

SYMBOLS, LISTS AND TEMPLATES



This book takes you through the different symbol types in PC|Automation, that is their functionalities as well as their design rules. Design rules include references to standards for symbols for use in electrical documentation.

Drawing headers – title blocks – are also symbols, and will also be treated here, including list symbols.

Finally, there are descriptions of how to create templates to use in the program

Developed by PCSCHMATIC A/S

Last edit June 2026



PREFACE

Components are illustrated with symbols that show the component name and connections for the components in diagrams. There are many different symbol types, each of which has a function that makes data from these components appear in diagrams and lists in an appropriate way. This booklet reviews the different symbol types and their functionality and lists design rules for them.

Symbols for use in the program come in several different types:

- Electrical symbols for use in diagrams, where there are many different types intended for different component types
- Mechanical symbols for placement in panel layouts
- Drawing headers and lists are also created as symbols, where there are also different rules that must be observed if lists are to function
- The Panelbuilder tool also places requirements on the design of the symbols if automatic layouts are to be generated
- Finally, we have programs such as Cabledim, which also have special symbol types, but this is not reviewed here

This booklet reviews the design of the different types, and a review of how the symbol is related to the component database.

The booklet is used partly as a manual and partly as training material.



Contents

Preface	3
When you place a symbol	9
Where does the different kinds of data come from	9
Setting the Text/symbol defaults	10
Symbols in the program	11
Rules for designing symbols	13
Standards for electrical symbols	13
Basic design rules for electrical symbols.....	13
Module size, symbol size, text size and line thickness etc	14
When creating your own symbols.....	15
Basic symbols	15
Additional symbols – Qualifiers.....	15
Combined symbols	15
Symbols for sensors and instruments	16
Be careful – don’t make too many details	16
Working in Automation’s Symbol Editor.....	17
The fixed symbol data fields.....	17
Connection point data fields.....	18
Symbol settings	19
Symbol type.....	19
Selection.....	19
Advanced settings.....	19
Text positions	20
Reference	20
Titles.....	20
Settings on connection points	21
Connection details	21
Symboltypes.....	22
How to – design electrical symbols.....	25
Open the Symbol editor	26
Editing options in the Symbol Editor	27
Pickmenu in the Symbol editor.....	27
Create a new symbol using the Symbol Generator	27
Create the symbol	28
Create a symbol using an existing symbol	28
Normal electrical symbol	30
Create the symbol	30
Edit symbol.....	30
Combined symbol	30
Follow connected.....	31
Symbols with states	31
Place a symbol with states	33
Interaction with the database	33
Symbols for Relays, NO (Open), NC (Close) and NO/NC (Switch).....	34
Interaction with the database	34
Reference symbol (NO/NC ref. cross)	35
Create new reference symbol.....	35
Data symbol	36
Create Symbol data fields.....	36
Create a data symbol	37

Add symbol data fields to a diagram symbol – in the symbol menu	37
Add symbol data fields on a single symbol – in the project	37
Terminal symbols	38
Other components on the terminal list	39
Distribution blocks	40
Busbar connections	40
Plc symbols	41
Create the symbols as a set – example 1	41
Save the two symbols	43
Create the symbols as a set – example 2	43
Other symbols for plc components	44
Nice – and need – to know, when you create your own symbols	44
Cable symbol	46
Busbar	47
Signal symbol	48
Join-symbol or Multi signal symbol	49
Wire number	50
Insertion point	51
Line connection symbol	52
Using the line connection symbol for models (subdrawings)	52
Support symbol – Accessory	53
Header (List on page)	54
Combined symbols – Symbols on the Portal	55
Symbol documentation	56
Explain you symbols – also when you combine standard symbols	56
Mechanical symbols	57
Use the Symbol Generator	57
Create mechanical components based on an AutoCAD drawing	58
Mechanical symbol for terminals	59
Mechanical symbol for multi-layer terminals	59
Non-conducting symbol	59
Symbols for the Panelbuilder	60
Component symbols for Panelbuilder	60
Symbols for the panel in Panelbuilder	60
Headers (Title blocks), lists and templates	61
Create your own drawing header	61
The size of the drawing header	61
Where to place the * the reference point	61
Drawing headers with states and optional language codes	62
Data fields in normal drawing headers	62
Tips & tricks when creating drawing headers	62
Create your own lists	63
Rules for data fields in lists	63
Tips & tricks when you create lists	64
Create a template	66
Create a DIA-template	66
Drawing header and other page data	66
Zones (current paths), reference symbol etc	67
Insert potentials	67
Insertion points	67
The page contains a list	68
Diagram pages for other diagram types / symbols	68



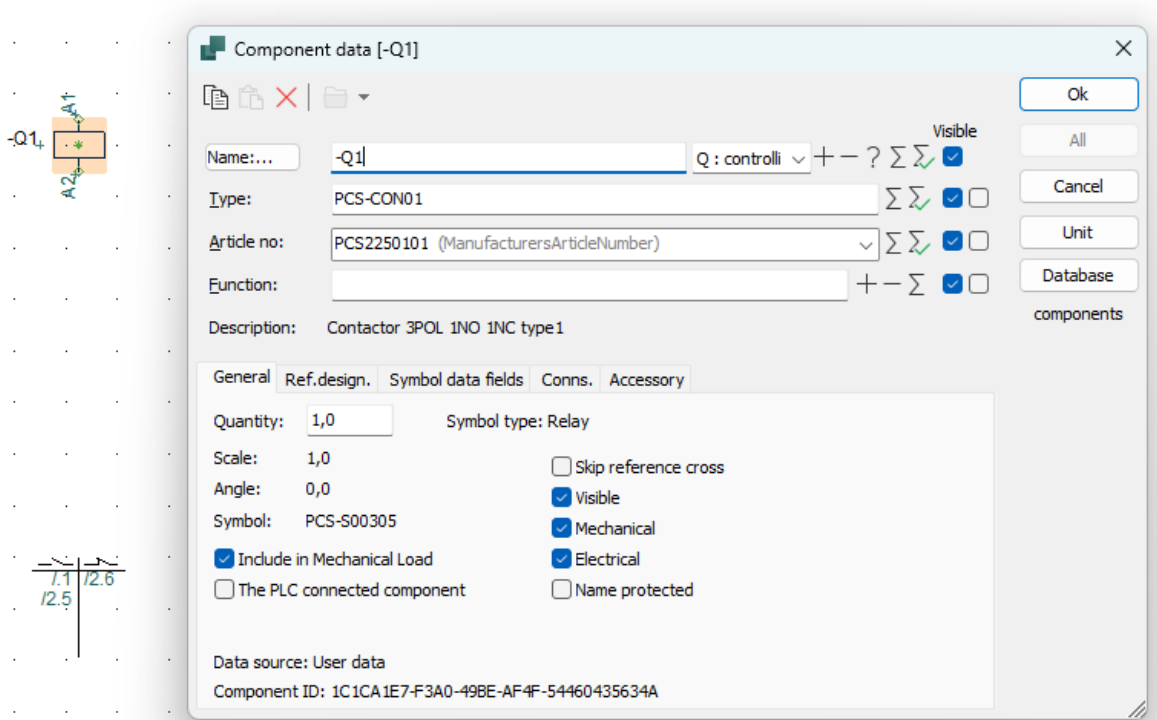
Create a GRP template.....	69
Tavler og komponenter i modulmål	69
Template with a list	70
Create the template	70
Listeopsætning.....	70
Templates for graphical plans	72
Create a Project template	72
My notes	74



WHEN YOU PLACE A SYMBOL ...

When you place a symbol in a diagram, different things happen. Some depend on your own settings, others are general functions, which depend on how your installation is set up, i.e. your Symbol settings, your Database settings and of course on which Component (or symbol) you choose.

In the image below, the coil for a contactor has been placed.

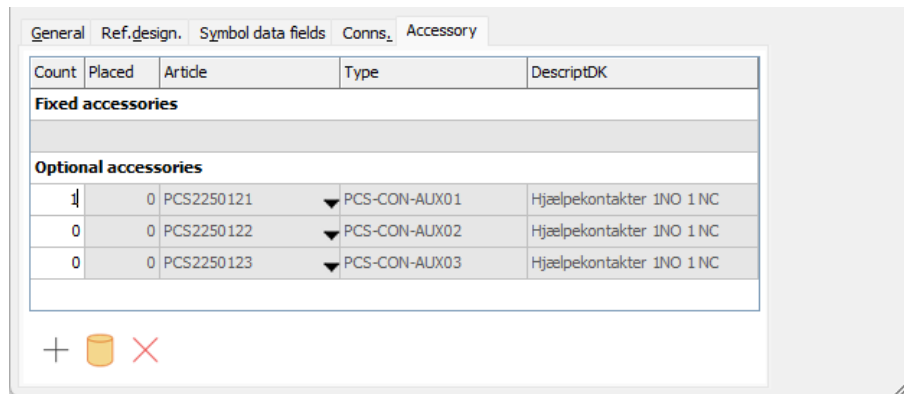


Where does the different kinds of data come from

When we place symbols from a component, different things happen in the program. Below is an overview of what happens and where this is set.

- When we place symbols from a component, we select one of the component's already defined symbols. If we insert a symbol on the selected part number that is not defined in the database, we get an error message.
- Most symbols are created with a default name (here K), but since a specific component has been selected, this one is called -Qxx, since the component's RefID field contains this value. A value in the database overrides the symbol's own name.
- A '-' is placed in front of the symbol name. This is selected under Settings|Cursor/Screen. '-' means that it is a product.
- The NO/NC cross is placed automatically, as it follows the symbol type 'relay'. When the corresponding NO and NC symbols are placed in the project, references to these are displayed.
- The location of the NO/NC cross is defined under Page settings|References. There are different Symbol Types in the program, and their different functions are reviewed in the booklet here from page 22
- When the component comes from the database, Type, Article No. and Component ID will be filled in. Type and article no. can be used on parts and component lists.
- The connection names on the displayed part also come from the database.

- If the database contains information about accessories, you can also see and select it on the component – and place the symbols accordingly:



Count	Placed	Article	Type	DescriptDK
Fixed accessories				
Optional accessories				
1	0	PCS2250121	PCS-CON-AUX01	Hjælpekontakter INO 1 NC
0	0	PCS2250122	PCS-CON-AUX02	Hjælpekontakter INO 1 NC
0	0	PCS2250123	PCS-CON-AUX03	Hjælpekontakter INO 1 NC

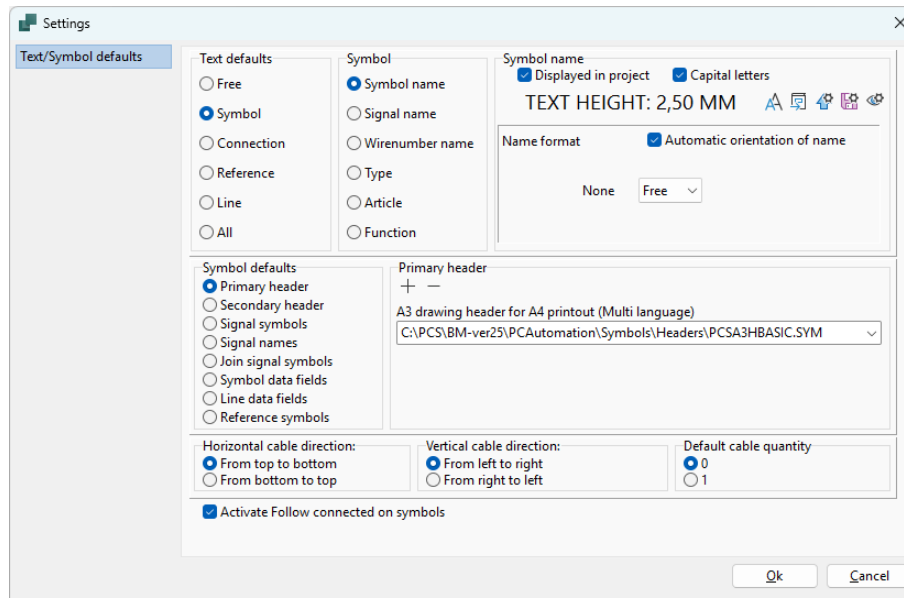
Everything that has to do with database settings is described in the Database Manual. From ver23, most database settings are fixed, for example Type always comes from the Type field. This also applies to setting up fields for article numbers, accessories, connection names, etc.; in short, everything that has to do with an item number comes from the database, and the setting up of fields is fixed.

Setting the Text/symbol defaults

Setting how data from the component is displayed in the program is done under Settings |Text/Symbol defaults.

Here, you can set the font, color, and visibility in the project for the different symbol types and the fixed symbol texts.

Generally, all texts should be 2.5 mm, which corresponds to the module size used. Symbol and connection



names should be visible – so you can see where the wires are to be mounted – and various types and part numbers may be invisible (default – they don't exist in the documentation standards).

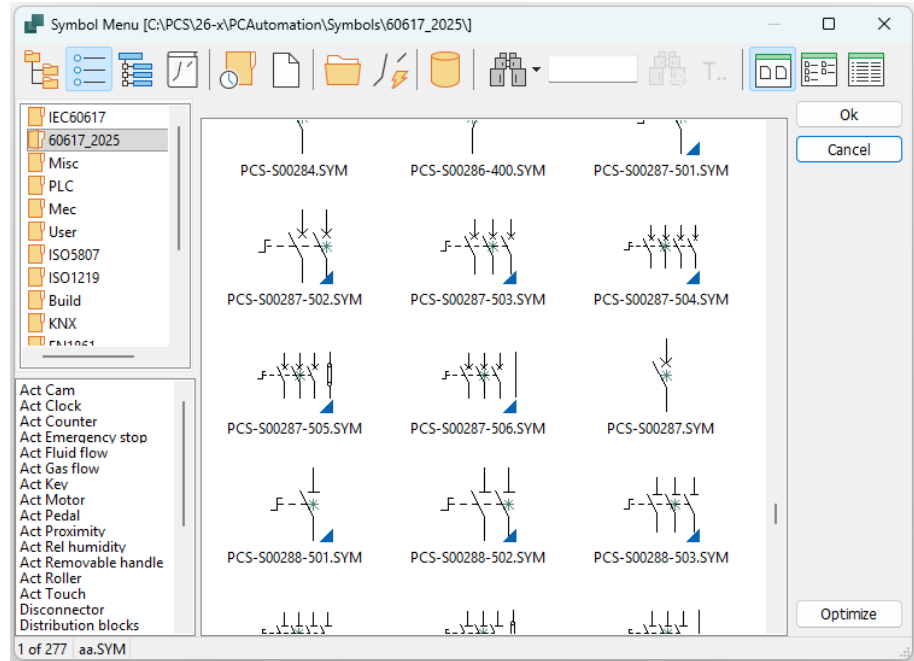
The settings that are selected are reflected in the project and in the symbol editor. Settings can be copied between projects.



Symbols in the program

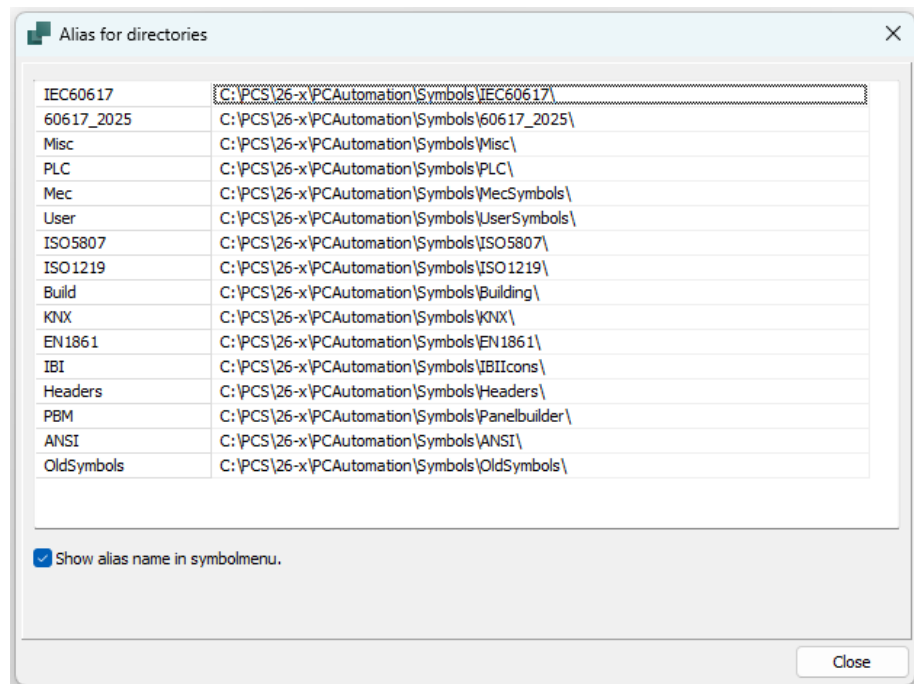
When you press the Symbol Menu button, you can see all the symbols that you have access to in the program.

The symbols are located in folders with Alias names that are defined under Settings|Libraries.



In the picture, all symbols are located locally, but they can – of course – also be located on a shared drive.

The symbols for a component are retrieved from one of the folders shown. Which folder depends on how the Alias list is set up: File names are searched for in the folders according to their order in the list. You can change the order by dragging the folder to the correct position.



(Picture from Settings|Directories|Show Alias folders).

The database and the pickmenus can only find symbols located in the Alias folders!



RULES FOR DESIGNING SYMBOLS

This section reviews rules for how to design your own symbols according to standards, as well as how to do it in Automation.

The most important folder with symbols in Automation is called IEC60617. Here, the symbols are made so that the position and size of the texts match the standard IEC 60617 (electric diagram symbols) and IEC/ISO 81714-2 (design of symbols).

In general, we recommend that you follow guidelines from the standard(s) that symbols must follow. The following is primarily based on the fact that the symbols are to be used in electrical documentation, i.e. that the symbols are used together with symbols from EN 60617, but you can also use other standards as a starting point.

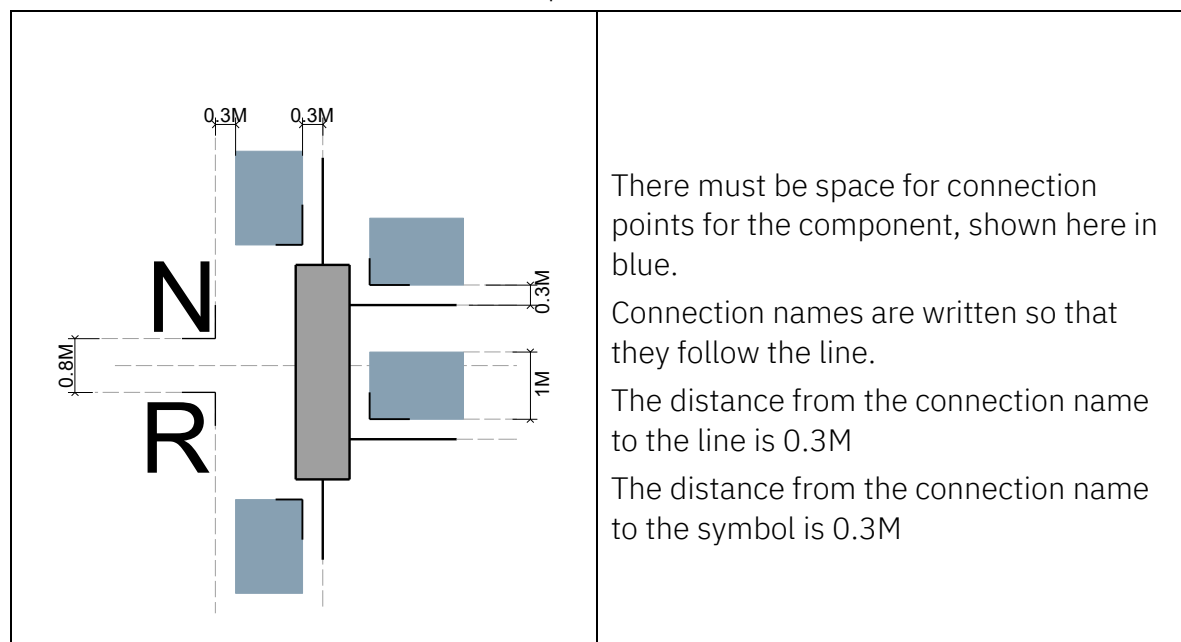
Standards for electrical symbols

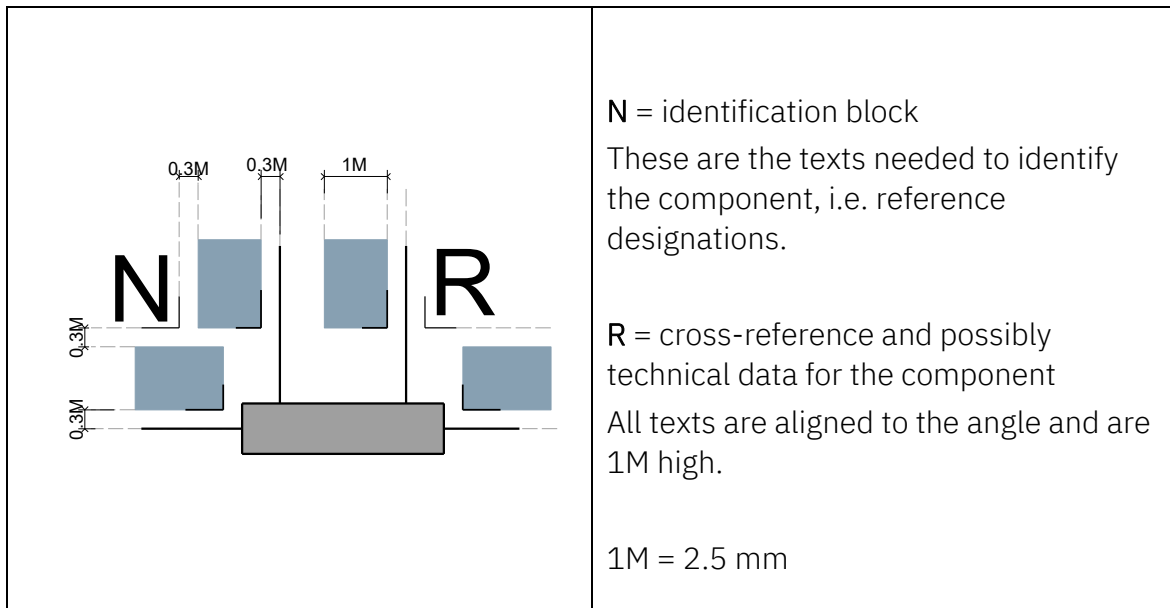
When creating symbols for electrical documentation, these are the relevant standards for symbol design:

- IEC 61082-1 on general drawing rules in electrical documentation
- IEC/ISO 81714-2 on the design of symbols
- IEC 60617 for electrical symbols
- IEC 62711TR on additional symbols etc. for measuring relays and instruments
- ISO 14617 for non-electrical objects in diagrams
- ISO 5807 for flow-chart symbols

Basic design rules for electrical symbols

Below is a brief overview of the most important rules:





Module size, symbol size, text size and line thickness etc

Symbols are constructed with the module size $M = 2.5$ mm, and this size is also used in Automation. This is the recommended size in the standard and also the size shown in IEC 60617.

This means, among other things, that distances between connecting lines, lines and points are a multiple of the mentioned module. The minimum size of a symbol must be adapted so that the rules on line thickness, line spacing, and rules on writing, etc. can be observed.

The existing symbols in the IEC60617, MISC and PLC libraries have been created according to these guidelines.

Lines in symbols

The line thickness is $0.1M$, i.e. 0.25 mm.

The symbols must be drawn in black with a line width of 0.25 mm and with the attribute 'Follow connected'. 'Follow connected' means that the symbol's lines can take on the color and thickness of the (last) connected line. The function can be turned off in the program.

Font and text sizes

The font should be selected as OSI font (or Arial). The font height should be set to $1M$, i.e. 2.5 mm. The width should be set to AUTO (the ratio between height and width is $2/3$).

Connection points and reference point

Connection points in the symbol are ALWAYS placed on the grid.

All dimensions for symbols must be divisible by the module dimension. The standard says at least $2M$ (i.e. 5 mm) between connections to provide sufficient space for terminal designation.

Reference point (*) and connection points should be placed with a distance between them that is divisible by 2.5 mm.



When creating your own symbols

You can create symbols by

- starting from scratch by opening the symbol editor, drawing the symbol and moving the symbol data fields into place yourself
- or you can work from an existing symbol
- or start by using the Symbol Generator

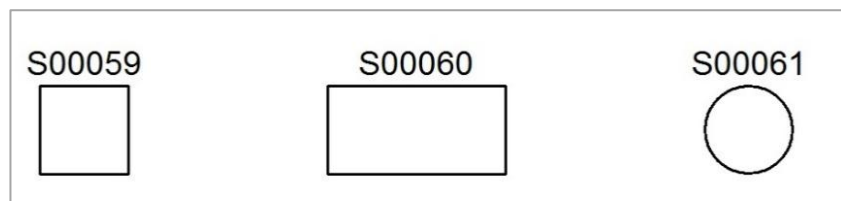
As a starting point, choose one of the last two options - it is easier, and the result is usually better.

Basic symbols

There may be situations where you cannot find a specific symbol because it is not standardized. This can be, for example, symbols for PLCs and safety relays.

You can use one of the three general symbols S00059, S00060 and S00061, which you can see below.

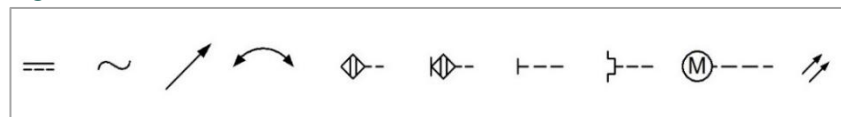
The basic symbols can be created by the Symbol Generator, or downloaded as an existing symbol.



Additional symbols – Qualifiers

The basic symbols are intended to be combined with other

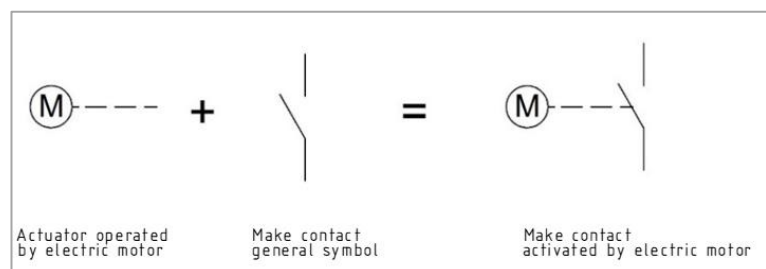
information, it can be one of the additional symbols in IEC 60617 or other standards, especially ISO 14617 or possibly just an explanatory text.



Combined symbols

In the IEC 60617 standard there are quite a few symbols, namely additional symbols, which are intended to be combined with another symbol to provide additional

information. These are called compound symbols. When you combine two standard symbols, you generally do not need to explain their meaning in the documentation or on an information sheet. But it is better to do it once too much than once too little. Remember that what you write in the Title will appear in the Symbol documentation, see page 56.



Symbols for sensors and instruments

Often symbols are missing for different sensors and instruments of different types. The IEC 62711TR standard, TR means Technical Report, shows examples of how to build symbols for these.

The basic symbols are shown below. S00327 is from group B (convert variable to signal), the other three are from group P (present information).



These symbols form the basis for a number of different sensor types and instruments, which are already found in IEC 60617. IEC 62711TR specifies additional symbols and letters, so that you can continue if you need to.

The small star in the middle simply needs to be replaced with the relevant additional symbols or letters, e.g. a voltmeter is an S00910 + a V. The symbol is called S00913. A short summary is that designing symbols where you simply use the existing ones is a safe and relatively easy way to go. Look up the symbol standards, find the basic symbols, and put them together so that they cover the need..

Be careful – don't make too many details

The only thing you need to be careful about is that you don't put too much information into the symbols. The biggest 'danger' is that you put too much information in, and in that way make the purpose of the symbol unclear.

The symbol should simply show the main purpose of the component and the reference designation on the symbol should help to find the type via the component list. With the product type it is now possible to see the technical information that may be relevant in a given situation, for example when troubleshooting the component.

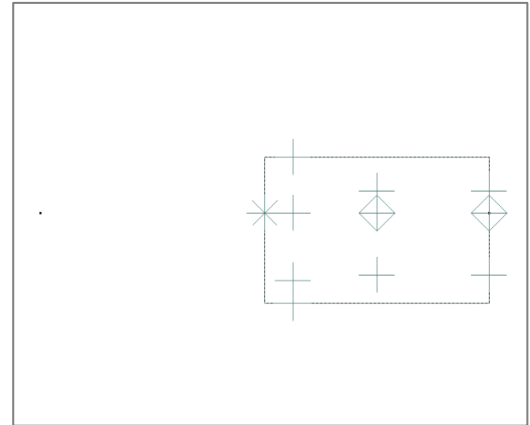


Working in Automation's Symbol Editor

When you open the symbol editor in Automation (F8 and New), it looks like this:

There is a *, which is the symbol's reference point, and the fixed symbol data fields (+'s), which must be moved to the correct location on the finished symbol.

Here you can also see two connection points (◇) and the texts that belong to them.



The fixed symbol data fields

A little explanation about the fixed data fields:

S.name

The data field is equal to N in the drawing above.

The name is specified with a single or multiple letters, which are found in the IEC 81346-2 standard and which depends on the function of the component.

Format, size, etc. are set under Settings|text/symbol default. The default is uppercase letters and that it is always displayed.

'-' is placed automatically by the program (set under Cursor/Screen).

S.Name also contains the object's ref. designation in diagrams.

Ref

The data field is part of R in the drawing above.

Cross-reference with page and path to other symbols with the same name. It is set to '/' in front of ref. to page and '.' in front of reference to path by default.

Format, size etc. are set under Settings|text/symbol default.

The field is only visible when there is content.

S.func

The data field is part of R in the drawing above.

Function text for the component.

Format, size etc. are set under Settings|text/symbol default. Normally visible.

S.type

Type name for the component. Retrieved from the database if the component is created here. Visibility is set under Settings|text/symbol default. Normally not visible, but belongs to R in the figure above if it is displayed.

S.articleno

The part number for the component. Retrieved from the database if the component is created here. Visibility is set under Settings|text/symbol default. Normally not visible, but belongs to R in the figure above if it is displayed.

R

In the figure on page 13 is composed of – from top – Ref, S.Funk, S.Type, S.Vare. Any other symbol data fields (e.g. Setting) also belong here.

Connection point data fields

Each connection point has the following data fields – the first two are required by the standard:

T.name

The name of the connection point, i.e. what is written on the 'pin'. In the figure above, these are the blue fields.

The text height is one module, i.e. 2.5 mm.

The text is placed so that it is on top of the connected conductor.

The setting in the program is that T.name is visible when there is content. Content can be retrieved either via the database (PINDATA) or by entering it together with the symbol name, e.g.: P1:X1,X2.

The order of the pin number is determined by the numerical value, which also has significance when the connection names are retrieved from the database.



The symbols must be drawn with increasing connection names 1, 2, 3, 4, etc. The numerical order determines where the correct connection names are placed from the database. The connection names are placed on the symbols according to the examples below:

All connection names in 'our' symbols (IEC60617) are invisible. The program makes them visible when data is entered, either manually or from the database. This way it is easy to see whether the symbol contains data from the database.

T.func

Function text for the connection point. For PLCs, the field for the I/O address of the connection is used. The text is usually visible.

T.label

One of two additional texts that are mostly (only) used for PLCs. Mostly a short text.

T.desc

The second of two additional texts that are mostly (only) used for PLCs. Mostly a longer text.



Symbol settings

While the symbol is saved with name, title (description – in multiple languages) and type, there are other settings than the symbol type that need to be considered.

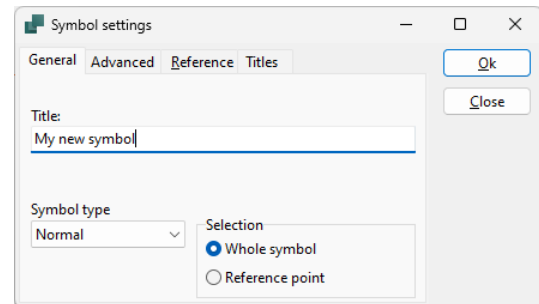
Symbol type

Here you select the symbol type – read more from page 22.

Selection

Symbols can be selected either on the Entire symbol or on the Reference point.

Selection on the reference point may be relevant for larger (mechanical) symbols, e.g. base plates, panel fronts and drawing headers to the page. Other symbols are selected on the Entire symbol.

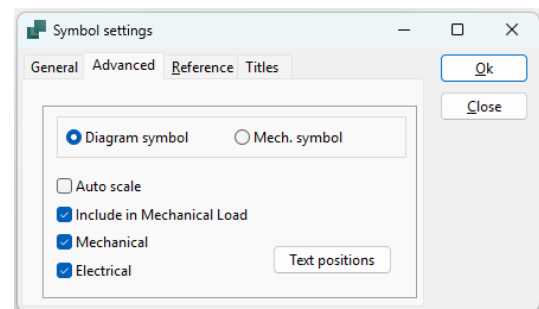


Advanced settings

The default settings on the tab are shown here, as most symbols are saved with these settings.

Auto scale

A special function for (some) Non-conductive symbols. It makes the symbol scale to fit the line it is placed on. The symbol must be designed with a line width of 10 mm.



See the two symbols T-conn.sym and Corner.sym in the MISC folder to see the functionality.

Diagram or Mechanical symbol

Diagram symbols have the same size independent of page size and scale. Meant for DIA pages.

Mechanical symbols will always scale according to the page's scale. Meant for MEC/GRP pages.

Include in Mechanical load

If you create symbols for components that are not to be in the panel, you can turn this option off for their electrical symbols. This way, the symbol for e.g. a motor or sensor will not be included in Mechanical load.

Same checkbox is found on the symbol in the project.

Mechanical

Is normally on, so that the component referenced by the symbol is included in component and parts lists.

Same checkbox is found on the symbol in the project.

Electrical

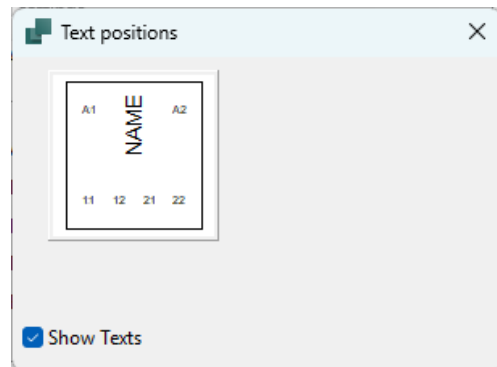
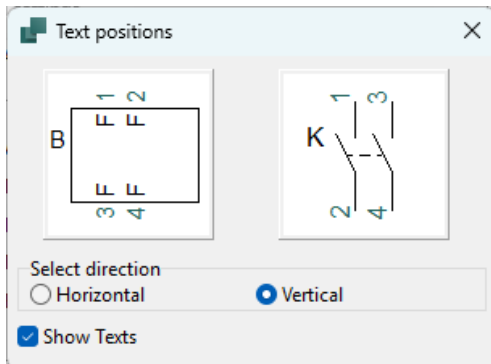
Is normally on, so that the component referenced by the symbol is included in the 'electrical' connection lists, namely terminal, cable, PLC and net lists.

Same checkbox is found on the symbol in the project.

Text positions

A tool to adjust all text positions and font, color and size to defaults.

Depending on whether the symbol is a Diagram symbol or a Mechanical symbol, you have different options.



Reference

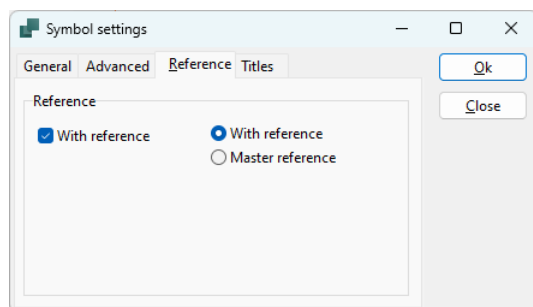
Symbols can be created with active cross-reference, either on this tab or when the symbol is placed.

With reference

Means that the symbol refers to the previous and next symbol with the same name, or possibly only to the symbol that has the Master reference and the same name.

Master reference

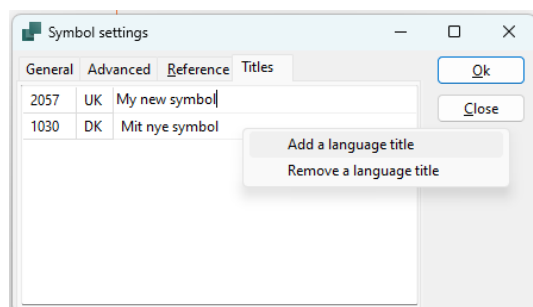
Means that a reference is shown to all other symbols with the same name.



Titles

All symbols work in all languages. If you create your own symbol, the title (description) is entered in the field corresponding to the language of the program – in the example, the program is in Danish, therefore the first description is in Danish. You can then add a description for the symbol in other languages.

If a language title is missing, it is replaced by US (1033) (if it exists, otherwise the one that exists).



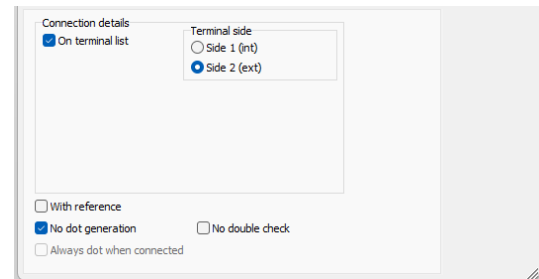
Settings on connection points

Both when you create a symbol and when you use a symbol, you have the option of different settings on the connection points themselves.

Connection details

If you want a component to appear on the terminal list – for example a relay in a terminal row – you can click on the individual connections and get their connections on the list.

The symbol can also be created with this property, and it can also be set via the database.



With reference

Used in connection with plc symbols, see from page 41.

No dot generation

When two leading lines meet, a dot automatically appears; also on connection points, if you haven't set this option.

SYMBOLTYPES

When designing symbols, you need to consider which component type they will be used for. The different symbol types are treated differently by the program and are a fundamental part of the intelligence in the program. Below is a list of the different symbol types and what the settings mean in general. You can read more about the different symbol types and their possible special rules in the individual sections later.

Symbol type	Description
Normal See page 30	Symbols that do not need to have any special status. These are symbols for, eg., sensors, lamps and motors; symbols that are 'just' needed on parts and component lists and that are connected components on the various connection lists. Mechanical symbols and common drawing headers and lists are also Normal symbols.
Mechanical symbols See page 57	Mechanical symbols that are placed on arrangement pages. They are saved as type Normal. Mechanical symbols are the same size as the component they represent. There are various rules that must be followed if the arrangement is to work well.
Normal drawing headers (Title blocks) and lists See page 61	The regular drawing headers/frames/title blocks and corresponding lists are created as Normal symbols. However, there are various rules that must be followed if drawing headers and lists are to work, as you can see described.
Relay See page 34	The symbol will have a make/break cross when placed on the page. Activation coils have this type. There can only be one relay with a given name in a project. A component can contain multiple sets of coils and contact sets, each set having a sub-name.
Open (NO) See page 34	The symbol will be interpreted as a make contact that is referenced to the make side in the NO/NC cross for the component it is part of. If a relay symbol is not inserted, multiple make or break contacts will refer to each other.
Close (NC) See page 34	The symbol will be perceived as a break contact, and behaves similarly to a make contact.
Switch (NONC) See page 34	The symbol will be perceived as a switch, and will be included as both a closing and a breaking function, with = in between, and otherwise as above.
Reference See page 35	Symbol type for NO/NC cross. There can only be one Reference Symbol in the list under Text/Symbol Standard!
Data See page 36	Symbol used to add information about a component already placed on the drawing.
Signal See page 48	Symbol type for a signal that terminates conductive lines and thus 'extends' the line. Two signal symbols with the same name, e.g. L1, will have a reference to each other. Signal symbols are placed on conductive lines and in this way give it a signal name. Named signals can be excluded from automatic wire numbering.



Symbol type	Description
	You can create a list of preferred signal symbols and names under Text/symbol defaults.
Terminal See page 36	A terminal symbol is a symbol where all connections have the same name and thus the same potential. The terminal has its own connection type list – the terminal list – and the terminal graphical plan. In order to differ between the connections, they have properties referring to Side1 (Internal) or Side 2 (External), so that Terminal lists show the right connections to other components. There are also considerations about the mechanical symbols for terminals and finally about creating terminals in the database.
Cable See page 46	The symbol and its connections are shown on cable lists and plans. It is directional and this is used for sorting. The symbol may have a connection point for the shield. The symbol itself has a variable size to suit different cable types.
Plc reference See page 41	The symbol is used as a reference/overview symbol for PLCs. The symbol 'controls' all addresses on the PLC symbols. All (relevant) connections per channel are specified on the PLC-ref symbol so that they can be transferred to PLC symbols. The symbol and its connections (via PLC symbols) are displayed on PLC lists.
Plc See page 41	The symbol contains the connection points that must be connected in a PLC for each channel/address. The content of the symbol is a mirror of the PLC reference symbol and there is automatic reference to the PLC reference symbol, so that all data may be transferred between the two symbols.
Wire number See page 50	Symbol used to indicate wire numbers. Used for automatic and manual placement of wire numbers, where the program suggests the next available wire number. Appears in the wire number list. Content format is defined under the Wire Numbering function.
Insertion point See page 51	Symbol that functions as a delimiter in connection with sub-drawings: On diagram pages they define the area in which sub-drawings can be inserted; on the sub-drawings they define the size of these.
Line connection See page 52	Symbol used in connection with part drawings with model selection. When the part drawing is placed on the page, the symbol is converted to a line of the desired length.
Non-conducting See page 57	Symbol that is connected to a non-conducting line (wall), ie a door or a window.
Head See page 54	Parts, component, or plc list that can be placed on a DIA or GRP-page.
Busbar See page 47	The symbol is used where you do not want to draw potentials as lines, but instead as the 'busbars' they actually are.



HOW TO – DESIGN ELECTRICAL SYMBOLS

Below, the general functions for creating symbols are first reviewed, based on Normal symbols. Then, later in the document, you can see how the more special symbols are created.

The order is

- how symbols are created
- any special settings
- how the symbols are processed by the program and the database

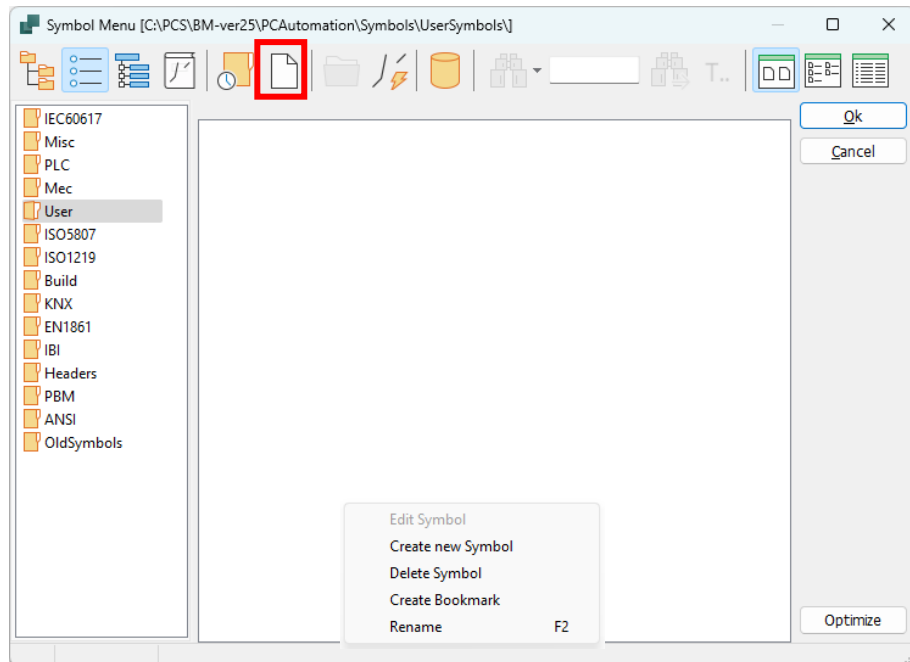
The different options for editing, composing, creating states and the interaction with the database are shown together with the review of a normal electrical symbol.

Open the Symbol editor

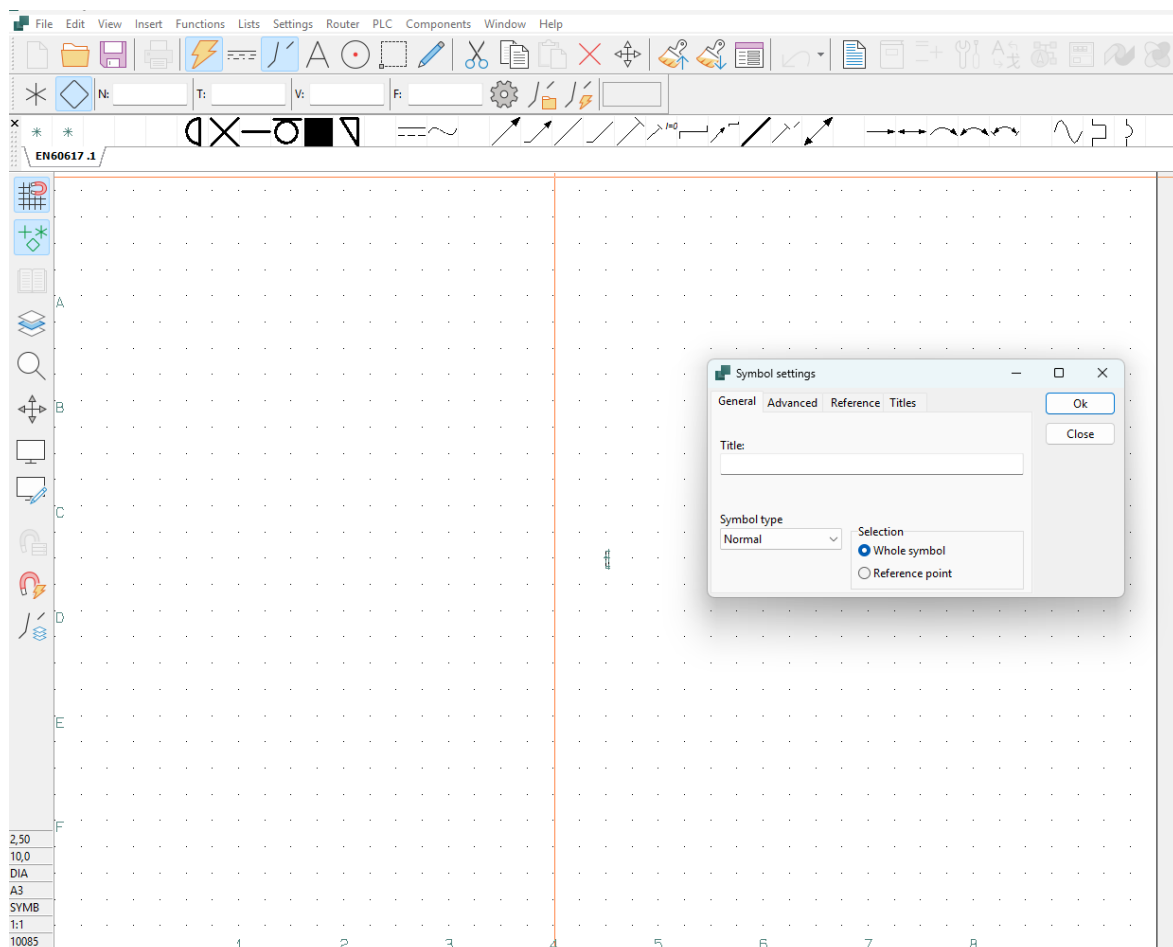
Start by opening a new and empty project.

Press F8 or the button Symbol menu ... This window opens.

Then you either rightclick or press the Icon new in the toolbar.



The symbol editor looks (a lot) like the main program, and you have the same drawing features here – and you use a couple more when you design symbols, than when you make diagrams.



Editing options in the Symbol Editor

When an object is selected or in the crosshairs, it can be

- placed on the drawing using <click>
- flipped/rotated using <space>
- mirrored around the x/y axis using <icon on the toolbar>
- deleted, copied or moved using <common Windows commands>
- aligned, e.g. connection points or texts to these
- distributed, e.g. connection points or texts to these
- transferred properties between objects
- swapped, e.g. you can swap two connection points or texts.

In general, you have the same options in the editor as you have in the program itself. You also have the different line types that you use when/if you create or edit a symbol. Remember that when you put symbols together, it is 'exploded' so that it consists of the original objects: lines, circles and texts.

Pickmenu in the Symbol editor

The Symbol Editor also has a pick menu. You can also customize this pick menu to have more tabs and more symbols.

Tip

Create a bunch of 'building blocks' and place them in the pick menu so you can pick from them when you create your own symbols. These can be connection points, with preset texts and positions, they can be line segments, qualifiers, etc.

Create a new symbol using the Symbol Generator

If you need to create new symbols based on a basic symbol – S00059, S00060 or S00061 – you can use the Symbol Generator as a starting point.

The Symbol Generator uses your text settings regarding font, color and size.

The Symbol Generator places all texts according to the rules in the mentioned standards, see page 13.

You can find the Symbol Generator in the Symbol toolbar or in the toolbar in the Symbol menu.



In the Symbol Generator, enter the symbol size, the desired number of connections, their location and numbering, the symbol name, etc.

The dialog box continuously shows what the changes mean.

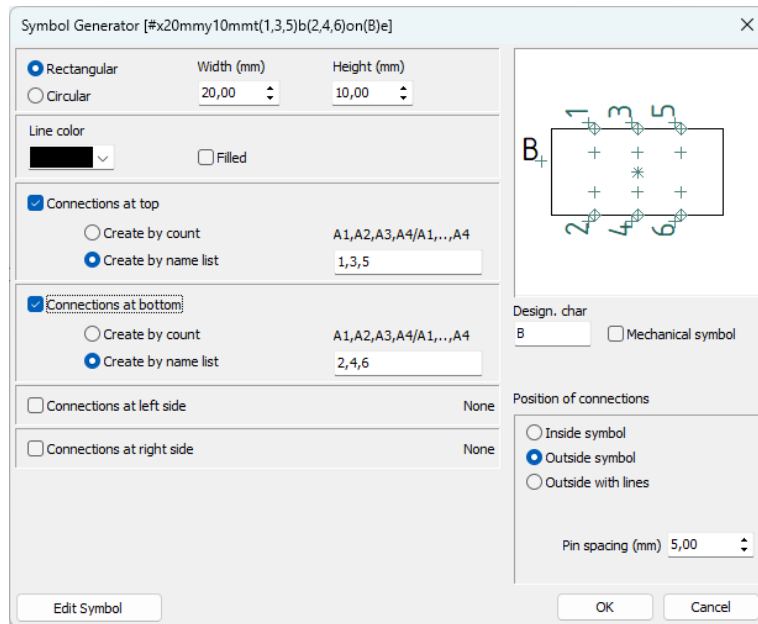
If you press the Edit symbol button, you will go directly to the Symbol Editor, where you can insert additional symbols, etc.

It is also possible to set the distance between terminals.

Symbols like the one shown can be placed directly in the project; you can also select Edit symbol and go to the symbol editor.

Create the symbol

1. Open the Symbol generator – press the icon
2. Make entries as shown in the image.
All texts are placed according to the rules, meaning that they are placed as with the ‘text position’ tool.
3. Text font, size and color follows the current project.



When you create the symbol using Symbol Generator – and in general – think about your diagram: you want to avoid crossing lines.

Therefore, you want to control where your connections are: here the odd connections are at the top, and the even ones at the bottom. This gives the best diagram in most cases.

Create a symbol using an existing symbol

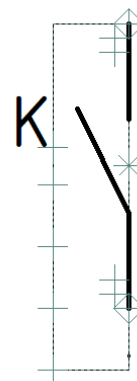
Find the symbol in the symbol folder – preferably the one shown – right-click and select Edit, or select Edit by right-clicking on the symbol directly in the diagram.

You will now enter the Symbol Editor for the symbol.

The symbol itself is clearly visible, the symbol's data fields are placed according to the rules: K is S.Name, which corresponds to N.

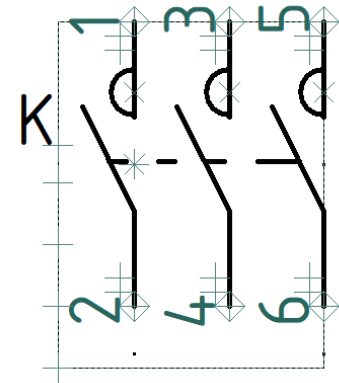
The other texts are from above Ref, S.Function, S.Type, S.Item.

Connection names are invisible, but are called 1 (at the top) and 2 (at the bottom). They are placed 0.3M (0.75 mm) from the symbol's connection line.



Using the drawing functions in the Symbol Editor, this symbol has been created:

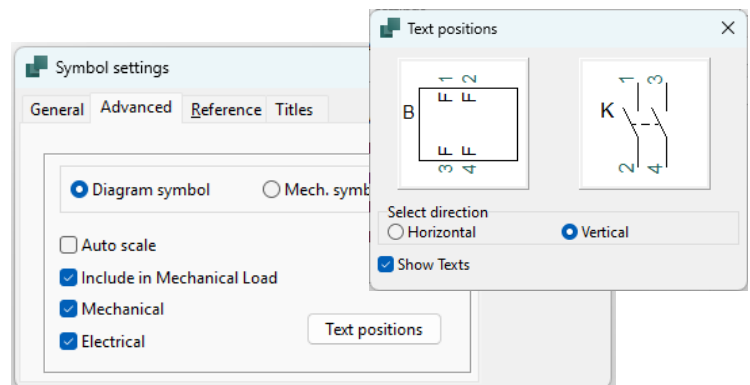
3 make contacts have been connected together, with a distance of 2M (5 mm). They are connected together with a dashed line to indicate simultaneous activation. The qualifying symbol for Contactor has also been added, so that the final symbol is a '3-pole contactor'. Connection names may be made invisible before saving the symbol.



The advantage of working this way is that you build on things that have already been agreed upon.

You can save the symbol under its own (new) name, preferably in the USER_SYMBOLS folder. Remember, the symbol type is Open.

Use the function Text positions on the Advanced tab, to place all symbol and connection texts correctly .



Please note, that from version 23 you cannot edit 'our' symbols and save them. However, you can save them with a different name.

Normal electrical symbol

Normal symbols are symbols that don't have a special behaviour or requirement. A normal symbol might refer to eg a lamp, a motor or a sensor.

A component may consist of more than one (normal) symbol.

Components that use normal symbols will occur on parts and components lists, and may be connected components in all connection type lists (terminal, cable, plc and net lists).

Create the symbol

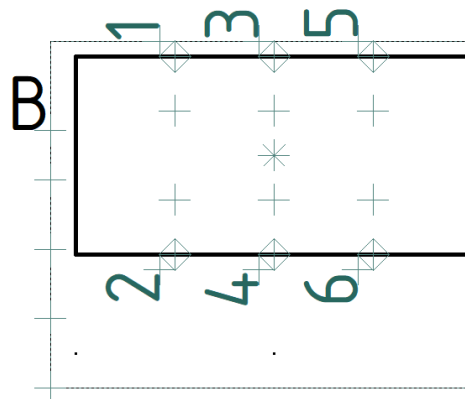
Create the symbol from above by using the Symbol Generator and select Edit symbol.

Now you are in the Symbol Editor.

Edit symbol

In the Symbol Editor you have all editing functions:

- you can draw/move/edit lines, texts, circles and connection points
- you can transfer properties
- you can move connection points if necessary. Use the commands 'Distribute' and 'Align'
- the connections can also change properties
- the reference point itself * can also be moved
- etc.

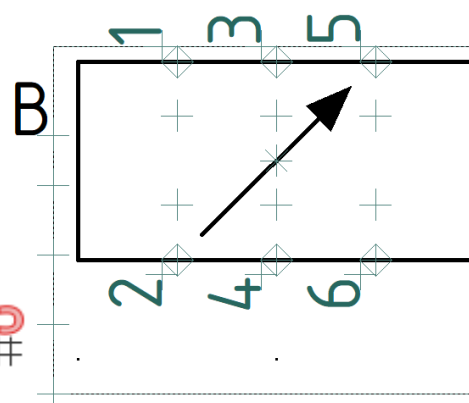


Combined symbol

You can add additional symbols, for example by retrieving them from the pick menu in the Symbol Editor.

You can use 'fine snap' when placing the additional symbol: Hold down Shift while placing it.

This is equivalent to pressing this button:



Invisible Connection Points

Connection points on symbols in Automation are named as shown above and are generally set to Non-visible. In the following, connection points are visible, as it is otherwise difficult to explain what is happening.

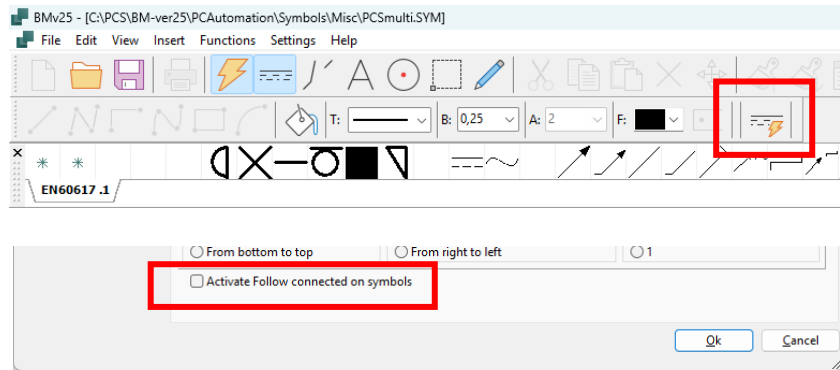


Follow connected

All lines and circles in symbols have (should have) the Follow connected property.

This way the symbol will change to Red if you connect a Red conducting line to it.

The function is by default disabled in the project, but can be enabled under Text/Symbol Defaults if desired



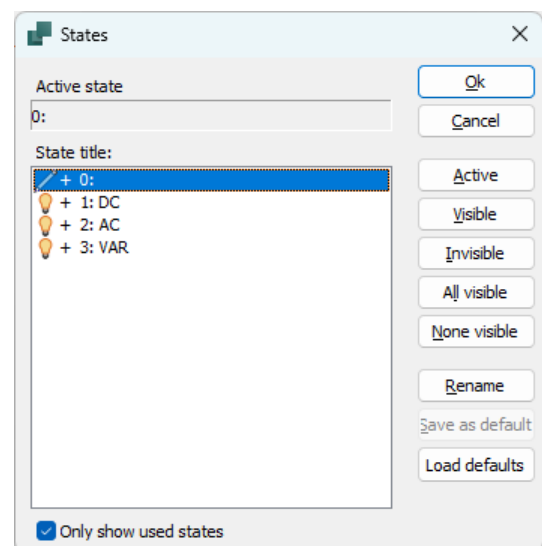
Symbols with states

You can use the 'States to Layer' function to create many symbols in one symbol.

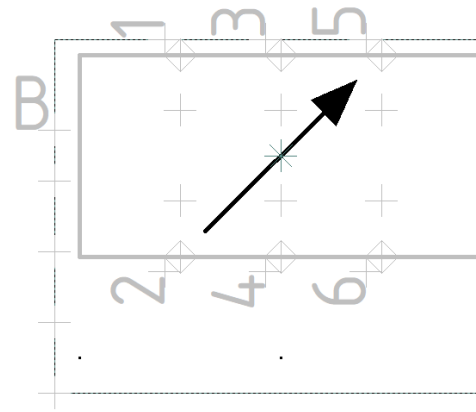
This can be used, for example, to create one symbol for all (2-wire) sensors, so that you can easily replace the component in the diagrams.

Here, however, we continue working with the symbol above

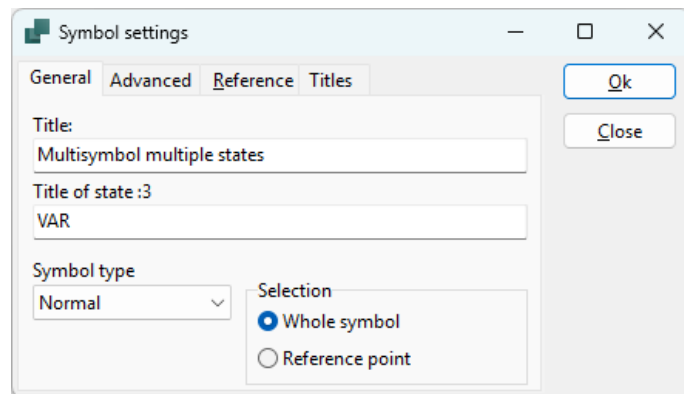
- Press the 'States to layer' icon 
- Press the icon 'State' 
- Select a state in the dialog – remember that you may only see the active layers; show all by removing the 'tick' at the bottom!
- This is what it looks like, when the symbol has three states
- Double-click on the desired state, here state 1



- Now the old symbol gets 'dimmed', as it is in state 0
- Place the new symbol (aka the qualifier) – the arrow (use the fine snap)



- Save the symbol; now you see the symbol title and the state's title
- Repeat if you want more states with individual names

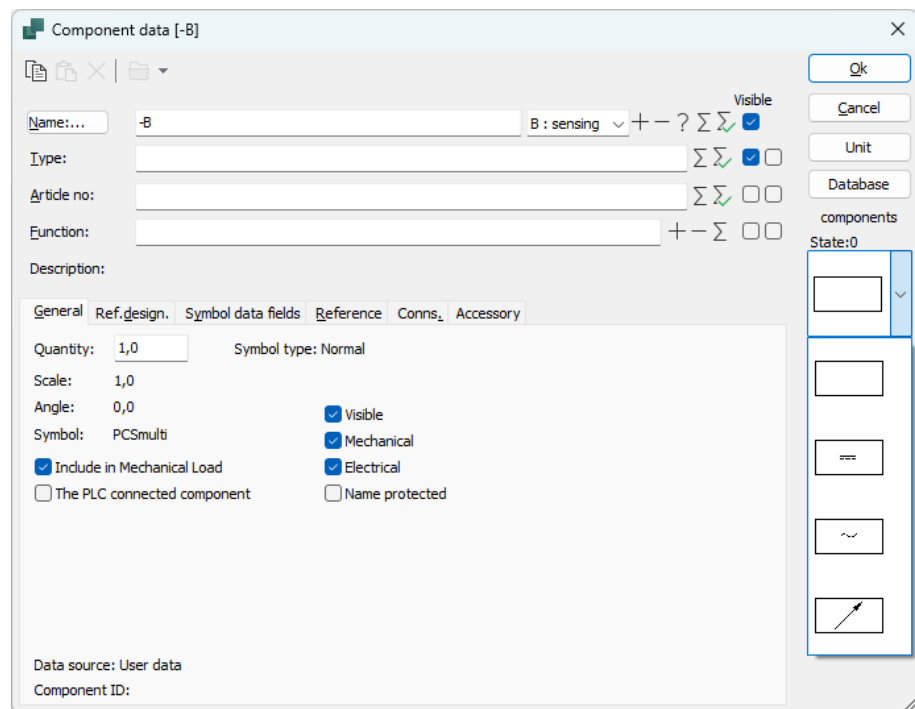


Place a symbol with states

When a symbol is created with states, a dark triangle appears in the corner of the symbol in the symbol menu itself. When you hover the mouse over the symbol, the titles of the different states appear.



When you place the symbol – directly from the Symbol menu and not from the database – you can select the desired state on the right in the dialog box.

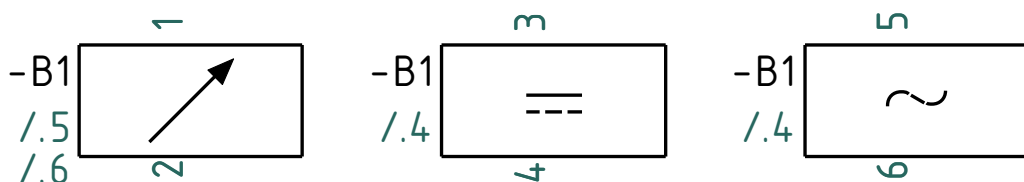


Interaction with the database

The database can control the symbol state, connection names, reference, etc.

If you use the Component Wizard and select the symbol for your component, you can make the symbols look like this.

This means that the component can look like this:



Symbols for Relays, NO (Open), NC (Close) and NO/NC (Switch)

Symbols for use with relays and contactors are created in the same way as Normal symbols, but they must be saved as Relay, Close, Open or Switch respectively.

These four symbol types are related, as they are part of the same component. The coil in the relay is given the symbol type Relay, and is the 'main symbol' in the component. It is also the coil that controls the name.

A symbol that is defined as Relay also draws a make/break cross.

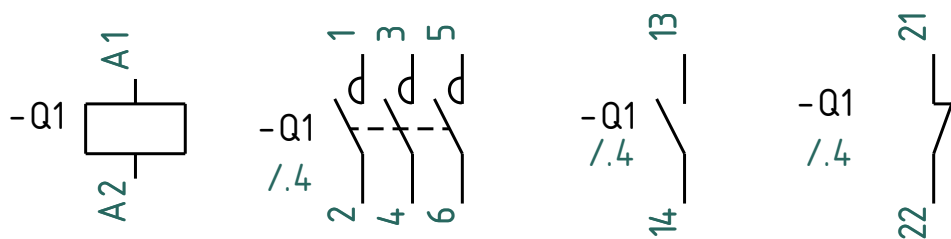
There are many different contact symbols in the symbol library (60617) that are normally used to create new contact symbols, e.g. 2- and 3-phase power circuit symbols, possibly combined with an activation.

Interaction with the database

Here are the three symbol types used for a contactor.

The symbols are 'born' with connection names 1 at the top and 2 at the bottom. This is the order that the database uses to write the component's connection names.

The database lists which symbols are used and their connection names. The component designation itself – Q – is also defined in the database.

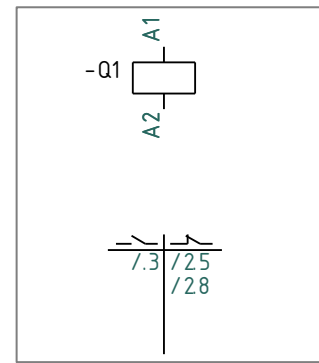


You can learn more about how to create components in the Component Guide manual, in the Getting Started with Automation booklet, in the database manual, and in various webinars and YouTube videos.



Reference symbol (NO/NC ref. cross)

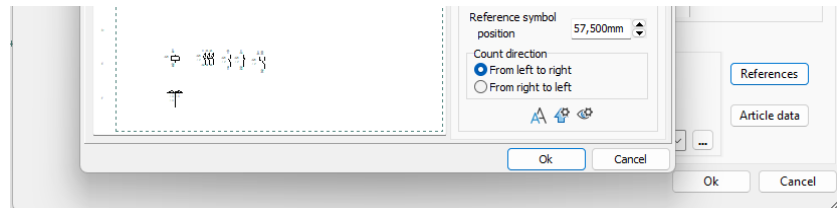
A symbol of the Reference type is a make/break cross. The make/break cross appears on the page when a symbol of the Relay type is placed.



Position for the ref.cross on the page

The position of the NO/NC cross on the page is set under Page data|References.

See also under the section Headers (Title blocks), lists and templates from page 61.



Create new reference symbol

If you want a make/break cross with NO/NC instead of the existing symbol, this is done (easily) in this way:

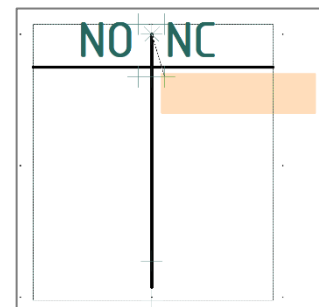
Find the existing symbol in the MISC folder, right-click and select Edit symbol.

Mark the contacts at the top (Lines) and delete.

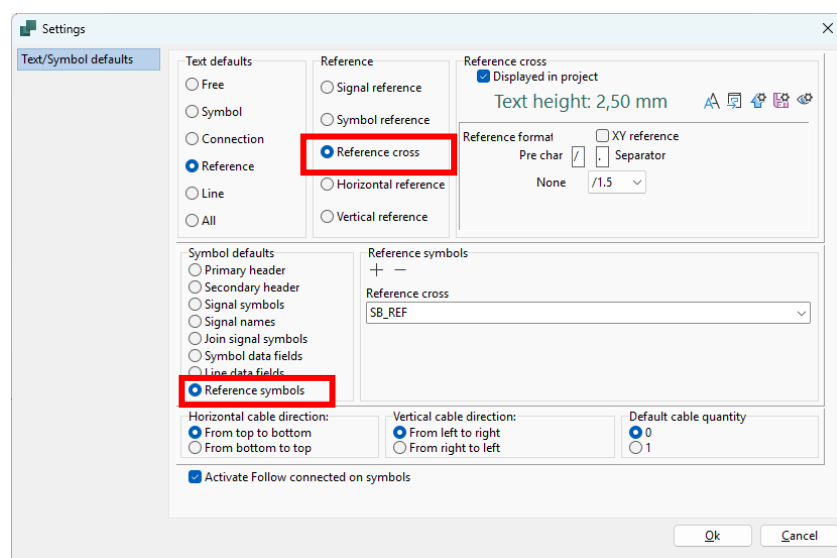
Place the texts NO and NC where the contacts were before.

For info, you can see the three references below – for Closers, Switches and Breakers respectively. Here the reference to the breakers is marked.

Save the new symbol, e.g. SB_REF NO_NC.sym.



Text formats and selection of ref.symbol are done under Test/symbol standard. Add the new symbol to the list and delete the old one. There must only be this one symbol in the list! Update the references under Functions |Special functions| Recreate references if you are in an existing project.



Data symbol

Data symbols are intended to display and/or enter additional data on components. The fixed symbol data fields such as Name, Type, Part No. and Function are not always enough to display relevant information about components in the projects. This can be solved in various ways, all of which are discussed below:

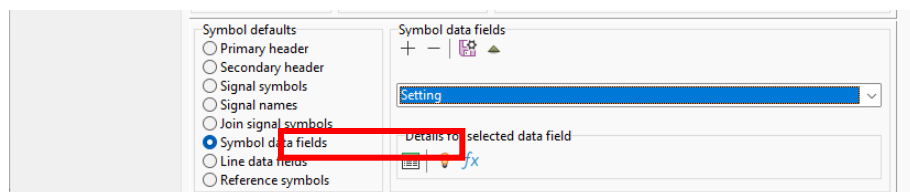
- you can create pure data symbols, which can contain additional data
- you can add additional data fields permanently to a symbol, possibly as part of a condition
- you can always add additional symbol data fields to symbols in the project.

Create Symbol data fields

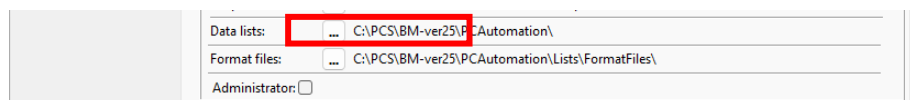
The desired extra data fields are created under Settings | Text/symbol default.

Here you can create extra symbol data fields, and they can be created so that they are either only present in the current project, or they can be present in all projects in the installation.

Symbol data fields can be created with a data list, possibly with a default value and the list can be locked for editing.



Data lists can be saved in their own (shared) folder. Selected in the Directory settings.



The Setting and Segregation fields are created by default; the first field is intended to specify a setting for the component in an optional unit; the second to specify traces on cables that are to be exported to Cablemanager.

The symbol data fields can also have a built-in formula; we have used this on the fixed (from ver23) fields on plc symbols (PlcID: plc-name.plc-net.plc-node.plc-slot).

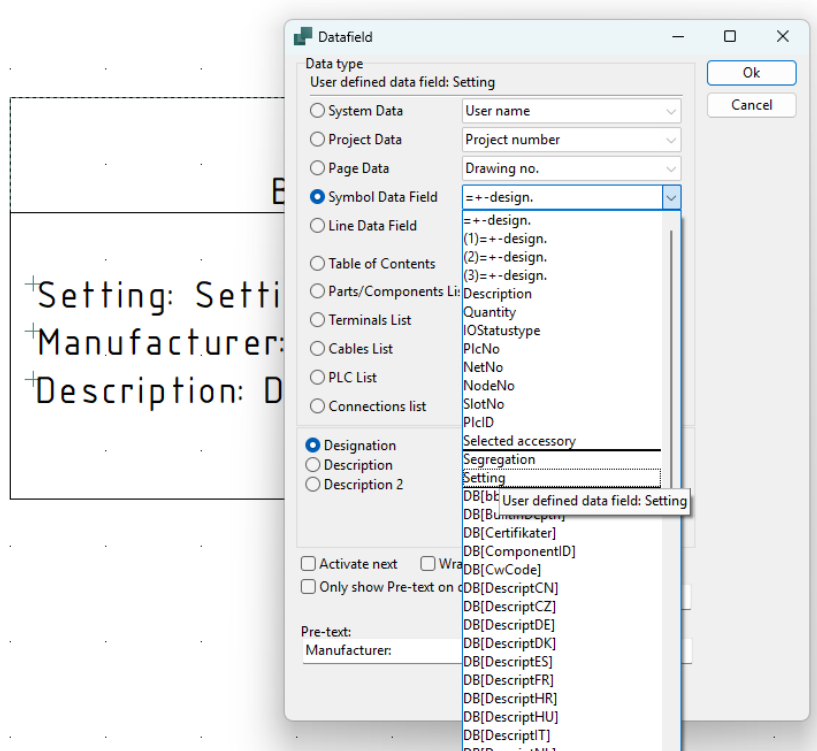
You can create line data fields in the same way, but it is not relevant in this context.



Create a data symbol

A pure data symbol simply contains extra data for a component. The symbol must have the same name as this component when placed in the project (same name = same component).

For example, you can use the Symbol Generator to create a simple, square symbol. In the symbol editor, you need to 'Insert Data Field' to insert one or more data fields into the symbol. You can select from the list and choose properties for the individual data fields. A pure data symbol simply contains extra data for a component. The symbol must have the same name as this component when placed in the project.



Add symbol data fields to a diagram symbol – in the symbol menu

You can add extra fields to all symbols.

Select edit symbol and place the data field as above, possibly in a given state, and then the symbol always has this extra field. The advantage is that you can control its placement.

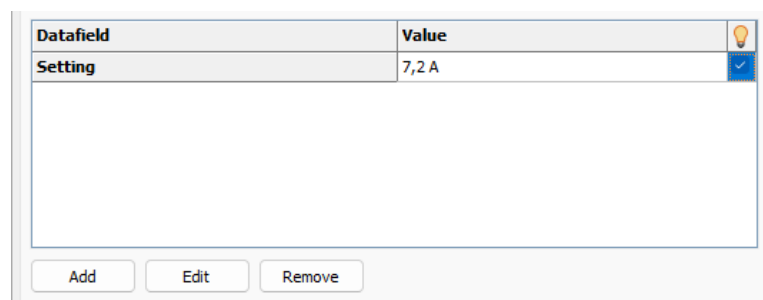
If you want to do this on many symbols, you can contact support, which has a tool – Symbol Manipulator – where you can process many symbols at once.

Add symbol data fields on a single symbol – in the project

All symbols have a tab (from ver. 18) where you can choose to add one or more symbol data fields.

The selected data fields can be made visible and in the project they can be moved.

The fields are of course also available in parts and component lists, so that they can also be used when you mark/label your components..



Terminal symbols

A terminal is a special symbol used to illustrate feed-through terminals. A feed-through terminal has the same potential on all connection points.

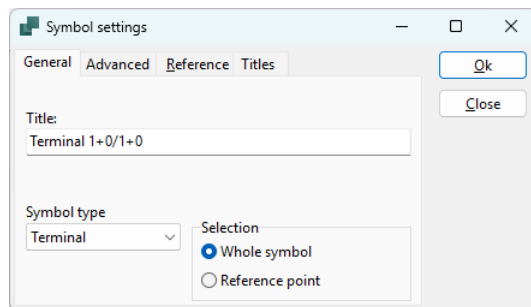
Terminal lists and diagrams can show connections to and from the terminal row.

In order to make both terminal lists, parts lists and the mechanical placement of components work, there are a number of rules associated with the creation of terminal *symbols* and terminal *components*.

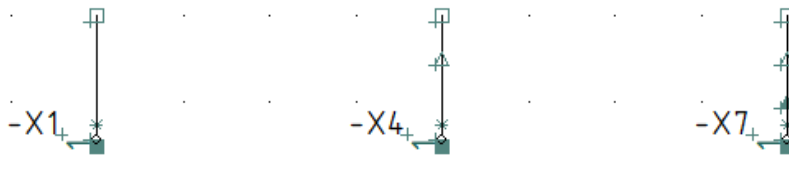
See also page 59, about how to create the mechanical terminal symbols.

Terminal *symbols* must be created in this way

- Start by selecting the symbol type = Terminal. This opens up for the functions described below.



- A terminal has the same electrical potential on all connection points. We illustrate this by giving all connections the same name, with only one of them being visible

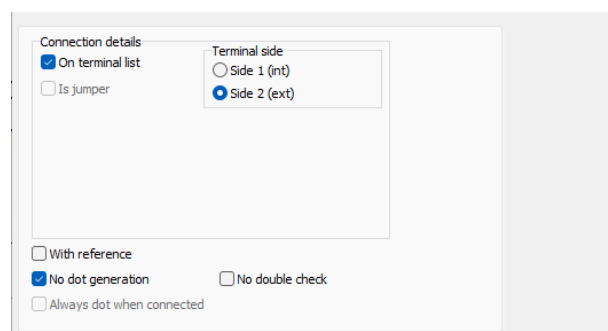


There must be at least one connection point on both sides: the empty connection point are on Side 1 (internal), the filled-out ones are on Side 2 (external).

- It is also possible to add jumper connection points on the symbol. Same rules, with the exception that only jumpers can be connected. Jumper connections are shown as triangles.

Side 1/side 2/jumper is selected on the individual connection point,

Side 1/side 2 means something in relation to getting the terminal list become useful.



With reference

If you place the same terminal twice in a project (one page for connections to the terminal, another page for connections from the terminal (copy identical)), you can set a visible reference between the symbol. The reference should be part of a general symbol, only the specific component.



No dot-generation

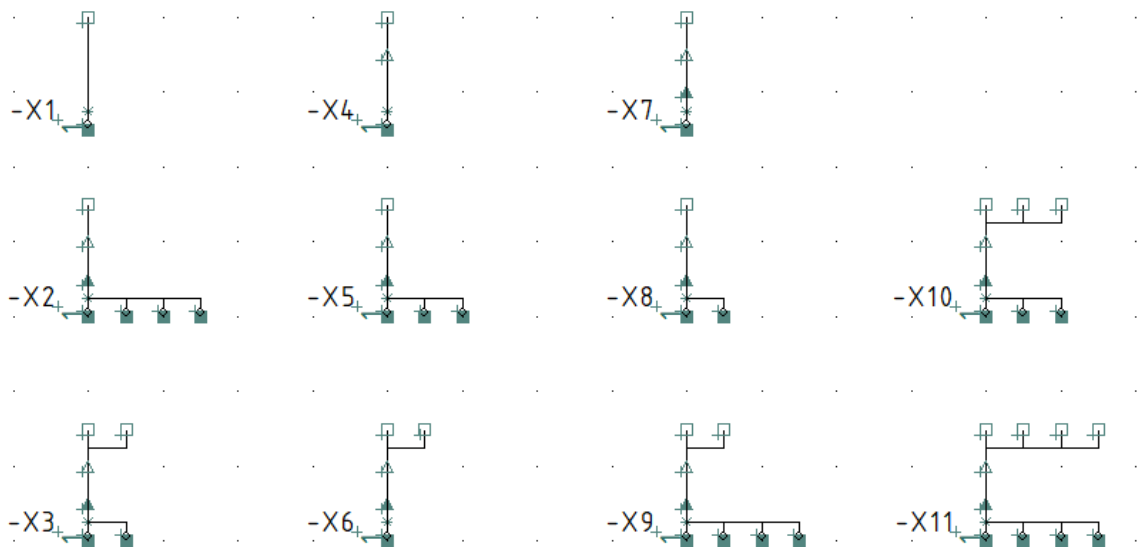
When two conducting wires meet, you automatically get a dot. But, when you select this option you won't get dots on connection points.

On terminal list should have **No dot generation**

Is jumper should have **Dot generation**.

The terminal symbols in symbol folder 60617_2025 look like below. For each basic type, there are 3 versions – without jumpers, 1 jumper (side1) and 2 jumpers (side 1 and side 2). The biggest change is that the visible text is on side 2.

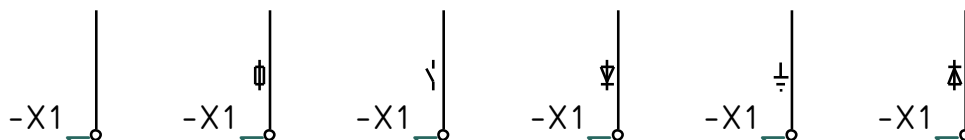
Their names are running from PCS-S00017-500.



We have also created special symbols for special terminals – terminals with/for fuse, separator, diode terminal etc. The different functions are in states. The symbols start with PCS-S00017-800.

We have created them with a terminal symbol, as they in reality have the same potential on all connection points!

More variants and more states might come.



Other components on the terminal list

You can select 'On terminal list' for all connections on all components in a project (individually) or you can set the option in the database.

Distribution blocks

Distribution blocks are Normal symbols that function like a terminal symbol:

All connections in a Distribution block have a unique name – like all other symbols.

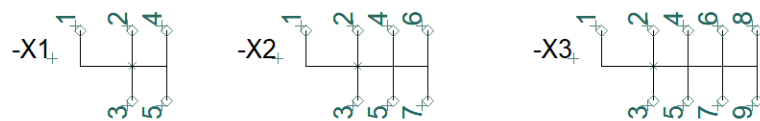
And all connections have the same potential – like terminal symbols.

That means, that

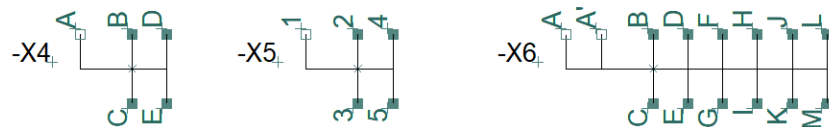
- when we create the symbol, we create a normal symbol
- when we create the component, we add the property Terminal to make it have the same potential on all connection points. (the Component Wizard has the component kind = Distribution block)

In the 60617_2025 folder we have those symbols

The top line are symbol – normal symbols with normal connection points (diamond shaped connection points).



The lower line are components – set to Terminal symbols, and connection points are on the Terminal list (square sized connection points).

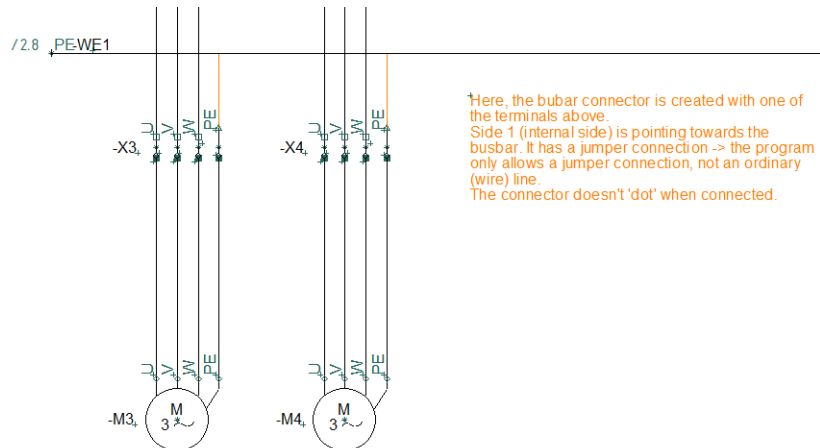


Busbar connections

We have made a special type of terminal intended for busbar connections.

The jumper connector does not 'dot'.

The symbols have an External connection – for wires or cables conductors or shields – and an internal connection, which only allows jumper connections. The busbar is connected with a jumper line from this connector.



Plc symbols

Plc-symbols are two different symbol types, and they show IO data only:

- the plc symbol, that connects to external components. They have an Input or Output status
- the plc reference symbol is a symbol used for overview of the cards/components used in the project. External components are not connected here, but maybe supply voltage

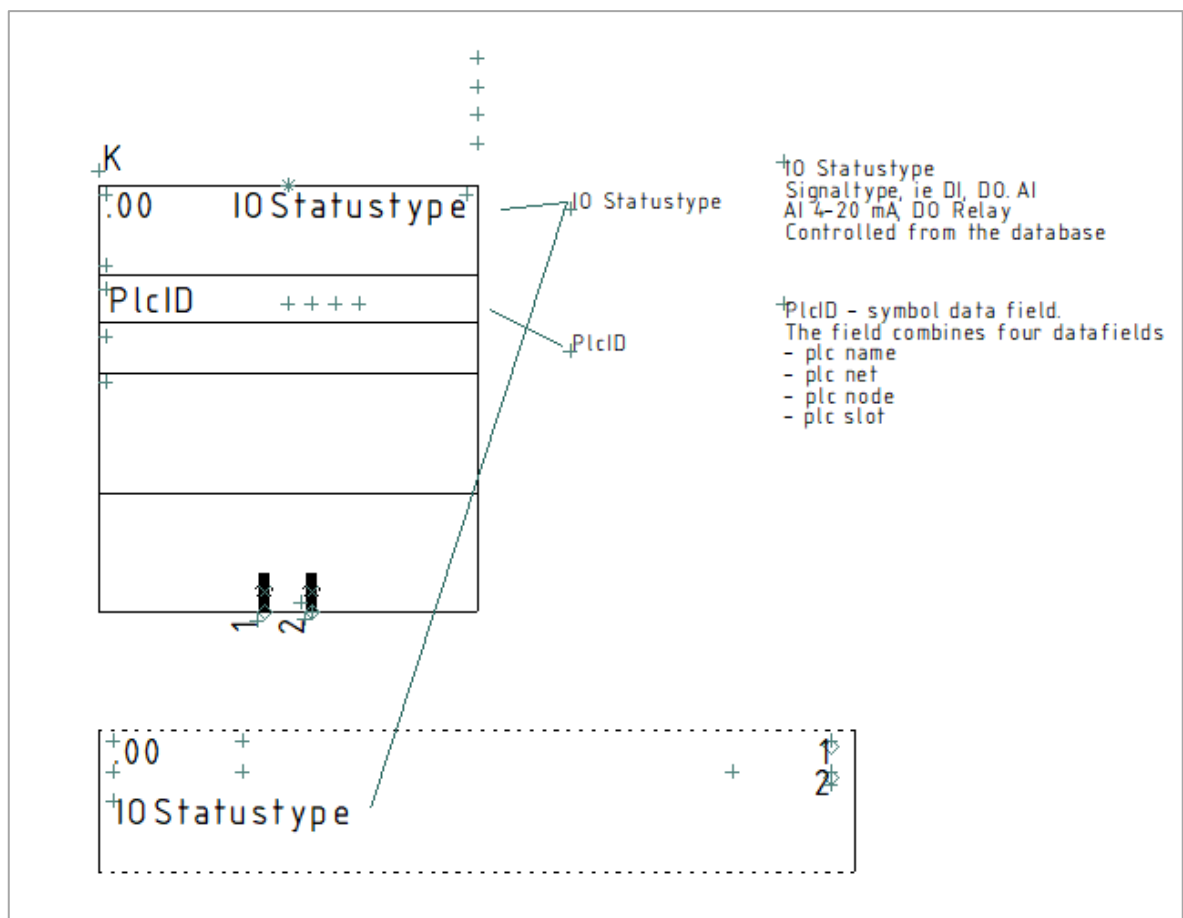
For the two symbol types to work together in the project, the symbols must be created correctly, and subsequently the components in the database must be created correctly. You can read more about the latter in the Component Wizard manual..

In connection with the release of ver 23, a new library of plc symbols has been created, and the following is based on these.

You can of course create your own symbols if you want a different graphic, but remember to comply with the other design rules!

Create the symbols as a set – example 1

Below are the symbols PLC-IN2 and PLC-REF-IN2 in the Symbol Editor. When creating a PLC component, several reference symbols are put together using Symbol Group - read more about it in the Component Wizard manual or see demo files.



To make the two symbols work together, the following rules must be observed

- there must be the same NUMBER of connections per address on both symbols, here two
- all connection points that belong to the same address must have the same IO address/function text

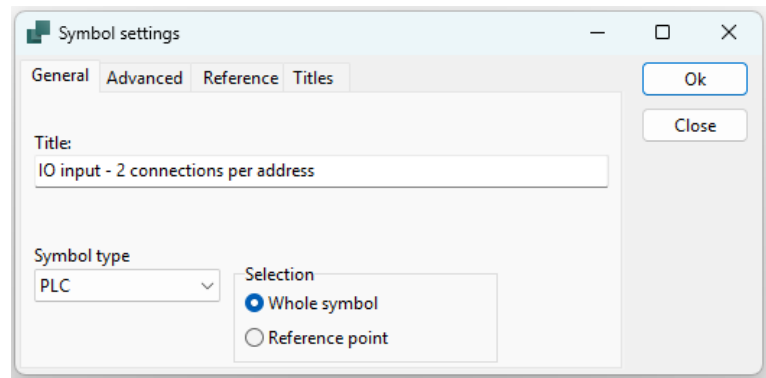
- the first connection point, i.e. the point that is most to the left of the plc symbol, must have IO status
- IO address (corresponds to function text below) must be set
- the point must be either Input or Output
- the point must have With reference, in order to be able to refer to the plc reference symbol
- status can be selected here, but it is better to set the status of the component in the database

- the extra connection points do not have IO status
- BUT remember the Function text, which MUST be the same as the IO address of the first point.
- this way all connections belong to the same address
- otherwise no check marks

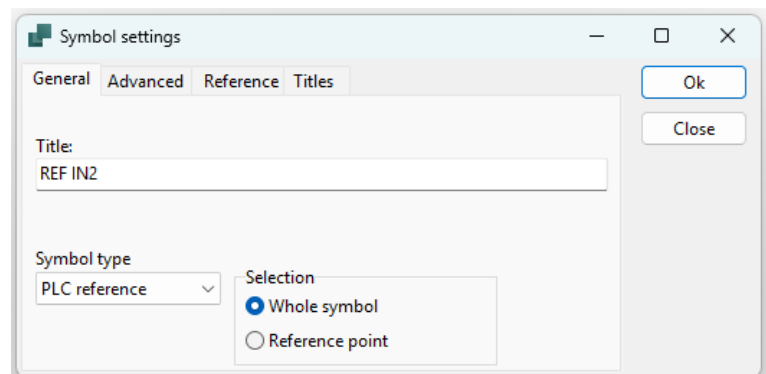


Save the two symbols

The Plc symbol is saved as symbol type PLC.



The Plc reference symbol must have the same connections with the same settings, and the symbol is saved as a PLC reference.



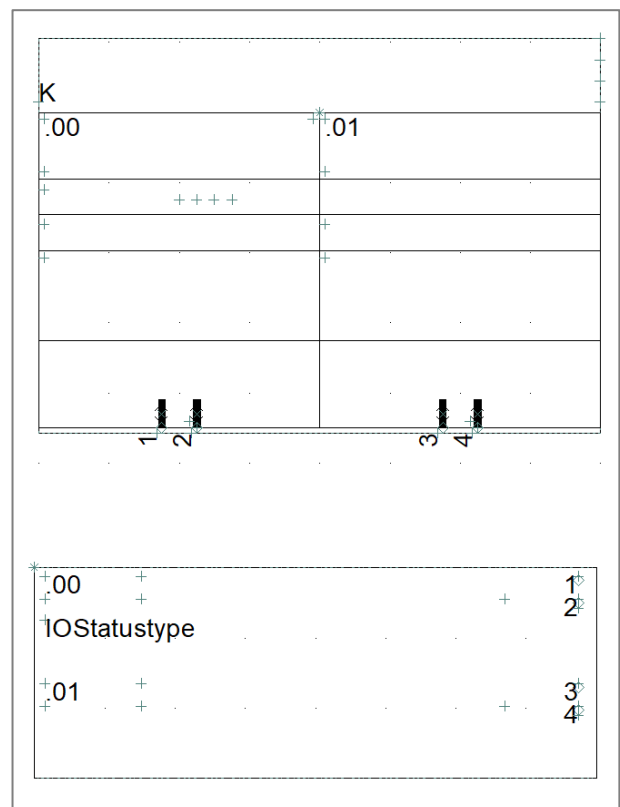
Create the symbols as a set – example 2

Below you can see the symbols 2XIN2 and PLC-REF-2X2IN in the symbol editor.

The two symbols are intended to be used as 'channels with two addresses. When the symbols are created, we use the same symbols as described above.

The symbols are made for 2 inputs or 2 outputs.

In the case, that you have a channel with an input and an output, we 'cheat' with the status types: DI/DO or DO/DI.



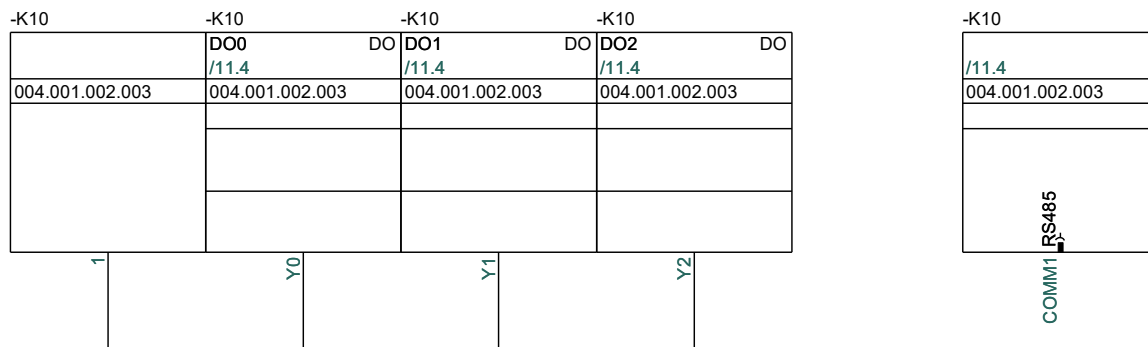
Other symbols for plc components

The plc symbols only show IO data. But the components can contain other data types as well.

From ver23, the plc symbol folder also contains symbols for some of the other functions, respectively IO-link, supply and top/bottom for the symbol groups. Some of those are made with Normal symbols.

Below are examples of symbols in a component from PCSplcdemo1.

The first symbol is an Aux – for supply and Common and ..., then three outputs and finally a Comm, where a connector for RS485 is shown, i.e. the technician must find a plug connector when the communication is to be connected.



The symbols for Aux etc are created as Normal symbols. If you need to give descriptions to IO-links etc (as you do with inputs/outputs) we have created some based on inputs/outputs, that are enclosed from ver25.

Nice – and need – to know, when you create your own symbols

If you need to create a plc with more than one ref-symbol

You can choose to create the plc with separate ref-symbols, ie for DI, AI, DO and AO etc.

If you do that, you must keep an eye on the number of addresses and the order of top and bottom symbols.

If you want to create your own design of plc symbols and our plc functions

You can – as always – create your own symbols, but in order to have the functionality in the program, there are some rules to obey to.

Requirements to the reference symbols

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.

The rules for grouping of ref. symbols –

Grouping in the project is based on how the component is created in the database

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 don't fit, the Next button is not active.

Cable symbol

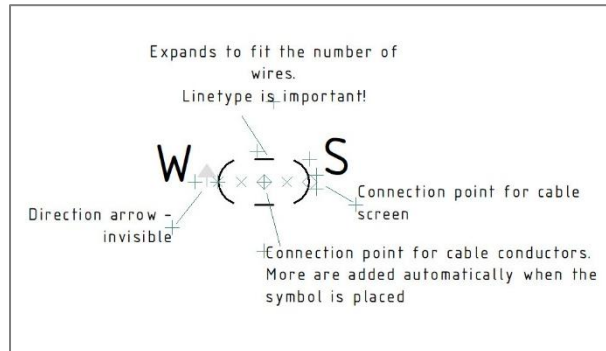
Cable symbols are special in that they adapt their size to the number of conductors. In addition, affected conductors (of course) appear in cable lists. Finally, you can determine the direction of the cable, which has an impact on which end of the cable is From or To in the cable list.

In the folders IEC60617 and MISC, there are various cable symbols.

The most commonly used ones are PCS-S0007, PCS-S00009 and PCS-S00010-001.

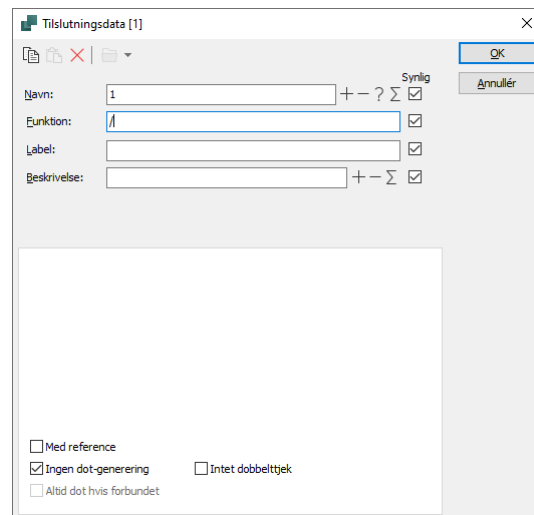
In general, it can be said that the symbols graphically enclose the conductors and they expand depending on the number of conductors.

Here you can see PCS-S0009 as it looks during creation: the ends are circular arcs, the middle is (here) dotted lines



If there should be a character where the connection point 'catches' the conductor, the character is written in the Function Text of the connection point.

It works best on PCS-S00010-001.



When designing the cable symbol, the first connection point (far left) is the location for the first cable conductor in the cable. A possible second (far right) connection point will be for a possible cable shield. The other connection points appear when conductors are added.

When designing the symbol, remember the following:

- the reference point must be placed to the LEFT of the connection point for the first conductor. The same applies to any lines that are to be used at the start of the symbol
- the rest of the cable symbol must be drawn to the RIGHT of the conductor's connection point, the rest of the cable symbol is drawn.
- the far right is the connection point for any cable shield.
- when the symbol is drawn, it has a default direction of UP. This can be reversed using Right-click|Reverse direction on the drawing itself or set under Settings|Text/symbol defaults.



Busbar

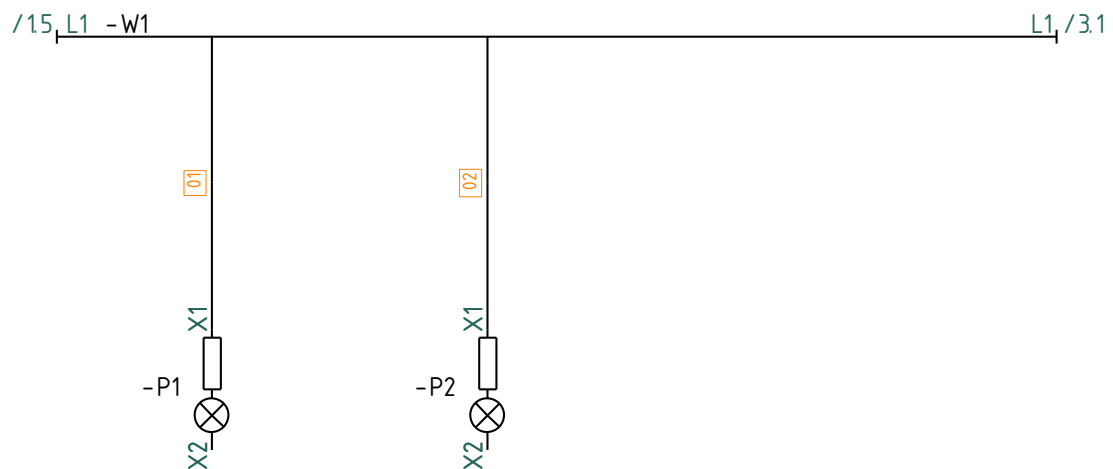
A busbar is a symbol type where connection points are given the name of the signal they represent, when the symbol is placed in the project.

The symbol looks like and is drawn – like a line, meaning that its length is variable.

When you connect lines to the busbar, you don't get dots!

Wires connected to a busbar (may) get wire numbers – they are wires connected to a component. If you connect cable shields, however, you might want to use the special terminals and a jumper line, see page 40..

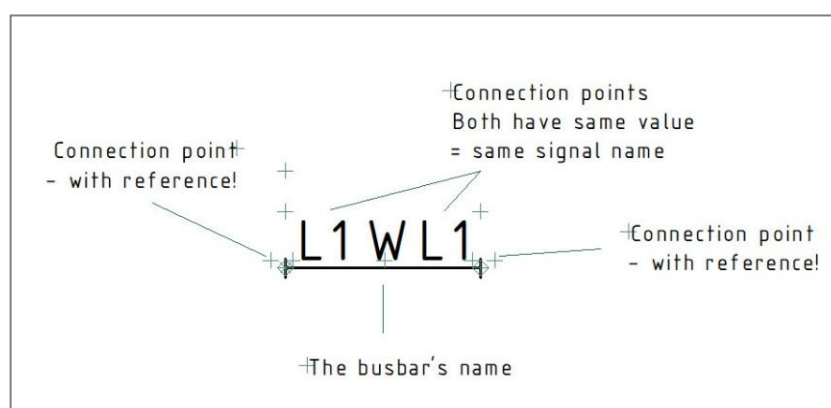
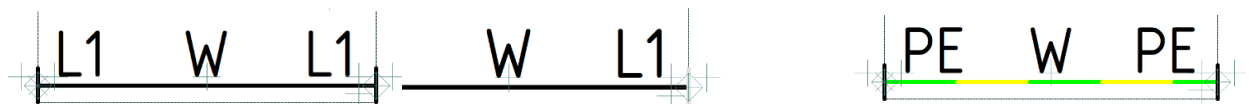
You can 'look through' the busbar (as with the terminals) with the Netnavigator, or you can decide, that it is a new potential (as with terminals).



You can find three different busbars in the 60617-folder: PCS-S00001-001, PCS-S00001-002 and PCS-S00001-003.

The first and the last have vertical lines in the ends, which the middle one doesn't.

On all three symbols, there is Active reference on all connection points.



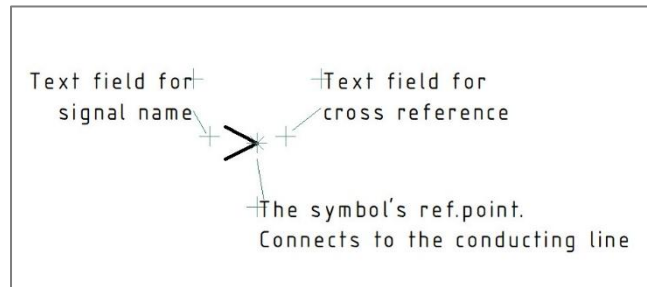
Signal symbol

Signal symbols are used to provide an electrical connection between two conductive lines, mostly when you go from one page to another.

Unlike other symbols, Signal symbols only contain a reference point, symbol name and page reference.

The actual creation of the symbol is done in the same way as other symbols.

The signal symbol shown is created with an arrow (SGPILRight), but often a signal symbol without graphics (SGSIGNAL) is preferred in the documentation.



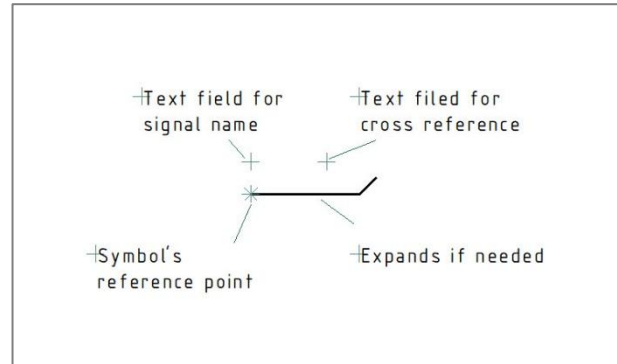
When the symbol is placed, the reference point is connected to the conducting line, the cross-reference is filled in when another signal symbol with the same name is placed - and an electrical connection is thus created between the two lines.

Under Settings|Text/symbol standards, you can create a list of preferred signal names, and another list of preferred signal symbols.signalsymboler.



Join-symbol or Multi signal symbol

Join symbols are a type of signal symbol. They are created in the same way as cable symbols: the reference point is placed on the left and the symbol itself is drawn on the right. When placed on the drawing, the horizontal lines are extended and the other lines are moved according to the extension, but retain their appearance. The symbol is saved as the SIGNAL type.

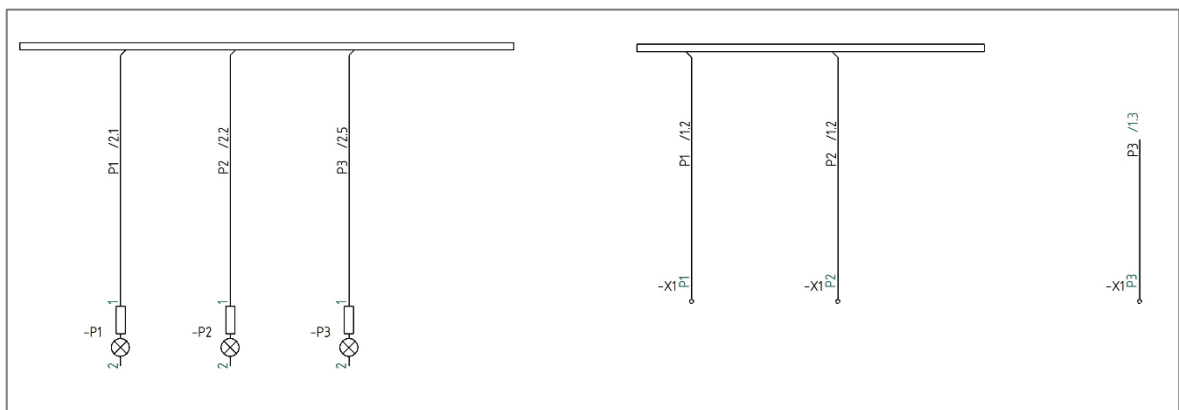


Using the join symbol for bus connections

The join symbol can be used in this way:

Wide lines can show bus connections

- draw a wide (non-conducting) line
- place a number of symbols that are to be connected to the bus.
- select the wide line, right-click and select Connect signal and select one of the Join symbols
- give it a name
- possibly go to the next page and repeat, but use the same names
- go to object properties and set Reference active (and possibly change the text font and color)



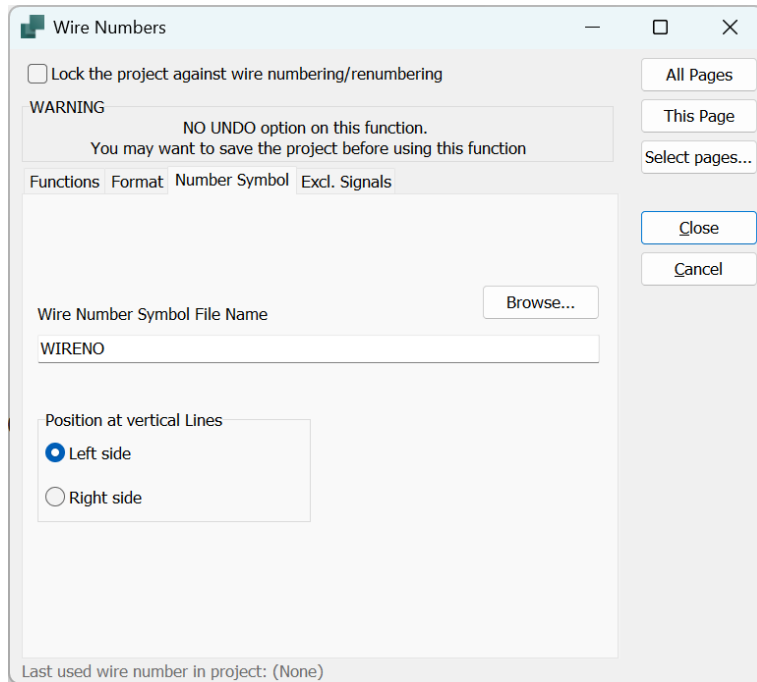
Wire number

The function Wire numbering places symbols of the type WireNumber in the diagram.

They may also be placed manually.

Which symbol and where it is placed is selected in the wirenumber function.

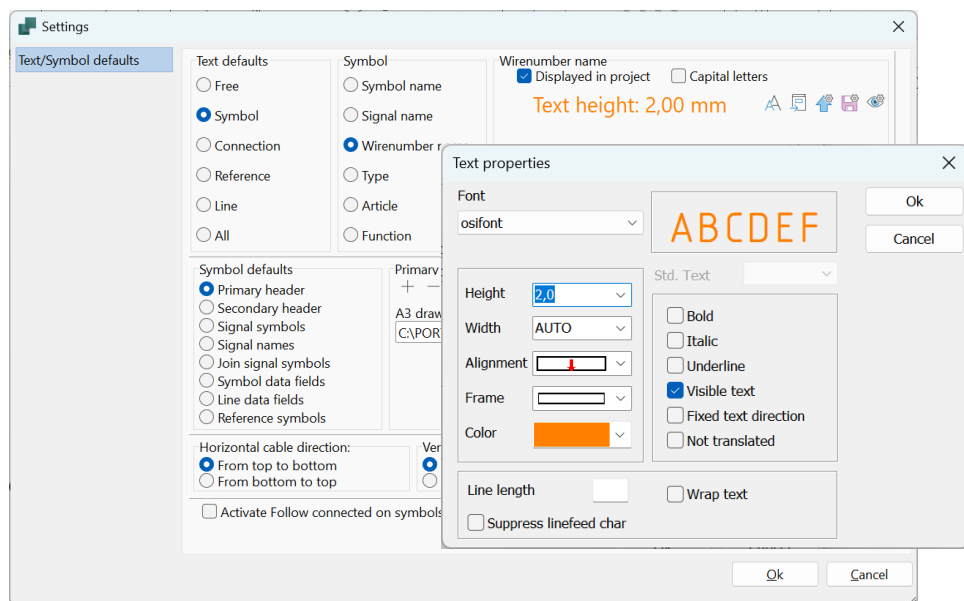
Wire numbers' names may be printed by using the List files: Wire number list or Connection lists.



The symbol itself consist of the reference point and S.Name.

Color and font etc is set in the Text/Symbol defaults dialog.

Here – orange and a frame.



Insertion point

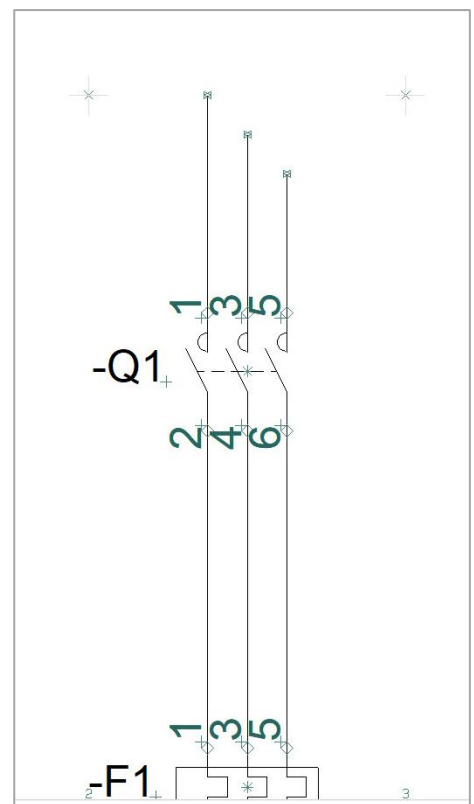
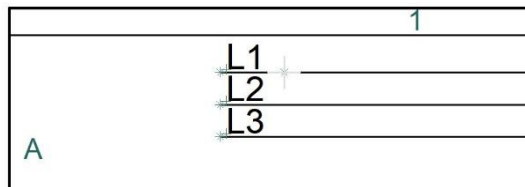
The insertion point symbol is a small symbol that the program uses to place subdrawings in the right position:

On a diagram page, we place two symbols to indicate where we can auto-place subdrawings FROM and TO. Below you can see the FROM on a diagram page; the TO is at the other end.

All subdrawings – also see below – have two Insertion symbols to define the width. This symbol works with the functions Subdrawings, Drag'n Draw and the Project Generator.

The symbol has the color NP and is not seen on print outs etc.

See how to use it in the section about Templates from page 67.



Line connection symbol

The line connection symbol is a special symbol that allows you to make variable connections lengths to phases or other lines. In the symbol library there are two of these symbols, namely for three and four phases respectively (LineConnect3 and LineConnect4 in the misc folder). You can create other symbols for other purposes, for example with connection points as an X, which are also controlled by variables. However, for all symbols of this type, they must be used together with models, and the procedure for doing so is showed below.

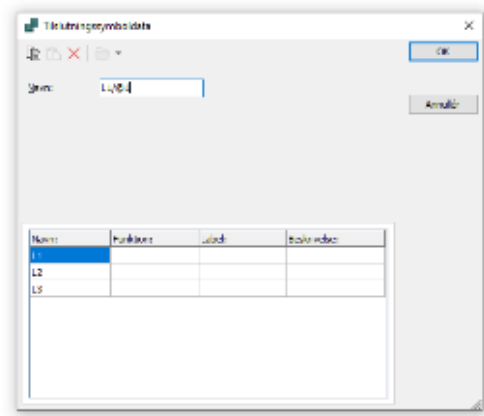
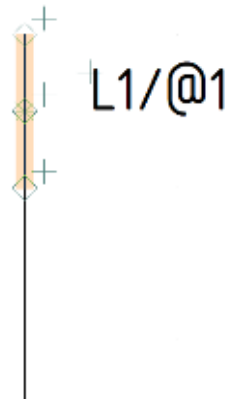
Using the line connection symbol for models (subdrawings)

See the sub-drawing pcsLamp.pro. The line connection symbol is placed so that it matches the connections to the potentials L1, L2 and L3.

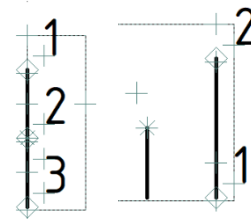
The symbol itself is called L1/@1 (@1 indicates that the name is a variable).

In the dialog box, the connection names are set to L1, L2 and L3 respectively.

When you place the drawing, the symbol, i.e. the line, is extended so that it captures the mentioned phase.



To the right is shown how LineConnect3 looks in the Symbol editor, and next to it is shown how you could possibly create a line connection symbol that connects to the phases as L1L2L3 or L3L2L1 respectively.



Support symbol – Accessory

When you have components with accessories in your project, it is possible to use a Support symbol and add article no and quantity of accessory to it. This is what was done earlier.

For the current versions, we recommend to use the accessory tab and select the accessories here. In that way, the program knows that the parts belong together, and it is possible to copy the complet 'set'.

The accessory – fixed and/or optional – is created the database, and may be added to other components here, regardless of whether it has symbols or not.

Component data [-P1]

Name: -P1 P : presenti + - ? Σ Σ Visible

Type: PCS-RDLAMP 1 Σ Σ

Article no: PCS500001 (ManufacturersArticleNumber) Σ Σ

Function: + - Σ

Description: Red lamp with built-in resistor

General Ref.design. Symbol data fields Reference Conns. Accessory

Count	Placed	Article	Type
Fixed accessories			
Optional accessories			
1	0	PCS500005	PCS-LAMPSOCKET1
0	0	PCS500006	PCS-LAMPSOCKET2

+ [Icon] X

Ok Cancel Unit Database components

The selected accessory's, quantity, type and article number may be visible in the project if you add this symbol data field.

General Ref.design. Symbol data fields Reference Conns. Accessory

Datafield	Value
Selected accessory	~1, PCS500005, PCS-LAMPSOCKET1

Add Edit Remove

From ver23 we – PCSCHMATIC – haven't created components containing Support symbols. That means that components from the Component Portal don't contain Support symbols.

The 'old' support symbol is found in the MISC folder.

Header (List on page)

Symbols of the Drawing Header type can contain a mini parts, component or plc list, which can be inserted on a page.

If this is desired, remember to check the option 'Contains list [type]' under Page settings on the diagram page.

See also the symbols pcsPgxxx in the Headers folder.

They are created according to the same rules as all other lists, see page 63, but must be saved as type Headers.

Header		
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type
Name	Article No.	Type

The symbols in the Headers folder fit the page, when it has a Scale 1:1.
If you use another page scale, use the Symbol scale (in the pickmenu) to compensate, ie if the page scale is 1:5, set the symbol scale to 5.
This is done in the template pcsStart.



COMBINED SYMBOLS – SYMBOLS ON THE PORTAL

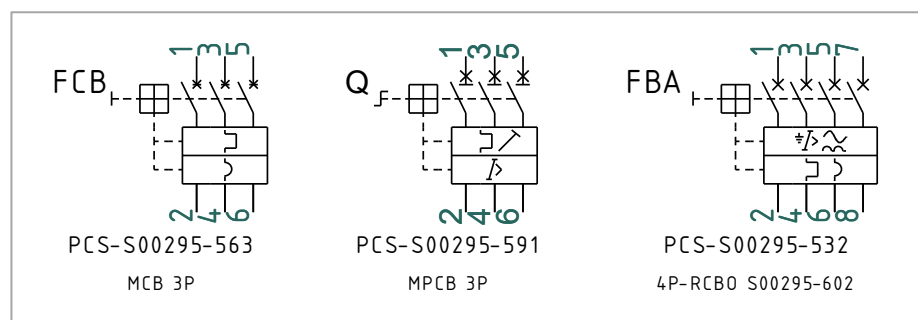
When we create symbols for components on our Component Portal, we need to create a lot of new symbols. And we create them according to the rules laid down in various standards – see earlier in this document – and knowledge about the components.

We try to be as consequent as possible, meaning that we try to use the same symbol for the same type of component,

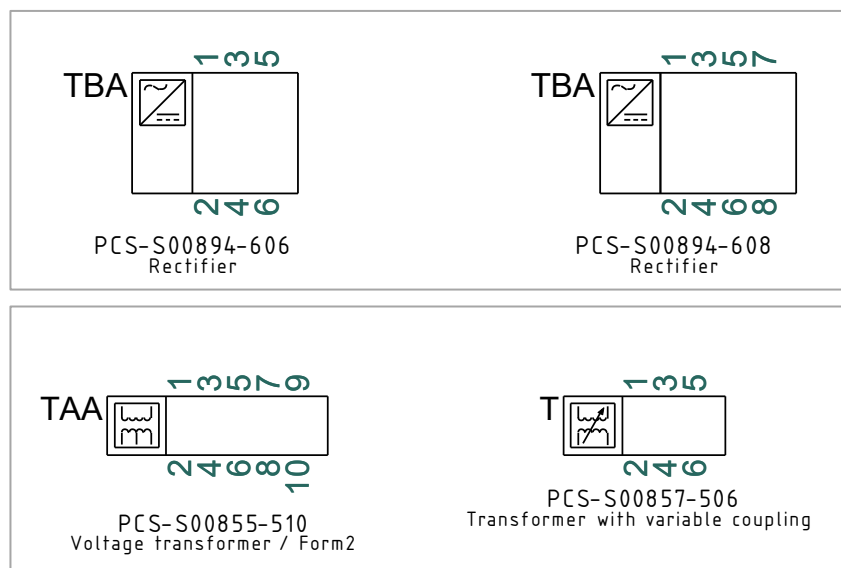
Therefore we have laid down our own rules for many component types, written down and shown in our Design Guide.

The main parts for symbol design follow here:

- We use the IEC 60617 symbol if possible
 - We name the symbols PCS-Sxxxx for the standard symbols
- IEC 60617 symbols can be combined as described earlier
 - Such a symbol is named after the main symbol, ie PCS-Sxxxx-001, PCS-Sxxxx-002 etc
- In this document, you can see examples of our new terminal symbols and their names
- All Panelbuilder diagram symbols are made by using the same standard contact symbols, and then filled out with qualifiers depending on component type.
 - You can read more about this in the Design Manual



- A lot of components – power supplies, sensors, fuse blocks, SPD's etc – are made by using a box with the right number of connections, and qualifiers placed in a section to the left.



SYMBOL DOCUMENTATION

In the Tools menu you can find the tool Symbol documentation.

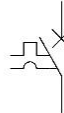
This tool can document the symbols in a project or in a selected folder. It generates a project with the selected symbols with symbol name and title for each symbol, and an optional list with the same information.

The title is shown in the same language as you run the program in (or default UK (1033) if local language is missing.)

Explain you symbols – also when you combine standard symbols

Even when you use standard symbols, there might not be only one answer to combinations. Below an example from the IEC 61082-1 standard showing the construction of a symbol for an MCB.

And we have decided a different solution, see above.

There is not a specific symbol for a MCB. Since it is a small power breaker, you may use the symbol S000287.	 S00287 Load circuit breaker
You may also combine the general contact symbol with the automatic tripping symbol. It doesn't show the power breaker function, but useful.	 S00227 Make contact general symbol +  S00222 Automatic tripping function = 
Here the load circuit breaker and the automatic tripping symbols are combined. Now both function are there.	 S00287 Load circuit breaker +  S00222 Automatic tripping function = 
The MCB – Miniature Circuit Breaker – is based on a thermal and an electro magnetic tripping. Here combined with the general breaker symbol.	 S00227 Make contact general symbol +  S00120 Thermal effect +  S00121 Electro- magnetic effect = 
The MCB – Miniature Circuit Breaker – is based on a thermal and an electro magnetic tripping. Here combined with the power breaker symbol.	 S00287 Load circuit breaker +  S00120 Thermal effect +  S00121 Electro- magnetic effect = 



MECHANICAL SYMBOLS

Mechanical symbols for components are not part of the diagrams, but are placed on layout and arrangement pages. This way you can see where the individual components are placed in the panel and whether they can fit into it 😊

All mechanical symbols must be made to exact dimensions; and if they have connection points, they must have exactly the same names as the electrical connections in the diagram, and they should be placed at the correct XY coordinates. The Z position of the connections is controlled from the database's BuiltInDepth field. When the mechanical symbols are placed on the page, their size is determined by the page scale.

Below we will go over how to create the symbols best and easiest.

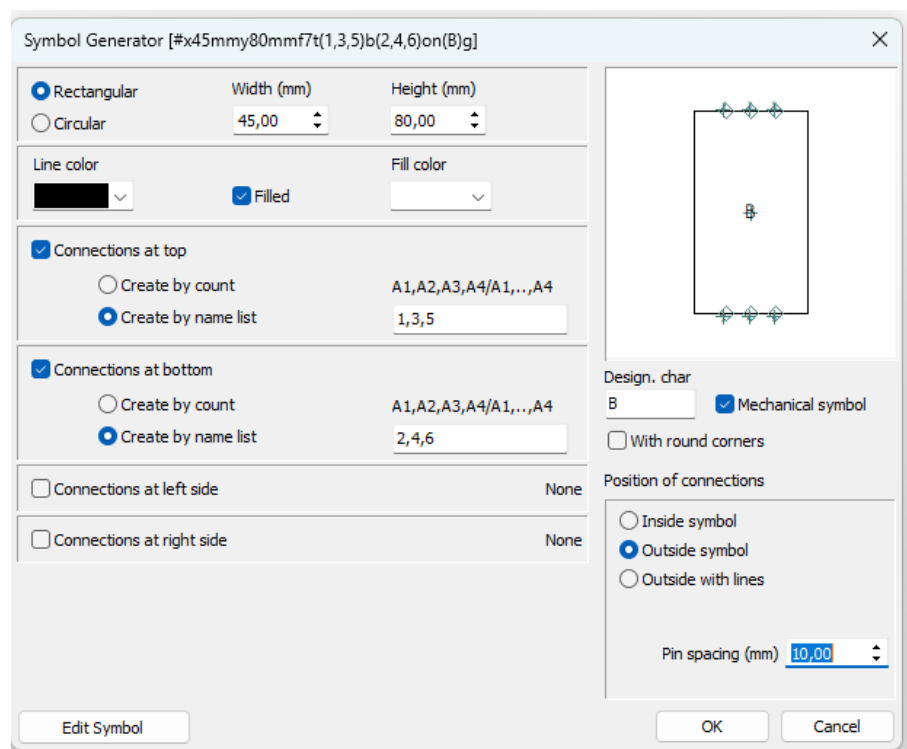
As a rule, you create mechanical symbols on mechanical pages and as the Normal type. Special functions are covered on the following pages

Use the Symbol Generator

Start by using the Symbol Generator to create the symbol so that it has the correct dimensions.

Select Filled, so that a DIN-rail will be hidden behind the component. Select Mechanical symbol so that the name is placed in the center.

You can create the connection points here; however, they will be placed on the edge of the symbol - they will be moved later. The name is 5 mm high.

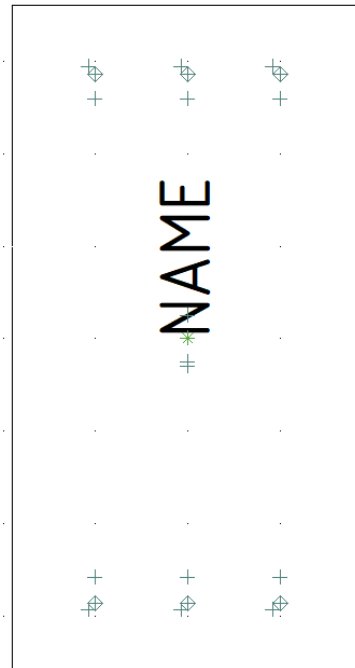


You can also just make a box without connections, but all mechanical symbols on the Component Portal are made as described here.

If you want to use the Panel Router, the symbols MUST have the correct connection names.

Continue to Edit the symbol in the Symbol Editor by pressing the Edit Symbol button.
Make sure you are on a mechanical page!

- The reference point * should be placed in the middle of the suspension point on the DIN rail, so it may need to be moved.
 - When all * represent the middle of the DIN rail, the symbols can be aligned here.
 - Magnetic snap will also align according to *
 - This is why you should process the symbol in the editor and not only with the symbol generator
- Name is rotated and centered so that it grows upwards from *
- Use the function in Symbol Settings
- The symbol should have a name after 81346-2 and not 'Name'.
- All connection points are moved to the correct XY position and made invisible.
- When everything is ok, save the symbol with the type Normal. Give it a name after Manufacturer_ManufacturersArticleNumber if desired - this is how it is on the portal.



Tip

If necessary, use the Move Coordinates function (ctrl+I) to move the connection points

More tips

In the Symbol Editor, you can, for example, insert a dotted line that represents the respect distances for the component.

Even more tips

Components that are not to be placed on a DIN rail – such as a power supply – can have holes on another layer.

This way, you can also export a dxf to a drilling machine.

Create mechanical components based on an AutoCAD drawing

If you wish a symbols that looks very much like the component, then it is possible to use a dwg or a dxf file as a starting point.

In the Symbol Editor you can open a *.dwg or a *.dxf-file.

Use it as a starting point. Remember to check the scale. And follow the steps above.



Mechanical symbol for terminals

Mechanical symbols for terminals must be saved as symbol type **TERMINAL**. The connection points must also have status as input/output type **TERM**.

For common 1-layer feed-through terminals, the terminals in the symbol must be named with 1 or 0 – as in the electrical symbol.

Mechanical symbol for multi-layer terminals

Mechanical symbols for multi-layer terminals are also saved as the type **TERMINAL**: If you want to display connection names on the mechanical symbol for multi-layer terminals, the connection names must be named with layer no.

Here layers 1, 2 and 3, which here each have an input and an output.

Any jumper connections can also be placed in the same way.



Non-conducting symbol

Non-conducting symbols are symbols that can open a non-conducting line. The symbols may be doors or windows, that are placed on a non-conducting line = wall. The symbols are drawn to scale, and maybe you need to put a scale to the page that you are drawing on.

You add a connection point in both ends of the symbol, corresponding to where the line is to be broken.

SYMBOLS FOR THE PANELBUILDER

The small tool – the Panelbuilder, that can generate small IEC 61439-3 panels (BDOs) – uses symbols that are either generated automatically (most of them and our recommendation), but it is also possible to create your own symbols as described below.

These are only the symbols that appear on the arrangement. The symbols for the single-line diagram are the standard IEC60617 symbols, but of the Normal type, so that no reference is made to make/break cross-sections

Component symbols for Panelbuilder

Component symbols for Panelbuilder have a connection point OUT at the top in the middle of the edge of the symbol and a corresponding IN at the bottom. And no other connection points.

If you want the option to show the board closed, i.e. to hide the component that is under the lid, the symbol is drawn in two layers:

- What you see when the lid is closed, is drawn in layer 0
- What is under the lid, is drawn in layer 27.

Symbols for the panel in Panelbuilder

The panel symbol itself is also drawn on dimensions. Component symbols can automatically align, i.e. the reference points are aligned, within a defined area, which is the cutout itself.

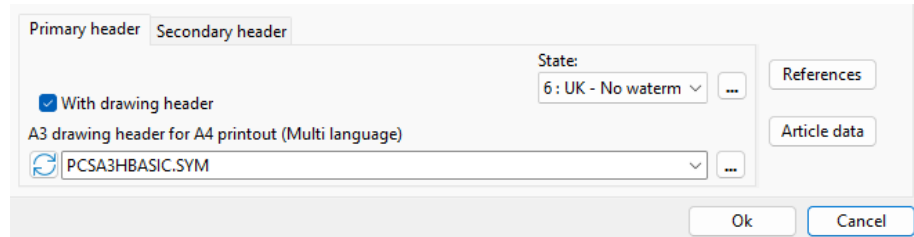
Cutout is made by creating a SQUARE with the line type NP in Layer 255.

See also the symbol PCSCHEMATIC_Panel1 in the Mec folder.



HEADERS (TITLE BLOCKS), LISTS AND TEMPLATES

Symbols for drawing headers (title blocks) have the symbol type Normal; they are selected as Drawing header in the Page data settings.



Create your own drawing header

Designing your own drawing header is simple, as they only consist of lines and data fields. The difficult thing is to get an overview of the symbol as it has the same size as the page itself.

As a starting point, Automation uses one drawing header on diagrams and two on list pages. The primary drawing header is the frame that is (may be) used on all pages, whereas the secondary header contains the list itself.

The size of the drawing header

Regarding the size of the drawing headers, there must be 5 mm to the edge of the paper and 20 mm on the side where the punch holes are.

You select the page size under Settings|Page settings.

On the Settings|Pointer/Screen tab, you can activate a Help frame that shows where the drawing header should be if you print on the same paper size.

The normal thing in Automation is that diagrams are made on A3, but are printed on A4. This means that the drawing frame is made on A3, but the punch holes are placed as on A4. Therefore, the frame does not fit completely.

The drawing header must be made on the page size it will be used on. The 'normal' drawing header used on diagram pages is an A3 drawing header set up for printing on A4 – pcsA3Hbasic.

There are different drawing headers in the Headers folder, for A1, A2, A3, A4. The easiest way is to use one of these as a starting point if you are going to design a drawing head yourself.

Where to place the * the reference point

The reference point of for drawing header symbol must be placed precisely on the Help frame.

The Help frame can be turned on/off under Settings|Pointer/Screen.

The location of the reference point determines where the drawing head is inserted on the page.



Drawing headers with states and optional language codes

PCSCHEMATIC's own drawing headers are created with states, that describe a status and in different languages.

The program will automatically select the (first) same state as the selected user language. The user language is set in Settings|System.

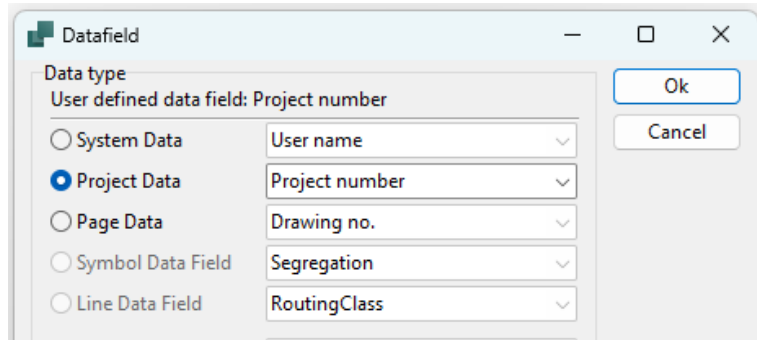
Data fields in normal drawing headers

Data fields in normal drawing headers – the frame itself – are selected among the two sections Project data and Page data.

You select the data fields from Insert|Insert data field.

You can create your own data fields in the two

sections, not here but in the Settings menu for Project and Page data, respectively.



Tips & tricks when creating drawing headers

The frame and reference point itself fit the different page sizes, and the text field has a size and content that corresponds to the requirements in standards regarding this.

Therefore, it is easiest for you to take one of ours as a starting point if you want to make your own.

In the long run, you may be able to just replace the logo in the project: The logo fields are called Logo1 and Logo2, respectively, and in our drawing header they are placed to the right and left of the logo field, respectively.

Their size and alignment are set using Text Properties, and it is best to leave them as they are once they are set! If your logo does not fit nicely in the field, you should instead edit the logo, e.g. in Paint, where you can either cut it cleanly or insert white. Try opening our logos to see examples of both.

On the other text fields, you can consider alignment and possibly max. length.

Drawing headers are saved as symbol type Normal or Headers.



Create your own lists

Lists contain data fields showing different data about components and/or connections.

Parts and Component lists show information and quantities for components

Terminal, Cable, PLC and Connection lists shown connections to/from the mentioned component type

To make the lists work in the projects, there are various rules that must be followed, and they are described below.

Below you can see the (part of the) component list pcsComponents1 in the symbol editor.

Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description
Name	Article No.	Type	Function	Description

Rules for data fields in lists

- Data fields in lists – the content part is specific: i.e. if you want to make a Parts list you must use data fields from the Parts and Component Lists section; if you want to make a Terminal list, you must use lists from the Terminal Lists section, etc
- Data fields may come from both the project and the component database
- You can use Project and page data fields, including Symbol data fields and Line data fields for all list types
- The first data field must have the property Activate next – shown as the small filled out diamond on the first data field
- Many data fields have extra functions, here quantities and ref. designations; on terminal lists for example, these are functions that refer to the internal/external side of the terminal
- All data fields for a record MUST be in line!
- The reference point must be placed as described under the common drawing headers.

The screenshot shows a 'List Configuration' dialog box. On the left, there are radio buttons for different list types: 'Table of Contents', 'Parts/Components List' (selected), 'Terminals List', 'Cables List', 'PLC List', and 'Connections list'. To the right of these are dropdown menus for selecting data fields: 'Drawing no.', 'Name', 'T.name', 'Cb.name', 'PLC name', and 'Signal'. Below these, there are checkboxes for 'With Quantity', 'Function aspect' (checked), 'Location aspect' (checked), and 'Product aspect' (checked). At the bottom, there are checkboxes for 'Activate next' (checked), 'Wrap text', 'Keep line break (^)', and 'Only show Pre-text on data'. A 'Fill character' input field is also present.

Tips & tricks when you create lists

Again, it is easiest to make your own list if you take one of ours as a starting point, among other things the reference point is placed correctly.

If you start from scratch, you place a row with the desired data fields, the text properties of the texts, e.g. length are set, and all data fields are aligned! And the alignment line is determined from the texts' reference points (+).

If necessary, put a line underneath, and copy the entire row until the page is full.

If necessary, place headings above; in ours, the headings are made in states, each representing a different language. And it is a good idea to have the same order in all lists...

If you want to edit a list, you can select an entire column of data fields and replace them at once.

Some of the data fields – e.g. Description in the example above – come from the project, but are filled with the description in the language you have selected in your database settings.



CREATE A TEMPLATE

Templates are empty pages and projects, that contain various settings. Those settings are used in the project.

You may have several sets of templates, so that they fit with different requirements, settings and project types.

In Settings|Directories you may change the folder containing your templates.

Considerations before you create your templates

- Page settings of zones (current paths), meaning your module size that is used for templates and for subdrawings
- Different page types
- Position of potentials/signals on the diagram pages
- Lists with different settings, ie repetitions and sorting rules for parts and component lists
- Project templates
- Default folders for templates

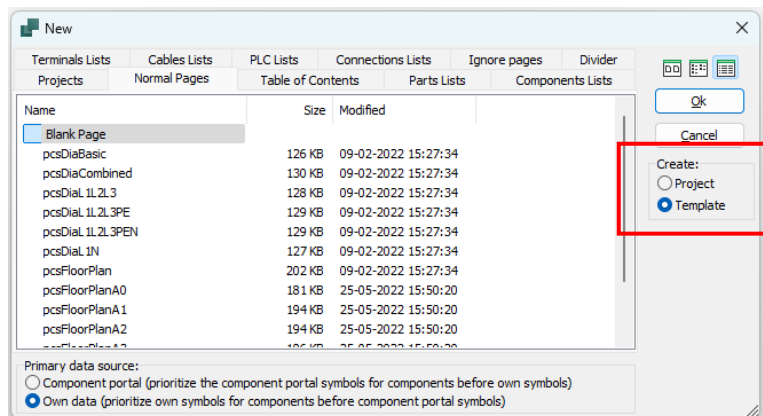
Create a DIA-template

Press New (or go to Files|New) to create a template.

Here you can select the page type that you want to make a template for.

Select the blank page if you want to start from scratch, or use an existing as a starting point.

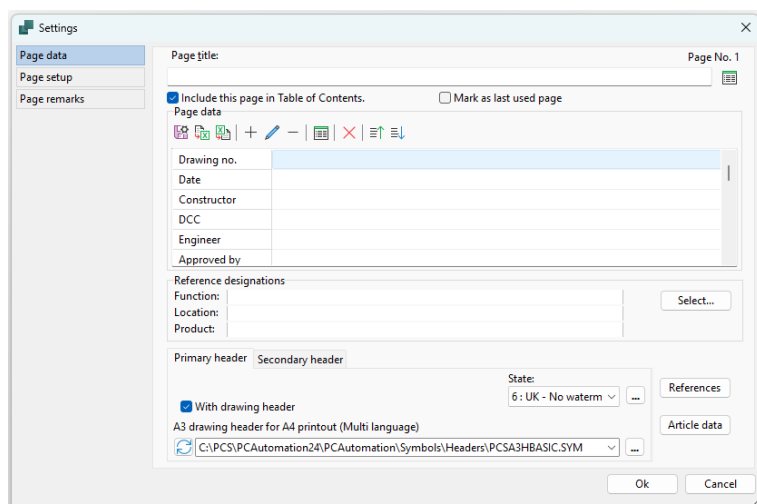
Here you see existing page templates for normal pages (DIA or GRP).



Drawing header and other page data

Page data contains the page header and default page data, ie Page title.

Our header has language codes, and they may also be default selected – and changed in the project.



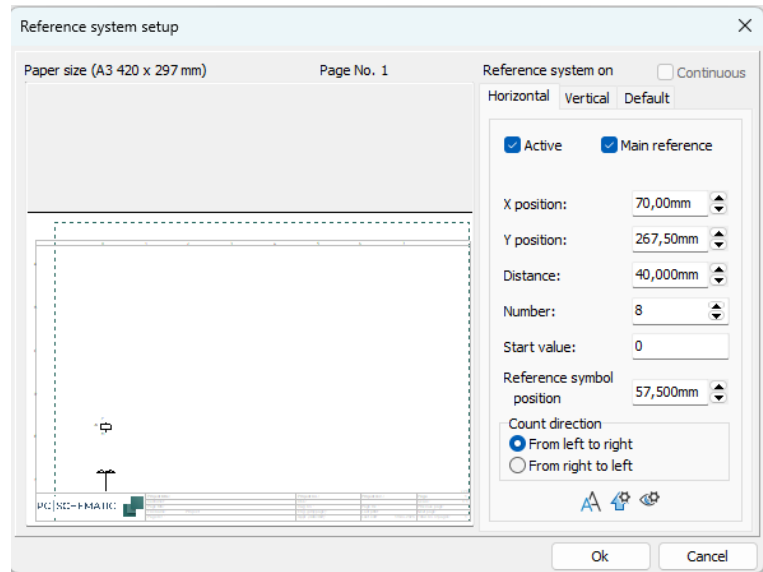
Zones (current paths), reference symbol etc

The References button provides access to set up current paths, both vertically and horizontally.

A distance of 40 mm is the recommended distance according to the IEC 61082-1 standard.

The location of the reference symbol is also determined here.

You can save your setup as Standard, so that your future projects will have this setup – used when you don't start with a project template – or you can update the current project with new settings on the Standard tab.



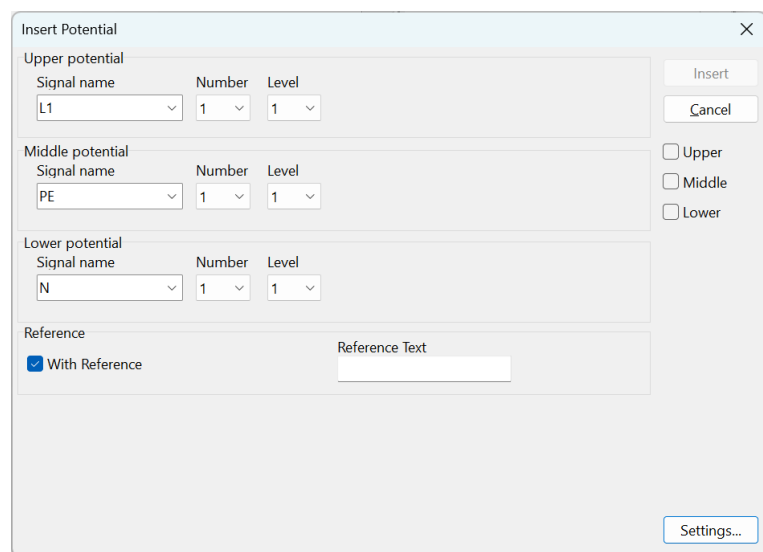
Insert potentials



You can insert potentials on the page when you press this button.

You can place potentials in three preset positions.

In Settings, you make your desired setup, regarding names, signal symbols and positions.

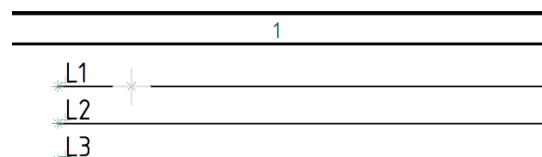


Insertion points

If you want to use the template in connection with subdrawings and project generator, you need to place two insertion points on the page: one where you want to start placing subdrawings *from*, and another where you want to place *to*.

The two symbols must be aligned – horizontally or vertically.

Here they are placed a half zone – 20 mm – before and after the first and last zone number, respectively.



The page contains a list

If you want a list on the page – parts, component or plc-list – you can check the box. The selected list must be of type Header.

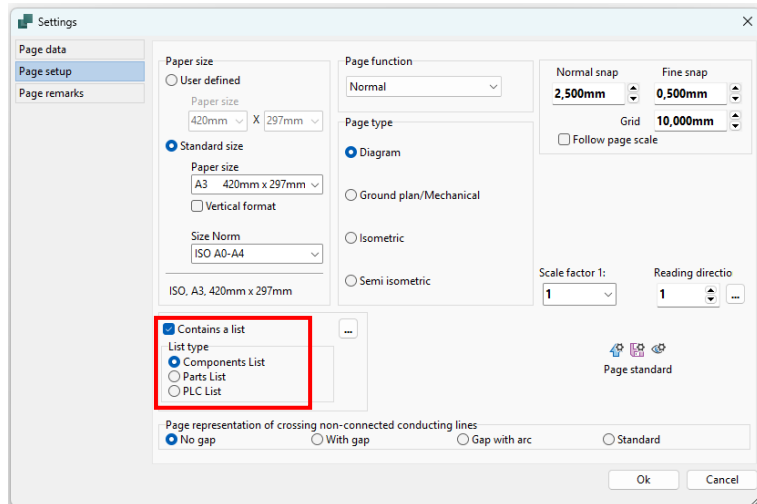
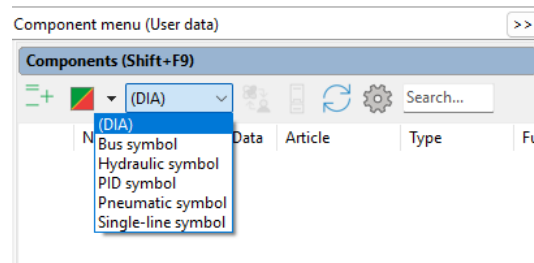


Diagram pages for other diagram types / symbols

You can create your project template to automatically select symbols for other diagram types, ie Single line diagrams. From version 23, this option is always present, so you don't have to create specific page templates.



Create a GRP template

When creating GRP pages – mechanical pages – you must follow the same procedure as shown above for DIA pages. However, the section on potentials is not relevant.

The GRP function comes when you select Page Type in the Page Settings tab.

If you want to maintain dimensional accuracy on a mechanical page, it is important that the page is printed on the same paper size as the page in the project. This means that if you want to print on A4, the page must also be A4 and the drawing header must also be A4.

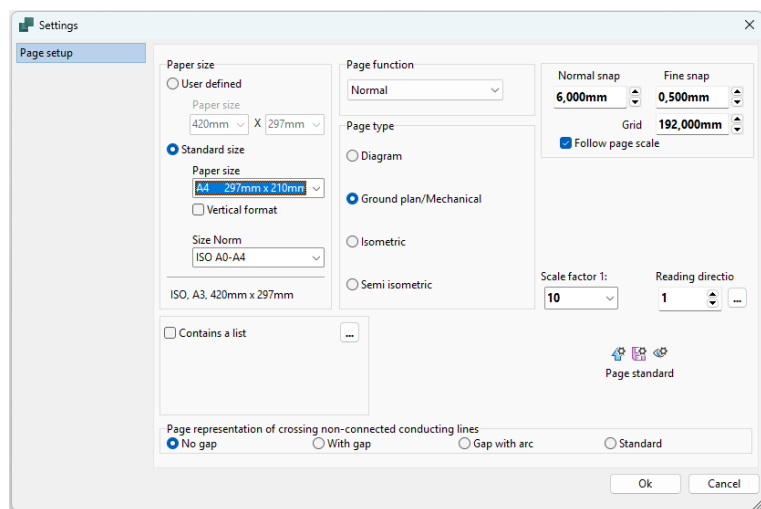
The scale on the page is selected for the individual template, but can also be changed in the project.

Tavler og komponenter i modulmål

You can create GRP pages using module dimensions from the panels.

Set your snap and grid values to match the module dimensions.

If you select Follow page scale, the grid on the page will show the correct dimensions, not the paper dimensions.

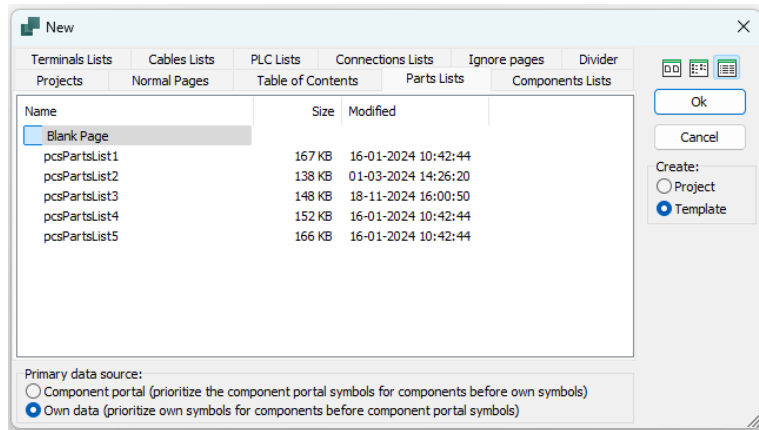


Template with a list

When you create a template with a specific list type, start by selecting the list type in Files|New.

You can choose between

- Table of contents (TOC)
- Parts list (PAR)
- Components list (COM)
- Terminals list (TER)
- Cable list (CAB)
- Plc list (PLC)
- Connection list (NET)



Create the template

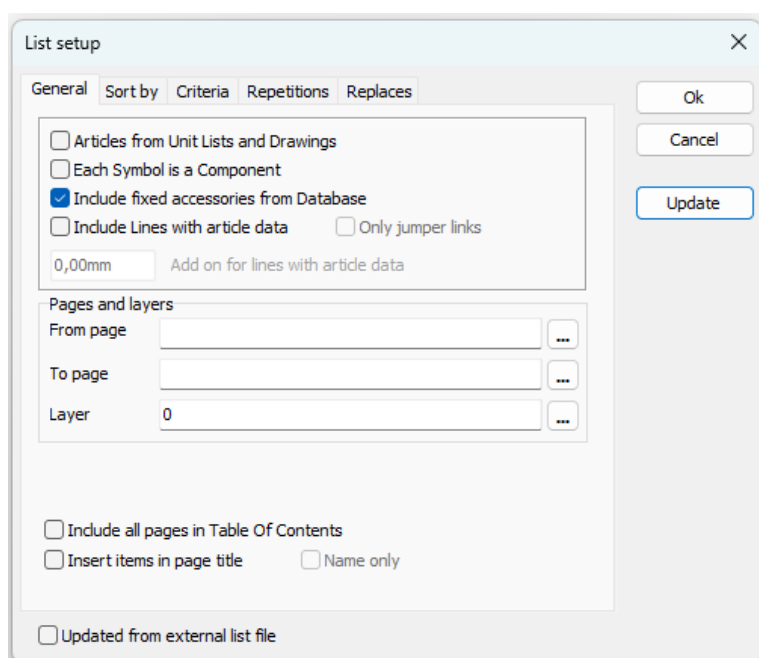
If you start from scratch, follow the instruction below,

If you use an existing list as a starting point, you can skip some of the steps. Vælg listetype

- Select Template
- Select Blank page
- Select a Primary drawing header – often the same as for the diagram pages
Select the Secondary drawing header, that MUST be created with data fields from the right section!!

Listeopsætning

The list has different settings: criteria, sorting, repetitions etc.



If you go through the different tabs, you can see the different options.

The options look alike for the different list types, but they also contain differences.

The settings are part of the template itself.

If you have started as described, the template is automatically saved in the correct folder.

Templates have the extension *.std.

It is a good idea to save a description of the template; you do this in Project Data's Project Title. That way, a popup with the description will appear when you hover your mouse over the template.

Templates for graphical plans

You can create templates for graphical plans in this way:

For example, create a graphic terminal plan. When you have made all the settings you want, generate the plan.

Copy the first page, as it contains all the settings.

Create a new single page, e.g. a terminal list – then the template will be in this folder - insert the copied page, delete the first page, and save as a template.

Check that the page types are TEP, CAP and CNP, respectively.

The advantage of having templates for the graphical plans is that you can have different settings for them, depending on your and your customers' needs. In addition, the project generator can update these in the context of generating a project.

Create a Project template

Once you have created the necessary individual pages that you need in your projects, you can create a project template.

Go to File|New. Select the Projects and Template tab.

Use the Page menu to insert pages into the project and to give page numbers.

Save the template under a sensible name.

Remember that if you use module dimensions for the board drawing, you can only have one module dimension in each template, as page settings follow the page type.

If you want naming according to page flow path in some projects, you can also put this into the template. And at the same time have start templates without.



MY NOTES











