CADMATIC Electrical
Electrical design for buildings

Contents

1. Introduction

This guide introduces you to the Layout toolset of CADMATIC Electrical. Via designing the electrics for an industrial building, you will learn the basics of adding electrical components to a building starting from preparing the floor plan drawing to printing the complete drawing. You will also learn about reference drawings, different symbol and wiring functions, and quantity calculations.

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

Some of the functions may not be available in Electrical Basic, or they work differently. This is always mentioned in connection with these functions.

2. Starting the exercise

First, we will save the floor plan to be used as a template for the drawing. Then, we will create an electrical drawing and insert the floor plan as a reference drawing to it.

2.1. Open the floor plan and save it as a DWG file

Let's open the floor plan from the sample project directory. We will save it as a DWG file in its own directory, to which we will collect all drawings created during this exercise.

Typically when saving to DWG, the *colors.cnv* conversion file is used. This file changes the pen numbers so that the colors in the DWG and DRW drawings remain identical.

Do the following:

1. Select **File** > **Open**. The **Open file** dialog opens.
2. Navigate to *[CADMATIC directory]\Samples\Electrical\Electrical Example Project\REF*.
3. Select the *A-0-FP-01-101_Production.dwg* file.
4. Click **Open**. The program opens the drawing on a new document tab.
5. Select **File** > **Save as**. The **Save As** dialog opens.
6. Navigate to *[CADMATIC directory]\Samples\Electrical*.
7. Create a new folder called *Layout exercise*, and open it.
8. Create a new folder called *REF*, and open it.
9. For file type, select **AutoCAD 2018-2025 Drawing (*.dwg)**.
10. Click **Settings**. The **General settings** dialog opens.
11. Check that the following settings below **DWG and DFX** are selected:
    - The **DWG save format** is **AutoCAD 2018-2025 Drawing**.
    - Below **Use conversion files for**, **DWG drawings** is selected, and the selected file is *colors.cnv*.
12. Return to the directory by clicking **OK**.
13. Click **Save**.

Before moving on, save the *A-0-FP-01-201_Production.dwg* file in the *REF* folder as described.

2.2. Check the scale

Let's check that the drawing has been drawn in real scale.

Tip: It is often enough to insert a linear light fixture and compare it to a door, for example. If the linear light fixture is bigger than the whole house, the drawing is likely 50 or 100 times too small. You should then scale it according to the following instructions.

Do the following:

1. Select **Tools** tab > **Utilities** group > **Calculate** menu > **Calculate distance**.
2. Turn the **End** snap on by pressing Shift down, right-clicking and selecting **End** to lock the cursor to the end points of lines.
3. In the drawing, find a distance whose correct length you know. This distance can be the width of a door frame or a window or the distance between two walls of a room.
4. Indicate the start point of the distance.
5. Indicate the end point of the distance. The program shows you the distance in millimeters on the command line.
6. Check if the length is the same as the selected distance. If the length is what you expected, the drawing is in the correct scale.

If the length is larger or smaller than expected, scale the drawing to the correct scale as follows:

1. Select **Home** tab > **Modify** group > **Scale** menu > **Scale drawing**.
2. Indicate the start point of a distance whose correct length you know.
3. Indicate the end point of the distance. The program shows you the distance on the command line.
4. Enter the correct length in millimeters on the command line.
5. Press Enter. The program scales the drawing according to the distance you entered.

2.3. Create an Electrical drawing

Let's create a new Electrical drawing, and attach the floor plan we previously saved as a reference drawing.

It is important to draw arrangements in real scale in the design mode. In addition, the print scale is needed, as well as the symbol scale factors in order to have the symbols in the desired size.

These settings make drawing easier, in addition to which they affect the final outcome – the drawing creation function goes through the settings systematically. By default, this function is activated when you start a new drawing.

Do the following:

1. Select **Electrical** tab > **Documents** group > **New**. The program opens a new document tab, and the **Start a new Electrical drawing** dialog opens.
2. Define the following:
    a. As the drawing type, select **Arrangement** below **Layout**.
    b. For the project directory, select *[CADMATIC directory]\Samples\Electrical\Layout exercise*.
    c. For the drawing name, enter *ELEC_2100.drw*.
3. Click **OK**. The **New Layout Drawing** dialog opens.
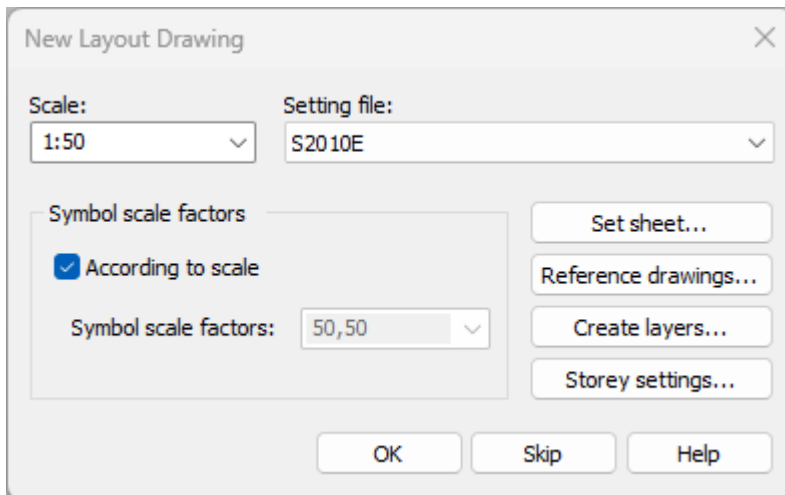
4. Define the following:
    a. For scale, select *1:50*.
    b. Select a setting file.

      c. Set the symbol scale factors according to scale.

5. Attach the previous floor plan as a reference drawing as follows:

      a. Click **Reference drawings**.

      b. Navigate to *[CADMATIC directory]\Samples\Electrical\Layout exercise\REF*.

      c. Double-click the *A-0-FP-01-101_Production.dwg* file. The **Attach reference drawing** dialog opens.
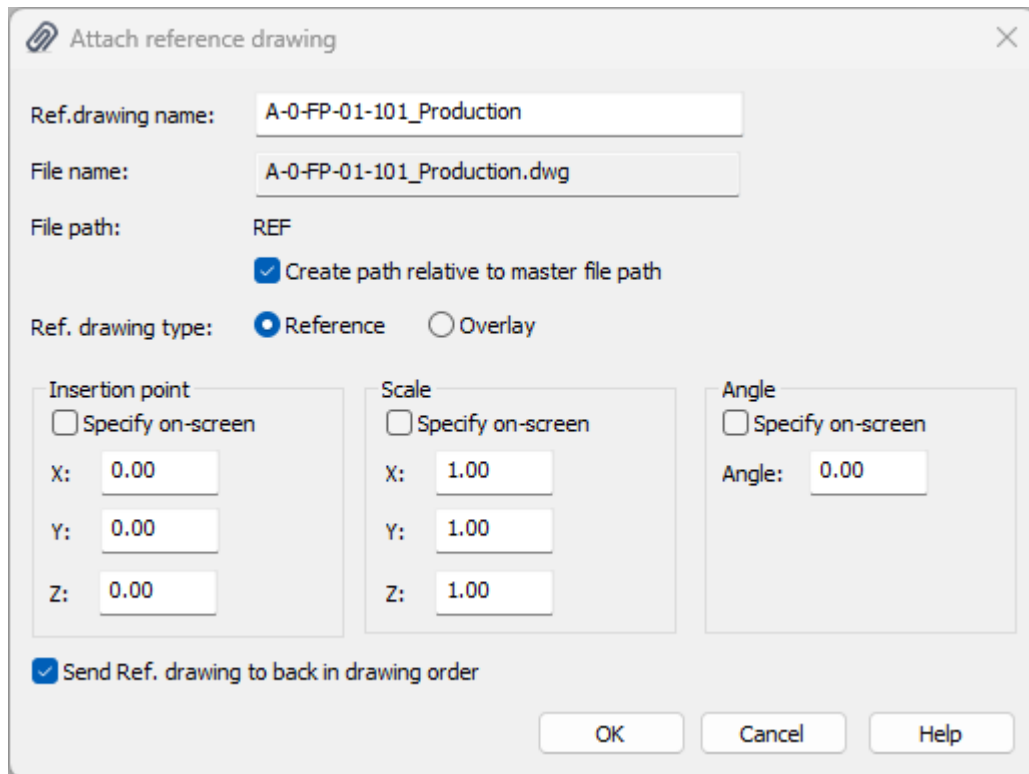
      d. In the **Ref. drawing name** field, enter *Floor plan 1st floor* as the description. This will not affect the file name.

      e. Select **Create path relative to master file path**. The relative reference drawing will be shown for everyone handling the master drawing, as long as they have the reference drawing and it has been saved to the same directory as the master drawing. The reference drawing type is an absolute directory location so it might not show correctly for everyone.

      f. If you do not want the reference drawing attached to the reference drawing to be shown, select **Overlay**.

      g. Accept the default settings for the reference drawing by clicking **OK**. The program attaches the floor plan as a reference drawing to the new drawing.

You can also add reference drawings via the **External references** window .

6. Click **OK**. Check that the **Layout** tab is active (underlined). If not, you need to change the drawing type by selecting **Electrical** tab > **Documents** group > **Drawing type** and then **Arrangement** below **Layout**.

Before moving on, create another drawing file by the name of *ELEC_2200.drw* and add *A-0-FP-01-201_Production.dwg* as the reference drawing as described.

2.4. Define storey settings

If the building has several storeys, the storey settings define which drawing represents each storey. Furthermore, other storey information can be included. You can define several planning areas for one storey.

Do the following:

1. Open story settings by selecting **Electrical** tab > **Settings** group > **Storey**. The **Project's storey settings** dialog opens.

2. Give the site a name. You can also define the origin and add a site plan. You can define several buildings for one site.

3. In the **Buildings** section, add a building with **Add**. You can define a name and an origin, and optionally IFC angle size and shape.

4. In the **Storeys** section, add the first storey with **Add**, add the drawing file *ELEC_2100.drw* for it and enter *1st floor* as the description.
   You can also define elevation and storey height. The storey ID will be created automatically.

5. Add the second storey in the same way. The elevation for the second storey will be created automatically based on the first storey's elevation.

6. Stop defining storey settings by clicking **OK**.

2.5. Turn off and lock layers

Next, let's turn off the unnecessary layers in the floor plan drawing to make editing easier – provided that everything has been drawn to their own layers in the reference drawing. Such unnecessary elements could be distance lines, room area information, or furniture.

We will also lock the floor plan layers, to prevent us from accidentally editing the reference drawing.

Do the following:

1. Turn off the unnecessary objects by selecting **Home** tab > **Layers** group > (**Turn off**).

2. Select the elements you want to turn off:
   - Room area information
   - Distances inside the building
   - Door codes
   - Window codes

   The application turns the layers of these objects off, so they become invisible.

3. Lock the reference drawing by selecting a file in the **External references** window, right-clicking it and selecting **Lock layer**.

The important functions related to starting electrical drawings are now done and we can move on to drawing.

3. Cableways

We will start by designing the cable routes, i.e. drawing cableways and trunkings. All the cableways are drawn using the same function by selecting the desired cableway type.

3.1. Draw cable tray

Let's draw a cable tray.

Do the following:

1. Select **Layout** tab > **Cableways** group > **Draw**. In the **Cableway** window, you can define settings for the cableway to be drawn. You can draw all the cableways via this window.
2. For the cableway type, select **Cable tray**.
3. For the type, select **KS80**.
4. For width/depth, enter *300*.
5. For elevation, enter *2900*, and select **Bottom of** from the drop-down menu.
6. For drawing line, select the left-most option and define *50* as the distance.
7. In the **Geometry** section, select **2D** and then the raster and corner as in the image below:
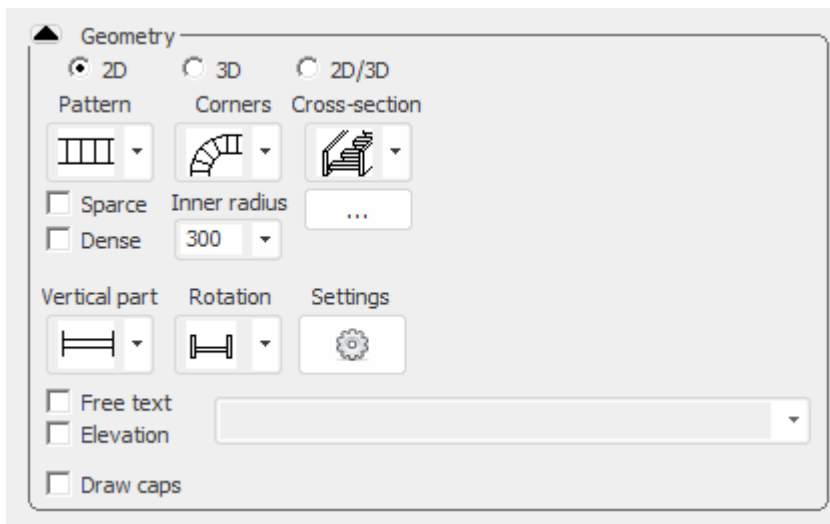
8. Start drawing. You can change the values (the width to **400**, for example) while you draw.

Cable tray T branch

If you need to draw a T branch, start drawing again and start from the desired spot. The program adds the branch to the indicated spot automatically.

The T branch is automatically added also when you end the cable tray to another tray in the drawing, as long as the elevation is the same.

Vertical cable tray

A vertical cable tray can be drawn by changing the elevation and selecting the angle. You do not need to stop drawing or use a different function.
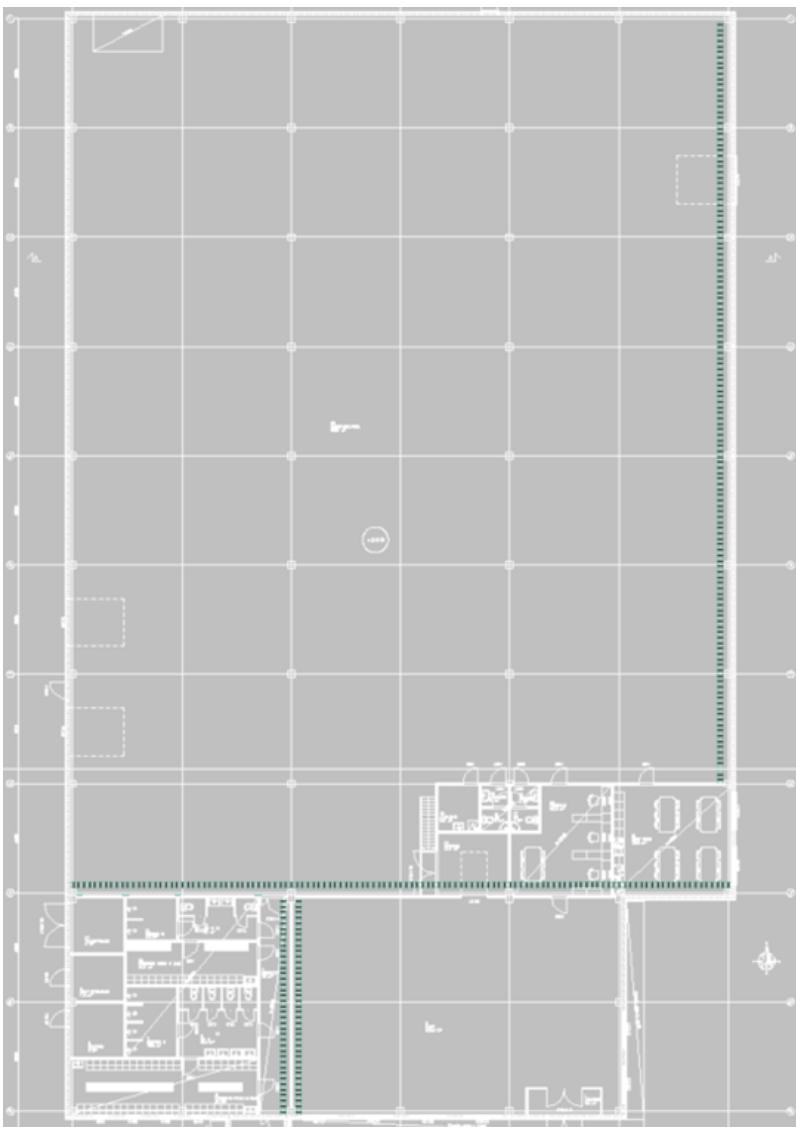
3.2. Make cableway go over

Next, we will add an ascent to a cableway.

Do the following:

1. Select **Layout** tab > **Cableways** group > **Over/under**.

2. Select the cable tray to which you want to make the ascent.

3. Indicate the ascent start point.

4. Indicate the ascent end point. The **Definition of cableway ascent** dialog opens.

5. As the elevation, enter *3500*.
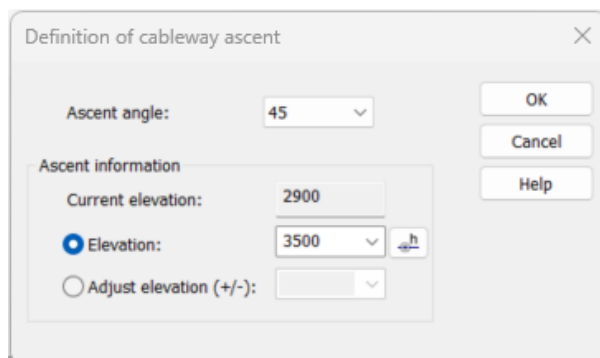
6. Click **OK**. The program adds the ascent to the cableway.

3.3. Draw vertical cableways

Next, we will insert vertical cableways.

Using F8 to rotate the symbol sometimes makes inserting easier, as you can rotate the symbol angle as you wish. The angle is based on the ortho angle.

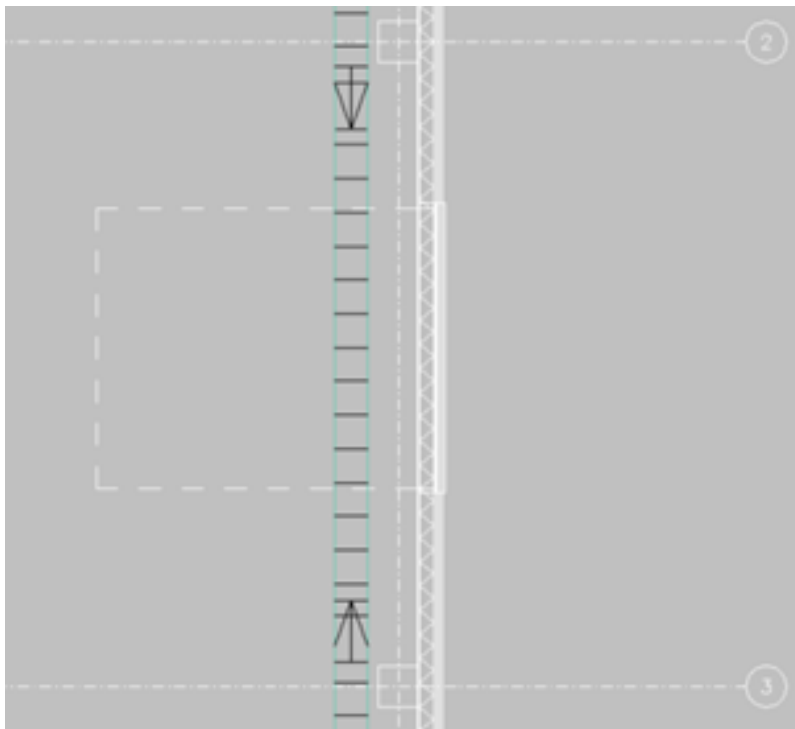You can change the insertion point in the symbol with the Tab key. Each press will move from one point to another.

You can easily draw straight line with ortho. If ortho is disabled, you can temporarily enable it with the Shift key (enabled as long as Shift pressed down).

Do the following:

1. Select **Layout** tab > **Cableways** group > **Draw** menu > **Vertical cableway**. The **Draw a vertical cableway** dialog opens.
2. Define the following:
   a. For the cableway type, select **Trunking**.
   b. For **Width/Depth**, enter *65*.
   c. For **Height**, enter *144*.
   d. For **Start elevation**, enter *0*.
   e. For **End elevation**, enter *2500*.
   f. Select the cross-section and vertical part symbols.
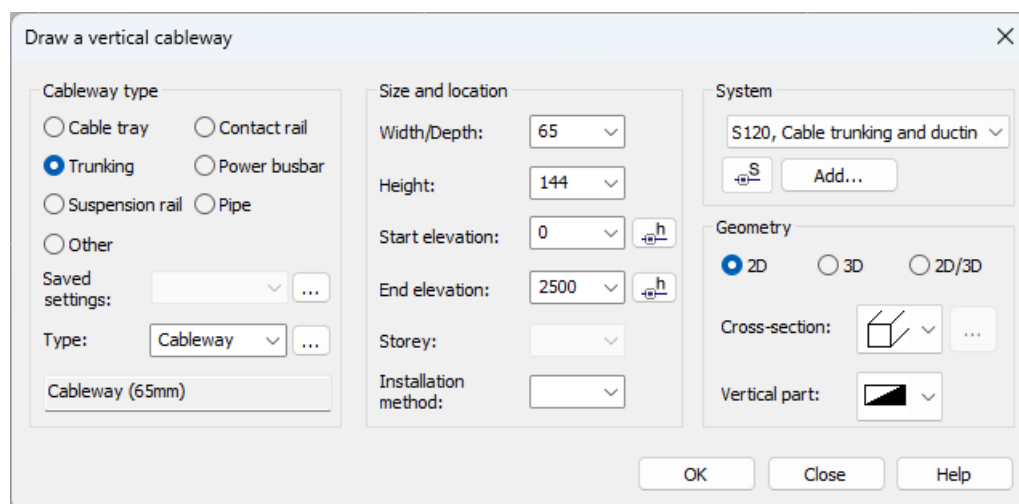
3. Start inserting by clicking **OK**.
4. Press the Tab key until the cursor is in the bottom right corner of the symbol.
5. Insert the vertical cableway to the door frame using the **End** snap.

6. Accept the 0 angle by right-clicking.

7. Start inserting another cableway by repeating the previous command by right-clicking.

8. Accept the size information by clicking **OK**.

9. Rotate the symbol to the correct angle by pressing F8.

10. Insert the vertical cableway to the wall using the **Nearest** snap.

11. After inserting the vertical cableways, select to draw cableways and draw trunking with the same sizes. Set to draw according to the top and set elevation as 1000 mm.
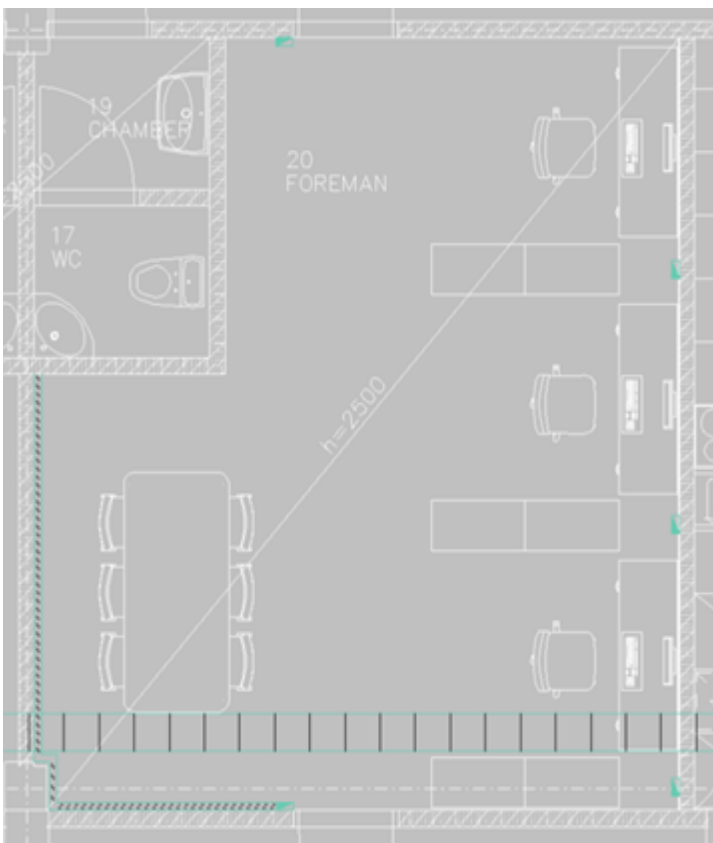
4. Symbols

In this section, we will explore the different ways of inserting symbols as well as auxiliary functions related to them.

4.1. Insert symbols

You can insert symbols via the **Symbols** window.

Note: The locale setting affects the symbols available – always select the ones suitable for your location.

Before insertion, we recommend to set the **Nearest** snap on as that will help inserting the symbols at equal distances. To set it on, select the **Tools** tab and then **Osnap to nearest** from the **Osnap** menu.
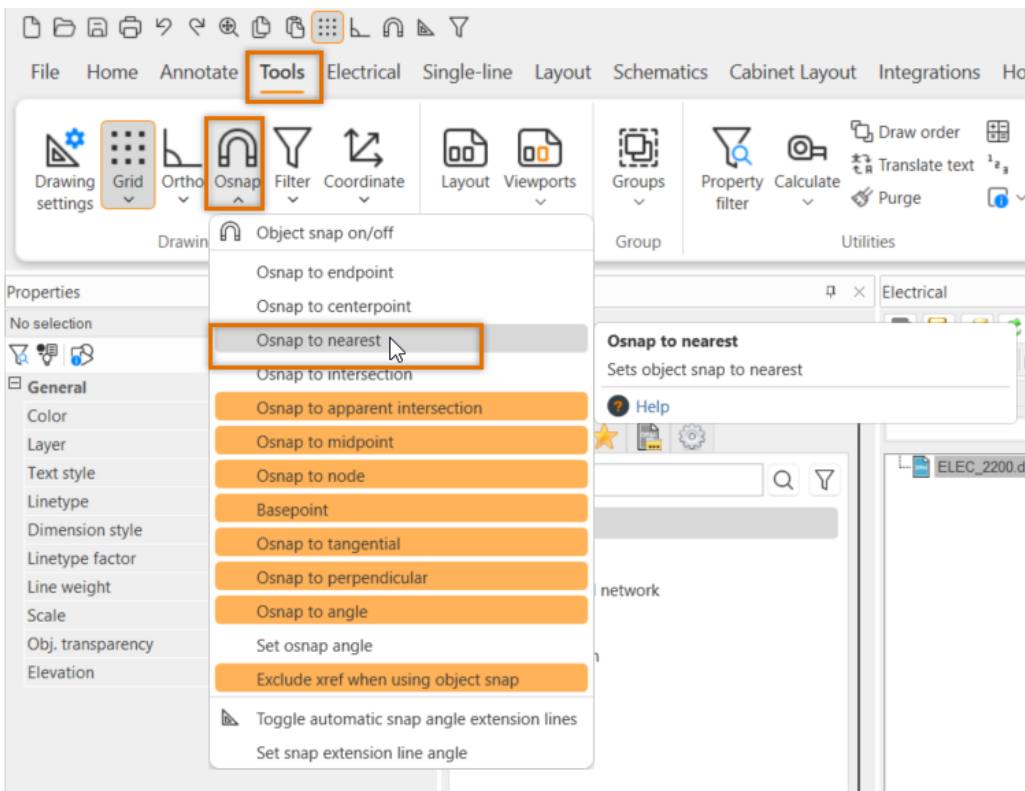
4.1.1. Insert a symbol

The instructions below contain the basic instructions for inserting a symbol, and you can use them later when you insert symbols in any drawing.

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

Do the following:

1. Open the **Symbols** window by selecting **Layout** tab > **Devices** group > **Symbols**. The window stays on the screen until you choose to close it.
2. From the **Electric** drop-down menu, select **Switches > Common**.
3. Insert the **Switch, 1-pole, flush** symbol by double-clicking it or by right-clicking it and selecting **Insert symbol to drawing**:
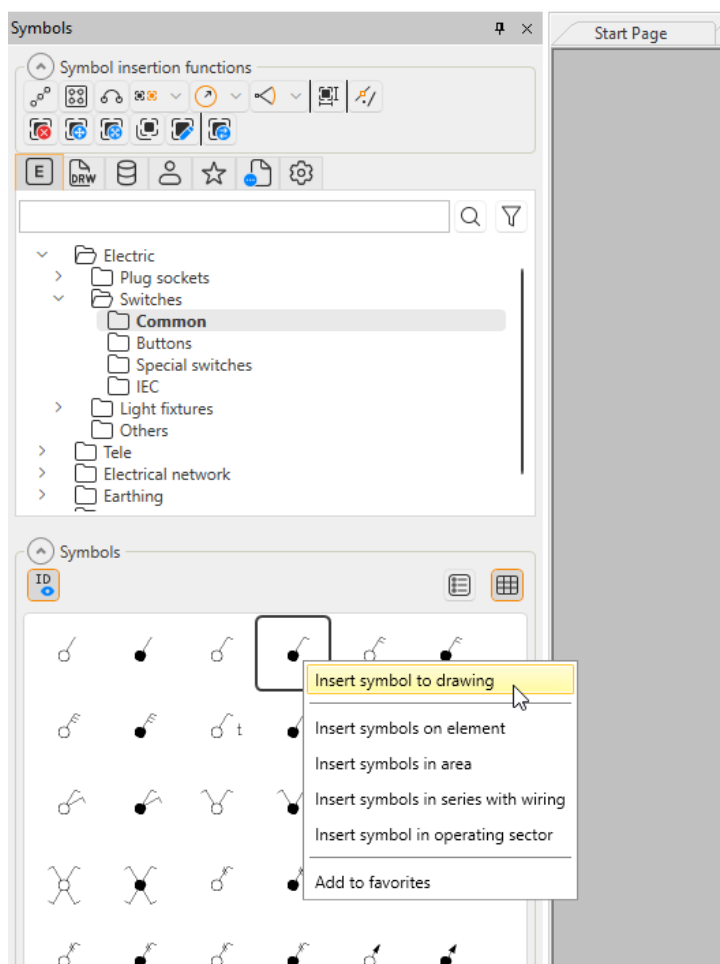
4. Insert the symbol:

    a. If necessary, rotate the symbol with F8.

    b. Indicate the location of the symbol.

    c. Accept the angle by right-clicking.

    d. Indicate the location of the next symbol.

    e. Accept the angle by right-clicking.

5. In the **Symbols** window, select **Electric > Switches > Common** and insert the **Crown switch, flush** symbol:
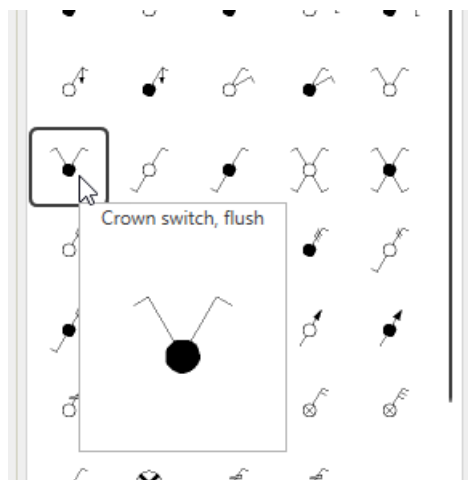
Tip: You can change the symbol in the cross-hair simply by selecting another symbol. If necessary, you can stop insertion with Esc or by double-tapping the mouse wheel.

4.1.2. Insert a symbol in the middle of a rectangle

Let's now insert a symbol in the middle of a rectangle.

Do the following:

1. In the **Symbols** window, select **Electric > Light fixtures > Common** and double-click the **Light fixture, junction box** symbol:
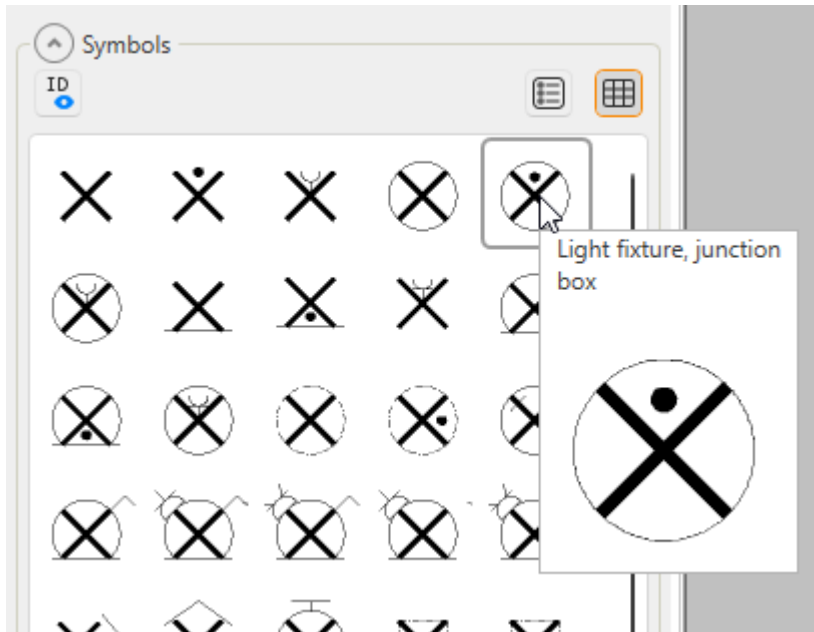
2. If necessary, rotate the symbol with F8.
3. Press Shift and right-click at the same time. A menu opens.
4. Select **Rectangle**.
5. Indicate the first corner of the room.
6. Indicate the second corner on the opposite corner of the room.
7. Indicate the angle of the symbol.
8. Accept the angle by right-clicking.
9. Repeat the steps for the other rooms.

4.1.3. Insert a symbol in the midpoint of a line

Next, we will insert a symbol in the middle of two points.

Tip: To make indicating corners easier, activate the **End** snap during this function: press Shift down, right-click and select **End**.

Do the following:

1. In the **Symbols** window, select **Electric > Light fixtures > Linear light fixtures** and double-click the **Linear light fixture, 650 mm, double plug socket, wall mounting** symbol:
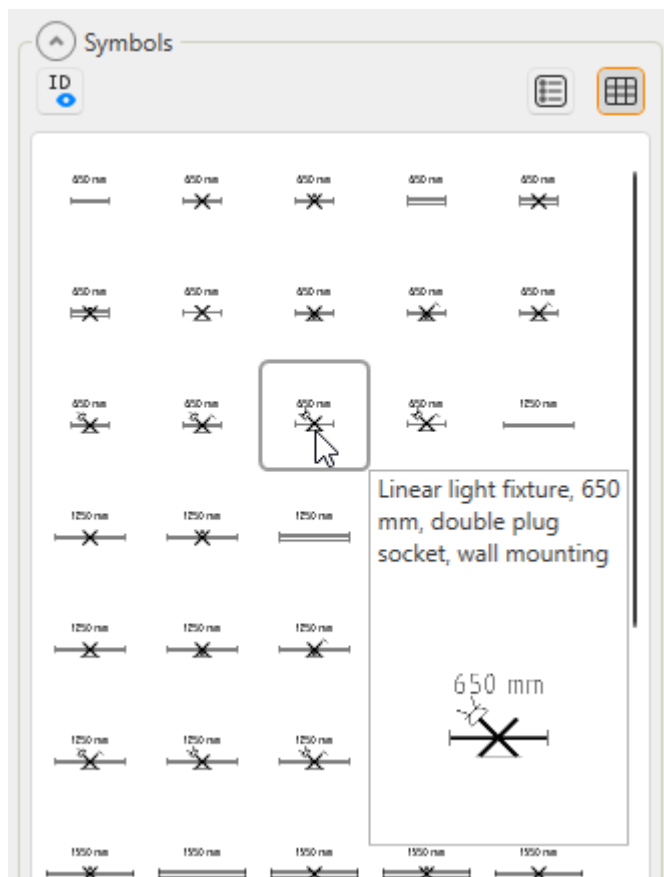
2. If necessary, rotate the symbol with F8.
3. Press Shift and right-click. A menu opens.
4. Select **Midpoint**.
5. Indicate the corner of the room.
6. Indicate another corner of the room.

7. Accept the angle of the symbol by right-clicking.

8. Repeat the steps for the other rooms.

4.1.4. Insert symbols an equal distance apart from each other

We will now insert light fixtures to a room with a specific distance.

Tip: To make indicating corners easier, activate the **End** snap during this function: press Shift down, right-click and select **End**.

Do the following:

1. In the **Symbols** window, select **Electric > Light fixtures > Common** and then the **Light fixture, flush (scalable)** symbol:
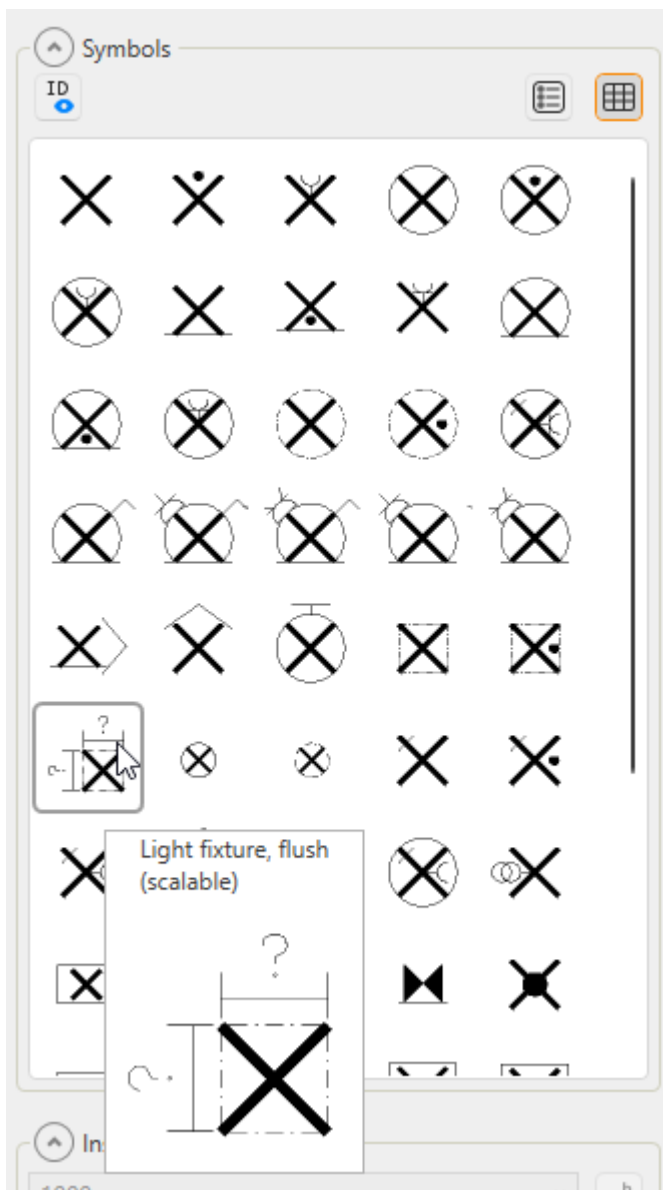
2. In the top part of the **Symbols** window, click the (**Insert symbols in area**) button:
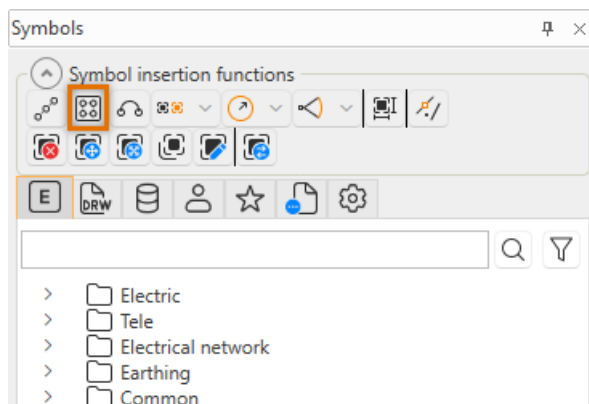
The **Areal insertion** dialog opens.

3. For **Symbol angle**, select **0**.

4. For **Insertion method**, select **With spacing**.

5. For spacing, enter *1800* in both fields.
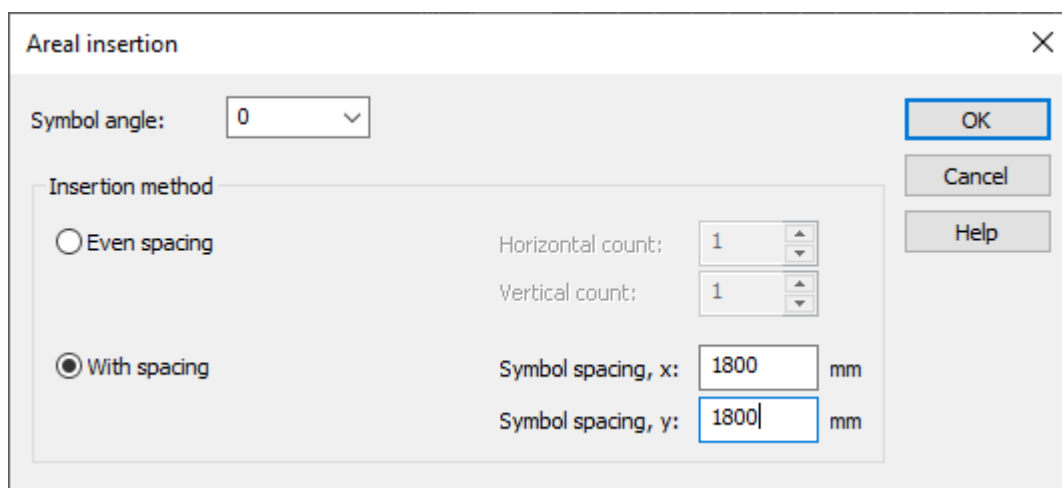
6. Click **OK**.

7. As the X size, enter *600* and click **OK**.

8. As the Y size, enter *600* and click **OK**.

9. Indicate the first corner of the area.

10. Indicate the second corner of the area.

4.1.5. Insert a symbol by copying

You can make drawing easier by copying symbols you have already inserted.

Do the following:

1. Select the symbols to be copied by forming a selection window around them from the bottom upwards.
2. Remove or add individual symbols by clicking them.
3. Accept the selection by right-clicking, and select **Copy**.
4. Indicate or enter the point you want to grab.
5. Indicate or enter the point to which to move the previous point.
6. Indicate or enter the next move, or stop copying with the Esc key.

> Tip: You can stop any function with Esc or by double-tapping the mouse wheel.
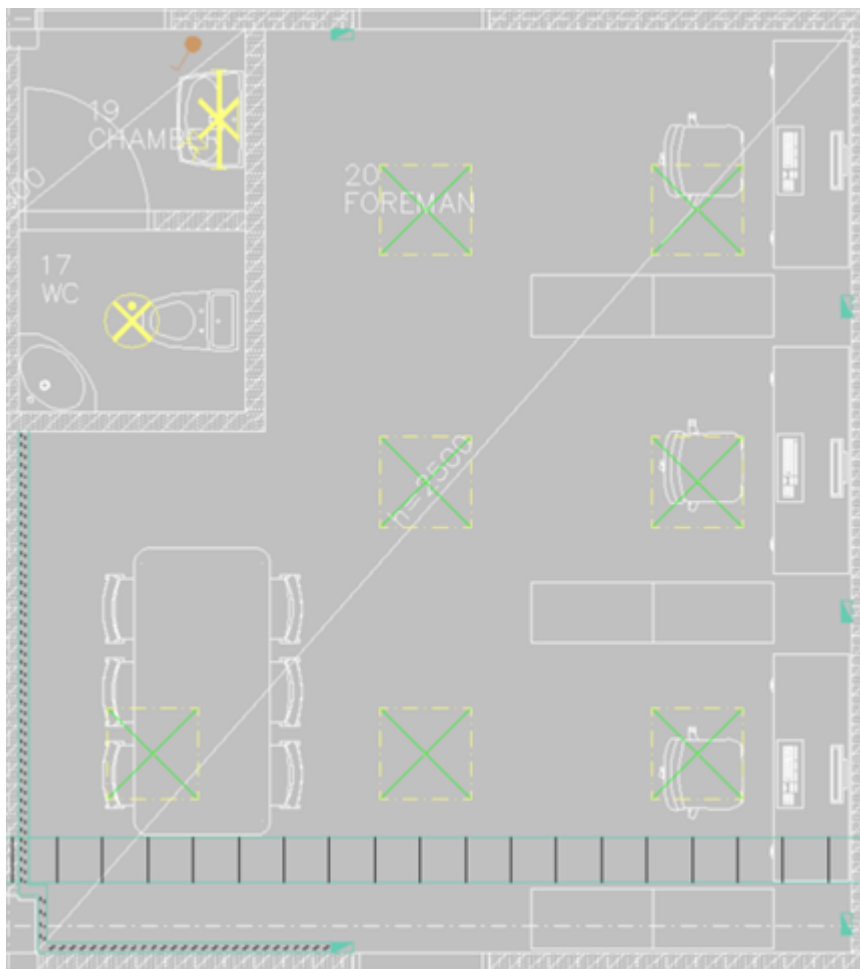
4.1.6. Insert symbols next to other symbols

Next, we will insert symbols next to other symbols.

With this function, you can insert symbols next to each other at a set distance. This makes inserting symbols next to each other as well as above each other easier. While this function is enabled, it is still possible to insert individual symbols.

Do the following:

1. Before using the function for the first time, define the spaces between symbols:

   a. From the top part of the **Symbols** window, click the ▣▣ (**Insert symbol next to another**) button and select **Set spaces between symbols**:
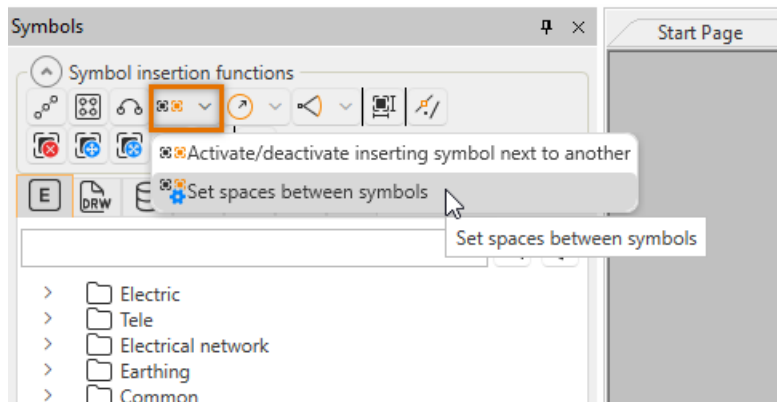
   The **Adjacent symbol inserting** dialog opens.

   b. As the space when inserting next to another symbol, define *250*.

   c. As the space when inserting above another symbol, define *250*.

   d. Click **OK**. You can change the values whenever necessary.

2. Set the function on in the **Symbols** window by clicking the ▣▣ (**Insert symbol next to another**) button and selecting **Activate/deactivate inserting symbol next to another**.

3. In the **Symbols** window, select **Electric > Plug sockets** and double-click the **Plug socket, 1-gang, protective contact, flush** symbol.

4. Take the mouse pointer close to the symbol next to which you want to insert the selected symbols. The direction will be based on the direction from which you are approaching the symbol.

5. Insert the symbol by clicking.

6. In the **Symbols** window, select **Electric > Plug sockets** and double-click the **Plug socket, 2-gang, protective contact, flush** symbol.

7. Right-click in order to insert the selected symbol with the basic symbol insertion function.

8. Insert the symbol for the drying cabinet.

9. Right-click in order to insert the selected symbol with the basic symbol insertion function.

10. Insert a symbol in the cableway using the **Nearest** snap.

11. Take the mouse pointer to the left of the plug you just inserted, and insert the new plug by clicking.

12. In the **Symbols** window, select **Tele > Data** and double-click the **Data plug socket, flush mounting** symbol.

13. Take the mouse pointer to the left of the plug you just inserted, and insert the new plug by clicking.

14. If necessary, set the function off in the **Symbols** window by clicking the **Insert symbol next to another** button and selecting **Activate/deactivate inserting symbol next to another**.
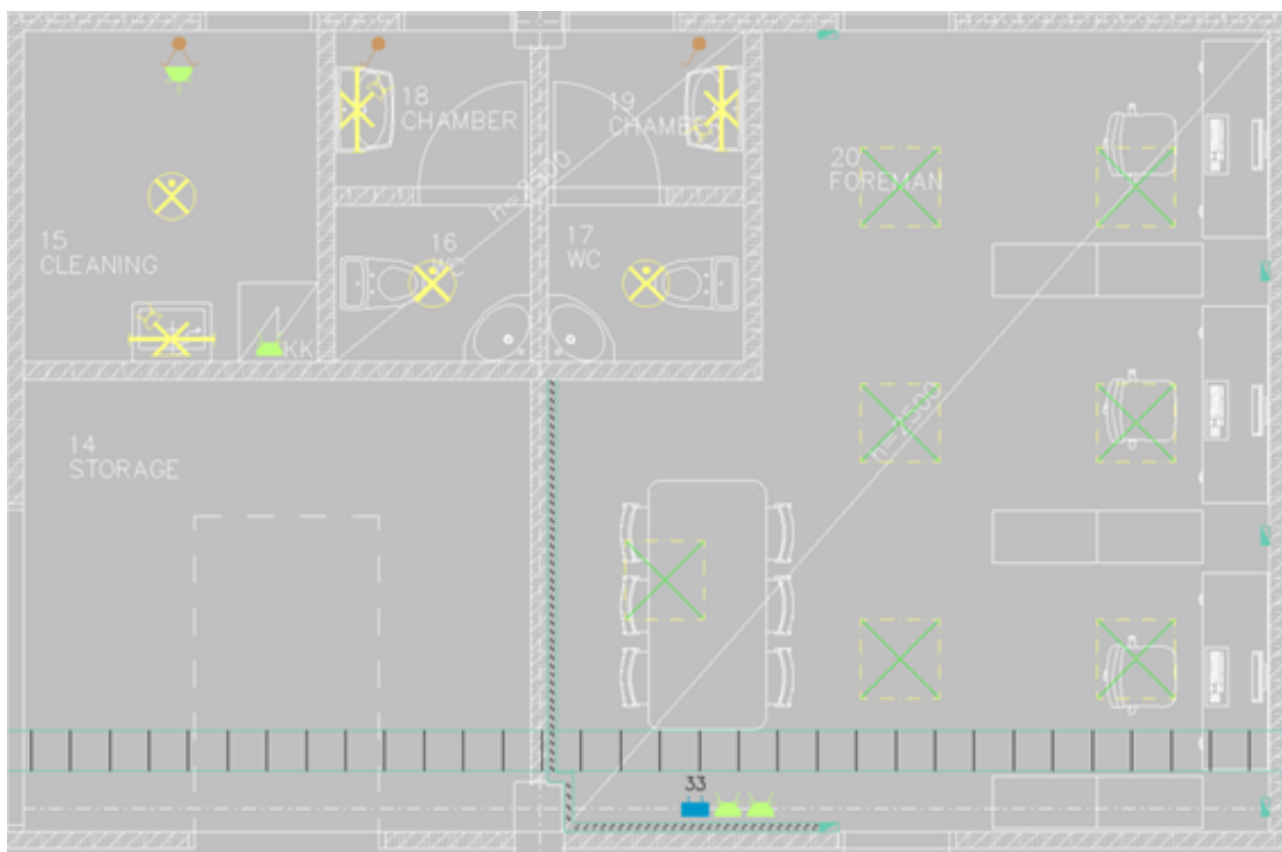
4.2. Move a symbol

Do the following:

1. Select the symbol you want to move.

2. Accept the selection by right-clicking, and then select **Move symbol** from the menu.

3. If necessary, rotate the symbol with F8.

4. Indicate or enter the displacement i.e. the point to which the symbol will be moved to.

   Alternatively, enter a relative displacement *@0,600*, which means that the symbol will be moved from the current location 0 mm in the X direction and 600 mm in the Y direction.

4.3. Change elevation data for the drawing

You can set default elevation values for different systems. When you insert a symbol with that system, it will get the elevation value set. Changes in these settings will not be applied to the symbols already inserted.

Do the following:

1. Select **Layout** tab > **Markings** group > **Elevation** menu > **Set default elevation values for current drawing**. The **Set default elevation values for this drawing** dialog opens.
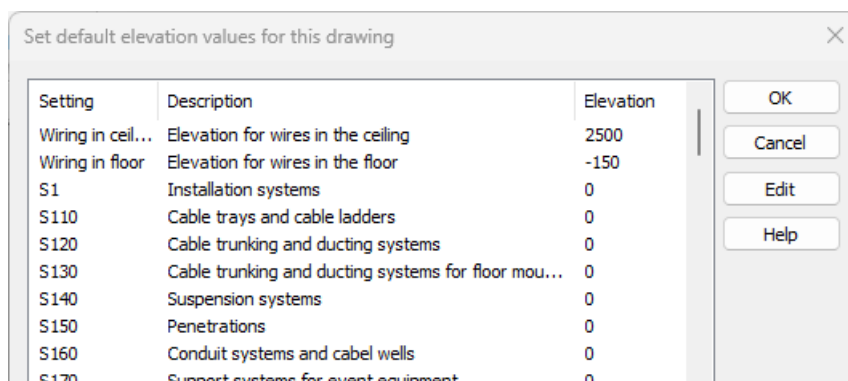
2. Select the system whose default elevation value you want to change.

3. Click **Edit**. A dialog opens.

4. Enter the new elevation for the system.

5. Click **OK**.

6. Accept the change by clicking **OK**. Alternatively, discard changes with **Cancel**.

4.4. Change elevation data for symbols

Next, we will change the elevation for symbols. You can change the elevation both for individual symbols and several symbols at once.

Do the following:

1. Select **Layout** tab > **Markings** group > **Elevation** menu > **Define element elevation data afterwards**.

2. Select the plugs for cleaning, and accept by right-clicking. The **Elevation change** dialog opens.
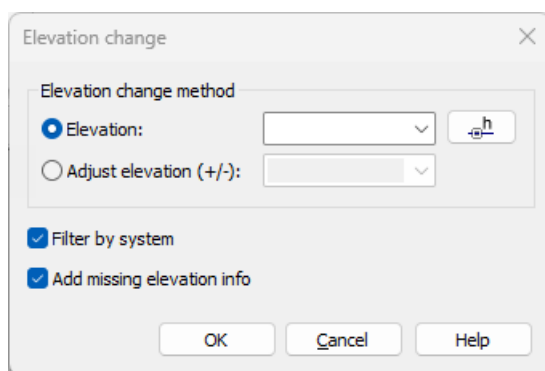
3. Enter *1700* as the new elevation, and click **OK**.

4. Repeat the previous function by right-clicking.

5. Select the cableway plugs and the data plug, and accept the selection by right-clicking.

6. Enter *940* as the new elevation, and click **OK**.

4.5. Mark elevation data to the drawing

Let's mark the elevation data of a symbol into the drawing and change the elevation of an object.

Do the following:

1. Select **Layout** tab > **Markings** group > **Elevation**.

2. Indicate the element for which you want to mark elevation.

3. Indicate the location of the leader arrow.

4. Repeat the previous function by right-clicking.

5. Indicate the drying cabinet plug.

6. Indicate the location of the leader arrow.

7. Double-click the marking text.

8. Edit the value to *h=2200*.

9. Accept with **OK**.
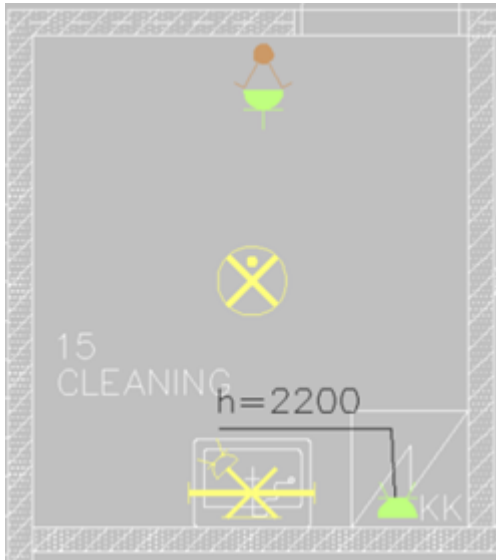
4.6. Save a symbol package

If you use certain symbol combinations often, it is beneficial to save them as symbol packages. This makes insertion easier, in addition to which you can avoid unnecessary elevation changes.

The program saves the elevation data for the symbols included in symbol packages. After insertion, symbol package symbols are handled as individual symbols so that quantity calculations, for example, will be correct.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbols** menu > **Save symbol package**.
2. Select the switch and the plug socket for cleaning to be saved in the symbol package.
3. Accept the selection by right-clicking. The **Save output package** dialog opens.
4. Enter *SO_PACKAGE1* as the name.
5. Click **OK**.
6. Indicate the base point for the symbol package near the wall on the centerline of the symbols with the **Nearest** snap. This way, the packages will be inserted as far away from the wall as the other symbols. The **Tip text for this symbol package** dialog opens.
7. For the tip text, enter *SO_PACKAGE1 1-switch and plug socket for cleaning*.
8. Click **OK**.
9. Indicate the corner points of an area that you want to see in the icon menu.

4.7. Insert a symbol package

Let's then insert the symbol package we just saved.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **User symbol packages**. An icon menu opens.

2. Select the **SO_PACKAGE1 1-switch and plug socket for cleaning** icon.

3. If necessary, rotate the symbol with F8.

4. Insert the symbol package in the middle of the vertical cableway.

5. Insert the same symbol package again by repeating the previous command by right-clicking.

6. If necessary, rotate the symbols with F8.

7. Insert the symbol package in the middle of the vertical cableway.

8. Insert the same symbol package again next to the break room door.

4.8. Change a symbol

First, insert switches into the cableways by the door in room 20. Then follow the instructions below.

Tip: You can easily change the snap mode by pressing down Shift and right-clicking. You can then select the snap mode to be used during the next function from the menu.

Do the following:

1. Open the **Symbols** window by selecting **Layout** tab > **Devices** group > **Symbols**.

2. Click the (**Change symbol**) button.

3. Indicate a switch in room 20.

4. In the **Symbols** window, select **Electric > Switches** and double-click the **Change-over switch, flush** symbol.

5. Select both of the switches in room 20.

6. Accept the selection by right-clicking.

4.9. Create and insert symbol groups

Next, we will create and insert symbol groups into the drawing.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **Symbol group**. The **Symbol group** dialog opens.

2. Click **New**. The **New symbol group** dialog opens.

3. Enter *Workstation 1* as the name.

4. Enter *PS + PS + RJ45* as the description.

5. Click **OK**.

6. Select the symbol group you just created from the left.
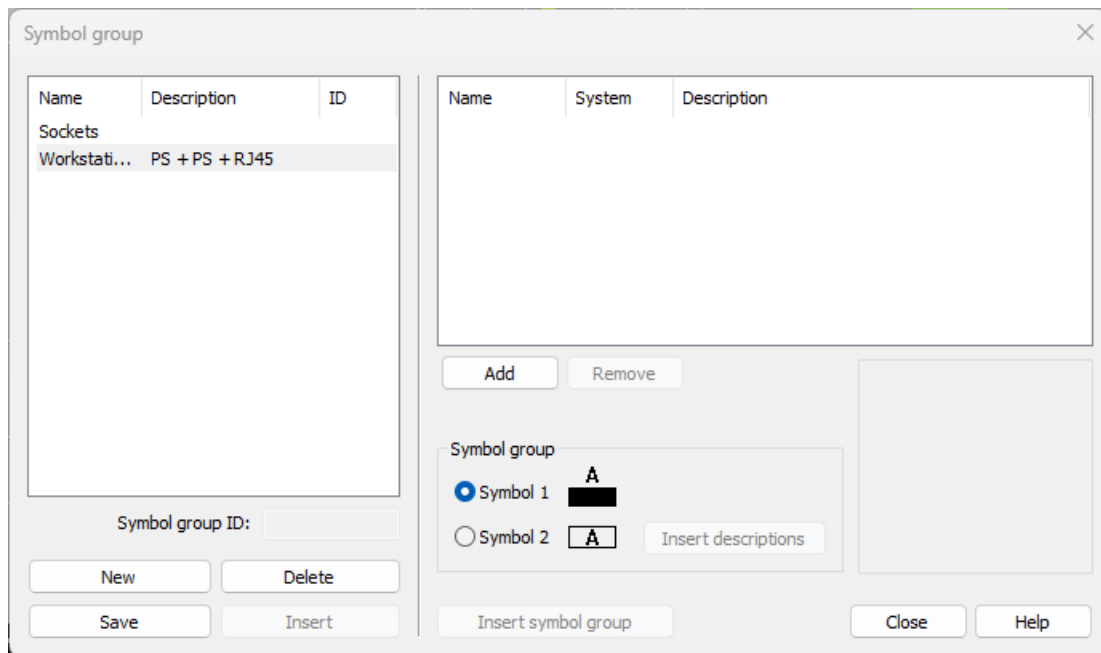
7. Select the symbols for the package:

    a. Click **Add**.

    b. In the **Symbols** window, select **Electric > Plug sockets** and double-click the **Plug socket, 2-gang, protective contact, flush** symbol.

    c. Click **Add**.

    d. In the **Symbols** window, select **Electric > Plug sockets** and double-click the **Plug socket, 2-gang, data, flush** symbol.

    e. Click **Add**.

f. In the **Symbols** window, select **Tele > Data** and double-click the **Data plug socket, flush mounting** symbol.

8. Click **Save**.

9. In the **Symbol group ID** field, enter *A*.

10. Click **Insert**.

11. If necessary, rotate the package symbol by pressing F8.

12. Insert the package symbol in the middle of the vertical cableways using the **Center** or **Midpoint** snap.

13. Stop inserting package symbols by pressing Esc.

14. Select **Symbol groups** from the **Symbol functions** menu again. The **Symbol groups** dialog opens.

15. Click **Insert descriptions**.

16. Insert the descriptions in the drawing outside the building.

4.10. Insert symbols on objects

Let's revise what we have learned so far and learn another useful method for inserting symbols, i.e. inserting symbols on objects.

We will now add suspension rails and light fixtures.

Tip: You can easily change the snap mode by pressing down Shift and right-clicking. You can then select the snap mode to be used during the next function from the menu.

## 4.10.1. Define and draw the suspension rail

Let's first add one suspension rail.

Do the following:

1. Start drawing cableways by selecting **Layout** tab > **Cableways** group > **Draw**.

2. In the **Cableway** dialog, enter the required information.

3. Start drawing the suspension rail as a relative displacement from the desired spot.

4. Press Shift and right-click. A menu opens.

5. Select **XY-offset**.

6. On the command line, enter *0* for X offset.

7. Enter *1500* for the Y offset.

8. For the offset basepoint, select the point where the module line and the wall intersect with the **Intersection** snap.

9. Indicate the end point of the suspension rail across the room on the right wall with the **Perpendicular** snap.

4.10.2. Insert light fixtures on the suspension rail

Let's insert light fixtures on the suspension rail. The suspension rail is still too long but we will shorten it later – this way we can utilize the function that divides the light fixtures at even spaces using the whole room width.

Do the following:

1. From the top part of the **Symbols** window, click the ⚬°° (**Insert symbols on element**) button.

2. In the **Symbols** window, select **Electric > Light fixtures > Linear light fixtures** and double-click the **Linear light fixture, 2x1250 mm** symbol. The **Insert symbols on element** dialog opens.

3. Select **Amount**.

4. Enter *6* as the amount.

5. For alignment, select the middle option where the program leaves space between the symbols and the ends of the line.

6. For angle, select **By element**.

7. Select **Electrical compatible symbol insertion (with coordinate z-values zeroed)**.

8. Select **Attach information for 3D direction**.

9. Select **Skip prompt attributes**.

10. Click **OK**.

11. Click the suspension rail. The application inserts six light fixtures on the suspension rail with equal distances.

12. End the function.

4.10.3. Copy the light fixture rail

Do the following:

1. Select the suspension rail and the light fixtures on it.

2. Right-click, and select **Copy**.

3. Indicate a base point anywhere in the drawing.

4. On the command line, enter *@0,3000* as the relative displacement. This means the basepoint will be moved from the indicated spot 0 mm in the X direction and 3 000 mm in the Y direction.

5. Press Enter.

6. Copy the rail and the light fixtures so that you will have four suspension rails with 3 meters between them.
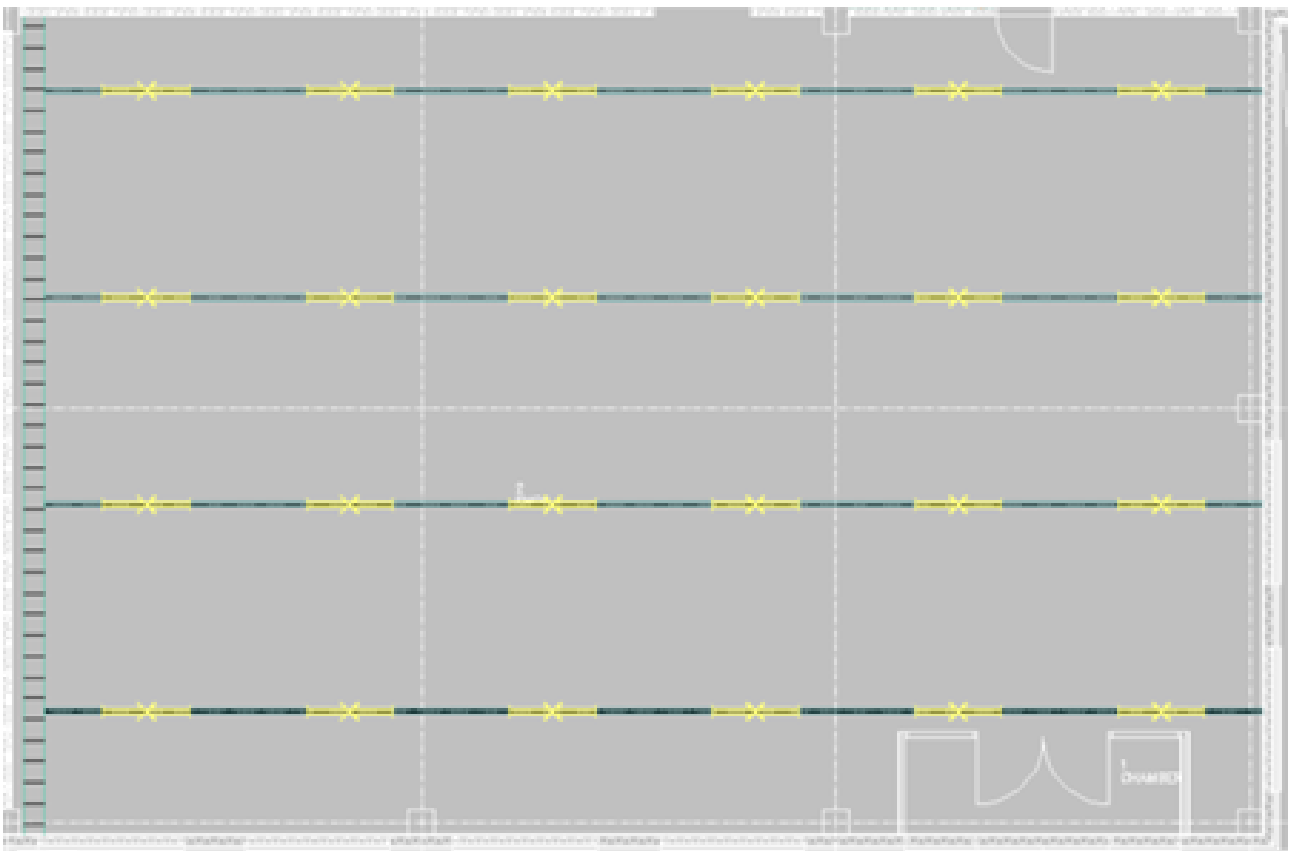
4.11. Add heaters

Let's add heaters next.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **Heaters** menu > **Heating radiators**.

2. Enter heater information:

   a. As power, enter *700*.

   b. As width, enter *1100*.

   c. As height, enter *400*.

   d. As depth, enter *40*.

   e. From **Thermostat**, select **Right**.

3. Click **OK**.

4. If necessary, rotate the symbol with F8.

5. Indicate the insertion point.

6. Accept the angle by right-clicking.

7. End the function by clicking **Close**.
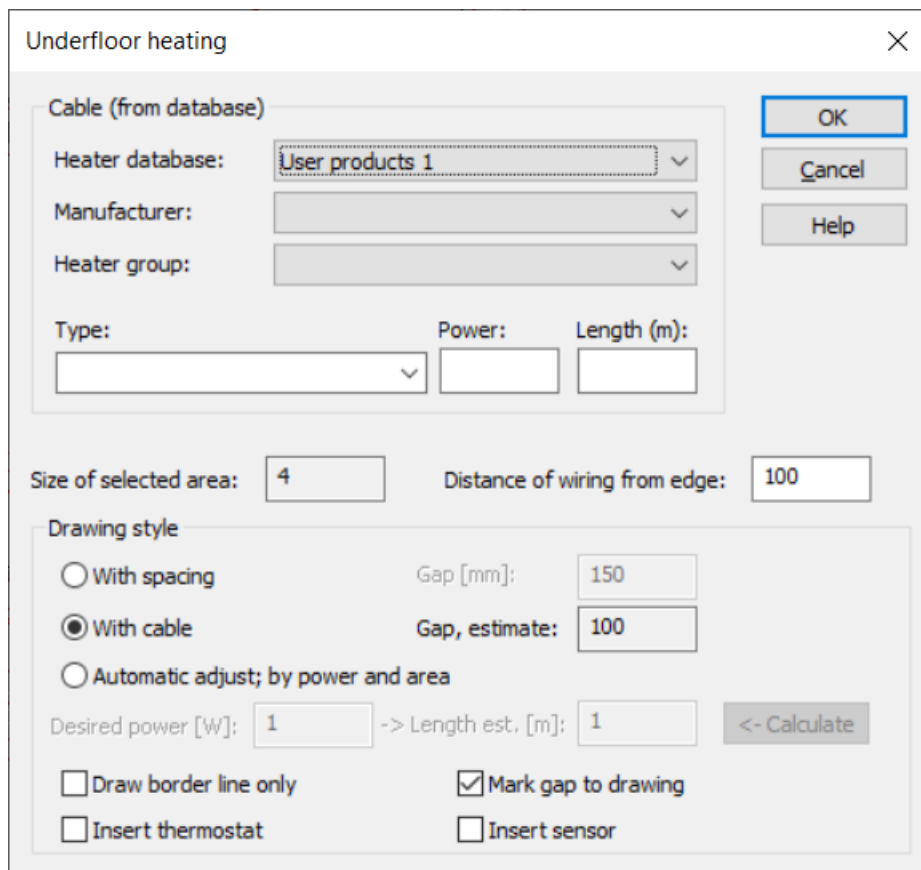
4.12. Add floor heating

To prevent the outdoor areas from freezing, we will add anti-frost cables with the floor heating function.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **Heaters** menu > **Cabling to rectangular area**.
2. Indicate the area to be heated:
   a. Indicate the first point close to the wall and near the side of the front door.
   b. On the command line, enter *@4000,-1000* as the relative displacement for the second point.

   The **Underfloor heating** dialog opens.

3. Select the cable type and manufacturer.Enter the cable type and the manufacturer.Select the cable type and manufacturer. In Electrical Basic, you need to enter the information manually.

- **Heater database** – Sähkönumerot.fiUser products 1–9Sähkönumerot.fi or User products 1–9 in Electrical Basic
- **Manufacturer** – DEVIflex
- **Heater group** – 81 Heaters
- **Type** – DEVIflex 18T 59m 1075m 230V

4. For **Distance of wiring from edge**, enter *50*.

5. For **Drawing style**, select **With cable**.

6. Select **Mark gap to drawing**.

7. Click **OK**.

8. Select the short side of the inner rectangle.

9. Insert the leader text by indicating the start and end points of the leader.

10. Copy the floor heating under the first cabling with 1 meter space between them:

   a. Select the cabling and the leader.

   b. Right-click, and select **Copy**.

   c. Indicate a base point somewhere in the drawing.

   d. Enter *@0,-1000* as the relative displacement.

   e. Enter *@0,-1000* as the new displacement.

   > **Tip:** Press the arrow key up to easily get the latest command line entries.

   f. Stop copying objects by pressing Esc.

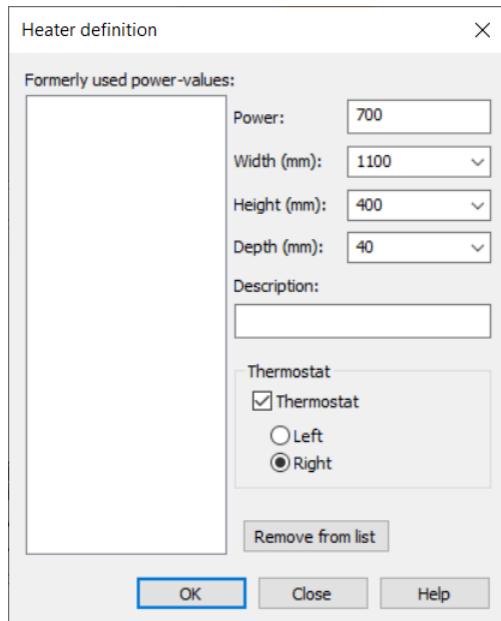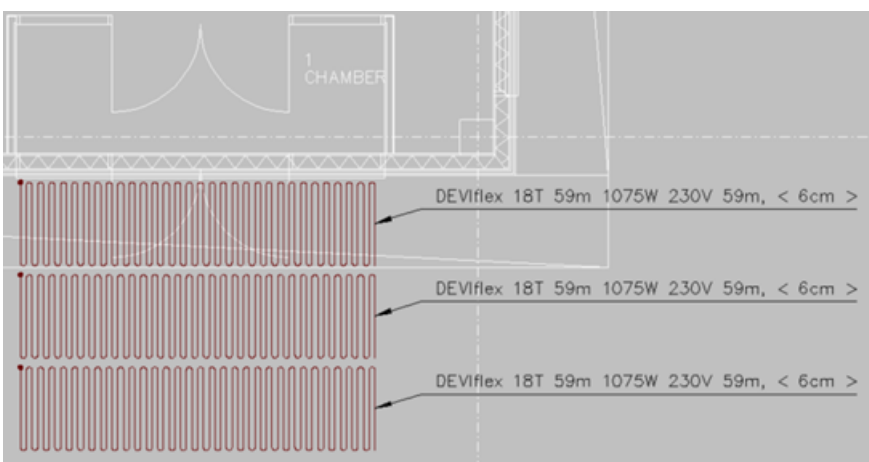# 4.13. Add thermostat and sensor wiring

Let's then add a thermostat, and sensor wiring.

Do the following:

1. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **Heaters** menu > **Thermostat**.
2. If necessary, turn the symbol by pressing F8.
3. Indicate the insertion point.
4. Draw a cable from the thermostat to the floor heating.
5. Select **Layout** tab > **Devices** group > **Symbol functions** menu > **Heaters** menu > **Sensor and wiring**.
6. If necessary, turn the symbol by pressing F8.
7. Indicate the insertion point.
8. Accept the angle by right-clicking.
9. Select the start point for the sensor wire.
10. Indicate the next point. Using ortho helps inserting the wire.

   Tip: If ortho is not on, you can activate it with Shift. Ortho will then be on for as long as Shift is pressed down. This also works vice versa, i.e. when ortho is on, you can temporarily turn it off with Shift.

11. Indicate the last point of the wire to the thermostat.

5. Wiring

You can draw wiring in two different ways: with or without wiring information. Both are done via the **Wiring** window.

- In this exercise, we draw with wiring information. Wiring with wiring information is more "intelligent", and the type of the wire or cable can be given in connection with drawing. You can define wiring type afterwards as well.

  The program does not take undefined wirings into account in quantity calculations. When you draw wiring, make sure to connect the wire to a wiring point in the symbol. This way, the wiring gets the correct elevation data from the symbol, for example.

  Wiring functions include predefined wire sets. You can select the desired cables from the cable database. The selected cables are project-specific in Electrical Premium and user-specific in Electrical Basic.

- Whenever the wiring function is active, the **Wiring** window is displayed and you can select the desired wiring style. You can also change the wiring height and wiring information during wiring. Wiring symbols are also available in the **Wiring** window. You can start wiring from a wiring symbol or end it on a wiring symbol.

- While wiring, both the wire/cable and the elevation are shown on the status bar. You should keep note of them during wiring, you can change the elevation and type during wiring.

5.1. Draw flush wiring

Next, we will draw flush wiring and select some cables as the cable types to be used in this project.

Do the following:

1. Select **Layout** tab > **Wiring** group > **Draw**. The **Wiring** window opens.

2. For style, select **Curve**.

3. Select **Single wiring**.

4. For elevation, select **Other** and define *3000* as the value. If you cannot change the value, first clear the selection on **Autom**.
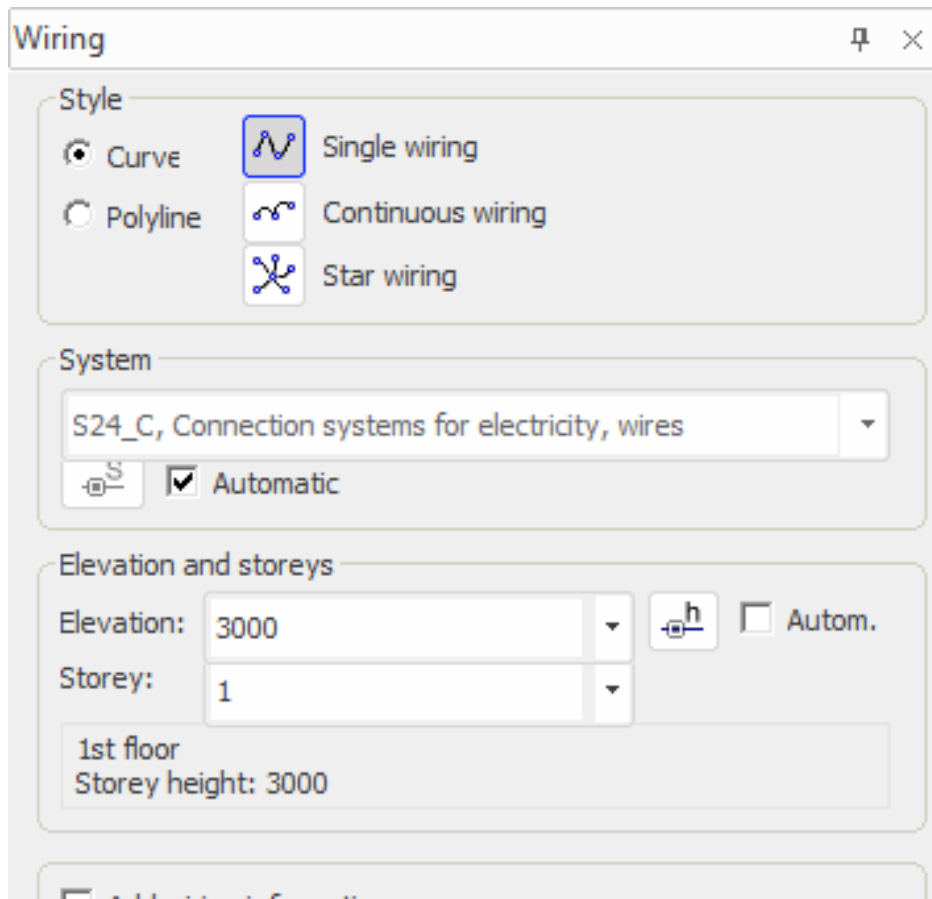
5. Define wiring information:

   a. Select **Add wiring information**.

   b. Click the ⋯ button. The **Wiring properties** dialog opens.

   c. Click **Cable type management**. The **Cable type management** dialog opens.

   d. In the **Global filter** field, enter *MMJ 3x*.

e. Select the **MMJ 3x1,5 S** cable type from the filtered list.

f. Add the cable to the project by double-clicking or by clicking **Add to project**.

g. Add the **MMJ 3x1,5 N**, **MMJ 5x2.5 S** and **MMJ 3x2.5 S** cable types to the project in the same way.
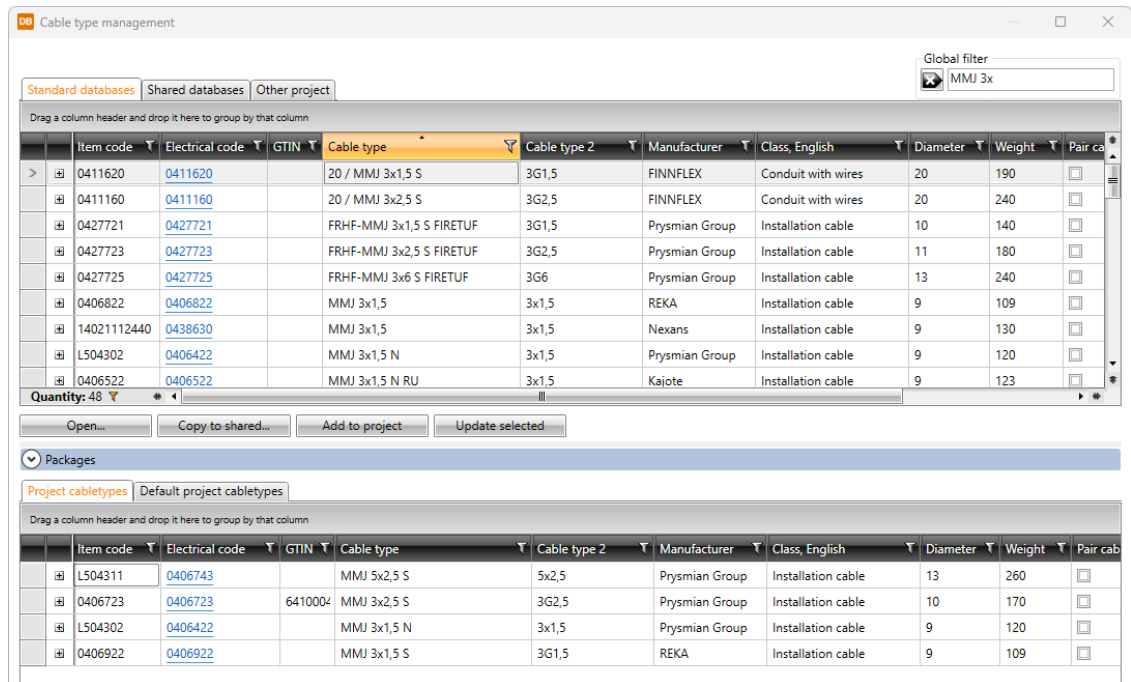
h. Click **Close**.

6. In the **Wiring properties** dialog, select **MMJ 3x1,5 S** as the cable type and click **OK**.

7. Indicate the start point of the wiring on the light fixture.

8. Indicate the middle point, i.e. the point through which the curve goes.

9. Indicate the end point.

10. Define wiring to go through a joint box by adding the joint box at the end of the wiring:

a. Start wiring from the symbol, and add the middle point.

b. In the **Wiring** dialog, click **Symbol**.

c. Select the joint box symbol.

5.2. Define wiring or cable information for the elements

We just wired all wires with the same cable – let's now change all the wires going to switches to the correct type.

Do the following:

1. Select **Layout** tab > **Wiring** group > **Edit**.

2. Select both the wires that connect to the switches.

3. Confirm the selection by right-clicking. The **Wiring properties** dialog opens.

4. Select **MMJ 3x1,5 S** as the cable type.

5. Clear the selection for **Neutral conductor**.

6. Click **OK**.

If the cable type is shown with the wire marking, you can turn it off in attribute management or by changing the project setting **Wiring markings (Layout drawings)** > **Data shown in attribute type** > **Cross-sectional area**.

5.3. Draw continuous flush wiring

Do the following:

1. Select **Layout** tab > **Wiring** group > **Draw**. The **Wiring** window opens.
2. Select **Continuous wiring** as the style.
3. From the bottom of the window, select **Wire marking**.
4. Select cable type and elevation.
5. Indicate the start point of the wiring (curve).
6. Indicate the middle or the end point for the curve. The middle point is the point through which the curve goes.
7. Indicate the next lighting fixture as the end point.
8. Indicate the middle or end point for the next curve.
9. Wire all the lighting fixtures in room 20 in the same way.
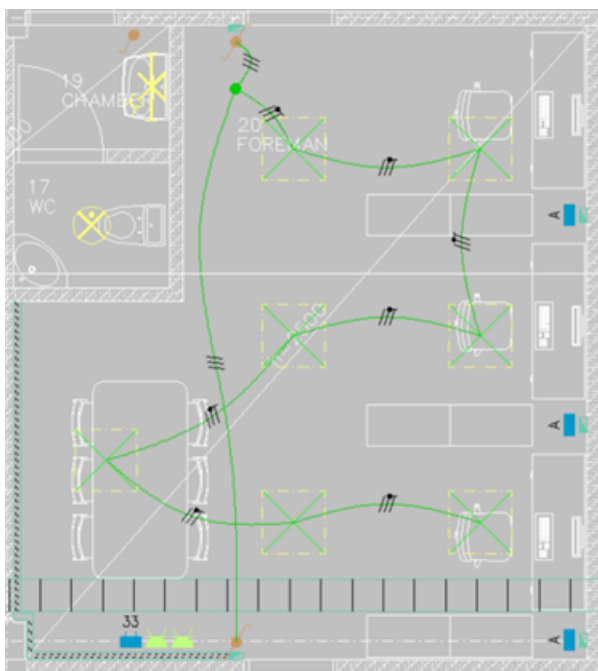
5.4. Wire from one point to multiple points

Let's first draw wiring from one point to multiple points, and then add the wire sets **2xML 1.5** and **3XML 1.5**.

**Do the following:**

1. Select **Layout** tab > **Wiring** group > **Draw**. The **Wiring** window opens.

2. Select **Star wiring**.

3. Select the wiring type and elevation.

4. Indicate the start point of the wiring (for example, the light fixture in room 105).

5. Indicate the middle or the end point for the curve. The middle point is the point through which the curve goes.

6. Indicate the next lighting fixture as the end point.

7. Indicate the light fixture in room 103 as the end point.

8. Add wire sets to the wiring:

   a. In the **Add wiring information** section, click the ⋯ button.
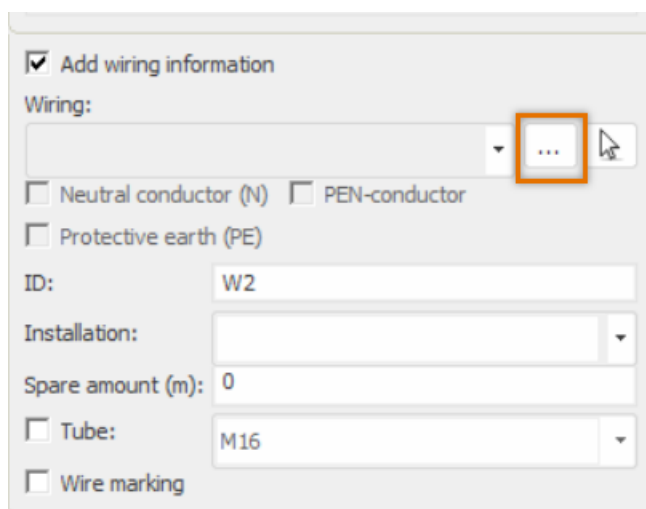
   The **Wiring properties** dialog opens.

   b. Click **Cable type management**. The **Cable type management** dialog opens.

   c. In the **Global filter** field, enter *ML 1.5*.

   d. Add the cable to the project by double-clicking it or by clicking **Add to project**.

   e. Select wire sets **2xML 1.5** and **3XML 1.5**.

   f. Accept the change by clicking **OK**.

g. Change **2XML 1.5** as the wiring.

h. Indicate the switch in room 18 as the curve end point.

i. End wiring from this point by right-clicking.

j. Indicate the light fixture in room 16 as the new wiring point.

k. Indicate the switch in room 18 as the curve end point.

l. Change the wire back to **ML 3x1.5** and select **(N)** and **(PE)**.

m. Indicate the light fixture on the wall in room 18 as the curve end point.

n. Indicate the light fixture in the ceiling in room 15 as the next curve's end point.

o. End wiring from this point by right-clicking.

5.5. Wire with automatic angles

Next, we will wire the light fixtures in the suspension rails.

Do the following:

1. Select **Layout** tab > **Wiring** group > **Draw**. The **Wiring** window opens.

2. For style, select **Polyline**.

3. Select **Continuous wiring**.
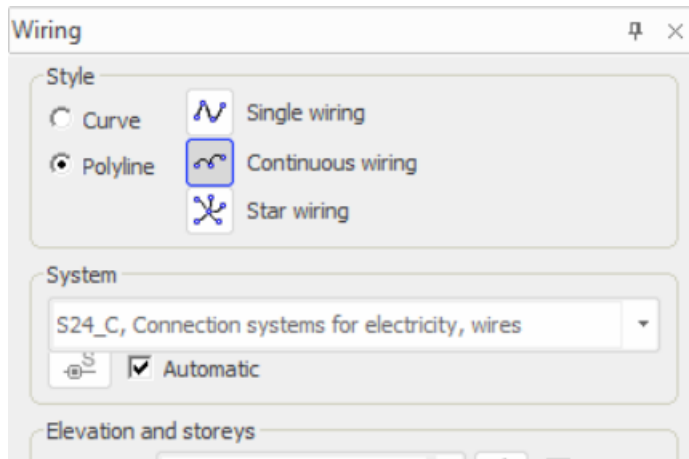
4. For cable type, select **MMJ 5x2.5 S**.

5. Select **Wiring marking**.

6. Indicate the symbol from which you want to start wiring.

7. For elevation, enter *2900*. The elevation is available in the drop-down menu for this session.

8. Indicate the next symbol to which you want to wire.

9. Indicate another symbol to which you want to wire.

10. In the same way, indicate all the suspension rail light fixtures.

11. Stop wiring by right-clicking.

12. Repeat the same steps for the other suspension rails.

5.6. Add junction boxes

You can add junction boxes when wiring or like any other symbols. Let's now ass the junction boxes needed in the cable tray.

Do the following:

1. In the **Symbols** window, select **Electric > Others** and double-click the **Junction box in wall, surface mounting** symbol:
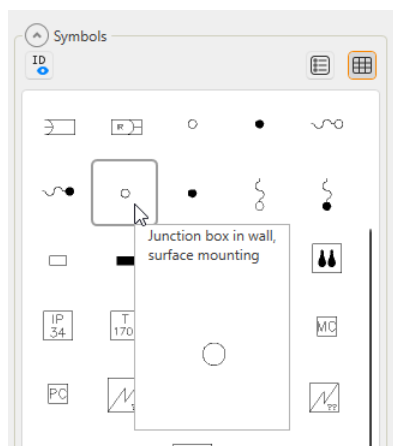
The **System selection** dialog opens.

2. Select **S251_C** as the system.
3. Accept the system by clicking **OK**.
4. Indicate the location of the symbol.
5. Accept the angle by right-clicking.
6. Insert the other junction boxes in the same way.

5.7. Draw surface wiring

Let's finalize the wirings in the building by adding the missing wires with surface wiring.

Do the following:
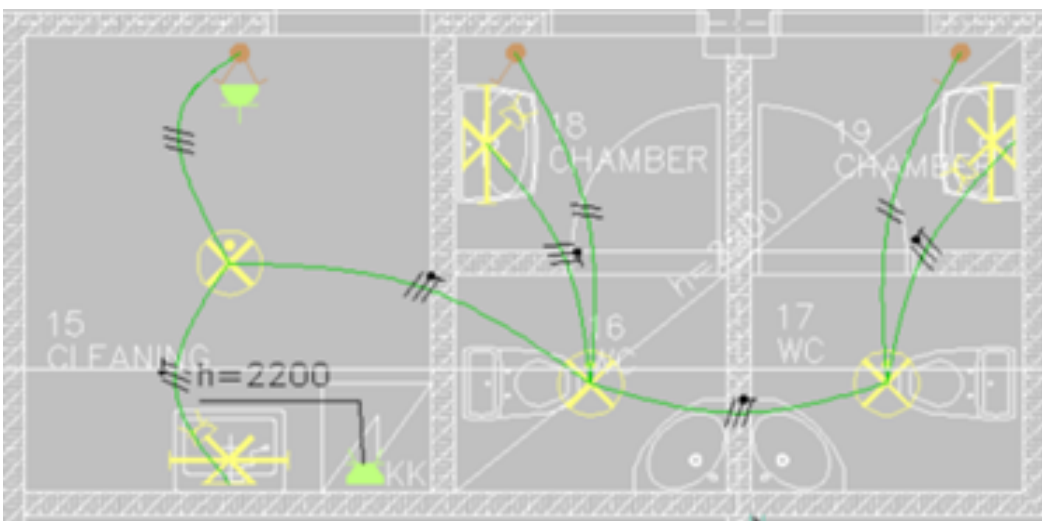
1. Select **Layout** tab > **Wiring** group > **Draw**. The **Wiring** window opens.
2. Select **Polyline** as the wiring style.
3. Select **MMJ 5x2,5 S** as the cable type.
4. Indicate wire start point from the bottom of the junction box.
5. Indicate the wire end point from the top of the other junction box.
6. End the wire by right-clicking.
7. Indicate the lighting fixture as the start point for a new wire.
8. Indicate the horizontal part of the wire in the drawing.
9. Indicate a junction box as the wire's end point.
10. Stop wiring by right-clicking.
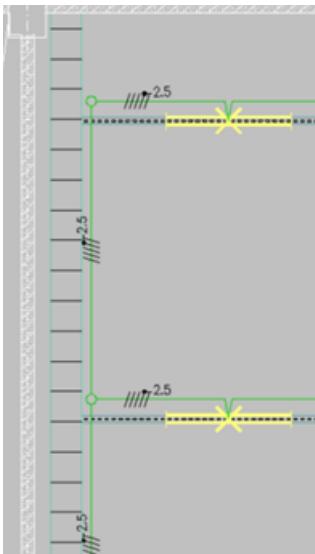
5.8. Add a distribution board

Before wiring and creating groups you need to create a distribution board. You can then add sub-devices, such as busbars and protective devices, for the distribution board.

Do the following:

1. Select **Layout** tab > **Cabinets and feeders** group > **Cabinet**.

   You can also create a distribution board via the **Electrical** window by selecting **Distribution boards and feeders**, right-clicking the project name or drawing file and selecting **New distribution board**.

   The **Distribution board properties** dialog opens.
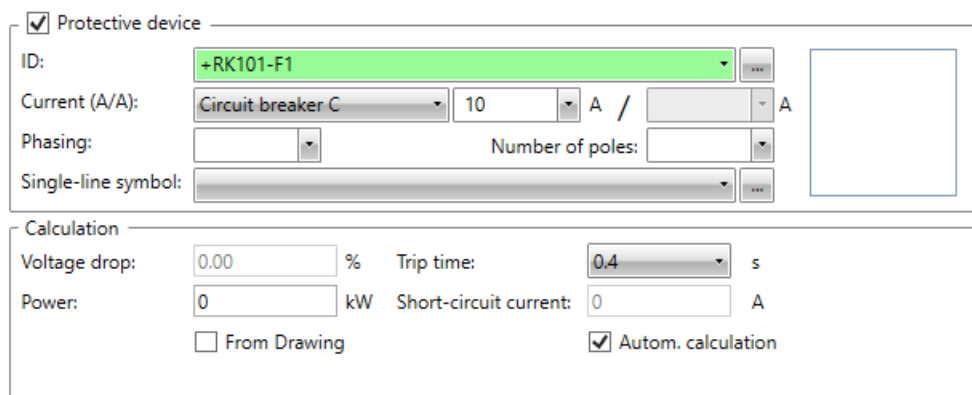
2. In the **Electrical position** field, enter *RK101* as the distribution board ID.

3. Enter size information.

4. If you want to calculate short-circuits, for example, define the feeder.

5. Select the **Symbols** tab and select the 2D and 3D symbols for the distribution board with **Add**.

6. Stop editing the distribution board properties by clicking **OK**.

7. Insert the distribution board symbol via the **Electrical** window: select **Distribution boards and feeders**, right-click the distribution board **RK101** and select **Insert this distribution board in the drawing**.

8. If necessary, rotate the symbol with F8.

9. Indicate symbol insertion point.

10. Accept the angle by right-clicking.

5.9. Insert a group marking

Next, we will add group markings in the rooms. A group can consist of several incoming and outgoing feeders.

Do the following:

1. Select **Layout** tab > **Cabinets and feeders** group > **Feeder**. The **New group** dialog opens.

2. As the group number, enter *3*. If desired, also enter a group description.

3. Make sure the distribution board ID is **+RK101**.

4. Make sure the feeder type is **Outgoing**.

5. As the group address, enter *Light fixtures* in the **Feeder text 1** field and *h. 2, store* in the **Feeder text 2** field.

6. Select **Protective device** and then the device information:

   a. Select the ID from the drop-down menu or with the [...] button.

   b. From the **Current (A/A)** drop-down menu, select **Circuit breaker C**.

   c. As current, select **10**.

   d. From the **Calculation** section, select **0.4** as the operating time.

7. Select **Wiring**, and define the wiring information.

   Note: If the cable ID is not defined, it will be automatically added with consecutive numbering.

8. Select the system from the drawing: click the button and indicate the desired wiring.

9. On the **Symbols** tab, select the 2D symbol for the feeder by clicking **Select**.

10. Close the dialog by clicking **OK**.

All the group feeders are shown on the **Feeders** tab in the top part of the group dialog. you can create several feeders for the group by copying the group marks in the layout drawing.

You can view the distribution boards and groups in the **Electrical** window, in the **Distribution boards and feeders** tree. Expand the tree by clicking the + signs, to find the group mark below the **<SUPPLY>** device. This is the busbar in the distribution board, and you can change the ID as desired.

The ◆ icon indicates an incoming feeder. Each busbar needs an incoming feeder.

The ◆ icon indicates an outgoing feeder. A busbar can have individual feeders that are not groups.

An incoming or an outgoing feeder becomes a group when it is given a group number. A group can have several incoming and outgoing feeders, and the all have the same group number.

Select a group from the tree, and right-click the group number or the feeder and select **Insert group marking into drawing**.

5.10. Wiring recap

You can practice the wiring functions used so far by finalizing the grouping.
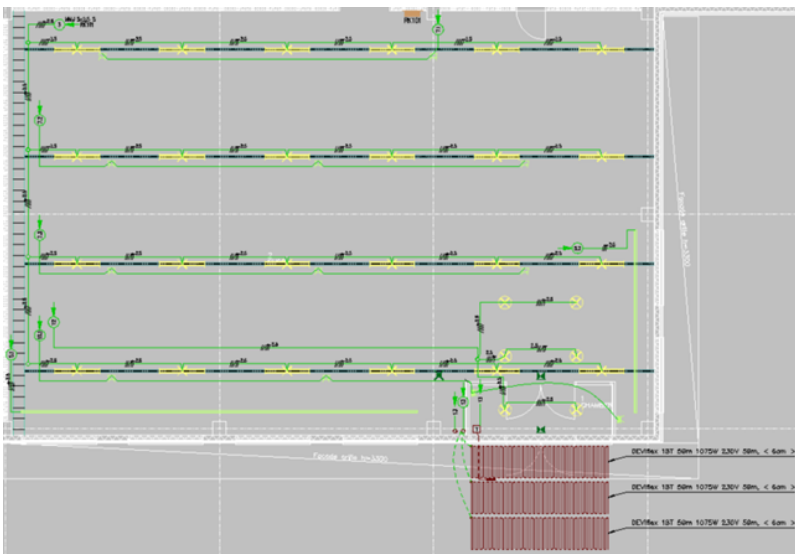
This section introduced functions related to wiring. Next, we will learn more about setting files and quantity calculation.

6. Setting files

When inserting, symbols are placed on a specific layer and the symbol receives a system ID based on that layer. The layer depends on the settings file in use as well as the automatic layer change option.

You can define multiple different setting files and change them as you design. In a setting file, you can define at a system basis which layer the program places the symbols on. You can also affect the color of the layer, the linetype, and the color that the program uses to draw the symbol. The layer color does not need to be the same as the symbol color.

You can enable or disable the automatic layer change in the Layout settings. If the automatic layer change is disabled, the symbol is placed onto the active layer. If, in turn, the automatic layer change is enabled, the layer is checked from the settings file.

When you insert a symbol, the program gives the symbol a system ID regardless of whether automatic layer change is enabled or disabled. With this information, the program knows the system of the symbol, no matter which layer it is placed on.

Next, we will practice using setting files with a sample drawing using the default setting file. Let's first create a new setting file, and then change the symbol colors in the drawing with it.

6.1. Create a new setting file

Do the following:

1. Select **Electrical** tab > **Settings** group > **Setting files**. The **Setting files management** dialog opens.
2. Click **New**. The **Setting file** dialog opens.
3. Enter *Exercise* as the name.
4. For **Template setting file**, select **E-2021T1**, based on which you will edit your own file.
5. Click **OK**. The new file is now shown in the list.

6.2. Edit the setting file

Let's continue by editing the setting file. You can close editing at any stage you want – you can always come back by selecting **Electrical** tab > **Settings** group > **Setting files management** and start editing by clicking **Edit**.

Do the following:

1. Select the **Exercise** file you just created.
2. Click **Edit**. The **Setting file [File path]** dialog opens. The **Setting** column shows the system ID. We will utilize the naming logic in filtering.
3. In the **Global filter** field, enter *wires*.
4. Click the **Setting** column header. The program sorts the list alphabetically based on the name of the setting. The first setting is now **S2_C**.
5. Select the first row, and with Shift pressed down select the last row starting with the letter S.
6. Right-click, select **4** from the **Color** drop-down menu.
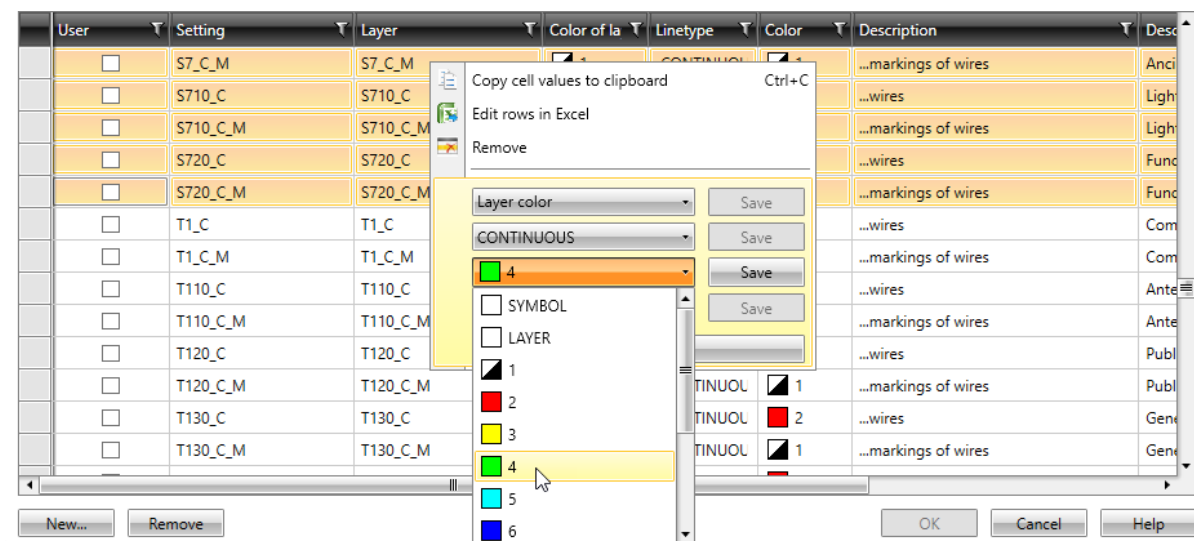
7. Click **Save**.
8. Click the **Setting** column header again. The first setting is now **T850_C**.
9. Select the first row, and with Shift pressed down select the last row starting with the letter T.
10. Right-click, select **7** from the **Color** drop-down menu, and click **Save**.
11. Stop editing the file by clicking **OK**.

6.3. Select the setting file and convert the drawing according to it

We can now change the setting file used. Changing the setting file does not affect the elements already in the drawing; the new file will only be applied to new symbols inserted. When we want to apply the new settings file to the elements already in the drawing, we use a specific function.

Do the following:

1. Select **Electrical** tab > **Settings** group > **Setting files**. The **Setting files management** dialog opens.
2. Select the **Exercise** file.
3. Click **Set as current**.
4. Click the arrow in the **Close** button and select **Close and update drawing**. Alternatively, select Select **Electrical** tab > **Settings** group > **Setting files** menu > **Convert drawing elements according to current setting file**. The **Select properties to change** dialog opens.
5. Select what to change. In this exercise, element colors would suffice but selecting all is the easiest option.
6. Click **OK**. The program goes through all the layers in the drawing and changes the information according to the setting file.

You cannot cancel the conversion but you can always convert to the old setting file..

7. Quantity calculation

With the quantity calculation function, you can count the number of symbols and objects. You can either count from the whole drawing or from the selected elements. The other auxiliary functions in the menu are meant for handling the list of already counted items.

Next, we will practice quantity calculation with the example drawing.

Do the following:

1. Select **Layout** tab > **Other functions** group > **List functions** menu > **List functions**.
2. Select **Count quantities from whole drawing**. The **List of electrical symbols** dialog opens.
3. Under **List**, select **List symbols and objects with length**.
4. Under **List, desired information**, select the following:
   - **System (setting)**
   - Next to **System (setting)**, **Use long description**
   - Next to **Symbol name**, **Use long description**
   - **IP class**
   - **Text in symbol**
   - **Graphical picture**
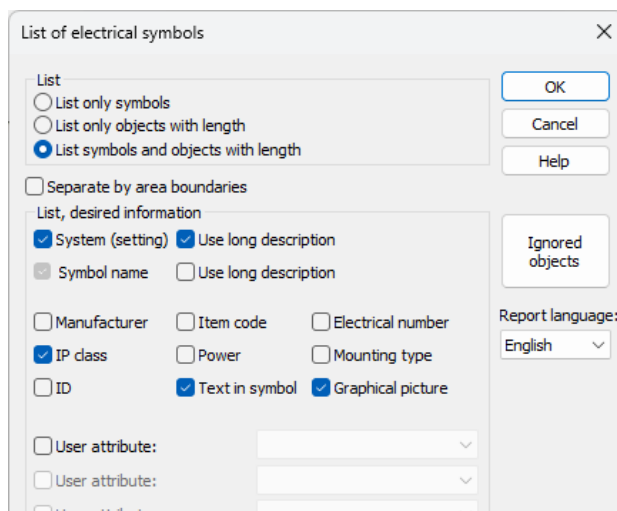
5. Under **Objects with length**, select **Count wires** and **Count tubes**.
6. For **List condition**, select **Include only if System information found**.
7. Click **OK**. The program calculates the objects.
8. Insert the list somewhere in the drawing.

8. Positioning

The position information is always attached to the lighting fixture or heater symbol position attribute. You can enter the attribute manually, or use a specific function to position multiple light fixtures and heaters afterwards. At the same time, you can insert a position mark for all the desired light fixtures.

You can utilize the Electrical DB product information databases in positioning.In Electrical Premium, you can utilize the Electrical DB product information databases in positioning. You can select the correct lighting fixture type for the position from the database already when inserting the mark. You can then, for example, create a light fixture list from the database automatically.

Product information is attached via symbol properties.In Electrical Premium, youYou can simultaneously manage lighting fixture position numbers. Adding the position number adds the position information to all the similar symbols in the drawing.

In Electrical Premium, youYou can open the Electrical DB tool by selecting **Electrical** tab > **Projects** group > **Open**. You can then view project information, manage device databases and create reports, for example.

Next, we will practice positioning by inserting lighting fixture position marks in the drawing.

Do the following:

1. Select **Layout** tab > **Positioning** group > **Device**. The **Position** dialog opens.
2. For **Equipment type**, select **Lighting equipment**.
3. For **New position**, enter *6*.
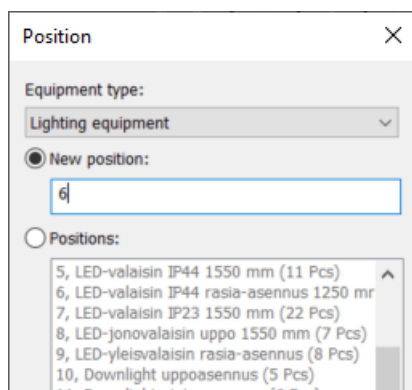
4. Click **OK**.
5. In room 109, select the four light fixtures inside the room near the front door.

6. Press Enter. The **Position markings** icon menu opens.
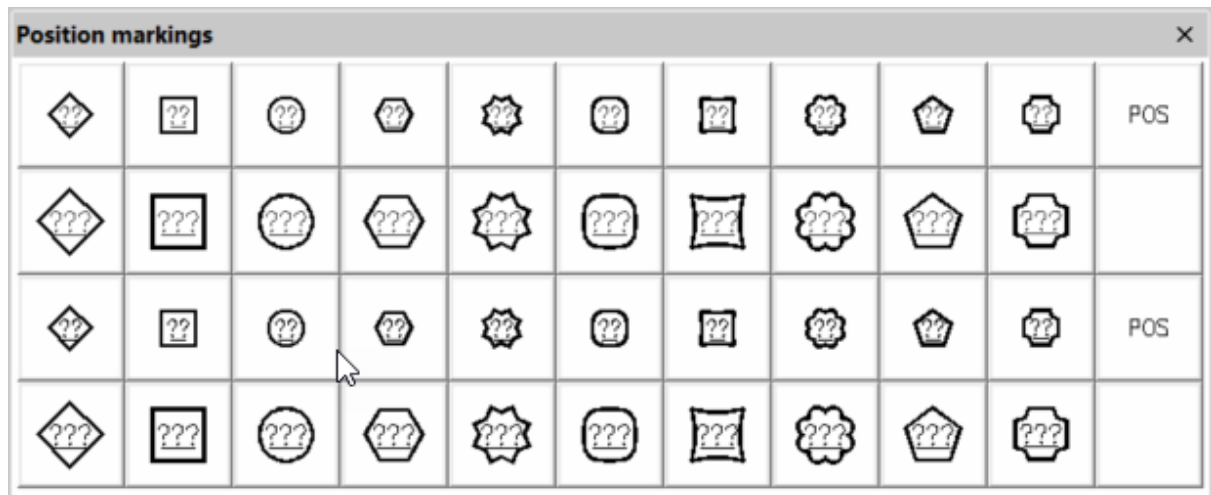
7. Select the position mark to use (upper row, on the left).

8. Insert other four markings in the same way with the position number 7.

9. Insert two markings for the lighting fixtures in the entry.

9. Finalizing the drawings

In this section, we finish the drawing by placing a drawing sheet on it. You also learn how to use viewports.

The drawing in the design mode was drawn in real scale, so for printing, you have to scale the drawing down so that it fits on a sheet of paper. To scale the drawing, you use drawing sheets and viewports.

- Drawing sheets are usually same kinds of objects as any other object (lines, circles, symbols, etc.). In a Layout drawing, you usually place a drawing sheet in the print mode. You can handle objects in the print mode the same way as you can in the design mode.
- A viewport is a view from the print mode into the design mode. When you work inside a viewport in the design mode in Layout, you can affect what part of the design mode is visible in the viewport and in what scale.

9.1. Set the drawing sheet

In this exercise, create a layout and place a drawing sheet on it.

Do the following:

1. Select **Electrical** tab > **Frames and labels** group > **Drawing sheet**. The **Set sheet** dialog opens.
2. Select the **A1, landscape: 841, 594** sheet size.
3. For **Frame**, select **Fold marks**.
4. Select **Add sheets to paper space**.
5. For **Destination layout**, enter *ELEC_2101*.
6. Select **Add viewports to sheets**.
7. Select **Use margins**.
8. Click **Margins**. The **Margins** dialog opens.
9. For the top, bottom, and left margins, enter *10*, and for the right margin, enter *190*.
10. Click **OK**.

11. Select **Set viewport layer state as 'Do not print'**.
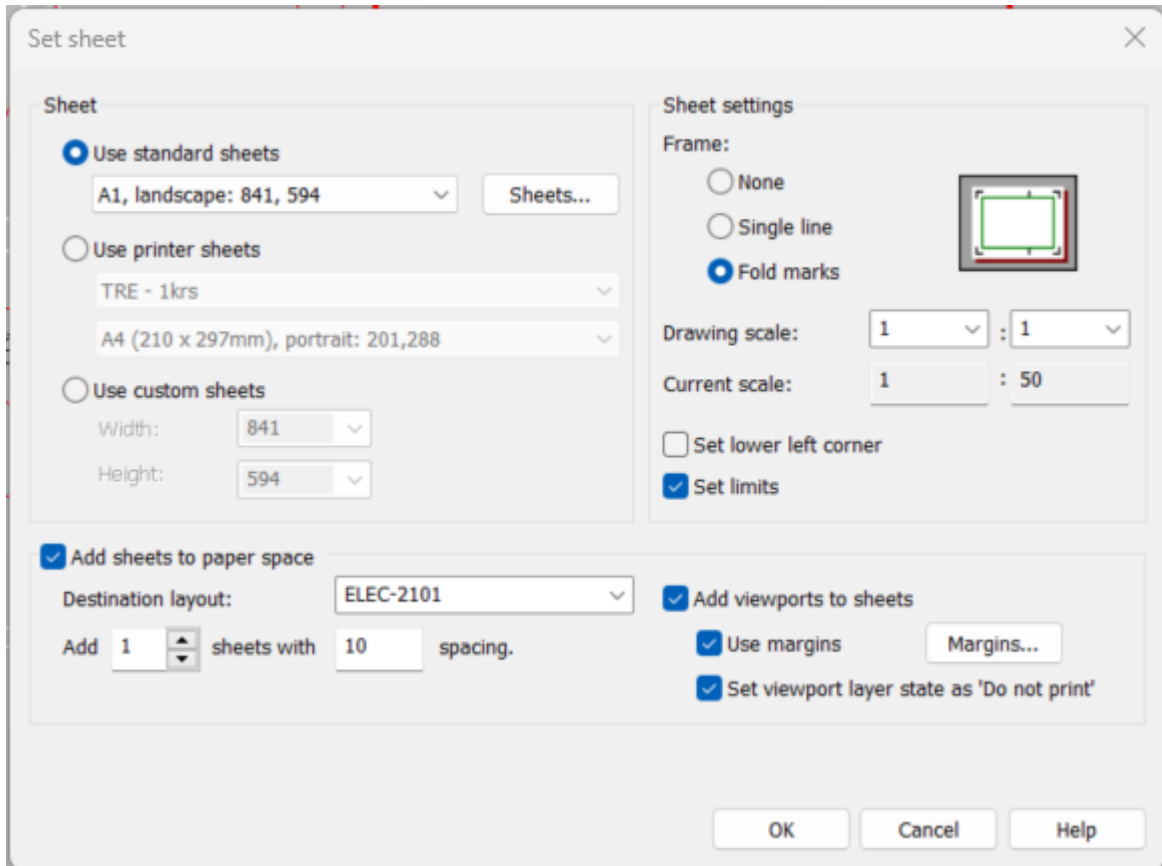
12. Click **OK**.

9.2. Add a drawing label

In this exercise, add a label on the drawing sheet.

Do the following:

1. Select the **ELEC_2101** layout.
2. Select **Electrical** tab > **Frames and labels** group > **Insert**. The **System selection** dialog opens.
3. Select **Filter from all systems**.
4. Select **Other systems**.
5. On the list, select **Drawing label**.
6. Click **OK**.
7. Indicate the location of the label near the bottom right corner of the frame. The program inserts the label in the drawing.
8. Select the label.

9. Right-click, and select **Edit the drawing label**. The **Fill drawing label** dialog opens.

10. Fill in the drawing label information.

11. Click **OK**.

9.3. Create a new layout

In this exercise, create a new layout because the drawing of the building does not fit on one layout.

Do the following:

1. Select **Tools** tab > **Layout** group > **Layout**. The **Layout** dialog opens.
2. Click **New**. The **New layout** dialog opens.
3. Enter *ELEC_2102* as the name, and click **OK**.
4. Select the layout you just created.
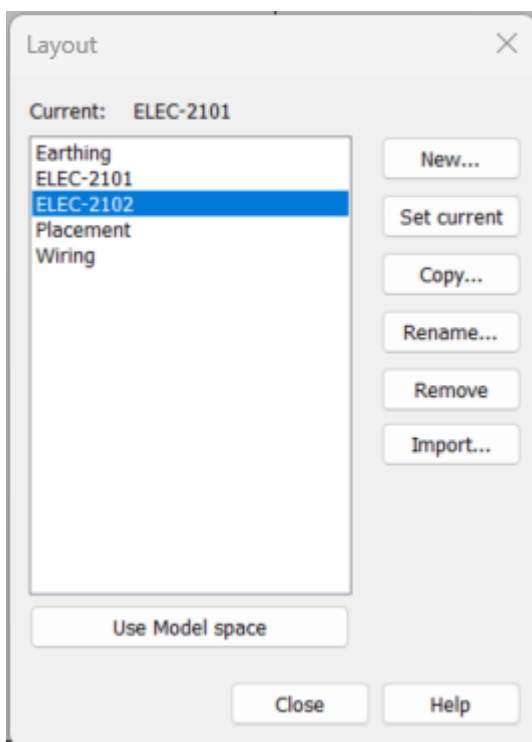
5. Click **Set current**.
6. Close the dialog.
7. Select **Electrical** tab > **Frames and labels** group > **Drawing sheet**. The **Set sheet** dialog opens.
8. Enter the information for the drawing sheet.
9. Insert a drawing label on the drawing sheet, and fill the information on it.

9.4. Copy a layout

Let's then copy the layout just created.

Do the following:

1. Select **Tools** tab > **Layout** group > **Layout**. The **Layout** dialog opens.

2. Select **ELEC_2102**.

3. Click **Copy**. The **Copy: 'ELEC_2102'** dialog opens.

4. Enter *ELEC_2103* as the name, and click **OK**.

5. Change the layout as the current layout by selecting the layout and clicking **Set current**.

6. Close the dialog.

7. Edit the necessary information in the label to be correct.

9.5. Change the view of the viewport

Next, we will edit the viewport view as desired, i.e. we will set the template to the desired spot in the viewport and change the scale of this viewport.

Do the following:

1. Move inside the viewport by double-clicking the printer icon on the bottom right corner of the screen.
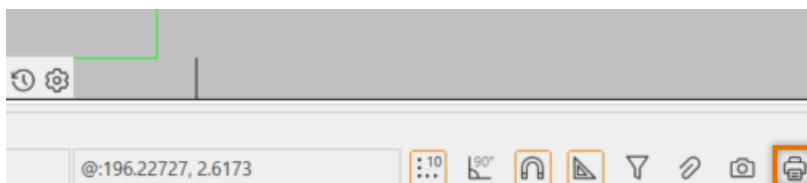
2. Zoom and pan the view so that tha building is completely visible in the viewport.

3. Select **Tools** tab > **Layout** group > **Viewports** menu > **Zoom 1:50**.

4. Pan the zoomed view in the viewport so that a part of the factory hall is visible in the viewport.

   Note: If you now turn the wheel of the mouse, the view is no longer to scale. You can reset the scale by repeating step 3.

   Tip: You can lock the viewport by changing the mode to design mode by double-clicking the same icon as in step 1, selecting the viewport and changing its **Locked** property to **Yes**.

CADMATIC SOFTWARE SOLUTIONS

9.6. Change viewport size

In this exercise, change the size of the viewport.

Do the following:

1. Move to the **ELEC_2101** layout.
2. Move inside the viewport by double-clicking the printer icon.
3. Zoom and pan the view so that the lower part of the building is visible in the viewport.
4. Move back to the print mode by double-clicking the pen icon.
5. Select the viewport.
6. Click the handle on the top right corner of the viewport.
7. Indicate the new location of the top corner.
8. Move the bottom right corner by repeating steps 6–7.

The program changes the size of the viewport, so more of the design is visible.

9.7. Copy a viewport

Let's copy a viewport on the drawing sheet so that it shows the module markers of the design that are cut out of the first viewport.

Do the following:

1. Select the viewport and right-click.
2. Select **Copy**.
3. Move inside the viewport by double-clicking the printer icon.
4. Zoom and pan the view so that the module lines, or the horizontal and vertical lines, are visible in the viewport.
5. Select **Tools** tab > **Layout** group > **Viewports** menu > **Zoom 1:50**.
6. Move back to the print mode.
7. Change the size and the location of the viewports so that they fit on one drawing sheet as shown in the image below.
8. Move back inside the viewport.
9. Pan the views in the viewports so that the top viewport shows the edge of the module lines and the module lines continue straight to the bottom viewport.

9.8. Cut the viewport

Let's cut the viewport using a polyline so that one room is hidden from view. These premises are shown in layout ELEC_2102.

Do the following:

1. Move into the **ELEC_2103** layout.
2. Draw a polyline that follows the boundaries of the viewport but leaves out the resting area.
3. Select **Tools** tab > **Layout** group > **Viewports** menu > **Clip**.
4. Select the old viewport.
5. Select the polyline.

9.9. Use setting groups

With setting groups, you can manage the layers shown in the drawing. Setting groups are based on the system IDs, and are therefore not dependent on layer name or the settings file in use. Therefore, you can utilize them during design. However, the setting groups are most valuable with print modes.

Next, we'll practice using the setting groups and create a new layout.

Do the following:

1. Select **Tools** tab > **Layout** group > **Layout**. The **Layout** dialog opens.
2. Select the **ELEC_2101** layout, and click **Copy**.
3. Enter *ELEC_2104* as the name, and click **OK**.
4. Click **Close**.
5. Set **ELEC_2104** as the current layout.
6. Change the information on the label:
    a. Change **2104** as the drawing number.
    b. In the fourth description field, enter *Electrical locations*.
7. Move inside the viewport by double-clicking **PRINT** located in the lower right corner of the status bar.
8. Select **Layout** tab > **3D and view** group > ⬙ (**Handle setting groups**). The **Setting groups** dialog opens.

9. In the **Application setting groups** list, select **Wirings, all**.
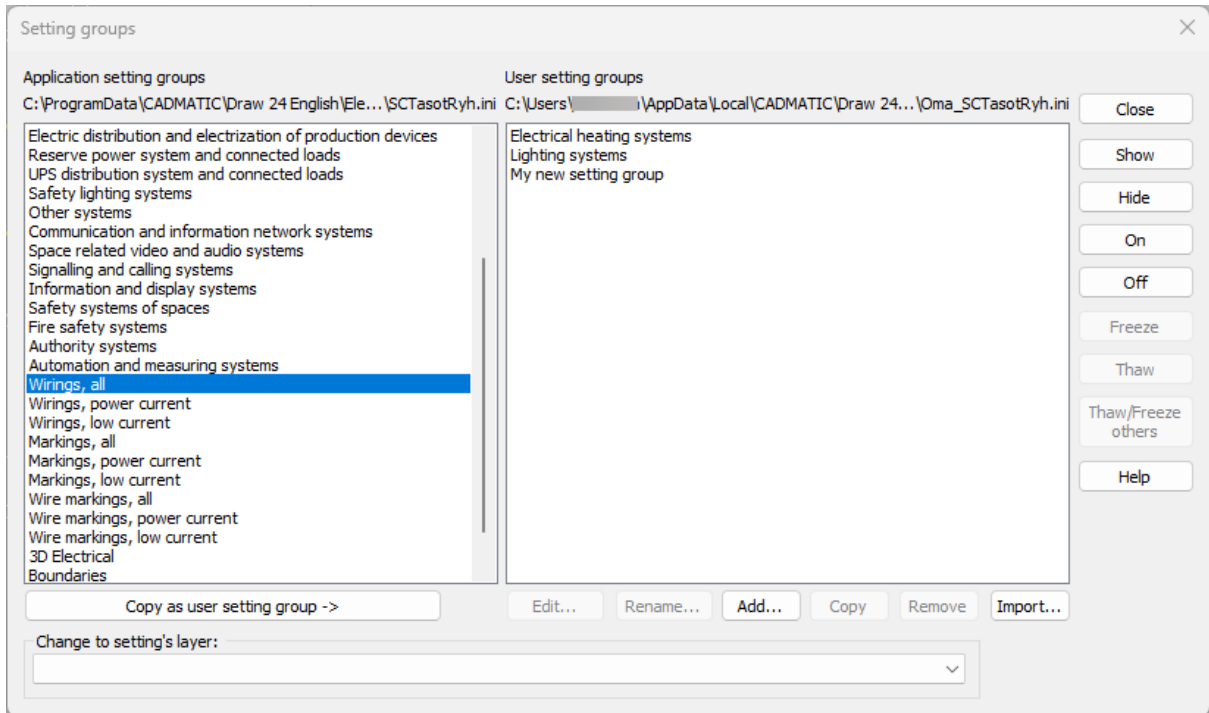
10. Click **Freeze**.

11. Close the dialog.

12. Move back into the print mode by double-clicking the pen icon located in the lower right corner of the status bar.

9.10. Print the frames

When printing, you need to define the printer to which you want to print and how you want to print. You can also define, for example, the style i.e. the color and line width and the scale for printing.

Typically, the scale used in print mode is 1:1. In print mode, the viewport was already scaled so that the building fits on the paper and the paper, in turn, was drawn with real measurements.

When printing into a print file (PLT) or PDF, you can also select the path to where the file is stored.

When printing arrangement drawing frames, it is best to use the function designed exactly for that. The Print frames from drawing function allows you to print all frames from the selected drawings at once.

Do the following:

1. Select **File** > **Print** > **Print frames from drawings**. The **Print frames from drawings** dialogs opens.

2. Select *[CADMATIC directory]\Samples\Electrical\Layout exercise\ELEC_2100.drw*.

3. Click **Open**. The **Print frames from drawings** dialog opens.

4. Select the printer.

5. For **Paper**, select **A1** or select a paper size that fits the largest frame on the drawing.

6. For **Print style table**, select **BW**.

7. For **Scale**, select **Drawing sheet scale**.

8. Select **Print file**.

9. Select **Print PDF** and **Combine each drawing into single PDF file**.

10. For **Print directory**, select **Drawing directory\PLT**.

11. Print the frames by clicking **OK**.

Tip: You can open the drawing directory with the key combination Ctrl + E.

10. Distribution board table

Next, we will draw a distribution board table. The distribution board table functions are available on the **Single-line** tab.

10.1. Create a distribution board table drawing

Let's start by creating a new drawing file as a basis for the distribution board table.

Do the following:

1. Select **Electrical** tab > **Documents** group > **New**. The program opens a new document tab, and the **Start a new Electrical drawing** dialog opens.

2. For **Drawing type**, select **Distribution Board (Table)** below **Single-line**.

3. Next to the **Project directory** field, click the [...] button. The **Select folder** dialog opens.

4. Navigate to *[CADMATIC directory]\Samples\Electrical*.

5. Create a new folder named *Distribution board table exercise*.

6. Click **Open**.

7. For name, enter *ELEC_4511.drw*.

8. Click **OK**. The **Distribution Board** tab opens automatically, based on the selected drawing type.

10.2. Create a cover sheet

Next, we will create a cover sheet on the drawing and fill it.

Do the following:

1. Select **Distribution Board** tab > **Frames and labels** group > **Frames**. The **Distribution Board Schema frames** dialog opens.

2. Select **Cover sheet 1**.

3. Click **OK**. The program inserts the frame on the drawing, and the **Distribution Board (object ID and el.position) for drawing label** dialog opens.

4. Enter *123* as the object ID and *P* as the electrical position. You can also leave these empty, and fill them in later.

5. Click **OK**. The fill function for the cover sheet starts.

> **Tip:** You can easily move to any field with your mouse. Alternatively, you can browse the fields with the arrow keys.

6. Enter the desired values in the fields, for example:

   - 400 V

   - 125 A

   - 50 Hz

7. Double-click the check boxes to select them, for example:

   - Control voltage bar OFF

   - AC busbar or wires: L1, L2, L3, N, PE

8. Stop the filling function with Esc.

10.3. Create a schema sheet

We will now create a new schema sheet and add a frame for it.

Do the following:

1. Select **Distribution Board** tab > **Frames and sheets** group > (Create new sheet). The program opens an empty sheet.

2. Select **Distribution Board** tab > **Frames and labels** group > **Frames**. The **Distribution Board Schema frames** dialog opens.

3. Select **Distribution Board Schema frame, columns for calculation information**.

4. Click **OK**.

The program inserts a sheet frame on the drawing and starts the filling function. The command line shows you the functions and their key combinations. In the following exercises, you learn to use the filling functions.

10.4. Fill the table
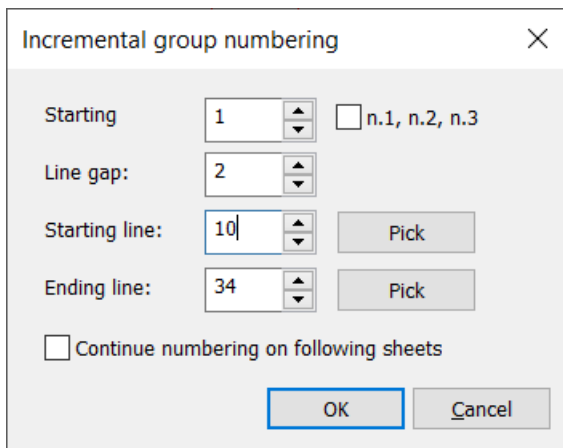
Next, we will learn to use the filling functions.

10.4.1. F5: Add incremental numbering

In this exercise, add incremental numbering to the frame.

Do the following:

1. Press F5. The **Incremental group numbering** dialog opens.
2. Enter the following values:
   a. **Starting number** – *1*
   b. **Line gap** – *2*
   c. **Starting line** – *10*

3. Click **OK**.

The program adds incremental numbering to the frame.

10.4.2. F8: Add symbols for inputs and outputs

Let's add symbols for inputs and outputs on the frame.

Do the following:

1. Press F8. The **Select Distribution Board symbol** dialog opens.

2. As the main group, select **Inputs and supplies**.

3. As the sub group, select **Inputs (no measuring)**.

4. Select **TN-S input, 3-p**.
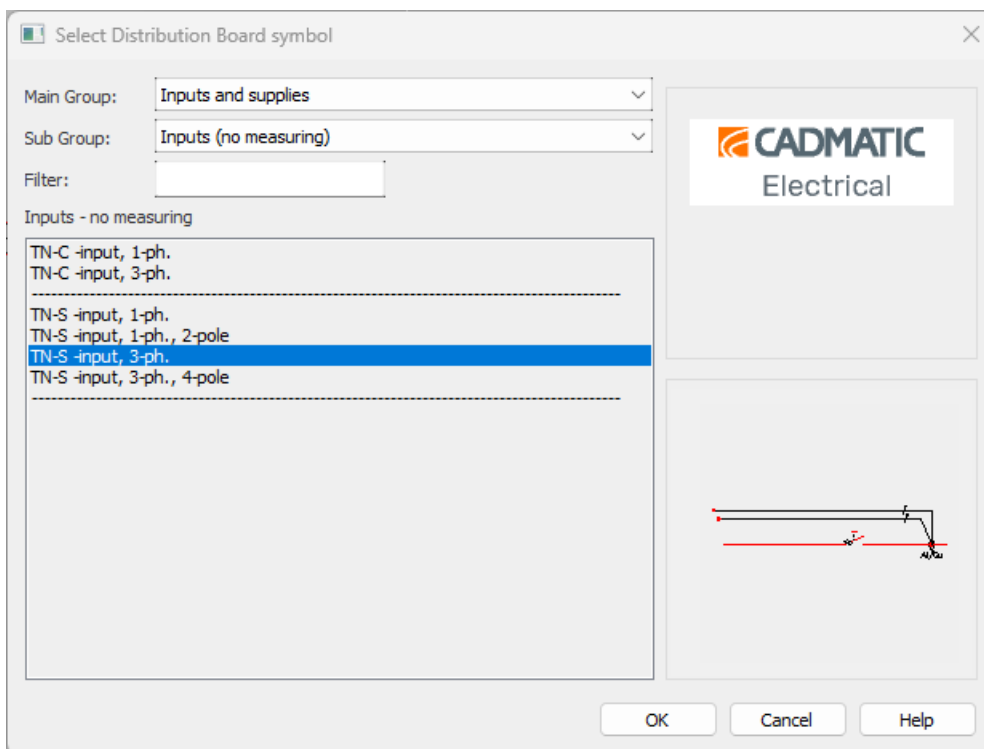
5. Click **OK**.

6. Indicate the row to which you want to add the symbol.

7. Select a new symbol from the library:

   a. Press F8.

   b. As the main group, select **Inputs and supplies**.

   c. As the sub group, select **Earthing**.

   d. Select the **Protective earth for Distribution Board** symbol.

   e. Click **OK**.

8. Insert the symbol above the previous symbol.

9. Select another symbol from the library:

   a. Press F8.

   b. As the main group, select **Outputs - system without fuses**.

   c. As the sub group, select **Distribution Board devices**.

   d. Select the **Socket-outlet, 1x3ph.** symbol.

   e. Click **OK**.

10. Insert the symbol on the group 1 row.

11. Select the same symbol again by pressing F8, and insert it to group 2.

12. Select another symbol from the library:

   a. Press F8.

   b. As the main group, select **Outputs - system without fuses**.

   c. As the sub group, select **Distribution Board devices**.

   d. Select the **Socket-outlet, 1x1-ph. + 1x3-ph.** symbol.

   e. Click **OK**.

13. Insert the symbol on the group 3 row.

10.4.3. Move, copy, and paste symbols

With the fill function enabled, you can easily edit the rows with the familiar **Cut** and **Paste** commands, with which you can cut and paste row symbols and texts. Let's now move and copy a plug socket for groups 4 and 5.

Do the following:

1. Select a row by clicking.
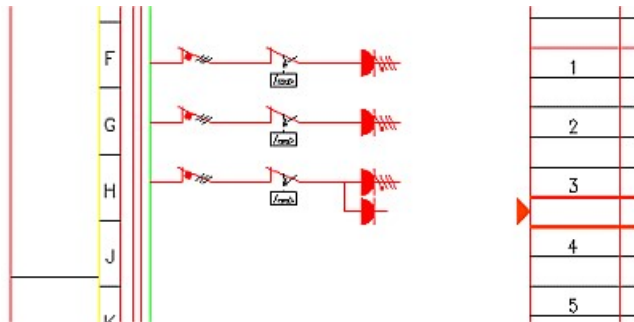
2. Press Ctrl + X.

3. With the arrow keys, move one row lower to group 4 and paste the plug socket with Ctrl + V.

4. Move to group 5 row, and paste the plug socket again with Ctrl + V.

In order to copy/cut, symbol graphics need to be linked to the row. Link by selecting **Distribution Board** tab > **Cabinets and feeders** group > **Auxiliary functions** menu > **Link graphics to Distribution Board Schema rows**. Follow the command line for instructions.

10.4.4. F8: Align symbols horizontally

If the program does not have a pre-made symbol for an input that you need, you can insert symbols individually on the schema and combine them to form the input symbol that you need. When inserting the symbols you can easily move the alignment horizontally.

Let's insert another plug socket by utilizin this function.

Do the following:

1. Press F8. The **Select Distribution Board symbol** dialog opens.
2. Select the symbol:
   a. As the main group, select **Symbols**.
   b. As the sub group, select **Distribution Board devices**.
   c. Select the **Plug socket, 1-gang, protective contact, flush** symbol.
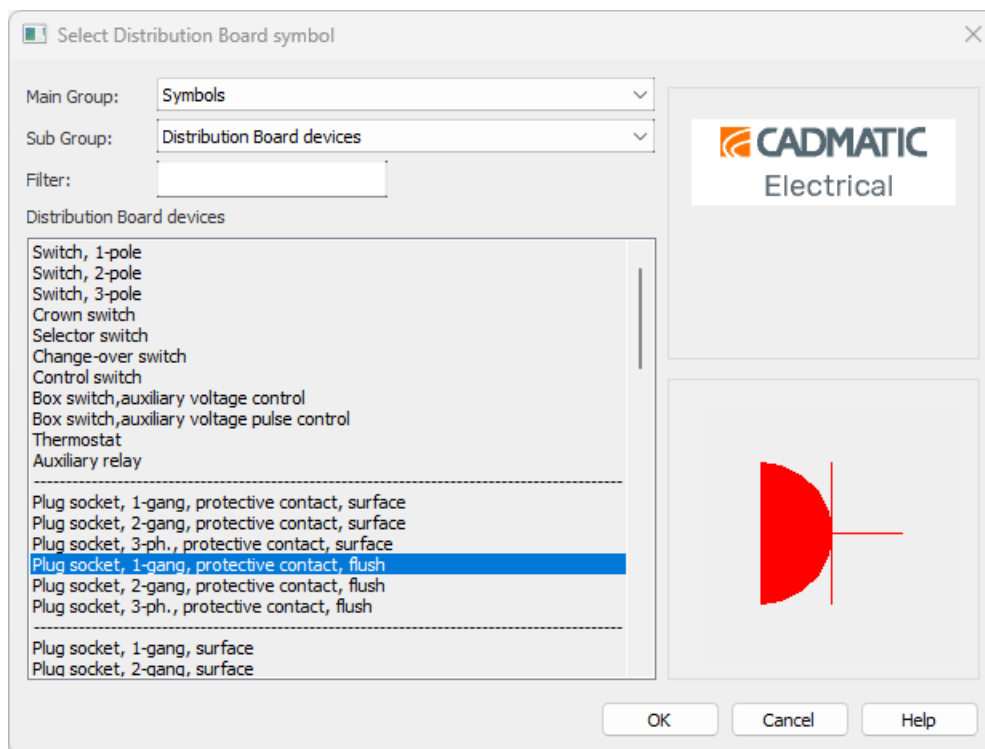
3. Click **OK**.
4. Change the horizontal alignment of the symbol by pressing Ctrl + F2 two times.
5. Insert the symbol on the group 6 row.
6. Stop the fill function with Esc.

10.4.5. Edit the symbols

When the filling function is not active, you can edit the graphics with regular drawing functions. The inputs and outputs are symbol packages so they can easily be edited afterwards.

Let's edit the drawing to meet our needs.

Do the following:

1. Remove the RCD from group 1:
   a. Select the symbol.
   b. Right-click and select **Delete symbol**.
2. Stretch the line connecting the plug sockets to the right length:
   a. With window indication, select the line.
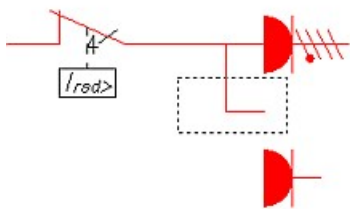
   b. Right-click and select **Stretch**.
   c. Indicate the point you want to grab.
   d. Indicate the offset. Stretch the line all the way to the plug socket in group 6.
3. Add connecting lines to the other plug sockets:
   a. Select **Distribution Board** tab > **Wiring** group > **Draw wiring**.
   b. Indicate the start point on the vertical line.
   c. Indicate the end point on the plug socket.
   d. Do the same for the other plug socket.
   e. Press Esc.

4. Edit the symbol according to the following image:
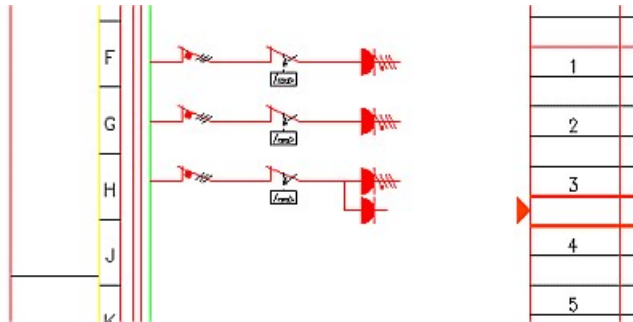
10.4.6. Fill the rows

It is easy to fill the distribution board schema rows with the fill function. By double-clicking a row you can enter values. The values are saved, and you can select values previously added from the drop-down menu.

Tip: When entering the texts, select the closest one from the drop-down menu and edit it as needed.

Note: When filling the group/feeder information, you can create a new feeder. Feeders created this way are also available in the layout drawing project tree.

Note: In Electrical Premium, you can create a new feeder when filling the group/feeder information. Feeders created this way are also available in the layout drawing project tree. In Electrical Basic, there is no database for merging the information from separate drawing files.

Do the following:

1. Select **Distribution Board** tab > **Frames and labels** group > **Edit**.
2. Double-click the incoming feeder row.
3. Enter the following information:
   - **Address** – *Input from Distribution Board PK*.
   - **Wiring, type** – *AMCMK 4x70+21Cu*.
4. Click **OK**.
5. Double-click the outgoing feeder row.

6. Enter the following information:

- **Address** – *Input to Distribution Board PRK1.2*
- **Wiring, type** – Select **AMCMK 4x70+21Cu** on the drop-down menu.

7. Click **OK**.

8. For group 1, enter the following information:

- **Address** – *3~ plug socket 63A*
- **Current (A/A)** – *C63A*

9. Click **OK**.

10. For group 2, enter the following information:

- **Address** – *3~ plug socket 32A*
- **Current (A/A)** – *C32A*

11. Click **OK**.

12. For group 3, enter the following information:

- **Address** – *3~ plug socket 16A*
- **Current (A/A)** – *C16A*

13. Click **OK**.

14. For group 4, enter the following information:

- **Group number** – *3.1*
- **Address** – *1~ plug socket 16A*
- **Current (A/A)** – **C16A**

15. Click **OK**.

16. For group 5, enter the following information:

- **Group number** – *3.2*
- **Address** – *1~ plug socket 16A*
- **Current (A/A)** – **C16A**

17. Click **OK**.

18. For group 6, enter the following information:

- **Group number** – *3.3*
- **Address** – *1~ plug socket 16A*
- **Current (A/A)** – **C16A**

19. Click **OK**.

10.4.7. Remove rows

Let's now remove unnecessary rows from the schema.

Do the following:

1. Select the group 7 row.
2. Press Ctrl + D. The **Edit Distribution Board Schema** dialog opens.
3. Enter *12* as the number of rows to remove.
4. Click **OK**. The **Edit Distribution Board Schema** dialog opens.
5. Click **No**. The program removes all but one row from the schema.
6. Remove the final row by pressing F4.
7. Stop the filling function by pressing Esc.

10.5. Edit attributes and create input package

Next, we will edit the symbol attributes and create an input package of the symbols.

10.5.1. Edit attributes

Let's add some more information for the symbols in the drawing with attributes. You can either fill the attributes in the properties dialog or with separate functions. We will now use both ways.

Do the following:

1. Edit the value of the attribute:
    a. Select **Home** tab > **Symbol** group > **Attributes** menu > **Edit attributes**.
    b. Select the main switch symbol.
    c. Right-click or press Enter. The **Edit Attributes** dialog opens.
    d. For **Rated current**, enter *125A*.
    e. Click **OK**.
2. Edit text properties:
    a. Select **Home** tab > **Symbol** group > **Attributes** menu > **Edit properties**.
    b. Select the **Al/Cu** text below the symbol. The **Modify** dialog opens.
    c. For **Value**, enter *2xAl/Cu*.
    d. Click **Move**.
    e. Indicate a new position for the text.
    f. Click **OK**.

3. Edit the properties in the **Properties** window:

   a. Select the first three-gang plug socket.

   b. In the **Properties** window, enter *63A* as the rated current.

   c. Press Enter.

   d. Exit the selection mode by pressing Esc.

   e. Select the RCD symbol.

   f. As the rated current, enter *40A*.

   g. As the information, enter *30mA*.

   h. Press Enter.

   i. Exit the selection mode by pressing Esc.

   j. Select the second three-gang plug socket.

   k. As the rated current, enter *32A*.

   l. Press Enter.

   m. Exit the selection mode by pressing Esc.

   n. Select the second three-gang plug socket.

   o. As the rated current, enter *16A*.

   p. Press Enter.

   q. Exit the selection mode by pressing Esc.

4. Copy attribute values:

   a. Select **Home** tab > **Symbol** group > **Attributes** menu > **Copy attribute values**.

   b. Select the first residual current device. The **Select attributes to copy** dialog opens.

   c. Select **Residual current** and **Information**.

   d. Click **Copy**.

   e. Select the other residual current device.

   f. Confirm the selection by right-clicking.

   g. Start the function again by right-clicking.

   h. Select the 16A three-gang plug socket.

   i. Confirm the selection by right-clicking.

   j. Select **Residual current**.

   k. Click **Copy**.

   l. Select all the 1-gang plug sockets.

   m. Confirm the selection by right-clicking.

10.5.2. Save an input package

It is recommended to save the edited inputs/outputs as their own packages for later use. You can select your own packages fro the library in the same way as the ones that come with the program. You can also save rows in which case rows are filled with texts from the package when inserting the package into a drawing.

Let's now save the edited input as its own input package.

Do the following:

1. Select **Distribution Board** tab > **Symbols** group > **Symbols** menu > **Save user's input package**.
2. Select the text rows of the input.
3. Use upwards area selection to select the symbols and their terminal blocks.
4. Confirm the selection by right-clicking. The **Name of the symbol packet** dialog opens.
5. Enter *SO_PRKinput125A* as the name.
6. Click **OK**.
7. Indicate the base point of the symbol on the connection point to the voltage busbar. The **Tip text for this symbol packet** dialog opens.
8. Enter *Plug socket distribution board input, chainable 125A*.
9. Click **OK**.
10. Indicate the corner points of the area that you want to see in the icon menu so that you can recognize the symbol.

The program saves the package to the symbol library, from where you can insert it to a drawing.

You can find the package in the following way:

1. Click **Distribution Board** tab > **Symbols** group > **Symbols** menu > **Symbol packages**. The **Select Distribution Board Schema symbol** dialog opens.
2. As the main group, select **User's own**.
3. As the sub group, select **Input packages**. Your package is now on the list.

10.6. Finalize the drawings

Finally, let's fill the drawing labels and print the distribution board schema as a PDF file.

10.6.1. Fill the drawing label

Let's fill the drawing label for the distribution board schema.

Do the following:

1. Select the drawing frame.
2. Right-click, and select **Sheet management**. The **Sheet management** dialog opens.
3. Enter the following information:
   - **Drawing number** – *4511*
   - **Job no.** – *1234*
   - **Object ID** – *123*
   - **Electrical position** – *PRK1.1*
   - **Drawing name (1st line)** – *Plug socket distribution board*
   - **Drawing name (2nd line)** – *PRK1.1*
   - **Drawing name (3rd line)** – *Main schema*

10.6.2. Copy drawing label information

Let's copy the label information to the other sheet as well and update the number of sheets.

Do the following:

1. In sheet management, select the sheet's first row.
2. Click **Copy**.
3. Select the sheet's second row.
4. Click **Paste**.

10.6.3. Queue print as a PDF file

In this exercise, print the complete drawing as a PDF file to the project directory.

Do the following:

1. Select **File** > **Print** > **Queue printing from drawing sheets**. The **Print sheets** dialog opens.
2. Click **Select all**.
3. Click **OK**. The **Electrical queue printing** dialog opens.
4. For **Print device**, select **Microsoft Print to PDF**.
5. For **Print area**, select **Limits**.
6. Select **PDF file**.
7. For **Printing directory**, select **Drawing directory**.
8. Print the drawing by clicking **OK**.