# NEWS IN AUTOMATION – VERSION 26

This document describes the news in PCSCHEMATIC Automation version 26.

Last edit May 2026.

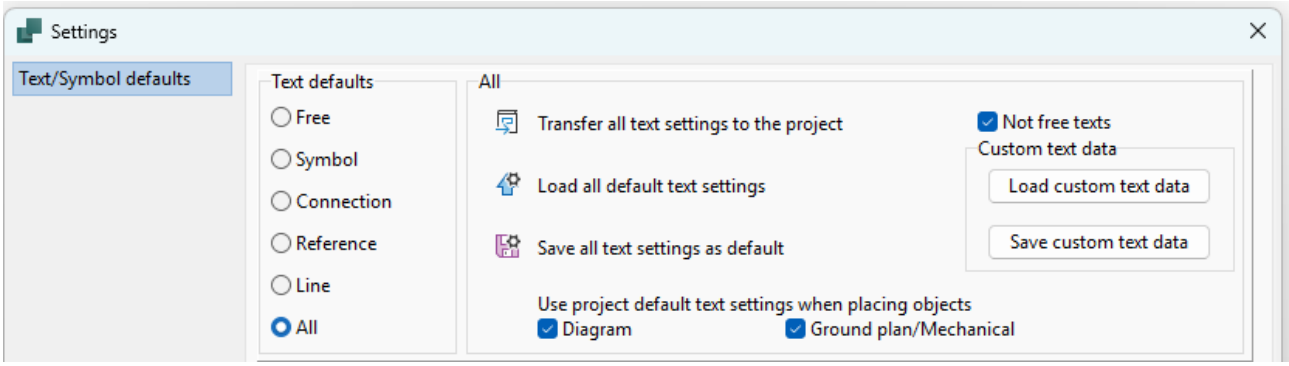
# CONTENTS

# TEXT / SYMBOL DEFAULTS

Many versions ago we made a function to let you easily update a project with your text settings.

It works in the way that you can copy text settings from one project and transfer them to another. It is possible to save text settings to 'custom text data' files to allow multiple settings 'with a click'.

The transfer text settings transfers to DIAgram pages.




It is also possible to apply the project's text settings when you place symbols in the project.

Until now, it has been on all pages or on no pages, but from ver26, it is possible to exclude GRP-pages from the default settings.

In this way, you can create mechanical symbols with texts that are fitting for the page's scale to optimize readability.

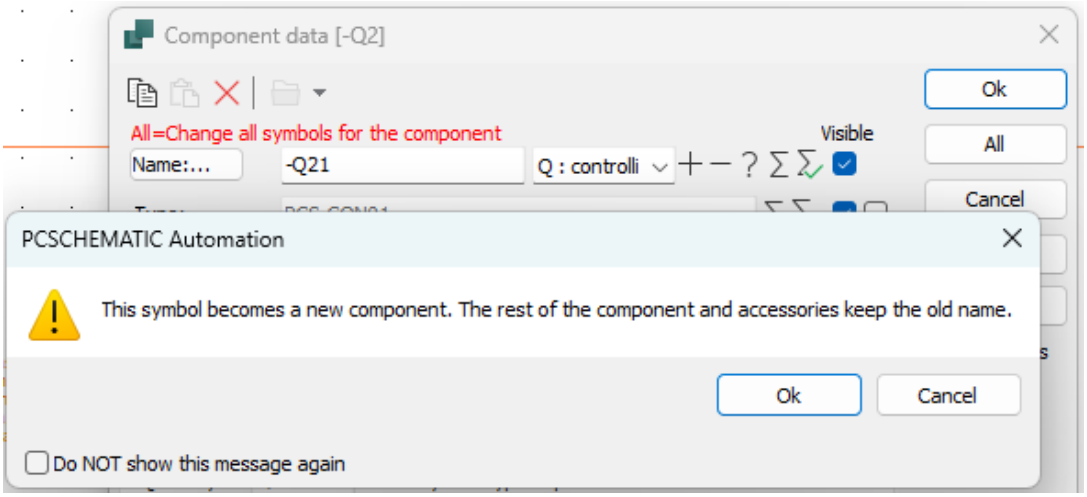
## OK OR ALL?

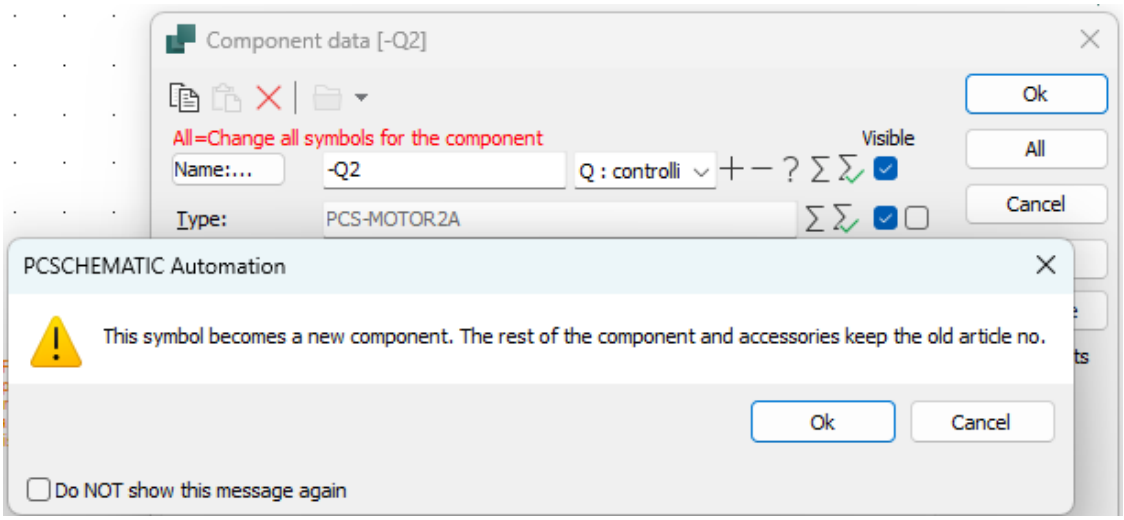
There is still confusion among some of our customers about which button to push – Ok or All – when you change name or article number on a *symbol.*

- A *symbol* is the drawing function and the object you place in the drawing
- A *component* may consist of many symbols – on the diagram and / or in the layout and graphical plans.

If you change anything, it is most likely that you want to change the *component* and not only the *symbol.* You get a warning written in red (and this warning has been there for many versions…). This is also the reason why we changed the program's default button on Enter to be All.

If you press the 'wrong' button, your parts list will be incorrect. Therefore, we have made a bigger warning:

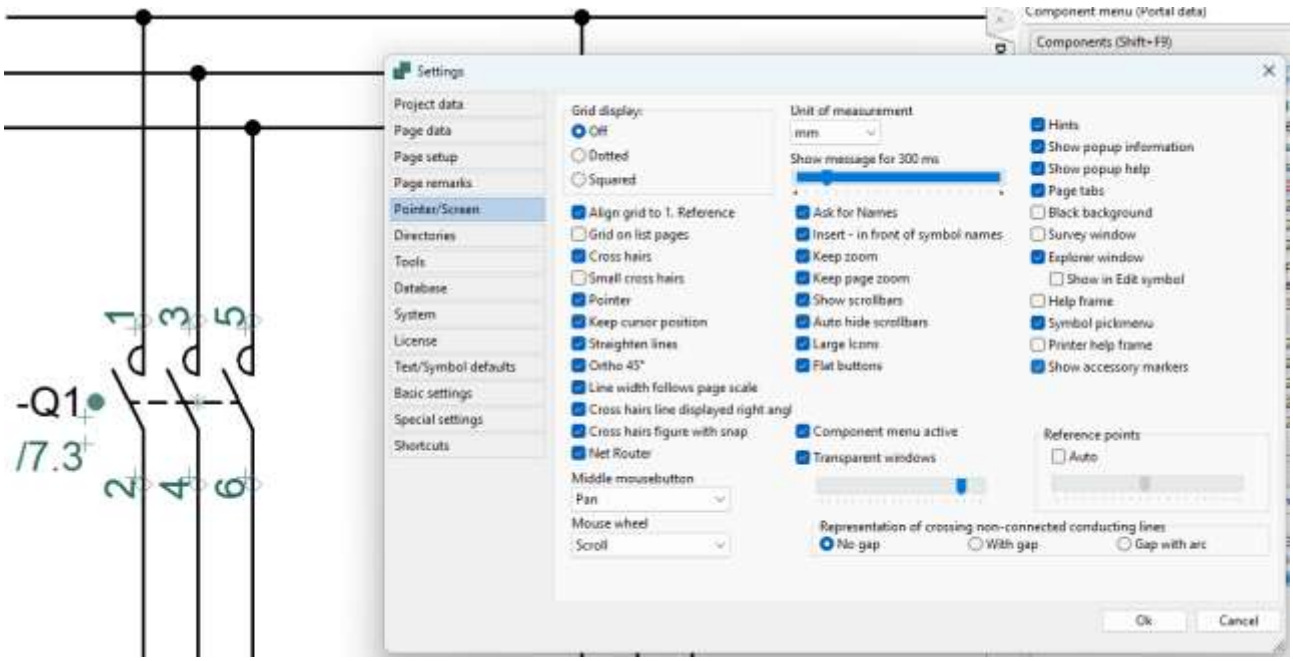



If you choose 'Do NOT show this message again', it will disappear, but popup again after updating the program, just to make sure, that you don't forget 😉

# DOT OR NOT?

A new check box for Show accessory markers has been added to the Pointer/Screen settings tab.
In that way it is possible to deselect the dot that indicates selected accessories to components.

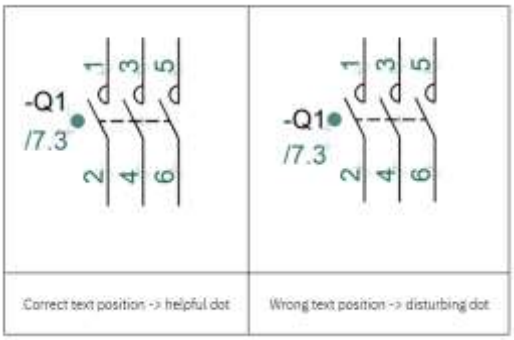



## The dot's position

The dot is placed automatically when *optional* accessories are selected to a component. The position is aligned to the *.

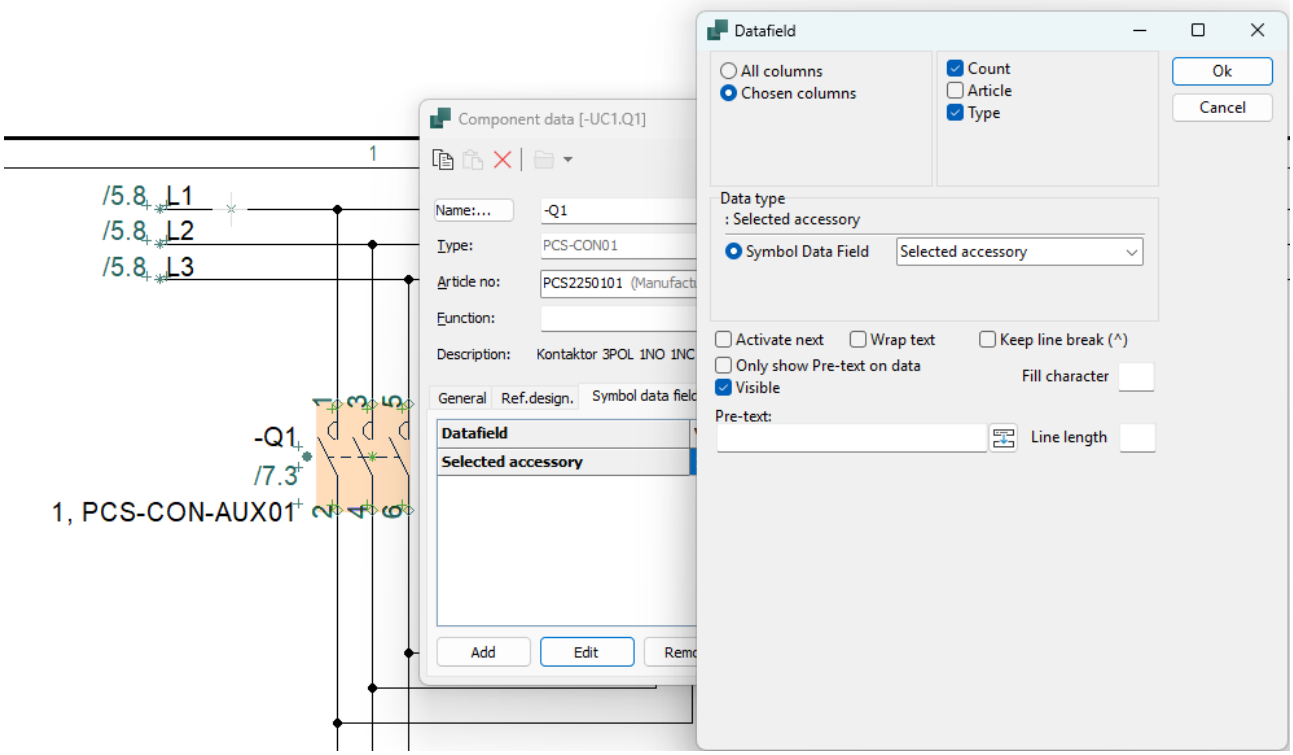
The general rule for symbol design – which is not automatically followed for all symbols ... - is that that position is free from texts. If that's not the case, then the dot might disturb more than it might help.

Remember, the dot is not printed.



# SHOW SELECTED ACCESSORY

The data field Show selected accessories can now be adjusted to show what you find relevant.
In previous versions all columns from the Accessory tab would show; now you can limit to your preference.
Default – and for updates – will be All columns (as earlier), which can be changed in the individual projects.



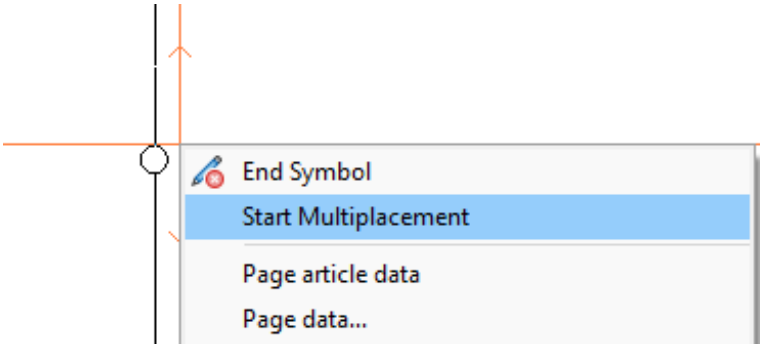

There will be one line per type/article no of selected accessories.

The selection is done for all instances of the data field in the project and is valid for the full project, meaning that also lists with the data field will follow the project setting.

| Name | Type | Selected accessory |
|---|---|---|
| -M1 | PCS-MOTOR2A | |
| -S1 | | |
| -S2 | | |
| -F1 | PCS-MCB-017 | |
| -F1 | PCS-MV001A | |
| -Q1 | PCS-CON01 | 1, PCS-CON-AUX01 |
| -Q1 | PCS-CON-AUX01 | |
| -Q2 | PCS-CON01 | |

# TERMINAL SYMBOLS AND COMPONENTS

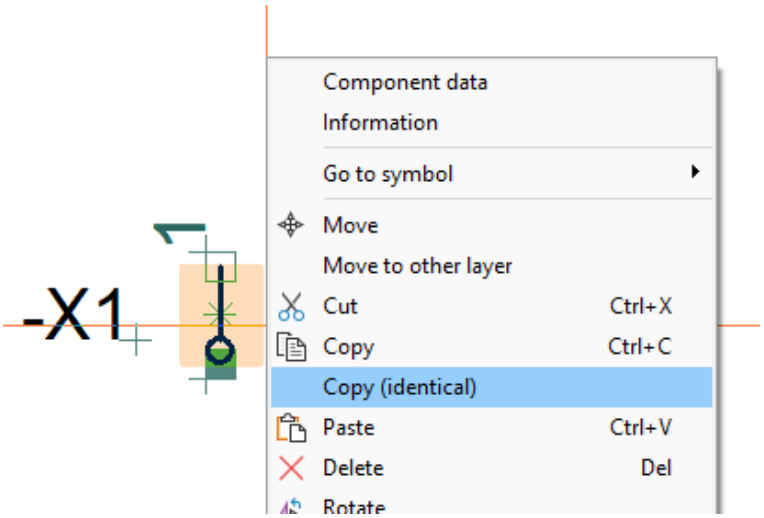
Terminal are special symbols and components.




That also means that you sometimes need to have special commands for the components, and two of those are

- Multi placement
- Copy identical

Those two functions are now only for terminals – as they were intended from the beginning.

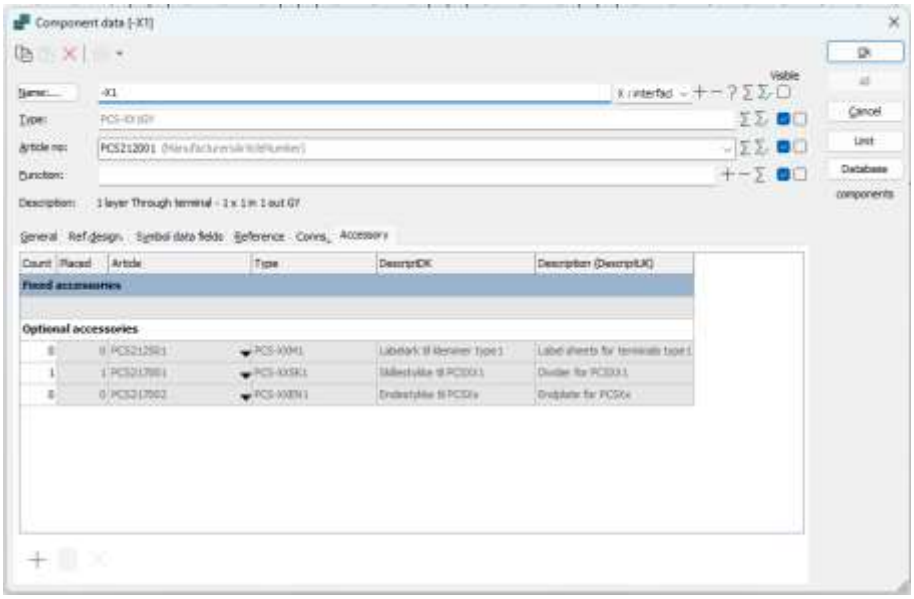


# TERMINALS – PLACE ACCESSORIES AUTOMATICALLY

When you select accessories for a terminal row, you often have a lot of parts that you want to connect to individual components and subsequently place in a logical position in the layout.

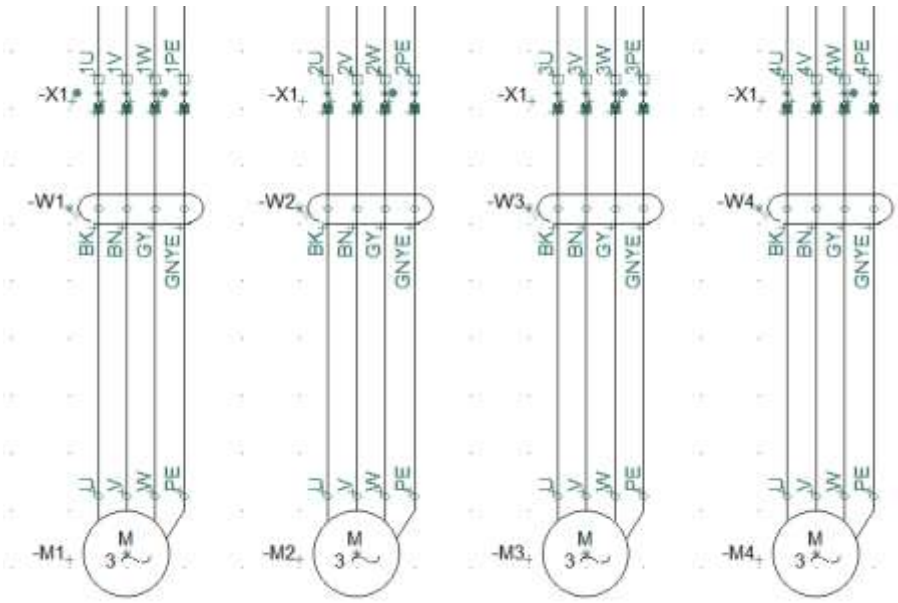
## In the diagram

You can select accessories to the individual component on the accessory tab, and when you do that, it is included in the project and its parts and components lists.




You can choose to see a data field that shows the component's accessory – see page 8.

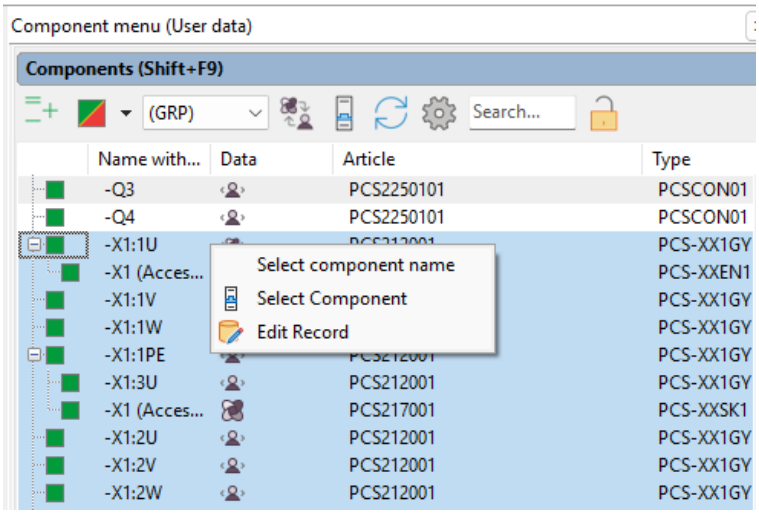
A dot pops up automatically – see page 7 – when you have selected accessory to a component.



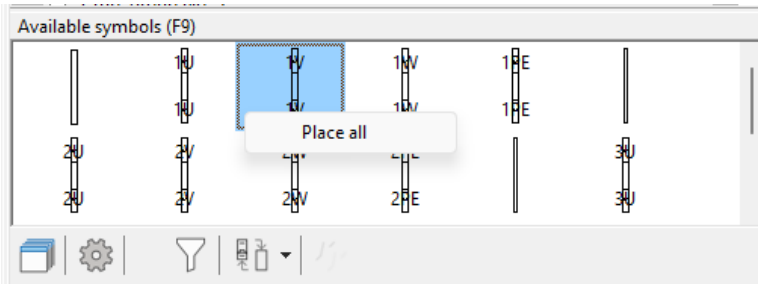
## On the layout page

When you want to place the mechanical symbols for the terminal, right-click and select the component name.

All components in the terminal row are selected – including the accessories.



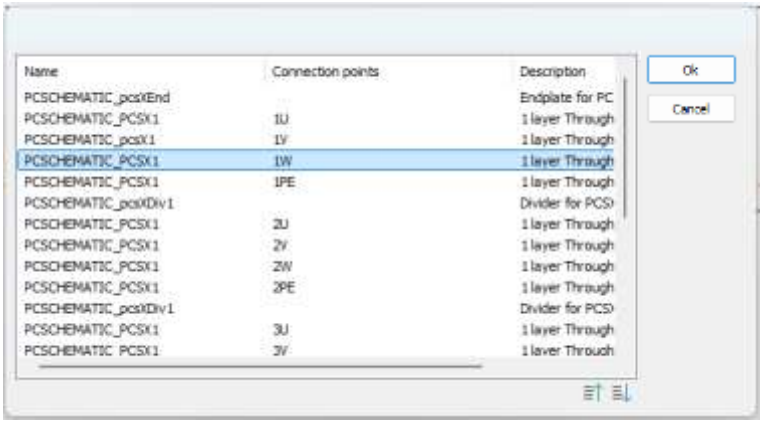

In the Available symbols window, you see all components with the selected name. Right-click and select Place all.




You get window with all components listed in the default order:

- Accessories to the FIRST component is placed in front of it
- Accessories to all other components is placed after it

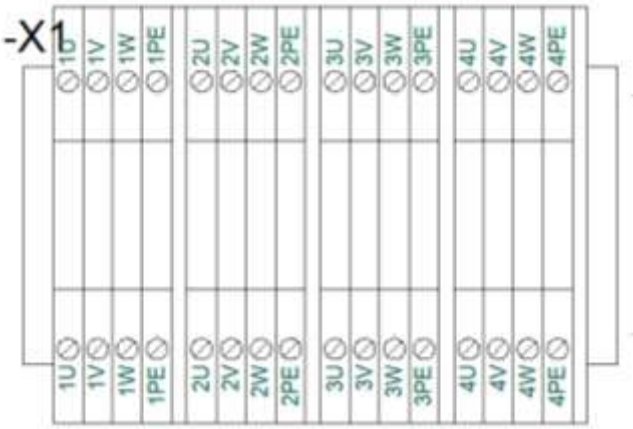
IF you want to move a part, you select it and use the arrows at the bottom.




## The result

All components belonging to the terminal row are placed in the selected order.

The name is placed only once.



# COMPONENT WIZARD

We are improving our Component Wizard tool, so that creation of components becomes more and more easy for customers – and ourselves.

## Create component | Terminal

The Component Wizard has a few improvements in ver 26.

One is an improvement on how to create Terminals, where you now have only ONE field per symbol to add a value, no matter how many connections a terminal symbol has.

Remember the basic rules for terminal diagram symbols:

- The symbol type is Terminal
- All connections on a terminal symbol have the same value
- All connections on a terminal symbol have the same electrical potential
- A terminal symbol has at least two connections: one internal, one external. Connections may have jumper status.

A terminal component can consist of more symbols that follow the above rules.

All symbols in the component are Terminals.

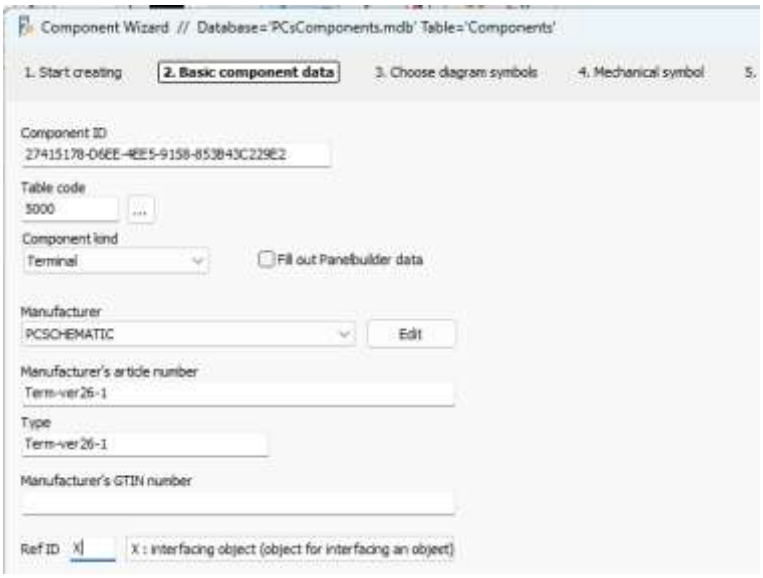
A terminal's mechanic symbol may have more connections with different names: 1 = layer 1, 2 = layer2, 3= layer3. To do this, the symbol MUST be created on a mechanical page!

We have created a new Terminal demo-file in which we show examples of some of the more tricky components that are found also on our Component Portal.

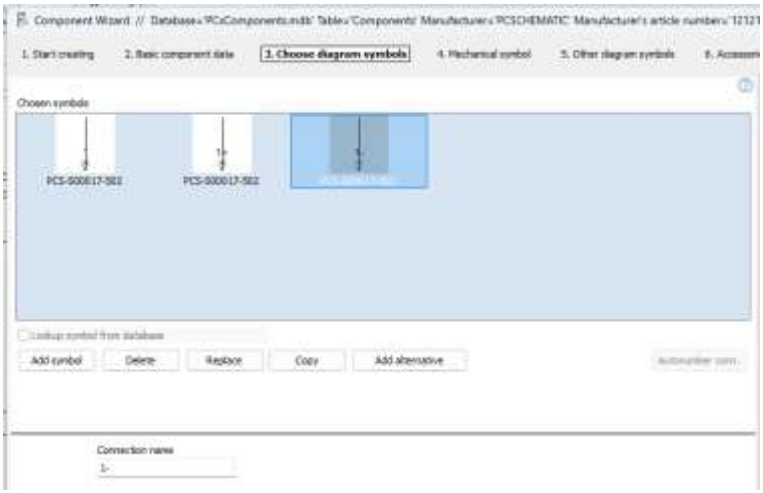
## Example

Create a 3-layer terminal

- Select component kind = Terminal

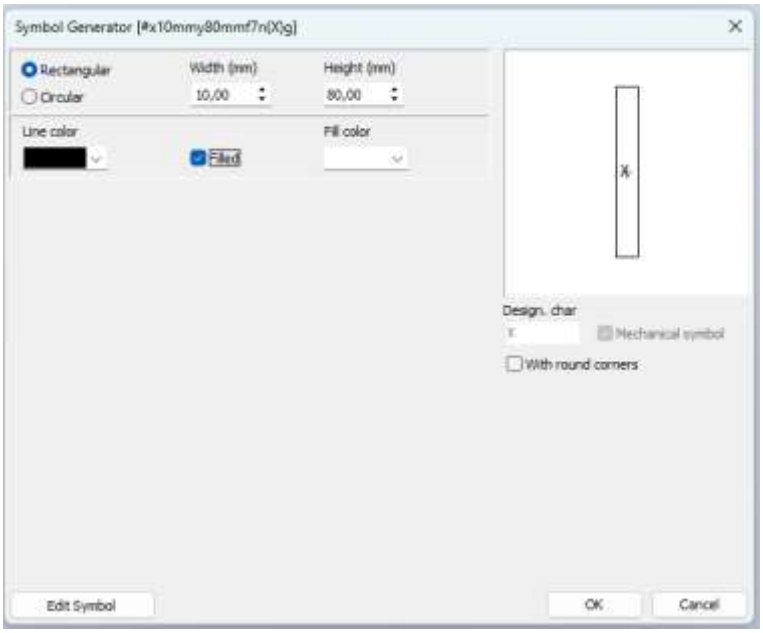


- Select diagram symbols
  - There is one small input field below for naming the individual symbol
  - Here I call them 1, 1+ and 1- so that I can easily see where + and – belongs 😉




It is possible to create a simple mechanical symbol, also for terminals. The symbol type is Terminal, as mechanical symbols for terminals should be.

If you want to see the individual connections, you need to create a mechanical symbol with the right connection names: 1,2,3 referring to the layers / positions that are defined in the diagram section.



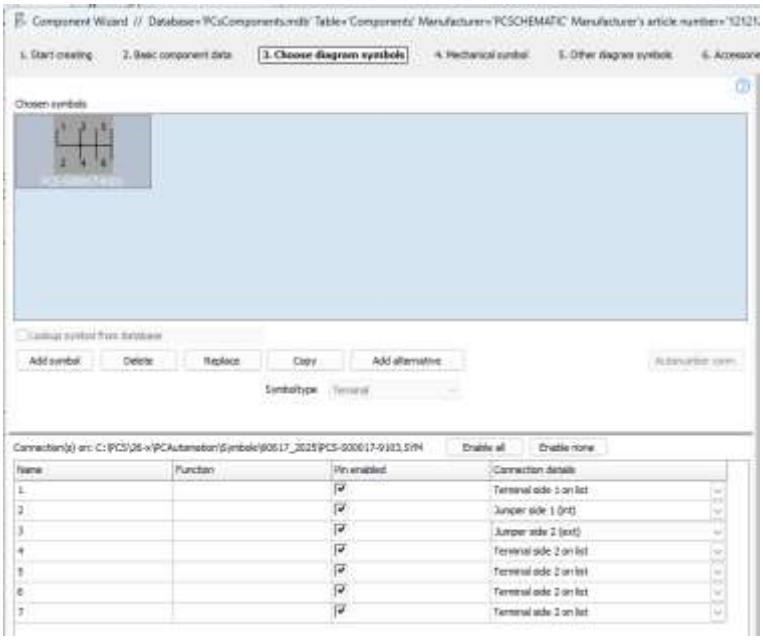
## Create component | Distribution block

Another improvement of the Component Wizard is a new component kind – the distribution block.

The distribution block is functionally a terminal, with some exceptions.

The basic rules for distribution block diagram symbols:

- The symbol type is Normal
- All connections on a distribution block have different names (1,2,3 etc)
- All connections on a distribution block have the same electrical potential



A distribution block component can consist of more symbols that follow the above rules

The component's symbol is Terminal (controlled by the database)

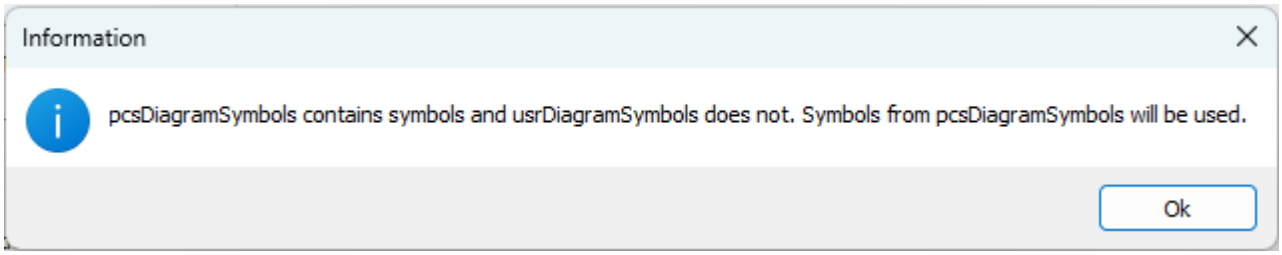
The component symbols may have internal, external or jumper status, in the same way as other terminals.

We have created a new Terminal demo-file in which we show examples of some of the more tricky components that are found also on our Component Portal

## General improvements in the Component Wizard

One issue that pops up is that some components lack symbols in some aspects.

We have information – see below – about No symbols, but prevention is better and therefore we start by copying all symbols from the portal to the user fields, and in that way ensuring that there are symbols in (at least) the same aspects as what the Component Portal provides.




We have also fixed some bugs and annoyances from the wizard, ie splitters that move without consent, symbols that change state upon copy and a couple of other issues.[i]

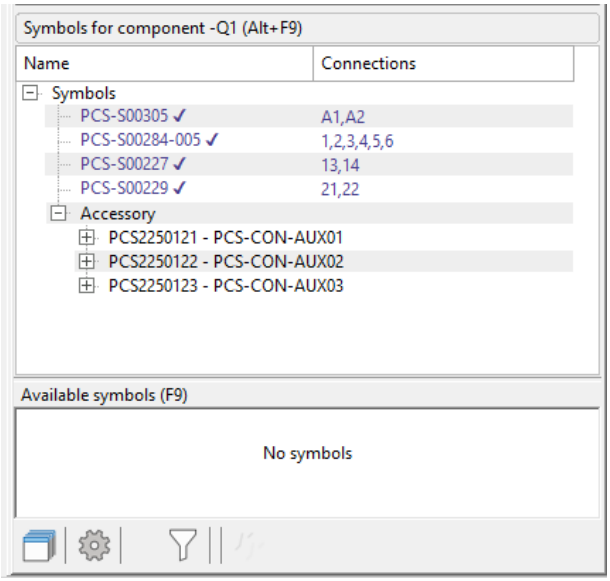
# INFO IN AVAILABLE SYMBOLS: NO SYMBOLS

Sometimes it is difficult to know whether you should expect a symbol for a component in the selected drawing aspect.

We have now added a simple text saying ‘No symbols’ when

- All symbols in the selected aspect are placed
- There a no symbols in the selected aspect to place

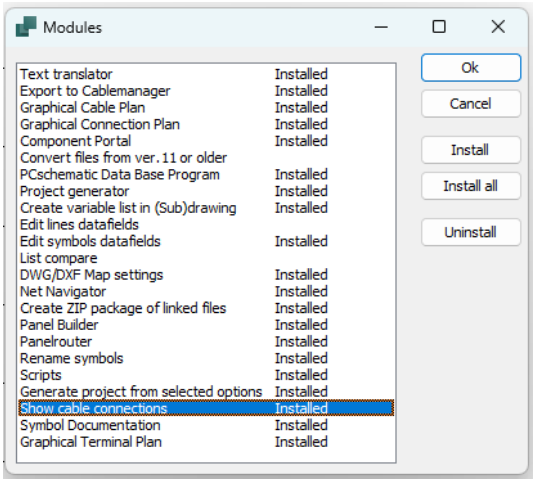
In case of all symbols placed, you can see check marks next to them in the window above.

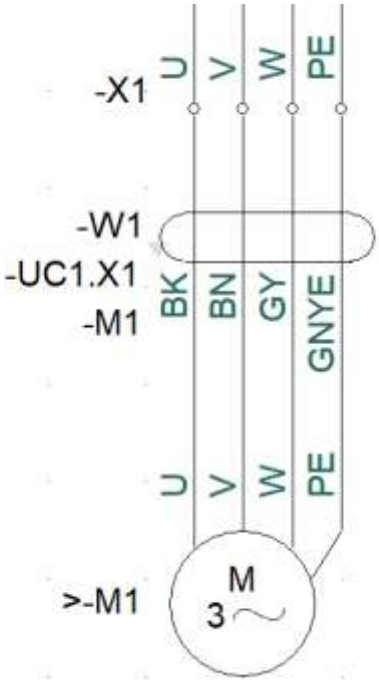


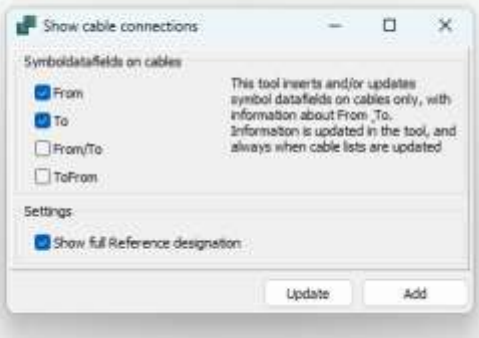
# SHOW CABLE CONNECTIONS

A new tool – Show cable connections – has been added to the program. More customers have wished for a function which automatically shows a cable's connection, so that it can be read directly in the diagram.

You install it by going to Files|Modules and click Install. Then the TOOL (found in the Tool menu) can Update and Add one or more data fields to a cable symbol.

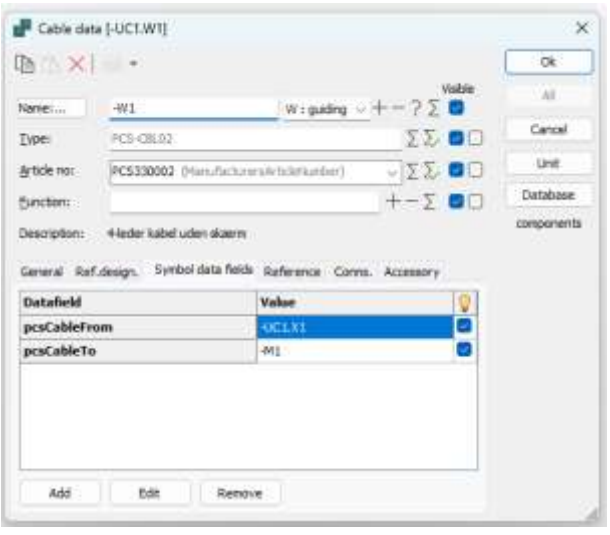



You can choose between

- From and/or To (in separate data fields)
- From/To (combined data fields)
- To/From (combined data fields)



## Show full reference designation

There is a checkmark below the data fields, to allow you to insert the full reference designations of the From and To components.

It works similar to the checkmark on all components' rds-tab, meaning that it either shows the full designation or follows the rules according to 81346-series about reference frames etc: you will see *either* the exact same information as in the diagram *or* the components' full reference designation.

## Update or Add

The Update button updates all present data fields, maintaining their last known positions. The default position is to the top right of the symbol.

The Add button adds the selected data field(s) to ALL cable symbols in the project.

## Create your own cable symbol – with data fields

In the picture above, the data fields are moved to the left side of the symbol, which is the preferred position for symbol data.

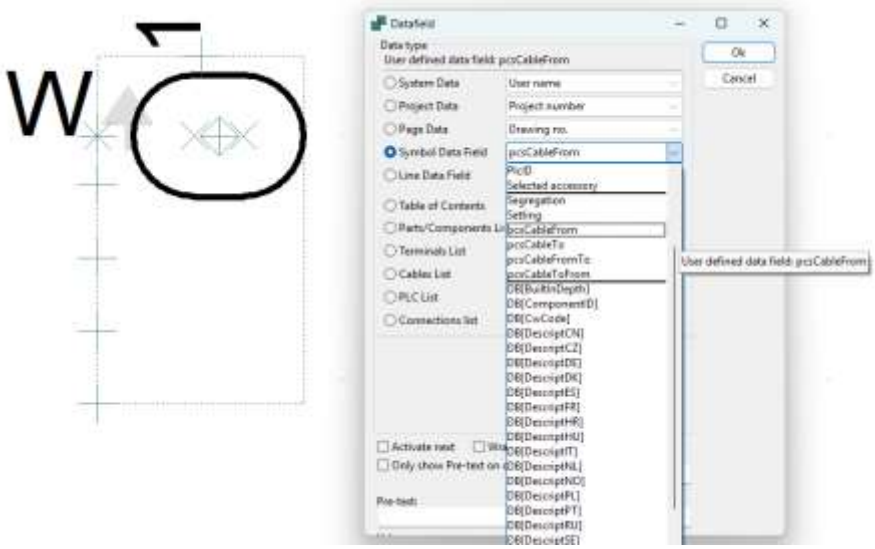
And when you update, the data fields will remain there.

It is also possible to create your own version of the symbol, that includes the desired data fields in the right position. In that way, you can decide which symbols that have the desired data fields in the right positions.

The new data fields have been created in new installations and in updated installations, and they are found section with user-defined data fields.

In that way they can be used with all CABLE SYMBOLS.

And cable symbols only!



# WIRENUMBERING – NEW FORMAT

The function Wirenumbering adds small wireno symbols with values on the wires in the diagram. The value in the symbols can be printed with the dedicated wire no list and the connection list on ie label printers.

Basically, you have to different ways of numbering:

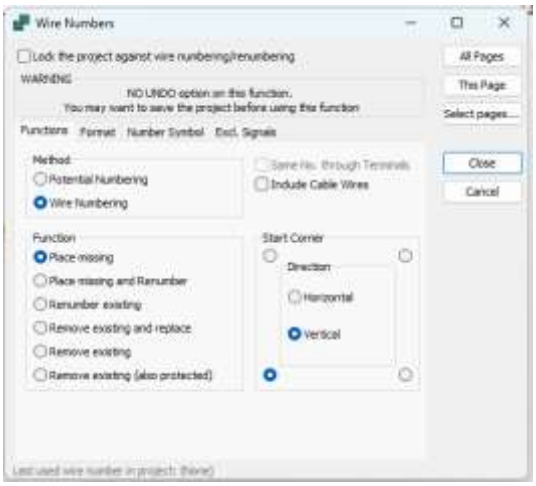
- Per connection / wire (method Wire numbering) (new format option from ver26)
- Per net / potential (method Potential numbering)

## Wire numbering – method

When you choose the method Wire numbering, you add an individual number to each wire / connection in the diagram.

The program automatically excludes jumpers, and you can select to add numbers to cable wires.

On the Excl. signals tab you can choose to include / exclude connections with signal symbols.

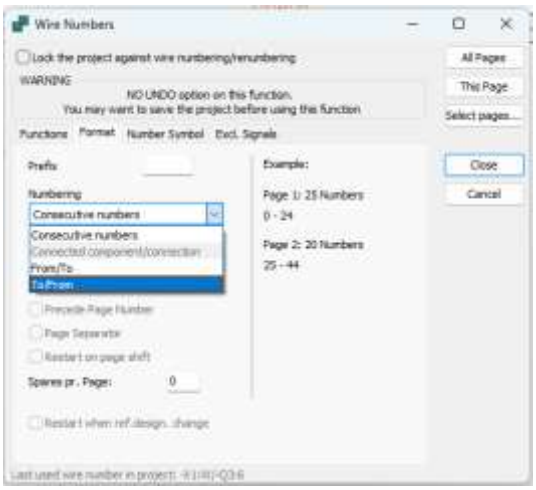



## Format

On the Format tab, you select the format.

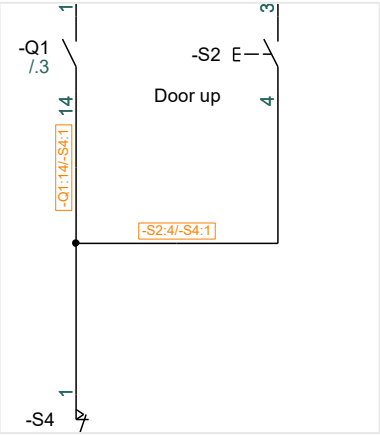
From ver26, you can choose between

- Consecutive number (as before)
- From/To – new format
- To/From – new format



## The result

From / To will show Name: connection / Name: connection for the From and To components, respectively.



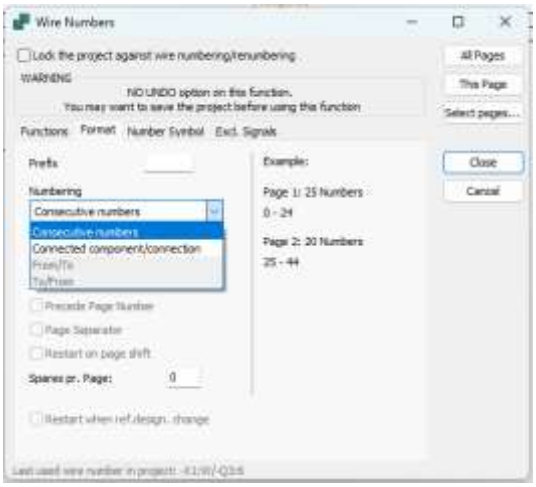

Beware – as it says in the warning about 'dots' and wire numbering: the program can only guess when you use dots and not mounting correct bends!

## Potential numbering

When you choose the method Potential numbering, you add an individual number to each potential / net in the diagram.

The program automatically excludes jumpers, and you can select to add numbers to cable wires, and you can choose to have the same number through terminals.

On the Excl. signals tab you can choose to include / exclude connections with signal symbols.

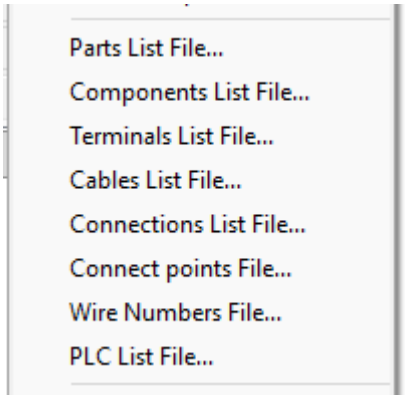



## Format

There are no new options here, so you can choose from – as earlier:

- Consecutive numbers
- Connected component:connection (the first symbol in the net, selected here)

## Export to third party programs – general function

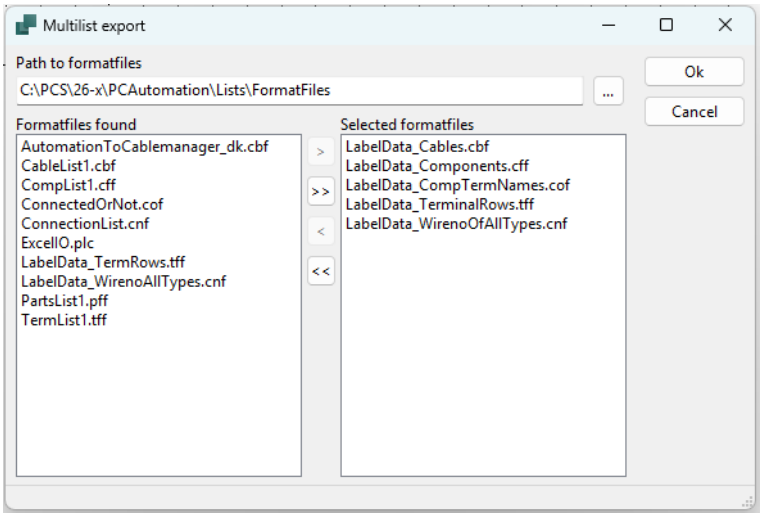



The export-to-list file function is a general function that allows you to use information from the project directly for third-party programs, ie label printers, when you export Excel-files. Below examples of use:

- Components lists – used for labels on components and on panel front (name and function texts)
- Terminal lists - used for labels on terminal rows. When component grouping is correct, you have the right connection names on each layer.
- Cable lists – used for marking of cables, name and from/to component
- Connection lists - used for marking of wires: ALL information about from/to is here, including wire numbers
- Connection points file – used for marking of wires with FROM information (terminal marking – CL)
- Wire numbers file: the contents of the wire numbers are exported to a file (special format).

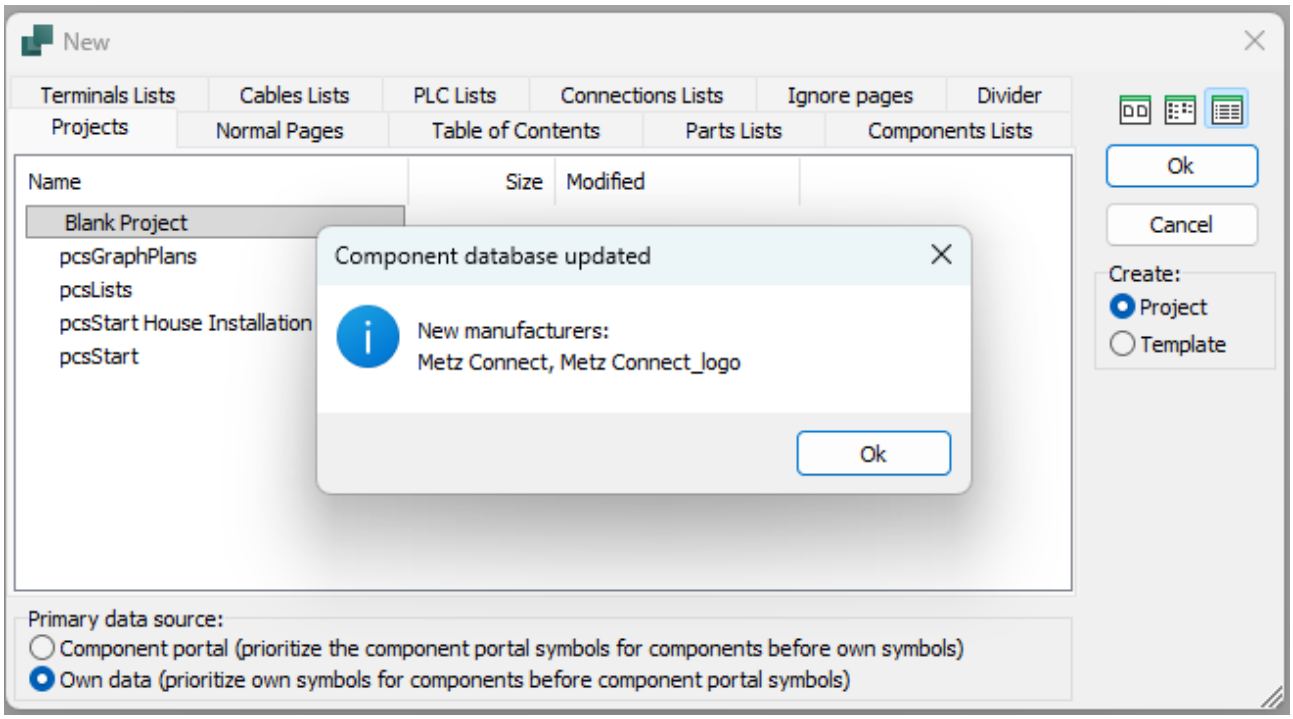
We have added new format files for export of all label types. They are all prefixed LabelData, so that they are easy to find with Multilist export:

Format files with too long names and old format files (missing data) are not shown in the window.



# UPDATED LIST OF MANUFACTURERS

We keep adding new manufacturers to the Component Portal, so that everybody is updated. You simply get an update – if any – each day when you open the program.



# GENERAL IMPROVEMENTS

## Program optimizations

We have made optimizations to allow the program to handle more pictures before it runs out of memory. This is not an invitation to add more pictures to your projects 😉

We have also optimized the drawing routines, which means that (very) large projects will load and save faster.

## New demo projects

We have created two new demo projects – one is a motor control system with a plc, the other is a demo showing different terminal types; what they should look like symbol-wise and how they could be used in projects.

## New demo components

The Portal's manufacturer PCSCHEMATIC will have an increased number of components for downloading. In this way we will communicate best practices of how to create various kinds of components. Some of the components will be new components to the demo-database, others will updated components for it.

## NOTES