



CADMATIC

CADMATIC Electrical

Release notes

2023T1

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	6
Schematics	8
New and improved features	8
Bug fixes	12
Electrical DB (Premium)	13
New and improved features	13
Bug fixes	14

Release notes

The release notes for CADMATIC Electrical version 2023T1 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

Symbol standard	A new function, Symbol standard and location , has been added to the Cabinet Layout menu. With this function you can select a new symbol standard/locale for the project. The standard determines what kind of symbol menus will open from applications.
Symbol selection	A completely new tool is available for symbol selection in Electrical Premium and Electrical Basic: <ul style="list-style-type: none">• The tool allows you to view symbols in three ways:<ul style="list-style-type: none">◦ All symbols used by the program◦ All symbols in the current file◦ All symbols in the project• Inserting symbols into drawings is easy via the Symbols window.• The symbol selection window can be docked just like the properties window, so it can be kept open on the screen while working.• For the time being, the previously used symbol menus can still be used alongside the new tool.
Information box	The information box that appears when hovering over a part in the drawing has been redesigned. In addition to the appearance, a lot of attention has been paid to the readability and order of information displayed.
Copying objects	The size of the Edit values for copied objects dialog opening when copying and pasting objects can now be changed as needed.

Bug fixes

- The **Draw cabinet** function inserted cabinets with a 50 mm space between them.
- If there was no frame in the drawing, the default drawing frame was inserted when drawing a cabinet.

Distribution Board

New and improved features

Symbol standard	A new function, Symbol standard and location , has been added to the Distribution Board menu. With this function you can select a new symbol standard/locale for the project. The standard determines what kind of symbol menus will open from applications.
Symbol selection	A completely new tool is available for symbol selection in Electrical Premium and Electrical Basic: <ul style="list-style-type: none"> • The tool allows you to view symbols in three ways: <ul style="list-style-type: none"> ◦ All symbols used by the program ◦ All symbols in the current file ◦ All symbols in the project • Inserting symbols into drawings is easy via the Symbols window. • The symbol selection window can be docked just like the properties window, so it can be kept open on the screen while working. • For the time being, the previously used symbol menus can still be used alongside the new tool.
Information box	The information box that appears when hovering over a part in the drawing has been redesigned. In addition to the appearance, a lot of attention has been paid to the readability and order of information displayed.
Copying objects	The size of the Edit values for copied objects dialog opening when copying and pasting objects can now be changed as needed.
Lite application level	The Electrical Lite application level has been published. Compatibility with the other Electrical application levels enables smooth collaboration between users using different application levels (for example, designer and contractor).

Bug fixes

- It was not possible to create your own symbols with the **Create user symbol** function.
- Mass editing distribution board symbols resulted in two symbols for each group.
- Power information was erased from distribution board schema rows when the drawing was not in a project.
- Distribution board schema frame cover sheets included outdated standard information.

Layout

New and improved features

Symbol standard	A new function, Symbol standard and location , has been added to the Layout menu. With this function you can select a new symbol standard/locale for the project. The standard determines what kind of symbol menus will open from applications.
Symbol selection	<p>A completely new tool is available for symbol selection in Electrical Premium and Electrical Basic:</p> <ul style="list-style-type: none"> • The tool allows you to view symbols in three ways: <ul style="list-style-type: none"> ◦ All symbols used by the program ◦ All symbols in the current file ◦ All symbols in the project • All symbol insertion functions remain unchanged, but it is now a lot easier to use them: all functions are conveniently available in the symbol selection tool. • The insertion method, elevation and system can be defined through a single interface when inserting the symbols. • The symbol selection window can be docked just like the properties window, so it can be kept open on the screen while working. • For the time being, the previously used symbol menus can still be used alongside the new tool.
Information box	The information box that appears when hovering over a part in the drawing has been redesigned. In addition to the appearance, a lot of attention has been paid to the readability and order of information

	displayed.
Copying objects	The size of the Edit values for copied objects dialog opening when copying and pasting objects can now be changed as needed.
Space functions	The following changes have been done to space functions: <ul style="list-style-type: none"> • Spaces are no longer implemented as groups, instead symbols are created. This makes it easier to manage and edit spaces. The new symbol-based functionality still supports groups, but new spaces are automatically created according to the symbol method. • The user interface for editing space information has been renewed.
Cables	It is now possible to edit cables by right-clicking them in the drawing and selecting Edit cable .
Earthing	The earthing symbols have been updated as follows: <ul style="list-style-type: none"> • The default value has been removed from the earthing diagram symbol IDs. In addition, attributes have been moved to the side of symbols. • In Layout symbols, the position of the ID and the text have been reversed and the ID has been set visible by default.
Project tree	In the Documents project tree, the right-click menu functions were different in Design mode and Print mode. The functions have now been unified.
Lite application level	The Electrical Lite application level has been published. Compatibility with the other Electrical application levels enables smooth collaboration between users using different application levels (for example, designer and contractor).

Bug fixes

- When copying groups and their cables from a drawing not in a project, the cable types were changed in the copy.
- The protective device current information was not always shown when opening the **Group / feeder properties** dialog for the first time.

- The program crashed when removing a feeder with empty wiring information and selecting the **Wiring** option for another feeder in the same group.
- When using the **Insert symbols in area** function, IDs were not assigned for symbols.
- When inserting a device from the project tree, space information was not assigned unless the drawing was reopened or the device moved after insertion.
- When protective device information changed, the feeder occurrences protected by that device were not updated.
- When a symbol was assigned a position number in a drawing that was not in a project, it was no longer possible to edit the attributes.
- When changing a symbol, attributes at a 180-degree angle were unnecessarily moved.
- In Distribution Board and Group management, changing the order of groups/feeders did not work correctly.
- Comparing document versions was sometimes slow.
- There were issues with distribution board symbols: sometimes it was not possible to change the symbol (both 2D and 3D), it was allowed to assign the symbol to the location instead of the distribution board, and unintentionally created symbols were not removed.
- When a symbol was replaced with the same symbol, the symbol scale was also reset.
- Mass editing distribution board symbols resulted in two symbols for each group.
- It was possible to select a location other than distribution board for a group.
- Copying two distribution boards without wire occurrences resulted in wiring duplicates lacking either the **From** or **To** information.
- When a column had been hidden in Distribution Board and Group management by dragging the width to zero, it stayed hidden even when restarting the dialog or using the **Show columns** function.
- In the **Group / feeder properties** dialog, the description given to a feeder was not saved when the drawing was not in a project.
- 3D symbols changed for a product model did not update in the drawing.
- Changing incoming feeders to outgoing feeders did not change cable's direction.
- If the system/layer was changed after the first wiring point had already been selected, the cable went to the original layer instead of the layer of the newly selected system.
- When wiring between storeys without group mark, the wiring direction was sometimes turned which led to removal of feeders and groups.
- Changing a cable to a cable package broke the hierarchy between two distribution boards.

- Some functions opened by right-clicking from the project tree caused issues.
- Filtering made in the **Distribution properties** dialog also affected the **Group / feeder properties** dialog.
- Exporting to IFC resulted in an error if other settings were defined before clicking **Storeys**.
- When a project was copied in SQL, project check-up was not done for the cable reports.
- There were issues with the **Insert object info to drawing** function: symbol texts and wiring information were only shown in functions after insertion, **Text 4** was not grouped with **Text 1** and **Text 2** and there was no **Text 3**, and phasing information was missing.
- Location texts and device texts were mixed when editing between Layout and Schematics.

Schematics

New and improved features

<p>Drawing frames</p>	<p>New large drawing frames are available:</p> <ul style="list-style-type: none"> • A0, landscape and portrait with title block • A1, landscape and portrait with title block • A2, landscape and portrait with title block <p>In addition, a new A3 landscape frame with circuit information has been added.</p>
<p>Symbol standard</p>	<p>A new function, Symbol standard and location, has been added to the Schematics menu. With this function you can select a new symbol standard/locale for the project. The standard determines what kind of symbol menus will open from applications.</p>
<p>Settings</p>	<p>The following new settings have been added to the Schematics-specific settings:</p> <ul style="list-style-type: none"> • Use osnap nearest – When this setting is enabled, snap will force points to the nearest point of object. This is useful when wiring from boundary to boundary, for example, instead of from wiring point to wiring point. • Show the wiring dialog on modifying (Electrical Premium) – When this setting is enabled and you click a wire, cable or

	<p>cable package, it is selected and the Wiring window opens. When this setting is disabled, the wire, cable or cable package is selected but the Wiring window will not open.</p>
<p>Project information/settings</p>	<p>The following new settings have been added to the Cable markings section in project settings:</p> <ul style="list-style-type: none"> • Visibility of Type in drawing – When this setting has been set to According to visibility setting of symbol's attribute, you can select to hide the cable type information in the Cable and Cable combing dialogs and in the Wiring window when drawing a cable (Premium level). If the setting has been set to Always visible or Not visible, hiding is disabled in the aforementioned functions. • Default symbol of cable markings – When a symbol has been selected, you can select whether to use it when drawing a cable (Premium level).
<p>Project tree</p>	<p>In the Wirings project tree, the following functions have been added to the right-click menu of the Cables node:</p> <ul style="list-style-type: none"> • Connect and mark cable and its wires • Draw cable • Cable markings • Cable combing
<p>Symbol selection</p>	<p>A completely new tool is available for symbol selection in Electrical Premium and Electrical Basic:</p> <ul style="list-style-type: none"> • The tool allows you to view symbols in three ways: <ul style="list-style-type: none"> ◦ All symbols used by the program ◦ All symbols in the current file ◦ All symbols in the project • Inserting symbols into drawings is easy via the Symbols window. • The symbol selection window can be docked just like the properties window, so it can be kept open on the screen

	<p>while working.</p> <ul style="list-style-type: none"> • For the time being, the previously used symbol menus can still be used alongside the new tool.
Information box	<p>The information box that appears when hovering over a part in the drawing has been redesigned. In addition to the appearance, a lot of attention has been paid to the readability and order of information displayed.</p>
Copying objects	<p>The size of the Edit values for copied objects dialog opening when copying and pasting objects can now be changed as needed.</p>
Wiring	<p>The following changes have been made to the wiring functions:</p> <ul style="list-style-type: none"> • The program now remembers the settings defined previously in the Cable combing dialog. • When the project's cable marking setting Visibility of Type in drawing has been set to According to visibility setting of symbol's attribute, you can select to hide the cable type information in the Cable and Cable combing dialogs and when drawing a cable. If the setting has been set to Always visible or Not visible, hiding is disabled in the aforementioned functions. • When a symbol has been defined to the project's cable marking setting Default symbol of cable markings, you can select whether to use it when drawing a cable (Premium level). • The Redraw the wire and Extend wire functions now allow defining a reference when ending the wire at an empty spot in the drawing. • The "wiring set" term has been changed to "cable package". In connection with the term change, the following improvements have been made: <ul style="list-style-type: none"> ◦ In the Cable package properties dialog, you can manage cable packages. Adding, removing and renaming cables is easy, in addition to which you can

	<p>define additional information, product information and plates for the packages.</p> <ul style="list-style-type: none"> ○ In the Wirings project tree, the following cable package insertion and editing functions have been added to the right-click menu: <ul style="list-style-type: none"> ▪ Copy cable package ▪ Draw cable package occurrence ▪ Insert wiring marking ▪ Reverse wiring direction ○ It is possible to add position for a cable package definition. The cable markings are then included in the description while only the position of the cable package is marked in the drawing – this improves the readability of the drawing.
Devices	In Electrical Premium, the Space name field has been renamed to Space info in the Device properties dialog. In addition, the space number (if defined) is now added after the space name.
Cables	In Electrical Premium, the Space name field has been renamed to Space info in the Cable properties dialog. In addition, the space number (if defined) is now added after the space name.
Locations	In Electrical Premium, the Space name field has been renamed to Space info in the location property dialogs (such as Station properties). In addition, the space number (if defined) is now added after the space name.
Terminal blocks	Symbols for terminal blocks, plug connectors and device connectors have been added. They can be inserted from the Symbols menu added to the Terminal block functions toolbar. It is also possible to create your own terminal blocks via the new menus.
Lite application level	The Electrical Lite application level has been published. Compatibility with the other Electrical application levels enables smooth collaboration between users using different application levels (for example, designer and contractor).

Bug fixes

- In the wiring function after indicating the first point, opening cable management or color settings caused problems.
- When marking wiring references, the application stayed on the target sheet even when the **Stay on target sheet** option was not selected in the **Wiring reference** dialog.
- When drawing a cable across multiple boundaries, all cables were assigned the same ID.
- After first adding I/O channel symbols and then inserting the I/O card symbol resulted in I/O texts disappearing.
- Reference pair IDs were not checked when adding documents to project, resulting in unwanted pairing of unpaired references.
- When returning to the drawing from the cable edit dialog, the cable was sometimes unnecessarily saved when saving the drawing.
- Cable type and other information was not shown correctly for cables without wires.
- In Electrical Premium, if the user clicked the **New** button while drawing a cable the previously selected cable type was not taken into account for the new cable.
- In cable combing, the cable shield was misplaced when the **Pair shield** setting was enabled but there was no pair shield.
- Redrawing did not work when cables had been drawn as lines instead of polylines.
- When changing from one project to another, the **Wiring** window listed cables from the previous project.
- The **Extend wire** function threw an exception when extending to another document.
- The **Redraw wire** function threw an exception when continuing to another document.
- The I/O tag attribute E_IOTAGNAME[0] for channel 0 did not update.
- Disconnecting a wiring reference sometimes resulted in an error.
- Sometimes a terminal strip and its terminal blocks ended up in different occurrences.
- Setting your own drawing frame as the default frame resulted in an error.
- Changing drawing frame language sometimes resulted in duplicates.
- Replacing a drawing frame inserted the new frame twice.
- Location texts and device texts were mixed when editing between Layout and Schematics.

Electrical DB (Premium)

New and improved features

Editing functions	<p>A new option, Paste from clipboard, has been added to the right-click menu in the various grids. With this function, you can paste values copied with the Copy cell values to clipboard function or key combination Ctrl + C to other cells.</p> <p>Alternatively, you can still use Ctrl + V.</p>
Search and replace	<p>The function has been improved as follows:</p> <ul style="list-style-type: none"> • If the user has selected one or more columns with all their cells in the grid before starting the function, the Search from columns option is automatically enabled on both the Search and the Replace tabs. • If the user has selected individual cells from the grid before starting the function, the Only from items selected in the grid option on the Replace tab is automatically enabled. <p>Only one of the options mentioned above can be active at a time.</p>
Devices	<p>In the Device properties dialog, the Space name field has been renamed to Space info. In addition, the space number (if defined) is now added after the space name.</p>
Cables	<p>In the Cable properties dialog, the Space name field has been renamed to Space info. In addition, the space number (if defined) is now added after the space name.</p>
Locations	<p>In the location property dialogs (such as Station properties), the Space name field has been renamed to Space info. In addition, the space number (if defined) is now added after the space name.</p>
Plates	<p>In plate definitions, it is now possible to define additional information for plates whose parent type is Feeder.</p>
Functions	<p>In the new Uninserted symbols section in the Clean up project dialog, it is now possible to select symbols that have not been inserted to be removed from devices, distribution boards, and feeders. However, symbols originating</p>

	from product models will not be removed.
--	--

Bug fixes

- In DB reports, including values with forbidden characters resulted in an error.
- It was not possible to use *Electric.DB.Updater.exe* with SQL Server credentials.
- Channel address entered in the **I/O properties** dialog was shown in the **I/O Address** column instead of the **Channel address** column.
- In Excel import, fill mode did not work properly if the cable type was not linked.
- The program crashed when removing a feeder with empty wiring information and selecting the **Wiring** option for another feeder in the same group.
- In Product information management, importing did not work.
- When adding new terminal blocks after the existing ones, they were not consecutively numbered.
- In SQL Server Project Management, the global filter did not work as should.
- In SQL Server Project Management, updating the database was not possible because of missing version number.
- Clean-up removed incoming and outgoing feeders incorrectly.
- When importing from Excel, hierarchy bindings created new hierarchies instead of merging into the existing ones.
- In the **Project information / settings** dialog, selecting **Plate definitions** resulted in an error.
- A single quotation mark as the only value in a field sometimes resulted in an error when importing product information.
- If several devices in Excel import referred to a product model that did not exist, an error with an unclear message was shown for each device.
- In reports, the **All name plates** list type did not list wire plates.
- A new product database was not available until reopening product information selection.
- In SQL Server project management, shared database selection did not work.
- In Excel import, existing additional information was replaced with new information when using the **Fill** mode.
- Date format defined in project settings was not taken into account in modular generation.
- In SQL, creating a new protective device resulted in an error.
- In picking lists, your own additional info was shown as code instead of the name.