# NEWS IN PC|AUTOMATION VERSION 24

This document descrinbes new features of the PC|Automation version 24.

Last edit: February 2024

# Contents

Symbol editor ..... 5
Diagram symbols ..... 5
Mechanical symbol ..... 6
The Advanced tab ..... 6
Symbol generator – Text placement ..... 7
Symbol generator – mechanical symbol with round corners ..... 7
Upgrade projects from ver22 ..... 8
Old settings ..... 9
Unknown manufacturers ..... 9
Old symbols ..... 10
Unknown_xxx ..... 10
In case you have imported your ver22 Alias list ... ..... 11
The CW creates Panelbuilder components ..... 12
When components are used in the Panelbuilder ..... 14
Load old Panelbuilder projects ..... 15
Fetch new components from the Component portal ..... 15
No more Ok/All confusion! ..... 16
Multi letter codes for components ..... 17
A little more about PLCs in Automation ..... 18
When you place a PLC ..... 18
How to distribute the adresses ..... 18
Balanced ..... 18
Forced to use ..... 19
A little more about the new settings ..... 20
PLC with alternatives – Change to another alternative ..... 20
Creating PLCs in the database ..... 21
Info from Support: PLC's and other components in the Component Portal ..... 23
Check your Database components ..... 24
The Component menu – Errors are grouped ..... 25
The 'Generate project from options' module ..... 26
Import Settings ..... 28
Text/Symbol defaults ..... 28
Import user settings ..... 28
User defined symbol datafields ..... 29
News on the Component Portal ..... 30
From 2024: Access to the Component Portal requires a valid MA ..... 31
Various small functions ... ..... 32
More fields in the Component Database ..... 32
Subdrawing – models ..... 32
More manufacturers are added ..... 32
The Automation Service Program ..... 32
Page setup ..... 33
Crossing lines ..... 33
Page snap and grid ..... 33
Insert Potentials – with middle position ..... 34
Automatic renumbering of pages ..... 35

’Printed copy’ on prints ..... 35
Replace symbol ..... 36
More columns in Object Lister ..... 36
List setup ..... 36
Copy line with name ..... 37
The Pickmenu ..... 37
Panelrouter: Output to wire cutting machines ..... 37
Barcodes on parts and component lists ..... 38
Default setting of Text Translator is changed ..... 39
All manuals are online ..... 40
My notes ..... 41

# SYMBOL EDITOR

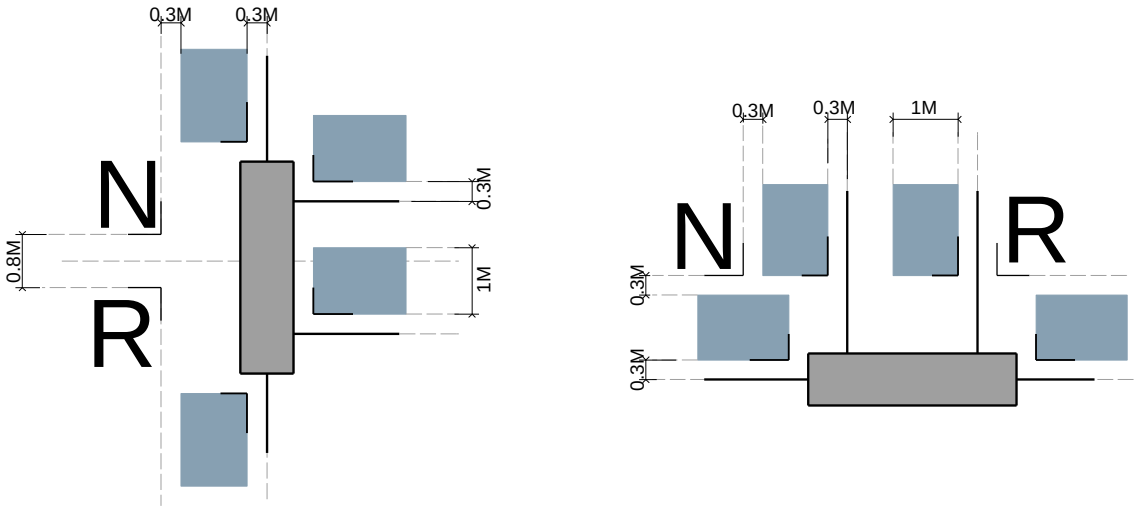
When you create new symbols, you want all texts to be aligned nicely. You don't always succeed the first time ...

The Symbol editoren has been expanded with an extra tab, in which you can adjust all symbol texts. The function works on diagrams symbols and mechanical symbols.

## Diagram symbols

The texts are adjusted according to

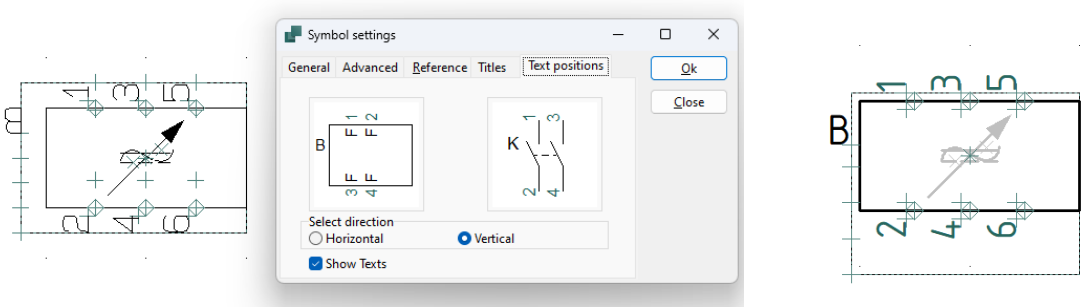
- Distance to 'symbol body' 0,3M = 0,75 mm
- Font and color according to user setting!
  - We recommend to follow the standard, meaning that all texts are 1M = 2.5mm and font= Osifont eller Arial
- Symbol type: Box or line / vertical or horizontal




Below, you see an example with an old symbol from <ver 22. The old symbol at the left has its name set upside/down, and the font was PCSCHEMATIC.

The symbol type is a box and the direction is vertical.

The corrected symbol is at the right.



## Mechanical symbol

Mechanical symbols have the same physical measurements as the component itself.

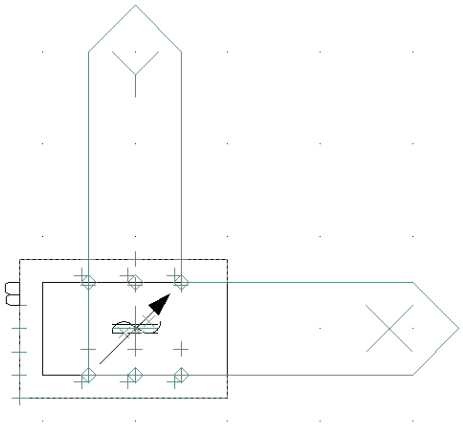
The * (reference point) is places – as always – in the middle fo the DIN-rail.

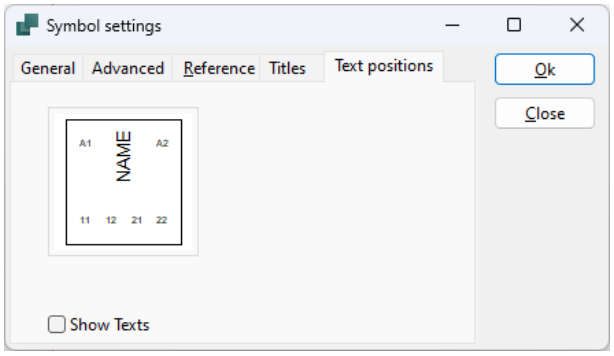
The new function also allows to align mechanical symbols:

You need to be on a mechanical page; if you are, the XY (origo) symbol places itself on the *,

All names are replaced and adjusted

- S.name is aligned in the middle, grows upwards
- S.name size is 2M
- C.names are aligned in the middle of the connection point





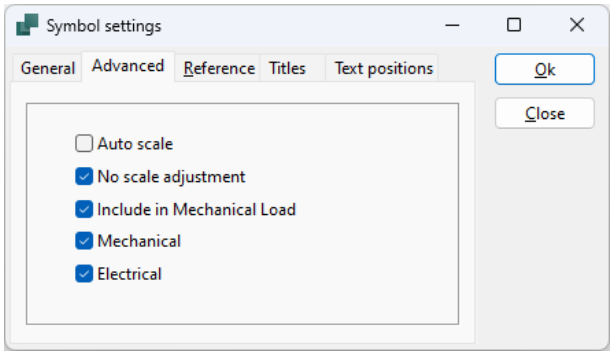

**REMEMBER**

Text/symbol default settings (color, font, size) are set according to the current project's settings!

## The Advanced tab

When you create symbols, it is now possible to select the property 'Include in Mechanical Load' directly on the symbol (same property as in Component Data).

If you create a sensor, you will probably deselect it 😊

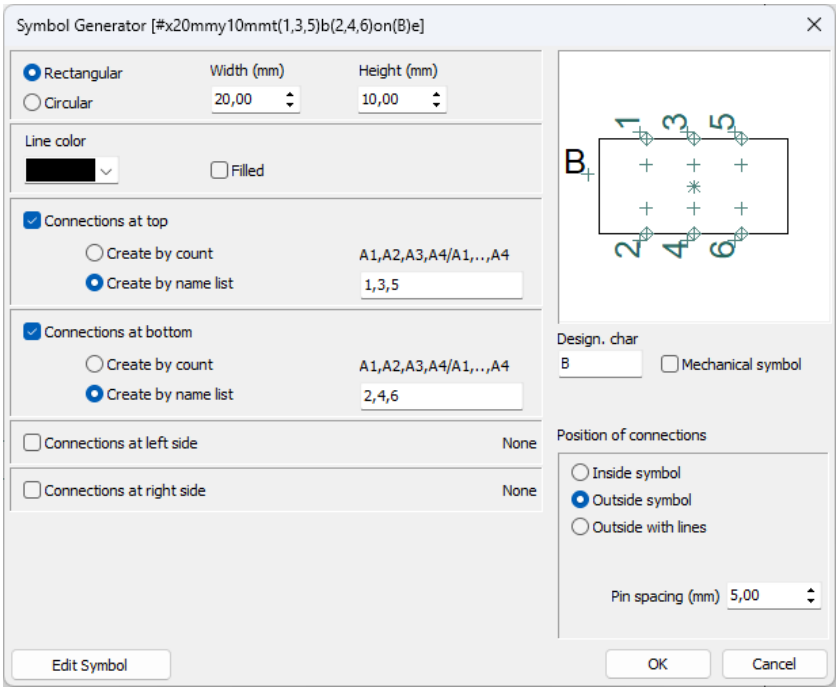


## Symbol generator – Text placement

The Symbol Generator uses the same rules for text placement as we use when we create new symbols.

When you create a symbol like this with connections at top/bottom and connection names outside the symbol, the texts are placed correctly, when you press the OK button and place the symbol in the diagram,

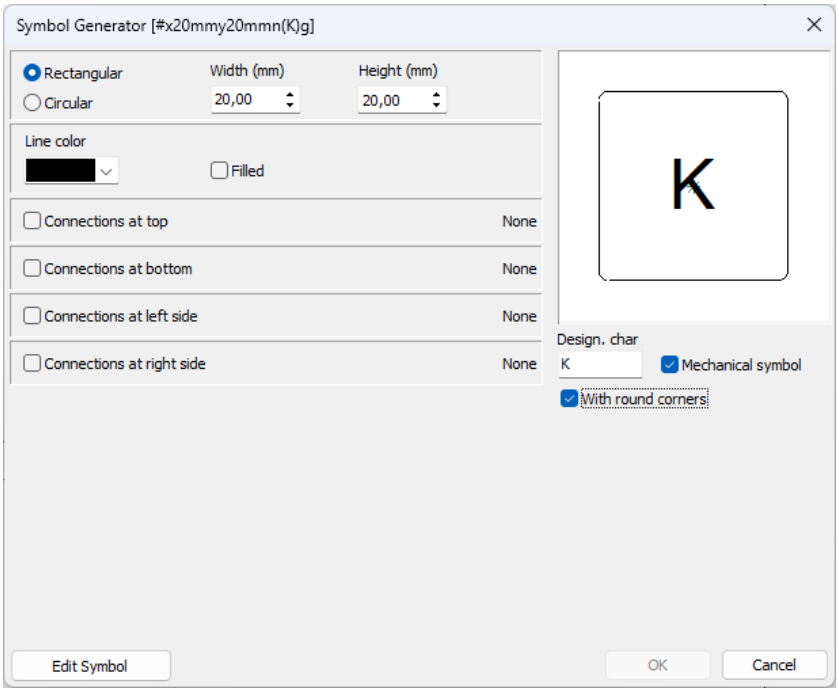
If you want to make something else or edit this, you use the function in the symbol editor.




## Symbol generator – mechanical symbol with round corners

You can create a mechanical symbol with round corners.

If you do it, you must go via the Symbol editor to save it, as the #XY-syntax doesn't allow round corners.



# UPGRADE PROJECTS FROM VER22

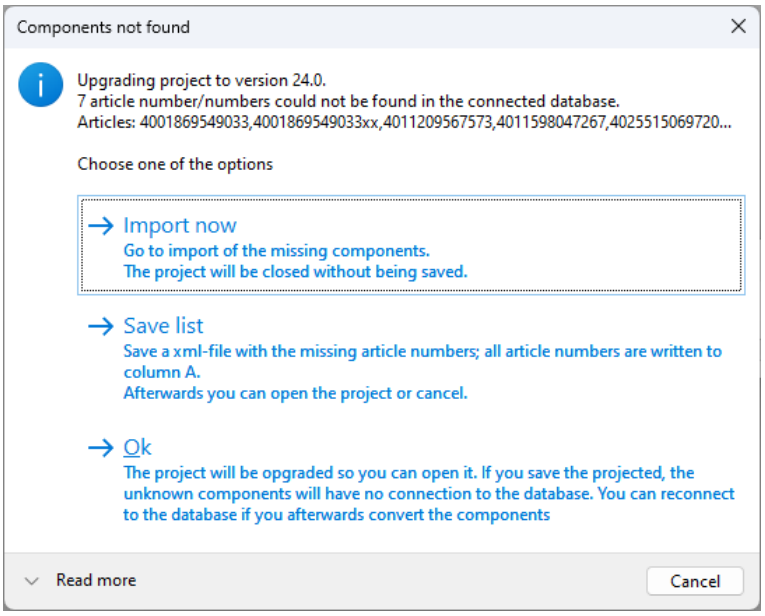
We have changed the dialogs and some of the functions in relation to upgrading of old projects.

You can read more about this below.

You get this dialog, when you open an old project that contains components that aren't found in the connected database.

You can go directly to import of the non-found components, or you can save a list in the same way as you did in ver23 😉

When you select the Import now, you go directly to the import wizard, which is changed slightly.




**REMEMBER**

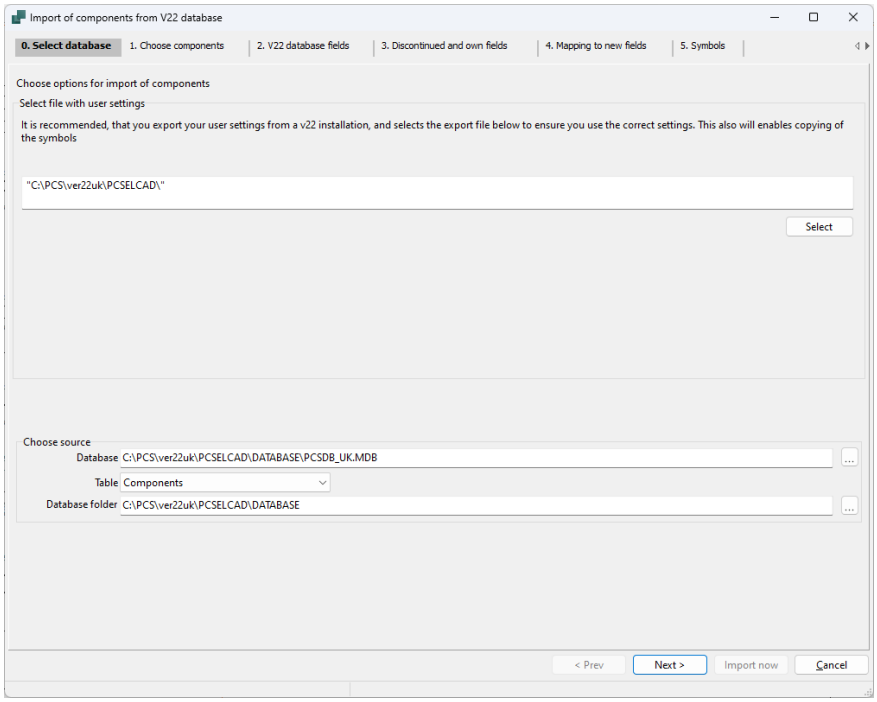
You use your old settings and symbols from ver22, when you upgrade the old projects. So don't delete your old installation as long as you have files that you want to upgrade.

We have made a new guide that describes how you upgrade projects and components from old to new version.

You find it in our webpage.

## Old settings

The first time, you import, you must (should) select the file with ver22 settings. In this way, the 'old' database and your old symbols (and other settings) are found automatically.
On the first page you see the database automatically, that you used in your old installation.
You can change to another database.

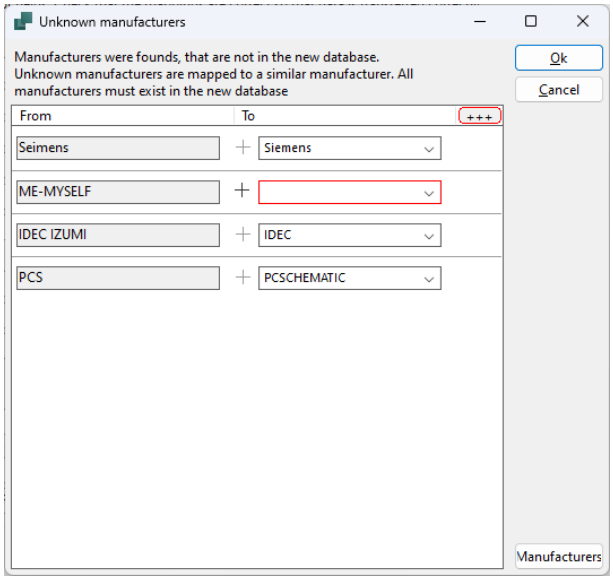



If all the other settings are made (and they are next time, you import), you can press the 'Import now' button and the components are imported immediately into your new database.

## Unknown manufacturers

You still need to consider creating new and unknown manufacturers.

There is a new button to create all unknown manufacturers in one click: if you press the button, all unknown manufacturers are created with their previous names.

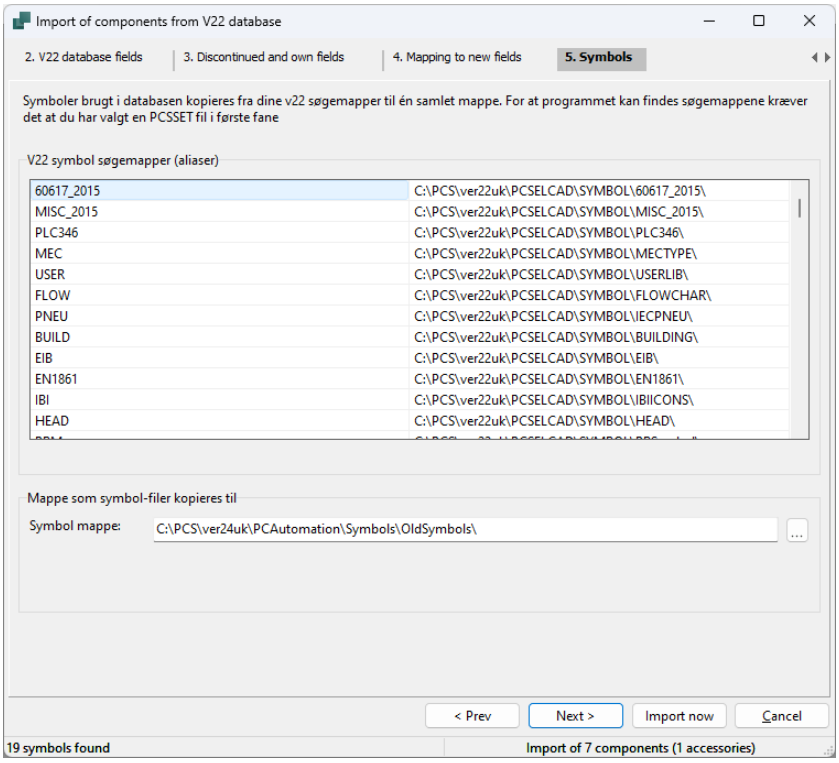


## Old symbols

The main reason that you should select the file with old settings is, that it also contains the paths to your old symbols.

And in the new import guide, we also fetch the old symbols from the shown Alias'es.

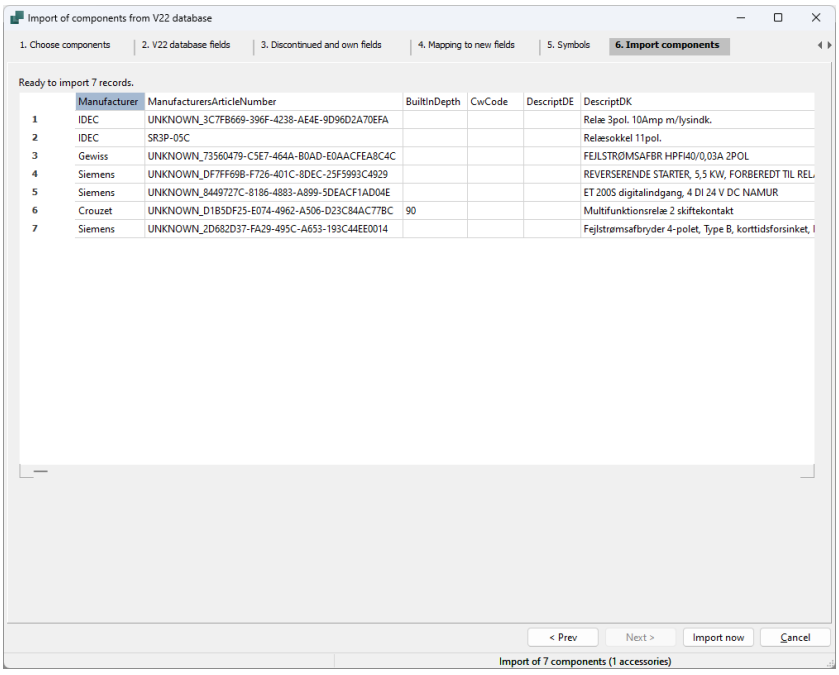
All old symbols are copied into a new folder – OldSymbols – so that all old components have their own old symbols after import.




## Unknown_xxx

If components in this list has Unknown_xxx in their Manufacturers ArticleNumbers, it means that either

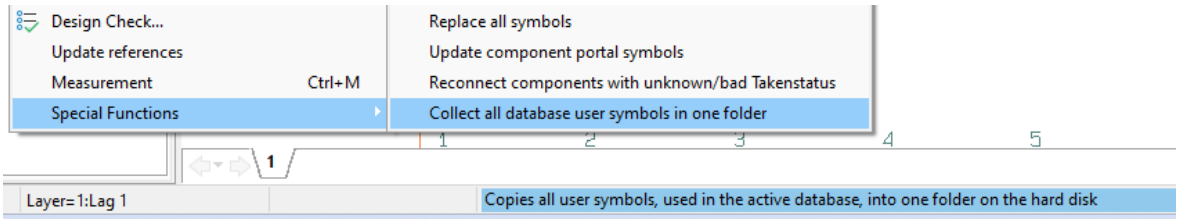
- we haven't created the component in our old database, or
- we did have it in our old database, but the component is obsolete.



## In case you have imported your ver22 Alias list …

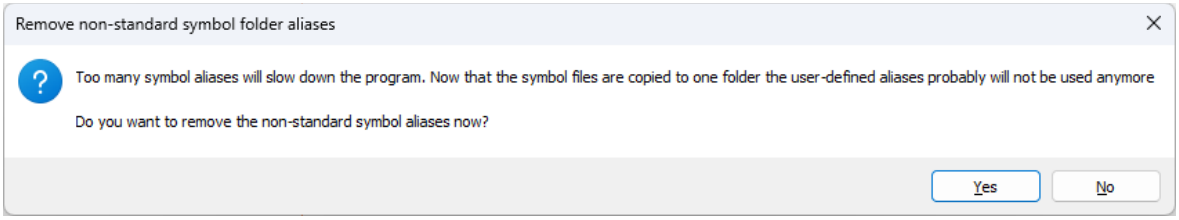
If you have imported you old Alias'es into your ver23, and therefore have a long list, then you can copy all your old symbols into the OldSymbols folder in this way:

In the menu Functions| Special Functions, you select 'Collect aææ database user symbols in one folder'.




You should also clean up the Alias list itself, as a long list will slow down the program. You can do it yourself, or you can let the program do it automatically as a part of this function.

You don't delete symbols or folders, only the Alias list!



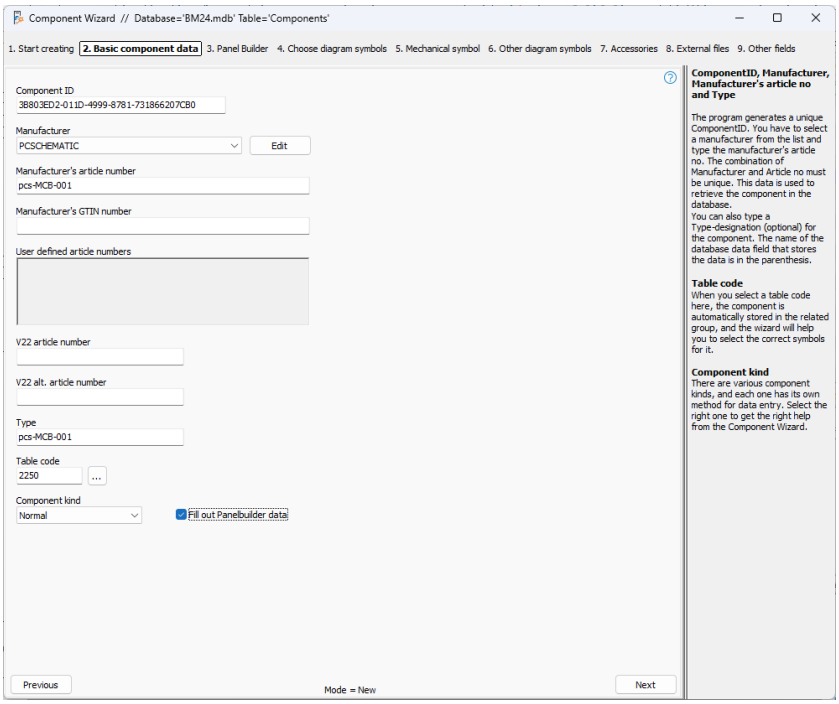
# THE CW CREATES PANELBUILDER COMPONENTS

From ver24 it is possible to create your own Panelbuilder equipment by using the Component Wizard, meaning that you don't need to contact us to create components with powerloss data for the Panelbuilder module.

And just to emphasize the point: we use this tool, too, when we create components for the Panelbuilder.

If you want to be able to use the component with the Panelbuilder module, you select that on the first tab,

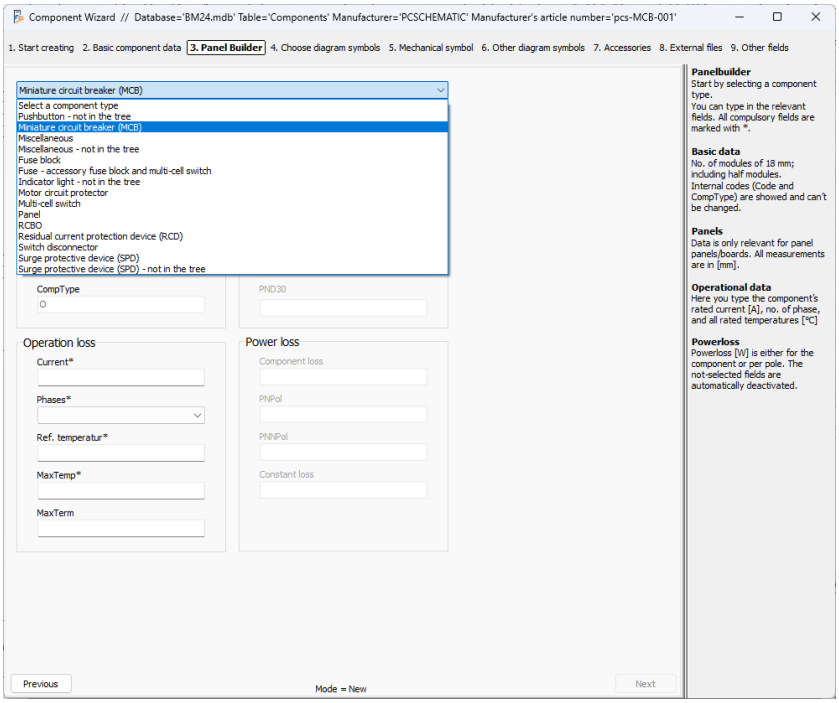
You can select Panelbuilder data for Normal, Terminal and PLC.




When you press Next, you get to this tab.

Start by selecting Component type in the drop down list.

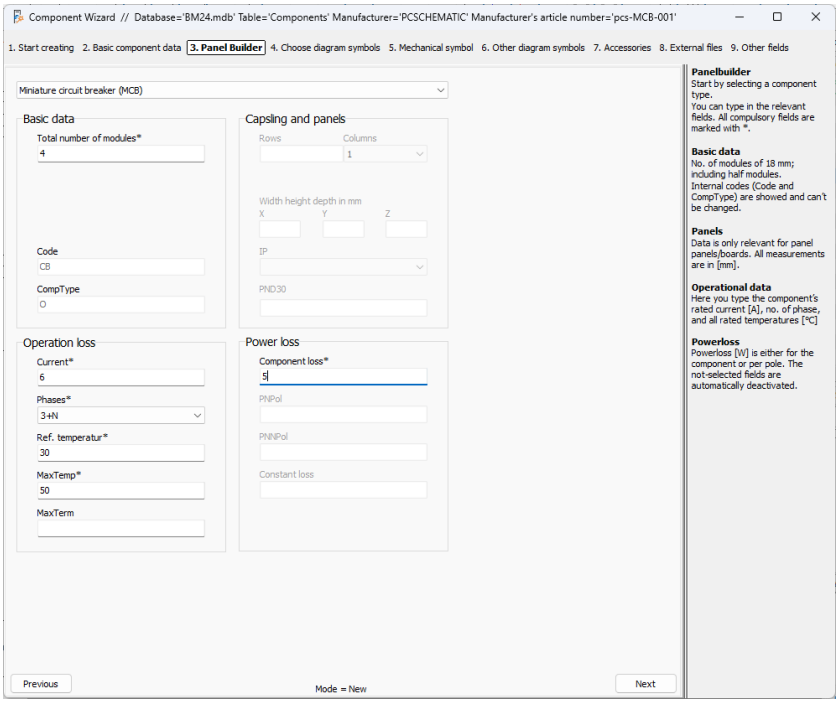
The result is saved in the usrPBData database field; if you fetch a Panel builder component from the portal, the data will be in the pcsPBData database field.

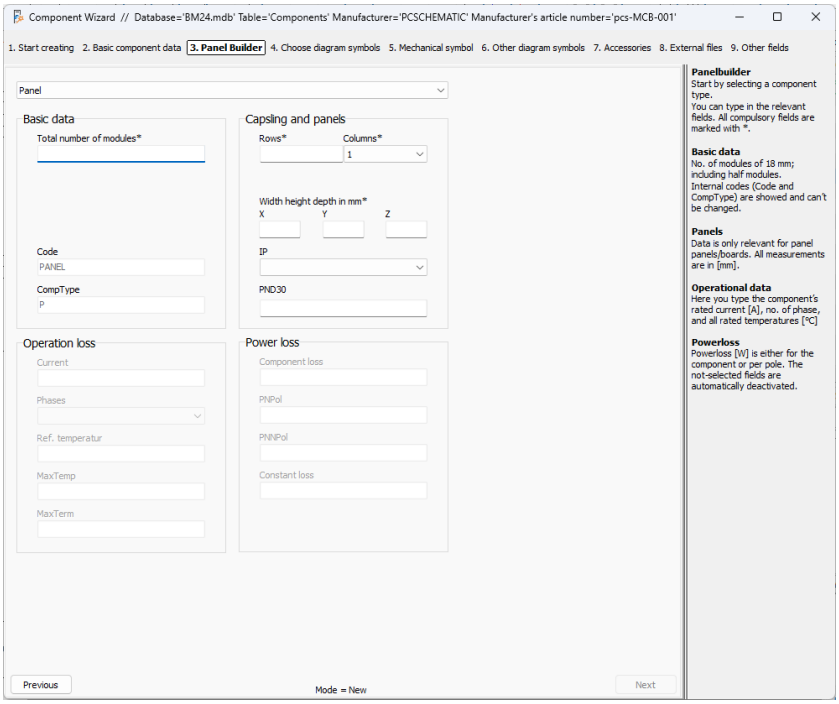



You can see more details about creating components in the manuals for the Component Wizard and the database program.

For each component type, the window shows the relevant data fields, and you can't continue, until all fields are filled out.

Fields marked with * must be filled out, the other fields may be filled out.




Basically, there are four component types:

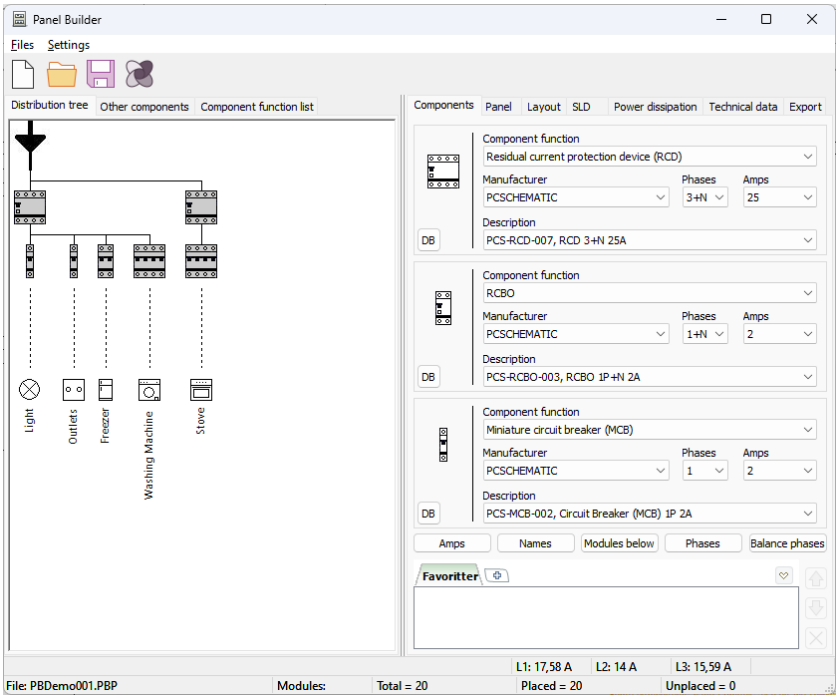
- Panels that contain all components
- Components that are part of the tree, meaning components that are included in the power dissipation calculation with their actual load. The are many different component types in this group
- Component that are not part of the tree, but take up space in the panel. They can dissipate heat and they also have a max temperature. There are many different types in this group
- Accessory, mainly fuses

The input window reflects which data that must be typed for each component type, making it possible to make the power dissipation calculation as stipulated 61349-3.

## When components are used in the Panelbuilder

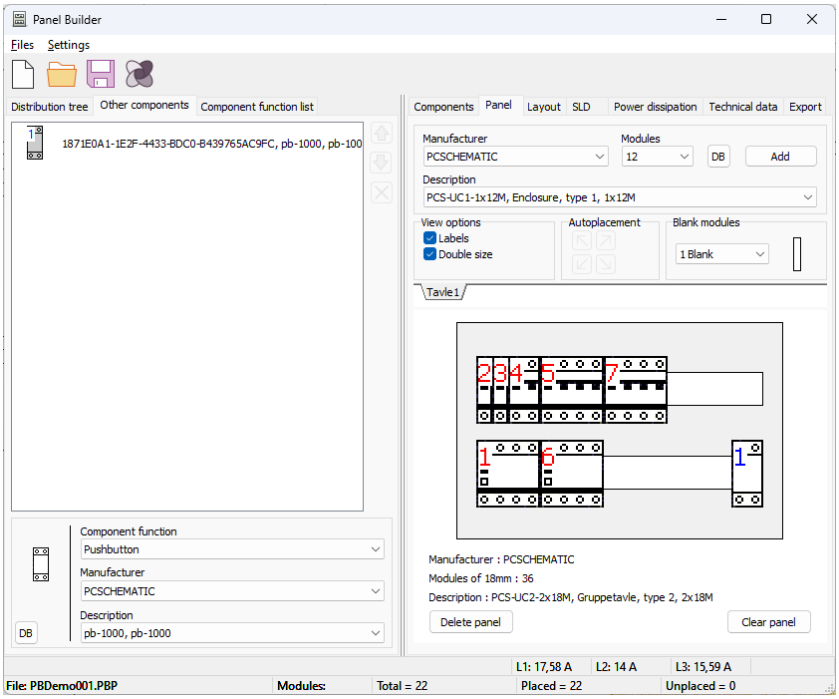
Components that represent the individual groups,  are in the tree,

Also see, that the component names are updated.




Components that 'only' takes up space in the panel: they can dissipate heat and also be sensitive to heat.

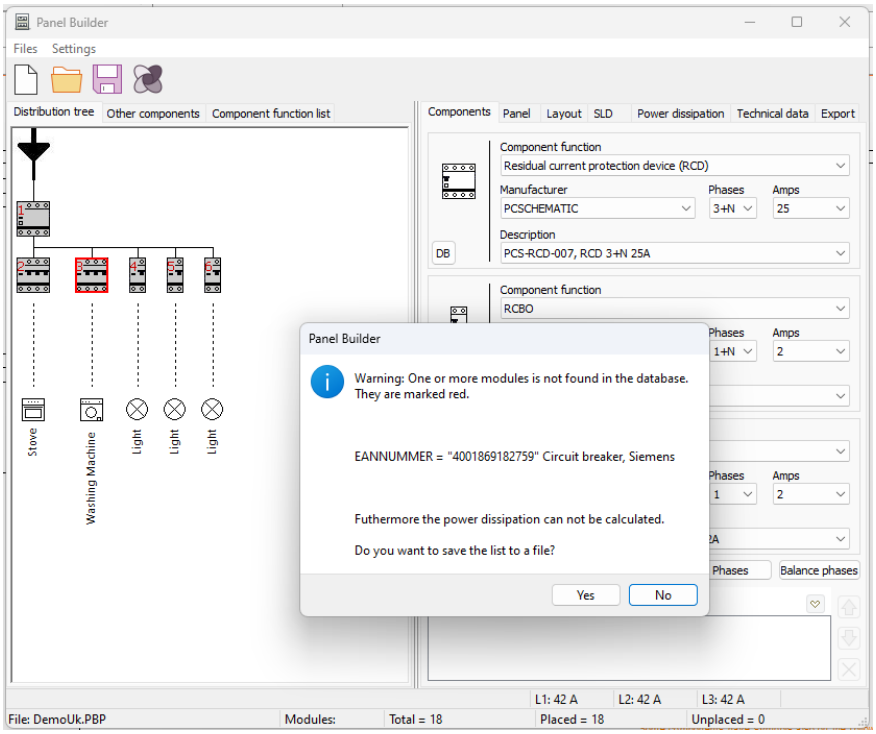
They are found and selected in the Other components tab.



## Load old Panelbuilder projects

When you load old projects, that is projects older than ver23, you can save a list with the unknown components.

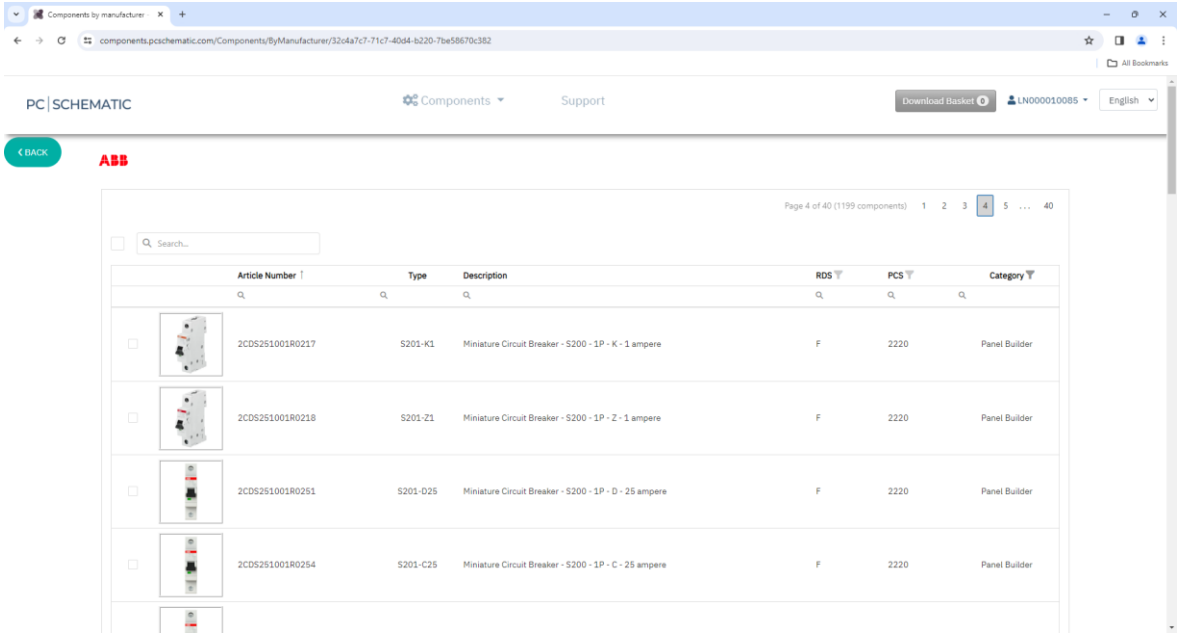
With the list, you can import the components from you old database as described in the section Upgrade projects from ver22 from page 8.




If you use 'old' components, you may need to set your article number to the 'old' article nymer (often EANNUMBER), else the lists will not be nice. Imported projects are ok, as they keep the old article number.

## Fetch new components from the Component portal

When you press the Component Portal icon in the Panelbuilder, you only search for components with Panelbuilder data.

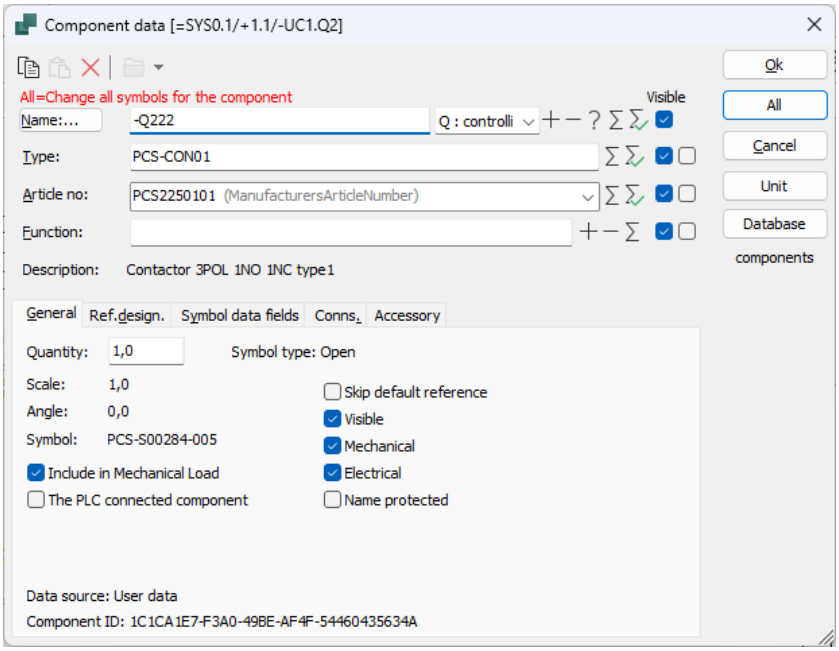


# NO MORE OK/ALL CONFUSION!

In ver24, only the All button is active, when you change something in the Component data dialog.

That means, that if you press Enter, you change whatever on *ALL* symbols in the component.

In ver23 and all previous versions, both the Ok and the All button looked active = confusion, but if you pressed Enter you only changed data for the *selected* symbol.

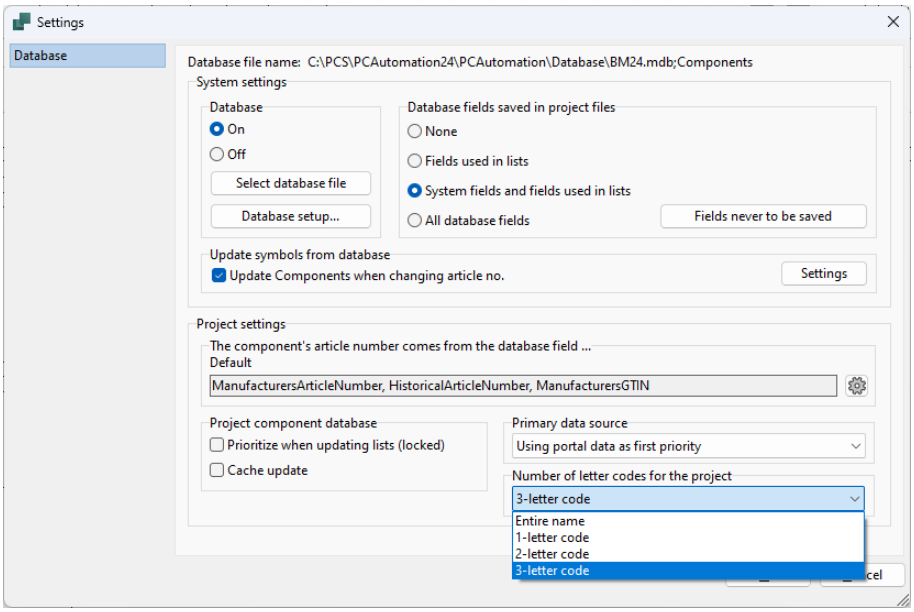


# MULTI LETTER CODES FOR COMPONENTS

You can choose to show multi letters codes on components in the project. You choose so in the Database Settings.

The letter codes are taken – by default – from the RefIdIec database field (set up in the Basis mapping tab).

In the project, you can chose to have the full name, as in the database field or you can limit it to 1, 2 or 3 letters.




The setting is valid from the the time you make it, and it doesn't change anything in the project. Finally, ou can delete or add letters to the individual component.

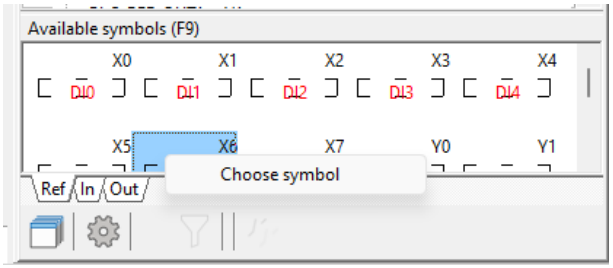
# A LITTLE MORE ABOUT PLCS IN AUTOMATION

We made a fair amount of PLC functions in ver23, mostly new symbols and also new functions that fit to the new symbols, one of which is symbol grouping of reference symbols.

## When you place a PLC

From ver24, left click on a reference symbol – *the main rule* – means that you get the symbol groups.

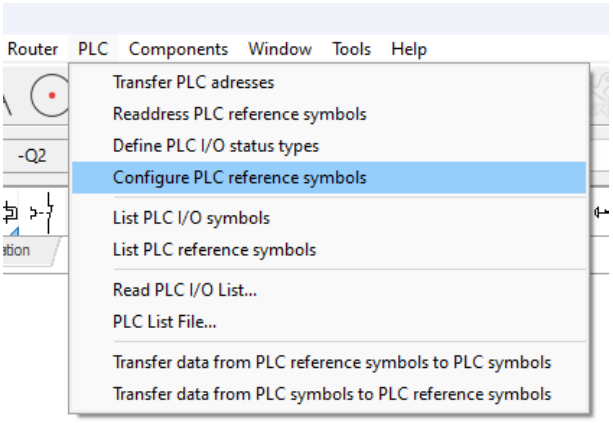
If you right click on a reference symbol – *the exception* – you select a single symbol.




## How to distribute the adresses

When you place the PLC, we often place the ref-symbol first.

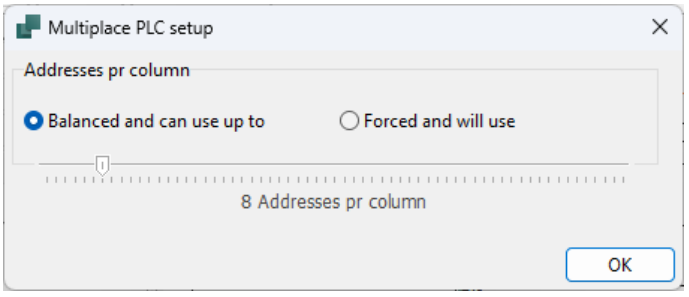
In ver24, you have a few extra option, that you find in the PLC menu.




## Balanced

In ver23 ref.symbols are placed as 'Balanced':

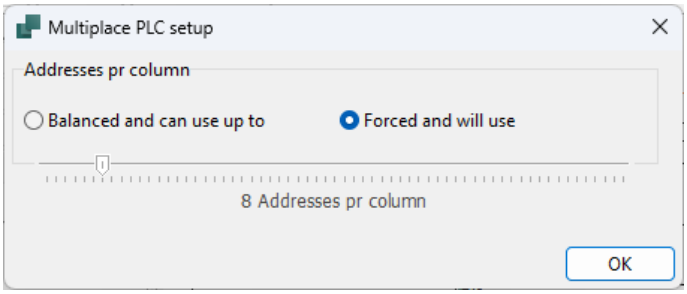
That means, that you have a max of 8 adresses pr column (our standard). If a component only has ie 2 or 4 addresses, it may look 'uneven'.



| -K7 | | | -K8 | | -K9 | | |
|---|---|---|---|---|---|---|---|
| +I1 | -I1 | 1 2 | +R1 | 1 | +R1 | +RL1 | 1 2 |
| AI | | | AI | | AI | -R1 | 3 |
| | | | | | | -RL1 | 4 |
| +I2 | -I2 | 5 6 | +R2 | 2 | +R2 | +RL2 | 5 6 |
| AI | | | AI | | AI | -R2 | 7 |
| | | | | | | -RL2 | 8 |
| | | | +R3 | 3 | | | |
| | | | AI | | | | |
| | | | +R4 | 4 | | | |
| | | | AI | | | | |
| | | | +R5 | 5 | | | |
| | | | AI | | | | |
| | | | +R6 | 6 | | | |
| | | | AI | | | | |
| | | | +R7 | 7 | | | |
| | | | AI | | | | |
| | | | +R8 | 8 | | | |
| | | | AI | | | | |

## Forced to use

In ver24, you get the option to force the ref.symbols to have a fixed size, and in this way the ref.symbols above will look like this instead:




| -K4 | | | -K5 | | -K6 | | |
|---|---|---|---|---|---|---|---|
| +I1 | -I1 | 1 2 | +R1 | 1 | +R1 | +RL1 | 1 2 |
| AI | | | AI | | AI | -R1 | 3 |
| | | | | | | -RL1 | 4 |
| +I2 | -I2 | 5 6 | +R2 | 2 | +R2 | +RL2 | 5 6 |
| AI | | | AI | | AI | -R2 | 7 |
| | | | | | | -RL2 | 8 |
| | | | +R3 | 3 | | | |
| | | | AI | | | | |
| | | | +R4 | 4 | | | |
| | | | AI | | | | |
| | | | +R5 | 5 | | | |
| | | | AI | | | | |
| | | | +R6 | 6 | | | |
| | | | AI | | | | |
| | | | +R7 | 7 | | | |
| | | | AI | | | | |
| | | | +R8 | 8 | | | |
| | | | AI | | | | |

## A little more about the new settings

### Balanced and may use up to xx addresses

The program tries to distribute the addresses in the columns that Top/Bottom allows. You adjust the number of addresses by using the slider.

### Forced to use xx addresses per columns

All columns have the number of set addresses. If there are fewer addresses in the component, than selected, the space is filled out with Empty symbols.

### If Top/Bottom symbols are missing

The addresses are distributed according to last setting.

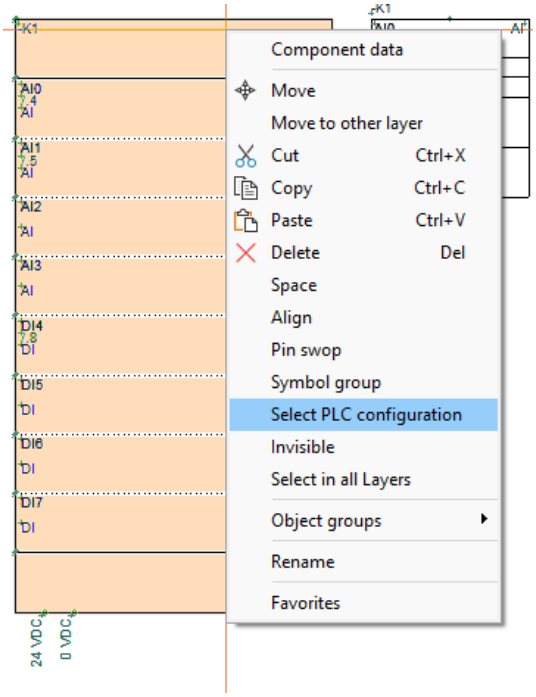
## PLC with alternatives – Change to another alternative

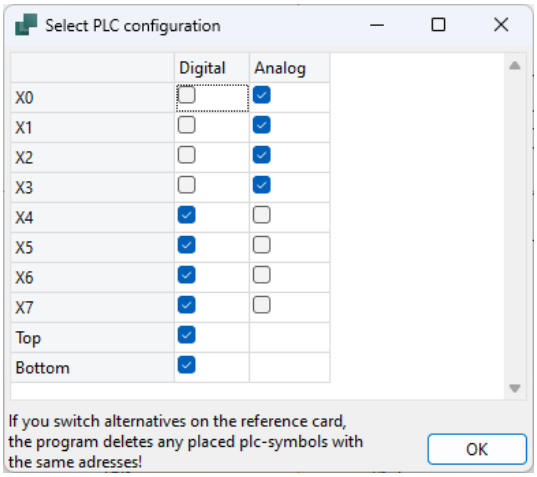
From ver23 it has been easy to create PLCs with alternative settings per address.

It has been easy to choose an alternativ, but difficult to change it later.

For that reason, we have made a new functiton in ver24, which allows you to change to another alternative.

You find the function in your right click menu, and you simply enter the same grid, as you did when you first selected. Be aware that any placed IO symbols are deleted, ifn they are changed to another alternative.

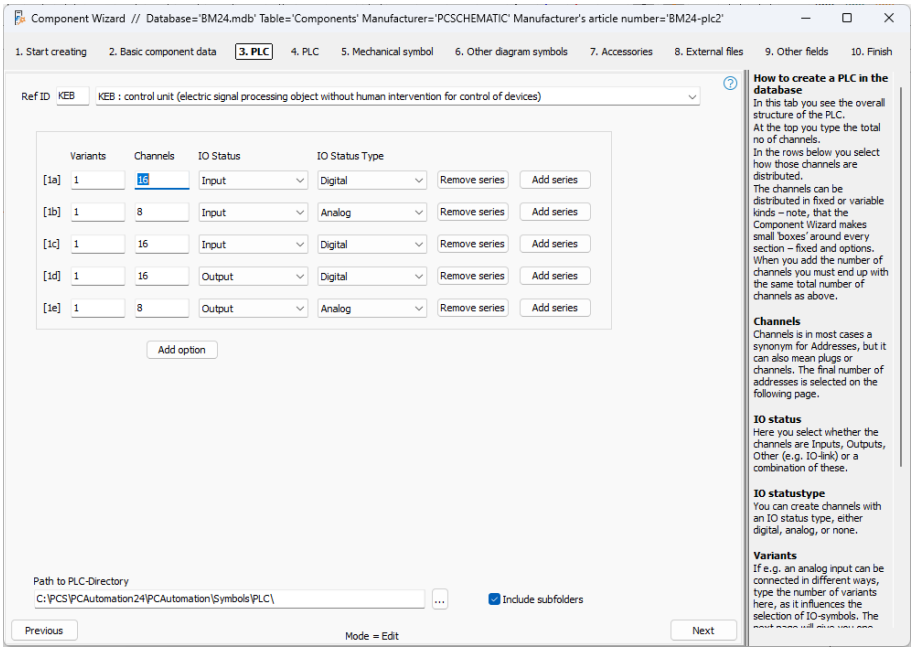


## Creating PLCs in the database

In ver23, the PLC reference symbols could only be combined in a few ways and max 32 adresses per symbol. From ver24 the user can decide more, as long as the rules described below are followed.

Start by selecting a number of channels/ addresses.




### The width of ref.symbols is defined by the Top and Bottom symbols

When you creaste a PLC, there *must* be a set of Top/Bottom symbols, that defines the width of the grouped ref.symbol:

Top/Bottom = 1 column
Top2/Bottom2= 2 columns
Top3/Bottom3= 3 columns
Top4/Bottom4= 4 columns
Top and Bottom must fit together, ie top2 fits with bottom2.

### The component is created to have more ref. symbols

When you create your PLC, you can choose to have separate ref. symbols, ie one symbol for DI, DO, AI, AO etc.

If you do so, you have to control yourself how many addresses of each type that fits to the different tops and bottoms.

**The rules for grouping of ref. symbols are**

The sequence of TOPs decides the sequence of ref. symbols. The program looks for fitting BOTTOMS.

That means:

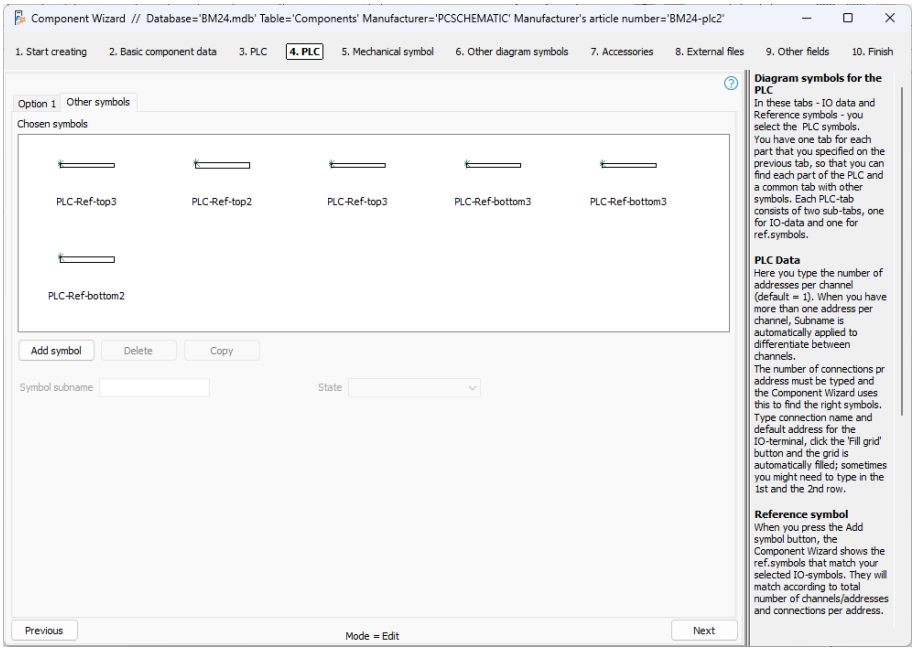
Top2 fits with – and needs – Bottom2

If the sequence is Top1; Top3; Top1; Bottom3; Bottom1; Bottom1 the result will be in the same sequence as the Top-symbols.

If the symbols dont fit, the Next button is not active.

Here we have the sequence Top3, Top2, Top3. The Bottom symbols sequence doesn't influence anything, but the symbols must be there.

The bottom symbols have connections that can be used, alternatively you may select an extra symbol for ie power supply or communication.




### Requirements to the reference symbols

If you want to create your own symbols that can be used and grouped in the same way as ours, there are some rules, that must be followed:

All symbols must have the same height. That goes for

- In
- Out
- Top
- Bottom
- Empty
- Ref.symbols for two addresses have 2 x height

The width of ref. in/out/empty must be multiplicable to the width of the different Tops and Bottoms.

For top-symbols:

- The filename must contain 'REF-TOP'
- The symbol must NOT contain connection points

For bottom-symbols:

- The filename must contain 'REF-BOTTOM'
- The symbol MUST contain at least 1 connection point
- Connection points must not have I/O status

And the symbol must – of course – be created with the right connection point settings as for all plc symbols!

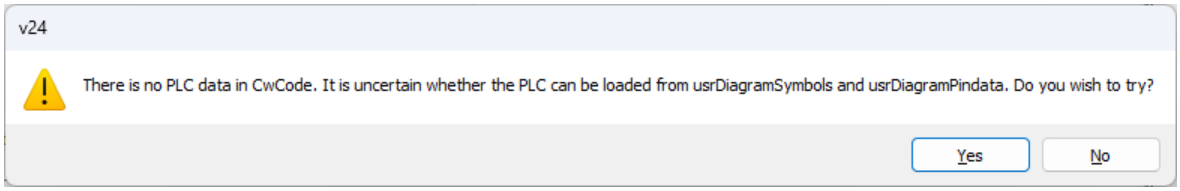
If you are unfamiliar with those, you can try to open the symbols and check their settings.

## Info from Support: PLC's and other components in the Component Portal

Almost all components on the portal have been created by using the Component Wizard. This means that a user has the same tools for creating components as we do, and thus can have the same uniform result when creating own components.

When a component is created by using the Component Wizard, there is a code in the database field cwcode. For most components, the code consists of a single letter, but for PLCs, you get a lot of information, that is meant to be used byt the CW if the component needs to be edited.

And ... unfortunately, we deleted a lot of those, when we uploaded the components to the portal. We have added a lot of those again, but you might get this messageif you try to edit/copy it:




### What should you do?

First, simply press the Yes button, and most times everything is ok.

If not, simply fetch the component once again on the Portal, and then you are 'back to start'.

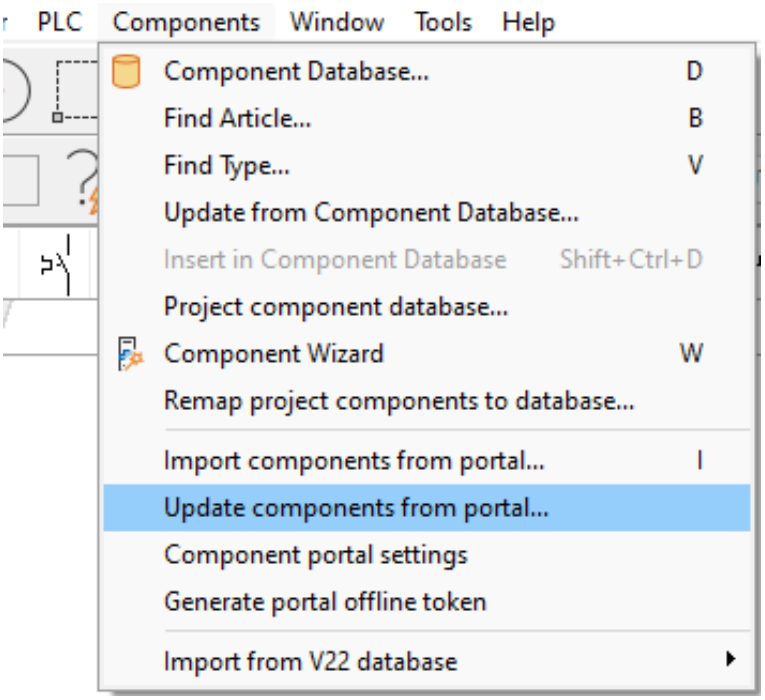
We would be happy if you let us know, soon you can send a message about this directly from the Portal.

# CHECK YOUR DATABASE COMPONENTS

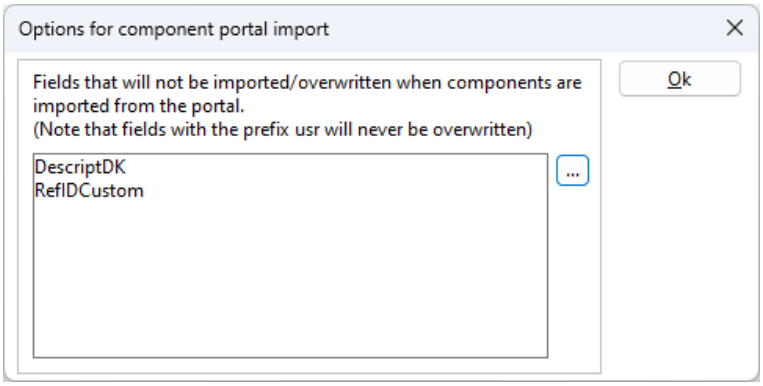
One of the purpose with the Component Portal is, that it makes it possible to fetch one component at a time.

We are doing our best when we create the components, but we do have faults and mishasps, and for that reason it should also be easy to update the downloaded components.

From ver24, you get an option in the Components menu to update your downloaded Portal components

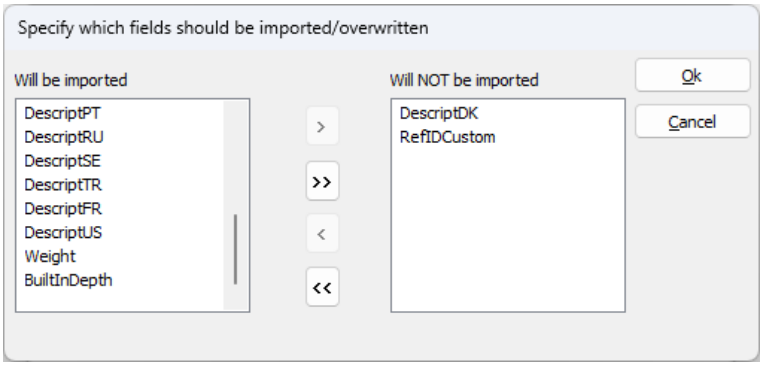



Basically, we update all pcs-data fields, among them all descriptions.




It is possible, thought, to deselect update of selected fields; maybe you want to maintain the German description yourself.

In the picture, the RefIdCustom is also deselected, as you may use it for your own reference letters.



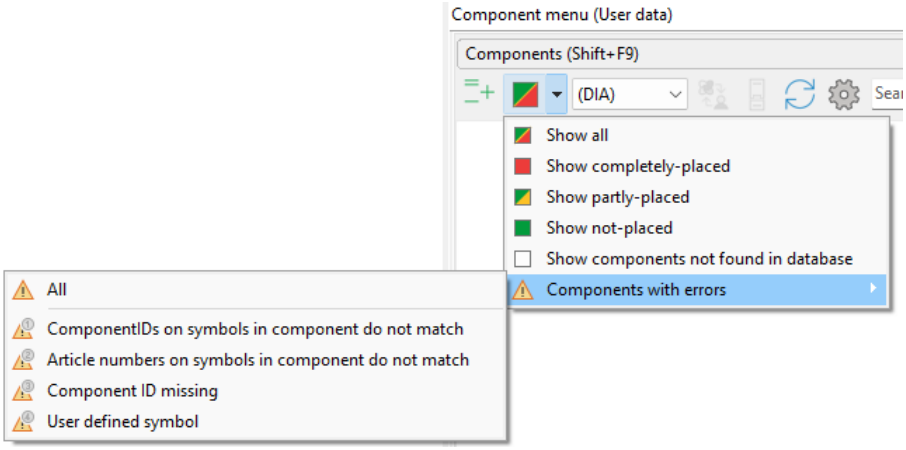

REMEMBER

We don't change any of your fields. Not the usr-fields, and not the fields created by you.

# THE COMPONENT MENU – ERRORS ARE GROUPED

The Component menu is linked to the database, and it also shows when the database and the project contains different data about the components.

In the last version, we showed that some components had errors, in this version we can show the different error types individually.




1. ComponentIDs on symbols in the component do not match:
    a. Article numbers on the symbols are the same, but …
    b. If you have fetched the component from the portal AND you have imported it from your old database, too, this can occur.
    c. We know – and can control – the components that we had created for the old database, but not the components created by customers, is Rockwell PLCs.
    d. Solution: All symbols in the same component MUST have the same ComponenID. Use the Object Lister to find them and then replace them!

2. Article numbers on symbols in component do not match
    a. Here you have errors in Component grouping
    b. Different article numbers are grouped to one component, typiocally the main component and the auxilliary component has been grouped together as one component. It also creates errors in the parts list.
    c. Solution: In Component Grouping (F7) you give each component its own component group.

3. Component ID missing
    a. Components are not in the database. Same as the ‘white box’.

4. User defined symbol
    a. The component has symbols from both sets; pcs and usr. If the symbols are the same, we need to clean it up – soon!
    b. If you have added ’user defined symbols’ that are unknown to the component in the database, you can also see it here.

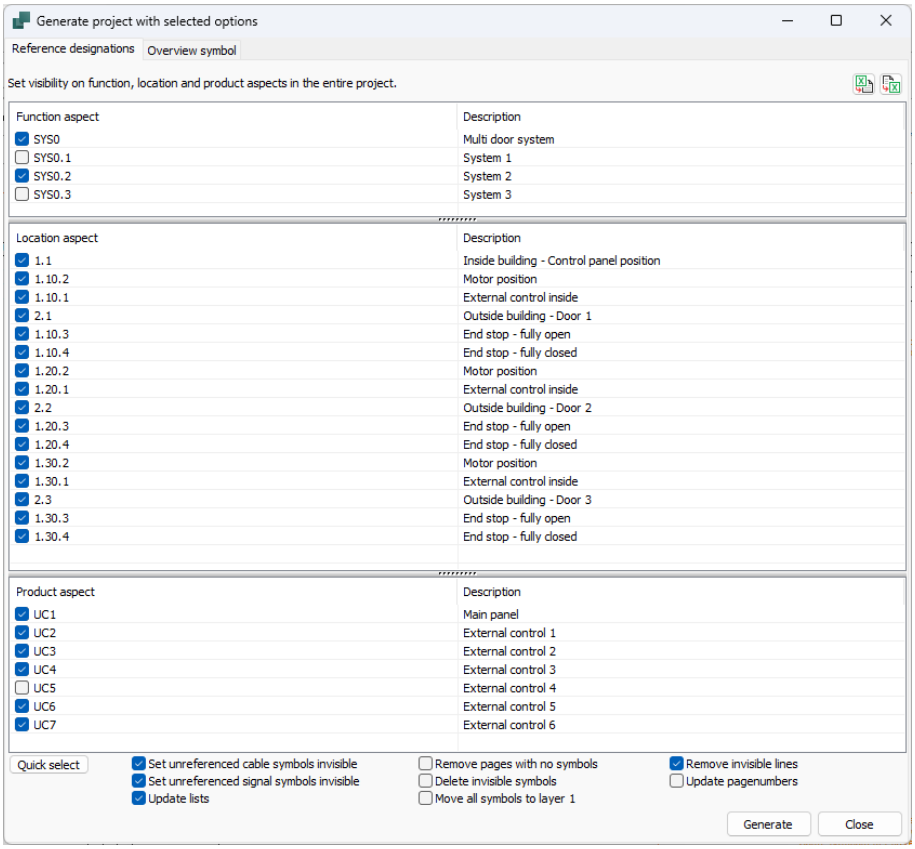
# THE ‘GENERATE PROJECT FROM OPTIONS’ MODULE

We have made a few extra functions in the Options module, that you can read more about below.

The module makes it possible to generate a project based on deselecting options, that are defined by means of reference designations.

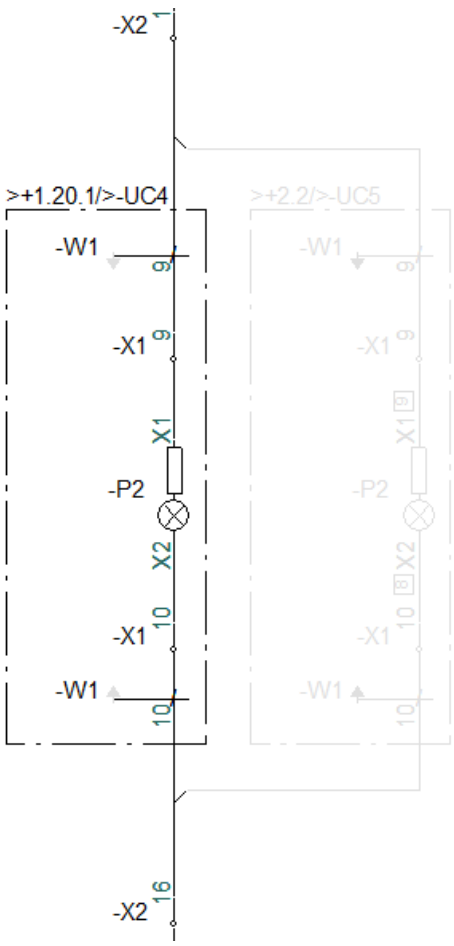
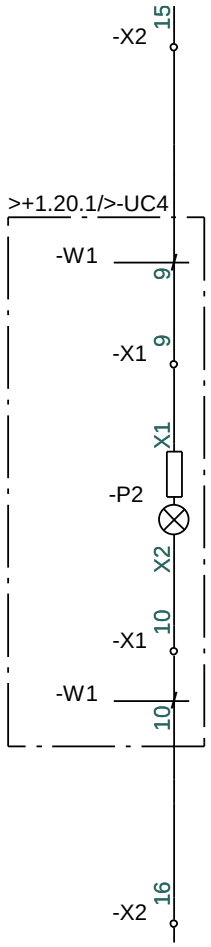
Beside, you can see a picture from our pcsMotordemo3, that illustrates that also small projects can contain a lot of (as in too many) rds-codes.

At the bottom of the window, you see the various options for tidying up after generating a new project.




The new functions are:

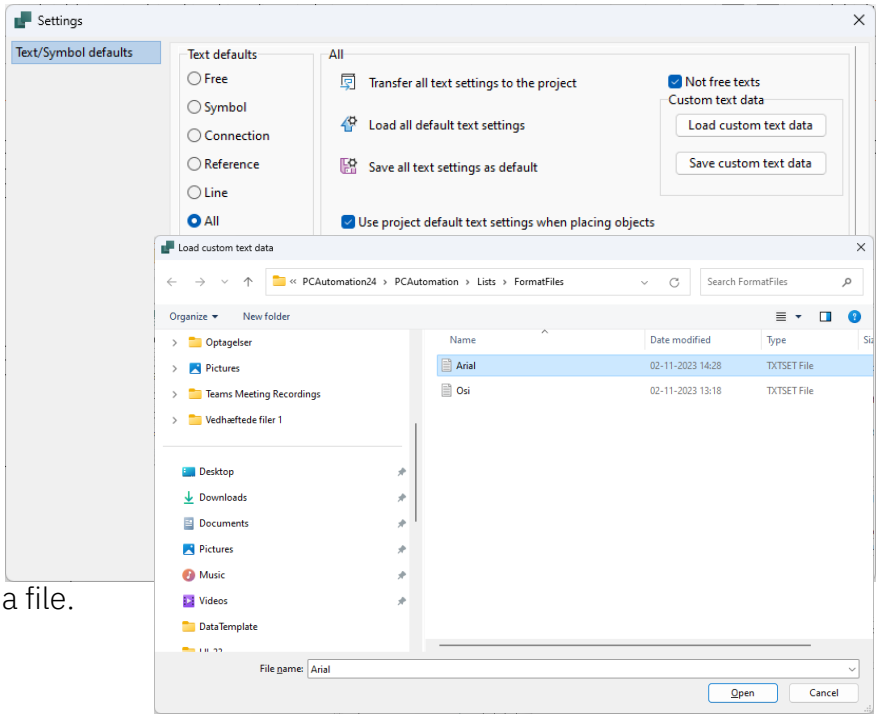
- The option to delete invisible line, in the diagrams and on the mechanical pages
- ’Left-overs’ of mounting correct bends are convertes to straight or angled lines
- Page number may be updated automatically as part of the generation
- Dividers may be deleted as part of the generation, also when they have a rds-code

| | |
|---|---|
|  |  |
| I ver23 you sees 'left-overs' of mounting correct bends and invisible lines | In ver24, there are no 'left-overs' and no invisible lines |

| | |
|---|---|
| Unreferenced cable symbols<br>(unreferenced cable conductors) | You can set reference between conductors in a cable. If it is active on the individual conductor, but without reference, the cable symbol and its conductors become invisible.<br>Refenced cable conductors are used when you want to show 'both ends' of a cable: to let the program understand that it is the same conductor and not new one, you make references between the two sets of the same conductor. |
| Unreferenced signal symbols | Signals, that don't continue to another page. |
| Delete invisible lines/symbols/pages | When you deselect an option, it is made invisible. The invisible objects, including full pages, can be deleted. |
| Update page numbers | You get holes in page numbers after deleting options, therefore you can choose to update the page numbers. |

# IMPORT SETTINGS

## Text/Symbol defaults



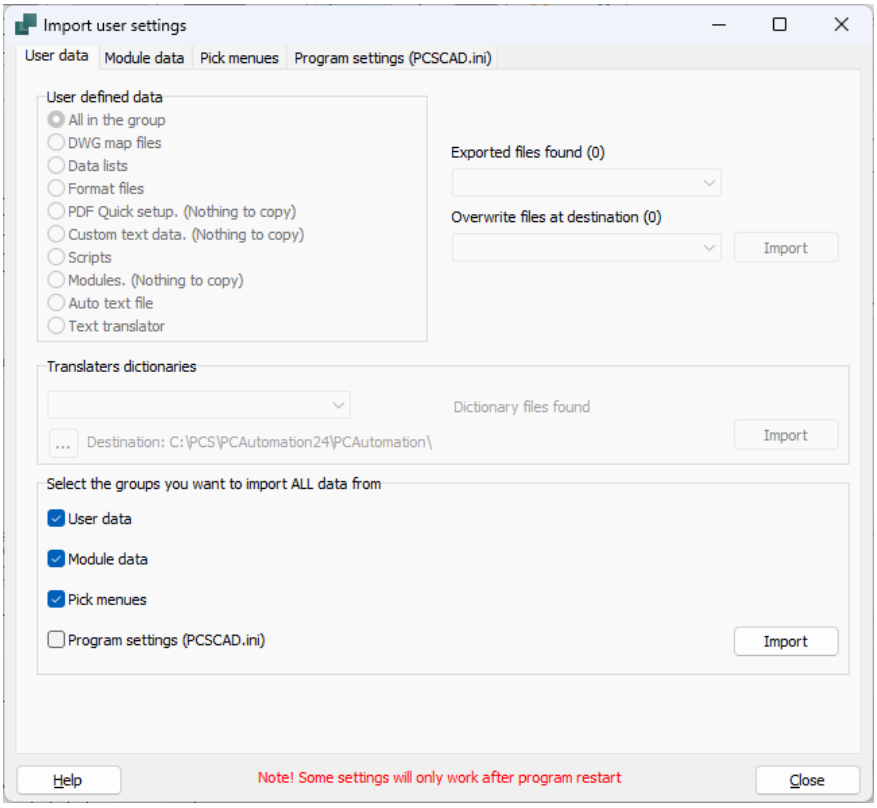

You can load a set of text/Symbol defaults that changes all texts in a project into another font and color.

The function has been available in the last couple of version, but it is easier from ver24.

You can load a file and save your project's settings into a file.

## Import user settings

In 'the old days', we saw that many customers copied their pcscad.ini-files to each other, because they thought they contained all settings.




It didn't and it doesn't!

Instead we have made the function Import/Export user settings in the System/License settings,

This function makes it possible to import different settings from another user, and from ver24, it is also possible to import 'all of Peter's settings'.

The first tab has been changed: We have maintained the different sections, allowing a flexible import, and the option to import everything in one operation.

# USER DEFINED SYMBOL DATAFIELDS

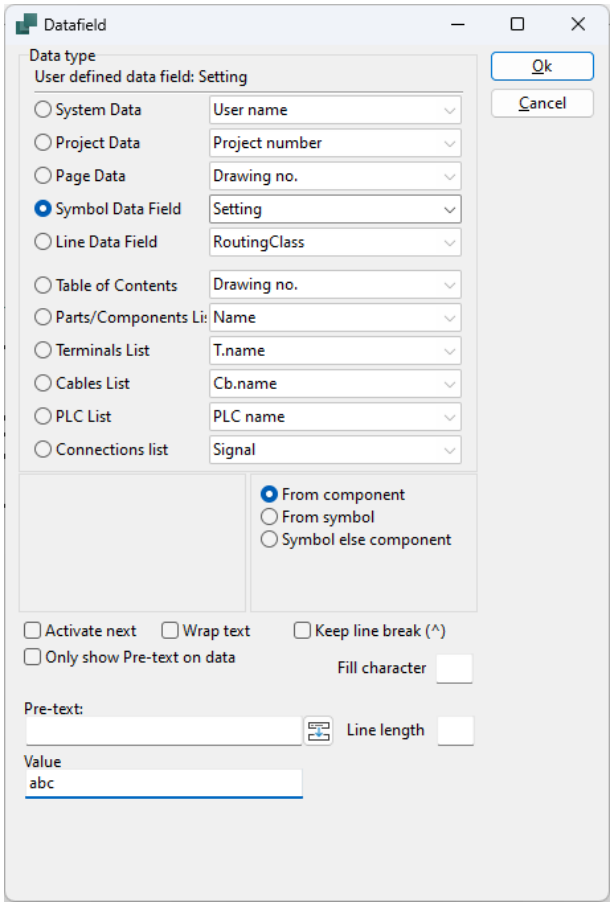
As a user, you can create your own symbol datafields that allow you to store different data in the project.

When you create them, you can – for many years – create a list of values for the field. The list is general and can be used everywhere you use the datafield.

There has been a wish to have default values that were related to a specific symbol and not to the datafield itself: As an example, you could have different settings depending on sensor type: temperature, pressure, humidity etc.

Having a list ensures that you use the same units/sizes everywhere – and with correct spelling

If you have a list already, this new list will override the old list.

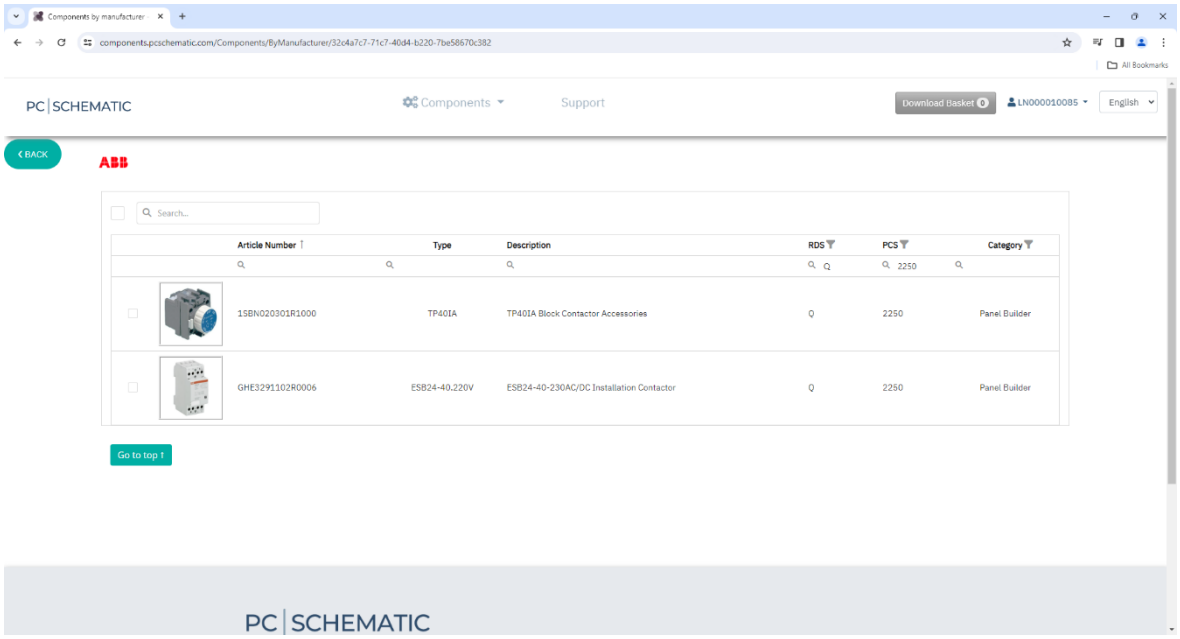


# NEWS ON THE COMPONENT PORTAL

New components and manufacturers are continuously added to the portal. And most of them come on the basis of requests from customers. Creation of components for the Portal is contained in the Maintenance Agreement.

We are continuously working on improving the user experience, and there are more things going on and more on the way.

- Pressing the icon in the program now opens All Manufacturers window
    - If you press the icon in the Panelbuilder, you only see components with Panelbuilderdata
- When you enter a specific manufacturer, you can search for RDS-code and DB-code: here ABB, Q, 2250 and Panelbuilder.




- Soon, it becomes possible to request new components directly on the portal
    - We make a form in which you can type Manufacturer, Type/Article numer, Description and link to a datasheet
- Iin the same way, we make a form in which you can report errors on individual components
- The window will show a fixed maximum of components per page
- You can select all components on the page with one click and add to the basket
- The basket can contain the same amount of components, as you see on the page

## From 2024: Access to the Component Portal requires a valid MA

If you meet this message on the Component Portal, it simply means that you don’t have access to it.

Access to the Component Portal has been free, since the launch more than two years ago. From 2024, we require that you have a valid Maintenance Agreement to access the Portal.

# VARIOUS SMALL FUNCTIONS ...

## More fields in the Component Database

Our pcsComponents database contains extra fields, or rather three of the previously invisible pcssys fields have become visible:

- One for usrPBData (pcsSys6)
- One for the component's width (X) (pcsSys1)
- One for the component's height (Y) (pcsSys2)

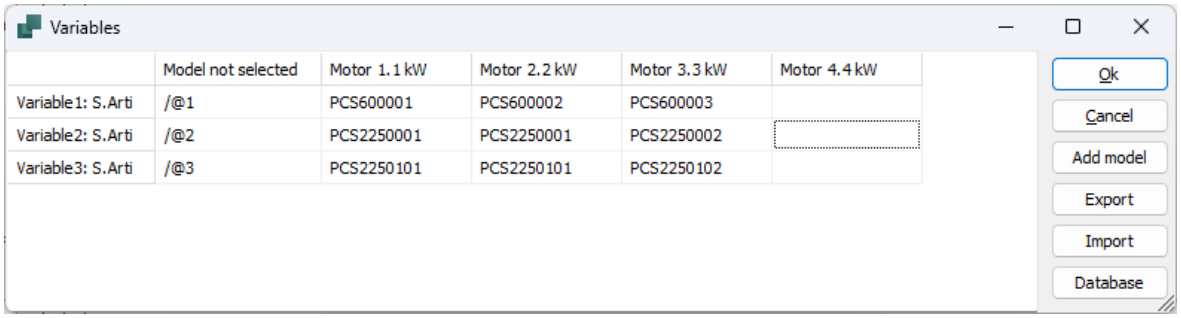
The depth (Z) is already in the BuildInDepth.

All measurements are in meter!!

Measurements can be used for export to label printers. When you export a label for ie a terminal row, it is important to include the width of separators.

## Subdrawing – models

The database button is no longer hidden: when you create models, you want to add article numbers on components. Earliner, you could only open the database with a right-clik; now you have a button. It becomes active, when you are in a type or article number field:




## More manufacturers are added

When you update to ver24, you get a message about (a lot of) new manufacturers. The manufacturer table is updated automatically with new manufacturers, if/when you get their components from the Portal.

## The Automation Service Program

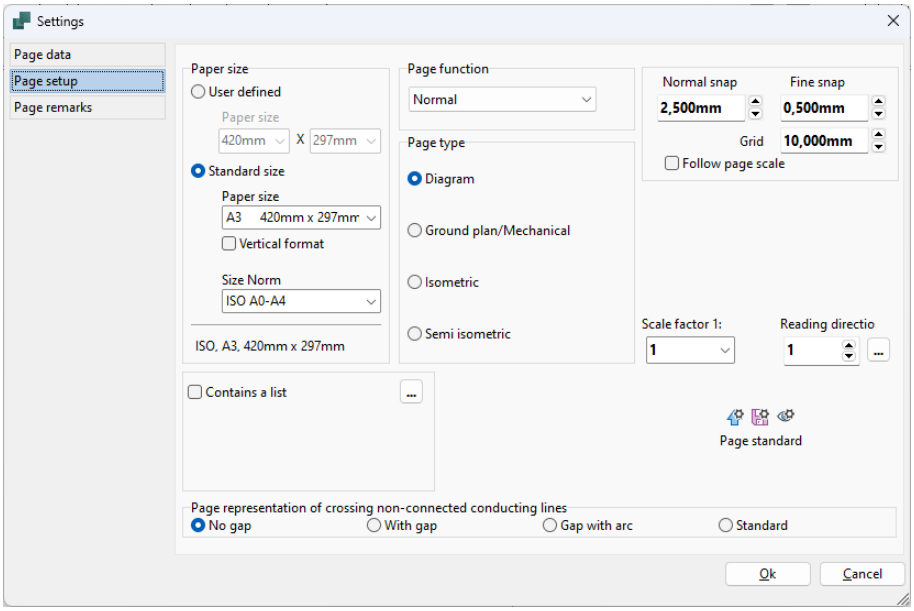
When you open the database record for a component, you see the same datafields as the Automation user. This means that the show/hide option follows the database and not the program.

## Page setup

We have moved som page setting to *page setup* – previously they were valid for all pages of the same kind, now they are only valid for the selected page(s).

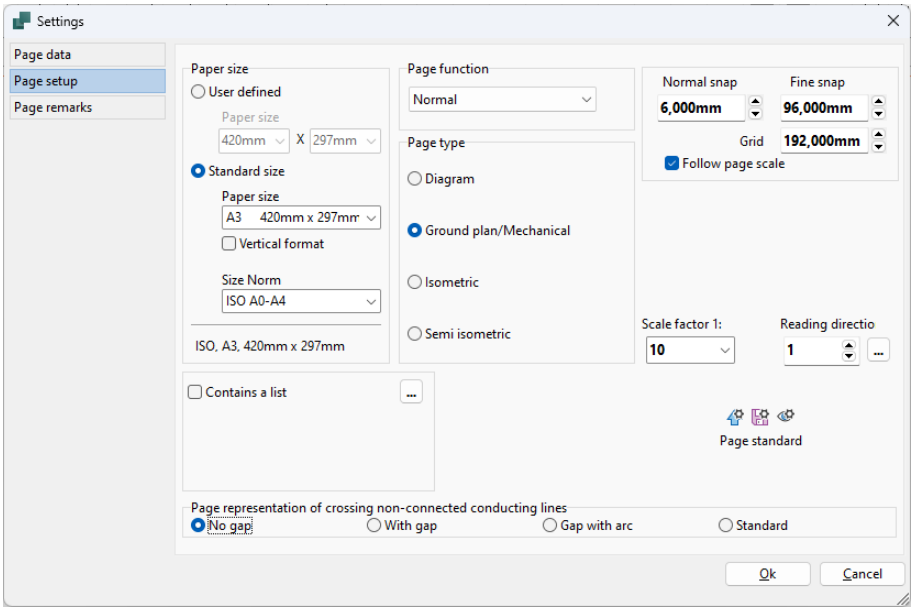
## Crossing lines

The option to show crossing lines with gap or arc, has up to now been a project setting, valid for all pages. From ver24, you can change the project default (from Pointer/ Screen) for each page, also for a template.




## Page snap and grid

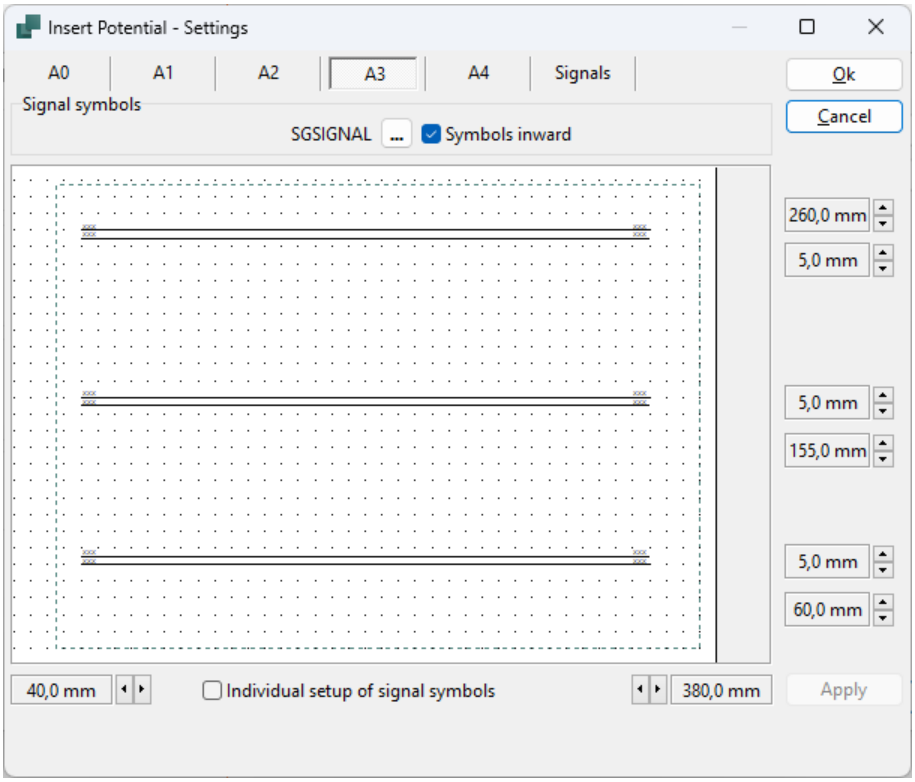
Page snap and grid is also a page setting, so that you can have different settings in the same project, ie different module sizes for different panel types.



## Insert Potentials – with middle position

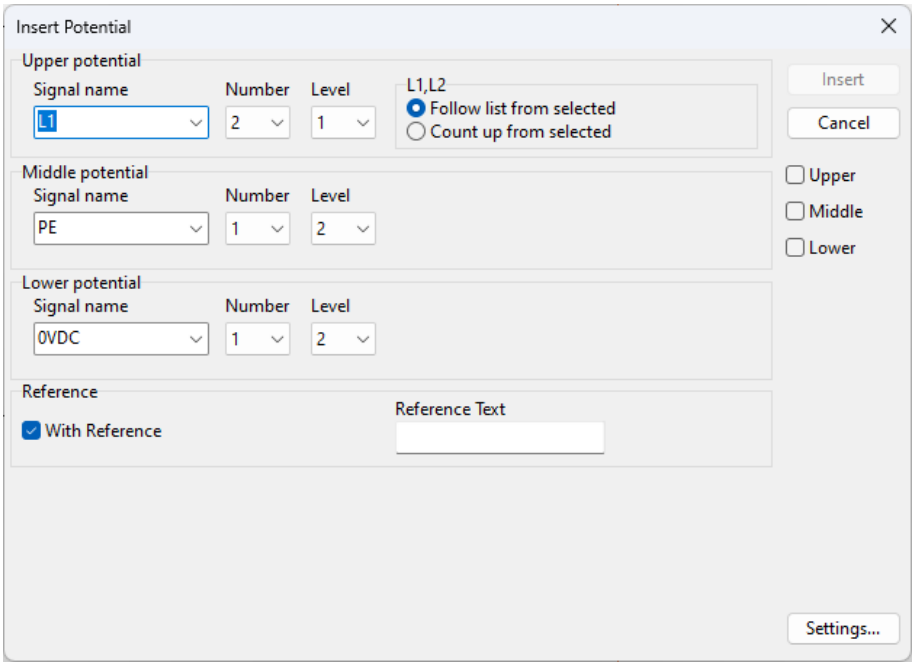
The function has been expanded to include a fixed middle position.

Its setup is done similarly to the the top and bottom setups: Setups for the different page sizes and signal names.




You also insert the potentials in the same way.

If you have a list, you see the list values before you place them, exactly like before.

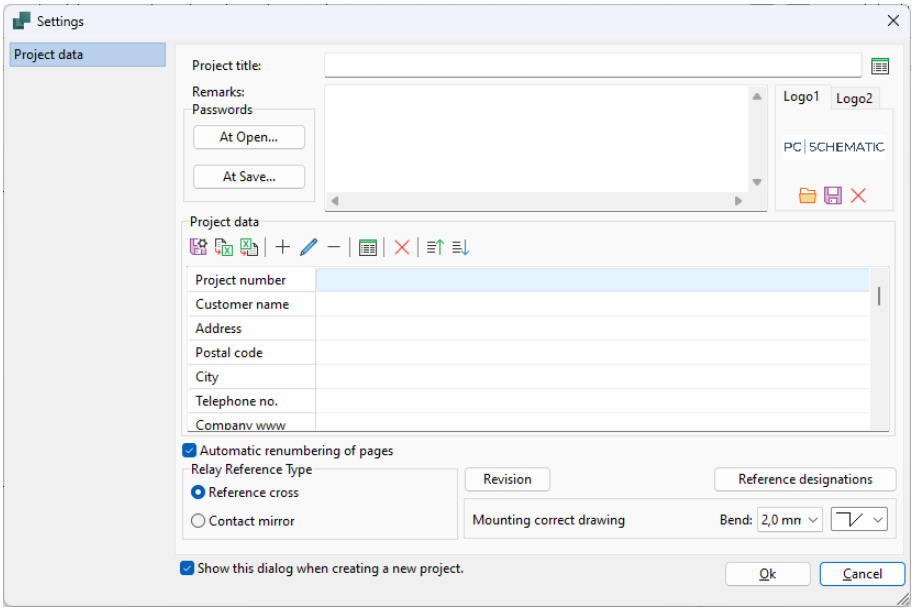


## Automatic renumbering of pages

When you copy a page, the inserted page is numbered as DIA(xx) – in all previous versions. Also when you have selected 'Automatic renumbering ...' in Project data.

When you copy and paste a single page, it automatically gets the next page number, if selected.

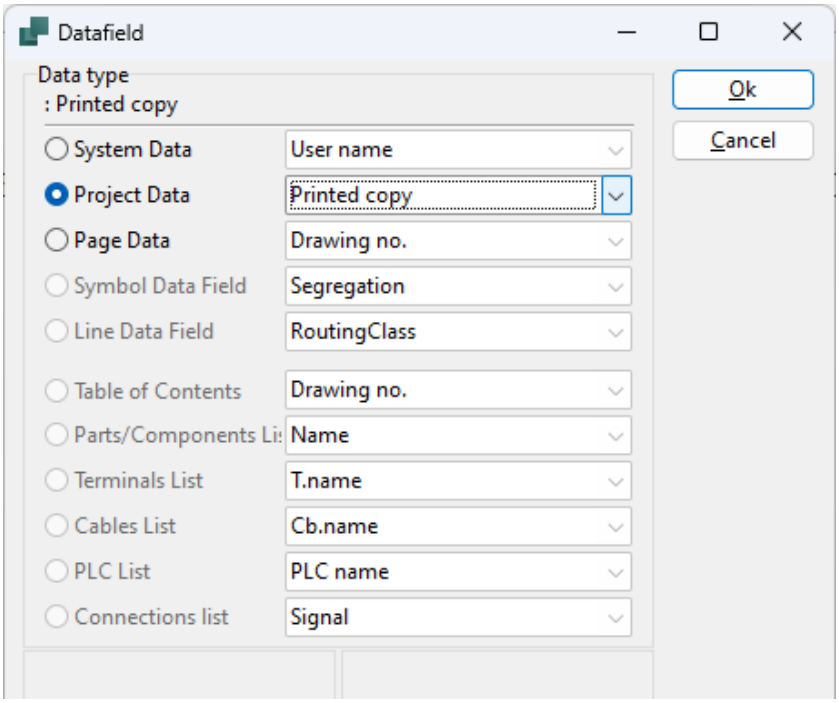
Accordingly, (mostly) list pages are also renumbered if updating the list means that it has fewer pages.




## 'Printed copy' on prints

We have created a new project datafield that can be inserted in drawing header or where you want to see which Printed copy you have.

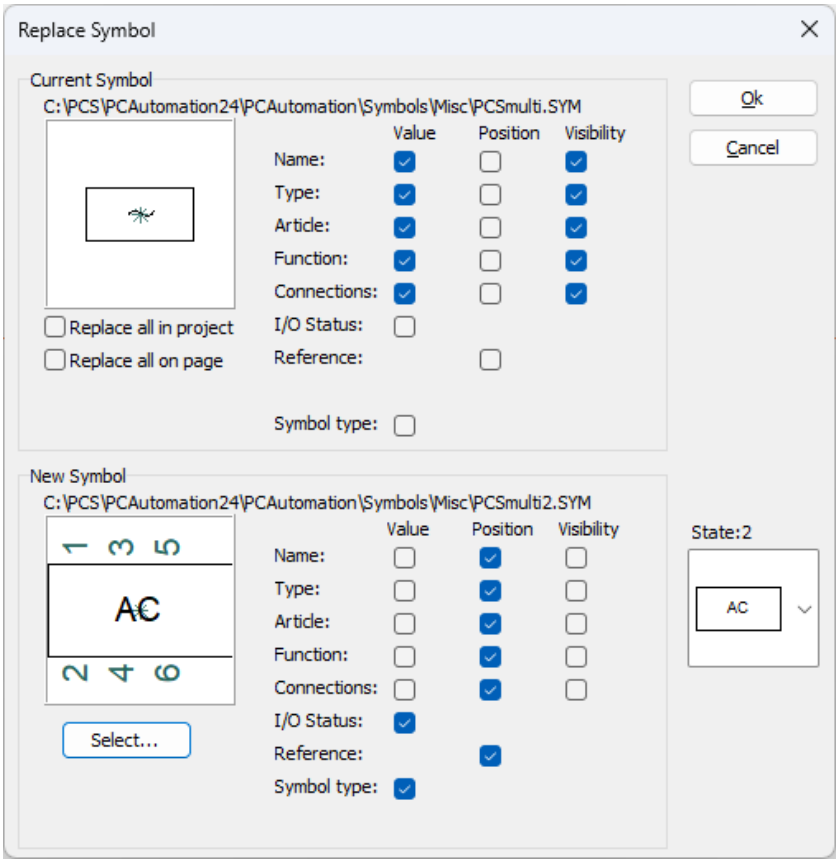
It is a supplementary to Printed date.



## Replace symbol

When you replace a symbol with states with another synbol with states, you can now keep the state.

The state is defined as a number, eg state 2. The description (AC) has no influence on the replace function.




## More columns in Object Lister

On the Pages tab, we have added the 'In Table of Contents' check.

On the Symbols tab, we have added the 'Full reference designation' check.

## List setup

All lists are in columns – some lists only have one column ...

## Copy line with name

Lines with names can be renumbered, and a line's new name follows the existing, ie Line001 -> Line002, Line101 -> Line102.

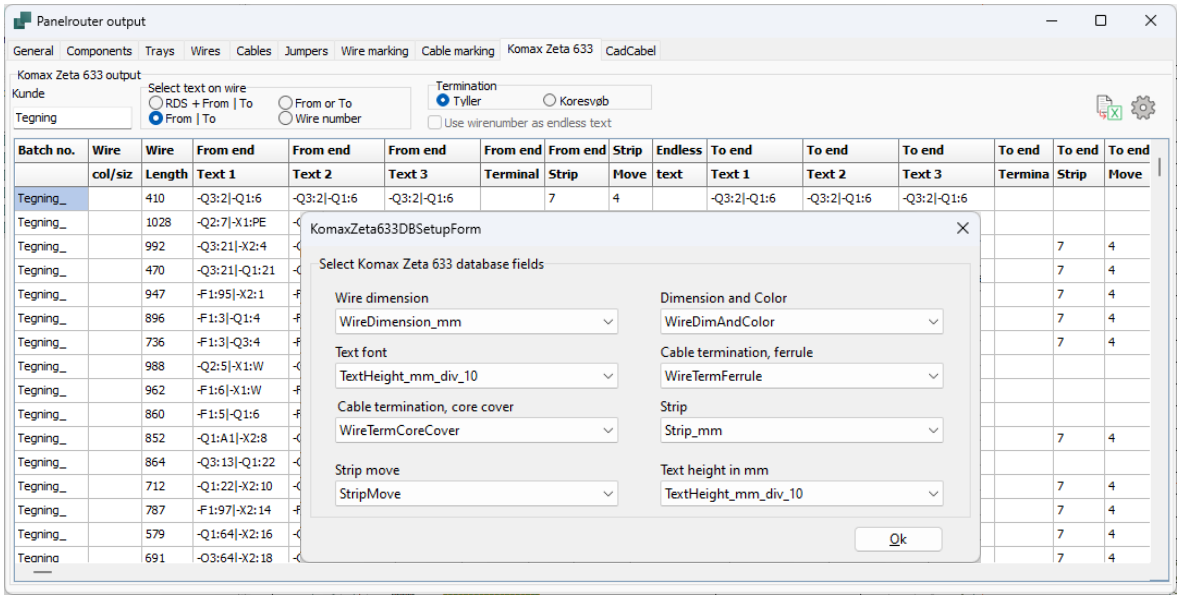
## The Pickmenu

Lines in the Pickmenu can also contain RoutingClass.

## Panelrouter: Output to wire cutting machines

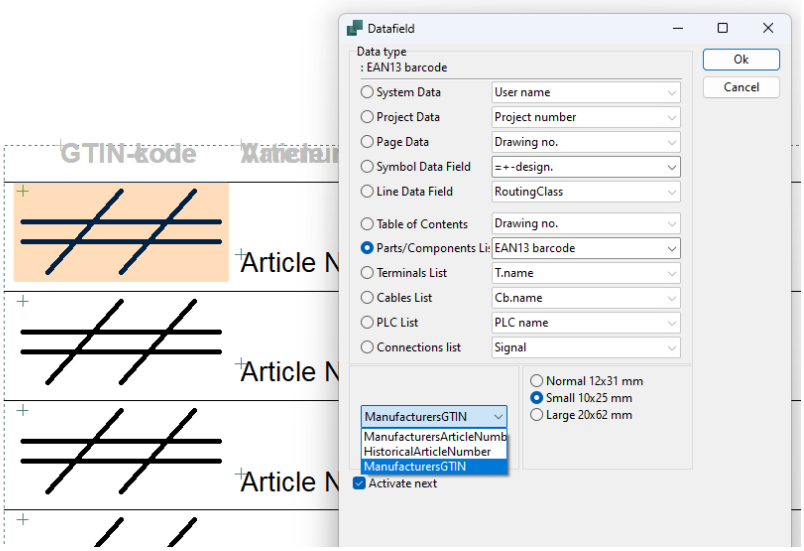
On the Komax Zeta 633 tab (a customer's Komax), the setup is changed to look into the database; previously it looked for settings in an Excel file.

The same method is used for – and has always been – for the export to the CadCable format.



## Barcodes on parts and component lists

It is possible to have a barcode in a parts list, and up to and including ver23, it has been connected to the article number used in the component's symbols. This was – and is – ok, as long as the article number is a barcode.




In 'the old days' we used the eannumber (now gtin number) as the primary article number, but today we use the Manufacturer's Article number.

From ver24, you can select which one of the component's article numbers, you want to use here. You can use the article numbers that are defined in the program.

We have chosen to – in the pcsParts3 header - map it to ManufacturersGTIN, but you can choose differently.

If you want to use another field, you should make your own symbol:

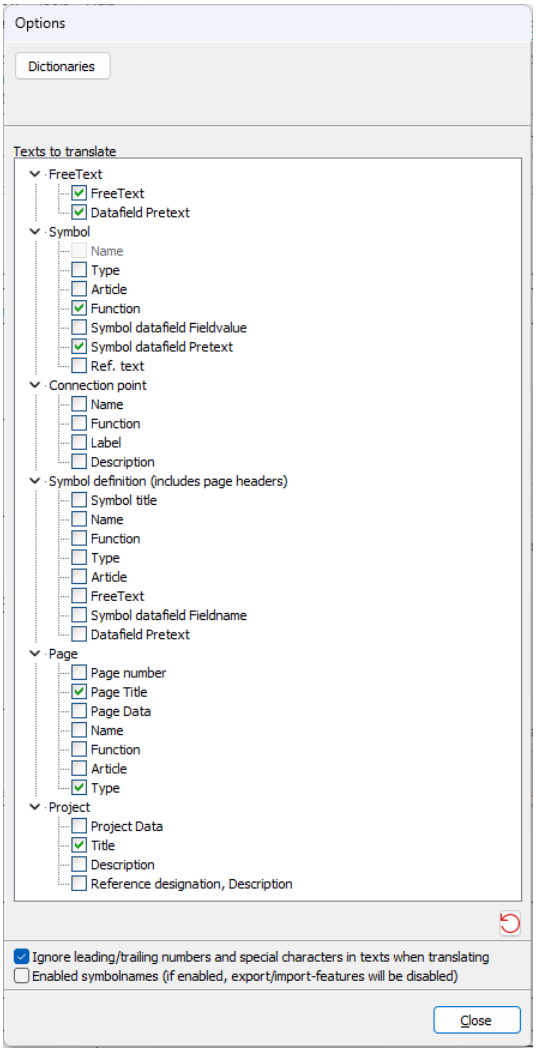
Open the (easiest method) pcsParts3 in the symbol editor. Select all barcodes, and change the mapping to the desired barcode. The barcode field must contain 13 digits to work. Save the symbol with a new name.

## Default setting of Text Translator is changed

The default setting of the Text Translator is changed (only new installations), so that only relevant texts are translated.

You can still change the settings, if you prefer.

Remarks to the new settings:




### FreeText

Everything is translated

### Symbol

Only function and pretext are translated.

### Connection point

Nothing is translated here,

Function, label and description are (mainly) used for plc's and here you would mostly update the IO-list ot translate.

### Symbol definition

Everything here is deselected, as you can change state (language) on (our) page headers.

### Page

We translate the page title

### Project

The project title is translated.

Ref.designations and descriptions are (may be) translated by importing/updating a list in the rds-function.

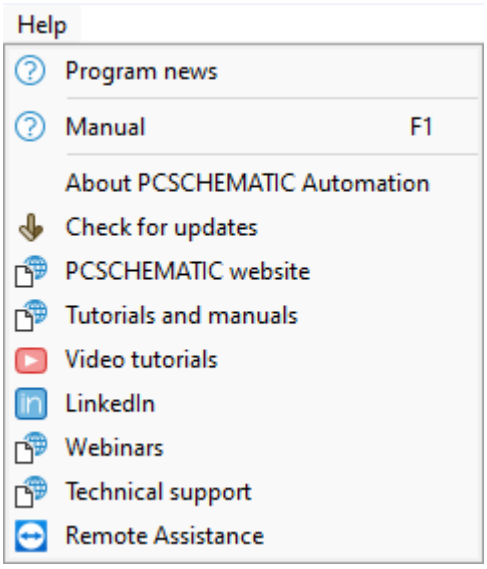
Numbers and special characters are ignored.

If you have numbers in side a text, you can setup a variable for those texts.

# ALL MANUALS ARE ONLINE

And you can see this in the Help menu in the program.

You find the news document in the menu, and links to different parts of our website.




When you press F1, you open a pdf-document with links to the part of our website with

- Changelog
- News
- Manuals

Depending on the program language, you go to different websites – a national website (if any) and PCSCHEMATICs uk website.

We update our manuals, but for the latest versions they haven't been included in the program itself, unfortunately.

From now, we update the manuals and upload them to the website.

## MY NOTES