CADMATIC Electrical

Release information | 2021T3

FINLAND THE NETHERLANDS CHINA HUNGARY INDIA ITALY RUSSIA SINGAPORE UAE

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

Contents

Release information

The release notes for CADMATIC Electrical version 2021T3 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

Symbolics	After the first installation when you first start the application, you are asked to define which symbol standard to use in projects. This affects the kind of symbol menus you will have. The selected standard will be used as the default for new projects. You can change it project-specifically in the general symbol settings.
Logos	When you insert a logo in the drawing, the program now checks whether the drawing already has a logo. If the program finds a logo, it asks if you want to replace the existing logo.
Project tree	In the **Find** project tree, context menus are now available showing the same functions as in the device, wiring and product model trees, depending on the selected node. You can, for example, easily insert a device into the drawing from your own selection list.
Handle sheets	The naming of customer drawing target files has been changed: • The file name is no longer automatically modified. If the source and target file names are identical, saving is not done. • The new **Add sheet number range to file name** option added for the **Create an array of sheets** setting adds the sheet number range to the file name.
Copying object	The settings dialog for copying object data has been renewed. The new **Paste**

data	dialog only shows the data the copied objects actually have. The default settings for the dialog are still defined in the general Electrical settings.
Inserting symbols	Inserting symbols and symbol IDs has been improved: When you place a non-scalable symbol in the drawing with the **Insert devices of the location** function, the ID of the symbol does not scale according to the symbol size but stays in the default size.
Product models	Importing pins and additional information for product information selected in the product model has been improved. • When selecting product information for a product model, you can now decide how to handle the pins and additional information: add missing information only, add missing and overwrite existing, or do not add. • With the **Import product information pins and additional infos** button, you can update pins and additional information for product information selected in the product model without opening the product information dialog.
Device and location boundaries	The following changes have been made to boundary functions: • You can now select the symbol for the boundary appearance from the Layout menu. • When you select a symbol for the boundary appearance from a menu, the symbol menu toolbar is shown first.

Bug fixes

- In the project tree context menu, there were functions (such as **Assign symbols in drawing to this device**) that did not work in print mode. They have now been removed.
- When inserting a symbol, the **OK** button was sometimes disabled when it should not have been.
- Removing an attribute at symbol creation caused the program to crash.
- Even if you had defined size information as product information, for example, for a terminal block, the **Draw terminal strip** settings still had the default values.
- The **Convert to product model's device** function did not fetch the product model symbol when it was not in the drawing.

Distribution Board

New and improved features

Symbolics	After the first installation when you first start the application, you are asked to define which symbol standard to use in projects. This affects the kind of symbol menus you will have. The selected standard will be used as the default for new projects. You can change it project-specifically in the general symbol settings.
Logos	When you insert a logo in the drawing, the program now checks whether the drawing already has a logo. If the program finds a logo, it asks if you want to replace the existing logo.
Project tree	In the **Find** project tree, context menus are now available showing the same functions as in the device, wiring and product model trees, depending on the selected node. You can, for example, easily insert a device into the drawing from your own selection list.
Handle sheets	The naming of customer drawing target files has been changed: • The file name is no longer automatically modified. If the source and target file names are identical, saving is not done. • The new **Add sheet number range to file name** option added for the **Create an array of sheets** setting adds the sheet number range to the file name.
Copying object data	The settings dialog for copying object data has been renewed. The new **Paste** dialog only shows the data the copied objects actually have. The default settings for the dialog are still defined in the general Electrical settings.

Bug fixes

- If there was more than one cover sheet in a drawing, the filling function did not work properly.
- In the **Distribution Board and Group management** dialog, showing/hiding the **Distribution Board** and **Group** columns did not work correctly.

Layout

New and improved features

Symbolics	After the first installation when you first start the application, you are asked to define which symbol standard to use in projects. This affects the kind of symbol menus you will have. The selected standard will be used as the default for new projects. You can change it project-specifically in the general symbol settings.
Logos	When you insert a logo in the drawing, the program now checks whether the drawing already has a logo. If the program finds a logo, it asks if you want to replace the existing logo.
Project tree	In the **Find** project tree, context menus are now available showing the same functions as in the device, wiring and product model trees, depending on the selected node. You can, for example, easily insert a device into the drawing from your own selection list.
Handle sheets	The naming of customer drawing target files has been changed: • The file name is no longer automatically modified. If the source and target file names are identical, saving is not done. • The new **Add sheet number range to file name** option added for the **Create an array of sheets** setting adds the sheet number range to the file name.
Copying object data	The settings dialog for copying object data has been renewed. The new **Paste** dialog only shows the data the copied objects actually have. The default settings for the dialog are still defined in the general Electrical settings.
Size data of equipment	Defining and presenting size data has been improved: • In the drawing, size data is presented as a scalable symbol which creates a sub-occurrence. • When the **Define size of devices** dialog opens, it shows the current size data for the device. • If data comes from the product model, size data is not set for the

	device. • The **Size data of equipment** and **Delete size data of equipment** functions have been added to symbol functions. • The **Draw/update size data of equipment** and **Deactivate size data of equipment** functions have been removed from lighting functions.
Elevations	Picking elevation has been changed so that if a 3D object is selected, elevation is taken from the sub-occurrences's insertion point.
Functions	The following changes have been made to functions: • You can now use the **Define element elevation data afterwards** function to change the elevation for holes. • The **Use the previous offset-value for inserting** setting has been added to the **Insert object info to drawing** function. ◦ The setting enables using the same offset easily in different drawings. ◦ When enabled, you do not need to insert the symbol. Instead, it is automatically inserted according to the saved offset. ◦ If the previous value has not been saved yet, the option is disabled. The offset value is saved after inserting the symbol, and it will then be available the next time.
Wiring and markings	The following changes have been made to the wiring functions: • The logic of drawing continuous wiring has been changed: when one continuous wiring is ended with Enter, the function is no longer closed. Instead, the program asks you to indicate the start point for a new wiring. Pressing Enter when defining the start point will close the function. • A new button, **Mark/pick wiring IDs, reference line**, has been added to the **Wiring and markings** toolbar. With this function, it is possible to add wiring IDs for the selected wires in a layout.
Inserting symbols	It is now possible to automatically create a device group from a symbol inserted next to another one, by selecting the **Automatically add symbol to ID device group when inserting left or right** option in the settings dialog.

Groups	You can now create additional information for a group with the **Additional info to objects** function.
Product models	Importing pins and additional information for product information selected in the product model has been improved. • When selecting product information for a product model, you can now decide how to handle the pins and additional information: add missing information only, add missing and overwrite existing, or do not add. • With the **Import product information pins and additional infos** button, you can update pins and additional information for product information selected in the product model without opening the product information dialog.
Floors and spaces	The **Floors and spaces** project tree now displays apartments. This makes it easier to manage spaces in the project tree, when a building consists of several spaces/floor (hotels and apartment buildings, for example). You can add multiple apartments to a floor, and you can add multiple spaces to an apartment. The project tree groups the spaces in the same apartment together.
IFC export	You can now export the storeys of a selected building into separate IFC files. The program names the files after the names of the building, the storey, and an optional planning area.
Plant Modeller integration	The following changes have been made to the integration: • It is now possible to use customer ID when exporting devices. This enables a common ID format between Electrical and the 3D model. The new setting in the mapping dialog affects the IDs for cables, devices, locations and cable targets. Furthermore, publishing to CADMATIC eShare and showing eShare information also follow this setting. • It is now possible to update location coordinates from Plant Modeller to Electrical. The new setting is available in the **Export/synchronize devices** dialog. When the setting is on, the drawings (layout drawings only) are opened and the changes are implemented, after which the

	drawings are left open. The drawings are not automatically saved. • In device import, there is a new column showing the Requires electricity attribute value, based on which the list can be sorted.

Bug fixes

- All cables in a wiring package had the same ID as the first cable, even when individual IDs had been defined.
- Counting quantities did not work, if you selected multiple options.
- The right side of a scalable symbol icon in product model edit was cut out.
- When inserting a device, the elevation value was 0 even when the value for the scalable symbol had been modified.
- In the **Distribution Board and Group management** dialog, showing/hiding the **Distribution Board** and **Group** columns did not work correctly.
- The feed length did not update in the **Group** dialog.
- When a product model with default system had been created based on a scalable symbol, the program always asked for a system when using the **Create new devices and insert them in selected area** function.
- Updating a value in an additional information boundary did not affect the **Insert object info to drawing** symbols.
- When inserting multiple IDs with the **Insert info to drawing** function, the IDs were shown correctly only after syncing.
- The **Insert object info to drawing** function did not work well with angled coordinates.
- Creating a product model from a scalable symbol added two 3D symbols to the product model.
- When using the **Save a copy of the drawing to the project** function, the 3D symbols of devices were changed to scalable boxes.
- When the system was left empty in the **Distribution Board** dialog and then selected in the **System selection** dialog, the system was not set for the board.
- After inserting a scalable symbol, the 3D symbol defined for it was not taken into account when creating the new device. Instead, the default 3D symbol was added to the device.
- A notification about print mode being active was shown inconsistently during IFC export.
- Quitting the wiring function was slow.

- When wiring was started, for example, from the project tree or group mark, wiring information was added even when **Add wiring information** had not been selected.
- In some cases, there were errors in the suspension rail 2D geometry.
- When editing raceways, the **Connect at the same angle by stretching** and **Connect at the same angle with new segment** functions did not work as should.
- The **Find text or ID** function crashed the program.
- When the cable type was changed for a cable with wiring marking, **Zoom extents** was activated.
- Device integration did not work if there was no 3D symbol in the drawing.
- Saving a copy of the current drawing caused problems in generating 3D raceways.
- Devices were sometimes assigned 3D symbols that were different from those in the product models.
- There were problems in rotating devices' 3D sub-occurrences.
- When coming back to Electrical from CADMATIC HVAC, the system selection dialog was empty and the language was changed.
- Publishing to eShare left the viewport active.

Schematics

New and improved features

Symbolics	The following changes have been made: • The ANSI (American National Standards Institute) standard symbols are now available in the applications. • After the first installation when you first start the application, you are asked to define which symbol standard/locale to use in projects. The symbol menus will then be based on this setting. The selected standard/locale will be used as the default for new projects. You can change it project-specifically in the general symbol settings.
Logos	When you insert a logo in the drawing, the program now checks whether the drawing already has a logo. If the program finds a logo, it asks if you want to replace the existing logo.
Project tree	In the **Find** project tree, context menus are now available showing the same functions as in the device, wiring and product model trees, depending on the selected node. You can, for example, easily insert a device into the drawing from your own selection list.
Handle sheets	The naming of customer drawing target files has been changed: • The file name is no longer automatically modified. If the source and target file names are identical, saving is not done. • The new **Add sheet number range to file name** option added for the **Create an array of sheets** setting adds the sheet number range to the file name.
Copying object data	The settings dialog for copying object data has been renewed. The new **Paste** dialog only shows the data the copied objects actually have. The default settings for the dialog are still defined in the general Electrical settings.
Wiring	The following changes have been made to wiring: • When a cable wire does not have any occurrences but it is connected,

	you can change a wire in the drawing to this cable by right-clicking and selecting **Define a wiring in drawing as this wire** or **Connect and mark cable and its wires**. Wire marking is also added. • With the**Extend wiring** function located in the context menu of an unpaired reference, you can extend wiring or select an existing wire or reference. In addition to another sheet, you can also continue wiring in another open drawing. • The **Connect and mark cable and its wires** function now offers cables based on the **From**/**To** information, which makes it easier to select wires for cable combining. Before, the function did not offer cables already in the drawing, and you had to select them one by one. • The **Insert object info to drawing** dialog now shows group information when you select a cable belonging to a group.
Functions	The following changes have been made to the **Insert object info to drawing** function: • The **Use the previous offset-value for inserting** setting has been added. ◦ The setting enables using the same offset easily in different drawings. ◦ When enabled, you do not need to insert the symbol. Instead, it is automatically inserted according to the saved offset. ◦ If the previous value has not been saved yet, the option is disabled. The offset value is saved after inserting the symbol, and it will then be available the next time. • The **Assign symbols in drawing to this device** function now works for the **Insert object info to drawing** symbols as well.
Single-line diagrams	The tools for creating single-line diagrams are now available in one toolbar, in addition to which new functions for creating symbols have also been added. Furthermore, symbol menus for the most common symbols are also available.
Product models	Importing pins and additional information for product information selected in the product model has been improved.

	• When selecting product information for a product model, you can now decide how to handle the pins and additional information: add missing information only, add missing and overwrite existing, or do not add. • With the **Import product information pins and additional infos** button, you can update pins and additional information for product information selected in the product model without opening the product information dialog.
Terminal block functions	When you select a terminal strip or a terminal block / connector from the drawing, you can insert terminal blocks or connectors from that same terminal strip with the **Insert terminal blocks of this strip in drawing** function.
Device, location and circuit boundaries	The following changes have been made to boundary functions: • You can now select the symbol for the boundary appearance from the Layout menu. • When you select a symbol for the boundary appearance from a menu, the symbol menu toolbar is shown first.
References	You can now connect a wire from cable combining to the **From**/**To** pins in the drawing with the new **Connect wire** function. The function is available in database projects when the wire has no occurrences in any drawings. If necessary, you can remove the connection with the **Clear wire's connection** function.
Plant Modeller integration	The following changes have been made to the integration: • It is now possible to use customer ID when exporting devices. This enables a common ID format between Electrical and the 3D model. The new setting in the mapping dialog affects the IDs for cables, devices, locations and cable targets. Furthermore, publishing to CADMATIC eShare and showing eShare information also follow this setting. • It is now possible to update location coordinates from Plant Modeller to Electrical. The new setting is available in the **Export/synchronize devices** dialog. When the setting is on, the drawings (layout drawings only) are opened and the changes are implemented, after which the

	drawings are left open. The drawings are not automatically saved. • In device import, there is a new column showing the Requires electricity attribute value, based on which the list can be sorted.

Bug fixes

- Flipping the wiring reference arrow did not work for multiple objects. Now, the **Flip wiring reference arrow** function is only available when only one object has been selected.
- If a circuit had more than one drawing template and the templates were generated into one file, generation did not work properly.
- Selecting a wiring reference pair did not connect the wires in a clipboard project.
- The **AutoCAD drawings to Electrical drawing sheets** function asked for the line type name in most drawings.
- When editing certain devices, the program notifies they do not exist.
- When creating wiring references from one drawing to another, the function crashed.
- When generating drawings, identical cable IDs from different templates were not generated correctly into the target project.
- In queue printing and adding revision change arrows, for example, the dialogs opened to the wrong screen. The dialogs now always open to the middle of the Draw dialog, regardless of the screen.
- Location was mistaken for electrical position when it had been set that the prefix is not shown with locations and the **Wrap Full IDS to multiple lines** option had not been selected (the location was only marked to the E_ELPOSID attribute).
- Copying subdevices inside a device boundary resulted in symbol duplicates.
- Device ID was added to device boundary as a common attribute.
- Changes to information added via the **Insert object info to drawing** dialog were not updated in the drawing.
- When editing a symbol added with the **Insert object info to drawing** function, your own additional information was not shown.
- The formatting defined in the **Insert object info to drawing** dialog did not work for location.
- In a cable schema, location was divided to multiple overlapping parts instead of one boundary.
- Cable combing references were generated randomly when it was not clear whether the cable combining belonged to the start or the end of the cable.

- Data for the new I/O card end symbols was inadequate.
- Setting the channel number did not work in Excel generation.
- Symbol selection in product model prompted an error when selecting a Schema symbol.
- Several functions were missing from the context menus in different grids.
- When several users synced changes and saved, an error was prompted.
- Starting modular generation without a project prompted the CADS.Common error.
- When adding a drawing to a project, project settings were only saved to the 0 project.
- Device symbols were created to the database even when there was no device occurrence in the current document.
- Symbols (e.g. SPKIO049 and SPKIO050) were missing from the *Symbolit.vlb* file.
- One of the contacts for the switch SPKYT01216 was faulty.
- The **Represents multiple device parts** symbol type was set based on the main type, even when the symbol had the **Represents a single part of a device** subtype.
- The **Change drawing frame** function did not show all drawing frames.
- The *PurgeDeviceSymbols* command did not work as should.
- When copying a symbol without changing the ID and then using the **Assign symbols in drawing to this device** function, the original symbol referenced the copy.
- Cable combing did not create connection reference if only one wire was selected.
- Editing wire reference properties crashed the application.
- Product information disappeared when device boundary was bound to another location.
- Syncing changes from the database to the drawing sometimes resulted in an error.

Electrical DB

New and improved features

Symbolics	After the first installation when you first start the application, you are asked to define which symbol standard to use in projects. This affects the kind of symbol menus you will have. The selected standard will be used as the default for new projects. You can change it project-specifically in the general symbol settings.
Viewing, selecting	Before, right-clicking a column header automatically selected the whole

and project information	column. Now, the selected cell or row stays selected and you can then choose to select the whole column with the new **Select column** function.
Copying object data	The following changes have been made to copying object data: • The functions for copying and pasting object data have been renamed: ◦ **Copy related data** has been changed to **Copy object data**. ◦ **Paste related data** has been changed to **Paste object data** • The settings dialog for copying object data has been renewed: ◦ The new **Paste** dialog only shows the data the copied objects actually have. ◦ Sub-objects are copied to the target object according to the selections made in the dialog. Product model information will also be copied. ◦ If the target object already has a product model, it is changed to the source object product model by default. If the product model has not been set to update automatically, a dialog where you can select the data to be copied opens.
Devices	The **Product model description** and **Directory (Project tree)** columns have been added to the **Devices** grid.
Reports	Opening the report tool is now faster.
Clipboard project ELEC-2457	The following changes have been made to the clipboard project: • If the wire type is something else than cable wire, import mode is always set to **New**. • If the wire type is internal wire, the value in the **Number** field is automatically increased, so that the number is always unique in the location. The changes also apply to modular generation and Excel import.
Import	Hiding default columns for an import definition is now saved along with the definition, i.e. the hidden columns are now kept hidden until you decide to show them again.

Modular generation	The following changes have been made to modular generation: • Modular generation uses the previously used module list by default, if the module root folder is the same (unless the list has already been selected, for example, via the command line). • The **DrawingFile** column is highlighted in yellow if the file already exists. • If **Merge schemas** is selected and target files already exist, a warning about the merge is shown after clicking **Generate**. • Block and strip modules are now supported. ◦ A module can be smaller than one sheet. In practice, this means that different variations can be created with less modules. ◦ The strips can be generated in a schema either vertically or horizontally, and the size can be defined separately for each strip. ◦ If the **X-Offset** or **Y-Offset** column has a value in the Excel file, all the objects in the drawing are moved according to that value. ◦ If the **Sheet** column has a value in the Excel file, the SLEHTI layer will be changed according to that value. Automatic sheet selection is then ignored in the merge as well, and the drawing is imported from the module as it is without changing the SLEHTI layers.
I/O functions	I/O functions have been improved as follows: • In the **I/O card** field in the **I/O properties** dialog, you can now see how many channels there are and how many of them are in use. In addition, the icon shows whether the cards have free or broken connections. See Editing I/Os. • In the **I/O** grid, the **Channel number** column shows the channels available for an I/O card and their connection status. The column also shows, if several I/Os have been connected to the same channel. See Connecting I/Os with channels and disconnecting them. • The following I/O attributes are now available for drawings:

	◦ **Field device part** – E_IOFIELDPART ◦ **I/O signal range** – E_IOSIGNALRANGE ◦ **Security group** – E_IOSECURITYGROUP ◦ **Voltage** – E_VOLTAGE ◦ **I/O polarity** – E_IOPOLARITY ◦ **Comment** – E_COMMENT ◦ **I/O Circuit code**, **I/O Circuit number**, **Customer circuit name** – E_IOCIRCUIT, E_IOCIRCUITNUMBER, E_IOCUSTCIRCUIT ◦ **I/O Circuit's fuse's ID** – E_IOFUSEID, E_IOFUSEFULLID, E_ IOFUSECUSTID • The following I/O card attributes are now available for drawings: ◦ **Peripheral Interface Controller number** – E_IOCARDPIC ◦ **Field Bus Controller Number** – E_IOCARDFBC ◦ **I/O station** – E_IOSTATION ◦ **Subslot** – E_IOSUBSLOT ◦ **Subslot 2**– E_IOSUBSLOT2 ◦ **Cross connection board ID** – E_CCID, E_CCFULLID, E_CCCUSTID, E_CCLOCATIONID, E_CCCUSTLOCATION ◦ **Cross connection board 2 ID** – E_CC2ID, E_CC2FULLID, E_ CC2CUSTID, E_CC2LOCATIONID, E_CC2CUSTLOCATION
SQL Server project management	The following changes have been made to SQL Server project management: • The filtering function has been added to all the tabs in the project management. Filtering is applied to all the columns with values and visible in the grid. • Project information and additional information are now shown in the grid, and they can be edited. The red highlight indicates that the database connection was not made. • In the dynamic project tree, you can select how to group data. You can also save the tree. • Access projects can use SQL Server's shared databases. The feature can be taken into by selecting the new **Default shared databases** setting.

	• Database settings now include the **Product version** field which prevents using the wrong database when multiple versions are in use.

Bug fixes

- Entering a value for additional information originating from product information resulted in an error.
- Loading database rows to the report function took a long time when using your own report settings.
- In addition to wire occurrences, cable occurrences and cable's other wire occurrences were also included in the **Cable connections list** and **Cable connections list by locations** reports.
- Copying a cable in the shared database resulted in an error.
- Changing the device product model did not update the device pin information.
- Location occurrences were not included in the part lists.
- The **Item** field was filled incorrectly in the part list.
- In the part list, only the product model manufacturer was shown while the product information manufacturer was missing.
- When selecting a product model for a device, the program added **Electric** as the default design area value, even if that field was empty in the product model.
- The length and width values were mixed up in Excel import.
- Updating the project database timed out.
- When importing product information from Excel, the program crashed.
- The program crashed when copying and pasting object information if location, for example, had been selected from the project tree.
- After opening a newly created piece of additional information for editing, clicking **Cancel** removed it.
- It was not possible to add imported items to user's selection list, if the option **Imported** was selected from the project tree.
- When editing an I/O in the **I/O properties** dialog, the channel number value was not updated after changing the I/O card to one that did not have the connected channel.
- Changing the station ID for an I/O card caused an error when the channel was incorrectly connected.
- Exporting to a temporary project failed when project name was empty or the name had a space at the end.

- Reading the ETIM database occasionally failed.
- The context menu for the column header row showed functions that should not have been shown.
- In modular generation, the sheets were on top of each other in some cases.
- When sheet numbers were set in Excel, modular generation did not set them in drawings or the database.
- When updating the database, the connection string was not set if it was pasted.
- Modular generation failed if the user name consisted of more than eight characters.
- Saving grid column width did not work.
- When syncing was not done, drawing a cable schema resulted in duplicate cables.
- Adding cable types from the project to the shared database resulted in an error.
- In the SQL database, new rows with project number 0 were generated in the **Devices** table.
- Entering a customized database description creates a connection string in *EDBUserCommon.mdb*. If necessary, the field can be cleared in the product database settings by right-clicking and selecting **Clear connection string**.
- In the DB settings, columns were missing from the **Product information visibility** list.
- Functions were missing from the I/O context menu.
- Adding and deleting product information for cables sometimes resulted in errors.
- Additional information was not shown in the **Devices** and **Additional information** grids, in addition to which information coming from a product model was not highlighted in blue.
- When importing from Excel, the **Start address IN**, **Start address OUT** and **Bit address offset** values were not imported to I/O cards.
- In modular generation, wiring references did not work if the module had no database.
- Customer IDs were not shown in the **Plates** grid, if dynamic customer ids were used.
- When updating device information, attribute stacking did not work with device boundary.
- Copying a cable type with additional information prompted a confirmation request about replacing the additional information definition.
- In some grids, the rows were sorted incorrectly in alphabetical order.
- Pressing Esc after starting the import resulted in an error.
- Linking IO to channels did not update symbols in the drawing until reopened.
- Adding additional information to product information resulted in an error.
- Copying a cable type from one project to another removed additional information from the cable type.

- When product information was replaced, the information was not updated to the drawing or the database.
- In the **I/O cards** grid, the **Import rows from Excel** function was missing from the context menu.
- Copying cells to the grids via the clipboard sometimes resulted in errors.
- Importing a clipboard project sometimes resulted in errors.
- Opening, for example, a cable type definition from another project showed empty fields and wrong information.
- Reimporting I/O cards with a different product model did not update channel addresses if the rule changed.
- In wiring set management, when you first removed a wiring set used in the project and then added it again, the wiring set was saved in the database but was not shown in the user interface.
- When updating additional information for a device, the value did not always update if the same value had already been entered to that field earlier.
- When a device was created based on a product model and the symbol for that product model was changed without changing the tag, device properties showed the old icon.
- When the parent was something else than device or product model, the cleanup function removed all rows from the **Symbols** table.
- When addresses were created for Siemens' I/O cards based on the address creation rules, bit addresses were erroneously added to analog cards.
- In the **Symbols** table, some plates had multiple symbols.
- Copying pin data from product information to product information lost them from the source.
- When adding a cable ID in the **Cables** grid, the application did not check whether the ID was unique.