

TUTORIALS – GET A GREAT START WITH PC|AUTOMATION



The book contains the tutorials, that are used in relation with Basic Training in PC|Automation.

Developed by PCSCHMATIC A/S

Last change March 2023

PREFACE

The book collects the tutorials, that are used in relation with training in PC|Automation, in order to have a single document.

The individual tutorials can also be found and downloaded from our website.

Throughout the book, you find 'green' boxes. They contain extra information and tips that can help you ahead.

More materials on www.pcschematic.com

In <https://www.pcschematic.com/en/academy/materials/> you can find all manuals and tutorials to the program.

Videos in YouTube

In our YouTube channel <https://www.youtube.com/user/PCSCHEMATIC> you can see a lot of videos about how you can work with PC|Automation; one of the playlists takes you through the creation of a motor control project, another takes you through PLC functions etc.

Demo files

In PC|Automation we have included a few demo files, that show some examples of how to make projects. The files also have some explanations in them, that elaborates on some topics.

Tutorials in the PC|Automation series

The tutorials marked with * are included in this book. Most of them are in Danish and English

- Quick start *
- Motor control project *
- PLC project *
- Component wizard *
- Labelling and Marking (engelsk) *
- House installation
- Panelbuilder
- Automation Service
- The Mounting Assistant *
- Panelrouter

Contents

Preface	3
The workarea in Automation.....	11
About projects and pages	13
What does Project oriented mean?	13
What does a project consist of?.....	13
To insert new pages	13
How to select pages and chapters	13
The Page menu	14
Drawing objects.....	15
What is a drawing project?	15
Automatic command change.....	15
How to work with drawing objects	15
How to select multiple objects	16
To move, copy, and delete objects	16
To select different object types at the same time	16
Lines	17
Texts.....	19
How to place a text	19
How to correct a text	19
There are different types of text	19
Symbols	20
Briefly about symbols.....	20
Briefly about the Pick menu	20
To delete a symbol	21
Components	22
What is the relationship between symbols and components?	22
Component Database.....	22
Component menu.....	23
How to easily select a free symbol in a component	23
Component Symbols window	23
Jump between symbols for the same component	23
How to jump to another symbol for the component.....	24
Automatic jump to the reference from the reference cross/contact mirror.....	24
Component Portal.....	24
Circles and circular arcs	25
Router	25
See shortcuts in the Router menu	25
Shortcuts.....	26
Custom shortcuts	27
Right mouse key	27
Start a new project.....	31
The power circuit diagram	32
Place a Motor	32
Place directly from the database.....	34
A few tips to save time	36

Connect the motor to the phases	37
Insert motor protection	39
Insert Contactor.....	40
Circuit diagram.....	41
Component menu - show the available symbols.....	41
Relay, coil and control circuit switches	42
Accessories (not used here).....	43
Insert terminal row with item data from the pick menu	44
Insert cable	45
Link between the pages	46
Arrangement drawing.....	47
Page scale.....	48
Measurement.....	48
Lists in the project	49
Object Lister can fix page titles... and much more.....	49
The final finish.....	50
Insert new pages	50
Delete pages you don't need	50
New page numbers.....	50
Language settings.....	51
A little more info about terminals and cables	52
Direction of the terminals.....	52
Show terminal row	52
Component grouping.....	53
Direction of cables	55
Quick placement of a cable.....	55
Cables on the control current diagram.....	55
Conductors with screen	56
Marking and Wire Numbering.....	57
Wire Numbers function	58
Printing wiring numbers.....	59
Start a new project	63
Place a PLC from the database	64
Place the PLC inputs	66
Readdresses the PLC-card	66
Read the IO list	67
If the PLC has multiple options.....	69
If you use a PLC without component data.....	70
Get data from the reference symbol.....	71
Select PLC	72
PLC projects with Subdrawings	73
Example 1	73
How to do – method 1	74
How to do – method 2	74
Other filters.....	75
Readdressing reference symbols	76
A little more info about the PLC symbols	77
Create PLC component in the database	80

What can your PLC do	81
Open the component wizard	87
Create a new component – Normal.....	88
Choose diagram symbols	89
Get symbols using database lookups	89
Get symbols by going to the Symbol Menu	91
Symbol settings	92
Alternative symbol selection	93
Mechanical symbol	95
Other diagram symbols	96
Accessories for components.....	97
Accessories.....	97
Optional accessories	97
External files	98
Other fields	98
Finish	99
First tab – again	99
First tab – again again.....	99
Create Jumper link.....	100
Create wire	101
Create wiretray	102
Draw with jumpers, wires and wire trays	102
Create cable	103
Download file	103
Last conductor is GNYE	103
Cable accessories	103
Create PLC	104
Example 1: Cards with 8 digital inputs and separate supply	104
Definitions	104
Start creation	105
Enter PLC data and select PLC symbol	105
Select PLC reference symbol	106
Other symbols – Top and bottom etc.....	107
Example 2: Compact PLC with 8 inputs and 8 outputs	108
Select PLC symbols and PLC-ref. symbols	108
Example 3: A card that can be either 8 outputs or 8 inputs.....	109
Place the component in the project.....	109
Example 4: A card that can have different status types per address	110
Place the component in the project.....	110
Example 5: Compact card with communication	111
Place the component in the project.....	111
Example 6: Card that refer to different connectors (sub-name)	112
Place the component in the project.....	113
Copy of an existing component	114
Edit an existing component.....	115
Edit multiple components at once	116
Advanced filter	116

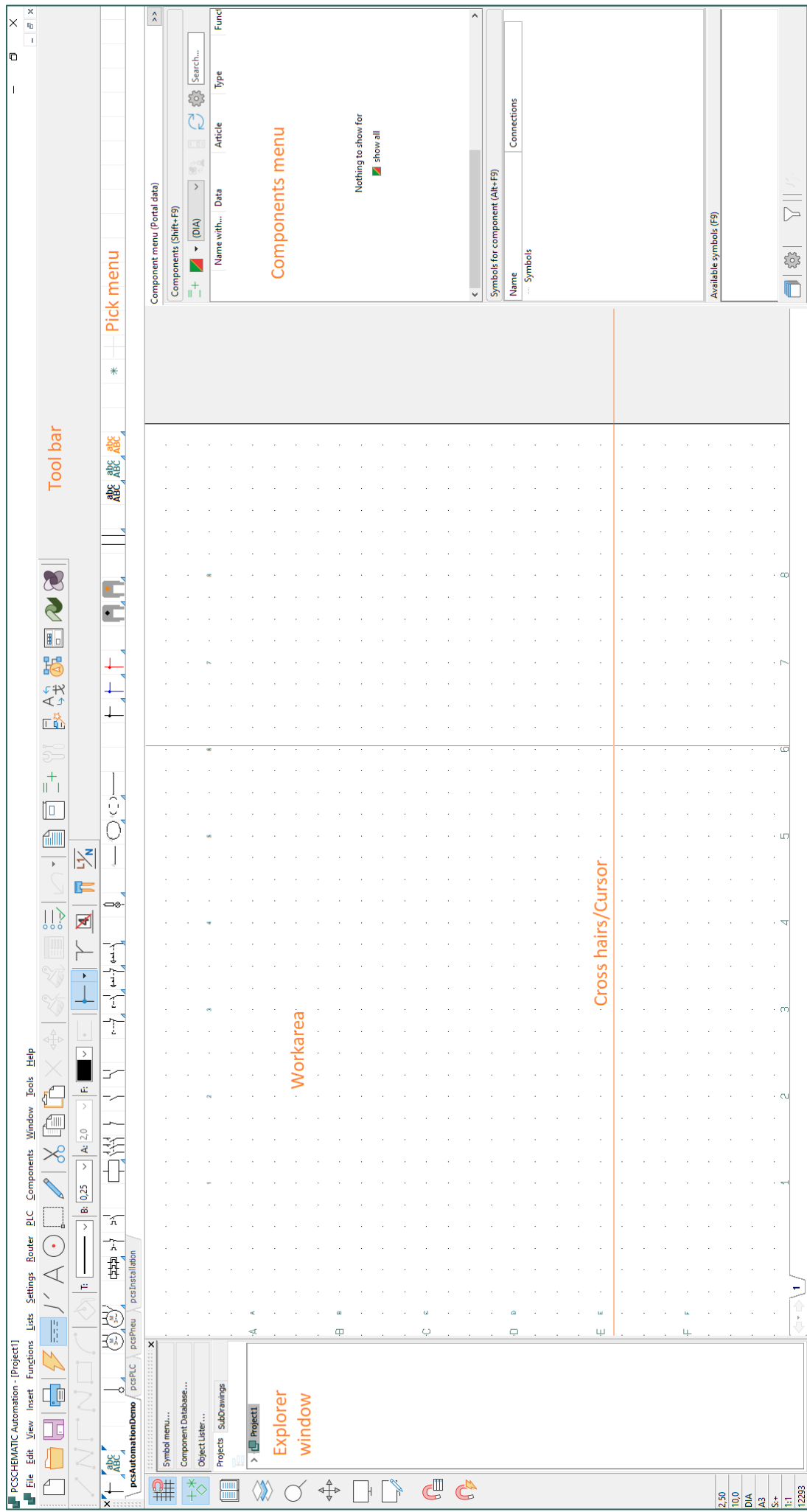
Continue editing multiple	117
Delete components	118
Setting up component wizard and database.....	119
Article numbers	119
URL-links	120
Other fields.....	120
Purpose of the Mounting Assistant	124
Get started.....	125
If the order number is not listed	125
The Mounting Assistant opens.....	126
General information about the Mounting Assistant and functions	126
Display window	126
The mounting assist window – list	127
Other settings	128
The Arrangement Tab	129
Why this tab	129
Diagram or Arrangement	129
Workflow on the Arrangement tab.....	130
What do you see in the list	130
Display window	131
Make a note.....	132
Connections tab	133
Why this tab	133
Connections tab workflow.....	134
What do you see in the list	135
Display window	136
Component tab	137
Why this tab	137
Diagram or Component tab.....	137
Workflow on the Component tab	138
Display window	139
What do you see in the list	139
When there are changes in the project.....	140
If there are changes in the project.....	140
Making changes with the Mounting Assistant open	141
If the installer has made notes	141
How is data stored from Mounting Assistant	141
When the project changes its name	141

QUICK START

This chapter gives you a quick introduction to how to work with the program Automation from PCSCHEMATIC. Among other issues, you can read about:

- What does projects consist of
- How do you insert new pages
- How do you draw diagrams
- Useful shortcuts

... and a lot more.



THE WORKAREAⁱ IN AUTOMATION

When you start Automation, you will be given the option to choose one of the included templates. Selecting a blank project opens a blank project with one page, with no drawing header or anything else. Here you can start all over again. If you select pcsStart, a project opens that contains a series of pages that are normally part of a project.

Demo projects can be found under File => Open and select the Demo folder. Here are examples of what projects can look like, so you can quickly get started drawing yourself.

Menu Bar

The menu bar at the top consists of the menus, that contain all the features of the program.

Program Tool Bar

Here you can choose the common file and print options as well as the general drawing and editing tools.

Command Tool Bar

Appearance is different depending on which features are active.

Pick Menu

Here you can place the symbols and components you use the most, so you can download them directly into your drawings. By clicking on the tabs at the bottom of the pick menu, you can switch between the created pick menus.

File Explorer Window

Provides direct access to the Symbol Menu, Component Database, and Object Lists. In addition, there are two tabs: Projects and Sub-drawings.

Projects tab in File Explorer Window

Here you will find information about all open projects. You get an overview of all the pages in the projects, as well as all components on the relevant pages. When you click on an icon for a project page, the page is displayed on the screen.

Sub-drawing tab in File Explorer Window

This is used to handle e.g. placement, editing and display of partial drawings. Here you can drag entire pages or sub-drawings into the project.

The vertical Tool Bar

Displays shortcuts to various page and zoom functions and at the very bottom information about the page's settings, status, license and version number.

Drawing area

Corresponds to the size of paper you have chosen to draw on. The paper size can be selected in Settings => Page Settings.

Page History buttons

By clicking on the Page History buttons, you can jump to the most recently used pages within the active project.



Status Bar

This includes coordinates, layer title and help text, as well as various messages. When you place the mouse pointer over, e.g., a button on the screen or a symbol in the pick menu, a drop-down text is displayed here.

Current path numbers

Current path numbers are displayed both where you specify, they should be placed on the page, and at the bottom of the screen. The current path numbers at the bottom of the screen are also displayed when you zoom in on that part of the page.

Page tabs

Here you can click to switch between the pages. The chapter pages can open/close the chapter.

Components window

In the window you can see all the components used in the project. On the red/yellow/green button, you can choose which ones.

Free symbols for selected component in the Components window

After selecting a component in the Components window, you can see the used and available symbols for the component, in a list at the top and as a symbol at the bottom.

Chapter tabs

When you click on a tab, you jump to the first page of the selected chapter.

ABOUT PROJECTS AND PAGES

What does Project oriented mean?

Automation is a project-oriented program. That means all the information you need for a project is in one and the same file.

You do not need to change programs to create e.g. part or component lists, as these are part of the same project file as your drawings.

What does a project consist of?

A project can typically consist of a cover page, tables of contents, chapter covers, pages with diagrams, and pages with different kinds of lists. In addition to this, a project can contain pages with mechanical symbols for the components used.

All these parts are located on the appropriate pages in the same project.

Example of a project:

You can find some project samples in the C:\PCAutomation\Projects\Demo folder. In some of the projects you will find explanations of the different page types etc.

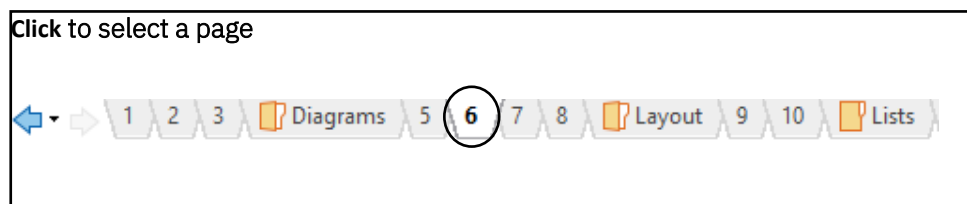
Diagrams are the starting point

The starting point for your project is the diagrams. They are made by placing symbols, texts, and lines on one or more pages of the project. Here you also associate item data with the different symbols in the forms. This item data can be retrieved from the program's component database or written manually and can then be automatically transferred to the project's lists.

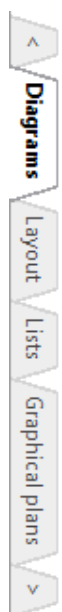
To insert new pages

How to select pages and chapters

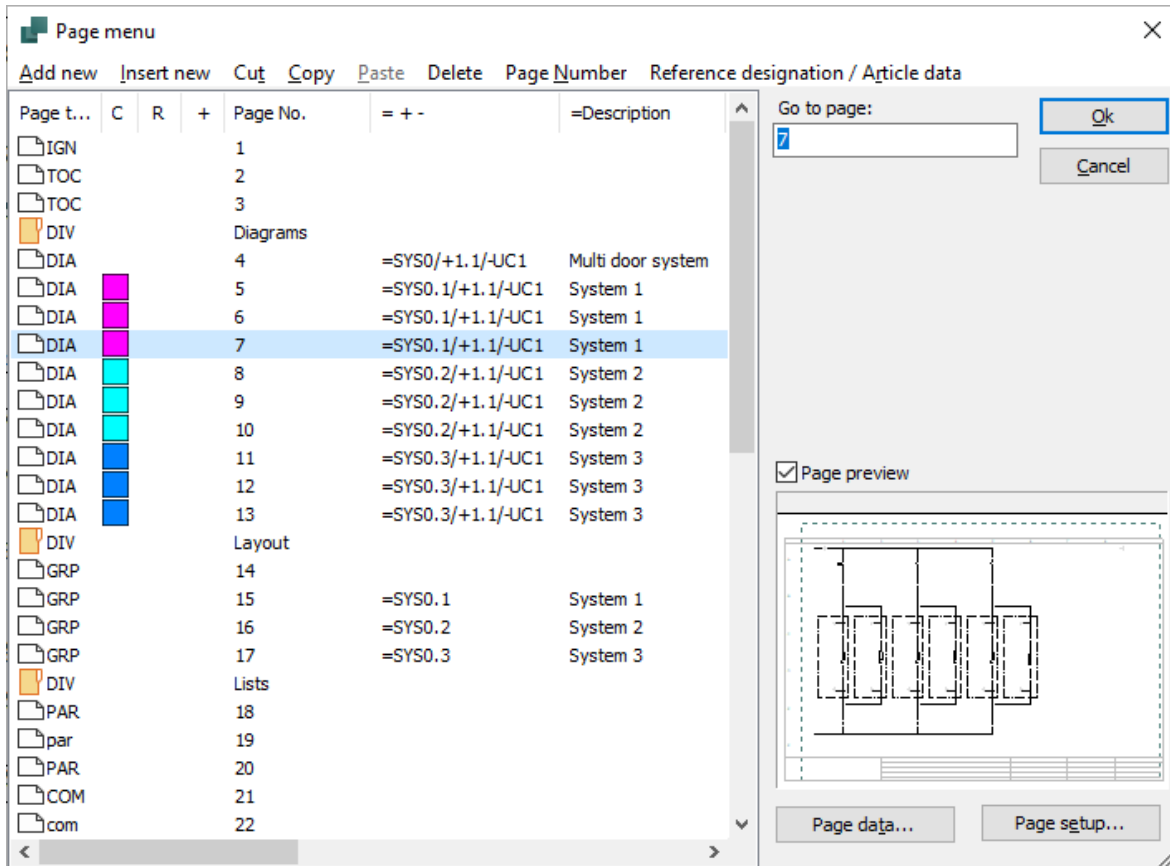
Open one of our Demo Projects to try out what you're reading. You can choose from the project pages by clicking on the pages' page tabs at the bottom of the screen:



Clicking on the chapter tabs on the right side of the screen will display the first page of the selected chapter. You can also use the [PageDown] and [PageUp] keys to flip back and forth between the pages of the project.



The Page menu



Here you can insert new pages, delete or move pages, copy pages, and change the setup of the pages you created. Or you can use it to choose which page you want to work with. The page menu is a great place to get an overview of a large project because all page titles are listed here.

It is possible to mark selected pages and copy them to another project, change the page number of all pages or delete selected pages. On the Page Data button, you can fill in the various data for your project and on the Page Settings button you can, e.g. determine the page type and paper size.

In front of each page, an icon indicates whether the page has vertical or horizontal page format. Next, you'll see the page's page function. A folder in front of a page indicates the start of a new chapter.

If colors are selected for page tabs, these are displayed in the **C** column to the right of the page type. By right-clicking a page in the page menu and then choosing Color, you can also change the colors of the page tab for the page. If there are selected states on pages in the project, the name of the selected state appears in parentheses after the page title.

Pages marked with **+** have been changed since you last saved the project. Pages marked with **!** have been changed since the project was opened.

Pages marked with **R** have been modified since the start of the revision check.

DRAWING OBJECTS








What is a drawing project?

Automation has four different types of drawing objects: **Symbols**, **Texts**, **Lines**, and **Circles**. Everything you put onto a drawing will always be one of these four types of drawing objects.

There is also an **Area** command that makes it possible to work with all the object types at the same time — e.g. when copying, moving and rotating.

How to change object type

To work with a specific type of drawing object, start by clicking one of these buttons (the keyboard shortcut appears in parentheses):

Symbols [S]	Texts [T]	Lines [L]	Circles [C]	Area ⁱⁱ [O]	Draw [INS]	Auto command [Shift+Z]
						

When you click one of these buttons, the program changes state:

- The Program tool bar changes
- The options in the Program menu changes
- You are only allowed to work with objects of the selected type
- Instead of the icons, you can also de-/activate the Pencil by pressing the shortcut twice

Automatic command change

If you want **Automation** to switch object type for you automatically, click the **Auto Command Change** button.



When the feature is turned on, Automation automatically switches to the object type of the selected object. The feature is disabled by clicking the **Automatic command change** button again.

How to work with drawing objects

There are two different states to work in. The state in which you work is determined by the **Draw** button — also known as the **Pencil**. You can either:

- Draw/Place new objects (Pencil enabled)
- Edit already placed objects (Pencil is NOT enabled) — for example, when you move, copy, delete, or edit objects. The objects are selected by clicking them, or by clicking and dragging the mouse over the objects you want.



How to select multiple objects

Start by opening a Demo Project and simultaneously try out what you're reading. When you need to work with already, placed objects, you must select them first:

1. Click the icon that corresponds to the object type the object has (Texts)
2. Make sure the Pencil is not active - press <Esc>
3. Click in a corner of the area you have selected
4. Hold down the mouse button and drag the mouse across the screen until a rectangle is drawn around the objects you have selected.
5. Release the mouse button and all the objects in the area are selected.



If you want to deselect some of the selected objects, or select objects outside the area, hold down <Ctrl> and click them.

To move, copy, and delete objects

After selecting the objects, click the **Copy**, **Move** or **Delete** button. After placing the copied objects, you will be asked how the symbols must be renamed.

Press <Esc> to remove the objects from the Crosshair.

You can undo your actions by clicking the Undo button.



To select different object types at the same time

If you need to select different types of objects at the same time, first click the **Areas** button, and then do as described above.



How much you choose when selecting the area depends on whether you drag the mouse from left to right or from right to left when the area is selected.

- When you go from right to left, you only get what is completely inside your marking
- When you go from left to right, you get everything you touch.

LINES

Conductive and non-conductive lines

There are two different kinds of lines in **Automation**:

Conductive (electric) lines

Non conductive lines / free lines

Is the **Conducting lines** button active (see image below), the line is perceived as an electric/conductive connection.

If it is not active, the line is not perceived as an electrical connection.



How to draw lines

1. Click the **Lines** button or press the shortcut key [L].



2. The Tool bar for lines appears:

3. Need to draw lines: Click on the **Pencil** so that it is active.



4. Select the line type - e.g. **Straight lines**.

5. Click where you want the line to begin and click once every time the line should change direction.



When you're done drawing lines, you can press <Esc> to disable the **Pencil**.

Signal names and conductive (electric) lines

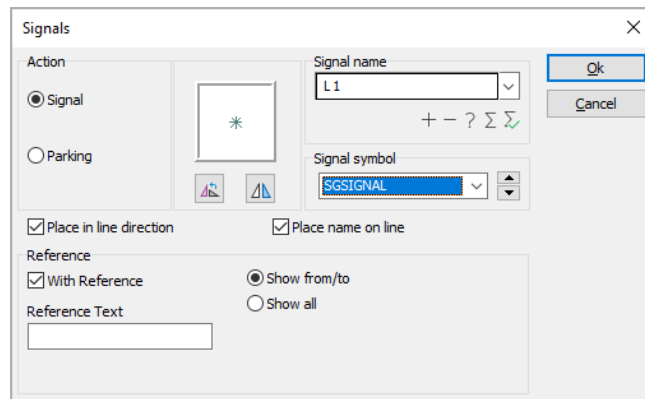
The first time you click the page to draw **Conducting Lines**, you will enter the **Signals** dialog box.



This is because conductive (electric) lines can only start and end at an electrical point. An electrical point can be another conductive line, a connection point on a symbol or a signal symbol.

Therefore, when you start or end an electric line without electrical connection, choose a signal name, or choose for the line to be parked (without signal name).

Once you have done that, you will be allowed to draw the line.



What are signal names used for?

The signal name of a signal symbol is used to indicate an electrical connection to another signal symbol with the same signal name. That means, all electrical points with the same signal names are the same electrical potential.

What does it mean to Park a line?

If you want to wait to decide what to connect a lead line to, choose **Parking**. The line gets no electrical connection. However, this is only a temporary solution, as there should never be a parking in a finished project.

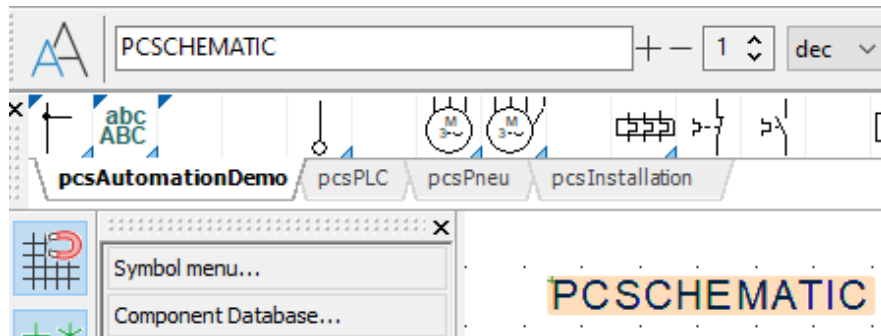
TEXTS

How to place a text

1. Click the **Texts** button or press the shortcut key [T].



2. The text tool bar appears:

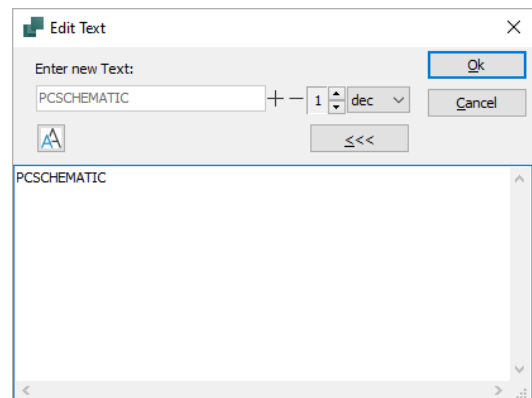


3. Enter the text as shown above and type <Enter>.
4. You now have the text in the crosshairs. Click where you want to place it.

How to correct a text

If you want to correct a text in a project, do the following:

1. Click on the **Text** button, and turn the **Pencil** off by pressing <Esc>
2. Double click the text
3. Change the text and tap <Enter> - or click **OK**
4. To enter the large window, press >>>



There are different types of text

There are several different types of texts in Automation:

Text type	Description
Free texts	Texts that can be freely placed anywhere in the project.
Symbol texts	Texts associated with each symbol and containing information about the component the symbol represents.
Connection texts	Texts associated with each of a symbols connection points.
Data fields	Text fields that are filled automatically.

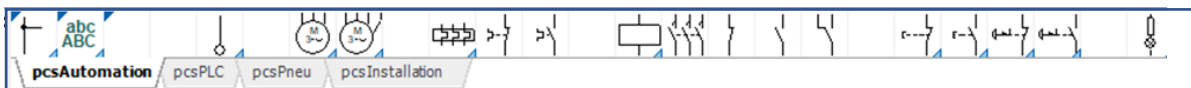
SYMBOLS

Briefly about symbols

In **Automation**, you use symbols when you need to show a specific electrical function in a drawing. For example, if you want to place a lamp, you will find the symbol for a lamp and then insert it in the drawing. Components with multiple electrical functions may have multiple symbols associated with them.

Briefly about the Pick menu

When working in Automation, a **Pick menu** appears on the screen. Here you can place the symbols and other drawing objects that you use the most.



How to place a symbol from the Pick menu

1. Click on the desired symbol in the pick menu. The program automatically switches to working with **Symbols**, and the **Pencil** is activated.
2. You will now get the symbol in the crosshair: Click where you want to place the symbol.
3. You will now enter the **Component Data** dialog box:



4. Enter component name, as well as Type, Article number and/or function description, and click **OK**.
5. The symbol is now placed in the diagram, and has been assigned article data.

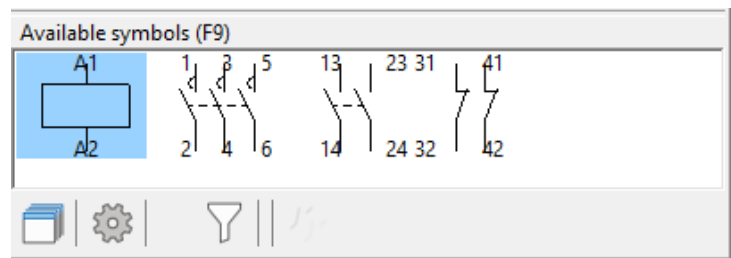
Press <Esc> to remove the symbol from the crosshair.

Symbols with article data in the Pick menu

The symbols in the pick menu can be mapped to existing components in the database.

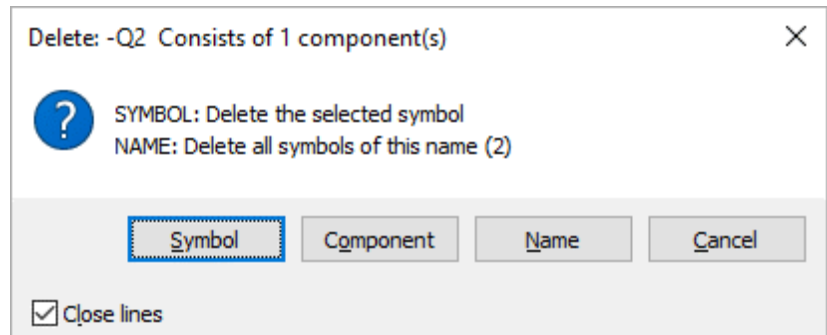
1. When you place such a symbol in a project page, it automatically has the component's article data attached to it on the diagram

2. If the component in the database contains more functions, the symbols are automatically seen in the Available Symbols window at the bottom of the Component Menu.



To delete a symbol

1. Click the Symbol and press Delete

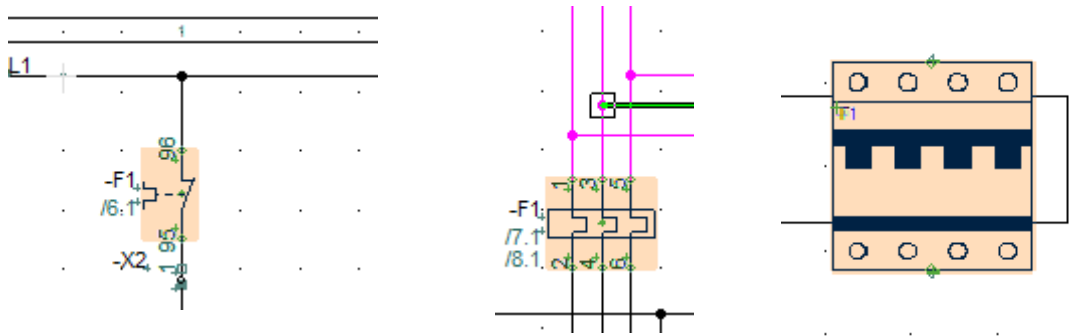


2. You now have the following options:
 Click **Symbol**: Only the selected symbol is deleted.
 Click **Component**: All symbols for the component are deleted.
 Click **Name**: All symbols with this name are deleted.
 Click **Cancel**: Nothing is deleted
 If you check **Close lines**, the lines close for the symbols that are deleted. If you don't check **Close lines**, the lines are parked.

COMPONENTS

What is the relationship between symbols and components?

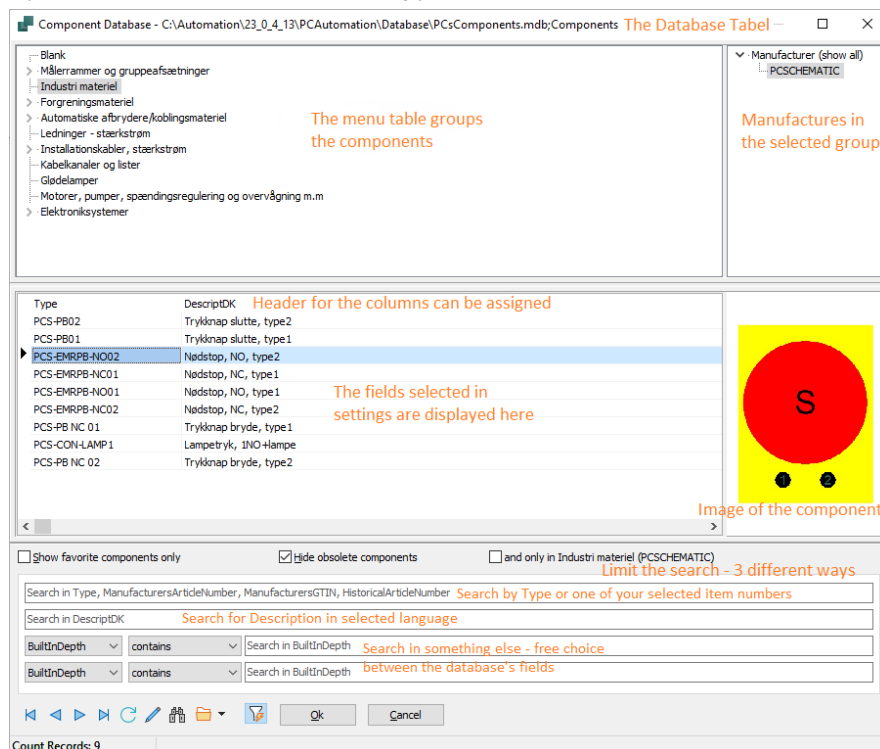
1. A symbol shows an electrical function in a diagram.
2. A **Component** is one or more symbols that have a specified type and Article number. When a component has multiple electrical functions, the component has one symbol for each function. To indicate that two symbols belong to the same component, the symbols must have the same **Component name**.
3. You pick **Components** from the Component Database and it has the shortcut D.



Symbols for -F1, located on different project pages (Two electrical symbols and one mechanical)

Component Database

You can pick components from the database. The window looks like this when opened. You can search for Type and other data for the individual components.



You can create your own components in your Database – see the Component Guide booklet.

Component menu

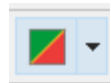
How to easily select a free symbol in a component

When you select a component from the database, you automatically get its electrical symbols in the Available symbols window and can freely place these in the project diagrams.

Automation keeps track of which symbols belong to which component and which types of diagrams the symbols belong to.

Once you have used components from the database, you can easily get an overview of all components that still have unused symbols:

- Component menu shows all components in the project
 - You can choose to show only a sample of these, e.g. based on whether they have available symbols or not, by using the color codes.
- In the Component menu it is possible to see the used and available symbols of the components.
- The component menu consists of 3 windows: Components, Symbols for Component and any possible accessories and Available symbols.



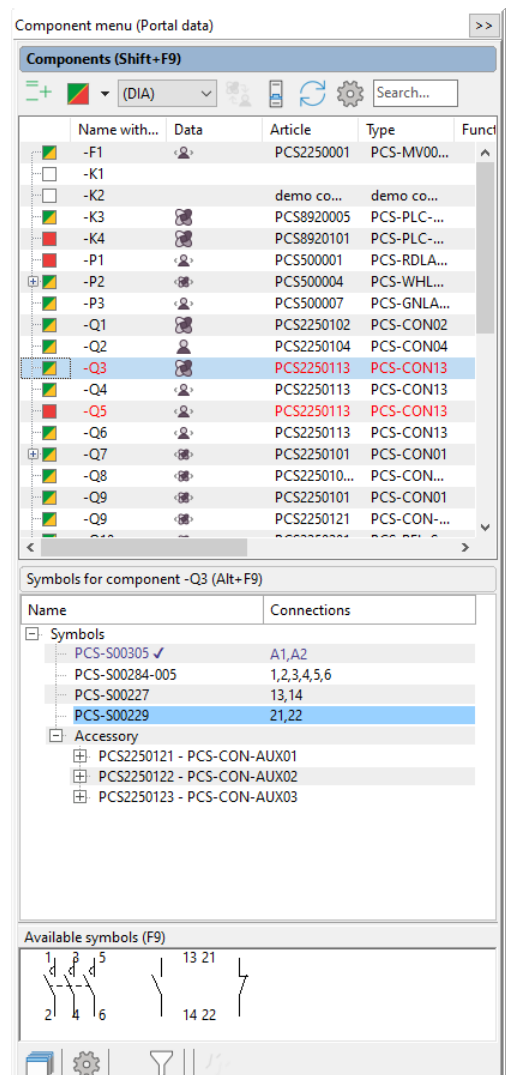
Component Symbols window

When you click a component in the list in the Components window, you get access to all its symbols.

If you are on a Diagram page, the electrical symbols of the components are displayed. If you are on a Mechanical page, the mechanical symbols of the components are displayed. Possible accessories are also displayed in the window.

Jump between symbols for the same component

When you place multiple symbols for the same component, those symbols are automatically created references on the project pages. Therefore, next to the individual symbols, you can see references to other symbols for the component.



How to jump to another symbol for the component

To automatically jump over to the symbol being referenced, do the following:

- Double-click the reference(s).
- The program now automatically jumps over to the page of the project on which the referenced symbol exists. The symbol is designated by the crosshairs.

Automatic jump to the reference from the reference cross/contact mirror

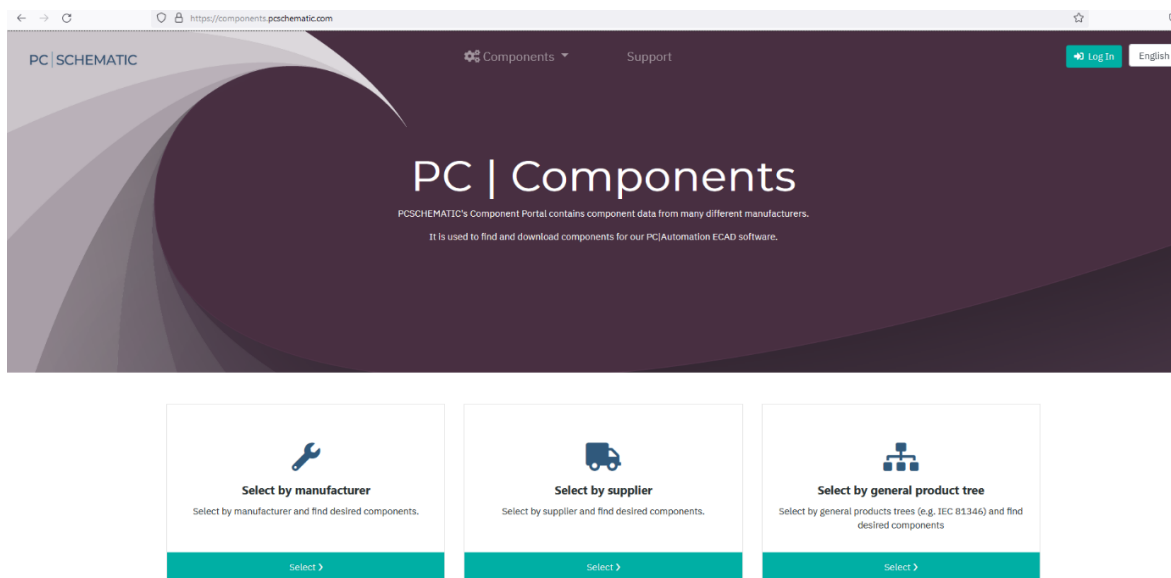
- Similarly, you can double-click on the reference in the reference cross or contact mirror.
- If you double-click the symbol's reference in the reference cross, you will return to the symbol.

COMPONENT PORTAL

If the component is not in the Component Database, you can look for it in the Component Portal. Clicking the icon at the top right of the tool bar opens the Portal.



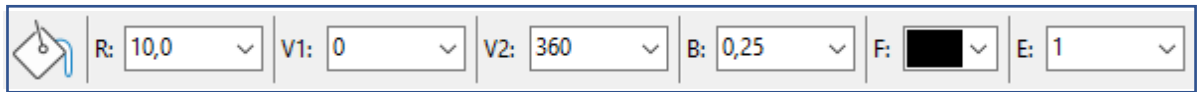
If you choose via manufacturer or via supplier, you get the opportunity to find the component(s) you need:



Drag a single component directly into your project — it's imported into your Database and ready to use right away. Or select multiple components and add them to the 'basket', which can be downloaded and imported into your Database.

CIRCLES AND CIRCULAR ARCS

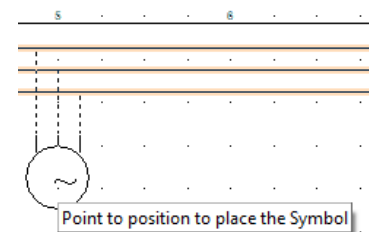
To draw circles/circle arcs, select **Circles** and activate **the Pencil**. Circles have the shortcut key [C]. In the circle tool bar, you have these options:



You can make whole circles, circle slices from selected degrees and ellipses using E.

ROUTER

When placing symbols in a project, **Automation** can automatically draw the connections of the symbol. To do so, enable the program's **Router** - shortcut [R]. To turn off **the Router**, press the shortcut again.

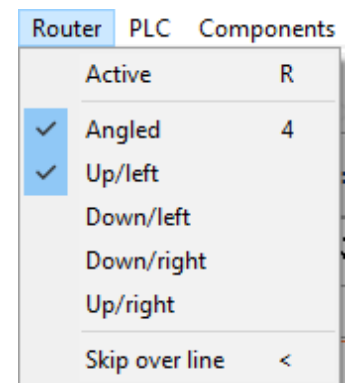


See shortcuts in the Router menu

In the **Router** menu you can see what options you have for the Router and shortcut keys.

It's only when you use the shortcuts together with the Router that it becomes a really powerful tool.

The Router is used when working with Lines and with Symbols.



SHORTCUTS

A lot of **Automation's** commands have shortcuts. Here you see the most common:

BASIC DRAWING FUNCTIONS			
Symbols	S	Texts	T
Lines	L	Areas	A
Draw /Pen	Insert	Circles	C
Potential	P	Auto-command change	Shift+Z
Rotate object	Space	Change mount.correct bend	Space
Fine snap	Shift-button down	Change Text alignment	Shift+Space
Edit selected text	K		
ROUTER			
Active	R	Route	1
Nearest direction	2	Reverse order	3
Angled line	4	Skip line	<
COMPONENTS			
Component Database	D	Component Wizard (own components)	W
Component grouping	Ctrl+F7	Select component	Shift+C
SYMBOLS			
Symbol menu	F8	Replace symbol	F4
Next/Previous s.text	F5/F6		
PAGES AND NAVIGATION			
New project	Ctrl+N	Open Project	Ctrl+O
Page menu	Shift+P	Previously selected page	F10
Zoom	Z or Ctrl+mouse	Scroll	Ctrl+arrow keys or mouse
Zoom full page	Home	Previous zoom	Tab
TOOLS			
Netnavigator	N	Object lister	F7
Mounting Assistant	Shift+M	Panelbuilder	Alt+P
Create own shortcuts	Shift+S	Panelrouter	Shift+R

Custom shortcuts

In **Automation**, you can freely define your own shortcuts. This is done by going into **Settings|Shortcuts** - or by using the shortcut <Shift+S> - where you easily can create the shortcuts that suit your everyday life. Shortcuts can also be just a single key, such as <S> for **Symbol**.

Right mouse key

To get additional options when working with the program, you can right-click with the mouse on an object. It can be one on a symbol, on the pick menu, on a whole page or something else. Right-clicking will in many cases produce a menu that contains functions and/or data about the object.

If you are in doubt about something, you can always try to see if you get something out of right-clicking.

MY FIRST PROJECT - MOTOR CONTROL

This chapter gives you a quick introduction to how to work with PC|Automation. Among other things, you can read about:

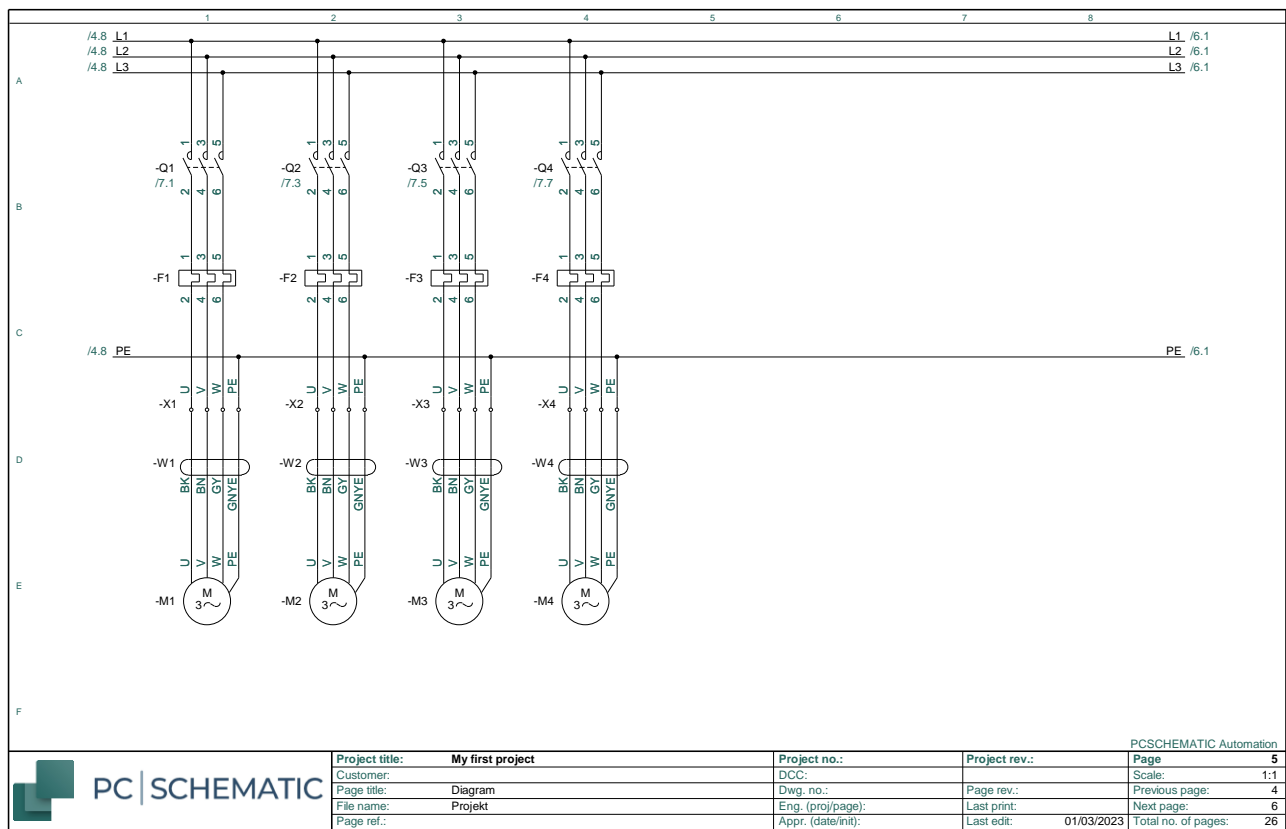
- Use the most common drawing functions
- Use the database to find components
- Use the router to make automatic lines
- Place terminal and cable symbols
- Create a panel layout

... and much more.

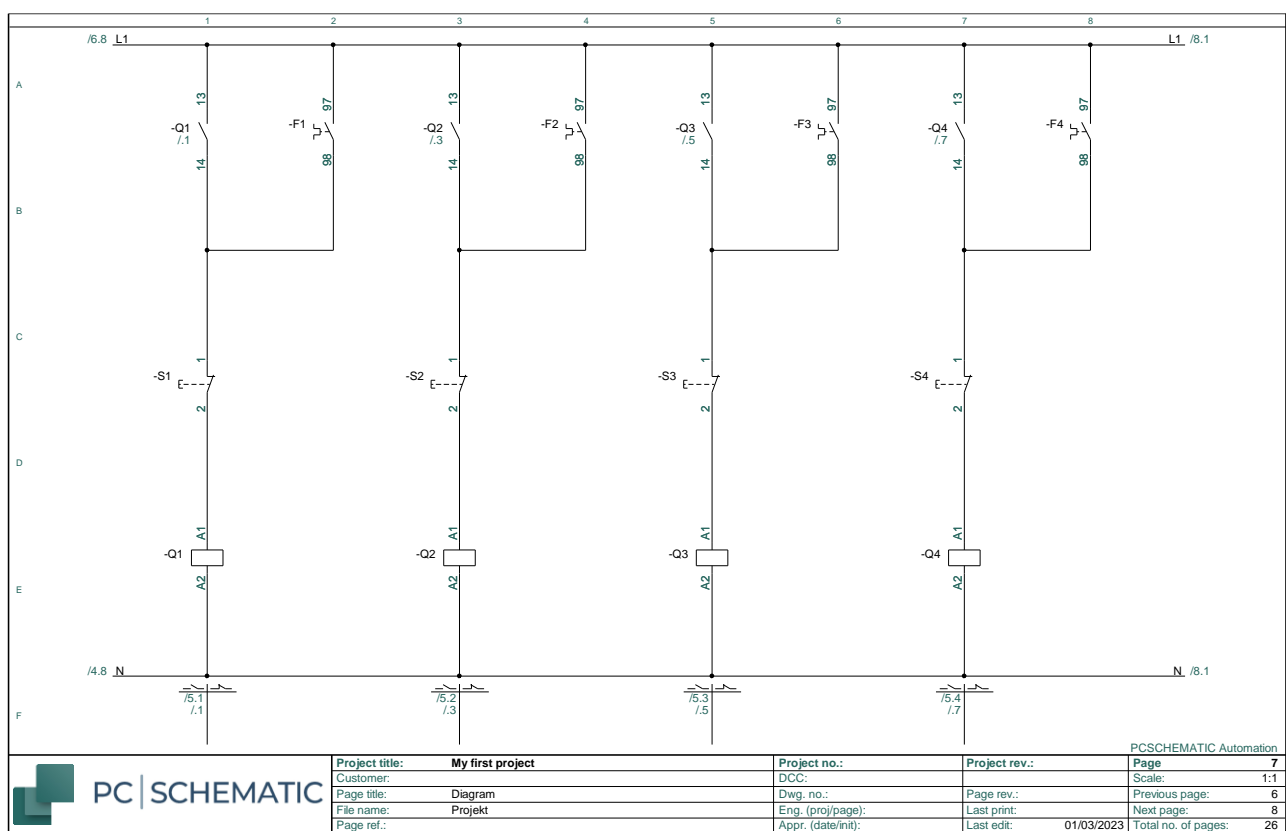
You will be guided through the workflows needed to document a small control system with both power and control circuits using the templates, functions and components found in PC|Automation and the connected database.

Components in the project are demo components found in the program's included database. The finished project may look like the example on the next page; Then you can always check if you got it all.

This is what the power circuit might look like

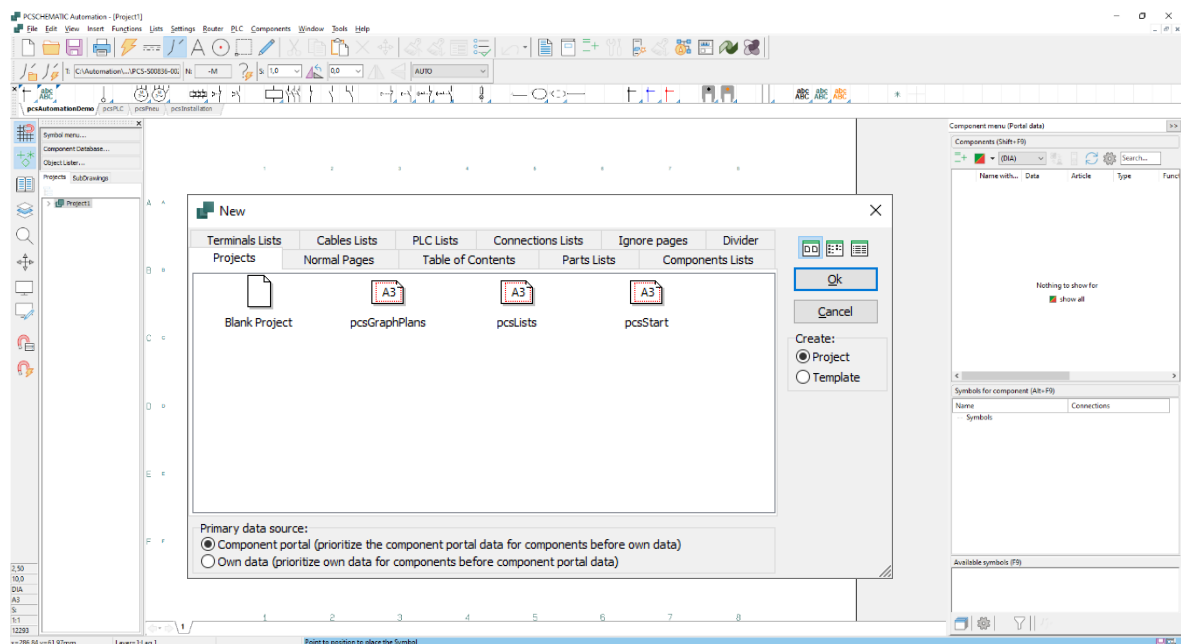


This is what the control circuit might look like

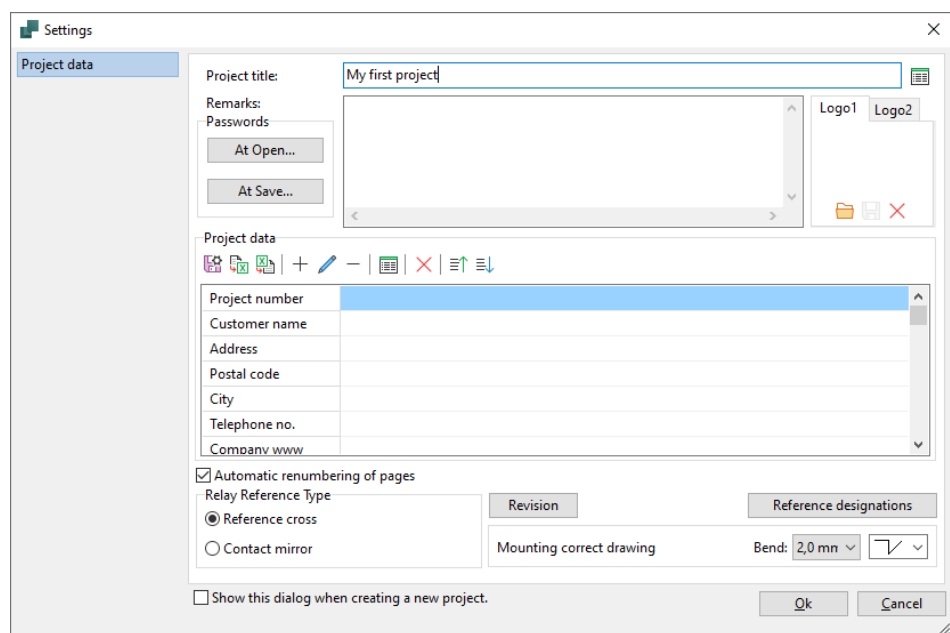


START A NEW PROJECT

Open File|New or click the new document icon and select pcsStart, which is a project template that contains the pages that a project can consist of.



When you select the template, the Project Data dialog appears. Give the project a title, e.g. My first project. When you press OK, you can see the title on the front of your project and in the drawing header on all pages.



You can also open Project Data via the toolbar button to enter the remaining Information.

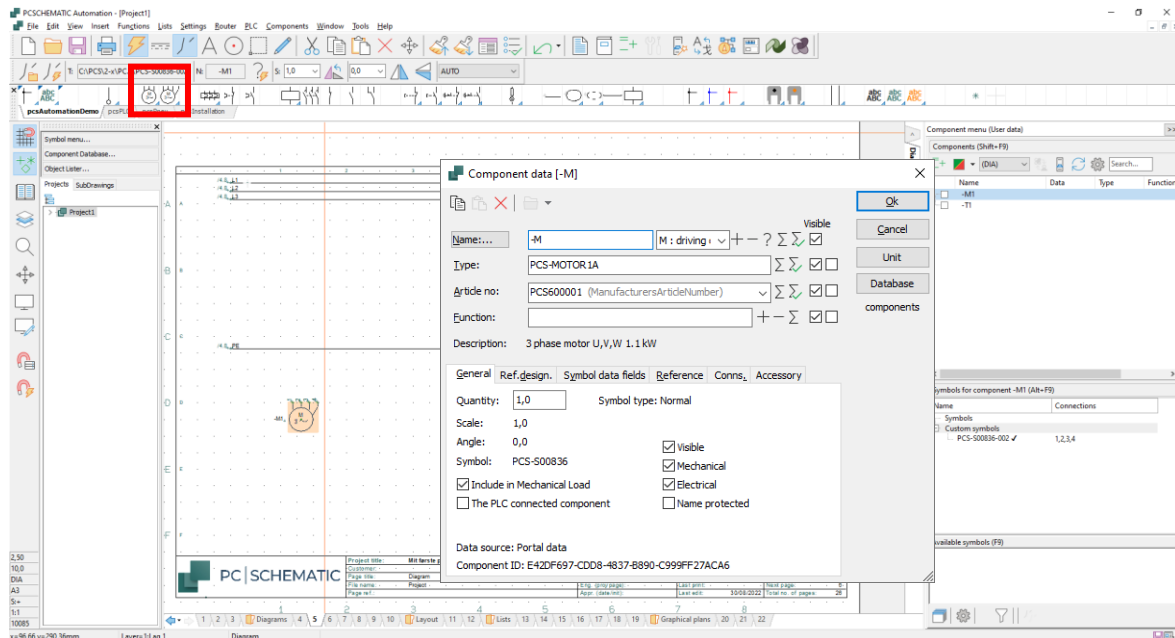


THE POWER CIRCUIT DIAGRAM

Start to make the power circuit diagram.

Place a Motor

Start by clicking on one of the motor symbols in the pickmenu — the row of symbols, components, lines, and more below the toolbars — and place it in the power circuit diagram. I have chosen the one with four connections.



The name "-M" will appear automatically, but you will need to select a number. This is done, for example, by clicking the ? to get the next available number.

In the dialog box, you can

- Enter the Type and Article number directly, or
- Select the Database button to go directly to the component database to select the Type and Article number.

Regardless of method to get the Type and Article number for the components, data comes in the various lists. If the component comes from the database, you can also get other data from here in the lists, such as manufacturer and description.

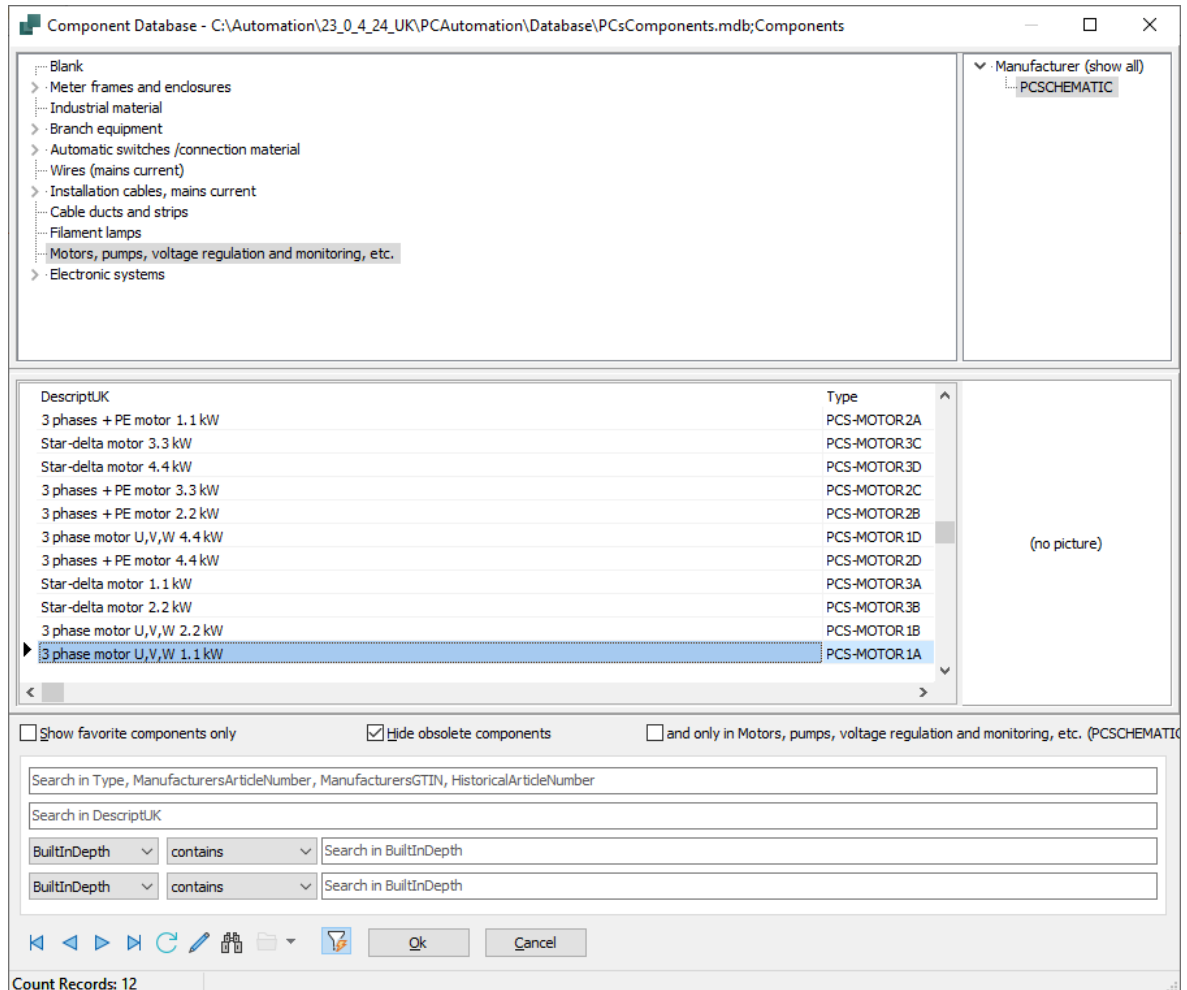
In this example, we have chosen to use only components that are in the database, therefore the Database button is selected. When you click the database button, the program looks in the database for components that use the symbol you placed.

- **Components** are items that are identified by Type and Article number.
- **Symbols** show only electrical functions.

The electrical functions of components are displayed by one or more symbols. The database contains components and various information about them, including the electrical and mechanical symbols that are used to draw them.

Here, I have pressed the database button: The program uses the symbol as the search key in the database.

Be aware, that as a starting point it only finds components with usr-symbols (which are components that you have created or edited); here I have also set it to search in 'our' symbols (pcs-symbols – components that come directly from the component portal):

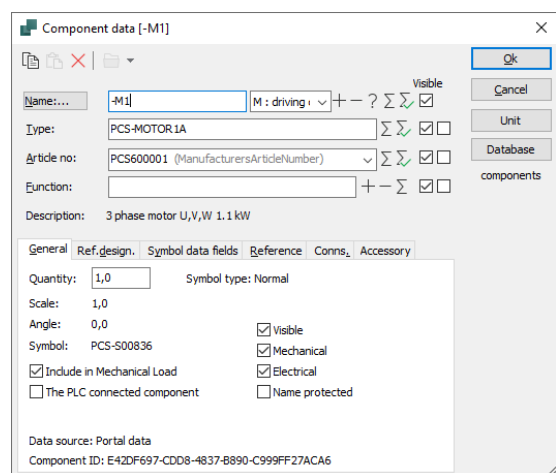


iii

Here you can choose between different motors. Choose the 1.1 kW motor from PCSHEMATIC.

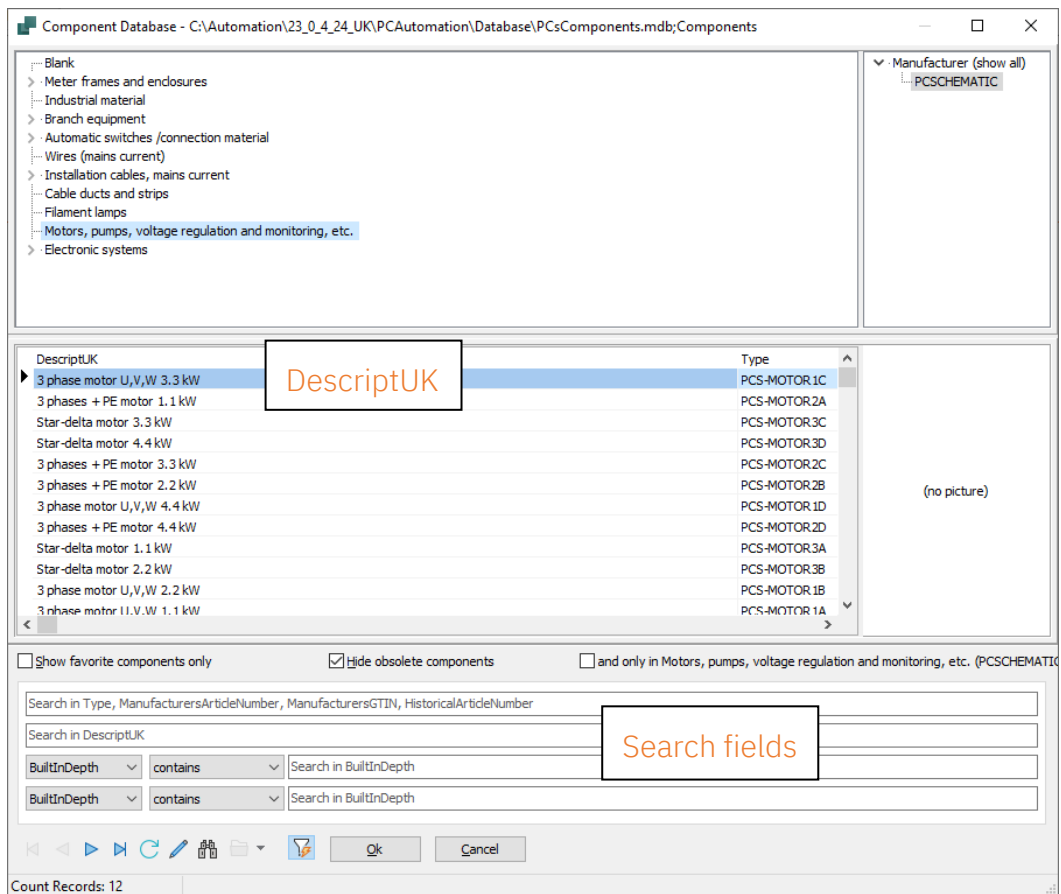
When you press OK, you return to Automation and the dialog box looks like this:

The Type and Article number are filled in with data from the database, the description is also seen, and in the diagram the connection names of the motor are also included.



Place directly from the database

Click the Component Database button... on the left of your screen or press the shortcut [D] to open the database.



You enter this window where you can select components based on

- Manufacturer – click on the desired Manufacturer in the list on the right
- Component groups – click the desired item groups in the list on the left
- Description – search the DescriptUK field.
- Type, Article number – search the Type, ManufacturesArticleNumber field.

You can also use the two additional search fields, where you can search all fields in the database.

First, select the item group Motors, pumps..... and then Manufacturer PCSCHEMATIC.
Now you can see all the motors that are in the database and come from PCSCHEMATIC.

- Find a 3-phase motor with PE and 1.1 kW
- Dobbel-click or tap OK

You come back to Automation, this time with the motor symbol in the crosshair, so you now only need to place this.

When you place the symbol in the drawing, the Type and Article no. dialog box appears.

- Press ? to insert the next available number automatically.
- Press OK, and then you can connect the motor with lines.

Component data [-M]

Name: M M : driving i + - ? Σ Visible

Type: PCS-MOTOR2A Σ Σ Visible

Article no: PCS600005 (Unknown) ✓ Σ Σ Visible

Function: + - Σ Visible

Description: 3-faser + PE motor 1.1 kW

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

Quantity: 1,0 Symbol type: Normal

Scale: 1,0

Angle: 0,0 ☒ Visible

Symbol: PCS-S00836 ☒ Mechanical

☒ Include in Mechanical Load ☒ Electrical

☐ The PLC connected component ☐ Name protected

Data source: Portal data

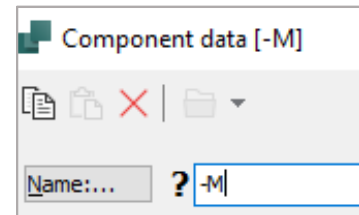
Component ID: 2089487B-A036-43E0-81A5-836B961B9903

OK Cancel Unit Database components

A few tips to save time

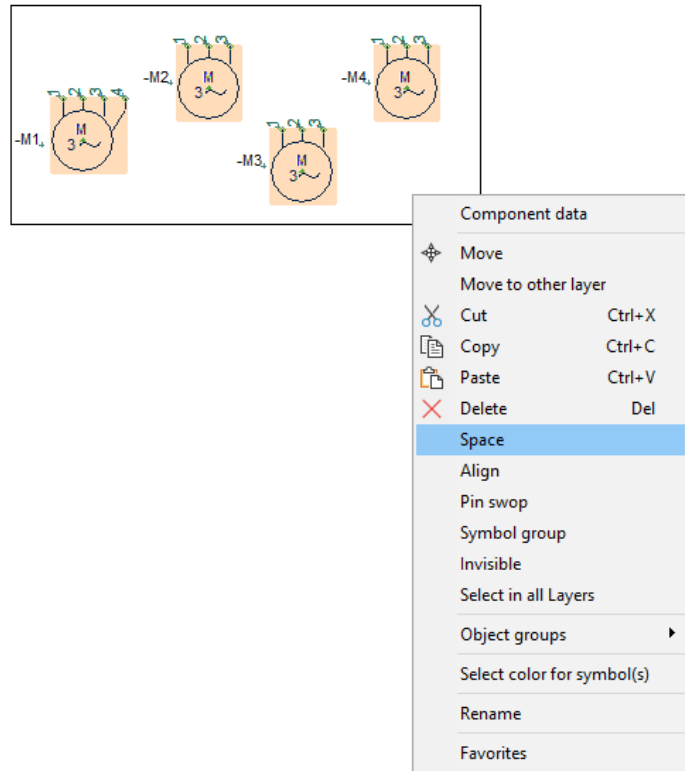
Place 4 motors in the diagram, but don't spend time placing them accurately. When you place them, you give them the next available number by pressing Ctrl + ?

An ? is shown in front of the name.



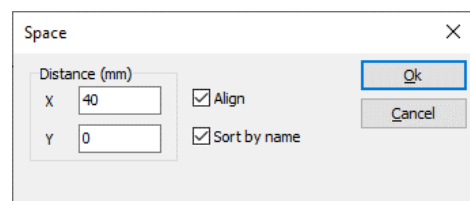
Once all four motors are located in the diagram, select them with the mouse, right-click inside the 'box' and select Space.

Click the mouse where the first motor should be placed – preferably directly in the power path.



Choose a distance of 40 mm in the x-direction and select 'Align' and 'Sort by name'. Click OK.

In this way, all four motors are placed in a row in separate current paths.

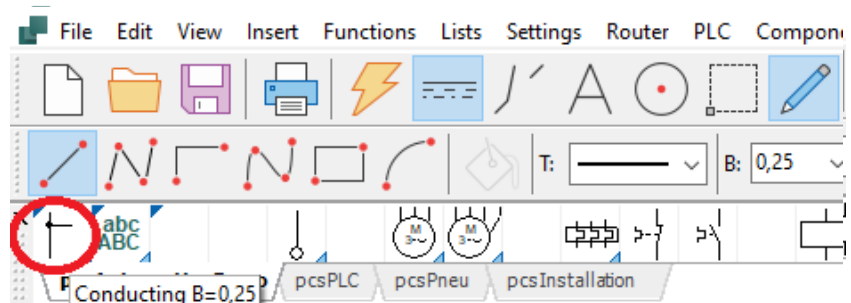


Connect the motor to the phases

The motor must be connected to the phases. This should be done with an conducting line, since in this way there will be an electrical connection between the phases and the motor.

The simplest method is to select the conducting line in the pick menu:

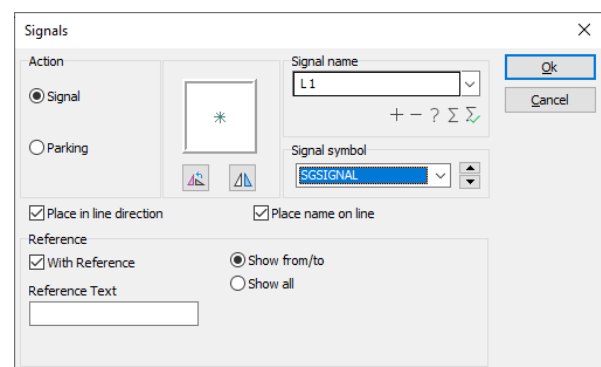
When you click here, the pencil is also activated in the toolbar – you can only draw when the pencil is active.



To switch manually between the different drawing commands, see the Quick start booklet under Drawing Objects.

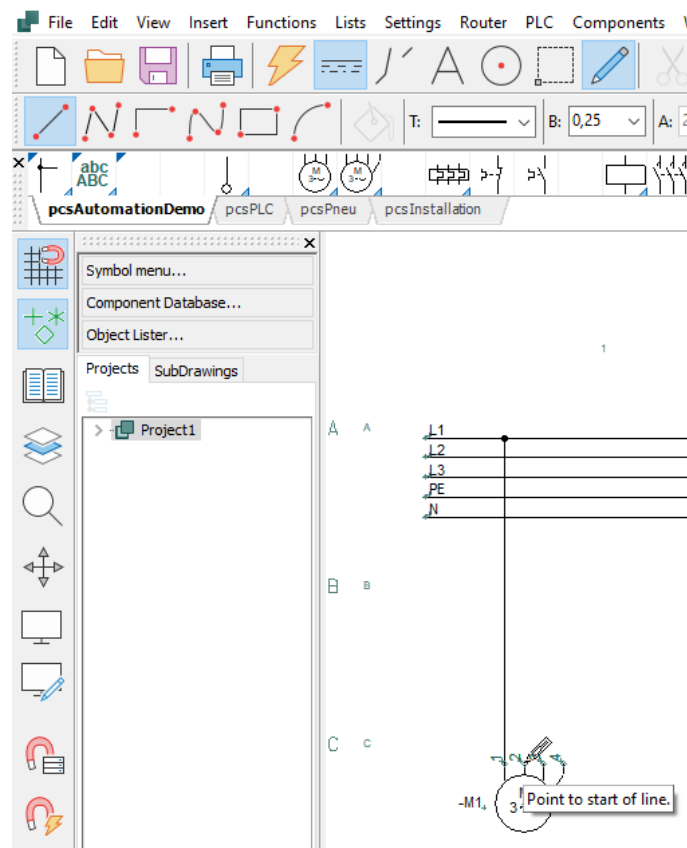
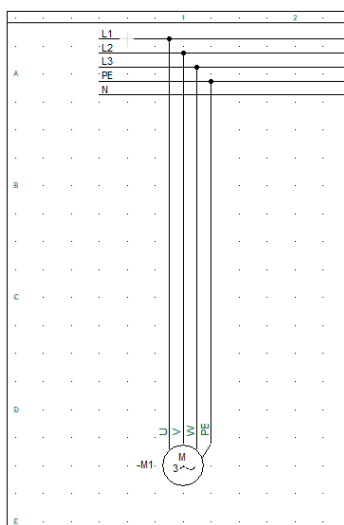
Start by putting the line on the motor. When you see the small square, you have a hold of the connection point itself.

If you don't hit a connection point, this dialogue will come up. Press Esc and try again.



Click here and move the pencil up to L1 and click again. The line ends automatically when you click on L1, and a black 'dot' appears, which tells you that there is an electrical connection.

Continue to connect the other phases to the motor.
The lines later become actual cable conductors.



Maybe your page looks different; But the important thing is that you connect the motor to the supply.

TIPS

You can also use the **Router** to connect the motors:

Click on an motor and release your mouse! (so you don't risk moving the motor while typing)

Press the **R** key. Now you can see where the motor can be connected.

Press the **<** key to skip a line,

If necessary, press **3** to reverse the order.

When you are satisfied, press the **1** key to make the connections that are currently selected.

To turn it off again, press the **R** key.

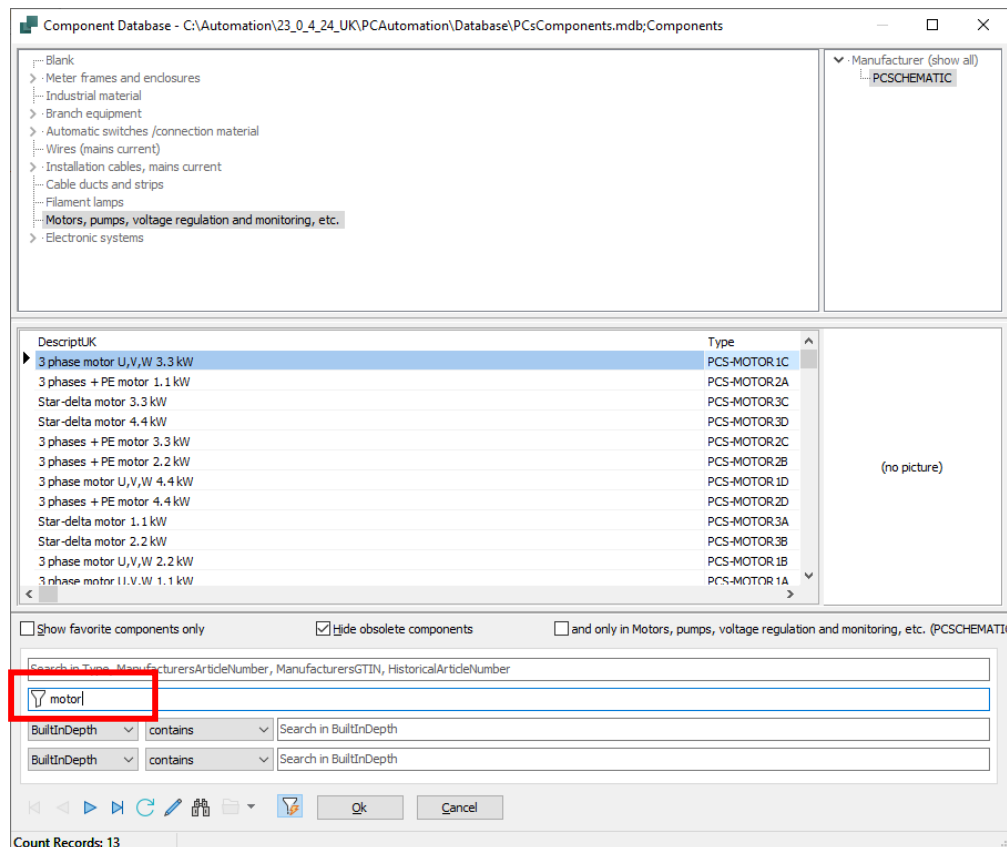
You can also activate the Router before placing the symbols, then the lines will be there immediately! When the feature is active, you can see that there are lines when you move the mouse.

2 finds the closest connection.

4 makes angled lines.

Insert motor protection

Press the [Component Database] button or press [D] to open the database.

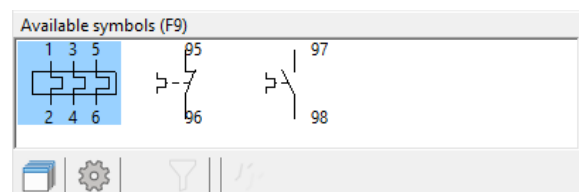


If you don't know the component, you can search DescriptUK in the database:

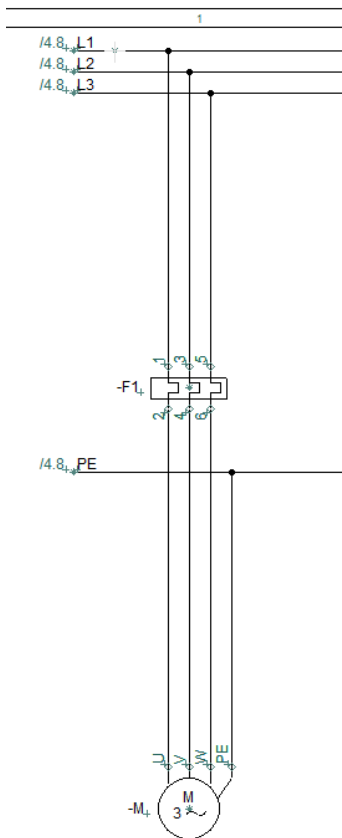
Type the beginning of the word 'MOTOR PROTECTION' and the different ones will appear in the window. You can also search the Type, ManufacturersArticleNumber, ManufacturersGTIN field,

If you choose motor protection Type 1a,

you will get these symbols:
Click on the first symbol – it will now be placed in the diagram.



The window shows all electrical symbols for this component. The symbols have numbers on the connection points corresponding to the terminals of the component itself.



Component data [-F1]

Name: -F1 F : protecti + - ? Σ Visible

Type: PCS-MV001A Σ ✓ □

Article no: PCS2250001 (ManufacturersArticleNumber) Σ ✓ □

Function: + - Σ ✓ □

Description: Thermal overload type 1a

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

Datafield	Value

Add Edit Remove

Ok Cancel Unit Database components

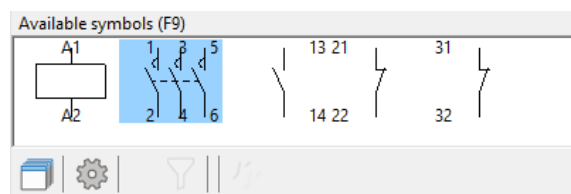
When you place it, the dialog box will open automatically. In the dialog box, you can name the component using ? then it will be called -F1.

Insert Contactor

Then return to the database using [D].

Select the search box "Search Type...". Type 'CON06' in the *Type field*. Click OK.

You will get these symbols in the 'Available symbols' window. The component contains all these electrical symbols. Place the 3-pin contact and name it -Q1.



CIRCUIT DIAGRAM

Go to the L1 and N page of the project.

Here you can place and connect (some of) the last symbols of the motor protection and the contactors, which are displayed in the control circuit. You can see the finished diagrams on page 30 here in the booklet.

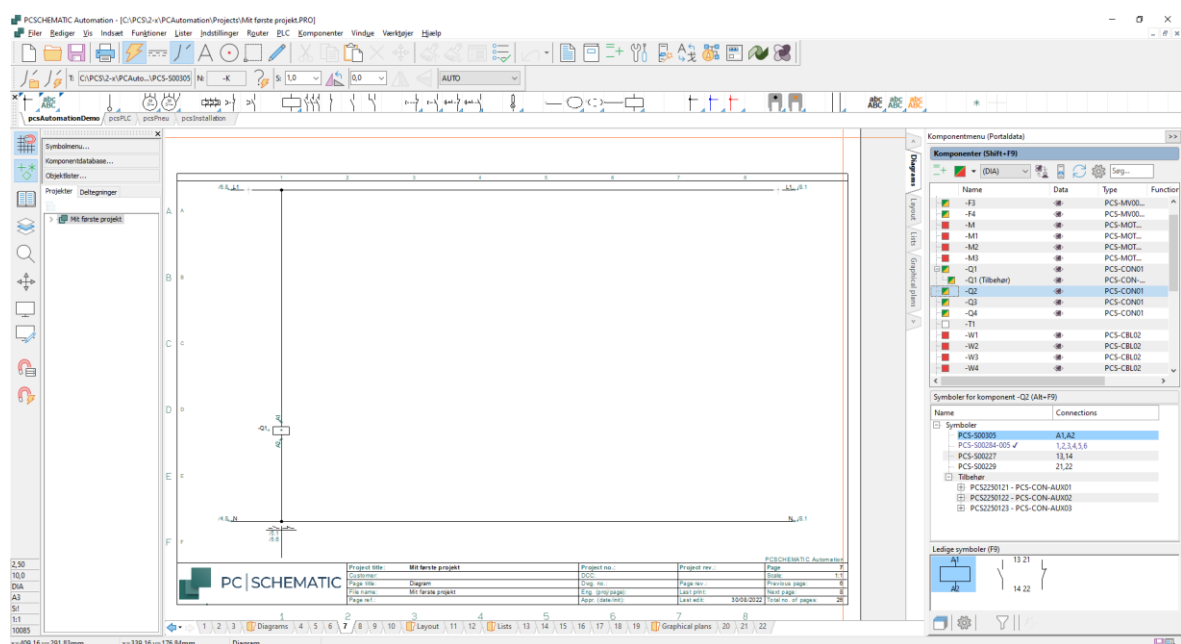
Component menu - show the available symbols

You can get help from the program to remember what you have already placed in the project:

On the right side of the screen, you have the Component menu.

When you select a component in the list, in the lower part you can see its free symbols. You can see them both as file names with connection names and as symbols.

You can choose to display all components or only those with available symbols (red/yellow/green/white button). To sort the list, tap the top of the columns. You can filter using reference designations. In this way, you can control what is relevant to be showed while you work.



BEWARE!!

If you go to the database (again) to find the control circuit switches it will go wrong!! It's like putting another component into the project.

Instead, use all functions of the component you selected for the power circuit scheme.

Relay, coil and control circuit switches

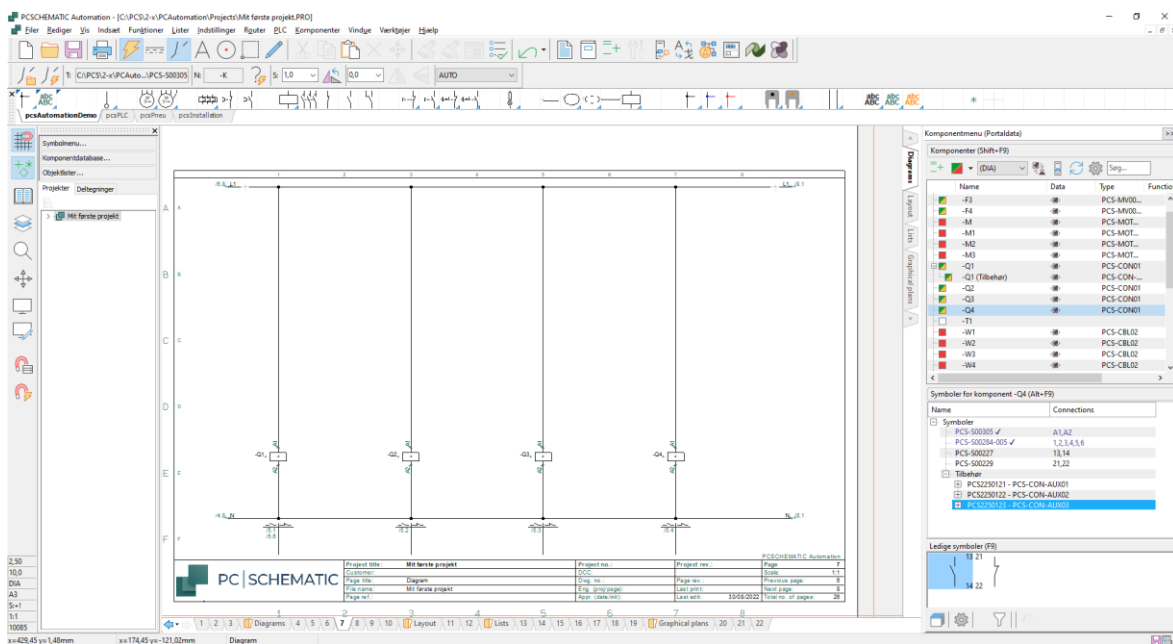
Start by connecting the coil to the relay –Q1.

You can select -Q1 in the upper part of the window.

At the bottom of the window comes a window with the symbols that are not yet connected in the component.

Select the first symbol – the activation coil – and place it on the drawing.

When you place an activation coil, a reference symbol – a reference cross – will automatically appear showing where the different end and refractive contacts in the relay are.



Now the control circuit diagram must be made so that the relays are connected.

Place all four coils. If your Router is active, it makes the connections immediately.

You can place the four coils, mark them, select Space and Align by 40 mm.

The lines are included, so you must 'Align' before inserting more symbols.

Continue to place and connect the various available symbols until you finish the control circuit. If you use all symbols in a component, it will disappear from the list....

Also use the Router here; it makes it faster.

R starts The Router, 2 finds the nearest connection, 4 makes angled lines.

You can also insert hand-operated push buttons by using the symbols from the pick menu, press the Database button in Component Data and select one of those displayed in the database.

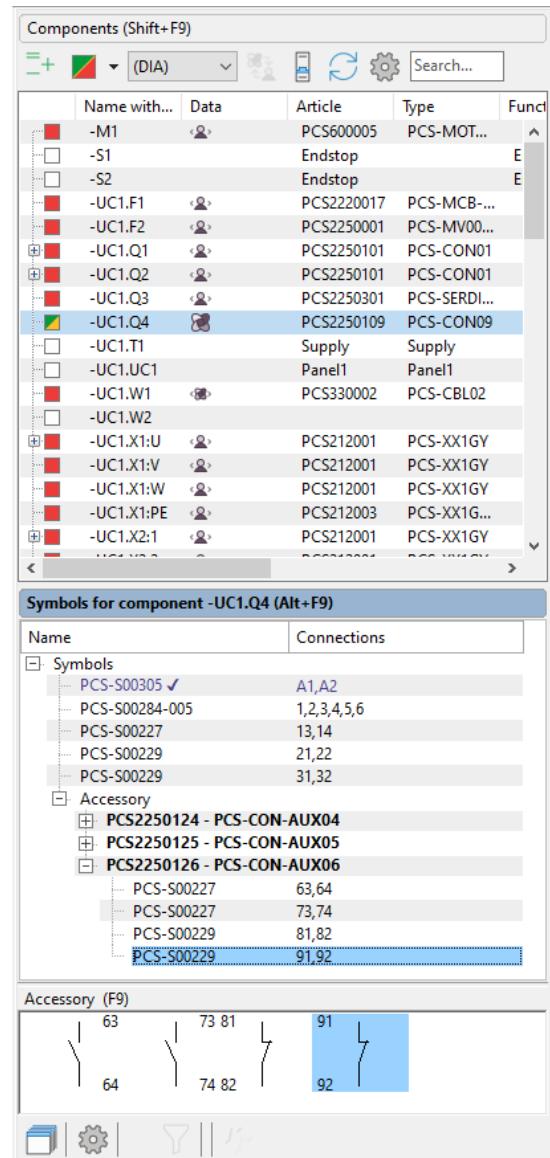
Accessories (not used here)

If you need contact functions for your relay, you can download help contacts from the database. A helper switch is an additional component, i.e. a new Article number.

In the Component menu, you can see the symbols and other accessories defined for that component.

When you select the desired accessory, you can see its symbols in the lower window.

Pick and place the desired symbol, and a line is created under the main component, with the same name (this is the same activation coil) and marked with (accessory). A similar function for mechanical accessories is available on the mechanical page and in the Accessories tab on component data.



You can read more about how to work with drawing objects

- Line
- Symbol
- Text

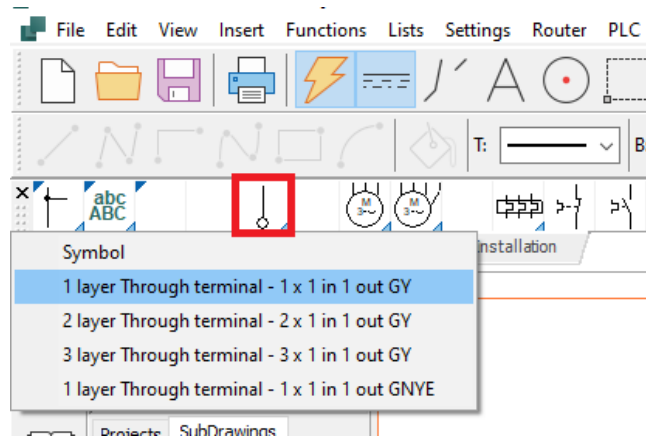
in the Quick start book and in the manual.

Insert terminal row with item data from the pick menu

You need to go back to the power circuit diagram to place the last components, i.e. terminal rows and cables.

You can place terminals directly on individual conductors, one at a time, but this is done faster with the Multiplacement feature:

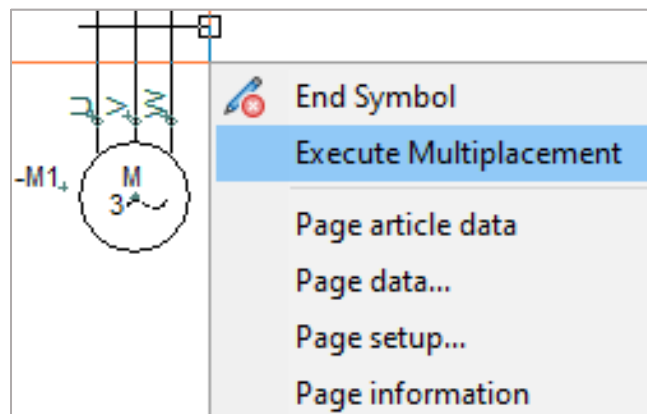
Pick a terminal from the Pick menu, select the terminal as shown here:



Place your crosshair to the left of the first conductor, right-click and select Start Multiplacement.

You get a 'point-out' line in the crosshairs, which you pull over all the leaders.

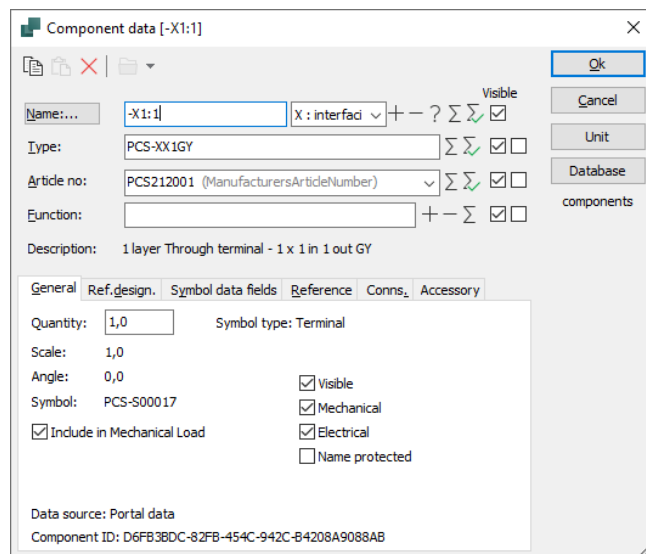
When the line crosses all desired conductors, again right-click and select Execute Multiplacement.



Now the regular component dialog appears, and here you can name all the terminals in the terminal block at once, with consecutive terminal numbers.

Type '-X1:1'. -X1 is the terminal block name :1 is the first terminal number on the row.

Enable automatic counting by holding down <Ctrl> while clicking ? with the mouse. Then you get a ? in front of the name, which shows that there is automatic counting. Press OK.

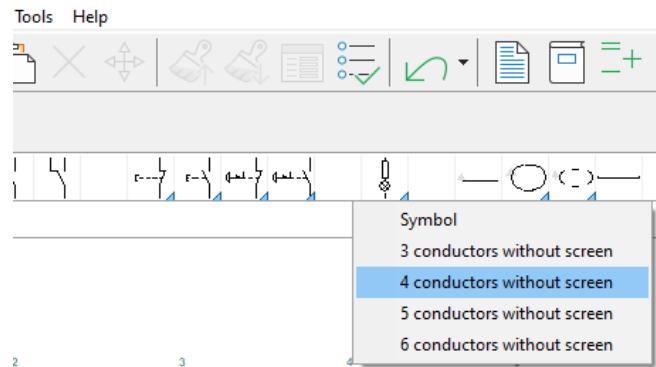


If you type '-X1:U' and do as above, the terminals will be named U,V,W,PE

Insert cable

Pick a 4-wire cable without screen in the Pick menu.

When the window scrolls down, click on the desired cable.

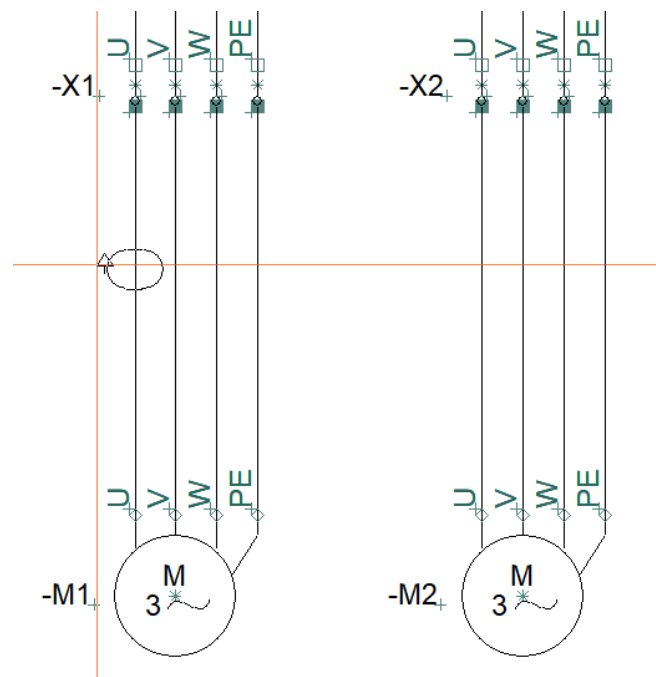


You will get the cable symbol in the crosshair.

Place it slightly to the left of the first conductor in the cable.

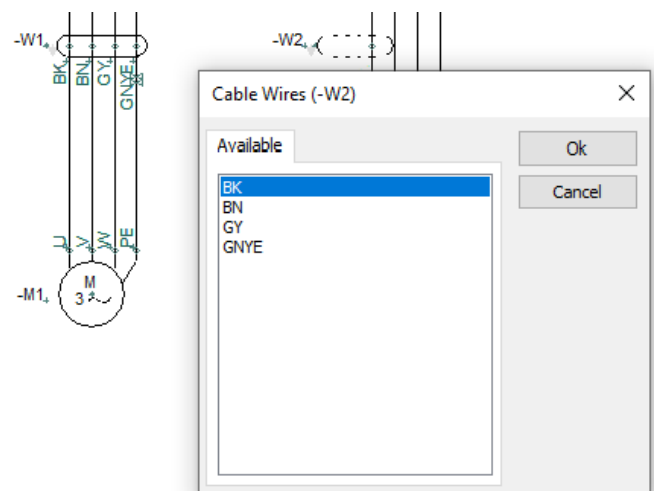
When you place the symbol, the regular dialog box appears, where you can see, Type, Article number and number of conductors in the cable.

Name the cable –W1.



After clicking OK, a 'point-out' line will appear, which you will use to designate the individual leaders in the cable.

Since you downloaded the cable from the database, each conductor also has color or number. Now designate the individual lines, one at a time. Once the leaders have been appointed, press <Ok>.



Repeat this for the –W2, -W3, and –W4 cables.

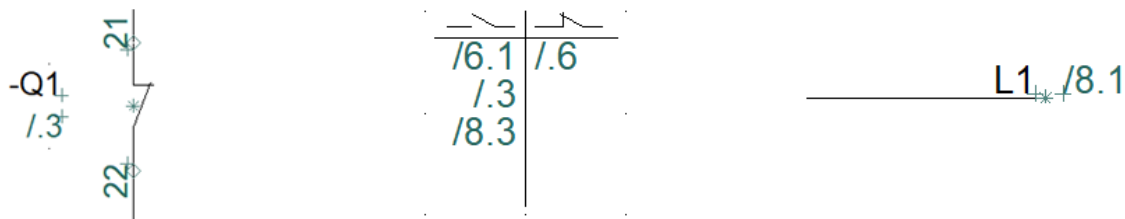
LINK BETWEEN THE PAGES

Once the symbols of the different components are connected on both the power circuit and control circuit diagrams, you can jump between the individual parts of the components.

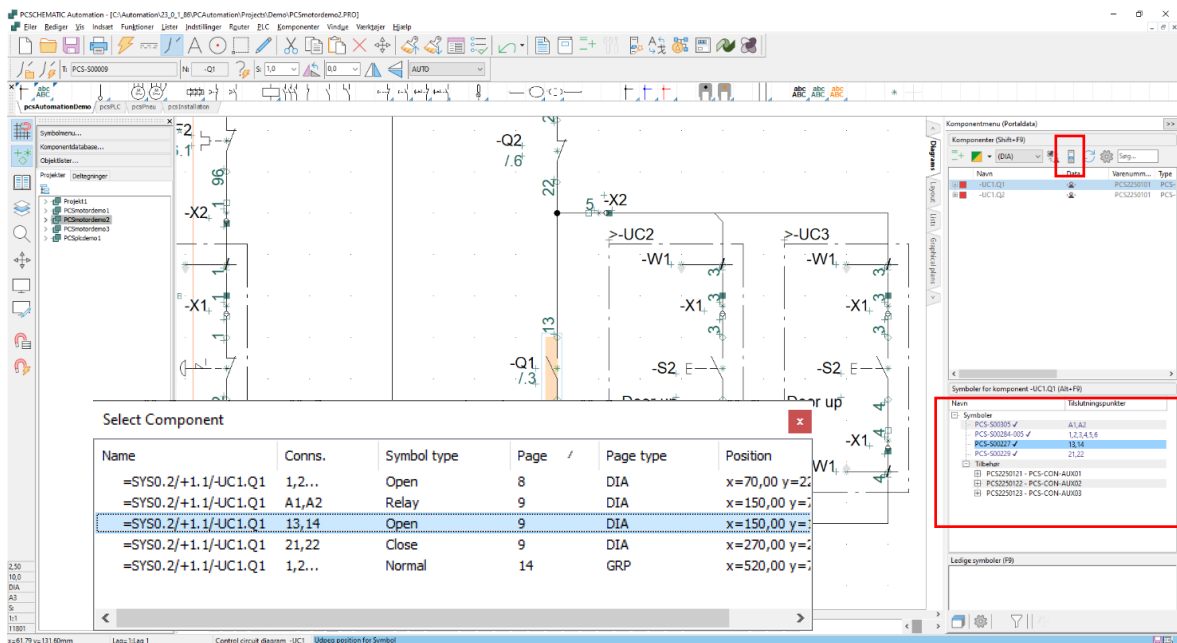
The end-reference cross is a reference symbol, i.e. it contains references to the other symbols. The references are active links you can jump from.

There is reference between the symbols in a component, and there is reference between signal names in a project.

'/' is reference to page, '.' is reference to a power path.



Also, try selecting one of the components in the Component menu. Then tap on the icon for Select component. When you do this, a box appears showing all the symbols placed in the component and their location as hyperlinks: clicking on one of them will take you to it. This window can also be used to jump on to mechanical symbols on the arrangement drawing, if there is such one.



ARRANGEMENT DRAWING

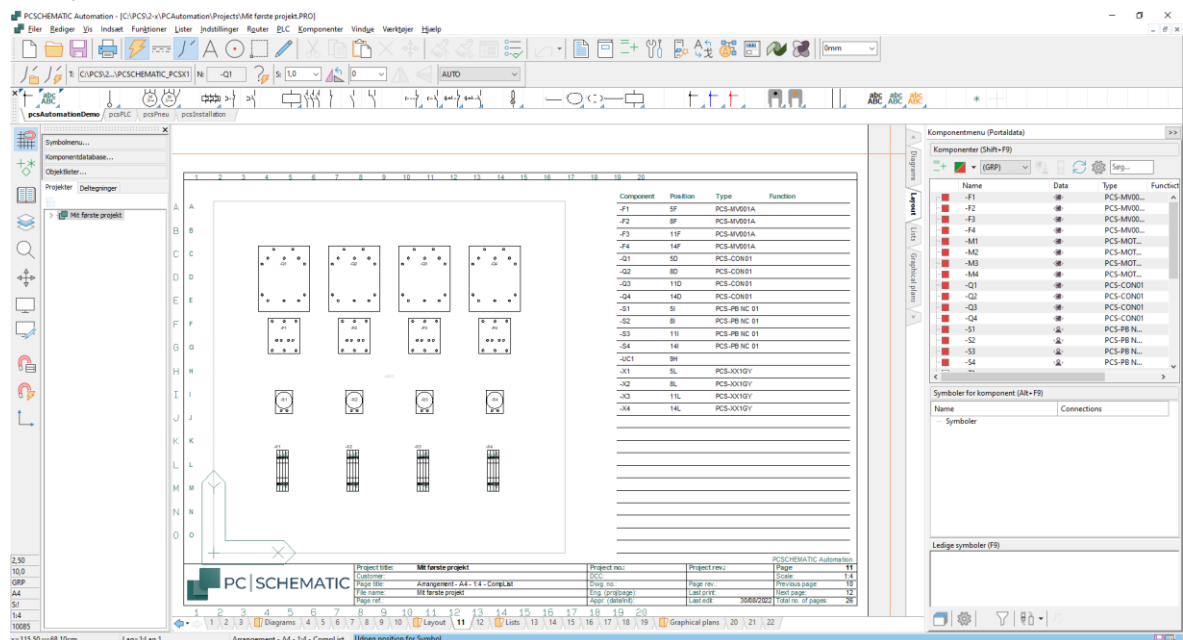
Once you have placed components from the database in your diagrams, you can make an arrangement drawing i.e. a drawing of what the board looks like.

Start by going to the page after the Layout tab. This is a mechanical page (GRP=floor plan). In the lower left corner of the paper, you see the Origo or (0.0) – the corner of the board. You can move this point if it fits better with your project.

On the right side you can see a list of the components that can be placed.

In the lower part, you can see the symbol itself, which is picked and placed. You can also double-click the name in the list. You can activate magnetic snap to make the components lie close.

If you have a terminal row, which consists of several components, you can right-click and select component name, then all the terminals will appear in the lower window and can be selected via right-click and then placed. Same method if you have a component with accessories.

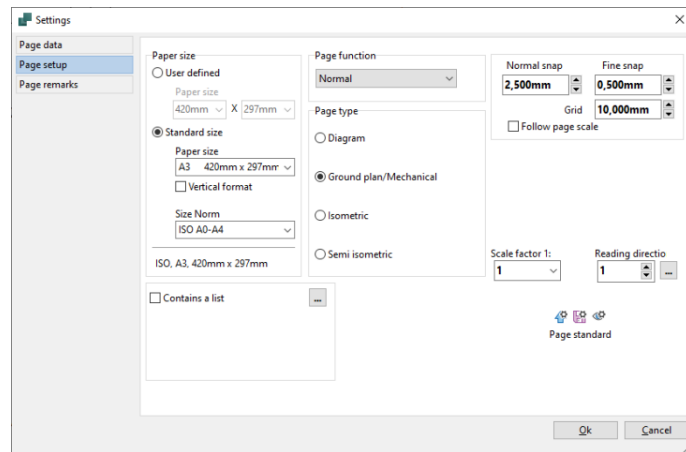


To get this result, I have done the following:

1. Place a board symbol on layer 2.
 - a. The board is made with the Symbol Generator, and has the size 600 x 600 mm.
2. On layer 1, I place components by selecting in the list and double-clicking the name.
3. Some of the components I have alligned ...
4. Finally, I have updated the list by going to Lists|Update All Lists

Everything you do on a mechanical page is determined. If you can't have all components on the page, you can change the page scale:

Click Settings | Page settings and switch to desired page scale.



Measurement

Everything is determined on a mechanical page, i.e. you can set right goals for different purposes. Measurement can be found under the Tools menu.

When you select the feature, you will get a 'point-out-pencil' in the crosshair.

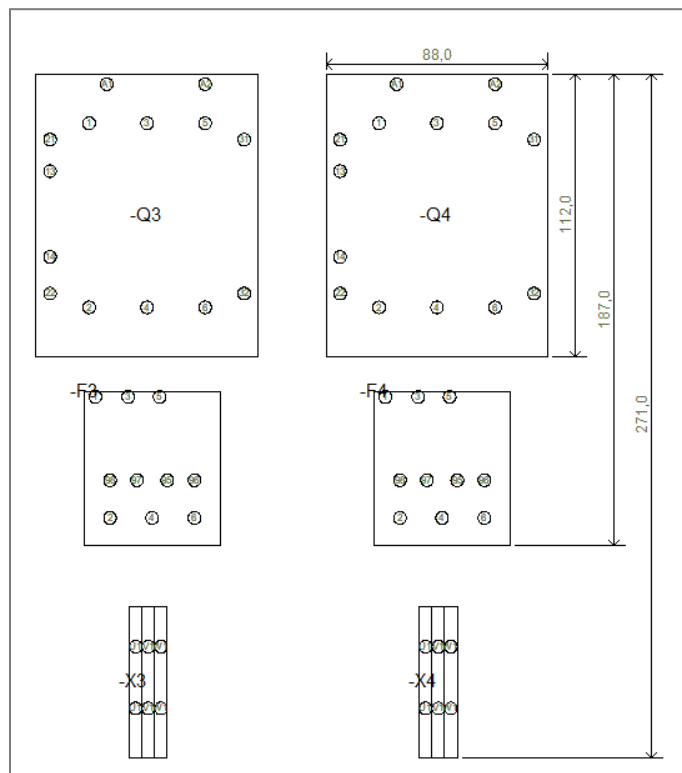
You need to point out where you want to start from.

It is automatically snapped to the end and center of the various lines.

You can hold down <Ctrl> while clicking the mouse to the target anywhere.

If your target is skewed, you can switch between skewed/vertical/horizontal objectives using <spaces> before clicking the mouse.

After setting the different measurements, disconnect the function using <Esc>.



LISTS IN THE PROJECT

In the pcsStart template, there are pages with different lists that should be included in a project, such as table of contents, parts lists and cables lists.

You can update all of these lists by going to the Menu Lists|Update All Lists; or you can update a single list, either here or by right-clicking on the page.

There are also links via the various lists:

- Via the table of contents, you can jump to a page.
- From the component list to a component.
- Via the terminal list to a terminal.
- Etc..

Object Lister can fix page titles... and much more

Select the table of contents on page 3 of the project. All the diagram pages have the same title. If you want to correct a page title, you can go into the individual page's page data and correct it.



Alternatively, you can use the Object Lister for this (shortcut [F7]):

Go to the Pages tab.

Here you can see different page data – more can be found by right-clicking on the header.

1. Activate the pencil.
2. Select the pages that you want to be named the same in the field in 'Title' and type the title and press <Enter>.
3. Close Object Lister.

Object	Name with	Conns.	Article	Type	Function	Sym
1.T1 (0)	-T1	L1,L2,L3,N	Supply	Supply		#x35
	-T1	L1,L2,L3,N	Supply	Supply		#x35
+1.1/-UC1.F1 (1)	-F1	1,2,3,4,5,6,7	PCS2220017	PCS-MCB		PCS
	-F1	1,2,3,4,5,6,7	PCS2220017	PCS-MCB		PCS
+1.1/-UC1.F2 (1)	-F2	1,2,3,4,5,6	PCS2250001	PCS-MV00		PCS
	-F2	1,2,3,4,5,6,9	PCS2250001	PCS-MV00		PCS
	-F2	95,96	PCS2250001	PCS-MV00		PCS

Update the lists again. See table of contents.

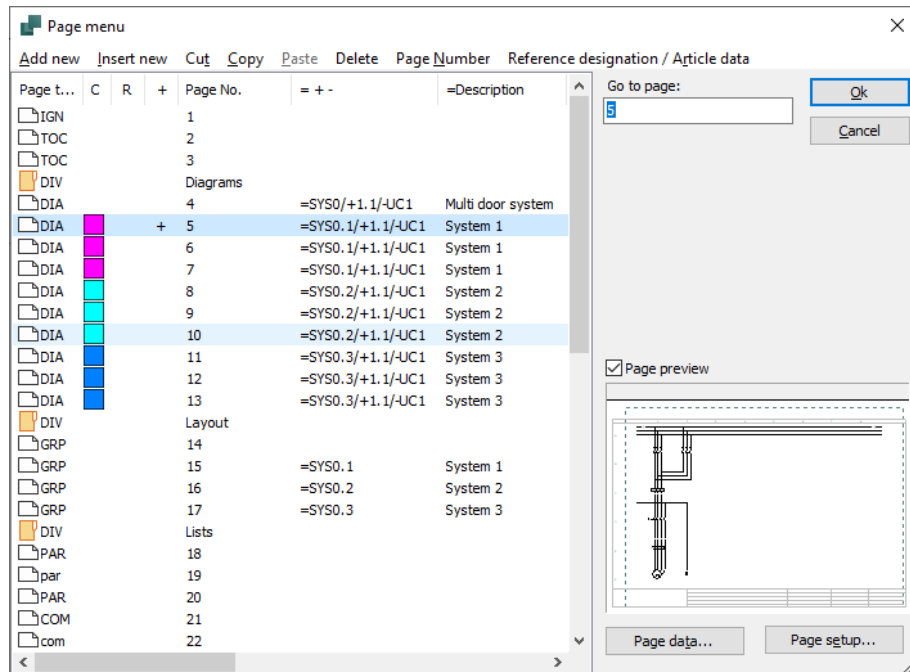
You can fix a lot of other data with Object Lister. Give it a try....

If you fix something with Object Lister, you correct it simultaneously in the project, that means, you don't have to save or do anything else. It's already there.

THE FINAL FINISH

You have started working in a template. This means that sometimes you have to put in extra pages, other times you have to delete pages. And then you need to give new page numbers for the entire project.

All these tasks can be done via the Page menu. Tap the icon in the vertical toolbar on the left of your screen.



Insert new pages

You can insert pages here.

Delete pages you don't need

You can delete pages here.

New page numbers

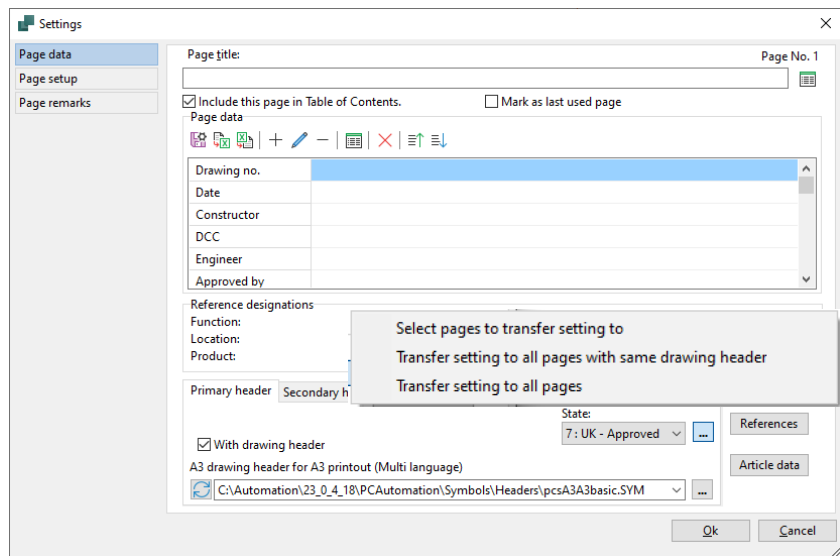
If you select one or more pages, you can change the page number of the selected ones. All commands are in the top bar of the dialog box.

Language settings

All our drawing heads are made with texts in many languages and watermarks, which can be selected on Page data in the dropdown box.

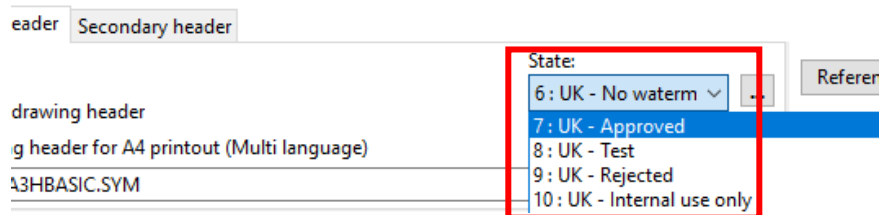
On the button ... you can then choose to transfer a setting to other pages.

The primary drawing head – the frame – is the same on all pages, except the arrangement drawing. You need to change it with 'all pages with the same drawing header'.



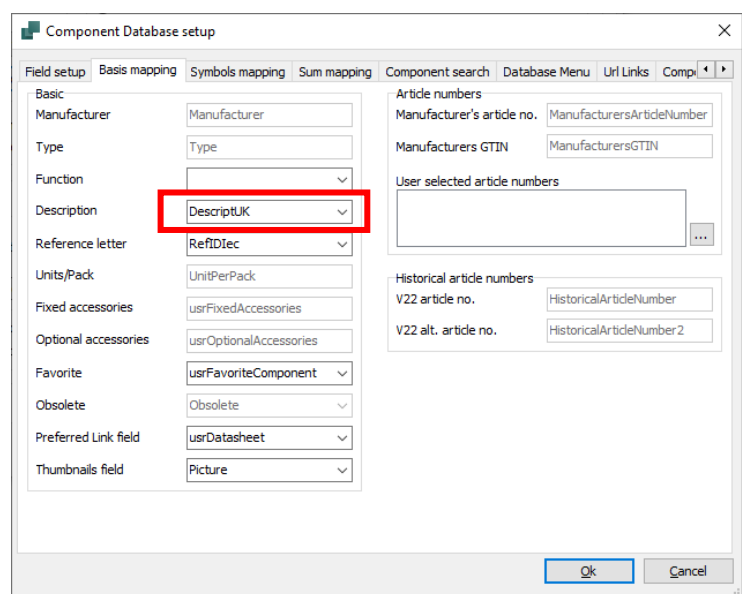
On the list pages there are two drawing heads – the frame and the list itself;.

Here you switch, and transfer to all pages.



Descriptions of components in the lists come from the database.

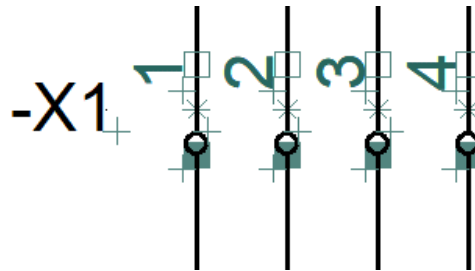
You can change the language by going to Settings|Database|Database settings.



A LITTLE MORE INFO ABOUT TERMINALS AND CABLES

Direction of the terminals

After placing your terminals, you can see that all terminals have two connection points and that one is filled in.



This is because it is convenient to be able to tell the difference between the two sides of the terminal row: the internal side (into the board) and the external side (out of the board).

The filled out part 'points out' of the board.

See also the figure on page 55.

The direction affects how terminal lists and terminal plan's are sorted. And it is only the terminals and cables that are in the project – that also include the power control – which will be included in the lists.

If the direction is wrong in relation to your project, you can select the individual terminal – or the entire terminal row – right-click and select Pin swap.

Show terminal row

In addition to the direction of the individual terminals in the diagram, it is also important to keep track of the order of the terminals in the row and which type of terminal you have selected:

Terminal rows can consist of many different terminals: 1-layer, 2-layer mm, and if the order is not ok, the terminal list becomes difficult to use during assembly.

Right click on a terminal and select 'Show terminal row'.

The window shows the order, and if it is not ok, you can drag the individual terminals to a new position.

You can also see if several terminal symbols go to a component – multilayer terminal – in the window.

The order of the terminals follows the component group. And the component group follows the order in which they are entered in the project.

That means, the first terminal located in a terminal row is the first, the next becomes the second one, etc.

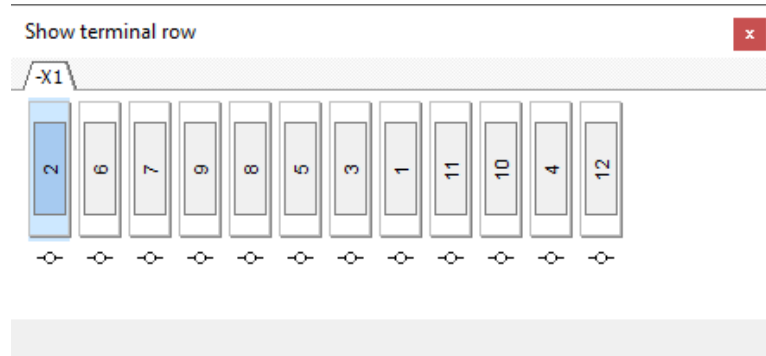
Component grouping

If the number or order doesn't match, you can use the Component Grouping feature.

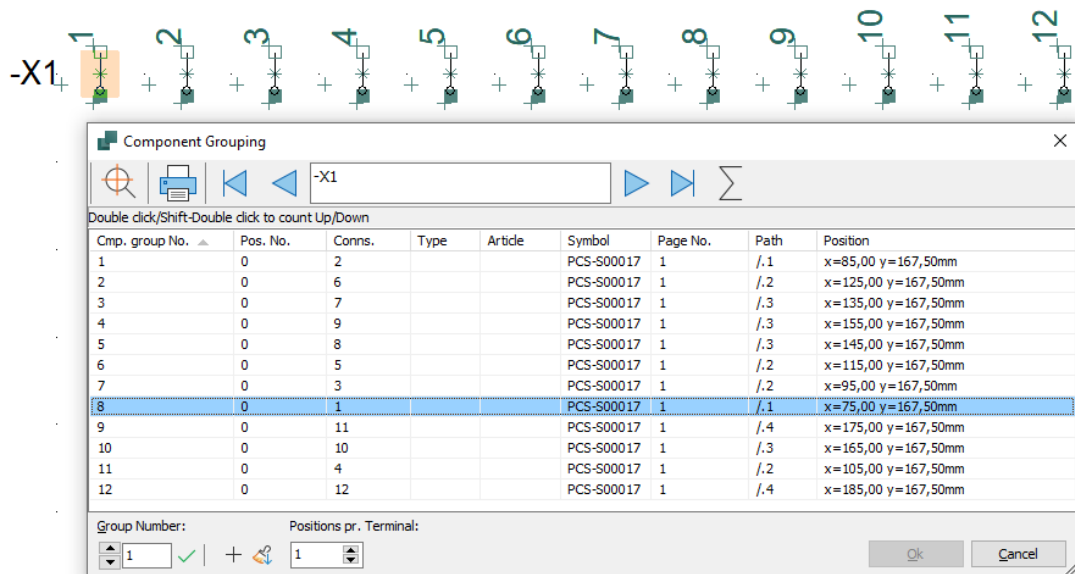
Here are placed 12 terminals in a project, in any order. Which is equivalent to inserting parts of a terminal row in a not too structured order.

After that, the entire terminal row is renumbered, after which the symbols of the project have been given the correct numbers, but the order has been incorrect.

At first, it looks right in the project, but in the Show terminal row you can see that the order is wrong.



Right-click a terminal and select Component Grouping (Ctrl+F7):



Here you can see that there are 12 component groups, corresponding to 12 components.

The terminal row order follows the component group numbers, not the connection names.

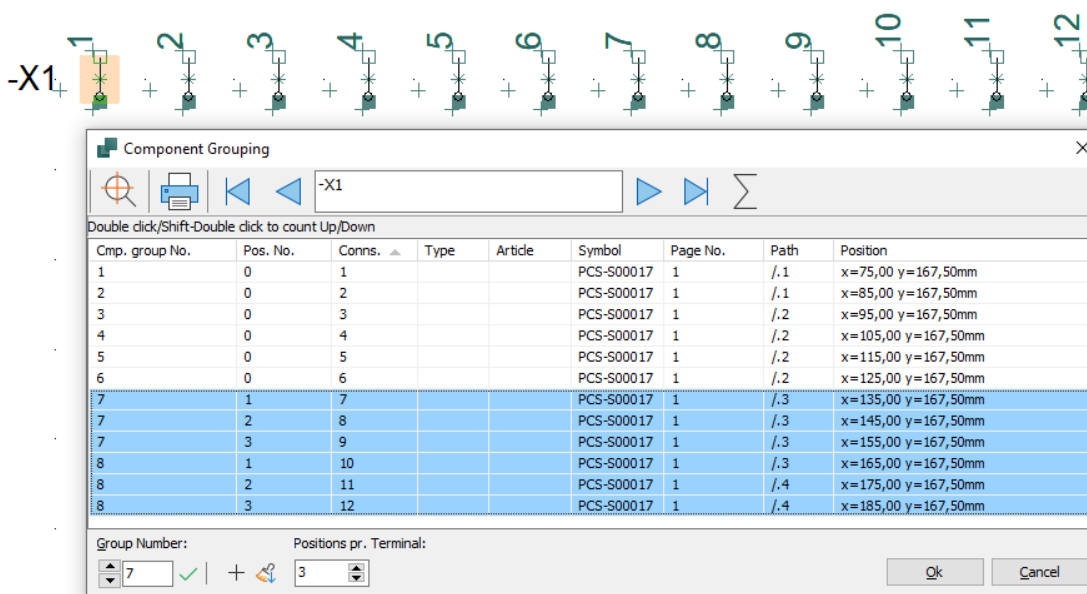
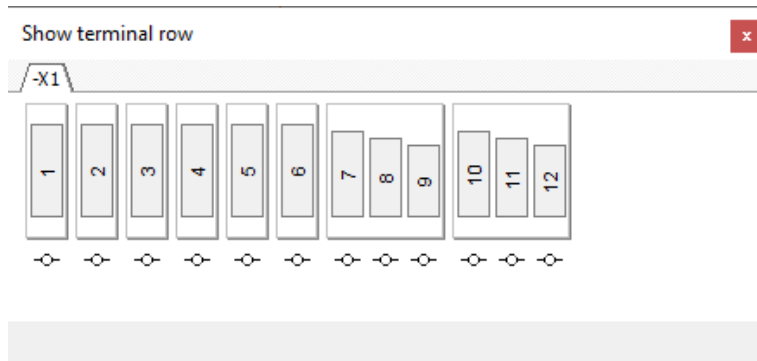
You can sort the terminal row based on e.g. the Connection Name or Position column, maybe only one selection using Shift or Ctrl selection.

Then select the Number of positions per terminal, which is transferred to the terminal row. 1 position for 1-layer terminals, 2 for 2-layers, etc...

Here the terminal row has been regrouped:

Terminals 1-6 are 1-layer terminals, which each have their own consecutive component group number and Pos.nr. 0

Terminals 7-12 are 3-layer terminals, in which there are three terminals around a component group number, and pos.nr. reflects the layers on the terminal.



Component Grouping

Double click/Shift-Double click to count Up/Down

Cmp. group No.	Pos. No.	Conns.	Type	Artide	Symbol	Page No.	Path	Position
1	0	1			PCS-S00017	1	/1	x=75,00 y=167,50mm
2	0	2			PCS-S00017	1	/1	x=85,00 y=167,50mm
3	0	3			PCS-S00017	1	/2	x=95,00 y=167,50mm
4	0	4			PCS-S00017	1	/2	x=105,00 y=167,50mm
5	0	5			PCS-S00017	1	/2	x=115,00 y=167,50mm
6	0	6			PCS-S00017	1	/2	x=125,00 y=167,50mm
7	1	7			PCS-S00017	1	/3	x=135,00 y=167,50mm
7	2	8			PCS-S00017	1	/3	x=145,00 y=167,50mm
7	3	9			PCS-S00017	1	/3	x=155,00 y=167,50mm
8	1	10			PCS-S00017	1	/3	x=165,00 y=167,50mm
8	2	11			PCS-S00017	1	/4	x=175,00 y=167,50mm
8	3	12			PCS-S00017	1	/4	x=185,00 y=167,50mm

Group Number: 7 Positions pr. Terminal: 3

Ok Cancel

When you multi-place terminals, a new component group comes in for each time a symbol appears in the diagram. This means that if you multi-place a 3-layer terminal, you get 3 times as many components as you need.

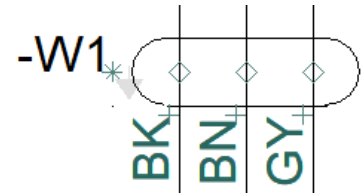
Therefore, you need to component group the terminal row afterwards if the Part list must fit.

In the demo project PCSCompMenu.pro there are examples of multilayer terminals.

Direction of cables

Notice the small arrow next to the cable name. This is a directional arrow: here it shows that the cable comes from above and goes downwards, which is important for sorting cable lists and plans.

If the orientation is wrong for your project, select it, right-click, and choose Reverse direction.



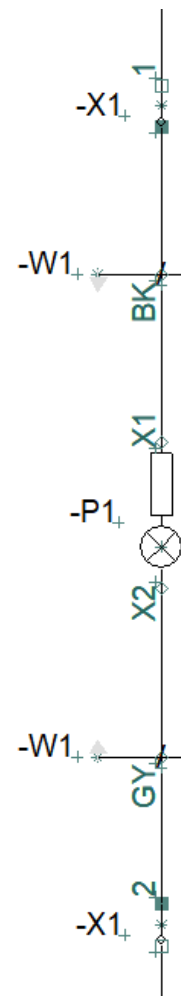
Quick placement of a cable

If the conductor color or conductor number matches your cable, you can now place all conductors on the cable by:

- Place the cable symbol
- Click and drag the mouse over all the conductors
- Now the conductors are placed in the order in which they are located in the database – as in the image above.

Cables on the control current diagram

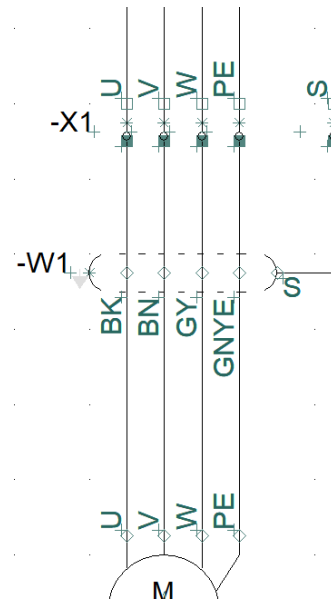
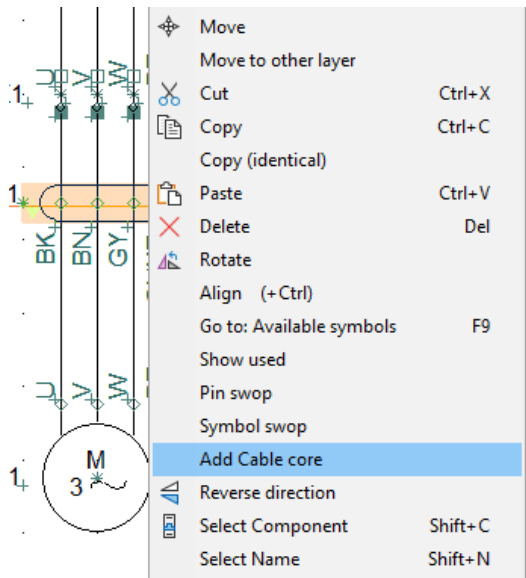
On the control current diagram, place the first conductor as described, then go down vertically and place the second conductor – without Escape. Then they turn correct – as in the picture here:



Conductors with screen

The conductor symbol may be prepared for a conductor screen: the 'dotted' symbol is intended for conductor with screens, where the full drawn line symbol is without.

To put the screen on, right-click on the symbol and 'Add Cable core'. A connection point appears on the side of the symbol, and the beginning of a line that connects in the diagram.



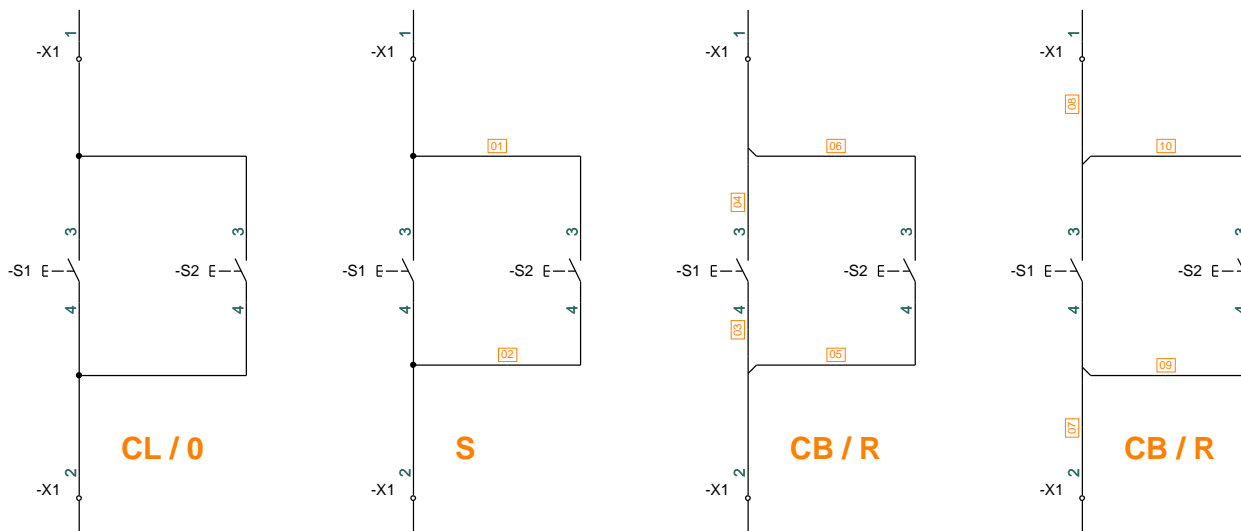
MARKING AND WIRE NUMBERING

Marking wires and cables is an important part of building a control system, switchboards, or machine.

This makes different demands on the level of detail of the diagrams, but there may be good maintenance arguments for making high demands on the markup.

The standard IEC 62491, shows different ways of doing it:

Method	Meaning
0	No marking If all the wires can be followed visually, then you do not need to label them. And no, there aren't that many of those kinds of connections in reality. ...
CL	Connection Local – terminal markup In the example below, the wires will be labeled with the connection name, e.g. S1:3, S2:4 or possibly just 3 or 4. Here it is fine to use dots.
S	Signal markup All wires are labeled with the signal it represents. Here it is fine to use dots. That way, you can see which wires are connected. We also use the method of Potential Marking, i.e. where all wires in a network have the same potential. In the example below, there will be four wires, where two wires have the potential 01 and 02. Often you will mark phases after this method. However, you should also assign a group name, as this way you can see which group the phase belongs to, e.g. L1; F1, L1; F3, etc..
R	Identification marking, means each conductor has it's own name. If you want to use this method, you should have control over the individual connections, i.e. you should make your diagrams with assembly-correct branches. In the examples below, you can see the difference between two different branches – and if dots had been used, you could not clearly see where the connections were.
CB	Connection both ends – both ends connections are shown at both ends (there is also a method called CR – Connection Remote) You can replace the continuous wiring number with the display of Local and Remote ends. In the examples below, it would be, for example, -S1:4/-S2:4 og -S1:4/-X1:2. As you can see, drawing with branches is required for this to make sense, and the direction of the branches also matters.
A	Cable manufacturer's marking If connections were in a cable, it would be sufficient to label with cable name and conductor color or no. Not relevant in the examples below.
Mixture	And you can perform it in several different ways, and possibly mix the methods to suit the task. E.g Potential + terminal markup – then you can still draw with dots and still get a very informative markup.



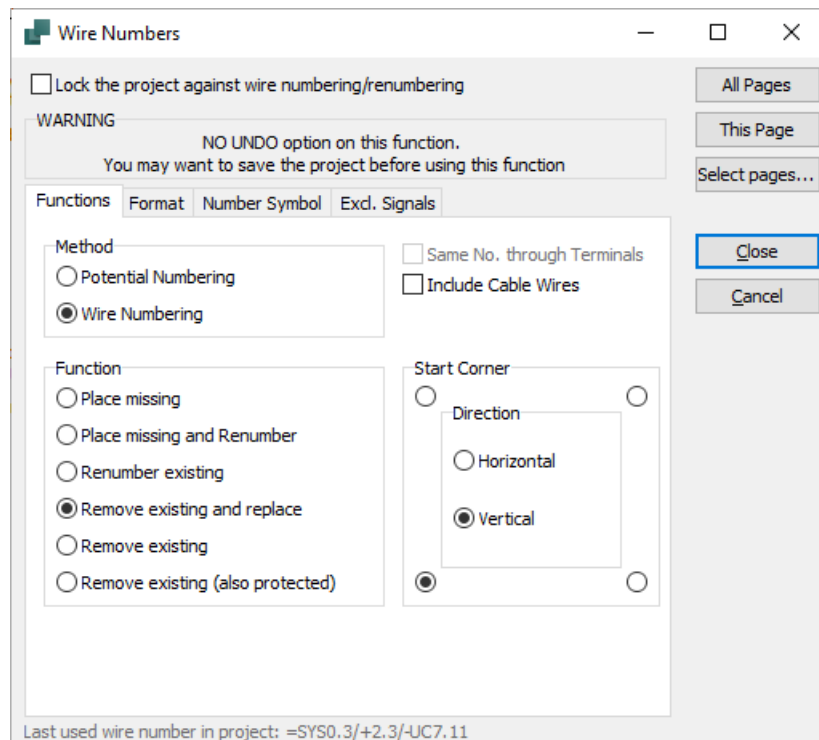
When drawing, you can choose whether you use 'dots' or 'branches' to indicate how the connections are made.

Electrically, it usually means the same thing, but when it comes to marking the wires, it is of great importance whether the diagram is made one way or the other.

Wire Numbers function

If you want to tag up wiring numbers as in the last three examples above, you can either insert the symbols manually or – best – use this feature.

You choose the method on the front page, and on the subsequent tabs you can choose the format and exclude signals.



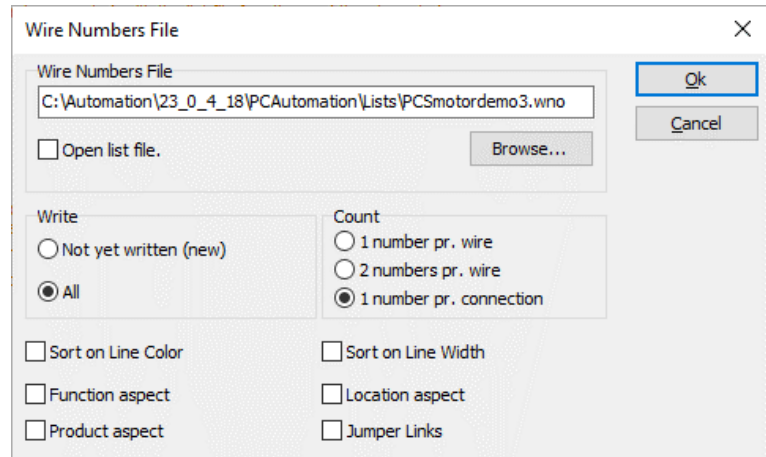
Printing wiring numbers

Depending on the method, you can write out the wiring numbers via different lists.

From the Lists menu, you can choose Wire Numbers File.

Here you can get out a csv list that contains the number of tracks you have selected.

Used for methods S and R.



The 'Wire Numbers File' dialog box is shown. It has a title bar with a close button. The 'Wire Numbers File' section contains a text box with the path 'C:\Automation\23_0_4_18\PCAutomation\Lists\PCSmotordemo3.wno' and a 'Browse...' button. Below this is a checkbox for 'Open list file.'. The 'Write' section has two radio buttons: 'Not yet written (new)' and 'All' (which is selected). The 'Count' section has three radio buttons: '1 number pr. wire', '2 numbers pr. wire', and '1 number pr. connection' (which is selected). There are four checkboxes: 'Sort on Line Color', 'Sort on Line Width', 'Function aspect', and 'Product aspect'. On the right side, there are 'Ok' and 'Cancel' buttons.

You can also go to Connections List File or Connect points File to make a suitable list.

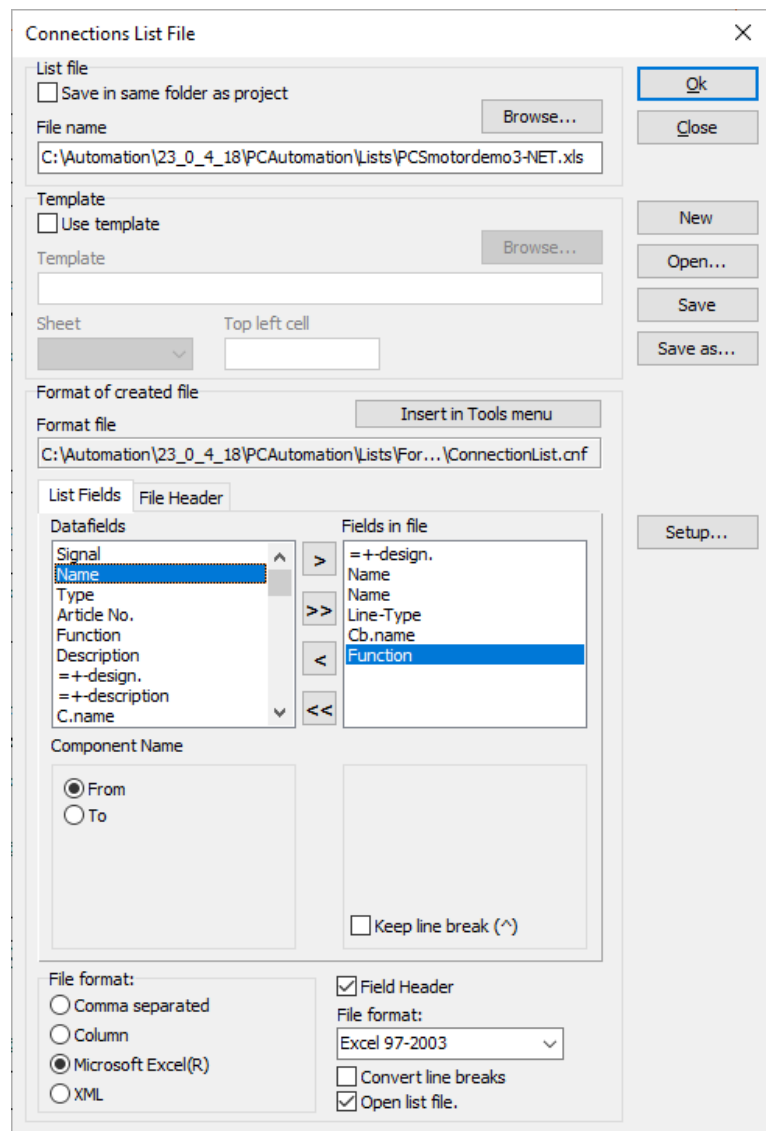
Here you first make a Format file, which determines which data from the project to export.

Here it is the connection names for method CL.

Here I have chosen to take the RDS code in a column so that I can sort the list in *the printer*.

I have also chosen to have a column with

Name:Connection and another where the connection stands on its own. Then you can choose how much information should be on the wire.



The 'Connections List File' dialog box is shown. It has a title bar with a close button. The 'List file' section has a checkbox for 'Save in same folder as project' and a 'Browse...' button. The 'File name' text box contains 'C:\Automation\23_0_4_18\PCAutomation\Lists\PCSmotordemo3-NET.xls'. The 'Template' section has a checkbox for 'Use template' and a 'Browse...' button. The 'Sheet' dropdown is set to 'Top left cell'. The 'Format of created file' section has an 'Insert in Tools menu' button and a 'Format file' text box with the path 'C:\Automation\23_0_4_18\PCAutomation\Lists\For... \ConnectionList.cnf'. The 'List Fields' section has two panes: 'Datafields' and 'Fields in file'. The 'Datafields' pane lists 'Signal', 'Name', 'Type', 'Article No.', 'Function', 'Description', '=+-design.', '=+-description', and 'C.name'. The 'Fields in file' pane lists '=+-design.', 'Name', 'Line-Type', 'Cb.name', and 'Function' (which is selected). The 'Component Name' section has two radio buttons: 'From' (selected) and 'To'. There is a checkbox for 'Keep line break (^)'. The 'File format' section has four radio buttons: 'Comma separated', 'Column', 'Microsoft Excel(R)' (selected), and 'XML'. The 'File format' dropdown is set to 'Excel 97-2003'. There are checkboxes for 'Field Header', 'Convert line breaks', and 'Open list file.'. On the right side, there are 'Ok', 'Close', 'New', 'Open...', 'Save', 'Save as...', and 'Setup...' buttons.

MY FIRST PROJECT – WITH PLC

This chapter describes how to document PLC installations with PC|Automation. You can, among other things, read about:

- How to retrieve and place different types of PLC in the database
- How to load the I/O list with miscellaneous info
- How how to create your own PLC if it is not in the database

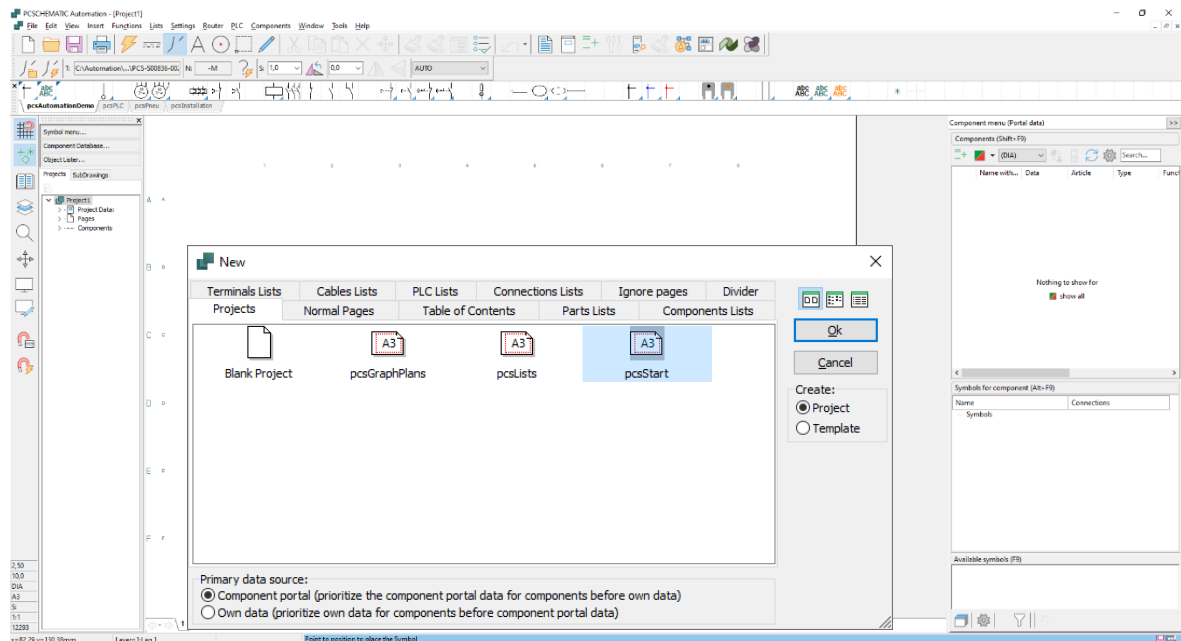
As you read the chapter, follow the examples. Once you have done that, you will have tried the most common functions of this part of the program and will be able to make your own small project.

All examples are based on demo components that are stored in the program's standard database.

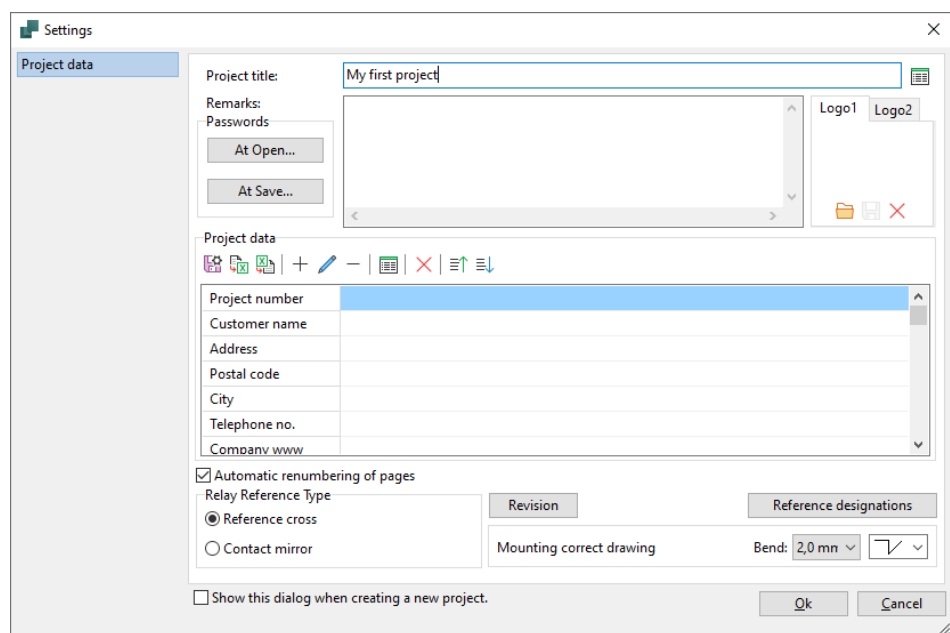
When you make larger projects, you will often work with partial drawings and/or copy from old projects. Here you will need to group the individual symbols into components across many pages. This is also reviewed and the examples used are in the SubDrawings folder.

START A NEW PROJECT

Go to File|New or click the new document icon and select pcsStart, which is a project template that contains the pages a project can consist of.



When you select the template, the Project Data dialog appears. Give the project a title, e.g. My first project. When you press OK, you can see the title on the front of your project and in the drawing header on all pages.



You can also open Project Data via the toolbar button to edit the remaining Information

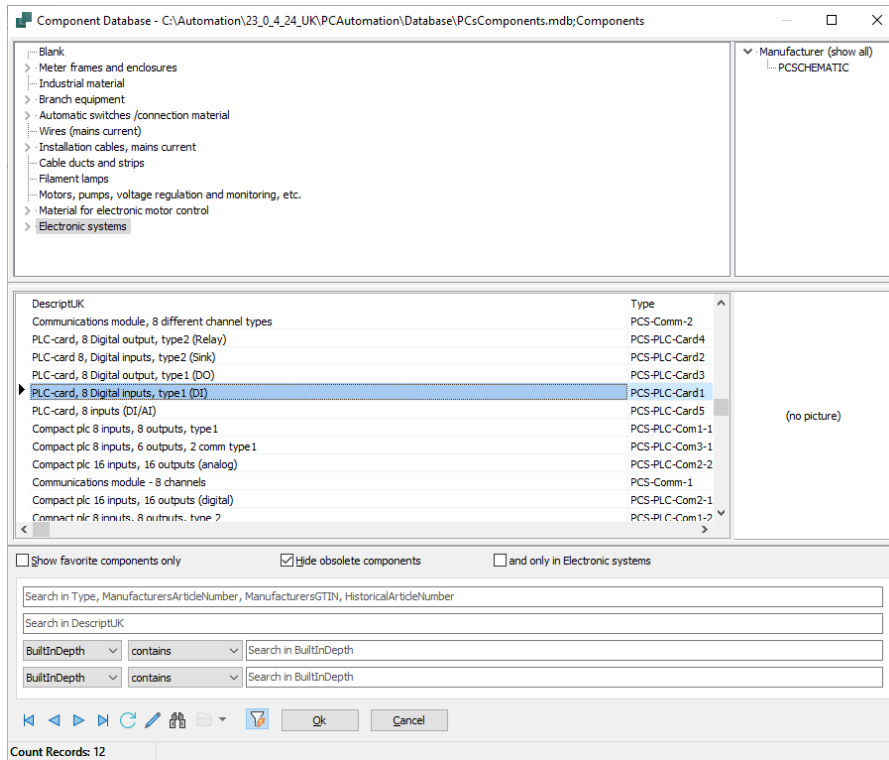


Place a PLC from the database

Go to a page in the template where you make the control current diagram.

Press the <Component Database> button and open the database.

You will find PLC's in the product group Electronics systems; select the Type PCS-PLC-Card1, which is a card with eight digital inputs.



A little about the symbols

Working with PLCs in PC|Automation uses two different symbol types, namely PLC symbols and PLC reference symbols:

- PLC symbols (IO-symbols) are symbols placed for each address to which the external components are connected.
- PLC-ref. symbols are used to show an overview of the PLC's functions and they also show where the individual inputs and outputs are located in the project.
- The PLC-ref. symbols are put together by one symbol per address and a common top and bottom.

Both symbol types show the connecting points of the PLC, i.e. the terminals to which one connects in both the project and on the board. Program wise, it is made so that there is a reference between the two, from the PLC-ref. symbol you can 'jump' directly over to the IO symbol itself and back again.

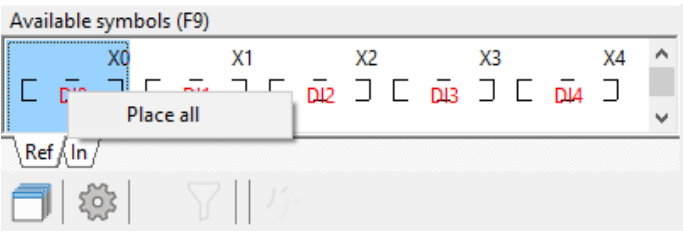
You only connects to the PLC symbol itself! At the bottom, however, there may be a connection for supply etc., which you can connect to.

A PLC can also be created with separate symbols for supply, communication, etc..

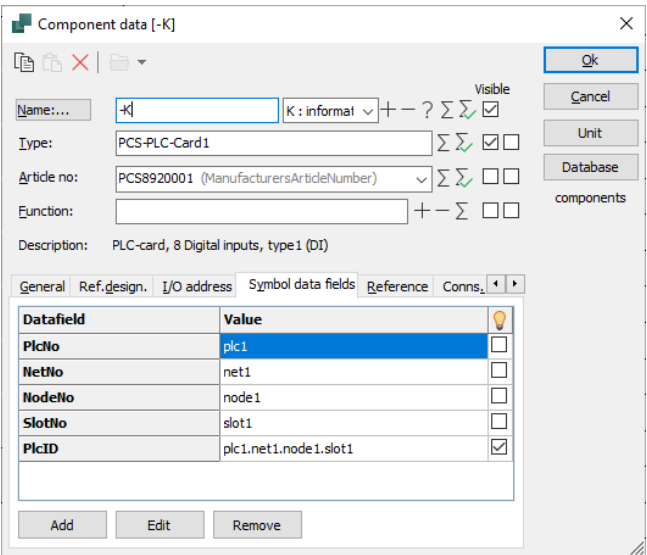
You can read more about the PLC symbols from page 77.

In the component menu, select the tab with Ref symbols, and here you right-click and select 'Place All'.

Now you get all small reference symbols grouped into one symbol, which is placed in the diagram:



1. Name it – here as -K1
2. You can enter a PLCID on the Symbol Data Fields tab, which appears on all of the component symbols. The ID is generated from the entries in the first four fields.
3. If you have entered PLCID data, it will be displayed on the ref. symbol – see below
4. Place the ref. symbol on the page



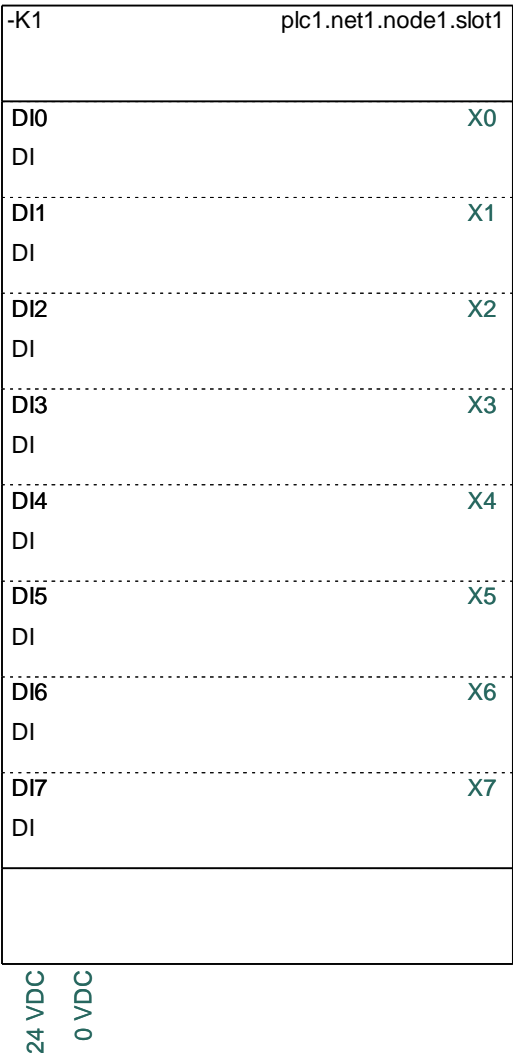
The symbol looks like this:

There is a Top which shows Name and PLCID. The top shown is one column wide.

There is one row per address, which shows the address and its connection name(s), address and input type, here Digital Input – DI.

There is a bottom which shows connections for supply. The bottom is also one column wide.

The PLC is pre-addressed; you can change this manually, and/or by loading an IO list.




Place the PLC inputs

The Component menu displays all available symbols for the project components.

For PLCs, it looks like the picture.

By tapping the gear icon,

you can choose which columns to display.

You can also 'stack' the symbols by pressing 

If the component has both inputs and outputs, they are on separate tabs. Any other symbols also have their own tab.



Symbols for component - K1 (Alt+F9)			
Name	PLC address	Connections	IO
Symbols			
PLC-REF-IN1 ✓	DI0	X0	Ref
PLC-REF-IN1 ✓	DI1	X1	Ref
PLC-REF-IN1 ✓	DI2	X2	Ref
PLC-REF-IN1 ✓	DI3	X3	Ref
PLC-REF-IN1 ✓	DI4	X4	Ref
PLC-REF-IN1 ✓	DI5	X5	Ref
PLC-REF-IN1 ✓	DI6	X6	Ref
PLC-REF-IN1 ✓	DI7	X7	Ref
PLC-IN1	DI0	X0	In
PLC-IN1	DI1	X1	In
PLC-IN1	DI2	X2	In
Available symbols (F9)			

Readdresses the PLC-card

The PLC-card is pre-addressed from the database. You can change this on the symbol's I/O addresses tab, where you type the card's home address; or by loading an IO list as described below.

Component data [-K]

Name:...

+K|

K : informat

Visible

+ - ? Σ Σ ✓

Type:

PCS-PLC-Card1

Σ Σ ✓ ✓ □

Article no:

PCS8920001 (ManufacturersArticleNumber)

Σ Σ ✓ □ □

Function:

+ - Σ □ □

Description:

PLC-card, 8 Digital inputs, type1 (DI)

General

Ref.design.

I/O address

Symbol data fields

Reference

Conns.

Inputs:

DI0

Mode:

From left/high

From left/low

From right/low

From right/high

Current order

Apply addresses

Num. System:

Oct

1

Ok

Cancel

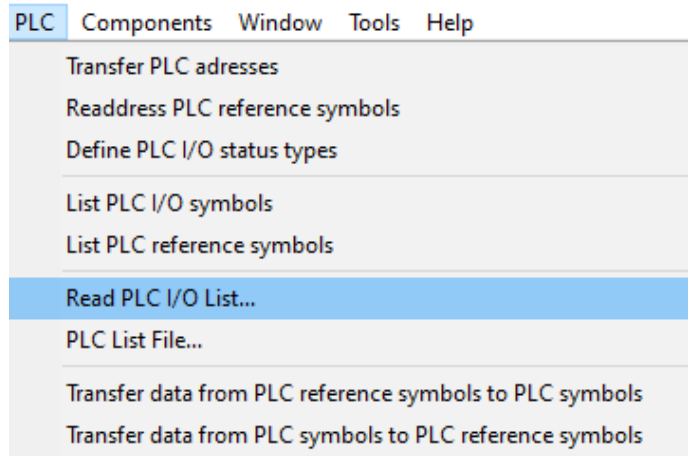
Unit

Database

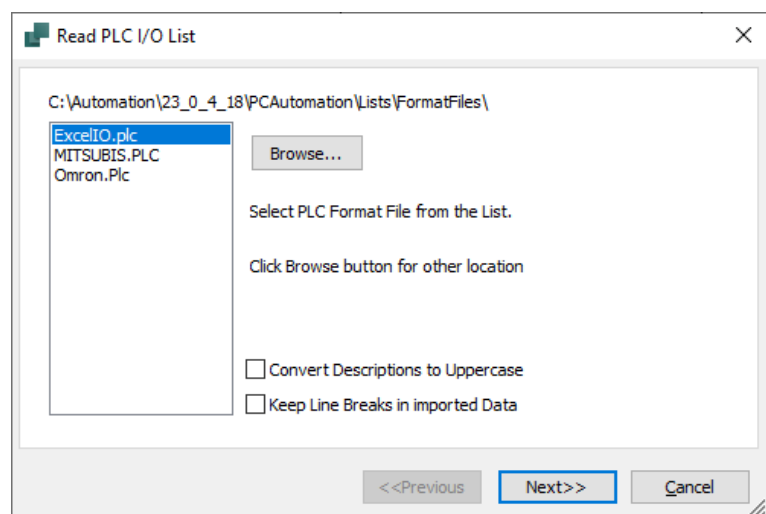
components

Read the IO list

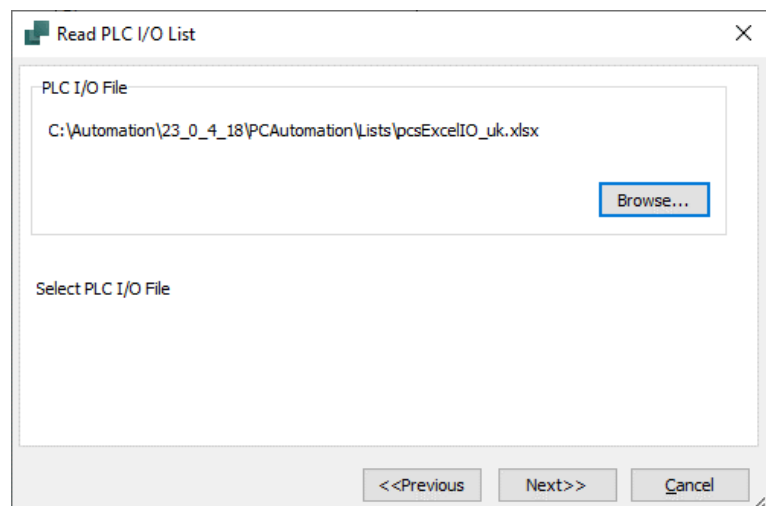
In the PLC menu, select 'Read PLC I/O List...



First, format file is selected. The format file tells you what data is in which columns of the Excel file you need to read.



You need to select the displayed list. You'll probably need to press the Browse button to find it.

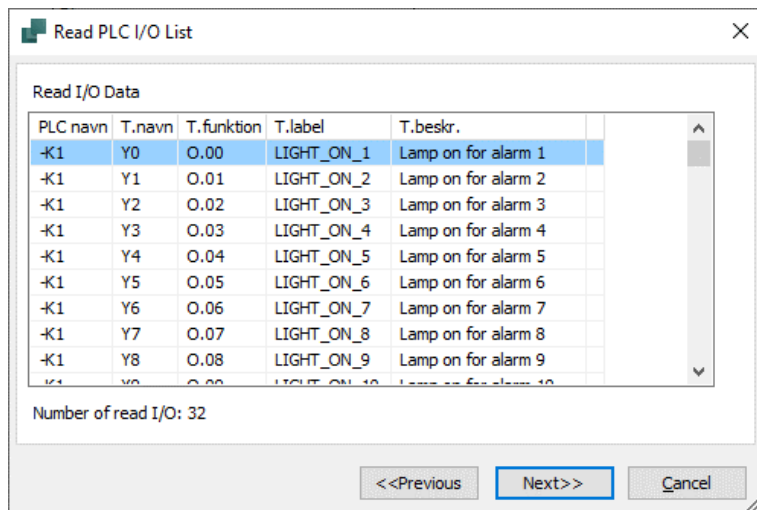


You can make your own format files to suit your PLC's and setup: under the PLC menu, you can make a 'PLC list file', and you can save the setup under your own name.

When you press Next, you can see data in the grid here.

The list MUST contain the same PLC name and t-name as in the project; the rest can be read via the list.

The PLC's connection name (T-name) contains all the data about the individual address. Therefore, in order to exchange data, the combination of PLC name and t-name must be the same in both locations..



PLC navn	T.navn	T.funktion	T.label	T.beskr.
-K1	Y0	O.00	LIGHT_ON_1	Lamp on for alarm 1
-K1	Y1	O.01	LIGHT_ON_2	Lamp on for alarm 2
-K1	Y2	O.02	LIGHT_ON_3	Lamp on for alarm 3
-K1	Y3	O.03	LIGHT_ON_4	Lamp on for alarm 4
-K1	Y4	O.04	LIGHT_ON_5	Lamp on for alarm 5
-K1	Y5	O.05	LIGHT_ON_6	Lamp on for alarm 6
-K1	Y6	O.06	LIGHT_ON_7	Lamp on for alarm 7
-K1	Y7	O.07	LIGHT_ON_8	Lamp on for alarm 8
-K1	Y8	O.08	LIGHT_ON_9	Lamp on for alarm 9
-K1	Y9	O.09	LIGHT_ON_10	Lamp on for alarm 10

Number of read I/O: 32

<<Previous Next>> Cancel

Alternatively, you can write the text for label and description directly into the symbols' fields. If so, go to the Connection Points tab.



Address	Label	Description	Input/Output
I.00	ALARM_ON_1	Alarm button activated 1	X0
I.01	ALARM_ON_2	Alarm button activated 2	X1
I.02	ALARM_ON_3	Alarm button activated 3	X2
I.03	ALARM_ON_4	Alarm button activated 4	X3
I.04	ALARM_ON_5	Alarm button activated 5	X4
I.05	ALARM_ON_6	Alarm button activated 6	X5
I.06	ALARM_ON_7	Alarm button activated 7	X6
I.07	ALARM_ON_8	Alarm button activated 8	X7

24 VDC 0 VDC

When you make electrical documentation, you make diagrams that show connections between your components – hardware.

When you make a PLC program – software – you tell which addresses perform which functions.

When reading the list, you must match the PLC program with the hardware, and therefore the PLC name and connection name must be exactly the same in diagram and IO list respectively.

In this way, you can see on the individual cards and the individual addresses what is happening.

If the PLC has multiple options

If you have a PLC where you can program it for different signal types, you can create it as shown from page 80.

When placing such a component, a grid appears where you choose how to use it in the project.

The reference symbol will look like this, and the relevant PLC symbols are in the Component Menu Accessible symbols.

	Analog Input 4-20mA	Analog Input Temp
X0,0,0.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X1,1,1.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X2,2,2.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X3,3,3.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X4,4	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X5,5	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X6,6	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X7,7	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Top	<input checked="" type="checkbox"/>	
Bottom	<input checked="" type="checkbox"/>	

If you have a card where *all* addresses are either one or the other, the grid looks like this:

	Analog Input	Digital Input
X0,X1,X2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Top	<input checked="" type="checkbox"/>	
Bottom	<input checked="" type="checkbox"/>	

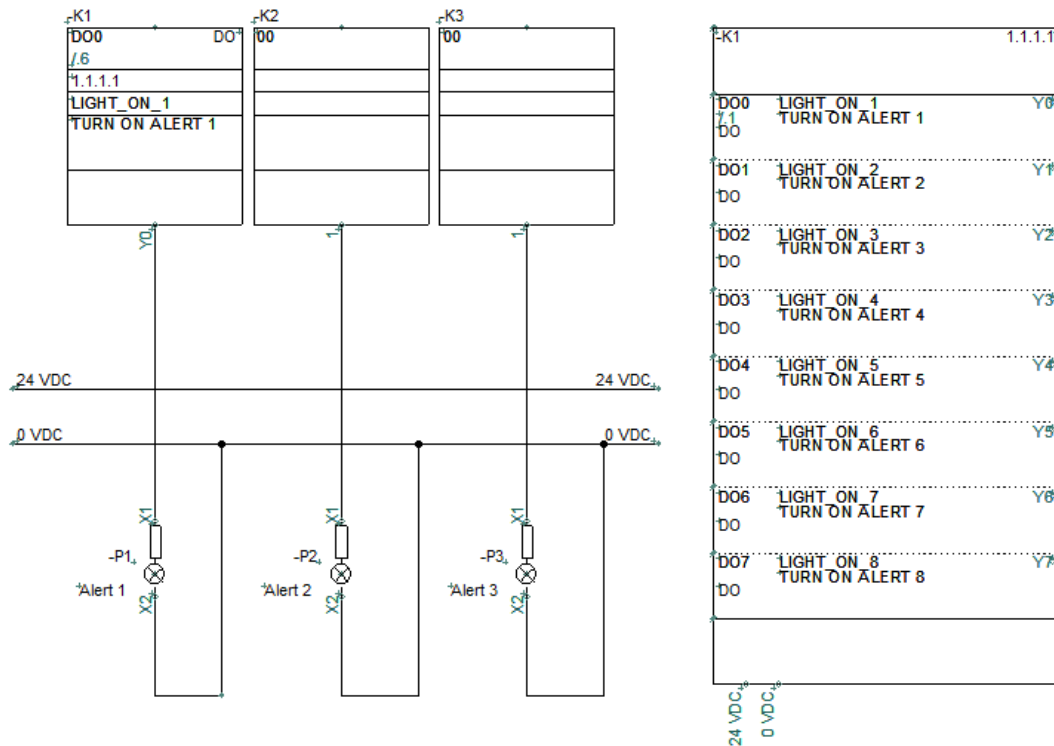
-K1				plc1.net1.node1.slot1			
AI0		X0	AO0		Y0		
AI		0+	AO	+	0.		
Temp		0-	Ampere				
<hr/>							
AI1		X1	AO1		Y1		
AI		1+	AO	+	1.		
Temp		1-	Ampere				
<hr/>							
AI2		X2	AO2		Y2		
AI		2+	AO	+	2.		
Temp		2-	Ampere				
<hr/>							
AI3		X3	AO3		Y3		
AI		3+	AO	+	3.		
Temp		3-	Ampere				
<hr/>							
AI4		X4	AO4		Y4		
AI	+	4	AO	+	4.		
4-20mA			Ampere				
<hr/>							
AI5		X5	AO5		Y5		
AI	+	5	AO	+	5.		
4-20mA			Ampere				
<hr/>							
AI6		X6	AO6		Y6		
AI	+	6	AO	+	6.		
4-20mA			Ampere				
<hr/>							
AI7		X7	AO7		Y7		
AI	+	7	AO	+	7.		
4-20mA			Ampere				
<hr/>							

24 VDC
0 VDC

IF YOU USE A PLC WITHOUT COMPONENT DATA

Often you don't know which components you're working with; at least not which item numbers are involved.

If you have made a diagram as below, you know the item number on the ref. symbol, and you want to transfer that item number to the three outputs with lamps on.



Get data from the reference symbol

Double-click one of the output symbols.

Click the I/O addr button...

Component data [-K3]

Name: -K3 K: informal + - ? Σ Σ Visible ☒

Type: PCS-PLC-Card3 Σ Σ ☒ ☐

Article no: PCS8920003 (ManufacturersArticleNumber) Σ Σ ☒ ☐

Function: + - Σ ☐ ☐

Description: PLC-card, 8 Digital output, type1 (DO)

General Ref. design. I/O address Symbol data fields Reference Conns. ▾ ▸

Quantity: 1,0 Symbol type: PLC

Scale: 1,0

Angle: 0,0

Symbol: PLC-OUT1 ☒ Visible ☒ Mechanical ☒ Electrical ☐ Name protected

☒ Include in Mechanical Load

Data source: Portal data

Component ID: 5A8F6718-CCD8-46CB-870E-86A7E8D7788A

Ok All Cancel Unit Database components State:0 I/O addr....

Select PLC

Now you enter the list of all ref. symbols Outputs in the project.

If there are multiple PLC's, you can choose the correct one from the dropdown list.

The addresses already in use are marked with a +.

Select another address by double-clicking.

Outputs:

Name	Function	Connection point	Description	Label	I/O Status type
-K3	DO0	X0			Digital
-K3	DO1	X1			Digital
-K3	DO2	X2			Digital
-K3	DO3	X3			Digital
-K3	DO4	X4			Digital
-K3	DO5	X5			Digital
-K3	DO6	X6			Digital
-K3	DO7	X7			Digital

OK
Cancel
-K3

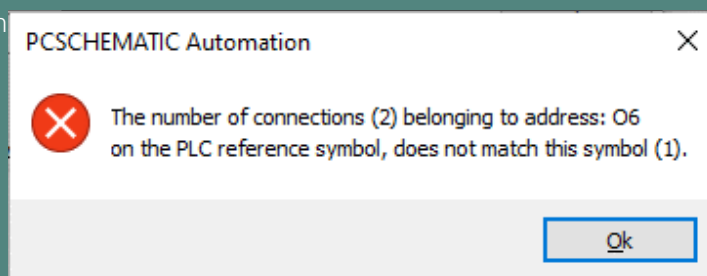
I/O
☒ All
☐ +
☐ -

It is important that the symbols in a component match:

the corresponding part of the ref. sym have two connections.

In that way, it is possible to transfer both address and other connection data for all connection points on the PLC.

If you do not choose correctly, you will have a warning.



PLC projects with Subdrawings

When you make 'real projects', you often work with modules, and in our world, modules will often be made as subdrawings.

For example, PLC inputs with different sensors and associated cables, and similar outputs that activate some equipment.

It can also be a 'function', e.g. a motor with frequency converter with associated inputs and outputs to the various signals.

Common is that in these projects you do not always know which and how big a PLC you end up with in the project.

That's why we have a feature called Transfer PLC Addresses, and it's reviewed below.

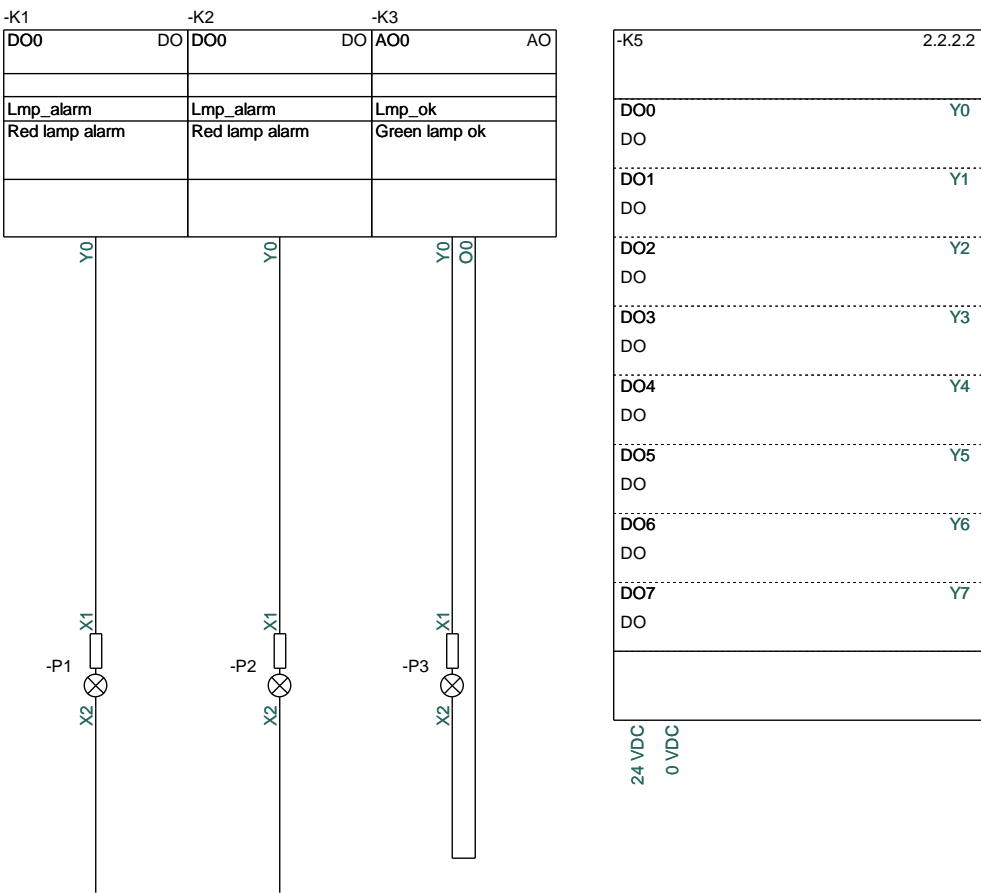
Example 1

In the SubDrawings tab, the Templates folder|Data Template have several examples of sub-drawings.

Here are placed two pcsPLCDO1 – digital output with red lamp – and a pcsPLCAO2 – analog output with green lamp.

When you have drawn the sub-drawings into the project, all PLC symbols have their own component name. In addition, connection names and addresses are the same.

The example is to match the two sub-drawings – and definitely not the third – with an output card from the database; here is selected the Type PLC-PLC-Card3.



How to do – method 1

Open the menu tab
PLC|Transfer PLC
addresses.

The function can
transfer data between
reference cards and
the PLC symbols – the
direction is shown in
the image.

In order not to send
incorrect data, the
filter is applied to the
I/O status type,
because I only want to
transfer data related to
digital addresses.

In this way, only the
two digital addresses
are chosen, and not
the analog.

Tap Execute to transfer
data.

Transfer PLC addresses, No Undo!

Select addressing method

☒ I/O Status type

☐ Address prefix

I/O Status type (5)

All

Search methode

☒ In columns

☐ In rows

Select reference designation:

Function: (0)

All

Location: (0)

All

Product: (0)

All

Result of selected method

Plc reference inputs = 56

Plc inputs = 4

Plc reference outputs = 46

Plc outputs = 5

Plc reference

Symbol

Name -> Name

Type -> Type

Article -> Article

Function | Function

Connection point

Name -> Name

I/O address -> I/O address

Label <- Label

Description <- Description

☐ Lock I/O address

Execute

How to do – method 2

Alternatively, you can
filter on Address prefix.
In this example, the
result is the same, but
it won't always be.

Transfer PLC addresses, No Undo!

Select addressing method

☐ I/O Status type

☒ Address prefix

Address prefix (6)

DO

Search methode

☒ In columns

☐ In rows

Select reference designation:

Function: (0)

All

Location: (0)

All

Product: (0)

All

Result of selected method

Plc reference inputs = 0

Plc inputs = 0

Plc reference outputs = 38

Plc outputs = 4

Plc reference

Symbol

Name -> Name

Type -> Type

Article -> Article

Function | Function

Connection point

Name -> Name

I/O address -> I/O address

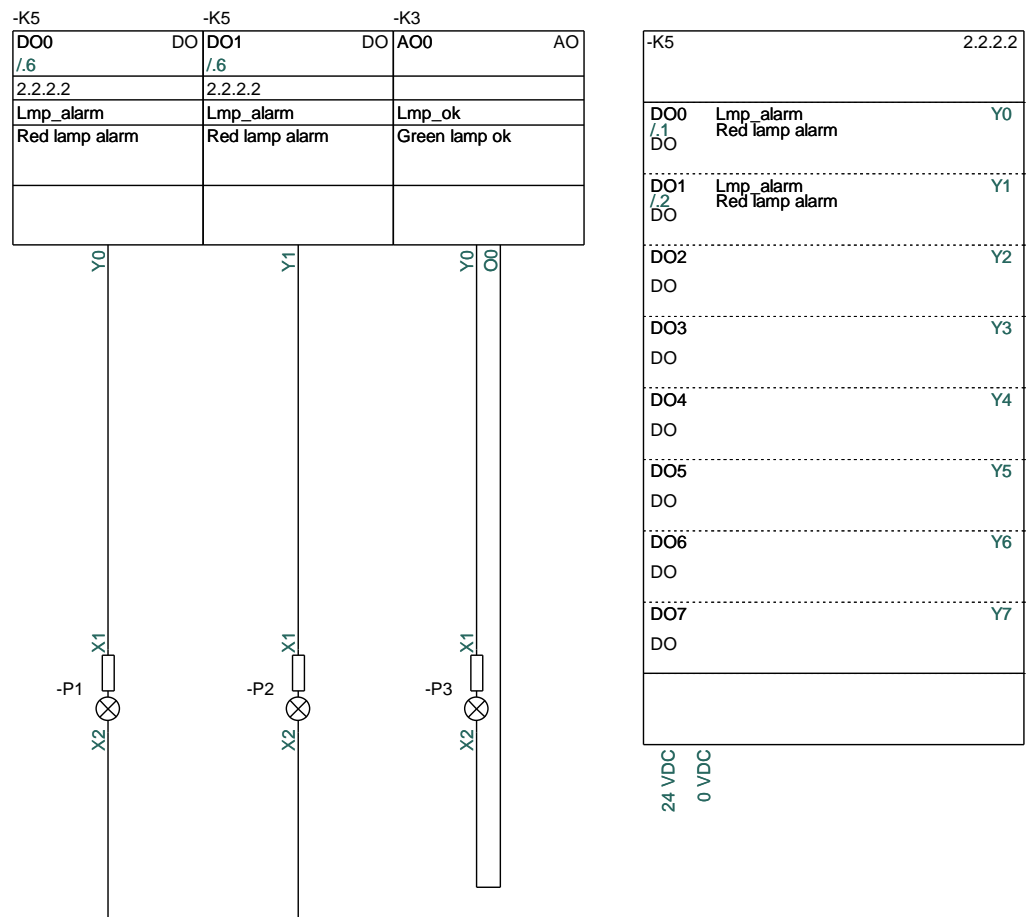
Label <- Label

Description <- Description

☐ Lock I/O address

Execute

This is how the result looks like:



Other filters

You can select the relevant PLC cards using I/O status types, address prefix and project reference designations.

In this way, you can select exactly the right addresses.

Transfer PLC addresses, No Undo!

Select addressing method

☒ I/O Status type

☐ Address prefix

I/O Status type (1)

All

Search method

☒ In columns

☐ In rows

Result of selected method

Plc reference inputs = 0

Plc inputs = 0

Plc reference outputs = 8

Plc outputs = 10

Select reference designation:

Function: (0)

All

Location: (0)

All

Product: (0)

All

Ok

Cancel

If the line with PLC-ref turns red:

There are more of the selected IO type than there are ref symbols for.

Numbers in (parentheses) refer to the number of addresses that are locked, i.e. the addresses do not change when Execute is pressed.

When you transfer addresses, you can choose to lock them so that they don't change if you expand your project.

You can only select filters that are on the reference symbols.

Readdressing reference symbols

If you have a large project, you may need to transfer many addresses to the project, and this also means that there are many reference symbols that need to be addressed.

Of course, you can go into the individual symbol and address it, but if there are many of the same kind, these can be addressed at once..

Readdress PLC reference symbols, No Undo!

Select addressing method:
☒ I/O Status type
☐ Address prefix

I/O Status type (5)
All

Select reference designation:
Function: (0)
All
Location: (0)
All
Product: (0)
All

Search method:
☒ In columns
☐ In rows

Result of selected method
Plc reference inputs = 56
Plc reference outputs = 0

Address start: 0.00
Num. System: Oct
Select component: All

☐ Restart address at new symbol

Start address for:
☒ Inputs:
☐ Outputs:

I/O Status type	Count Value
All	1

Execute

Dialogue PLC|Readdress PLC reference symbols is made for this purpose.

You can address the cards using the same filters as described on the previous pages, e.g. I/O status types, Address prefix and reference designations.

You can transfer the following information to one or more components:

- Address Start
- Number System
- Count Value

A LITTLE MORE INFO ABOUT THE PLC SYMBOLS

The PLC symbols found in the PC|Automation from version 23 works in the same way as all previous versions, but the design has been tightened up.

Below we review their features; both graphically and functionally.

The graphical appearance of the symbols

The IO symbols themselves and the other symbols in the 'field' have a uniform appearance:

-K1	-K1	-K1	-K1	-K1
.00 I.7	I.01 I.7	O.02 I.7	O I.7	
Plc1.Net2.Node3.Slot4	Plc1.Net2.Node3.Slot4	Plc1.Net2.Node3.Slot4	Plc1.Net2.Node3.Slot4	Plc1.Net2.Node3.Slot4
Label - Input	Label - Output	Label		
Description. Standard "I" for Input	Output. Signal type is also visible	Output with 4 connections	Symbol for communication on the PLC	Symbol for supply
	24 VDC	0 VDC 24 VDC PE		
1	2 3	4 5 6 7	8	9

- 1 current path in width (40 mm)
- All connections point downwards (that means they are inside the board)
- New symbol data field – PLCID (composed of PLC.Net.Node.Slot) – on all symbols
- Space for label and description
- Space for description on connections
- All texts in Arial, 2.5 mm

The reference symbols are also similar:

- 2 current paths wide (80 mm), i.e. there can be four paths on one side
- 15/30 mm high per address
- A common top and bottom, is automatically grouped together with the symbols per address.
- All texts in Arial, 2.5 mm

-K1	Plc 1.Net2.Node3.Slot4	
.00 I.2 I	Label - Input Description. Standard "I" for Input	1
.01 I.3 O	Label - Output Output. Signal type is also visible	2 3
.02 I.4 O	Label Output with 4 connections	4 5 6 7
CH0 I.5	Symbol for communication on the PLC	8

REMEMBER

For all symbols, the first connection is the one that has PLC status. And that this pin must be connected in the diagram to be shown on the PLC list and the IO list.

Overview of the symbols – Inputs and Outputs

14 sets of symbols have been created for inputs and outputs:

- PLC-In1 .. PLC-In7 – symbols with 1 to 7 pins for PLC-inputs
- PLC-Ref-In1 .. PLC-Ref-In7 – symbols with 1 to 7 pins for PLC-reference symbols for inputs
- PLC-Out1 .. PLC-Out7 – symbols with 1 to 7 pins for PLC-outputs
- PLC-Ref-Out1 .. PLC-Ref-Out7 – symbols with 1 to 7 pins for PLC-reference symbols for outputs

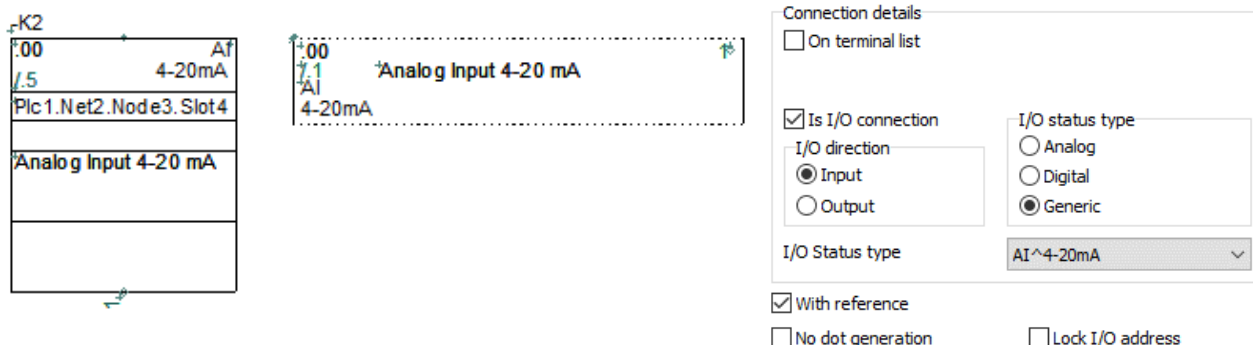
These symbols match in pairs: PLC-ref-In1 matches PLC-In1, etc.

There is Input status and With reference on the first pin of all the symbols.

All symbols have a new data field for PLC.Net.Node.Slot, which shows where the PLC fits into the PLC network.

All symbols have a new data field – PLCSignalType – which can display the selected signal type. By default, I or O is displayed when nothing is selected on the connection. If e.g. Digital Input is selected, it will be displayed in the field as DI.

A list of different signal types has been created, which you can choose from if you want to specialize further, e.g. AI 4-20 mA. You can create your own signal types in the list, and these can also be used when creating components in the database.



Overview of the symbols – PLC top and bottom symbols

Four symbols have been made for the top and bottom, where they can cover 1-4 columns.

The top shows the Component Name, and the PLC's ID in the form of PLC.Net.Node.Slot.

The bottom contains 10 connection points that can be used for e.g. supply. The bottom can also just be a nice graphic finish.

If you want to make your own PLC symbols, the easiest thing to do would be to start by researching ours, and then make your own in the same way.

Remember that the first must be PLC-pin and that all other pins must have the same value in the function text in order to work at the same address.

Overview of the symbols – PLC Comm symbols

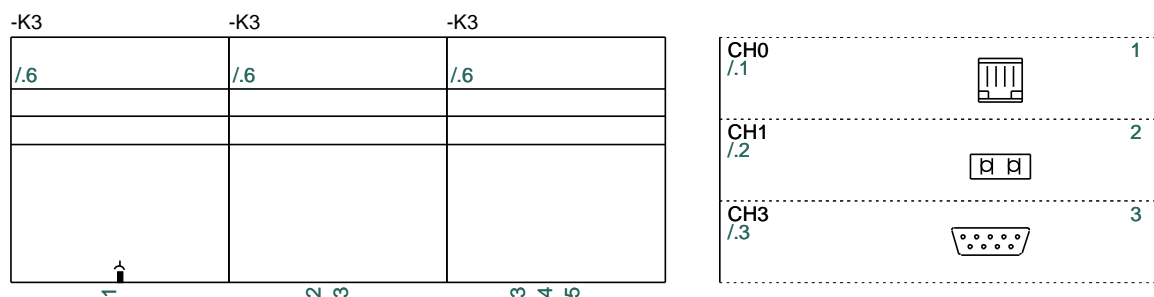
There is a set of symbols for use in PLC communication:

- PLC-Comm1 .. PLC-Comm7 – symbols with 1 to 7 pins for connections to PLC communication
- PLC-Ref-Comm – reference symbols (with states) with 1 pin, which fits all PLC-Comm symbols (previous IO-link)

All PLC-Comm symbols refer to PLC-Ref-Comm. First pin on PLC-CommX symbols has With reference.

The PLC-Comm symbols are a simplified version of the PLC-In/Out.

PLC-Ref-Comm has states so that you can display the current type of connector/connection on the ref symbol. Currently, it has six states, but we can expand that if the need arises. And at that time, we will send this symbol from the Portal.



Overview of the symbols – PLC Aux symbols

A set of Aux symbols in the same series has been made, which can be used for e.g. supply:

- PLC-Aux1 .. PLC-Aux7 – symbols with 1 to 7 pins for PLC aux connections, e.g. power supply.

These symbols have no reference card, and therefore no pin references.

They are simplified symbols and are simply made to have a uniformity for all PLC functions.

Alternatively, you can use the connections found in the PLC-Bottom symbols.

The PLC components of the Portal are created with these symbols.

CREATE PLC COMPONENT IN THE DATABASE

You may need to create your own components in the database. The Component Wizard is made as a tool for this.

Below is a review of how to create a PLC.

See the next section for more examples of how to create components in your database.

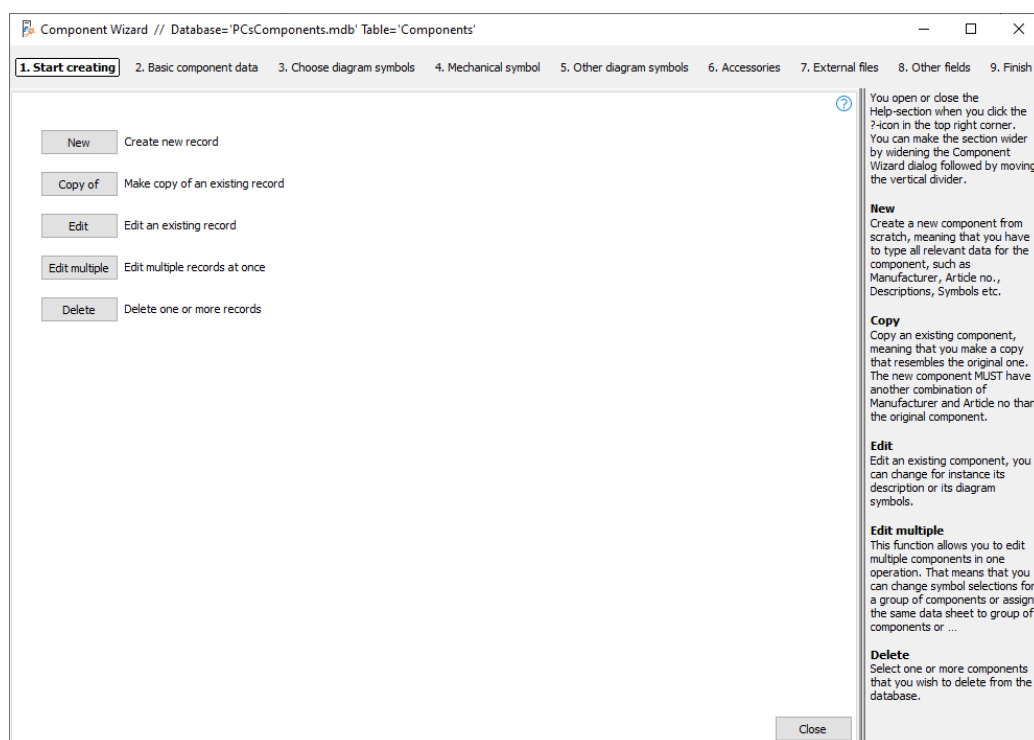
Start the Component Wizard

You will find the Component Wizard in the tool bar.

The wizard is connected to the database you are currently working in; you can see which one in the title bar.

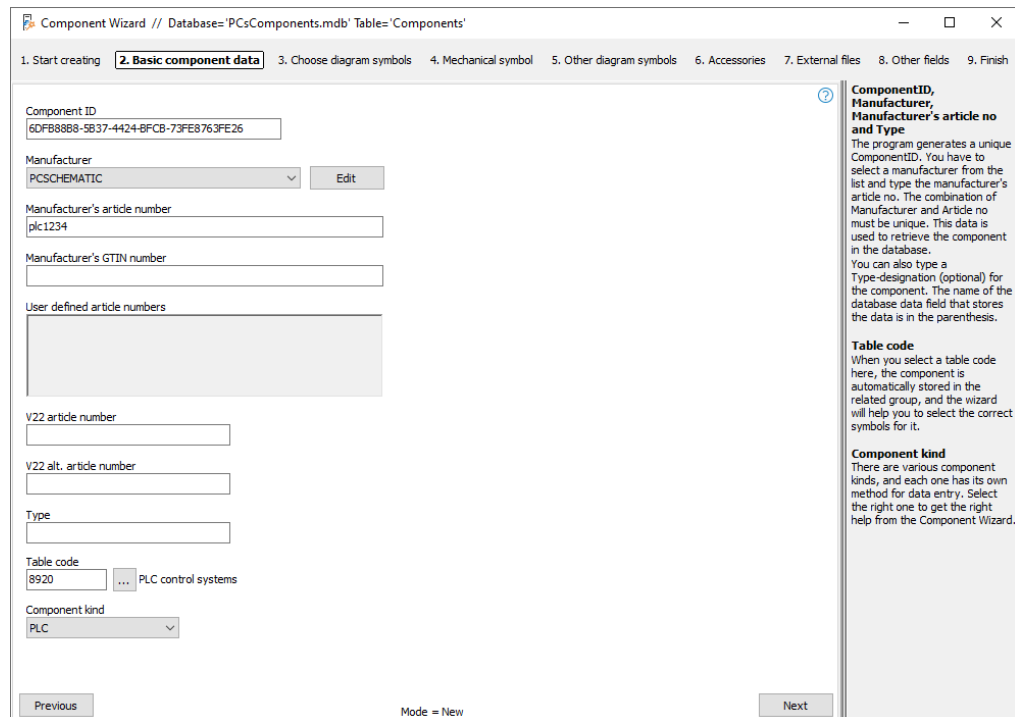
The wizard will take you through all the necessary steps related to creating different types of components. Below you can see how to create a PLC.

Press the <New> button to create a new component.



Manufacturer and Manufacturer's Article number

You start by selecting Manufacturer and Manufacturer Article number. If the manufacturer is not on the list, you can edit the list. The other data is optional. To create a PLC, select PLC as the Component kind.



The screenshot shows the 'Component Wizard' window with the 'Basic component data' tab selected. The window title is 'Component Wizard // Database='PCsComponents.mdb' Table='Components''. The progress bar at the top shows steps from 1 to 9, with step 2 being the current tab.

Fields and values:

- Component ID: 60FB8888-5B37-4424-8FCB-73FE8763FE26
- Manufacturer: PCSCHMATIC (with an 'Edit' button)
- Manufacturer's article number: plc1234
- Manufacturer's GTIN number: (empty)
- User defined article numbers: (empty list)
- V22 article number: (empty)
- V22 alt. article number: (empty)
- Type: (empty)
- Table code: 8920 (with a dropdown arrow and 'PLC control systems' text)
- Component kind: PLC (with a dropdown arrow)

Buttons: 'Previous' and 'Next' are at the bottom. 'Mode = New' is displayed in the center.

ComponentID, Manufacturer, Manufacturer's article no and Type
The program generates a unique ComponentID. You have to select a manufacturer from the list and type the manufacturer's article no. The combination of Manufacturer and Article no must be unique. This data is used to retrieve the component in the database.
You can also type a Type-designation (optional) for the component. The name of the database data field that stores the data is in the parenthesis.

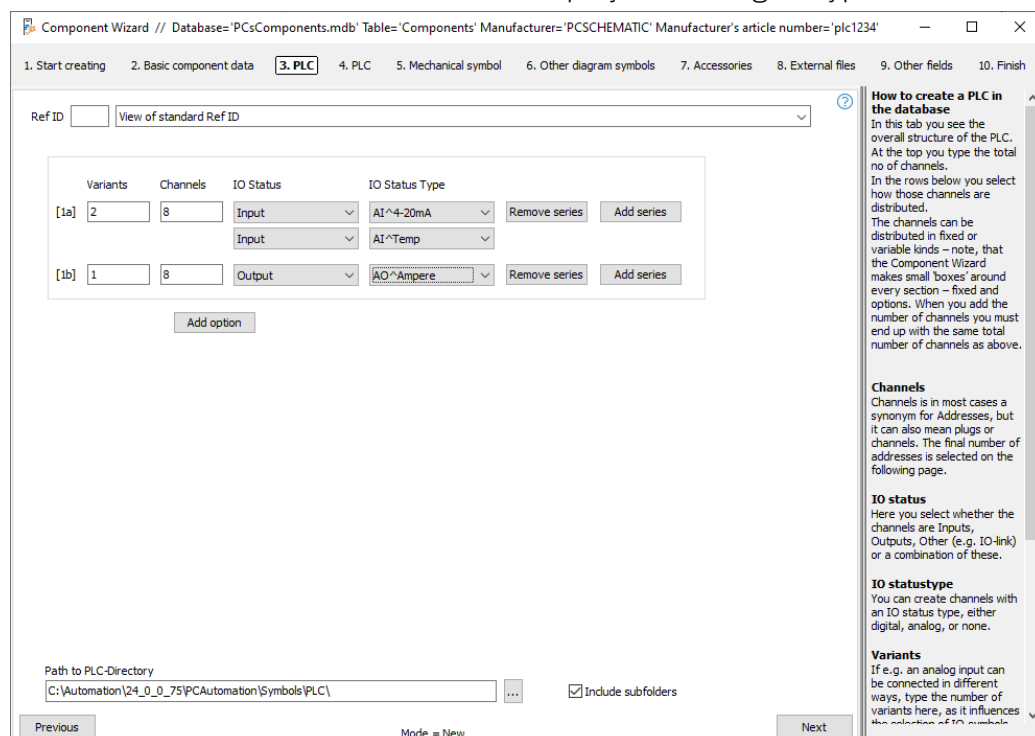
Table code
When you select a table code here, the component is automatically stored in the related group, and the wizard will help you to select the correct symbols for it.

Component kind
There are various component kinds, and each one has its own method for data entry. Select the right one to get the right help from the Component Wizard.

What can your PLC do

To get the right options i.e. symbols, when you put the PLC into your project, you need to create it with the right features. And you do that on this tab.

Here is selected a PLC with analog outputs and inputs, which can be either 2- or 3-wire. The different addresses are also displayed with signal type.



The screenshot shows the 'Component Wizard' window with the 'PLC' tab selected. The window title is 'Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article numbers='plc1234''. The progress bar at the top shows steps from 1 to 10, with step 3 being the current tab.

Fields and values:

- Ref ID: (empty) (with a dropdown arrow and 'View of standard Ref ID' text)

Table with 4 columns: Variants, Channels, IO Status, IO Status Type

Variants	Channels	IO Status	IO Status Type		
[1a]	2	8	Input	AI^4-20mA	Remove series Add series
			Input	AI^Temp	
[1b]	1	8	Output	AO^Ampere	Remove series Add series

Buttons: 'Add option' is below the table.

Path to PLC-Directory: C:\Automation\24_0_75\PCAutomation\Symbols\PLC\ (with a dropdown arrow)

☒ Include subfolders

Buttons: 'Previous' and 'Next' are at the bottom. 'Mode = New' is displayed in the center.

How to create a PLC in the database
In this tab you see the overall structure of the PLC. At the top you type the total no of channels.
In the rows below you select how those channels are distributed.
The channels can be distributed in fixed or variable kinds – note, that the Component Wizard makes small 'boxes' around every section – fixed and options. When you add the number of channels you must end up with the same total number of channels as above.

Channels
Channels is in most cases a synonym for Addresses, but it can also mean plugs or channels. The final number of addresses is selected on the following page.

IO status
Here you select whether the channels are Inputs, Outputs, Other (e.g. IO-link) or a combination of these.

IO status type
You can create channels with an IO status type, either digital, analog, or none.

Variants
If e.g. an analog input can be connected in different ways, type the number of variants here, as it influences the selection of IO symbols.

The right connection names

On the next tab, for each type, there is a sub-tab for PLC data and reference symbols.

On each subtab, you can select:

- Addresses per channel. Select 1
- Number of connections per address – here is selected 4-20 mA, which is a 2-wire signal; Therefore, 2 connections are selected per address
- IO-symbol(s) appears
- In the grid, all the connection names of the component are displayed if all inputs are connected as 4-20 mA. Enter the data of the first address and point your mouse over <Fill grid> and press the button if it is ok.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='plc1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Option 1 Other symbols

[1a] var1 - 8 Input AI^4-20mA [1a] var2 - 8 Input AI^Temp [1b] - 8 Output AO^Ampere

PLC Data Reference symbols

Addr. pr. channel: 1 Terminals pr. addr.: 2 Address number system: OCT

Fill grid Clear grid Subname

Fill the grid automatically

Symbol (1/1)

Channel	IO	Address	Term2	Func2
1	X0	AI0	0	+
2	X1	AI1	1	+
3	X2	AI2	2	+
4	X3	AI3	3	+
5	X4	AI4	4	+
6	X5	AI5	5	+
7	X6	AI6	6	+
8	X7	AI7	7	+

Symbol: PLC-IN2 Title: IO input - 2 connections per address State: 0

Diagram symbols for the PLC

In these tabs - IO data and Reference symbols - you select the PLC symbols. You have one tab for each part that you specified on the previous tab, so that you can find each part of the PLC and a common tab with other symbols. Each PLC-tab consists of two sub-tabs, one for IO-data and one for ref.symbols.

PLC Data

Here you type the number of addresses per channel (default = 1). When you have more than one address per channel, Subname is automatically applied to differentiate between channels. The number of connections or address must be typed and the Component Wizard uses this to find the right symbols. Type connection name and default address for the IO-terminal, click the Fill grid button and the grid is automatically filled; sometimes you might need to type in the 1st and the 2nd row.

Reference symbol

When you press the Add symbol button, the Component Wizard shows the ref.symbols that match your selected IO-symbols. They will match according to total number of

Previous Mode = New Next

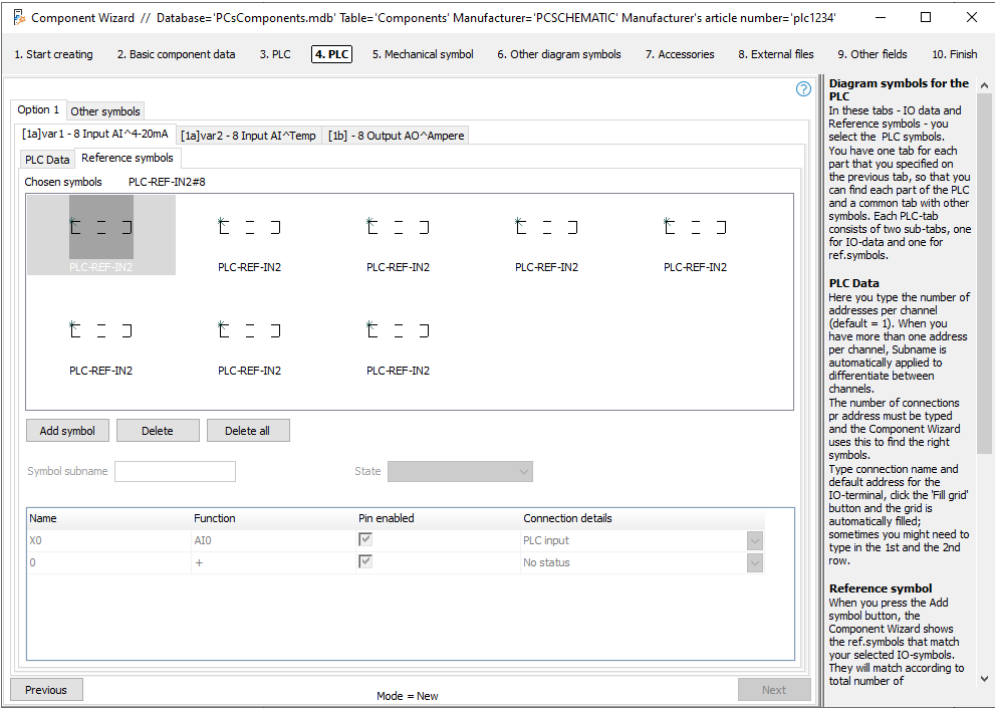
The first connection point is the point that contains all PLC data. This means that the IO-pin MUST be connected when the address in question is in the project. Otherwise, there will be no data on either the PLC list or the IO list.

There is a 'set' of PLC data and reference symbols for each option on the PLC.

You can only move forward with the <Next> button once all tabs are filled in.

Reference symbols

When the PLCs are created in the database, there must be reference symbols. They are selected on this tab. You are only presented with the symbol(s) that match the choice from the PLC-Data tab, and all data also comes from here.

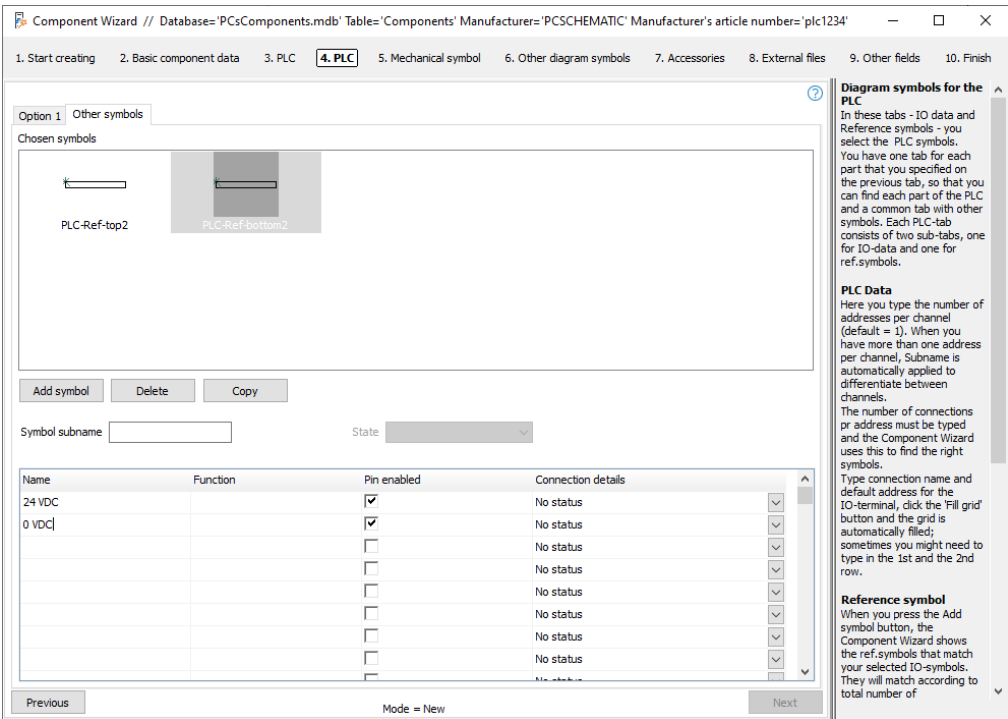


Other symbols

On this tab, select Top and Bottom and other symbols.

You can – like here – enter connection points for supply on the bottom symbol, or you can choose to insert an extra symbol – it is your choice.

The number of pairs determines how many symbols you get for diagram.



COMPONENT WIZARD: CREATE YOUR OWN COMPONENTS IN THE DATABASE

This chapter shows you how to use the Component Wizard to create components in PC|Automation database. The Component Wizard is a part of the PC|Automation from version 16, and new options are continuously added to it. This booklet is prepared in version 23.

The guide has various functions, it can, among other things:

- Create an entire new component, including special component types
- Create a new component as a copy of an existing one
- Edit an existing component
- Edit multiple components

The different options are reviewed below.

A few definitions ...

To use the Component Wizard, you must have a database associated with it.

COMPONENTS are in the database and always have a unique ID, which in our database is a combination of the Manufacturer and its Article number (ManufacturersArticleNumber).

COMPONENTS can consist of one or more SYMBOLS – and a lot of other info.

Most of the information about setting up the database is fixed; Read more from page 119. However, you also have the option of adding your own data and data fields in the database, which you can read more about in the Database Manual.

The Component Wizard allows you to easily enter data about the components so that they will work in your projects. How to do this, you can read much more about in this manual.

REMEMBER that ...

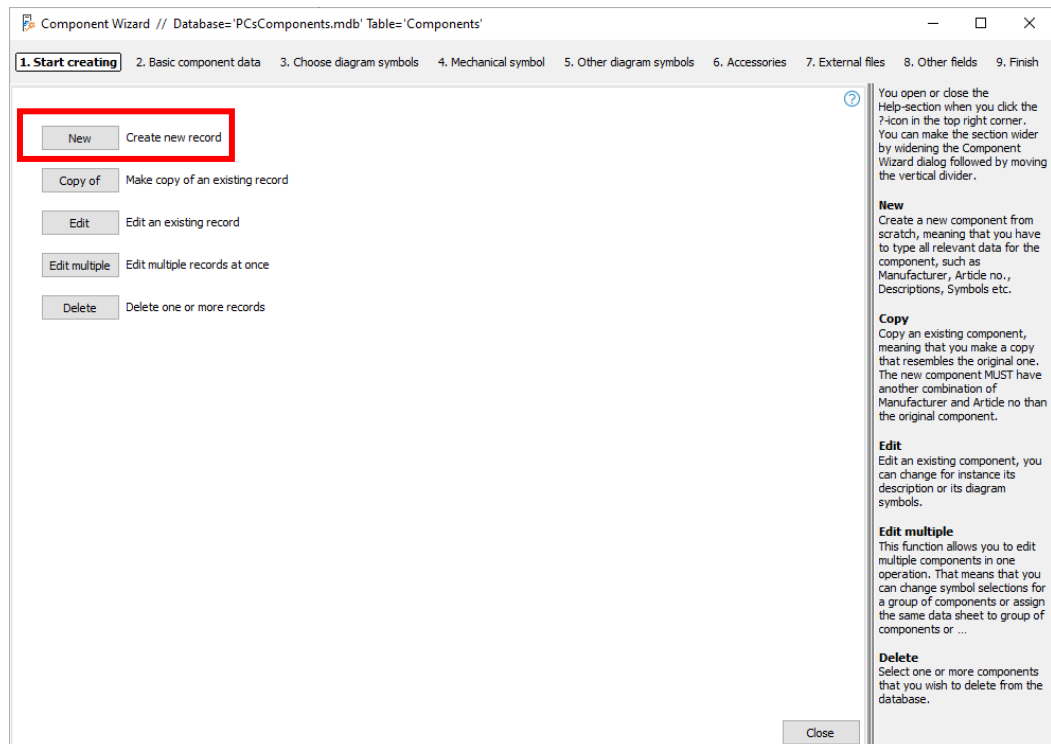
It is always a good idea to make your own database, preferably as a copy of ours.

In this way, you also make it easier to maintain your data, which you can also use the Component Wizard for.

The component wizard uses your database settings. On page 123, you can see default settings for the database – with a few additions. If your Component Wizard looks different, it's because your setup is different119.

OPEN THE COMPONENT WIZARD

Open the Component Wizard using the icon in the top tool bar or from the Components menu|Component Wizard or shortcut [W].



In the top bar, you can see which database and table you have access to. This is important if you work in different databases.

In the next line, you can see the steps you go through to create a component.

In the middle of the image you can see your options with the wizard i.e. Create New, Copy, Edit, etc..

On the far right, there's a help text for each step you go through. You can close the help text by tapping the icon in the upper right corner.

The Wizard can always be closed at the intersection in the upper right corner.

When you create a component, your selections are stored in the usr fields. Read more about the database fields from page 119 and in the database manual.

Create a new component – Normal

When you press the New button, you enter this window, where you start by entering various basic data on the component:

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files 8. Other fields 9. Finish

Component ID
BE157EBF-943E-41CF-8E58-98D7B6718ECF

Manufacturer
PCSCHEMATIC Edit

Manufacturer's article number
Test1234

Manufacturer's GTIN number

User defined article numbers

V22 article number

V22 alt. article number

Type
Test1234

Table code
5000 ... Filament lamps

Component kind
Normal

Previous Mode = New Next

ComponentID, Manufacturer, Manufacturer's article no and Type
The program generates a unique ComponentID. You have to select a manufacturer from the list and type the manufacturer's article no. The combination of Manufacturer and Article no must be unique. This data is used to retrieve the component in the database.
You can also type a Type-designation (optional) for the component. The name of the database data field that stores the data is in the parenthesis.

Table code
When you select a table code here, the component is automatically stored in the related group, and the wizard will help you to select the correct symbols for it.

Component kind
There are various component kinds, and each one has its own method for data entry. Select the right one to get the right help from the Component Wizard.

ComponentID

The unique ID is created automatically. That should not be changed here!

Manufacturer and Manufacturer's article number

All components MUST be created with a Manufacturer and the Manufacturer's article number. The combination of the two fields must be unique.

Manufacturer's GTIN number

If you know the manufacturer's GTIN number (barcode, former EANnr) you can enter this here.

User-selected article numbers

If you have your own article numbers, e.g. in Navision, you can create field(s) for these in the database and have them entered here (see how to do in the database manual).

V22 article numbers

Components you also have in version 22 have their old article number (most often EANnr) here. A possible 'V22 alt. article number' gives a number in this field.

Type

The component may also have a Type that you can enter here.

Table code

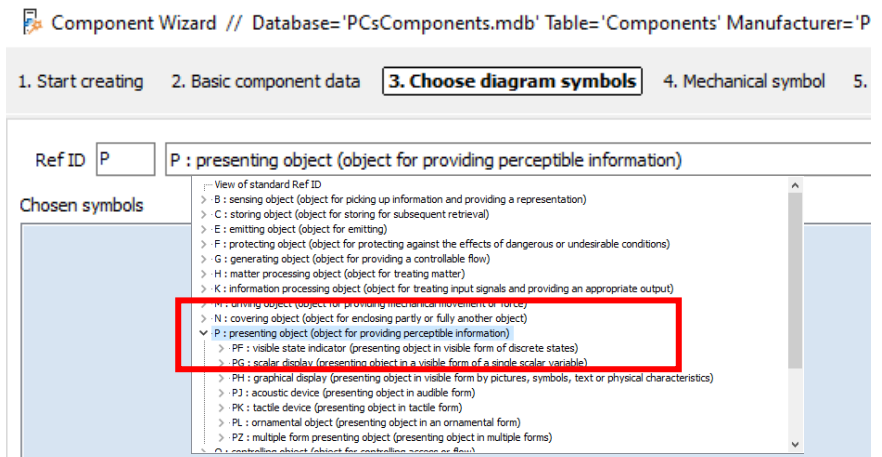
It is convenient that components of the same kind are in the same group, so you can easily find them in the database. The item group refers to the menu table/ OrganizationTree that is used. By default, it is OrganizationTree1 that corresponds to the VELTEK-menu.

Component kind

The Component Wizard behaves differently depending on the selected component kind. In this first section, you'll see the basics of creating new components. Later you can see examples of creating other component kinds.

Choose diagram symbols

In this window, start by selecting the reference ID; It is optional but recommended. The letter code in the list follows the standard ISO IEC 81346-2. You can choose 1-, 2-, or 3-letter code.



After that, select electrical symbols for the component. You can do this in two ways, which are shown below.

Get symbols using database lookups

From the start, the wizard has a check mark in the box above the Add symbol button. This is an easy way to retrieve symbols, as you often need the same symbols for components in the same article group.



Example 1: Component with one symbol

When you press the button, the symbols that are already in the article group are displayed.

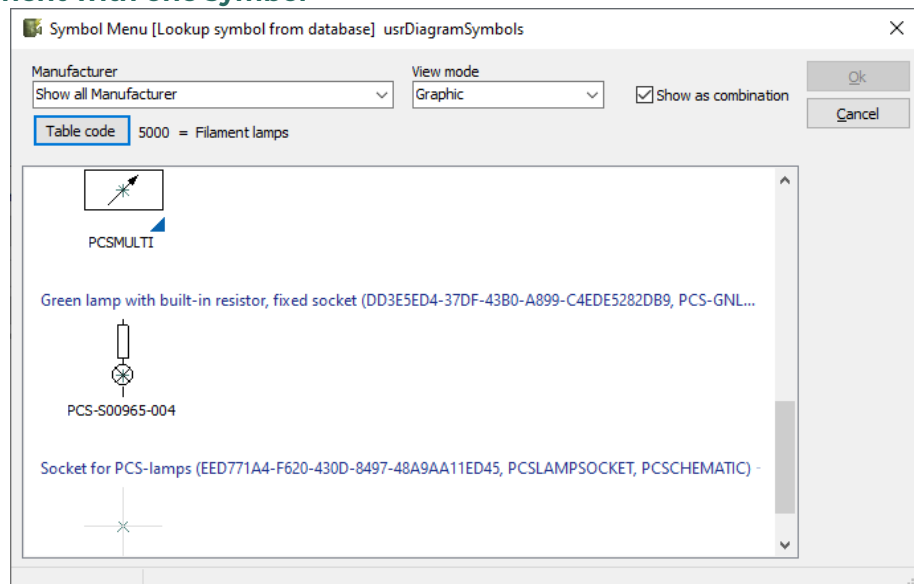
You can select a specific manufacturer or all.

Here there are multiple options:

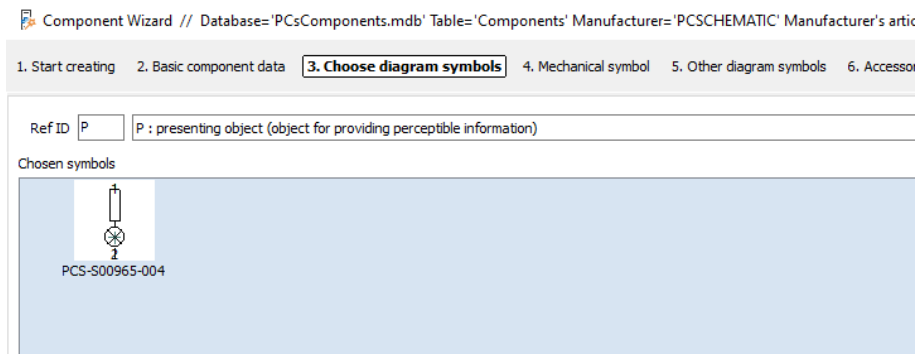
If you want to

create a

component with the same electrical function as the white lamp, select the lamp symbol and double-click on it.

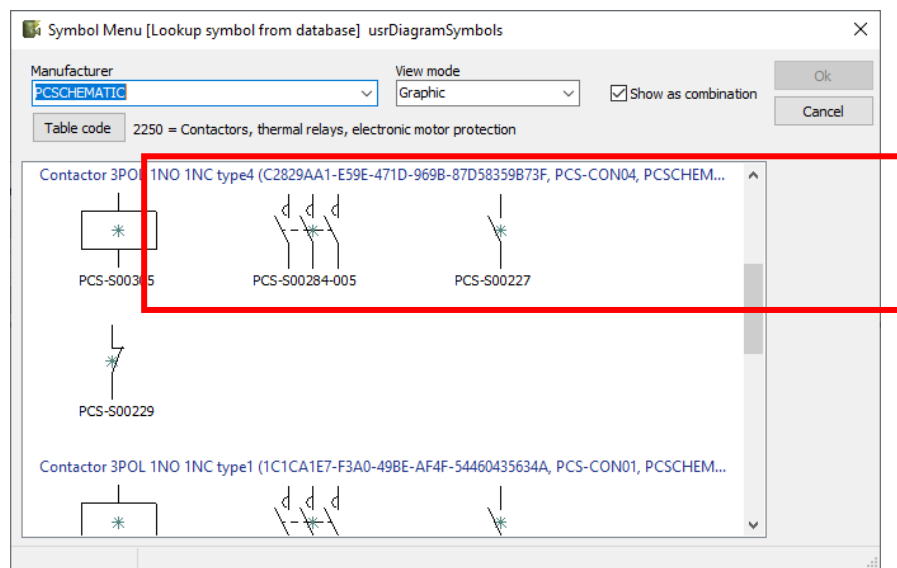


When you select the symbol in this way, you get it including connection names into your window with chosen symbols.

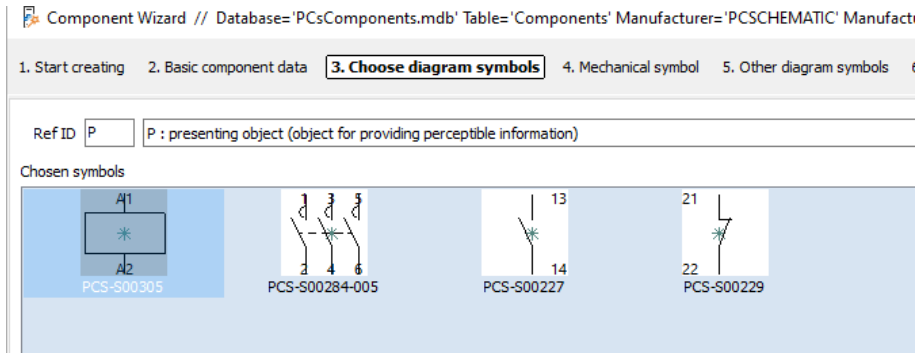


Example 2: Component with more symbols

If you instead want to create a component with multiple electrical symbols, such as a contactor, your windows will look like this:



When you choose one of these symbol sets, you also get everything: However, you must choose refID yourself, but the rest, i.e. all symbols with connection names and symbol properties in general, are included.

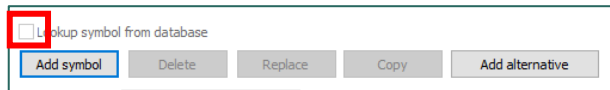


If this is ok, click next to proceed to the selection of mechanical and other symbols.

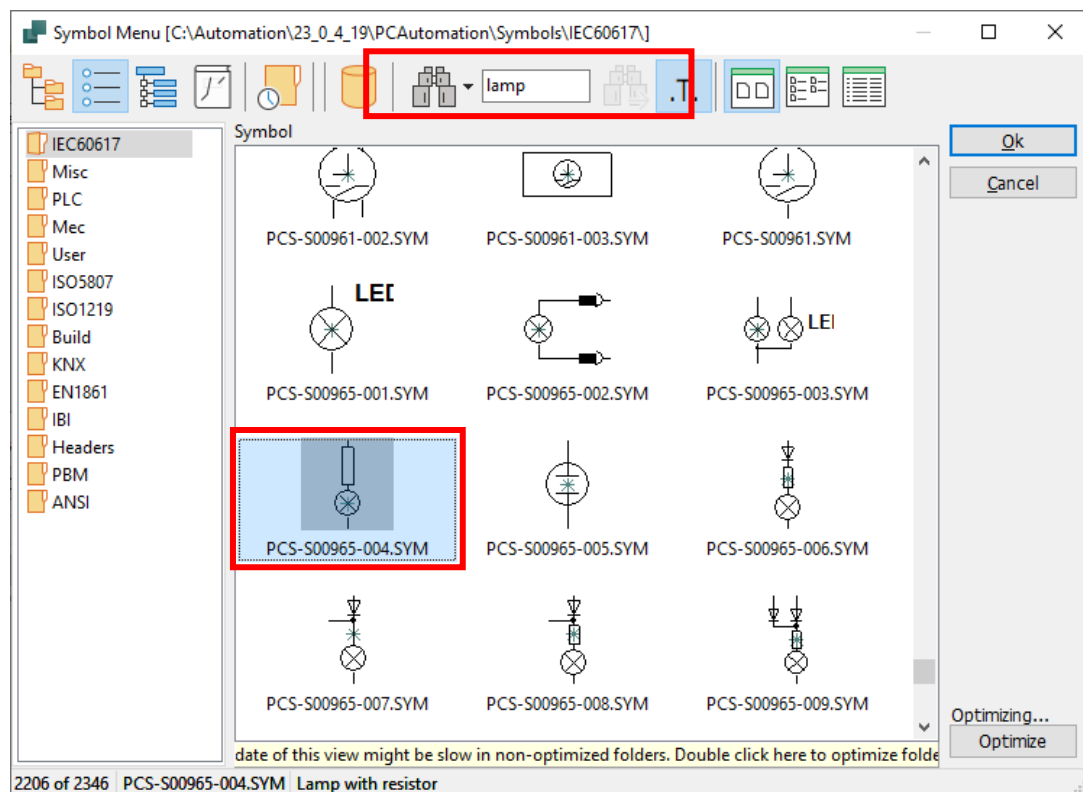
Get symbols by going to the Symbol Menu

If you don't have a component of the same kind that helps you with symbol selection, you can download a symbol from your symbol library instead.

Turn off the check mark and tap the Add Symbol button.

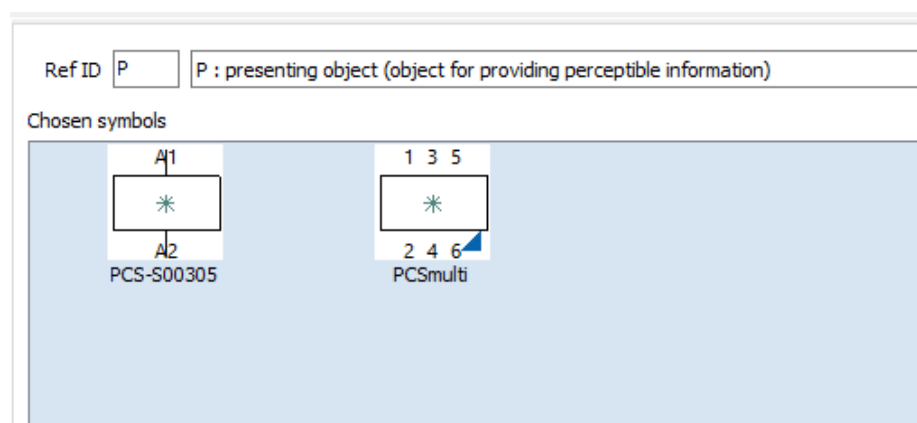


You enter your symbol library, and – if you are in the process of making a lamp as in the example – you will find the lamp symbol in the IEC60617 folder.



Once you have selected the symbol, you can see it in the window. Here is also selected the symbol PCSmulti from the Misc folder.

You can select the individual symbol and delete, replace and copy it, or choose an alternative symbol, which you can read about later.



Symbol settings

Component Wizard // Database='PCComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data **3. Choose diagram symbols** 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files 8. Other fields 9. Finish

Ref ID S S : manual interacting object (object for detecting a manual action and providing an appropriate response)

Chosen symbols

PCSMulti PCS-S00965-004

☐ Lookup symbol from database

Add symbol Delete Replace Copy Add alternative

Symbol subtype s Symboltype Normal

State 3 : VAR

Connection(s) on: 1 : DC 2 : AC 3 : VAR

Name	Pin enabled	Connection details
	<input type="checkbox"/>	No status
	<input type="checkbox"/>	No status
A	<input checked="" type="checkbox"/>	On list
B	<input checked="" type="checkbox"/>	No status
	<input type="checkbox"/>	No status
	<input type="checkbox"/>	No status

Previous Mode = New Next

Choose diagram symbols
In this tab you select the symbols that represent the component in the electrical connection diagrams. You have the following options on the tab:

RefID
Letter codes for component names, according to the standard ISO/IEC 81346-2, table 2. Any value in this datafield will overrule the symbols' default values.

Add symbol
When the 'Lookup symbol from database' is active, the Component Wizard will suggest symbol set(s) from components with the same table code. If it is inactive, you will have to select your own symbols from the Symbol menu.

You can delete a symbol, replace it with another one, copy it, and drag it to a new position.

For each individual symbol, you can select extra functionality: subtype, state, change symbol type and reference settings.

For all symbols, you can change connection names, optionally delete them, and you can select to have the connections on the terminal list by default.

If a function can be illustrated in another way – as an alternative – you can create alternatives. This means that a special symbol

Once the symbol is selected, you have the following options, which depend on the selected symbol(s). In general, you have the same options for the individual symbol as when you create a new symbol.

In addition, there are these options:

- You can give the symbol a subtype: For example, you want to make a print with a lamp. The entire component must be called S (the primary function of the component), the contact part gets the subtype S, the lamp part gets the subtype P.
- If you have selected a symbol with states, you can select one of the modes for the current component.
- You can change the type of symbol (rarely a good idea).
- You can choose reference – with reference or main reference.
- You can change the connection name, possibly 'turn it off', and see the result in the window above.
- You can choose to have the connection details on list (on the external page).
- You can copy the symbol – with the settings you have given it.

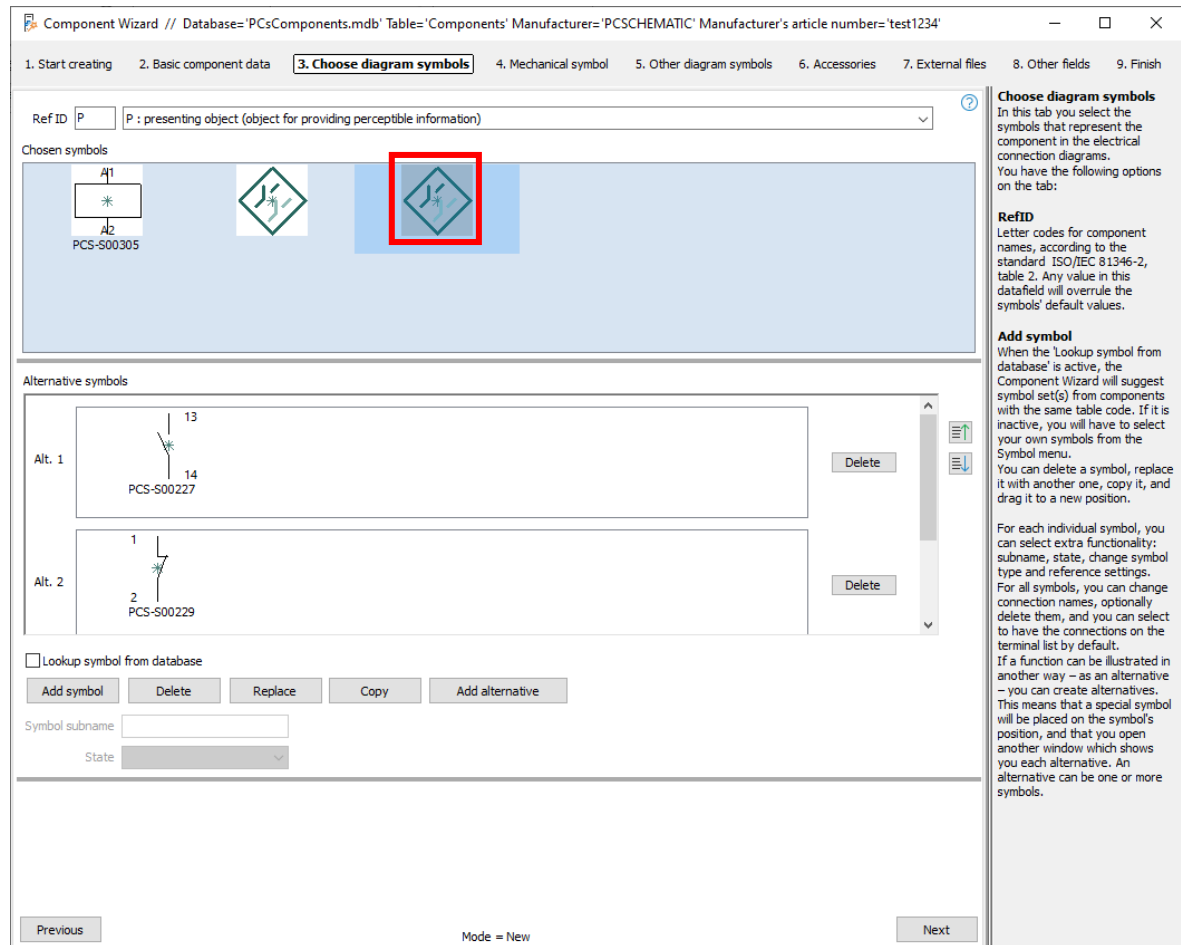
Alternative symbol selection

Some component symbols are displayed in different ways, reflecting the functions contained in the component.

In this example, it is a relay that contains a coil — positioned as shown earlier — and two contact sets, each of which can be displayed as either a termination, a breaker, or a toggle switch.

To create the alternate symbol selection, place the first option, and then click the Add Alternate button.

More alternatives are added by clicking on the bottom Add alternative.

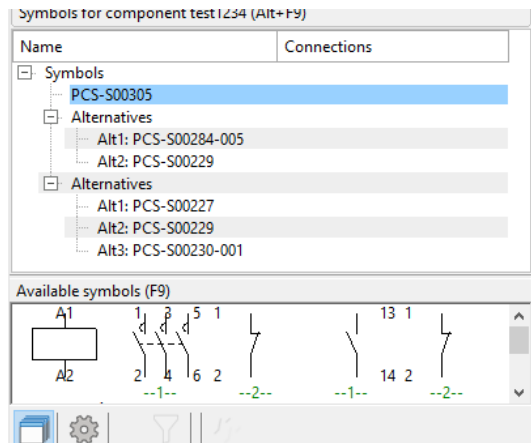


Now an ALT symbol is replaced by the first symbol, and underneath comes each of the – in this case – three alternative symbol choices.

When you retrieve the component from the database, it looks like this:

The window displays a coil, then two contact sets, each of which can be displayed as either a circuit switch, a refractive switch, or a toggle switch. When choosing one alternative, the other two fall away.

You can select from the list at the top, or via the symbols at the bottom.



Mechanical symbol

After selecting electrical symbols, you can select mechanical symbols for the component.

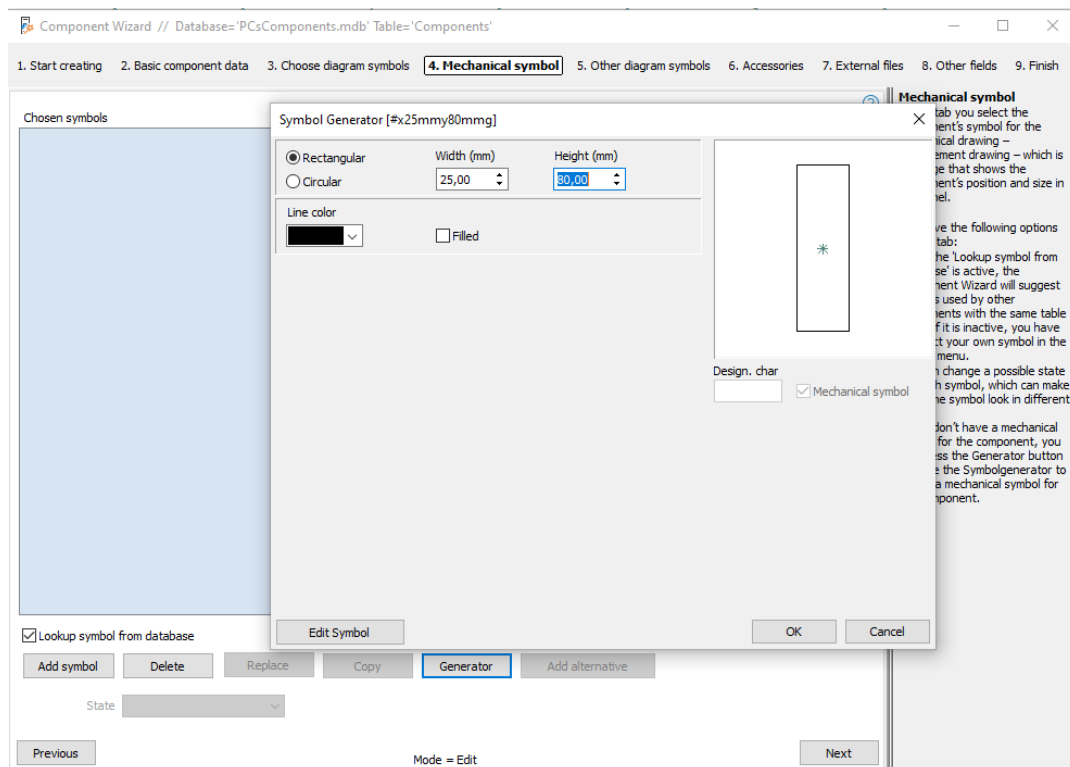
Here you also have the option to look up the database to be able to use the same symbols that have already been used in the article group or select a symbol from the symbol menu.

Alternatively, you can click the Generator button and call the Symbol Generator directly. If you have assigned a RefID in the previous window, it is automatically included here.

Here you can make a rectangular or a circular symbol without connections or anything else. Remember that it must have the right measurement!

Please note that the generator only stores symbols without connection points when you use it here.

If you want to have connection points on – which is a great idea – you can create a mechanical symbol using the Symbol Generator, which you save via Edit Symbol and then select it using the Add Symbol button.



Other diagram symbols

As of version 23, the database contains fields for different diagram types. This means that if your component is to be placed on single-line diagrams, it must have a symbol in the SingleLineSymbol field.

You can then select this field and also the symbol on each diagram page.

The screenshot shows the 'Component Wizard' window with the title bar 'Component Wizard // Database='PCsComponents.mdb' Table='Components''. The window has a tabbed interface with tabs numbered 1 to 9. Tab 5, '5. Other diagram symbols', is selected. The main area contains six sections, each with a text input field and three buttons: 'Add symbol', 'Delete', and 'Generator'. The sections are labeled 'usrSingleLineSymbol', 'usrInstSymbols', 'usrBusSymbols', 'usrPidSymbols', 'usrPneuSymbols', and 'usrHydSymbols'. Each section also has a checkbox labeled 'Lookup symbol from database' which is checked. On the right side, there is a vertical scroll bar and a help text box that reads: 'Here you select a symbol for the representation of the component in other diagram types, for instance in single line diagrams. The number of data fields here depends on your database setup of Component data|Other symbols.' At the bottom of the window, there are 'Previous' and 'Next' buttons, and a status bar that says 'Mode = Edit'.

By default, DIA pages are bound to the diagram Symbols field and GRP pages to MecSymbols. If you want to make single line diagrams, you can change the database field on the page – there is a drop-down in the component menu – in that way you get the right symbol for this diagram. Similarly, to the other diagram symbols seen above.

ACCESSORIES FOR COMPONENTS

Accessories are standalone components created in the database that can be associated with each component. Accessories come in two categories:

Accessories

That is, a component that automatically comes with the main component. Component ID is stored in the `usrFixedAccessories` field. You can choose a number for each accessory.

Optional accessories

That is, a component that can be added to the main component. ComponentID is stored in the `Optional Accessories` field. The number itself is selected in the project.

To add the accessory in the right category, tap the Add button. You can only select something that has already been created in the database.

The accessory contains the symbol(s) associated with it, and they can be placed in the project in the same way as the main components.

Component Wizard // Database='PCsComponents.mdb' Table='Components'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols **6. Accessories** 7. External files 8. Other fields 9. Finish

Accessories [usrFixedAccessories]

Count	Component ID	Type	Description
-------	--------------	------	-------------

Add Remove

Optional accessories [usrOptionalAccessories]

Component ID	Type	Description
EED771A4-F620-43	PCS-LAMPSOCKET1	Sokkel til PCS lamper og tryk, type 1
7D3C2F70-16FD-4	PCS-LAMPSOCKET2	Sokkel til PCS lamper og tryk, type 2

Add Remove

Previous Mode = Edit Next

Accessory
Fixed accessory, which means that it cannot be selected/deselected per component in the project. Fixed accessory is not (necessarily) visible in diagrams or arrangements. The accessory is included in parts- and component lists, when the List settings is made to do so.

Optional accessory
A list of selectable accessories. An accessory can have electrical, mechanical and/or other diagram symbols that can be placed in the project. Selected optional accessories are always included in the parts and component lists.

EXTERNAL FILES

On this tab you can attach external files, e.g. pictures and data sheets.

The program looks up one of the defined Alias folders, and when you select a file from one of these folders, the Alias name is written; If you choose to select the file from somewhere else, write the entire path.

Component Wizard // Database='PCsComponents.mdb' Table='Components'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories **7. External files** 8. Other fields 9. Finish

Picture (Picture)
%PCSPIC%PCSEmrcy.jpg

Datasheet (usrDatasheet)

Choose alias

☒ PCSPIC
☐ PCSDOC

Ok Cancel

Previous Mode = Edit Next

External files
In this tab you assign external files to the component, they can be datasheets or photos. Later you can make a zip-file of all related datasheets to a project. If the selected files are located in an ALIAS-folder, the Component Wizard writes the ALIAS-name instead of the full path.

Other fields

On the last tab are the last fields selected in the database settings. See more about this on page 119.

Component Wizard // Database='PCsComponents.mdb' Table='Components'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files **8. Other fields** 9. Finish

DescriptDK
Testkomponent

DescriptUK
Test component

BuiltInDepth
0,07

Weight
0,125

UnitPerPack
1

UnitForUPP
pcd

usrFavoriteComponent
☐

Obsolete
☐

usrFavoriteSupplier

Previous Mode = Edit Next

Other fields
On the tab you can type extra information about the component. The tab contains all mapped data fields that you haven't seen in the previous tabs, as well as the data fields that you selected in your Database settings.

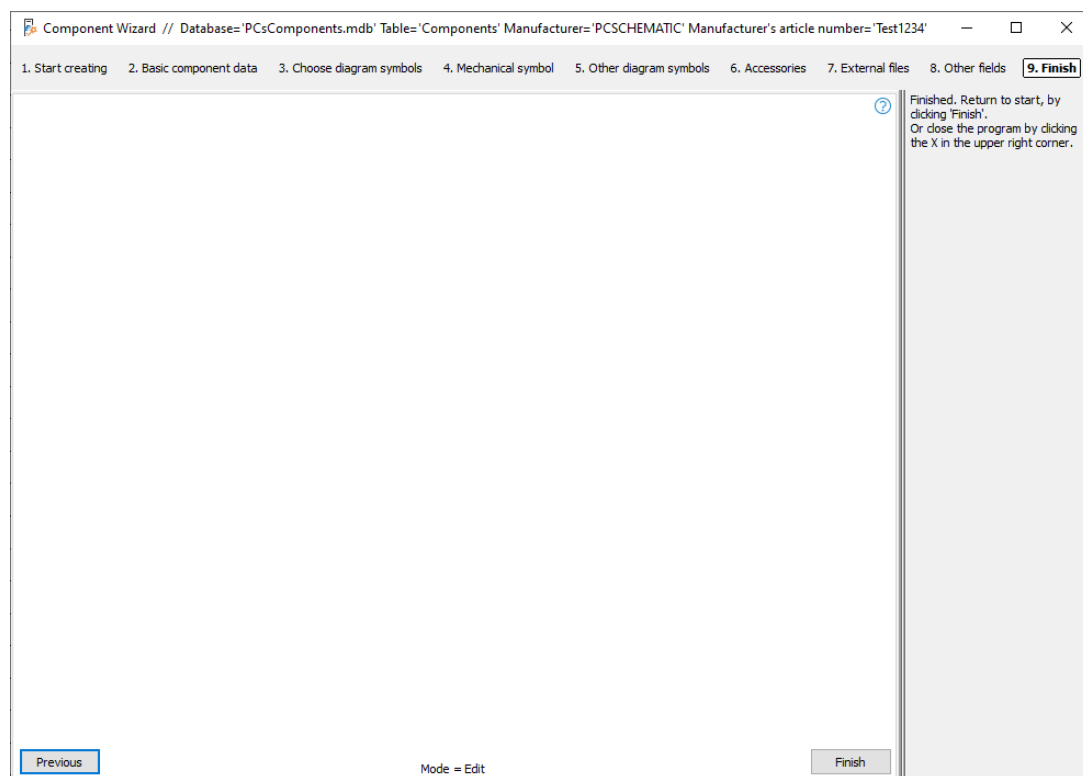
Here you can basically enter descriptions, dimensions and weight and whatever else you find relevant.

Dimensions and weights are entered in SI units, i.e. Dimensions must be entered in meters and Weight in kilograms!

You can press the All Fields button, and the entire record will open.
Press Next and you're done,

Finish

If you press Finish now, you will return 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 – again

When you are back here, you have the same choice as at the beginning.

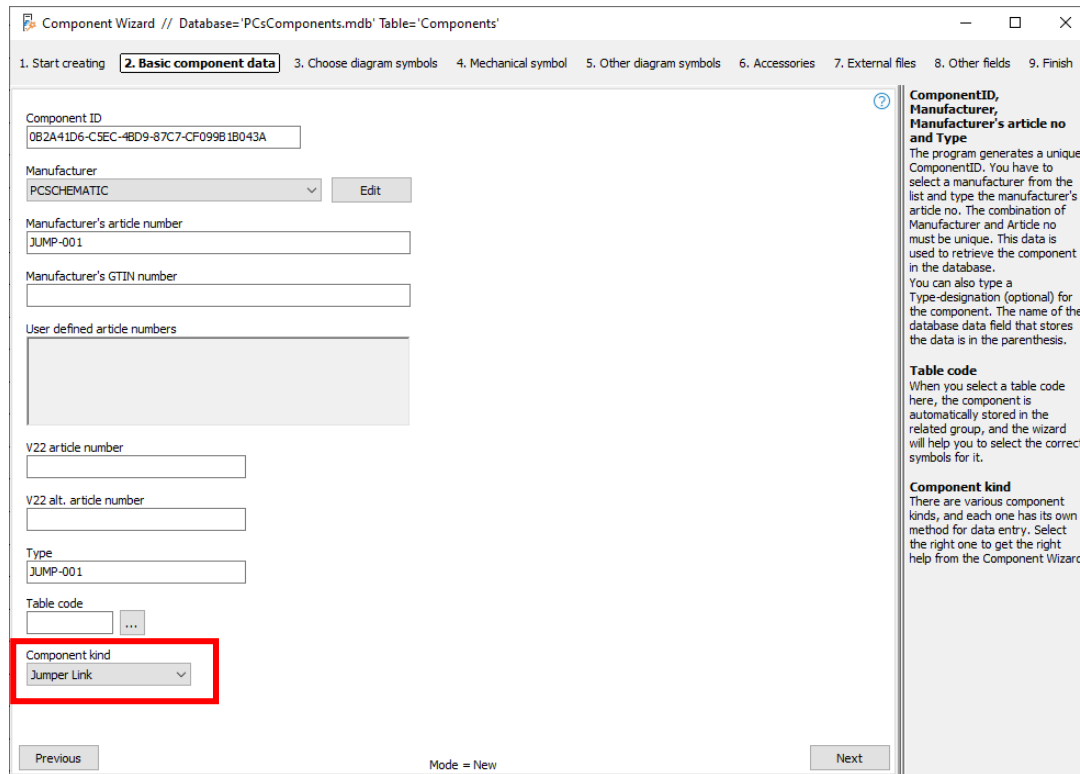
First tab – again again

If you interrupt in the middle of creating a component, the Wizard will remember that you were working on a given component and therefore suggests that you continue with this.

CREATE JUMPER LINK

When you create a new component using the Wizard, you have the option to select different component kinds. Below you can see examples of different kinds of components and how they are created in the database.

Start by choosing new and Jumper.



The screenshot shows the 'Component Wizard' window, specifically the '2. Basic component data' step. The window title is 'Component Wizard // Database: 'PCsComponents.mdb' Table: 'Components''. The progress bar at the top shows steps: 1. Start creating, 2. Basic component data (selected), 3. Choose diagram symbols, 4. Mechanical symbol, 5. Other diagram symbols, 6. Accessories, 7. External files, 8. Other fields, 9. Finish.

Fields and values:

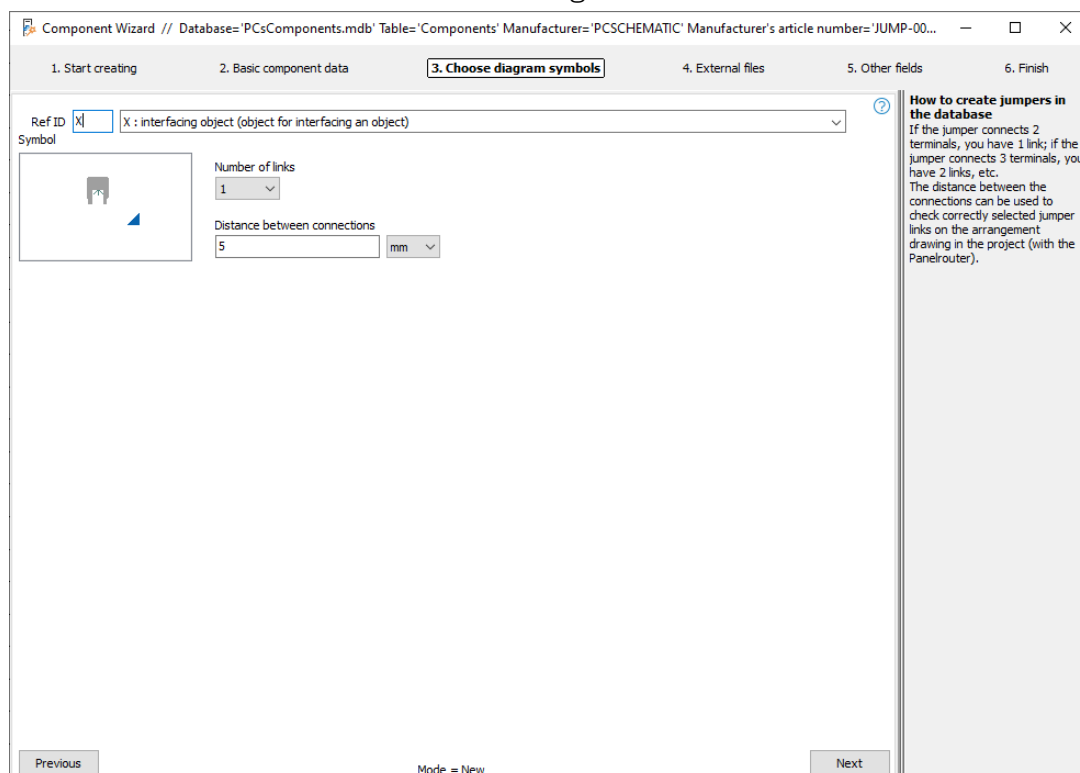
- Component ID: 0B2A41D6-C5EC-4BD9-87C7-CF099B1B043A
- Manufacturer: PCSCHEMATIC (with an 'Edit' button)
- Manufacturer's article number: JUMP-001
- Manufacturer's GTIN number: (empty)
- User defined article numbers: (empty list)
- V22 article number: (empty)
- V22 alt. article number: (empty)
- Type: JUMP-001
- Table code: (empty)
- Component kind: Jumper Link (highlighted with a red rectangle)

Buttons: 'Previous' and 'Next' are at the bottom. 'Mode = New' is displayed in the center.

Help text on the right:

- ComponentID, Manufacturer, Manufacturer's article no and Type**
The program generates a unique ComponentID. You have to select a manufacturer from the list and type the manufacturer's article no. The combination of Manufacturer and Article no must be unique. This data is used to retrieve the component in the database. You can also type a Type-designation (optional) for the component. The name of the database data field that stores the data is in the parenthesis.
- Table code**
When you select a table code here, the component is automatically stored in the related group, and the wizard will help you to select the correct symbols for it.
- Component kind**
There are various component kinds, and each one has its own method for data entry. Select the right one to get the right help from the Component Wizard.

When creating jumpers in the database, select (and display) the number of connections for the jumper and the distance between each connection. The Panel Router can check that the mechanical length matches the article number.



The screenshot shows the 'Component Wizard' window, specifically the '3. Choose diagram symbols' step. The window title is 'Component Wizard // Database: 'PCsComponents.mdb' Table: 'Components' Manufacturer: 'PCSCHEMATIC' Manufacturer's article number: 'JUMP-001...'. The progress bar at the top shows steps: 1. Start creating, 2. Basic component data, 3. Choose diagram symbols (selected), 4. External files, 5. Other fields, 6. Finish.

Fields and values:

- Ref ID: X (with a dropdown arrow)
- Symbol: (empty)
- Number of links: 1 (with a dropdown arrow)
- Distance between connections: 5 mm (with a dropdown arrow)

Buttons: 'Previous' and 'Next' are at the bottom. 'Mode = New' is displayed in the center.

Help text on the right:

- How to create jumpers in the database**
If the jumper connects 2 terminals, you have 1 link; if the jumper connects 3 terminals, you have 2 links, etc. The distance between the connections can be used to check correctly selected jumper links on the arrangement drawing in the project (with the Panelrouter).

CREATE WIRE

Start by choosing new and wire.

Component ID
0B2A41D6-C5EC-4BD9-87C7-CF09981B043A

Manufacturer
PCSCHEMATIC

Manufacturer's article number
Wire-001

Manufacturer's GTIN number

User defined article numbers

V22 article number

V22 alt. article number

Type
Wire-001

Table code

Blank

Component kind
Wire

Previous

Mode = New

Next

ComponentID, Manufacturer, Manufacturer's article no and Type
The program generates a unique ComponentID. You have to select a manufacturer from the list and type the manufacturer's article no. The combination of Manufacturer and Article no must be unique. This data is used to retrieve the component in the database.
You can also type a Type-designation (optional) for the component. The name of the database data field that stores the data is in the parenthesis.

Table code
When you select a table code here, the component is automatically stored in the related group, and the wizard will help you to select the correct symbols for it.

Component kind
There are various component kinds, and each one has its own method for data entry. Select the right one to get the right help from the Component Wizard.

When creating wires in the database, enter the diameter (mm or ") and number of meters per reel/drum. The diameter is used to calculate the capacity of the wiring channels in the Panel Router. The number of meters can be used in bills of materials (BOMs), so the number of rolls can be calculated based on the project's consumption.

Ref ID W W : guiding object (object for leading from one place to another)

Symbol

Diameter
5 mm

Meters per reel/drum
100

Previous

Mode = New

Next

How to create wires in the database
Type the outer diameter, that is the wire's diameter including its insulation.
This makes it possible to check the fill percentage of wire trays on the project's arrangement page (with the Panelrouter).

CREATE WIRETRAY

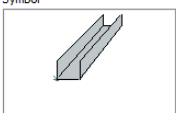
When creating a wiretray in the database, enter the internal height and width (mm or ") and length (m) of the tray.

The size of the tray is used when calculating capacity in the Panel Router, and to calculate how many pieces should be in the BOM.

Component Wizard // Database: 'PCsComponents.mdb' Table: 'Components' Manufacturer: 'PCSCHEMATIC' Manufacturer's article number: 'Wire-001'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Accessories 5. External files 6. Other fields 7. Finish

Ref ID: W W : guiding object (object for leading from one place to another)

Symbol: 

Height: 25 Width: 50 mm

Length (m): 2

How to create wire trays in the database
Type the height and width for the wire tray.
This makes it possible to check the fill percentage of wire trays on the project's arrangement page (with the Panelrouter).

Previous Mode = New Next

Draw with jumpers, wires and wire trays

Jumpers, wires and wire trays are primarily created in the database so that the Panel Router can use data to calculate the capacity of the wire trays.

If you want to draw with a specific wire type, you cannot retrieve a 'line' in the database, but you must pick the wire via the Pick menu:

Place a line of the desired type and color in the pick menu, e.g. a blue leading line for a regular wire or a pink line with a jumper-status for jumpers. To put component data on the line, right-click the line and select Component Article Data.

Then select the right data in the dialog box that appears. There can be several different Article numbers on the individual line in the pick menu.

If you create a wire tray in the pick menu, then you alone need to create the type and color of the line. The width itself is controlled from the database, and it aligns with the scale of the page.

Description	Article no:	Type	Fu
	2000	250	Ikk
> Wire tray 25 x 25 x 700 mm	PCS219001	PCST-25-25-700	
Wire tray 25 x 50 x 700 mm	PCS219002	PCST-25-50-700	
Wire tray 50 x 25 x 700 mm	PCS219003	PCST-50-25-700	
Wire tray 50 x 50 x 700 mm	PCS219004	PCST-50-50-700	

Ok Cancel

CREATE CABLE

When creating cables, select a cable symbol, either by looking up the database or directly in your symbol menu. The Wizard only accepts cable symbols!

You can enter diameter (mm or ") and number of meters per reel. Diameter is used to calculate the capacity of wiring channels in the Panel Router.

In addition, you must choose the number of conductors and how the individual conductors are labeled. You can choose between colors and numbers, where the Wizard helps to fill in, or you can write your own text. The last option is chosen, where you have both numbers and colors.

Codes for conductor colors are shown in the help text.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='test1234'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Accessories 5. External files 6. Other fields 7. Finish

Ref ID W W : guiding object (object for leading from one place to another)

Symbol

Diameter 7,5 mm

Meters per reel/drum 100

☐ Lookup symbol from database

Add symbol Delete

Number of conductors incl. ground 5

Cable conductors

☐ Colors

☐ Numbers

☒ Texts

Last conductor is GNYE

Load file

Conductor color/number
1
2
3
4
GNYE

Previous Mode = New Next

How to create cables in the database

Add symbol

The selected symbol must be of the Cable type.

Type in the cable's outer diameter, that is the diameter incl. insulation. This makes it possible to check the fill percentage of wire trays on the project's arrangement page (with the Panelrouter).

Type the number of cable conductors and select whether to identify them by color or number or a free text.

The last conductor can be selected as Green/Yellow. Conductor colors can also be selected from existing cable files (*.cbl, that contains a list of colors according to specific cable standards).

Conductor colors

BK: Black
BN: Brown
GT: Grey
BU: Blue
WH: White
RD: Red
VT: Violet
PK: Pink
TQ: Turquoise
GN: Green
YE: Yellow
OG: Orange
GNYE: Green Yellow
BUWH: Blue White
WHBU: White Blue

Download file

You can create cables with specific conductor colors by downloading an already existing file. Cable files are made in notepad or similar, they (often) follow a cable standard, have one conductor color per line, and they are saved as type *.cbl in the database folder. A sample file is included with the program (pcscable.cbl).

Last conductor is GNYE

By checking this box, the last conductor becomes GNYE.

Cable accessories

On the accessories tab you can assign fixed accessories for cables either per running meter or per cable.

The optional accessories are created as a list of selectable Article numbers, which in the project are selected per running meter or per cable.

CREATE PLC

The Component Wizard can also create PLC components in the database. You start by selecting the Component kind PLC, after which you enter a special PLC page.

Below are examples of creating different kinds of PLC so that you can see how each is created in the database.

The first example takes you through all the dialogues so you can see what options you have. The following examples show further possibilities.

Example 1: Cards with 8 digital inputs and separate supply

On the first tab, you must show how the PLC is built with inputs and outputs.

The screenshot shows the 'Component Wizard' dialog box, specifically the '3. PLC' tab. The title bar indicates the database is 'PCComponents.mdb', the table is 'Components', the manufacturer is 'PCSCHMATIC', and the article number is 'Test1234'. The tab bar at the top shows steps from '1. Start creating' to '10. Finish', with '3. PLC' currently selected. The main area contains a 'Ref ID' field with a dropdown menu set to 'View of standard Ref ID'. Below this is a table with columns: 'Variants', 'Channels', 'IO Status', and 'IO Status Type'. The first row has values '1', '8', 'Input', and 'Digital'. To the right of the table is an 'Add series' button. Below the table is an 'Add option' button. At the bottom, there is a 'Path to PLC-Directory' field containing 'C:\Automation\23_0_4_18\PCAutomation\Symbols\PLC\' and a checked 'Include subfolders' checkbox. 'Previous' and 'Next' buttons are at the bottom left and right respectively. A 'Mode = New' label is at the bottom center. On the right side of the dialog, there is a help panel titled 'How to create a PLC in the database' with instructions on how to use the wizard, including details about channels, IO status, IO status type, and variants.

Definitions

Variants

In how many different ways can you use your PLC? Used to create alternatives per channel/address.

Channels

We have chosen to use the word channels, which in most cases is synonymous with addresses, but which sometimes means e.g. connectors, or channels.

IO Status

Here you choose whether the channels can be input, output or something else.

IO Status type

You can create your channels with different status types, e.g. digital, analog, 4-20 mA, etc. The different status types are used to filter and find the right PLC when you want to transfer addresses to the individual IOs in the project.

Path to PLC folder

You can choose which folder you select symbols from. Default is selected PLC folder.

Start creation

Start by typing the total number of channels, here 8 digital inputs, as shown in the picture. Click next.

Enter PLC data and select PLC symbol

Enter how many addresses there are per channel (default = 1), then how many connection points there are per address, and finally the numeral system.

If there is more than one symbol that matches your entry, you can choose from them using the arrow keys.

If the symbol has states, you can display – and select one of – the states in the window. Type the first connection name - the one with IO status - type the first address and hover the mouse over 'Fill grid' to see what the program suggests. If it is OK, press the button and the addressing is complete.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Option 1 Other symbols

8 Input Digital

PLC Data Reference symbols

Addr. pr. channel 1 Terminals pr. addr. 2 Address number system OCT

Fill grid Clear grid Subname

Fill the grid automatically

Channel	IO	Address	Term2	Func2
1	X0	DI0	0	+
2	X1	DI1	1	+
3	X2	DI2	2	+
4	X3	DI3	3	+
5	X4	DI4	4	+
6	X5	DI5	5	+
7	X6	DI6	6	+
8	X7	DI7	7	+

Symbol (1/1)

Symbol: PLC-IN2

Title: IO input - 2 connections per address

State: 0

Diagram symbols for the PLC

In these tabs - IO data and Reference symbols - you select the PLC symbols. You have one tab for each part that you specified on the previous tab, so that you can find each part of the PLC and a common tab with other symbols. Each PLC-tab consists of two sub-tabs, one for IO-data and one for ref.symbols.

PLC Data

Here you type the number of addresses per channel (default = 1). When you have more than one address per channel, Subname is automatically applied to differentiate between channels. The number of connections pr address must be typed and the Component Wizard uses this to find the right symbols. Type connection name and default address for the IO-terminal, click the 'Fill grid' button and the grid is automatically filled; sometimes you might need to type in the 1st and the 2nd row.

Reference symbol

When you press the Add symbol button, the Component Wizard shows the ref.symbols that match your selected IO-symbols. They will match according to total number of

Previous Mode = New Next

PLC symbols MUST have a PLC status at first connection point! It's a rule. Only symbols that meet this can be selected here.

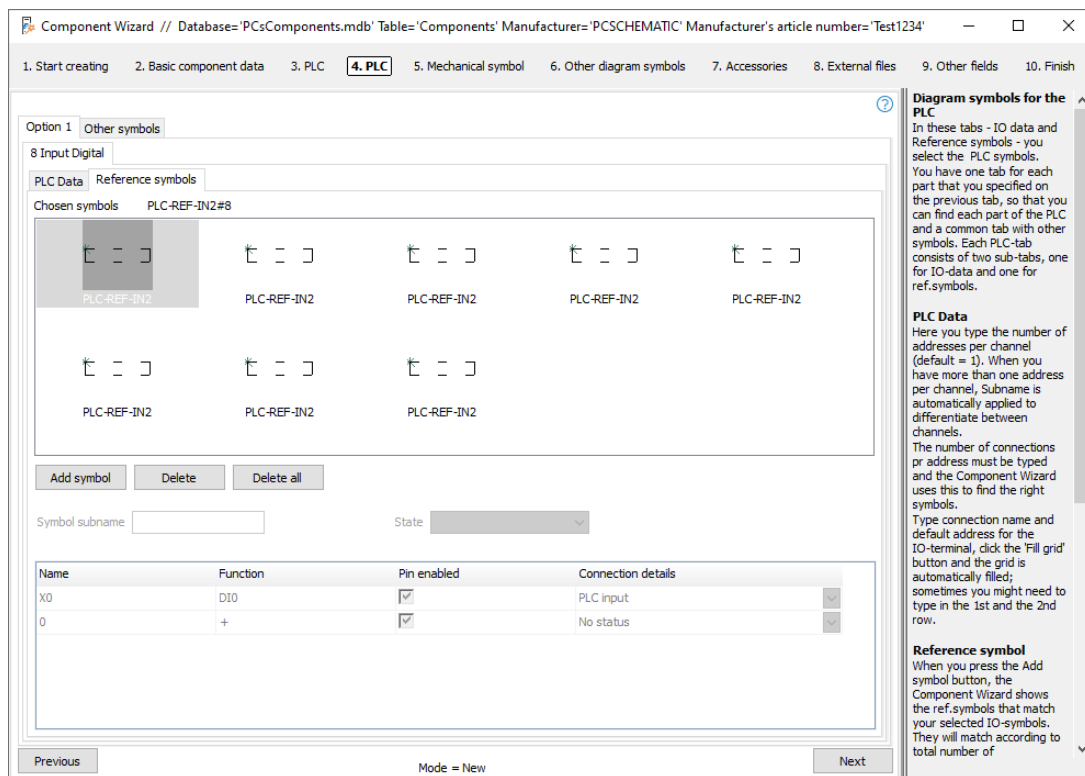
The file pcsPLCDemo contains several examples of PLCs and how they can be created. It is a good idea to see this and possibly open them in the Component Wizard to see how they are created.

Select PLC reference symbol

When you press the Add Symbol button, the Component Wizard has selected the reference symbols that match the selected PLC symbol.

Once you have eight inputs, you can choose between PLC-ref. symbols with 8, 4, 2 or 1 input. The PLC-ref. symbols must also match in terms of number of connection points per address, and this is where it is difficult if you have to choose manually.

In the folder of symbols created for ver23, there is only one choice. Here are PLC-ref. symbols made as small symbols, each covering one address.



The eight individual PLC-ref. symbols each contain the same PLC data that was entered on the previous tab – so that the symbols match.

Rules for PLC symbols

IO status MUST be the same for PLC symbols and PLC ref. symbols, i.e. 'inputs to inputs, and outputs to outputs'.

There must be the same number of connection points per address – otherwise you cannot transfer data between the symbols in the project.

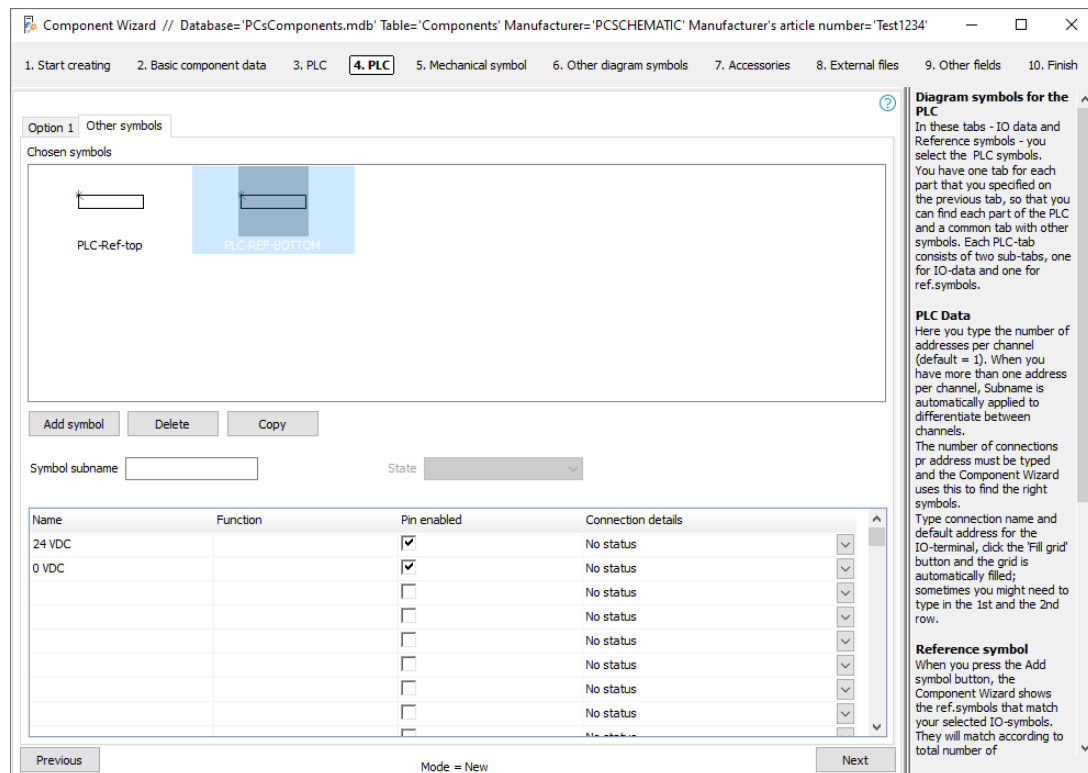
The first connection point must have IO status and an address, the other connection points must have the same value in the function text as the first has as an address. Otherwise, they do not hang at the address.

Other symbols – Top and bottom etc.

On the last tab you can select other symbols.

The new symbols from ver23 should have symbols for top and bottom respectively. This is also how the components are created on the Component Portal.

In the example, it has been chosen to show supply on the bottom symbol. Alternatively, a separate symbol could have been chosen for this.



Now you have finished selecting symbols for diagrams, press Next to select mechanical symbols, accessories, data sheets, etc. that the database can hold for the components.

There are four sets of top and bottom, for 1, 2, 3 or 4 bars respectively.

If you want to force your PLC to have more PLC-ref. symbols, you can put more top/bottom sets in here.

As a starting point, the program divides the component into inputs and outputs, or when there are more than 32 addresses on the PLC; 16 addresses if they are large.

Example 2: Compact PLC with 8 inputs and 8 outputs

On the first tab you select 8 digital inputs and 8 digital outputs. Press Next.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Ref ID View of standard Ref ID

Variables Channels IO Status IO Status Type

[1a] 1 8 Input Digital Remove series Add series

[1b] 1 8 Input Digital Remove series Add series

Add option

Path to PLC-Directory
C:\Automation\23_0_4_18\PCAutomation\Symbols\PLC\ Include subfolders

Previous Mode = New Next

How to create a PLC in the database
In this tab you see the overall structure of the PLC. At the top you type the total no of channels. In the rows below you select how those channels are distributed. The channels can be distributed in fixed or variable kinds – note, that the Component Wizard makes small 'boxes' around every section – fixed and options. When you add the number of channels you must end up with the same total number of channels as above.

Channels
Channels is in most cases a synonym for Addresses, but it can also mean plugs or channels. The final number of addresses is selected on the following page.

IO status
Here you select whether the channels are Inputs, Outputs, Other (e.g. IO-link) or a combination of these.

IO status type
You can create channels with an IO status type, either digital, analog, or none.

Variants
If e.g. an analog input can be connected in different ways, type the number of variants here, as it influences the selection of IO symbols.

Select PLC symbols and PLC-ref. symbols

Now there are two tabs where you select PLC symbols: one for input and one for output. The function of the two sets of tabs is as described in example 1, the selection of PLC symbol and the selection of PLC-ref. symbol.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Option 1 Other symbols

[1a] - 8 Input Digital [1b] - 8 Input Digital

PLC Data Reference symbols

Addr. pr. channel Terminals pr. addr. Address number system

1 2 OCT

Fill grid Clear grid Subname

Symbol: PLC-IN2
Title: IO input - 2 connections per address
State: 0

Symbol (1/1)

Channel	IO	Address	Term2	Func2
1	Y0	DO0	0.	-
2	Y1	DO1	1.	-
3	Y2	DO2	2.	-
4	Y3	DO3	3.	-
5	Y4	DO4	4.	-
6	Y5	DO5	5.	-
7	Y6	DO6	6.	-
8	Y7	DO7	7.	-

Previous Mode = New Next

Diagram symbols for the PLC
In these tabs - IO data and Reference symbols - you select the PLC symbols. You have one tab for each part that you specified on the previous tab, so that you can find each part of the PLC and a common tab with other symbols. Each PLC-tab consists of two sub-tabs, one for IO-data and one for ref.symbols.

PLC Data
Here you type the number of addresses per channel (default = 1). When you have more than one address per channel, Subname is automatically applied to differentiate between channels. The number of connections pr address must be typed and the Component Wizard uses this to find the right symbols. Type connection name and default address for the IO-terminal, click the 'Fill grid' button and the grid is automatically filled; sometimes you might need to type in the 1st and the 2nd row.

Reference symbol
When you press the Add symbol button, the Component Wizard shows the ref.symbols that match your selected IO-symbols. They will match according to total number of

Example 3: A card that can be either 8 outputs or 8 inputs

The entries on the first tab determine how the PLC is created.

This is where you choose how many addresses the component has and how they combine or exclude each other.

You must notice the small 'boxes' around the individual blocks. The boxes mark a part of the PLC and can either be a fixed part of it or an option.

When there are more blocks, each block gets a name, and that name repeats itself in the next tabs where you choose symbols.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test132'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Ref ID View of standard Ref ID

Option 1

Variables	Channels	IO Status	IO Status Type	<input type="checkbox"/> Fixed
[1a] 1	8	Input	Digital	<input type="button" value="Add series"/>

Option 2

Variables	Channels	IO Status	IO Status Type	<input type="checkbox"/> Fixed
[2a] 1	8	Output	Analog	<input type="button" value="Add series"/>

Path to PLC-Directory

☒ Include subfolders

Mode = New

How to create a PLC in the database
In this tab you see the overall structure of the PLC. At the top you type the total no of channels. In the rows below you select how those channels are distributed. The channels can be distributed in fixed or variable kinds – note, that the Component Wizard makes small 'boxes' around every section – fixed and options. When you add the number of channels you must end up with the same total number of channels as above.

Channels
Channels is in most cases a synonym for Addresses, but it can also mean plugs or channels. The final number of addresses is selected on the following page.

IO status
Here you select whether the channels are Inputs, Outputs, Other (e.g. IO-link) or a combination of these.

IO status type
You can create channels with an IO status type, either digital, analog, or none.

Variables
If e.g. an analog input can be connected in different ways, type the number of variants here, as it influences the selection of IO symbols

This example is based on a card where, for example, by setting a jumper, you turn the entire card into inputs or outputs.

Option 1 contains 8 inputs, option 2 contains 8 outputs.

When you get to the selection of PLC and PLC ref. symbols, the dialog looks like in the previous examples.

Place the component in the project

When placing the component in the project, you are given the choice between the two options: depending on what you choose, you can see the connection names^{iv}:

Select PLC configuration

	Analog Input	Digital Input
X0,X1,X2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Bottom	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Select PLC configuration

	Analog Input	Digital Input
Y0,Y1,Y2	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Bottom	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Example 4: A card that can have different status types per address

This example is based on a card where you determine its IO status type for each address. This is based on an analog card that can be programmed/wired to different signal types.

Each address has two variants, i.e. there are two lines with the 8 addresses:

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test134'

1. Start creating 2. Basic component data **3. PLC** 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Ref ID View of standard Ref ID

Variables	Channels	IO Status	IO Status Type	
<input type="text" value="2"/>	<input type="text" value="8"/>	<input type="text" value="Input"/>	<input type="text" value="AI^4-20mA"/>	<input type="button" value="Add series"/>
		<input type="text" value="Input"/>	<input type="text" value="AI^Temp"/>	

Path to PLC-Directory

☒ Include subfolders

The entry means that two sets of tabs are created where you can select PLC symbol and PLC ref symbol; exactly as in the previous examples.

Place the component in the project

When the component is to be placed in the project, you get a grid that shows the options that the component contains.

Here you put a ✓ in the desired option, after here you see the connection points.

When you place the symbols, the status type is also displayed, at least when using the default symbols.

	Analog Input 4-20mA	Analog Input Temp
X0,0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X1,1	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X2,2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X3,3	<input checked="" type="checkbox"/>	<input type="checkbox"/>
X4,4,4.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X5,5,5.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X6,6,6.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
X7,7,7.	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Top	<input checked="" type="checkbox"/>	
Bottom	<input checked="" type="checkbox"/>	

OK

-K4
AI0
/ .8
+
XO 0

-K4
AI4
/ .8
+
X4 4.

Example 5: Compact card with communication

Here a PLC is created, where there are both inputs, outputs and communication channels.

Component Wizard // Database='PCComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Ref ID: View of standard Ref ID:

Variables	Channels	IO Status	IO Status Type		
[1a]	2	8	Input	AI~4-20mA	Remove series Add series
			Input	AI~Temp	
[1b]	1	6	Output	DO~Relay	Remove series Add series
[1c]	1	1	Other	None	Remove series Add series

Add option

Path to PLC-Directory: C:\Automation\23_0_4_18\PCAutomation\Symbols\PLC\ ... ☒ Include subfolders

Previous Mode = Edit Next

How to create a PLC in the database

In this tab you see the overall structure of the PLC. At the top you type the total no of channels. In the rows below you select how those channels are distributed.

The channels can be distributed in fixed or variable kinds – note, that the Component Wizard makes small 'boxes' around every section – fixed and options. When you add the number of channels you must end up with the same total number of channels as above.

Channels

Channels is in most cases a synonym for Addresses, but it can also mean plugs or channels. The final number of addresses is selected on the following page.

IO status

Here you select whether the channels are Inputs, Outputs, Other (e.g. IO-link) or a combination of these.

IO status type

You can create channels with an IO status type, either digital, analog, or none.

Variables

If e.g. an analog input can be connected in different ways, type the number of variants here, as it influences the calculation of I/O modules.

Place the component in the project

The number of channels goes up by eight, therefore an empty PLC-ref. symbol is placed.

The Comm channel is selected as generic Communication on the ref symbol and a connector on the external symbol.

In addition, I have chosen to show the supply on a separate symbol instead of at the bottom of the reference symbol.

-K5			
AI0	+	X0	DO0
AI Temp	-	0.	DO Relay

AI1	+	X1	DO1
AI Temp	-	1.	DO Relay

AI2	+	X2	DO2
AI Temp	-	2.	DO Relay

AI3	+	X3	DO3
AI Temp	-	3.	DO Relay

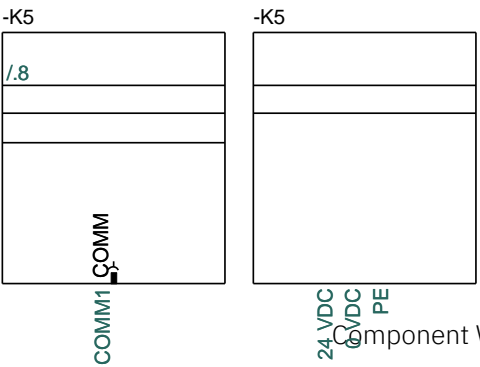
AI4	+	X4	DO4
AI 4-20mA		4	DO Relay

AI5	+	X5	DO5
AI 4-20mA		5	DO Relay

AI6	+	X6	COMM
AI 4-20mA		6	

AI7	+	X7	
AI 4-20mA		7	

			COMM1
			COMMUNICATION



Example 6: Card that refer to different connectors (sub-name)

This example is based on a component that per channel/connector can have either 2 inputs, 2 outputs or 1 IO-link.

When we create it, we choose to create 4 channels, where one channel corresponds to one connector.

Component Wizard // Database='PCComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Ref ID View of standard Ref ID

Variants: 3 Channels: 4 IO Status: Input IO Status Type: Digital

Add series

Add option

Path to PLC-Directory: C:\Automation\23_0_4_18\PCAutomation\Symbols\PLC\

Include subfolders ☒

Previous Mode = Edit Next

How to create a PLC in the database
In this tab you see the overall structure of the PLC. At the top you type the total no of channels. In the rows below you select how those channels are distributed. The channels can be distributed in fixed or variable kinds - note, that the Component Wizard makes small 'boxes' around every section - fixed and options. When you add the number of channels you must end up with the same total number of channels as above.

Channels
Channels is in most cases a synonym for Addresses, but it can also mean plugs or channels. The final number of addresses is selected on the following page.

IO status
Here you select whether the channels are Inputs, Outputs, Other (e.g. IO-link) or a combination of these.

IO status type
You can create channels with an IO status type, either digital, analog, or none.

Variants
If e.g. an analog input can be connected in different ways, type the number of variants here, as it influences the selection of IO number.

Since each channel/connector contains multiple addresses, and because each channel is the same, we use sub-names to separate these. Each channel gets an X-subname. Fill in the first connector, press Fill grid and this is the result:

Component Wizard // Database='PCComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating 2. Basic component data 3. PLC 4. PLC 5. Mechanical symbol 6. Other diagram symbols 7. Accessories 8. External files 9. Other fields 10. Finish

Option 1 Other symbols

var 1 - 4 Input Digital var 2 - 4 Output Digital var 3 - 4 Other None

PLC Data Reference symbols

Addr. pr. channel: 2 Terminals pr. addr.: 2 Address number system: OCT

Fill grid Clear grid Subname ☒

Symbol: PLC-2XIN2 Title: IO input - 2 connections per address - x2 State: 0

Symbol (1/1)

Channel	Subname	IO	Address	Term2	Func2
1	X1	1	DI0	2	+
		5	DI1	6	+
2	X2	1	DI0	2	+
		5	DI1	6	+
3	X3	1	DI0	2	+
		5	DI1	6	+
4	X4	1	DI0	2	+
		5	DI1	6	+

Previous Mode = Edit Next

Diagram symbols for the PLC
In these tabs - IO data and Reference symbols - you select the PLC symbols. You have one tab for each part that you specified on the previous tab, so that you can find each part of the PLC and a common tab with other symbols. Each PLC-tab consists of two sub-tabs, one for IO-data and one for ref.symbols.

PLC Data
Here you type the number of addresses per channel (default = 1). When you have more than one address per channel, Subname is automatically applied to differentiate between channels. The number of connections per address must be typed and the Component Wizard uses this to find the right symbols. Type connection name and default address for the IO-terminal, click the 'Fill grid' button and the grid is automatically filled; sometimes you might need to type in the 1st and the 2nd row.

Reference symbol
When you press the Add symbol button, the Component Wizard shows the ref.symbols that match your selected IO-symbols. They will match according to total number of

Place the component in the project

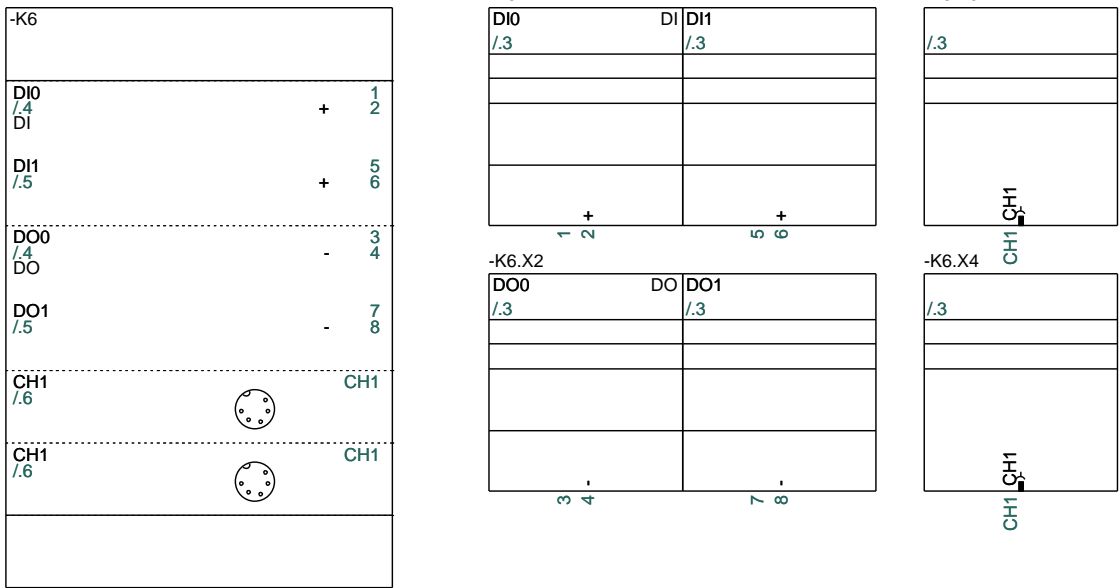
When you choose to place the component, the grid comes up so you can choose which channels you want to use:

Select PLC configuration

	Digital	Digital	Alternative 1
1,2,5,6	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
3,4,7,8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
CH1	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
CH1	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Top	<input checked="" type="checkbox"/>		
Bottom	<input checked="" type="checkbox"/>		

OK

The result will look like this:



COPY OF AN EXISTING COMPONENT

When you choose to make a copy of an existing component, you must first go to the database to select the 'original'. After that, you will enter this window where you need to give the 'copy' its own data. Since the original and copy often have the type and Article number, which are close to each other, the Wizard suggests the values of the original. Then you continue as with 'New'.

However, you will not be allowed to continue 'Copy of' unless you enter a new, non-existent Article number. Alternatively, the Wizard suggests editing instead.

Component Wizard // Database='PCsComponents.mdb' Table='Components' Manufacturer='PCSCHEMATIC' Manufacturer's article number='Test1234'

1. Start creating **2. Basic component data** 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files 8. Other fields 9. Finish

Make copy of this comp. (ComponentID) 2A128EA5-863F-475C-9EF7-59FE65C9A62F	Component ID 2208E945-8F2F-4FC5-9763-117A2B6971A5
Manufacturer PCSCHEMATIC	Manufacturer PCSCHEMATIC <input type="button" value="Edit"/>
Manufacturer's article number Test1234	Manufacturer's article number Test5678
Manufacturer's GTIN number 1234	Manufacturer's GTIN number 5678
User defined article numbers	User defined article numbers
V22 article number	V22 article number
V22 alt. article number	V22 alt. article number
Type Test1234	Type Test5678
Table code 5000 <input type="button" value="..."/> Filament lamps	
Component kind Normal	

Previous Mode = Copy of Next

Copy a component
Click the Database icon to select the component, you wish to make a copy of.

Select a manufacturer and manufacturer's article no and optionally type, that you wish to give to the copy. The combination of Manufacturer and Article number must be unique.

EDIT AN EXISTING COMPONENT

If you select the edit function, you must also select the desired component in the database. After that, proceed with the same tabs as with 'New'.

The screenshot shows the 'Component Wizard' dialog box with the title bar indicating the database is 'PCsComponents.mdb', the table is 'Components', the manufacturer is 'PCSCHEMATIC', and the manufacturer's article number is 'Test1234'. The '2. Basic component data' tab is selected. The form contains the following fields and controls:

- Component ID:** 2A128EAS-863F-475C-9EF7-55FE65C9A62F (with a database icon)
- Manufacturer:** PCSCHEMATIC (with an 'Edit' button)
- Manufacturer's article number:** Test1234
- Manufacturer's GTIN number:** 1234
- User defined article numbers:** (empty text area)
- V22 article number:** (empty text field)
- V22 alt. article number:** (empty text field)
- Type:** Test1234
- Table code:** 5000 (with a button to select from a list containing 'Filament lamps')
- Component kind:** Normal (dropdown menu)

At the bottom, there are three buttons: 'Previous', 'Mode = Edit' (highlighted with a red rectangle), and 'Next'. On the right side, there is a panel titled 'Edit a component' with the instruction: 'Click the Database icon to select the component, you wish to modify.'

Notice the bottom of the dialog – here you can see if you're creating a new one, copying or editing.

EDIT MULTIPLE COMPONENTS AT ONCE

When you have an (old) database, you need to be able to maintain data, and in this way you often need to be able to edit many components at once. With this feature, you can select a series of existing components based on one or more criteria and assign them common data, e.g. the same electrical symbols or a common data sheet or the status of Approved or Discontinued. In short, the functions that today can only be performed using the database program's built-in SQL function.

Select Edit more. You select components by using a filter. The filter can be applied to all fields in the selected database.

Component Wizard // Database: 'PCComponents.mdb' Table: 'Components'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files 8. Other fields 9. Finish

Filter settings

DescriptUK contains lamp

"DescriptUK" contains "lamp"

"Type" begins with "PCS"

Delete filter

Add filter

☒ Advanced ((DescriptUK LIKE '%lamp%') AND (Type LIKE 'PCS%'))

Apply filter

Selected components Count: 8

ComponentID	Manufacturer	ManufacturersArticleNumber	ManufacturersGTIN	HistoricalArticleNumber	Historical
47CF004F-EEC5-4D91-BD88-1720987117E7	PCSCHEMATIC	PCS500004		PCS500004	
7D3C2F70-16FD-4256-A2FB-0A23360E0838	PCSCHEMATIC	PCS500006			
9E1E36FD-30AC-4DA5-B762-5515E59EB0FA	PCSCHEMATIC	PCS500001		PCS500001	
9AAACC62-1E74-413A-A120-9AF5B79D6FF9	PCSCHEMATIC	PCS500002		PCS500002	
D190284C-EF7E-46A4-969C-037080DDE3E7	PCSCHEMATIC	PCS2250202		PCS2250202	
DD3E5ED4-37DF-43B0-A899-C4EDE5282DB9	PCSCHEMATIC	PCS500007			
EBBFAD16-EA5D-4165-B21F-C45DDEA85DD8	PCSCHEMATIC	PCS500003		PCS500003	
EED771A4-F620-430D-8497-4BA9AA11ED45	PCSCHEMATIC	PCS500005		PCS500005	

Previous Mode = Edit multiple Next

Advanced filter

Two filters are set up here: The English description of the components must contain the word 'lamp' and the Type must begin with 'PCS'.

This query is basically an AND filter, i.e. selected components must meet both Filter1 AND Filter2 AND etc..

When you select 'Advanced', you get access to see the query as an SQL. Here you have the opportunity to replace AND with OR, and thus set up a completely different filter, which has previously been a little complicated to make on your own.

If you want to select Favorite components or other Boolean fields, set the filter to 'contain 1'.

Continue editing multiple

Once you have set up your filters, move on by pressing Next. If something in these windows is 'Grayed out', it means that the selected components do NOT have the same data in the selected database field. If you keep it, this particular data will not be overwritten. If you want to overwrite, activate the field by clicking in it.

The screenshot shows the 'Component Wizard' window for the 'Components' table in the 'PCsComponents.mdb' database. The window has a tabbed interface with the following tabs: 1. Start creating, 2. Basic component data, 3. Choose diagram symbols, 4. Mechanical symbol, 5. Other diagram symbols, 6. Accessories, 7. External files, 8. Other fields (active), and 9. Finish. The '8. Other fields' tab contains several input fields: 'DescriptDK', 'DescriptUK', 'BuiltInDepth', 'Weight', 'UnitPerPack', 'UnitForUPP', 'usrFavoriteComponent' (with a checkbox), 'Obsolete' (with a checkbox), and 'usrFavoriteSupplier'. A help icon (?) is located next to the 'DescriptDK' field. A text box on the right side of the window provides instructions: 'On the tab you can type extra information about the component. The tab contains all mapped data fields that you haven't seen in the previous tabs, as well as the data fields that you selected in your Database settings.' At the bottom of the window, there are 'Previous' and 'Next' buttons, and a status bar indicating 'Mode = Edit multiple'.

Remember that you are editing directly in the database and with this function you have access to many components.

When you work in the database, you have no Undo function!!! Therefore, we recommend that you consider taking a copy of your database BEFORE using this function.

DELETE COMPONENTS

The Component Wizard can also be used to delete components.

The delete function contains the same filter functions as under edit and edit multiple, so you start by selecting the component(s) to be deleted based on various criteria, and finish by pressing the Delete button in the lower right corner.

Component Wizard // Database: 'PCsComponents.mdb' Table: 'Components'

1. Start creating 2. Basic component data 3. Choose diagram symbols 4. Mechanical symbol 5. Other diagram symbols 6. Accessories 7. External files 8. Other fields 9. Finish

Filter settings

Manufacturer not equal pcsch

Advanced (Manufacturer <> 'pcsch')

Apply filter

Selected components Count: 194

ComponentID	Manufacturer	ManufacturersArticleNumber	ManufacturersGTIN	HistoricalArticleNumber	Historic
5E7B786E-AA4C-4F1C-97B4-B647F8929F9A	PCSCHEMATIC	PCS2220002			
60A5CA20-0125-4004-9FD3-DC622D971C79	PCSCHEMATIC	PCS320008		PCS320008	
61FD9E87-C274-457B-BA5C-EF5E580CE36C	PCSCHEMATIC	PCS02210017			
65620471-1C4D-4351-A295-3E2766A5CD82	PCSCHEMATIC	PCS02210010			
67AE532B-3030-43BC-9D5D-CC48D188FB44	PCSCHEMATIC	PCS320017		PCS320017	
6A8B253A-CBF5-4096-8D87-E4A5239846AD	PCSCHEMATIC	PCS320014		PCS320014	
6B0FD7D1-D974-48ED-8E6C-2017A72E0061	PCSCHEMATIC	PCS212501		PCS212501	
0001FA14-CD98-40A3-B37B-09A604FB7109	PCSCHEMATIC	PCS320012		PCS320012	
01816DEF-1AEF-4CF4-8B60-DADD833E8281	PCSCHEMATIC	PCS320002		PCS320002	
036C9933-DF2A-47C7-964C-1CDEE794F994	PCSCHEMATIC	PCS2250121		PCS2250121	
045C87E5-DB01-4DF7-8EEB-AB3AFAF165FA	PCSCHEMATIC	PCS2250109		PCS2250109	
04D42C88-3FFC-4A9D-B211-A91FC6960580	PCSCHEMATIC	PCS1640003		PCS1640003	
0591FBCD-0E90-4C8B-A284-6D98E0586D66	PCSCHEMATIC	PCS2250108		PCS2250108	
071B3B6C-DC70-46A1-9BB6-1BA476DE5B37	PCSCHEMATIC	PCS2250002		PCS2250002	
07BDB7B4-2F08-43D0-8D5B-CBDA83FCAEC8	PCSCHEMATIC	PCS360001		PCS360001	

Previous Mode = Delete Delete

SETTING UP COMPONENT WIZARD AND DATABASE

Under Settings|Database you can choose the database, and thus also which database the Component Wizard is associated with. As of ver 23, most settings have already been made – accessories, symbols for various diagrams, etc. – but you do have a few options for your own customization, as described below.

Article numbers

If you have your own article number, then the correct thing to do would be to create a field for this in the database, and then add it as an article number on this tab. In this way, you can both search it in the database and have it selected as the Article number of components in the project. And you will automatically be asked about it on the first tab during creation.

The screenshot shows the 'Component Database setup' dialog box with the 'Component search' tab selected. The 'Basic' section on the left contains fields for Manufacturer, Type, Function, Description, Reference letter, Units/Pack, Fixed accessories, Optional accessories, Favorite, Obsolete, Preferred Link field, and Thumbnails field. The 'Article numbers' section on the right includes fields for Manufacturer's article no., Manufacturers GTIN, User selected article numbers (with a 'My field' entry and a button to add more), and Historical article numbers (with fields for V22 article no. and V22 alt. article no.).

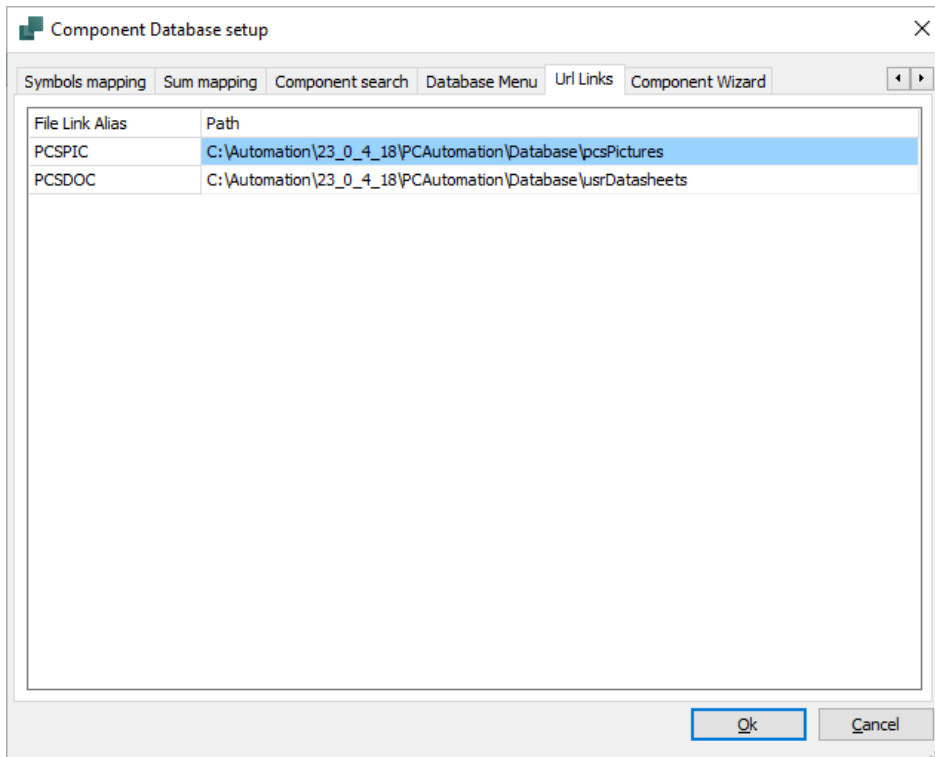
And you will automatically be asked about it on the first tab during creation:

The screenshot shows the 'Component Wizard' dialog box with the '2. Basic component data' tab selected. The 'Component ID' field is populated with 'A4DACBCC-0151-44E6-AF43-CF647A332CCB'. The 'Manufacturer' field is a dropdown menu with an 'Edit' button. The 'Manufacturer's article number' and 'Manufacturer's GTIN number' fields are empty. The 'User defined article numbers' section includes a 'My field' entry with an empty text box.

Remember that everything you create yourself in the database will be stored in the usr-fields. Content in the PCS-fields comes from the Component Portal.

URL-links

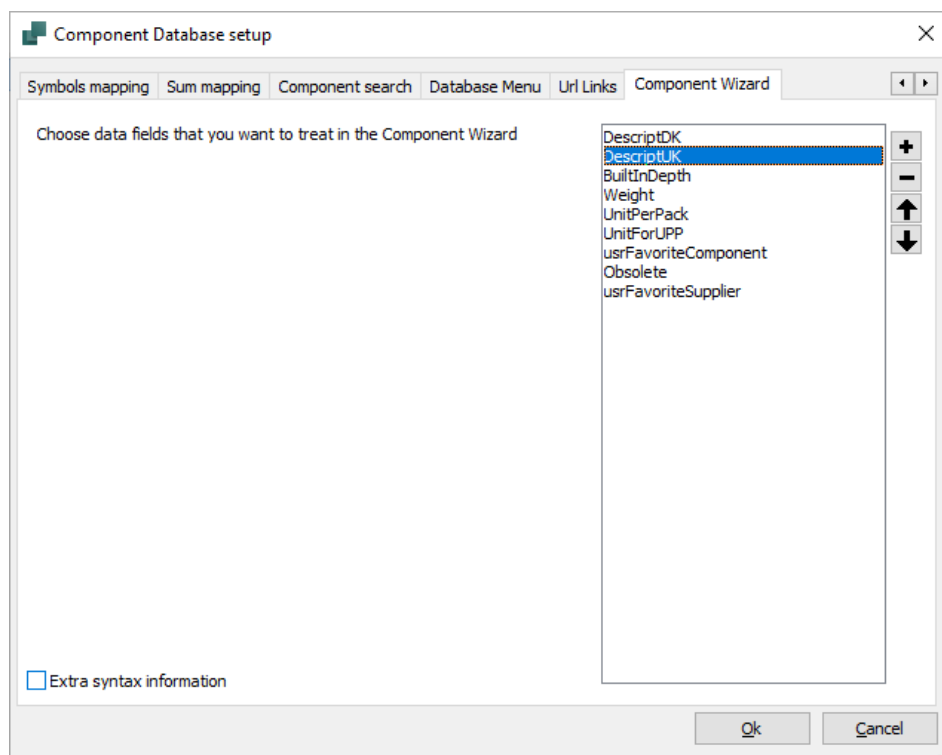
If you have several places where you store data sheets etc., then you should create links to these here. When you then add images and data sheets to the component, you are given the choice between the created aliases.



Other fields

On the last tab, you can select other fields that you want to enter.

You can only select fields that are not already mapped, e.g. descriptions in other languages.



MOUNTING ASSISTANT

The mounting assistant is a tool used when installing the electrical equipment. It leads the way to the paperless workshop when it keeps track of all components and connections in the diagrams and logs all completed tasks in a separate database.

Purpose of the Mounting Assistant

There are several purposes for using the Mounting Assistant: You save time in mounting, as the diagram is always updated, so you do not have to keep the paper folder up to date, and in this way you are also on your way to the paperless workshop with full quality assurance of the work done.

In headings, the purposes can also be listed in this way:

- To control the production of a Panel without the use of paper
 - The Program reads the diagrams and displays the individual components and connections in a list for which the user decides the sorting
- It will assist in the placement and control of the placed components
 - The Program generates different lists, which help the installer to see all components and to structure and control all parts
- It will help control the installation of wires, cables, jumpers, etc.
 - All connections can be sorted by kind, type and other, so you get a useful order for the installation
- It will help with the finishing of components
 - If components need to be post-processed, e.g. setting a timer, this also appears in the window
- All actions are documented in a database
 - The database logs user, time and action – this is quality assurance of the installation
- This can be done for prototypes, one-off productions and series productions
 - The program does not see any differences in how many times a diagram must be used
- Each Panel in a series production is documented in its own database with its own production number
 - All mounting tasks are logged in your own database based on production number
- Multiple boards can be mounted at the same time using the same project file
 - Since all cases have their own database with a new name, the same diagram can be used for several different cases
- There may be several people installing in the same project at the same time
 - Large projects often require several installers. This is possible as each installer's completed tasks are logged in the database with the user's
- Automation Service is an independent program in which you can NOT correct the drawing
 - The program is available in PC|Automation, which is used to make circuit diagrams. The program is also available in PC|Automation Service, which is a so-called workshop license and does not allow editing of the diagram.

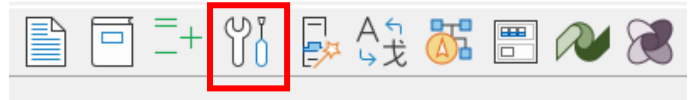
In the rest of this chapter, you can see and read how to work in the program.

You can also watch a series of videos on our YouTube channels – PCSCHMATICdk and PCSCHMATIC

GET STARTED

Start by opening the file that we want to work with. In this booklet, PCSmotordemo3 is used, as it contains examples of most of what the Mounting Assistant can be used for.

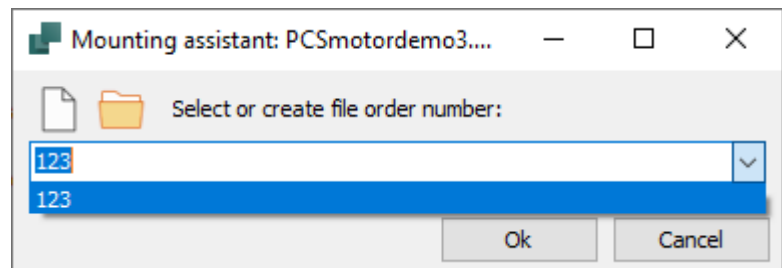
Start the Mounting Assistant using the shortcut Ctrl+M or by pressing the icon in the toolbar.



When you start the mounting, you select or write an order number for the task.

The same file can be used for many different orders: for example, we can use the selected demo file for both customer 1 and customer 2, but customers will each have their own order number.

There is a dropdown list of used order numbers for the active project file.



If the order number is not listed

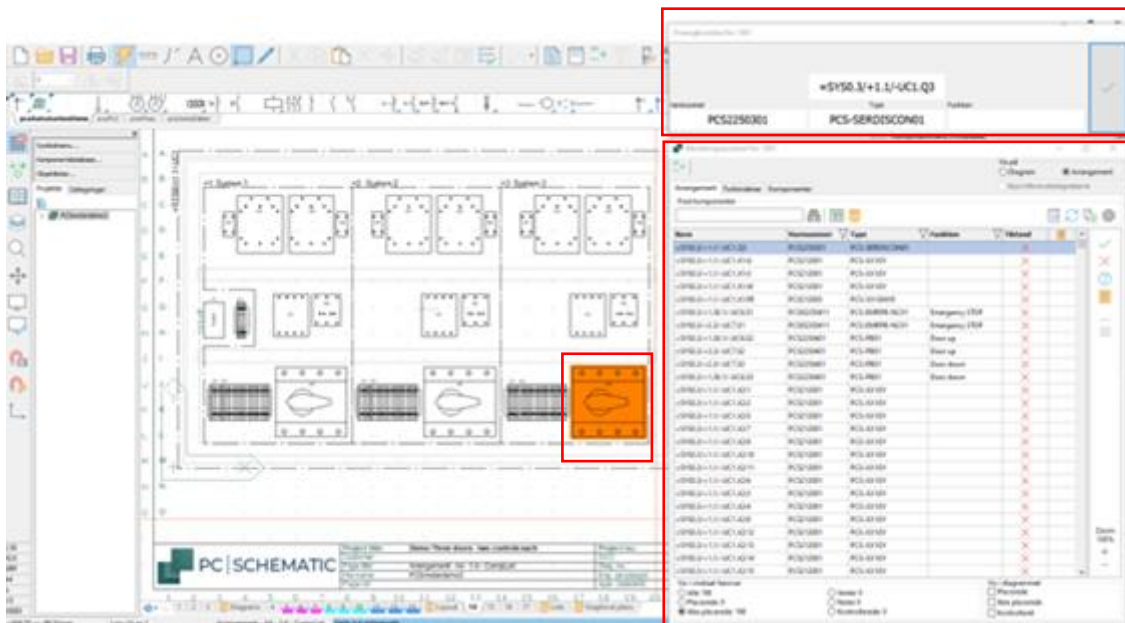
If the number you want isn't in the list, you can type it to create a new number.

If you use audit management or otherwise save your project files under new names, the dropdown list will be empty when the Mounting Assistant is opened in that file.

You can open the database used by the Mounting Assistant by opening the Folder icon. That way, you can find the different databases.

The Mounting Assistant opens

The Mounting Assistant opens on the Arrangement tab, simply because it provides a logical workflow in relation to the mounting. Read more about this tab in the next section.



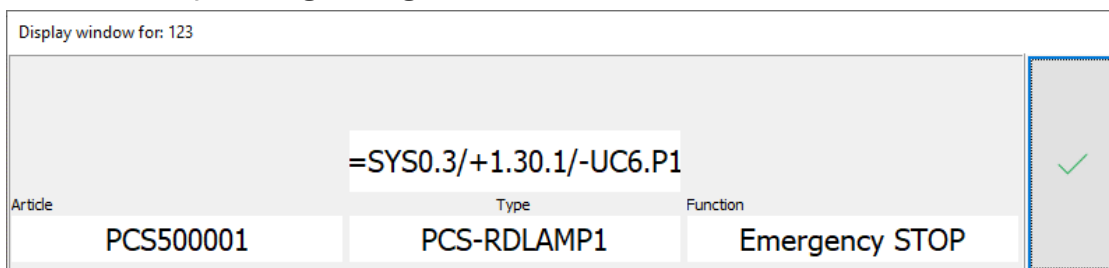
General information about the Mounting Assistant and functions

The Mounting Assistant consists of two windows, the Display window and the Mounting Assistant window.

Display window

The Display window—the small window in the upper-right corner—shows details about the object you selected in the list window below.

You 'mount' by clicking the big button, either with a mouse or on a touch screen.



You can choose what to display in the top two fields, see the Settings section below. The bottom left corner always has the diagram pop-up window, including accessories if you place your mouse over it.

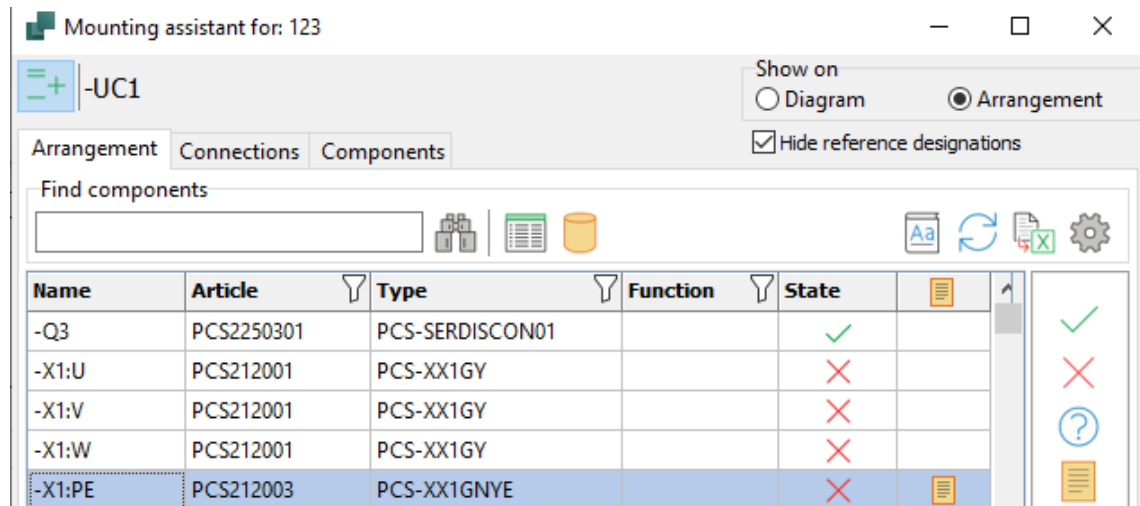
Zoom

You can zoom in and out on the page using + and – in the window and using + and – on the keyboard. That way, you see a section of the page when you select objects in the list.

The mounting assist window – list

The Mounting Assistant window itself displays a list of objects to mount. Depending on the tab chosen, there are different options, but they are so similar that most of it is explained below.

In short, it is often a long list, which means that you need to be able to sort and filter, which is described below.



Reference designations

If you have structured your project with reference designations, you can use these to filter your project, e.g. in the example you can choose to show only one board, e.g. - UC1.

If you use this feature, you can choose to see only the component name itself and hide the rest of the reference designation. It makes the overview more manageable.

Search

You can look for one or more specific components by entering some of the name in the window

View additional data from the Object List or the Database

You can press either of the two buttons to see additional data about the object in either the Object List or the Database.

Filters on Type, Article number, Function

You can put filters on applied values, such as a type designation. When you press the filter button, a list of applied values appears in the column.

Work instruction – 'How to do it when we install panels'

You can load a guide by clicking the button here. You decide where the document is located.

Refresh the list

The list is not automatically updated when there are multiple users. You can force an update to the list by pressing the button.

Export to Excel

The list can be exported to Excel, with the same content displayed on the screen.

Multiple users

If there is more than one user on the project, the names of all users are displayed at the bottom of the window.

Mark as placed/mounted

You can either acknowledge in the Summary window or in the list window when you process the objects. In this way, it disappears from the list of not placed.

Their state is displayed with a small icon in the list when viewing all objects.

Mark as not done

You can put an object back as not placed/mounted by tapping the icon.

Mark as waiting

You can mark an object to wait by tapping the icon. In this way, it disappears from the list of not placed.

Note

You can't change the diagram with the Mounting Assistant, but you can write a note on each object that can be opened later, see page 132.

Mark all as

You can select all objects in the list at once and switch modes by tapping the icon.


Mark as ignored

You can ignore a connection by tapping the icon.

Mark as partially mounted

You can mark that one end of a connection is mounted by selecting the end in the list and then clicking the icon, or by selecting the mounted end in the view window. The mounted end turns green, the non-mounted one turns yellow. See more about this on page 136.

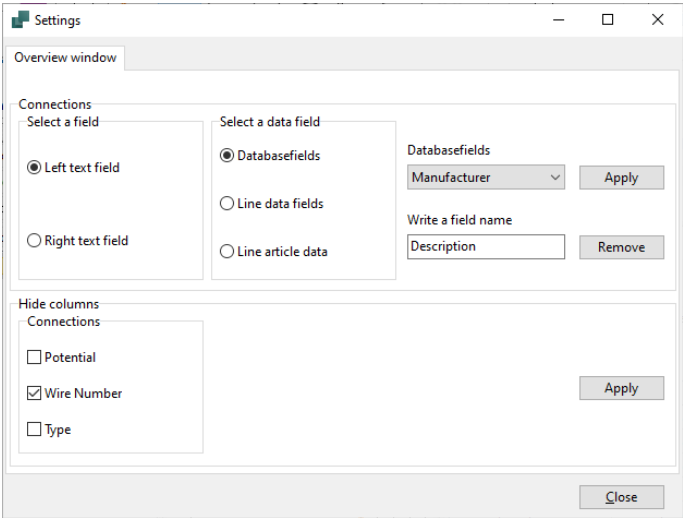
Other settings

You'll get these options when you click Settings: 

Here you can select fields for the Summary window, either from the database or symbol data fields, and give them a name.

Finally, you can choose to hide one or more columns in the Mounting Assistant window.

Here the manufacturer and description from the database are shown in the top two fields.



The screenshot shows a 'Settings' window with a title bar containing a gear icon and standard window controls. The window is divided into two main sections. The top section, titled 'Overview window', contains two columns of settings. The left column, 'Connections', has a 'Select a field' section with radio buttons for 'Left text field' (selected) and 'Right text field'. The right column, 'Select a data field', has radio buttons for 'Databasefields' (selected), 'Line data fields', and 'Line article data'. To the right of these is a 'Databasefields' section with a dropdown menu showing 'Manufacturer' and an 'Apply' button. Below that is a 'Write a field name' section with a text input containing 'Description' and a 'Remove' button. The bottom section, 'Hide columns', has a 'Connections' label and a list of checkboxes: 'Potential' (unchecked), 'Wire Number' (checked), and 'Type' (unchecked). An 'Apply' button is to the right of this list. A 'Close' button is at the bottom right of the window.

The text size adjusts automatically when you click Apply or drag the window.

THE ARRANGEMENT TAB

When using the Mounting Assistant, we recommend that you make a layout, also called arrangement drawing. Such a page makes it easy to communicate with the fitter, as it shows the location, size ratio, etc. of the individual components.

Why this tab

The purpose of this tab is to place the components in the panel and to check that the right components are placed.

The workflow – both in the program and in the description – is based on starting on this tab.

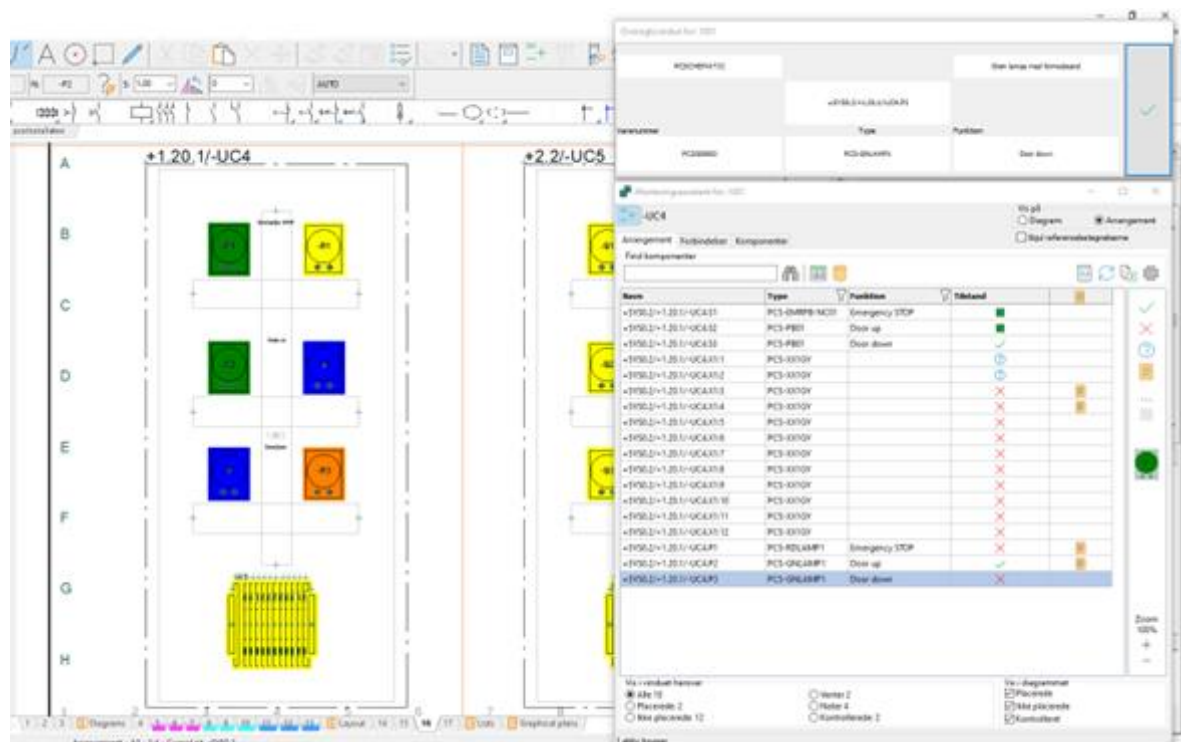


Diagram or Arrangement

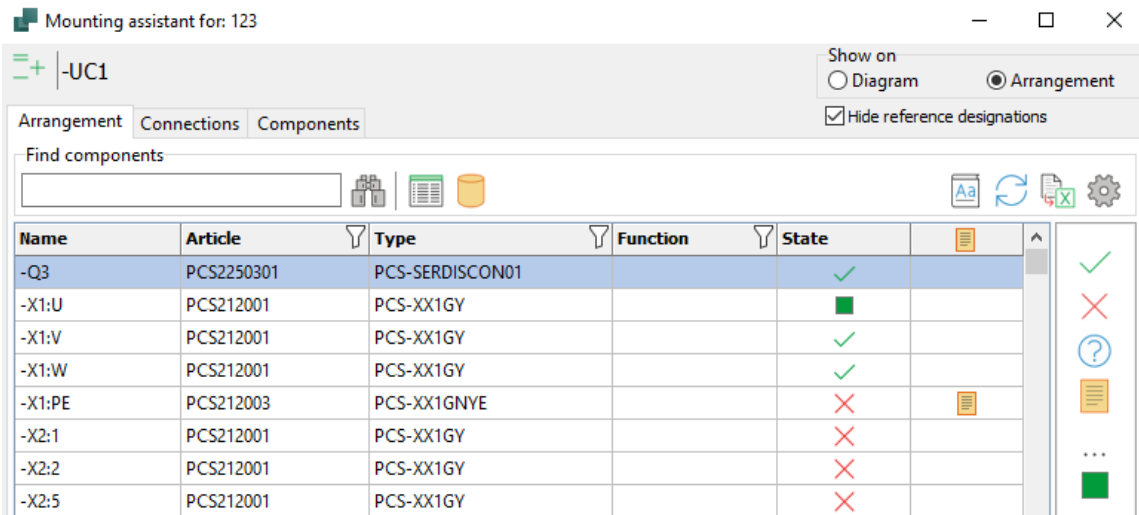
The list shows components from the project. They are listed in the same order as they appear in the diagram.

If you have not created an arrangement page, and/or if you have components that are not on this page, e.g. sensors, choose Diagram.

Use filters!

Workflow on the Arrangement tab

We see the workflow here in two steps, namely a clean location, then a check that it is the right component, e.g. right type and right markup.



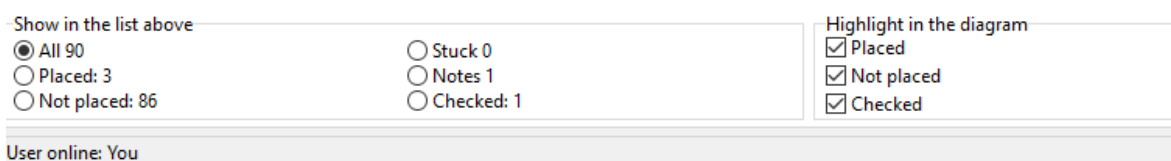
1. First, *place* the components in the panel. This is done optimally based on the arrangement drawing, which shows the physical location of the individual components.
 - a. You select the component by standing on it in the list, or by marking it on the page - either the arrangement or the diagram.
 - b. If the component has an image in the database, the image appears on the right side of the list.
 - c. To place the selected component, press ✓ either in the Display window or in the list window.

When the components are *placed* in the panel, you can *check* them.

- d. You can only check already placed components: check e.g. that they are correctly marked. When everything is ok, press ■
 - e. The big button has been switched to show this icon.

What do you see in the list

You decide what appears in the list: As you process the objects, they may disappear from the list, or you may choose to keep them. In some of the windows there are several options.



Display window

The display window—the small window in the top right—displays details about the component you selected in the list window below.

You 'mount' by clicking the big button, either with a mouse or on the touch screen.

Display window for: 123		
Manufacturer		Description
PCSCHMATIC		Kontaktor 3POL 1NO 1NC type1
	-Q2:1	
Article	Type	Function
PCS2250101	PCS-CON01	

✓

When the components are *mounted*, the button changes its appearance to *control*.

Display window for: 123		
Manufacturer		Description
PCSCHMATIC		Kontaktor 3POL 1NO 1NC type1
	-Q2:1	
Article	Type	Function
PCS2250101	PCS-CON01	

■

You can choose what to display in the top two boxes, see the Settings section on page 128. The bottom left corner has the diagram pop-up window, which also shows any accessories and symbol data fields if you place your mouse over it.

Highlight the components

Placed components are marked with Blue.

Not placed components are marked with Yellow.

Checked components are marked with Green.

The selected component is always marked with Orange.

The different colors appear on all parts of the component throughout the project, i.e. both in the diagram and on the arrangement.

Make a note

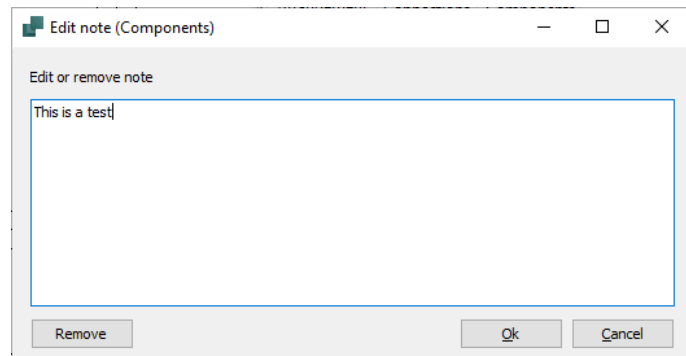
You can't change anything in the project when working in the Mounting Assistant! If changes are to be made, they must be made in Automation, and then the project must be read in again, see more page 140.

As a user of the Mounting Assistant, you can make notes instead.

Start by pressing the 

Then the window opens where you can write a message.

The message can be edited/deleted by pressing the button again.



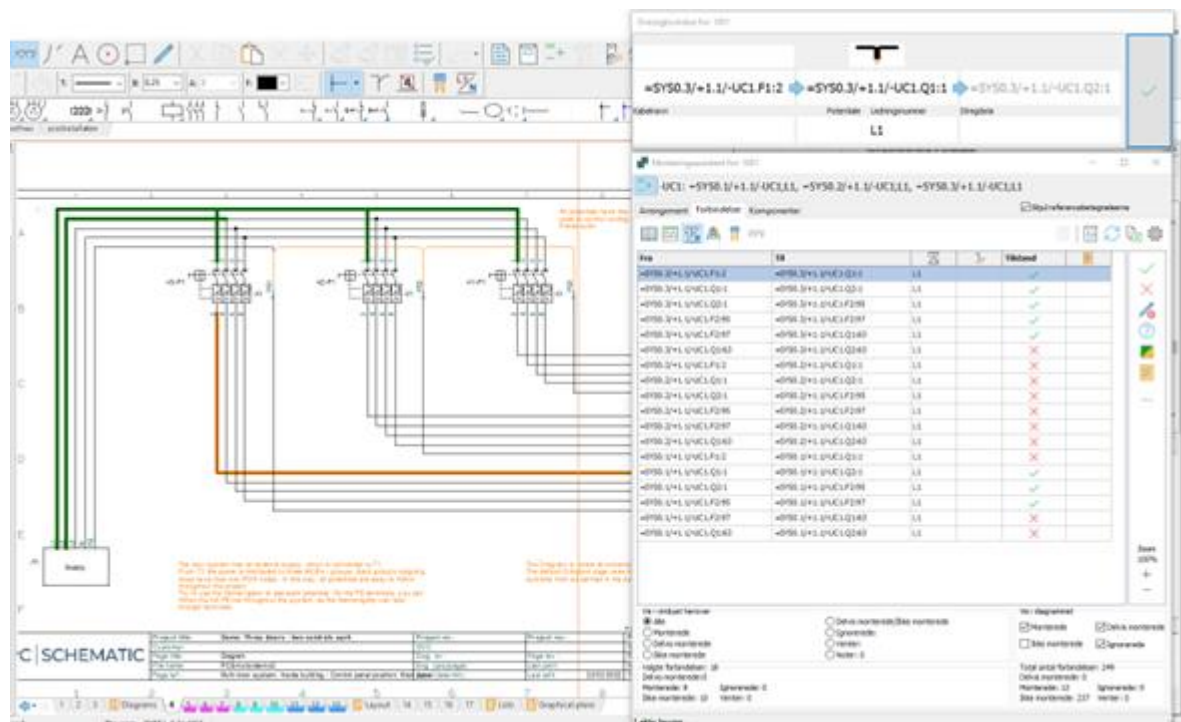
Once you're done here, or don't want any more right now, you can move on to the next tab.

CONNECTIONS TAB

When the components are in their place, it is time to connect them by mounting wires, cables, jumpers etc.

Why this tab

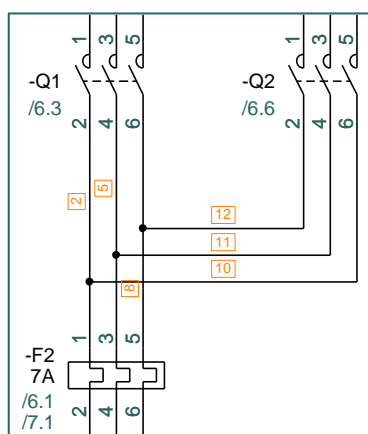
This tab helps keep track of the montage of connections in the project. Connections in the diagram are made with lines, but the line itself can cover many different types of physical connections: wires, cables, jumpers, hoses, etc.



When you mount connections between components, Mounting Assistant is based on the diagram. That is, it interprets the connections of the diagrams:

Where do connections go from and to?

If you have drawn with 'Dots', the drawing will be interpreted. The number of lines will be correct, but there may be differences in how the connection is understood. It is not necessary to draw the mounting correctly, but if there are doubts, individual connections should be made like this.

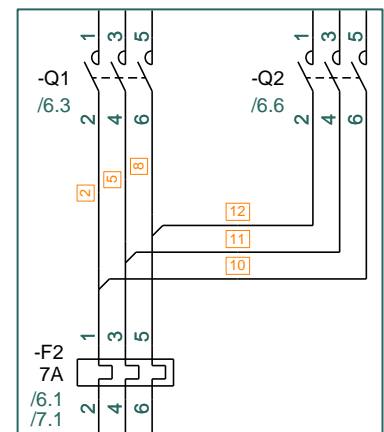


Two different drawing methods

There is a wire number on the lines where the wire is alone, and no number where there are two wires.

Left: 'Dots' does not provide optimal interpretation

Right: Correctly installed branches eliminate doubts about the connections



- All lines have different data and properties, and these you need to use to get through the list in a sensible way:
 - If a line has a cable symbol on it, is it a cable and can be found using the cable filter
 - If a line is connected to a signal symbol, is it a potential and can be found using the potential filter
 - If a line has color, item data or wire numbers, you can filter the list by this, etc.

Connections tab workflow

When you are on the Connections tab, all connections from the diagram come up in the list. The selected connection can be seen in the Display window.

Mounting assistant for: 123

=SYS0.3

Arrangement Connections Components

Hide reference designations

From	To	L1	L2	State
+1.1/-UC1.T1:L1	=SYS0.3/+1.1/-UC1.F1:1			Mounted
=SYS0.3/+1.1/-UC1.F1:1	=SYS0.2/+1.1/-UC1.F1:1			Mounted
+1.1/-UC1.T1:L2	=SYS0.3/+1.1/-UC1.F1:3			Not mounted
=SYS0.3/+1.1/-UC1.F1:3	=SYS0.2/+1.1/-UC1.F1:3			Not mounted
+1.1/-UC1.T1:L3	=SYS0.3/+1.1/-UC1.F1:5			Not mounted
=SYS0.3/+1.1/-UC1.F1:5	=SYS0.2/+1.1/-UC1.F1:5			Not mounted
+1.1/-UC1.T1:N	=SYS0.3/+1.1/-UC1.F1:7			Not mounted
=SYS0.3/+1.1/-UC1.F1:7	=SYS0.2/+1.1/-UC1.F1:7			Mounted
=SYS0.3/+1.1/-UC1.F1:2	=SYS0.3/+1.1/-UC1.Q1:1	L1		Not mounted
=SYS0.3/+1.1/-UC1.Q1:1	=SYS0.3/+1.1/-UC1.Q2:1	L1		Not mounted
=SYS0.3/+1.1/-UC1.Q2:1	=SYS0.3/+1.1/-UC1.F2:95	L1		Not mounted
=SYS0.3/+1.1/-UC1.F2:95	=SYS0.3/+1.1/-UC1.F2:97	L1		Not mounted
=SYS0.3/+1.1/-UC1.F2:97	=SYS0.3/+1.1/-UC1.Q1:63	L1		Not mounted

Show in the list above

☒ All
☐ Mounted
☐ Partly mounted
☐ Not mounted

☐ Partly mounted/Not mounted
☐ Ignored:
☐ Stuck:
☐ Notes: 0

Highlight in the diagram

☐ Mounted
☐ Not mounted
☐ Partly mounted
☐ Ignored

Selected links: 110
 Partly mounted: 3
 Mounted: 0
 Not mounted: 107

Ignored: 0
 Stuck: 0

Total number of links: 110
 Partly mounted: 3
 Mounted: 0
 Not mounted: 107

Ignored: 0
 Stuck: 0

1 User online: You

To get a sensible list, start by selecting a portion of the panel's connections. You can select connectors from the icons in the toolbar:

- Reference designations used in the projekt
- One or more selected diagram pages
- Selected components
- Potentials, i.e wires connected to signal symbols, e.g. phases and other supplies
- Cables, i.e wires that have cable symbols on them in the diagram
- Jumpers, i.e. wires that have the property 'jumper' in the diagram
- Line color, Type and/or wire number as shown in the diagram

In the image above, I have first chosen to concentrate on one of the services by using reference designations.



Next, I have chosen a cable with many conductors. The cables are displayed by pressing

Use filters!

Then you can concentrate on one part at a time, e.g. the main panel or one of the controls, as here. If it's a large panel, you may also want to filter so that you only have one type of connection.

What do you see in the list

You decide what appears in the list: As you process the objects, they may disappear from the list, or you may choose to keep them.

You can also highlight the connections, to have an overview in the diagram.

Show in the list above		Highlight in the diagram	
<input type="radio"/> All	<input type="radio"/> Partly mounted/Not mounted	<input checked="" type="checkbox"/> Mounted	<input type="checkbox"/> Partly mounted
<input type="radio"/> Mounted	<input type="radio"/> Ignored:	<input type="checkbox"/> Not mounted	<input type="checkbox"/> Ignored
<input type="radio"/> Partly mounted	<input type="radio"/> Stuck:		
<input checked="" type="radio"/> Not mounted	<input type="radio"/> Notes: 0		
Selected links: 321		Total number of links: 321	
Partly mounted: 0		Partly mounted: 0	
Mounted: 0	Ignored: 0	Mounted: 0	Ignored: 0
Not mounted: 321	Stuck: 0	Not mounted: 321	Stuck: 0

Highlight the connections

Mounted connections are marked with Green

Not-mounted connections are marked yellow

Partially mounted connections are marked with Blue

Ignored connections are marked with Gray

The selected connection is always marked with Orange.

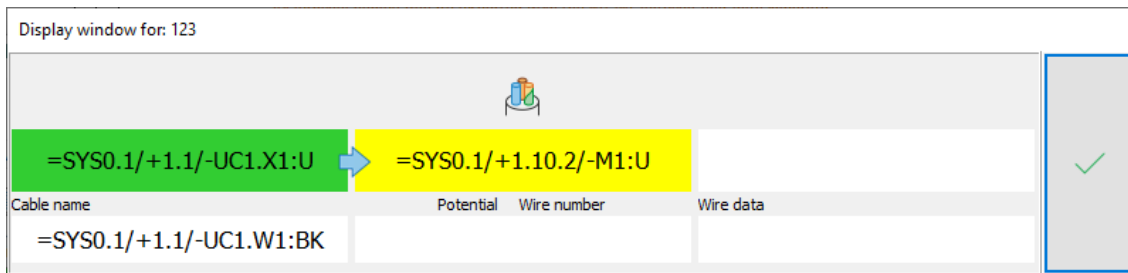
The different colors appear on every part of the connection, even when it spans multiple pages of your project.

Display window

The display window — the small window in the top right — shows details about the connection you selected in the list window below.

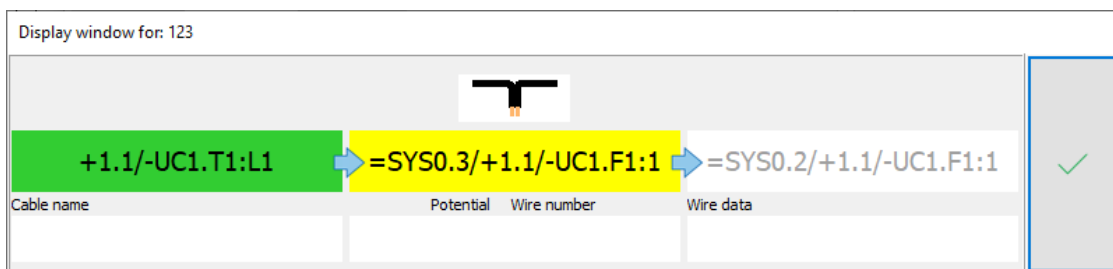
You 'mount' by clicking the big button, either with a mouse or on the touch screen.

You can choose what to display in the top two boxes, see the Settings section on page 128. The lower-left corner has the diagram pop-up window, including line data fields if you hover the mouse.



When connecting cables, the small cable icon appears at the top.

You can connect partially, i.e. you can connect at one end, which turns green. To do this, click on the end to be mounted.



In the case of wires, the color of the line appears on the icon.

Once you are done here, or don't want any more right now, you can move on to the next tab.

COMPONENT TAB

When the right components are placed in the panel and/or on the machine, and when they have subsequently been fitted with all connections, it is time to make the final finish.

Why this tab

The purpose of this tab is that there is always some finishing of the components when they sit in their place. For example, setting timers and motor protection, tightening parts, etc..

The desired post-processing may be shown in the project itself, alternatively it may appear in the database.

On this tab you can keep track of the fact that you have processed all components.

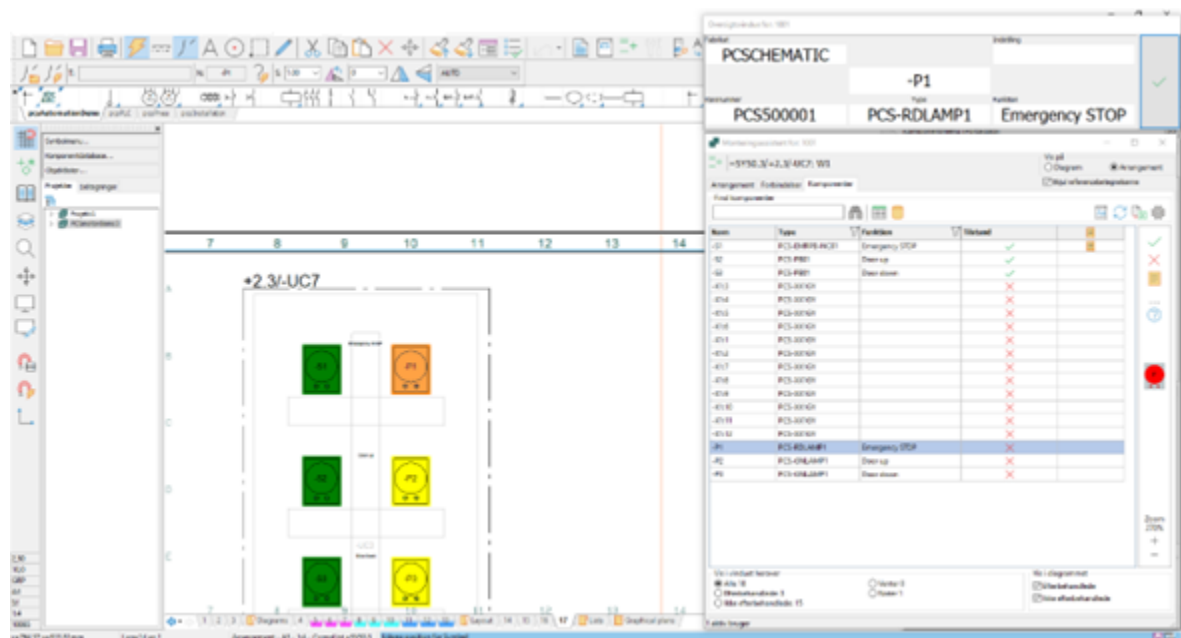


Diagram or Component tab

The list shows components from the project. They are listed in the same order as they appear in the diagram.

If you have not created an arrangement page, and/or if you have components that are not on this page, e.g. sensors, you choose Diagram.

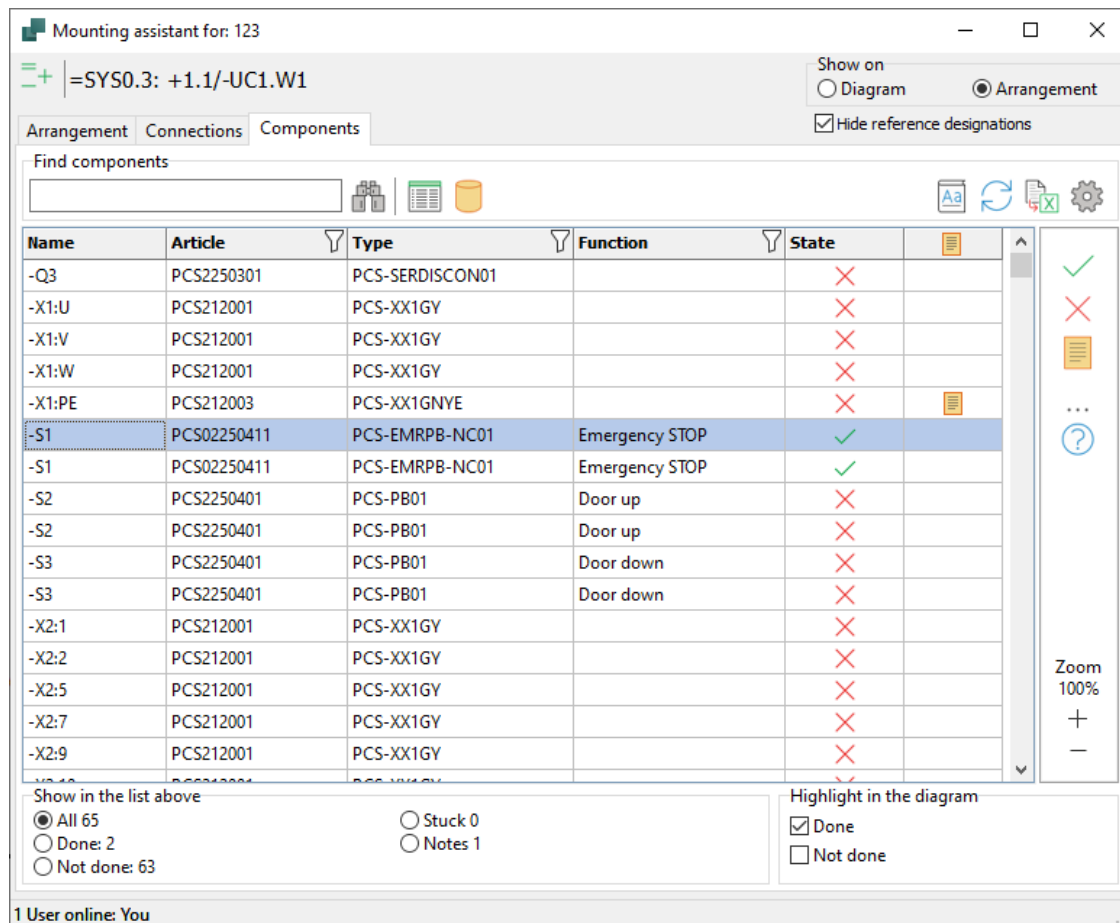
Use filters!

Then you can concentrate on one part at a time, e.g. one of the controls, as here.

When processing, the same thing often has to happen with components of the same type or function, so it can also be good to filter on this.


Workflow on the Component tab


On this tab there is a long list of all components of the project. The selected component is also displayed in the Display window.



Finishing components can be many things; setting timers, tightening torque, etc..

Information about post-processing can appear on the diagram in the form of e.g. text in a symbol data field or be hidden in the database, possibly on a data sheet.

Each company can create its own guide, which can be downloaded by pressing 

1. Select the component that you want to finish
 - a. Tap the Object Lister button, which displays data from the project about each component. 

Component	Name with Conns.	Article	Type	Function	Symbol	Page No.	Cmp. grou	Visible	Electrical	Mechanical	Accessory
-S1	1,2	PCS02250411	PCS-EMRP		PCS-50025	12	1	✓	✓	✓	
-S1	1,2	PCS02250411	PCS-EMRP	Emergency STOP	PCS-50025	17	1	✓	✓	✓	

If you right-click on the top line (the one with field names) you can set which columns to display.

1. You can also (or instead) press the database button to open the database and view its information about the selected component. If there is a data sheet, you can open it from here; if a field has been created for Finishing, you can see it here, etc.

The Record Viewer window displays the following information:

- ComponentID: 3079401E-6334-49EF-8895-2DDCDB0EC0DC
- Manufacturer: PCS02250411
- ManufacturersArticleNumber: PCS02250411
- ManufacturersGTIN:
- HistoricalArticleNumber:
- HistoricalArticleNumber2:
- Type: PCS-EMRPB-NC01
- usrDatasheet:

In the Automation program, you can decide which fields to display.
You can also create your own fields for the database in Automation.

Display window

The window displays details about the selected component. After processing, you acknowledge it and the component disappears from the list.

Display window for: 123

Manufacturer		Description
PCS02250411		Trykknapp slutte, type1
-S2		
Article	Type	Function
PCS2250401	PCS-PB01	Door up

A green checkmark button is located on the right side of the window.

What do you see in the list

You decide what appears in the list: As you process the objects, they may disappear from the list, or you may choose to keep them.

You can also highlight the components so that you have an overview of the project.

Options for displaying components:

- Show in the list above:
 - ☐ All 65
 - ☐ Done: 2
 - ☒ Not done: 63
- Highlight in the diagram:
 - ☒ Done
 - ☐ Not done

Additional options: ☐ Stuck 0, ☐ Notes 1

1 User online: You

Highlight the connections

Finished components are marked with Green.

Non-processed components are marked the Yellow .

The selected component is always marked with Orange.

The different colors appear on all pages of the project, both diagram and arrangement.

WHEN THERE ARE CHANGES IN THE PROJECT

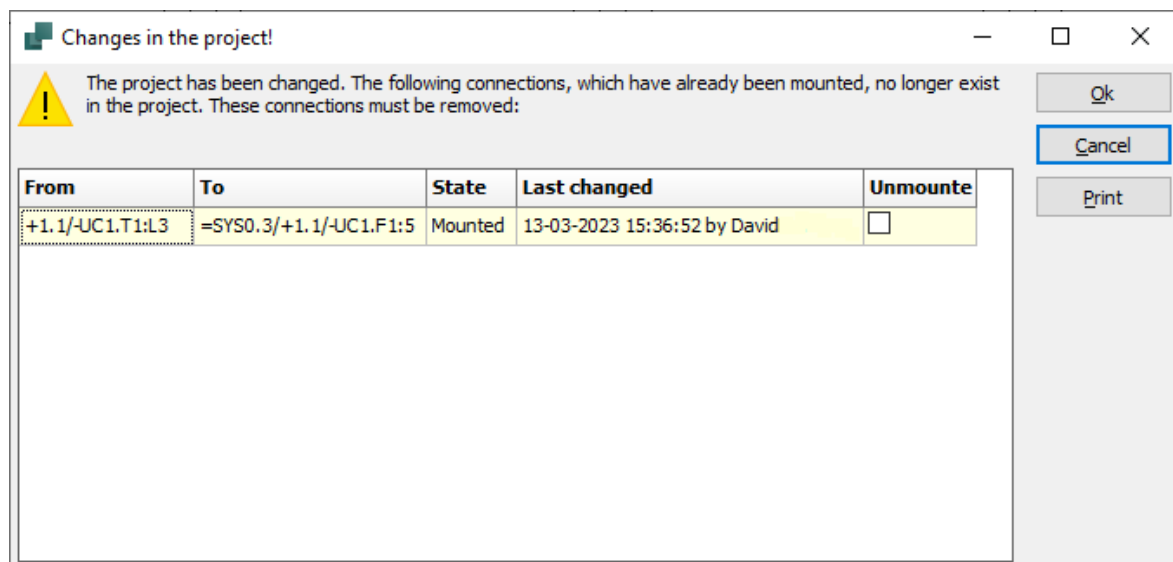
The Mounting Assistant is a tool where you can *only* see the project and acknowledge the installation of components and connections carried out.

You can't change, correct or delete anything in the diagram! This means that if you as an installer have questions or comments about the project, you can make notes that the constructor can later see.

If there are changes in the project

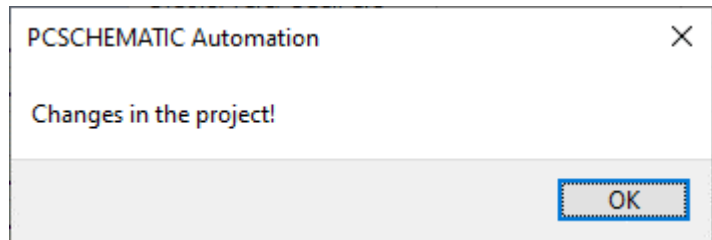
If the constructor has made changes to the project that you are installing, the following happens:

- The constructor has created several pages, or has deleted something that you have not yet come to
 - Nothing happens except that you may notice that there are more or fewer completed tasks at the bottom of the Mounting Assistant window
 - This just means that the installer always has the latest version of the project!
- The constructor has deleted something that you have already mounted
 - When you next open the project, you'll get a message saying there are removed connections.
 - Again, this just means that the installer always has the latest version of the project!



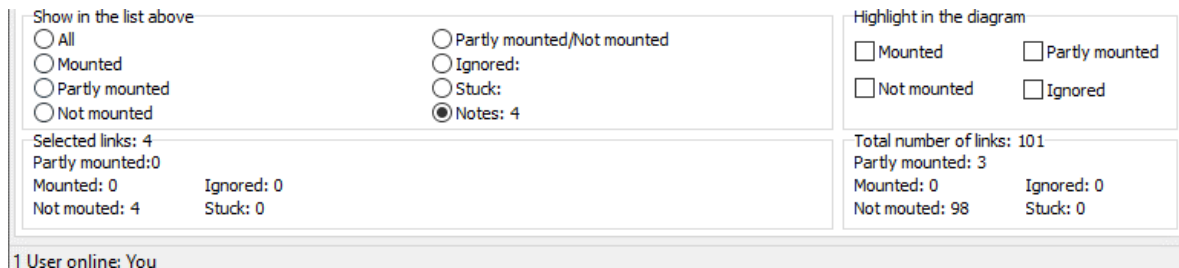
Making changes with the Mounting Assistant open

If you have the Mounting Assistant open in Automation, you can change the project, but it will forcibly close the Mounting Assistant with this message.



If the installer has made notes

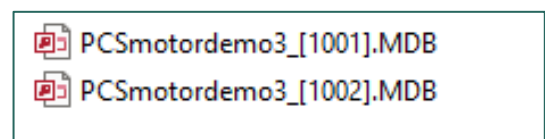
The constructor is not automatically notified of notes etc., but must keep an eye on this himself. At the bottom of the Mounting Assistant window you can find all objects with notes.



How is data stored from Mounting Assistant

The Automation project itself (*.PRO) is not affected when working with the Mounting Assist. Instead, data is stored in a database, as it is precisely data about the assembly that is interesting in this context. The database is stored in the same folder as the project.

Naming follows the rule: Filnavn_[Order number].mdb.



You can open the Access database and print various reports if you wish; Reports that you have to design yourself.

When the project changes its name

If audit management is used, the project changes its name. But the assembly project itself is still stored in the 'old' database. In this case, you choose the old database (above) in the new project.

MY NOTES

Here are a few pages where you can write your own notes.











