



AUTOMATION

News in version 21

15/10-2019

TABLE OF CONTENTS

PROJECTS WITH REFERENCE DESIGNATIONS	7
LET AUTOMATION CORRECT THE REFERENCE DESIGNATIONS	7
CORRECTING REFERENCE DESIGNATIONS DURING PROJECT LOAD	7
THE WRITING STYLE MATTERS – PART 1	8
THE WRITING STYLE MATTERS – PART 2	9
CHANGES IN "REFERENCE DESIGNATIONS - SETTINGS"	10
REFERENCE FRAMES AND DESCRIPTIONS FOR REF. DESIGNATIONS	10
REFERENCE FRAME SETTINGS MEMORIZED	10
NEW BUTTON FOR EASY SELECTION OF PAGE REF. DESIGNATIONS	11
UNDERSTANDING NAMES, COMPONENTS AND SYMBOLS	11
LISTS	13
NEW SORTING OPTION FOR PARTS AND COMPONENT LISTS	13
PLC LIST IMPORT/EXPORT WITH DATA FIELDS	14
CONNECTION LIST AS LIST IN THE PROJECT	15
NEW TEMPLATE FOLDERS IN FILES => NEW	15
HOW TO USE DRAWING HEADERS WITH MULTIPLE LANGUAGES (ML)	16
HOW TO CHANGE LANGUAGE IN THE DRAWING HEADER	16
'SILENT' FUNCTION IN LIST-TO-FILE FROM THE TOOLS MENU	17
LINE DATA FIELDS IN LISTS	18
NEW DATA FIELDS IN THE TABLE OF CONTENTS	18
NEW FIXED SYMBOL DATA FIELD - QUANTITY	18
SELECTING AND SHOWING ACCESSORIES	19
ADDING ADDITIONAL MECHANICAL ACCESSORIES	19
SUPPORT SYMBOLS AND ACCESSORIES	20
SUPPORT SYMBOLS IN PROJECTS FROM PREVIOUS VERSIONS	20
SUPPORT SYMBOLS IN THE COMPONENT MENU IN VERSION 20	20
SUPPORT SYMBOLS IGNORED IN THE COMPONENT MENU IN VERSION 21	20
ON TERMINALS	21
DRAWING LINES THROUGH TERMINALS	21
MULTI PLACING MOTOR TERMINALS WITH UVW	21
TERMINALS WITH ACCESSORIES AND TERMINALS IN COMPONENT GROUP 0 ...	22
A BRIEF HISTORY LESSON ON COMPONENT GROUP 0	22
TERMINALS WITH ACCESSORIES NOT PLACED ON MECHANICAL PAGES	23
TERMINALS WITH ACCESSORIES PLACED ON MECHANICAL PAGES	23
ACCESSORIES SELECTED AS SUPPORT SYMBOLS	23
ACCESSORIES CHOSEN FROM THE MECHANICAL ACCESSORIES TAB	24
HOW TO CORRECT ERRORS DUE TO COMPONENT GROUP 0	24

REPLACING ALL SYMBOLS IN A PROJECT25

REPLACE ALL SYMBOLS WITH SYMBOLS FROM ANOTHER DIRECTORY 25

EXISTING SYMBOL DATAFIELDS ARE INCLUDED 25

 ADDITIONAL INFORMATION ABOUT REPLACING SYMBOLS25

LINES.....26

CALCULATED LINE DATA FIELDS (FORMULA EDITOR) 26

 CREATE THE NECESSARY LINE DATA FIELDS26

 CREATE A CALCULATED LINE DATA FIELD26

TEXT ANCHOR FOR LINE DATA 27

CONTROLLING GAP ARCS USING PRIORITIES ON LINES 28

LINE DATA FIELDS IN LISTS 28

TEXT ADJUSTMENT29

LOCKING DATA LISTS FOR SYMBOL DATA FIELDS 30

THE LOCK IS ALSO DISPLAYED IN COMPONENT DATA 30

ENHANCED ALIGN FUNCTION.....31

ALIGNING SYMBOLS ACROSS PROJECT PAGES 31

ALIGNING - WITH AND WITHOUT <CTRL> 31

LICENSE SYSTEM IMPROVEMENTS32

TANKING LIMITATIONS 32

AUTOMATION SERVICE 32

NEW LICENSE SERVER 32

ENHANCED ADMINISTRATOR PROGRAM.....33

TEXT SETTINGS: THE ALL FUNCTION CONTROLLED BY ADMINISTRATOR 33

FOLLOW CONNECTED 33

MINOR CHANGES34

MENU ITEMS REARRANGED 34

THE PAGE MENU 34

RECTANGLES WITHOUT CONNECTIONS ARE NOW NON-CONDUCTING 34

EXTRA DESCRIPTION ON REFERENCE DESIGNATIONS 34

THE TABS IN PANELROUTER 35

INSERTING STATES AND REMOVING EMPTY STATES 35

THE NEW ICON AND FILE => NEW HAVE IDENTICAL FUNCTIONALITY 36

SELECTING IO ADDRESSES 36

THE UPDATE FROM DATABASE FUNCTION 37

ROTATING OBJECTS WITH 10° 38

DEFAULT CABLE QUANTITY 38

OBJECT LISTER - NEW DEFAULT SETTINGS..... 39
OBJECT LISTER - SELECT SYMBOL TYPES ON SYMBOLS TAB..... 39
WARNING WHEN OPENING WRITE PROTECTED FILES..... 39

THE MOUNTING ASSISTANT40

MULTIPLE USERS CAN WORK ON THE SAME PANEL 40
ALL CREATED CASE NUMBERS IN THE PROJECT ARE DISPLAYED..... 40
THE SURVEY WINDOW NOW HAS A "MOUNTED" BUTTON..... 40
CHANGING STATUS FOR ALL WIRES IN A SINGLE OPERATION 41

THE COMPONENT MENU42

PROJECTS WITH REFERENCE DESIGNATIONS

When you load a project with reference designations and the project is older than version 18, the program will now 'translate' the reference designations to the new – version 18 – format.

Before version 18, it was possible to create many formats that did not all meet the standard (IEC ISO 81346). We generally have the attitude that we do not want to change projects, but when you load an unstructured project, the output also becomes unstructured. Our intent was to make it easy to load old projects and have these formats corrected, but we have experienced problems implementing this.

The **Reference designations - Settings** dialog box will now show a lot of check marks that are set when you load an 'old' project. The intention is that you can then remove them all and, in this way, make the designations work according to the standard.

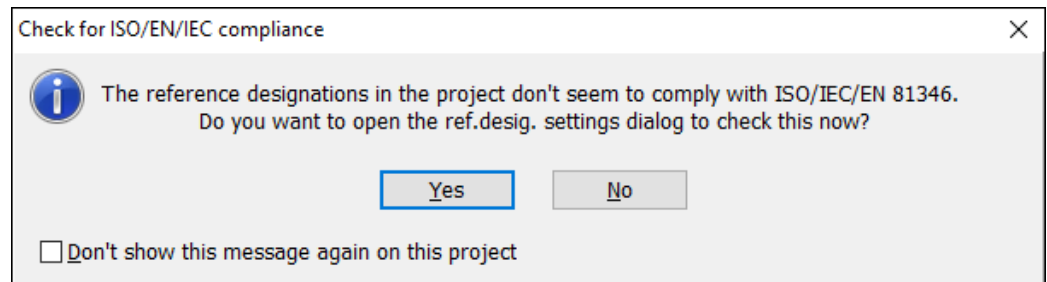
LET AUTOMATION CORRECT THE REFERENCE DESIGNATIONS

In version 21 it is possible to let **Automation** correct the reference designations to comply with the standards.

This way, you will for instance be able to avoid situations where some designations have double prefix.

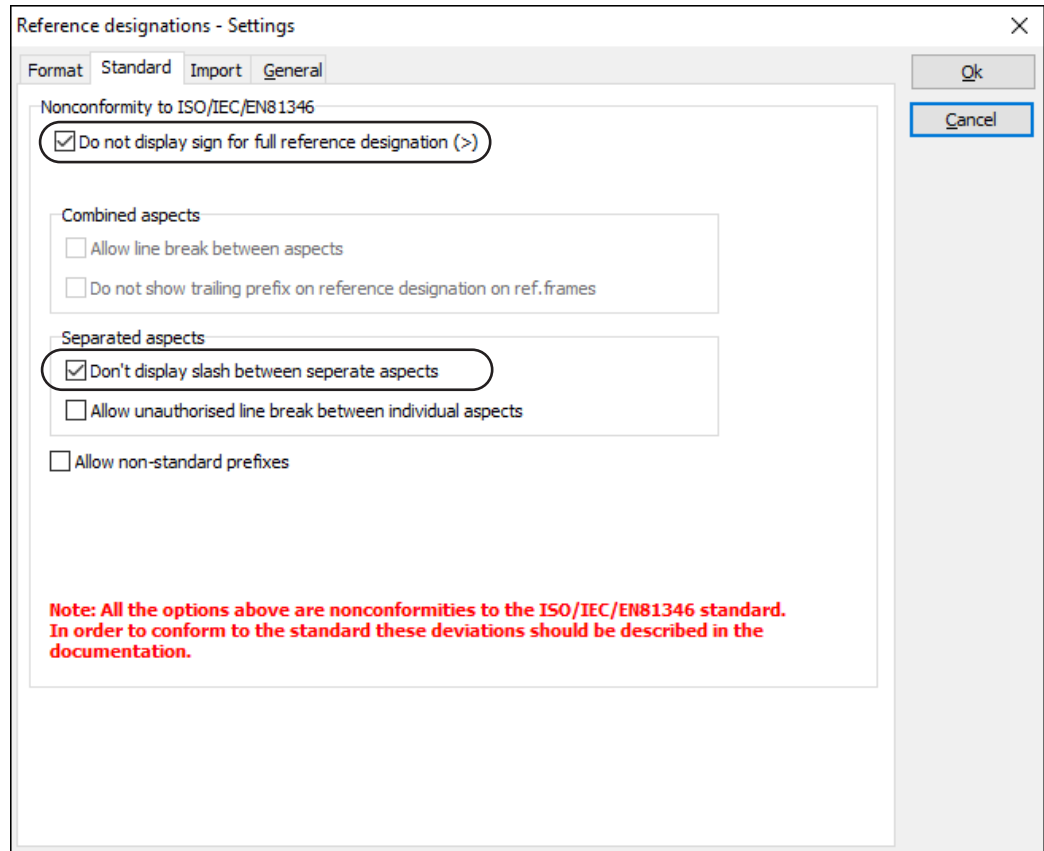
Correcting reference designations during project load

When you open a project, **Automation** checks the reference designation settings. When deviations from the standard are detected, a dialog box appears:



Here you can either click **Yes** to go directly to the **Reference designations - Settings** dialog box — where you can correct the reference designations settings — or click **No** to open the project as it is.

In the **Reference designations - Settings** dialog box all check marks represent deviations from the standard. You can now clean up the deviations by removing the check marks:



As you remove the check marks you are informed, that you are about to correct the reference designations, and when you have removed all of the check marks, the reference designations are in compliance with the standards.

We recommend applying **Separated aspects** (more about this later), but your overall choice in this relation is not affected by settings/check marks on this tab.

THE WRITING STYLE MATTERS – PART 1

Please be aware that it does matter, how you show reference designations in projects:

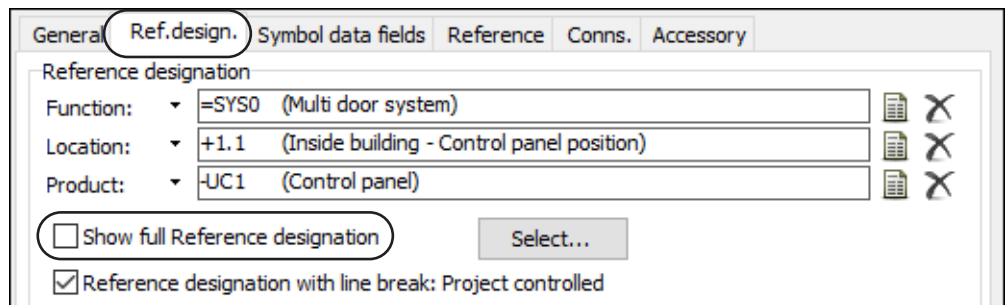
THE SET OF REFERENCE DESIGNATIONS CONTAINS	REFERENCE DESIGNATIONS ON INDIVIDUAL LINES	REFERENCE DESIGNATIONS IN A SINGLE LINE
THREE REFERENCE DESIGNATIONS (SEPARATE ASPECTS)	=A2 -B2 +C3	=A1/-B2/+C3
A SINGLE REFERENCE DESIGNATION (COMBINED ASPECTS)	=A1-B2+C3	=A1-B2+C3
TWO REFERENCE DESIGNATIONS (HYBRID)	=A1-B2 +C3	=A1-B2/+C3

The easiest solution will often also be the best solution, simply because it is difficult to know when the function depends on the location or when the opposite is the case. When you select separate designations, this will not affect your component marking.

In short, our recommendation is to use the default selection – which is also the default in the standard – which is Separate aspects.

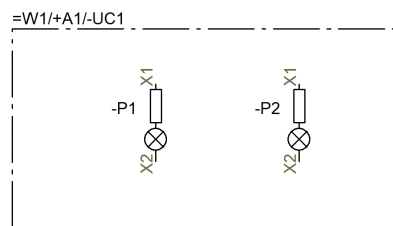
THE WRITING STYLE MATTERS – PART 2

When you place a symbol in a diagram, you enter the **Component data** dialog box. If you here choose **Show full reference designation** in the **Ref.design.** tab, this changes how the reference designation is displayed in the diagram — and hereby also the *meaning* of what you see in the diagram.



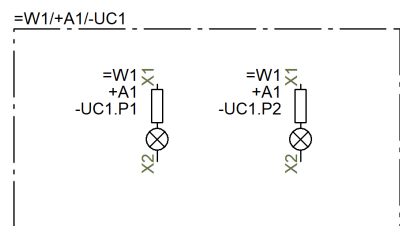
The recommendation is to use the default setting — which is *not* to choose **Show full reference designation**. This is in compliance with the standards.

Whether you use the reference frames or the reference designation for the page, the reading conventions about concatenating names within the same aspect apply as seen below:



The full reference designations of the components are:
 =W1 / +A1 / -UC1.P1
 =W1 / +A1 / -UC1.P2

The common part is outside the ref.frame

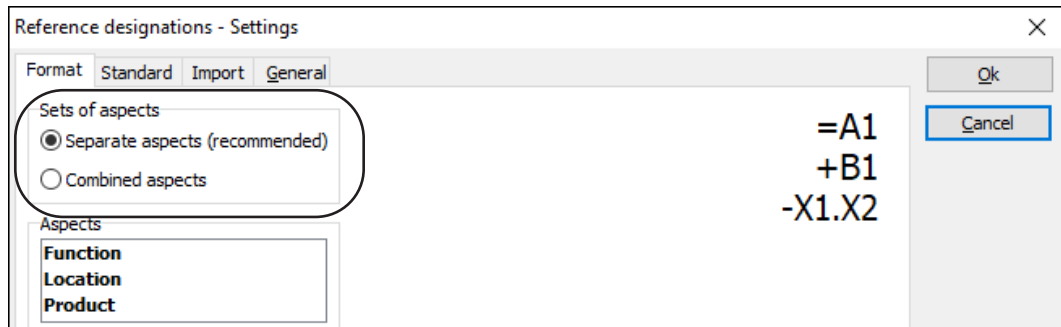


The full reference designations of the components are:
 =W1.W1 / +A1.A1 / -UC1.UC1.P1
 =W1.W1 / +A1.A1 / -UC1.UC1.P1

The common part is outside the ref.frame
 AND
 you read = with =, + with +, - with -

CHANGES IN "REFERENCE DESIGNATIONS - SETTINGS"

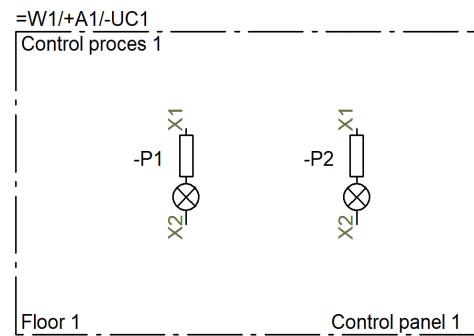
Apart from the tidy-up function, we have changed the order of the Sets of aspects, meaning that the recommended **Separate aspects** setting is now at the top:



REFERENCE FRAMES AND DESCRIPTIONS FOR REF. DESIGNATIONS

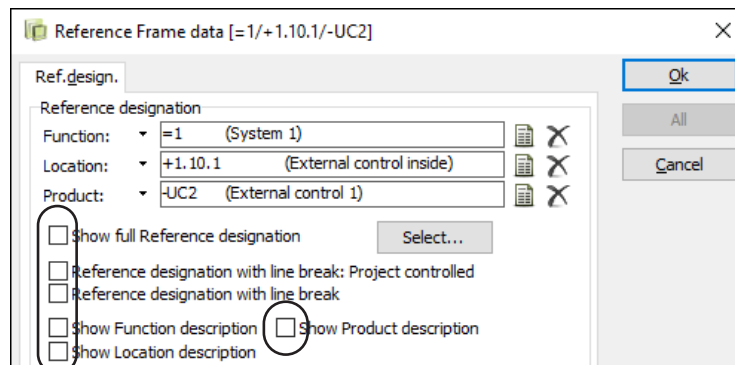
When you use reference frames in your projects it is possible to show descriptions for the designations.

From version 21, the texts will follow their 'corners' when you change the frame's size or position.



Reference Frame settings memorized

The program now remembers the last selections in the **Reference Frame data** dialog box.

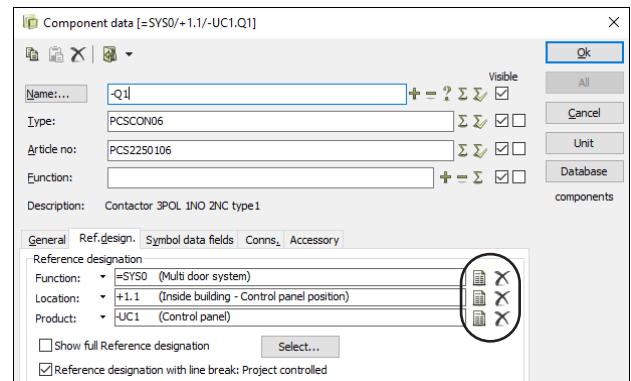


But please be aware that some of these settings are not only about graphics, they also have a significance attached to them. — See “The writing style matters – part 2” on page 9.

NEW BUTTON FOR EASY SELECTION OF PAGE REF. DESIGNATIONS

You can now quickly delete a reference designation or to assign the reference designation for the page to an object.

This is done by using the new buttons to the right of each aspect.



UNDERSTANDING NAMES, COMPONENTS AND SYMBOLS

The following is to clarify some basic concepts in **Automation**:

A **symbol** is a part of a **component** which can be part of a **name**.

A **symbol** is a drawing object that shows function and connections of a component. The component symbols can have different (symbol) properties, which makes them fit the component types they represent.

A **component** is a physical unit, which you 'can buy'; that means a unit defined by one or more symbols with the same ID (name), type, article no and component group no in the projects.

In the ideal world, the article number exist in the connected component database.

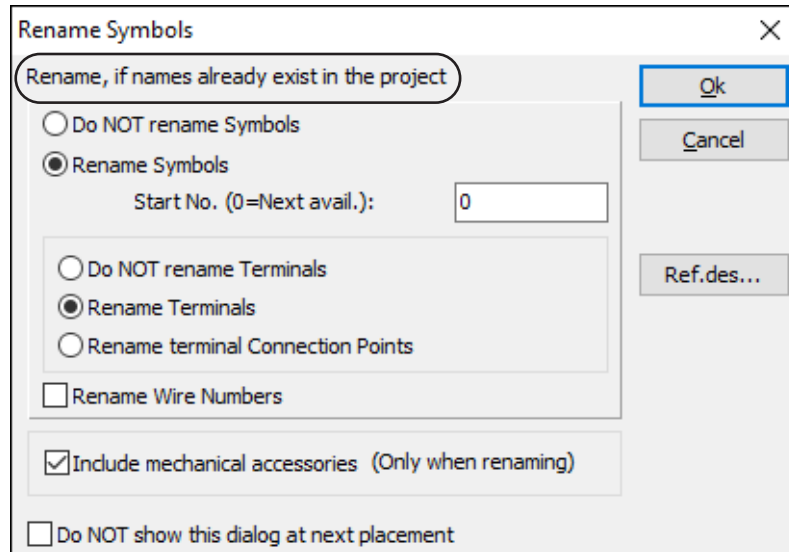
A **name** is a combination of more components, and often one or more of those components are accessories to other components.

When we make electrical documentation we need to show connections between components, so that you can make assemblies of the components and get the desired function; in case of repair and maintenance etc, you use the documentation to see where to disconnect the components and connect them (or new ones) again and obtain the same functionality.

In order to be able to identify all objects, it is necessary to have unique names on components as well as for their connection terminals, that are to be found in the panel and the corresponding documentation project.

So, the option NOT to rename a component when you copy/paste should NOT be used – simply because it is a new component! When it is a new component it needs a new name, or often the same name, but a new reference designation – maybe a temporary one.

Read more about terminals and accessories on pages “Selecting and showing accessories” on page 19 and “Terminals with accessories and Terminals in component group 0” on page 22.



The 'Rename Symbols' dialog box has the following options:

- Rename, if names already exist in the project
- Do NOT rename Symbols
- Rename Symbols
 - Start No. (0=Next avail.):
- Do NOT rename Terminals
- Rename Terminals
- Rename terminal Connection Points
- Rename Wire Numbers
- Include mechanical accessories (Only when renaming)
- Do NOT show this dialog at next placement

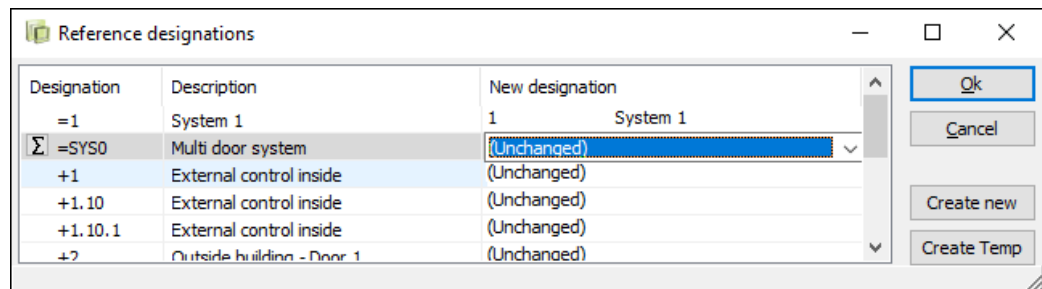
Buttons: Ok, Cancel, Ref.des...

If you want more symbols with the same name – ie plc or terminal symbols – you do that later:

The function Transfer Plc-addresses will also group the individual IOs into full PLCs.

Component Grouping groups terminals into components resulting in correct quantities on the parts list.

The Component Grouping function also assigns individual connection terminals to correct physical layers.



Designation	Description	New designation
=1	System 1	1 System 1
Σ =SYS0	Multi door system	(Unchanged)
+1	External control inside	(Unchanged)
+1.10	External control inside	(Unchanged)
+1.10.1	External control inside	(Unchanged)
+2	Outside building - Door 1	(Unchanged)

Buttons: Ok, Cancel, Create new, Create Temp

When you work with reference designations, the program will assist you in selecting a new designation for your copied objects.

If you don't have a new designation, select a temporary designation. Which can be changed at any time.



REMEMBER, ALIKE IS NOT IDENTICAL!

SAME ARTICLE NUMBER IS NOT SAME COMPONENT NAME.

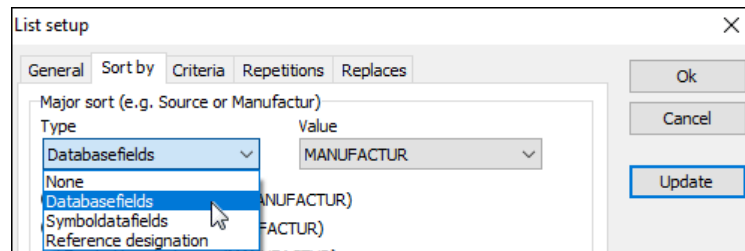
SAME COMPONENT NAME IS NOT THE SAME REFERENCE DESIGNATIONS.

LISTS

In version 21 we have added extra functionality to list settings, created more data fields and made a few other improvements that are explained below.

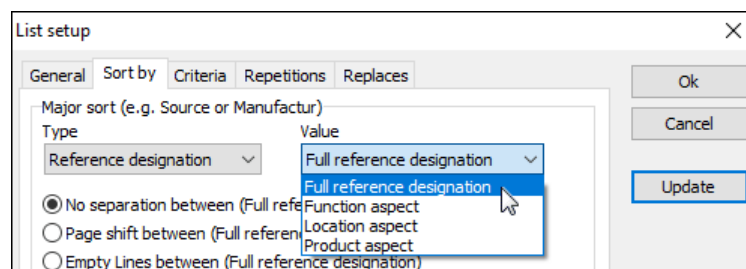
NEW SORTING OPTION FOR PARTS AND COMPONENT LISTS

The sorting functions for parts and component lists have been improved:



It is now possible to make a **Major sort** based on:

- None — meaning that you will sort according to the **Minor sort** specified at the bottom of the dialog box
- Database fields (the same function as in version 20)
- Symbol datafields
- Reference designations



You only get the relevant options. This means that you need to be connected to a database in order to be able to sort according to for instance **Manufacturer**.

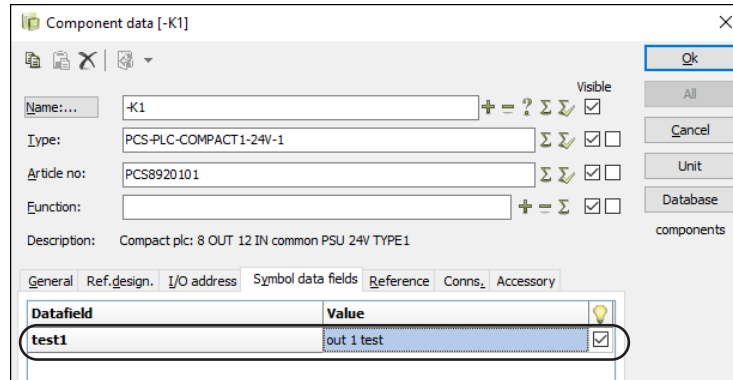
Correspondingly, you can only sort according to **Symbols datafields** or **Reference designations** when these are applied in the project.

PLC LIST IMPORT/EXPORT WITH DATA FIELDS

When you work with plcs, you often need to transfer more data than what you find room for in the fixed data fields for the symbols.

You will frequently need extra user-defined symbol data fields, and you will want to be able to import and export this data together with the IO-list.

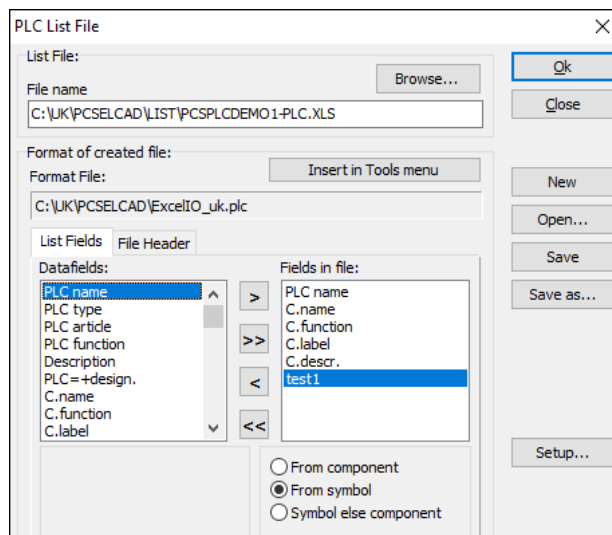
This is now possible.



As in previous versions, it is only possible to import data that is related to the plc. This means that you can't use the IO-list to update data on the components connected to the plc.

Please remember, that the symbol data fields here belong to the *symbol* and not the *component*, as you want the value from the individual plc symbols.

Also remember, that the plc symbols must have the relevant symbol data fields attached to them, before you can import data to the symbol data fields.



CONNECTION LIST AS LIST IN THE PROJECT

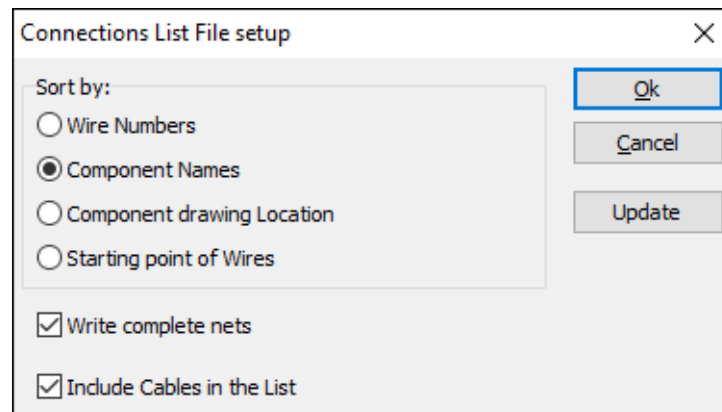
In previous versions, the connections list has only existed as a 'List to file'.



In version 21, you can also include the list type as project pages, and it has its own tab in the **Template** folder, that you find when you choose **File => New**.

The data fields, that you use in the list, are the same as those that earlier only existed in the List to file. This means that you can make a variety of different connections lists, such as lists for wires.

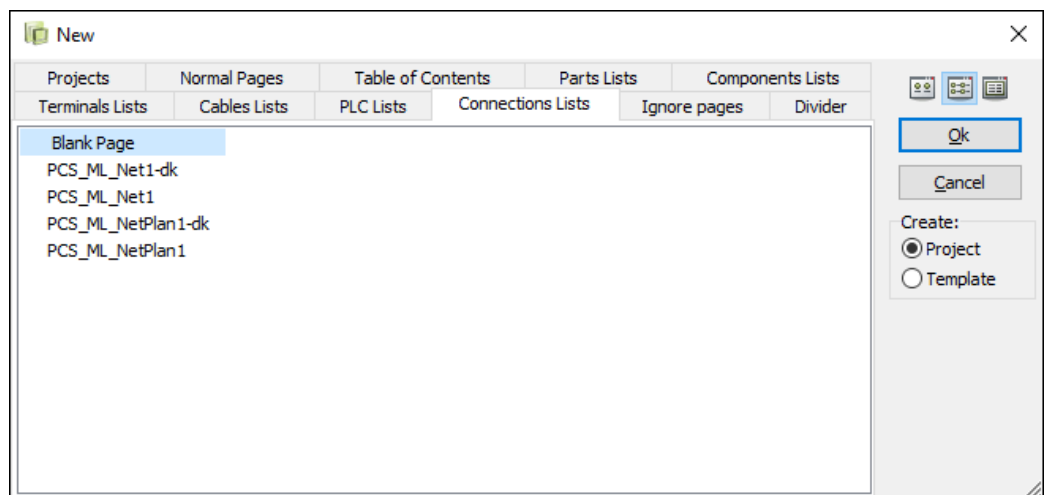
The list can be sorted according to wire numbers, component names or locations or according to the starting point of the wires.



New template folders in Files => New

There are new tabs and folders for **Connections lists** and **Ignored pages**.

The Connections lists template is new, and templates for connection plans / net plans also appear in this tab.



The **Terminals lists** folder also contains templates for terminal plans, the **Cables list** folder contains cable plans.

All new templates are created using multi-lingual (ML) drawing headers.

How to use drawing headers with multiple languages (ML)

When you start a new, empty project with a single empty page and select a *multi language (ML) drawing header* on the page, the program will automatically select the same language as you have specified for the user-interface in **Settings => System => Select language**.

If this language does not exist in the drawing header, the program will select State 0. When you use our pre-installed drawing headers, this means that the data fields inserted in the drawing headers will only display data from the project, but not the specified pre-texts for the data fields.

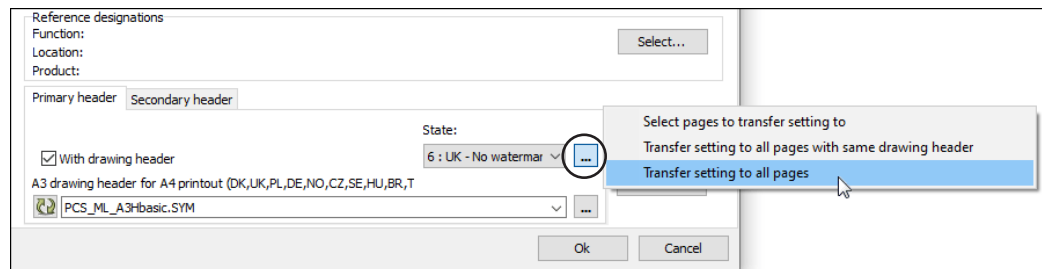
When you select a project template in order to create a new project, the selected language in the template pages will be the language specified in **Settings => System => Select language**. However, you might want to have your user interface in Danish, while the drawing headers are in English. This is easy to change, as you can see below.

Please be aware, that the predefined ML list pages use two headers – one for the main title block (and frame) and one for the list contents.

How to change language in the drawing header

When you select a ML drawing header (using *states*), you can see its languages (states) in a list in the **Page data** tab.

Click the ... button and select the appropriate command.



You need to do this for the title block as well as for the list contents.

In our drawing header we have included descriptions for all languages and the states have the same order in all languages.

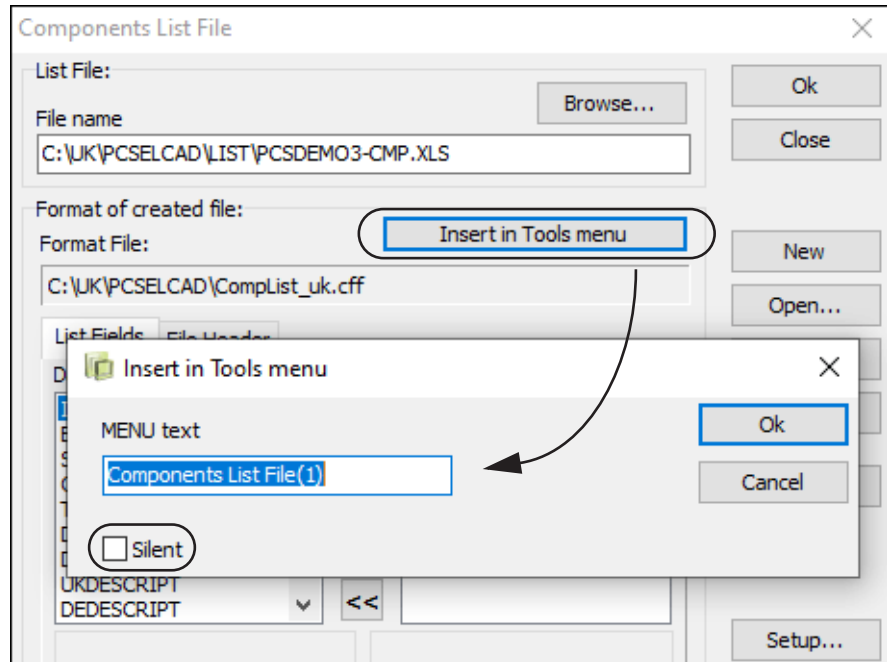
The order of the languages is the same in all lists.

That means that you can change all primary drawing headers / title blocks in a single operation (as long as they use the same header) and all list contents on another operation.

This is a lot faster than translating all texts. And you get the same result every time.

'SILENT' FUNCTION IN LIST-TO-FILE FROM THE TOOLS MENU

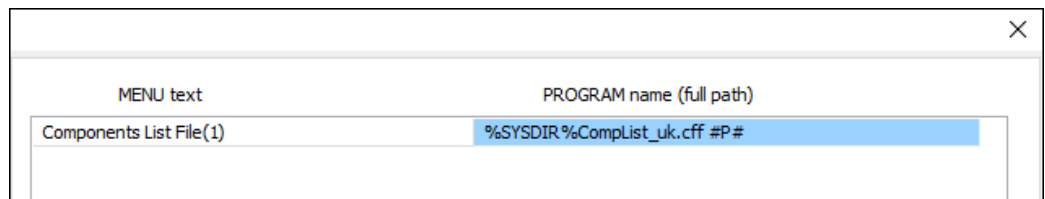
For many versions, it has been possible to add your favorite List-to-file formats to the **Tools** menu. In version 21 it is now possible to export these lists **Silent**. This means that the lists are exported as specified in the format file right away, without asking further questions.



You activate it **Silent** function like this: In the **List File** dialog box you click the **Insert in Tools menu** button and then select **Silent** in the **Insert in Tools menu** dialog box and click **OK**.

Please be aware that you can give the menu item another title when you insert it into the menu.

If you select **Settings => Tools** it looks like this:



LINE DATA FIELDS IN LISTS

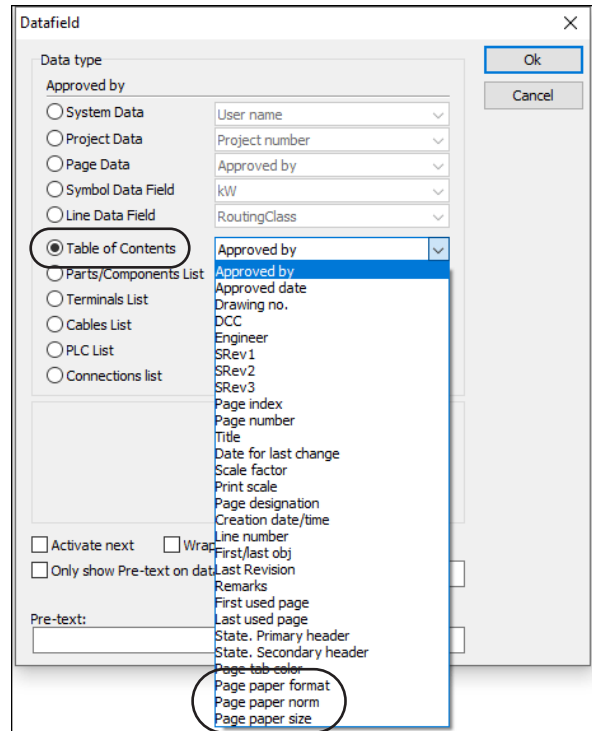
Line data fields can now be included in lists. This can be used in the various connection type lists, such as terminals, cables, plcs and connections lists. Line data fields are created in the **Settings => Text/symbol defaults**.

NEW DATA FIELDS IN THE TABLE OF CONTENTS

There are three new data fields for **Table of contents**:

- **Page paper format**
- **Page paper norm**
- **Page paper size**

The contents of the data fields can for instance be part of the exchange format to various PLM/PDM systems.



NEW FIXED SYMBOL DATA FIELD - QUANTITY

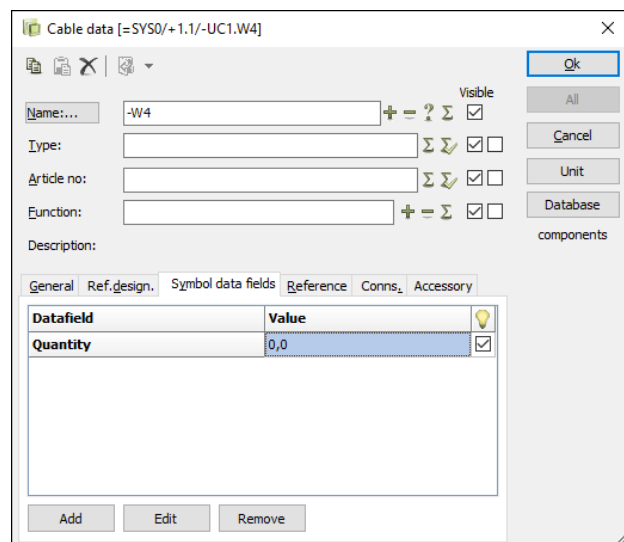
A new fixed symbol data field named **Quantity** has also been added.

The symbol data field shows the value that you type into the **Quantity** field on the **General** tab.

If you want to show the quantity (mainly on cables) you can set this field visible.

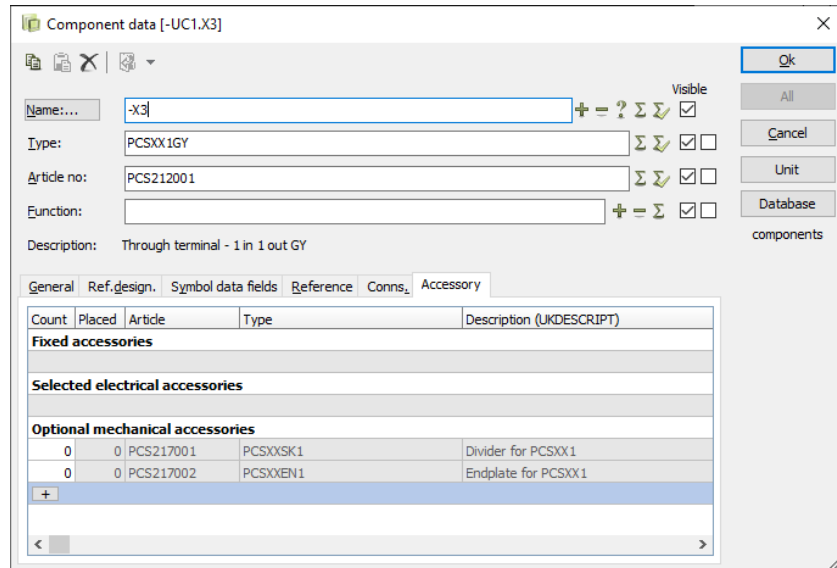
You can't type into this field.

The value can – as earlier – be seen in Object lists and other lists.



SELECTING AND SHOWING ACCESSORIES

The **Accessory** tab in the **Component data** dialog box – earlier known as **Mechanical accessories** – has changed in version 21, so that it now shows all selected accessories, and makes easy selection of mechanical accessories possible.



The tab is divided into three parts:

- **Fixed accessories** — which can't be selected here, but you get an overview over the accessories that are defined in the database.
- You can see a list of **Selected electrical accessories**. Likewise, they are not selected here but in the **Component menu**.
- A list of **Optional mechanical accessories**, as defined in the database. To select accessories, simply type in the desired amount.

Adding additional mechanical accessories

You can add additional mechanical accessories by clicking the **+** button at the bottom line for the **Optional mechanical accessories**. When you click it, you can add further accessories:

Optional mechanical accessories				
Count	Placed	Article	Type	Description (UKDESCRIPTION)
0	0	PCS217001	PCSXXSK1	Divider for PCSXX1
0	0	PCS217002	PCSXXEN1	Endplate for PCSXX1
1	0	1234		
1	0		DB	

- When you press the **DB** button, you can add components that are in the database. It adds a new line per article number. You can then change the quantity yourself. Type and description are automatically fetched from the database.
- Instead of pressing the **DB** button, you can type in an *Article number* (or *Type*) directly. If the component does not appear in the database, you will not get any *Description* for it.

The program always adds a new empty line with a **+** on it. You remove any previously selected accessories by setting the number for the accessory to **0**.

If the accessory is placed in the project (for instance on the mechanical page) you will not be allowed to delete it in the dialog box.

— Read more about how to select accessories in the **Component menu** in “Komponentmenüen” on page 42.

SUPPORT SYMBOLS AND ACCESSORIES

Support symbols was the method used for specifying accessories in older versions of **Automation**. This is what happens to these in version 21:

Support symbols in projects from previous versions

When you have a project where mechanical accessories are specified using support symbols, these components still remain in the project in version 21.

However, the following is changed in version 21:

- These accessories will no longer appear as accessories in the project, but as components at the same level as the main component.
- These accessories will no longer be displayed on the **Accessory** tab in the **Component data** dialog box, and will not appear as accessories in the **Component menu** (more about this later).
- When you copy a component with accessories represented by support symbols, these accessories will no longer be included — simply because the program no longer understands them as accessories.

Please note that *components represented by support symbols will still be included in parts and components lists.*

Support symbols in the Component menu in version 20

In version 20 the **Component menu** would contain a list of "not placed support symbols" representing the accessories that you had placed on mechanical pages from the **Component menu**. — And if you subsequently placed these support symbols, the project would contain too many accessories.

Support symbols ignored in the Component menu in version 21

If a component only has a support symbol specified as an electrical symbol in the database, this symbol is ignored in the **Component menu** in version 21.

This means that there are no available support symbols in this case, and there is therefore no risk of accidentally placing too many support symbols.

The support symbol is only ignored in the **Component menu** when it is represented as the only diagram symbol for the component.

ON TERMINALS

The terminal symbols are some of the smallest symbols in a project, which at times can be somewhat challenging. We have addressed some of these challenges in version 21.

DRAWING LINES THROUGH TERMINALS

When you draw a connection to a terminal without applying the Router, you could accidentally connect the line to the lower connection point in stead of the upper connection point, which would result in messing up the connections.

The algorithm is now changed, so that the program automatically connects the line to the upper connection point when you start drawing the connection from a point above the terminal symbol — and automatically connects the line to the lower connection point, when you start drawing the connection from below the terminal symbol.

MULTI PLACING MOTOR TERMINALS WITH UVW

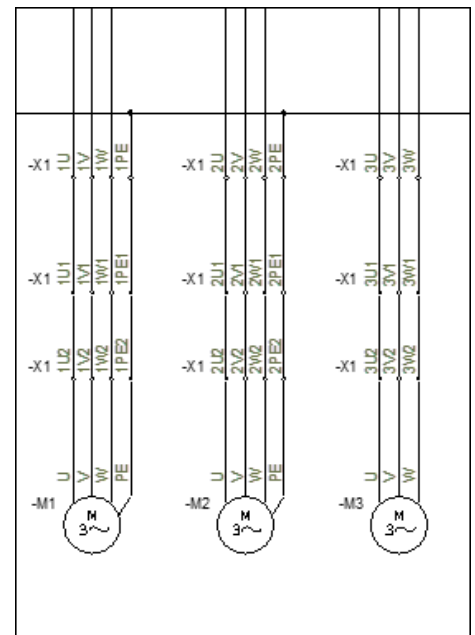
You can also multi place motor terminals with UVW.

When specifying the component name above in the **Component data** dialog box, do this:

Type in **-X1:1U**, **-X1:1U1** or **-X1:1U2** and subsequently press <Ctrl> while clicking the **+** or the **?** button on the right-hand side of the **Name** field.

The program automatically detects if a PE is needed.

If there are only two conductors, they will be named U and V, and the naming starts over again.





MARKING TERMINALS FOR INCOMING AND OUTGOING CONDUCTORS (EN 60445:2017)

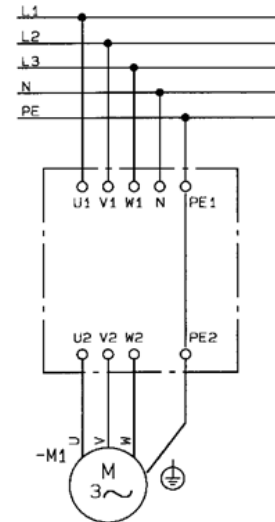
TERMINALS FOR **INCOMING** CONDUCTORS SHOULD BE MARKED AS **U1**, **V1** AND **W1**.

THIS SHOWS THAT THE THREE TERMINALS **U1**, **V1** AND **W1** ARE MEANT FOR CONNECTING TO THREE POWER SUPPLY PHASES.

TERMINALS FOR **OUTGOING** CONDUCTORS MUST BE MARKED AS **U2**, **V2** AND **W2**.

THIS SHOWS THAT THE THREE TERMINALS **U2**, **V2** AND **W2** ARE MEANT FOR CONNECTING TO THREE OUTGOING PHASES OF CIRCUIT SUPPLIED FROM THE CONTROL PANEL.

IF THERE ARE SEVERAL INCOMING OR OUTGOING CONDUCTORS/WIRES, YOU MUST USE SEQUENTIAL NUMBERING OF TERMINALS.



TERMINALS FOR INCOMING CONDUCTORS:

1U1, 1V1, 1W1 ... 3U1, 3V1, 3W1

TERMINALS FOR OUTGOING CONDUCTORS:

1U2, 1V2, 1W2 ... 3U2, 3V2, 3W2

TERMINALS WITH ACCESSORIES AND TERMINALS IN COMPONENT GROUP 0

In version 21 we have changed some of the rules for how to handle accessories — generally and for terminals in particular.

This means that you can no longer have terminals in component group 0. If this is not an issue that you have addressed earlier, the program has handled this for you automatically. So, please skip the remaining part of this section.

A brief history lesson on Component group 0

In previous versions of **Automation**, symbols — including terminals — were placed in project pages in component group 0. This just meant that you still had not decided which components to apply yet.

When you selected an article number, you could — for all components but multi layer terminals — keep this component group, and the parts list would still be correct. **Automation** would be able to make the right qualified guess (almost) every time.

For common terminals this meant that each terminal symbol was treated as a component, which meant that the components list would be ok. But when you placed the mechanical symbol for the terminal, you could not know which symbols that were connected. Therefore, you could not use the right-click function **Go to symbol** on other project pages.

Automation has not assigned component group 0 to any symbols for several versions, but some components using component group 0 this will still exist in projects, due to re-use of material from older projects.

What **Automation** has done in recent versions, is that components are assigned consecutive component group numbers, so that these can be applied for following the individual terminal in all project pages, because it assigns a unique id to each component. You can therefore apply the right-click function **Go to symbol** and track the symbols for the component on other project pages.

However, we have been facing an increasing number of challenges when attempting to make a qualified guess. Especially for terminals, and almost always when they have accessories. This is why we have started to phase out component group 0.

Terminals with accessories not placed on mechanical pages

If you have terminals with accessories, which are not placed on mechanical pages, you will not experience any difficulties. Parts and components lists will be correct, and you can continue working without any further ado.

But if you wish to continue working with these terminal rows — either by clicking on the components in diagram pages to open the **Component data** dialog box or by trying to insert the components on mechanical pages — you will have to apply the **Change** function.

This option appears in the **Component menu** under **Symbols for component** when you select a terminal with symbols in component group 0. Simply because you will not be able to edit a terminal row when its symbols are in component group 0.

When you click the **Change** button, all of the accessories in the terminal row are set to belong to one of the terminals in the selected terminal row, and the component group numbers are set to 1, 2, 3 e.t.c.

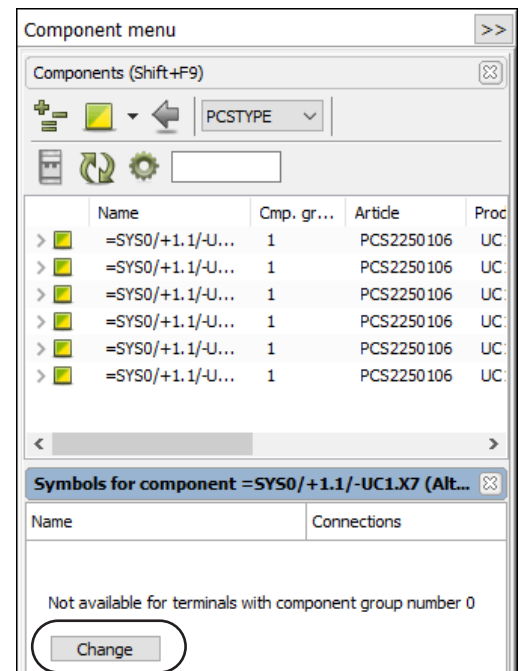
The parts list will still be correct, and you will be able to place the terminals correctly from the **Component menu** on mechanical pages.

Terminals with accessories placed on mechanical pages

When you have terminals with accessories which are placed in the project, the components list may be incorrect. Whether this is the case depends on how you have chosen the accessories and how you have placed the mechanical symbols for the accessories.

Accessories selected as support symbols

This is the old way of placing accessories — which can result in incorrect parts lists.



Accessories chosen from the Mechanical accessories tab

Since version 18 it has been possible to select accessories in the **Mechanical accessories** tab in the **Component data** dialog box. (In version 21 the tab is named **Accessory**). If you have subsequently changed component group numbers to 0, this has resulted in incorrect parts lists.

How to correct errors due to component group 0

To correct errors due to using component group 0 for terminals, do this:

- 1) On the mechanical layout page: Remove the terminal row.
- 2) On the diagram page: Select the terminal and click the **Change** button as described above.
- 3) On the mechanical layout page: In the **Component menu** in the **Components** window: Right-click on the component and choose **Select component name**.
- 4) In the **Component menu** in the **Available symbols** window: Right-click on a symbol and choose **Select all**. (This ensures that the terminals will be placed in the correct order).
- 5) Click to place the terminal symbols on the mechanical layout page.

After this the lists will be correct, and you can navigate back and forth between the mechanical page and the diagram for the selected component.

REPLACING ALL SYMBOLS IN A PROJECT

When a symbol is placed in a project, the symbol is saved under the *symbol file name* for the symbol (e.g. *Contacto1.sym*). Where the symbol originally came from then makes no difference to the program.

However, several symbols with the same symbol file name can occur in a project.

You can therefore use the function **Functions => Special functions => Replace all symbols**. It reloads all symbols in the active project based on the order of symbol libraries specified under **Settings => Directories => Alias for directories**.

REPLACE ALL SYMBOLS WITH SYMBOLS FROM ANOTHER DIRECTORY

This special function also enables you to replace all symbols in a project with similar symbols — symbols with the same *symbol file name* — that are placed in another symbol directory.

This requires that you change the order of the symbol libraries under **Alias for directories** and then choose **Functions => Special functions => Replace all symbols**.

EXISTING SYMBOL DATAFIELDS ARE INCLUDED

When a symbol contains symbol datafields, these symbol datafields are also created for the new symbol, and they are assigned the same values as for the original symbol, when the symbol is replaced.

If the new symbol contains symbol datafields of its own, the final result will contain both sets of datafields.

Additional information about replacing symbols

A project contains all symbol file names and their graphical appearance, including text fonts and the positions of the texts in the symbols.

This also means that when you place a symbol with a file name that already exists in the project, the placed symbol will in some cases become identical to the symbol already placed in the project.

You can change the font in **Settings => Text/Symbol defaults** where you select **All** in the **Text defaults section**. You can read more about your options here in the manual.

This will take you much of the way, but some of the symbol texts will also have to be moved. This will not happen until you replace the symbols, or move the texts manually.

LINES

When creating electrical documentation we have traditionally not placed much information on the lines. However, the need for this has increased. In version 20 we added a *RoutingClass* data field for lines, and in version 21 more options have become available.

You can see some of these below.

CALCULATED LINE DATA FIELDS (FORMULA EDITOR)

In version 21 you can create calculated line data fields, which can be a combination of line data, texts and other line data fields. Thus you can combine multiple line data fields in a single line data field.

You can create a calculated line data field like this:

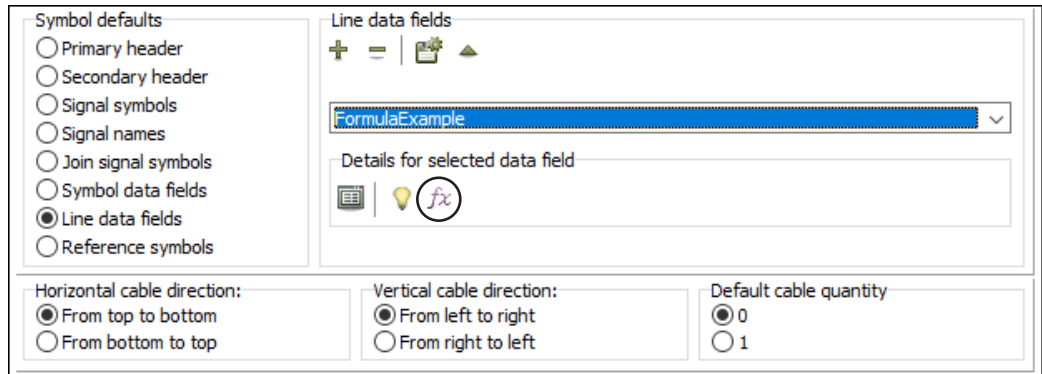
Create the necessary line data fields

- 1) Choose **Settings => Text/Symbol defaults**. In the **Symbol defaults** settings: Click on **Line data fields**.
- 2) Under **Line data fields**: Click the **Add** button and type in the name of the line data field — for instance *FieldA*. Also create the line data fields *FieldB* and *FormulaExample*.



Create a calculated line data field

- 3) Now click the **Formula editor** button:



- 4) You then enter the **Formula editor**, where you can specify how to create/calculate the line data field:

- 5) Select **Line data**, **Line data fields** and/or **Static text** and click **Append**, which adds the selected data fields / texts to the **Formula** line in the upper part of the dialog box — with the correct syntax. Under **Result** you can see how the calculated data field will appear when inserted on the project pages.
- 6) When the formula is complete: Click **OK**.

TEXT ANCHOR FOR LINE DATA

When you put data on lines and wish to show these in the project, you need to be able to control which line segment each text belongs to.

This is done by applying a text anchor. The text is then connected to the selected line segment. If you subsequently assign the text anchor to another line segment, the text will move to the middle of this other segment.

To show line data next to a specific line segment, do this:



- 1) Activate the **Lines** command and deselect the **Pencil**.
- 2) Right-click on the segment and select **Text anchor**. (If the text is already anchored to the selected segment, **Text anchor** will be dimmed).

CONTROLLING GAP ARCS USING PRIORITIES ON LINES

In **Settings => Pointer/Screen** under **Representation of crossing non-connected conducting lines** you can specify how to represent non-connected conducting lines, when they cross in diagram pages. There you have the following options:

- **No gap:** The lines cross.
- **With gap:** A gap is opened on one of the lines.
- **Gap with arc:** A gap is opened on one of the lines and an arc is drawn on the other line.

When you have selected **Gap on lines**, it is possible in version 21 to set priorities on lines, in order to further control how they behave when they cross.

To set a priority on a line, right-click on the line to enter the dialog box **Line article data**, where you set the **Priority** to **Low**, **Normal** or **High**.

The following rules then applies:

- When two lines with different priorities cross, the line with the highest priority is drawn without an arc.
- When two lines with the same priority setting cross, the horizontal line has the highest priority.

LINE DATA FIELDS IN LISTS

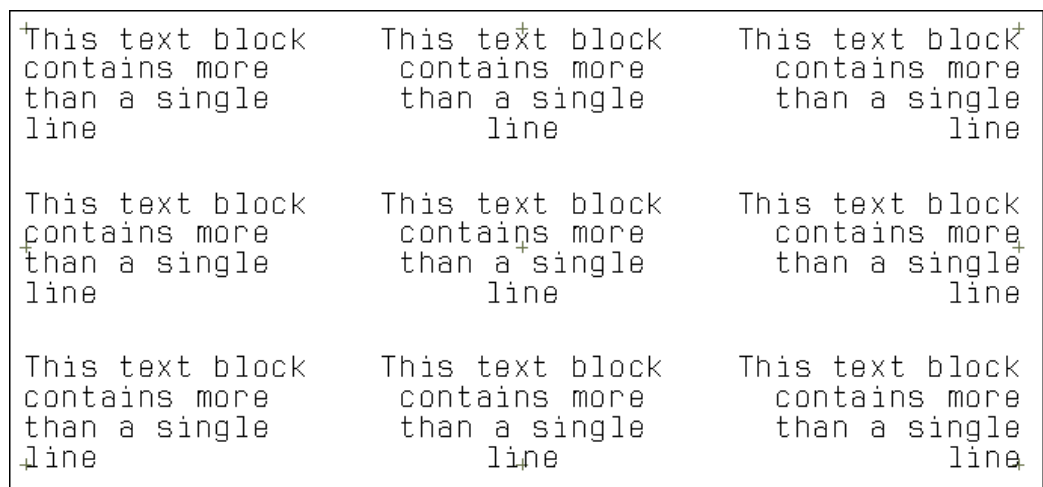
The contents of the created line data fields can also be included in lists. Read more about this in “Line data fields in lists” on page 18.

TEXT ADJUSTMENT

In earlier versions text adjustment only adjusted the first text line and not the full text block. Now we adjust the full text block.

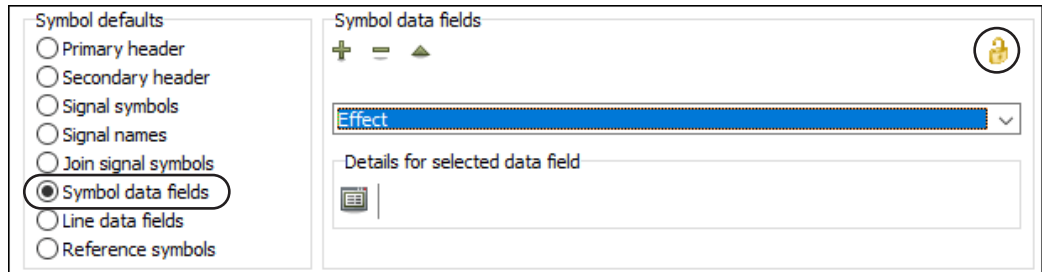
When you load a project from a previous version, the text location will be maintained, and its adjustment will be converted to the new type of adjustment.

In the picture below, you can see the small text anchors that indicate how the text is adjusted. The anchor is also the position from which the text is adjusted. In this way it is possible to let the text relate to a corner of an object and adjust it accordingly.



LOCKING DATA LISTS FOR SYMBOL DATA FIELDS

When you have created a data list for a symbol data field in **Settings => Text/Symbol defaults**, you have the option of locking the data list. You can do this by clicking the lock icon to the right below **Symbol data fields**.



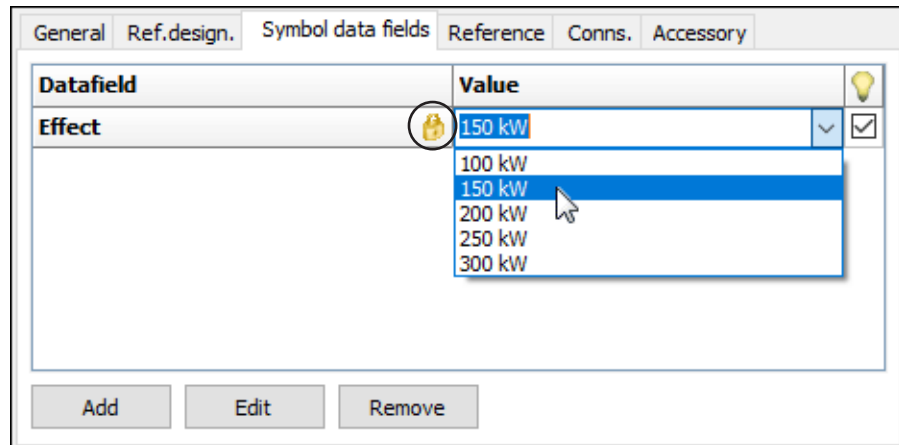
When the icon is displayed as a locked lock, it will only be possible on project pages to select the values specified in the data list.

When the icon is displayed as an open lock (as above), it will also be possible to type in your own values.

The lock icon is only visible when a data list is created for the selected data field.

THE LOCK IS ALSO DISPLAYED IN COMPONENT DATA

When you select a symbol data field with a locked data list attached, the lock is also displayed in the **Component data** dialog box:



ENHANCED ALIGN FUNCTION

ALIGNING SYMBOLS ACROSS PROJECT PAGES

In version 21 symbols placed on separate project pages can also be aligned:



- 1) Make sure that the **Symbols** button is activated, right-click on the symbol that you wish to align the other symbols to, and select **Align**.
- 2) Jump to another project page — for instance using the page tabs at the bottom of the screen
- 3) On the new project page a thin line now originates from where the selected symbol was placed on the previous project page, and ends in the cross-hairs: Now click on each individual symbol, that you wish to align to the selected symbol. This aligns them to the selected symbol.

It is not possible to do this for other types of objects.

ALIGNING - WITH AND WITHOUT <CTRL>

When you right-click on a symbol (A) and select **Align** and then click another symbol (B), A is aligned with B.

In version 21 you can also press down <Ctrl> while clicking B, which means that B will be aligned with A.

This is also possible for other types of objects.

LICENSE SYSTEM IMPROVEMENTS

Some improvements have also been made on the license system, which is used in both **Automation** and **Automation Service**.

TANKING LIMITATIONS

On net licenses using a USB license key you could set a limitation on how many of the licenses that could be tanked.

This is now also possible for software licenses.

AUTOMATION SERVICE

A net license for Automation Service is now also available.

NEW LICENSE SERVER

When you make a new installation in version 21, you will be able to specify a server name — either by server name or IP address — during the installation process.

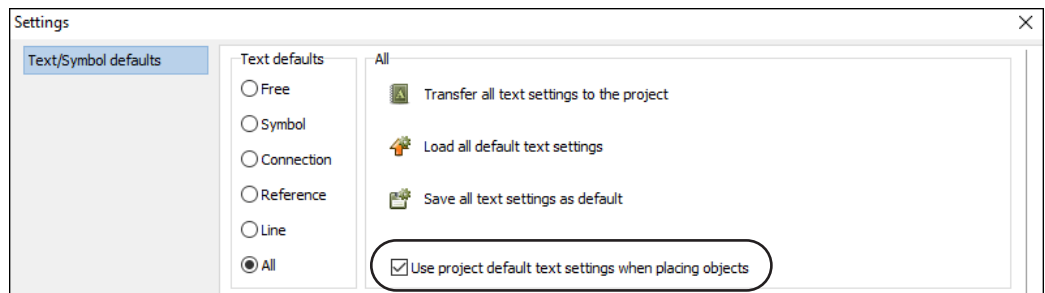
When you try to activate a license that is already being used, you will be informed about this, and are supported in solving the situation.

ENHANCED ADMINISTRATOR PROGRAM

The Administrator program can now control the following settings on the tab **Settings => Text/Symbol defaults**. When the administrator controls a function, the check box for the function is dimmed for the users.

TEXT SETTINGS: THE ALL FUNCTION CONTROLLED BY ADMINISTRATOR

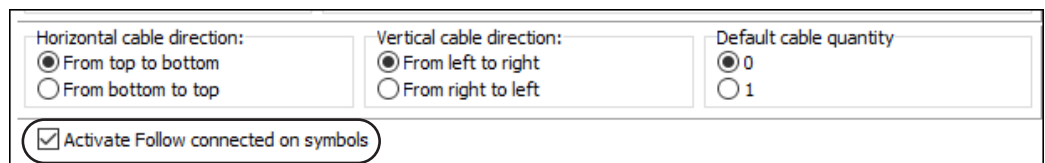
All text settings in the project can be set via the **All** function under **Text defaults** in the **System => Text/Symbol defaults** tab. This function can now be controlled from the Administrator program.



FOLLOW CONNECTED

The **Activate Follow connected on symbols** setting in the project can be set at the bottom of the **System => Text/Symbol defaults** tab. This function can now be controlled from the Administrator program.

The function causes all symbols to get the same color and line width as the connected lines.



MINOR CHANGES

Here is a list of minor changes in version 21.

MENU ITEMS REARRANGED

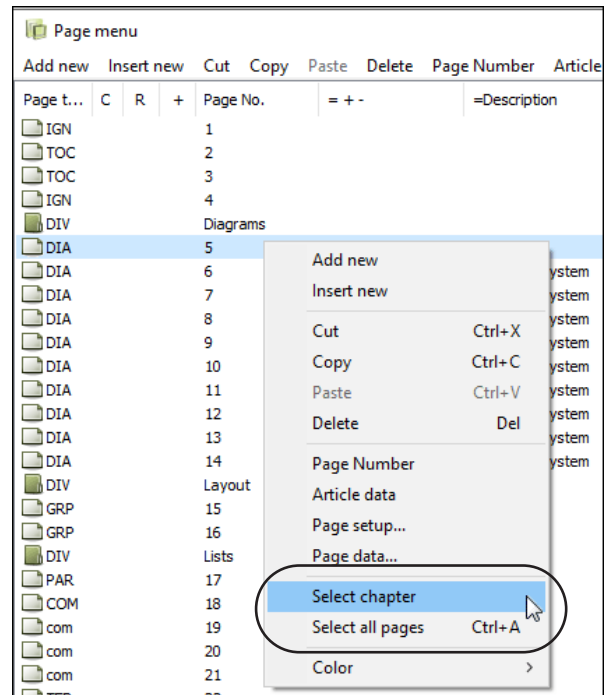
Several menu items have been rearranged and moved to other menus — for instance all plc-relevant menu items have been moved to the **Plc** menu. Other menu items have become obsolete and are therefore removed.

THE PAGE MENU

It is now possible to select all pages in the project or all pages in a chapter by right-clicking. This can for instance be quite handy when renumbering a project.



It is no longer possible to select all pages in the **Page menu** and subsequently delete them.



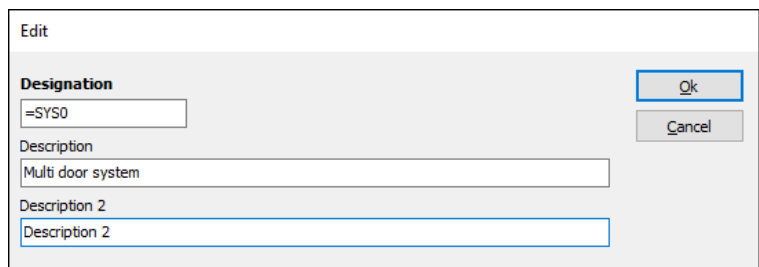
RECTANGLES WITHOUT CONNECTIONS ARE NOW NON-CONDUCTING

In **Automation** it is possible to draw rectangles using conducting lines, which doesn't always make sense. These rectangles without connections are now automatically set to non-conducting, so that they for instance are no longer assigned wire numbers.

EXTRA DESCRIPTION ON REFERENCE DESIGNATIONS

Reference designations have now been assigned an extra description data field.

This can for instance be applied for

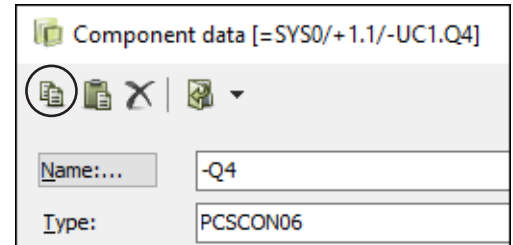


descriptions in other languages or numbers referring to ERP systems.

The contents of this data field can also be included in the overview over reference designations and in components lists.

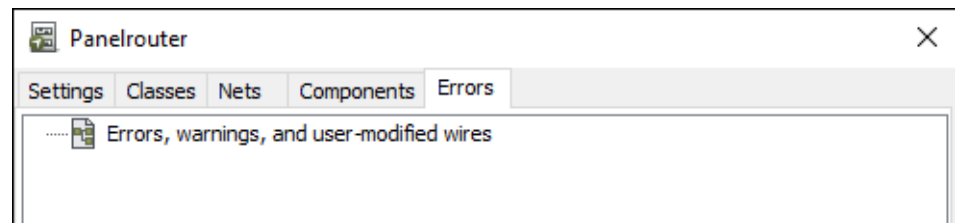
THE COPY FUNCTION IN THE COMPONENT DATA DIALOG BOX

The copy function in the dialog box **Component data** now also includes the contents of any attached symbol data fields when copying.



THE TABS IN PANELROUTER

The tabs in the **Panelrouter** window have been rearranged, so that the **Settings** tab is the first of the tabs. A new **Errors** tab has also been added. It displays a list of errors, warnings and user-modified wires.



INSERTING STATES AND REMOVING EMPTY STATES

When working with multiple symbols for which you wish to apply the same state numbers for identical states, it may become necessary to adapt which layers are used for specific purposes for the individual symbols. Therefore, you can now right-click on a layer in the **States** dialog box and choose **Insert state** or **Remove empty state**, which makes it easier to adapt state numbers for individual symbols.



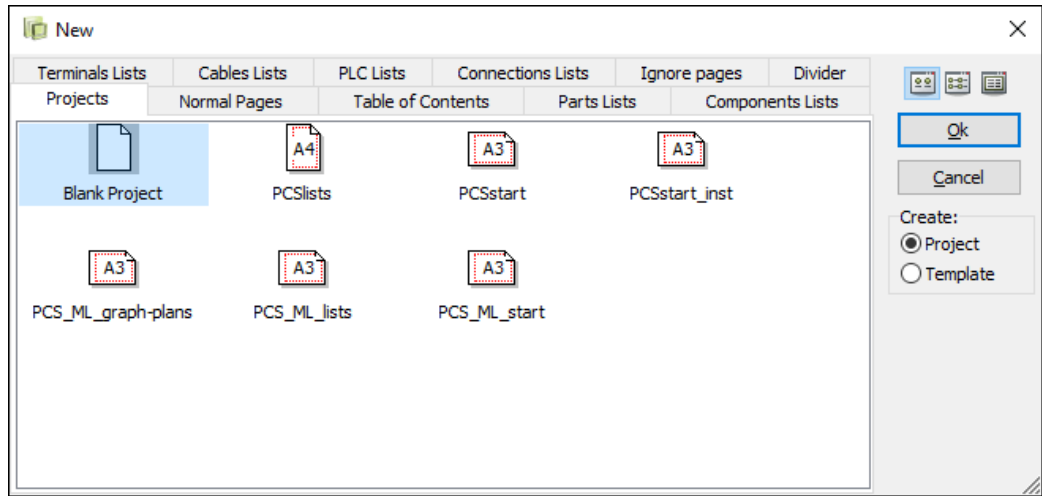
FOR THE DESIGN PHASE ONLY

PLEASE NOTE, THAT THIS IS MEANT FOR THE DESIGN PHASE ONLY. IF YOU, AT A LATER STAGE, REPLACE A SYMBOL IN A PROJECT WITH A SYMBOL WHERE THE STATE NUMBERS / LAYERS HAVE OTHER MEANINGS, THE APPLICATION OF THE STATES WILL NO LONGER MAKE SENSE.

THE NEW ICON AND FILE => NEW HAVE IDENTICAL FUNCTIONALITY



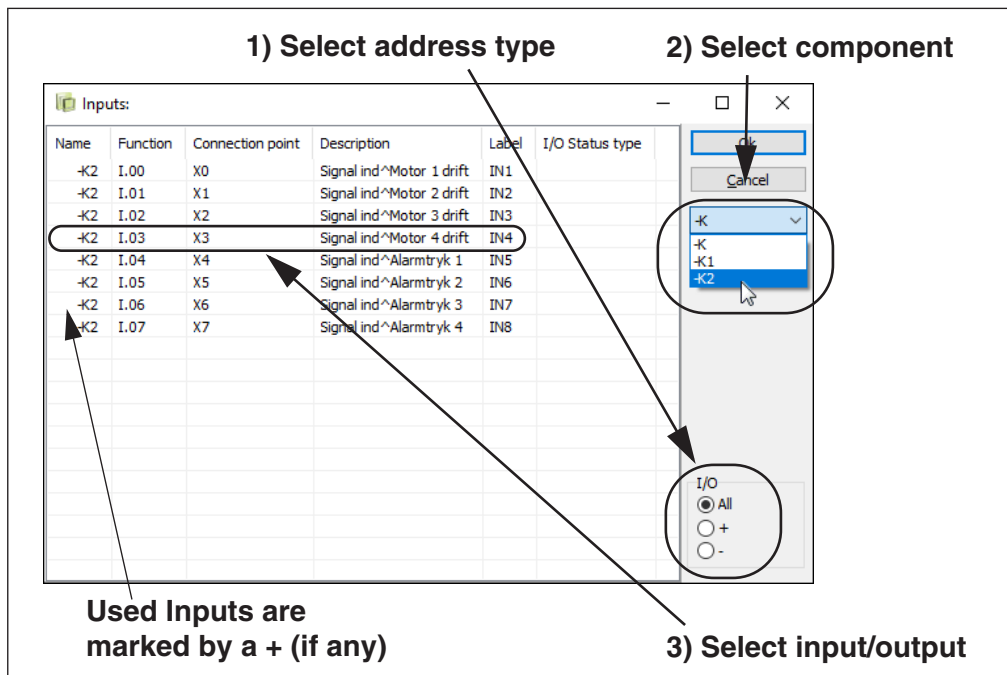
When you click the **New document** icon in version 21, you now open the dialog box **New** with the **Projects** tab opened — just like when you choose **File => New**:



SELECTING IO ADDRESSES

It has now become easier to select the right I/O address using the **I/O addr...** button in the right-hand side of the **Component data** dialog box for plcs:

- When you place a plc symbol in a diagram, you enter the dialog box **Component data**. Here you can now click the **I/O addr** button in the lower-right corner of the dialog box.
- You then enter the dialog box **Inputs** (or **Outputs**):



- Select the desired PLC in the right-hand side of the dialog box, and then select the desired input (or output) in the list, and click **OK**. Please note that used inputs/outputs are marked by a preceding **+**. Under **I/O** in the lower-right corner of the dialog box you can choose either to

show all inputs and outputs in the list (**All**), to show all used inputs and outputs (+) or only to show the available inputs and outputs (-).



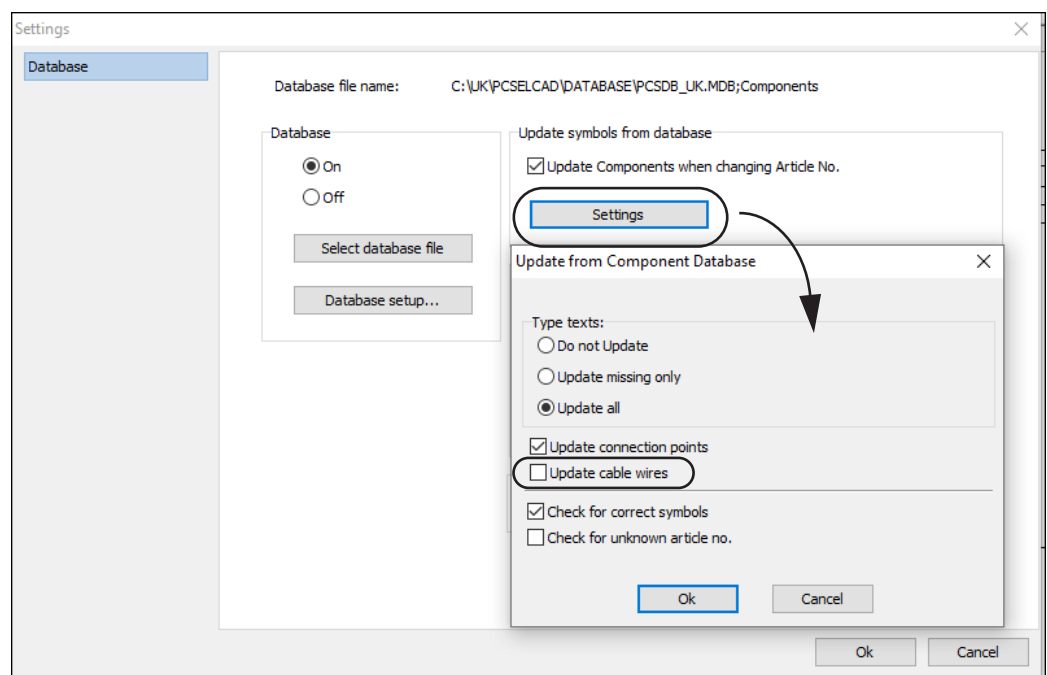
CHOOSE INPUTS/OUTPUTS FOR ALL PLCs IN THE PROJECT

IF YOU CHOOSE **-K** IN THE **NAME** FIELD IN THE RIGHT-HAND SIDE OF THE DIALOG BOX, INPUTS/OUTPUTS FOR ALL PLCs IN THE PROJECT WITH NAMES BEGINNING WITH **-K** ARE DISPLAYED.

THE UPDATE FROM DATABASE FUNCTION

When you updated from the component database in version 20, you were asked whether to update cable wires.

In version 21 this is now a fixed setting in **Settings => Database => Settings**.

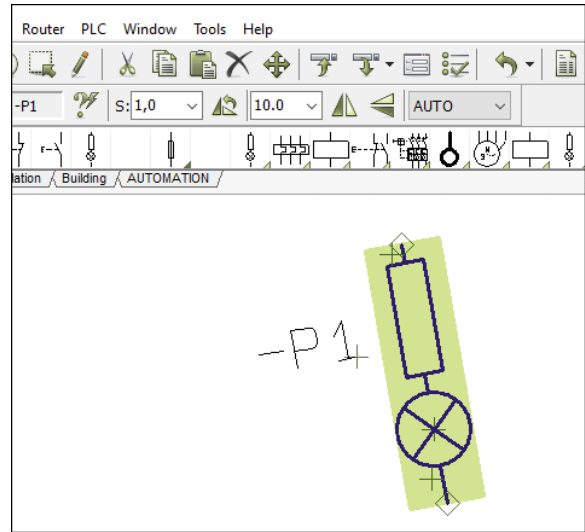


The Drag'n Draw settings (sub drawings) contains the same options, and these can be applied for both sub drawings and by the **Project generator**.

ROTATING OBJECTS WITH 10°

When you select an object and press <Spacebar>, the object is rotated 90°.

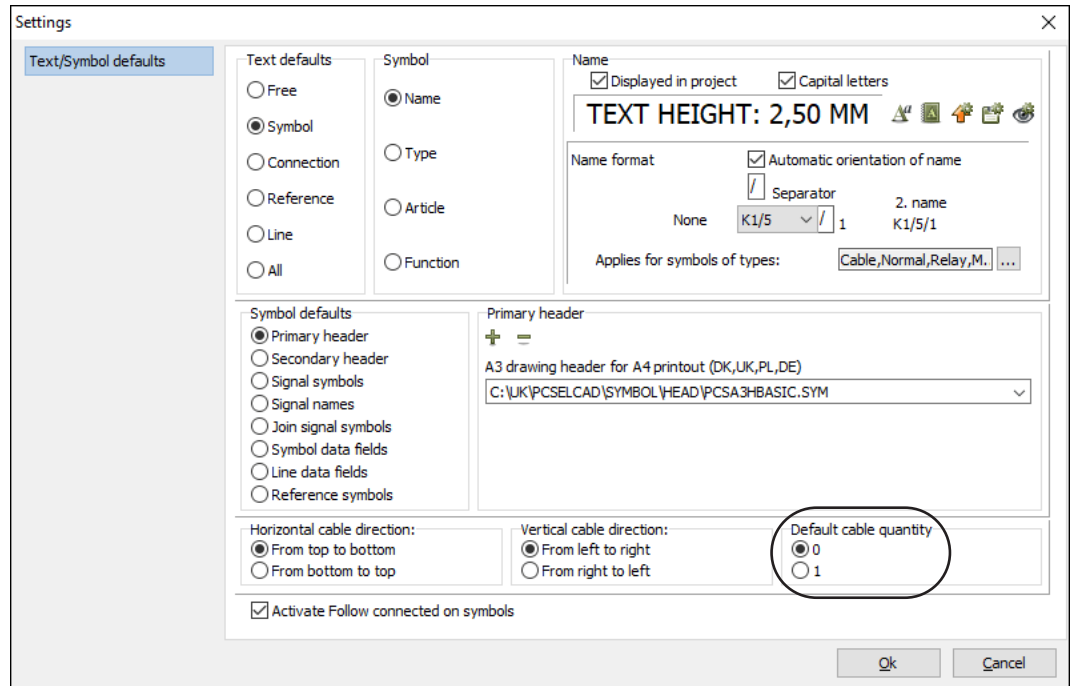
If you press <Ctrl> while pressing <Spacebar>, the object is now rotated by 10°.



DEFAULT CABLE QUANTITY

At the bottom of the tab **Settings** => **Text/Symbol defaults** you can now specify whether the **Default cable quantity** is **0** or **1**:

- When you choose **0**: The default content of the **Quantity** field in the **Component data** dialog box is **0**. Choose this setting when you use the **Quantity** field for typing in the lengths of the individual cables, which can then be accumulated in the project lists.
- When you choose **1**: The default content of the **Quantity** field in the **Component data** dialog box is **1**. Choose this setting when you use the **Quantity** field for counting the number of cables (cable symbols) in the project.



Please note that this is a system setting, which means that the **Quantity** field for any new cable that you place in any project from now on, will be set to the chosen **Default cable quantity**.

No changes are made in existing projects.

OBJECT LISTER - NEW DEFAULT SETTINGS

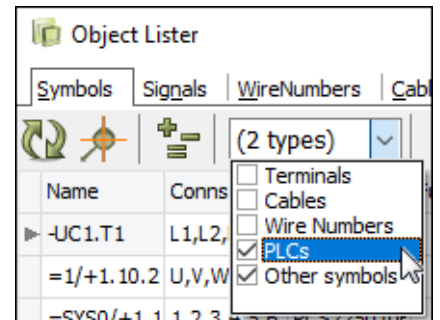
The default settings for the **Object Lister** has changed when installing **Automation**. When you *update Automation* the settings are not changed.

Please remember that you can always change the settings for the **Object Lister** by clicking the column headers and dragging them to the desired positions.

OBJECT LISTER - SELECT SYMBOL TYPES ON SYMBOLS TAB

You can specify which symbol types to display by selecting the relevant symbol types in the toolbar in the **Object Lister**.

This means that information about for instance terminals and cables can be displayed among information about the 'ordinary' symbol types.



WARNING WHEN OPENING WRITE PROTECTED FILES

When opening a Write protected file, you will get a warning telling you that it is Write protected.

THE MOUNTING ASSISTANT

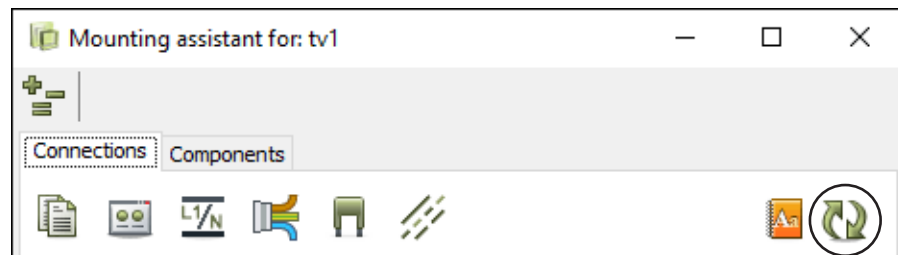
The **Mounting assistant** has been expanded with a couple of extra functions.

MULTIPLE USERS CAN WORK ON THE SAME PANEL

When you place your project on a commonly accessible drive and create a case number, multiple users will be able to open both the project and the case.

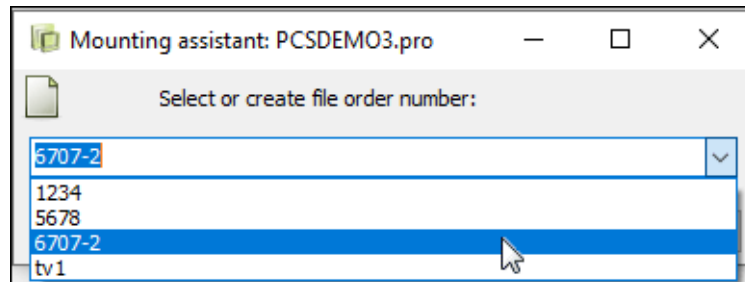


When multiple users work on the project simultaneously, a **Refresh** button will appear, so that all users can update the changes in the mounted connections in the project whenever necessary.



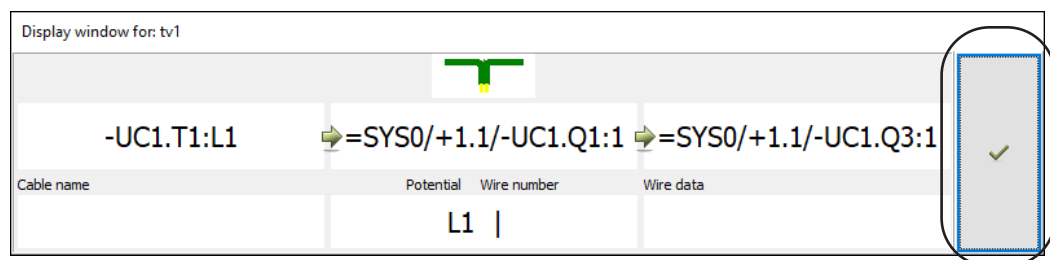
ALL CREATED CASE NUMBERS IN THE PROJECT ARE DISPLAYED

All created case numbers in the project are now displayed in a list when opening the **Mounting assistant**.



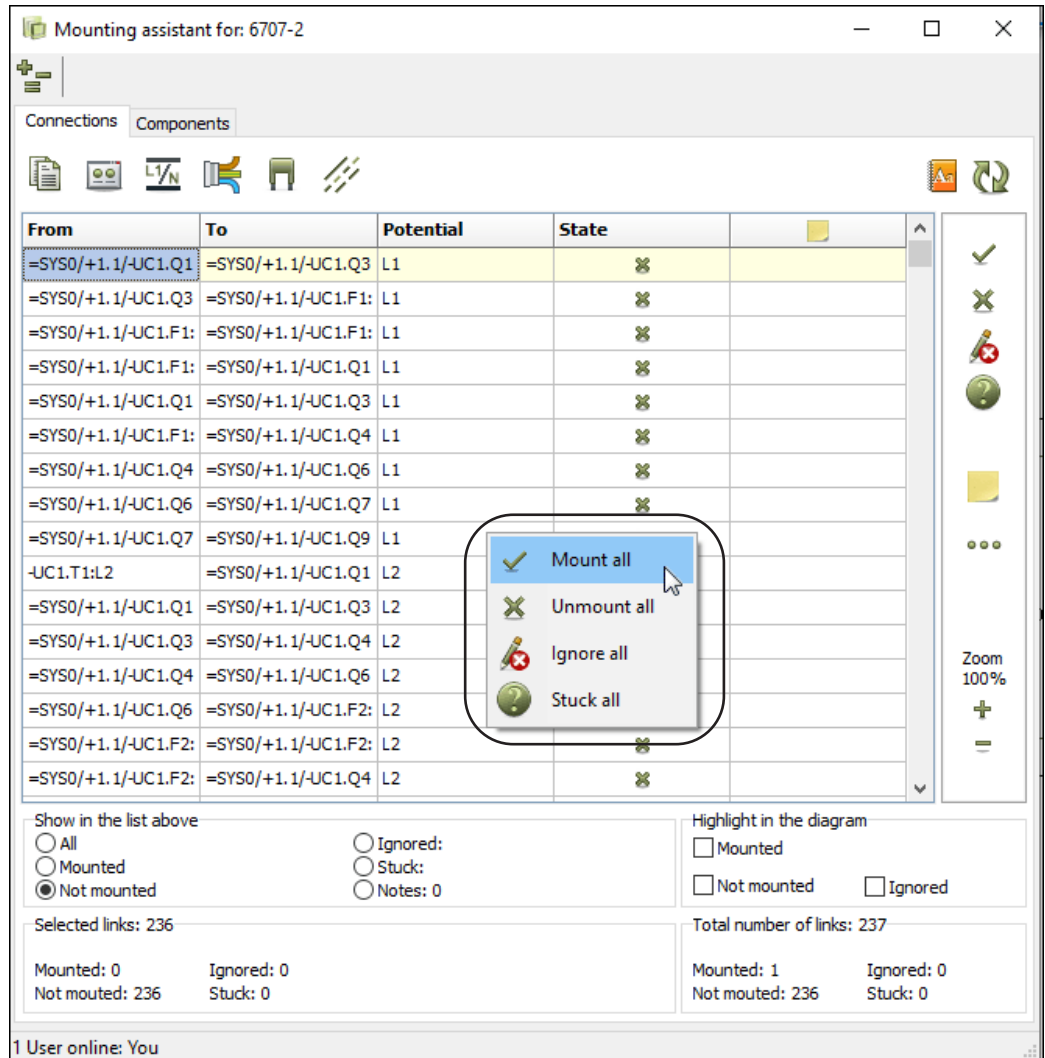
THE SURVEY WINDOW NOW HAS A "MOUNTED" BUTTON

When using the **Mounting assistant** on a tablet or a pressure sensitive screen, you can now specify that a wire has been mounted by pressing the big button in the top-right corner of the **Display window**.



CHANGING STATUS FOR ALL WIRES IN A SINGLE OPERATION

When you right-click on the lines on the **Connections** tab in the **Mounting assistant** you can change status for all connections in a single operation. This is also possible in the **Components** tab.



THE COMPONENT MENU

In version 20 quite a lot of improvements were made to the **Show available** function. This resulted in what we in version 21 have named the **Component menu**.

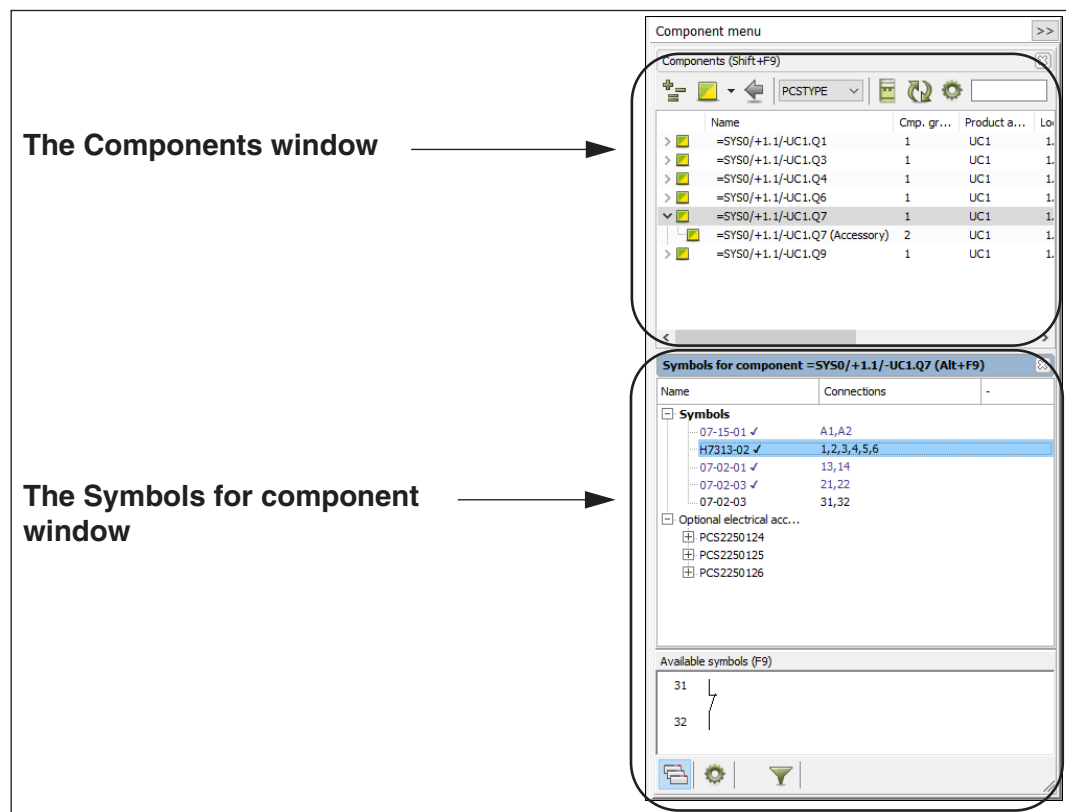
The **Component menu** is placed in the right-hand side of the work area, and gives you an overview over the applied components in the project.

The **Component menu** only displays symbols from the symbol category (such as electrical, mechanical, pneumatic or hydraulic), which are selected for the active page in the project:

- When you work on a *Diagram page*, only components with *electrical* symbols are displayed.
- When you work on a *Mechanical page*, only components with *mechanical* symbols are displayed.
- If you work with another symbol category on the page, only symbols from this category are displayed.

The **Component menu** contains two independent windows:

- The **Components** window, which contains a list of the applied components in the project and
- The **Symbols for component** window, which contains a tree structure displaying all of the symbols and optional accessories for the selected component, as well as a pickmenu containing the available symbols for the component.



You can read more about working with the **Component menu** in the main manual for **Automation**. (Page 254 in the A4 version / page 445 in the online manual).