

NEWS IN AUTOMATION – VERSION 25



This document describes the news in PCSHEMATIC Automation version 25.
Some of the news were released in ver 24.0.4 and these are marked with a * in the document.

Last edit March 2025

INDHOLD

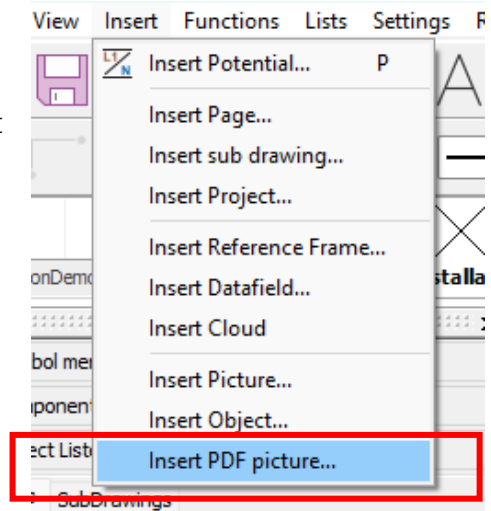
INSERT PDF IN THE PROJECT	5
House and alarm installations	7
DATABASE AND THE COMPONENT PORTAL – NEWS AND IMPROVEMENTS	9
Easier search for components	9
Component search – search locally and on the Portal	10
Search for a component	10
Get a component	11
How can I import components into my local database? *	12
Download the project components ...*	13
News on the Component Portal *	14
Component requests	15
Report component error *	16
Download components via file *	17
LOCKED OR UNLOCKED PROJECT DATABASE?	18
A LITTLE ABOUT SYMBOLS	19
The most requested news *	19
The component has accessories	19
Prioritized article numbers	21
Do not include in Mechanical load	22
Symbol settings – Symbol editor	24
More symbols for PLCs, switches etc *	26
New symbols for terminals – with jumpers *	27
COMPONENT WIZARD *	28
Automatic numbering of connection points *	28
Enable all/none *	28
The Component Wizard automatically jumps to tabs with errors/missing info *	29
The Component Wizard always searches in symbol folders and not in project *	29
CREATE A ZIP-FILE *	30
COPYING WITH REFERENCE DESIGNATIONS *	31
EXPORT TO PXC WIRE ASSIST FROM THE PANELROUTER *	32

INSERT PDF IN THE PROJECT

We have improved the function that inserts PDF documents in our projects.

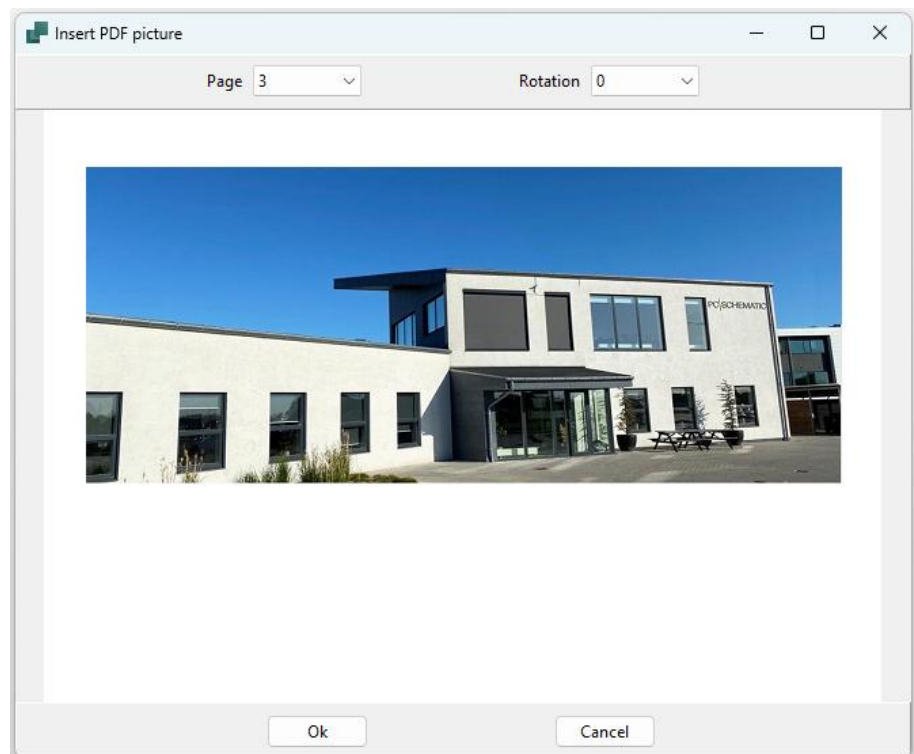
Up to an including ver24, we can insert pdf-pages by using the Insert object function. It only inserts one-page documents, and it is not possible to PDF the final document, because the resolution is not good enough.

For those reasons, we have created a new function, which allows you to insert the PDF as a picture, thus maintaining a good resolution.



The function works in this way:

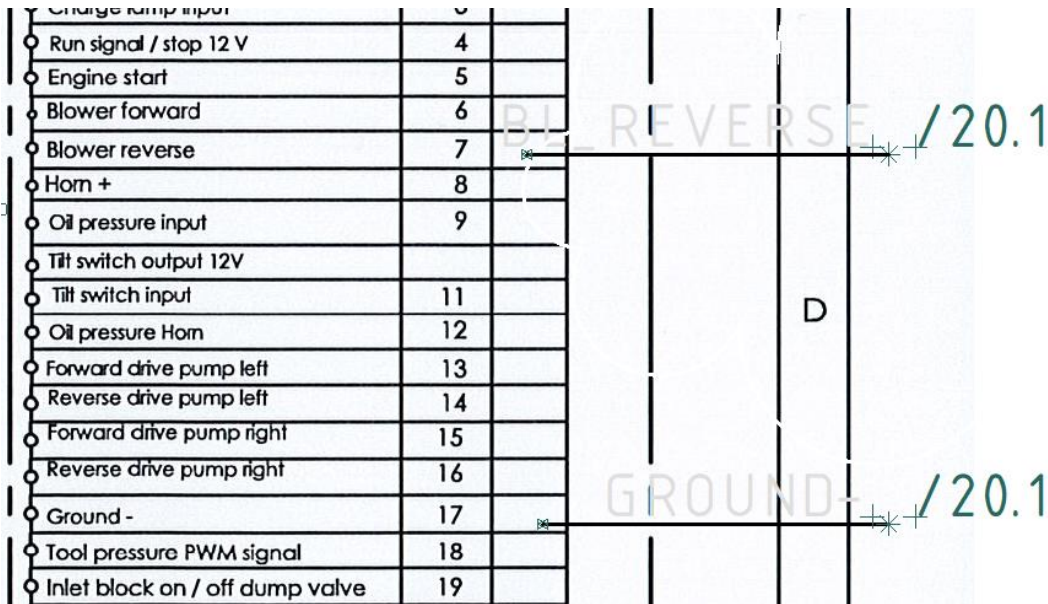
1. Go to an empty page with scale 1:1
2. Go to Insert|Insert PDF picture ...
3. Select your pdf-file
4. Now you can choose between the pages in the document – in 'the old days' you could only insert one page! It is also possible to rotate the page by 90°, 180° or 270°.



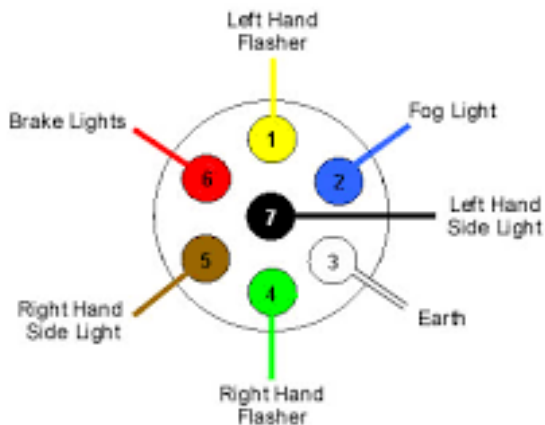
PC | SCHEMATIC

In this way it is easy to insert pages in your documents, either diagram (parts), illustrations or floorplans.

If it is a diagram, you can insert the PDF on the page and have cross references to other pages as shown below. An you can read the inserted part.



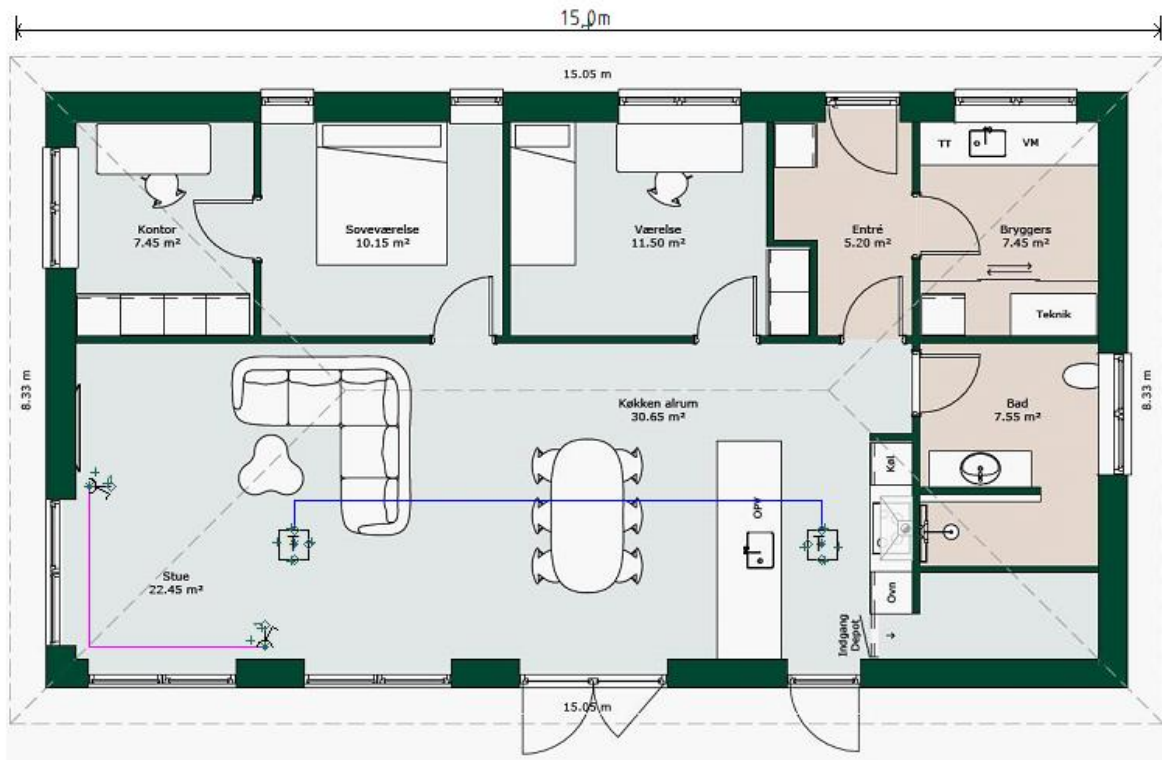
If it is an illustration from ie a datasheet, you can also insert this with high resolution.



House and alarm installations

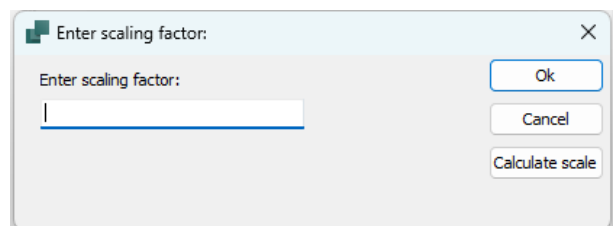
The new import PDF function will improve how we create projects with various house and alarm installations.

We will make an updated demo-file which shows the new functions.



Here you see a small example with a family home. We have placed two power outlets and two detectors.

1. Go to an empty page or insert an empty page into the project. If you want to make a floorplan like here, use a GRP-page type.
2. Scale must be 1:1.
3. If you need to use the page for measurements, ie cable lengths, you need to have the page in the right scale:
 - a. Select the inserted Area (use the Area command)
 - b. Go to Edit|Scale.
 - c. Click the Calculate scale
 - d. You are asked to find two points – preferably as far from each other as possible – and enter the distance into the window, ie 5.7m
 - e. Now the program calculates the scale, and the inserted Area is placed again.
4. When the drawing and the scale are correct, you can place power outlets, detectors and cables. Here you can see a couple of detectors and some power outlets.



Because power outlets, detectors and cables are inserted on a fixed-size page (scaled), parts and component lists can also be created so that they contain the correct lengths.

Here, a component list where pcsComponents3 is used.

Component	Qty	Total qty	Article
-D1	1	2	thermo detector
-D2	1	2	thermo detector
-X1	1	2	Power outlet
-X2	1	2	Power outlet
w1	4.99	4.99	pink
w2	7.70	7.70	blue

DATABASE AND THE COMPONENT PORTAL – NEWS AND IMPROVEMENTS

We have had our Component Portal since ver23, and for this version we have made some improvements. In particular, we have improved the search options; a little along the way, and a big one in ver25.

We have also worked on using the information from the Portal and the database actively to ensure better quality in the projects.

Some of the improvements came in the latest release of ver24 – they are marked with * in the following, and some will be released in ver25.

Easier search for components

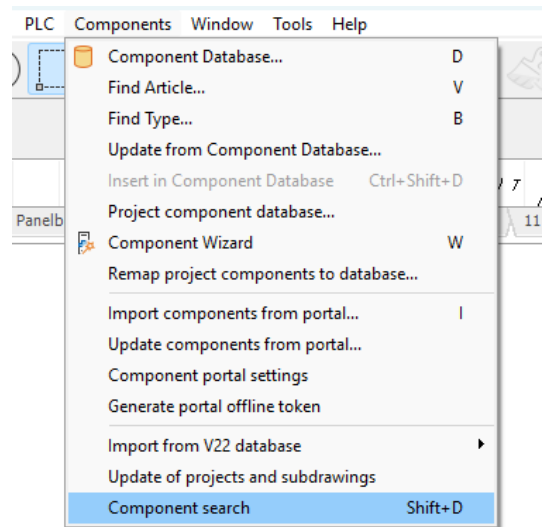
One of the big requests has been that it should be easier to find a component on the Portal.

We have created a new search function, which is located within Automation, and which solves most of customers' wishes for search and other functions on the Portal.

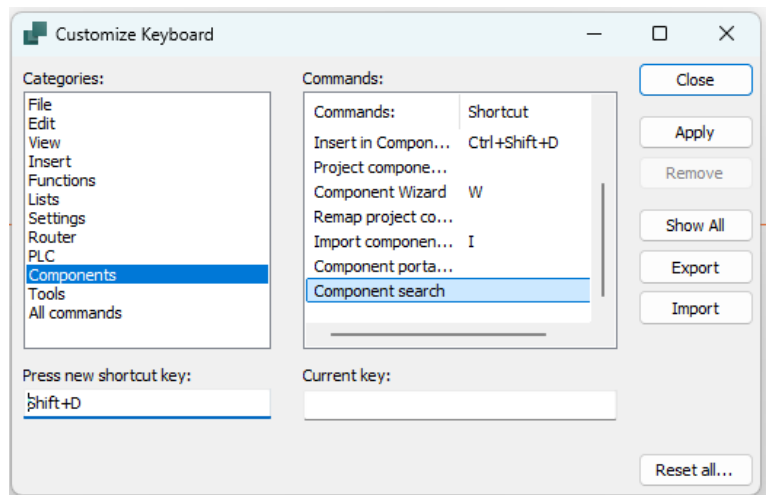
There are many customers who want to 'download everything' – which is not and will not be possible. The reason for the customers' request is that it can be difficult to search for individual components, and even more difficult that you can only download 30 at a time.

BUT - from version 25 we will integrate the old database and the Portal more and more, and the first step is included this time.

In the Components menu, there is a new menu item - Component search.



It is possible to create a shortcut to the function

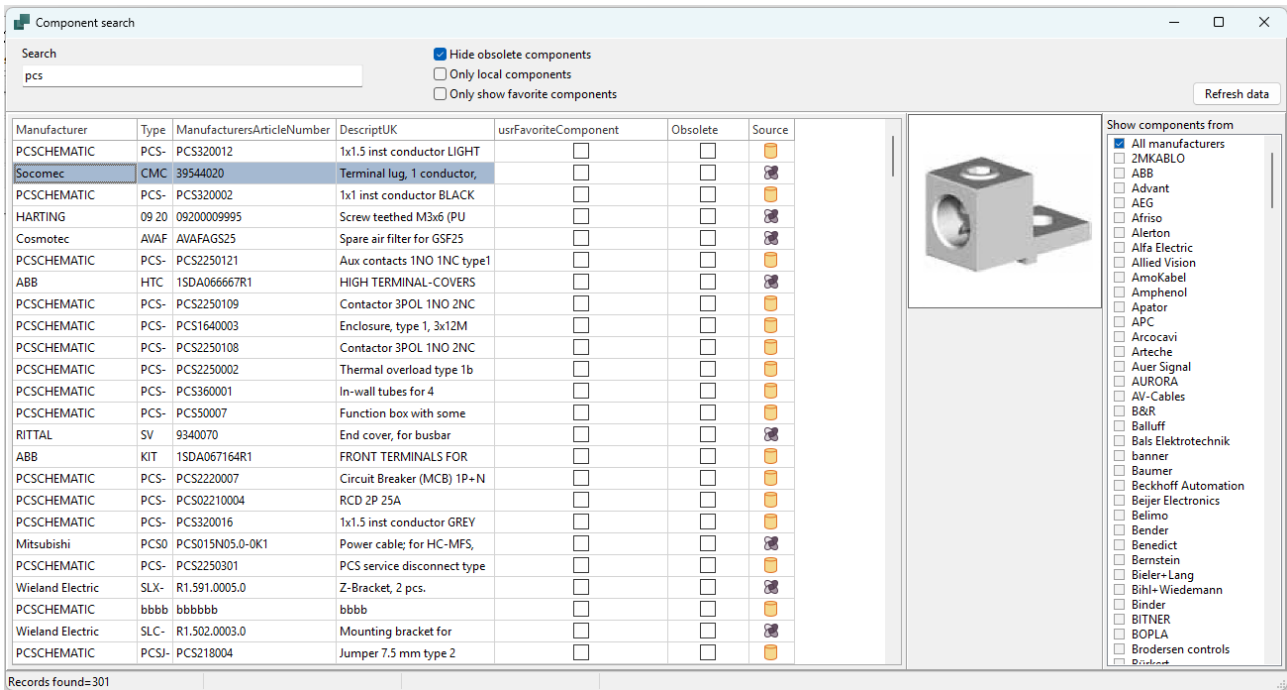


Component search – search locally and on the Portal

When you press the button, the program retrieves information about all the components that the installation knows, i.e. both locally in your own database and globally in the Component Portal.

When the data is ready – the first time about 5 seconds – a window opens showing all known components, i.e. both the components that are located locally and those that are only located on the Component Portal.

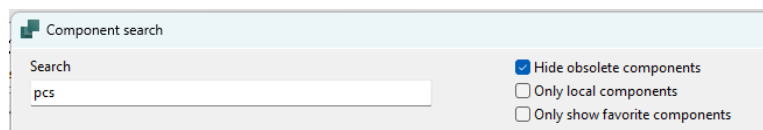
The window has some fixed columns with data, a picture of the selected component and a list of all manufacturers.



Search for a component

The window has the following fixed columns:

- Manufacturer
- Type
- Manufacturer's Article number
- Description – in any local language and always in UK
- Favorite
- Discontinued
- Source



When you type in the SEARCH field, all the displayed fields are searched, both at the beginning and in the actual content of the fields.

You can search more specifically by checking one or more boxes next to the search field, or by selecting all or a single manufacturer.

Get a component

When the desired component is visible in the window, it is retrieved by double-clicking on it. The window is closed, you return to the project, and the component symbols are ready – in exactly the same way as when you retrieve a component from the database.

There is no difference whether you click on a local component or one that is only in the Portal. But the next time you search in the window, the component in question will be local 😊

Refresh data

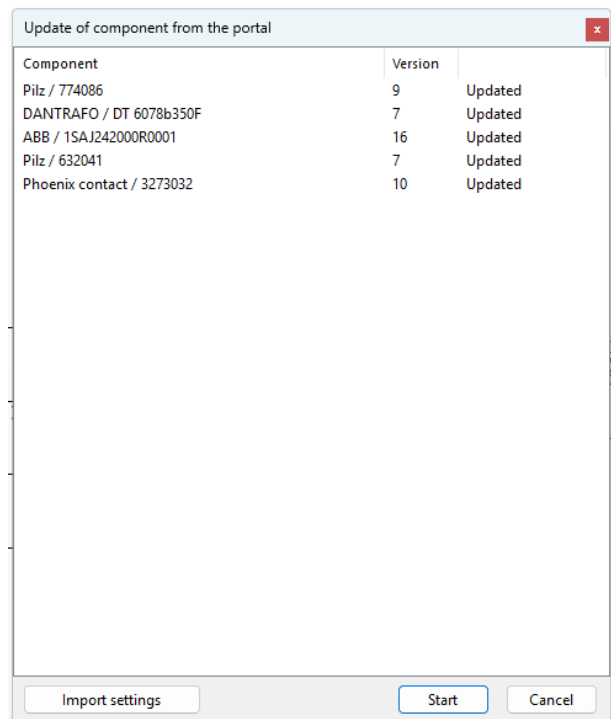
Data is retrieved once per session, i.e. once for each time you have opened the program. If you know that there are new components on the Portal or you have edited a component in your own database, you can update the list by pressing the 'Update data' button.

How can I import components into my local database? *

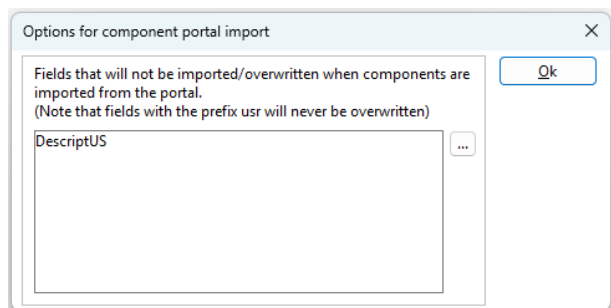
The new function above will make it easier to download a new component on the Portal, but there are also other ways to get new components into your database.

They are listed here:

- ✓ You can download components from the old ver22 database. You import the components, and their existing symbols will be added to the OldSymbols folder – and that's all for now. There is a description with examples and pictures here: <https://cdn.pcschematic.com/pdf/en/Converting projects from ver22 to new format.pdf>
- ✓ You can create components yourself using the Component Guide – there are a few new features that will be shown later in this document. The Component Guide manual is part of 'Getting started...' which can be downloaded here: https://cdn.pcschematic.com/pdf/en/UK_ver23_Tutorials.pdf
- ✓ You can import components from a locked Project Database – as before
- ✓ You can download a single component or a basket full of components directly on the Portal - a basket can contain 30 components at a time.
- ✓ Components that are already in the database can be updated, e.g. more descriptions, symbol changes, etc. You can deselect some fields from updating – for example, I want to use US description for 'my' descriptions
 - You cannot deselect fields that are permanently linked, e.g. type and item numbers (nor GTIN)
 - The portal cannot update USR fields – these are the user's own fields!!
 - And remember – it looks at the ComponentID's version number!



Updated components appear in a list in the window (the list is missing type/article number and manufacturer).

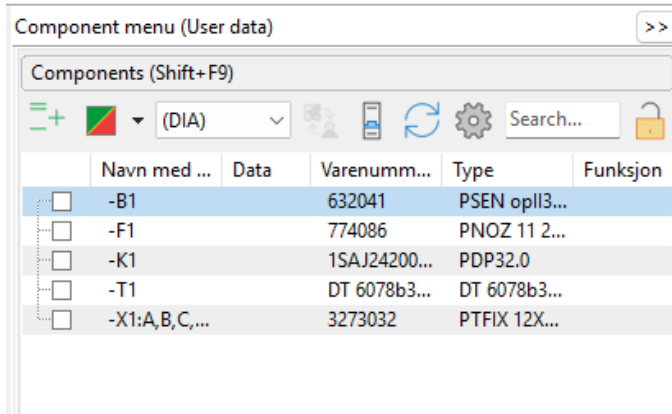


Download the project components ...*

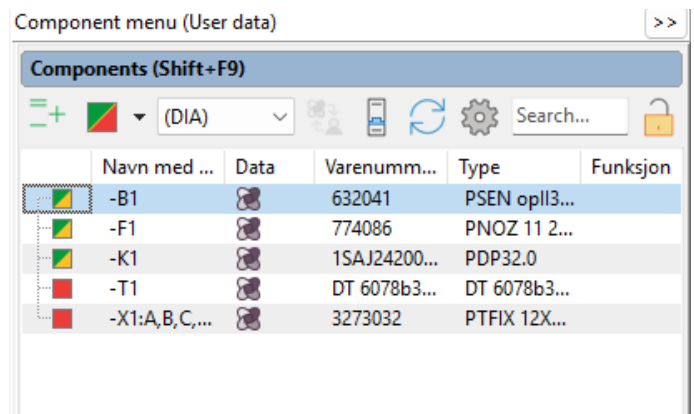
Components that originally come from the portal can be updated AND DOWNLOADED directly with the same function. This means that if you receive a file that contains components from the Portal, you can download these components directly via the file:

BEFORE and AFTER

(white square means 'not in the associated database')



	Navn med ...	Data	Varenumm...	Type	Funksjon
<input type="checkbox"/>	-B1		632041	PSEN oppl3...	
<input type="checkbox"/>	-F1		774086	PNOZ 11 2...	
<input type="checkbox"/>	-K1		1SAJ24200...	PDP32.0	
<input type="checkbox"/>	-T1		DT 6078b3...	DT 6078b3...	
<input type="checkbox"/>	-X1:A,B,C,...		3273032	PTFIX 12X...	



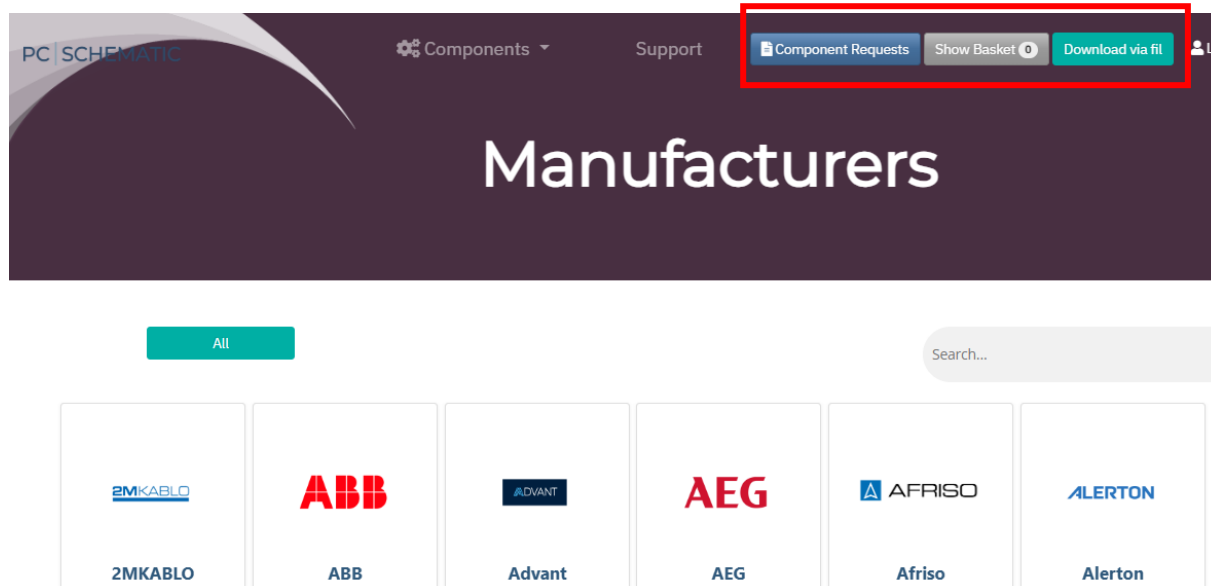
	Navn med ...	Data	Varenumm...	Type	Funksjon
<input checked="" type="checkbox"/>	-B1		632041	PSEN oppl3...	
<input checked="" type="checkbox"/>	-F1		774086	PNOZ 11 2...	
<input checked="" type="checkbox"/>	-K1		1SAJ24200...	PDP32.0	
<input type="checkbox"/>	-T1		DT 6078b3...	DT 6078b3...	
<input type="checkbox"/>	-X1:A,B,C,...		3273032	PTFIX 12X...	

News on the Component Portal *

There are three new features on the Portal:

- You can request to have new components created
- You can report errors on components
- You can retrieve a large number of components using a list with manufacturer and part number.

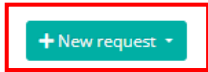
The functions require – of course – that you are logged in to the Portal via Automation.



Component requests

When you press the button, you can choose between regular components and Panelbuilder components.

Note! Expected time for creation of component can be up to **8 weeks** from today.



#	Article number	Manufacturer	Category	Status	Created
1	12345	ABB	-	Pending	Mar 10 2025

The front page shows the status of your own requests.

Panelbuilder components contain data so that they can be used in our Panelbuilder tool (residential panels with heat loss calculation).

The following must be completed for a regular component

Manufacturer *

Article number *
Description (DK) *
Description (UK)

Type *

Link to the component's datasheet *

Create as series

Name *

E-mail *

Please fill out your name and email so we can contact you directly, in case of issues with the request.

The following must be completed for a Panelbuilder component *

Manufacturer *

Article number *
Description (DK) *
Description (UK)

Type *

Link to the component's datasheet *

Create as series

Number of modules **Ref. temperature (°C)** **Width (mm)**

IP-code **Max Temperature (°C)** **Height (mm)**

Number of rows **PowerLoss (W)** **Depth (mm)**

Rated current (A)

Name *

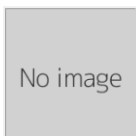
E-mail *

Please fill out you name and email so we can contact you directly, in case of issues with the request.

Report component error *

You can now report errors on specific components directly on the Portal

Manufacturer: PCSchematic
Article Number: PCS1640001
Type: PCS-UC1-1x12M



Description:
Enclosure, type 1, 1x12M

Properties

Dimensions & Weight	
Length(X)	
Height(Y)	
Depth(Z)	
Weight	

Error in component ✕

Enclosure, type 1, 1x12M
Article Number: PCS1640001
Type: PCS-UC1-1x12M
PCSchematic

Name

Email

Message

Download components via file *

The last option we have for downloading components to our own database is here.

If you need to download a lot of components at once, so that they can be local – some do not have internet access – then you can use this function to do this.

Upload an Excel or CSV file to download multiple components.

You can download an excel template with the headers filled here:

[Download template](#)

Choose an excel or csv file that has a list of components you wish to download.

[Choose file](#) No file chosen.

[Download](#)

Example

There can be used either the article number or the type number

Manufacturer	Article Number	Type
Beper Electronics		M9372
Chiffel	179609	
Hensel	KV 1509	
ABB		39-0
Benedict		ESU L24RL
Festo	MA-15-10-M5	
Rockwell Automation	100-C09001	

* There needs to be written "Manufacture" in the first row of the first column, as well as "Article Number" in the second column and "Type" in the third column.

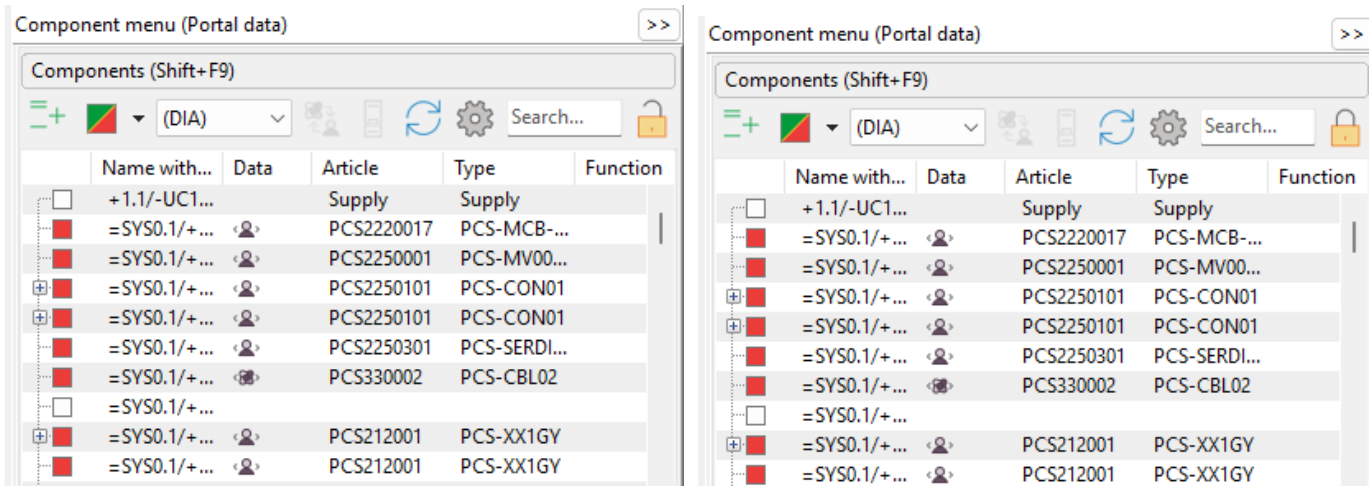
1. Start by downloading the template
2. Fill it out with Manufacturer and type and/or Article number
3. Select the file
4. Download the zip-file that can be imported directly into the database

LOCKED OR UNLOCKED PROJECT DATABASE?

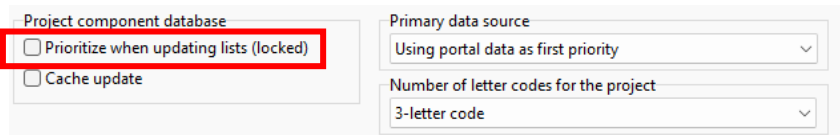
When exchanging files, it is often a good idea to lock the Project Database, so that you can update lists after copying or deleting components in a file.

But ... if you want to be able to continue working with the file, the Project Database must be unlocked - preferably after the contained components have been imported into your own database.

We wanted to make the status more visible, so there is now a padlock on the far right in the Component Menu.



You change the status in Settings | Database



A LITTLE ABOUT SYMBOLS

The most requested news *

OK or All?

Up until version 23, both the OK and All buttons were marked as active in the Component data dialog.

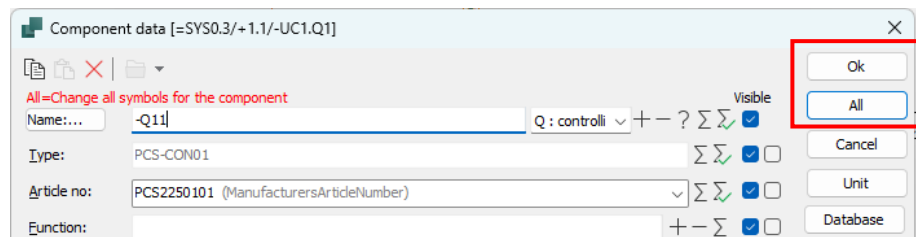
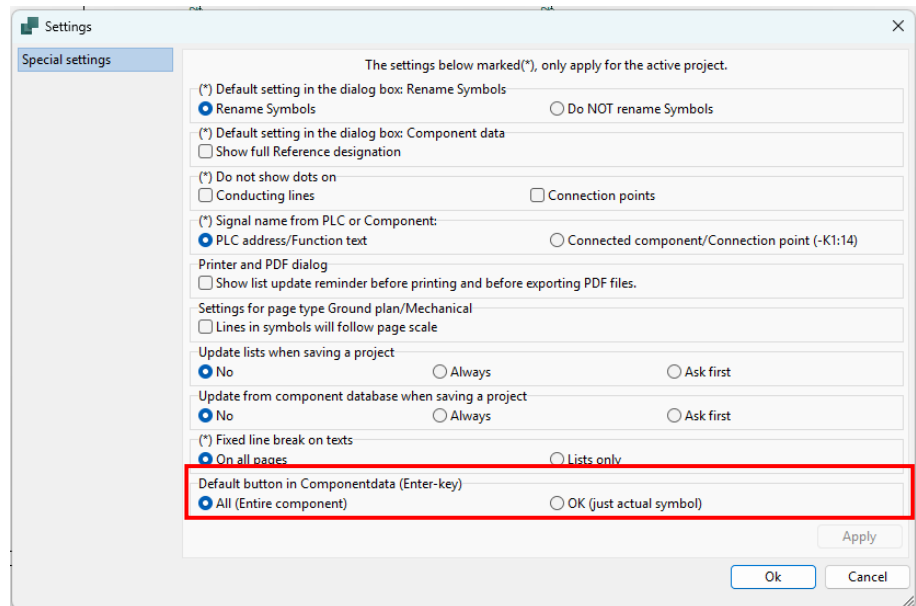
'Old' users knew that Enter meant OK.

From version 24, the default is 'All', so you ensure that the entire component automatically changes, for example, name or item number when you press Enter.

However, not all users have been enthusiastic, so from the next patch you can choose whether the default should be OK or All 😊

Located as an option in Special Settings.

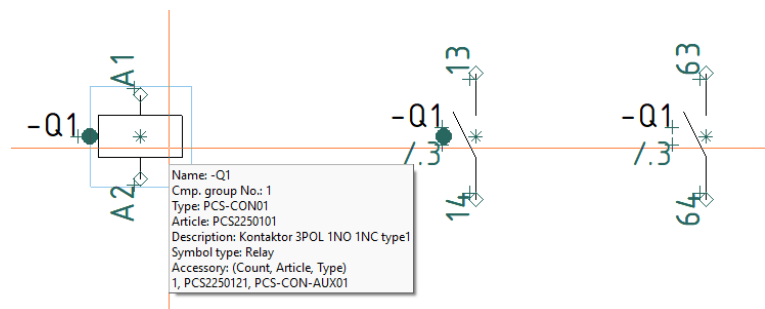
Default is All.



The component has accessories

If a component has accessories, there have been many requests to make this more visible.

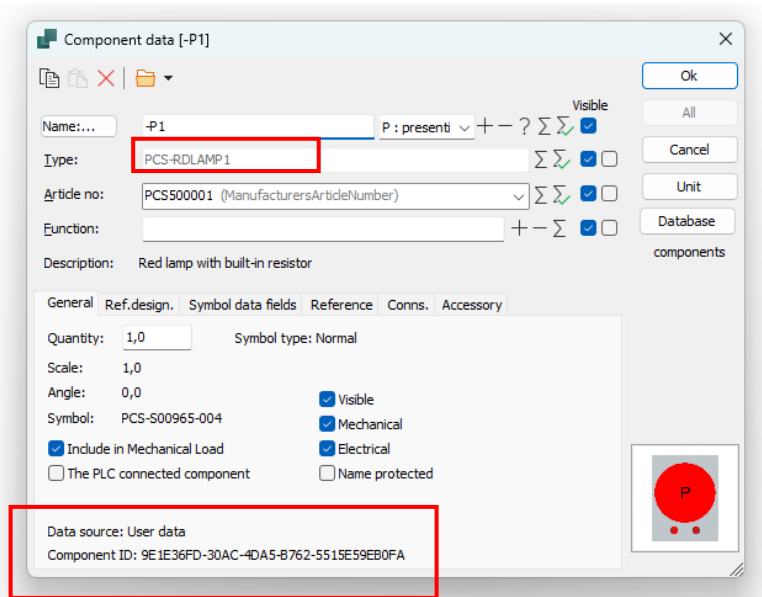
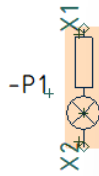
The information is found in the popup, but from ver25 you can also see a small dot by all the main component symbols.



When the component comes from the database ...

The database contains the truth about components. Therefore, you should not correct what comes from there!

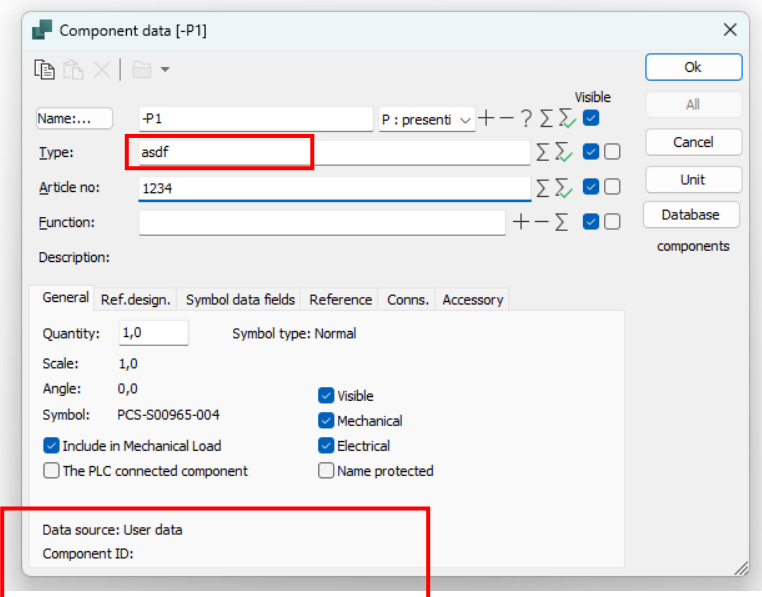
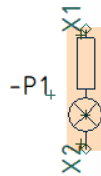
When a component comes from the database, the Type field is locked for editing. This is done so that you cannot enter a type designation for the component that does not match the selected item number and thus get errors in parts and component lists.



This means that when a component has a ComponentID – see at the very bottom of the dialog box, it comes from the database, where it is defined with type, item numbers, descriptions, image, etc.

And then you should not be able to change the type, either intentionally or accidentally.

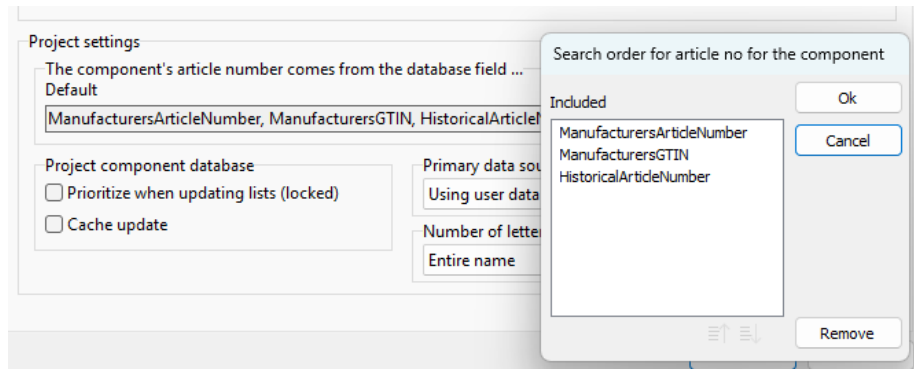
Of course, you can still freely write in both type and item number if you place a symbol, i.e. something that is not in the database and therefore does not have a ComponentID.



Prioritized article numbers

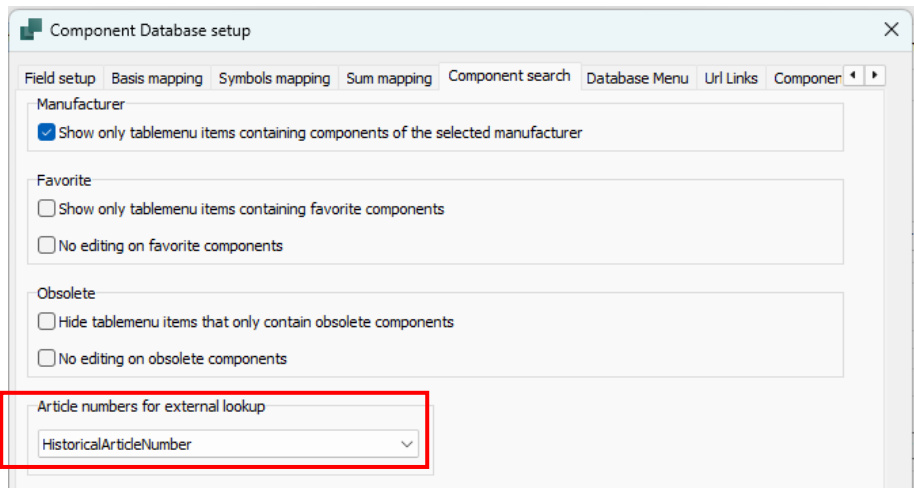
From version 23 we have been able to handle multiple article numbers from the database, so that you can, for example, have your own article number included in the parts list, or alternatively the historical or manufacturer's item number.

The setting is made under Settings|Database.



It has worked fine when talking about new projects, but if you have used various lists to retrieve components, you have been locked to the article number that is in the list.

And you had to choose that article number yourself under Settings|Database| Database settings| Component search.



You don't have to do that anymore!!

And the drop-down box is gone in version 25.

In the future, the priority will also apply to:

- Input from lists – Load component list, parts list, plc list
- Output from Panelrouter
- Project generator

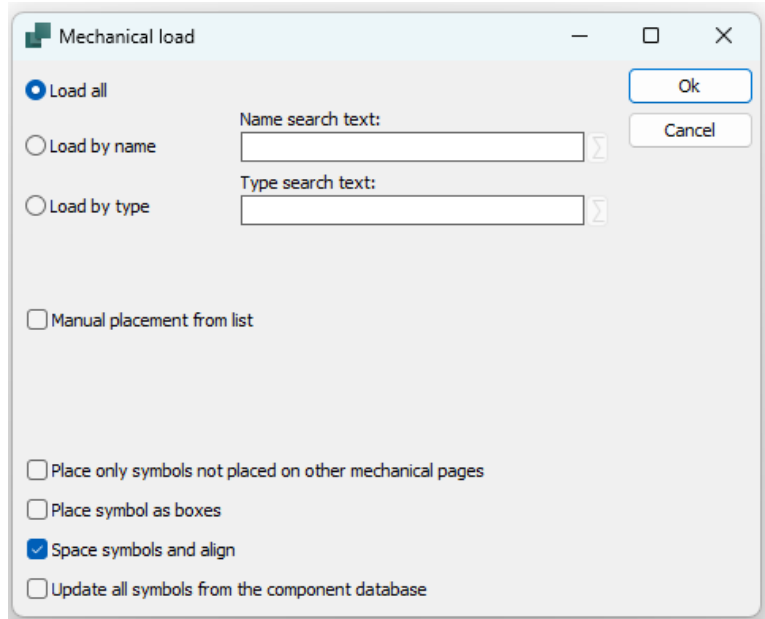
Regarding the Project generator, it is (always) the case that the settings on the first page of the project apply to the entire generated project:

- This means that if the first page is set to HistoricalArticleNumber and PCSHEMATIC font, the finished result will also contain this.
- On the other hand, if the page is set with prioritized order and OSI font, then this will apply.

Do not include in Mechanical load

When creating a Layout, you can either select components in the Component menu (see below).

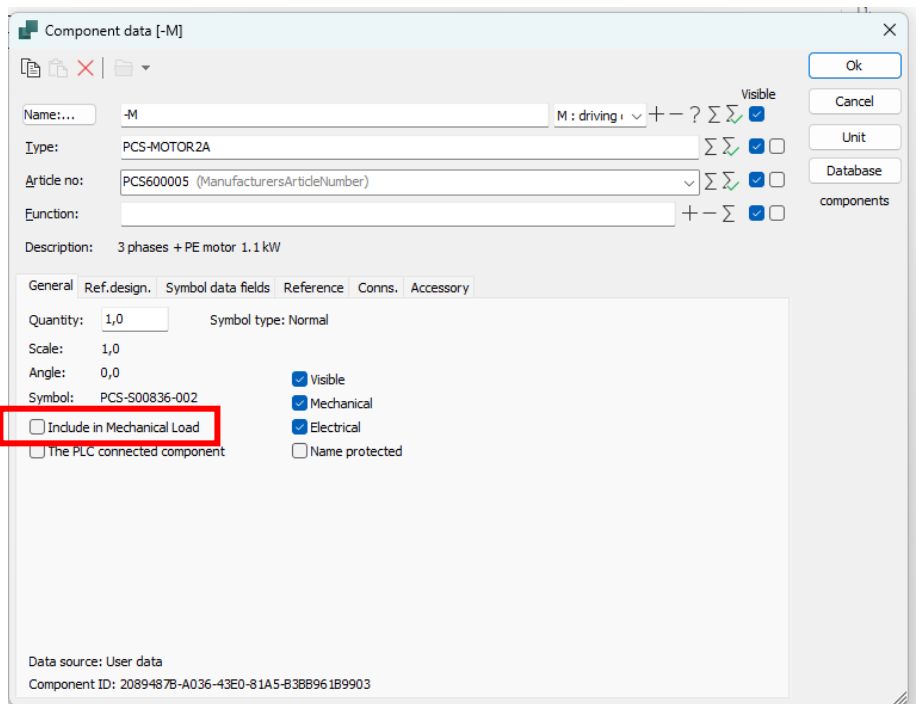
Alternatively, on the layout page, you can select Mechanical load, and then a list of selected components will appear.



However, some components are never intended to be placed in the layout, so it may be smart to deselect them in the diagram.

This selection means that the component in question disappears from both Mechanical load – see above – and from the Component menu on mechanical pages – see below.

When creating the symbol, this setting can be selected as the default.



Lines in the Component menu with red icons represent components that are either placed or do not have a mechanical symbol.

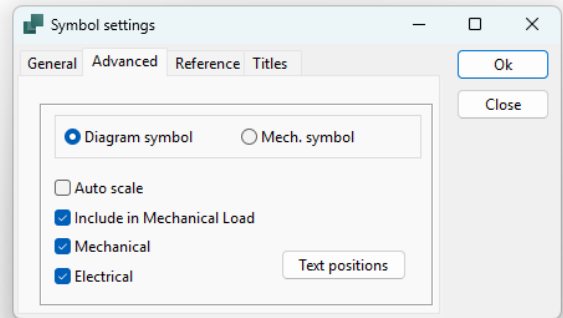
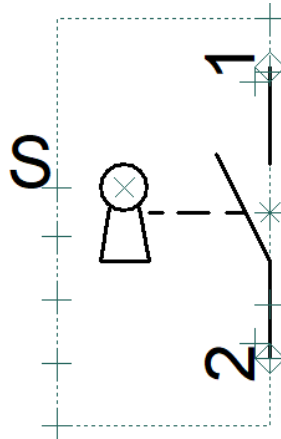
Here – taken from the Layout page – M1 has disappeared, as Mechanical placement is deselected above. .

	Name with...	Data	Article	Type	Function
■	-F1	⟨Ⓜ⟩	PCS2250001	PCSMV001A	
■	-F2	⟨Ⓜ⟩	PCS2250001	PCSMV001A	
■	-F3	⟨Ⓜ⟩	PCS2250001	PCSMV001A	
■	-F4	⟨Ⓜ⟩	PCS2250001	PCSMV001A	
■	-M2	⟨Ⓜ⟩	PCS600001	PCSMOTO...	
■	-M3	⟨Ⓜ⟩	PCS600001	PCSMOTO...	
■	-M4	⟨Ⓜ⟩	PCS600001	PCSMOTO...	
■	-Q1	⟨Ⓜ⟩	PCS2250101	PCSCON01	
■	-Q2	⟨Ⓜ⟩	PCS2250101	PCSCON01	
■	-Q3	⟨Ⓜ⟩	PCS2250101	PCSCON01	
■	-Q4	⟨Ⓜ⟩	PCS2250101	PCSCON01	

Symbol settings – Symbol editor

We have tidied up the Symbol Settings in the Symbol Editor and clarified what (some of) the advanced settings actually do:

A **diagram symbol** should always have the same size on the paper, regardless of page scale. Symbols for regular diagrams are made with a number of modules of 2.5 mm. And that size should be kept. On the screen they look a little different, depending on whether you are on an A3 page (standard) or choose another format. But if you have set your grid to 10x10, it will fill the same space on all page formats.

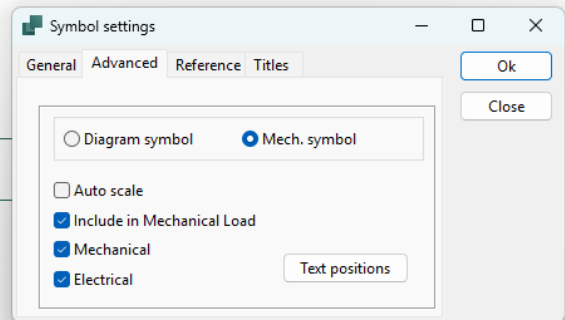


The new Alarm Symbols are diagram symbols.

A **mechanical symbol** is intended to be inserted on a mechanical page, and here it is important that it adapts to the scale of the page.

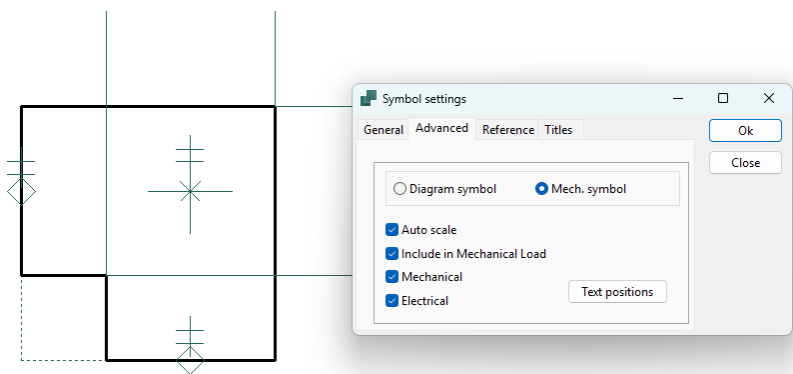
I find that it sometimes helps with understanding if you make a mechanical symbol on a mechanical page, i.e. to see the XY symbol.

Here is a terminal that MUST be made on a mechanical page if you want to be able to show connection names for more than 1 layer: the connections are placed, and the layers are called 1, 2, 3, etc.



Autoscale is rarely used.

Here is the symbol `Corner.sym`, which is a symbol made with a wide line in size 10 mm, connection points at both ends, and the Autoscale property is set. This symbol can be placed in the corner between two wide lines, e.g. wire trays, and make a nice corner that matches the lines on the page.



More symbols for PLCs, switches etc *

We create symbols for **COMM** (communication) and **COM** (common) based on existing Input and Output functions. We have investigated whether we should create a new type – Other – with the same function as I/O, but we do not dare. Therefore, we create some symbols that graphically differ from input and output (i.e. they resemble the current COMM symbols), but which have the same link functions and options for transferring data as input and output.

COM for inputs or outputs are created on input and output symbols, respectively.

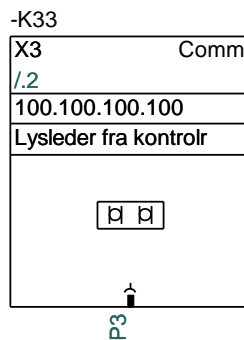
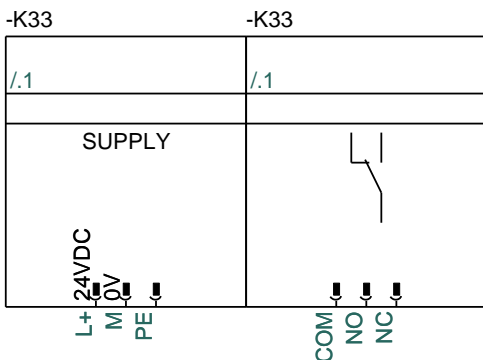
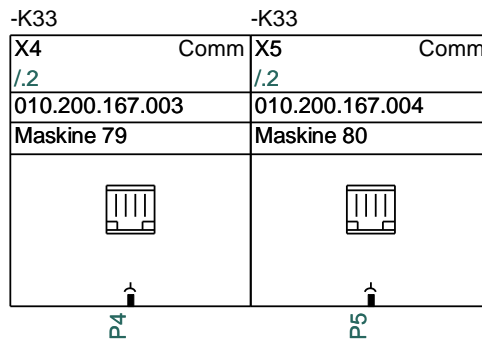
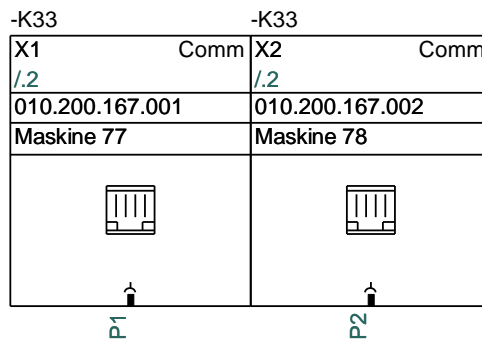
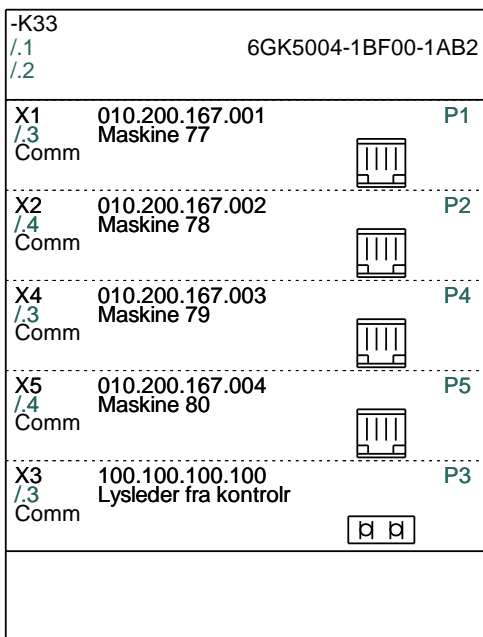
We have made it into some special components, and this means that the COM symbols/channels are located together with the corresponding addresses.

COMM for Communication is created on Input symbols.

They should graphically resemble our current COMM symbols, with the addition that the small icons from the ref symbol are also included in the 'IO' symbol.

4 sets are created: 1-4 pins. We can create more if/when the need arises.

The symbols are downloaded automatically when the components are downloaded from the Portal.

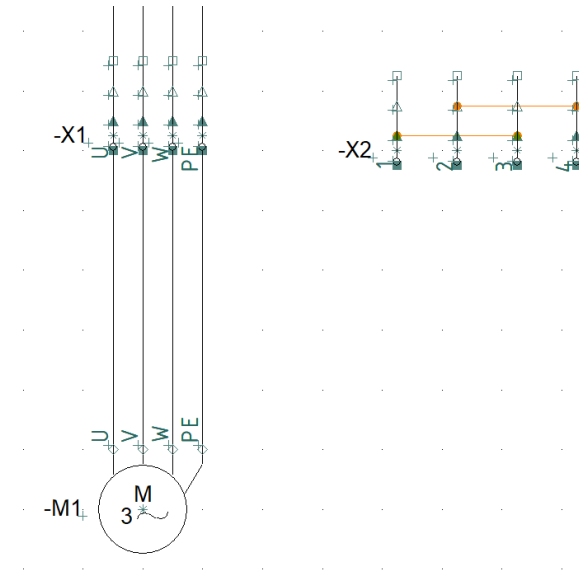


New symbols for terminals – with jumpers *

Many feedthrough terminals have jumpers, and we would like to show them on separate connection points.

In connection with cleaning up/improving symbols, we are adding space for jumpers, and they will 'dot' when connected.

If you already have your own terminal symbols with jumpers, the functionality will be included in the upcoming patch.



COMPONENT WIZARD *

You create new components – primarily – using the Component Wizard!

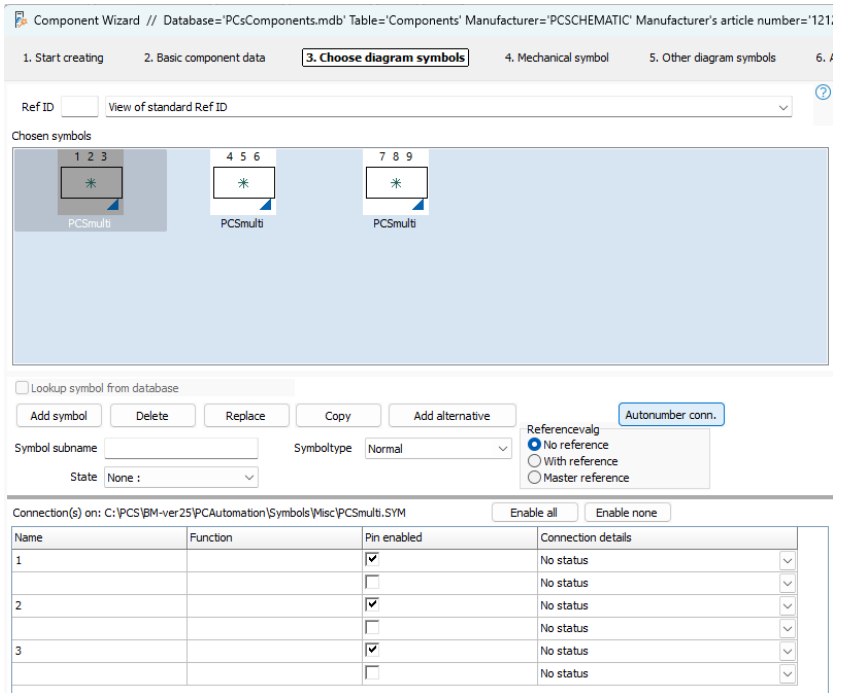
Then the various rules for entry are respected. For example, , or ; or [or ?

This also means that we have to keep up with the functions that are necessary. So if something is missing, please get back to us!

Automatic numbering of connection points *

If you are making components with many symbols, you may need a numbering function.

It is provided here:

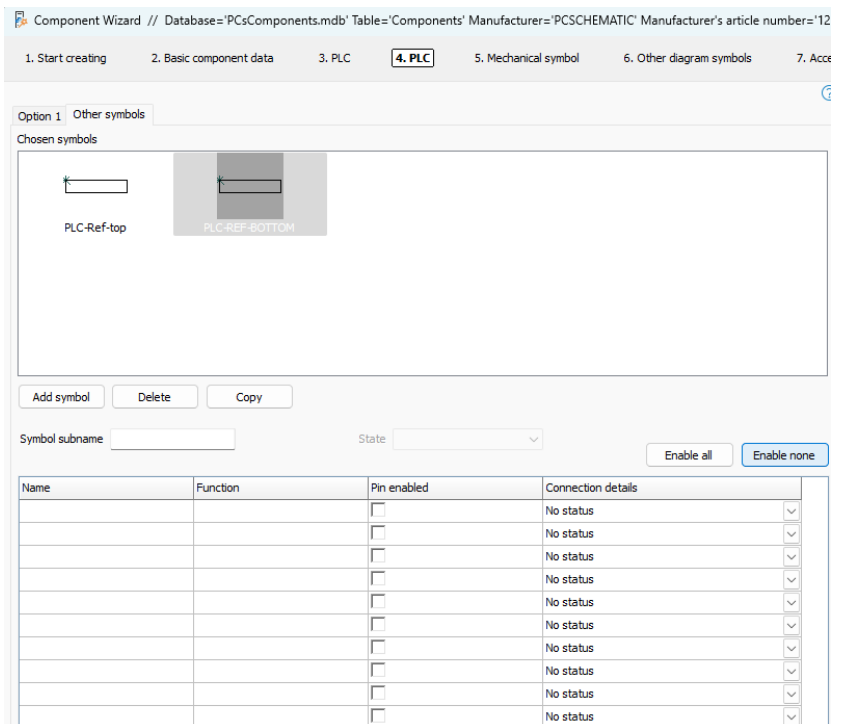


Enable all/none *

When making – especially – PLCs, you often want to disable/activate all connections.

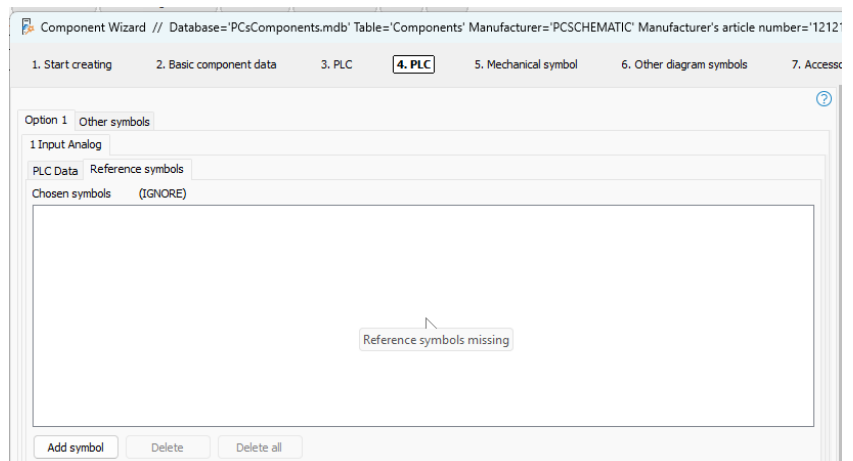
You can now do that at once 😊

See it both here and in the picture above.



The Component Wizard automatically jumps to tabs with errors/missing info *

In the example here, the reference symbols are missing on the PLC.



The Component Wizard always searches in symbol folders and not in project *

In the earlier versions, the Component Wizard has looked for symbols in the current project – as the program does – now it looks in the symbol folders

Remember, the program looks for symbols according to this priority

- ✓ In the project
- ✓ According to the ALIAS list

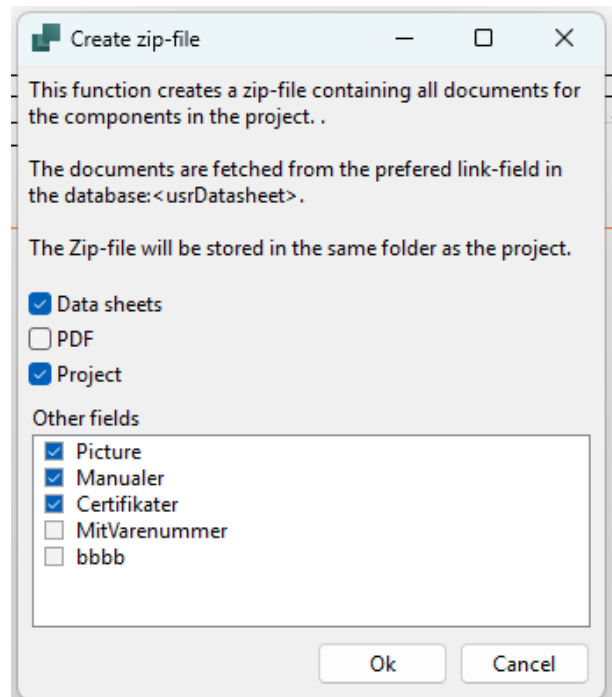
CREATE A ZIP-FILE *

When you download components from the Portal, we have created them with pictures.

You also have the option to add data sheets to an already defined field, and you can create your own fields for various other files that belong to the component.

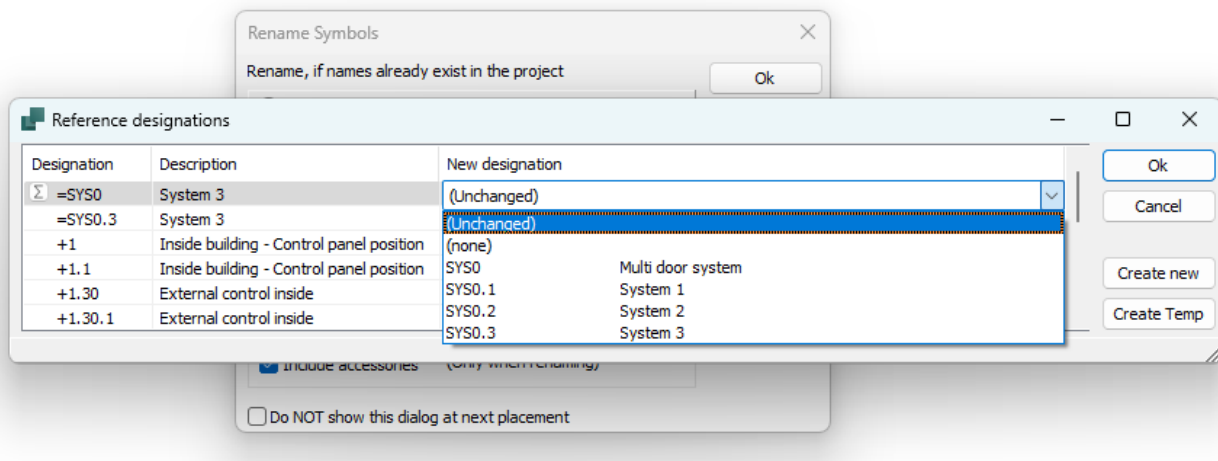
What is new is that you can create a combined zip file with these linked documents:

You can only add data to the extra fields by selecting them on the last tab in the database settings, i.e. where you select fields for the last page of the Component Guide. Or directly in the database dialog.



COPYING WITH REFERENCE DESIGNATIONS *

Now the window allows space for long descriptions. And you can drag the window even longer, so that it fits the contents.



EXPORT TO PXC WIRE ASSIST FROM THE PANELROUTER

*

All settings of relevant wire data, ie color, dimensions etc, are set by referring to relevant database data fields and not to 'loose files'.

Apart from that, the Panelrouter can export to

- ✓ Komax – need for other formats?
- ✓ CadCabel
- ✓ Phoenix Contact Wire Assist – new in ver24

