

The logo for the E3 series is positioned in the middle-left area of the page. It features a large, stylized red 'E3' with a motion blur effect on its right side. Below the 'E3', the word 'series' is written in a bold, black, sans-serif font.

# New Features

## 2015-16.00



### **Technical Inquiries**

Please contact our support department!

E-Mail: [e3-support@zuken.com](mailto:e3-support@zuken.com)

### **Note:**

Zuken is not responsible for any errors, which may appear in this documentation. Liability, due to direct and indirect losses resulting from the delivery or use of this documentation, is excluded to the extent permitted by law. This documentation contains copyrighted information.

All rights, especially those pertaining to the duplication and distribution as well as the translation, are reserved. This documentation, whether wholly or in part, may not be reproduced in any form (photocopy, microfilm, etc.), or processed, duplicated or distributed using an electronic system without Zuken's prior written consent.

### **Contact**

Zuken E3 GmbH  
Lämmerweg 55  
D-89079 Ulm/Einsingen

Phone: 07305/9309-0

Fax: 07305/9309-99

Web: <http://www.zuken.com>

E-Mail: [e3-info@Zuken.com](mailto:e3-info@Zuken.com)

# E<sup>3</sup>. series Releasenotes

Date: 29.05.2015

This manual describes the following functions of version:

## 2015 Build 16.00

### **!! Please Note when Switching to New Version !!**

Projects and databases are converted when opening them with this new **E<sup>3</sup>.series** Release. Please consider that these projects cannot be edited with an older version afterwards. That's why we recommend to backup projects before.

## Update Information of Version 2015

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

### Windows Support

The following Windows systems are supported with **E3.series** 2015:

- Windows Vista
- Windows 7
- Windows 8
- Windows Server 2003/2008

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** 2015 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** 2015, so copies should be used for **E3.series** 2015.

### 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 2015, a project can no longer be opened with an earlier version.

## Multi-User Installation

Existing multi-user projects from Version 2014A can directly be opened with Version 2015. They are automatically converted upon opening. However, this cannot be undone, and the projects can no longer be opened with Version 2014A.

That's why we recommend installing Version 2015 multi-user projects in parallel to the existing version, so that it's also possible to still access these project with **E3.series** Version 2014A.

To transfer the projects to Version 2015, these projects must be saved in Version 2014A as Single-User projects. They can then be stored in the multi-user environment of Version 2015.

In order to run the Multi-User databases in parallel, a new SQL Server process must be installed for Microsoft SQL Server and a new database user account for ORACLE.

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 2015 Build 16.00

The following functionality is new in Version 2015 Build 16.00:

- **Cables / Conductors / Wires**
  - Cables / Conductors / Wires - Miscellaneous Enhancements
  - Cabling Table
  
- **COM**
  - New Trigger AfterCreateDevice after Creating a Device
  - New Trigger AfterDeleteDevice after Deleting a Device
  - New Triggers for Signals and Signal Classes
  - The Trigger BeforeCheckOutSheet Now Receives a Parameter
  - COM Command Create2DView Receives New Value for Parameter 'modi' in DeviceInterface
  - New COM Commands in the NetInterface for Changing Signal Names
  - New Trigger BeforeModifyAttribute
  - Enhancement to ComponentInterface for Returning the Defined View Symbols of a Component with Their User-Specific View Numbers
  - New Trigger AfterConnectMUProject
  - Enhancement to JobInterface for Displaying a List of Drawing Elements in a Predefined Color with PDF Export
  - Enhancement to SymbolInterface for returning Duplicate Attribute Text Template Symbols whose Polygon Points refer to Origin's Coordinates
  - Enhancement within the PinInterface to set the maximum number of connections on a pin
  
- **Connections/Buses, Signals/Logic Lines, Supply**
  - **Conductor Logic Lines**
    - Conductor Logic Lines Can Now Be Displayed for Selective Signals
    - Logic Lines of Not Connected Conductors Can Now Be Displayed as Arcs
  - **New Functionality in E3 'Reorder Conductors'**
  - **New Functionality in E3 'Signal Format'**
  - **Misc**
    - Alternative Template Symbol Can Now Be Defined in the Settings
    - New Check Box in Settings for Deleting Signals of Not Connected View Symbols
    - Define whether the Signal on a Pin Shall Be Removed when Disconnecting Conductors/Wires on a Pin
  
- **Connectors**
  - Connector Pins Can Now Be Placed as Single Pins when Changing a Component or Updating a Project

- User-Defined View Numbers for Connector Views
- New Options for Assigning Numbers to Connectors and Mating Connectors in the Settings
  
- Fluid
  - 'Electric Settings' and 'Fluid Settings' Can Now Be Called Up in the Settings
  - Special Options with Separate Values for the Respective Discipline
  - Enhancement to JobInterface: COM Command 'SetDisplayConnectPoints' with Own Settings for Fluid
  
- Formboard
  - New Options for Inserting Nodes using the Formboard Dialog 'Add Node'
  - Pins without Placed Conductors/Wires Can Now Be Displayed in the Formboard Connector Table
  - Two Additional Possibilities Now for Automatically Routing Wires Assigned Options in Formboard
  
- Graphics / Dimensions
  - Three New Dimension Line Types Available
  - New Functionality 'Running Dimension'
  - Enhancements to DimensionInterface for Generating and Adapting Running Dimensions
  - Dimension Text of Running Dimensions Can Be Rotated
  - Enhancements to DimensionInterface for Displaying Dimension Text of Running Dimensions as Rotated
  
- Import/Export (subcircuits)
  - Connection Direction of Conductors/Wires Can Be Ignored when Importing Subcircuits
  - DWG/DXF Import/Export Now Compatible with AutoCAD Version 2013
  
- Panel
  - New Section '2D View Options' in 'Insert New Sheet' Dialog in E3.panel
  - Already Placed 2D Views Can Now Be Deleted
  - Symbol Properties Can Now Be Opened when Placing Models on Panel Sheets
  - New '3D View' Toolbar in Panel
  - Models Can Now Be Set as Transparent or Invisible in the 3D View

- **References**
  - **New Drop-Down Menu 'Format:' in the Settings for Defining How the Text Type 'Sheet Reference' Can Be Displayed**
  
- **Text**
  - **Two New Text Types in E3 for Referencing between Symbols in Circuit Diagrams and Components on Panel Sheets**
  - **Display of Multiline Texts Can Now Be Defined in the Symbol Properties for Symbols of the Type 'Attribute Text Template'**
  - **Lines Can Now Be Added to Text Boxes**
  - **Parameter Values Can Now Be Defined using Placeholders for Translated Text Nodes from the Translation Table**
  - **Ten New Sheet Text Types in the Database**
  
- **User Interface**
  - **'Full Screen Mode' for Workspace in E3.series**
  
- **Variants / Options**
  - **Variants/Options Tree Can Now Be Structured using Folders**
  - **All Elements in Variants/Options Tree Can Now Be Sorted in Ascending or Descending Order**
  
- **Views**
  - **Dialogs 'New View' and 'Place on Formboard' Have Been Enhanced**
  - **Control whether Active Connector Pin Terminal Symbol Shall Be Displayed for Views in the Device Properties**
  - **Keep pinview symbols at project update**



## Cables / Conductors / Wires - Overview

The cable list is now available in **E<sup>3</sup>.series**. The following chapters describe the new functionality and other enhancements:

- **Cabling Table**
  
- **Cables / Conductors / Wires - Miscellaneous Enhancements**
  - **Conductors/Wires Can Now Be Easily Moved from Pin to Pin**
  - **Wire Groups Can Now Be Displayed with Alternative Names in the Drop-Down List**
  - **Additional Connector Pin Terminal Information Can Now Be Defined on Symbols**

## Cabling Table

In project mode in **E<sup>3</sup>.series** there is now a Cabling Table, in which all connected graphical connections are listed with their devices in the project. The individual conductors of a cable are listed. Wires, hoses, tubes and jumpers are not displayed in the Cabling Table. With the help of this Cabling Table cables can be placed more effectively on connections.

Detailed information can be found in the following chapters:



- [Displaying the Cabling Table](#)
- [Adding/Deleting Columns](#)
- [Configuring the Cabling Table](#)
- [Global Search in the Cabling Table](#)
- ['Jump to' Functionality in the Cabling Table](#)
- [Display Ignored Connection Lines in the Cabling Table](#)
- [Context Menu Commands in the Cabling Table](#)
- [Setting/Resetting the Preselection](#)
- [New Check Method 'Checking graphical connections - \(0050\) None unique'](#)
- [Enhancement to JobInterface for Checking Ambiguous Connections in the Project](#)
- [Enhancement to NetSegmentInterface for Marking Connections Which Shall Not Be Displayed in the Cabling List](#)

### Displaying the Cabling Table

The Cabling Table can be called up as follows:

- click on the command **View -> Cabling Table** in the main menu, or
  - right-click on a connection line and select the command **Cabling Table...** in the displayed context menu.
- Note:** This context menu command is not displayed for a connection line in **E<sup>3</sup>.panel**.

Each row in the (Excel-like) table describes a pin to pin graphical connection and is divided into two levels:

- In the first level (Device Level) the connections are displayed from device to device (single pin/single row). Moreover, the status of the cabling state on the graphical connection is displayed; the following is valid:
    - **Red:** No cabling
    - **Yellow:** Partial cabling exists
    - **Green:** Complete cabling
  - In the second level (Pin Level) the connections are displayed on the pin level (multi pin/multi row). Only pins are displayed which have no wires, hoses and tubes applied
- Note:** This level can be displayed by clicking on the  button and hidden with the  button.

Cabling Table												
Assignment - From		Location - From		Device designation - From		Assignment - To		Location - To		Device designation - To	Cable Name	Cabling State
<All>		<All>		<All>		<All>		<All>		<All>	<All>	<All>
				-X1						-X2	-W2	
From Pin	From Pin Position	Conductor Name	Conductor Position	Bundle Name	To Pin	To Pin Position						
1	/1.C3				1	/1.C3						
2	/1.C3				2	/1.C3						
3	/1.C3	3			3	/1.C3						
4	/1.C3	4			4	/1.C3						
				-X3						-X4	-W13	
From Pin	From Pin Position	Conductor Name	Conductor Position	Bundle Name	To Pin	To Pin Position						
1	/1.C3	1	/1.C3		1	/1.C3						
4	/1.C3	2	/1.C3		4	/1.C3						
				-X5						-X6		
From Pin	From Pin Position	Conductor Name	Conductor Position	Bundle Name	To Pin	To Pin Position						
1	/1.D3				1	/1.D3						
2	/1.D3				2	/1.D3						
3	/1.D3				3	/1.D3						
4	/1.D3				4	/1.D3						
				-X7						-X8	-W8, -W9	

By default the columns in the first level display the following information:

**Note:** These columns cannot be deleted.

- Assignment - From
- Location - From
- Device designation - From
- Assignment - To
- Location - To
- Device designation - To
- Cable Name
- Cabling State

By default the columns in the second level display the following information:

**Note:** These columns cannot be deleted.

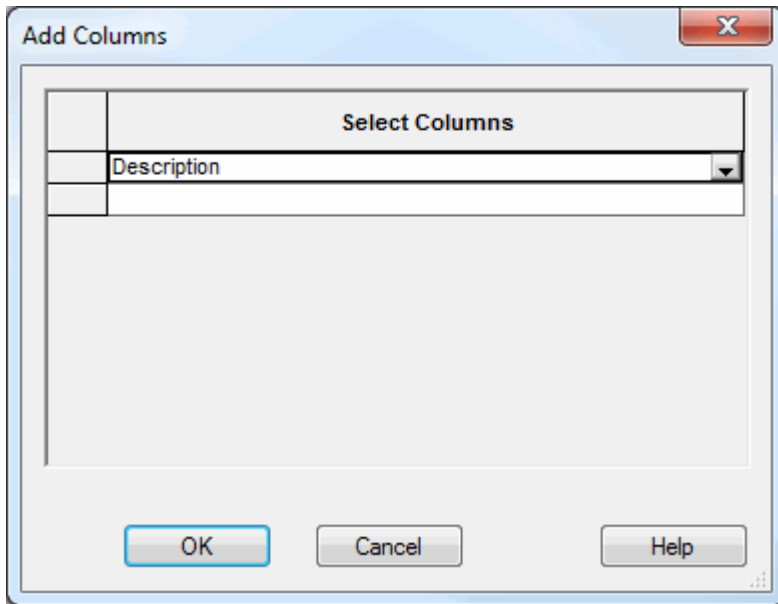
- From Pin
- From Pin Position
- Conductor Name
- Conductor Position
- Bundle Name
- To Pin
- To Pin Position

### Adding/Deleting Columns

All attributes can be displayed in the Cabling Table according to their database definitions. These attributes can be displayed in additional columns.

To do so, select the table column header and then the context menu command **Add Column**.

The dialog box **Add Columns** appears:



If an attribute appears more than once on an owner, the attribute can be selected repeatedly as a new column in the drop-down list. The headers of columns with the same attribute names appear with the number of the copy in brackets.

The command **Remove Column** deletes the column.

### Configuring the Table

The columns can be configured as follows:

- Columns of tables can be configured and arranged in any order. The columns of a table can be sorted based on the contents of a column using the command **Sort Ascending** or **Sort Descending**. By double-clicking on a column header the column can be sorted in ascending or descending order as well..
- The individual columns can be moved anywhere using Drag&Drop.
- By specifying a filter the output can be more selective (limited). Right-click on one of the two uppermost cells in first column of the window and select the command **Show Filter**. **Note:** When searching, it's recommended to type an asterisk in front of the expression to be searched: **\*Search Term**  
When resetting the filter, just type an asterisk in the cell: **\***
- The configuration of the tables, i.e. the displayed columns, width and order of the columns as well as the visibility of filtered rows, is saved in the project. This configuration is user-specific in a multi-user project.

### Global Search in the Cabling Table

It's possible to perform a global search in the Cabling Table. Not only does it search a fixed column but all locations in the Cabling Table.

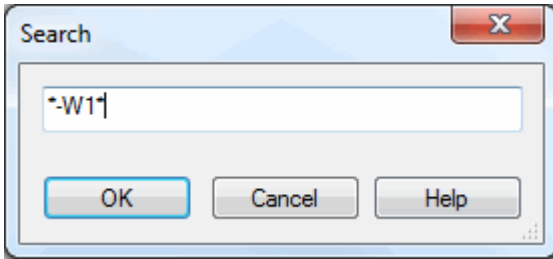
**To start the search:**

- right-click on one of the two uppermost cells in the first column of the window and select the command **Search**

or

click on the **F7** key

- The **Search** dialog window opens:



## Search

### OK / Cancel

The term to be searched in the table can be entered in the text field.

**Note:** A question mark '?' (for an individual character) and an asterisk '\*' (for several characters) can be used as wildcards for the search.

**OK** executes the search.

**Cancel** aborts the search.

All results are displayed in the project, where this search entry appears. Furthermore, the search term is enclosed in brackets after the cabling table header:

Cabling Table <span style="border: 1px solid red; padding: 2px;">*~W1*</span>				
	Cable Name	Device designation - From	Device designation - To	Cabling State
	<All>	<All>	<All>	<All>
<input type="checkbox"/>	-W1	-X3	-X4	3
<input type="checkbox"/>	-W1	-X5	-X6	3
<input type="checkbox"/>	-W1	-X9	-X10	3

### To reset the search:

- right-click on one of the two uppermost cells in the first column and select the command **Reset Search**  
or  
click on the key combination **SHIFT + F7**

## 'Jump to' Functionality in the Cabling Table

'Jump to' functionality is available in the Cabling Table. The 'Jump to' functionality offers the possibility to easily and quickly jump to objects within the project. Different 'Jump to' commands are available for selection depending on the level in the Cabling Table.

On the first level (Device/Cable Level) are the following *Jump* commands:

- **Jump to Schematic**
- **Jump to Panel**
- **Jump to Terminal Plan**
- **Jump to Device Tree**
- **Jump to Panel Tree (Placed)**

On the second level (Pin/Conductor/Net Level) are the following *Jump* commands:

- **Jump to Schematic**
- **Jump to Panel**
- **Jump to Device Tree**
- **Jump to Signal Tree**

**To execute the 'Jump to' command:**

- right-click on the desired cell in the Cabling Table and select the command **Jump to** in the displayed context menu,
- click on the corresponding **Jump to** command in the opened context menu.

The sheet with the marked symbol is opened automatically. If it is already opened, it is moved to the foreground. The drawing area is correspondingly adapted in order to have a clear overview of the selected object. Furthermore, the object is highlighted with a rectangle.

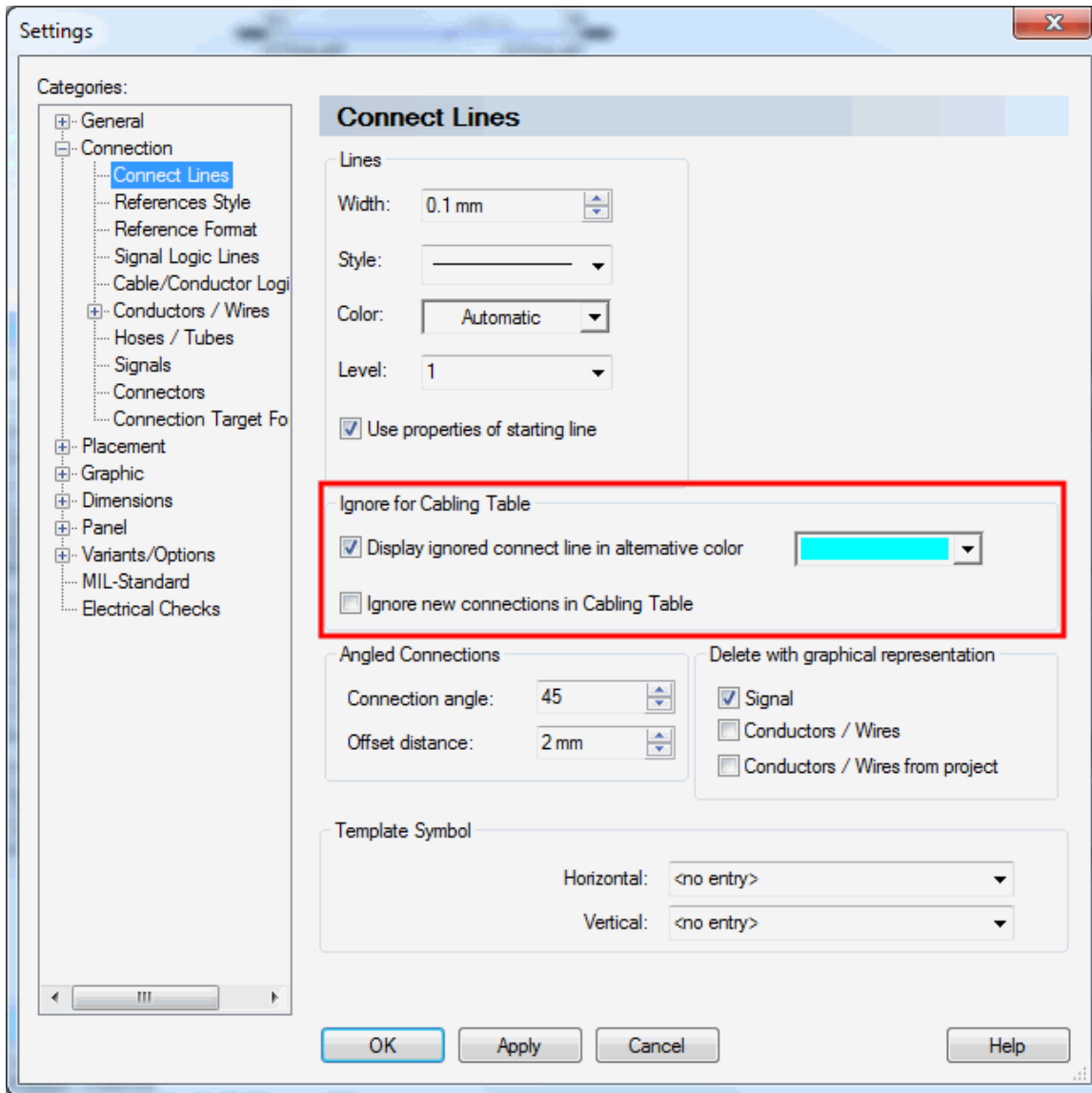
To reset the highlighting,

- click on the command **Edit -> Reset Highlight** in the main menu,  
or
- press the key combination **SHIFT+F3** .

## Display Ignored Connection Lines in the Cabling Table

It's possible to ignore connection lines in the Cabling Table. Moreover, the connection lines to be ignored can be highlighted using an alternative color. This is valid for all connection lines, on which no conductor is placed.

To do so, there is the new section **Ignore for Cabling Table** in the Settings under **Connection -> Connection Lines**.



## Connection - Connect Lines

### Ignore for Cabling Table

#### Display ignored connect line in alternative color

When checked, the desired connection lines can be ignored in the Cabling Table. These ignored connection lines are displayed in the schematic with the alternative color.

**Note:** The connection line is also displayed in the alternative color if only wires, tubes and hoses are located on the connection line.


For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**Ignore new connections in Cabling Table**

**DISPLAY\_IGNORED\_CONNECT\_LINE**

This check box is *user-specific* in a multi-user project.

When checked, all newly added connections in the Cabling Table can be ignored.

Alternatively, this functionality can also be controlled using the new icon  **Ignore in cabling table** (in the **Connection** toolbar).

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**IGNORE\_NEW\_CONNECT\_LINE\_IN\_CABLING\_TABLE**

This check box is *user-specific* in a multi-user project.

**Color**

Defines the color for the ignored connection lines.

If 'Automatic' is selected, the color defined in the **Color Definitions** dialog box (called up in the **Database Editor** under **Format-> Color Definitions...**) is used.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**COLOUR\_IGNORED\_CONNECT\_LINE**

This check box is *project-specific* in a multi-user project.

**Example:**

To ignore a connection line or the thereupon connected device that shall not appear in the Cabling Table:

- right-click on the desired connection line or the device, which shall be ignored in the Cabling Table,
- then select the command **Ignore for Cabling Table** in the opened context menu
- the connection line or the thereupon connected device are suppressed in the Cabling Table.

**Note:** Connections with conductors are permanently displayed in the Cabling Table. As soon as all the conductors are disconnected on the connection, the connection line or the thereupon connected device is suppressed in the Cabling Table.



To no longer ignore the connection line or the thereupon connected device:

- right-click on the ignored connection line or the ignored device,
- then select the command **Allow for Cabling Table** in the opened context menu,
- the connection line or the thereupon connected device is displayed again in the Cabling Table.

### Context Menu Commands in the Cabling Table

Depending on where one clicks in the Cabling Table, on which level (see [Displaying the Cabling Table](#)) and column with the right mouse button, different context menu commands are available:

#### Insert Template Symbol

**Note:** This context menu command can only be called up in the first column of the **Pin Level** or in the columns **Conductor Name**, **Conductor Position** or **Bundle Name**.

After executing the context menu command **Insert Template Symbol**, the system jumps to the selected connection line, on which the template symbol shall be inserted.

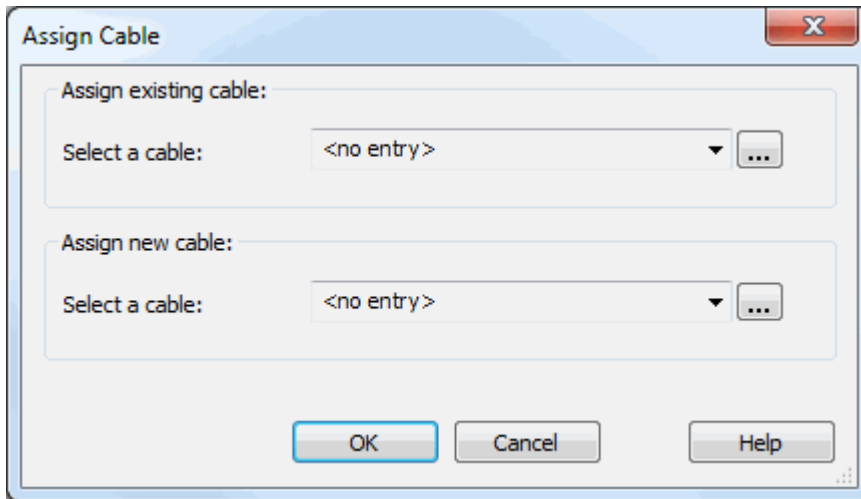
#### Unconnect Conductors

- If the context menu command is executed in the **Device Level**, all conductors are unconnected on the device.
- If the context menu command is executed in the first column of the **Pin Level** or in the columns **Conductor Name**, **Conductor Position** or **Bundle Name**, the conductor is unconnected on the pin.

#### Assign Cable...

- If the context menu command is executed in the **Device Level**, the cable's conductors are assigned to the complete device.
- If the context menu command is executed in the **Pin Level**, one cable conductor is assigned to the pin.

Executing this context menu command opens the **Assign Cable** dialog:




## Assign Cable

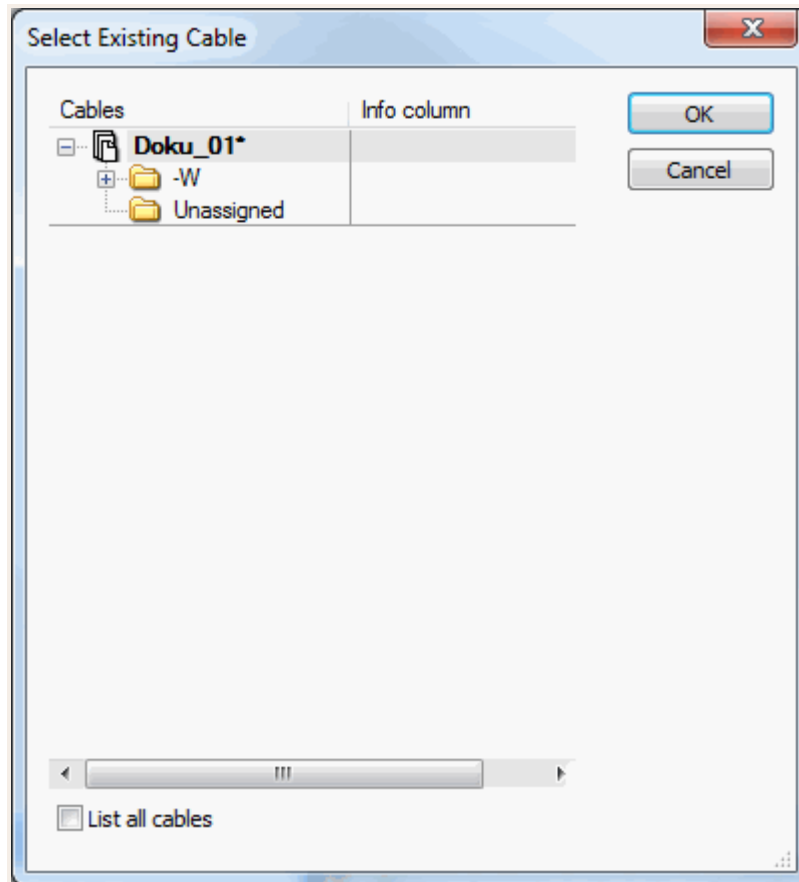
### Assign existing cable:

#### Select a cable:

All cables are displayed in the drop-down list, which fulfill the following prerequisites in the project:

- The cable has at least as many conductors as the number of pins available on the connection line.
- The conductors of the cable fulfill the cross-section specifications of the pins.

Clicking on this button  opens the **Select Existing Cable** dialog:



All useable cables within a project are displayed in the form of a tree structure here.


By checking the check box **List all cables** all cables in the project are displayed (as well as already placed).

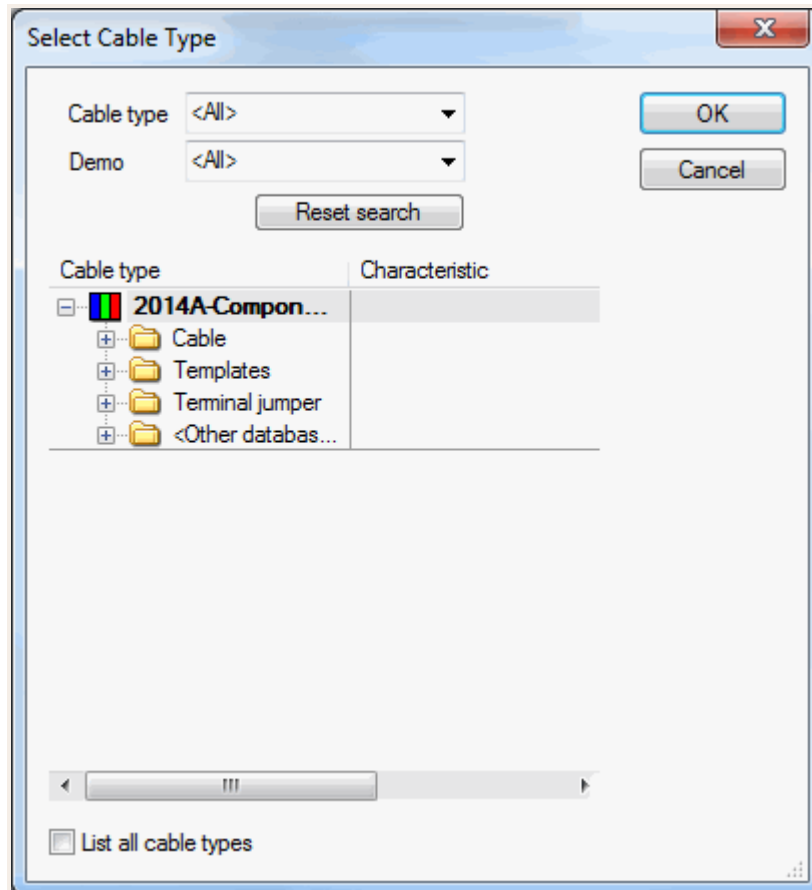
#### Assign new cable:

##### Select a cable:

All cables within a project and from the database are displayed in the drop-down list, which fulfill the following prerequisites:

- The cable has at least as many conductors as the number of pins available on the connection line.
- The conductors of the cable fulfill the cross-section specifications of the pins.

Clicking on the button  opens the **Select Cable Type** dialog:



The following cables are displayed here in the form of a tree structure:

- all useable cable types (number of conductors / cross-sections that meet the pin requirements) from the database
- all useable cables (number of conductors / cross-sections that meet the pin requirements) within a project

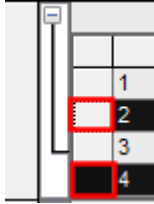
By checking the check box **List all cable types** all cable types from the database and the project are displayed.

## Swap Conductors

**Note:** This context menu command can only be executed in the **Pin Level**

- o if both conductors to be swapped are from the same cable
- o if only one conductor on a free pin shall be swapped
- o To do so, press and hold the **SHIFT** or **Ctrl** key and then left-click on the desired conductor in the first column. Afterward, right-click anywhere and select the context

menu command **Swap Conductors**:



After executing the context menu command **Swap Conductors**, both conductors are exchanged with one another.

### Device Properties...

**Note:** This context menu command can only be executed in the **Device Level**.

After executing the context menu command **Device Properties...**, the **Device Properties - Device** dialog opens.

### Pin Properties...

**Note:** This context menu command can only be called up in the first column of the **Pin Level** or in the columns **From Pin**, **From Pin Position**, **To Pin** or **To Pin Position**.

After executing the context menu command **Pin Properties...**, the **Device Properties - Pins** dialog opens.

### Conductor Properties...

**Note:** This context menu command can only be called up in the first column of the **Pin Level** or in the columns **Conductor Name**, **Conductor Position** or **Bundle Name**. Additionally, one conductor must be placed.

After executing the context menu command **Conductor Properties...**, the **Device Properties - Wire Information** dialog opens.

## Setting/Resetting the Preselection

### Defining the Preselection for the Cabling Table

By filtering using preselected connections and devices the Cabling Table can be defined accordingly. The filtering action gives the user a better overview of the Cabling Table because only the desired connections and devices are displayed.

To define the preselection for the desired connected graphical connections, which shall be displayed in the Cabling Table:

- pressing the SHIFT key and select the desired connection line or device in the schematic,
- right-click on one element of the selection,

- select the command **Cabling Table...** in the displayed context menu,
- this opens the Cabling Table with the preselected selection:

	Cable Name	Device designation - From	
	<All>	<All>	
-W1		-X3	
	From Pin	From Pin Position	Conductor
1		/1.C2	GN
2		/1.C2	YE
3		/1.C2	GY
4		/1.C2	PK
-W1		-X5	
	From Pin	From Pin Position	Conductor
1		/1.C2	BU
2		/1.C2	RD

### Resetting the Preselection for the Cabling Table

The filtering in the Cabling Table using preselected connection lines or devices can be reset.

To reset the preselected Cabling Table to its original state:

- right-click anywhere in the first column of the Cabling Table,

	Cable Name	Device designation - From		Device designation - To		
	<All>	<All>		<All>		
		-X3		-X4		
		From Pin Position	Conductor	Conductor Pos	Bundle Name	To Pin
1		/1.C2	GN	/1.C2		1
2		/1.C2	YE	/1.C2		2
3		/1.C2	GY	/1.C2		3
4		/1.C2	PK	/1.C2		4
-W1		-X5		-X6		
	From Pin	From Pin Position	Conductor	Conductor Pos	Bundle Name	To Pin
1		/1.C2	BU	/1.C2		1
2		/1.C2	RD	/1.C2		2

- then select the command **Reset Preselection** in the displayed context menu.

After resetting, the Cabling Table appears with all connected graphical connections in the project.

**Reference(s):** Designer-25598

### New Check Method 'Checking graphical connections - (0050) None unique'

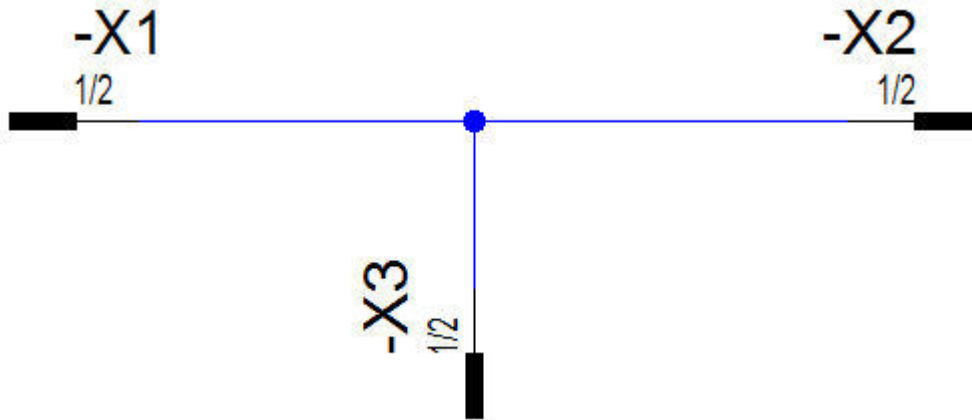
Additionally, there is the new check method **Checking graphical connections - (0050) None unique** in the standard check programs of **E3.series** (called up using **Tools -> Verify**). When executing the check method, all non-unique connections are listed in the 'Results Window'.

**Example:**



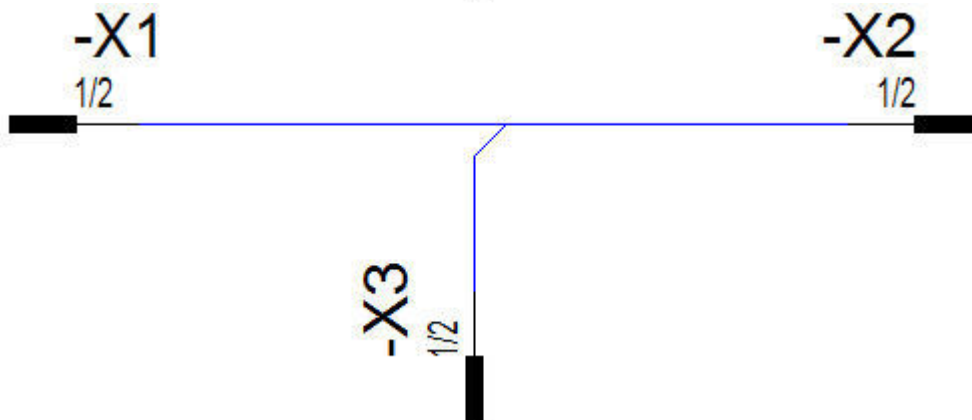
Unique graphical connection:

The connection pathway is unique because there is only one placement possibility.



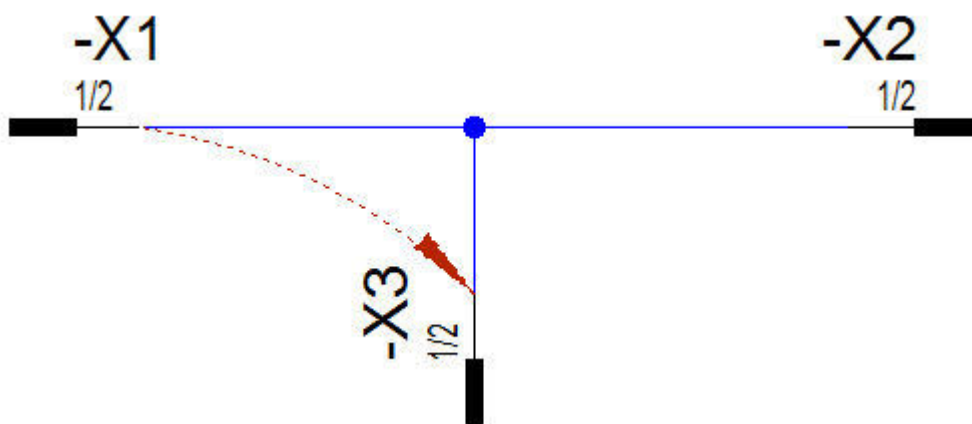
No unique graphical connection:

The connection path is not unique because there are several possibilities.



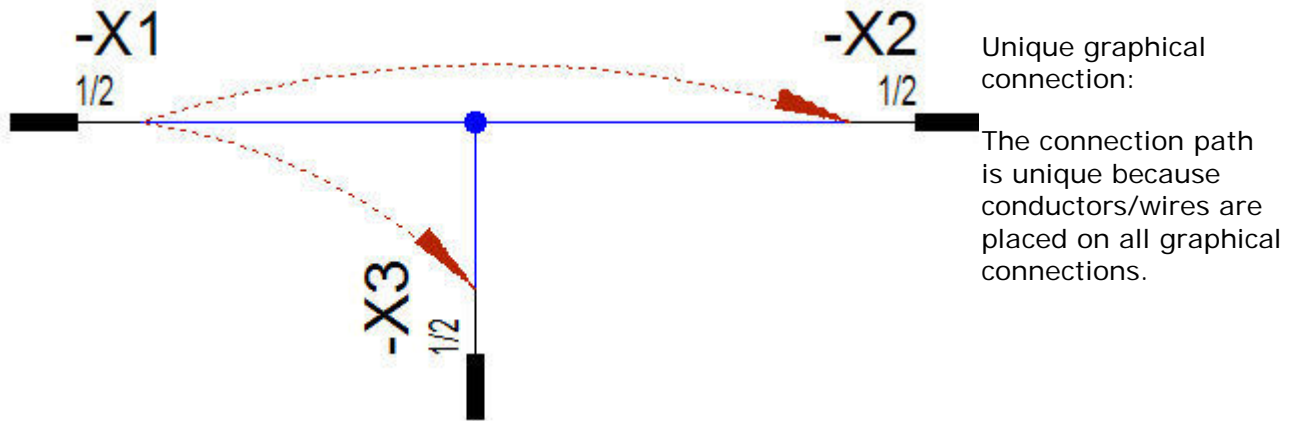
Unique graphical connection:

The connection path is unique based on the "forced wiring".



No unique graphical connection:

The connection path is not unique because conductors/wires are not placed on all graphical connection lines.



**Note:** Please make sure the XML check file contains the new check method. The XML check file is called up in the Registry (HKEY\_CURRENT\_USER\SOFTWARE\Wow6432Node\Zuken\E3.series\<version>) with the character string VerifyXMLPath for **E3.series** or VerifyXMLPath\_Logic for **E3.logic**.

### Enhancement to JobInterface for Checking Ambiguous Connections in the Project

There are the following new COM commands in the **JobInterface**:

- **cnt = job.GetAllConnectionIds ( ids [,flags] )**
- **cnt = job.GetSelectedConnectionIds ( ids [,flags] )**
  - o flags has been enhanced:
  - o **flags = 0x4** when the flag is set, formboard connections are ignored.

**Reference(s):** Designer-25609

### Enhancement to NetSegmentInterface for Marking Connections that Shall Not Be Displayed in the Cabling List

There is the following new COM command in the **NetSegmentInterface**:

- **ret = ns.SetIgnoreForCablingTable ( set )**
  - o **set 1** Connections are ignored in the cabling list.
  - o **set 0** Property reset
  - o **ret** The return value can have the following values:
    - **0:** The property was previously not set on the net segment.
    - **1:** The property was previously set on the net segment.
    - **-1:** No project
    - **-2:** Item is not a net segment.
    - **-3:** The net segment is part of a formboard or topology net.

**Reference(s):** Designer-25603



## Cables / Conductors / Wires - Miscellaneous Enhancements

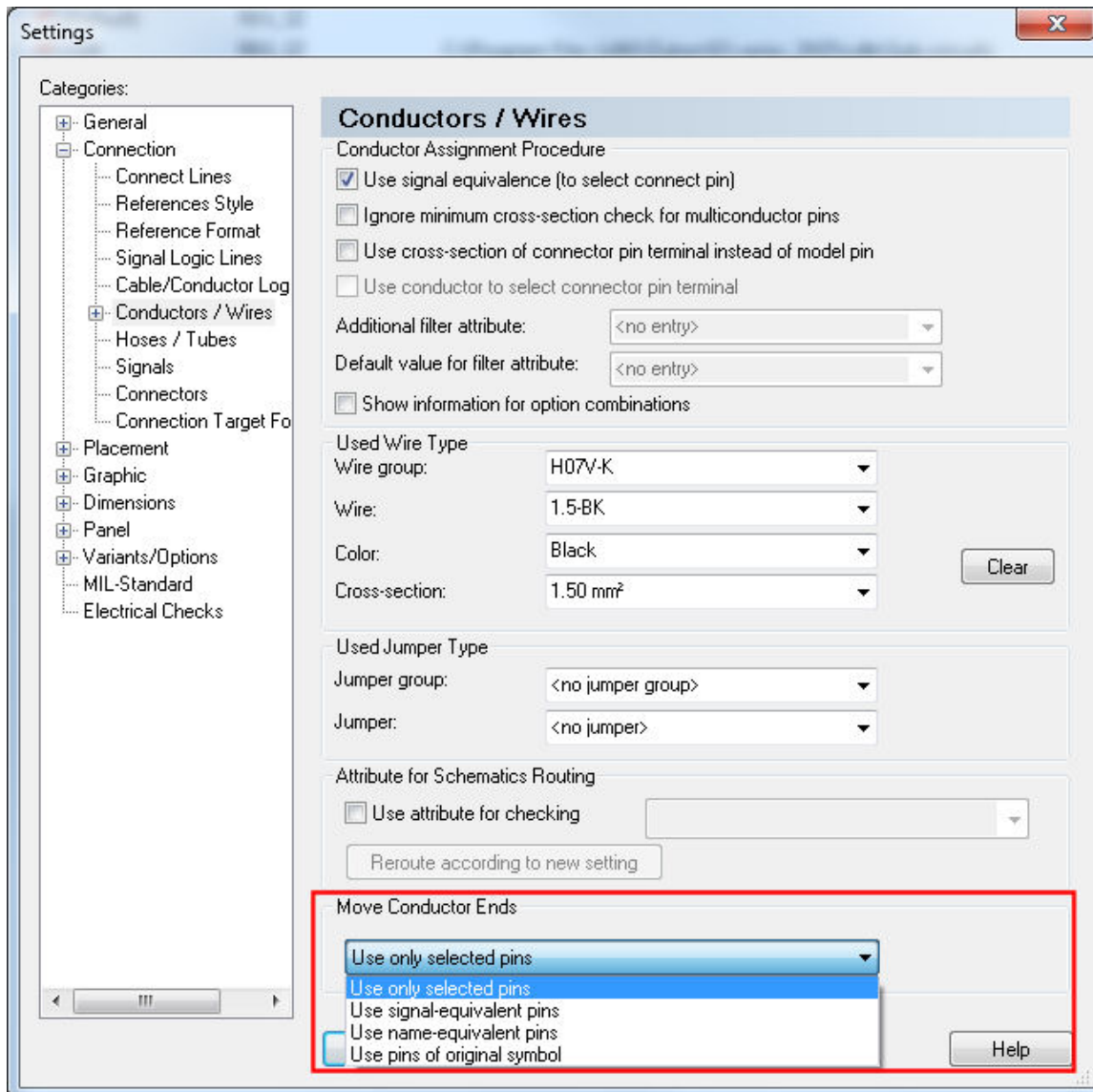
- [Conductors/Wires Can Now Be Easily Moved from Pin to Pin](#)
- [Wire Groups Can Now Be Displayed with Alternative Names in the Drop-Down List](#)
- [Additional Connector Pin Terminal Information Can Now Be Defined on Symbols](#)

### Conductors/Wires Can Now Be Easily Moved from Pin to Pin

Now it's possible to move conductors/wires easily from pin to pin in **E<sup>3</sup>.series**. The following options are available for determining which shall be used when moving:

- **Use only selected pins**
- **Use signal-equivalent pins**
- **Use name-equivalent pins**
- **Use pins of original symbol**

The desired option must be defined **before** the move takes place. To do so, use the new drop-down list in the section **Move Conductor Ends** under **Connection - Conductors/Wires**.



## Conductors/Wires

**Move Conductor Ends** Determining which pins shall be used when moving can be defined here:

- **Use only selected pins**
  - The command tries to move conductors/wires only on directly selected pins.
- **Use signal-equivalent pins**
  - The command tries to move conductors/wires on directly selected pins as well as on all corresponding signal equivalent pins.
- **Use name-equivalent pins**
  - The command tries to move conductors/wires directly on selected pins as well as on all corresponding name equivalent pins.
- **Use pins of original symbol**
  - The command tries to move conductors/wires directly on selected pins as well as on all pins which are available on the symbol of the original view.  
**Note:** If the original symbols are equal, the assignment of symbols will also be swapped.

For the COM methods `prj.GetSettingValue` and `prj.SetSettingValue` the following `setting_name` can be used: `MOVE_CONDUCTOR_ENDS`.

This check box is *user-specific* in a multi-user project.


### Note:

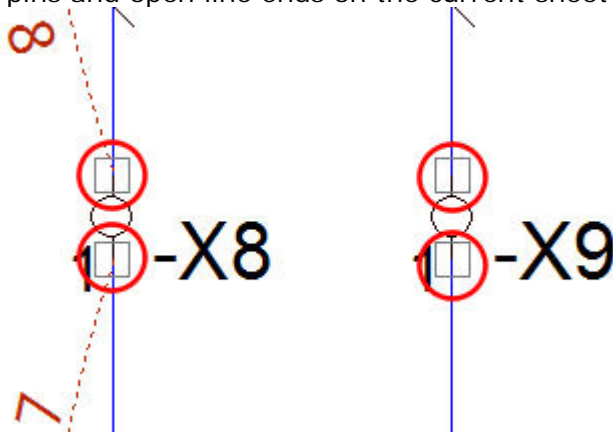
- Starting and target pins can also be selected in reverse order.
- If conductors/wires are placed on both the starting and the target pins, they are swapped.
- If conductor/wire ends cannot be moved (e.g. due to invalid cross-sections), they remain on the original pin.
- If there are overlaps at the mass of the selected starting and target pins, an attempt will be made to move the conductor/wire ends further.

**To move conductors/wires from one pin to another:**

Use the new **Move conductors** icon in the toolbar - category **Connect** to start the command.

**Step 1:**

- Left-click on the **Move conductors** icon  to activate the command.  
**Note:** double-clicking on the icon allows the user to repeat the command as often as he likes.
- All pins and open line ends on the current sheet are displayed by a square:



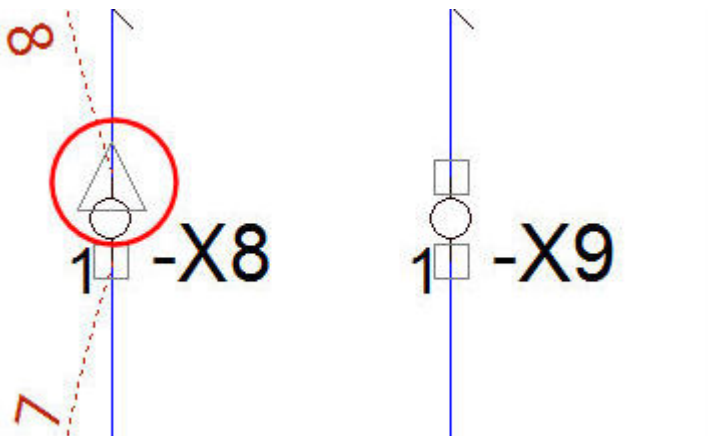
**Step 2:**

Select the starting pin, on which the conductor/wire ends are connected and that shall be moved to another target pin.

- Left-click on the desired pin or if several pins need to be selected,
- drag a rectangle around the desired pins while pressing the left mouse button.

**Note:**

- Depending on the setting in the **Move conductor ends** drop-down list (see above), the corresponding pins are added to the selection automatically.
- The selected starting pins are marked with a triangle (pointing upwards).

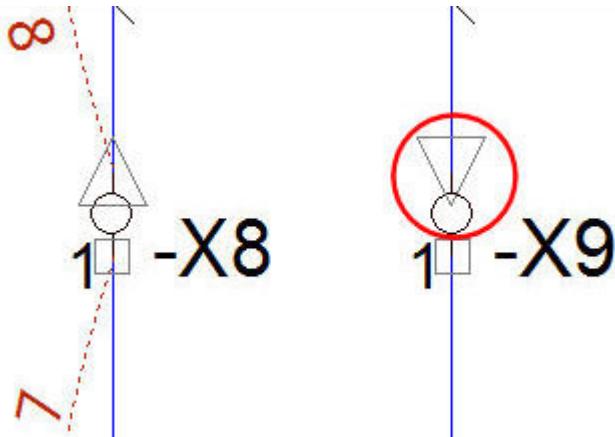


### Step 3:

- Left-click on the target pin where the conductor/wire ends should be moved or if several pins need to be selected,
- drag a rectangle around the desired pins with pressing the left mouse button.

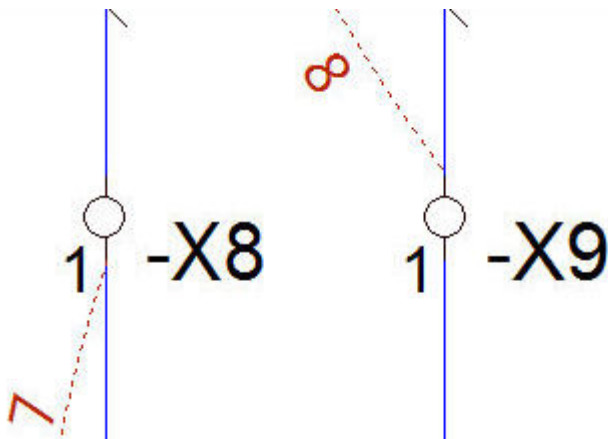
### Note:

- Depending on the setting in the **Move conductor ends** drop-down list (see above), the corresponding pins are added to the selection automatically.
- The selected target pins are marked with a triangle (pointing downwards).



### Step 4:

After the target pin has been selected, the new assignment of the conductor/wire ends occurs.



### Note:

- If several conductor/wire ends exist, the assignment is based on the position of the pins on the sheet.  
The sorting takes place first in ascending **Y** order and then in ascending **X** order.
- All new connected conductors/wires are displayed in Results Window after moving.

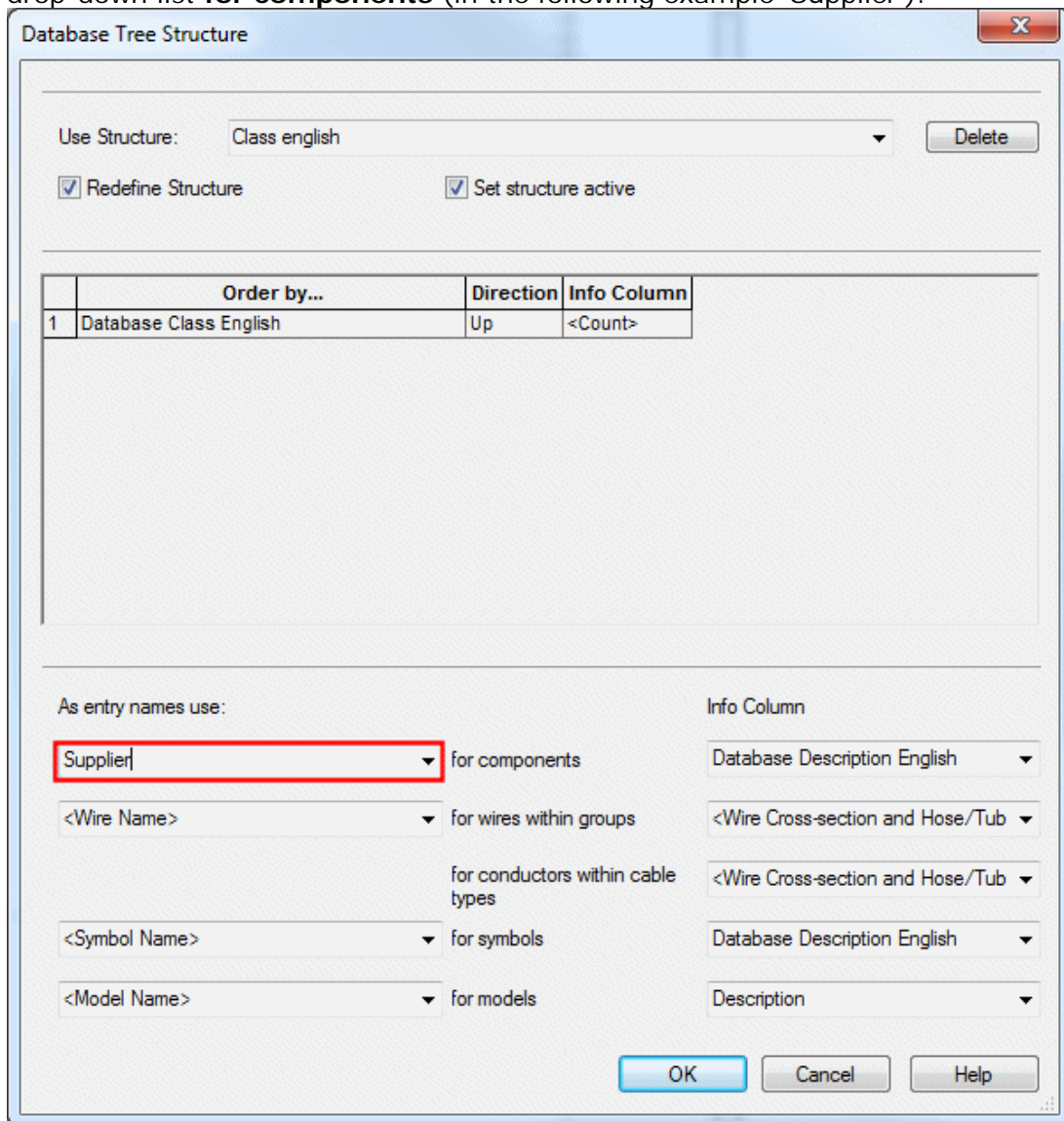
**Reference(s):** Designer-25501

## Wire Groups Can Now Be Displayed with Alternative Names in the Drop-Down List

In **E<sup>3</sup>.series** the value of an attribute can be defined in the tree structure, which is then displayed in the drop-down list instead of the component name. This was not true for wire groups.

From now on the functionality is also available for wire groups.

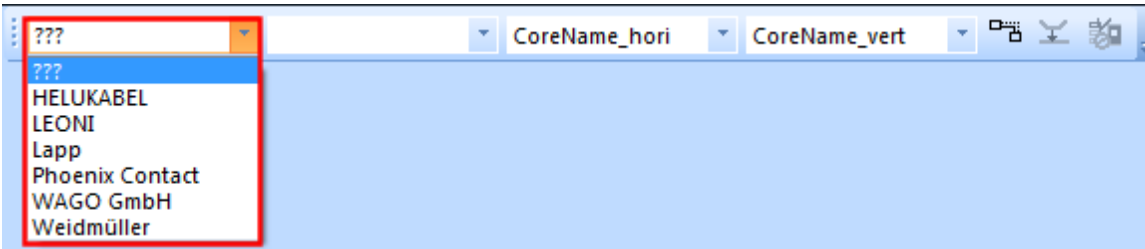
- To do so, right-click on the name of the component database in the component tree structure,
- select the context menu command **Tree Structure**,
- select a structure to use under **Use Structure** in the Database Tree Structure dialog,
- check the check box **Redefine Structure** and then select the desired attribute in the drop-down list **for components** (in the following example 'Supplier'):



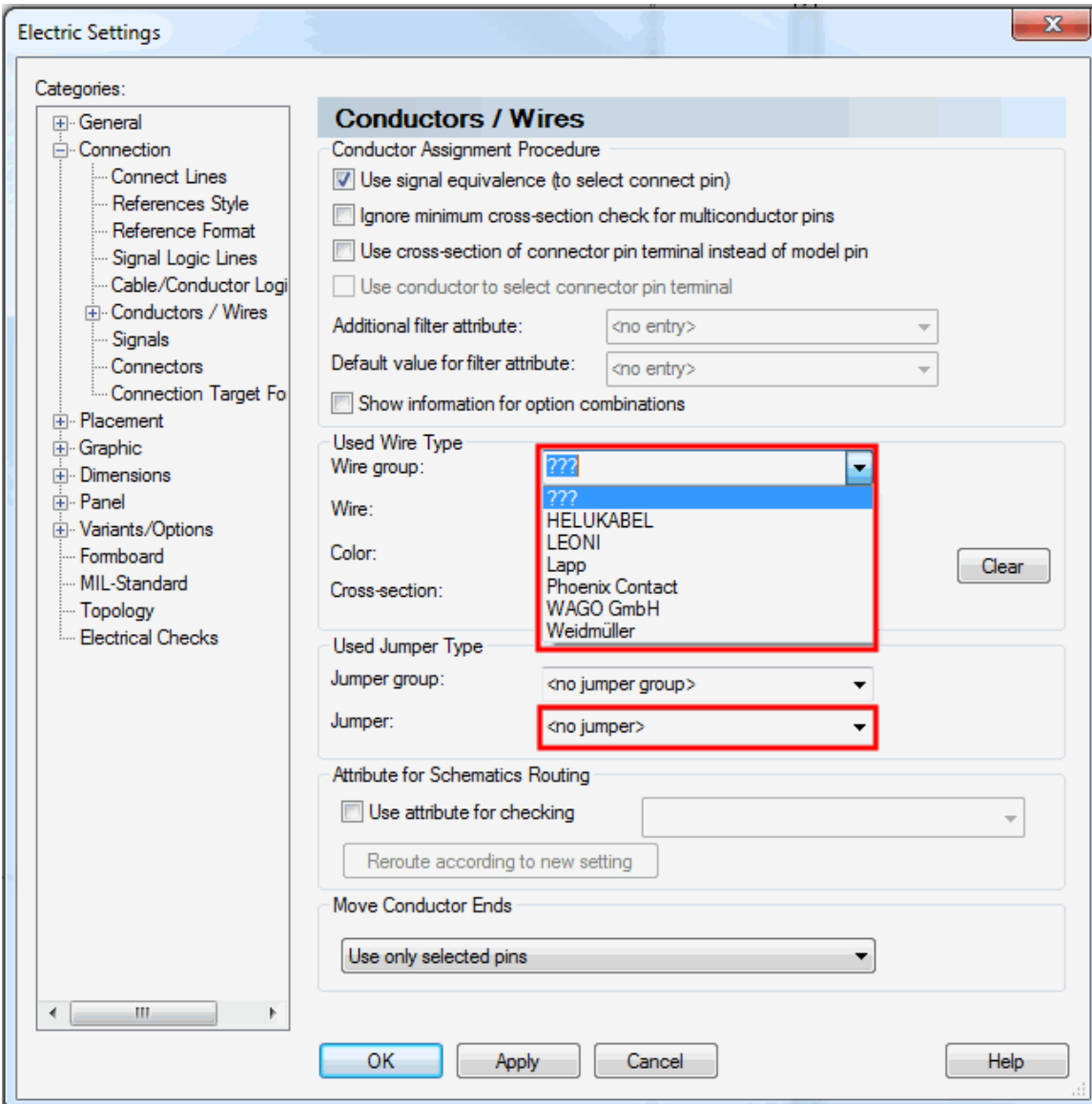
- update the component structure view by clicking the **F5** key

The wire groups corresponding with the attribute value (in this case according to 'Supplier') are now listed in the wire group drop-down list:

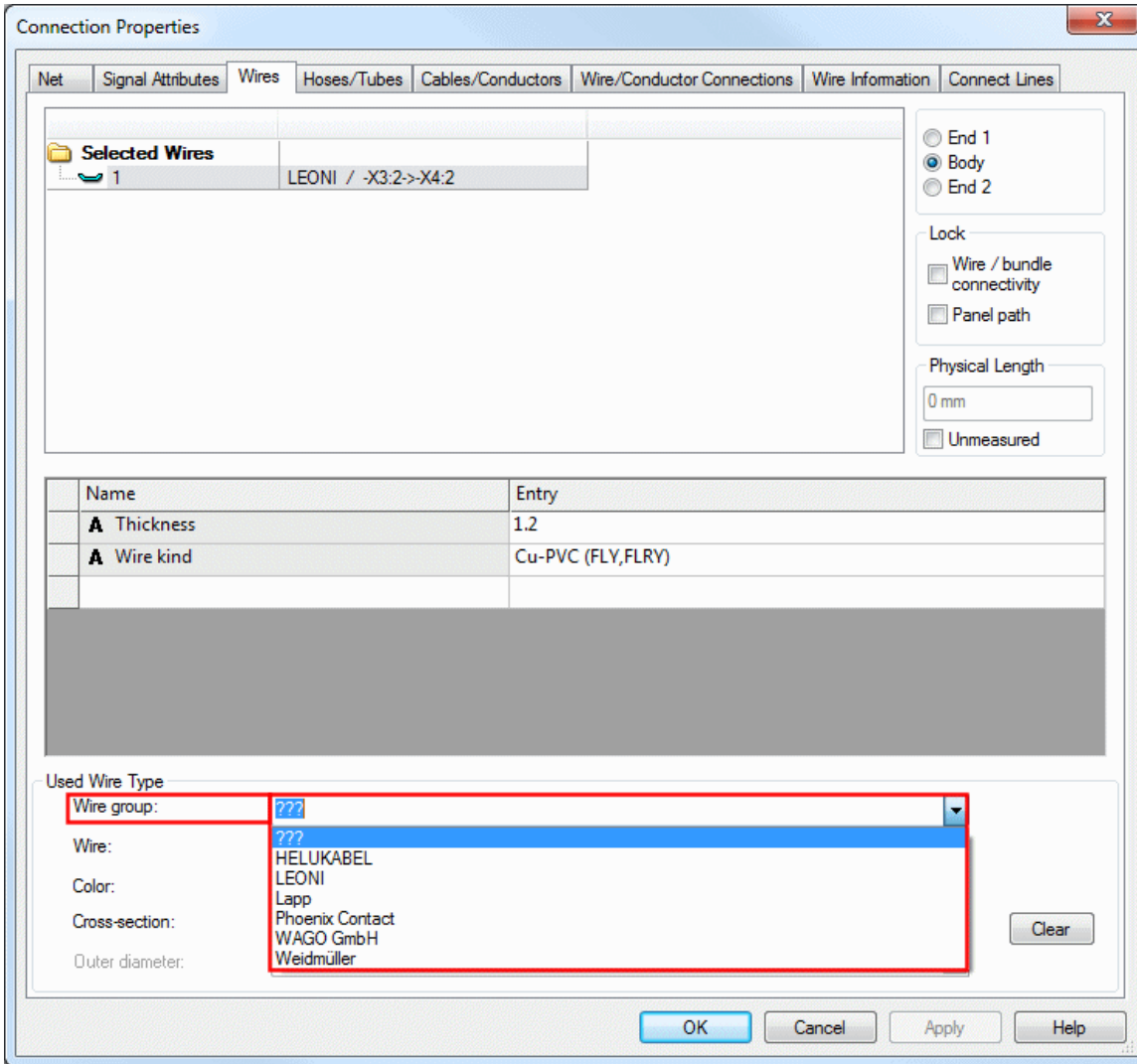
In the **Wire Setting toolbar**:



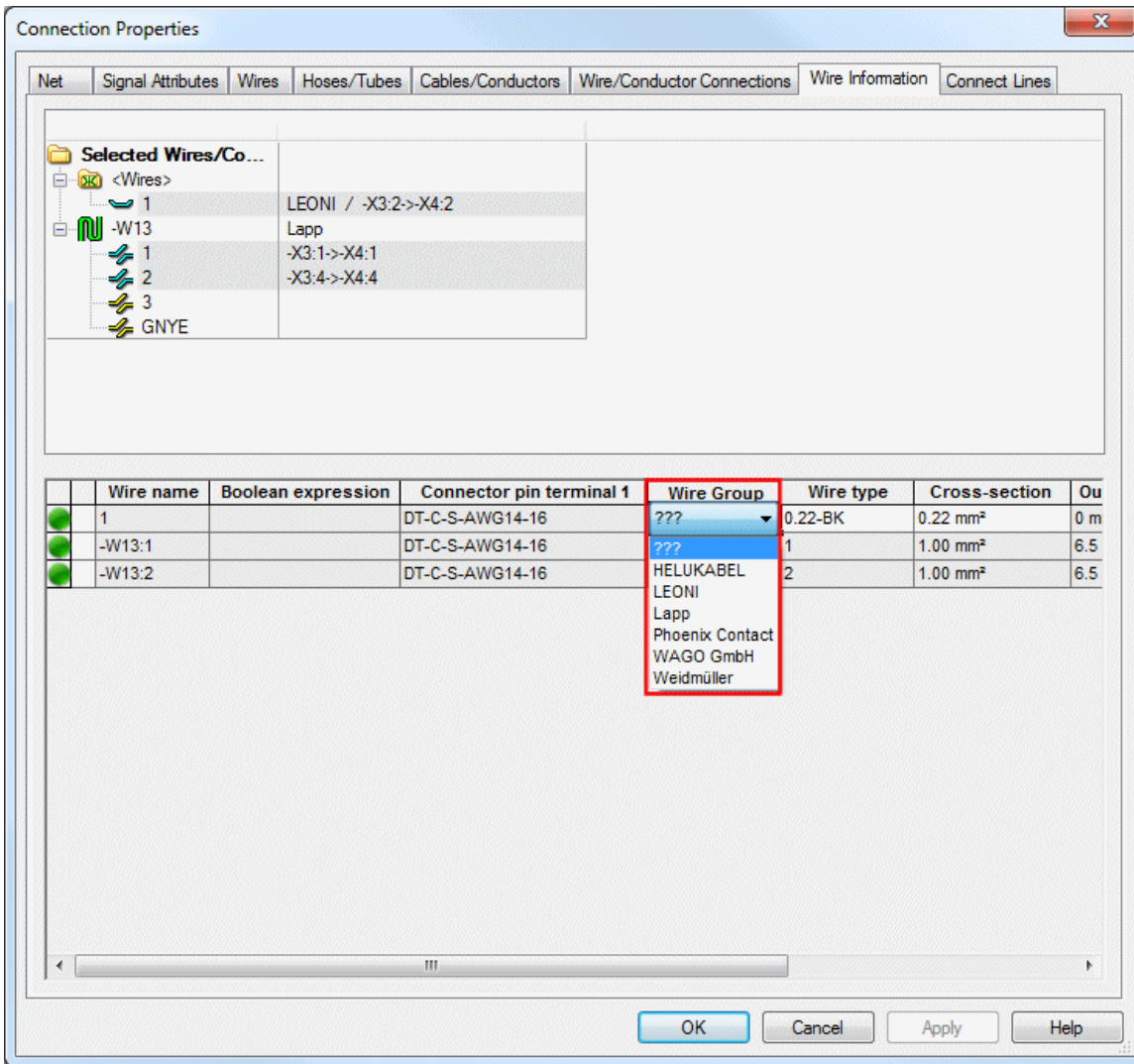
In the Settings **Connection - Conductors /Wires**:



In the **Wires** tab of the Connection Properties:



In the **Wire Information** tab of the Connection Properties:



Reference(s): Designer-18237

### Additional Connector Pin Terminal Information Can Now Be Defined on Symbols

Additional information can now be displayed on the active connector pin terminal of a pin using a text type.

For defining this additional connector pin terminal information there is the new option **Use from active pin terminal** in the **Edit Text Types** dialog in Database Editor Mode:



Text type	Text name	Max. length	Sheet text	Copy from selected sheet	Changeable by script only	Connection target	Use from active pin terminal	Assigned attribute
1	Position	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
3	Pin number	64	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
4	Symbol designation	12	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
5	Component code	64	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
6	PCB grid designation	8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
7	Signal name	200	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
9	Pin symbol text - fix	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
11	Text fix	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
12	Device designation	252	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
13	Text variable	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
14	Higher level assignment	252	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
15	Location	252	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
16	Reference to master	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
18	Reference to slave	255	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
19	External pin number	64	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
20	Drawn by	40	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
21	Document number	20	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
22	Customer	50	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
23	Drawing number	20	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
24	Sheet number	55	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
25	Special note	15	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
26	Date	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
27	Issue	6	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
28	Change	10	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
29	Date of modification	10	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
30	Modified by	6	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
31	Number of sheets	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
32	Device designation	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
33	Function	50	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>

**Note:**

An attribute must also be selected under **Assigned attribute**, which references the information for the active connector pin terminal later. This attribute must have the owner 'Component'.

If the check box **Use from active pin terminal** is checked, the text type is evaluated for...

- connector symbols/block connector symbols (must be connected with the symbol pin)
- normal symbols (must be connected with the symbol pin)
- formboard tables
- attribute text template symbols.

If the pin...

- on the node
- on the attribute text template symbol
- in the row of the Formboard Table

doesn't have a unique connector pin terminal, the text is not filled in.

**Reference(s):** Designer-22515

## COM-Interface

- **New Trigger AfterCreateDevice after Creating a Device**
- **New Trigger AfterDeleteDevice after Deleting a Device**
- **New Triggers for Signals and Signal Classes**
- **Trigger BeforeCheckOutSheet Now Receives a Parameter**
- **COM Command Create2DView Receives New Value for Parameter 'modi' in DeviceInterface**
- **New COM Commands in NetInterface for Changing Signal Names**
- **New Trigger BeforeModifyAttribute**
- **Enhancement to ComponentInterface for Returning the Defined View Symbols of a Component with Their User-Specific View Numbers**
- **New Trigger AfterConnectMUProject**
- **Enhancement to JobInterface for Displaying a List of Drawing Elements in a Predefined Color with PDF Export**
- **Enhancement to SymbolInterface for returning Duplicate Attribute Text Template Symbols whose Polygon Points refer to Origin's Coordinates**
- **Enhancement within the PinInterface to set the maximum number of connections on a pin**

### New Trigger AfterCreateDevice after Creating a Device

The trigger is activated after generating a device.

Can be accessed using the following parameter:

- **1:** ID of the newly generated device

**Reference(s):** Designer-20169

### New Trigger AfterDeleteDevice after Deleting a Device

The trigger is activated after deleting a device.

An array with the GID and object ID is returned as a parameter:

- **Array(0):** GID of the deleted device
- **Array(1):** Object ID of the deleted device

#### Note:

Afterward, the returned IDs are no longer valid and cannot be reused within the COM interface.

These IDs can be used to update external references, e.g. in Excel files or databases.

#### Example:

Start

```
Sub Start()
    Set e3 = CreateObject("CT.Application")

    args = e3.GetScriptArguments
    For i = 0 to UBound(args)

        MyArray = Split(args(i), ",", -1, 1)
```

```

        e3.PutInfo 0, "Item has been deleted: "
        e3.PutInfo 0, "      GID: " & MyArray(0)
        e3.PutInfo 0, "      ObjectID: " & MyArray(1)

    Next
    Set e3 = Nothing
End Sub

```

**Reference(s):** Designer-25495

## New Triggers for Signals and Signal Classes

There are the following new triggers:

### AfterCreateSignal

The trigger is executed after generating a signal or signal class.

### AfterModifySignalName

The trigger is executed after changing the name of a signal or signal class.

### AfterModifyReferenceItemsOfSignal

The trigger is executed after adding or deleting objects (e.g. pins, nodes, net nodes, core/wires) of a signal.

All triggers can access the following arguments:

- **1:** all signal IDs and signal class IDs

### Example:

Start

```

Sub Start()
    Set e3 = CreateObject("CT.Application")
    Set prj = e3.CreateJobObject()
    Set sig = prj.CreateSignalObject()
    Set sigClass = prj.CreateSignalClassObject()

    args = e3.GetScriptArguments
    For i = 0 to UBound(args)
        signalItem = sig.SetId( args(i) )

        If signalItem = 0 Then
            sigClass.SetId args(i)
            e3.PutInfo 1, "Signal class: " & sigClass.GetName() & " changed"
        Else
            e3.PutInfo 1, "Signal: " & sig.GetName() & " changed"
        End If
    Next
    Set e3 = Nothing
End Sub

```

**Reference(s):** Designer-13562

### Trigger BeforeCheckOutSheet Now Receives a Parameter

The trigger **BeforeCheckOutSheet** receives an argument as a parameter specifying which type of checkout shall be performed:

- **1** for Read/Write
- **2** for ReadOnly

**Reference(s):** Designer-13930

### COM Command Create2DView Receives New Value for Parameter 'modi' in DeviceInterface

In the **DeviceInterface** the following COM commands have been changed:

- **ret = dev.Create2DView ( modi, name, symbol, position, before )**
  - o **modi 0x000** Previously, the value was always 0. The display in the "base sheet" disappears as soon as the component is displayed on a 2D view sheet.
  - o **modi 0x001** When generating a 2D view, the component is still visible on the "base sheet", i.e. the cell appears twice.
  
- **ret = dev.GetPanelLevel**
  - o The return value has been changed:
  - o **ret** Returns the level **same level for all cells**, otherwise -1.

**Reference(s):** Designer-19431

### New COM Commands in NetInterface for Changing Signal Names

In the **NetInterface** there are the following new COM commands:

- **ret = net.GetSignalName**
  - o **ret** Returns the signal name of the current net.
  
- **ret = net.GetTranslatedSignalName**
  - o **ret** Returns the translated text for the signal name. If there is no translation, the internal signal name is returned.
  
- **ret = net.SetSignalName ( name )**
  - o **name** New signal name of the net
  - o **ret** The return value is 1 if successful, otherwise 0.
  
- **ret = net.SetSignalNameOnLocalNet ( name )**

- o name New signal name of the net. If additional nets are connected by references, the signal is not changed there.
- o ret The return value is 1 if successful, otherwise 0.

In the **ConnectionInterface** the following COM command has been changed:

- ret = con.SetSignalNameOnLocalNet ( name )
  - o This command is replaced by the COM command with the same name in the NetInterface.
  - o It now behaves like con.SetSignalname(name). The specified signal is transferred over the entire net. The signal is also inherited over references.

**Reference(s):** Designer-25939

### New Trigger BeforeModifyAttribute

The trigger is executed if one of the attributes to be monitored is changed/deleted or added to an object. The trigger is not executed if the owner is deleted (e.g. device is deleted).

If no attributes to be monitored are specified for the trigger definition, the trigger is not executed. The attributes to be monitored for the trigger definition are enclosed in angled brackets and separated by commas.

**Example:** D:\exampleTrigger.vbs<Attribute\_1,Attribute\_2,...>

Arguments are assigned to the trigger in the form of a tuple; each tuple can be interpreted as follows:

- Attribute owner (e.g. sheet, device...)
- Attribute name
- Old attribute value
- New attribute value

### Example of evaluating the tuple:

```
Set e3 = CreateObject( "CT.Application" )
Set prj = e3.CreateJobObject()

args = e3.GetScriptArguments
cnt = 0

For Each arg in args
  modResult = cnt Mod 4
  cnt = cnt + 1

  If modResult = 0 Then
    e3.PutInfo 0, "Attribute owner: " & arg
  ElseIf modResult = 1 Then
    e3.PutInfo 0, "Attribute name: " & arg
  ElseIf modResult = 2 Then
    e3.PutInfo 0, "Old value: " & arg
  ElseIf modResult = 3 Then
    e3.PutInfo 0, "New value: " & arg
  End If
Next
```

```
Set prj = Nothing
Set e3 = Nothing
```

The trigger can set a return value using the function **e3.SetTriggerReturn()**. The following return values are possible:

Value	Symbol Name	Meaning for BeforeModifyAttribute
1 (Default)	vbOK	The attribute modifications have been implemented.
6	vbYes	
4	vbRetry	All modifications are rejected.
2	vbCancel	
3	vbAbort	
5	vbIgnore	
7	vbNo	

**Note:** No other values can be set with **e3.SetTriggerReturn()** or returned with **e3.GetTriggerReturn()**.

**Reference(s):** Designer-26028

### Enhancement to ComponentInterface for Returning the Defined View Symbols of a Component with Their User-Specific View Numbers

There is the following new COM command in the ComponentInterface:

- **ret = comp.GetViewDefinitions ( viewDefinitions )**
  - o **ret** Number of definitions returned
  - o **viewDefinitions** Two-dimensional array with the existing view information
    - 1. Column = Symbol ID
    - 2. Column = User-defined view number; 0 if no user-defined view number is defined.

**Example:**

Symbol ID	User-Defined View Number
35876	0
35446	5
34923	33

The return value **ret** = 3

**Reference(s):** Designer-26026

### New Trigger AfterConnectMUProject

The trigger is executed when first opening a multi-user project and accessing a multi-user project.

The trigger is always called up with 3 parameters:

- 1. Parameter: The following values are possible:
  - o "Create" when creating a multi-user project
  - o "Open" when opening a multi-user project
  - o "Delete" when deleting a multi-user project
- 2. Parameter: The following values are possible:
  - o **1** for the first user, which opens the multi-user project
  - o **0** for all others with access
- 3. Parameter: Name of multi-user project

**Reference(s):** Designer-16579

### Enhancement to JobInterface for Displaying a List of Drawing Elements in a Predefined Color with PDF Export

In the **JobInterface** there is the following new COM command:

- **ret = job.ExportPDFEx ( file, shtids, options, itemListType, items, alternativeColour, imageBrightness [,password] )**
  - o The parameters 'file, shtids, password' are the same as **job.ExportPDF**
  - o **options** has changed as follows (this is also valid for **job.ExportPDF**):
    - **&h0000** Default Acrobat version (currently 8)
    - **&h0004** n/a: Version 4 no longer supported

- New:
  - **&h0008** Acrobat Version 8
  - **&h0009** Acrobat Version 9
  - **&h000A** Acrobat Version 10
- o The following parameters have been added:
  - **itemListType Flag** How the objects shall be output:
    - **0** All objects, which are **NOT** in the items list, receive the specified color (alternative color) and (brightness)
    - **1** All objects in the item list receive the specified color (alternative color) and brightness.
  - **items** List with objects (object numbers) from the E3 project
  - **alternativeColour** Color, which the objects shall receive (0 up to and including 255).
  - **imageBrightness** Brightness, which the images shall have. This can be a whole number between -100 and 100 (the smaller the number the darker; the larger the number the brighter).
- o **ret** The following return values are possible:
  - **1** Export successful
  - **-1** Unknown error
  - **-2** No project opened
  - **-3** Missing license
  - **-4** Invalid file path or missing authorization
  - **-5** Invalid sheet
  - **-6** Invalid Acrobat version
  - **-7** Invalid object list type
  - **-8** Invalid color (outside of value range)
  - **-9** Invalid luminosity (outside of value range)
  - **-10** Invalid password

**Reference(s):** Designer-25588

### Enhancement to SymbolInterface for Returning Duplicate Attribute Text Template Symbols whose Polygon Points refer to Origin's Coordinates

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

- **ret = sym.GetPlacedPolygon ( npnts, xarr, yarr )**
  - o Valid for duplicate attribute text template symbols
  - o **npnts** Number of polygon points, e.g. 4 points for a rectangle
  - o **xarr, yarr** Arrays with x/y coordinates of polygon points; arrays start with 1
  - o **ret** The return value is 1 if successful, otherwise 0.

**Reference(s):** Designer-24834



## Enhancement within the PinInterface to set the maximum number of connections on a pin

There is the following new COM command in the **PinInterface**:

- **ret = pin.SetPhysicalMaxConnections( conncount )**
  - o Sets the max. number of connections for this pin
  - o **conncount** is the max. number of connections for this pin
  - o Possible return values for **ret** see above list
    - **0** = Number has been set successfully
    - **-1** = Number hasn't been set
    - **-2** = No project opened
    - **-3** = Invalid Pin
    - **-4** = Invalid Number

### Restrictions:

- The pin must have a component assigned.
- The pin must have a physical description.
- When the **Use signal equivalence (to select connect pin)** option is checked in **Settings... -> Connection -> Conductors/Wires**, a physically described pin must exist with the same pin index.

### Note:

- The highest possible number is set to 999

**Reference(s):** Designer-26833

## Connections/Buses, Signals/Logic Lines, Supply - Overview

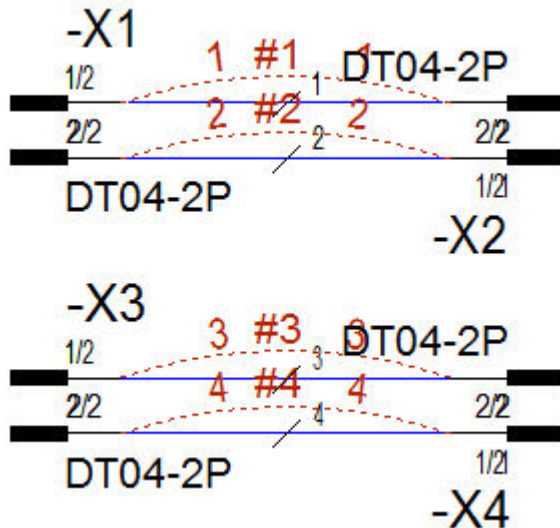
- **Conductor Logic Lines**
  - **Conductor Logic Lines Can Now Be Displayed for Selective Signals**
  - **Logic Lines of Not Connected Conductors Can Now Be Displayed as Arcs**
- **New Functionality in E3 'Reorder Conductors'**
- **New Functionality in E3 'Signal Format'**
- **Misc**
  - **Alternative Template Symbol Can Now Be Defined in the Settings**
  - **New Check Box in Settings for Deleting Signals of Not Connected View Symbols**
  - **Define whether the Signal on a Pin Shall Be Removed when Disconnecting Conductors/Wires on a Pin**

## Logic Lines

- **Conductor Logic Lines Can Now Be Displayed for Selective Signals**
- **Logic Lines of Not Connected Conductors Can Now Be Displayed as Arcs**



### Conductor Logic Lines Can Now Be Displayed for Selective Signals

Previously, when the check box **Display laid conductors also** in the Settings under **Connection - Cable/Conductor Logic Lines** in the section **Cable/Conductors** ) was checked, all conductor logic lines project-wide were displayed:



With this new option the user can now display the individual conductor logic lines of a signal on a pin or optionally all signals of a device. This enables the signal to be traced more easily over several sheets.

To do so, there are the following new icons in the **Display** toolbar:


-  **Display signal(s)**  
or alternatively by pressing the **F6** key
-  **Reset display signal(s)**  
or alternatively by pressing the key combination **SHIFT+F6**

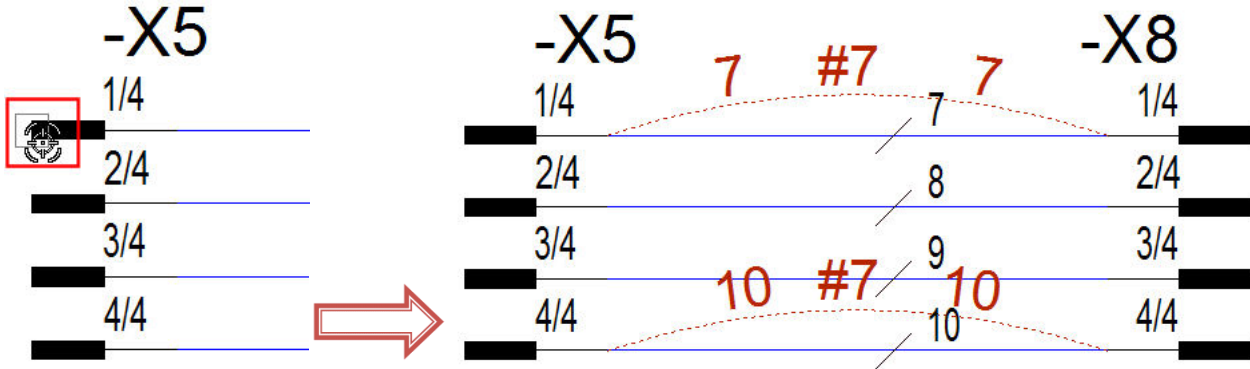
#### Note:

- At least one wire must be placed on the connection line.
- This functionality is inactive on panel sheets.
- When closing a project or reading/writing an E3D design file, already displayed conductor logic lines are not saved.

## Display Conductor Logic Lines of Signal on a Pin

To display the conductor logic lines of a signal on a pin:

- left-click on the icon  **Display signal(s)** or press the **F6** key,
- left-click with the mouse pointer on the area in front of the individual pin (all pins which are connected to a conductor/wire):



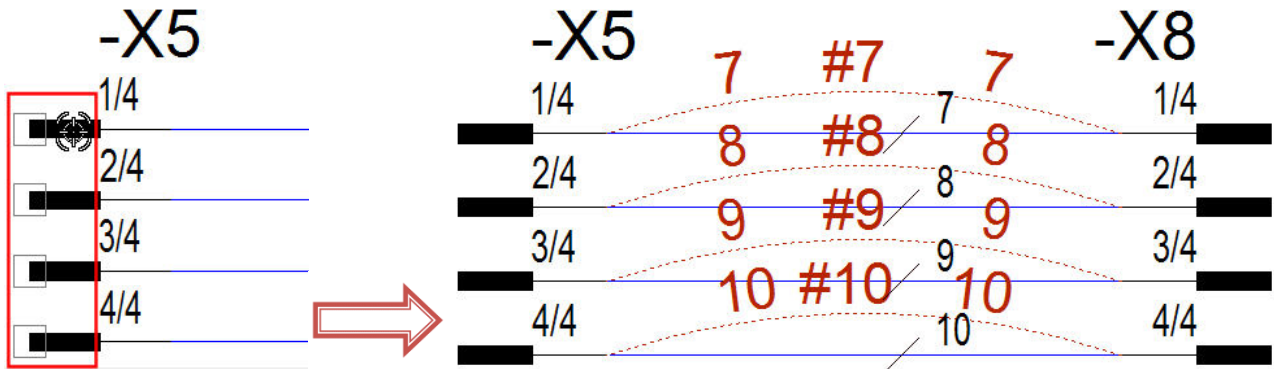
### Note:

- The conductor logic lines of the signals used (in the example the signal **#7**) are displayed project-wide.
- This functionality is also available in the Device Tree and the Pin Table:
  - o right-click on the desired pin symbol (multi-select also possible) and then select the context menu command **Highlight Conductor Logic Lines of Signal**

## Display Conductor Logic Lines of All Signals on a Device

To display the conductor logic lines of all signals on a device:

- left-click on the icon **Display signal(s)**  
or press the **F6** key,
- then left-click on a device symbol using the mouse pointer:



### Note:

- The conductor logic lines of the signals used (in the example the signals **#7**, **#8**, **#9**, **#10**) are displayed project-wide.
- This functionality is also available in the Device Tree and Device Table:
  - right-click on the desired device-symbol (multi-select also possible) and then select the context menu command **Highlight Conductor Logic Lines of Signal**

## Display Conductor Logic Lines of Signal using Signal Tree

To display the conductor logic lines of a signal using the signal tree:

- right-click on the desired symbol in the signal tree (multi-select also possible),
- then select the context menu command **Highlight Conductor Logic Lines of Signal**:

**Note:** The conductor logic lines of the selected signal are displayed project-wide.

## Suppress Conductor Logic Lines of All Signals

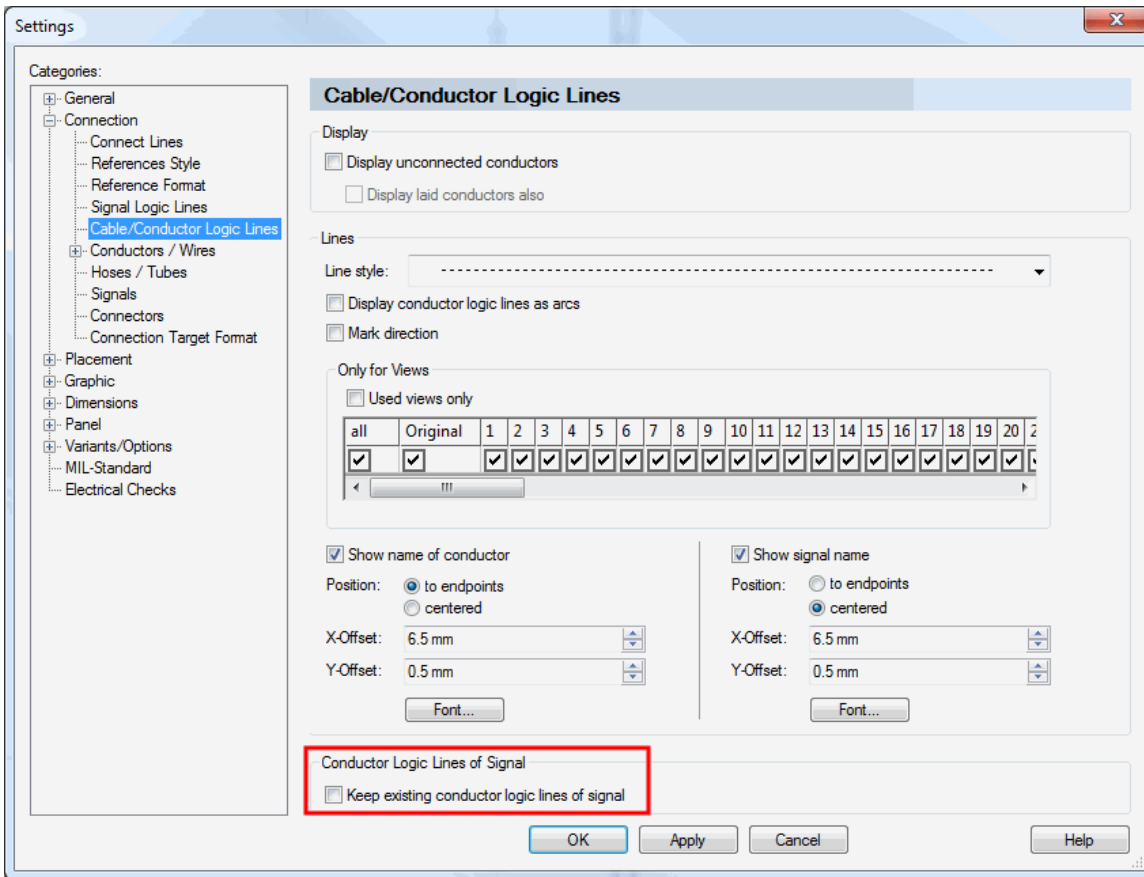
To suppress the conductor logic lines of all signals:

- left-click on the icon **Reset display signal(s)**  
or press the key combination **SHIFT+F6**

## Display Already Displayed Conductor Logic Lines

There is also the possibility to display already displayed conductor logic lines.

To do so, activate the new check box **Keep existing conductor logic lines of signal** in the section **Conductor Logic Lines of Signal** in the Settings under **Connection - Cable/Conductors Logic Lines**:



## Cable/Conductor Logic Lines

### Conductor Logic Lines of Signal

#### Keep existing conductor logic lines of signal

If this option is checked, the displayed signal logic lines are retained.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

KEEP\_EXISTING\_CORE\_LOGIC\_LINES\_OF\_SIGNAL

This check box is *user-specific* in a multi-user project.

For this there are the following new COM commands in the **SignalInterface** and **JobInterface**:

- **sig.HighlightCoreLogicLinesOfSignal ()**
  - o Display all conductor logic lines of a signal
- **job.ResetHighlightCoreLogicLinesOfSignals ()**

- o Reset the display of all conductor logic lines of all signals

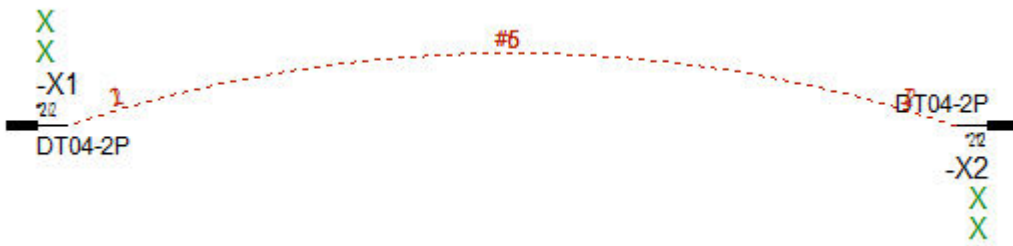
**Reference(s):** Designer-25376

### Logic Lines of Not Connected Conductors Can Now Be Displayed as Arcs

Previously, the logic lines of not connected conductors (without connection line) were always displayed as straight lines:

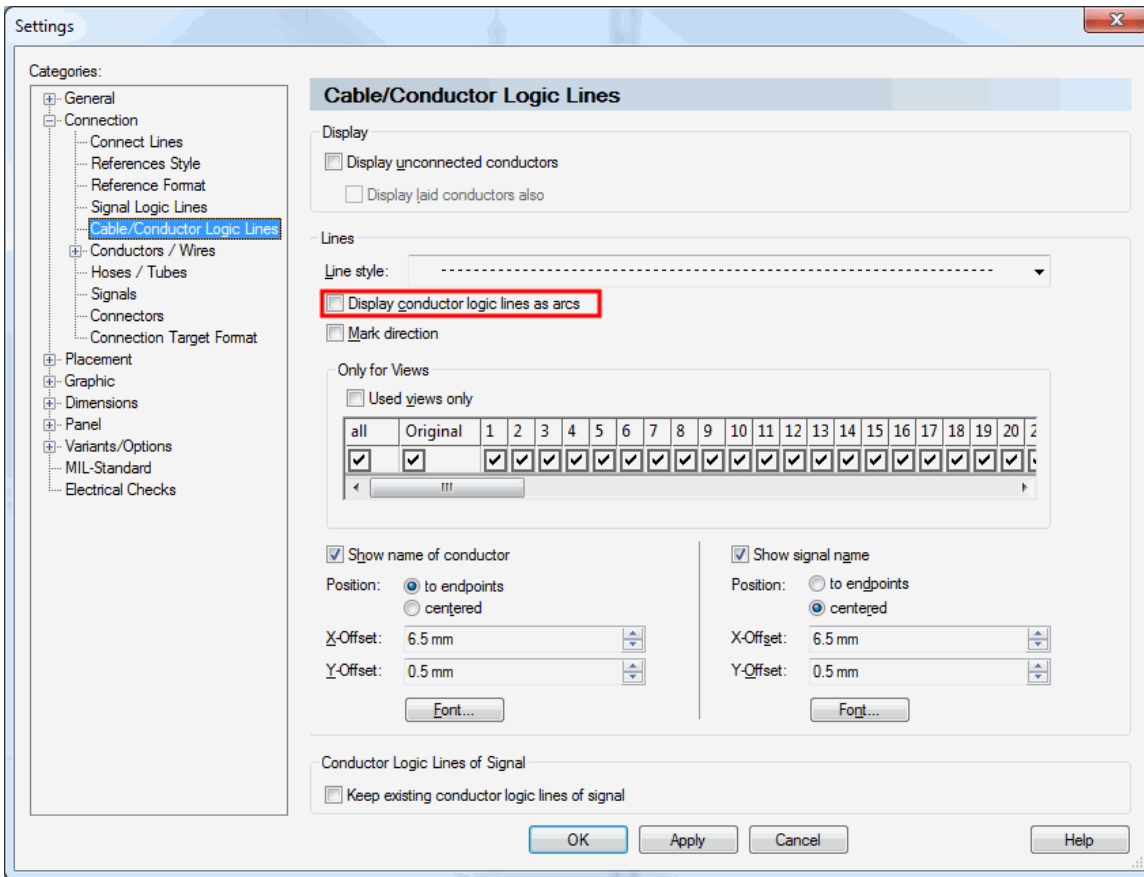


Now these logic lines can also be displayed as arcs:



To do so, there is the new check box **Display conductor logic lines as arcs** in the Settings under **Tools -> Settings -> Connection -> Cable/Conductor Logic Lines** in the section **Lines**.

**Note:** The section **Lines** has been enhanced for the already existing options **Line style** and **Mark direction**.



## Cable / Conductor Logic Lines

### Display conductor logic lines as arcs

When this option is active, all logic lines of unconnected conductors are shown as arcs.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used: AIRLINES

This check box is *user-specific* in a multi-user project.

**Reference(s):** Designer-25502



## Functionality 'Reorder Conductors'

- **New Functionality in E3 'Reorder Conductors'**

### New Functionality in E3 'Reorder Conductors'

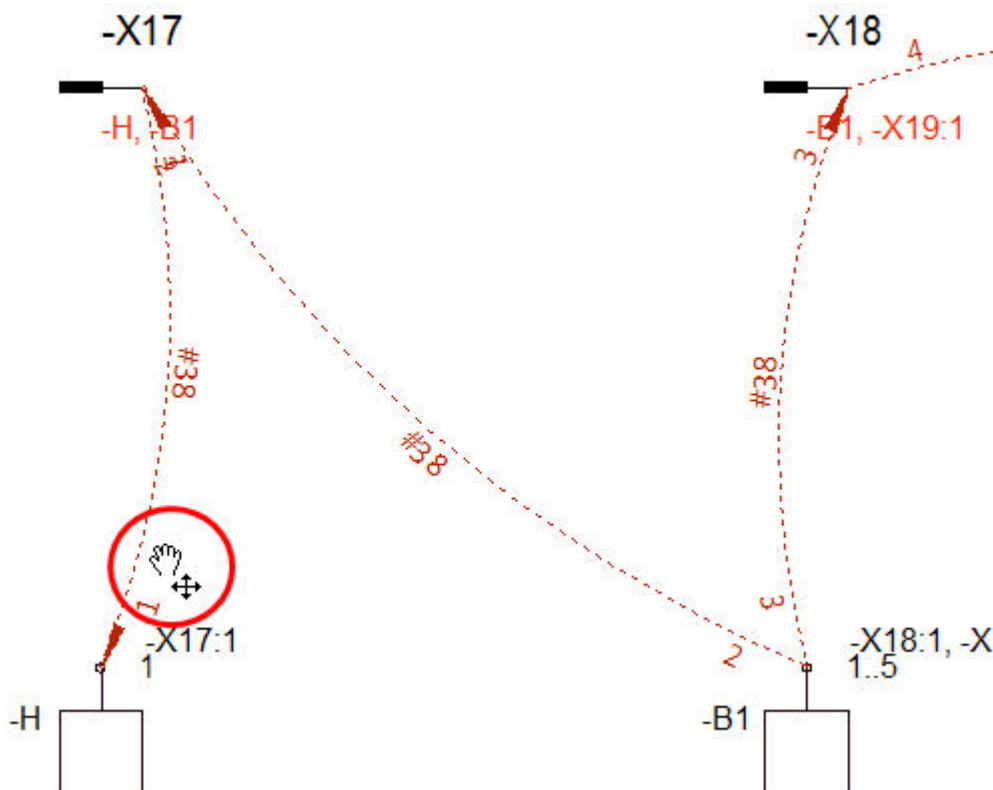
Previously, already connected conductors could only be reordered by executing the command **Reconnect Conductors**. Now conductors can be reordered within a connected conductor logic line segment (multiple conductors within a connection net) using the mouse pointer.

To do so, there is the new **Reorder conductors** icon in the Toolbar - Category **Connect**.

**Note:** The conductor logic lines must be visible for this function.

### How to change the connection order of conductors:

- Left-click on the icon **Reorder conductors**,
- the mouse pointer appears in the form of an open hand in the workspace,
- click on the desired end of the conductor logic line which shall be reconnected:

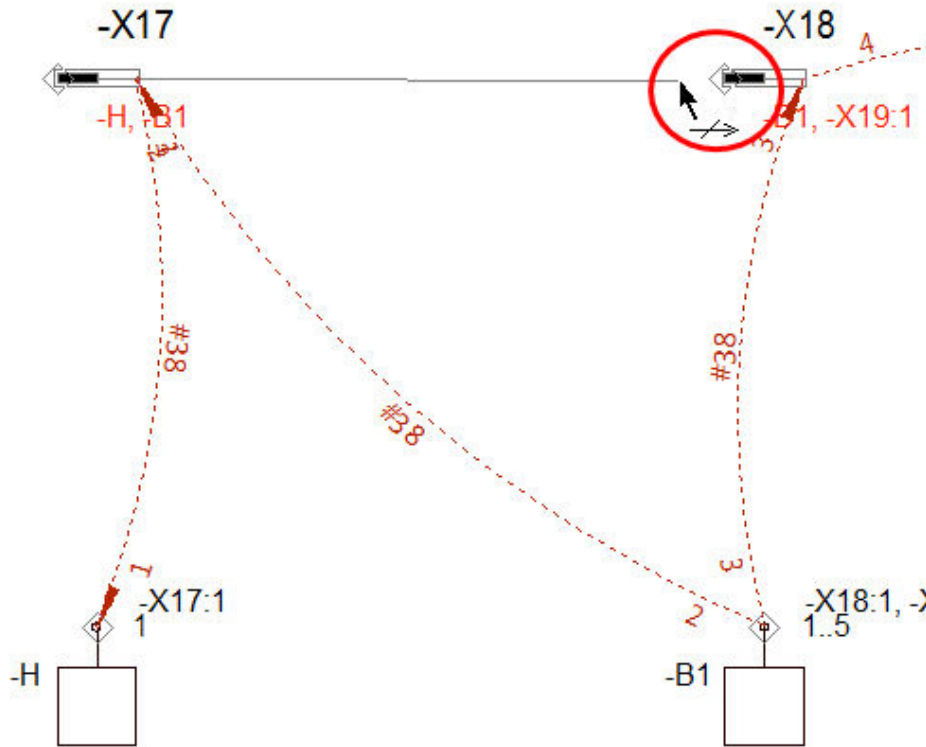


**Note:** The hand closes as soon as the core logic line is positioned within the snap area:



- Left-click on the pin where the conductor shall be reordered.

- All possible pins are identified by hash key symbols "#":



- **Note:**
- If necessary, additional conductor logic lines can be reordered accordingly after executing the command.
- After reordering all desired core logic lines, right-click anywhere within the work space
- the following context menu commands are available:
  - o **End**  
The command ends but keeps previous changes.
  - o **Abort**  
The command ends and rejects all changes made.  
The command can be terminated with the **ESC** tab.

**Note:**

- Once the pin no longer has contact to the previous core route, an appropriate message appears.
- If there is a conflict with the connection restrictions when connecting a core/wire to pin (e.g. max. number of cores/wires), an appropriate warning message appears in the status bar. The pin is marked with an exclamation point while the command is open.
  - If pins violate the pin restrictions, an appropriate warning message will appear after executing the command **Reorder conductors**.
  - The command **Reorder conductors** can only be canceled if the pin restrictions have been considered. The command can be undone optionally.
- While executing the command **Reorder conductors**, the context menu command **Open End** can be selected after right-clicking anywhere within the workspace. Then, the conductor logic line has an open end (temporarily).

**Reference(s):** Designer-25415

## Functionality 'Signal Format'

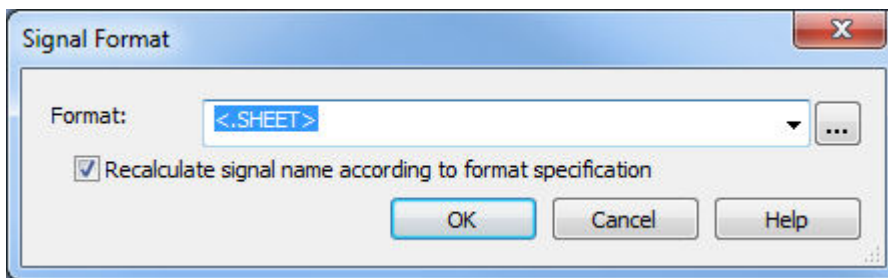
- [New Functionality in E3 'Signal Format'](#)

### New Functionality in E3 'Signal Format'

In **E<sup>3</sup>.series** a signal placeholder for the signal name can now be displayed on the signal. The signal name will automatically be generated either for one-time use or for relevant changes according to the format description after having placed a signal placeholder. Placing the signal placeholder occurs with the new functionality **Signal Format**.

How to place a signal placeholder in a signal text:

- right-click on the desired connection line and select the command **Signal Format...** in the displayed context menu  
or  
right-click with the mouse button on the desired signal within the signal tree and select the command **Signal Format...** in the displayed context menu  
or  
right-click on the desired symbol in the working area and select the command **Signal Format...** in the displayed context menu.
- the **Signal Format** dialog opens:



## Signal Format

### Format:

The signal placeholder can be entered on a signal here.

The following signal placeholders are available:

- **<.SHEET>**  
If this signal placeholder is specified, the sheet name on which the reference object is placed appears as the signal name.  
**Note:** If the sheet name is numeric, it will be displayed with at least three digits and zeros at the beginning.
- **<.GRID>**  
If this signal placeholder is specified, the current sheet coordinates on which the reference object is placed appear as the signal name.  
**Note:** The pin on the first sheet in the project structure with minimum **X** and maximum **Y** coordinate will be used as the reference object.  
The sheet coordinates are displayed according to the Setting **Format:** (in the section Settings under '**General -> Display -> Sheet Reference-> Format**)

- **<Attribute>**

If this signal placeholder is specified, a signal or a sheet attribute appears when available.

**Note:** The attribute must contain 'Signal' or 'Sheet' as the valid owner. If both 'Signal' and 'Sheet' are available, the attribute is determined by the signal.

Already created signal format combinations can be selected in the drop-down list.

**Note:**


- Additionally permitted characters can be placed before and after signal placeholders. The following characters are not allowed **< > , { }**.

**Example:** 123\_<.SHEET>-ABC

- Up to eight signal placeholders can be used for a signal. In doing so, a signal placeholder may not appear repeatedly.
- A signal placeholder must be unique.

**Examples of faults:**

- Different attributes used in the signal format have the same attribute value.  
(**Example:** *Attribute 1 with entry A - Attribute 2 with entry A*)
- The signal name generated by the signal format is already used as signal name.
- **Exception:** The signal name generated by the signal format <.GRID> from the sheet coordinates information is already used in another sheet. In this case the next possible free sheet coordinate will be used  
(**Example:** Instead of A1 -> A2).
- Using **<no entry>** resets the signal format on the signal.
- When determining the reference object, inactive objects are also considered for variants/options.

Clicking the button  opens the **Translation Table** dialog. This dialog displays all entries of the different languages.

The desired text is inserted.

**Recalculate signal name according to format specification**

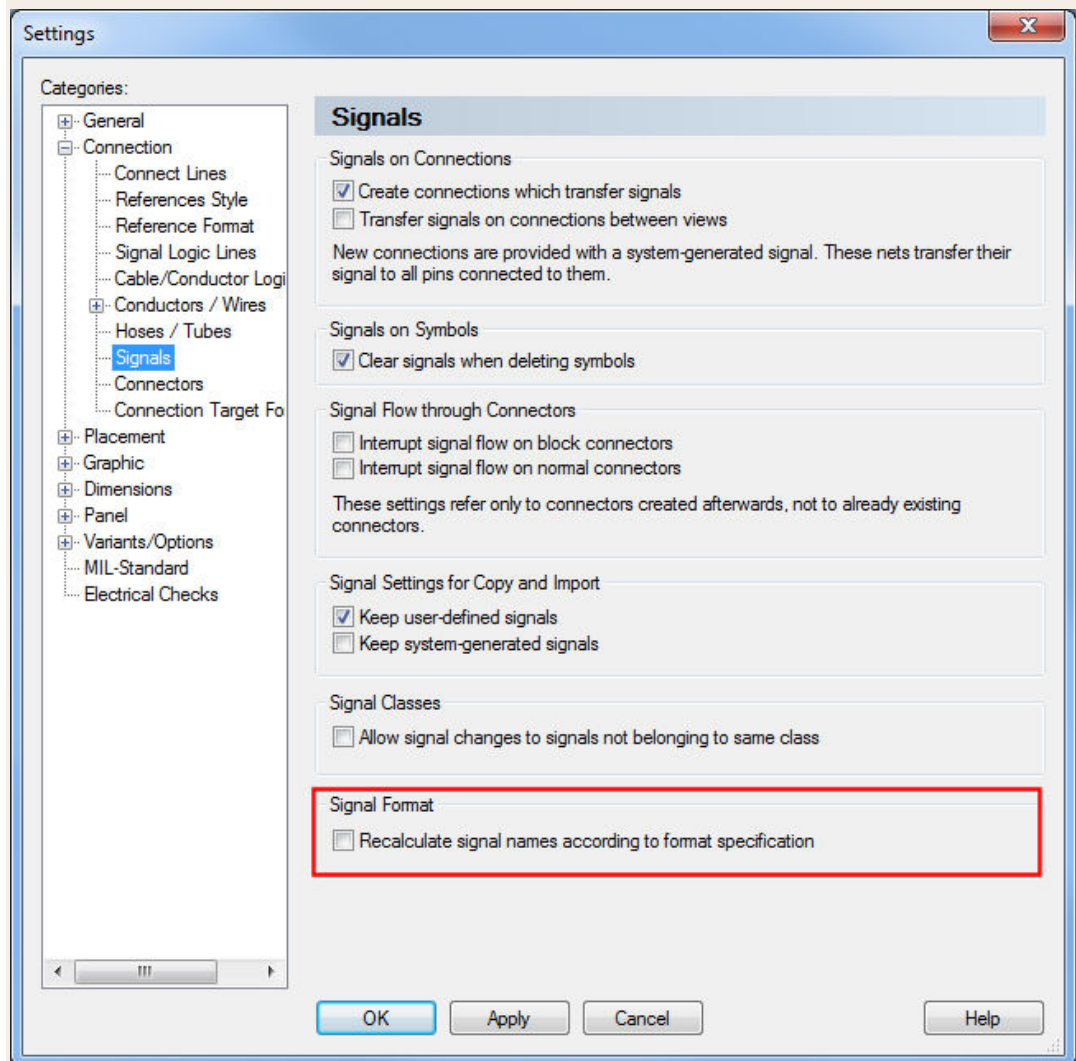
If the check box is active, all relevant changes to the signal formats will be recalculated automatically. In doing so, it's possible to control whether modifications to each signal can be individually updated or not.

Relevant changes are:

- Moving a sheet within the project structure if the reference object is changed.
- Moving connection lines or symbols if the reference object is changed.
- Changing an attribute value which is used as the signal format.
- Deleting signals
- Deleting or moving reference objects

A default value can be defined using the new check box **Recalculate signal names according to format specification** in the Settings (under

'Connection -> Signals -> Signal Format):

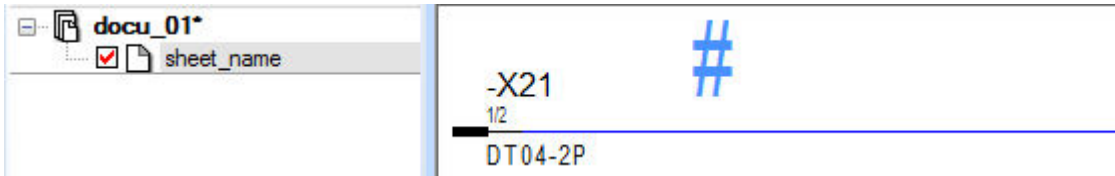


**Note:** It's also possible to set signal placeholders in texts of the type **TTYP\_SIGNAL** (#7) and **TTYP\_CONNECTION\_SIGNAL** (#1007) as follows:

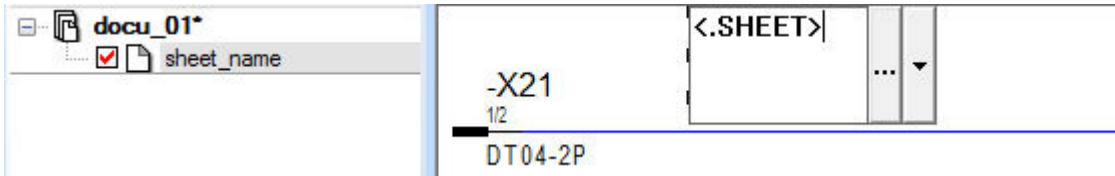
- by selecting the signal texts and clicking **F2** afterward, or
- right-clicking on the signal text and selecting the command **Text Properties...** in the displayed context menu.  
Select the tab **Text** in the **Text Properties** and insert the desired signal placeholder under **Text**.

Example for the signal placeholder **<.SHEET>**:

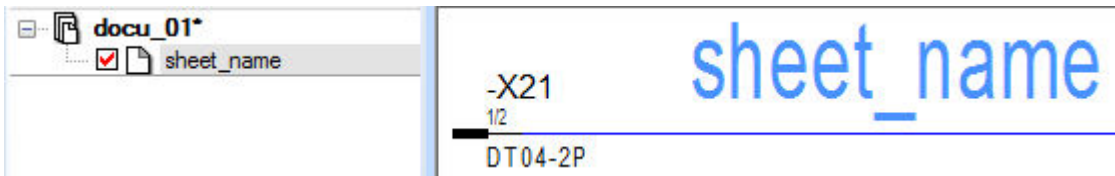
1. This example shows the sheet name 'sheet\_name':



2. The signal placeholder <.SHEET> is defined on the signal:

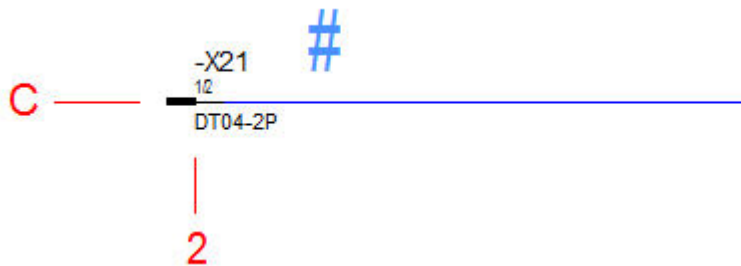


3. The signal now receives the current sheet name 'sheet\_name' as signal name:

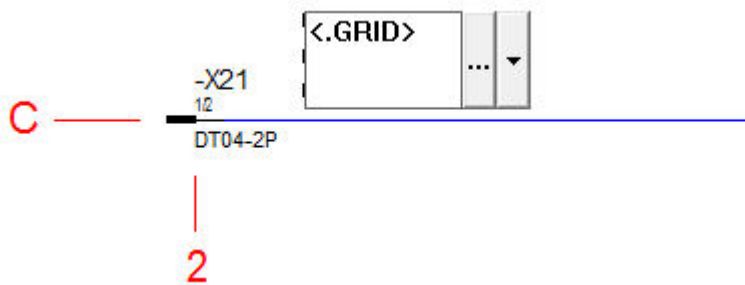


Example for the signal placeholder <.GRID>:

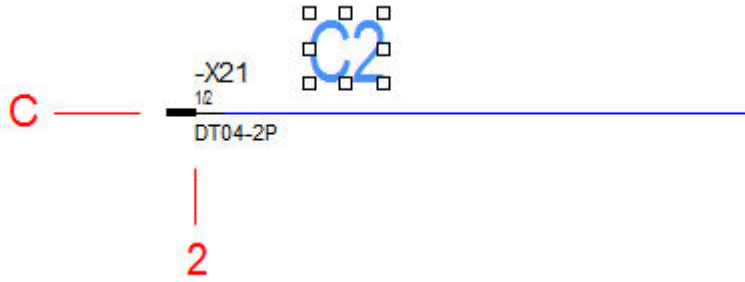
1. 'C2' is used as reference value for the sheet coordinate in following example :



2. The signal placeholder <.GRID> will be defined on signal:

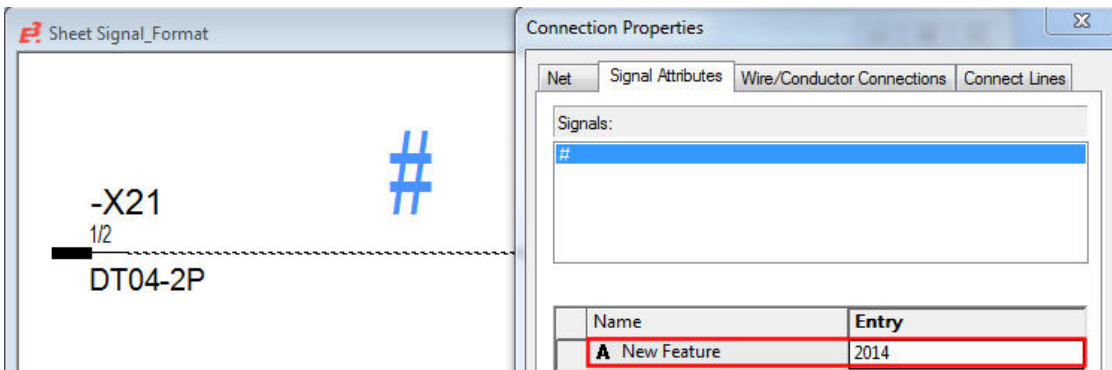


3. Now the signal receives the sheet coordinate 'C2' as signal name:

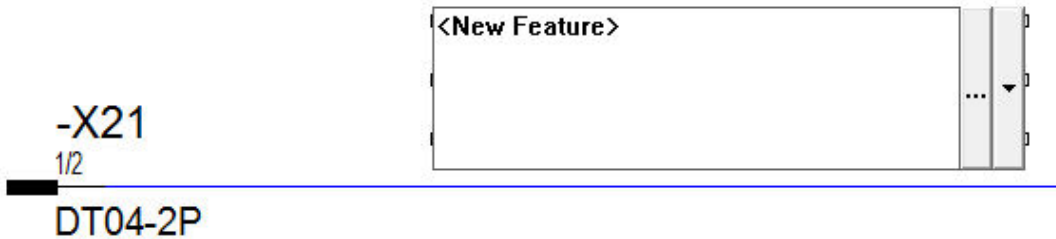


Example for the signal placeholder <ATTRIBUTE>:

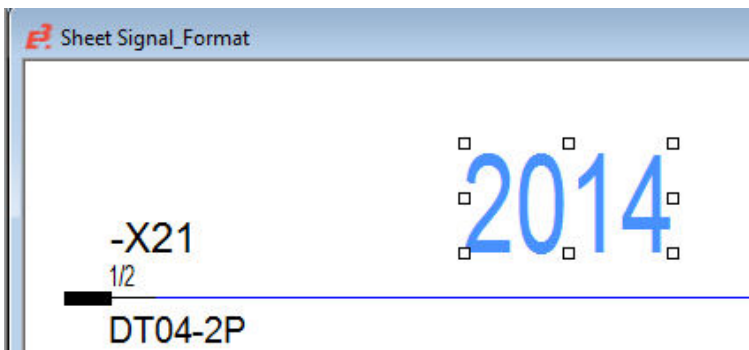
1. The signal attribute 'New Feature' is used as attribute with the Entry 2014':



2. The signal placeholder <New Feature> (<Name of attribute>) is defined on the signal:



3. The signal now receives the Entry 2014' from attribute Demo' as the signal name:



Reference(s): Designer-25414

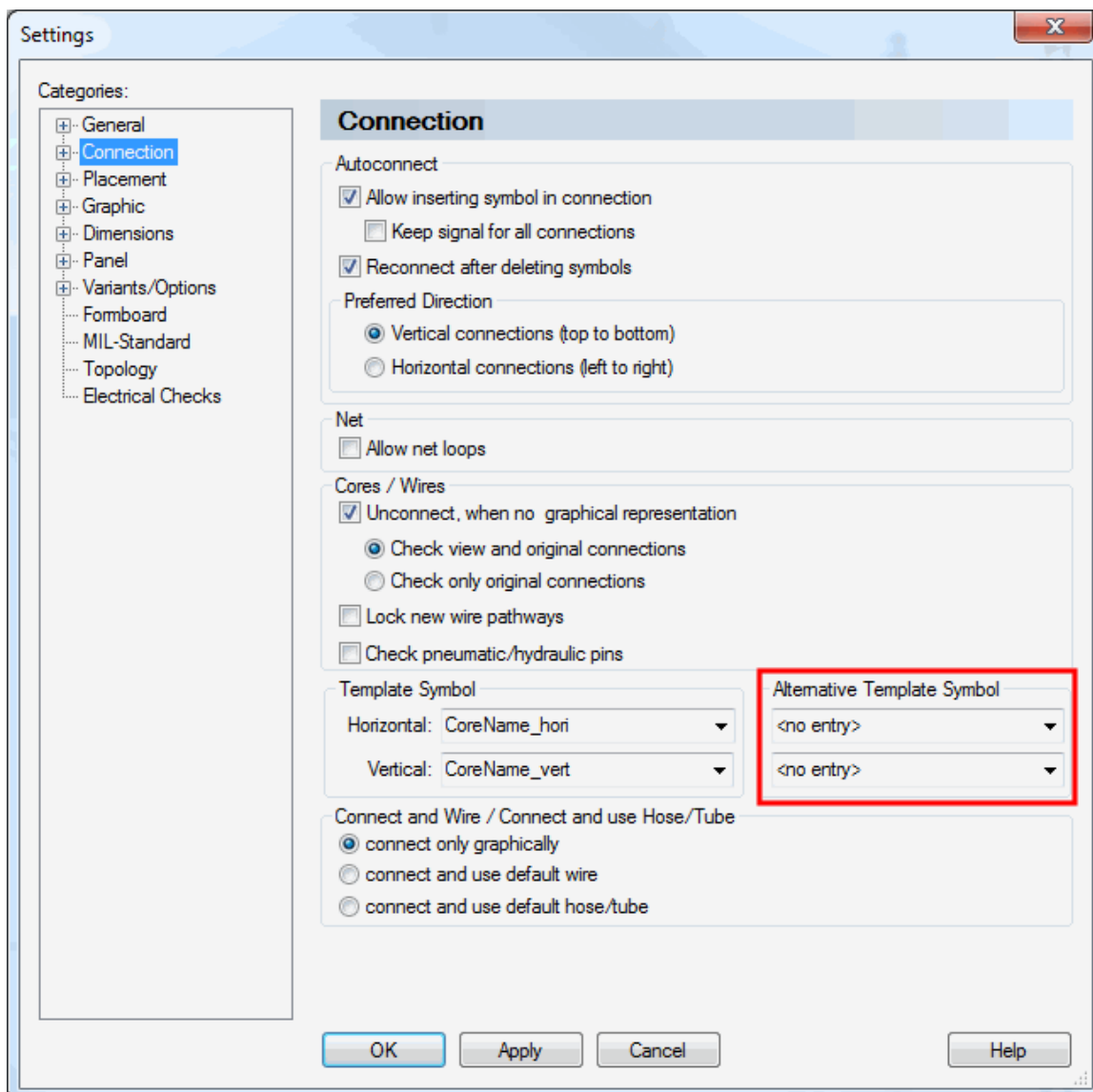
## Misc

- [Alternative Template Symbol Can Now Be Defined in the Settings](#)
- [New Check Box in Settings for Deleting Signals of Not Connected View Symbols](#)
- [Define whether the Signal on a Pin Shall Be Removed when Disconnecting Conductors/Wires on a Pin](#)

### Alternative Template Symbol Can Now Be Defined in the Settings

In **E3.series** an alternative template symbol can now be defined. When placing a conductor template symbol on a connection line, the alternative template symbol can be used.

To do so, there is the new section **Alternative Template Symbol** in the category **Connection**. A horizontal or vertical alternative template symbol can be defined here using the respective drop-down list:





## Connection

### Alternative Template Symbol

This option determines, which alternative template symbol shall be used when placing a conductor template symbol or a conductor on a connection (dragging & dropping a conductor). Depending on whether the connection line is horizontal or vertical, the corresponding symbol is placed.

**Note:** By pressing the **Alt** or **Alt Gr** key the alternative template symbol can be used for the following commands:

- o when placing a connection template symbol (symbol `Place connect template symbol` in the Placement toolbar)
- o when placing conductors/wires

For the COM methods `prj.GetSettingValue` and `prj.SetSettingValue` the following `setting_name` can be used:

- Horizontal: `WCOUNT_SYMBOL_HOR_ALT`
- Vertical: `WCOUNT_SYMBOL_VER_ALT`

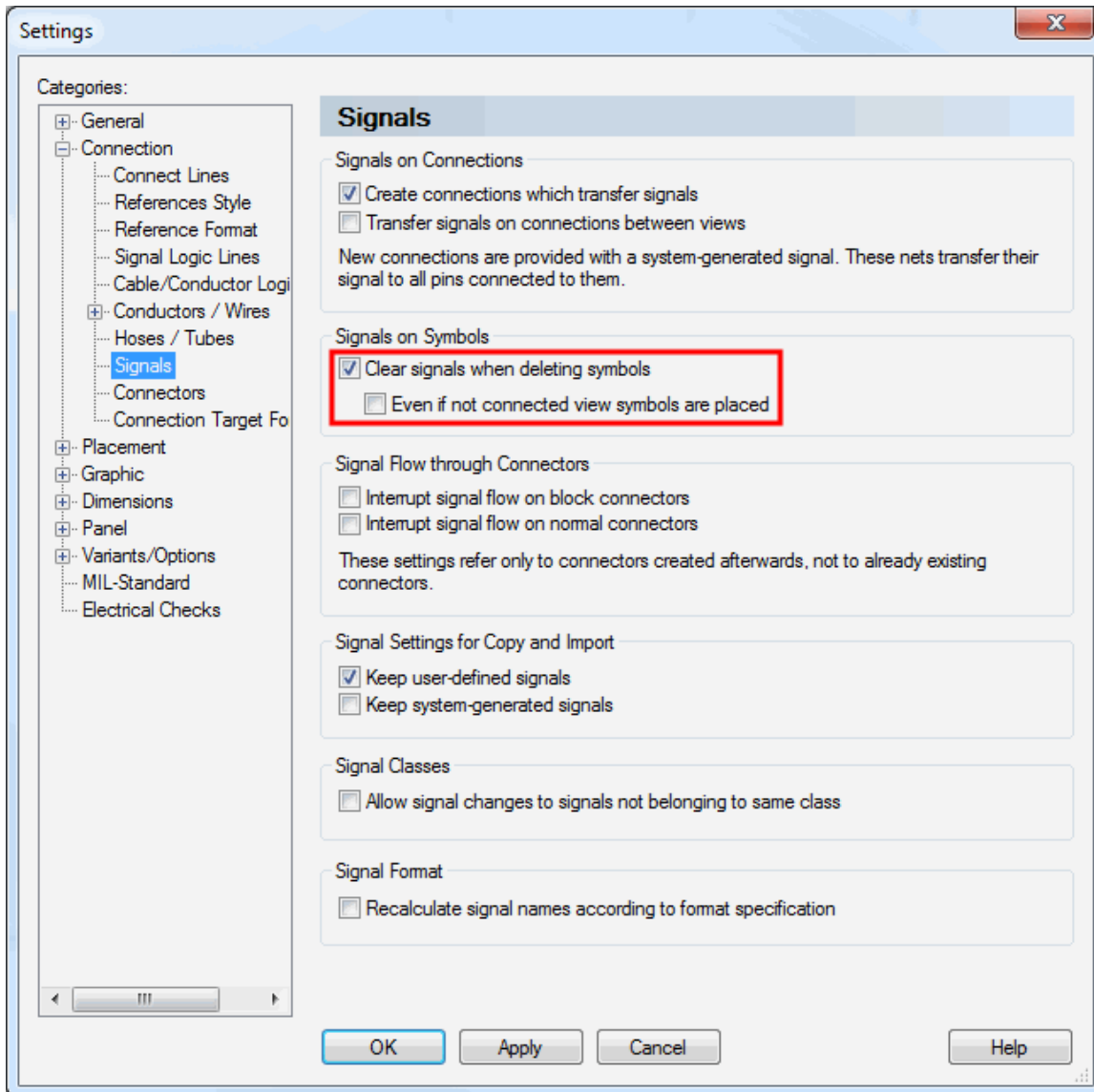
The alternative template symbol is *user-specific* in a multi-user project.

**Reference(s):** Designer-25605

### New Check Box in Settings for Deleting Signals of Not Connected View Symbols

Previously, signals were only deleted in **E3.series** when the check box **Clear signals when deleting symbols** was checked if there were no placed views of the corresponding device and no connected conductors. From now on signals can also be deleted if there are unconnected view symbols of this device.

To do so, there is a new check box **Even if not connected view symbols are placed** in the section **Signals on Symbols** under the category **Connection - Signals** in the Settings:



## Connection - Signals

**Even if not connected view symbols are placed**

**Note:** This check box can only be activated/deactivated if the check box **Clear signals when deleting symbols** is checked.

When the check box is checked and symbols are deleted, the signals are also deleted if there are unconnected view symbols of this device.

**Note:** The signal is not deleted if...

- the original device symbol is placed
- a device view with a graphical or physical connection is placed on the pin.
- a device view is placed and the pin is connected to another device
- the signal is also routed on a topology sheet
- a device view is placed and a wire is connected to the pin.

**Prerequisite:** The check box "**Unconnect, when no graphical**

**representation** is not active. (Not active in the Settings in the Category **Connection** in the section **Conductor/Wires**".

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**CLEAR\_SIG\_ON\_PLACED\_VIEW\_SYMBOL**

This check box is *user-specific* in a multi-user project.

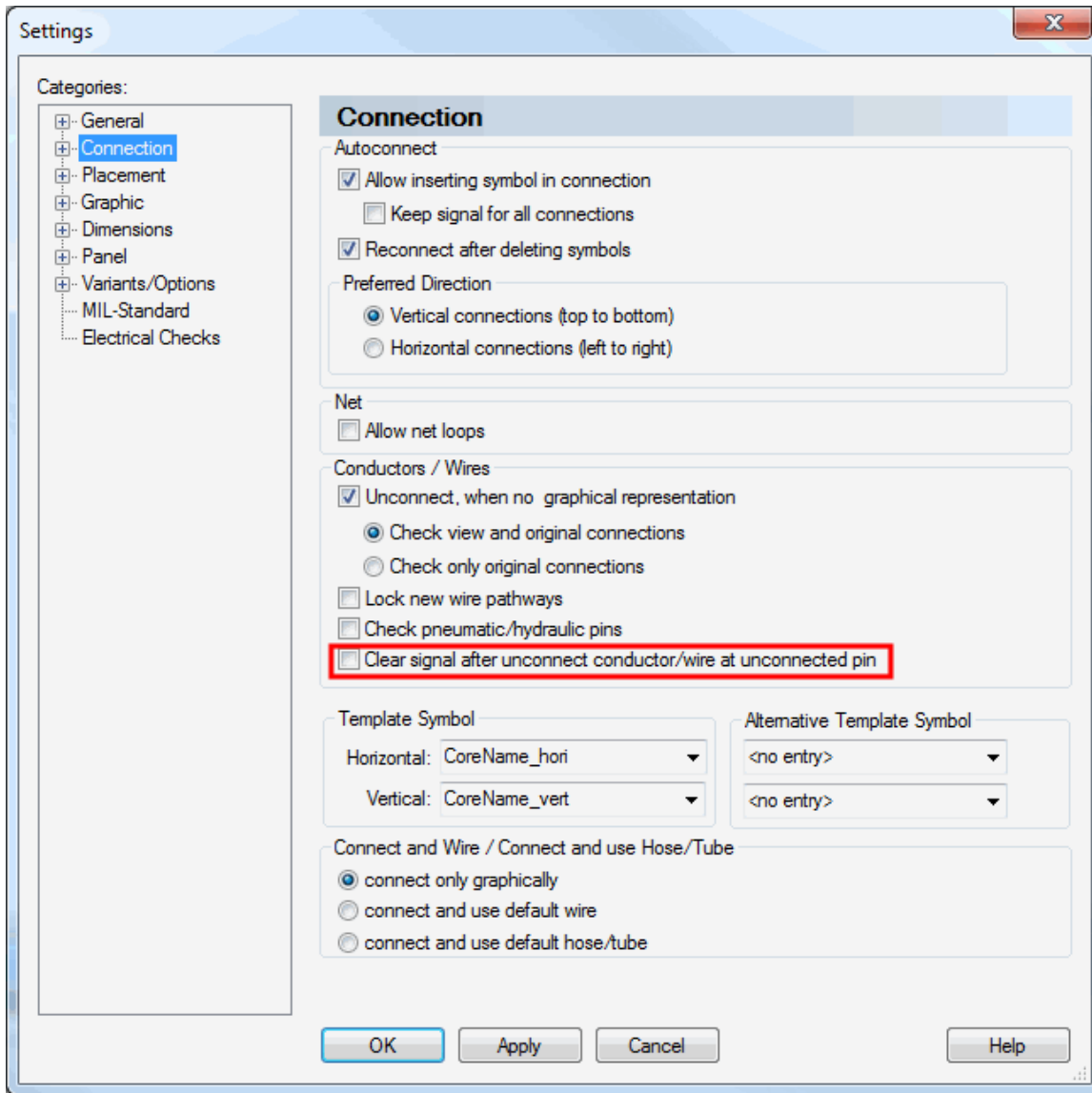
**Reference(s):** Designer-26078

### **Define whether the Signal on a Pin Shall Be Removed when Disconnecting Conductors/Wires on a Pin**

It's now possible to define whether the signal on a pin shall be removed when disconnecting conductors/wires on a pin.

**Note:** This is only valid if no additional conductors/wires and/or graphical connections are located on this pin and its pin views.

To do so, there is the new check box **Clear signal after disconnect conductor/wire at unconnected pin** in the Settings under the category **Connection** in the section **Conductors/Wires**:



## Connection

### Clear signal after unconnect conductor/wire at unconnected pin

If this check box is active, it checks to see whether additional conductors/wires are located on this pin when unconnecting a conductor/wire on a pin.

If no additional conductors/wires and/or graphical connections are placed on the pin and its pin views, the signal is removed.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**CLEAR\_SIGNAL\_AT\_PIN\_AFTER\_UNCONNECT\_CORE**

This check box is *user-specific* in a multi-user project.



**Reference(s):** Designer-25957

## Connectors

- **Connector Pins Can Now Be Placed as Single Pins when Changing a Component or Updating a Project**
- **User-Defined View Numbers for Connector Views**
- **New Options for Assigning Numbers to Connectors and Mating Connectors in the Settings**

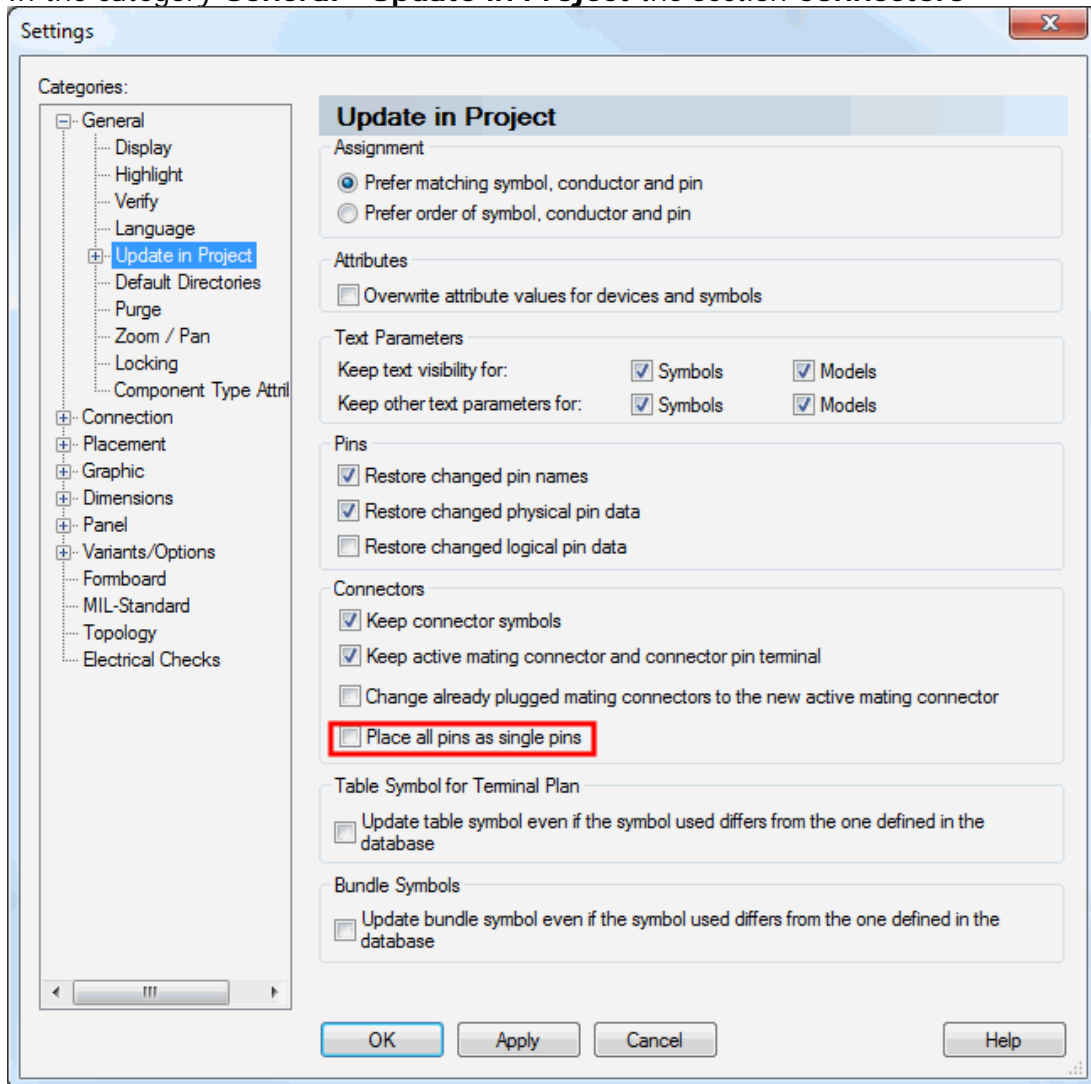
### Connector Pins Can Now Be Placed as Single Pins when Changing a Component or Updating a Project

In **E<sup>3</sup>.series** it's possible to control whether the connector pins shall be placed as single pins when

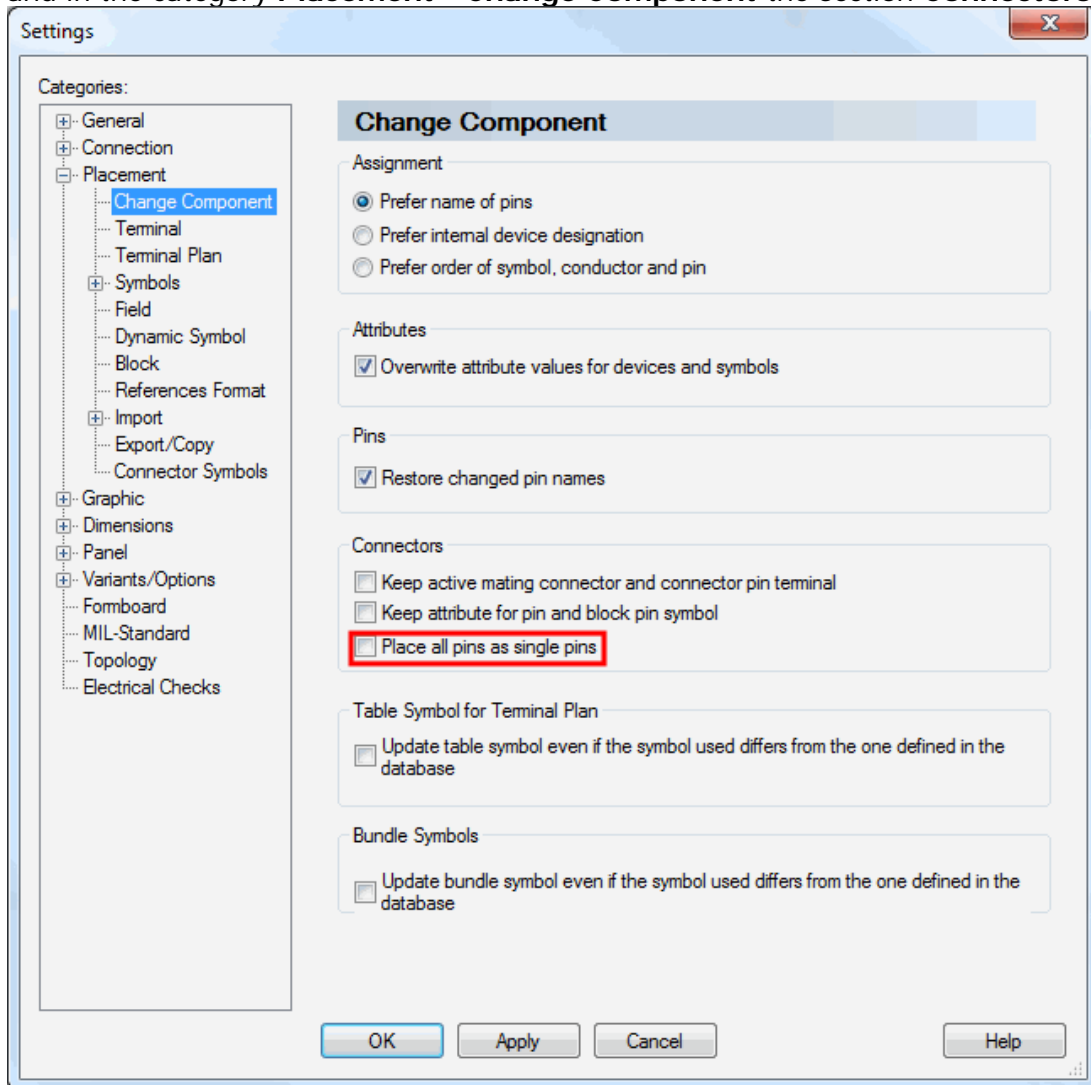
- updating the project, or
- changing a component

To do so, there is the check box **Place all pins as single pins** available in the following menus:

- In the category **General - Update in Project** the section **Connectors**



- and in the category **Placement - Change Component** the section **Connectors**



## General - Update in project

## Placement - Change Component

### Place all pins as single pins

If this check box is checked, the connector pins are placed as single pins when

- changing a component or
- updating a project

The old connector will be deleted and the new connector placed as single pins. The former position of the first (displayed) pin will help as indication. Existing spaces will be ignored and deleted when placing newly. Otherwise, the same rules apply as when interactively placing the entire connector as single pins.

### Prerequisite:

If all connector pins

- are placed as single pins
- have the same direction vector

- are placed without being interrupted by a symbol of another device

If the new connector has a different number of pins or symbols (with a feed through connector) as the previous connector, the following is valid:

- New pin symbols are placed automatically 'in addition'
- Symbols of no longer used pins are deleted automatically

**Note:**

- No difference is made between the original and the device view of the connector
- A group of symbols remains if **all** pins are part of the group
- Inserts are treated as general connectors at configurable connectors.

If the check box is not checked or the prerequisites are not fulfilled, the connector pins are handled like previously when

- changing a component or
- updating a project

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**UPDATE\_PLACE\_SINGLE\_PINS**

The check box is *project-specific* in a multi-user project.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**CC\_PLACE\_SINGLE\_PINS**

The check box is *project specific* in a multi-user project.

**Reference(s):** Designer-26110

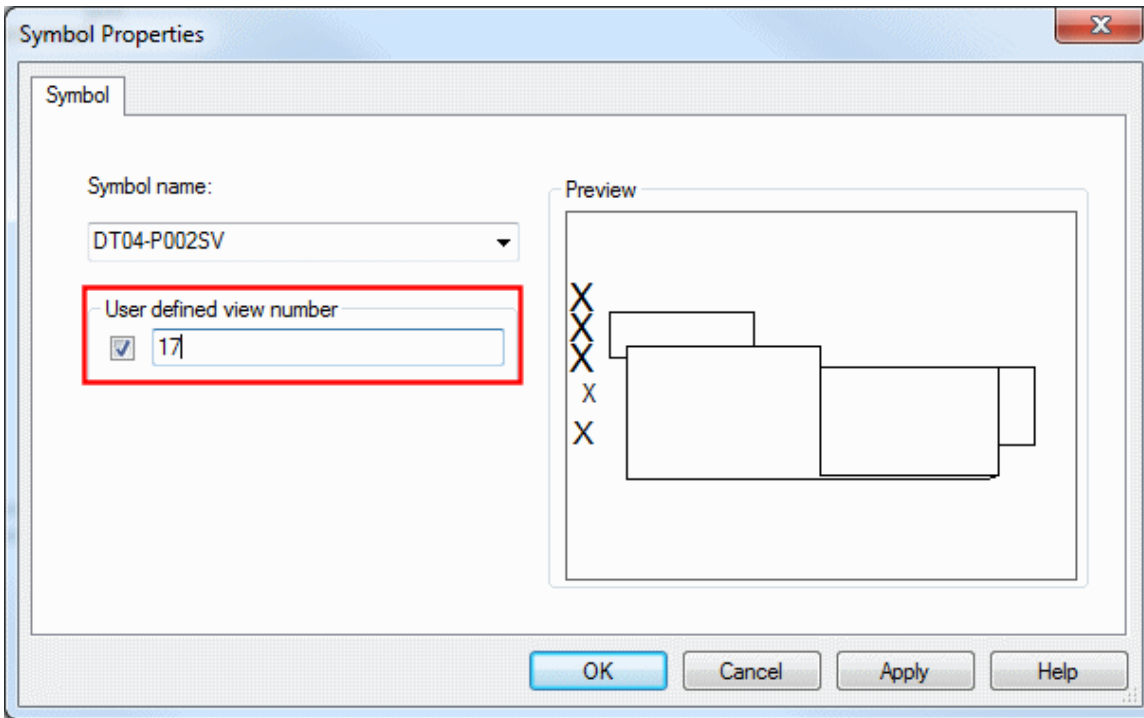
**User-Defined View Numbers for Connector Views**

Previously in **E3.series** a view number could only be defined for normal device views. From now on a user-defined view number can be specified for connector views.

To define the view number, there is the new input field **User defined view number** in the Symbol Properties in Database Editor Mode:

**This option is only available if a view on component sheet has been selected!**

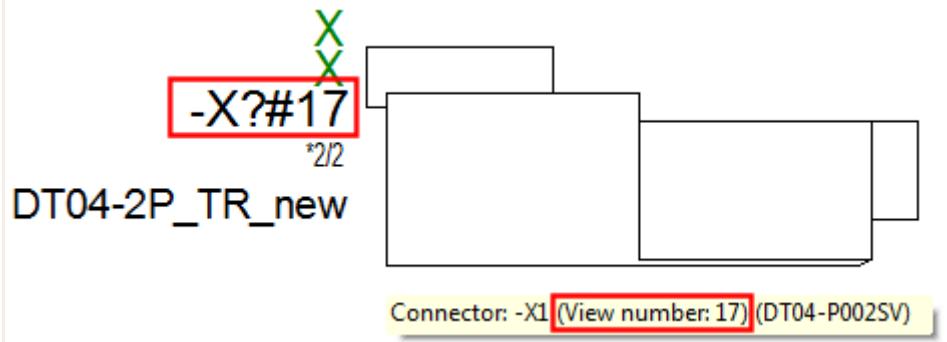




## Symbol Properties

### User defined view number

If the check box is checked, a user-defined view number can be specified for the view. This also appears in the tool tip view:

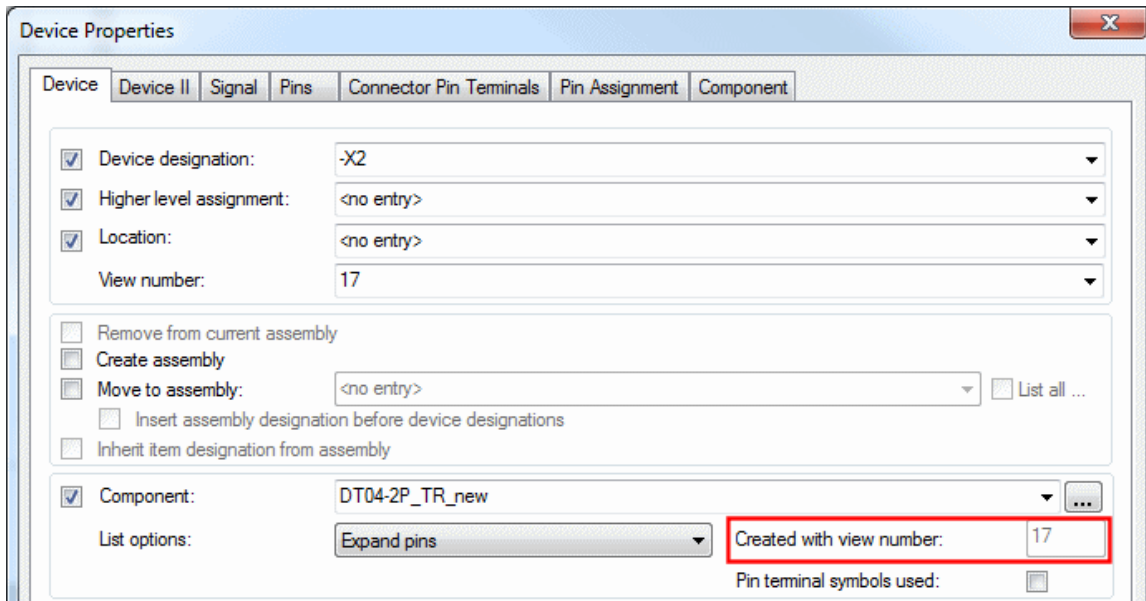


If the check box is not checked, a ? is displayed instead of the user-defined view number.

If a device view is placed on a sheet in the project with the attribute **.PREFERRED\_VIEW**, the user-defined view number appears after the symbol name (...#17).

(See **'New View' and 'Place on Formboard' Dialogs Have Been Enhanced**)

If one opens the device view's Device Properties, the user-defined view number is displayed in the field **Created with the view number**:

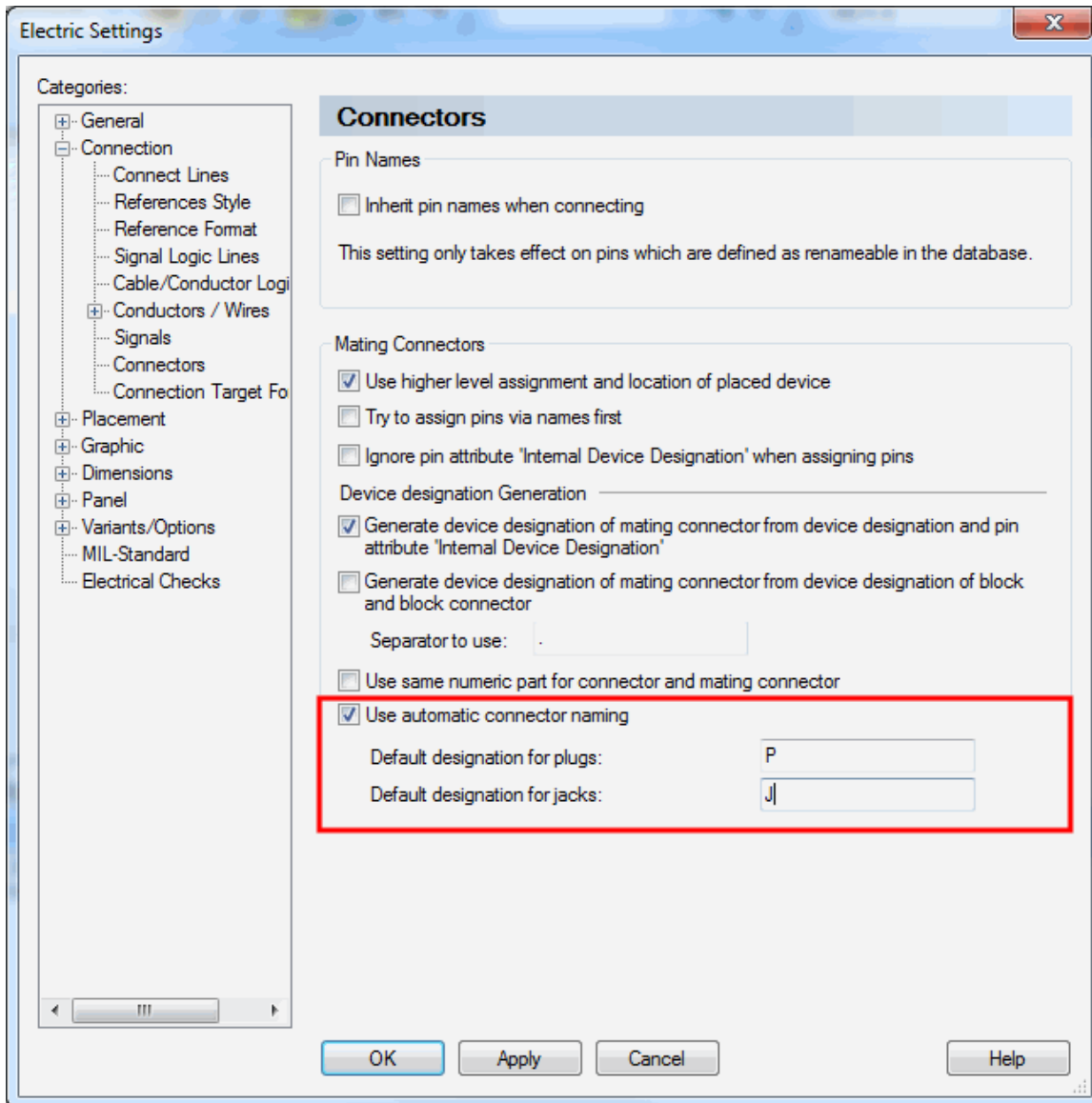


**Reference(s):** Designer-26026

## New Options for Assigning Numbers to Connectors and Mating Connectors in the Settings

In **E<sup>3</sup>.series** there are some new options for assigning numbers to connectors and mating connectors.

To do so, there are the following new functions in the Settings under the section **Device Designation Generation** in the category **Connection - Connectors**:



## Connection - Connectors

### Device designation Generation

#### Use same numeric part for connector and mating connector

If this option is active, the following is valid:

When placing a mating connector, it tries to use the same device designation as the corresponding connector.

**Note:** If it already has the same device designation (same letter code), then **<\_<nnn>** is also added. This ensures that the device designation is unique by adding **<nnn>**.

If the device designation of a connector in the plugged state is changed, the number of the mating connector is changed as well so that they both

have the same number.

**Example with different letters:**

connector: -X1 <--> mating connector: -B1 (letter B)

**Example with same letters:**

connector -X1 <--> mating connector: -X1\_1 (letter X)

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**USE\_SAME\_NUMERIC\_PART\_FOR\_MATING\_CONNECTOR.**

This check box is *project-specific* in a multi-user project.

**Use automatic connector naming**

If this option is active, a default designation must be defined for the connector and mating connector in the text fields **Default designation for plugs:** and **Default designation for jacks:**.

If a connector has a defined default designation, the defined default designation for the jack is used automatically when generating the mating connector. This is independent of which default designation has been defined in the database for the connector.

**Note:** If the numerical part of a connector is changed later, the numerical part of the mating connector is also correspondingly changed.

**Examples:**

Default designation for connector: P

Default designation for mating connector: J

connector: -P1 <--> mating connector: -J1

connector: -J100 <--> mating connector: -P100

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**USE\_ANSI\_STANDARD\_FOR\_MATING\_CONNECTOR.**

This check box is *project-specific* in a multi-user project.

**Default designation for**

**Note:** This text field can only be edited if this check box **Use automatic**

**plugs:**

**connector naming** is checked.

In this text field a default designation can be defined using a standard value.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**DEFAULT\_DESIGNATION\_FOR\_PLUGS.**

This check box is *project-specific* in a multi-user project.

**Default  
designation for  
jacks:**

**Note:** This text field can only be edited if this check box **Use automatic connector naming** is checked.

In this text field a default designation can be defined using a standard value.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**DEFAULT\_DESIGNATION\_FOR\_JACKS.**

This check box is *project-specific* in a multi-user project.

**Reference(s):** Designer-15125, Designer-26201

## Fluid

- 'Electric Settings' and 'Fluid Settings' Can Now Be Called Up in the Settings
- Special Options with Separate Values for the Respective Discipline
- Enhancement to JobInterface: COM Command 'SetDisplayConnectPoints' with Own Settings for Fluid

### 'Electric Settings' and 'Fluid Settings' Can Now Be Called Up in the Settings

From now on the settings in **E<sup>3</sup>.series** are separated into '**Electric Settings**' and '**Fluid Settings**'. From a functionality standpoint these setting dialogs are nearly identical. They only differ in that the functions and terms correspond with the respective discipline.

Using the main menu the desired settings can be called up:

- The electric settings are called up using **Tools -> Electric Settings....**
- The fluid settings are called up using **Tools -> Fluid Settings....**

The settings can also be called up in the main menu under **Tools -> Settings...** or by pressing the hot key **S**.

If one of the following schematic types...

- **Electric**
- **Process, measurement and control**
- **Panel Symbol**
- **Tubes + Instruments**
- **Single Line Diagram**

is checked in the Sheet Properties of the active sheet, the **Electric Settings** open.

If the schematic type '**Hydraulic**' and/or '**Pneumatic**' is selected in the Sheet Properties of the active sheet, the **Fluid Settings** open.

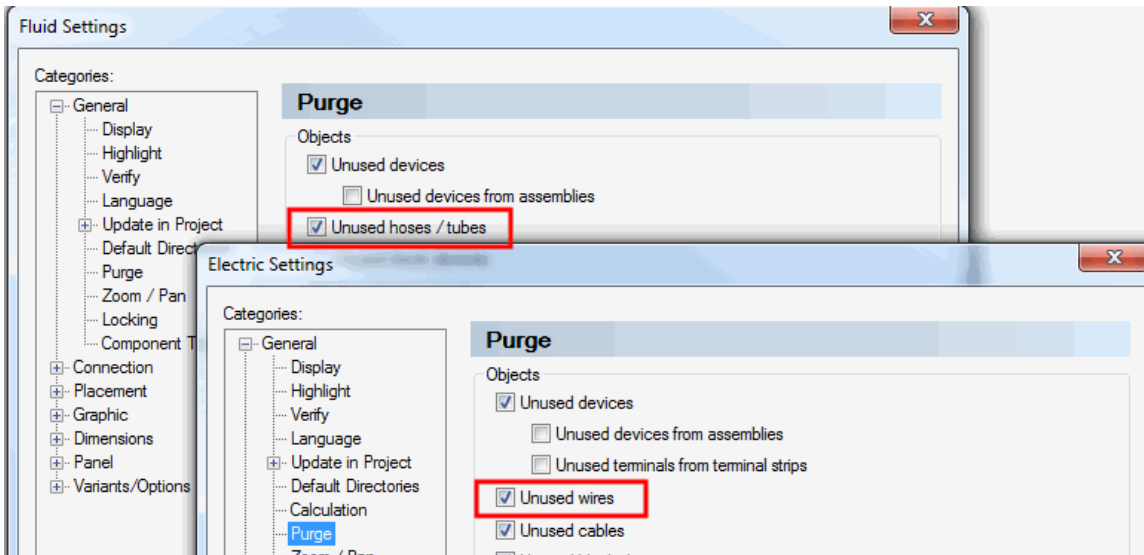
### Special Options with Separate Values for the Respective Discipline

Furthermore, there are special options in the settings, in which the defined value can be handled separately and saved for the respective discipline. These option are identified by *italics* in the Settings:

- *Grid size* (**General**)
- *Snap size* (**General**)
- *Mark unconnected nodes* (**General - Display**)
- *Internal/ external / jumper / seal representation for nodes* (**General - Display**)
- *T-Connections* (**General - Display**)
- *Forced wiring* (electric) / *Forced connections*(fluid) (**General - Display**)
- *Open line end* (**General - Display**)

#### Note:

All other options in the settings (with normal typeface) are globally valid for both disciplines. Even if they have designations which differ partly:



### Enhancement to JobInterface: COM Command 'SetDisplayConnectPoints' with Own Settings for Fluid

The following COM command in the JobInterface has been enhanced:

- **ret = job.SetDisplayConnectPoints ( newval )**
  - o **newval** The following values can be combined:
    - **&H01**: Pins are displayed in connection nets for T-connections and forced wiring. (Electric and now Fluid)
    - **&H02**: Pins are displayed in connection nets for T-connections. (Electric)
    - **&H04**: Pins are displayed in connection nets for forced wiring. (Electric)
    - **&H08**: Pins are displayed in connection nets for open line ends. (Electric)
    - **&H10**: Pins are displayed in connection nets for T-connections. (Fluid) **new**
    - **&H20**: Pins are displayed in connection nets for open line ends. (Fluid) **new**
    - **&H40**: Pins are displayed in connection nets for forced wiring. (Fluid) **new**
  - o **ret** The return value is the previous setting.

These values are also valid for the return value of the COM command **job.GetDisplayConnectPoints**

**Reference(s)**: Designer-26179

## Formboard

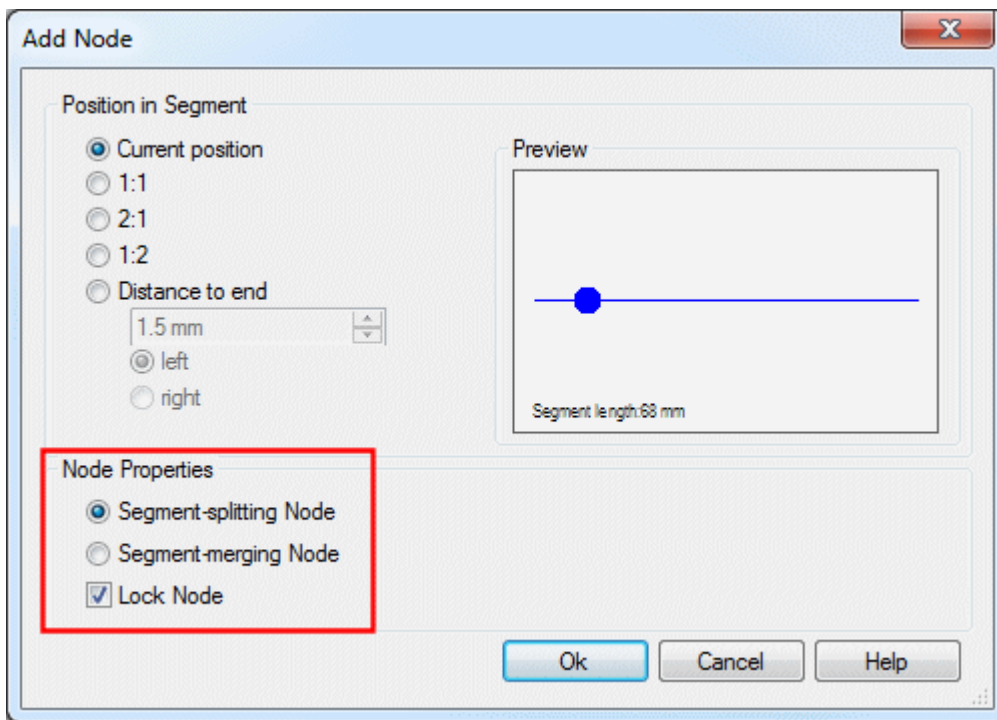
- **New Options for Inserting Nodes using the Formboard Dialog 'Add Node'**
- **Pins without Placed Conductors/Wires Can Now Be Displayed in the Formboard Connector Table**
- **Two Additional Possibilities Now for Automatically Routing Wires Assigned Options in Formboard**

### New Options for Inserting Nodes using the Formboard Dialog 'Add Node'

In the **Add Node** dialog there is the new section **Node Properties**:

#### Note:

- Previously, the selection fields **Segment-splitting node/Segment-merging node** as well as the check box **Lock node** were only context menu commands.
- The defined settings in the **Node Properties** remain unchanged when calling up the dialog again.



## Add Node

### Node Properties

#### Segment-splitting Node

This command is only active on a connected node of a segment. The appropriate segments are disconnected from each other afterward.

#### Segment-

This command is only active on a disconnected node of a segment. The



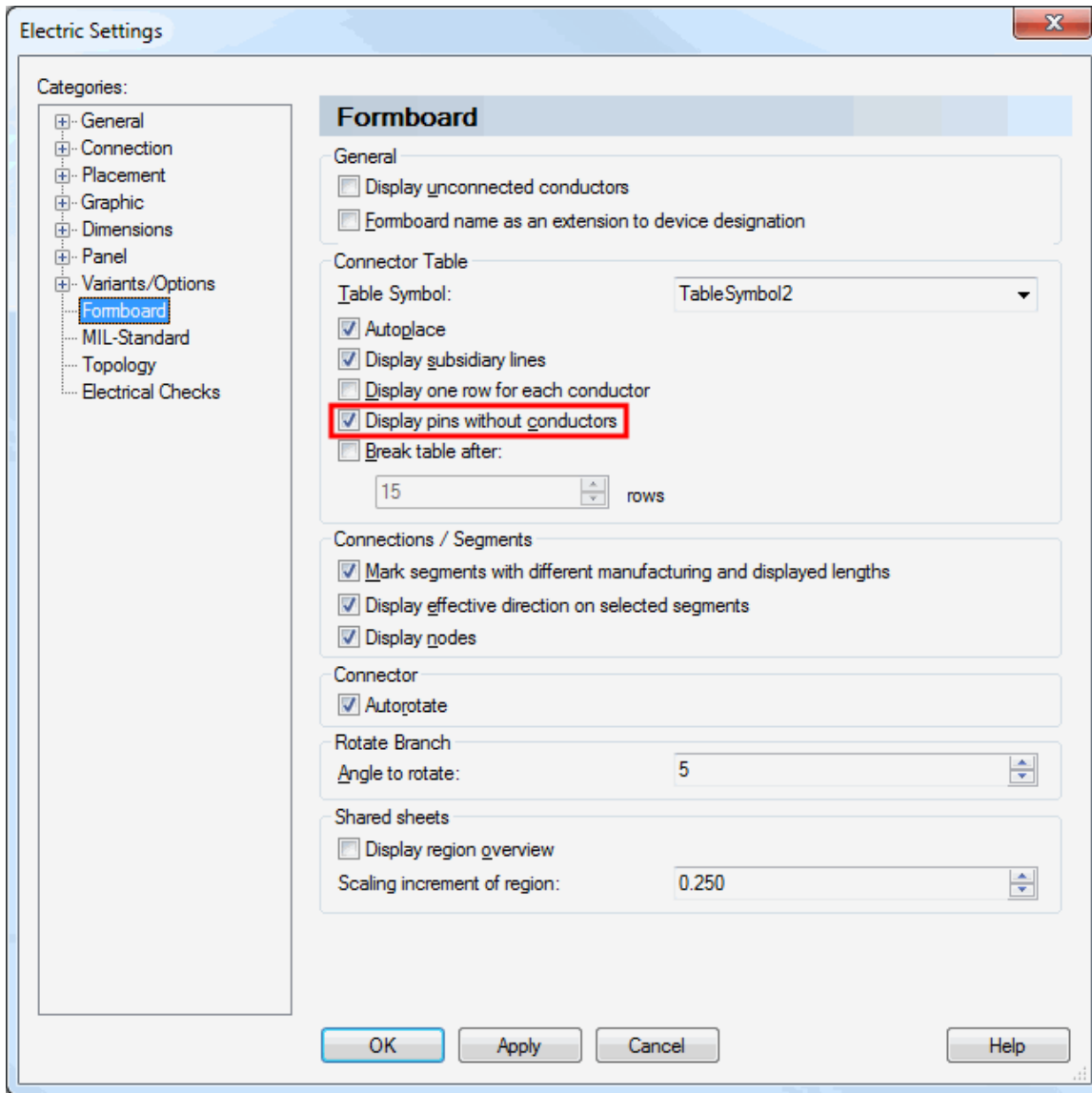
<b>merging Node</b>	appropriate segments are afterward connected to each other.
<b>Lock Node</b>	This command is active on nodes. The selected node is locked. As soon as a node has been fixed, it is shown grayed-out. A locked node will not be deleted when an added connection is deleted.

**Reference(s):** Designer-25617

### **Pins without Placed Conductors/Wires Can Now Be Displayed in the Formboard Connector Table**

Previously, when the check box **Display one row for each conductor** (called up using **Tools -> Settings -> Formboard** in the section '**Connector Table**) was checked, no pins without placed conductors/wires were displayed in the Formboard Connector Table. With the new functionality it's now possible to independently control whether pins without placed conductors/wires are displayed in the Connector Table.

To do so, there is the new check box **Display pins without conductors** under the category **Formboard** in the Settings:



## Formboard

### Connector Table

#### Display pins without conductors

If this option is checked, at least one row is displayed for every pin in the Formboard Table.

If this option is not checked, only pins with placed conductors/wires are displayed in the Formboard Table.

**Note:** This option is checked by default.

For the COM methods `prj.GetSettingValue` and `prj.SetSettingValue` the following `setting_name` can be used:

## FORMBOARD\_TABLE\_DISPLAY\_PINS\_WITHOUT\_CORES.

This check box is *user-specific* in a multi-user project.

Reference(s): Designer-18208

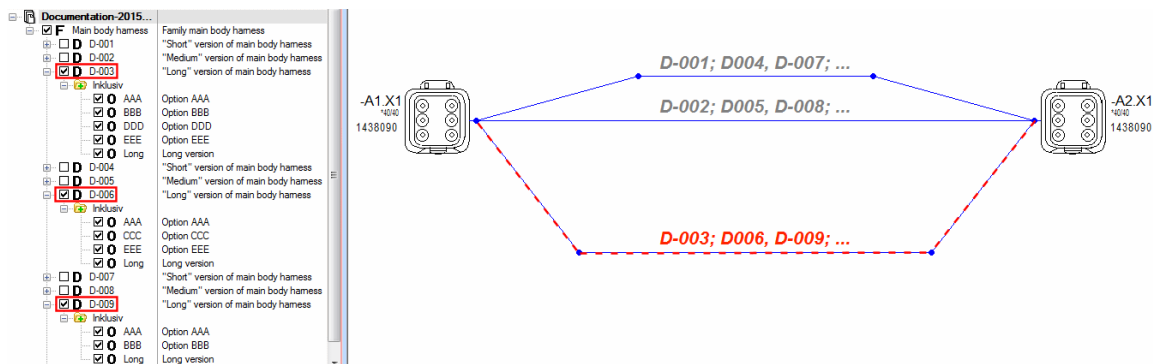
## Two Additional Possibilities Now for Automatically Routing Wires Assigned Options in Formboard

In total there are now three possibilities for automatically placing (routing) wires/conductors that are assigned options in formboard.

**Note:** To do so, the check box **Show information for option combinations** must be checked (in the settings under **Tools -> Settings -> Connection -> Conductors/Wires** in the section **Conductor Assignment Procedure**).

### Case 1: Routing Wires using Cable Harness Derivatives

Previously, cable harness derivatives were assigned directly to net segments. Wires with an appropriate option were automatically routed through these net segments.

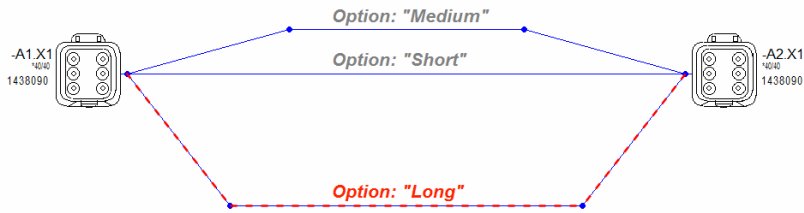
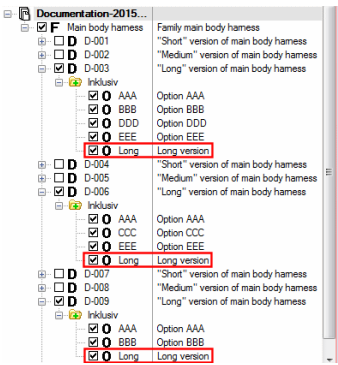


### Case 2: Routing Wires using Options (indirectly through cable harness derivatives)

Due to the fact that numerous cable harness derivatives are available in larger projects, routing wires directly may be extremely difficult to manage. That is why this routing method is recommended.

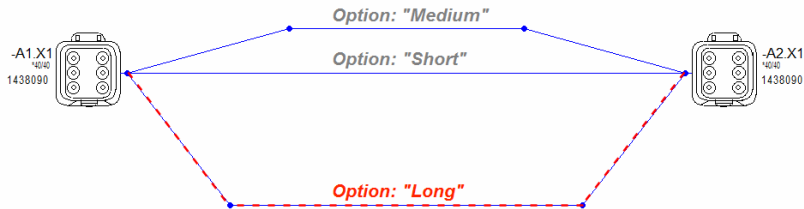
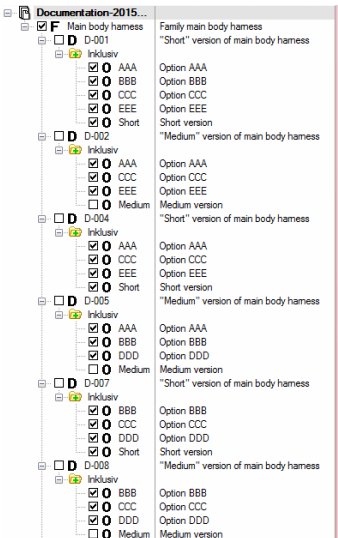
An option can now be assigned to a formboard net segment instead of a cable harness derivative. For this option all appropriate cable harness derivatives are determined automatically.

The wires are routed as if the determined cable harness derivatives have been assigned to the segment.



### Case 3: Routing Wires using Options (without cable harness derivatives)

If no appropriate cable harness derivative is found in the project, the automatic routing of wires takes place using options. For this the option combinations on the wires and net segments must be identical:



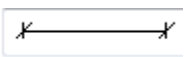
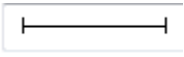
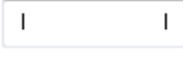
Reference(s): Designer-25427

## Graphics / Dimensions

- **Three New Dimension Line Types Available**
- **New Functionality 'Running Dimension'**
- **Enhancements to Dimension Interface for Generating and Adapting Running Dimensions**
- **Dimension Text of Running Dimensions Can Be Rotated**
- **Enhancements to Dimension Interface for Displaying Dimension Text of Running Dimensions as Rotated**

### Three New Dimension Line Types Available

The following three new line types are available for dimension lines:

Line type:	Description:
	Extension lines with dimension line and forward slashes at the ends instead of arrowheads
	Extension lines with dimension line and no arrowheads
	Extension lines without dimension line and no arrowheads

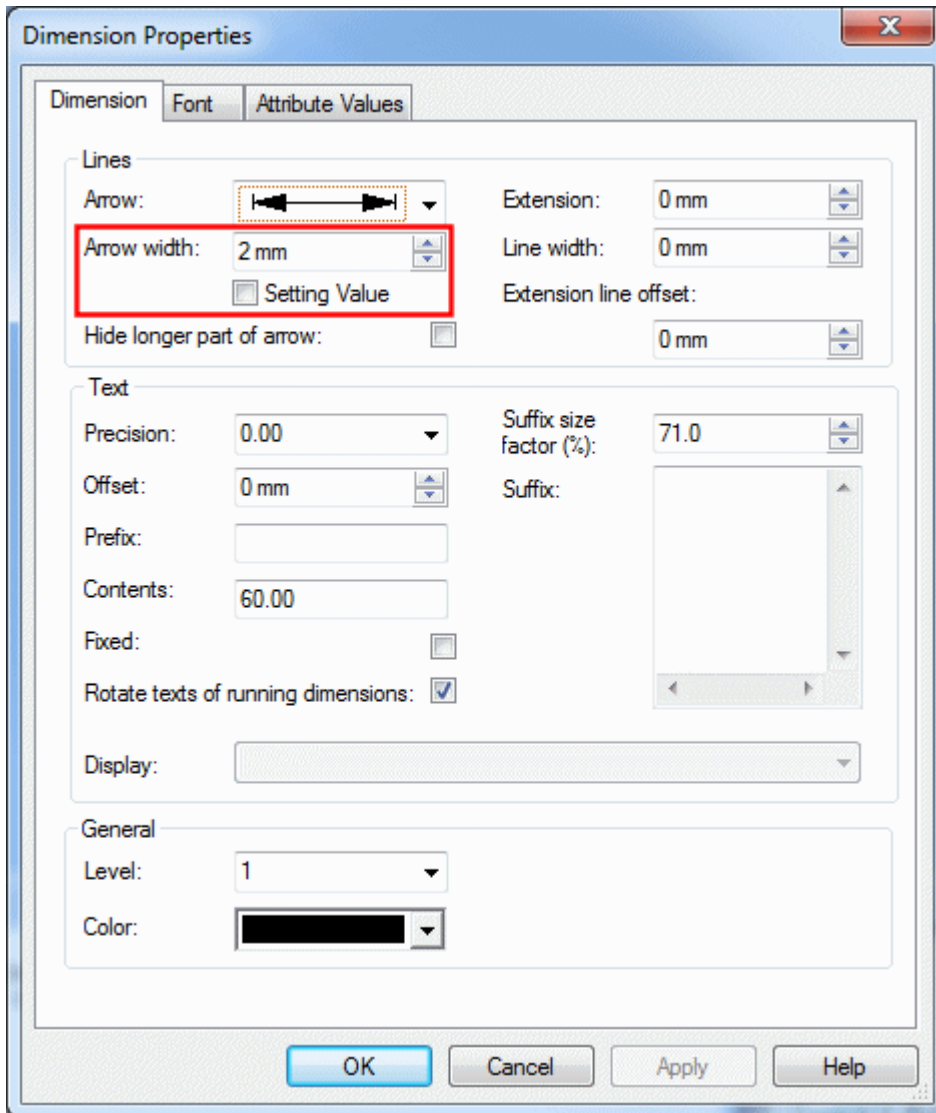
The arrow types can be defined

- o by default in the Settings (under **Tools -> Settings -> Dimension**)
- o and individually in the **Dimension Properties** in the tab **Dimension**

in the section **Lines** in the drop-down list **Arrow**.

Furthermore, the arrow width can now also be individually defined in the Dimension Properties.

To do so, there is the field **Arrow width**: under the **Dimension** tab of the **Dimension Properties**:



**Note:** The set arrow width from **Settings – Dimension** will be used as default value when the option **Setting Value** is active.

**Reference(s):** Designer-06049

### New Functionality 'Running Dimension'

Previously, a dimension could only be defined using two reference points (from starting point to end point). It's now possible to define dimensions using several reference points. To do so, there is the new function 'Running Dimension'.

#### To insert a running dimension:

- select the command **Insert -> Running Dimension** in the main menu, or
- click on the new icon  **Running Dimension** in the **Graphic** toolbar

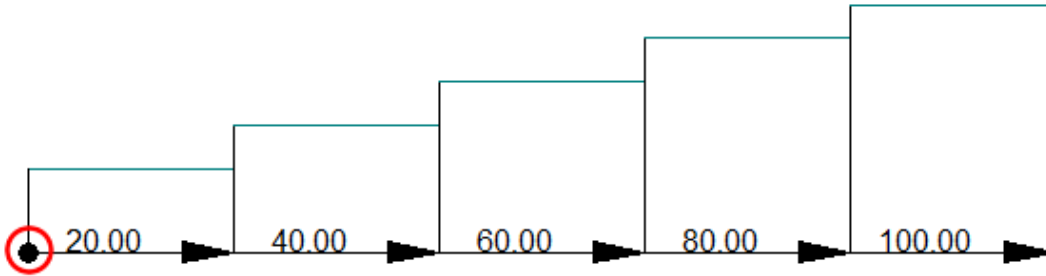
For determining the dimension's starting point left-click on the desired location.

For additional reference points simply left-click on the desired locations.

The dimension texts are centered automatically.

**Note:**

The filled circle denotes the starting point and reference point of the dimension. From there the individual length specifications are calculated:

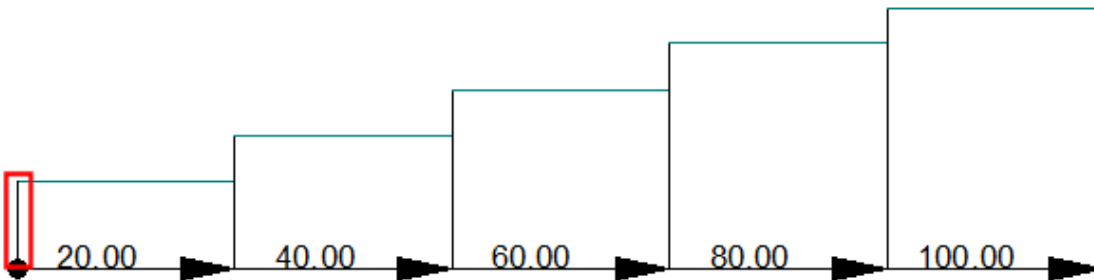


**To delete a running dimension:**

Parts of a running dimension can be individually deleted.

Select the desired elements of the running dimension by left-clicking and pressing the **SHIFT** key and then press the **DEL** key.

To select all parts of the running dimension, left-click directly on the first extension line at the starting point:



**Enhancements to DimensionInterface for Generating and Adapting Running Dimensions**

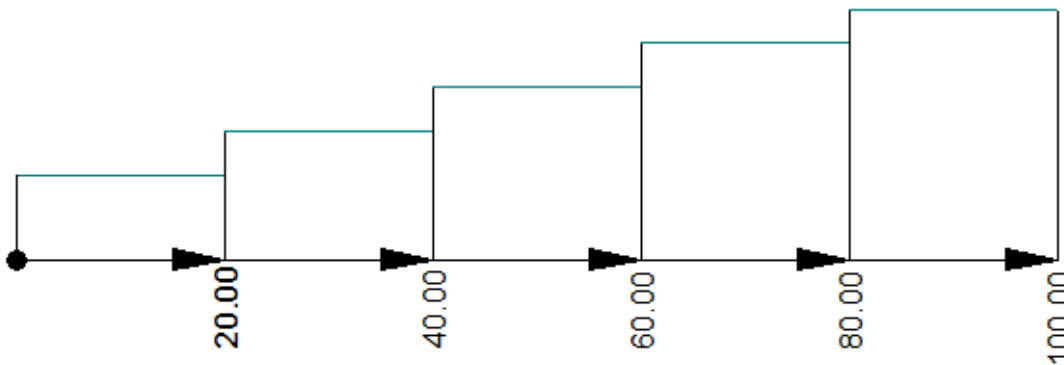
In the **DimensionInterface** there are the following new COM commands:

- **ret = dim.CreateEx ( dimtyp, sheet, x, y, flags )**
  - o **dimtyp** The following dimension types are supported:
    - 6 standard (horizontal or vertical) running dimension
    - 8 angled running dimension

- o **sheet** ID of sheet, on which the reference dimension shall be created
  - o **x, y** Array of all x, y coordinates of the click points
  - o **flags** currently not used
  - o **ret** The following return values are possible:
    - >0 ID of running dimension if successful
    - -1 invalid dimension type
    - -2 invalid sheet
    - -3 invalid starting point
    - -4 invalid point
- **ret = dim.GetPartIds ( ids )**
    - o **ids** Array with all IDs of part dimensions
    - o **ret** Number of IDs
  - **ret = dim.IsPartOfDimension**
    - o **ret** The return value is 1 if *dim* is a part of the running dimension, otherwise 0.
  - **ret = dim.IsRunningDimension**
    - o **ret** The return value is 1 if it's a running dimension, otherwise 0.

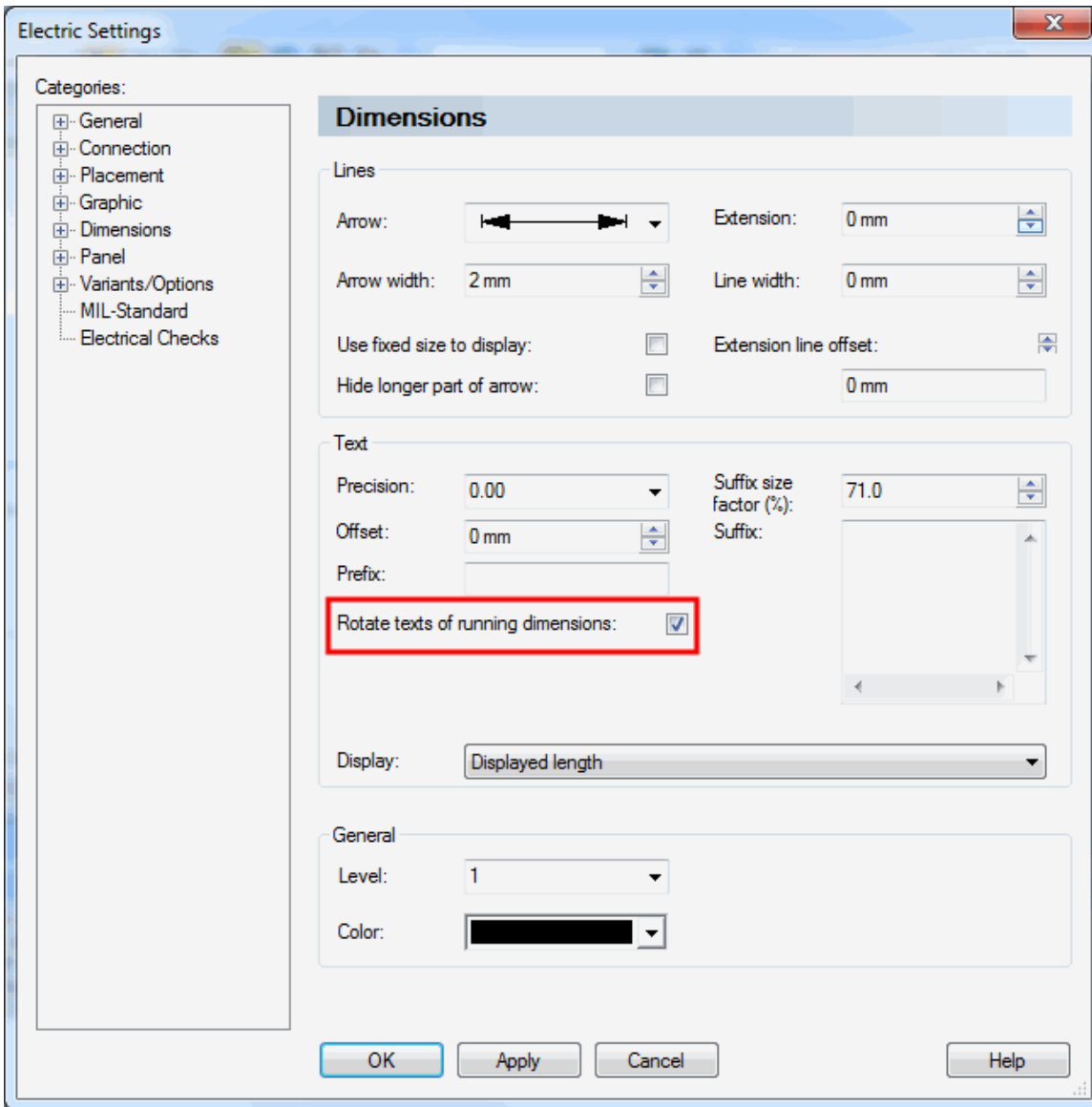
### Dimension Text of Running Dimensions Can Be Rotated

The dimension text of running dimensions can be rotated 90°:



To define the dimension text as rotated for all running dimensions by default, use the new check box **Rotate texts of running dimensions:** in the Settings under **Tools -> Settings -> Dimensions** in the section **Text:**





## Dimensions

### Rotate texts of running dimensions:

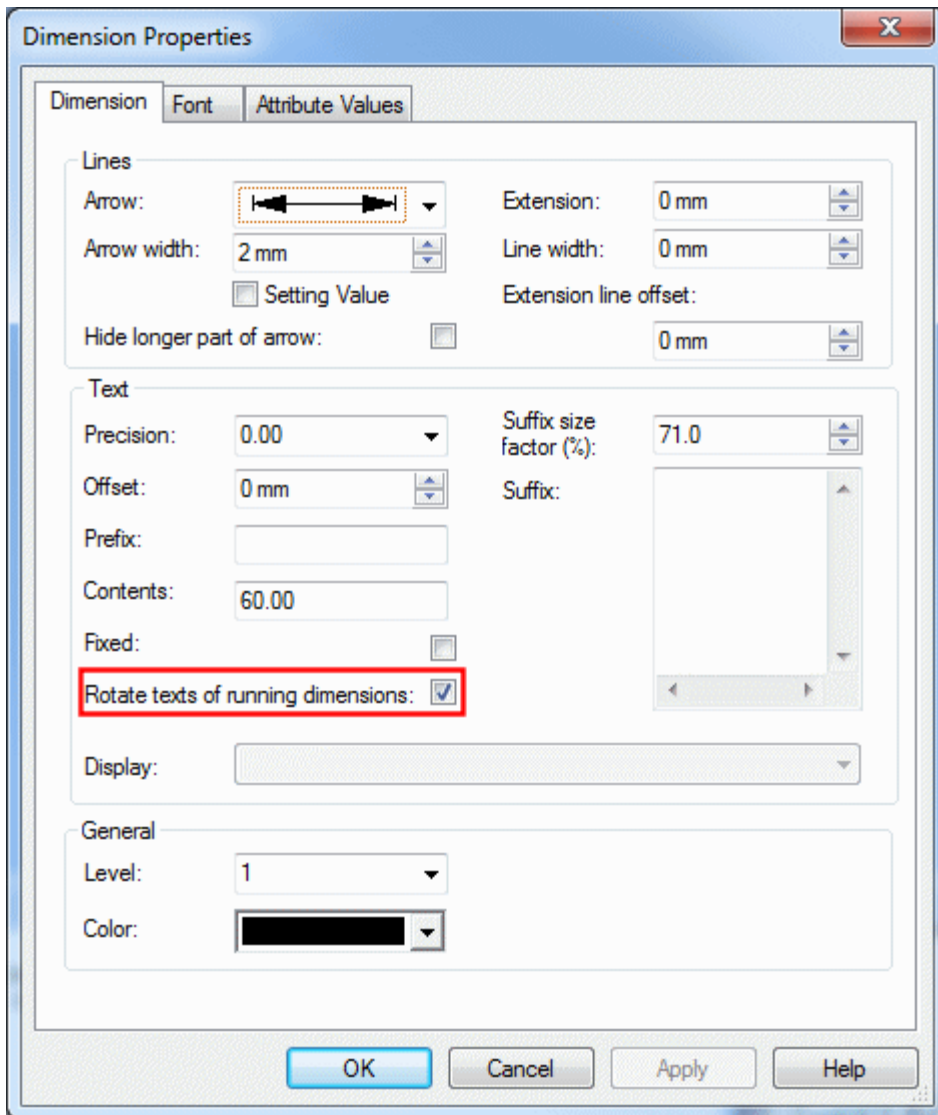
If checked, the dimension text of running dimensions is rotated 90°.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:

**DIM\_ROTATE\_TEXT\_OF\_RUNNING\_DIM.**

This check box is *user-specific* in a multiuser project.

Rotating the text of running dimensions can be specified later using the **Dimension Properties**:



## Dimension Properties - Dimension

### Rotate texts of running dimensions:

If checked, the dimension text for running dimensions is rotated 90°.

## Enhancements to DimensionInterface for Displaying Dimension Text of Running Dimensions as Rotated

In the **DimensionInterface** there are the following new COM commands:

- **ret = dim.GetRunningDimTextRotation**
  - o **ret** The return value can have the following values:
    - **-1** The function cannot be executed on this type of dimension.
    - **0** The dimension text is not rotated.
    - **1** The dimension text is rotated.
  
- **ret = dim.SetRunningDimTextRotation ( value )**
  - o **value** The following values are possible:
    - **0** The dimension text is not rotated.
    - **1** The dimension text is rotated.
  - o **ret** The return value is 0 if successful, otherwise -1.

**Reference(s):** Designer-03804, Designer-25881

## Import/Export (subcircuits)

- [Connection Direction of Conductors/Wires Can Be Ignored when Importing Subcircuits](#)
- [DWG/DXF Import/Export Now Compatible with AutoCAD Version 2013](#)

### Connection Direction of Conductors/Wires Can Be Ignored when Importing Subcircuits

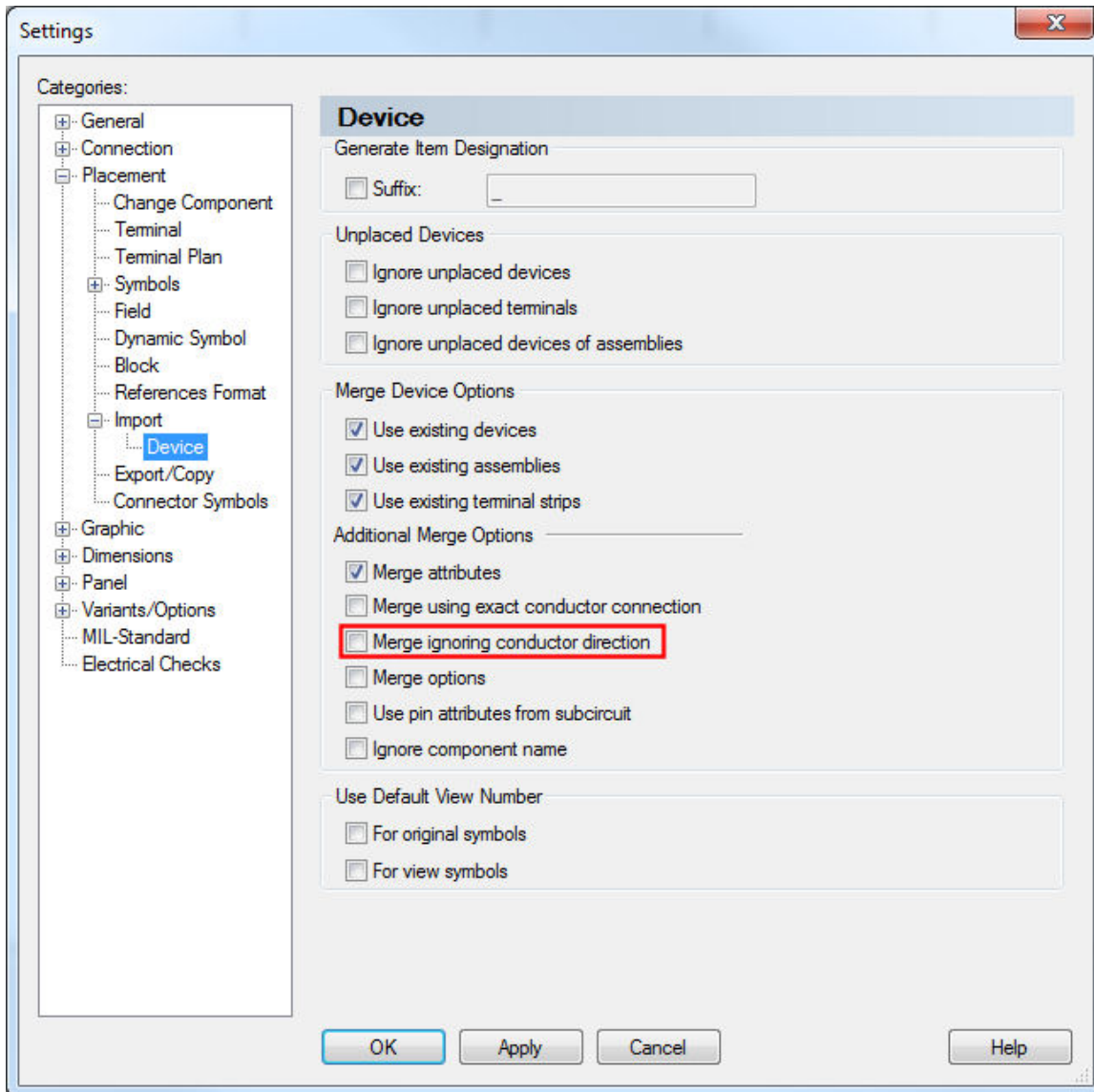
Previously when importing subcircuits, the connection direction of placed conductors/wires was checked automatically (from pin to pin).

If the conductors/wires in the subcircuit had different connection directions as elements already used in the project, new elements were generated for

- conductors/wires
- cables

This can be prevented by now ignoring the connection direction when importing a subcircuit. To do so, there is the new check box **Merge ignoring conductor direction**.

The check box **Merge ignoring conductor direction** can be defined by default in the Settings under **Tools -> Settings -> Placement -> Import - Device** in the section **Additional Merge Options** :



## Devices

### Merge ignoring conductor direction

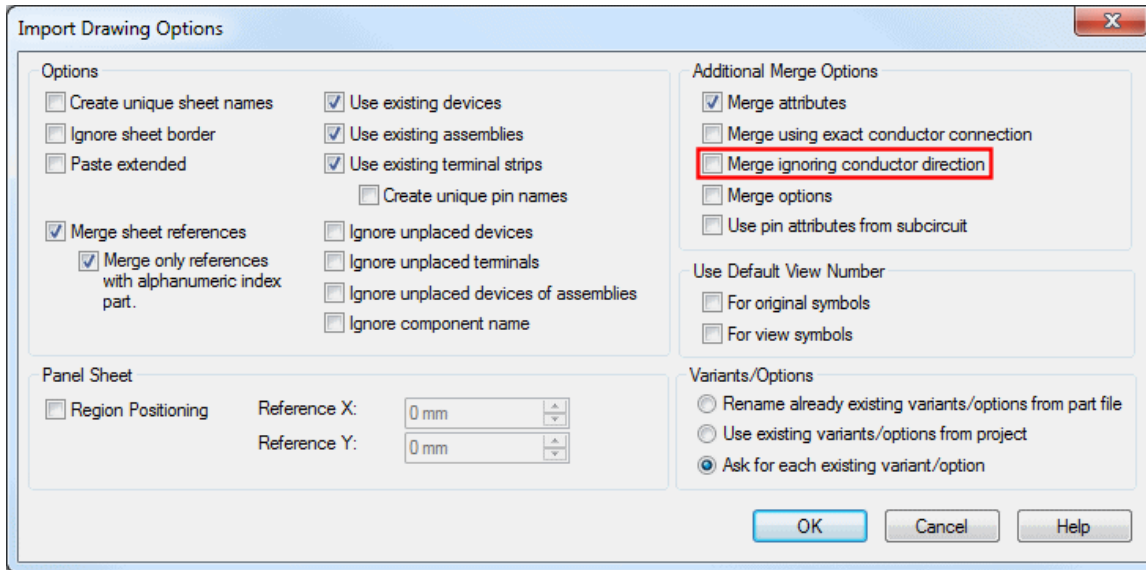
If checked, the connection direction of conductors/wires is ignored when merging a drawing or subcircuit.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used:  
MERGE\_IGNOREING\_CONDUCTOR\_DIRECTION

This check box is *user-specific* in a multi-user project.

The check box **Merge ignoring conductor direction** can be alternatively defined when importing a drawing **File -> Import -> Drawing... -> Options...**

**Note:** After clicking **OK**, the check box is changed accordingly in the Settings (and vice-versa).

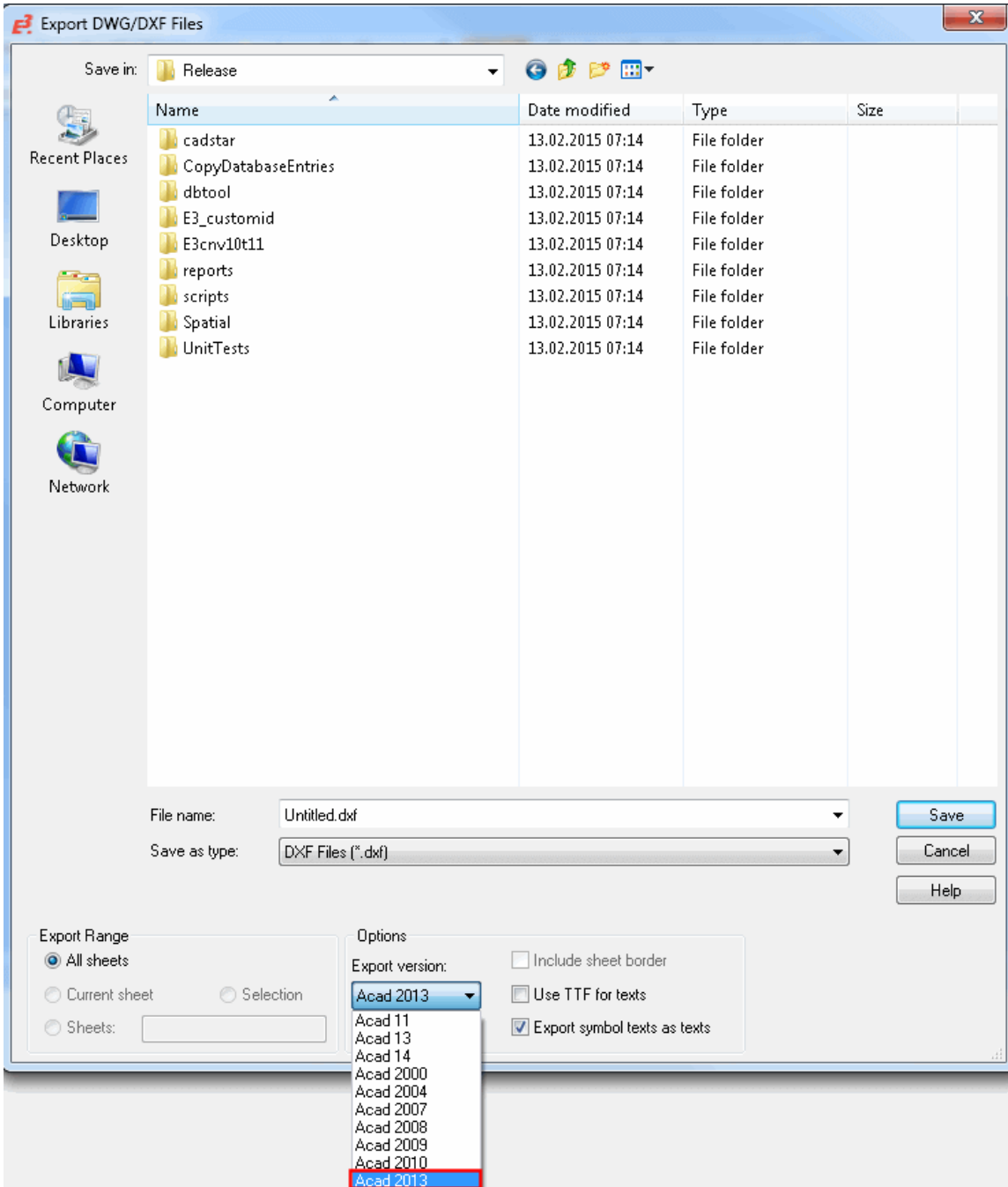


**Reference(s):** Designer-23119

## DWG/DXF Import/Export Now Compatible with AutoCAD Version 2013

DWG/DXF Import/Export is now also compatible with AutoCAD Version 2013.

In the **Export DWG/DXF Files** dialog (called up using **File -> Export -> DXF/DWG...**) there is the new entry **Acad 2013** in the **Export version:** drop-down list:



**Reference(s):** Designer-23294

## Panel

- **New Section '2D View Options' in 'Insert New Sheet' Dialog in E3.panel**
- **Already Placed 2D Views Can Now Be Deleted**
- **Symbol Properties Can Now Be Opened when Placing Models on Panel Sheets**
- **New '3D View' Toolbar in Panel**
- **Models Can Now Be Set as Transparent or Invisible in the 3D View**

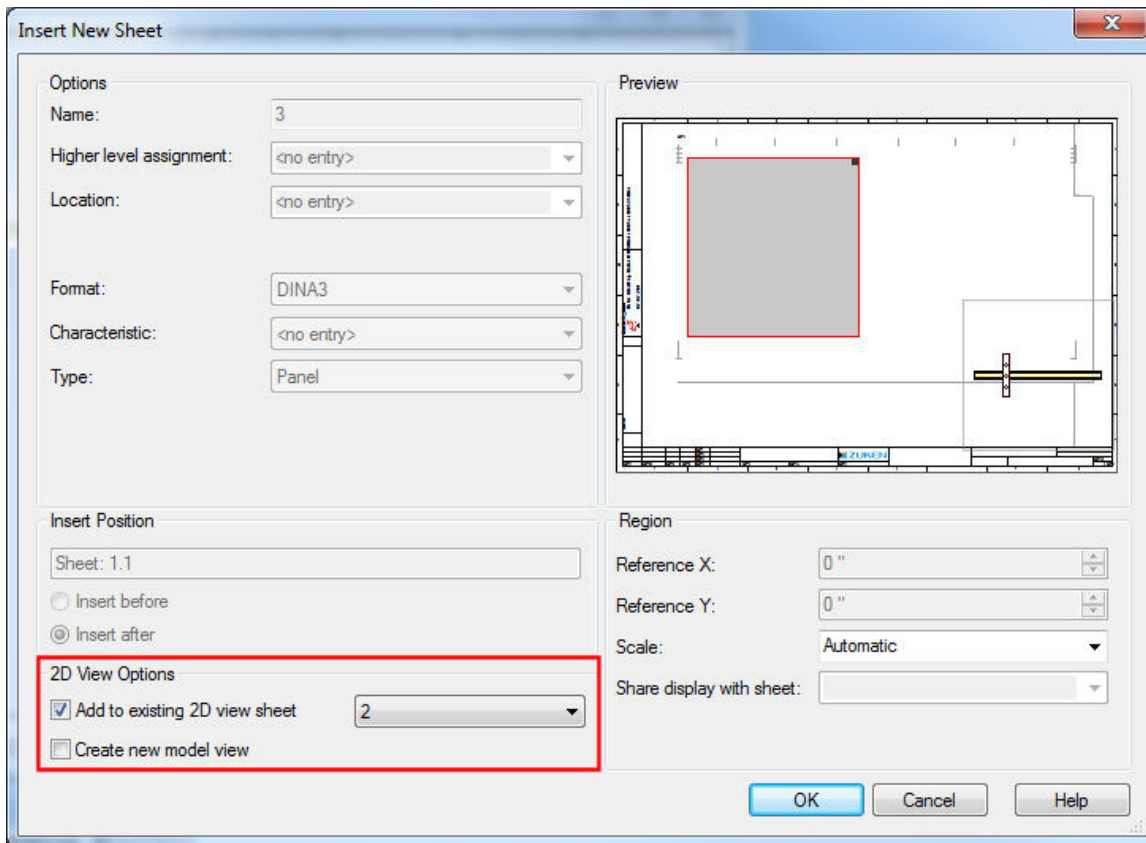
### New Section '2D View Options' in 'Insert New Sheet' Dialog in E3.panel

Previously, a new 2D view sheet was generated automatically when creating a 2D View. Now new 2D views can be added to already existing 2D view sheets.

To do so, there is the new section **2D View Options** in the **Insert New Sheet** dialog:

#### Note:

- The section **2D View Options** only appears in the dialog window if the context menu command **Create 2D View...** has been executed.



## Insert New Sheet

### 2D View Options



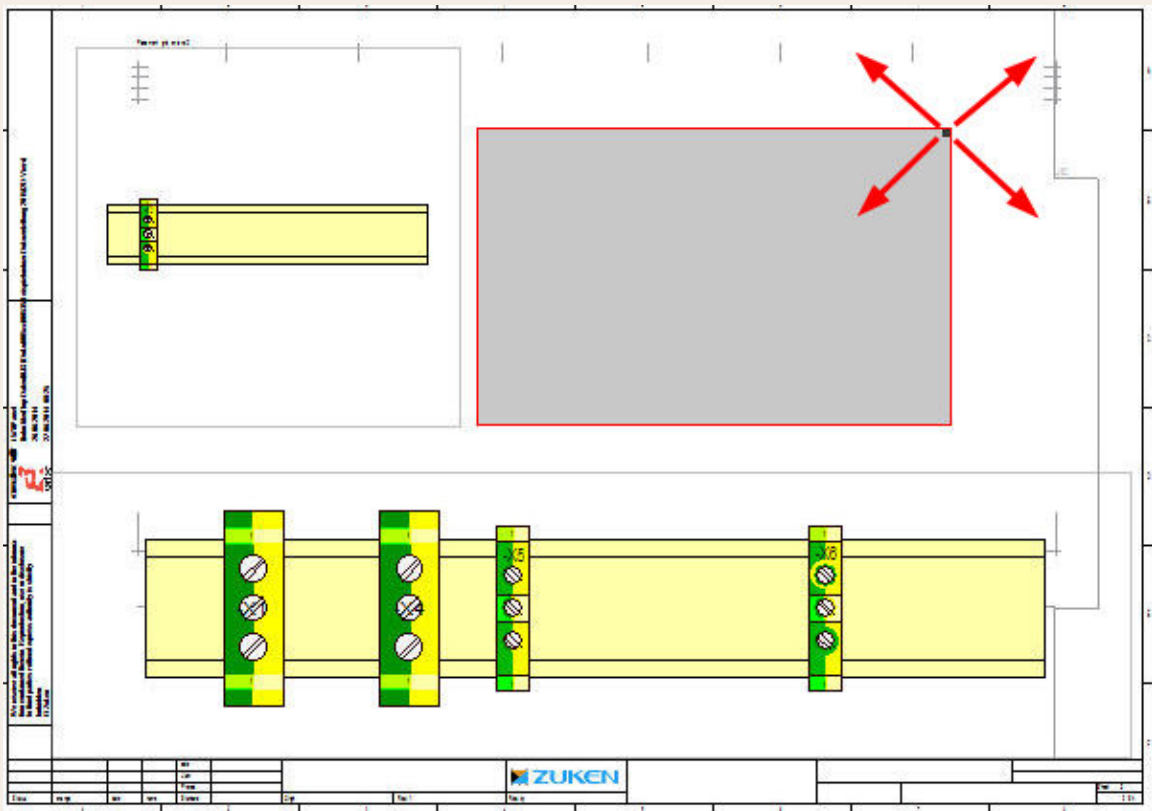
**Add to existing 2D view sheet**

The check box can only be activated if a sheet with a 2D view already exists. If the check box is inactive or no 2D view exists, a new 2D view sheet is created.

If the check box is checked, an already existing 2D view sheet can be selected within the drop-down list on which the new 2D view shall be added.

Under **Preview** the cutout of the 2D view can be adapted to

- the size (press and hold left mouse button on upper right corner)
- the position (press and hold left mouse button on the cutout)



**Create new model view**

The following definitions can be found after creating a 2D view of a model (and placed models on it):

- If the check box is active, the model is displayed both on the former sheet (on which the model is placed) and the new 2D view sheet.
- If the check box is inactive, the model will be switched to the 2D view sheet.

**Scale:**

Here, the option 'Automatic' has been added to the drop-down list.

If 'Automatic' is selected, the scale will be calculated according to the cutout.

### Note:

When importing panel sheets, it tries to merge models with 2D views.  
The following is valid:

- Only one 2D view per model
- Models and 2D views may not differ regarding the following points:
  - Position
  - Length (mounts, cable ducts)
  - Rotation/Tilting

When importing, panel sheets are **no** longer moved to a new position. This is valid for all imported panel sheets (not only 2D views).

Exceptions:

- If models/2D views collide and they cannot be inserted at the respective position
- By executing the command **Paste Extended**, the subcircuit sheets are moved to a new position.

### Already Placed 2D Views Can Now Be Deleted

If there are several 2D views on a 2D view sheet, the 2D view can be deleted using the new context menu command **Delete 2D View**.

#### How to delete a 2D view on the 2D view sheet:

- right-click on the 2D view and select the command within the displayed context menu **Delete 2D View**,
- the 2D view is deleted on the 2D view sheet

**Reference(s):** Designer-09310

### Symbol Properties Can Now Be Opened when Placing Models on Panel Sheets

It's now possible to open the Symbol Properties of models placed on panel sheets.

The tilting direction for the model symbol is displayed in curly brackets in the line **Symbol name** in the **Symbol Properties**.

**Example:** Symbol name: {Front}










**Reference(s):** Designer-19341

### New '3D View' Toolbar in Panel

In **E<sup>3</sup>.panel** it's now possible to rotate the respective view of an object when the 3D display mode is active.

The following symbols are available in the new toolbar '**3D View**' :



Symbol: Purpose:	
	<b>Top</b> Displays the top view
	<b>Bottom</b> Displays the bottom view
	<b>Left</b> Displays the left view
	<b>Right</b> Displays the right view
	<b>Front</b> Displays the front view
	<b>Back</b> Displays the back view
	<b>Isometric</b> Displays the view in isometric axonometric
	<b>Dimetric</b> Displays the view in dimetric axonometric
	<b>Animation</b> After changing the view an animation occurs



**Note:**

The toolbar is not displayed by default.

The toolbar can be displayed by:

- selecting **Tools -> Customize -> Tab Toolbar**,
- clicking **Toolbars:** and choosing **3D View** in the list

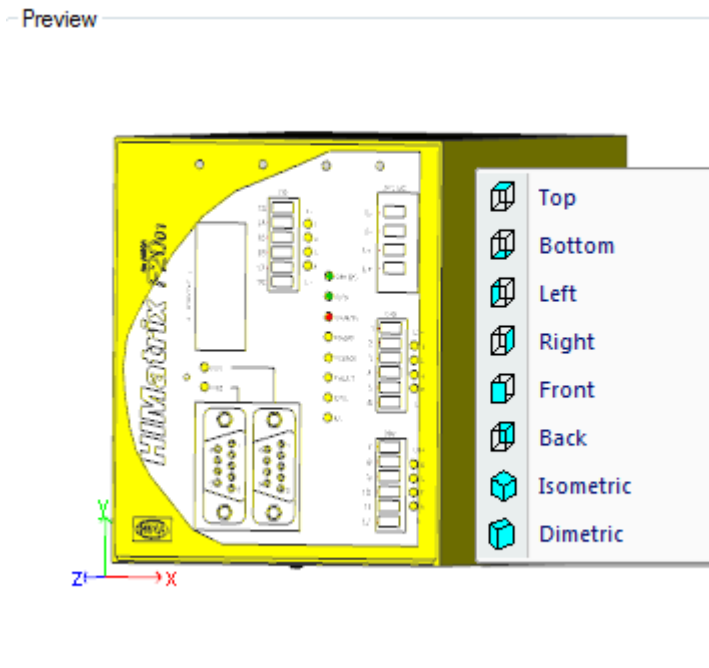
or

- right-clicking within the area of toolbars and
- marking the **3D View**.

This functionality is also available within the Preview Window of the following dialogs:

- **Device Properties - Shape**
- **Device Properties - Slots/Pins**
- **Device Properties - Restricted**
- **Model Properties - Shape**
- **Model Properties - Slots/Pins**
- **Component Table**

To do so, right-click anywhere in the Preview Window:



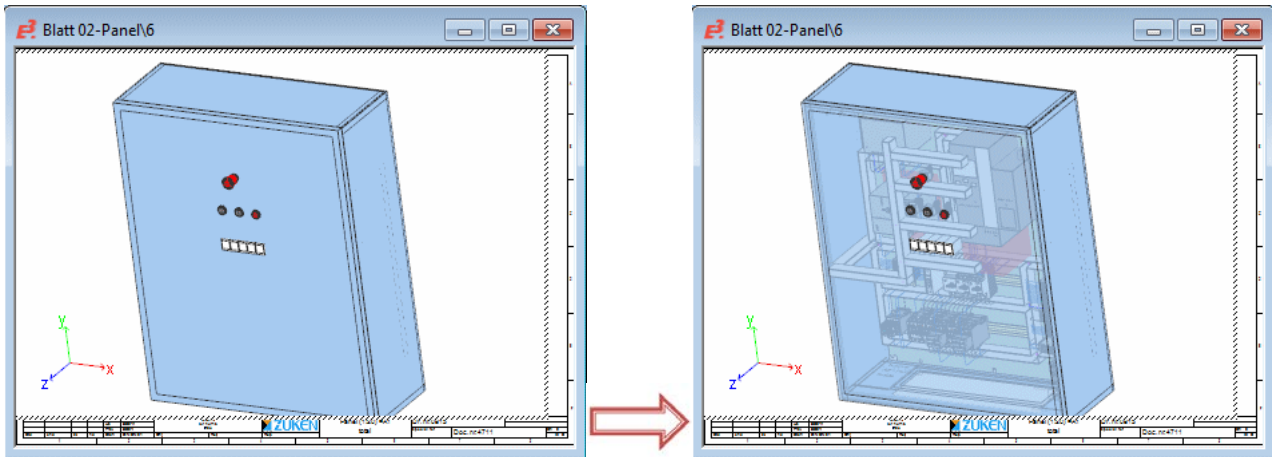
**References:** Designer-23135

## Models Can Now Be Set as Transparent or Invisible in the 3D View

In the 3D display models and their mounted devices can now be set as 'Transparent' or 'Invisible'. In doing so, panel doors or miscellaneous placed elements can be made invisible in a closed panel so that one has a better view of the desired area.

### Note:

- Connections on panel sheets cannot be set as 'Transparent' or 'Invisible'
- When importing subcircuits or using Cut&Paste/Copy&Paste, the state is set to visible once again.



To realize this functionality, there is the new context menu command **Transparency in 3D**.

**Prerequisite:** This is valid for the following models:

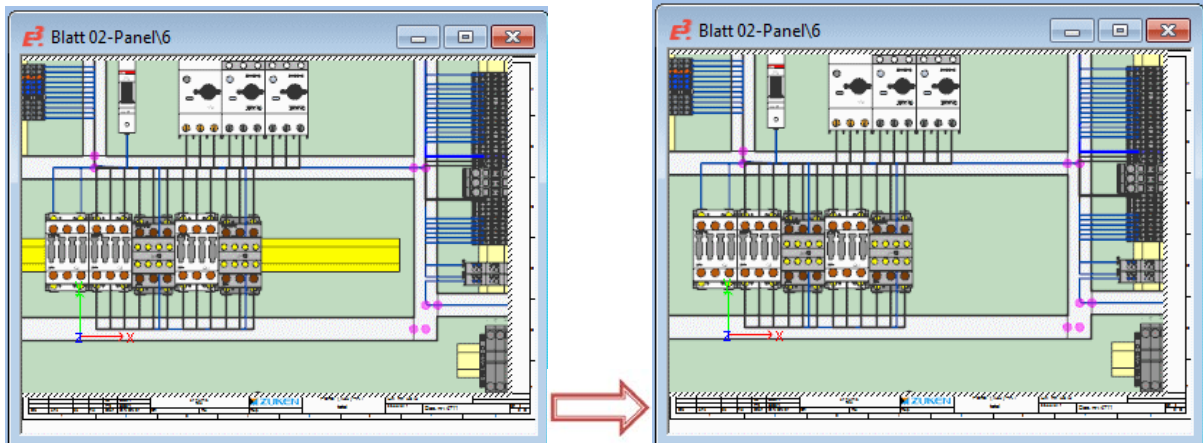
- For all models in the tree structure **Panel - Placed**
- For all models placed on the panel sheet in the 2D view
- For all models placed and visible on the panel sheet in the 3D view

**Note:** If the model placed on the panel sheet has already been set/defined as transparent or invisible, the context menu command can only be called up in the 2D view or in the tree structure **Panel - Placed**.

To set the visibility of a model:

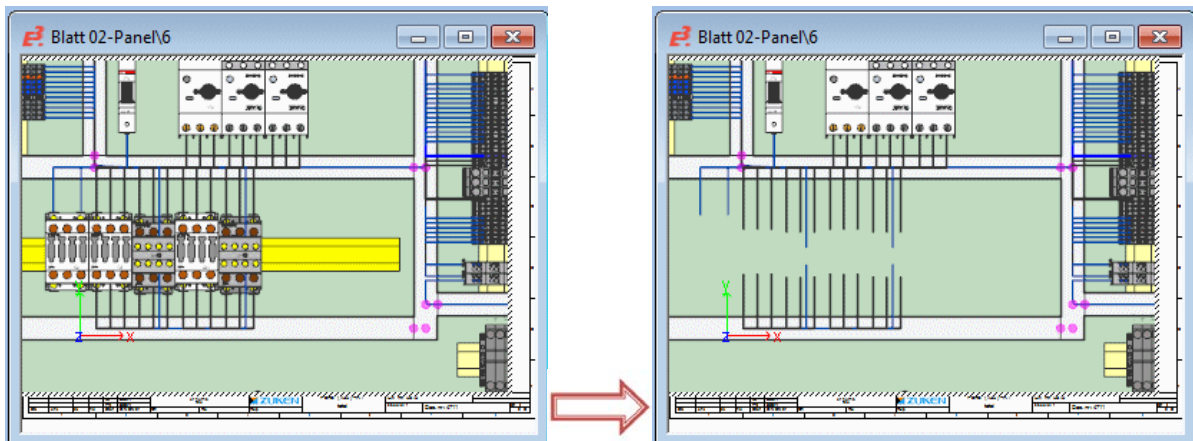
- right-click on the desired model(s) and move the mouse cursor over the context menu command **Transparency in 3D**
- the following modes are available for selection under **Transparency in 3D**:
- **Visible/ Transparent / Invisible**

**Example: "Invisible"**



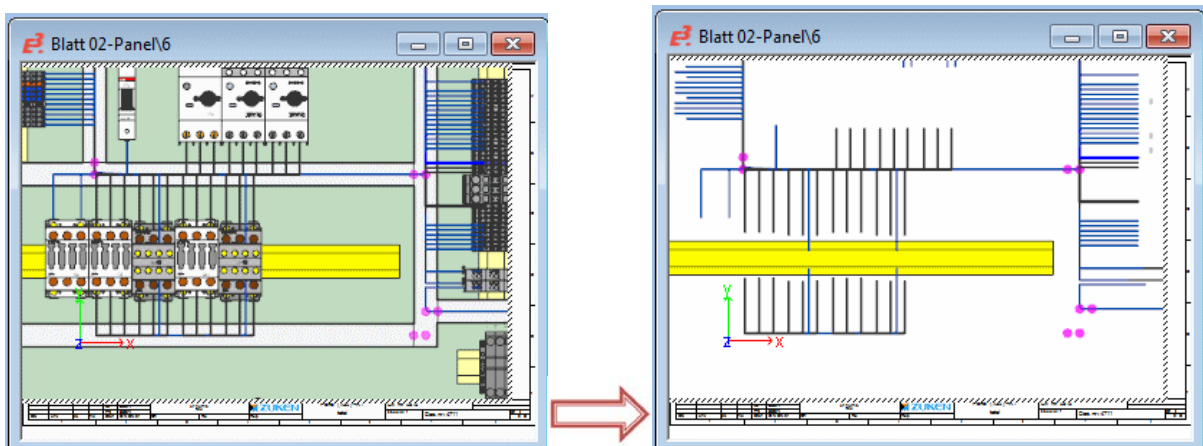
... with Mounted Devices *Only the selected model(s) with mounted devices are invisible in the corresponding mode.*

Example: "Invisible with Mounted Devices"



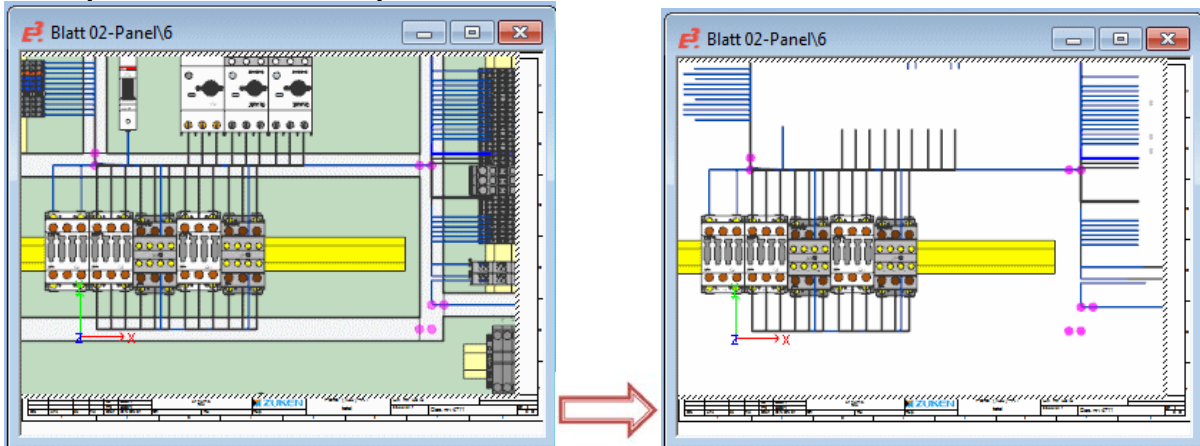
... except Selected Devices *All models, except the selected model(s) are invisible in the corresponding mode.*

Example: "Invisible except Selected Devices"



... except Selected and Mounted Devices *All models, except the selected model with mounted devices are invisible in the corresponding mode.*

Example: "Invisible except Selected and Mounted Devices"



In the tree structure **Panel - Placed** the icon changes according to which state is selected:

State	Model Symbol	Cable Duct Symbol	Mount Symbol	Connector Symbol	Terminal Symbol
Visible					
Transparent					
Invisible					

To select **all** models and mounted devices in the entire project, proceed as follows:

- right-click on the project name in the **Panel - Placed** tree,
- select the command **Visible / Transparent / Invisible** from the displayed context menu.

**References:** Designer-23224

## References

- [New Drop-Down Menu 'Format:' in the Settings for Defining How the Text Type 'Sheet Reference' Can Be Displayed](#)

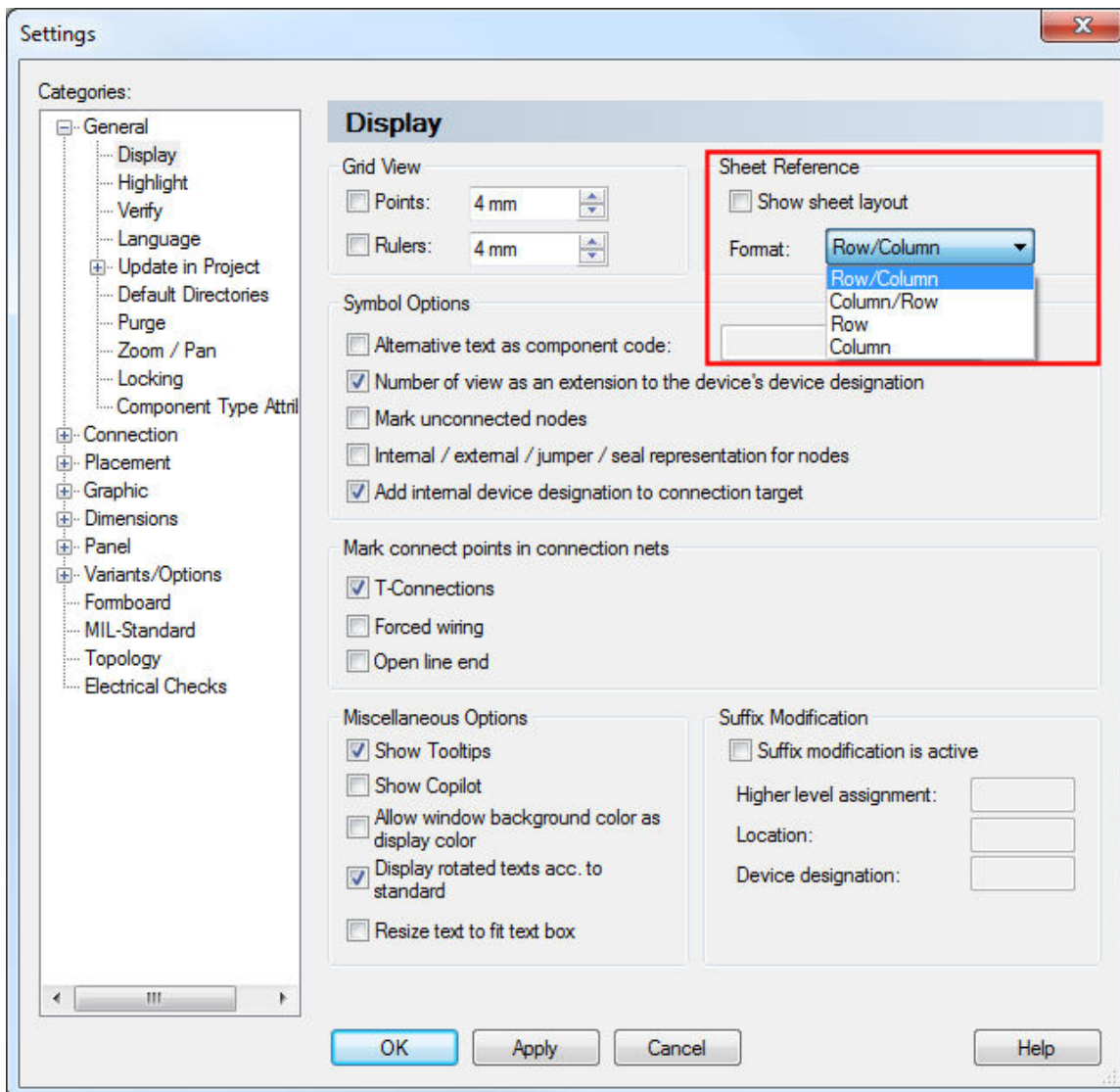
### New Drop-Down Menu 'Format:' in the Settings for Defining How the Text Type 'Sheet Reference' Can Be Displayed

In **E<sup>3</sup>.series** there are now several possibilities for displaying the text type 'Sheet reference':

- **Row/Column**
- **Column/Row**
- **Row**
- **Column**

To do so, there is the new drop-down menu **Format** in the Settings under **General - Display** in the section **Sheet Reference**.

**Note:** The already existing check box **Show sheet layout** has also been integrated in the section **Sheet Reference**.





## Display

### Format:

For the text type 'Sheet reference' the sheet coordinates of the reference target are displayed as follows:

- **Row/Column**
- **Column/Row**
- **Row**
- **Column**

**Note:** When changing the format, the display of sheet references is automatically updated. The display of the signal format <.GRID> is updated automatically provided that an automatic recalculation is active.

This check box is *project-specific* in a multi-user project.

For the COM methods **prj.GetSettingValue** and **prj.SetSettingValue** the following **setting\_name** can be used: **SHEETREF\_FORMAT**.

- **job.SetSettingValue ( SHEETREF\_FORMAT, value )**
- **value = job.GetSettingValue ( SHEETREF\_FORMAT )**
  - o **value** Permitted values:
    - **0:** Row/Column
    - **1:** Column/Row
    - **2:** Row
    - **3:** Column
    - If an invalid value is defined, Row/Column is used.

**Reference(s):** Designer-09802

## Text

- [Two New Text Types in E3 for Referencing between Symbols in Circuit Diagrams and Components on Panel Sheets](#)
- [Display of Multiline Texts Can Now Be Defined in the Symbol Properties for Symbols of the Type 'Attribute Text Template'](#)
- [Lines Can Now Be Added to Text Boxes](#)
- [Parameter Values Can Now Be Defined using Placeholders for Translated Text Nodes from the Translation Table](#)
- [Ten New Sheet Text Types in the Database](#)

### Two New Text Types in E3 for Referencing between Symbols in Circuit Diagrams and Components on Panel Sheets

The following two new text types are now available in E3:

- **Reference to Schematic** (#1141 )
- **Reference to Panel** (#1142 )

Both text types can be defined on symbols and components in database editor mode.

#### Reference to Schematic

This text type displays all positions of the master symbols which are defined on the device. If no master is defined on the device, the first symbol will be referenced. If no first symbol is placed, the referencing remains empty. If there are several reference destinations due to several defined master symbols, the reference symbols will not be displayed according to their position. The definition on the component will be used as sequence instead.

#### Reference to Panel

This text type displays all positions on which the model is visible.

**Reference(s):** Designer-14358

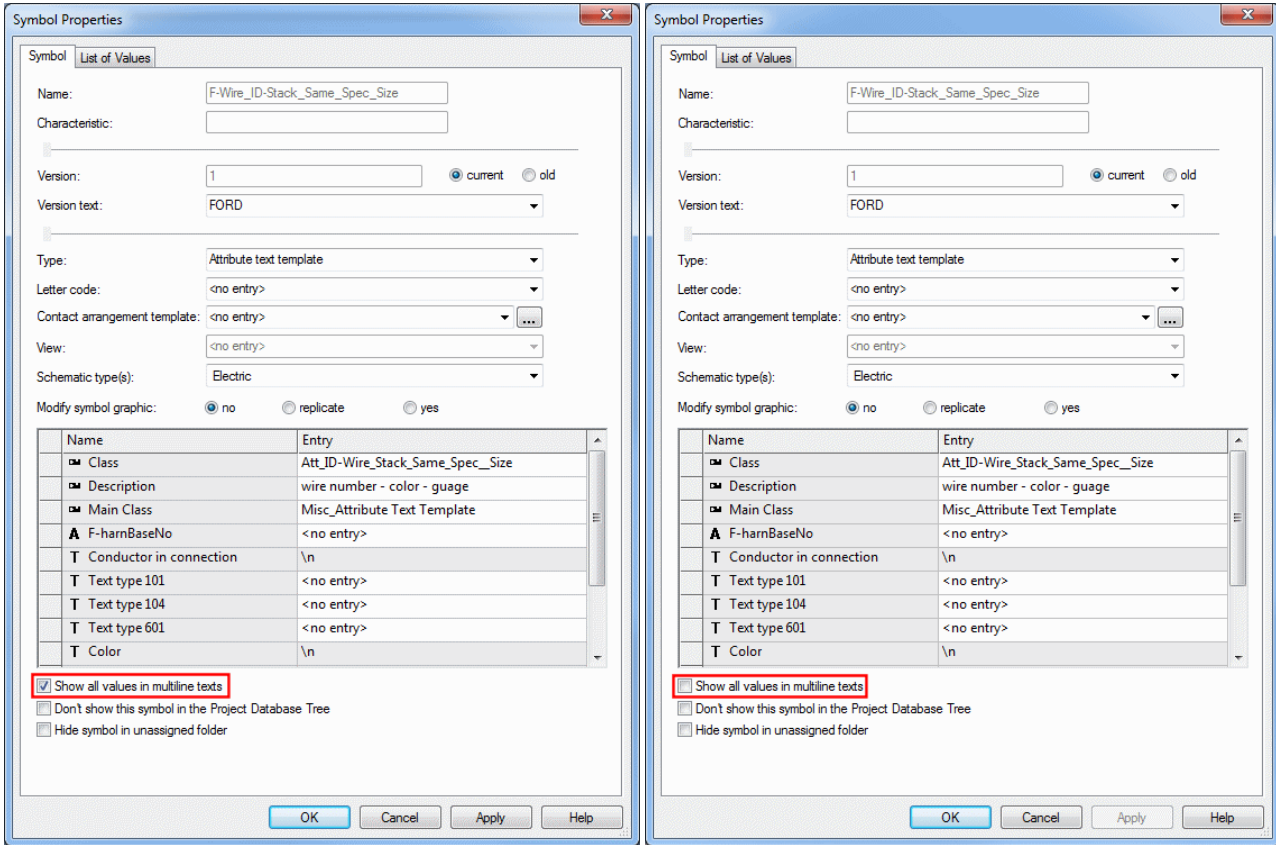
### Display of Multiline Texts Can Now Be Defined in the Symbol Properties for Symbols of the Type 'Attribute Text Template'

It's now possible to define how several values shall be displayed for symbols of the type **Attribute text template**.

**Note:** Several values may exist due to option combinations (i.e. different instances of a wire within a project).

To do so, there is the new check box **Show all values in multiline texts** in Project mode and Database Editor mode within the **Symbol Properties** in the tab **Symbol**:

**How the dialog is displayed within project mode:** **How the dialog is displayed within database editor mode:**

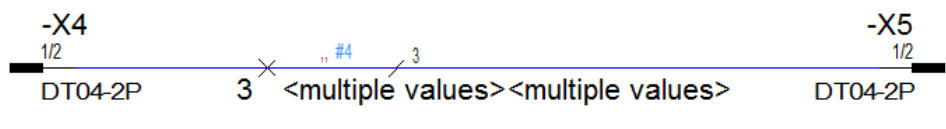


## Symbol Properties

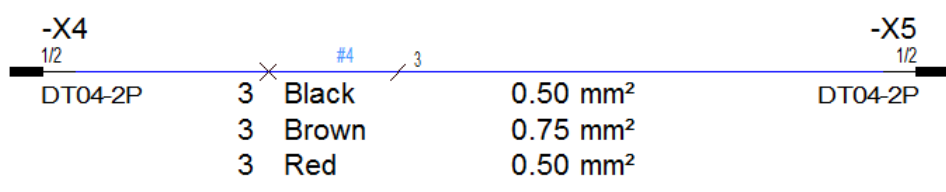
### Show all values in multiline texts

Several values are displayed on attribute text template symbols as follows when several instances of conductors/wires exist:

- If the check box is not checked, only one line with the entry **<multiple values>** is displayed:



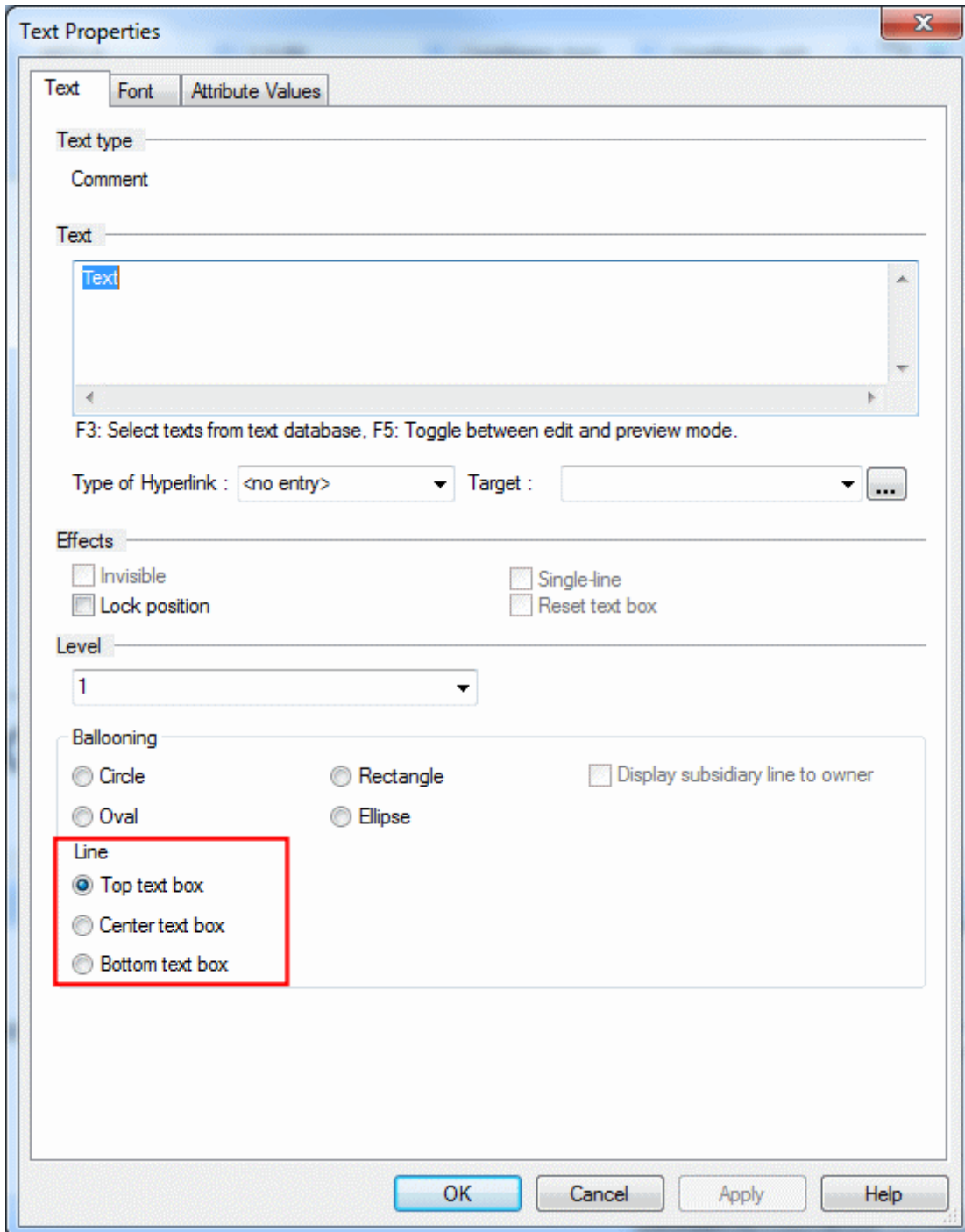
- If the check box is checked, the individual values are displayed in separate lines:



Reference(s): Designer-25426

**Lines Can Now Be Added to Text Boxes**

For texts a line can now be added to the text box. The line is placed at the 'top', 'center' or 'bottom' of the text box. To do so, there are three new radio buttons in the section **Line** in the tab **Text** of the **Text Properties**:



## Text Properties - Text

### Ballooning

#### Line

**Top text box** When activating this option, a line is added at the top of the text box.

**Center text box** When activating this option, a line is added in the middle of the text box.

**Bottom text box** When activating this option, a line is added at the bottom of the text box.

**Reference(s):** Designer-26481

### Parameter Values Can Now Be Defined using Placeholders for Translated Text Nodes from the Translation Table

A parameter value can now be defined for translated text nodes using a placeholder. This placeholder is created in the Translation Table in the Database Editor. In later projects the real parameter value is defined in the text.

#### To create the placeholder in the Database Editor:

- Start the Database Editor and select the command **Format -> Translation Table...** in the main menu
- Create the placeholder using the character combination **%s**  
**Note:** If **%s** is to be used as text and not as a placeholder, then use the following: **\%s**
- Enter the desired text in the required language along with its placeholder (see example with text node **&#1020;**):

Translation Table		
British English	American English	Deutsch
<All>	<All>	<All>
Motor %s Power consumption (kW) %s	Motor %s Power consumption (kW) %s	Leistungsaufnahme für Motor %s: %s kW
2 cylindars each	2 cylindars each	ie 2 Zylinder

- Save the modifications in the database.

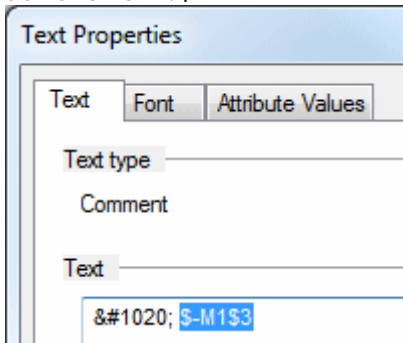
#### To define the parameter value in the project:

- Execute the command **Update configuration in project**

- Insert the translated text node (in the example **&#1020;**) in the project:

Motor -M1 Power consumption (kW) 3  
 Motor -M1 Power consumption (kW) 3  
 Leistungsaufnahme für Motor -M1: 3 kW

- The parameter value is defined using a dollar sign **\$**:  
**Note:** If the dollar sign **\$** is to be used as text and not as a parameter value, then do so as follows: **\\$**



- The text is now displayed with the defined parameter values:

Motor **-M1** Power consumption (kW) **3**  
 Motor **-M1** Power consumption (kW) **3**  
 Leistungsaufnahme für Motor **-M1**: **3** kW

**Note:**

- If no parameter value is defined, an empty space appears in the text.
- For multiline texts the parameter value must appear at the end of the text.

**Reference(s):** Designer-11332

**Ten New Sheet Text Types in the Database**

In **E<sup>3</sup>.series** there are ten new sheet text types under **Edit Text Types** in Database Editor Mode. With these text types one can define their own text nodes in the project to have a better overview when working with substructures.

The following new text types are available:

Text type	Text name	Max. length	Sheet text	Copy from selected sheet	Changeable by script only	Connection target	Use from active pin terminal	Assigned attribute
1150	Number of sheets in folder	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1151	Previous sheet in folder	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1152	Following sheet in folder	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1153	First sheet in folder	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1154	Last sheet in folder	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1155	Has following sheet in folder?	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1156	Previous sheet in project	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1157	First sheet in project	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1158	Last sheet in project	64	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>
1159	Has following sheet in project?	255	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<no entry>

Type	Name	Max. Length	Owner	Description
1150	Number of sheets in folder	255	Sheet	Shows the number of sheets in the folder.
1151	Previous sheet in the folder	64	Sheet	Shows the previous sheet in the folder.
1152	Following sheet in the folder	64	Sheet	Shows the next sheet in the folder.
1153	First sheet in folder	64	Sheet	Shows the first sheet in the folder.
1154	Last sheet in folder	64	Sheet	Shows the last sheet in the folder.
1155	Has following sheet in folder?	255	Sheet	<p><b>Plus sign (+):</b> A following sheet is available in the folder.</p> <p><b>Minus sign (-):</b> There is no following sheet in the folder.</p>
1156	Previous sheet in project	64	Sheet	Shows the previous sheet in the project.
1157	First sheet in project	64	Sheet	Shows the first sheet in the project.
1158	Last sheet in project	64	Sheet	Shows the last sheet in the project.
1159	Has following sheet in project?	255	Sheet	<p><b>Plus sign (+):</b> A following sheet is available in the project.</p> <p><b>Minus sign (-):</b> There is no following sheet in the project.</p>

The following text types are already there:

Type	Name	Max. Length	Owner	Description
------	------	-------------	-------	-------------

31	Number of sheets in project	255	Sheet	Shows the number of sheets in the project.
1020	Following sheet in project	64	Sheet	Shows the following sheet in the project.

**Reference(s):** Designer-08042



## User Interface

- **'Full Screen Mode' for Workspace in E3.series**

### 'Full Screen Mode' for Workspace in E3.series

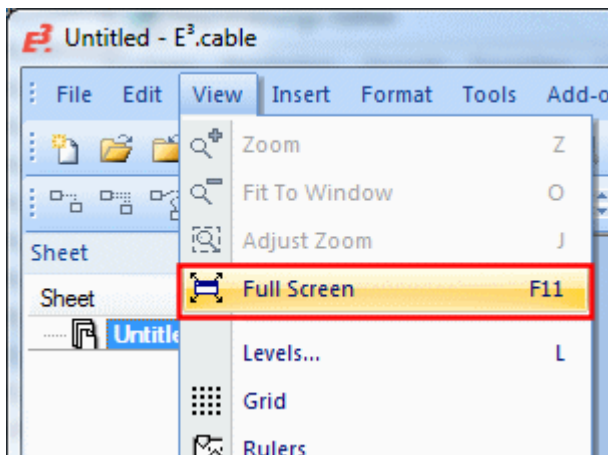
In **E<sup>3</sup>.series** there is now a full screen mode. In this full screen mode the workspace area of one or more sheets is projected on the complete screen. This is advantageous for drawings with numerous components.

**Note:** This functionality is only available for a project that is already opened.

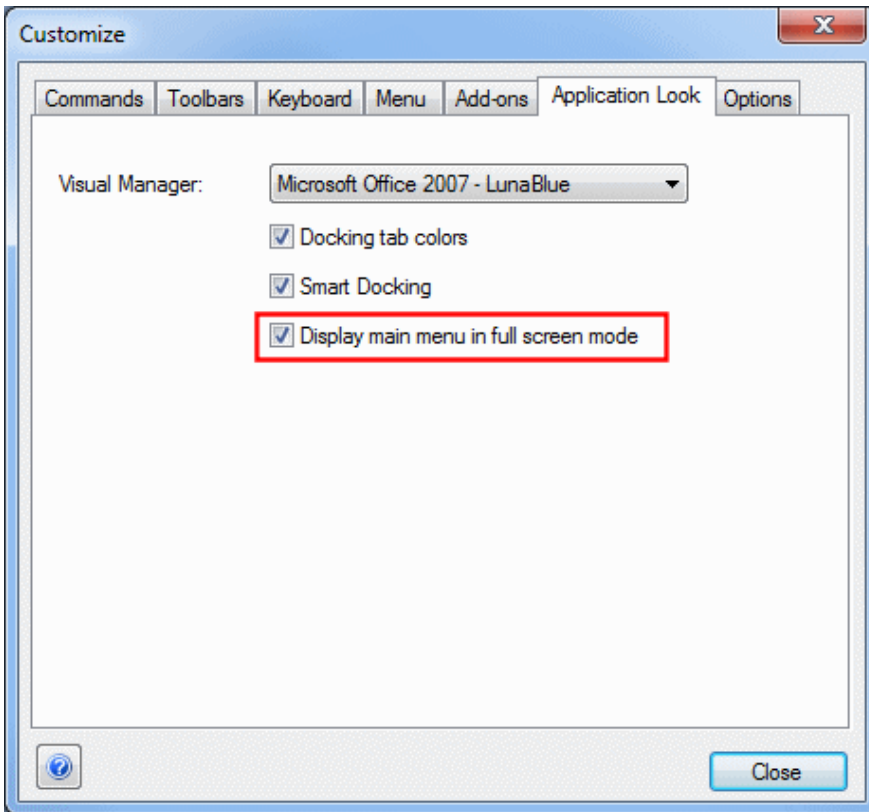
### Starting the Full Screen Mode

To start the full screen mode,

- select the command **View -> Full screen** in the main menu, or
- press the **F11** key.



In the **Customize** dialog (called up using **Tools -> Customize - Application Look**) there is the new check box **Display main menu in full screen mode**:



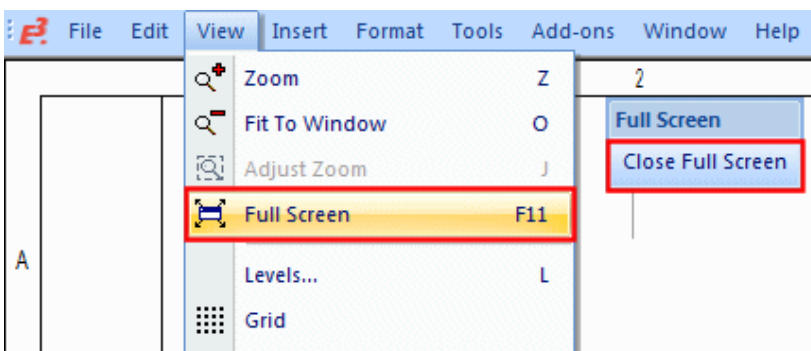
If checked, the main menu toolbar is displayed in full screen mode. Using the main menu toolbar the project, database and output windows as well as tables can be displayed in full screen mode.

**Note:** Changes to workspace configurations are not saved when closing the full screen mode. The saving and loading of workspace configurations is deactivated in full screen mode.

### Closing the Full Screen Mode

To close the full screen mode,

- select the command **View -> Full Screen** in the main menu, or
- select the command **Close Full Screen** in the **Full Screen** dialog, or
- press the **F11** key.



**Reference(s):** Designer-15129

## Variants / Options

- **Variants/Options Tree Can Now Be Structured using Folders**
- **All Elements in Variants/Options Tree Can Now Be Sorted in Ascending or Descending Order**

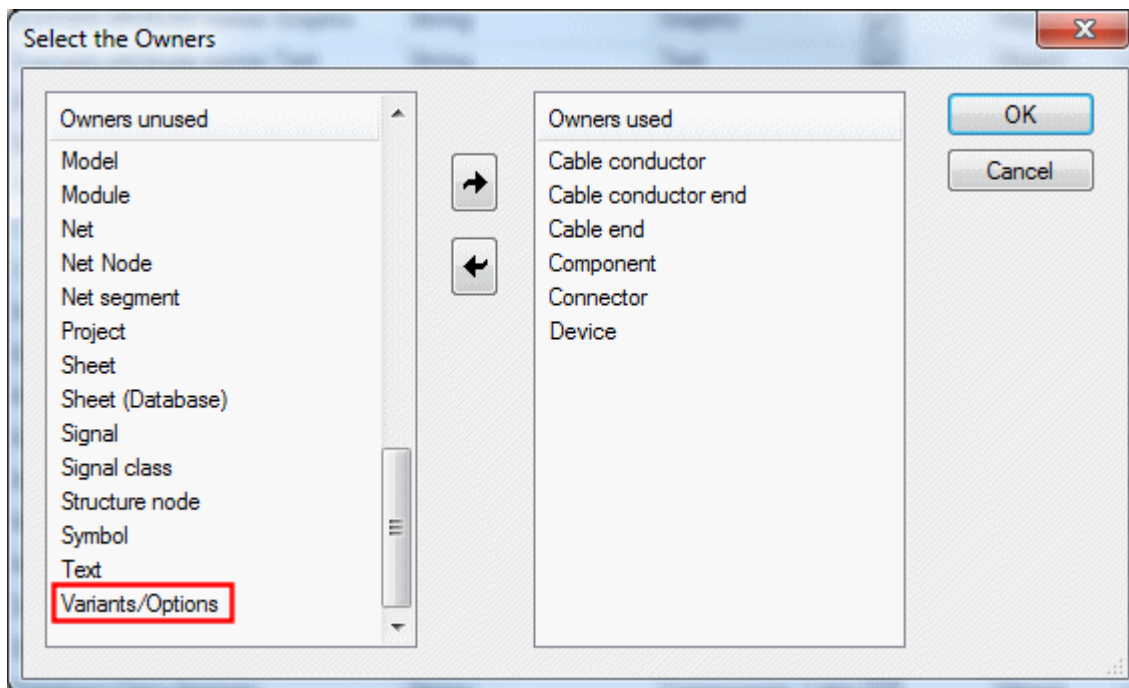
### Variants/Options Tree Can Now Be Structured using Folders

From now on the Variants/Options Tree can be displayed as a structured folder. This folder represents the values of the sorting attributes, which are created automatically after being defined. If one of the following objects...

- o variants
- o options
- o packages
- o configurations
- o harness families

has this sorting attribute, it is automatically sorted in the corresponding folder.

So that the attributes can be displayed in the Variants/Options Tree, the new owner '**Variants/Options**' must be defined on the respective attribute in Database Editor Mode under **Format -> Attribute Names...**:

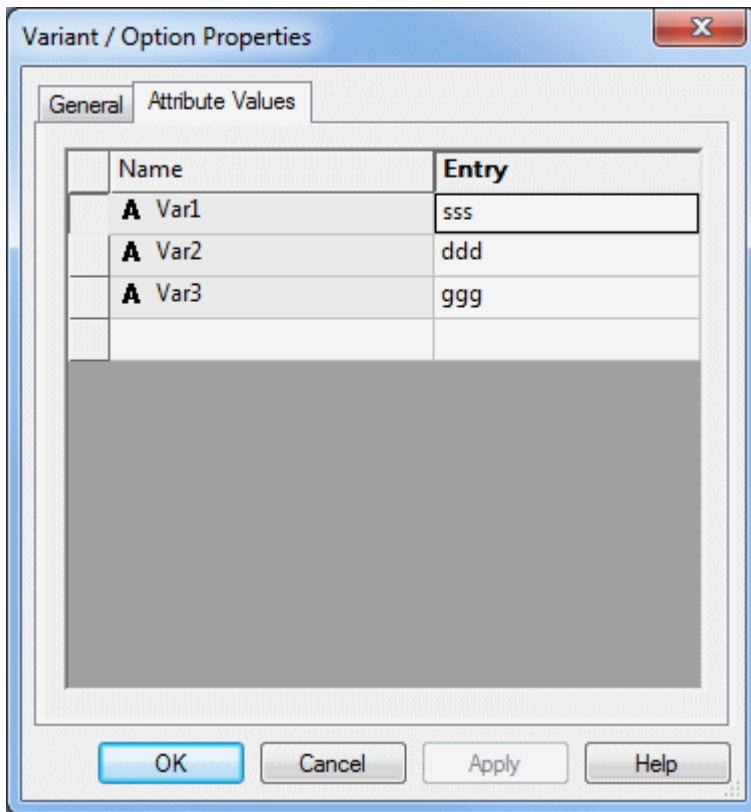


**Note:** For existing projects the configuration must be updated afterward in the project.

Now the required attributes must be defined for the objects (variants, options, packages, configurations and harness families).

- Right-click on the desired object in the Variants/Options Tree and select the context menu command **Properties...**,

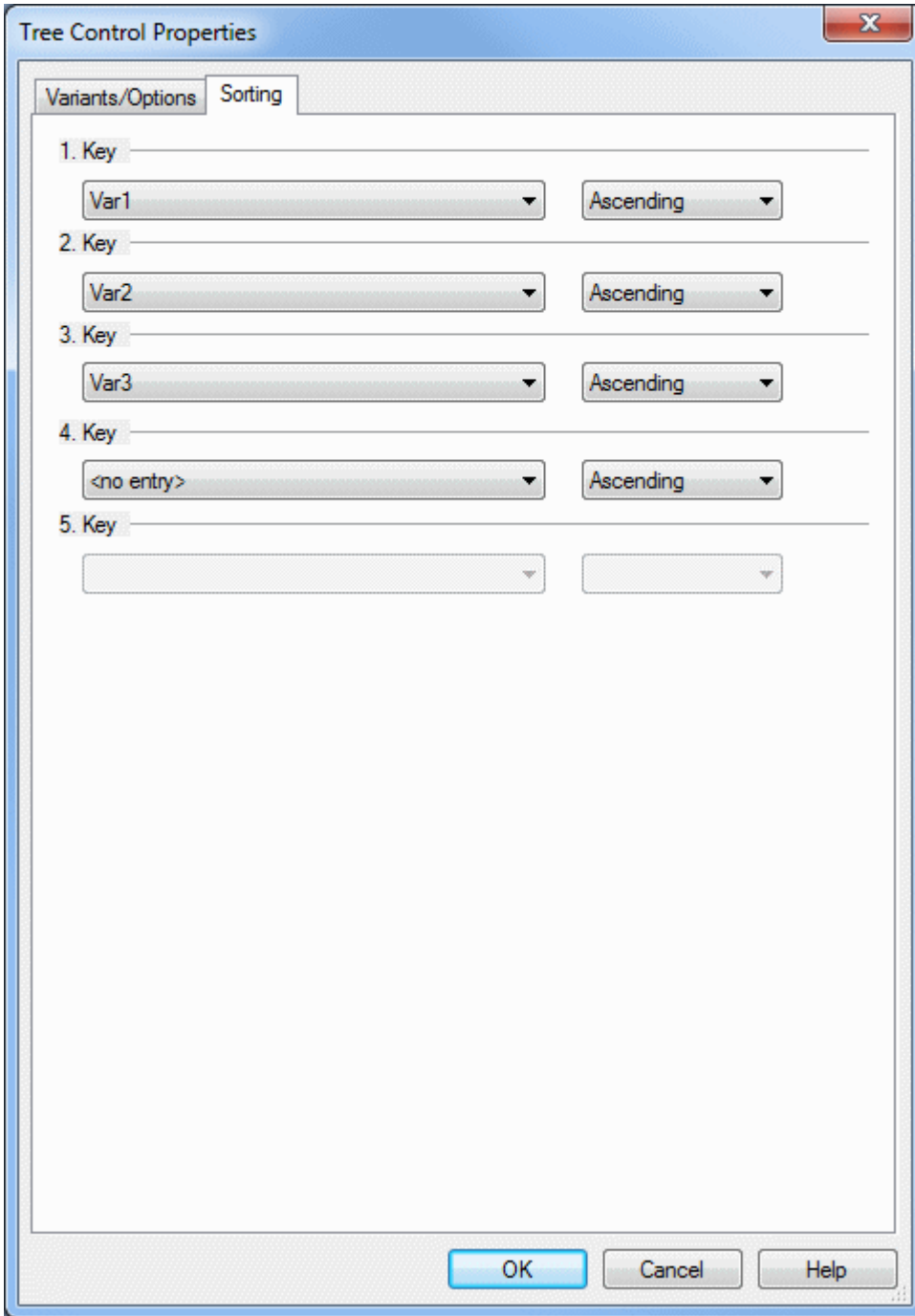
- next select the new **Attribute Values** tab in the 'Properties' dialog:



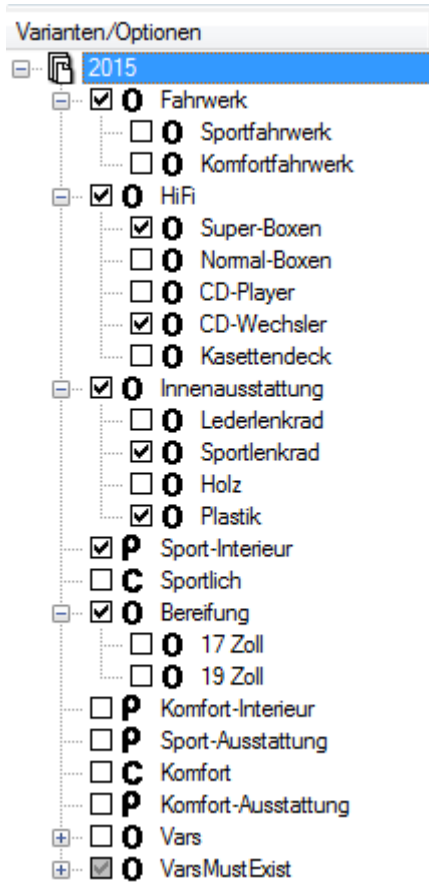
The next step is to define which attributes shall be used as sorting criteria in the Variants/Options Tree.

- Right-click in the Variants/Options Tree in the Project Window and select the context menu command **Tree Control Properties...**,
- select the new **Sorting** tab in the 'Tree Control Properties' dialog

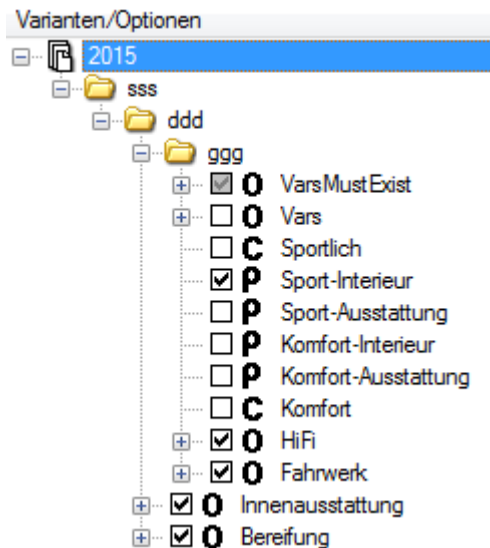
Using **Keys 1-5** a sorting attribute can be defined in the respective drop-down list for the Variants/Options Tree:



**Variants/Options Tree without Structuring:**



**Variants/Options Tree with Structuring:**



**Note:**

- When exporting/importing the structure, the attributes on the objects are transferred as well.
- When moving objects to another folder, they receive the attributes necessary for the structuring.
- When moving objects in a folder, they are sorted after the last object in the folder.
- Folders can be renamed. In doing so, the attribute values of all underlying objects are correspondingly adapted.

- The folder has a context menu command for creating objects. These receive the required attributes for the structuring and are sorted after the last object in the folder.

**Note:** If the structure is locked,

- o the folders are displayed as grayed-out in the tree
- o the objects can no longer be moved to another folder
- o the folders can no longer be renamed
- o the attribute values in the Properties can no longer be changed on the objects.

### New COM Commands in OptionInterface for Editing Attributes

The following new COM commands are now also available in the **OptionInterface**:

- **AddAttributeValue ( name, value )**
- **DeleteAttribute ( name )**
- **GetAttributeValue ( name )**
- **SetAttributeValue ( name, value )**
- **HasAttribute ( name )**
- **GetAttributeIds ( ids, [attnam] )**

**Reference(s):** Designer-24877

### All Elements in Variants/Options Tree Can Now Be Sorted in Ascending or Descending Order

From now on all elements (variants/options, packages/configurations, harness families/harness derivatives) in the Variants/Options Tree can be sorted in ascending or descending order. To do so, there are the new context menu commands **Sort ascending** / **Sort descending**.

**Note:** Depending on which level the context menu command is executed, all underlying (sub) elements are sorted.

### Enhancement to OptionInterface for Sorting Variants/Options in Project Tree

The following new COM command is available in the OptionInterface:

- **ret = opt.Sort ( ascending )**
  - o **ascending** The following values are possible:
    - **0** Sort descending (default)
    - **1** Sort ascending
  - o **ret** The return value is 1 if successful, otherwise 0.

**Reference(s):** Designer-14957

## Views

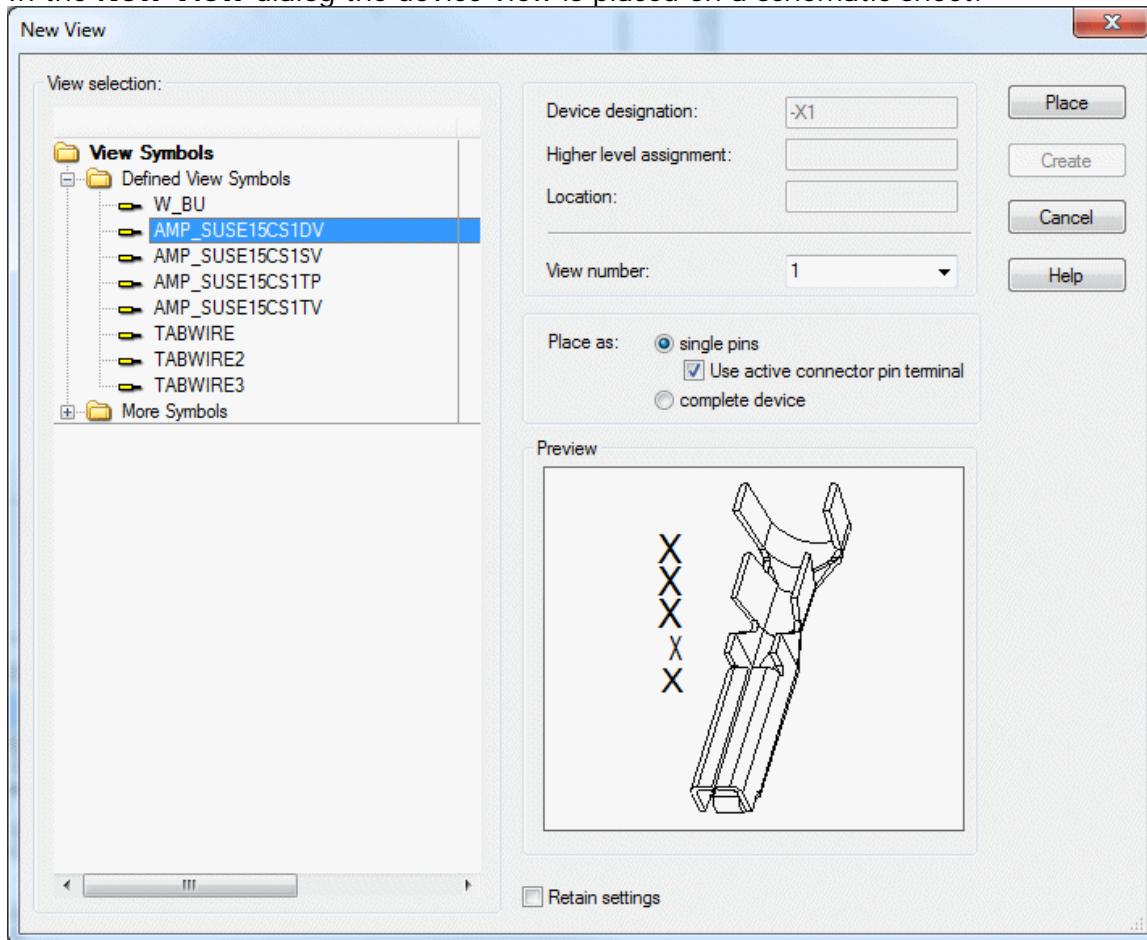
- **Dialogs 'New View' and 'Place on Formboard' Have Been Enhanced**
- **Control whether Active Connector Pin Terminal Symbol Shall Be Displayed for Views in the Device Properties**
- **Keep pinview symbols at project update**

### Dialogs 'New View' and 'Place on Formboard' Have Been Enhanced

When placing a new device view, the functionality in the following dialogs has been changed:

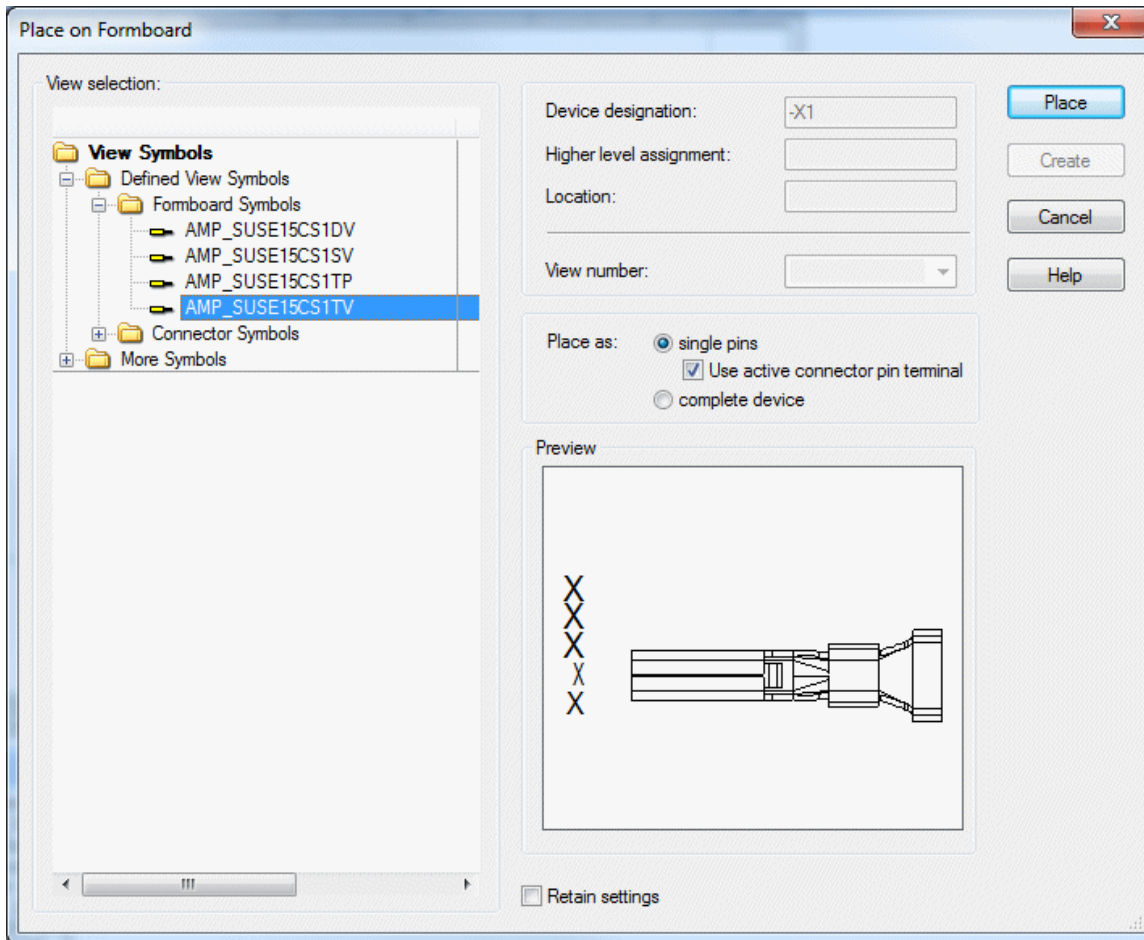
- **New View**
- **Place on Formboard**

In the **New View** dialog the device view is placed on a schematic sheet:



In the **Place on Formboard** dialog the device view is placed on a formboard sheet:





If the check box **Use active connector pin terminal** is checked, the symbol of the connector pin terminal is now used for the respective pin.

**Note:**

- If the same connector pin terminal is used on all pins, this connector pin terminal symbol appears in the Preview Window.
- If different connector pin terminals are used on the pins, a corresponding message appears in the Preview Window.
- In the tree structure the user-defined view number is now also displayed (if available).

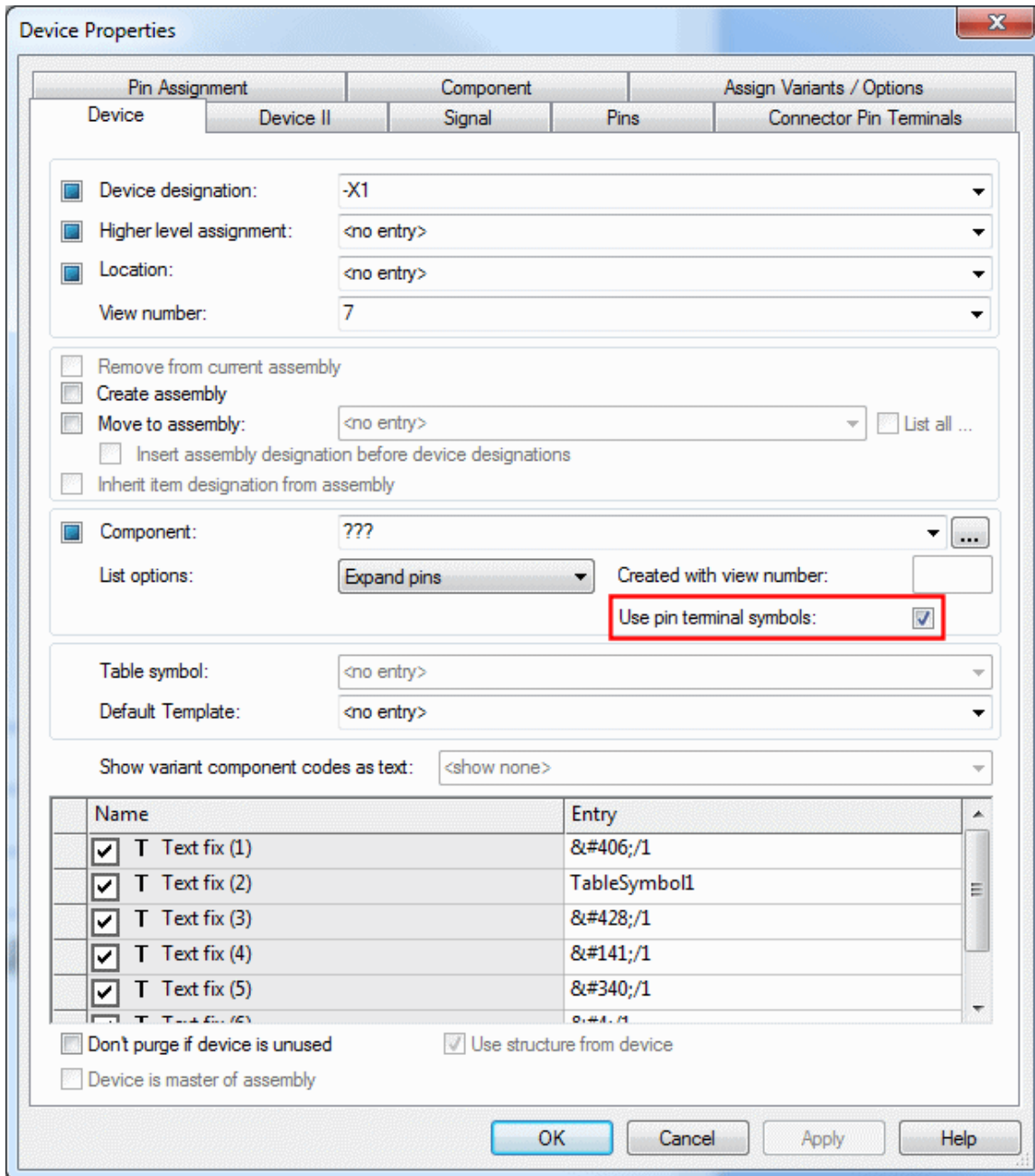
Furthermore, the previous selection field **complete connector** has been renamed **complete device** (So far, this functionality was only valid for connectors). If this selection field is active, the device view is placed as the complete device.

**Reference(s):** Designer-25703

**Control whether Active Connector Pin Terminal Symbol Shall Be Displayed for Views in the Device Properties**

For device views and pin views it's now possible to control whether the active connector pin terminal symbol shall be placed instead of the previous view symbol (in case a symbol is defined).

To do so, there is a new check box **Use pin terminal symbols** in the **Device** tab of the **Device Properties**.



If the check box **Use pin terminal symbols**: is checked, the presently set connector pin terminal symbol is used.

**Note:**

- This is only valid though in the single pin display. If a single pin is split from a complete connector, the active connector pin terminal symbol is used.
- Additionally, please note that the view of the symbol (top view, side view,...) is retained. If the previous view symbol was a formboard symbol, then a formboard symbol (if available) would be used for the active connector pin terminal.
- If the connector pin terminal symbol cannot be determined/swapped, a corresponding message appears (with the item designation and 'Jump to' functionality) in the message window.

- When placing a device view, the check box **Use pin terminal symbols:** is automatically set to active as soon as the options **Place as: single pins** and **Use active connector pin terminal** are activated in the **New View** dialog.
- When updating components in the project, the setting of the check box **Use pin terminal symbols:** is considered.

If the check box **Use pin terminal symbols:** is deactivated, the symbol momentarily placed is used and an automatic update is no longer possible.

## Multi-User Limitations

The following is valid in the multi-user environment:

If a variant/option (for example with pins, placed wires/conductors or assemblies) is involved with a device/pin view, the connector pin terminal symbol is not swapped and a warning appears in the message window instead.

Nevertheless, it is possible to get around this restriction in order to display the correct connector pin terminal symbol, e.g. for a printout.

To do so, there is the new icon  **Update pin terminal symbols** in the **Placement** toolbar.

**Note:** This symbol list is only active in multi-user projects.

## New COM Commands in DeviceInterface for the Setting 'Use pin terminal symbols'

In **DeviceInterface** there are the following new COM commands:

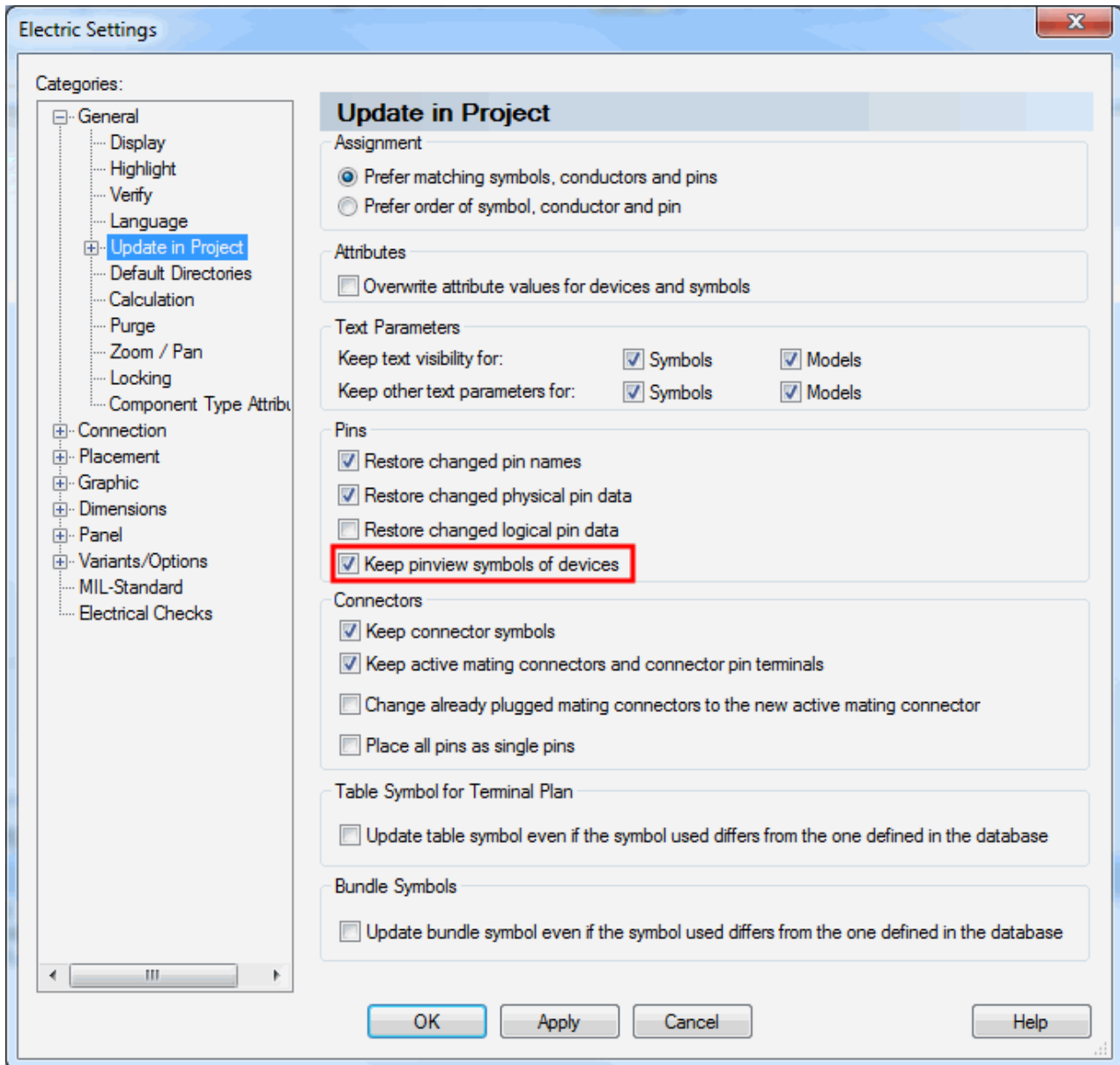
- **ret = dev.IsPinTerminalSymbolsUsed**
  - o **ret** The return value is 1 if the connector pin terminal symbols are used for the set device, otherwise 0.
- **ret = dev.SetPinTerminalSymbolsUsed ( use )**
  - o **use 1** The connector pin terminal symbols are used for the set device.
  - o **use 0** The normal view symbols are used for the set device.
  - o **ret** The return value is 0 if successful, otherwise -1.

**Reference(s):** Designer-22521

## Keep pinview symbols at project update

The setting Update in Project offers the possibility to control whether already placed pinview symbols shall be kept or not.

For using this function click the check box **Keep pinview symbols of devices** within the **Pins'** area in the settings category **General – Update in Project:**



## Update in project

### Keep pinview symbols of devices

With active option all pinview symbols for devices are kept at project update.

With inactive option no pinview symbols are kept for devices at project update but the symbol defined as attribute within the database is used.

**Note:** When no symbol definition is existing over an attribute on the pins, the set symbol (under **Settings -> Placement -> Symbols->Symbol Views-> For device pins** ) will be used.

The following setting\_name can be used for the COM-methods **prj.GetSettingValue** and **prj.SetSettingValue**:  
KEEP\_PINVIEW\_SYMBOLS\_OF\_DEVICE

This check box is *user-specific* for Multiuser-projects.

**Reference(s):** Designer-25350



## Americas

### North America

Zuken USA Inc.  
Westford, MA 01886, USA  
Tel: +1 978 692 4900

### Asia

#### Japan

Zuken (World Wide Head Office)  
Yokohama, Kanagawa 224-8585, Japan  
Tel: +81 45 942 1511

#### China

Zuken (Beijing Representative Office)  
ChaoYang District, Beijing, China, 100027  
Tel: +86 10 8447 5076

#### Korea

Zuken Korea Inc.  
Seoul 135-283, Korea  
Tel: +82 2 5648031

#### Singapore

Zuken Singapore Pte Ltd.  
#22-05 Gateway East, Singapore 189721  
Tel: +65 6392 5855

#### Taiwan

Zuken Taiwan Inc.  
Taipei 110, Taiwan  
Tel: +886 2 7718 1116



## Europe

### Germany

Zuken GmbH (European HQ)  
D-85399 Hallbergmoos, Germany  
Tel: +49 89 607969 00

Zuken E3 GmbH  
D-89079 Ulm, Germany  
Tel: +49 7305 9309 0

Zuken E3 GmbH  
D-30659 Hannover, Germany  
Tel: +49 511 8595 9489

### Switzerland

Zuken E3 GmbH  
CH-5506 Mägenwil, Switzerland  
Tel: +41 56 437 08 90

### United Kingdom

Zuken UK Ltd.  
Bristol, BS32 4RF, UK  
Tel: +44 1454 207 801

### France

Zuken S.A.  
#91974 Les Ulis Cédex, France  
Tel: +33 1 69 29 48 00

### Italy

Zuken S.r.l.  
20090 Milanofiori Assago, Milan, Italy  
Tel: +39 02 575 921

### Netherlands

Zuken GmbH  
NL-6075 HA Herkenbosch, The Netherlands  
Tel: +31 475 520 998