



# CADMATIC

## CADMATIC Electrical

Release notes

2023T2

---

# Contents

Release notes .....	3
Cabinet Layout .....	3
New and improved features .....	3
Bug fixes .....	4
Distribution Board .....	4
New and improved features .....	4
Bug fixes .....	5
Layout .....	5
New and improved features .....	5
Bug fixes .....	7
Schematics .....	8
New and improved features .....	8
Bug fixes .....	10
Electrical DB (Premium) .....	12
New and improved features .....	12
Bug fixes .....	13

## Release notes

---

The release notes for CADMATIC Electrical version 2023T2 are described in this document.

The following conventions apply to these release notes:

- Filenames, pathnames and environment variables are in *italics*.
- Commands, options, dialog names, and menu choices shown in the user interface are in **bold**.

## Cabinet Layout

---

### New and improved features

<p>Symbol selection</p>	<p>The following changes have been made to the symbol selection tool:</p> <ul style="list-style-type: none"> <li>• It is now possible to define your own favorite symbols after which they are easily available on the new <b>Favorite symbols</b> tab. The symbols can be selected from all the symbols: those in the current drawing or project as well as Electrical symbols.</li> <li>• On the new <b>Symbols in another drawing</b> tab, you can select another drawing from which to select symbols. You can then, for example, define these symbols as your favorites.</li> <li>• The search now also searches for a match in symbol names, in addition to the description.</li> <li>• On the <b>Symbols in current drawing</b> tab, you can right-click a symbol and select <b>Show occurrences from drawing</b>. The occurrences are then indicated with a yellow arrow.</li> <li>• In the <b>Symbol information</b> section, you can now copy the name and the description of the selected symbol to the clipboard.</li> </ul>
<p>Supported languages</p>	<p>Polish has been added to supported languages:</p> <ul style="list-style-type: none"> <li>• You can now select Polish as the design language in project settings.</li> <li>• You can now select Polish as the drawing frame/label and logo language.</li> </ul>

## Bug fixes

- Starting the **Draw cabinet into layout** function via the project tree crashed the application.
- When creating a product model, changing several symbols at once to user's own symbol crashed the application.
- Deactivating or deleting 3D layers did not work.
- When the user first closed the symbol selection window and then opened a new drawing, the symbol selection window was automatically reopened.

## Distribution Board

---

### New and improved features

<p>Symbol selection</p>	<p>The following changes have been made to the symbol selection tool:</p> <ul style="list-style-type: none"> <li>• It is now possible to define your own favorite symbols after which they are easily available on the new <b>Favorite symbols</b> tab. The symbols can be selected from all the symbols: those in the current drawing or project as well as Electrical symbols.</li> <li>• On the new <b>Symbols in another drawing</b> tab, you can select another drawing from which to select symbols. You can then, for example, define these symbols as your favorites.</li> <li>• The search now also searches for a match in symbol names, in addition to the description.</li> <li>• On the <b>Symbols in current drawing</b> tab, you can right-click a symbol and select <b>Show occurrences from drawing</b>. The occurrences are then indicated with a yellow arrow.</li> <li>• In the <b>Symbol information</b> section, you can now copy the name and the description of the selected symbol to the clipboard.</li> </ul>
<p>Supported languages</p>	<p>Polish has been added to supported languages:</p> <ul style="list-style-type: none"> <li>• You can now select Polish as the design language in project settings.</li> <li>• You can now select Polish as the drawing frame/label and logo language.</li> </ul>

## Bug fixes

- Updating a distribution board by importing new feeder data for an existing group resulted in an unwanted new feeder.
- When the user first closed the symbol selection window and then opened a new drawing, the symbol selection window was automatically reopened.
- The application suggested to change group numbering after editing a group.

## Layout

---

### New and improved features

Symbol selection	<p>The following changes have been made to the symbol selection tool:</p> <ul style="list-style-type: none"> <li>• It is now possible to define your own favorite symbols after which they are easily available on the new <b>Favorite symbols</b> tab. The symbols can be selected from all the symbols: those in the current drawing or project as well as Electrical symbols.</li> <li>• On the new <b>Symbols in another drawing</b> tab, you can select another drawing from which to select symbols. You can then, for example, define these symbols as your favorites.</li> <li>• The search now also searches for a match in symbol names, in addition to the description.</li> <li>• On the <b>Symbols in current drawing</b> tab, you can right-click a symbol and select <b>Show occurrences from drawing</b>. The occurrences are then indicated with a yellow arrow.</li> <li>• In the <b>Symbol information</b> section, you can now copy the name and the description of the selected symbol to the clipboard.</li> </ul>
Functions	<p>In addition to system and elevation, you can also use device type as a filter in the Additional information boundary function.</p>
Distribution boards and groups	<p>In Electrical Premium and Electrical Basic, the following changes have been made in the distribution board and group functions:</p> <ul style="list-style-type: none"> <li>• You can now create distribution board hierarchy i.e. define which distribution board supplies another via the <b>Distribution Boards and</b></li> </ul>

	<p><b>Groups</b> project tree. This is especially useful when distribution boards have not been inserted into drawings yet. The <b>Connect to a supplying distribution board</b> function is available in the right-click menu for distribution boards, busbars, groups, and incoming feeders.</p> <ul style="list-style-type: none"> <li>• Removing a symbol in the group dialog now removes it from the drawing as well.</li> <li>• In the group dialog, it is now easier to move from one field to another with the Tabulator.</li> </ul> <p>In addition, in Electrical Premium, the following changes have been made in the <b>Distribution board and group management</b> dialog:</p> <ul style="list-style-type: none"> <li>• The incoming feeders are now shown in the dialog. This makes short-circuit current calculations, for example, easier when both the incoming and outgoing short-circuit currents are easily available.</li> <li>• The <b>Feeder Type</b> column is now available in the grid.</li> </ul>
Wiring	<p>You can now define a reference pair for a singular rise symbol. After inserting the symbol, the <b>Select reference pair</b> and <b>Select reference pair from drawing</b> functions are available in the right-click menu.</p>
Markings	<p>The <b>Insert object info to drawing</b> function has been improved:</p> <ul style="list-style-type: none"> <li>• You can add the short-circuit current, voltage drop, and trip time from the outgoing feeder to the group mark.</li> <li>• You can add the short-circuit current and power from the incoming feeder to the distribution board.</li> </ul>
Storey functions	<p>In storey settings, it is now possible to copy both buildings and storeys.</p>
3D drawing functions	<p>In Electrical Premium, the following changes have been made to IFC export:</p> <ul style="list-style-type: none"> <li>• The active building is now selected by default.</li> <li>• There is a new <b>Info</b> field in the IFC export dialog, listing errors and notifications. The field now notifies the user if the selected building has no storeys.</li> </ul>

Supported languages	<p>Polish has been added to supported languages:</p> <ul style="list-style-type: none"><li>• You can now select Polish as the design language in project settings.</li><li>• A Polish settings file is now available.</li><li>• You can now select Polish as the drawing frame/label and logo language.</li></ul>
---------------------	---

## Bug fixes

- Removing feeders in distribution board properties sometimes resulted in an error.
- When the application was not already running, opening a drawing with raceways sometimes resulted in an error.
- It was not possible to generate raceways in an individual drawing if the drawing had initially been opened along with other drawings.
- Starting wiring from an object whose system had no layer information resulted in an error.
- If only the **From** and **To** information were defined for a cable without drawing the cable, cable length was not calculated.
- When automatic removal was allowed wirings with feeder occurrences in distribution board schema drawings were removed, resulting in missing information.
- Some of the project's target information was not updated to the frame.
- After changing the distribution board symbol, it was slow to close and reopen the **Distribution board properties** dialog.
- Space information sometimes disappeared if the drawing with the space boundaries and device was not also opened.
- When counting quantities from the drawing, selecting to insert a graphical picture resulted in an error.
- When changing a symbol, the width of an attribute (distribution board ID, for example) was sometimes unnecessarily changed.
- In the distribution board dialog, it was possible to delete a symbol with occurrences in the drawing.
- If the Layout-specific setting **Default wiring system** was not enabled, the program repeatedly asked for the system when opening the drawing.
- New spaces were not available when counting quantities.

- When mirroring a symbol, the symbol factor and attributes were not always correctly mirrored.
- When the symbol was selected from the **Other** folder in symbol selection tool, the application asked for symbol name when inserting it.
- When the user first closed the symbol selection window and then opened a new drawing, the symbol selection window was automatically reopened.
- The **Remove cable** function did not remove cable occurrences from the drawing.
- Consecutive numbering was applied when copying marking symbols.
- If filter was used when copying a symbol and left on, it was not possible to add other information for the occurrence.
- After generating 3D, moving 2D symbols resulted in unwanted behavior for device groups. If 3D was generated again, the 3D offset also disappeared from symbols.
- Importing lighting fixtures from DIALux resulted in an error.

## Schematics

---

### New and improved features

<p>Symbol selection</p>	<p>The following changes have been made to the symbol selection tool:</p> <ul style="list-style-type: none"> <li>• It is now possible to define your own favorite symbols after which they are easily available on the new <b>Favorite symbols</b> tab. The symbols can be selected from all the symbols: those in the current drawing or project as well as Electrical symbols.</li> <li>• On the new <b>Symbols in another drawing</b> tab, you can select another drawing from which to select symbols. You can then, for example, define these symbols as your favorites.</li> <li>• The search now also searches for a match in symbol names, in addition to the description.</li> <li>• It is now possible to insert terminal blocks, plug connectors, and device connectors via the symbol selection window, from the <b>Terminals</b> directory. Selecting to insert a terminal symbol opens the insert dialog.</li> <li>• On the <b>Symbols in current drawing</b> tab, you can right-click a</li> </ul>
-------------------------	---



	<p>symbol and select <b>Show occurrences from drawing</b>. The occurrences are then indicated with a yellow arrow.</p> <ul style="list-style-type: none"> <li>• In the <b>Symbol information</b> section, you can now copy the name and the description of the selected symbol to the clipboard.</li> </ul>
<p>Wiring functions</p>	<p>The following changes have been made to the wiring functions:</p> <ul style="list-style-type: none"> <li>• In Electrical Premium, the <b>From</b> and <b>To</b> fields have been removed from the <b>Cable</b> section in the <b>Wiring</b> window. Instead, the <b>From</b> and <b>To</b> information is now shown after the <b>Cable</b> section heading.</li> <li>• In Electrical Premium, the <b>From</b> and <b>To</b> information is now also available for wires in the <b>Wiring</b> window.</li> <li>• In Electrical Premium, there is a new option, <b>Next cable automatically</b>, in the <b>Wiring</b> window. Enabling this allows you to draw cables continuously with consecutive numbering.</li> <li>• In Electrical Premium, when a cable or a cable package was drawn across a boundary, it was automatically cut. Now, you can decide in the <b>Wiring</b> window whether to cut it or not. By default, the new <b>Cut cable when drawn through boundary / Cut cable package when drawn through boundary</b> setting is enabled i.e. the cable / cable package will be cut.</li> <li>• In Electrical Premium, the following changes have been made to the cable packages: <ul style="list-style-type: none"> <li>◦ Creating new cable packages now works similarly to creating new cables: a new package can be created with the <b>New</b> button in the <b>Wiring</b> window.</li> <li>◦ In <b>Project information / settings</b>, in the <b>Cable markings</b> section, you can now select what to show as type information for cable packages: <b>Type</b>, <b>Type 2</b>, <b>Position</b> or <b>Description</b>.</li> <li>◦ It is now possible to define additional information for cable package definitions. The additional information is also shown in the <b>Cable package properties</b> dialog.</li> </ul> </li> <li>• The <b>Extend wire</b> and <b>Extend cable</b> functions have been</li> </ul>

	<p>improved: when continuing in another drawing, the active drawing is no longer changed as often as before.</p> <ul style="list-style-type: none"> <li>• If there are old cable marking default symbols with attributes on top of each other, they are now automatically replaced in drawings with symbols where attributes are not on top of each other. The replacement is done when the drawing is added in the project. In addition, you can carry out the replacement from the auxiliary functions by selecting <b>Change drawings symbols to current versions symbols</b>.</li> </ul>
Terminal block functions	<p>Symbol selection has been removed from the terminal block functions menu in the main toolbar. Instead, you can now easily select the desired symbol when inserting a terminal block, either from a menu or from the current drawing.</p>
Supported languages	<p>Polish has been added to supported languages:</p> <ul style="list-style-type: none"> <li>• You can now select Polish as the design language in project settings.</li> <li>• You can now select Polish as the drawing frame/label and logo language.</li> </ul>

## Bug fixes

- Extending a wire and then clicking **Cancel** in the **Wiring reference** dialog crashed the application.
- Ortho set off in connection with the wiring function was not set back on after wiring.
- Previously deleted objects were not removed from the run-time database.
- Canceling product model creation sometimes crashed the application.
- The application sometimes added too many symbols for I/O card product models.
- Symbol selection resulted in an error if there were duplicate symbols whose names were the same except for the upper and lower case letters used.
- In the **Wire properties** dialog, it was possible to select cable or cable package as the connection type.
- After inserting a logic symbol via the new symbol selection window, the old **Logic symbols** menu opened.

- When editing symbols, it was not possible to change the color or reposition the attribute.
- When the customer ID had been selected to be shown in the **Customer IDs** project settings, the internal wire number or color were not shown in wire markings despite being selected.
- Some of the project's target information was not updated to the frame.
- Changing the wiring color from the Draw toolbar did not work correctly.
- If the drawing only included parts of the symbol representing multiple parts but not the actual symbol, references were not generated correctly.
- The symbol selection tool did not show all symbols correctly.
- Spare amount entered when adding a new cable or editing an existing one was not saved correctly.
- References were sometimes lost when they were updated.
- Right-clicking a symbol with a reference sometimes crashed the application.
- When the user first closed the symbol selection window and then opened a new drawing, the symbol selection window was automatically reopened.
- When a wire was drawn over an element and therefore automatically cut, it was shown on top of the element instead of behind it.
- When drawing a cable through a boundary and finishing at an empty spot, the reference creation dialog did not open.
- When merging documents, the sheet order was incorrect.
- When first removing product information from a device in a drawing and then copying product information from another device resulted in an error.
- In SQL Server, removing the header table object ID and saving the drawing resulted in an error.

# Electrical DB (Premium)

## New and improved features

<p>Devices</p>	<p>On the <b>Devices</b> tab, the auxiliary grids <b>Circuits</b>, <b>Parent devices</b> and <b>Locations</b> are now available. They are especially useful when mass editing device information; you can easily drag the desired information from the auxiliary grids to the main grid for several devices at once.</p>
<p>Cables</p>	<p>The following changes have been made to cables:</p> <ul style="list-style-type: none"> <li>• It is now possible to define product information for cable types. Whenever a cable of that type is then drawn, it automatically gets the information defined for it, and the information can also be utilized in reporting.</li> <li>• The following changes have been made to cable packages: <ul style="list-style-type: none"> <li>◦ It is now possible to define additional information for cable package definitions. The additional information is also shown in the <b>Cable package properties</b> dialog.</li> <li>◦ When copying cable package definitions, the value from the <b>Position</b> field is copied along with the description and the position symbol.</li> <li>◦ It is now possible to import cable packages with import definitions.</li> <li>◦ Additional information of a cable package are now shown for individual cables in cable lists.</li> </ul> </li> <li>• On the <b>Cables</b> tab, the auxiliary grids <b>Devices</b> and <b>Locations</b> are now available. They are especially useful when mass editing the <b>From</b> and <b>To</b> information for cables; you can easily drag the desired information from the auxiliary grids to the main grid for several cables at once.</li> </ul>
<p>I/O functions</p>	<p>The following changes have been made to the I/O functions:</p> <ul style="list-style-type: none"> <li>• I/O channels are now listed in their own grid on the new <b>I/O channels</b> tab. On this tab, you can create I/Os and connect them as well as edit</li> </ul>

	<p>connected I/Os and I/O cards, for example. can be easily defined and disconnected as a mass with the help of the <b>I/O</b> auxiliary grid.</p> <ul style="list-style-type: none"> <li>• The <b>Locations</b> auxiliary grid has been added to I/O cards tab. In addition, the auxiliary grid named <b>Protective devices</b> has been renamed to <b>Devices</b>, and information can be dragged to the main grid as the card's supply device or parent device information.</li> <li>• The following new rules have been added to I/O card address creation: <ul style="list-style-type: none"> <li>◦ {23} = Value from the channel's <b>Spare 1</b> field.</li> <li>◦ {24} = Value from the channel's <b>Spare 2</b> field.</li> </ul> </li> </ul>
Reports	<p>The following changes have been made to reports:</p> <ul style="list-style-type: none"> <li>• <b>Sum list of cable products</b> has been added to report types, listing product information. In addition to English, the report template is available in Swedish, Estonian, Latvian, and Lithuanian.</li> <li>• In cable type sum lists, cable package's additional information is now shown for cables included in the package.</li> <li>• It is now possible to import cable packages and groups/feeders with import definitions.</li> </ul>
Supported languages	<p>Polish has been added to supported languages:</p> <ul style="list-style-type: none"> <li>• You can now select Polish as the design language in project settings.</li> <li>• Report templates are available in Polish.</li> </ul>
Database updates	<p>The following databases have been updated:</p> <ul style="list-style-type: none"> <li>• Sähkönumerot.fi</li> <li>• Dreamled product database</li> </ul>

## Bug fixes

- When adding product information for a location, the size information was not updated accordingly.
- When creating a new import definition, linking between the source and the target was not possible, if no rows had been created in the target table beforehand.
- Modular generation was sometimes interrupted if a cable type was included both in the Excel and the template drawing.

- In modular generation, updating cable types sometimes resulted in an error.
- Opening another project and sorting the projects resulted in an error.
- In I/O card address creation, rules 15 and 16 did not work correctly. Therefore, the **Spare 1** and **Spare 2** values were not taken into account.
- It was possible to import full IDs and electrical positions even when they did not meet criteria set for them in the application.
- **Automatic removal allowed** was enabled by default for devices created in the DB tool.
- If the only edit made in Excel was the change of upper or lower case, it was not implemented for the device.
- Importing a temporary project was interrupted if the location was changed during the import.
- In **Cable type management**, editing the cable wire index resulted in an error even when the index in question was not in use.
- When entering the first character in a grid cell with a drop-down menu, a value was immediately set as selected.
- In **Reports**, the **Name** column was not available for the cable connections list and the cable connections list by location.
- Project clean-up was very slow when removing documents.
- Project clean-up removed panels that were not marked as to be automatically removed.
- Modular generation resulted in an error if information removed earlier was included.
- In projects using Access database, rules for creating dynamic customer IDs were lost when the database was updated or the DB tool was closed and reopened after creating the rule.
- If a new project was created by copying an existing project with all its sub-directories, modular generation drawings did not end up in the project.
- In report creation, it was not possible to link the **Approver** or the **Revision Date** field to the report.