

E³. series Releasenotes

Date: 6/26/2020

This manual describes the following functions of version:

2020 Build 21.00

!! Please Note when Switching to New Version !!

Projects and databases are converted when opening them with this new **E³.series** Release. Please consider that these projects cannot be edited with an older version afterward. That's why we recommend to backup projects before.

Note:

- The latest version of the license server must be installed when using a FlexNet license server. In particular, the provided new ECADLM.EXE must be used.
- Starting with Microsoft Office Version 2016, the TrueType font **'Arial Unicode MS'** is no longer included in the standard delivery of Microsoft Office due to financial reasons.

This font has been and is still in use for **E³.series**, **E³.** template projects and **E³.** example projects. You have most likely used this font to documents for **E³.** projects. Presently, Zuken is not aware of any comparable functionality that is free-of-charge for **'Arial Unicode MS'**.

To ensure that your new and older projects and the documents generated from them are displayed correctly on devices without older MS Office versions, we recommend you purchase and install the original font **'Arial Unicode MS Regular'** on your devices. Furthermore, we recommend you change your templates to another font style so that new projects no longer use this font.

- On Windows7 32 bit and Windows 10 32 bit, this E3 Version can be used with normal configuration (2GB user storage) or extended configuration (3GB user storage). In the case of the first configuration, the maximum project size is restricted to approx. 250MB. The extended configuration enables 450MB - 470MB.

For 3GB configuration, run the following command as Administrator:

```
bcdedit /set IncreaseUserVa 3072
```

Restart the computer afterwards.

To remove the extended configuration, run the following command as Administrator:

```
bcdedit /deletevalue IncreaseUserVa
```

Restart the computer afterwards.

- **New Features**
-

See also:

Update Information of Version 2020

- [Windows Support](#)
 - [Installation](#)
 - [Databases](#)
 - [Projects](#)
 - [Multi-User Installation](#)
 - [ORACLE](#)
 - [Microsoft SQL Server](#)
-

Windows Support

The following Windows systems are supported with **E³.series** 2020:

- Windows 7
- Windows 8
- Windows 10
- Windows Server 2008/2012/2016

For more information please refer to the installation description.

Installation

Before installing, we recommend you make a backup of your current installation (at least the data and databases).

The installation of **E³.series** 2020 is started using

SETUP.EXE

which can be found on the DVD's main directory.

The Version can be installed in addition to an existing **E³.series** Version.

The Default Directory is: `\Program Files\Zuken\E3.series_<version>`

Therefore, this version can also be used alongside existing versions of **E³.series**. However, note that you cannot use your existing database(s), or the existing projects, in both versions, as they are updated when opened in **E³.series** 2020, so copies should be used for **E³.series** 2020.

Databases

With this new version, a new default database is installed. Existing databases are not overwritten.

The new database contains new components and symbols.

Projects

Existing projects are automatically converted upon opening with the new version. So long as the file has not been saved in the new version, it can still be opened with an older version. After saving a project with **E³.series** Version 2020, a project can no longer be opened with an earlier version.

Multi-User Installation

Existing multi-user projects from Version 2019 can directly be opened with Version 2020. They are automatically converted upon opening. However, this cannot be undone, and the projects can no longer be opened with Version 2019.

That's why we recommend installing Version 2020 multi-user projects in parallel to the existing version, so that it's also possible to still access these project with **E³.series** Version 2019.

To transfer the projects to Version 2020, these projects must be saved in Version 2019 as Single-User projects. They can then be stored in the multi-user environment of Version 2020.

In order to run the Multi-User databases in parallel

- a new SQL Server process must be installed for Microsoft SQL Server or another server must be installed
- a new database user account for ORACLE must be used

The E3 Server process must then access the new database or the new database user account.

ORACLE

When setting up the E3-MU-Server, a new name must be specified for the Multi-User Administrator.

Thus a new database user account is created and both **E³.series** versions can work simultaneously in the multi-user environment.

Microsoft SQL Server

First, a new instance of the SQL Server processes must be installed.

Enter a new instance name, which must be used with the set-up installation of the E3-MU-Server.

The rest of the installation continues as described in the separate **MuSetup** installation description.

New Features 2020 Build 21.00

The following functionality is new in Version 2020 Build 21.00:

- **COM**

- **COM - Miscellaneous**

- **Behavior of COM Commands Changed in SheetInterface for Multiuser**
 - **Enhancement to SheetInterface for Setting Sheets as 'Read-only'**
 - **Enhancement to OutlineInterface for Determining whether a CutOut or Drill Hole from E3.CutOut Shall Be Considered**
 - **Enhancement to SymbolInterface, SheetInterface and JobInterface for Editing a Selection for Placed Objects**
 - **Enhancement to DeviceInterface, PinInterface and BundleInterface for Determining the Uppermost Overbraid Object**
 - **Enhancement to SymbolInterface for Assigning Components to Plugged (Block) Connector Symbols (also Shared PinGroups)**
 - **Enhancement to MultiuserCOMClient for Reading Users, Groups and Authorizations for Multiusers**

- **Importing and Exporting Data**

- **Embed Files in Circuit Diagrams via OLE**
 - **Export 3D Views to PDF**
 - **Export Component Tables as CSV Files**
 - **Transfer Levels of Symbol Graphics when Exporting as DXF**
 - **Generate PDF in Gray Scale**

- **Database Editor**

- **Create 2D Views of Model Areas**
 - **Component Wizard: Selection Lists for Attributes based on Existing Values**
 - **Change Component Type of Existing Block Connector**
 - **New Text Type *Device letter code***
 - **Sort Columns in the *Edit Text Types* Table**
 - **Translation Database: Mark Entries as Outdated**
 - **Translation Database: Filter by Outdated Entries**

- **Placement**

- **Control whether Imported Hierarchical Blocks Shall Be Merged with Existing Hierarchical Blocks or Be Newly Generated**

- **Project Handling**

- **3D Preview Window**
- **Delete Attributes No Longer Being Used Automatically**
- **Display Device Properties Automatically**
- **Consider Drill Holes for Devices in *E³.CutOut***
- **Unique Option Names independent of Folder Structure**
- **Messages associated with *Jump to Command* Highlighted**
- **MIL-Standard: Display Break-Up Edges Above and Below Connectors**
- **New Text Types Filled In Automatically**
- **Define Dynamic Restricted Areas and Cutouts in Projects**
- **Show Differences when Updating Components/Symbols**
- ***Jump to 2D View* if 2D View of Model Created**

- **Tables**

- **New Information in the *Connection Table***
- **New Column *Placement* in the *Pin Table***
- **Fit Column Width/Line Height for Multiline Texts**
- **Search using Boolean Expressions**

COM-Interface - Overview

The following functionality is now available in the COM Interface and described in these chapters:

- **COM-Interface - Miscellaneous**
 - **Behavior of COM Commands Changed in SheetInterface for Multiuser**
 - **Enhancement to SheetInterface for Setting Sheets as 'Read-only'**
 - **Enhancement to OutlineInterface for Determining whether a CutOut or Drill Hole from E3.CutOut Shall Be Considered**
 - **Enhancement to SymbolInterface, SheetInterface and JobInterface for Editing a Selection for Placed Objects**
 - **Enhancement to DeviceInterface, PinInterface and BundleInterface for Determining the Uppermost Overbraid Object**
 - **Enhancement to SymbolInterface for Assigning Components to Plugged (Block) Connector Symbols (also Shared PinGroups)**
 - **Enhancement to MultiuserCOMClient for Reading Users, Groups and Authorizations for Multiusers**

COM-Interface - Miscellaneous

- **Behavior of COM Commands Changed in SheetInterface for Multiuser**
- **Enhancement to SheetInterface for Setting Sheets as 'Read-only'**
- **Enhancement to OutlineInterface for Determining whether a CutOut or Drill Hole from E3.CutOut Shall Be Considered**
- **Enhancement to SymbolInterface, SheetInterface and JobInterface for Editing a Selection for Placed Objects**
- **Enhancement to DeviceInterface, PinInterface and BundleInterface for Determining the Upper-most Overbraid Object**
- **Enhancement to SymbolInterface for Assigning Components to Plugged (Block) Connector Symbols (also Shared PinGroups)**
- **Enhancement to MultiuserCOMClient for Reading Users, Groups and Authorizations for Multiusers**

Behavior of COM Commands Changed in SheetInterface for Multiuser 01 - Docu-00861

The following COM commands in the **SheetInterface** are affected and should no longer be used.

For compatibility reasons they are still available but their behavior has changed:

- **ret = sht.CheckOut ([lock [, sheetids]])**
 - **lock 1** locks sheet exclusively. This means the sheet can only be opened read-only by another client ("simulates" old behavior from E3.2015 and earlier)
- **ret = sht.IsOffline**
 - **ret 1** NOT exclusively locked

Reference(s): Designer-31487

Enhancement to SheetInterface for Setting Sheets as 'Read-only' 02 - Cat-01092

The following new COM commands are in the **SheetInterface**:

- **ret = sht.IsReadOnlyInProject**
 - **ret** The return value is 1 if the sheet is set as 'read-only' project-wide. Otherwise, the value is 0.
- **ret = sht.SetReadOnlyInProject (readonlyproject)**
 - **readonlyproject** The following values are possible:
 - **1** 'Read-only' activated!
 - **0** 'Read-only' deactivated!

- **ret** The following values are possible:
 - **1** The setting 'read-only' could not be changed!
 - **-1** No project or sheet set!
 - **-2** Sheet is locked!

Reference(s): Designer-34550

Enhancement to OutlineInterface for Determining whether a CutOut or Drill Hole from E3.CutOut Shall Be Considered 03 - Cat-01276

The following new COM command is in the **OutlineInterface**:

- **ret = outline.SetUseInE3CutOut (set)**
 - **set** Activates or deactivates the setting from **E3.CutOut**:
 - **0** The cutout or drill hole is not considered!
 - **1** The cutout or drill hole is considered!
 - **ret** The return value is 1 if successful. In case of an error, the following values are possible:
 - **-1** Doesn't exist in pool!
 - **-2** ID not set!
 - **-3** No outline object!
 - **-4** Outline is not the type 'CutOut'!

Reference(s): Designer-27456

Enhancement to SymbolInterface, SheetInterface and JobInterface for Editing a Selection for Placed Objects 04 - Cat-00951The

The following new COM command is in the **SymbolInterface**:

- **ret = sym.SetSelected (newval)**
 - Sets or removes a selection of symbols
 - **newval** The following values are possible:
 - **1** Selects the symbol if no symbol selected, or adds the selection if the previous selection is on the same sheet.

- **0** Removes the symbol from the selection.
- **ret** The return values are:
 - **1** Target state has been achieved (symbol not selected when newval = False, or selected, when newval = True)
 - **-1** No project opened!
 - **-2** No symbol ID set!
 - **-3** Symbol cannot be selected!
 - **-4** Selection exists on another sheet!
 - **-5** Symbol not currently displayed due to options/variants!

The following new COM command is in the **SheetInterface**:

- **ret = sht.SelectIds (symbolIds, selectedIds)**

- Selects the given symbols on this sheet and deletes the previous selection.
- **symbolIds** Array of symbol ids to select
- **selectedIds** If set, returns list of currently selected symbols.
- **ret** The return value is **1** if successful. If an error occurs, the following values are possible:
 - **-1** No pool available!
 - **-2 symbolIds** not set or empty!
 - **-3** Previous selection could not be cleared!
 - **-4** Could not select any of the given symbols!
 - **-5** Could only select some of the given symbols!

The following new COM command is in the **JobInterface**:

- **ret = job.ResetSelection**

- Deletes the entire selection in the project.
- **ret** The return value is **1** if successful, otherwise **-1**.

Reference(s): Designer-09714

Enhancement to DeviceInterface, PinInterface and BundleInterface for Determining the Uppermost Overbraid Object 05 - Cat-00517

The following new COM command is in the **DeviceInterface**, **PinInterface** and **BundleInterface**:

- **ret = xx.GetRootOverbraidId**

- **ret** The return value is the **Id** of the overbraid, even if it has no component code. In case of an error, the following values are possible:

- **-1** No project opened or no ID set!
- **-2** The set ID is not an element of the overbraid!
- **-3** An item with an invalid item type was transferred!
- **-4** (Only DeviceInterface) of the transferred item is the root overbraid!

Reference(s): Designer-34048

Enhancement to SymbolInterface for Assigning Components to Plugged (Block) Connector Symbols (also Shared PinGroups) 06 - Cat-00838

The following new COM command is in the **SymbolInterface**:

- **ret = sym.AssignToConnector (AssignToPinIds, AssignToPinIds_connected, flags)**

- **Prerequisite:** A connector or block connector symbol must be set!
- If both lists **AssignToPinIds** and **AssignToPinIds_connected** are empty, all existing pins are deassigned and the return value is 1.
- **AssignToPinIds** It tries to assign these connector pins to the symbol. If pins are already assigned to the symbol, these are deassigned so that only the transferred pins are assigned to the symbol.

The pins must exist and fit the symbol:

- for a connector symbol only connector pins
- for a block connector symbol only block connector pins of the same block
- all pins must be of the same view level
- **AssignToPinIds_connected:** If the symbol is already plugged, these pins are assigned. The same rules then apply. The mode set by `job.SetGetterOptionHandlingMode` is taken into account when determining the attached symbol for multiple insertion using different variants/options. The assigned pins are also plugged together according to the same rules as if the plug connection were made interactively by plugging in the symbols. If this is not possible (e.g. due to incompatible signals or other already existing plug connections), the whole action is rejected.
- **flags** Not currently used!
- **ret** The return value is **1** if successful or if both lists are empty. In case of an error, the project is not changed. The following values are possible:
 - **-1** No project opened!
 - **-2** No ID set!
 - **-3** The pins could not be assigned to the symbol!
 - **-4** The existing plug connection is not unique (variants/options)!
 - **-5** The pins assigned to the plugged symbols could not be plugged together!
 - **-6** Internal error!

Enhancement to MultiuserCOMClient for Reading Users, Groups and Authorizations for Multiusers 07 - Cat-01719

The following new COM command is in the **MultiuserCOMClient**:

- **ret = client.GetUsers(users)**

- **users** Array of user and/or group names
- **ret** Returns the number of names if successful, otherwise the following values are possible:
 - **-1** No connection to the server
 - **-2** An internal error has occurred!

- **ret = client.GetUsers(user)**

- **user** Name of user or group
- **ret** The following values are possible:
 - **if successful** returns the set of permissions of the **user** represented by a combination of these values:
 - **0** No permissions
 - **1** Create/Delete
 - **2** Export/Import
 - **4** Stop server process
 - **8** Transfer files
 - **16** AdminDictionary
 - **if not successful** returns one of the following error codes:
 - **-1** No connection to the server
 - **-2** An internal error has occurred
 - **-3 user** is not known
 - **-7** An error occurred while reading the data

- **ret = client.GetProjectPermissions(project, user)**

- **project** Name of the project
- **user** Name of user or group
- **ret** The following values are possible:

- **if successful** returns the set of permissions of the **user** represented by a combination of these values:
 - **0** No Permissions
 - **1** Modify
 - **2** View

- **if not successful** returns one of the following error codes:
 - **-1** No connection to the server!
 - **-2** An internal error has occurred!
 - **-3** User not known!
 - **-5** Project not known!
 - **-7** An error occurred while reading the data!

Reference(s): Designer-32385

Title: Importing and Exporting Data

Import Data

- [Embed Files in Circuit Diagrams via OLE](#)

Export Data

- [Export 3D Views to PDF](#)
- [Export Component Tables as CSV Files](#)
- [Transfer Levels of Symbol Graphics when Exporting as DXF](#)
- [Generate PDF in Gray Scale](#)

Embed Files in Circuit Diagrams via OLE

Files supported by OLE (**Object Linking and Embedding**) can be embedded in circuit diagrams.

This enables technical descriptions, tables from external sources, or similar to be displayed directly on the circuit diagram. OLE supports, among others, Windows application formats, AutoCAD files or common media formats.

To embed a file in a circuit diagram using OLE, select the command **Insert -> Object...** in the *Main Menu*. A dialog window with the following options opens:

- With **Create new** a new object can be created.
- With **Create from file** an existing file can be embedded.

Files that are embedded via OLE are displayed on the sheet in the form of a preview window. You can open, move, copy and paste embedded files directly to other parts of the project.

Export 3D Views to PDF

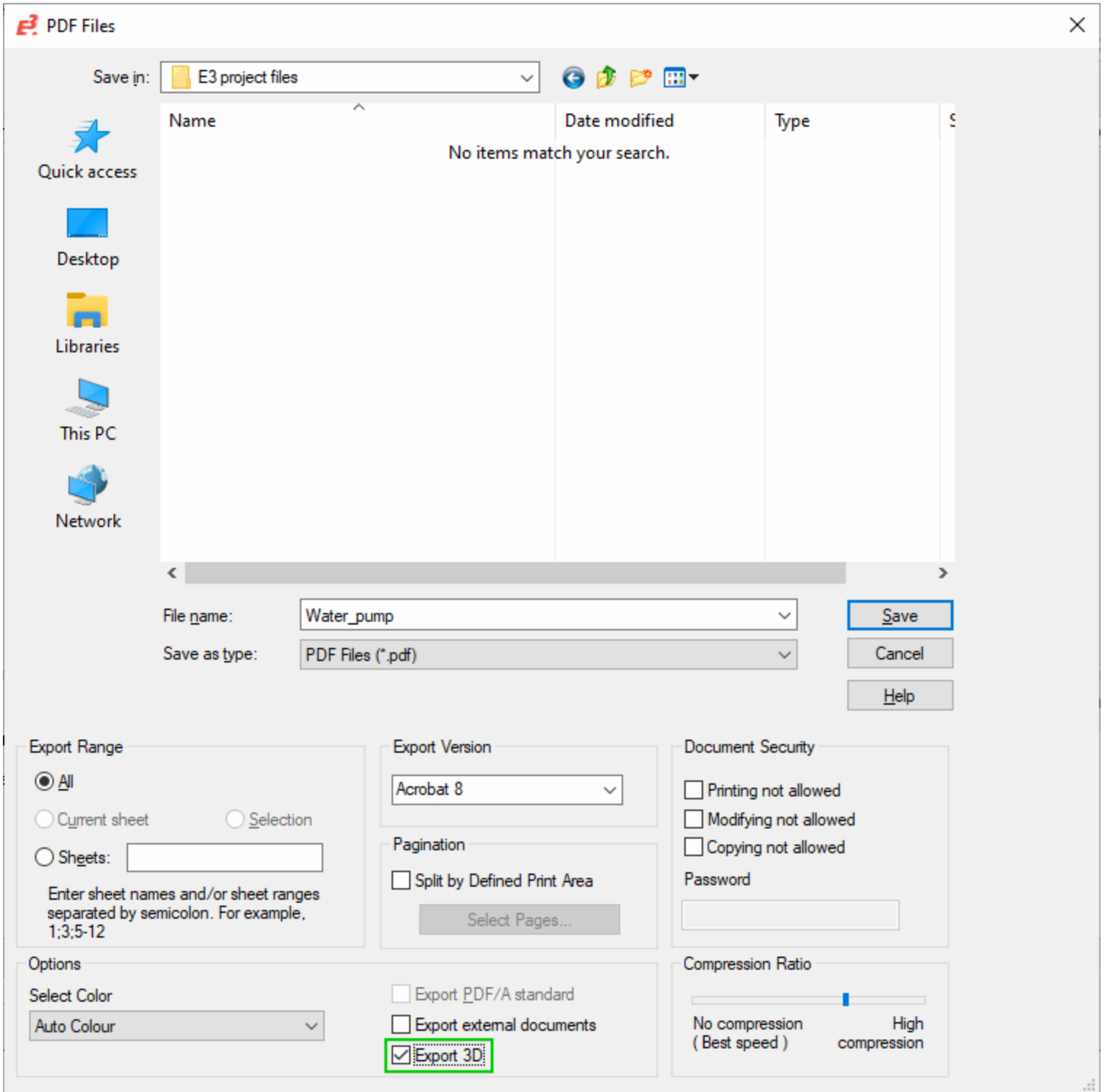
Prerequisite: To display objects correctly in PDF, the user requires a PDF program that can display 3D models.

Objects that are available as 3D models in **E³.series** can also be displayed in PDF as three-dimensional objects.

These objects can be rotated in PDF and viewed from all sides. For example, this allows complex structures to be better illustrated.

To export a 3D view as PDF, select the command **File -> Export -> PDF...** in the *Main Menu*. The PDF export dialog window opens.

Select **Export 3D**:



Export Component Tables as CSV Files

Component tables can be exported as CSV files.

For example, the CSV file can be edited or used for an additional order process.

To export a component table as a CSV file, first open a **Component Table**. Select the command **View - > Database Windows -> Component Table** in the *Main Menu*. The **Component Table** opens.

Right-click in the upper left cell of the **Component Table** and select **Export as CSV...**:

The screenshot shows the 'Component Table' window with a table of components. A context menu is open over the first row, with 'Export as CSV...' selected. The table has the following data:

Component	Version	Version text	Device letter code	Table symbol	Default template	Preview Symbol	Topology symbol	Number pins /
<All>	<All>	<All>	<All>	<All>	<All>	<All>	<All>	<All>
Zuken E3 Gm	X							18
Zuken E3 Gm	V							11
Zuken E3 Gm	V							7
Zuken E3 Gm	R							2

The 'Attributes and preview' section at the bottom is empty, showing a table with columns 'Name' and 'Entry'.

Transfer the Levels of Symbol Graphics when Exporting as DXF

When exporting projects as DXF, the levels defined on symbols can also be transferred.

By additionally exporting the levels of symbol graphics, the levels defined in the **E³** project are retained.

To transfer the levels of symbol graphics when exporting as DXF, a configuration file has to be adapted in the installation directory of **E³**.

Go to the directory, in which you have installed **E³.series**. **Note:** The default directory of **E³.series** is: Windows: **Program Files\Zuken\E³.series_<Version>**

Open the configuration file `DXFExport.CFG` in the installation directory. Enter the following line [Use symbol level] in the configuration file:

Note: If no configuration file exists yet, create a file named `DXFExport.CFG`.

Generate PDF in Gray Scale

Schematics from **E³.series** can be generated in gray scale when exporting as PDF.

Displaying in gray scale enables the user to distinguish differences in color much easier than in black and white.

To generate a PDF in gray scale, select the command **File -> Export -> PDF...** in the *Main Menu*. The PDF export dialog window opens.

Select **Gray Scale**:

Save in: E3 project files



- Quick access
- Desktop
- Libraries
- This PC
- Network

Name	Date modified	Type
No items match your search.		

File name: Water_pump

Save

Save as type: PDF Files (*.pdf)

Cancel

Help

Export Range

- All
- Current sheet
- Selection

Sheets:

Enter sheet names and/or sheet ranges separated by semicolon. For example, 1;3;5-12

Export Version

Acrobat 8

Pagination

Split by Defined Print Area

Select Pages...

Document Security

- Printing not allowed
- Modifying not allowed
- Copying not allowed

Password

Options

Select Color

- Auto Colour
- Auto Colour
- Black and White
- Gray Scale

- Export PDF/A standard
- Export external documents
- Export 3D

Compression Ratio

No compression (Best speed) High compression

Title: Database Editor


- **Create 2D Views of Model Areas**
- **Component Wizard: Selection Lists for Attributes based on Existing Values**
- **Change Component Type of Existing Block Connector**
- **New Text Type *Device letter code***
- **Sort Columns in the *Edit Text Types* Table**
- **Translation Database: Mark Entries as Outdated**
- **Translation Database: Filter by Outdated Entries**

Create 2D Views of Model Areas

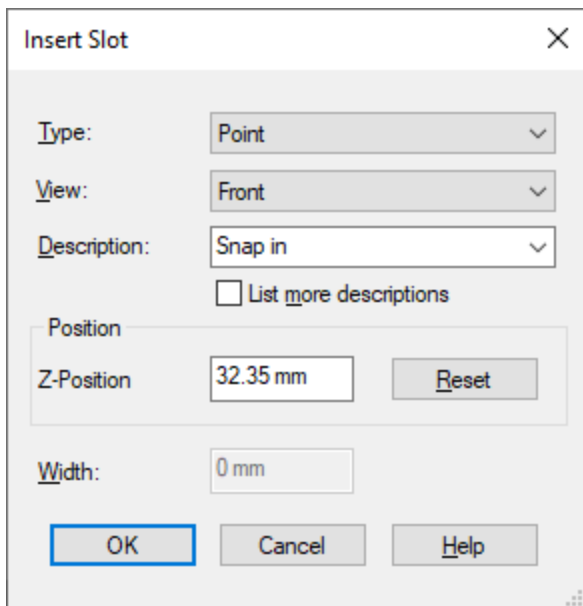
When editing models in the Database Editor, it's possible to create 2D views of different areas.

This allows you, for example, to create all the views you need for mounting plates that can be fitted from both sides.

To create a 2D view of a specific area, select **Tools -> Start Database Editor** in the *Main Menu*. The **Database Editor** opens.

Select a model that requires a 2D view of an area. Insert a slot in the model. To do so, select **Insert -> Slot...** in the *Main Menu* or the  icon in the *Database Model* toolbar. The **Insert Slot** window opens.

Select the corresponding area that you would like to generate a 2D view of under **Display::**



The screenshot shows the "Insert Slot" dialog box with the following settings:

- Type: Point
- View: Front
- Description: Snap in
- List more descriptions
- Position:
 - Z-Position: 32.35 mm (with a Reset button)
 - Width: 0 mm
- Buttons: OK, Cancel, Help

The desired 2D view is created in the top view.

Component Wizard: Selection Lists for Attributes based on Existing Values

When creating new components, additional attributes are available that can be filled with values created in the database. All possible values are listed in selection lists.

This allows for more information to be stored in the database as soon as new components are created and helps avoid typing mistakes by using selection lists.

To start the Component Wizard, right-click on a folder in the **Component** window and select the context menu command **New Component**. The Database Editor opens. If you have not yet started the database editor, it's started automatically.

Select the component you want to create and then fill in the information requested by the Component Wizard. Click **Next**. The **Component Wizard - Properties** are displayed.

A selection list opens in the **Component Wizard - Properties**, in which a list of entries for each attribute appears:

Component Wizard - Properties

Please enter the component-specific attributes.
Select the 'Name' column to add attributes.

Component name: NewComponent1
Version: 1

Name	Entry
Article number	<no entry>
Class	<no entry>
Description	<no entry>
Main Class	<no entry> ... ▼
Supplier	Electric
	Generic_E
	Generic_F
	Hydraulic
	Pneumatic

< Back Next > Finish Cancel

Note: The attributes and values displayed depend on the type of component being created.

Change Component Type of Existing Block Connector

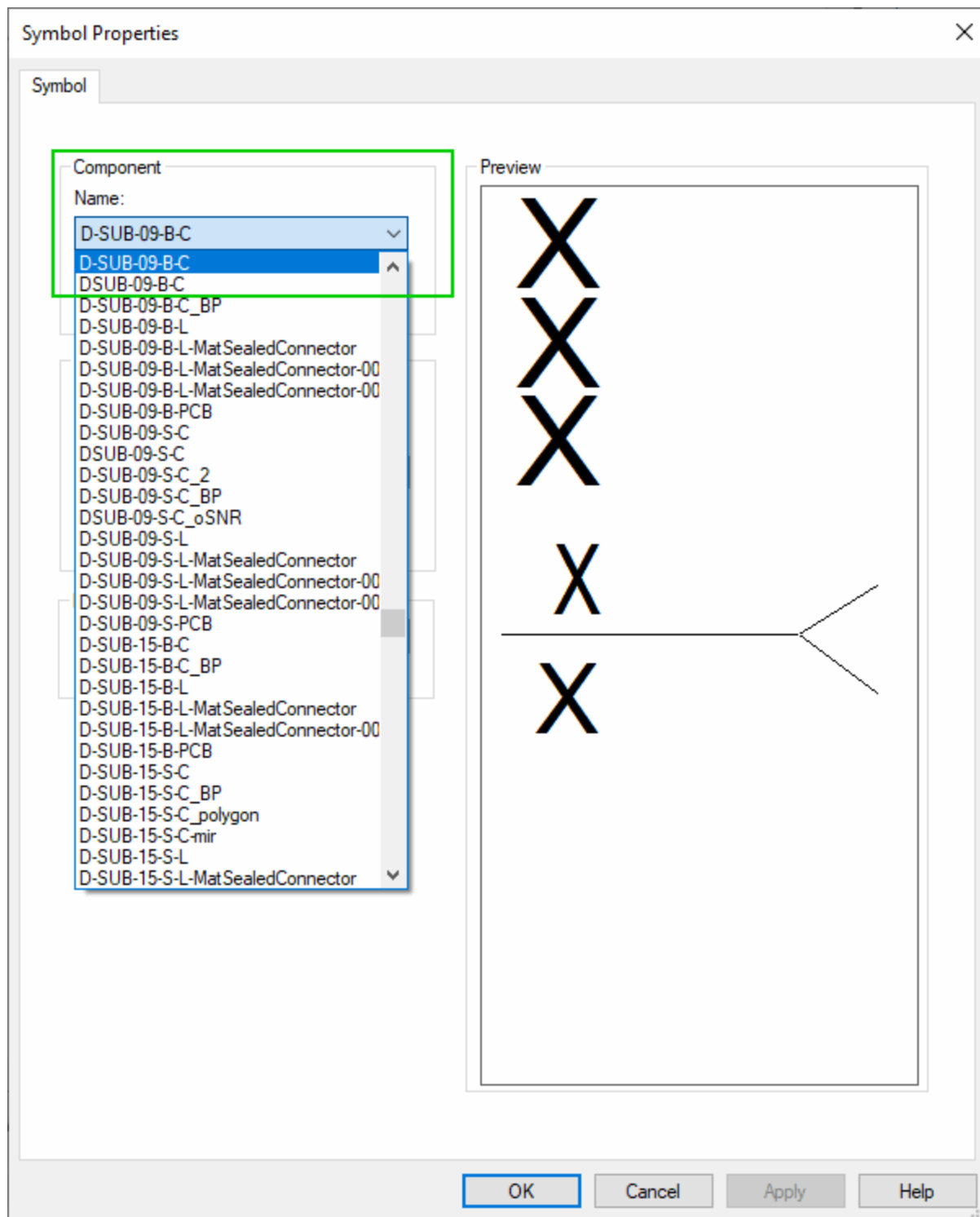
The component type of block connectors can be changed without deleting the existing block connector.

This enables block connectors to be changed directly without having to create a new block connector.

To change an existing block connector, select the command **Tools -> Start Database Editor** in the *Main Menu*. The **Database Editor** opens.

Open a project, in which the block connector is used. Next select the component, whose component type you want to change and then **Format -> Symbol Properties...** in the *Main Menu*. The **Symbol Properties** of the selected component opens.

Open the drop-down menu under **Component -> Name:** and select the new component type:



Sort Columns in the *Edit Text Types* Table

The columns in the *Edit Text Types* table can be sorted.

Sorting, for example, gives the user a better overview of the text types available.

To sort the text types, select the command **Tools -> Start Database Editor** in the *Main Menu*. The **Database Editor** opens.

Now select **Format -> Text Types...** in the *Main Menu*. The *Edit Text Types* table opens.

Click on a desired table column to sort it in ascending order. Click on the table column once again to reverse the sorting order.

New Text Type *Device letter code*

The new text type *Device letter code* is available in the Database Editor.

This text type shows only the device letter code, which can be used as an additional option for filtering.

All devices receive a corresponding device letter code automatically.

Translation Database: Mark Entries as Outdated

Entries in the translation database can be marked as outdated.

Outdated terms in the language database are no longer offered for selection. Outdated entries remain in existing projects but cannot be used in new projects.

To mark an entry as outdated, select **Tools -> Start Database Editor** in the *Main Menu*. The **Database Editor** opens.

Next select **Format -> Translation Table...** in the *Main Menu*. The **Translation Table** opens.

Click on the check box in the column **Outdated** to mark the term as outdated, otherwise the term is current:

Translation Table

	Outdated	Description	<mainClass>	<subClass>	British	American	Cast
⚙	<All>	<All>	<All>	<All>	<All>	<All>	<A
	<input type="checkbox"/>				cover	cover	portadi
	<input type="checkbox"/>				switch off	switch off	apagar
	<input type="checkbox"/>				cover panel	cover panel	portadi
	<input type="checkbox"/>				core	conductor	vena
	<input type="checkbox"/>				Number of co	Number of co	Númer
	<input type="checkbox"/>				alarm	alarm	alarma
	<input type="checkbox"/>				general	general	genera
	<input type="checkbox"/>				plant	plant	instala
	<input type="checkbox"/>				plant operatin	plant operatin	funcion
	<input type="checkbox"/>				plant designat	plant designat	design
	<input type="checkbox"/>				plant overvie	plant overvie	resum
	<input type="checkbox"/>				connection	connection	conexi
	<input type="checkbox"/>				No.	No.	No.
	<input type="checkbox"/>				article design	article design	design
	<input type="checkbox"/>				Article Numbe	Article Numbe	Númer
	<input type="checkbox"/>				divided into	divided into	dividid
	<input type="checkbox"/>				off	off	sin cor

OK Cancel Help

Translation Database: Filter by Outdated Entries

It's now possible to filter by outdated entries in the translation database.


This enables users to have an overview of all outdated terms or they can create a summary of currently valid entries.

To do so, select **Tools -> Start Database Editor** in the *Main Menu*. The **Database Editor** opens.

Next select **Format -> Translation Table...** in the *Main Menu*. The **Translation Table** opens.

- Enter "1" in the second line of the column **outdated** to display all outdated terms.
- Enter "0" in the second line of the column **outdated** to filter out all outdated terms.
- Enter "*" in the second line of the column **outdated** to reset the filter.

Translation Table ✕

	Outdated	Description	<mainClass>	<subClass>	British	American	Castellano
	1	<All>	<All>	<All>	<All>	<All>	<All>
	<input checked="" type="checkbox"/>				cover	cover	portada
	<input checked="" type="checkbox"/>				switch off	switch off	apagar
	<input checked="" type="checkbox"/>				cover panel	cover panel	portada de
	<input type="checkbox"/>						

<
>

OK
Cancel
Help

Placement / Purge

- **Control whether Imported Hierarchical Blocks Shall Be Merged with Existing Hierarchical Blocks or Be Newly Generated**

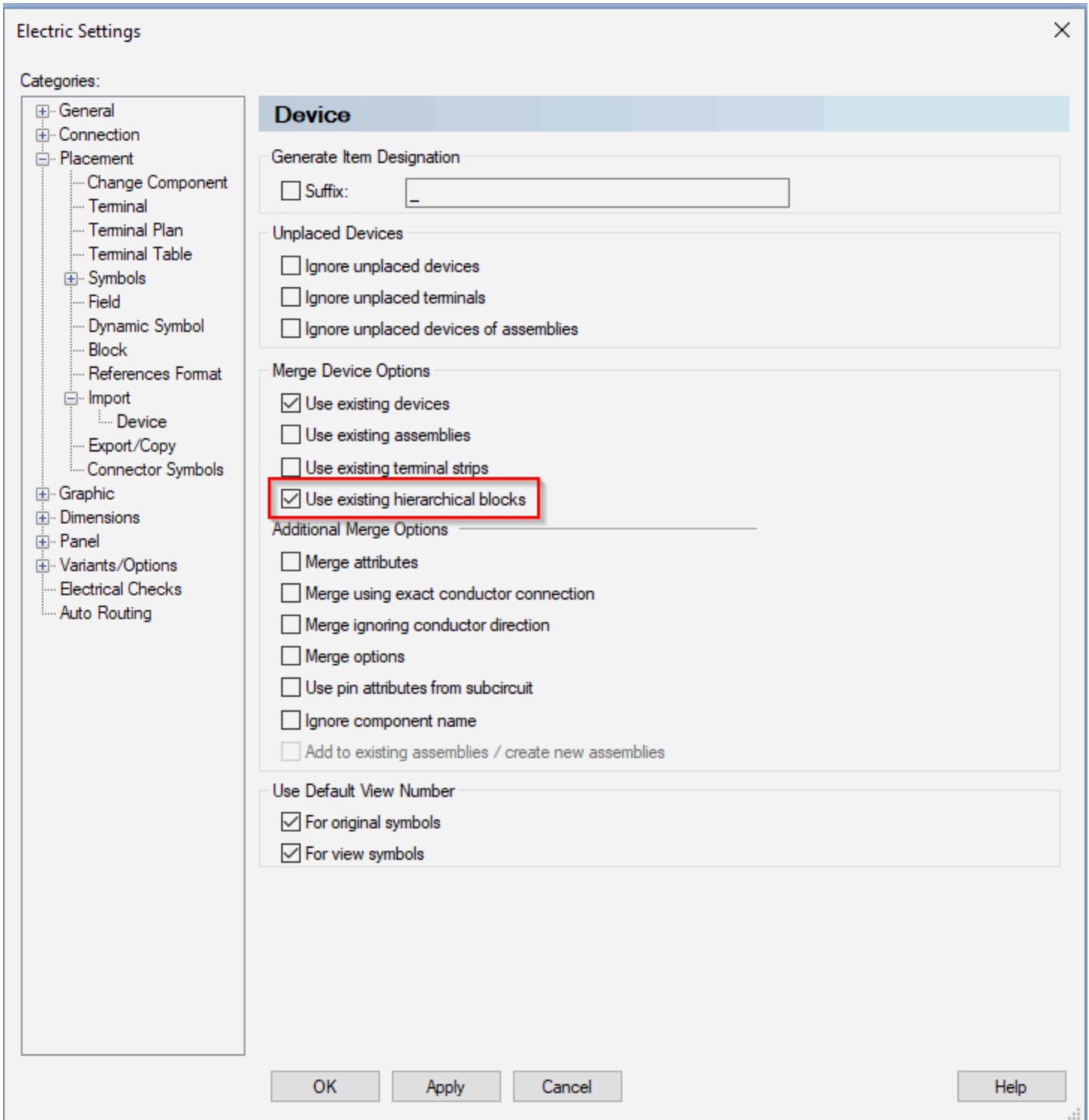
Control whether Imported Hierarchical Blocks Shall Be Merged with Existing Hierarchical Blocks or Be Newly Generated

Previously when importing hierarchical blocks in **E³.series** it wasn't possible to merge them with existing blocks.

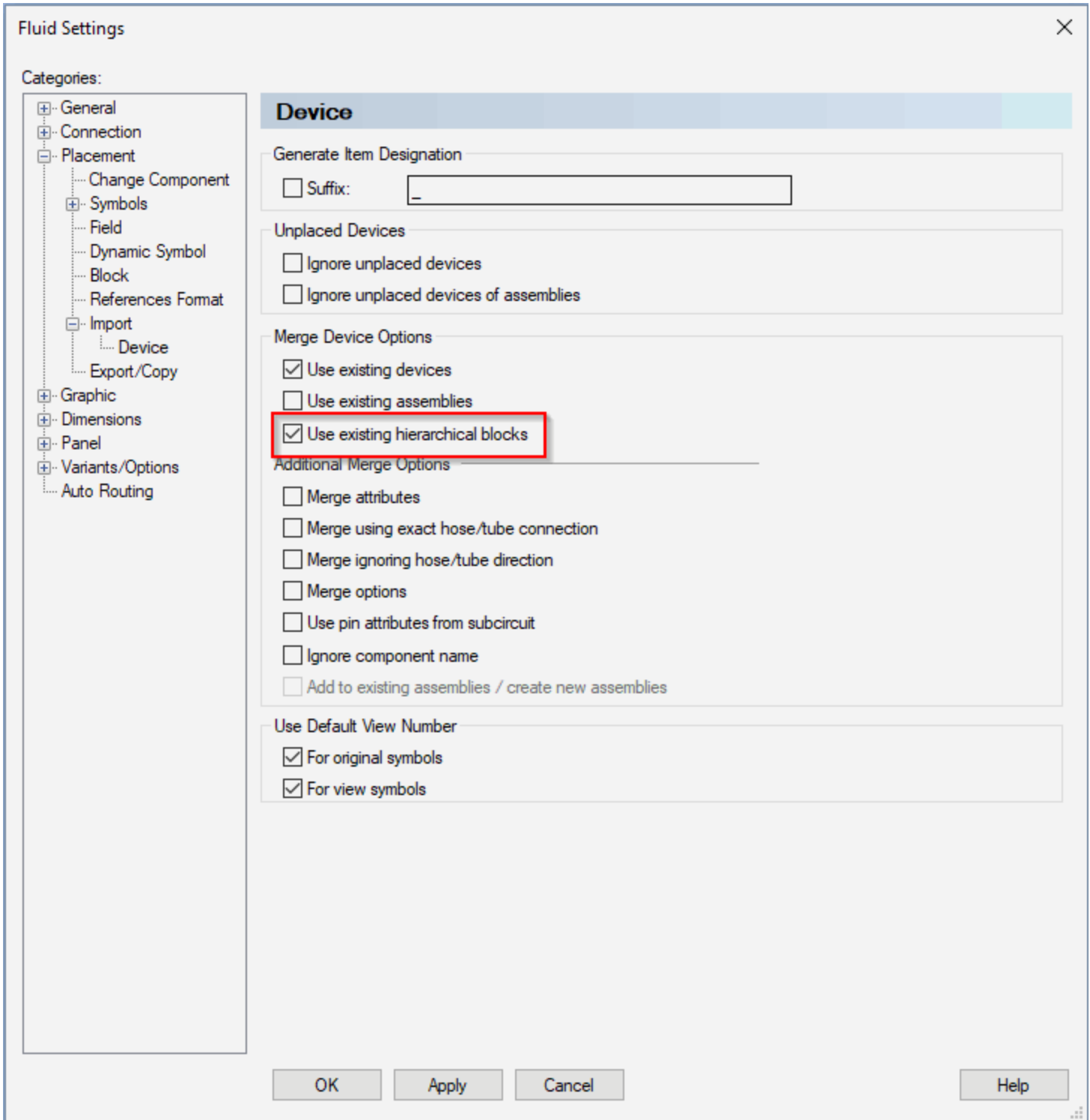
In **E³.series** the functionality for merging non-colliding hierarchical blocks has been enhanced. This will allow users to control whether hierarchical blocks in the clipboard with the same designation (as in the project) can be merged. Merging (Copy/Import) with the described rules also takes place without using the clipboard.

To do so, there is the new check box **Use existing hierarchical blocks** in the section **Merge Device Options** in the settings under **Placement -> Import -> Device**.

The dialog appears as follows in the 'Electric Settings':



The dialog appears as follows in the 'Fluid Settings':



Placement - Import - Device

Merge Device Options

Use existing hierarchical blocks

If the check box **Use existing hierarchical blocks** is activated, hierarchical blocks can be merged.

See examples:

- Example 1 - (Project with hierarchical blocks | Use existing hierarchical blocks active)
- Example 2 - (Project with hierarchical blocks | Use existing hierarchical blocks inactive)

Example 1 - Project with hierarchical blocks | **Use existing hierarchical blocks** is active

Devices	
Devices	Placement
MergeHierarchicalBlocks*	
-A	
-A1	/1.B3
Unassigned	
Clipboard [Manfred] [17.07...]	
-A	
-A1	/1.C3
Unassigned	

A hierarchical block is in the project and in the clipboard. Both have the same device designation.

No conflict is reported, there is a split block in the project:

Devices	
Devices	Placement
MergeHierarchicalBlocks*	
-A	
-A1	(2 Blocks)
Unassigned	

Example 2 - Project with hierarchical blocks | **Use existing hierarchical blocks** is inactive

Devices	
Devices	Placement
<ul style="list-style-type: none"> [-] MergeHierarchicalBlocks* <ul style="list-style-type: none"> [-] -A <ul style="list-style-type: none"> [+] -A1 [-] Unassigned 	/1.B3
<ul style="list-style-type: none"> [-] Clipboard [Manfred] [17.07... <ul style="list-style-type: none"> [-] -A <ul style="list-style-type: none"> [+] -A1 [-] Unassigned 	/1.C3

A hierarchical block is in the project and in the clipboard. Both have the same device designation.

A conflict is reported here. **Merge all non-colliding** cannot be executed.

Reference(s): Designer-34120

Title: Project Handling

- **3D Preview Window**
 - **Delete Attributes No Longer Being Used Automatically**
 - **Display Device Properties Automatically**
 - **Consider Drill Holes for Devices in *E³*.CutOut**
 - **Unique Option Names independent of Folder Structure**
 - **Messages associated with *Jump to Command* Highlighted**
 - **MIL-Standard: Display Break-Up Edges Above and Below Connectors**
 - **New Text Types Filled In Automatically**
 - **Define Dynamic Restricted Areas and Cutouts in Projects**
 - **Show Differences when Updating Components/Symbols**
 - ***Jump to 2D View* if 2D View of Model Created**
-

3D Preview Window

There is a 3D Preview Window in *E³*.series that can be placed anywhere the user wishes. The preview window is available in projects and the Database Editor.

Previewing the selected models gives the user an immediate overview of the models and a vivid impression of the planned project.

To activate the 3D preview window, select the command **View -> 3D Preview Window** in the *Main Menu*. The preview window opens.

The 3D preview window can be placed anywhere or docked to an area in the user interface.

The selected models are displayed in the 3D Preview Window. If several models are selected and placed, the 3D models are displayed as they are spatially related.

Delete Attributes No Longer Being Used Automatically

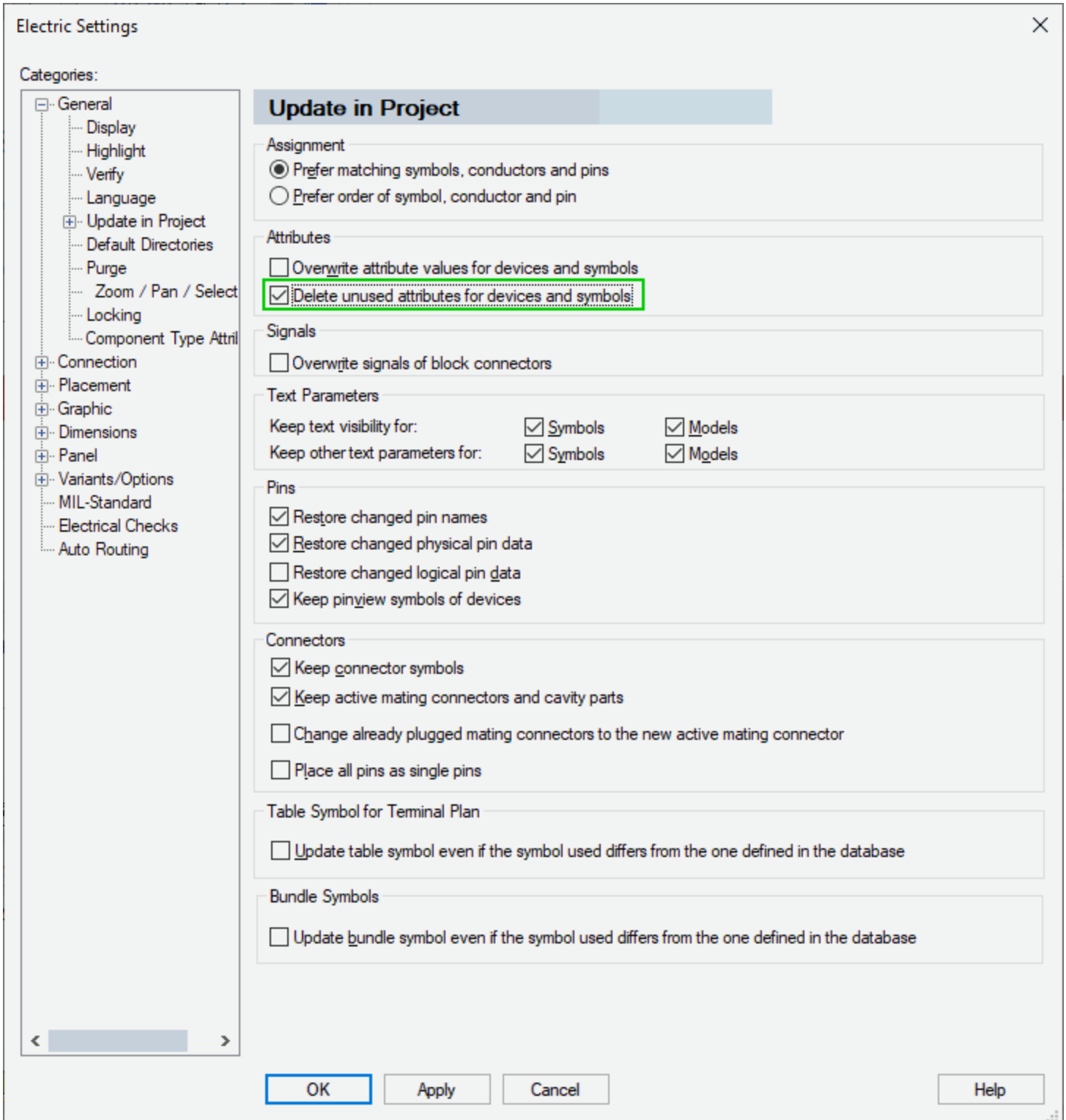
Attributes, which are no longer used in the project, can be deleted automatically.

This deletes unused and potentially superfluous data from projects and reduces the required storage space.

To automatically delete attributes not being used, select **Tools -> Settings...** in the *Main Menu*. The **Electric Settings** menu opens.

Note: This functionality is valid for the entire project and can be activated and deactivated using the **Electric Settings** menu as well as the **Fluid Settings** menu.

Go to the sub menu **General -> Update in Project** and activate the command **Delete unused attributes for devices and symbols:**



Attributes that no longer exist in the database are deleted. A corresponding message is displayed under the **Messages** tab in the *Output Window*.

Note: The command can also be activated and deactivated in the *Main Menu* under **Tools -> Settings... -> Placement -> Change Component**.

Display Device Properties Automatically

The device properties, which are selected on a sheet or in the Device Tree, are displayed automatically in the *Properties Window*.

This means you no longer have to use other menus to display the device properties.

Note: The device properties cannot be changed in the display.

To display the device properties, the *Properties Window* must be activated. To do so, select **View -> Properties** in the *Main Menu*.

Select one or more devices on a sheet or in the Device Tree. The properties of all selected devices are displayed in the *Properties Window*.

Note: The following restrictions apply to the automatic display:

- The properties of devices cannot be changed in the display.
 - The automatic display of properties are not valid for other objects, such as sheets, fields or symbols.
 - For devices that are part of a dynamic assembly and inherit the device designation of the assembly, the device designation is **not** displayed in the *Properties Window*.
 - The automatic display of device properties doesn't work if the view symbol of a device is selected.
 - Assigned variants/options of a device are not displayed.
-

Consider Drill Holes for Devices in **E³.CutOut**

In **E³.series** projects it's possible to set whether a drill hole shall be considered for each device in **E³.CutOut**.

In doing so, the user can directly specify for which devices the drill hole shall be considered in the project and which not.

To consider drill holes for devices in **E³.CutOut**, right-click on a device in the *Device Tree* and select **Device Properties...** The **Device Properties** open.

Go to the **Contour** tab and select an available **Drill hole**. **Note:** This tab is only displayed if a device with a model definition and a restricted area was selected.

Activate or deactivate the check box **Use in E3.CutOut** in order to consider or ignore the drill hole for the device in **E³.CutOut**:

Device Properties

Device | Device II | Component | Model | Shape | Contour

Selected devices

M_284957-P000004	-AP15
Drill-hole	

Type: Drill-hole

Z-Position: 0 mm Height: 13.5 mm

Radius: 12.5 mm

Use in E3.CutOut

Layer: on all layers

Restriction for

<input type="checkbox"/> Routes	<input type="checkbox"/> Pads
<input type="checkbox"/> Planes	<input type="checkbox"/> Vias
<input type="checkbox"/> Texts	<input type="checkbox"/> Components

Documentation Level

Fix (Restriction (18))

Free:

Preview

OK Cancel Apply Help

Unique Option Names independent of Folder Structure

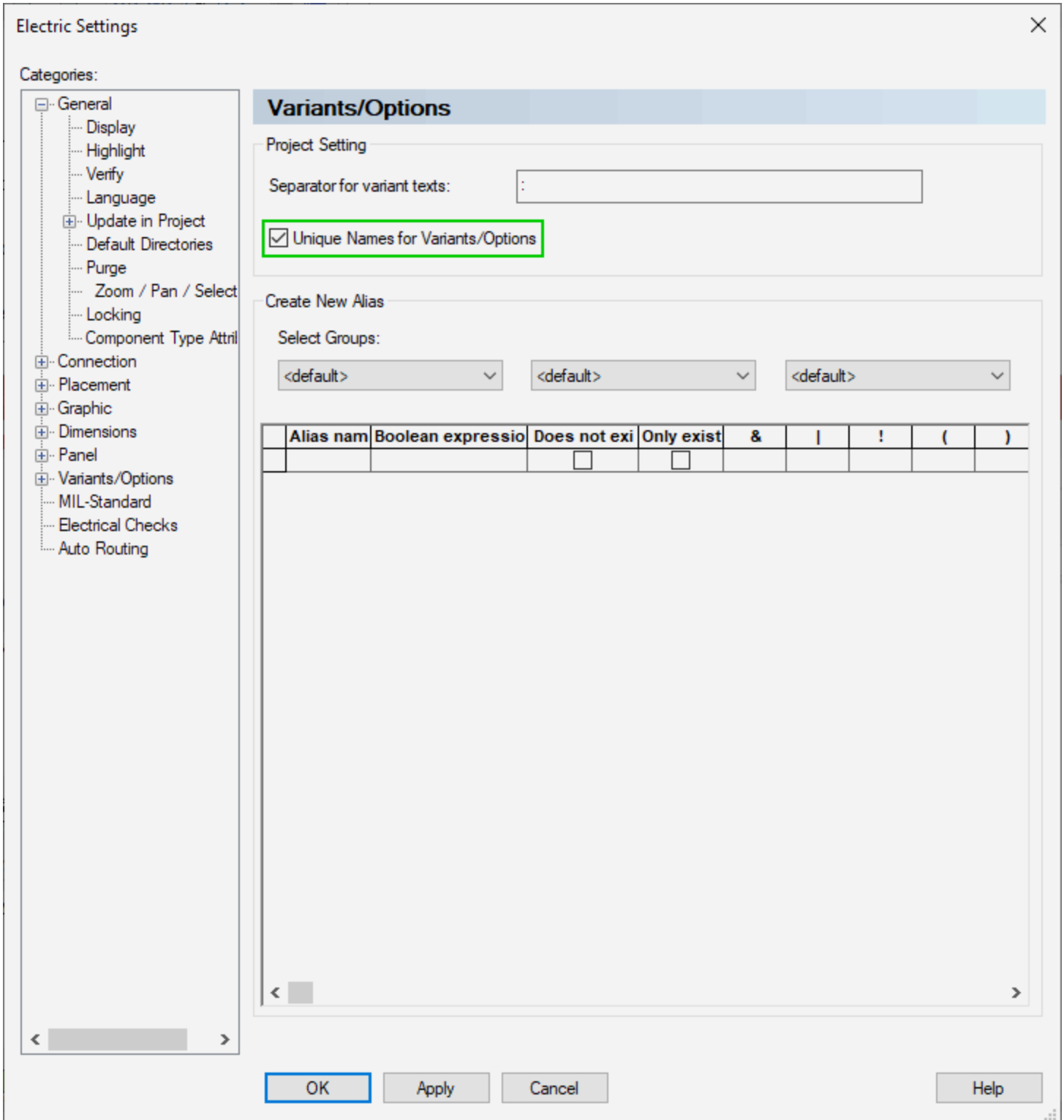
E³.series checks whether the assigned option names are unique. By default, the combination of folder and option names determines the uniqueness.

By checking for uniqueness, confusion and errors in planning can be avoided.

To check whether the option names are unique regardless of folder affiliation, select **Tools -> Settings...** in the *Main Menu*. The **Electric Settings** menu opens.

Note: The functionality is valid for the entire project and can be activated and deactivated in both the **Electric Settings** and **Fluid Settings**.

Go to the **Variants/Options** sub menu and activate the check box **Unique Names for Variants/Options**:



Variants or options, whose names are not unique, are displayed in the **Results** tab in the *Output Window*.

Messages associated with *Jump to Command* Highlighted

Some messages displayed in the *Output Window* are at times associated with a **Jump to** command.

Messages from which it is possible to jump to a specific point in the project are automatically highlighted in color. This makes it easier to identify messages with a jump destination.

Double-click on a highlighted message in the *Output Window* to jump to that destination.

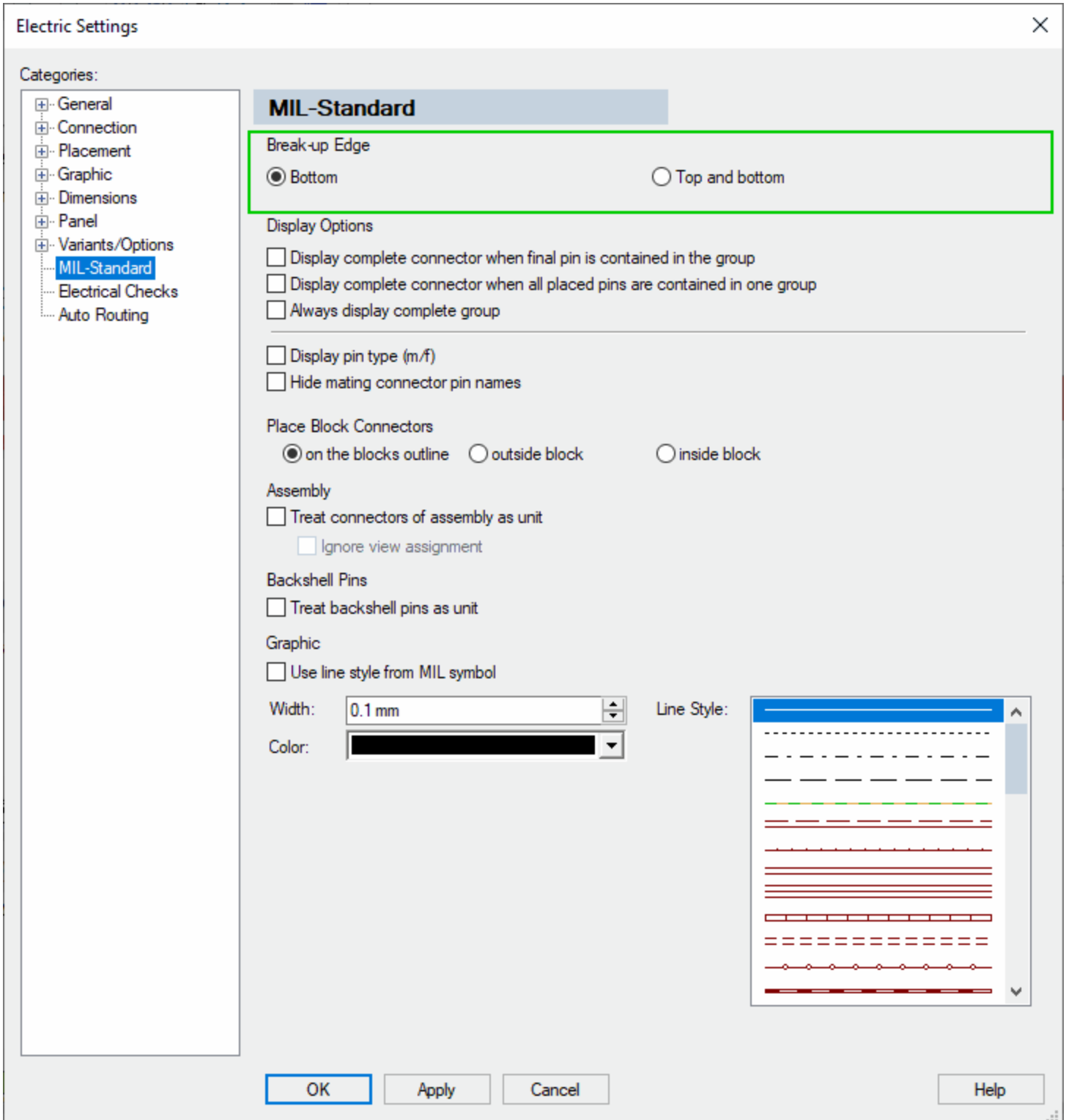
MIL-Standard: Display Break-Up Edges Above and Below Connectors

Connectors can be displayed with break-up edges if the connector is placed on a sheet and additional pins, which belong to the connector, are placed on other sheets.

This makes it easier to identify connectors that are not entirely on one sheet.

To display break-up edges, select **Tools -> Settings...** in the *Main Menu*. The **Electric Settings** menu opens.

Go to the section **Break-up Edge** in the **MIL-Standard** sub menu. Break-up edges can only be displayed on the **Bottom** or **Top and bottom** of connectors:



Note: There are three new characteristics in the Database Editor for displaying break-up edges **Top and bottom**. The new characteristics are:

- END_SPLIT_TOP: Lower end of connector with break-up edge above
 - MIDDLE_SPLIT: Connector with break-up edges above and below
 - SPLIT_TOP: Connector without defined end with break-up edge above
-

New Text Types Filled in Automatically

E³.series has new text types, which are filled in automatically.

The new text types link additional information to the project and symbols used. The following text types have been added:

- Model dimensions
- Outer diameter of conductors and cables
- Device letter code
- Sheet region references
- Sheet region scaling

If a symbol is used that can have one of these text types, the corresponding text type is assigned automatically and the value set.

Define Dynamic Restricted Areas and Cutouts in Projects


Restricted areas and cutouts can be defined directly in the project.

This allows the definitions of restricted areas and breakouts to be dynamically adapted to the project requirements.

To define a restricted area or a cutout dynamically in the project, select an individual model on a **Panel Sheet**.


Note: The model may not have any variants/options or be tilted.

Open the **Insert contour** menu. The menu can be opened in the following ways:

- right-click on the model and select **Insert contour** or
- **Insert -> Contour...** in the *Main Menu* or
- the  icon in the *Panel* toolbar.

Activate the **Create new contour** radio button, select the contour to be defined and then confirm with **OK**. The drawing mode for the contour opens.

In drawing mode define the shape and position of the restricted area or cutout!

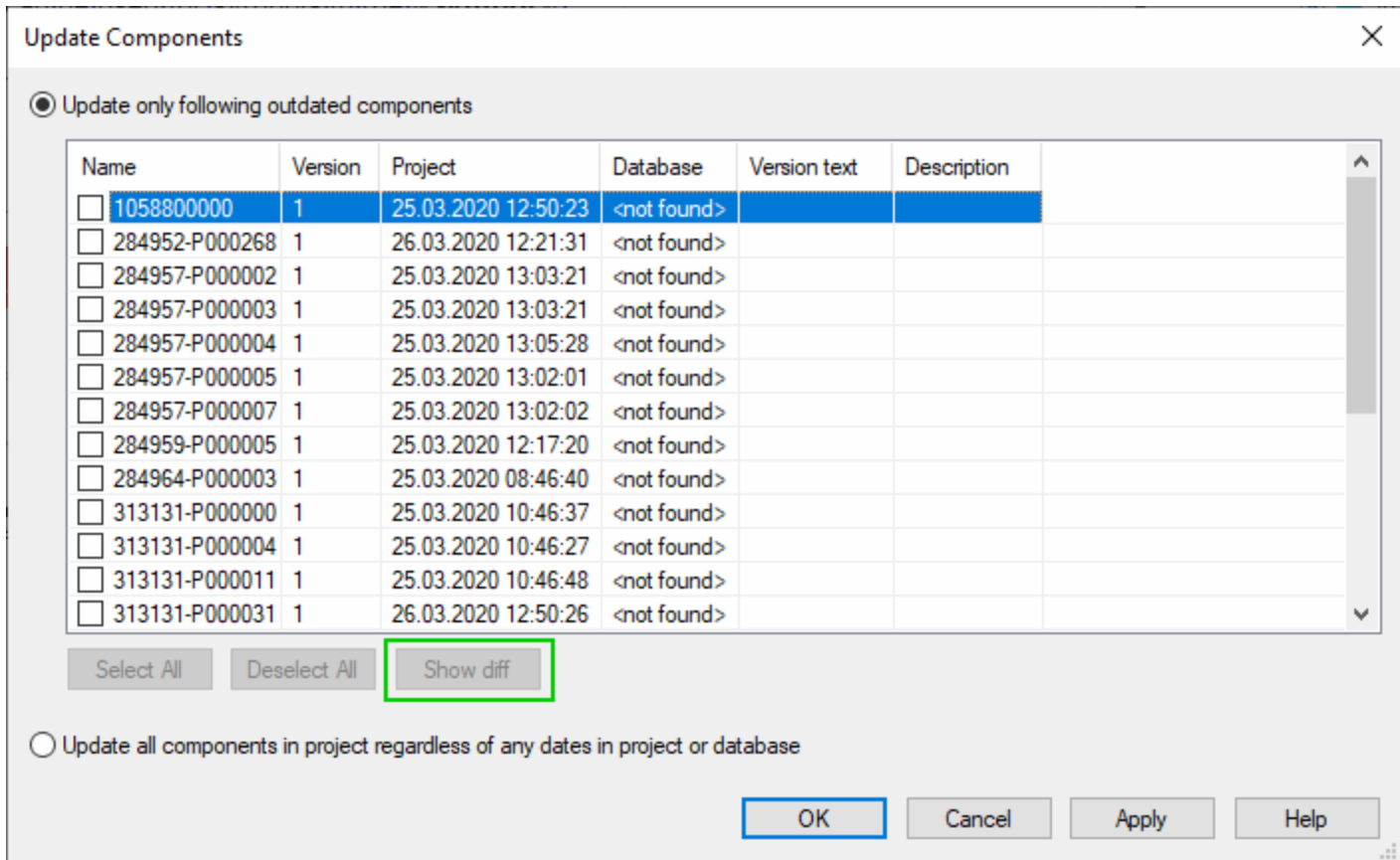
Restricted areas and cutouts, which are dynamically defined in the project, are additionally identified in the Device Properties by a blue *D*: 

Show Differences when Updating Components/Symbols

Before updating components or symbols, it's possible to display the changes.

This gives you an overview of the changes and allows you to check whether you want to update the affected components/symbols.

To display the differences before updating components, right-click on a folder in the **Component** tab in the *Data-base Window* and select the command **Update all components in project...** **Note:** This functionality can be used for symbols in the **Symbol** tab. The **Update Components** window opens:



Select **Show diff**. An overview of all changes to the selected components/symbols opens. The following table gives you an overview of the changes with an explanation of the highlighting:

Update Component - differences

ComponentsDbDiff [Items: 3]

- 280-601, 1
- 3RT10151AB01, 1
- H07V-K, 1

Project
3RT10151AB01, 1

Component

DeviceLetterCode	Q
Name	3RT10151AB01
InternalFlags	
IsSealed	<input type="checkbox"/>
Supply	
SuppressInDbTree	<input type="checkbox"/>

Version

Version	1
LastUpdate	12.01.2018 10:43:20
VersionStatus	Current
VersionText	Zuken E3 GmbH

Symbol Definition

TableSymbol	
DefaultTemplateSymbol	
PreviewSymbol	
TopologySymbol	

Symbols

Views

Attributes

ArticleNumber	3RT1015-1AB01
Class	Main contactor 3-pole AC

- Green highlighting: Line has been added!
- Yellow highlighting: Value has been changed!
- Red highlighting: Line was deleted!

Note: If a model is changed, every component connected to the model is also changed. The *LastUpdate* information is changed for all corresponding components. The changes to the model are not shown.

Jump to 2D View if 2D View of Model Created

From a model context menu it's possible to jump to a 2D view if a 2D view of the model has been created.

This enables you to determine rather quickly, for example, whether a 2D view of the selected model has already been created, and to jump directly to the 2D view for further editing of the model.

To jump to a 2D view of a model, open a panel sheet, on which the model is placed. Right-click on the model to open the context menu.

Select **Jump -> Jump to 2D View**.

Note: If **Jump to 2D View** is not listed in the context menu, a 2D view of the model hasn't been created yet. In this case, first select **Create 2D View** in the context menu. The **Insert New Sheet** window opens. Activate **Create new model view** under **2D View Options**.

Title: Tables


- **New Information in the *Connection Table***
- **New Column *Placement* in the *Pin Table***
- **Fit Column Width/Line Height for Multiline Texts**
- **Search using Boolean Expressions**

New Information in the *Connection Table*

The connection table displays new information by default.

The additional information provides an overview of the connections used in the project.

To display the information, select the command **View -> Connection Table** in the *Main Menu*. The **Connection Table** opens:

Connection Table									
	Wire/Hose/Tube type	From Device designation	From Assignment	From Pin terminal	From Wire seal	To Device designation	To Assignment	To Pin terminal	To Wire seal
	<All>	<All>	<All>	<All>	<All>	<All>	<All>	<All>	<All>
	H07V-K:2.5-BK	-F10	=A1						

From pin terminal: Shows which device the pin terminal has as a starting point.

To pin terminal: Shows which device the pin terminal has as a destination.

From wire seal: Shows which device the wire seal has as a starting point.

To wire seal: Shows which device the wire seal has as a destination.

New Column *Placement* in the *Pin Table*

There is a new column **Placement** in the **Pin Table**.

The column **Placement** shows on which sheet the pins are placed thereby making the search easier.

To display the **Placement** of the pins, first open a **Pin Table**. To do so, select the command **View -> Pin Table** in the *Main Menu*. The **Pin Table** opens.

Right-click on a column header in the **Pin Table** and select the command **Add Column**. A window opens in which you can add as many columns as you like using a drop-down menu. Select **< Placement >** as an additional column:

The image shows a software interface with a 'Pin Table' and an 'Add Columns' dialog box. The 'Pin Table' has the following data:

	Pin Name	Device	Assignr
	<All>	<All>	<All>
	U	-M10	=A1
	V	-M10	=A1
	W	-M10	=A1
	PE	-M10	=A1
	1	-F10	=A1
	2	-F10	=A1
	3	-F10	=A1
	4	-F10	=A1
	5	-F10	=A1
	6	-F10	=A1
	13	-F10	=A1

The 'Add Columns' dialog box shows a list of options to be added to the table. The selected option is '< Placement >'. Other options include '< Active wire seal >', '< Active connector pin terminal >', '(Class) Auxiliary Contacts', '(Class) Characteristic', '(Class) Core Identification', '(Class) Current', '(Class) Diameter', '(Class) Main Contacts', '(Class) Number of cores', '(Class) Number of poles', and '(Class) Operating voltage'.

The placement indicates on which sheet the pin is used and provides a rough orientation based on the coordinates displayed at the sheet margins. For example, the **Placement** specification */1.E4* means that the pin is placed on sheet 1 and located in the area where the coordinates E of the left/right sheet margin and 4 of the upper/lower sheet margin overlap.

Fit Column Width/Line Height for Multiline Texts

Multiline texts can be displayed in tables without moving the tables manually. This functionality is available in all tables.

This function can be used to avoid multiline texts being displayed as if they were single lines.

To fit the column width or line height, first open any table. Select the command **View** in the *Main Menu* and then a table. The corresponding table opens.

Right-click on the cogwheel symbol in the left hand margin of the table and select **Fit Line Height** or **Fit Column Width**:

Pin Name	Device	Assignment	Location	Component	Signal Name	Connection	(-20)
<All>	<All>	<All>	<All>	<All>	<All>	<All>	<All>
=A1	+F1	K21R-112-M2	#10	=A1+01-XT2:1c			
=A1	+F1	K21R-112-M2	#14	=A1+01-XT2:2c			
=A1	+F1	K21R-112-M2	#12	=A1+01-XT2:3c			
=A1	+F1	K21R-112-M2	PE	=A1+01-XT2:4			
=A1	+O1	3RV10214AA15	L1	=A1+01-S1:1, /1.			
=A1	+O1	3RV10214AA15	#7	=A1+01-Q10:1			
=A1	+O1	3RV10214AA15	L2	=A1+01-S1:3, /1.			
=A1	+O1	3RV10214AA15	#8	=A1+01-Q10:3			
=A1	+O1	3RV10214AA15	L3	=A1+01-S1:5, /1.			
=A1	+O1	3RV10214AA15	#9	=A1+01-Q10:5			
=A1	+O1	3RV10214AA15	L1				
=A1	+O1	3RV10214AA15	L1a				
=A1	+O1	3RV10214AA15					
=A1	+O1	3RV10214AA15					

Search using Boolean Expressions

Searching with Boolean expressions is supported in pin tables, device tables, sheet tables, translation tables and connection tables.

This functionality can be used to search for several terms within a table, which are connected with Boolean expressions. The following Boolean operations are available:

- the link with a logical AND (&),
- the link with a logical OR (|),
- the negation (!) and
- the combination of terms by parentheses ((to mark the beginning of a term and) to mark the end of a term).

To search in a table using Boolean expressions, first open any table. To do so, select the command **View** or **Format -> Translation Table** in the *Main Menu* and then a table. The corresponding table opens.

Press the **F7** key or right-click on the cogwheel symbol and select **Search**. The **Search** window opens.

Enter several search terms separated by Boolean expressions. The search results appear. The table contains the search command that was entered as a heading:

Device Table (+*&3*) ×

	Device designat	Assignment	Location	Component Co	Block
	<All>	<All>	<All>	<All>	<All>
	-F10	=A1	+O1	3RV10214AA15	
	-Q10	=A1	+O1	3RT10151AB01	
	-Q10	=A1	+O1	3RH19111HA12	
	-S1	=A1	+O1	3LD11040TB51	
	-SB10	=A1	+P1	3SB32050AA21	
	-SB11	=A1	+P1	3SB32070AA21	
	-P1	=A1	+T1	3SB32046AA20	

Contact

ZUKEN®

Zuken E3 GmbH

Laemmerweg 55

D-89079 Ulm

Phone: +49 7305 9309 0

Fax: +49 7305 9309 99

Email: e3-info@zuken.com

Web: www.zuken.com/e3.

For further contact information about Zuken E3 GmbH as well as our sales partners, please visit www.zuken.com/e3.