



3. Click **OK**.
4. Indicate the starting point for the busbar.
5. Indicate the ending point for the busbar.

6. Stop drawing by right-clicking.



You can assign an ID to a busbar, just like to any other device. The ID will later be found in the **Devices** project tree (see Edit IDs).

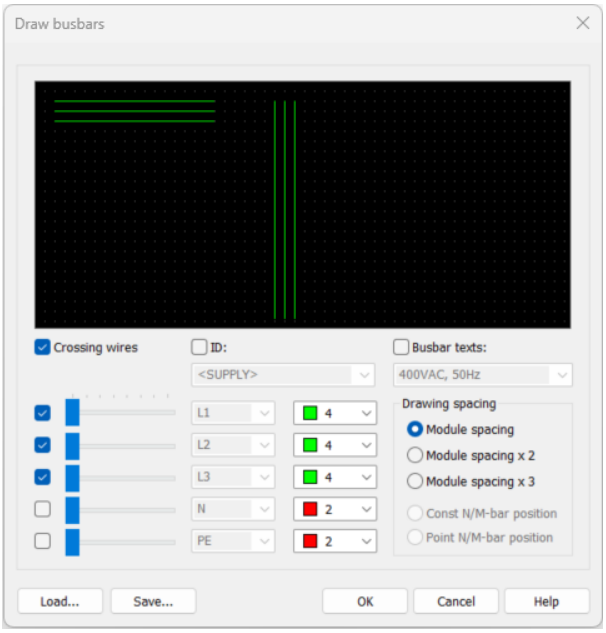
4.3. Draw 3-phase wiring

Let's continue by drawing a new busbar from the same starting point without the N and PE wires.

Do the following:

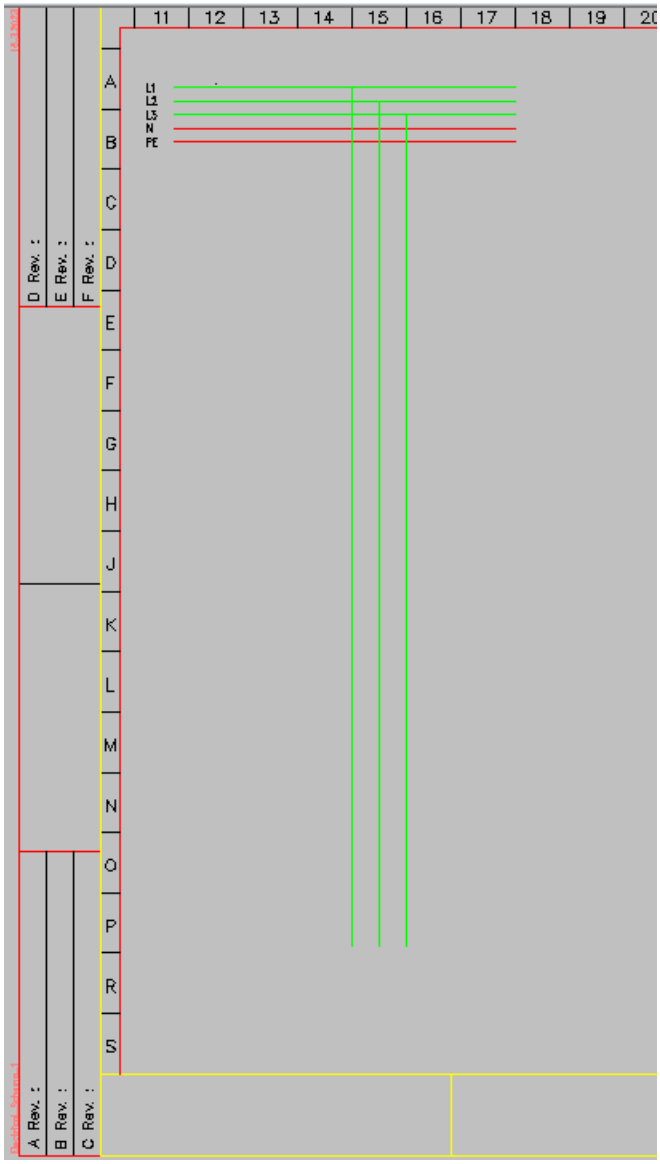
1. Select **Schematics** tab > **Wiring** group > **Draw busbars**. The **Draw busbars** dialog opens.
2. Define the settings according to the following image.

Note: Make sure to select **Crossing wires**.



3. Click **OK**.
4. Indicate the wiring point (= yellow square) for L1.
5. Move your mouse approximately halfway of the wire and click.

6. Move your mouse downwards, and change the space between wires bigger by pressing Shift + Tab.
7. Indicate the end of wiring to the bottom part (row R) of the drawing.



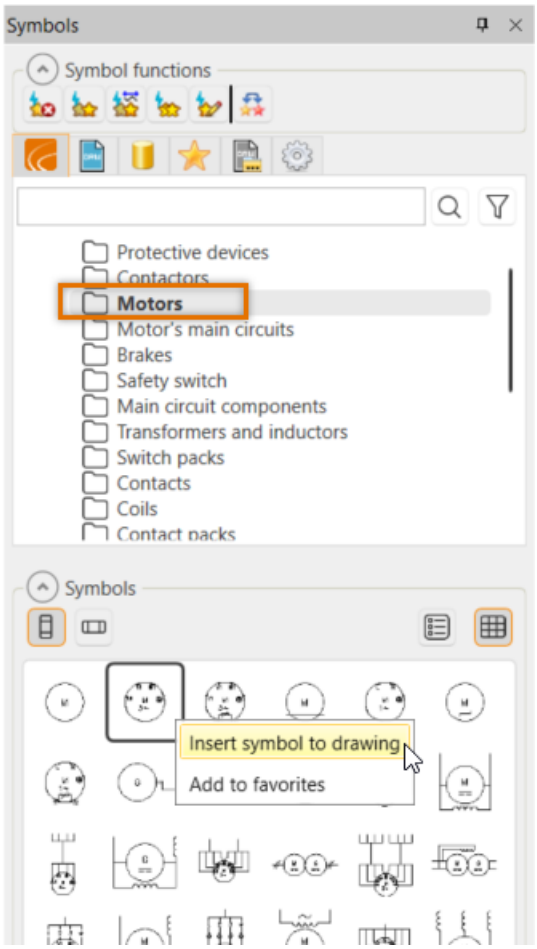
8. Stop drawing by right-clicking.

4.4. Insert symbols

Let's insert the main circuit symbols next. While doing that, follow the command line instructions (Tab and F8).

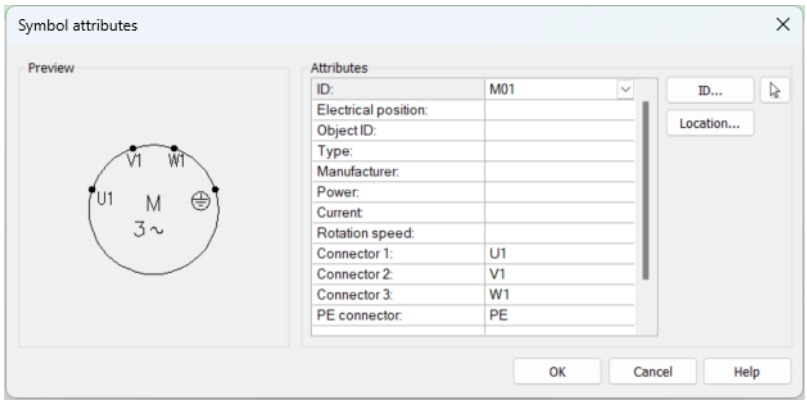
Do the following:

1. Open the **Symbols** window by clicking **Symbols** in the **Devices** group. The window stays open on the screen until you choose to close it.
2. Select **Motors**.
3. Insert the 3-phase motor symbol by double-clicking it, or by right-clicking it and selecting **Insert symbol to drawing**:

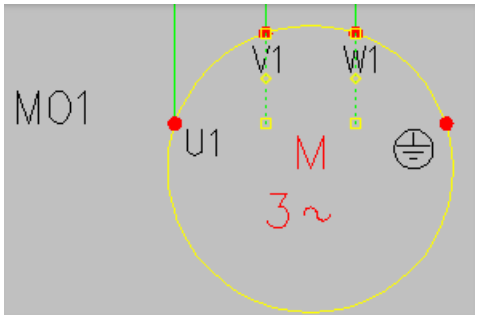


Note: When you insert a symbol on a wire, the program cuts the wire. Wire information is also removed.

4. Enter *M01* as the motor symbol ID, and click **OK**.



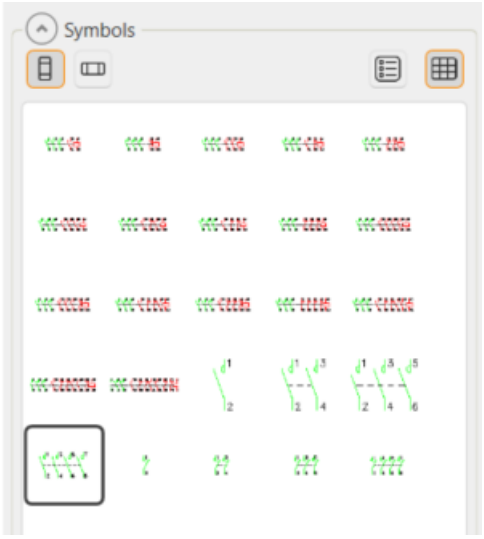
5. Remove the extra lines from the drawing.



6. In the **Symbols** window, select **Safety switch** and insert the safety switch symbol:

7. Enter *Q10* as the safety switch symbol ID, and click **OK**.

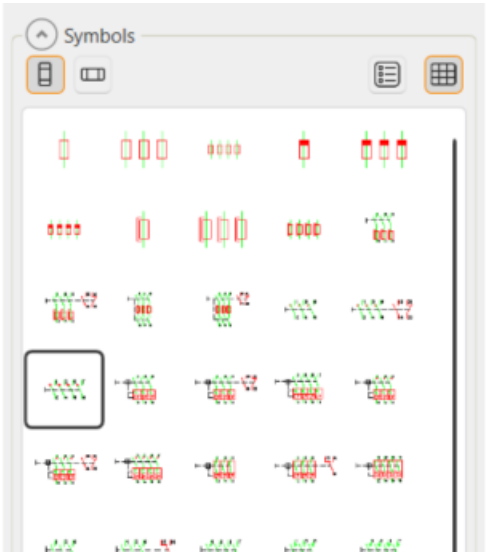
8. In the **Symbols** window, select **Contactors** and insert the contactor symbol:



9. Enter *Q01* as the contactor symbol ID, and click **OK**.
10. Insert another symbol into the drawing, and enter *Q02* as the ID.

> Tip: To insert the same symbol again, you do not need to go to the symbol menu. Instead, right-click to insert it.

11. In the **Symbols** window, select **Protective devices** and insert the protective device symbol:



12. Enter *Q11* as the protective device symbol ID, and click **OK**.

4.5. Draw wires

Let's draw wires from Q01 to Q02. We will later define them as distribution board's internal wires.

Important: You should always draw from wiring point to another, so that the application will find the right connections when updating the databases. During wiring, wiring points are indicated with yellow squares.

Do the following:

1. Select to draw main circuit:
    - In Premium: Select **Schematics** tab > **Wiring** group > **Draw wiring**, and then select main circuit as the wire type.
    - In Basic: Select **Schematics** tab > **Wiring** group > **Draw wiring** menu > **Draw wire (main circuit)**.
2. Select **Q01:1** as the starting point.
3. Select **Q02:1** as the ending point.
4. Stop drawing by right-clicking.
5. Restart wiring by right-clicking, and continue as follows:
    - **Q01:3** > **Q02:3**
    - **Q01:5** > **Q02:5**
    - **Q01:2** > **Q02:6**
    - **Q01:4** > **Q02:4**
    - **Q01:6** > **Q02:2**

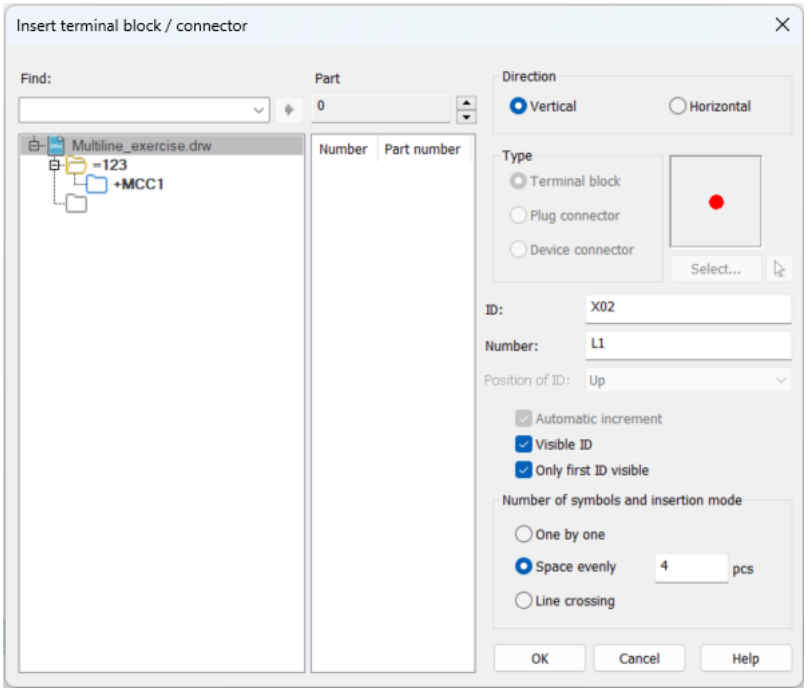
Tip: You can draw a chamfer by pressing Tab during drawing.

4.6. Insert terminal blocks

Let's insert the **X02** terminal blocks into the drawing. The **X02** ID is only visible in the **L1** terminal block but it still needs to be inserted to all terminal blocks.

Do the following:

1. In the **Symbols** window, select **Terminals**.
2. Insert the terminal block symbol by double-clicking it or by right-clicking it and selecting **Insert symbol to drawing**. The **Insert terminal block / connector** dialog opens.
3. Define the following information:
   - **ID** – *X02*
   - **Number** – *L1*
   - Select **Visible ID** and **Only first ID visible**.
   - **Number of symbols and insertion mode** – **Space evenly**, *4*



4. Click **OK**.
5. Insert the terminal block by indicating its location.
6. Insert the next terminal block.
7. Double-click the **L4** terminal block, and change the number to **PE**.

4.7. Draw protective earthing conductors

Let's draw two protective earthing conductors. Wiring can be drawn from the terminal blocks straight to the motor, because the wiring function knows to cut the wires when meeting security switches.

Do the following:

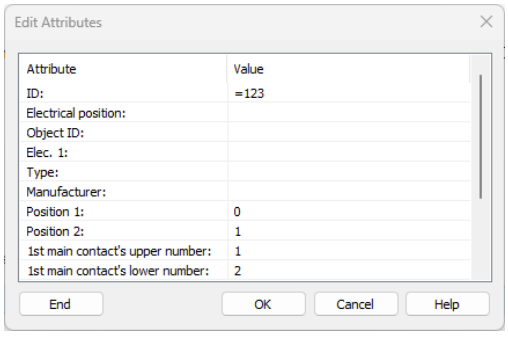
1. Select to draw main circuit:
    - In Premium: Select **Schematics** tab > **Wiring** group > **Draw wiring**, and then select main circuit as the wire type.
    - In Basic: Select **Schematics** tab > **Wiring** group > **Draw wiring** menu > **Draw wire (main circuit)**.
2. Select **X02:PE** as the starting point.
3. Select **M01:PE** as the ending point.

4.8. Edit IDs

Let's exclude the safety switch and the motor from the distribution board **+MCC1** by adding them the object ID **=123**. This way, they will not end up in the **+MCC1** device list.

Do the following:

1. Select the symbols **Q10** and **M01**.
2. Right-click and select **Attribs**. The **Edit Attributes** dialog opens.
3. Define **=123** as the object ID.



4. Click **OK**. The full IDs are now **=123-Q10** and **=123-M01**

The motor **M01** and the safety switch **Q10** now belong to the object ID **=123** but are not devices of the electrical position (distribution board) **+MCC1**.

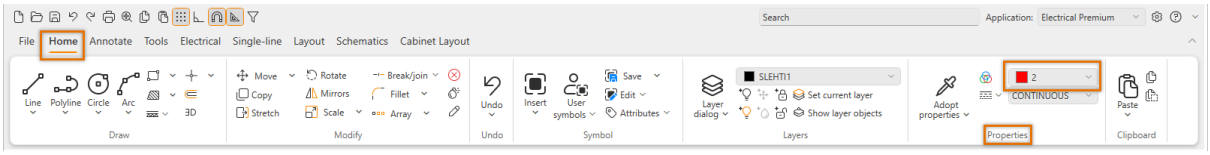
4.9. Draw area boundaries

Let's draw boundaries around the safety switch and the motor.

Note: The boundaries only indicate that the safety switch and the motor do not belong to the distribution board **+MCC1**. We only draw these boundaries to make the drawing easier to read; the application will not take it into account in any way. Components have been excluded from the distribution board with an ID: for example, **=123-M01** means that the motor **M01** belongs to **=123** but is not a component of **+MCC1**.

Do the following:

1. Select **Home** tab > **Draw** group > **Line**.
2. In the **Properties** group, select the color **2** (red) from the drop-down menu.



3. Below the color drop-down menu, select **DASHDOT** as the line type.
4. Draw the lines.

Note: After drawing, change the line type back to **CONTINUOUS**.

5. Draw control circuit

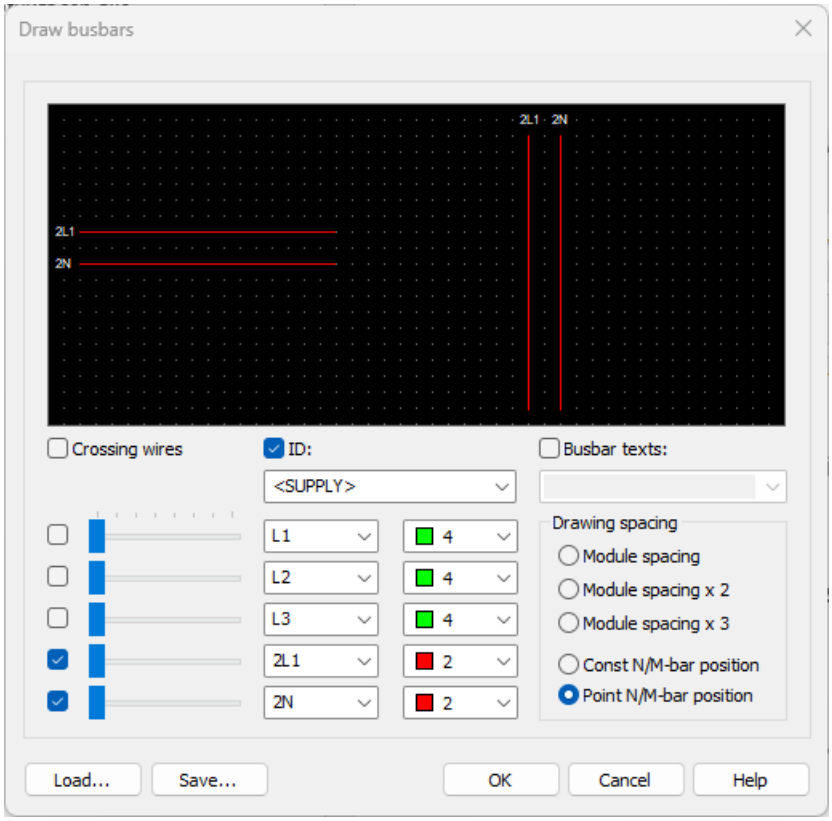
5.1. Draw control voltage busbar

We will start drawing the control circuit by first drawing a busbar, using the same function as for the 3-phase voltage supply bar.

Do the following:

1. Select **Schematics** tab > **Wiring** group > **Draw busbars**. The **Draw busbars** dialog opens.
2. Select settings according to the following image.

Note: Make sure to select **Point N/M-bar position**.



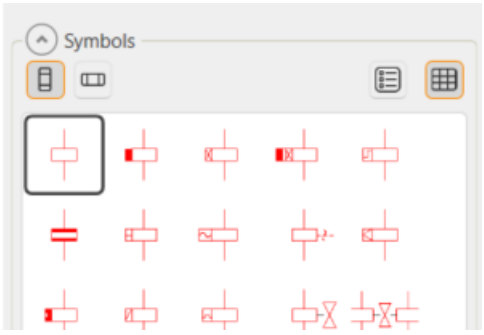
3. Indicate the starting point for **2L1**.
4. Indicate the ending point for **2L1**.
5. Indicate location for **2N** vertically as low as possible so that the auxiliary contact packs will still fit below it.

5.2. Insert coils

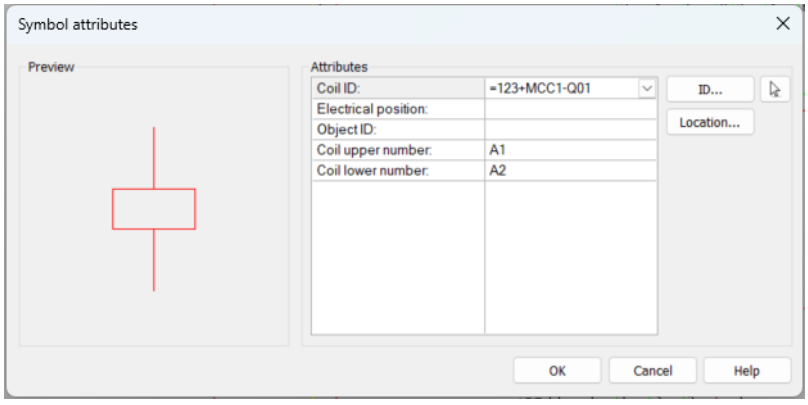
Next, we will insert control circuit symbols according to the example drawing. Let's start with coils.

Do the following:

1. In the In the **Symbols** window, select **Coils** and the coil symbol:



2. Insert the symbol two raster points (2x3,5mm) above **2N**.
3. Define the following:
   - **Coil ID** – From the drop-down menu, select **=123+MCC1-Q01**. The **=123+MCC1** part will be removed when the symbol is inserted because the same information is included in the drawing frame.
   - **Coil upper number** – *A1*
   - **Coil lower number** – *A2*



4. Click **OK**.
5. Insert the coils **Q02**, **K91** and **K92**. You cannot select **K91** and **K92** from the drop-down menu so you need to enter them manually.

5.3. Insert auxiliary contacts

Let's insert auxiliary contacts below the coils.

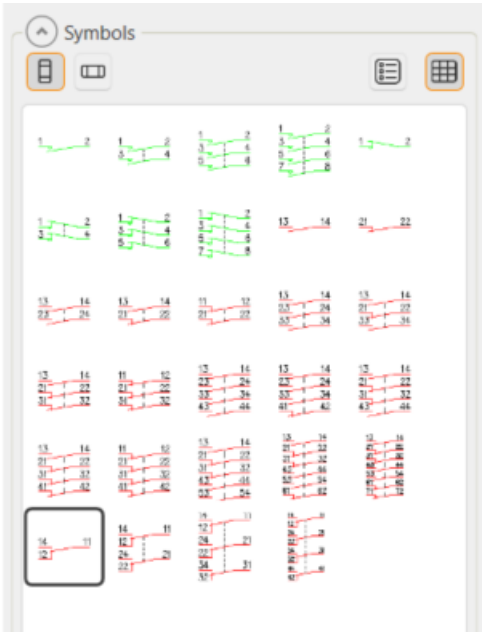
Do the following:

1. In the **Symbols** window, select **Contact packs** and insert the contact pack symbol:

2. Select **Q01** as the contact pack ID, and click **OK**.
3. Insert another symbol into the drawing, and select **Q02** as the ID.

> Tip: To insert the same symbol again, you do not need to go to the symbol menu. Instead, right-click to insert it.

4. In the **Symbols** window, select **Contact packs** and insert the contact pack symbol:



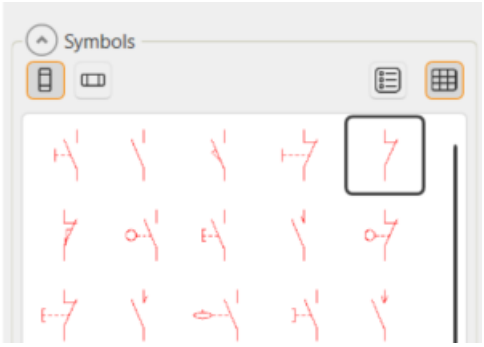
5. Enter *K91* as the contact pack ID, and click **OK**. You cannot select the ID from the drop-down menu so you need to enter it manually.
6. Insert another symbol into the drawing, and enter *K92* as the ID. You cannot select the ID from the drop-down menu so you need to enter it manually.

5.4. Insert contacts

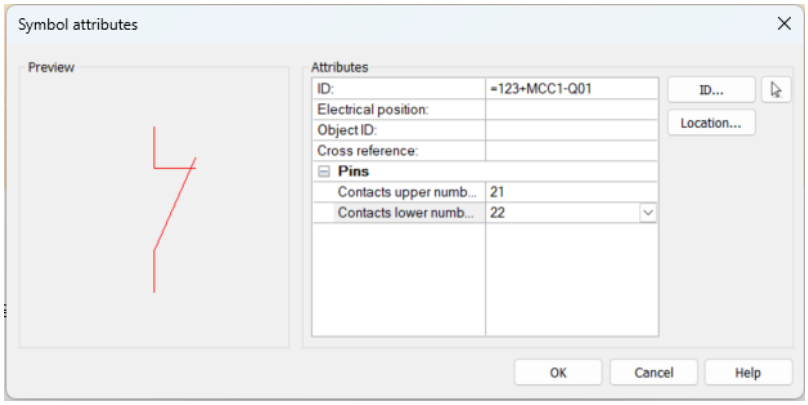
Let's insert contacts according to the example drawing. The cross-references for the auxiliary contact pack and the contact are automatically added.

Do the following:

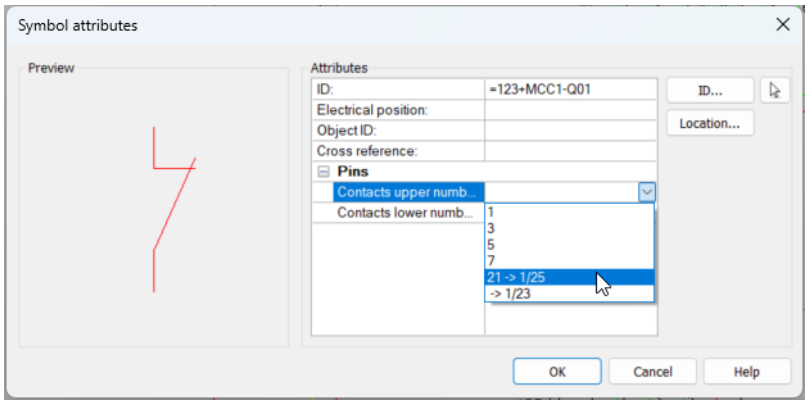
1. In the **Symbols** window, select **Contacts** and insert the contact symbol:



2. Define the following:
   - **ID** – From the drop-down menu, select **=123+MCC1-Q01**.
     The **=123+MCC1** part will be removed when the symbol is inserted because the same information is included in the drawing frame.
   - **Contacts upper number** – From the drop-down menu, select **21** (available after the auxiliary contact pack has been inserted).
   - **Contacts lower number** – From the drop-down menu, select **22** (available after the auxiliary contact pack has been inserted).



When inserting the next contact, you can see the contacts already inserted: 1/25 = sheet 1, wiring reference pair 25:



3. Insert the other contacts.

Tip: You can find the desired symbols quickly by entering the full symbol name in the **Symbols** window search field.

- Opening contact 21/22 for contactor **Q02**, symbol name **Normally-closed contact**
- Contactor 1/2 for the circuit breaker **F10**, symbol name **Circuit breaker**
- Normally-open contact 13/14 for the motor protection switch **Q11**, symbol name **Normally-open contact**
- Normally-open contact 13/14 for the safety switch **Q10**, symbol name **Normally-open contact**

  Note: The ID is left empty as it will be taken from the device boundary **=123-Q10**.

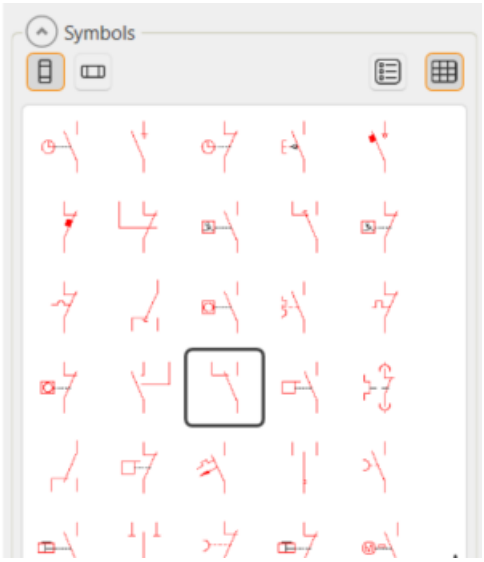
- Contact 19/20 for motor's thermal protector **F1**, symbol name **Thermal protector, normally-closed contact**

5.5. Insert change-over contacts

Let's insert change-over contacts next.

Do the following:

1. In the **Symbols** window, select **Contacts** and insert the change-over contact symbol:



2. Define the following:
    - **ID** – *K10, K20*
    - **Contact's 1st upper number** and **Contact's 2nd upper number** – *12* and *14*
    - **Contact's lower number** – *11*
3. Click **OK**.

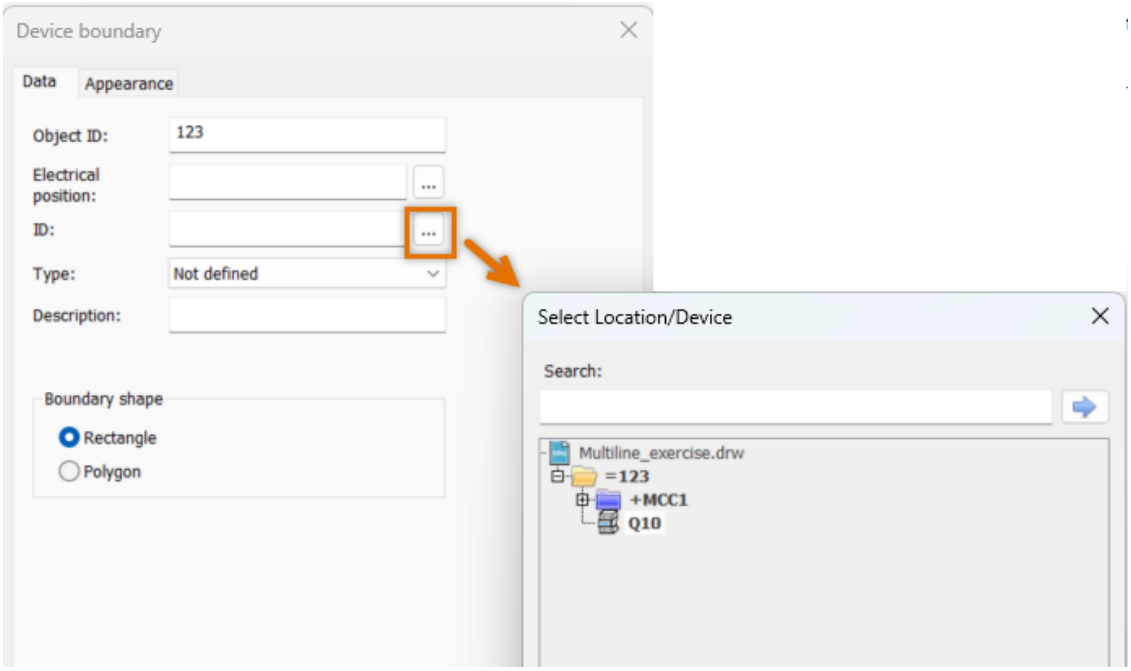
5.6. Draw device boundaries

Next, we will create device boundaries (=123-Q10 and =123-M01) with which you can define drawing elements for specific devices.

Tip: The ID for the bounded area can be added to the drawing as one text or three separate texts (objet ID, electrical position, and device ID). In addition, you can locate the text either inside or outside the bounded area. The bounded area can include all kinds of elements.

Do the following:

1. Select **Schematics** tab > **Other functions** group > **Boundaries** menu > **Device boundary**. The **Device boundary** dialog opens.
2. Define the following:
   - **Object ID** – *=123*
   - **ID** – *Q10*; with the … (**Browse**) button, you can easily select the device from the project tree:



3. Indicate the first corner of the bounded area.
4. Indicate the second corner of the bounded area.

Note: If you need to edit the bounded area (the size or the shape) later, select it, right-click and select **Redefine bounding area**.

5. Create device boundaries for =123-M01 as described.

5.7. Insert terminal blocks

Let's insert control circuit terminal blocks next. We will first insert eight terminal blocks at once, to location (distribution board) **+MCC1**.

Do the following:

1. In the **Symbols** window, select **Terminals** and insert the terminal block symbol. The **Insert terminal block / connector** dialog opens.
2. Define the following information:
   - **ID** – *X10*
   - **Number** – *1*
   - Select **Visible ID**.
   - Make sure **Only first ID visible** is not selected.
   - **Number of symbols and insertion mode** – **One by one**, *8* (all terminal blocks of terminal strip **X10**)
3. Click **OK**.
4. Indicate the base point for the terminal block (= location): **X10:1** inserted to **2N**.
5. With your mouse, indicate the direction where you want to add the terminal block number and click.
6. Insert the other terminal blocks one by one:
   - **X10:28**
   - **X10:PE** (2 pieces)

Note: Leave the device terminal block ID empty as it will be taken from the device boundary.

5.8. Draw internal wires

Let's draw distribution board's internal wires next. The function is otherwise similar to drawing a control circuit but we also add wire information which is needed when creating a wiring report.

Do the following:

1. Select to draw main circuit:
    - In Premium: Select **Schematics** tab > **Wiring** group > **Draw wiring**, and then select control circuit as the wire type.
    - In Basic: Select **Schematics** tab > **Wiring** group > **Draw wiring**.
2. Select the other end of **2L1** (= the yellow square) as the wire's starting point.

    Important: You should always draw from wiring point to another, so that the application will find the right connections when updating the databases. During wiring, wiring points are indicated with yellow squares.

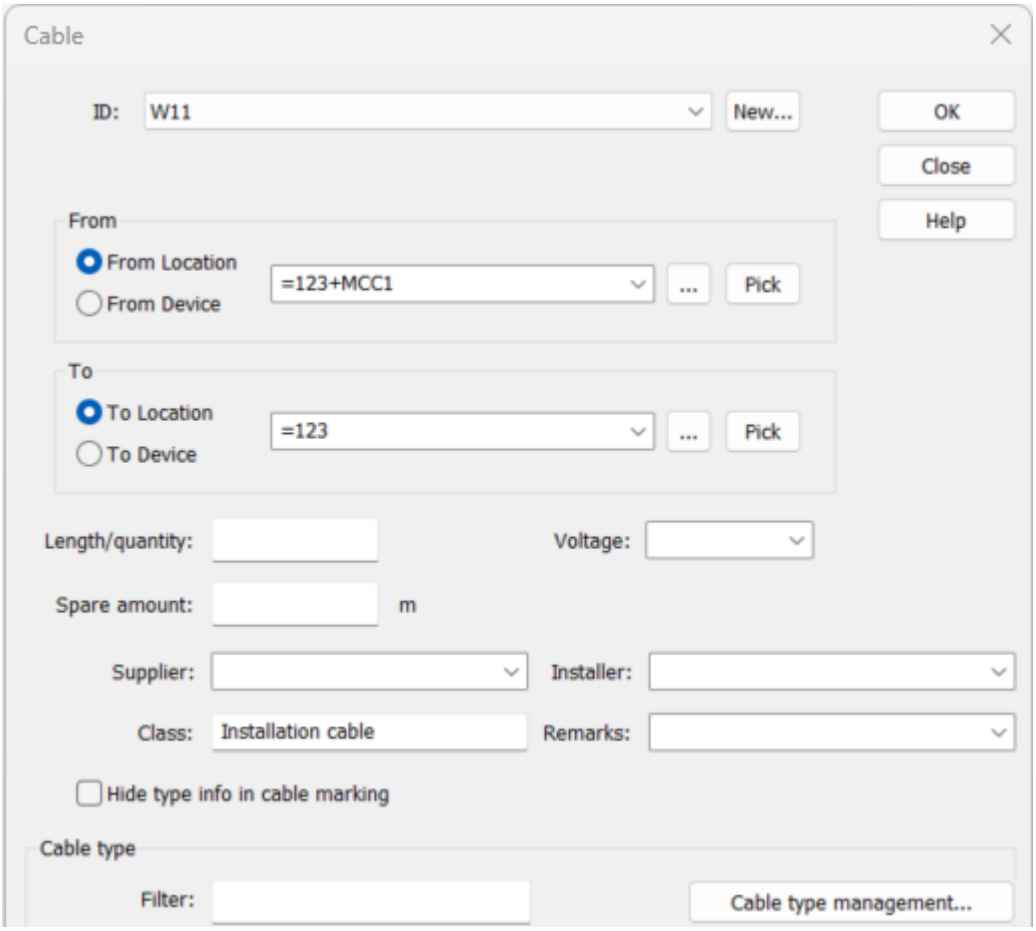
3. Select **X10:1** as the ending point. You can draw the wire across symbols as the functions knows to cut the wire, for example, **2L1** > **X10:3** > **F10:1**, **F10:2** > **Q11:13**, **Q11:14** > **X10:4**.
4. Finalize the wiring for the control circuit according to the example drawing.

6. Mark wires

Next, we'll add cable markings for cables **W10** and **W11** into the drawing.

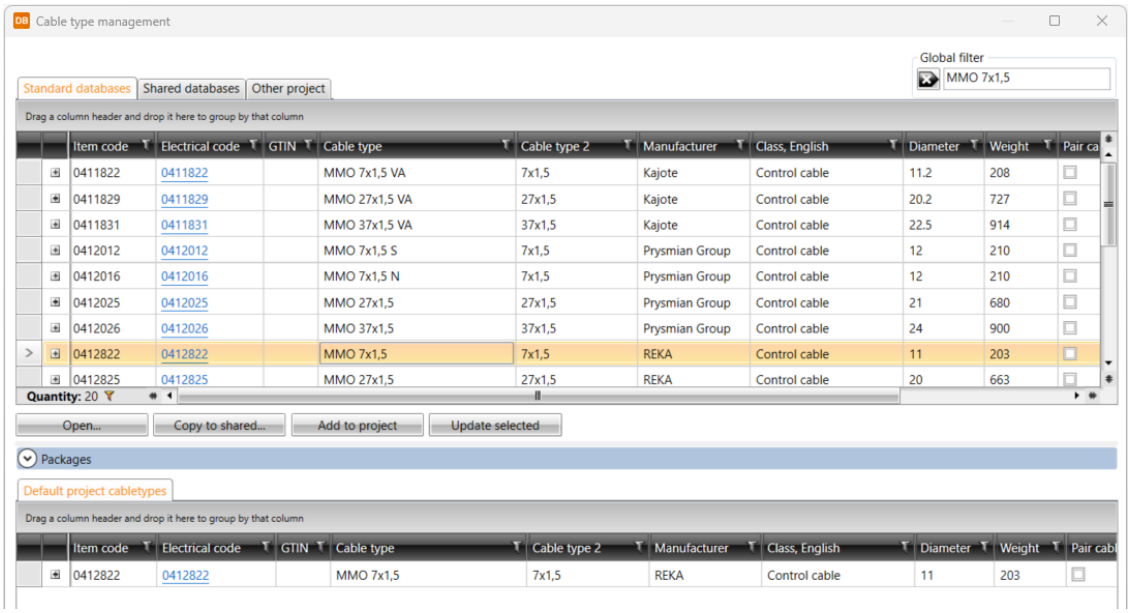
Do the following:

1. Select **Schematics** tab > **Markings** group > **Cable**.
2. Draw cable accross the wires: first indicate the first point from the right side of the wire, then indicate the second point from the left side.
3. Add **W11** as the ID, and define the other information according to the following image.



4. Because there are no cables defined for the project, add the cable:
    a. Click **Cable type management**.
    b. In the **Global filter** field, enter *MMO 7x1,5*.

c. Double-click the cable to add it to the project.



d. Close cable type management.

5. In the **Cable** dialog, click **OK**.
6. Assign the cable in the drawing to this cable, and click **OK**.
7. Insert the cable mark by following the command line instructions.
8. Mark the control circuit wires according to the example drawing.
9. Connect and mark cable **W10** in the same way, according to the example drawing.

7. Add cable marking

Next, we will connect and mark the cables **W01** and **W02**.

Do the following:

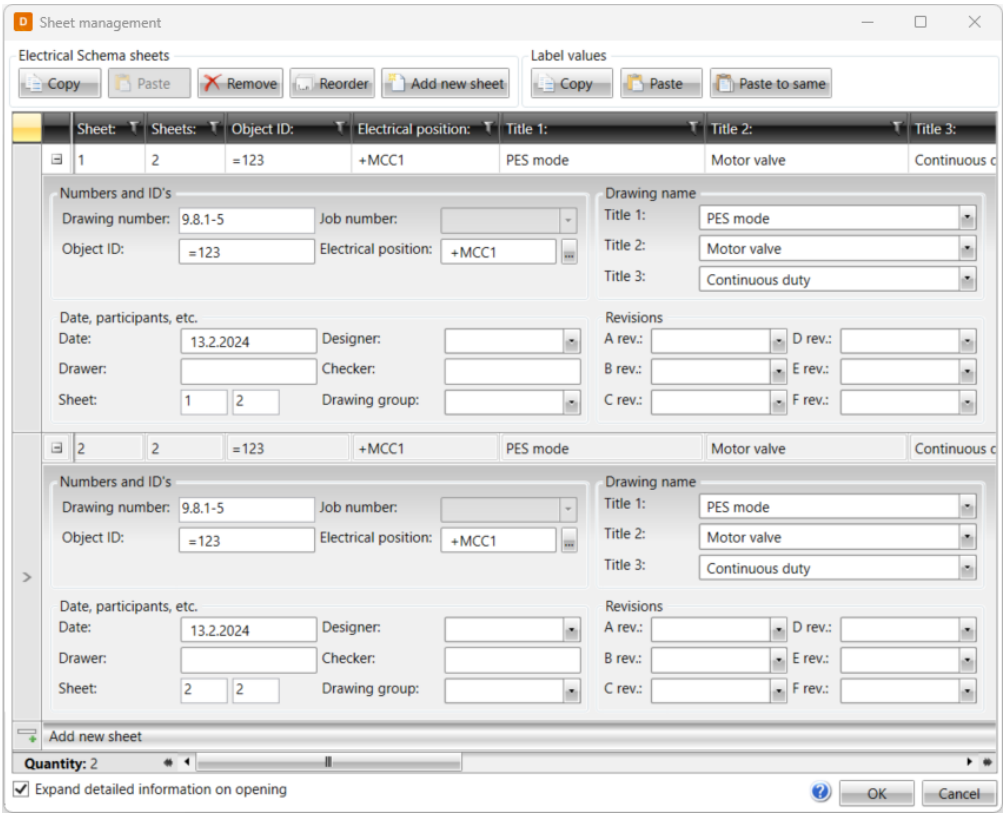
1. Select **Schematics** tab > **Markings** group > **Cable**.
2. Draw the cable line across the wires: indicate the first and second points.
3. Define the cable information:
    a. Select the safety switch cable **W01**.
    b. For **From** and **To**, select **From Device** and **To Device**.
    c. Select **Cable type management**, and find the cable type **MCMK 3x1,5/1,5**.
    d. Click **OK**.
4. Mark the cable and wire information to the drawing.
5. Indicate the ending point for the cable marking.
6. Connect and mark cable **W02** in the same way.

8. Create the next sheet

The first sheet (SLEHTI1) is now ready, and we can continue by drawing the second sheet (SLEHTI2). We will also copy label information from the first sheet to the second.

Do the following:

1. Select **Schematics** tab > **Frames and sheets** group > (**Create new sheet**).
2. Still in the **Frames and sheets** group, click the (**Edit frame/label information**) button. The **Sheet management** dialog opens.
3. Select the whole **Sheet 1** row.
4. In the **Label values** section, click **Copy**.
5. Select the whole **Sheet 2** row.
6. In the **Label values** section, click **Paste**.



7. Click **OK**.

9. Draw boundaries

9.1. Draw boundaries for locations

Elements inside location boundaries are defined to belong to that location. By default, all elements on a sheet belong to the location (= electrical position) defined in the drawing frame – with the bounding area you can define elements to a different location than that used by default.

Let's create the bounding area for +CS1.

Do the following:

1. Select **Schematics** tab > **Other functions** group > **Boundaries** menu > **Location boundary**.
2. Define the following:
    - **Object ID** – *+CS1*
    - **Description** – *CONTROL SYSTEM*
3. Click **OK**.
4. Indicate the first corner for the bounding area.
5. Indicate the second corner for the bounding area.

Note: If you need to edit the shape/size afterwards, select a boundary, right-click and select **Redefine bounding area**.

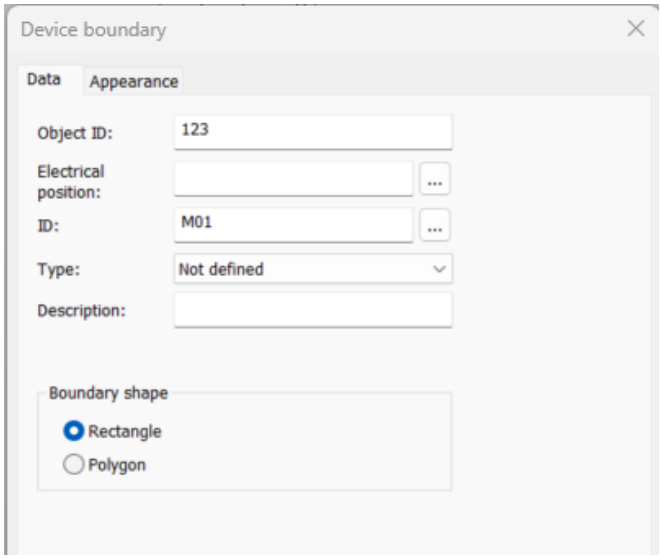
9.2. Draw boundaries for devices

With device boundaries, you can define elements belonging to specific devices.

Let's create device boundaries **=123-M01** and **-B1**.

Do the following:

1. Select **Schematics** tab > **Other functions** group > **Boundaries** menu > **Device boundary**.
2. Define the following:
    - **Object ID** – *=123*
    - **ID** – *M01*



3. Click **OK**.
4. Indicate the first corner for the bounding area.
5. Indicate the second corner for the bounding area.
6. Create boundaries for **-B1** as described.

Note: If you need to edit the shape/size afterwards, select a boundary, right-click and select **Redefine bounding area**.

---

10. Insert symbols

10.1. Insert terminal blocks and connectors

Let's insert terminal blocks for the control system **+CS1** and the distribution board **+MCC1**. Let's also insert device connectors for **B1**.

Do the following:

1. In the **Symbols** window, select **Terminals** and insert the terminal block symbol. The **Insert terminal block / connector** dialog opens.
2. Define the following for the control system **+CS1**:
   - **ID** – *X1*
   - **Number** – *1*
   - **Number of symbols and insertion mode** – **Space evenly**, *16*
3. Indicate the insertion point for the first terminal block (= location).
4. Indicate the connector number inside the location boundaries, and click.
5. Indicate the insertion point for the second terminal block (space between terminal blocks 6 raster points = 6 x 3.5 mm.
6. Move the terminal blocks (5–16) according to the example drawing:
   a. Select the terminal blocks to be moved (5-16) in the selection dialog.

      Note: Select the terminal blocks from bottom to the top.

   b. Right-click and select **Move**.
   c. Indicate the new location.
7. Insert the rest of the control system **+CS1** terminal blocks as described.
8. Insert the distribution board **+MCC1** terminal blocks **X11:10–14** and **17–18** as described.
   a. Define the following:
      - **ID** – *X11*
      - **Number** – *10*
      - **Number of symbols and insertion mode** – **One by one**, *5*

b. Insert terminal blocks 17 and 18.

> Tip: Align the **X11** terminal blocks with the **+CS1** terminal blocks.

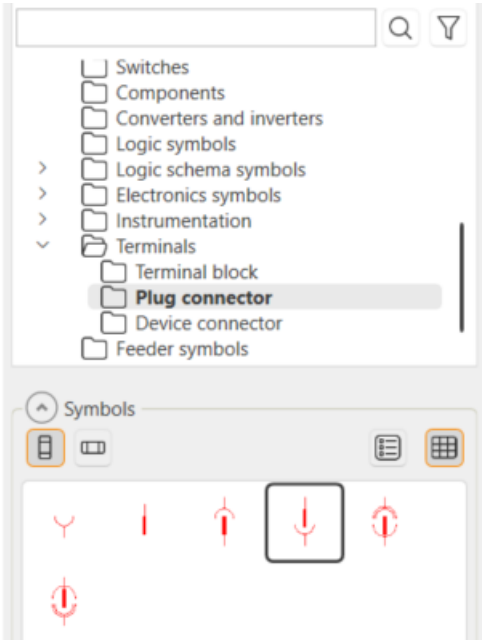
9. Insert **B1** device connectors - and +: first number **-/+**.

10.2. Insert plug connectors

Let's insert plug connectors inside the device bounding area **=123-M01**.

Do the following:

1. In the **Symbols** window, select **Plug connector** from the **Terminals** menu and insert the plug connector symbol:



   The **Insert terminal block / connector** dialog opens.

2. Define the following:
   - **ID** – Leave empty.
   - **Number** – *3*
   - **Number of symbols and insertion mode** – **Space evenly**, *10*

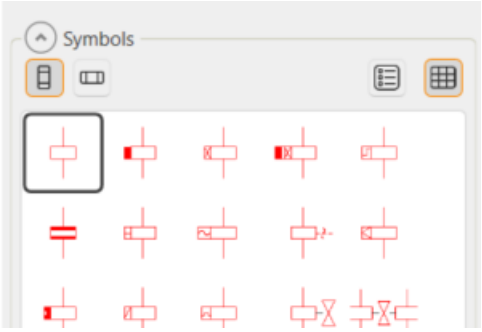
5. Edit the connector numbers according to the example drawing, and move the connectors to where they belong.

10.3. Insert symbols to the second sheet

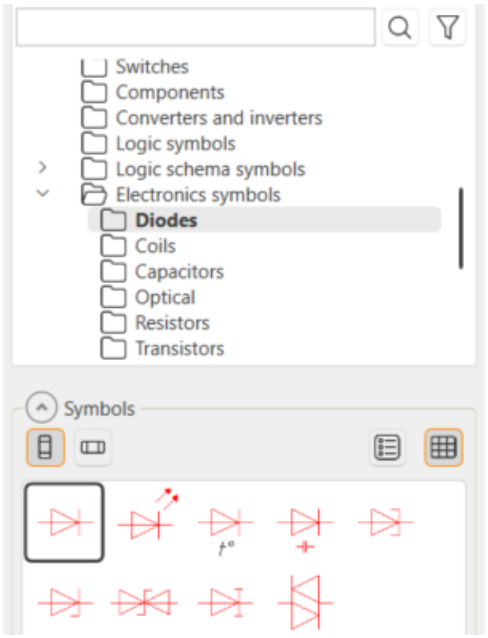
Let's insert symbols to the second sheet according to the example drawing.

Do the following:

1. Insert coils:
    a. In the **Symbols** window, select **Coils** and insert the coil symbol:



    b. Define the following:
        - **Coil ID** – From the drop-down menu, select **K10** and **K20**.
        - **Coil upper number** – *A1*
        - **Coil lower number** – *A2*
2. Insert the auxiliary contacts:
    a. In the **Symbols** window, select **Contact packs** and insert the contact symbol **1xchange-over contact**.
    b. Enter *K10* and *K20* as IDs.
3. Insert diodes next to the coils:
    a. In the **Symbols** window, select **Diodes** from the **Electronics symbols** menu and insert the diode symbol:



b. Insert the symbol into the drawing.

> Tip: Turn the diode with F8.

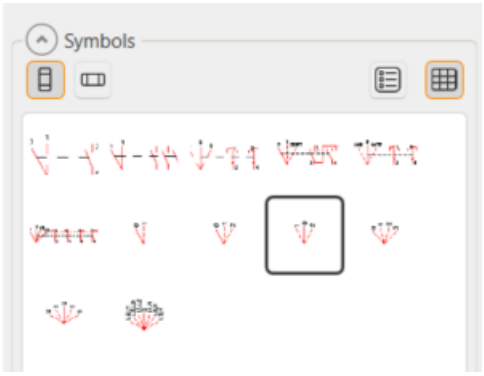
c. Leave component ID empty.

10.4. Insert a switch and contacts

Let's insert a knob for the **S01** switch, and then add for the **S01** switch.

Do the following:

1. In the **Symbols** window, select **Switches** and insert the switch symbol (OPEN/0/CLOSED):



2. Select the switch **S01** from the drawing.
3. Right-click, and select **Add auxiliary contact**. The **Adding contacts** menu opens.
4. Select a normally-open contact, and indicate the insertion point.

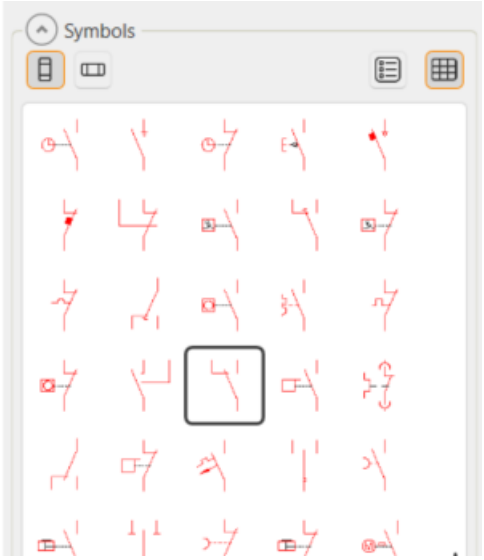
5. Define *1* as the upper number and *2* as the lower number.
6. Insert contact **S01:3/4** as described.
7. After inserting the contact symbols, click **OK** in the icon menu.

10.5. Insert change-over contacts

Let's insert change-over contacts and update cross-references.

Do the following:

1. In the **Symbols** window, select **Contacts** and insert the change-over contact symbol:



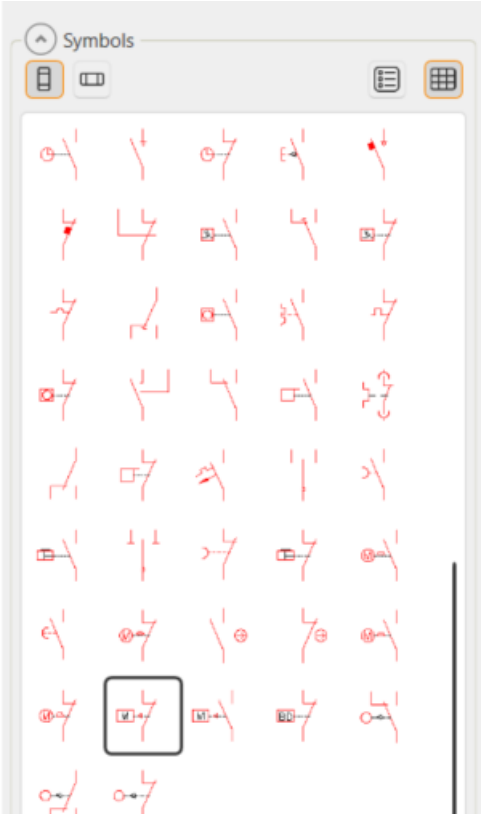
2. Define the following:
   - **ID** – *K92* and *K91*
   - **Contacts upper number** – *12* and *14*
   - **Contacts lower number** – *11*
3. Click **OK**.

10.6. Insert motor valve contacts

Next, let's insert motor valve contacts.

Do the following:

1. Insert a moment contact:
    a. In the **Symbols** window, select **Contacts** and insert the moment contact symbol:



    b. Define the following:
        - **ID** – *S1* and *S2*
        - **Contacts upper number**– *1*
        - **Contacts lower number**– *2*
2. Insert a position switch:

a. In the **Symbols** window, select **Contacts** and insert the normally-open contact symbol:

b. Define the following:
   - **ID** – *S3* and *S4*
   - **Contacts upper number** – *3*
   - **Contacts lower number** – *4*

The IDs and upper and lower numbers cannot be selected from the drop-down menus. Instead, you need to enter them manually.

11. Wiring

11.1. Draw wire

Important: You should always draw wires from wiring point to another so that the connection information is available when updating the databases. During wiring, wiring points are indicated with yellow squares.

You can draw over the symbols because this function knows how to cut the wire to separate wires.

Do the following:

1. Select to draw control circuit:
    - In Premium: Select **Schematics** tab > **Wiring** group > **Draw wiring**, and then select control circuit as the wire type.
    - In Basic: Select **Schematics** tab > **Wiring** group > **Draw wiring**.
2. Draw wires for diodes (**K10** and **K20**).
    - **=123-M01** internal wires
    - **S01:2** > **S01:4**
3. Continue wiring according to the example drawing.

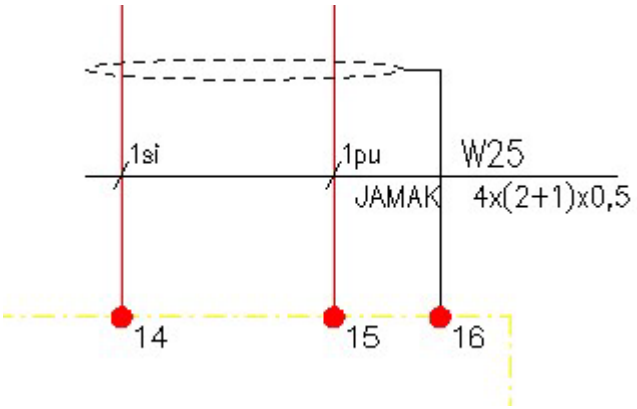
11.2. Draw cable sheathing

Next, let's draw cable sheathing.

Do the following:

1. Select **Schematics** tab > **Wiring** group > **Draw wiring** menu > **Draw cable shield**.
2. Indicate the start point for the ellipse representing the cable sheathing.
3. Indicate the end point for the ellipse.
4. Draw wire for the connector.
5. Stop drawing by right-clicking.

6. Leave the text field empty, and click **OK**.

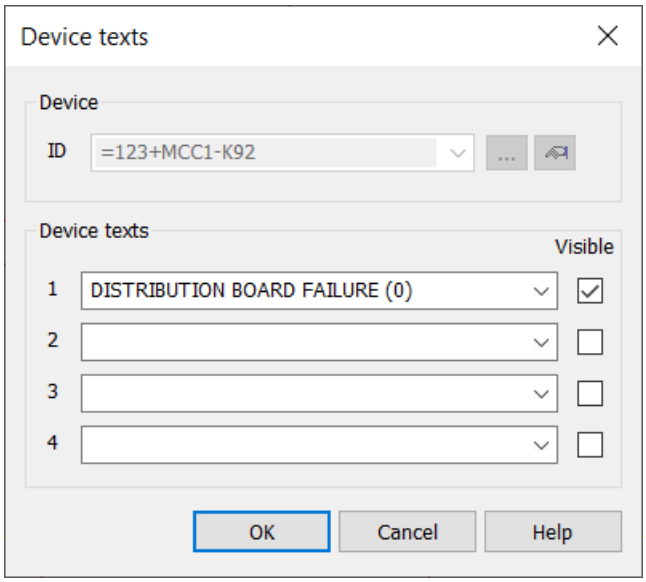


12. Create device texts

Finally, let's create the device texts for the drawing. Device texts (1–4) are free-form texts defined for devices, and they can be marked to the drawing, report or plate.

Do the following:

1. Select the change-over contact **K92**, right-click it and select **Insert devicetext to the drawing**. The **Device texts** dialog opens.
2. In the first field, enter *DISTRIBUTION BOARD FAILURE (0)* and select **Visible** for it.

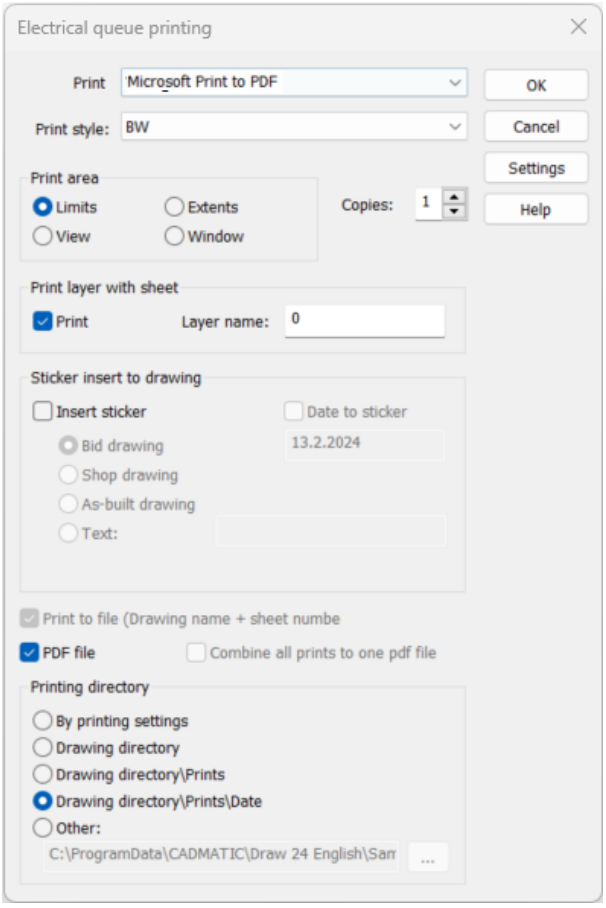


3. Click **OK**, and insert the device text in the drawing.
4. Insert device texts for each device as described: *FIELD FAILURE (0)*, *CLOSE MOMENT*, *OPEN MOMENT*, etc.

13. Print

Do the following:

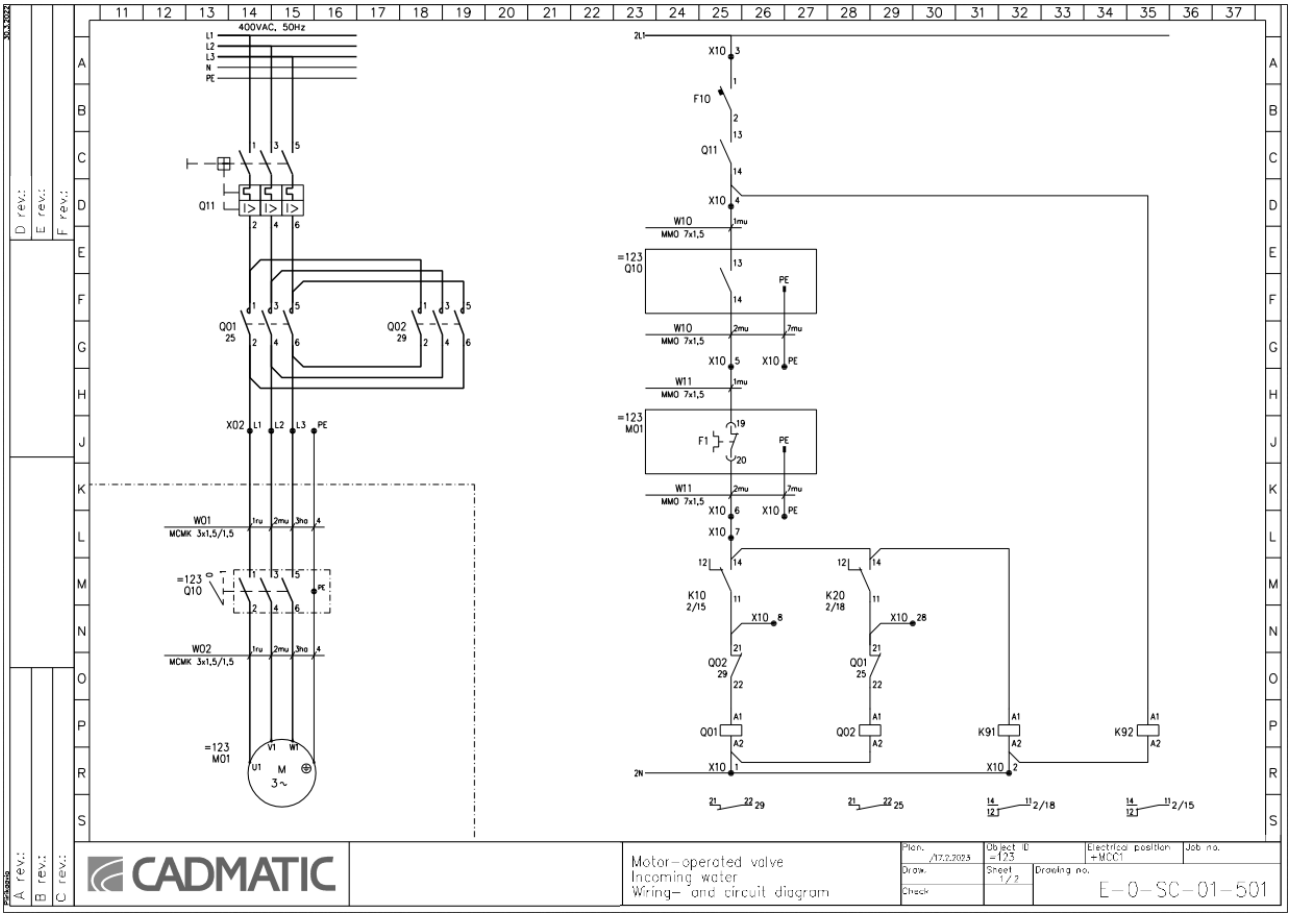
1. Select **File** > **Print** > **Queue printing from drawing sheets**. The **Print sheets** dialog opens.
2. Select sheets 1 and 2.
3. Close the dialog by clicking **OK**. The **Electrical queue printing** dialog opens.
4. Define settings according to the following image:

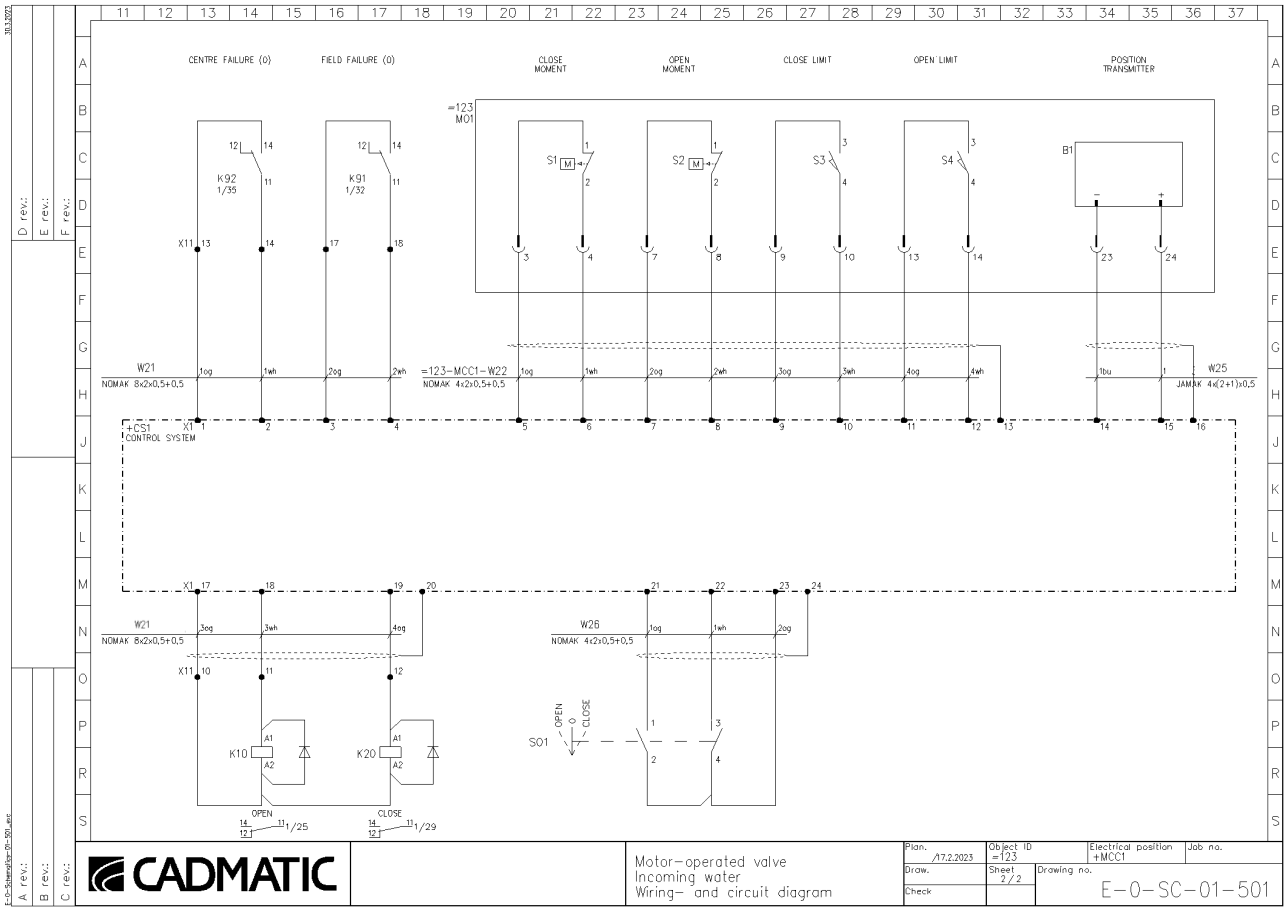


5. Click **OK**.
6. Check the line widths, and edit them if necessary.

You have now printed and completed the exercise.

14. Example drawing




CENTRE FAILURE (0)
FIELD FAILURE (0)
CLOSE MOMENT
OPEN MOMENT
CLOSE LIMIT
OPEN LIMIT
POSITION TRANSMITTER
=123 M01
K92 1/35
K91 1/32
S1
S2
S3
S4
B1
X11
W21
NOMAK 8x2x0,5+0,5
=123-MCC1-W22
NOMAK 4x2x0,5+0,5
W25
JAMAK 4x(2+1)x0,5
+CS1 CONTROL SYSTEM
X1
W26
NOMAK 4x2x0,5+0,5
K10
K20
SO1
OPEN
CLOSE
1/25
1/29
CADMATIC
Motor-operated valve
Incoming water
Wiring- and circuit diagram
Plan.
17.2.2023
Object ID
=123
Electrical position
+MCC1
Job no.
Draw.
Sheet
2/2
Drawing no.
Check
E-0-SC-01-501