# COMPONENT WIZARD

Made by PCSCHEMATIC A/S

June 2020

# PREFACE

This booklet shows you how to use the Component Wizard to create components in the PCSCHEMATIC database. The Component Wizard is part of Automation from version 16 and has been improved continuously.

The Wizard can be used for different functions, among them

- Create a totally new component
- Create a new component as a copy of an existing one
- Edit an existing component
- Edit a list of existing components
- Multi-create components by editing an Excel-file, which can later be imported into the database.

All functions are described in this booklet.

## A few definitions …

It is compulsory to connect a database if you want to use the wizard.

COMPONENTS are found in the database and they must have a unique number/id, which in our database is the EANNUMBER (the EANNUMBER is a barcode number, that identifies a component uniquely).

In our database setting we have connected the EANNUMBER from the database with the Article number in the Automation program, and we have connected the TYPE from the database with the Type in the Automation program.

You can see our default setting in page 41 (which also shows a few extras).

COMPONENTS consist of one or more diagram SYMBOLS – and a lot more information. (Some of) this information is tied together when using this wizard.

## REMEMBER that …

If you create components for your database, we recommend that you rename the default database or that you make a copy of it, which you give another name. In that way you can control the database's name and you do not risk having it replaced as part of a program update.

The Component Wizard uses *your* database settings. In page 41 you can see the default database settings – with a few extras. If your Component Wizard looks differently it is because your settings are different.

Books in the series about Automation:

Quick start

Motor control

Component Wizard

PLC-project

Labelling and Marking

Electrical Installation

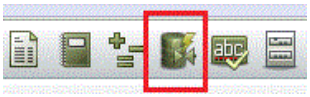
Panelbuilder

Automation Service

# Contents

# OPEN THE WIZARD

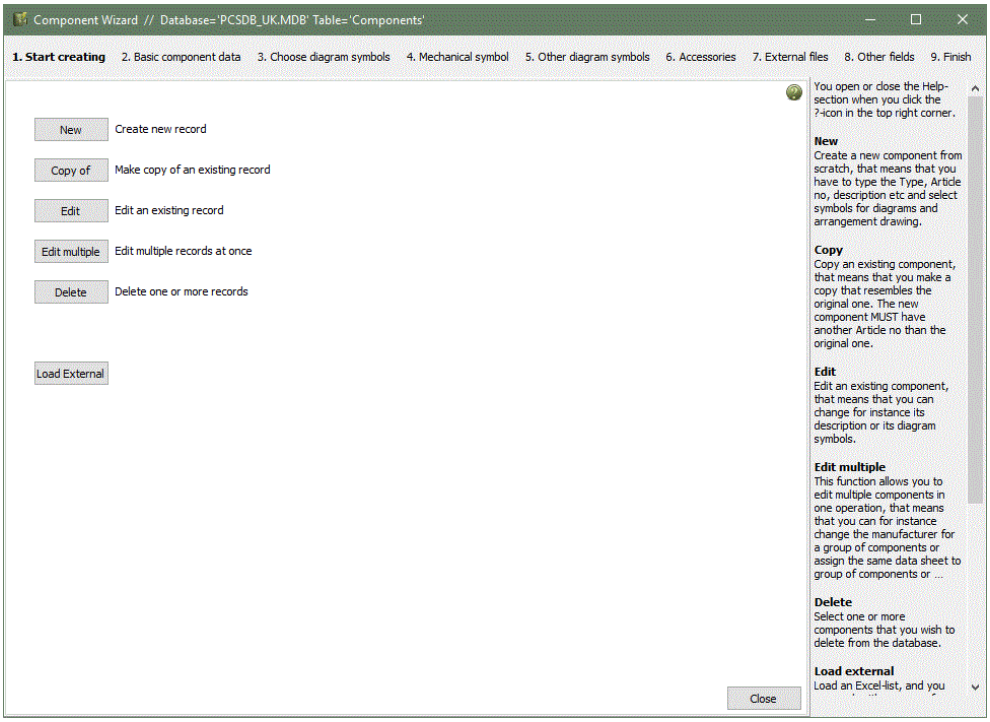
You can open the wizard by clicking the icon in the top toolbar or you can find it in Functions|Component Database|Component Wizard:



In the top line you can see the name of the attached database and the current table.

In the next line you can see the steps you are going through to create your component.

In the middle of the dialog you can select your options with the wizard.




At the right side you can see a help text for each step.

You can open or close this part of the window by clicking the icon.

At the button of the window you can – in the next tabs – see which main option you selected, i.e. Edit Component or New Component.

You can always close the wizard by clicking the X in the top right corner.

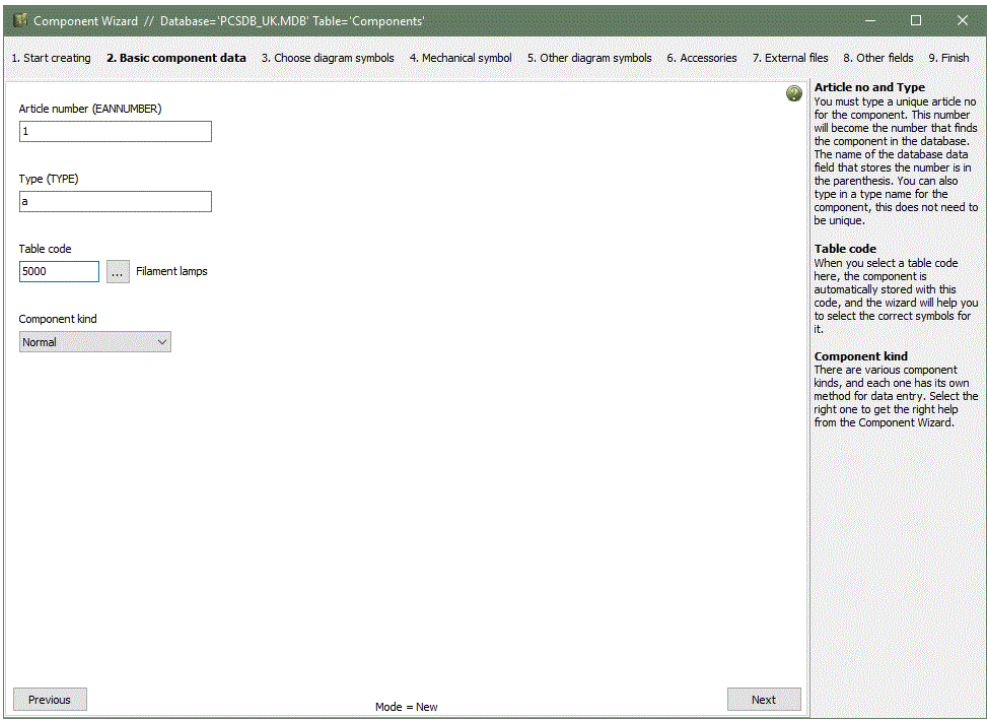
# CREATE A NEW NORMAL COMPONENT

When you click the button NEW you enter this dialog:

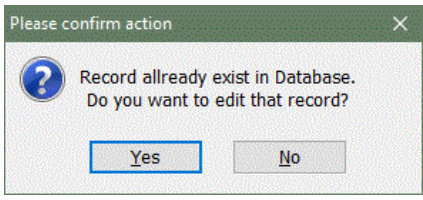
## Article Number and type

Here you type the EANNUMBER and TYPE for the new component.

In the default setup the Article number is mapped to EANNUMBER and Type is mapped to TYPE. That is the reason they are seen here in parenthesis.



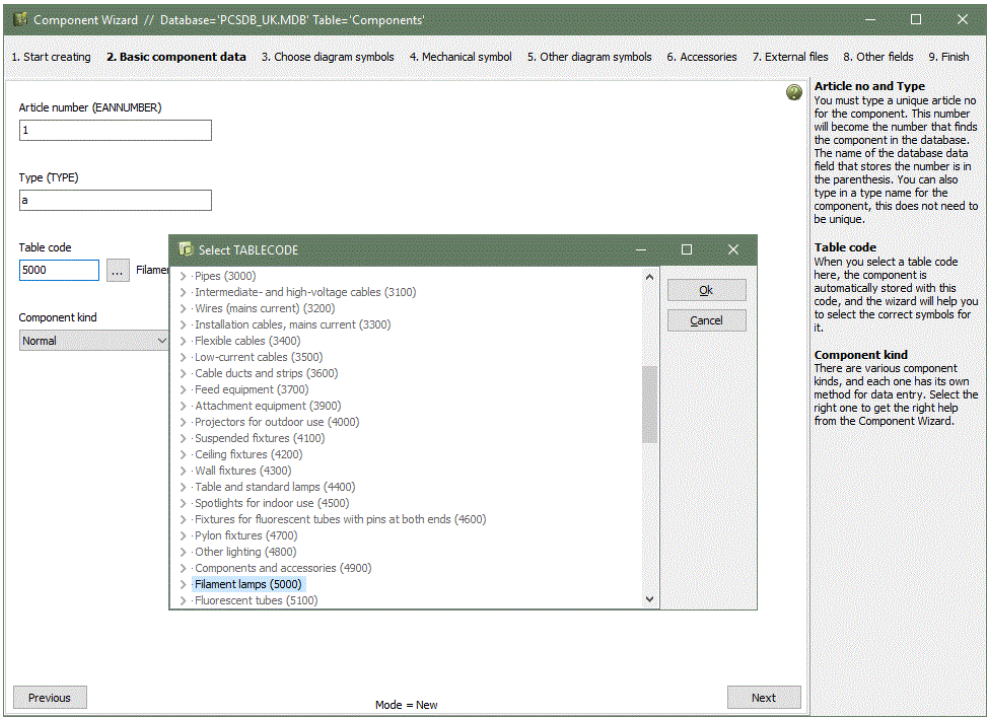

You can type any number, if your component does not have an EANNUMBER, but the number must not be used in another component. If that is the case, you will get this warning:



It is possible to select the appropriate Table code or Component group, for your new component

If you press the browse button you get a list of the table codes available in your database.

If you remember the number, you can also type it.



## Component kind

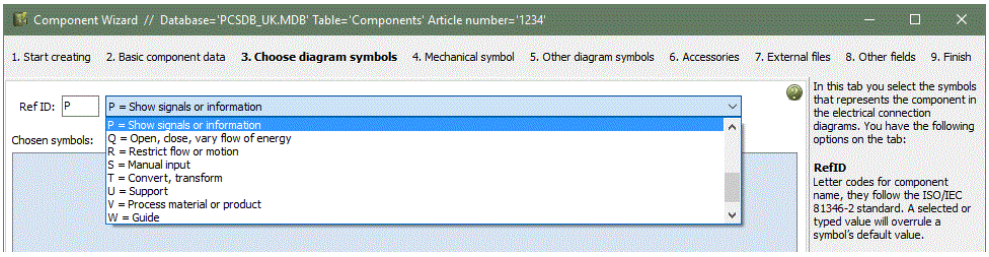
When you start creating a new component you also select the component kind.

Depending on the selected component kind, the Component Wizard will use different dialogs in the next tabs.


In this first section, you will see the basics of how to create components. In later sections you will see examples of special kinds.

## Select diagram symbols

In this window you start by selecting the reference ID for the component, it is optional but absolutely recommended.



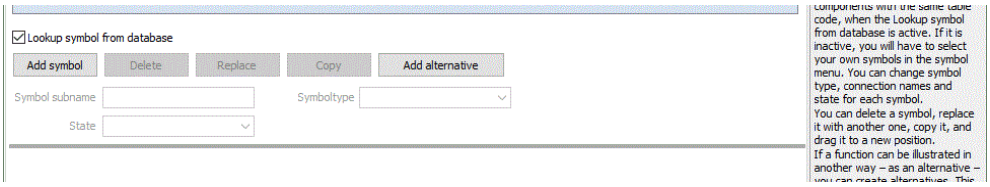

The letter code is based on ISO 81346-2.

And then you select the electrical symbols for the component. This can be done in two different ways, both of which can be seen below.

## Lookup symbols from the component database

The default setting of the wizard is that it looks up symbols in the component database.

It is an easy way to find the correct symbols, as you almost always use the same symbols for the same kind of components, ie components with the same tablecode use the same symbols.

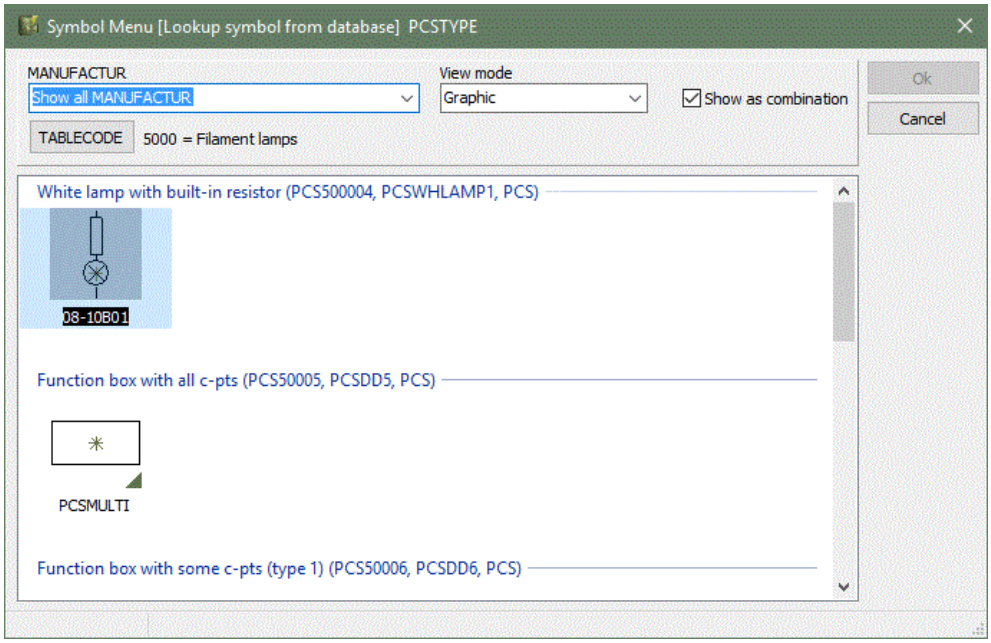



When you click the button, you enter this window:

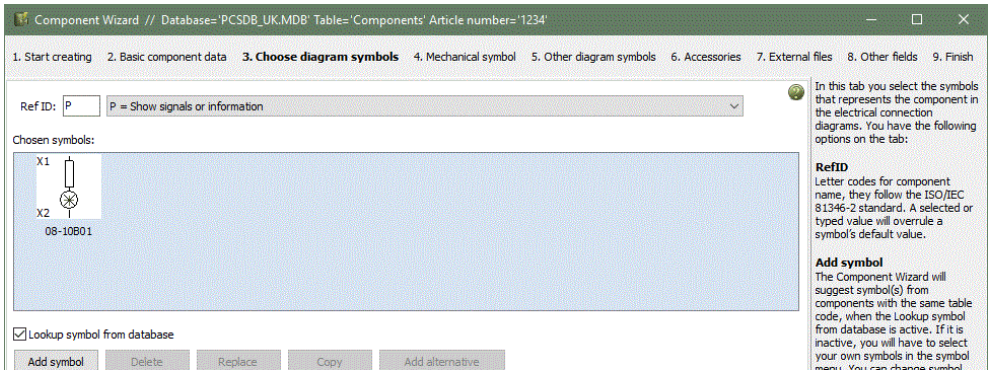
The wizard finds the used symbols in the component group/table code ‘Filament lamps. ‘Show as combination’ is selected

### Example 1

If you want to create a component with the same electrical function as one of the shown article numbers, ie *’PCS500004: White lamp with built-in resistor’* simply double click to select the lamp symbol.



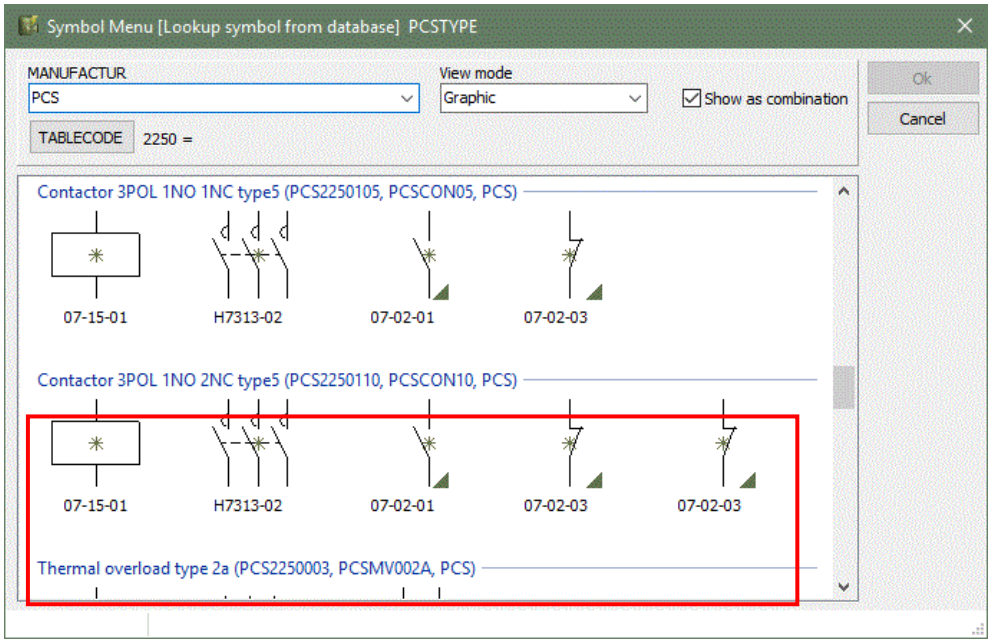
When you select the symbol in this way you also get connection names and possible connection point settings.




If this is ok, then click Next to select mechanical and other symbols for the component.

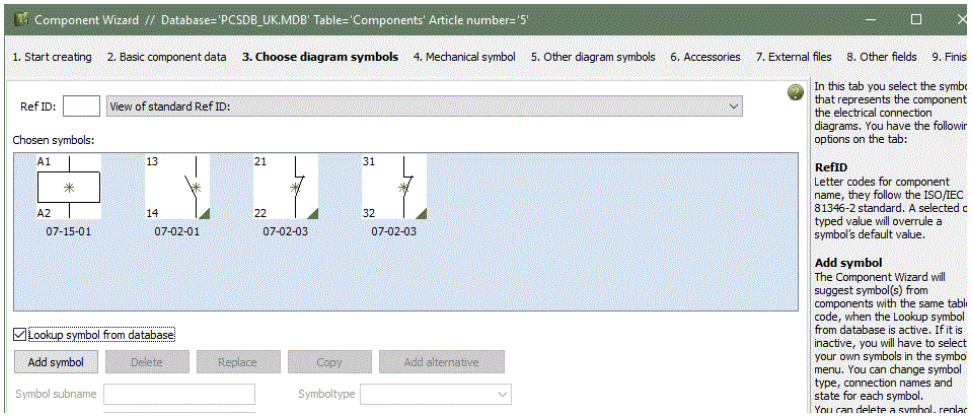
### Example 2

If you want to create a component with more electrical symbol, eg a contact, your windows will look like this:




When you select one of these sets of symbols, you also get it all:

You need to select the refID yourself, but the rest, meaning all symbols with connection names and properties are included.



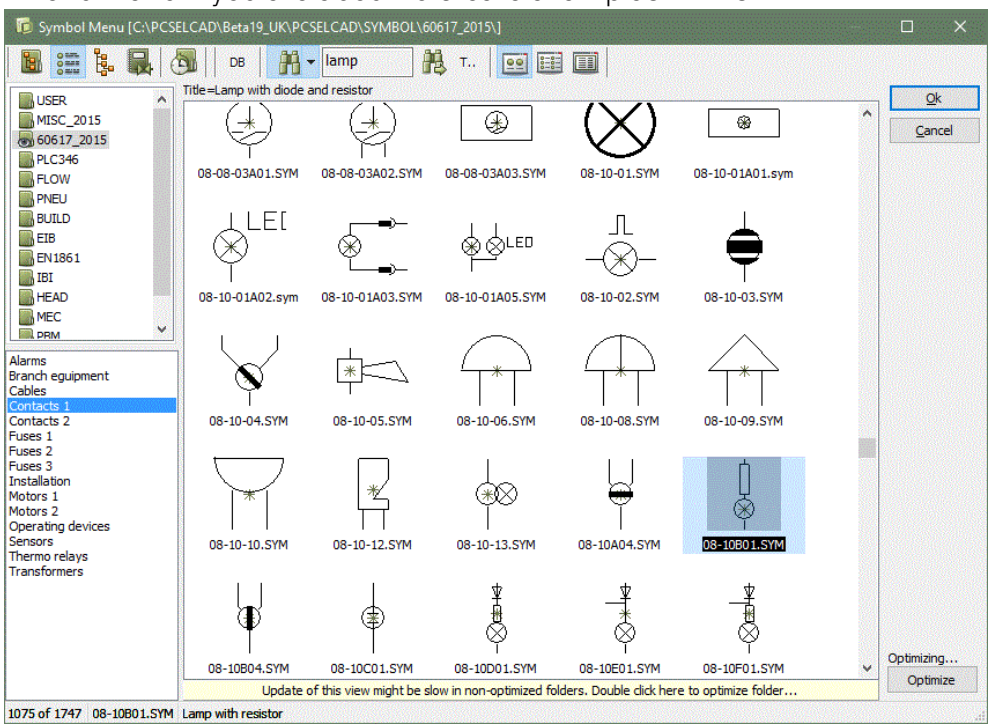
## Find symbols in the symbol menu

If you don't have a component of the same kind from which you can copy the electrical symbols, you can find the symbols in your symbol menu.

Deselect this setting and click the Choose symbol button.


Now you enter the symbol menu – and if you are about to create a lamp as in this example – you go to the 60617 folder to find a lamp symbol.

Here, the symbol for lamp with resistor.



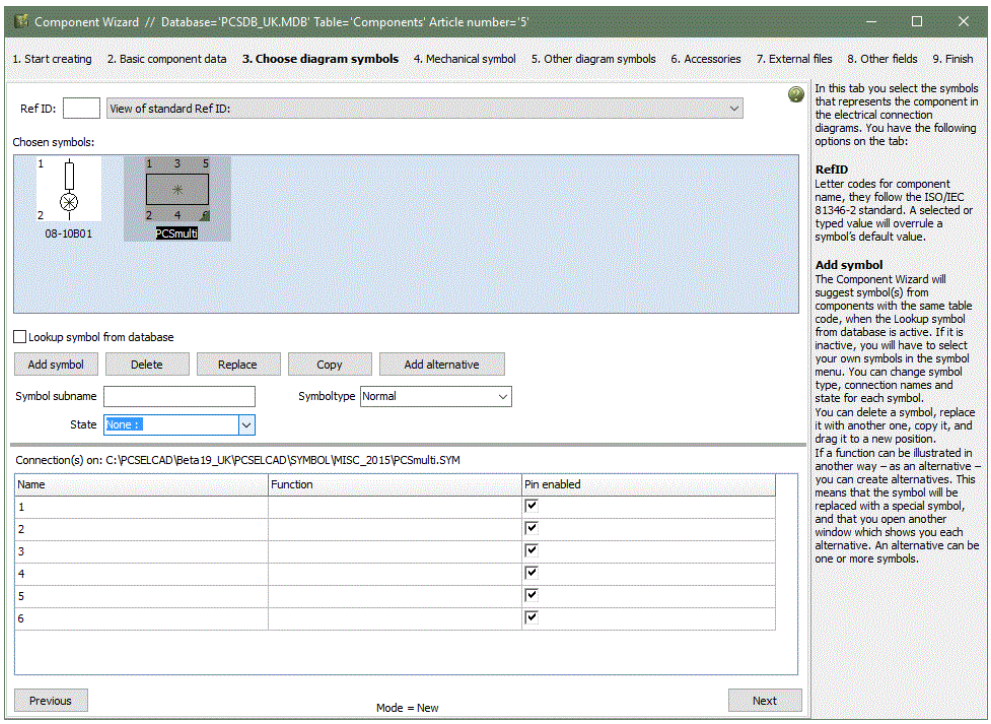

When you select the symbol you can see it in the window.

Here you can see the lamp from above and the PCSMulti from the MISC-folder.

When you mark a symbol, you can delete, replace or copy this symbol.

You can also select an alternative symbol, which you can read more about alter.

You cannot open the Symbol Generator to create diagram symbols directly from the Component Wizard.



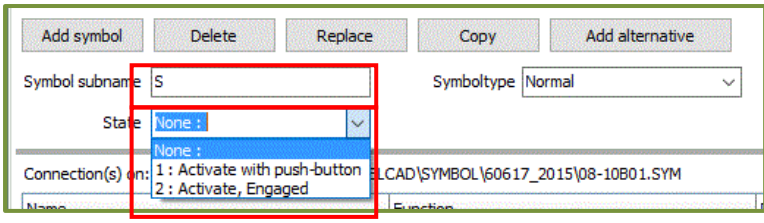
## Symbol settings

You have the following options when you selected the symbol, depending on the actual symbol(s).

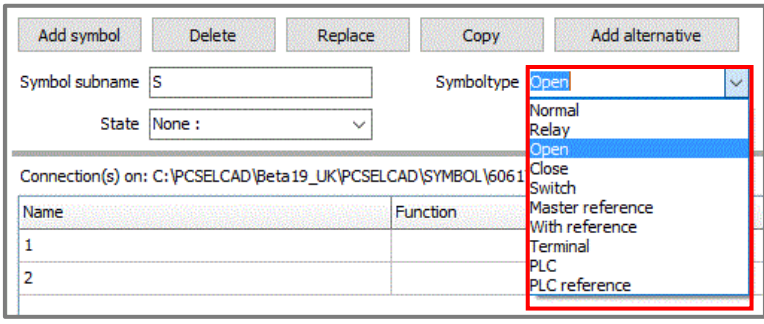
Generally, you have the same options for each symbol as you have when you design the symbol:

- You can name the symbol with a subname: if you are creating a switch with an indicating lamp, the refID of the component is S (the primary function of the component), the switch symbol gets the subname S and the lamp symbol get the subname P.

- If you selected a symbol with symbol states, you can select a specific state for the current component

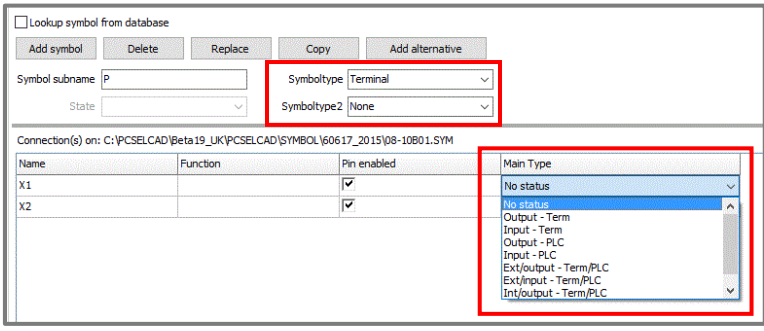



- You can change symbol type. If you select a type with subtypes you get further options




- You can copy the symbol – with all its settings.

- In the buttom part of the window you can type the connection names for each symbol. Depending on symboltype you will see extra options here as well.




If you type more than the data field size allows, the system automatically creates 'pcs' or 'pin'-files.

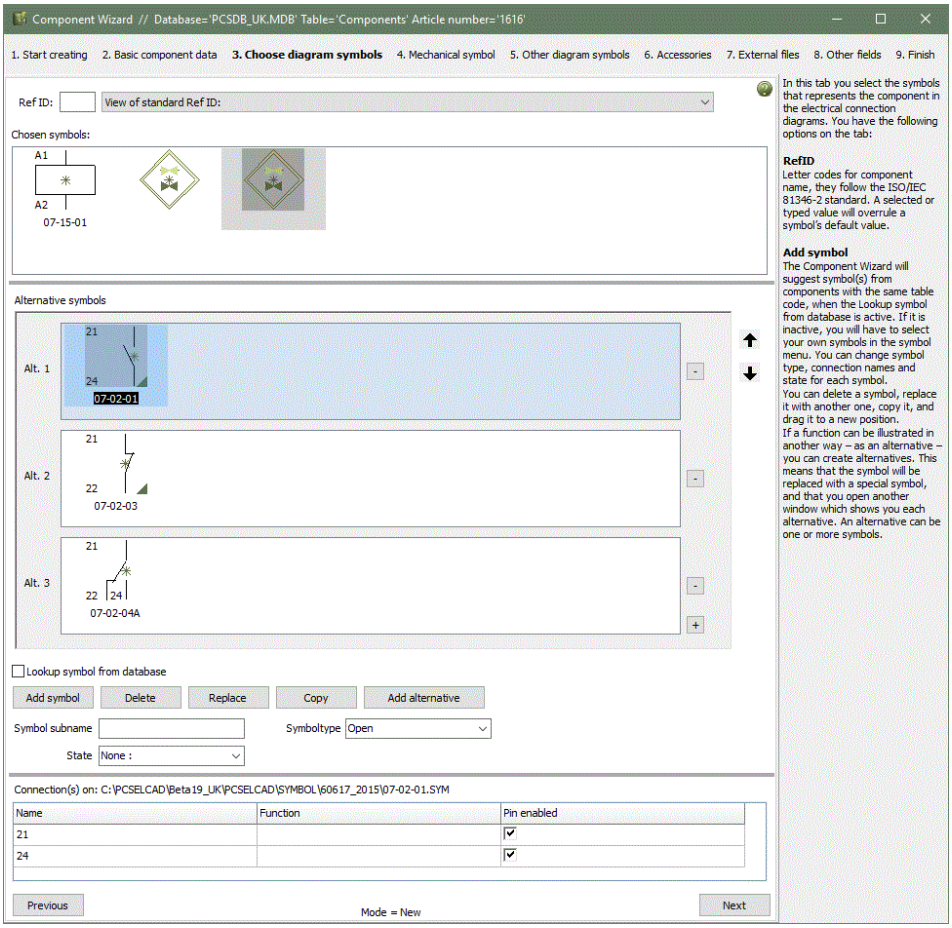
## Alternative symbol selection

For some components, you can select between symbols.

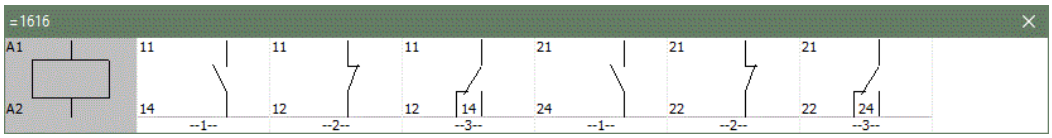
In this example, you see a contactor with an activating coil – selected as shown above – and two sets of contacts, that each are selected as *either* a make (NO), a break (NC) *or* a change-over contact.

You create the alternative symbol selection by placing the first selection, then mark it, and then click the Add Alternative button.

This will replace the symbol with the ALT-symbol and open a new window where you add the three alternatives.



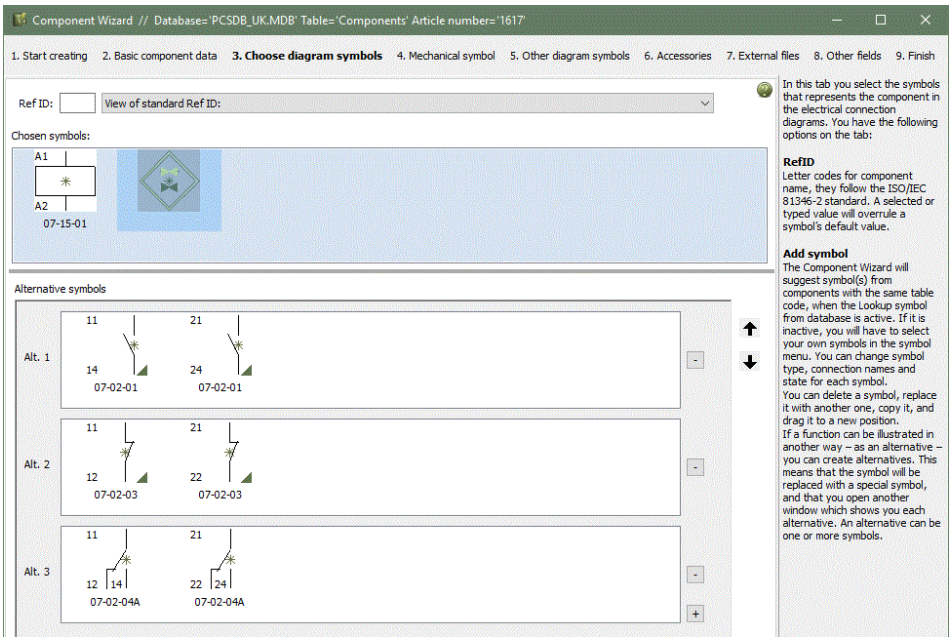

When you pick the component from the database, you will get this result:

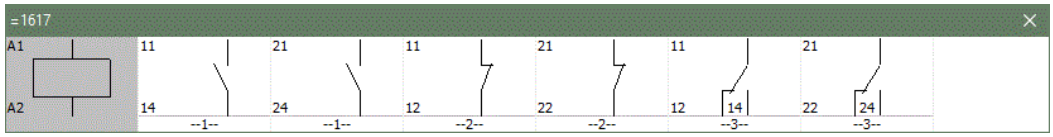



This window shows that first you have a coil, then you have two contacts, each with three alternative symbols: a make, a break, or a switch. When you select one, the others are gone.

Here you have a wrong creation of the component: It has *either* two make *or* two break *or* two change-over contacts.

When you pick it in the database, it will look this way: So, shortly, pay attention to each symbol position in the component.

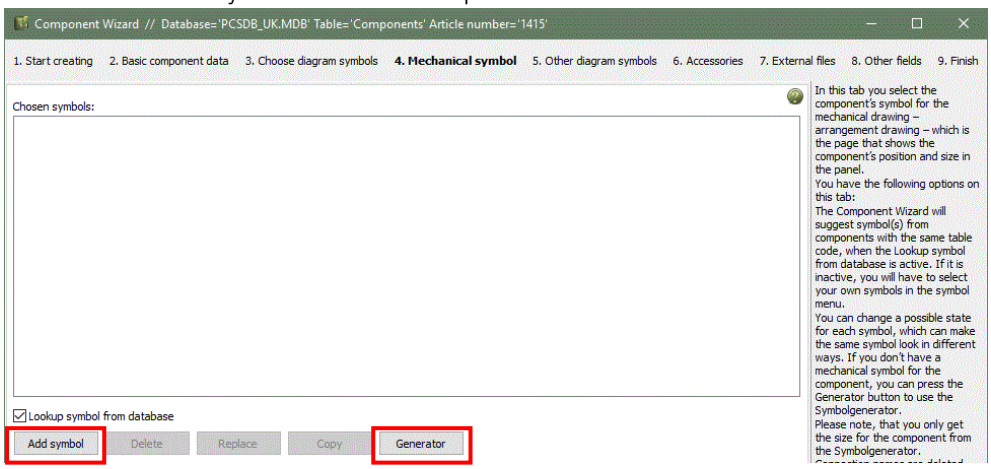


## Mechanical symbol

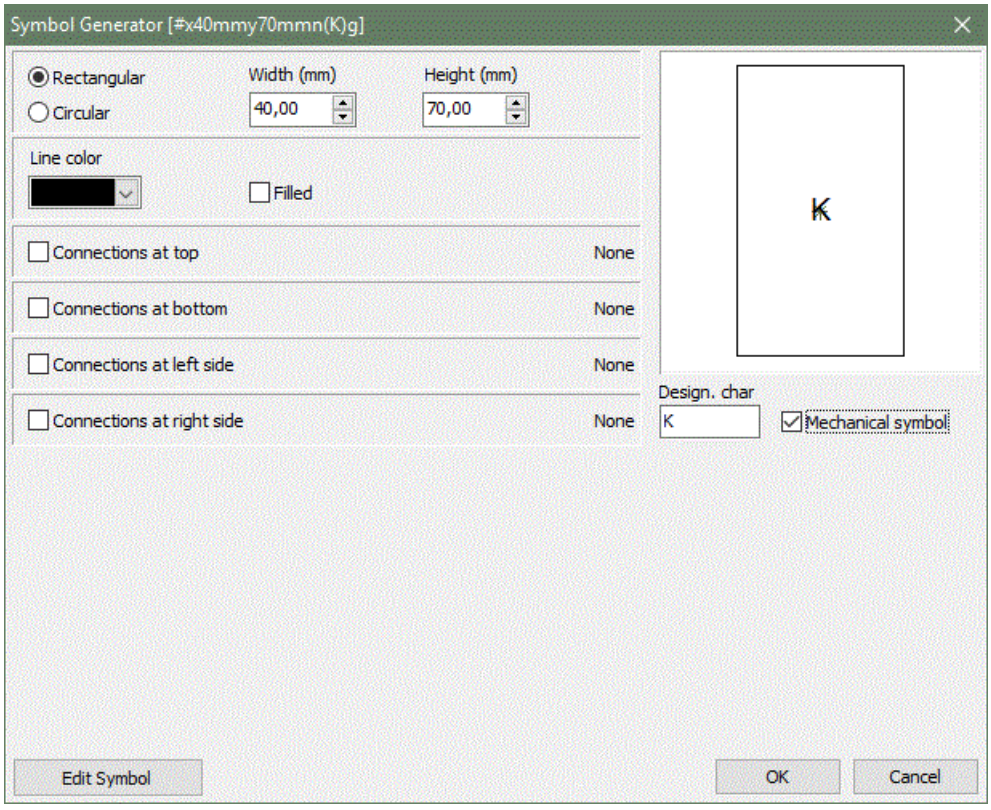
In this tab you can add a mechanical symbol to the component.

Database lookup is also possible when looking for mechanical symbols, and in this way, you can find the symbols that are in use for components in the selected component group.




Alternatively, you can click the Generator button to open the Symbol Generator.

Remember the RefID / Design character.



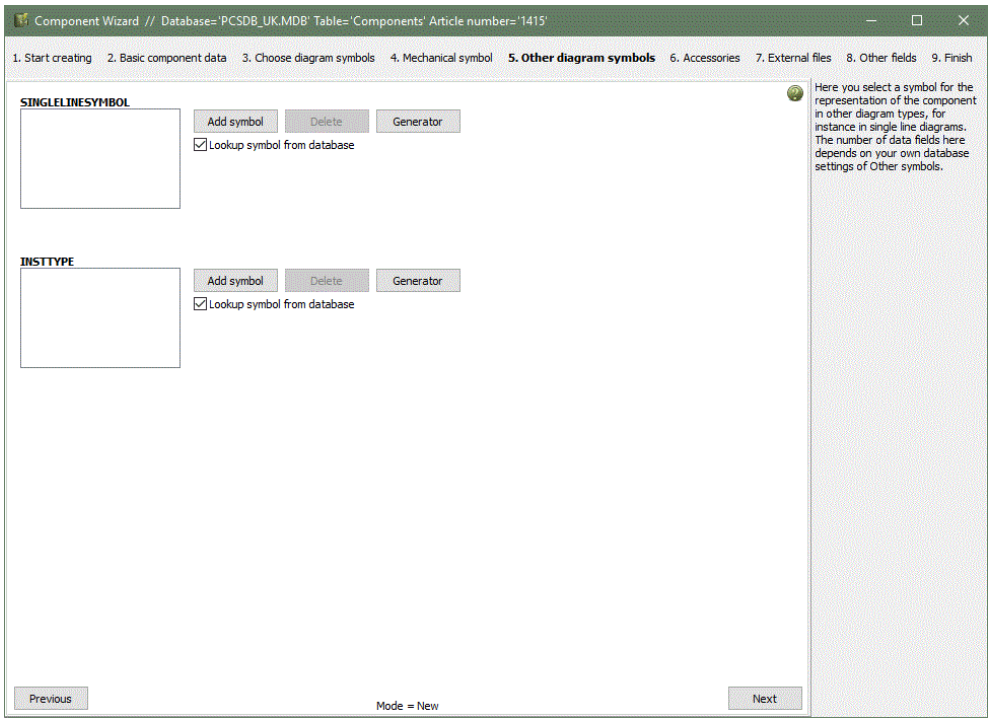

**Note** that the symbol generator only saves symbols without connection names, when you use it together with the Component Wizard.

If you want a mechanical symbol with connection terminals – and that is definitelly a recommended – then simply make a mechanical symbol by using the Symbol generator (and Symbol editor) to create a symbol with a name, and select this symbol with the Component Wizard, when you create the component.

## Other diagram symbols, e.g. SLD and Insttype

The Single line symbol field is this page. If you have need other types of diagram symbols and you have created data fields for those in your database and made them accessible as seen in page 41, you will can see the data fields in this tab.

You choose the symbols as seen in the previous pages, either by database lookup or by looking up in the symbol menu.




One way of using the other diagram symbols is to dedicate certain data fields to certain kinds of symbols, ie Single line symbols in one data field, communication symbols in another, power supply in a third etc., and then you select the data field on the diagram page. By default, the PCSTYPE is connected to DIA-pages and the MECTYPE is connected to GRP-pages.

It applies for all data fields, that you only see the data fields that are mapped in your installation. See more from page 41.

# COMPONENT ACCESSORIES

You can attach accessories to all components. Below you can read about the different types of accessories that are in the program and how they work.

Accessories come in three types.

## Fixed accessories

Fixed accessories are in the ACCESSORY data field.

The accessory is only seen in parts and component list. It inherits it name after the component it is attached to.

You can select/deselect fixed accessory from the part/component lists in each list's settings.

## Optional mechanical accessories

You find it in the OPT_ACCESSORY data field.

In earlier versions of the program, this kind of accessory was placed using the Support symbol. Optional accessory – where you can select between several possible articles. This might be dividers and end plates for terminal rows.

## Optional electrical accessories

You find it in the OPT_ADDON data field.

In earlier version you would place this as individual electrical symbols, eg auxiliary contacts, which were manually named after the main component. In this way, it was shown correctly in various lists and the arrangement drawing.

The new data field makes it possible to maintain info, telling whether an add-on component is valid or not for the main component.

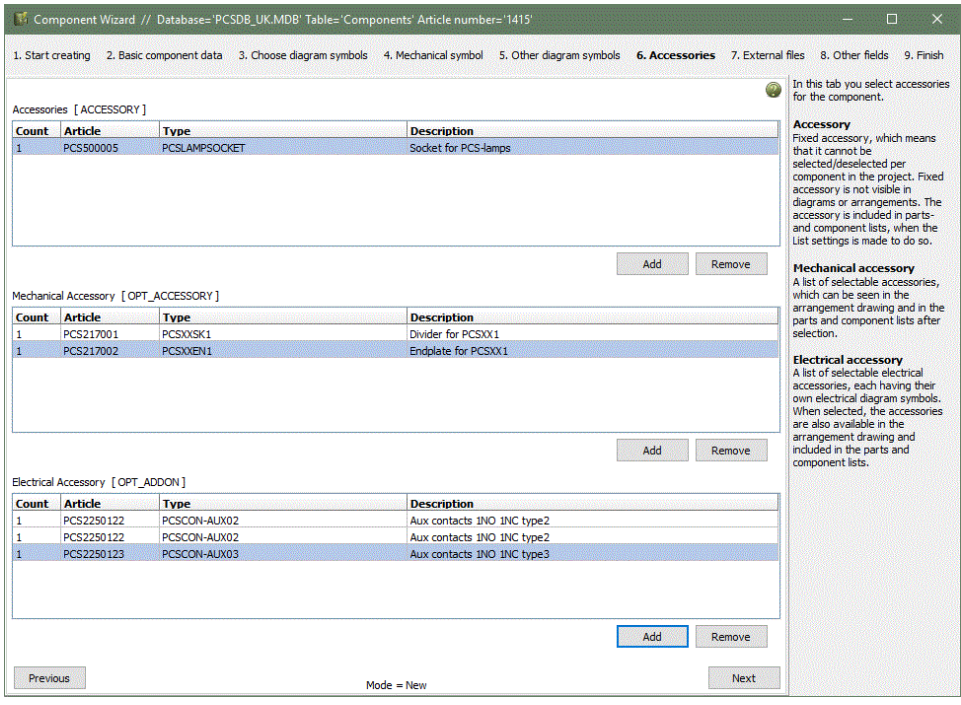
Optional electrical accessory is included in Show available on DIA and GRP-pages.

## How to select accessories

The desired accessory must be created as individual components in the database, that means it must have its own EANNUMBER.

For each type of accessory, simply select by clicking the Add button, give it a new count if it is not 1, and click the Remove button, in case you made a wrong selection. Counts can also be change on the project component.

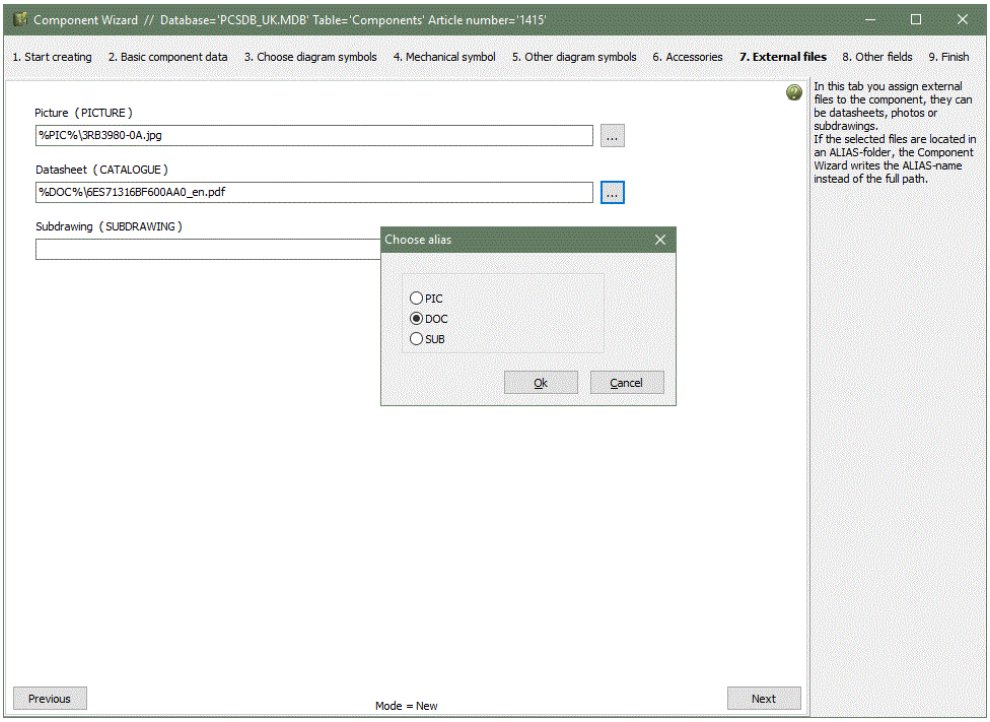
The picture is an example.



# EXTERNAL FILES, E.G. PICTURES, AND DATA SHEETS

In this tab you can attach external files, e.g. pictures or data sheets.

The program will look up in the preferred ALIAS folder for the data field and if you select a file from this folder, the ALIAS-name is automatically written in the data field. Alternatively, the program writes the full path.



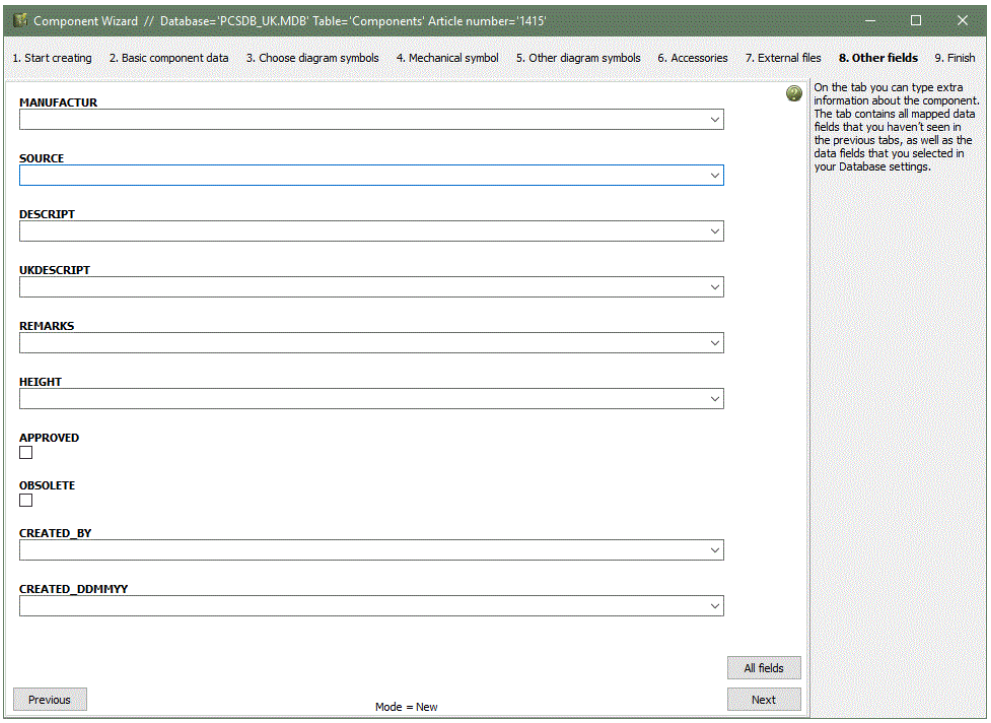

## Other fields 2

In this window you see the rest of the mapped data fields.

You can map other or more data fields in Settings|Database settings|Component wizard.

See more about this from page 41.

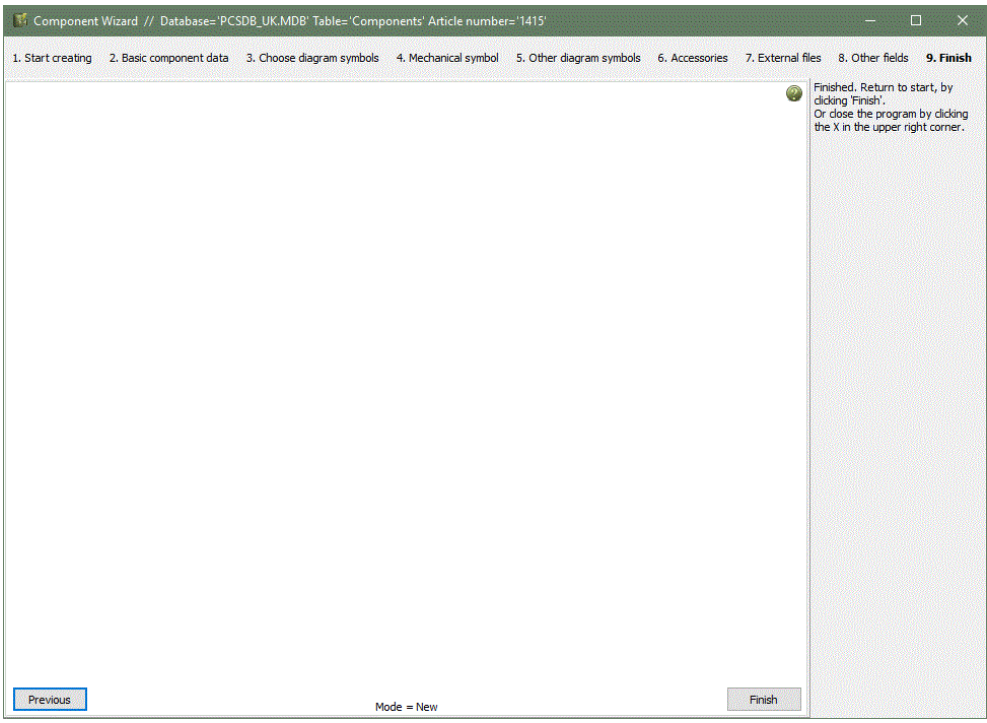
You can also press the ‘All fields’ button, after which the complete data record opens.



## Finish

If you exit now, you will be directed back to the first tab of the wizard.

And if you close the wizard and open the component database, the new component is at the top of the list.

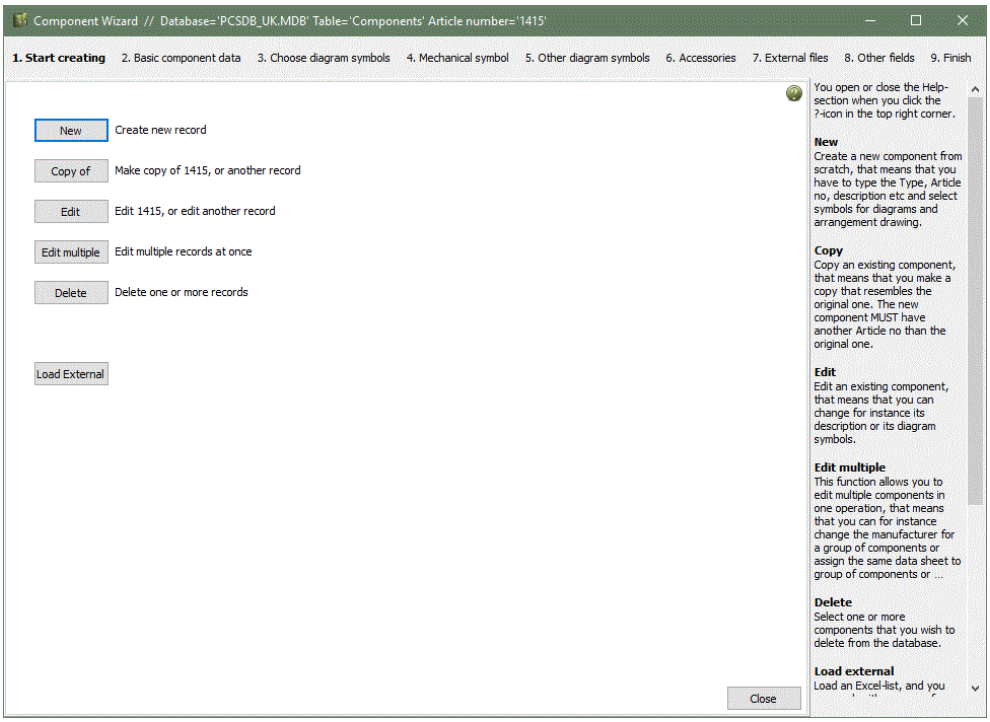


## First tab – once more

When you are back here you have the same choices as you did the first time.

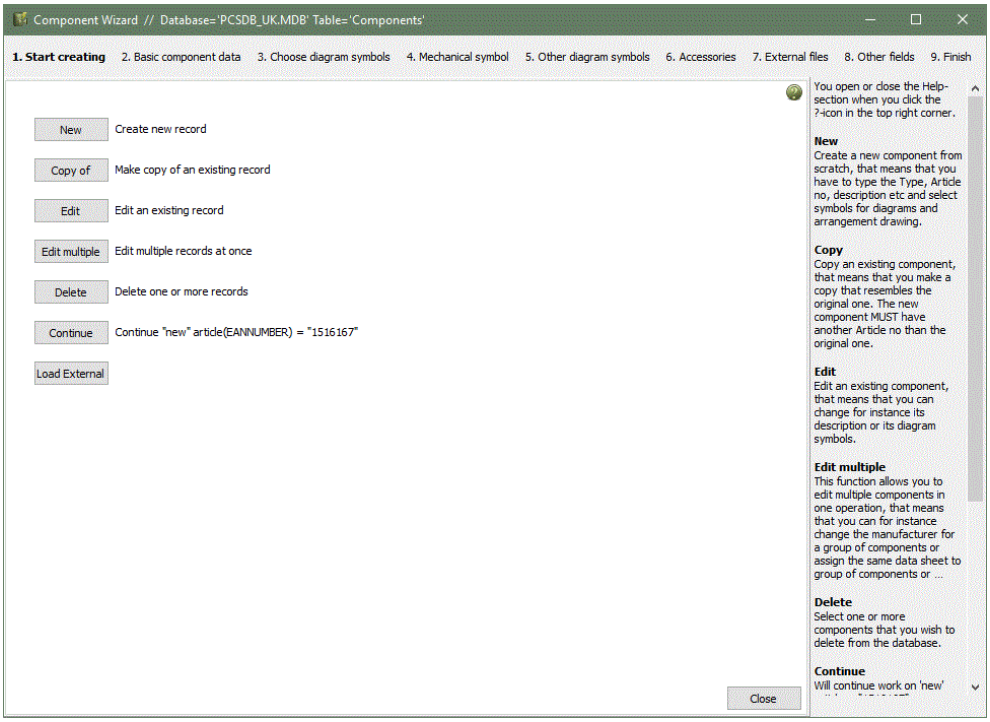
‘Copy of’ suggests that you make a copy of the component, you just made.

‘Edit’ suggests that you edit the component, you just made.



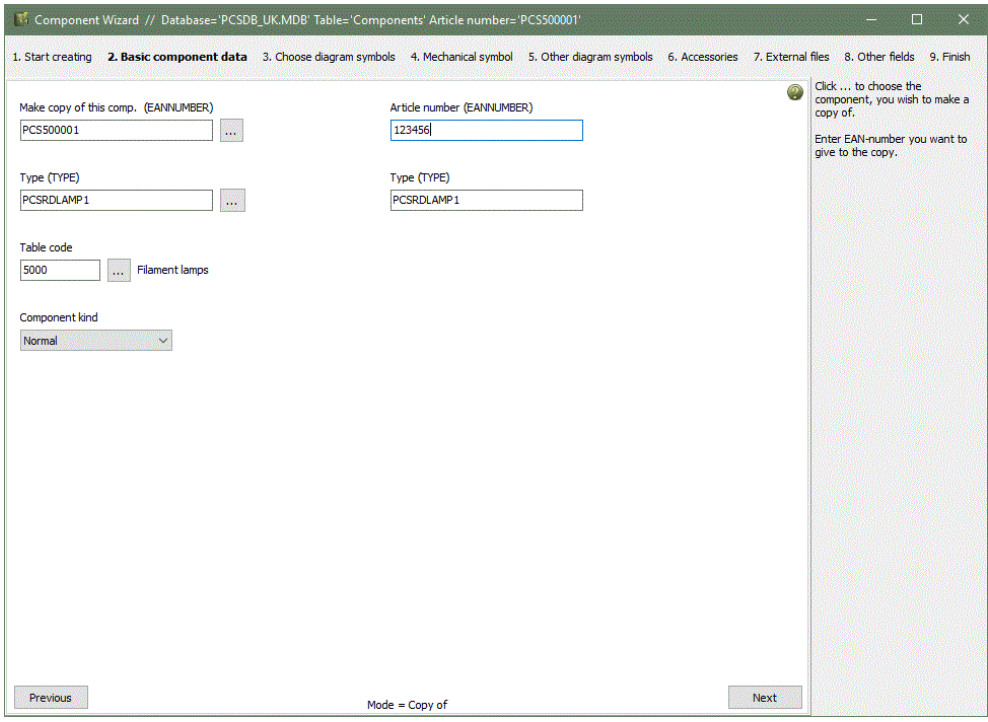

## First tab – once more - again

If you stop somewhere in the process of creating a component, the wizard will remember what you were doing - creating/ editing/ copying – and will suggest that you continue with this.



## COPY OF AN EXISTING COMPONENT

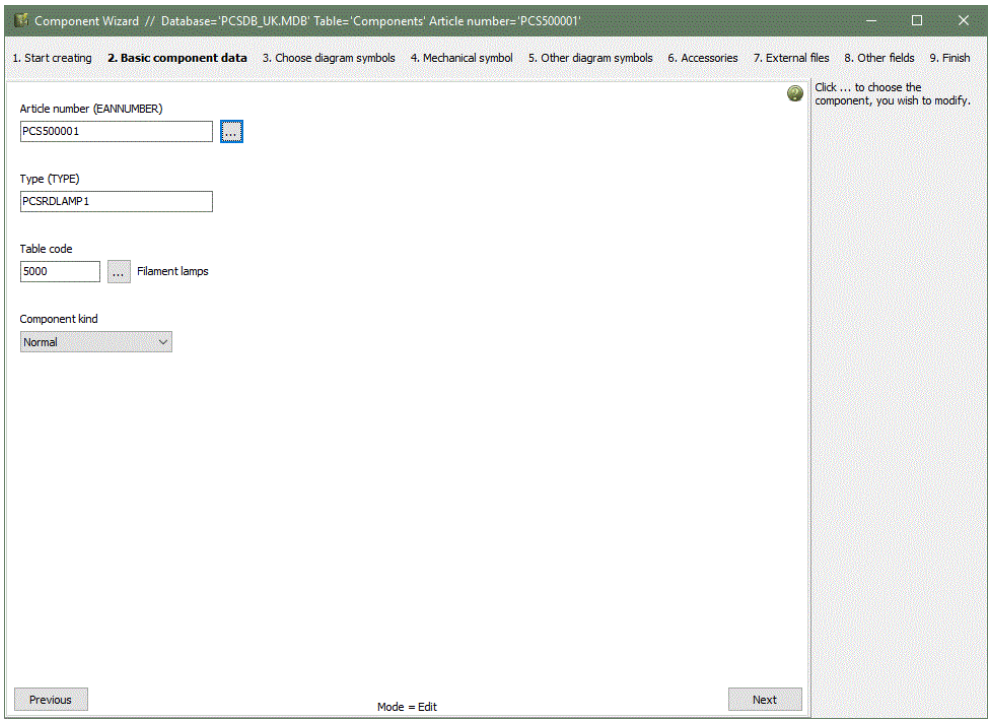
If you choose the Copy-option, you will get this window, where you download the "original" and give it the new name. As the 'copy' and the 'original' component often have type and item numbers that are very alike, the wizard suggests the original's values. Then continue as "New".




You are, however, not allowed to continue "copy of" unless you type a new, non-existent article number. Alternatively, the wizard suggests that you edit instead.

## EDIT EXISTING COMPONENT

If you choose this option, you must also fetch the component by clicking the '...' button. Then you continue with the same tabs as "New".



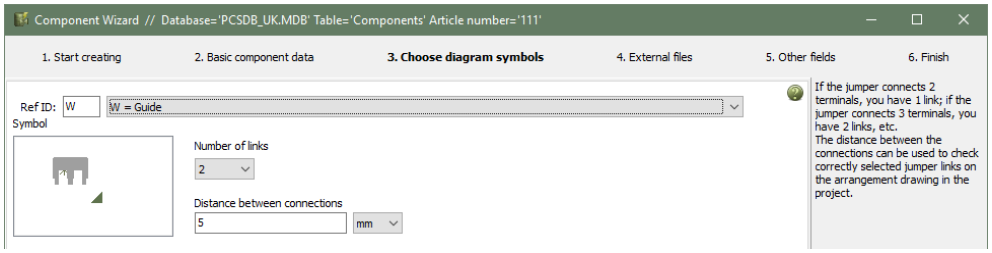
# CREATING OTHER COMPONENTS KINDS

When you create a new component using the Component Wizard, you can choose between different component kinds.

Below you will see examples of different component kinds and how they are created in the database.

## Create Jumper link

When you create jumper links in the database, select (and show) how many links it contains and type the distance between each link.

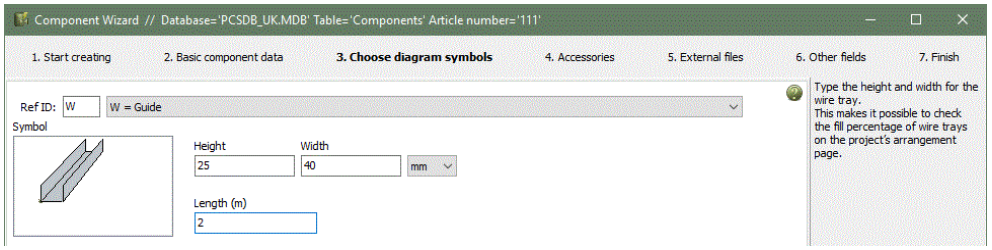



## Create Wire

When you create wires in the database, type its outer diameter (mm or “) and meters per reel/drum. The diameter is used to calculate capacity in wire trays when using the Panelrouter. The length is used to calculate quantities for the parts list.


## Create Wire tray

When you create wire trays in the database, you type height and width (mm or “) and the length of one section. The dimensions are used to calculate capacity when using the Panelrouter, the length is used to calculate quantities for the parts list.

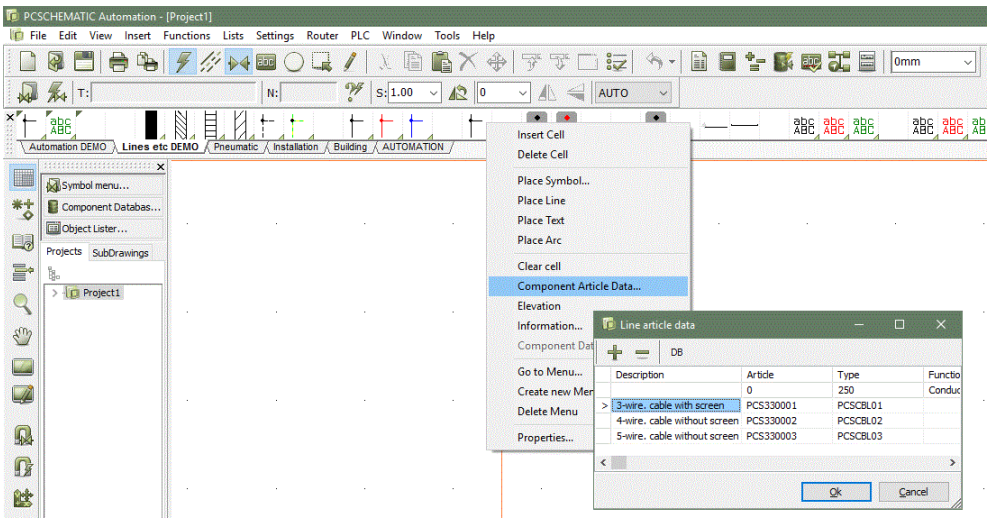


## How to draw jumper links, wires, and wire trays

The main reason for creating jumper links, wires and wire trays in the database is that the Panelrouter uses the data to calculate capacity in wire trays.

If you want your drawing to contain a certain wire type, you cannot pick it directly from the database, you need to pick it from the Pick menu:

Add a line with the desired type, width, and color to the pick menu, it could be a blue line for a normal conducting wire or a pink line with jumper status. You assign component data to the line by right-clicking on the line, select Component article data. The select the wires from the database – click DB. You can have several wires on one line in the pick menu.




If you create wire trays in the pick menu, you only select the line type and color, that data in the database – and the page scale – will decide the width on the drawing.

You can read more in the manual about adding components to the pick menu.

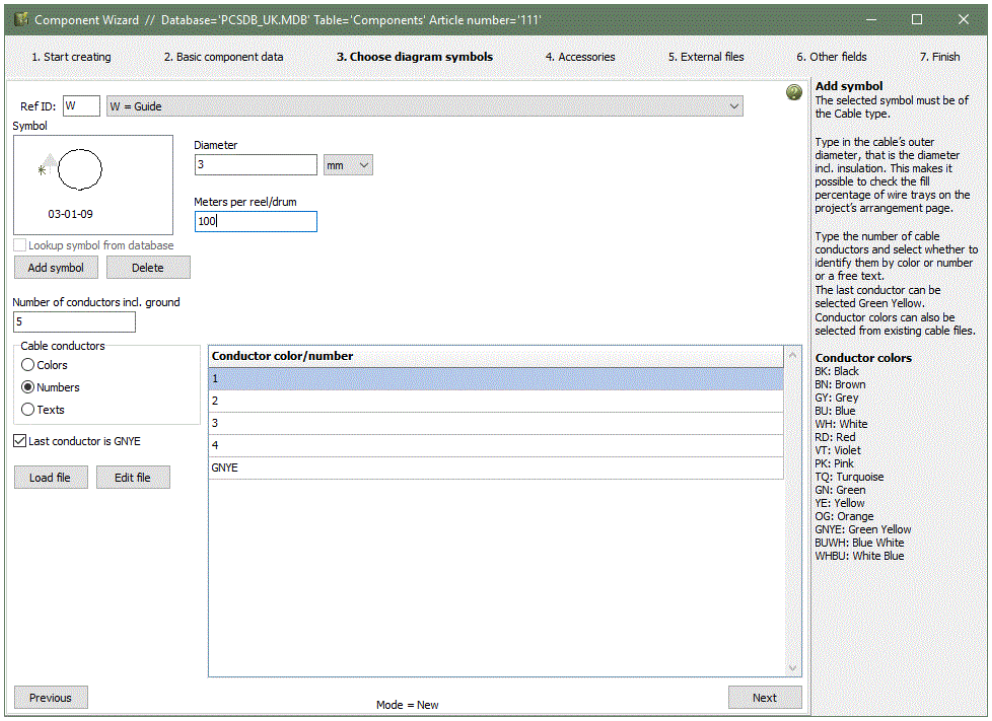
## Create Cable

When you create cables, you must select a cable symbol, either through database lookup or manually in the symbol menu. The Component Wizard only accepts Cable symbols!

You can type diameter (mm or “) and meters per drum. The diameter is used to calculate capacity in wire trays when using the Panelrouter.

After this, you must select the number of conductors and how to mark the conductors – colors, numbers, or free texts. The wizard helps you to automatically fill out the list with colors or numbers.

The help text shows color codes.




## Load file

You can create cables with specific conductor colors by loading an existing file. Cable files are made in Notepad or similar, and they (often) follow a cable standard, they have one conductor color or number per line, and they are saved as *.cbl-files in the database folder.

## Edit file

If a selected file is wrong, you can edit it. Beware that other cables might use the same file,

## Last conductor is GNYE

When you select this option, the last conductor will be GNYE.

## Plugs for cables – Accessory

When you go to the Accessory tab, you can assign Accessories per cable or per meter.

# CREATE PLC

The Component Wizard can also create PLCs in the database.

You start by selecting the PLC component kind, after which you enter a special PLC page.

Below you will find examples of various kinds of PLCs, which will show you how to create the different kinds of them in the database.

The first example will take you through all the dialogs, showing you the options, you have.

## Example 1: Card with 8 digital inputs and separate supply

### Select the number of inputs and outputs

Here you tell about the structure of the PLC.

#### Definitions

Channels – we have selected the word Channel, which mostly will be synonymous with Address, but sometimes it means plugs or channels.

Addresses – mostly it will be the same number as in Channels, but where you have plugs, single plugs or channels can have more addresses. You select the number of addresses on the next tab.

#### IO status

Here you choose whether your channels are inputs, outputs, other (e.g. IO-link) or a combination of those.

#### IO status type

You can create your channels with digital, analog or no status type. The status types can be used as filters, e.g. when you transfer plc addresses in the project.

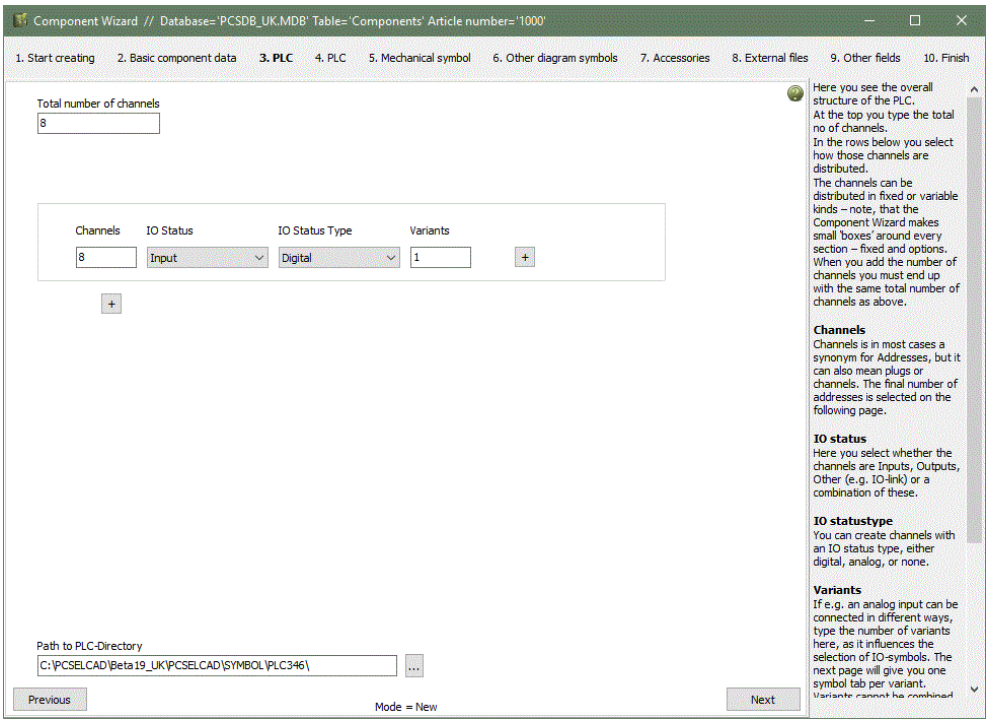
#### Variants

An analog input can be connected in different ways, depending on the equipment you connect and the input type you want. Here you type the number of different connection methods, as it influences the IO symbol selection. Variants cannot be combined with input/output combinations in IO status.

#### Path to PLC-directory

Here you select the directory from which you choose your plc symbols. Default selection is the PLC346 directory.

## Start creating



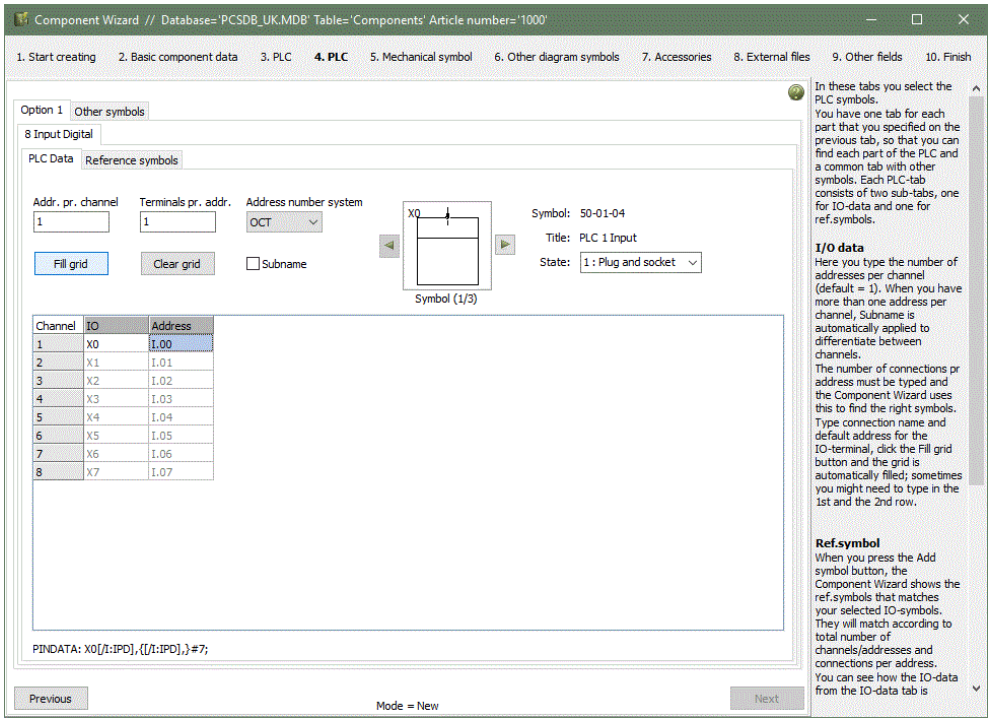

Start by typing the total number of channels, here 8.

Then you type the number of input channels – 8 again. Press the Next button.

This tab is the starting tab when create PLCs. Here you select its overall structure and whether you have fixed or variable functions on each address.

See the examples below to learn about the option you have in this tab.

## Select PLC-symbol




Type the number of addresses per channel (default = 1) and how many connection points you have per address.

With the combination in the picture, the program finds three possible symbols, which you can choose between using the arrow keys.

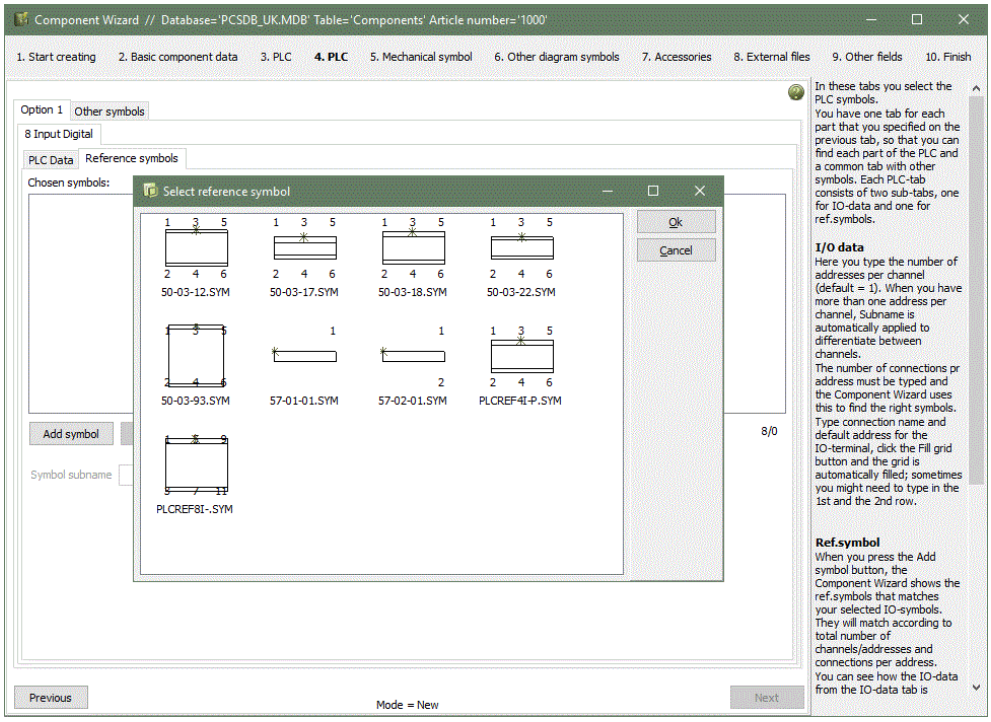
You can change to another IO symbol if you are on this tab. If the IO-symbol contains states, you can select one by using the drop-down list.

Type the first connection name – the connection with IO-status – type the first address and place the mouse on the 'Fill Grid' button to see what the program suggests, If it is ok, simply press the button, and you have addressed the IOs.

## Select PLC reference symbol

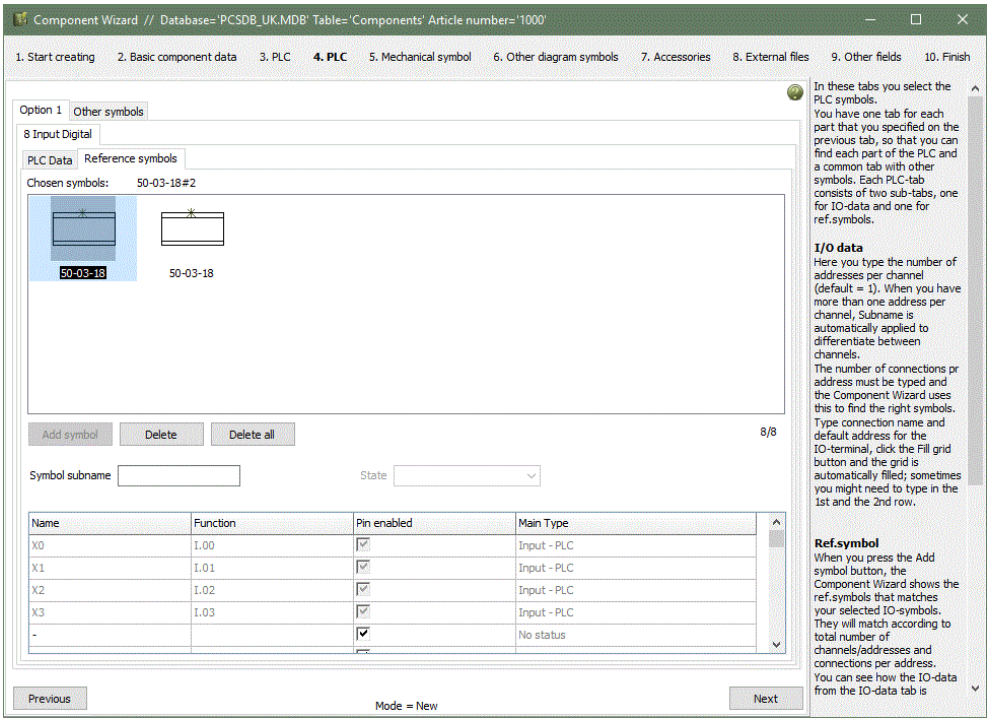
When you press the 'Add symbol' button, the Wizard has already found the reference symbols that match your selected IO-symbol.

When you have 8 inputs, you can choose between ref. symbols with 8, 4, 2 and 1 inputs. The ref. symbols must match in terms of connection points per address, which is difficult when you create the plc's manually.




Here the selection is 8 ref. symbols each with 4 addresses.

The 8 connection points with input status have the same names and addresses as the IO-symbols in the first tab – which means that they match. The only thing left for you to do, is to decide whether you want to use the connection points at the button of each symbol or not. They are often used for supply, if you want to do that, simply type e.g. 24V and 0V, alternatively disable them.



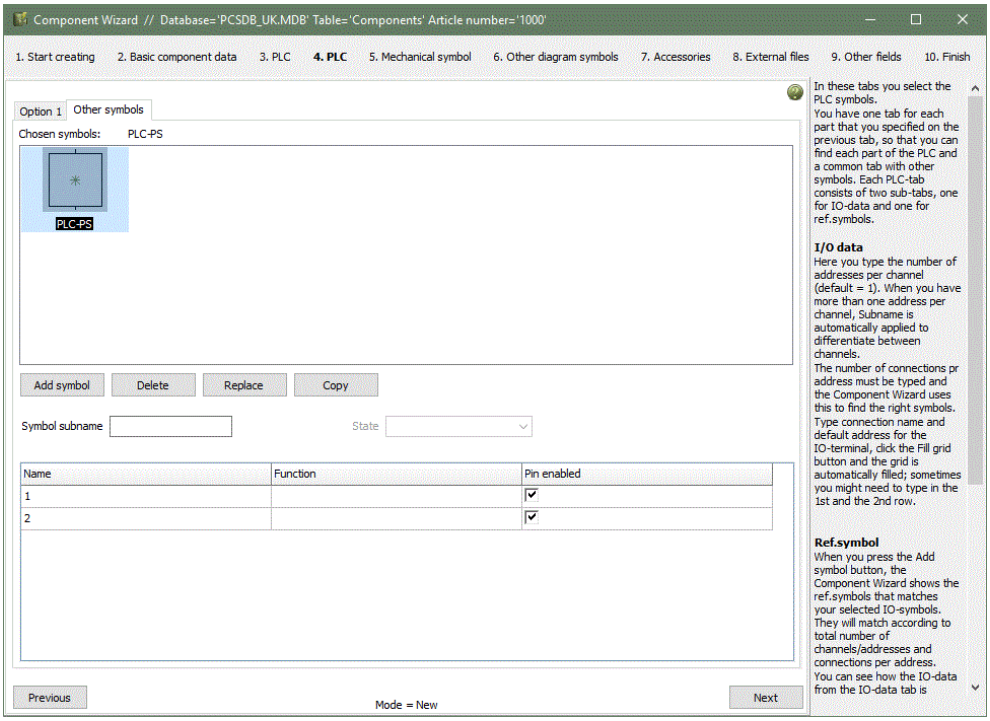
## Separate symbol for supply etc.

On the last tab you can select other symbols, for instance for supply and communication.

Press the ’Add symbol’ button and select a supply symbol. You can also select symbols for bus connections in the same way.

If you have made special data fields in the database – see pages 18 and 43 – you select the symbols on the ‘Other diagram symbols’ tab.

When you have selected all diagram symbols, press the Next button to go to the mechanical symbol, accessories, data sheet etc..



## Example 2: Compact PLC with 8 inputs and 8 outputs

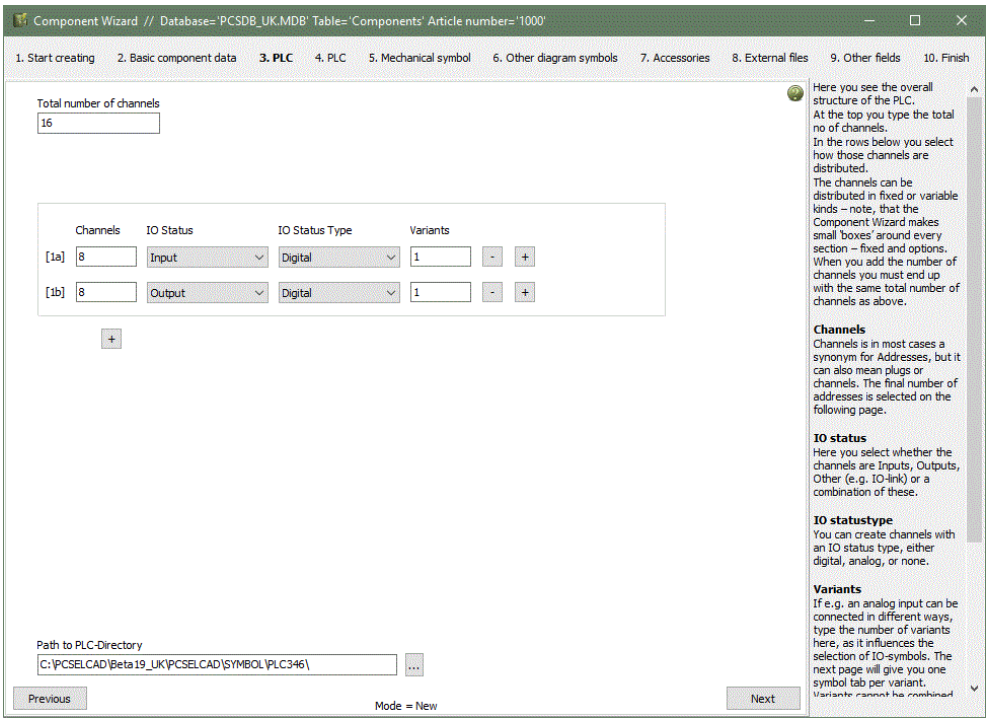
In this example you see a compact PLC, which consists of 8 fixed inputs and 8 fixed outputs.

Start again by selecting the component kind PLC.

Type 16 in the 'Total number of channels' field.

In the next section, you type 8 input and 8 output channels, which total to 16. If you do not get the correct total number, the Next button is inactive.

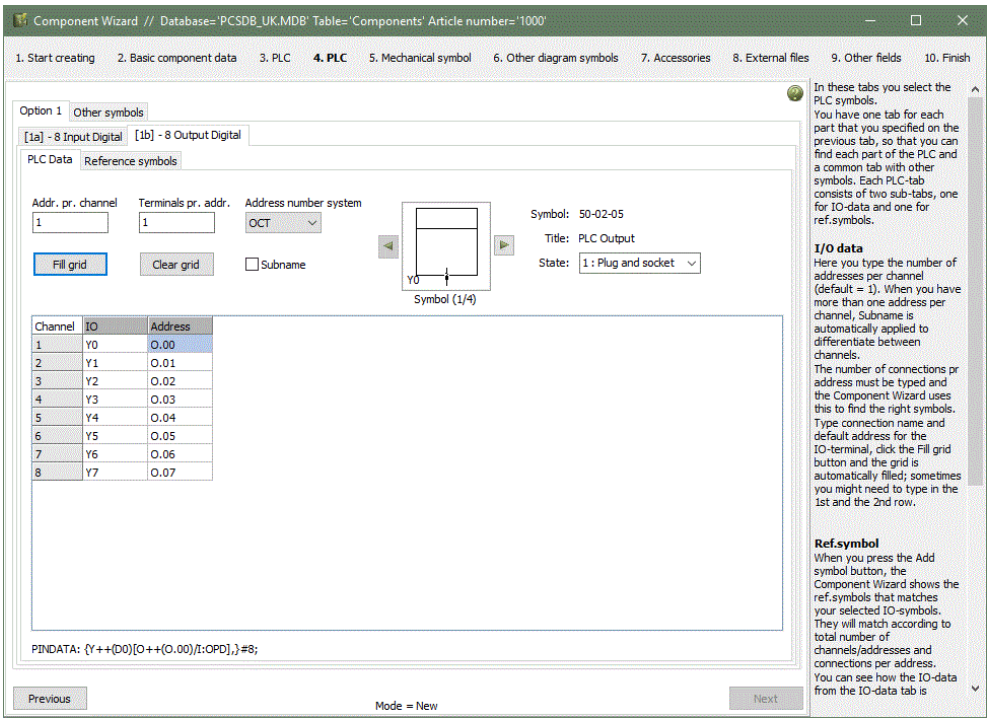
Press the Next button.




## Select PLC symbols and ref. symbols

Now you see 2 tabs in which you select PLC-symbols: one tab for inputs, one tab for outputs.

You use the 2 tabs in the same way as described in example 1, select PLC-symbol and PLC ref. symbol.



## Example 3: A card, that contains either 8 inputs or 8 outputs

The data on the first tab determines how the PLC is created.

That means that the number of channels and the combination of the channels are determined on this tab.

You need to notice the small ‘boxes’ around each section. The boxes indicate one part of the PLC and it can either a fixed section or an option.

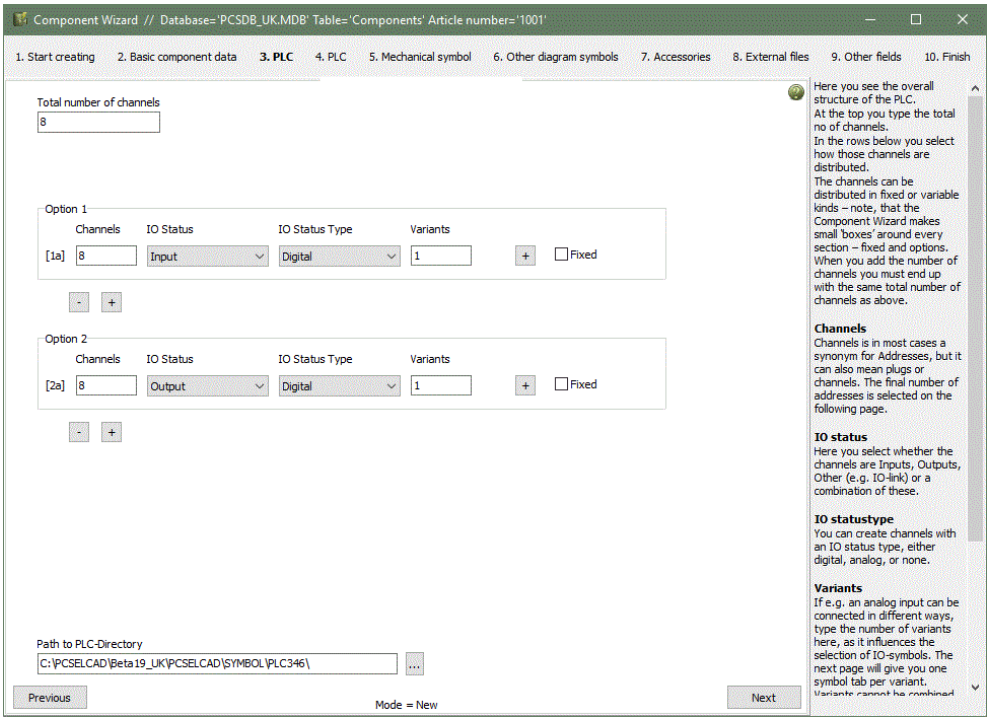
When you have more sections, each section gets a name, and this name is also found on the following tabs in which you select your symbols.

Below you will find more examples of how the first tab can be filled out.

This example is based on a card where you determine – e.g. by using a jumper – that the card contains inputs only or outputs only.

The total number of channels or addresses is 8.

Option 1 is 8 inputs.
Option 2 is 8 outputs.




When you continue to select symbols for PLCs and PLC references, the dialogs look exactly as in the previous examples.

## Example 4: A card, where each address is either input or output

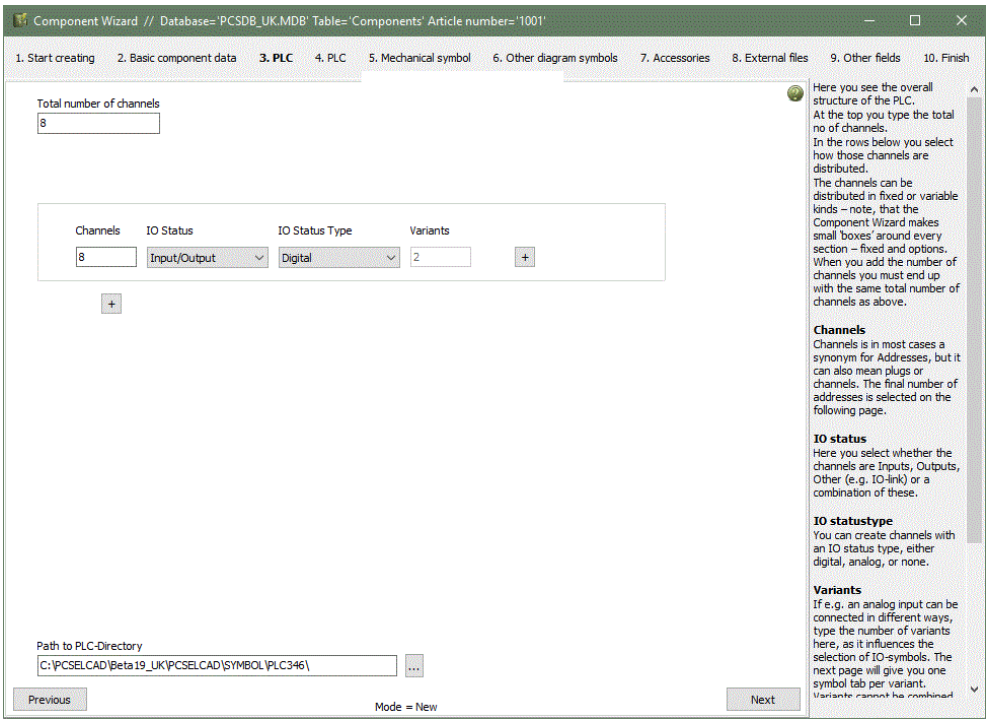
This example is based in a card where you address by address determine its IO-status.

The total number of channels is 8.

Option 1 is 8 input/output. It means that when you place a PLC-symbol you choose between input and output symbols for each channel.

This means 2 sets of tabs, where you select PLC symbols and PLC ref. symbols.

Selection of PLC ref. symbols is very easy: as you have a choice per channel, you also have 1 ref. symbol per channel.



## Example 5: Analog input card

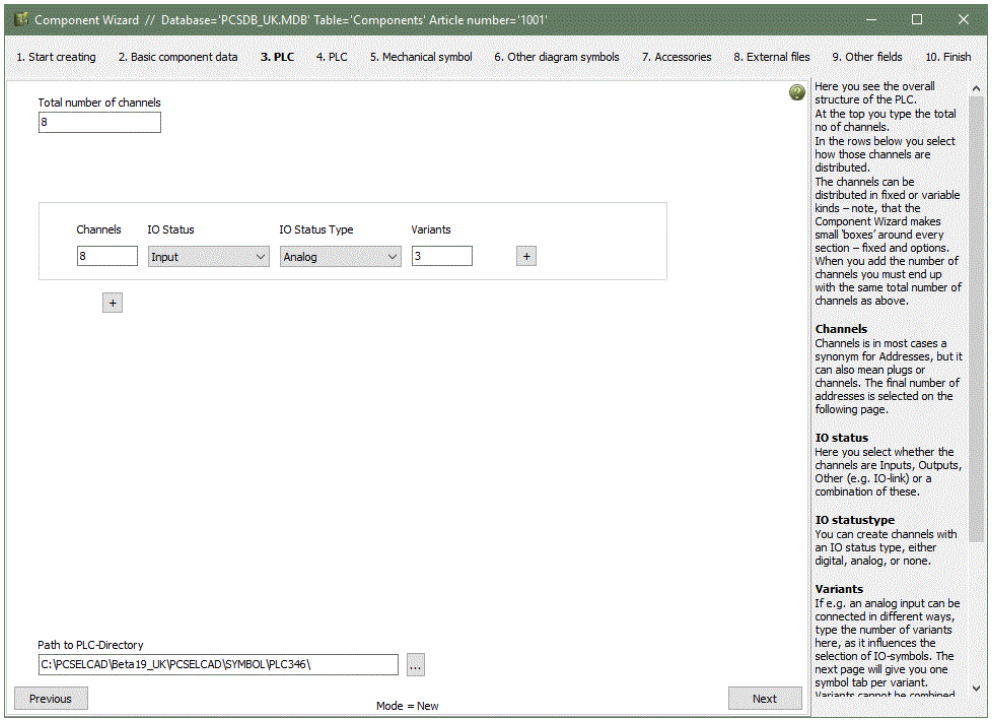
Analog cards can often be connected in different ways, determined by the measurements you want to get, e.g. a current value, a voltage value, or a resistance value. Which again means that you use different connection terminals per address/channel when you connect the sensor.

You can create the PLC with all its connection points in one symbol, but this will not work optimally in relation to PLC-lists etc. The solution is to make symbol sets per connection method, a solution which is very time consuming when you make it manually, but very easy, simple, and quick when you use the Component Wizard.

On this first tab, where you select what the PLC consists of, you type it this way:

This data means that you create 3 sets of tabs where you select PLC symbol and PLC ref. symbol.

As every channel has alternative symbols, you also get 1 ref. symbol per channel.



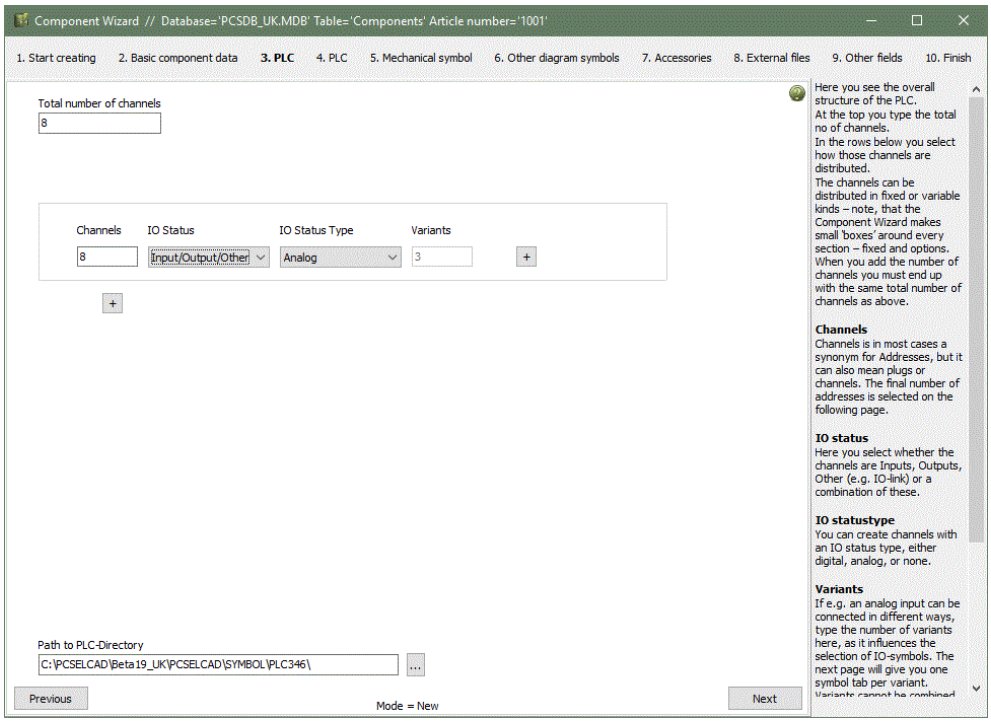
## Example 6: Cards, that contains different plugs (Sub name)

This example is based on a component with the following functions:

The component is a module, which per plug has either 2 inputs, 2 outputs or 1 IO-link. In addition to that it must fixed IO-links and 2 supply plugs.

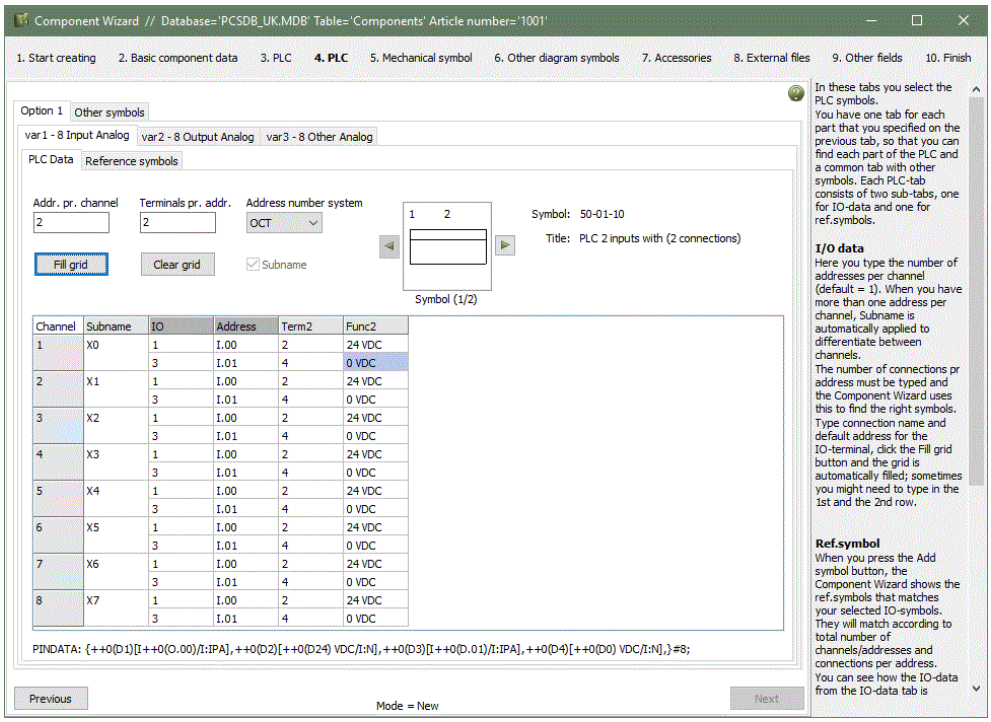
When we create it, we create 8 channels, where 1 channel relate to 1 plug.

Each channel can be input, output or other:



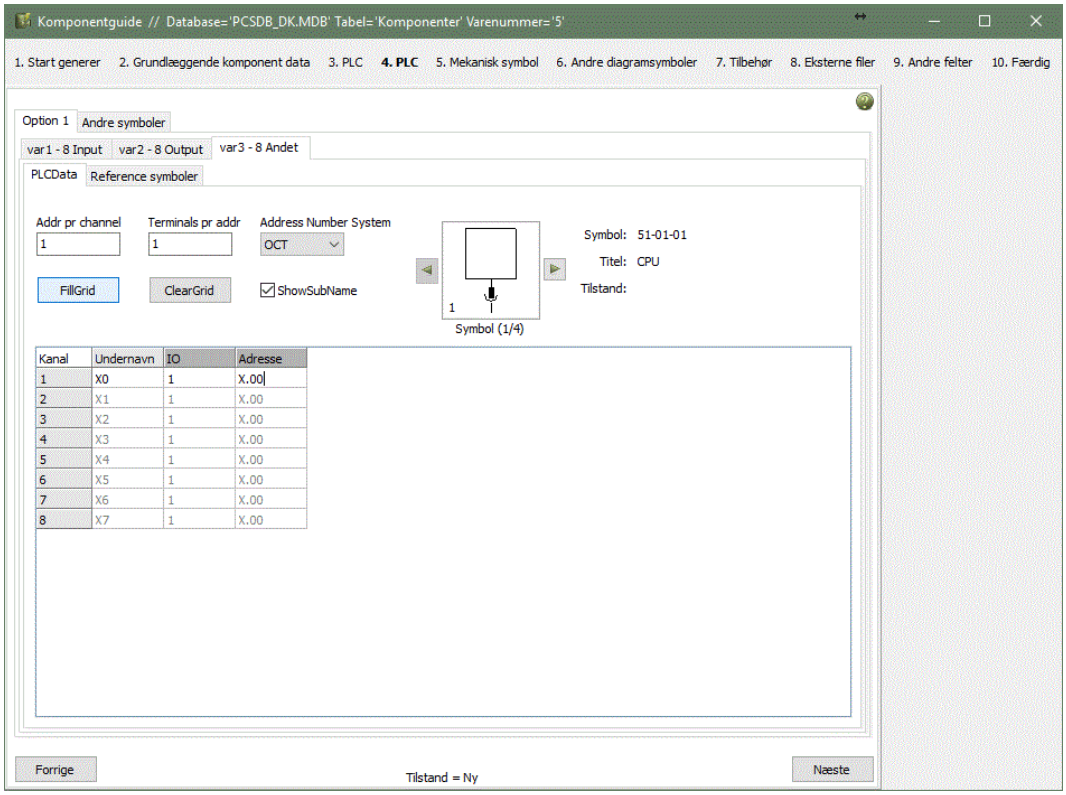

As each channel/plug contains more addresses and all channels are identical, we use sub names to differ between them:

Here, each channel has a X-sub name, and we typed the first channel and used the Fill Grid button for the rest.



The outputs are made the same way as the inputs, and below you can see, how the IO-links might look:

As every channel has alternative functions – and symbols – you will also find one ref. symbol per channel.

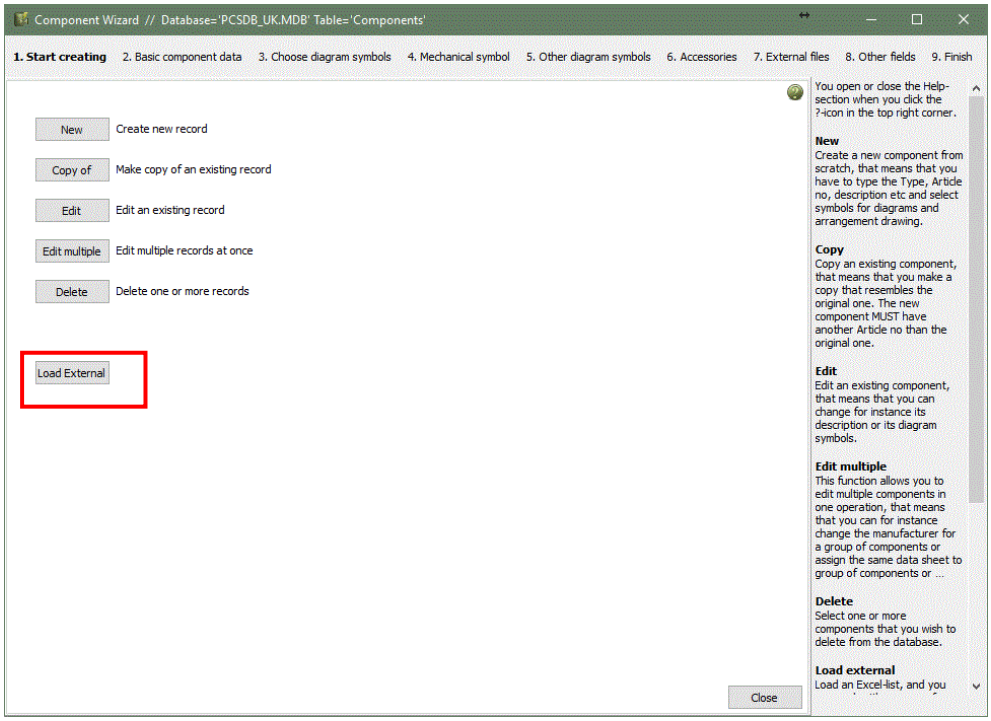


# LOAD EXTERNAL – OR CREATE MANY COMP. IN ONE OPERATION

With this function you can load an Excel list that contains many components.

It works in this way:

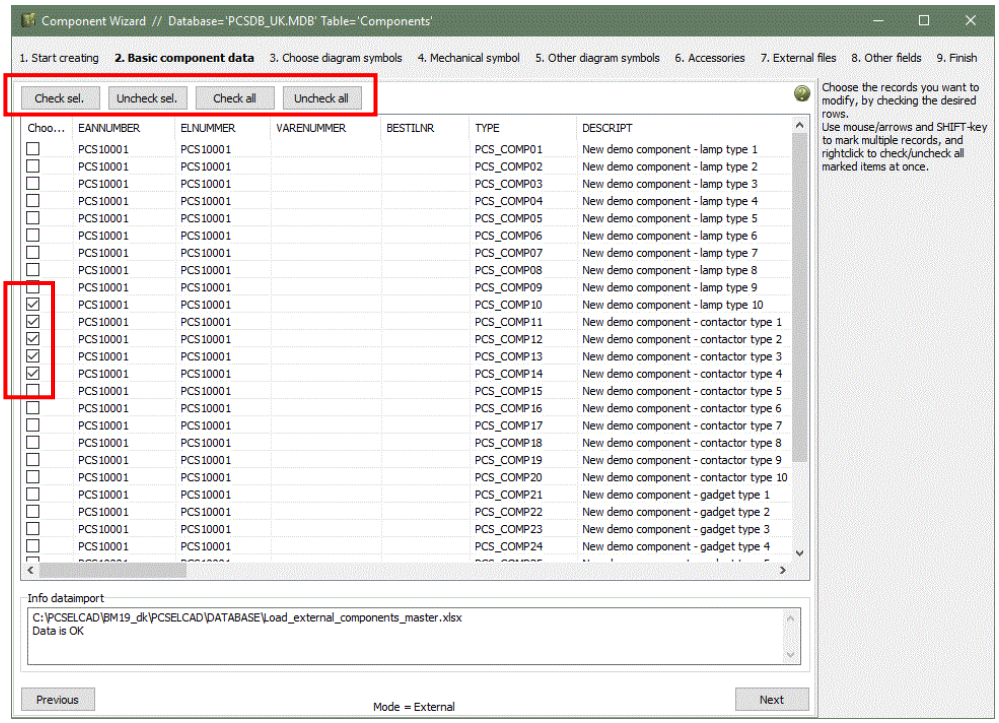
Columns in the list must have the *exact* same names as the data field names in the database. That means that you will have column named EANNUMBER, TYPE, PCSTYPE etc to use the function, because the wizard uses the column names to import the data correctly.




If you find it difficult to remember the names of the data fields, you can start by exporting the project's internal project database – this will give you an Excel file with the correct column names (including some, that you might not use ☺)

The function does NOT create the components into the database, but it makes a list which is ready to be imported into the database.

When you select the list, it is loaded into the wizard.




At the top you see the top row, which contains the names of the data fields. At the bottom you can see the file name of the list and you can see that the file is ok.

In this window you select one or more components. Then you will go through the same tabs as shown previously.

When you use the guide, you only see the data fields that are mapped. However, your Excel file may contain more data.

When you go through the wizard, you may encounter 'grayed out' data fields. This means that there is a difference between the components' contents in this data field. If you want to keep the different entries, simply press Next to continue to the next tab; if you want to change the contents, click in the grey window to select new data.

You have the same options with this function as with the 'normal' component creation functions. The big difference is that data is written into the external file and not into the database,

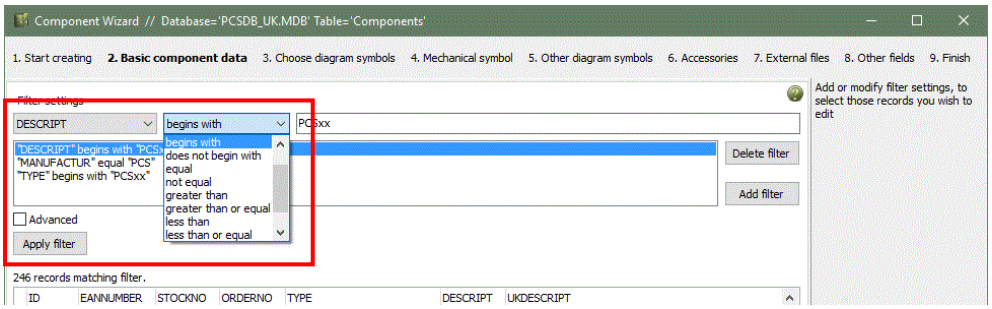
The purpose of the function is to make it easy to create a long list of components from a new supplier or manufacturer and then be able to select – simultaneously – the same symbols for (some of) the components.

# EDIT MORE COMPONENTS IN ONE OPERATION

When you have a (old) database you need to be able to maintain your data and, in that connection, you often need the possibility to edit more than one component in one operation.

When using the Multi-edit function you can select multiple components based on one or more criteria and apply the same data to all components at once, ie the same electrical diagram symbols or the same datasheet or the same status such as approved or obsolete. All together the functions that you also can do by means of the PCSCHEMATIC Database's built-in sql-function.

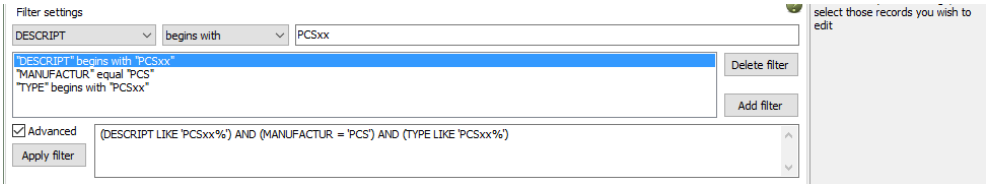
When you select the Multi-edit function, you enter this window, in which you can make your filter settings, which is how you select the components you want to edit.




You can filter by al data fields in the connected database, and you have a selection of conditions for all data fields, ie TYPE contains 'PCSxx' or MANUFACTUR contains 'PCS'.

## Advanced filter

When you select the Advanced option, you can see the query as a SQL.




The 'normal' filter is an AND-filter, that means that the components you find fulfil Filter1 AND Filter2 AND Filter3 etc.

With the Advanced setting, you can replace and AND with an OR and in this way make a very different query, which used to be difficult to make.

Note, that if you want to filter by using Boolean fields, e.g. find APPROVED components, you select the field to Contain 1 (=True).

When you have setup your filters, you go further by pressing Next. If anything is 'grayed out' in the tabs, it means that the selected components don't have identical data in the shown data field. If you want to keep the original data, simply continue (press Next), if you want to change the data, simply click in it to activate it, and select or type the new data.

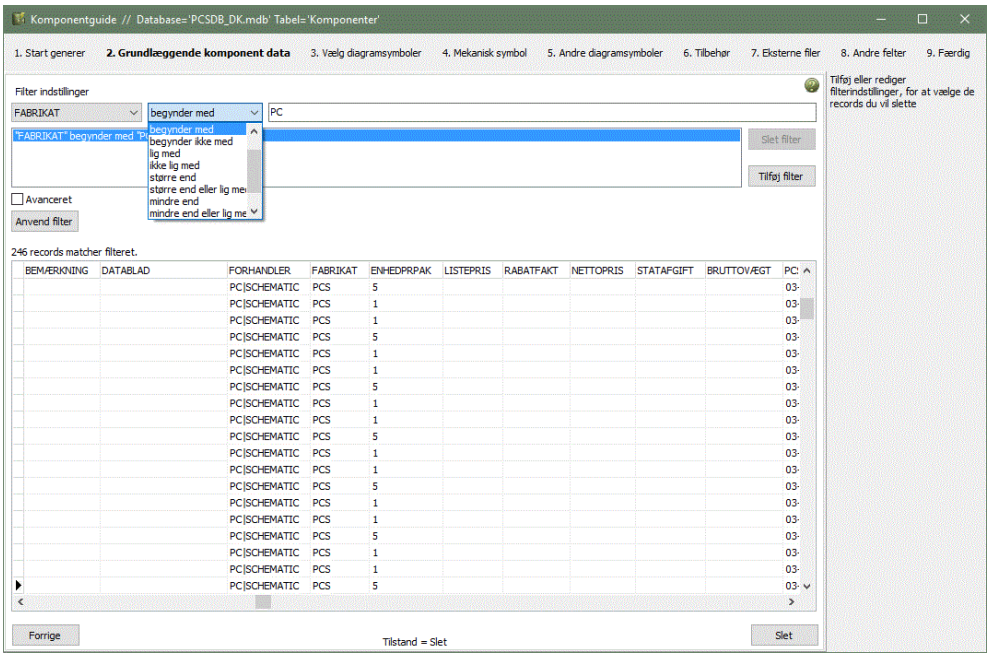
NOTE that this function operates directly in the database. And that means that you don't have any UNDO option!!!

Therefore we strongly recommend that you make a copy of your database BEFORE you use this function.

# DELETE COMPONENTS

You can also use the Component Wizard to delete selected components.

The Delete function contains the same filter functions as in Edit and Edit Multiple. This means that you start by selecting the component(s) that you want to delete, and you delete the selected one(s) – visible in the window – by clicking the Delete button at the lower right corner.



# COMPONENT DATABASE SETUP

In Settings|Database|Database setup you find this window, in which you tie – map – the two programs – Automation and Database – together.

That means that this is where you 'tell' the program in which data fields it finds various types of information.

## Make your own database

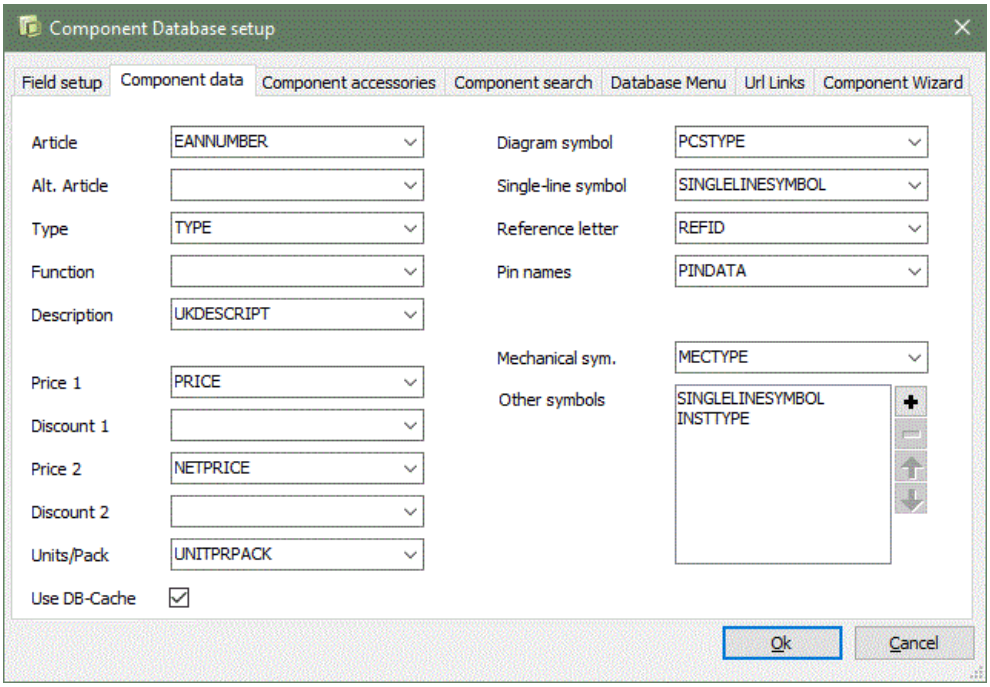
We recommend that you make your own database, which means a database with another filename.

See how to create your own database in the Database manual.

## Component data

In this example, diagram symbols are found in PCSTYPE and furthermore you can make 'connection diagrams' or mechanical layouts with MECTYPE, SINGLELINESYMBOL and INSTTYPE.

The data fields are all found in the normal database.



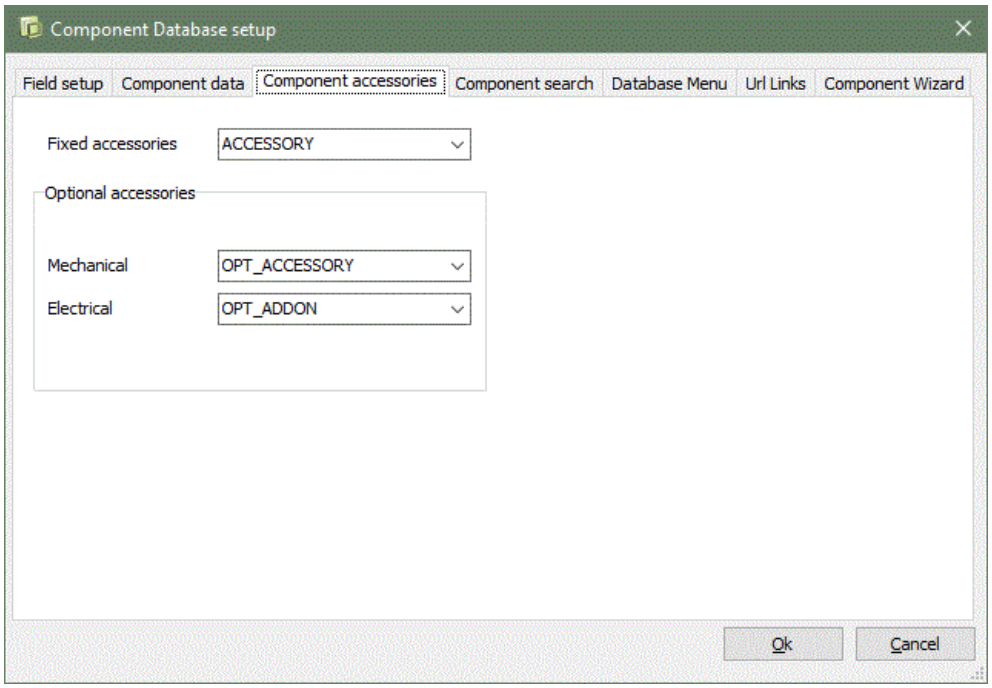

The field Other symbols is used to select data fields that contains symbols for either other diagram types than normal connection diagrams (bus connections, infrastructure, pneumatics, etc.) or other types of arrangement drawings. The fields must be created by yourself if you need these kinds of diagrams.

You can see how to create other data fields in the Database manual.

## Setup of Accessory data fields

Mapping of data fields for the various kinds of Accessories now has its own tab.

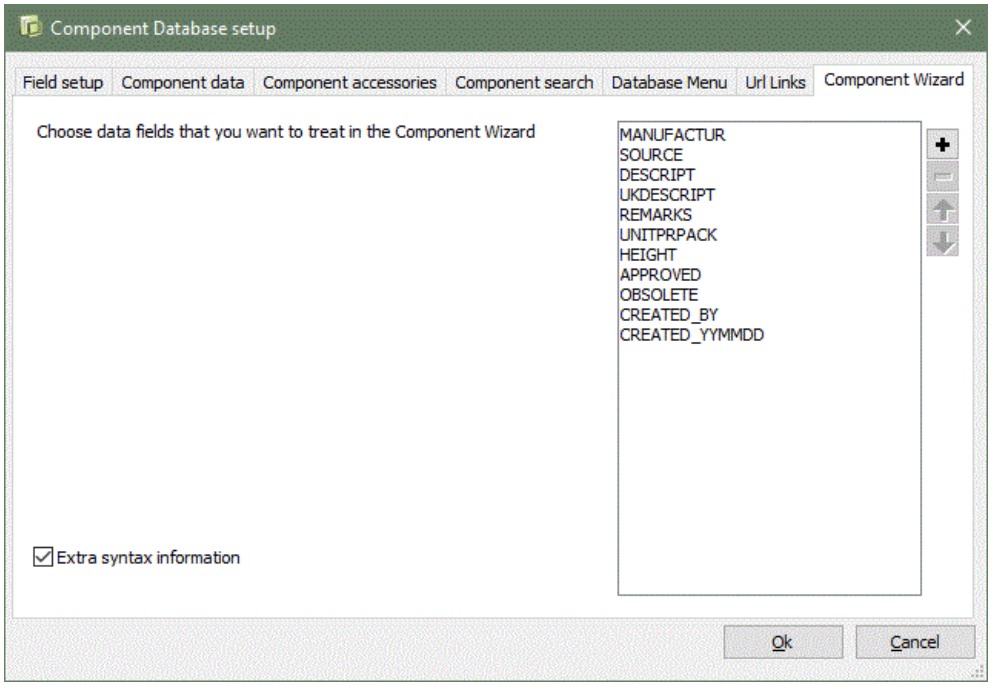
Read more about Accessories on page 19.




## Possible to edit in all selected data fields

When you work with the Component Wizard, you edit in the data fields that are mapped in Settings|Database Settings.

As a user, you might need to edit in other fields. To select those other fields, go to the Component Wizard to select them. Add by using the +, delete with – and change the order with the arrows.

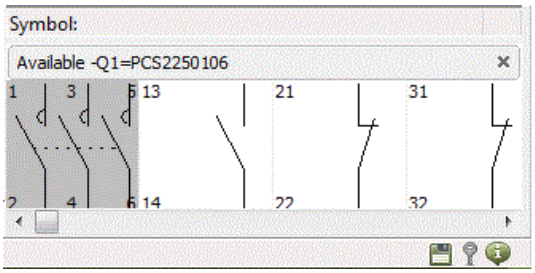



See the database manual for how to create your own data fields.

## Data fields mapped to a page

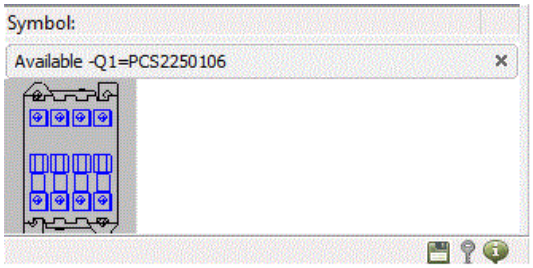
PCSTYPE is the standard selection for diagram pages (DIA).

When you fetch components in the database, the program finds the symbols in the data field PCSTYPE, and if you use the Show available window, you can see the unused/available symbols for the component in the current project.




When the program uses the default data field, you don't see the name of the data fields.

If you go to the layout page (GRP), it finds symbols in the MECTYPE data field.




This is the default function for Automation and the database.

If you have other component types, meaning that they are not represented in ordinary connection diagrams, it is possible to create extra data fields for this in the database.

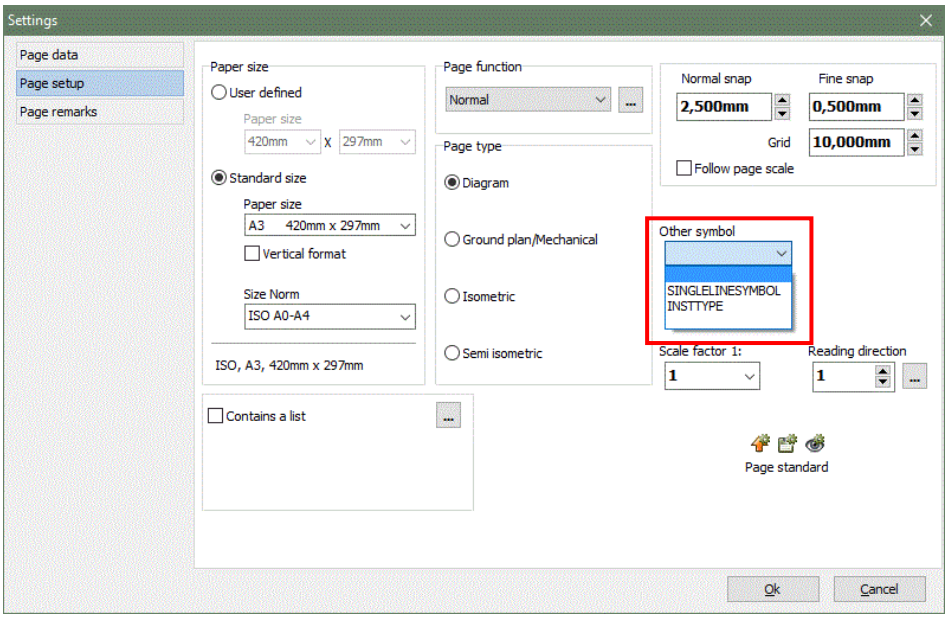
An example might be components for house installation diagrams. Symbols for installation components are not of a "connection diagram" type, so it would be more logical to have the symbols in an INSTTYPE data field.

Also, components that connect to a communication bus can have the bus symbol in the BUSTYPE data field. If the page has been setup to use this data field (see below), the Show available function will automatically show available bus connection symbols.

If you want to use one of the other data fields on a certain page, enter the Page setup, and select the data field.

The option is available on DIA and GRP pages.

You can do this at any time in a project and/or you can make a page template with the setup.



How do I use the database's symbols

You can get help from the database to retrieve different symbols for your components when you draw in Automation:

When you draw circuit diagrams, you use the Show available function to keep track of your available/unused symbols. The program retrieves the symbols from the datafield PCSTYPE, and the field contains all diagram symbols.

The mechanical symbols for arrangement drawings are in the MECTYPE, and they can also be retrieved from the Show available window.

How can I use the Other symbols settings

If you draw , eg., many PLCs, you may find it useful to be able to easily retrieve a symbol, which only shows the connections to your infrastructure/bus. Of course, the symbol can be part of the (long) list of normal diagram symbols, and then you just search for it, when you make the infrastructure overview.

But alternatively, you could make a datafield in the database which only contained this infrastructure/bus symbol. This would make it so much easier to find it.

And if you want to make a separate page with just this part of the documentation, you could link a page to this datafield as well. In that case the Show available function will show only this kind of symbol, eg the infrastructure/bus symbols.

See how to map the datafields on page 41 and 43.