CADMATIC Electrical

Release information | 2021T2

Contents

Release information

The release notes for CADMATIC Electrical version 2021T2 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

Printing functions	The dialog for the **Queue printing from drawing sheets** function now has columns that show sheet number, sheet names (1–3) and drawing number. These new columns make it easier to decide which sheets to print.

Fixed bugs

- When inserting a terminal block into a drawing, the size changed to the size of the symbol.

Distribution Board

New and improved features

Printing functions	The dialog for the **Queue printing from drawing sheets** function now has columns that show sheet number, sheet names (1–3) and drawing number. These new columns make it easier to decide which sheets to print.

Bug fixes

- In a wide distribution board schema frame, the default location for a symbol inserted using F8 was incorrect.
- In distribution board schema frames, the grid was changed to 0.5,0.5.

- When filling the distribution board schema, it was possible to enter an ending row that did not exist in the drawing, which resulted in Draw crashing.

Layout

New and improved features

Functions	The Insert device text to drawing function is now available in the **Functions** toolbar.
Setting files management	In setting file management, it is now possible to update changes to the drawing when closing the setting file management dialog.
Auxiliary functions	In the **Convert to Electrical symbol** function, you can now define elevation and system. • Both the elevation and the system can be picked from the drawing. • Defining the system is mandatory. • Selecting a system also sets the default elevation if one has been defined for the system (**Functions > Set default elevation values for this drawing**). In addition, you can select **Convert all symbols with the same name** to create IDs for all symbols with the same name. Otherwise, only the symbol type will be changed for the symbols not selected.
Markings	• The **Insert object info to drawing** function has been improved: ◦ The function remembers your definitions for text size and alignment the next time you open it. ◦ It is now possible to define more product information for drawings with the **Name 1**, **Name 2**, **Amount**, and **Planning unit** fields. ◦ When inserting, the symbol can be rotated with F8 to set the desired angle. The angle stays when updating, moving, or copying the symbol. ◦ Symbols added with this function can be edited by right-clicking and selecting **Edit symbol**. The changes can be updated to all the

	similar symbols, or to the selected symbol only. • When inserting, the address symbol can be rotated with F8 to set the desired angle. The angle stays when updating, moving, or copying the symbol.
Saving symbols	When saving symbols, you can now select the E_Elevation attribute, with which it is easy to add an elevation to a symbol.
Printing functions	The dialog for the **Queue printing from drawing sheets** function now has columns that show sheet number, sheet names (1–3) and drawing number. These new columns make it easier to decide which sheets to print.
Visualization window	The new  button added in the **3D drawing functions** toolbar opens the **Electrical visualization window** dialog. You can also open the dialog with the *EVWINDOW* command. When you select **Use visualization window**, a floating window opens with which you can, for example, inspect 3D objects on a second monitor. When the visualization window is in use, you can define the its layer states with setting groups. Ctrl and Shift can be used when selecting multiple setting groups. The function remembers the previously selected setting groups the next time you use it.

Bug fixes

- The **Group** dialog showed calculated length instead of the length manually defined for a cable.
- Rounding of raceway elevation did not work in some cases.
- When selecting a symbol from a drawing and trying to copy it, CADMATIC Draw was closed.
- Changing the color for a raceway from the color menu at the top of the screen did not change the raceway color in the drawing.
- When drawing a cable, the inserted symbol was assigned the _C system of the cable.
- Converting a symbol to an Electrical symbol did not work, because after filtering out the standard Electrical symbols there were no symbols left to convert.
- 3D generation of raceways made Draw close.

- Saving your own 3D symbol did not add to the icon menu, if **Save icon area by yourself** was selected.
- The **Define elevation** function did not update the reference.
- The **Insertion point of element's electrical outlet** function always asked for a system.
- Storey settings did not work in IFC export, if the drawing was not in the project.
- With raceways, there were problems related to the location of the reference line and reference mark values being cleared.
- Race-way color changed when copied.
- IFC export included empty information which resulted in errors.
- Saving in DWG lost position marking from motion detectors, if device occurrences did not have the E_POSID attribute.
- Changing the storey for wiring did not work, if the drawing was not in the project.
- Additional information removed from a product model were still shown for the product model and the devices inserted/added with it.
- The **Insert object info to drawing** symbol did not stay at the indicated location when it was moved together with a mirrored symbol.
- When converting from version 18 to version 19, device attributes were moved to their default positions.
- Locking device attribute editing did not always work as expected.

Schematics

New and improved features

Opening projects	Opening projects from project management is now faster. Project settings are read from all the project databases at the same time, and a project can be selected before all the project settings have been read.
Project tree	New options have been added to the **Find** project tree: • **Unconnected cables** – The selection lists cables that are not connected to any device's wiring point. • **Unconnected wire occurrences** – The selection lists cable wires, wire harness wires and internal wires and jumper bars, with one end unconnected and with empty **From** or **To** information.

Markings	The following changes have been made to markings: • The **Insert object info to drawing** function has been improved: ◦ The function remembers your definitions for text size and alignment the next time you open it. ◦ It is now possible to define more product information for drawings with the **Name 1**, **Name 2**, **Amount**, and **Planning unit** fields. ◦ Symbols added with this function can be edited by right-clicking and selecting **Edit symbol**. The changes can be updated to all the similar symbols, or to the selected symbol only. • The **Additional info to objects** function highlights a missing attribute in red. If you select additional information with missing attributes, you are notified.
Symbol functions	The **Create a symbol from Device Boundary and the objects in it** function has been added to the **Auxiliary functions** menu. With this function, you can create a new symbol of a device boundary and the symbols in it.
Selecting location/device	The **Select Location/Device** dialog used when defining ID now has a search field which makes finding the desired location/device easier.
Devices	The functions **Change device's location to surrounding boundary's location** and **Change device's circuit to surrounding boundary's circuit** have been added to the context menu of device occurrences. With these functions, you can reset the location or circuit back to that of the surrounding boundary.
Wiring	• The following changes have been made to the cable drawing functions: ◦ The functions **Draw cable**, **Redraw cable** and **Extend cable** are now available in the cable occurrence context menu. ◦ When a cable ends to a device or location boundary line, the **Reference to another sheet** dialog does not open. ◦ When a cable is drawn over a device or location boundary line, it is cut. ◦ The **Continue later** function has been added to the **Reference to**

	another sheet dialog. ◦ When creating the reference to another sheet, you can now also define the reference ID. ◦ Cut points are now supported. • When drawing a wire, you can now also define the reference ID when creating the reference to another sheet. • The order of steps in which the **Mark cable and its wires** function is carried out has changed: you first draw the cable across a wire line and only then define cable data. This way, the function can retrieve the **From** and **To** information for the cable from the drawing. You can also open this function via the wiring tree. However, the cable has then already been selected so you need to first define the cable data and then draw the cable.
I/O functions	The following changes have been made to the I/O functions: • A new function, **Create a new I/O to this channel**, has been added to the context menu for channel symbols. With this function, you can create a new I/O as well as connect it to the selected channel. • It is now possible to define the **Start address IN**, **Start address OUT** and **Bit address offset** information when creating an I/O card. With this information, addresses can be created to the card right away. • There is a new option, **Create a new I/O rack**, in the **I/O functions** toolbar. • When you right-click on an I/O card and select **Insert all card's channels into drawing**, the function checks whether channel symbols have already been inserted into the drawing. ◦ If symbols have been inserted, the user is prompted to select whether to insert all symbols or only the ones not inserted. ◦ If symbols have not been inserted, all symbols are inserted starting from the first channel. • The **Logic type** and **Logic elec. 1** attribute values now automatically come from the product model for I/O card input and output module symbols.

Printing functions	The dialog for the **Queue printing from drawing sheets** function now has columns that show sheet number, sheet names (1–3) and drawing number. These new columns make it easier to decide which sheets to print.
Reports	In the cable list, there are two new fields: **Sheets** and **Document types**.

Bug fixes

- Object ID / electrical position defined on the **Properties** tab changed the customer ID when the device had sub-devices.
- The **Assign symbols in drawing to this device** function did not change all occurrences.
- Location defined which wiring types were available in internal wiring.
- Drawing a cable wire did not update cable combining references.
- Cable combining lost wire connection data.
- When generating circuit drawings, templates generated separate drawings.
- Removing a cable without syncing is not possible even if the connected wirings have been removed.
- Opening and saving a drawing added wiring points in the database.
- Cable schema changed the cable connection.
- In device properties, a new position replaced the previously defined position.
- Changing the direction of a vertical reference arrow did not work.
- Reference texts were not copied.
- When the drawing was not in the project, the targets for wiring references changed when synchronized.
- When generating drawings from multiple circuits at the same time, the part number changed.
- When generating drawings based on Excel, all information was not imported if the template drawing was open when generation was started.
- If a contact pack only had one contact, moving to the other end of the reference did not work.
- In SQL project, extra electrical positions were created inside location boundaries.
- Coil cross-reference was not marked to coil pin's sub-reference occurrence for the primary object.
- When saving a drawing attachment to a new directory, the first pin description was removed

from each I/O channel.

- Creating and editing I/O card devices did not work as expected.

Electrical DB

New and improved features

I/O functions	The following changes have been made to the I/O functions: • It is now possible to define the **Start address IN**, **Start address OUT** and **Bit address offset** information when creating an I/O card. With this information, addresses can be created to the card right away. • In Schematics, there is a new option, **Create a new I/O rack**, in the **I/O functions** toolbar.
Project settings	Norwegian is now available as a design language in the general settings. Wire colors are also available in Norwegian.
Import functions	The following changes have been made to import functions: • If a linked column has been removed from an Excel file used in import definition, the link is highlighted with red in the target table. In this case, the link needs to be removed from the definition, or the column needs to be added to the Excel file again. • In Excel, Access and XML import, it is now possible to define an ID style for the objects imported as new. The **Increase** option increases the number in the ID, whereas the **Add** option adds a consecutive number in square brackets to the ID. • The following changes have been made to the clipboard project: ◦ Object ID and electrical positions are organized hierarchically. ◦ The function remembers the columns you have selected to show and hide in the **Show columns** dialog. ◦ The **Import mode** column is shown by default on all tabs. • When importing product information, more information is available if a row is not saved.
Picking lists	A picking list has been added for the **Spare part** field.

Selecting location, circuit or device	When using the tree view in location, circuit and device selection, it is now possible to create new locations, circuits or devices in the selection dialog.
Additional information	The ETIM database has been updated, and Estonian texts have been added.
Search and replace function	You can now select several columns with which to filter information.
Reports	The following new fields have been added to the cable list: • **Sheets** • **Document types** • **From Space Name** • **From Space Flat** • **From Space Number Text** • **From Space Number** • **To Space Name** • **To Space Flat** • **To Space Number Text** • **To Space Number**
Modular generation	In modular generation, the following changes have been done: • The Excel file can be entered with the MODULARGENERATECMD command. • Standard columns **StartAddressIN** and **StartAddressOUT** with versions from 1 to 10 are now available. • Module directory: ◦ If you open module generation without an open drawing, the module directory is taken from the project's default settings. ◦ The … button for module directory always opens the directory selection dialog instead of project settings, even for project databases. ◦ The selected module directory is interpreted as relative, if it is

	below the target project. ◦ The selected module directory is saved in the target project settings. • Linking parameters has been improved: ◦ Instead of the entire row, the selected cell is activated. This way, you can see which link you are about to create. ◦ In the generation dialog, you can select multiple cells that will then be linked one by one. ◦ If an attribute is read-only, linking is not possible and it will prompt a notification. ◦ The buttons and the link value stay visible while scrolling horizontally. ◦ You can add several pieces of product information in one cell with separators. ◦ It is now possible to link cable types (for example $CABLE TYPE [1]$). ◦ Mtext from drawings can be used as a link in generation. ◦ It is now possible to link product information without using product models. **DeviceProduct**, **AddDeviceProduct**, **LocationProduct** and **AddLocationProduct** columns are now available in standard columns.

Bug fixes

- Product information names were imported from Excel to the wrong product information database.
- The information selected from the product information grid were no longer selected in the Product information selection dialog.
- Updating the SQL Server database structure caused problems.
- When importing plates, additional information and names, duplicates were created for the parent.
- Attaching a product model to a project prompted an error due to incorrect link data.
- In import definition (Excel), the background color for the active tab was not updated correctly.

- In import definition (Excel), the background color for a column removed from the source file did not turn red.
- Removing a device selected from the tree did not work correctly.
- Clipboard project did not import product information for devices, if the information was already found for the project device. Therefore, product information was missing from copied devices.
- Importing product information databases from SQL server to Access resulted in erroneous product information rows.
- Excel import only imported one piece of additional information for devices.
- Changing the cable type caused errors.
- It was not possible to filter or sort a new additional information column.