MOTOR CONTROL

Made by PCSCHEMATIC A/S

June 2020

PREFACE

This booklet shows how to make a simple project using PCSCHEMATIC Automation.

PCSCHEMATIC has a series of booklets covering different subjects:

When you read the booklet, you can follow the example. When you have worked your way through this you have tried the most common functions in the program, and you will be able to make your own project.

All examples are based on demo components from the PCSDEMO database.

The examples are made in Automation 14. With a few exceptions all examples can be made in earlier versions.

There are text highlights in the book to indicate further explanations about some of the program functions. You don't need to read these explanations to make the project.

The booklet guides you through the work processes of creating the documentation of a small automation project using the templates, functions and components that are included in the program PC|SCHEMATIC Automation and its database.

The following subjects are covered:

- Use the basic drawing functions
- Use the database to select components
- Use the Router to make automatic connections
- How to place terminal and cable symbols
- How to make an arrangement drawing

Components used in the project are demo components that are in the database included in the program.

The finished project looks like PCSMOTORDEMO1.

Table of Contents

Preface 3

Power circuit diagram 7
Control circuit diagram 7

Start a new project 8

Power Circuit Diagram 9

Place a motor 9

Place the next motor directly from the database 11
A few tricks to save time..... 13
Connect the motor to the phases 14
Add motor protection – Thermal overload 16
Place contactor – 3-pole contact 17

Control Circuit diagram 18

The Show available function 18

Contactor – coil and control contactors 19

Place a terminal row with article data from the pick menu 21

Terminal row direction 21

Place cable 22

Cable direction 22

Relation between the pages 23

Arrangement drawing – Layout plan 24

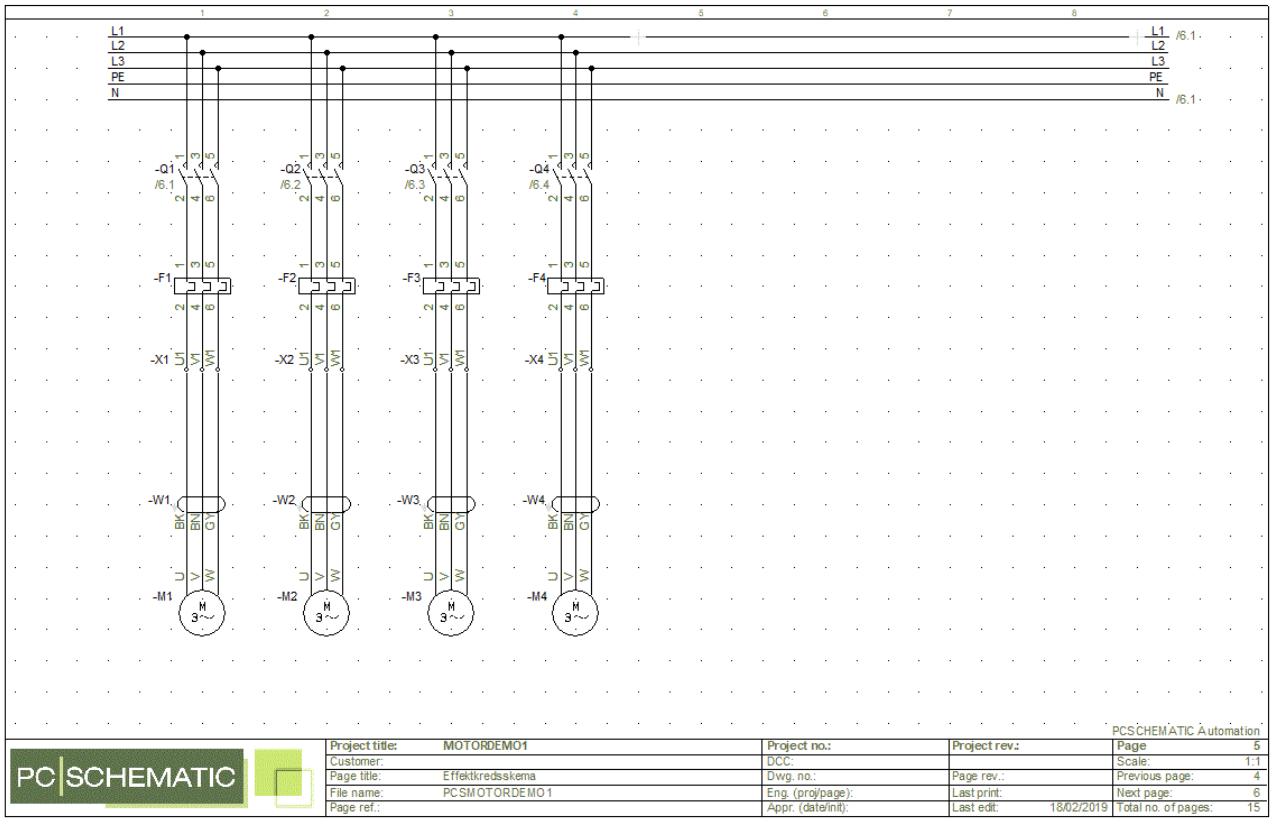
Measurement 25

Lists in the project 26

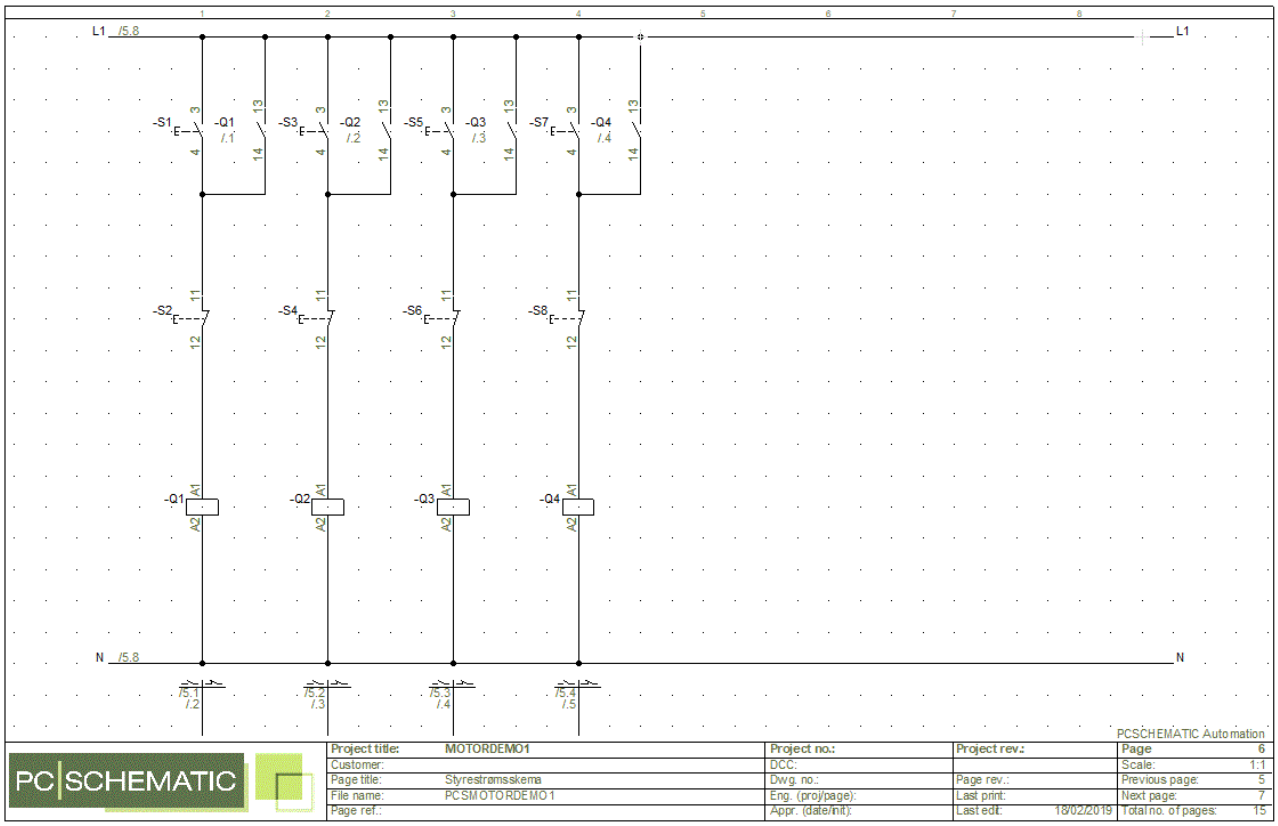
The last finish... 27

Add/Insert new pages 27
Delete excess pages 27
New page numbers 27

Power circuit diagram



Control circuit diagram

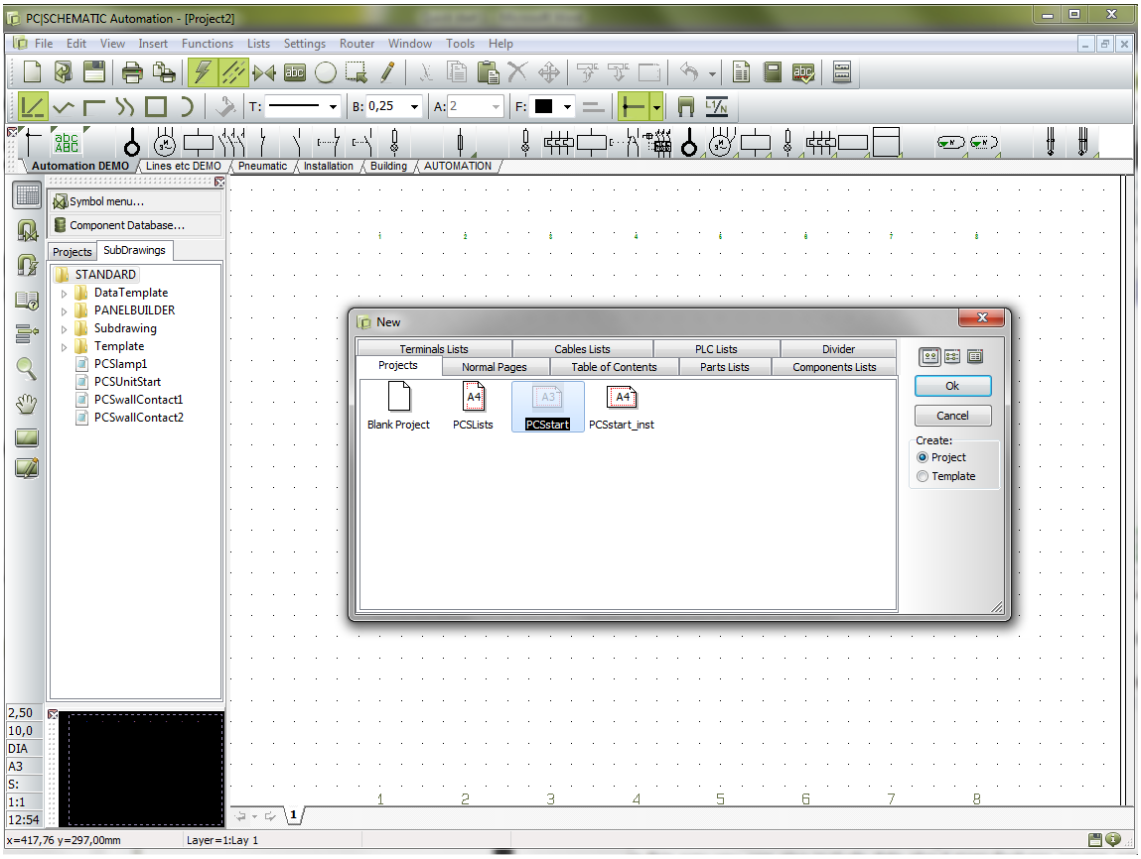


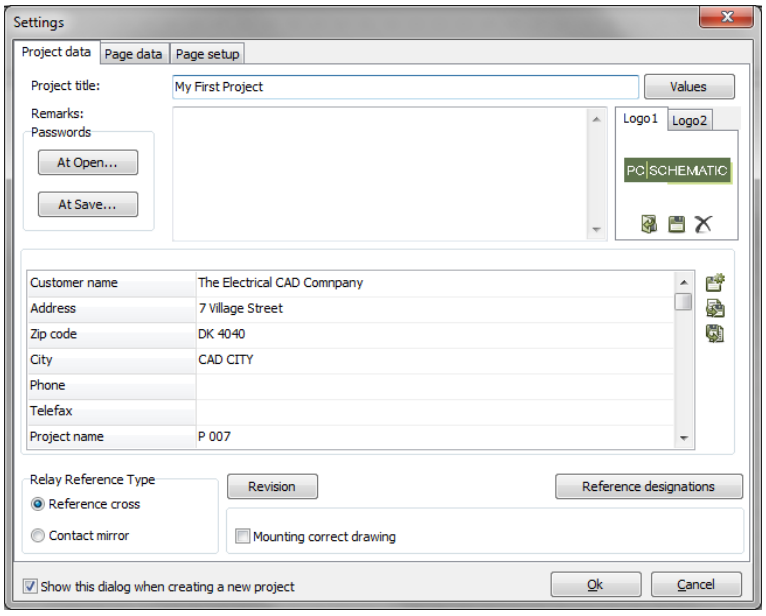
START A NEW PROJECT

You are now about to start a new project that has diagrams as those you can see on page 5 in this booklet.

1. Go to Files|New or click the icon New in the toolbar
2. Select PCSstart

PCSstart is a project template that contains the pages that make a project.





When you press <OK> this dialog will show.

Type “My first project” in the project title field.

You can also type in the other data now. If you want to do this later, you can enter the dialog by pressing the Project Data button in the toolbar.

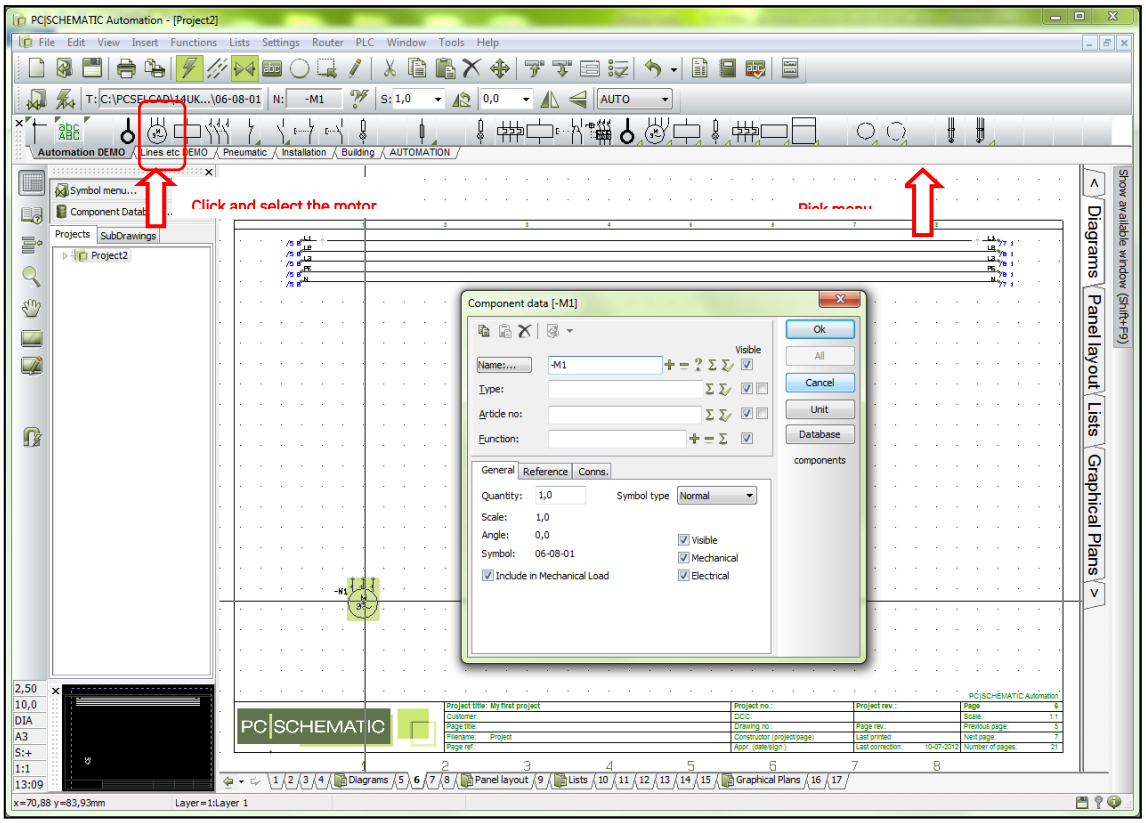
When you press <OK> you can see the title on the front page of the project.

POWER CIRCUIT DIAGRAM

There are different ways to start a project. If you know which diagrams you are going to make, the easiest way is to start with drawing the diagrams. Afterwards you can make the panel layout. This is the method used in the example below.

Go to page 6 in the project where you will find a page intended for power circuits. The page has been prepared with three phases and PE and N.

Place a motor



Start by placing the symbol for a motor: Select the symbol MOTOR in the pick menu Automation DEMO.

The name ”–M” pops up automatically, and your job is to give it a unique number. One way of doing this is by clicking the which will select the next available number.

In the dialog you have the following options now:

- type in type and article number directly
- in this way the data will be included in parts and components lists,

or

- press the Database button to enter the component database,
- in this way you can also include data about manufacturer, source, prices etc. in you parts and components lists.

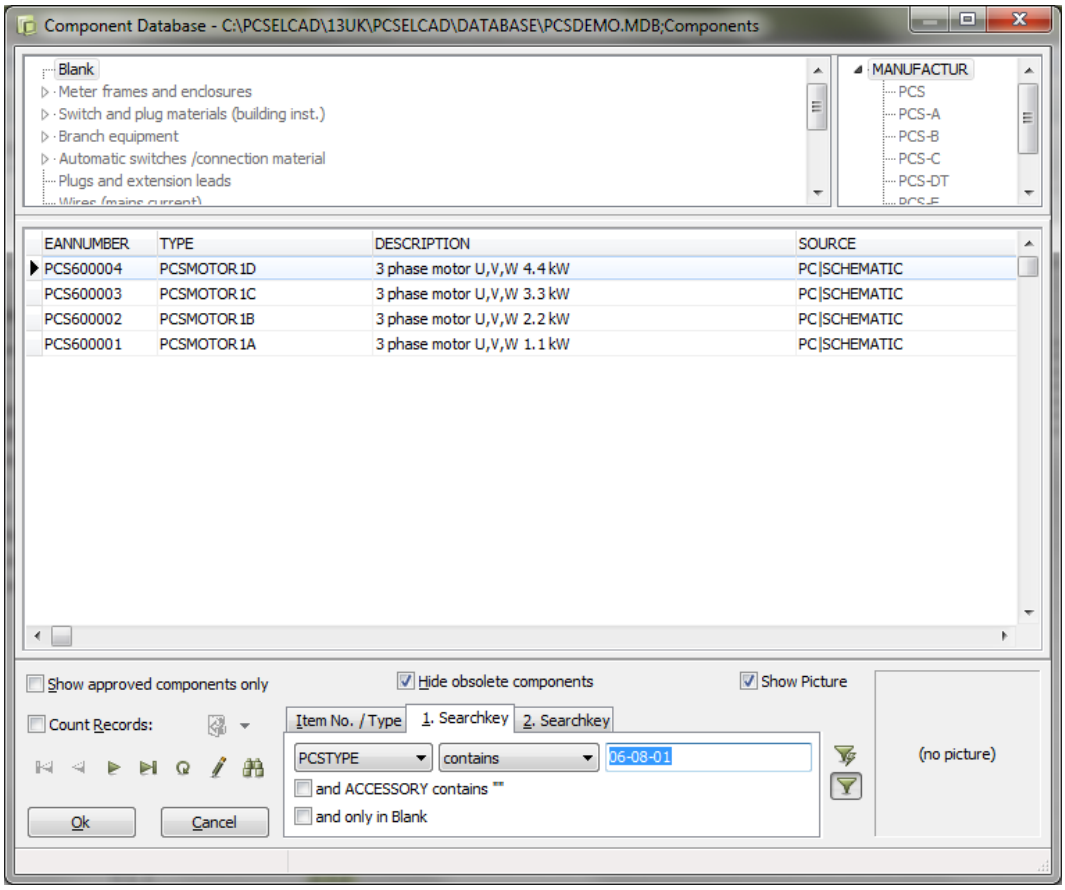
In this example we only uses components from the database. So here you press the database button. When you select the database button the program starts searching for *components* that use the *symbol* you have placed in the diagram:

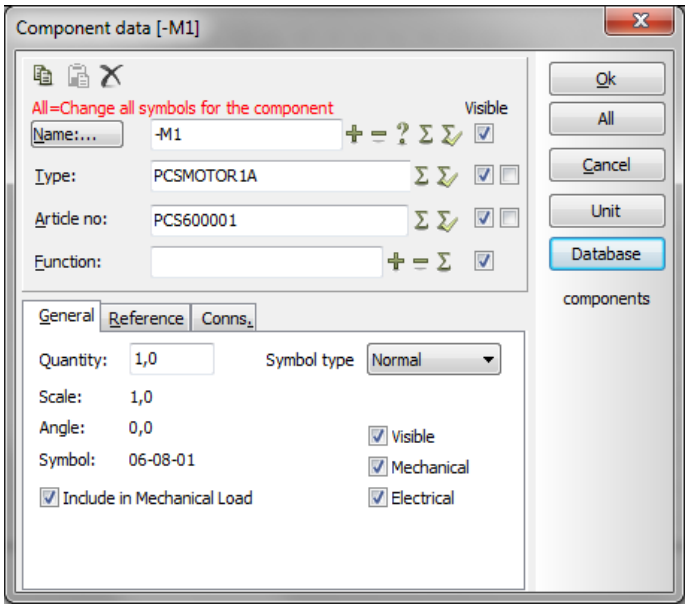
Components are physicala articles that can be identified by type and article number

Symbols illustrate electrical *functions*.

The electrical *functions* of a *component* is illustrated by one or more *symbols*.

The database contains *components*, and various data on these component, among those *symbols* for electrical functions and *symbols* that show their physical layout.





Here you can choose from four different motors. Select the motor with 1.1 kW. When you press OK you return to Automation and the dialog looks like this: Type and article number has been filled with data from the database, and the terminal names of the motor have also been added.

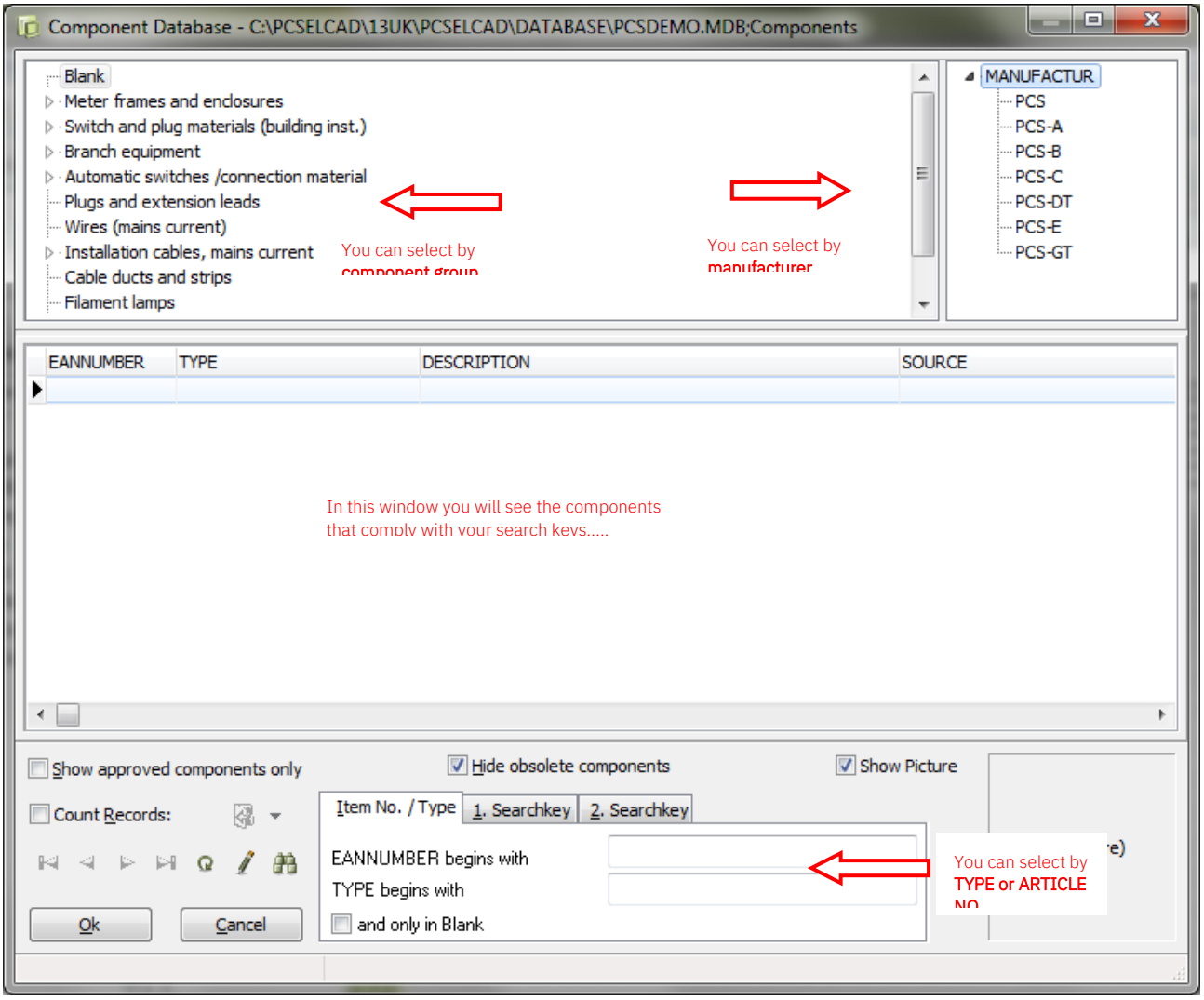
It is a good idea to press the ALL button when you change data. In this way you can ensure that all symbols of the component – electrical or chanical - are updated simultaniously.

Place the next motor directly from the database

You can also go directly to the database to fetch your components.

Click the button [Component Database] on top of the explorer window or press [D] to open the database.

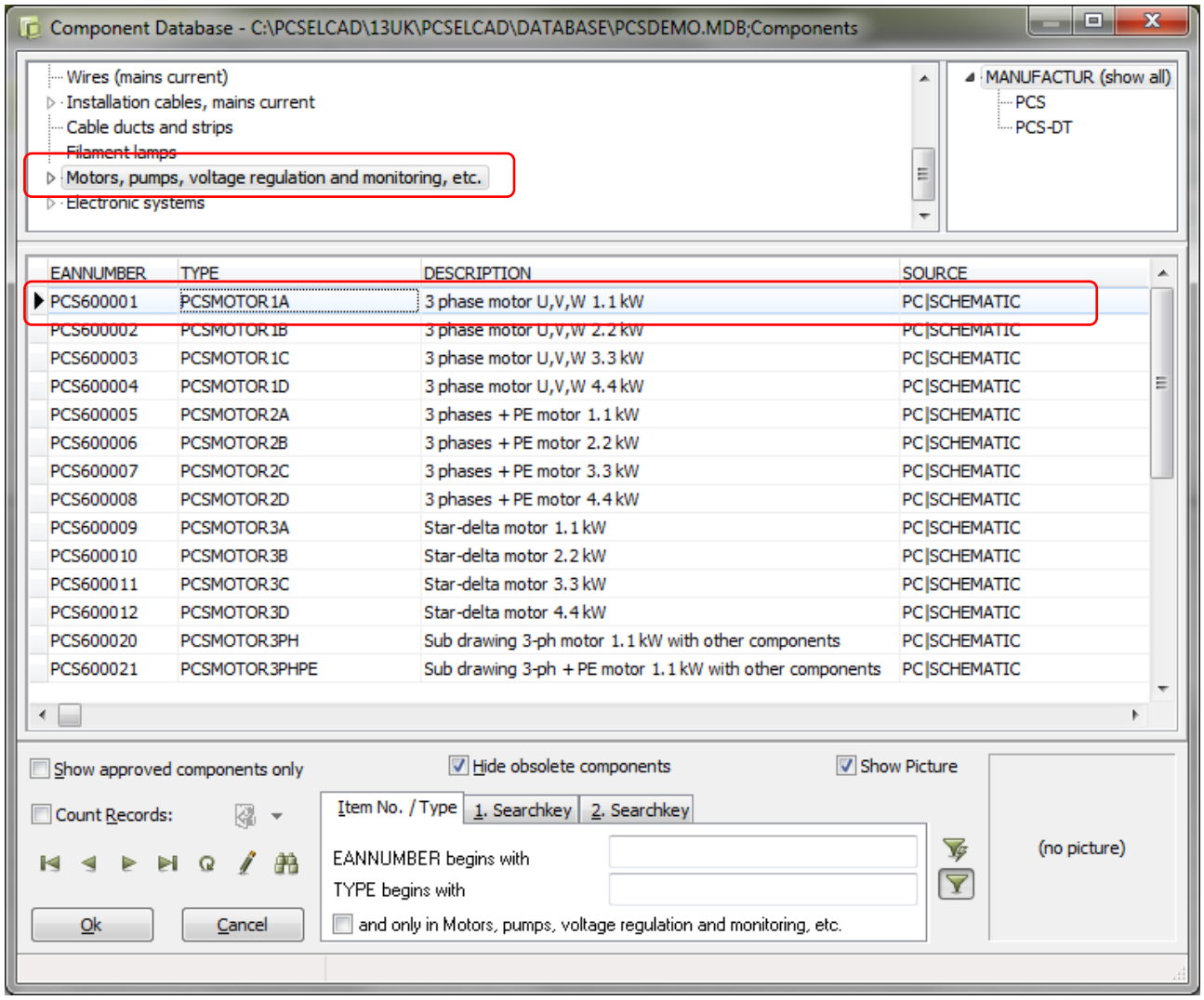
[D] is the shortcut for the Functions|Database|Database menu



You enter this window from which you can select components by

- manufacturer
- article group
- type
- article no
- or by any database field

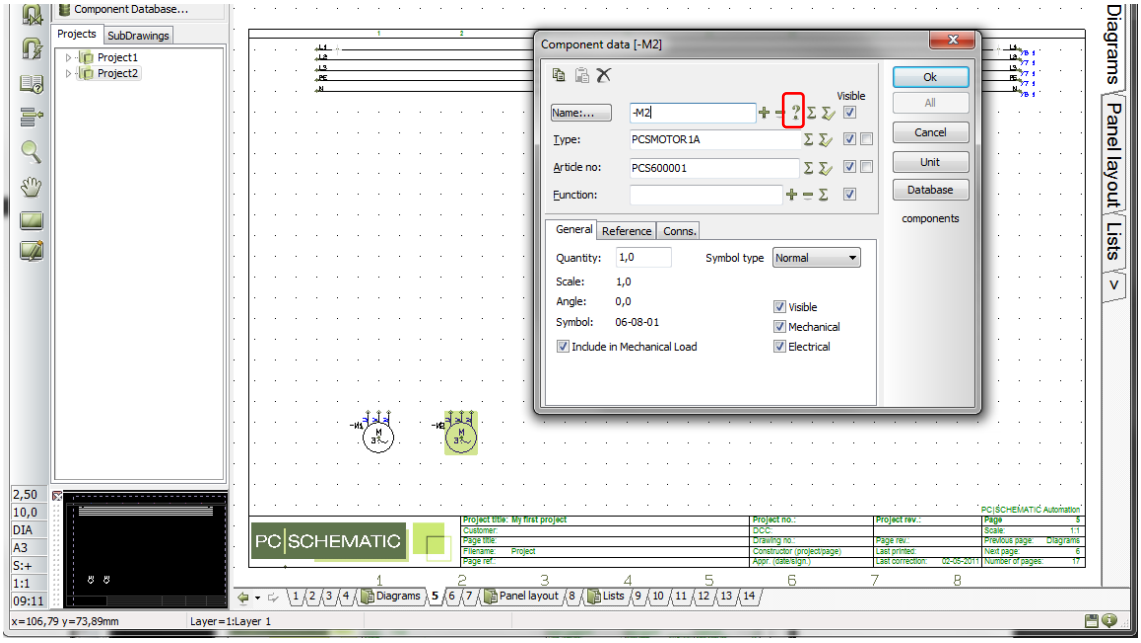
Select the article group *Motors, pumps……*



Now you can see the motors that are in the database

- Select a 3-phase motor 1,1 kW
- Double-click – or press OK

You return to Automation, this time with the motor symbol in the cross hairs, so all that remains doing is to place the symbol.



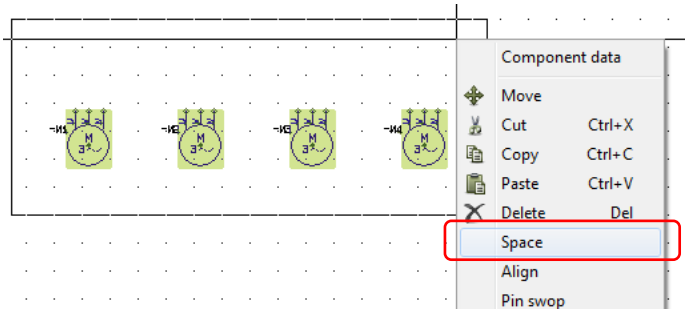
When you place the symbol on the page, the dialog pops up, now containing data about type and article no.

Press to get the next available number.

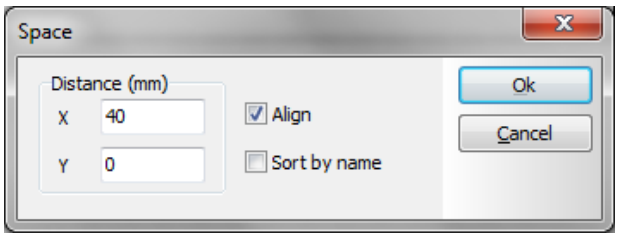
Press OK, and you are ready to connect the motor with electrical lines.

A few tricks to save time.....

Return to the database and select the last two motors and place them on the page. Do not spend time to space or align them manually.



When all four motors have been placed in the project, you mark them by dragging the mouse across them. Inside the “box” you right-click and select <Space>.

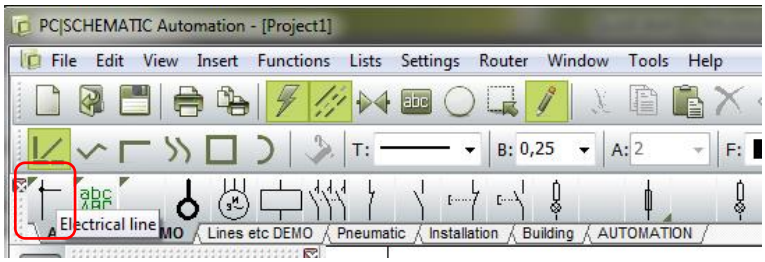


Use the mouse to point where you want to place the first motor – preferably centered in one of the current paths – and select a distance of 40 mm in the x-direction. Select “Align” and “in name order”. Press OK. In this way you have aligned and distributed all four motors.

Connect the motor to the phases

The motor must be connected to the phases. This is done with an electrical or rather conducting line, which makes an electrical connection between the phases and the motor.

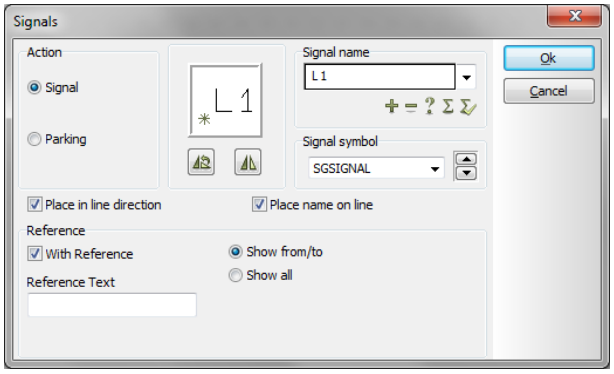
The simplest method is to select the electrical line in the pick menu:



When you press the line here, you also activate the pencil in the toolbar. And in this program, you can only draw when the pen is active ☺.

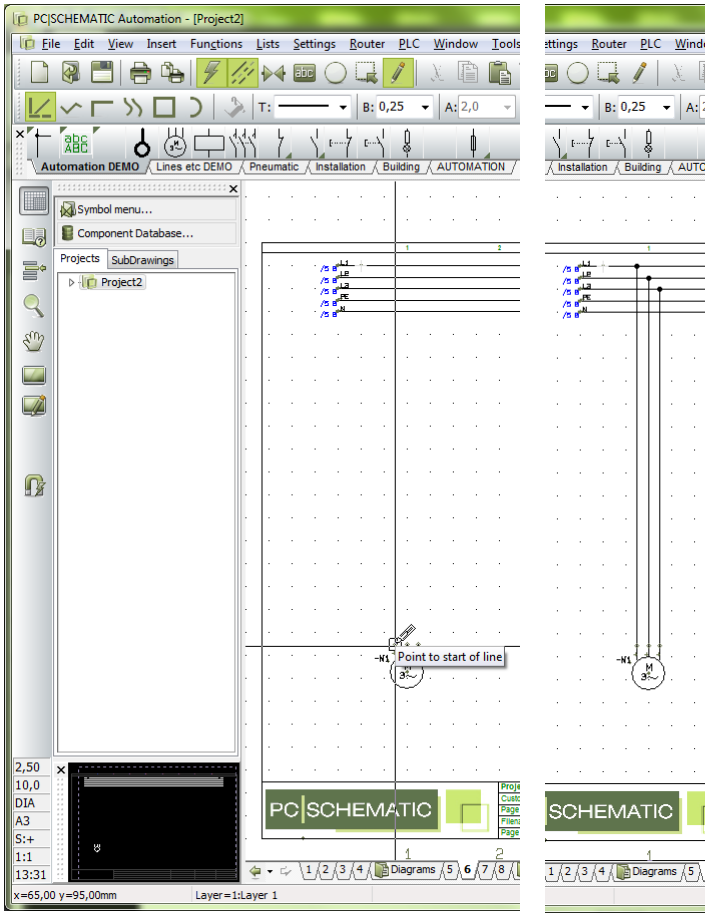
If you want to choose between the various drawing commands you can read more about them in the first Quick Start book. See in the Drawing Objects section.

Start by connecting the line to the motor, or rather to the connection points on the motor. When you see the ”small” rectangle, you have selected the connection point.



If you miss the connection point, this dialog will pop up. Press Esc and try again.

Click here and move the pen to L1 and click again. The line is ended automatically when you click L1 and a black “dot” will appear. This “dot” indicates that you have an electrical connection.



Continue by connecting the other two phases to the motor.

The lines are also conductors in a cable. How to add this information to the diagram, see page **Fejl! Bogmærke er ikke defineret.**.

TIP!

The program has an “automatic lines feature” also called The Router. You can use the router to connect the motor:

Click one of the motors, press the button [½]. Now you can see how the router will connect the motor.

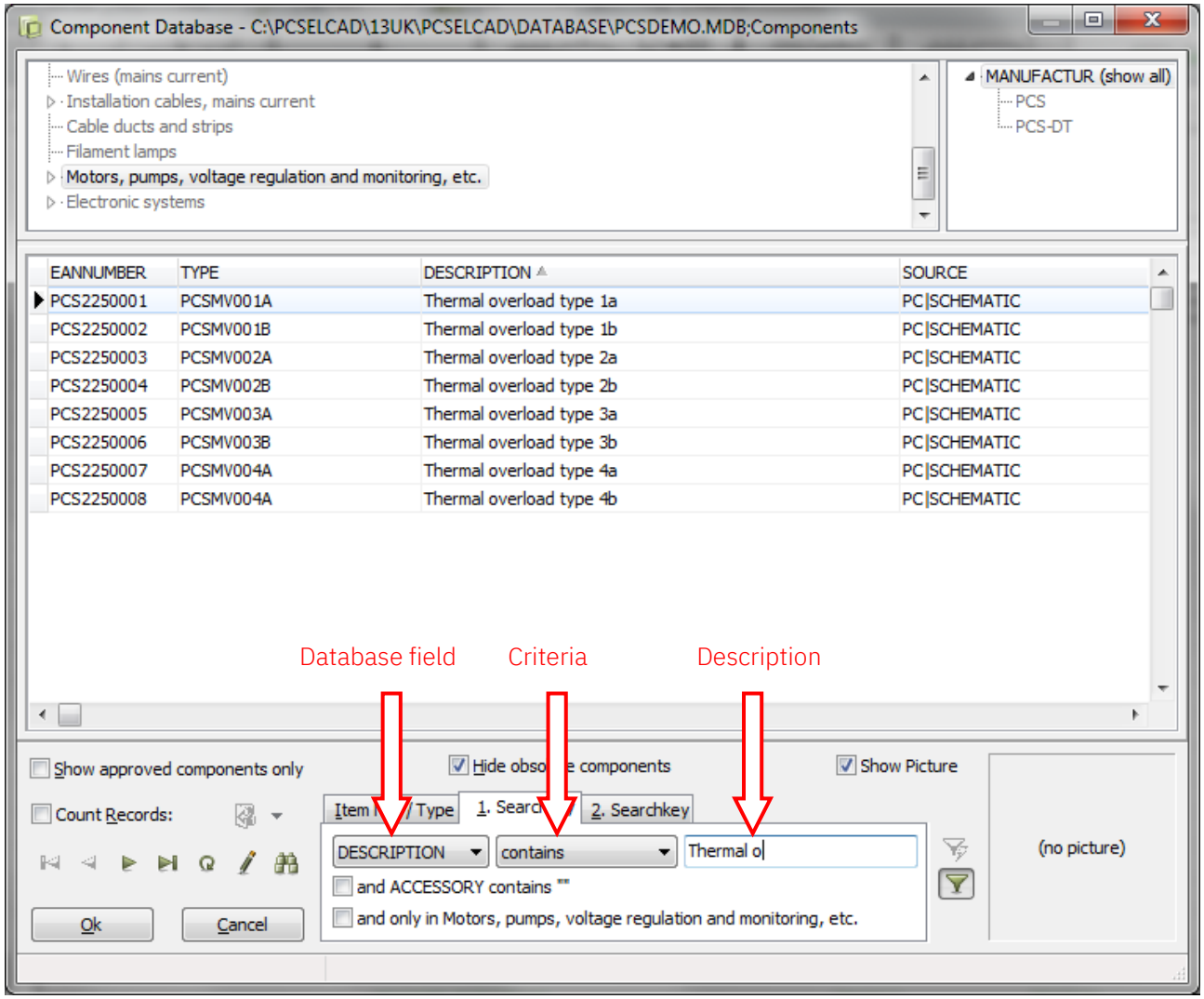
If you want other possibilities use [<] (jump to next connection), [3] (swap connections). When the connection is as wanted, press [1]. [1] makes the proposed connections.

You de/activate the router by pressing [½].

The router works in the Symbol drawing function and in the Line drawing function. You can read more about the router and its functions in the manual.

Add motor protection – Thermal overload

Press [D] to go to the database – or press the button above the explorer window.



If you do not know type or article no. of the component, you can search in the DESCRIPTION data field:

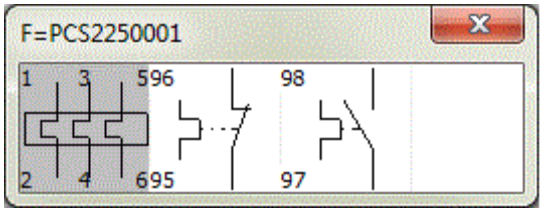
Start to select the tab ”1. Search key”

Select the database field DESCRIPTION.

Select a criteria, e.g. ”contains”.

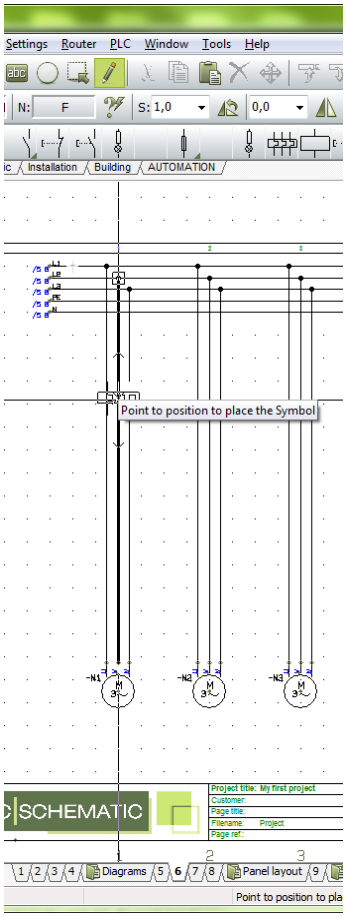
Start typing “Thermal overload” and you will see that the search routine starts, and the answer is seen in the window. Select type 1a.

In Automation you get this pick menu:



It shows all electrical symbols belonging to this component. The symbols have numbers/names on the connection points. The numbers are the same numbers as can be found on the component’s terminals.

Click on the first symbol – you will then place it in the power diagram:



When you place the symbol, the line will open automatically.

In the dialog you must name the component by using the . The component will then be “F1”.

.

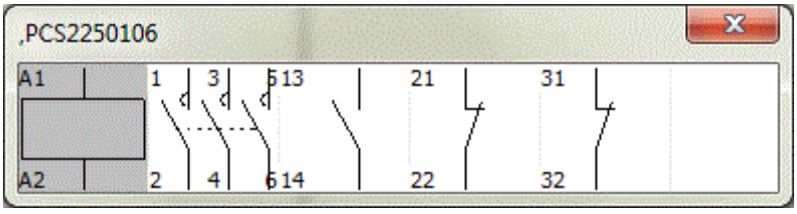
Place contactor – 3-pole contact

Return to the database. Use [D] or the button above the explorer window.

Select the tab ”Article no./Type”.

Type PCSCON06” in the *Type* field.

Click OK.



You now get this database pick menu. The component contains all those electrical symbols. Place the 3-pole contact (2nd from left). Call it –Q1.

Repeat the last two instructions, so that you place motor protection and 3-pole contacts on all motors.

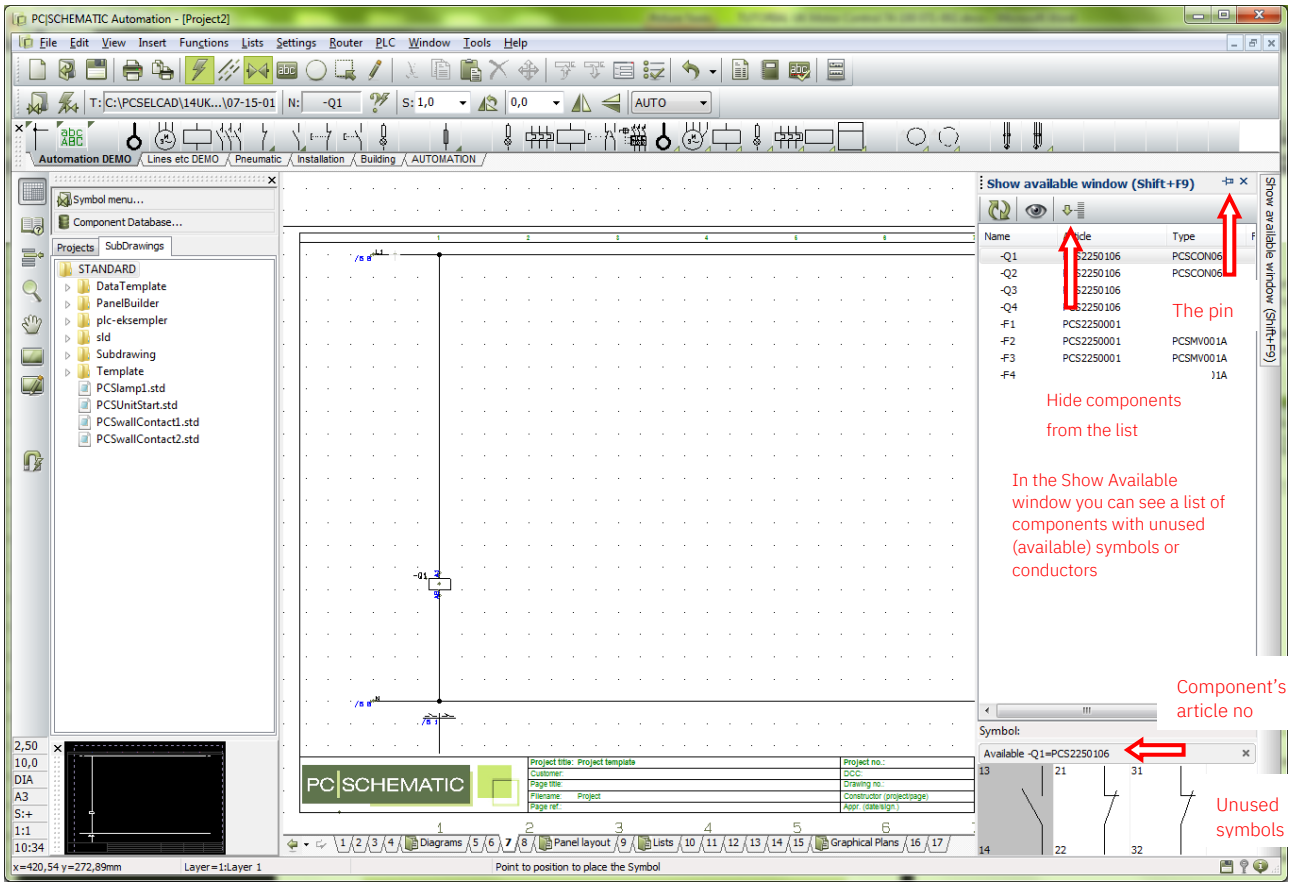
CONTROL CIRCUIT DIAGRAM

Go to page 7 in the project file.

On this page you are going to place the remaining symbols from the motor protection and the contactor, that is the symbols that belong in the control circuits. You can see the finished diagram on page 5 in this book.

The Show available function

Let the program help you to remember the components that you have placed in the project already.



In the right side of your drawing area you have e button **Show available window**. When you press this button you will get a roll-over menu containing all names of components with unused symbols (for cables: unused conductors). You can use the pin to keep the window open at all times.

You can sort the list by pressing the top of each column. You can hide components in the list by pressing the Hide button, and you can filter the list – see the options in the right-click menu.

BEWARE!!!!

If you return to the database to find the contacts for the control circuits you will do wrong!! This will be a little like going back to your stock, fetch another contactor in order til fit the control contactors!! Don't do that, instead you use the "show available" function to retrieve the functions of the component that you have not yet connected.

Contactor – coil and control contactors

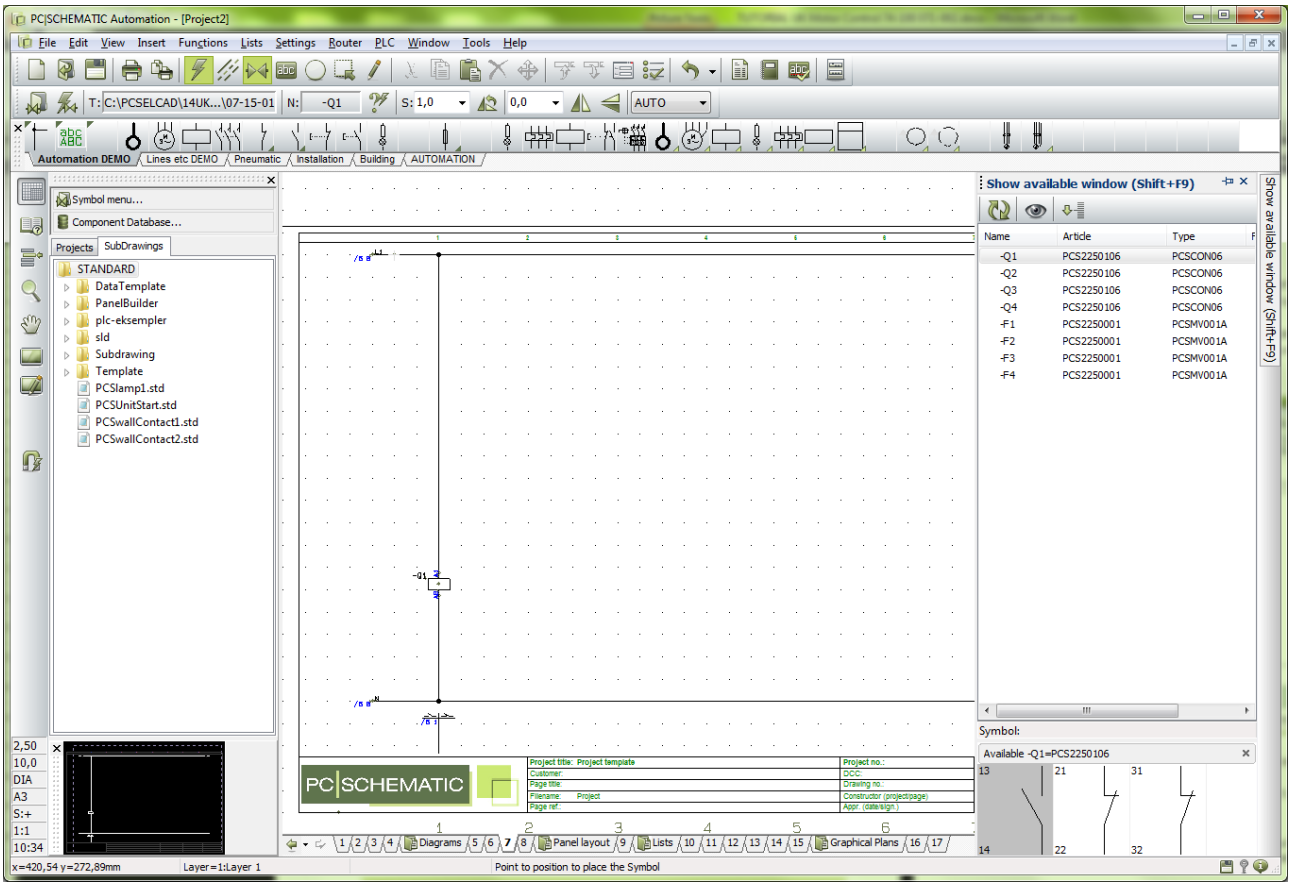
Start by placing the coil –Q1.

Double-click the name in the list to see the available symbols for this component.

In the bottom of the window you can see a smaller window with unused symbols for the selected component.

Select the 1st symbol – the activating coil – and place it on the page.

When you place an activating coil, it automatically comes with a reference symbol, that will show references to the corresponding make and break contacts.



Now you must complete the control circuits:

Place all four coils. If the Router is active, you should place the symbols exactly with 40 mm space – they can be aligned later; if the router is inactive, you can place all four coils and then select *Space* and distribute them with 40 mm. You also used this command on page Fejl! Bogmærke er ikke defineret..

When the coils have been placed, they must be connected to L1 and N.

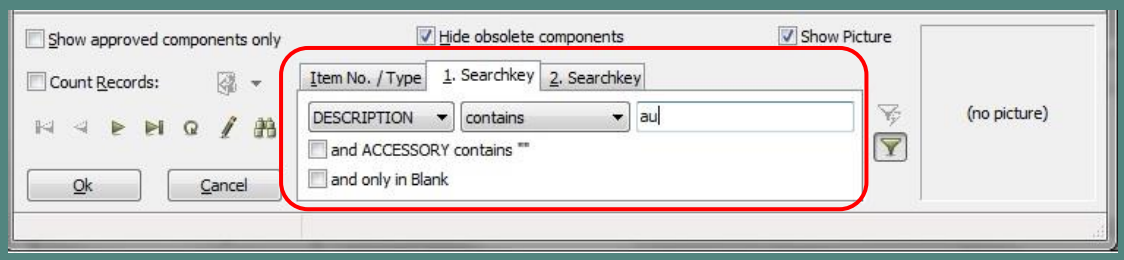
Continue placing and connecting the available symbols until you have finished the control circuit. If you place all symbols in a component, the name disappears from the list....

You can select other components from the database: the original project (PCSMOTORDEMO1) contains manual push-buttons.

Auxilliary contacts from the database (not used in this project)

If you lack contact functions for you contactor, you can fetch auxilliary contacts in the database. An auxilliary contact is an extra component, that is an extra article number.

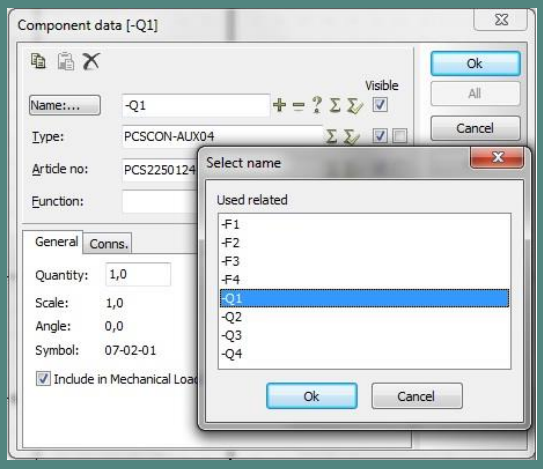
Go to the database (shortcut [D]).



Go to search keys, and search for components as shown above.

Select the aux.contact that fits, and place (one of the) contact(s) in the project.

If you have an aux.contact connected, you can see a reference to its position in the contactors reference symbol.



When you place the symbol, you must give it a name. Press the which will give you a list of possible names for the contact. In this way you are sure to select a valid name for the contact.

You can read more about how you work with the drawing objects

Line

Symbol

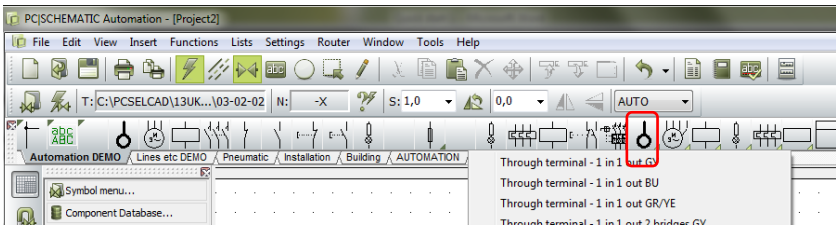
Text

in the QUICK START and, naturally, in the manual.

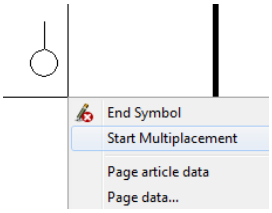
Place a terminal row with article data from the pick menu

Go back to the power circuit to place the last components, they are terminals and cables.

You can place the terminals one by one on the conductors, but it is much faster to use the function *Multiplacement*.

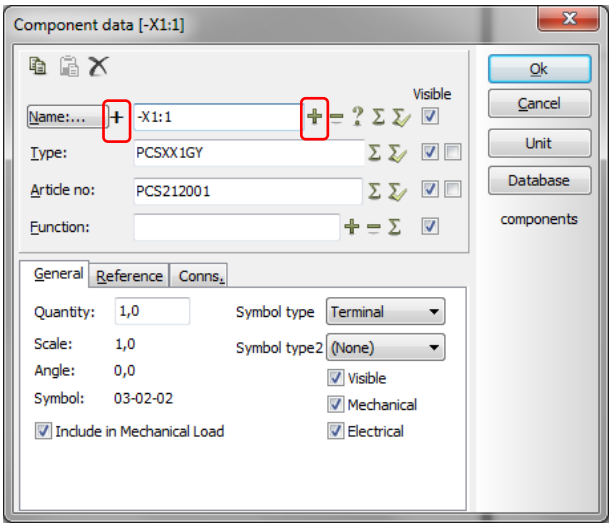


Select a terminal in the pick menu.



Place your cross-hair a little to the left of the first conductor, right-click and select *Start Multiplacement.*

You will not get a ”pointer line” in your cross-hair. Drag this line across all conductors.



When the line is dragged across all the desired conductors, you must right-click again and select *Execute Multiplacement.*

The normal component data dialog pops up, and you are going to name all terminals in the terminal row at once, with continuous terminal numbers:

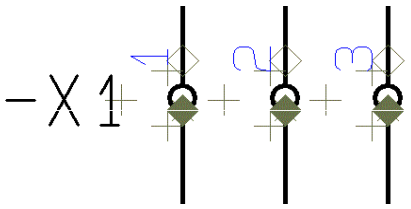
Type ”-X1:1”.

-X1 is the name of the terminal row

:1 is the first terminal name on the terminal row.

Activate automatic numbering by holding down the <Ctrl> key and click + the with the mouse. Then you see a + in front of the name. Click OK.

Terminal row direction



When your terminals have been placed you can see that all terminals have two connection points, and that one of them is filled.

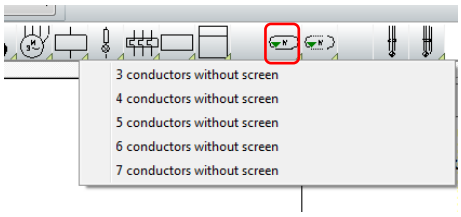
This is because it is practical to be able to differentiate between the internal (inside the panel) and the external side (outside the panel) of the terminal row.,

The direction is relevant when you sort your terminal lists and make terminal plans.

If the direction is wrong, you can mark the entire terminal row, right-click and select *Pin swop.*

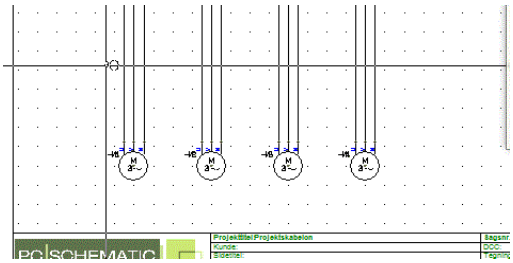
Place cable

The three wires/lines between the terminal row and the motor is a cable. This must be marked in the project.



You do it the following way:

Pick a 4-conductor cable without cable screen in the pick menu.

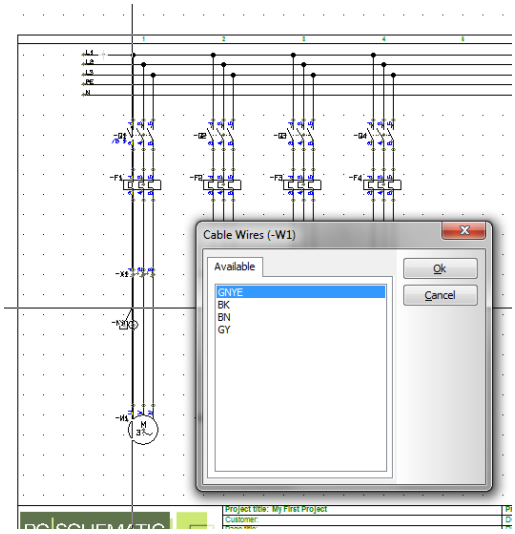


You will now have the cable symbol in your cross hairs.

Place it a little to the left of the first cable conductor.

When you place the symbol the normal component dialog pops up and you can check type, article no. and the number of conductors in the cable.

Name the cable –W1.



When you click OK you will see a ”pointer line”, which you are going to use to select the individual conductors in the cable.

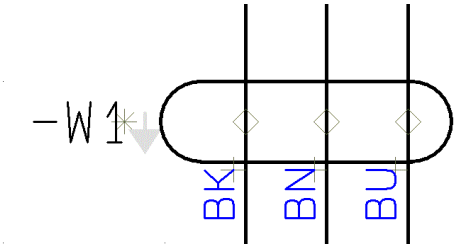
As you have found the cable in the database, the conductors also have a color (or a conductor no.).

Now select one conductor at a time, select a color. Avoid the green-yellow....

When all three conductors have been selected, you click <Esc>.

Repeat for cables –W2, -W3 og –W4.

Cable direction



See the small arrow beside the cable name. This a direction arrow, which influence the sorting of cable lists and plans.

If the direction is wrong, you simply mark the cable, right-click and select *Reverse direction*.

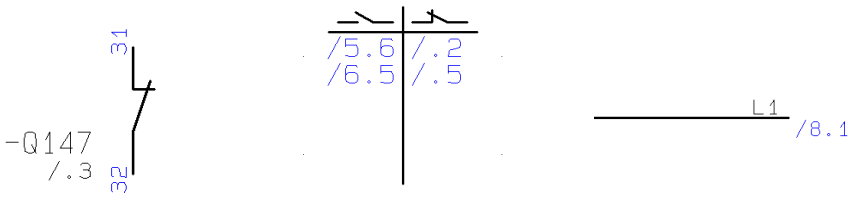
RELATION BETWEEN THE PAGES

When the symbols for the different components are connected in power and control circuits respectively, you can ”jump” between the individual parts of the components.

The reference cross is a reference symbol, meaning that it contains references to other symbols. The references are active links – hyperlinks.

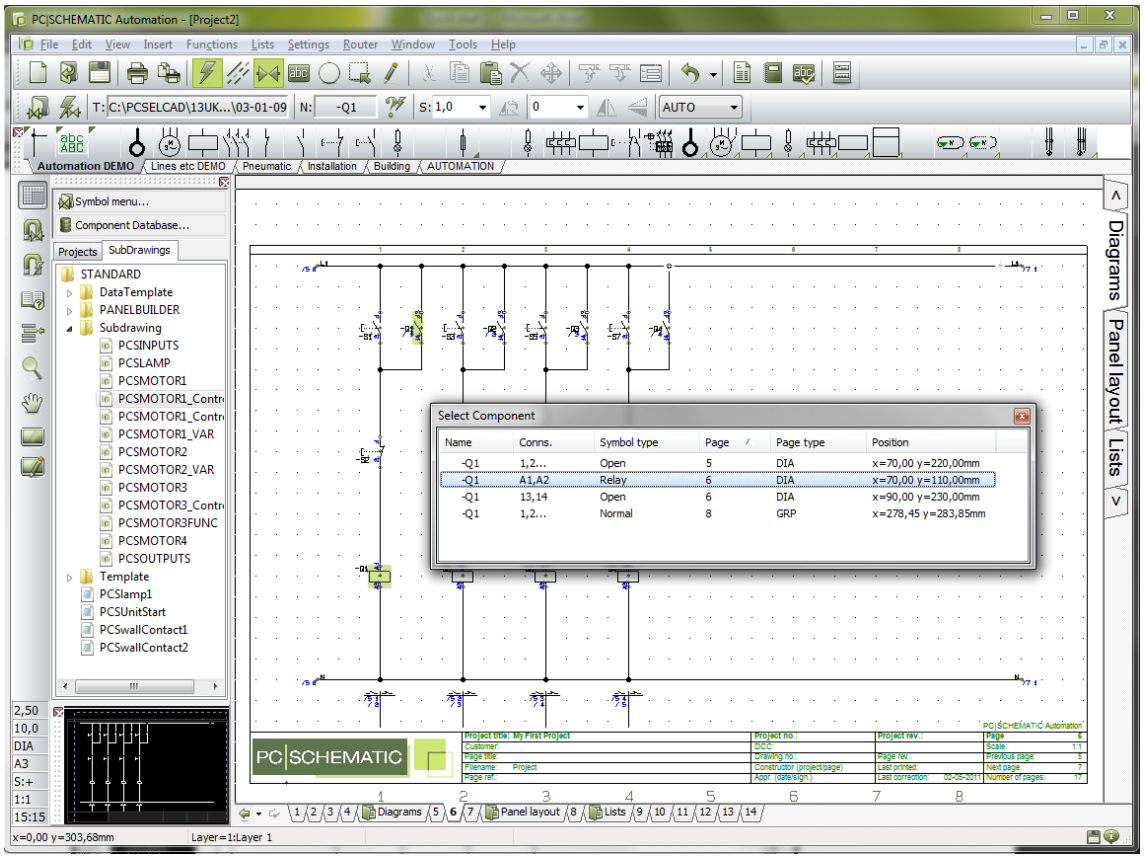
The are references between the symbols in a component, and there are references between signal names in a project.

Try ”jumping” some of the links.



Try to mark one of the symbols in the relay with right-click|Select component. When you do that you will get a window that shows all symbols in the component and their positions. If you click one you will jump to its position in the project.

This window can also be used to go to the mechanical symbols on the layout plan if there is a such in the project.



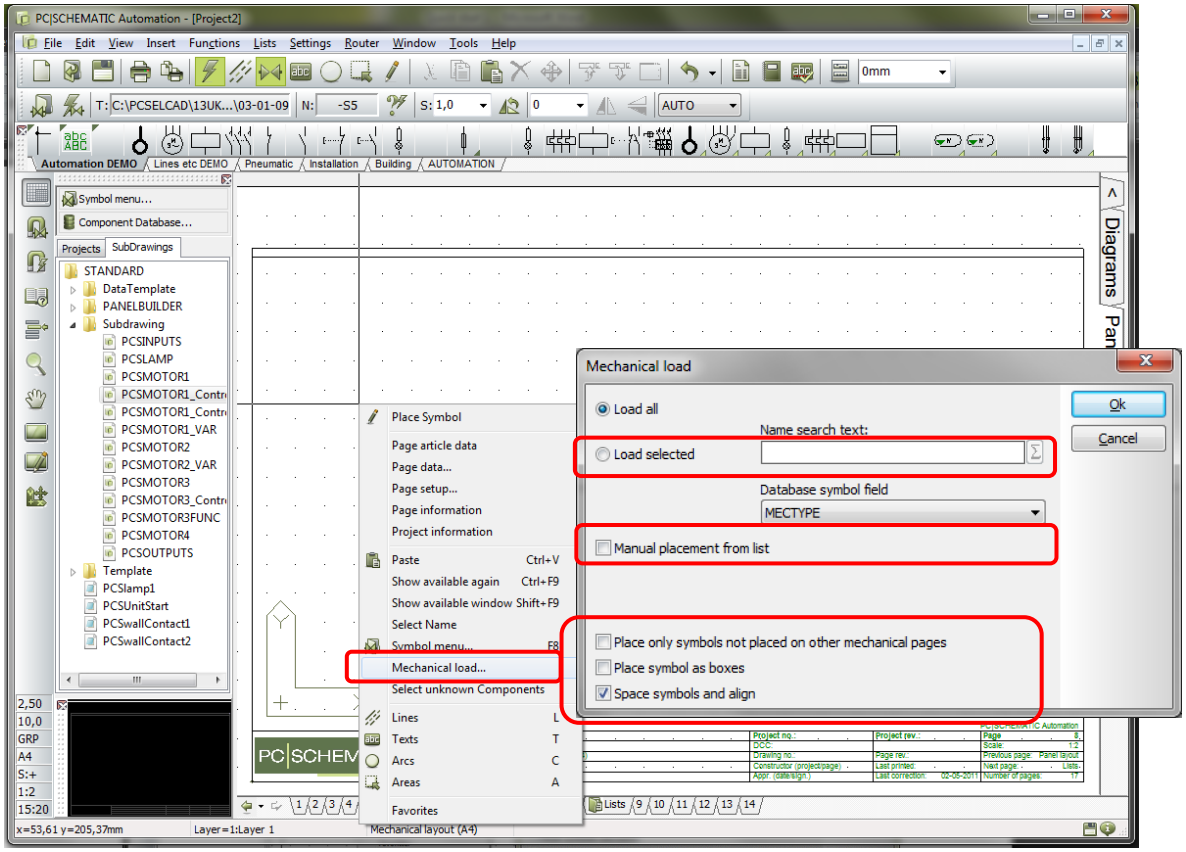
ARRANGEMENT DRAWING – LAYOUT PLAN

When you have places *components* from the database in your electrical circuit diagrams, you can arrange drawing which is a drawing of the panel.

The program has a dialog that assists you in doing this.

Start by going to page 8 in your project, which is a mechanical page (GRP=ground plane).

Right-click on the page and select Mechanical load.



In the dialog you can choose between different methods for placing your panel components. You can *Load all.* This will often be quite overwhelming, but if you combine it with *Manuel placement from list* it can be a fine method for smaller projects.

A good method can be selecting *Load selected.* Here you can select a range of components by using the joker *.

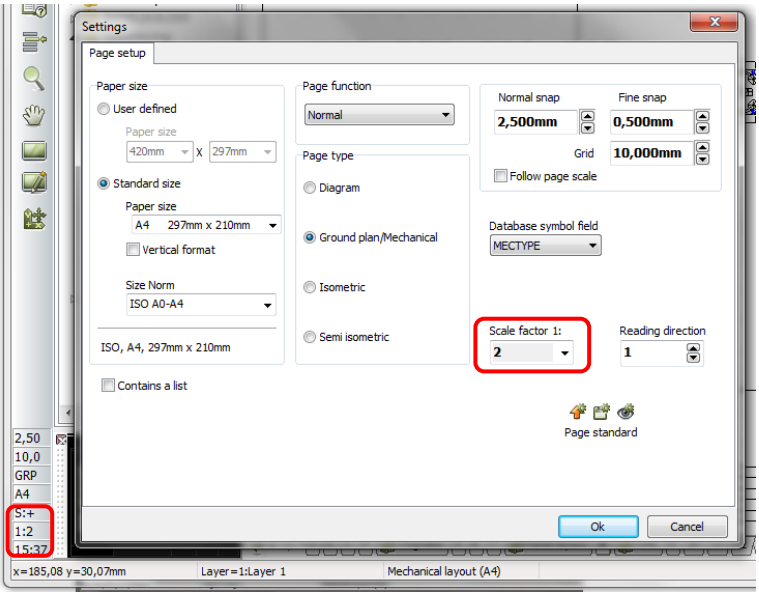
- If you write [-X*], you will get all terminals in the project and they are ready to place
- Then you can select all push buttons [-S*] asnd place thes and so on...

Group by group you place the various components from the project, and at the end you can check by selecting *Load all* possibly in combination with *Place only symbols not placed on other mechanical pages*.

You can try the different settings in the dialog.

The page is mapped to a datafield from the database that contains mechanical symbols for the components. If you try to select a component from the database when you are on this page you will get the mechanical symbol and not the electrical ones as you do on the diagram pages.

That means that you can start on this page by placing componens in the panel, and then make the circuit pages with the *Show available* functions.



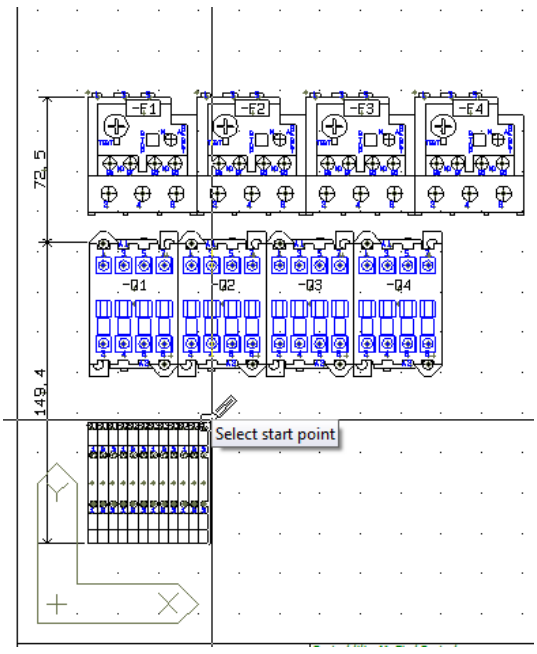
Everything you draw on this page has the right dimensions, meaning that the page also has a page scale.

If you do not have enough space for your components on the page you can change the scale factor:

- 1. Click the page scale in the vertical tool bar (1:2).
- 2. In Settings|Page setup you select the factor that fits.

Measurement

Everything on a mechanical page has a dimension, meaning that you can show the true measurement for various purposes. Measurement is found in the *Functions* menu



When you select the function you get the ”pointer pen” in your cross hairs.

Point where you want to start measuring.

The function “snaps” automatically to the end and middle of lines.

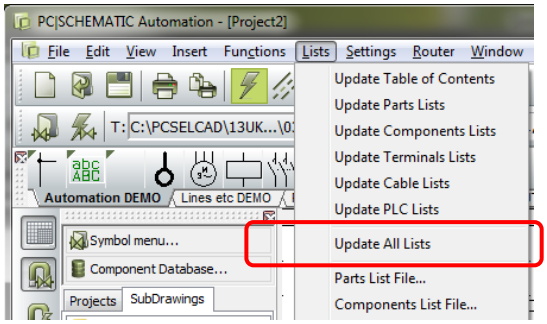
If you press <Ctrl> and click you can start measuring anywhere on the drawing.

If you measure “odd” you can change between Odd/Vertical/Horizontal with the <Space> bar *before* you click with the mouse.

Press <Esc> to finish.

LISTS IN THE PROJECT

In the Start template there are several list pages that are to be included in a project. There is chapter index, TOCs, parts list, components list, terminal list and cable lists.



All these lists can be updated at once by entering the menu Lists|Update all lists.

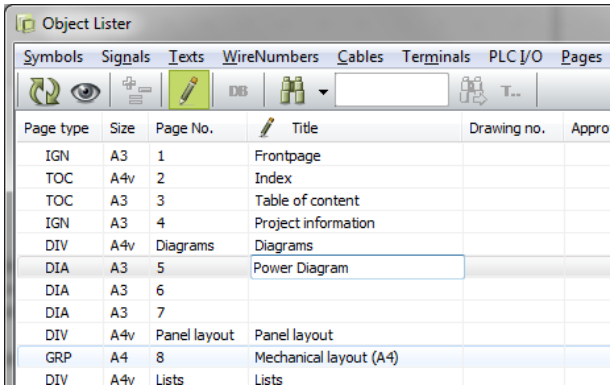
You can also update one list at a time, either via this menu or by right-clicking the individual list page.

The various lists contain links:

- From the TOC you can jump to individual pages
- From the components list you can jump to a specific component
- From the terminal list to a specific terminal
- And so on....

Select the Table of contents (page 3) in the project. You can see that there are no titles for the circuit diagram pages.

The easiest way to enter those is to use the Function|Objectlister (Shortcut [F7]).



Enter the titles this way:

Activate the pen.

Double click the Title for page 5.

Write ”Power Circuit Diagram” and press <Enter>.

Repeat for other pages without title.

Close the Object lister.

Update the lists again. See the TOC.

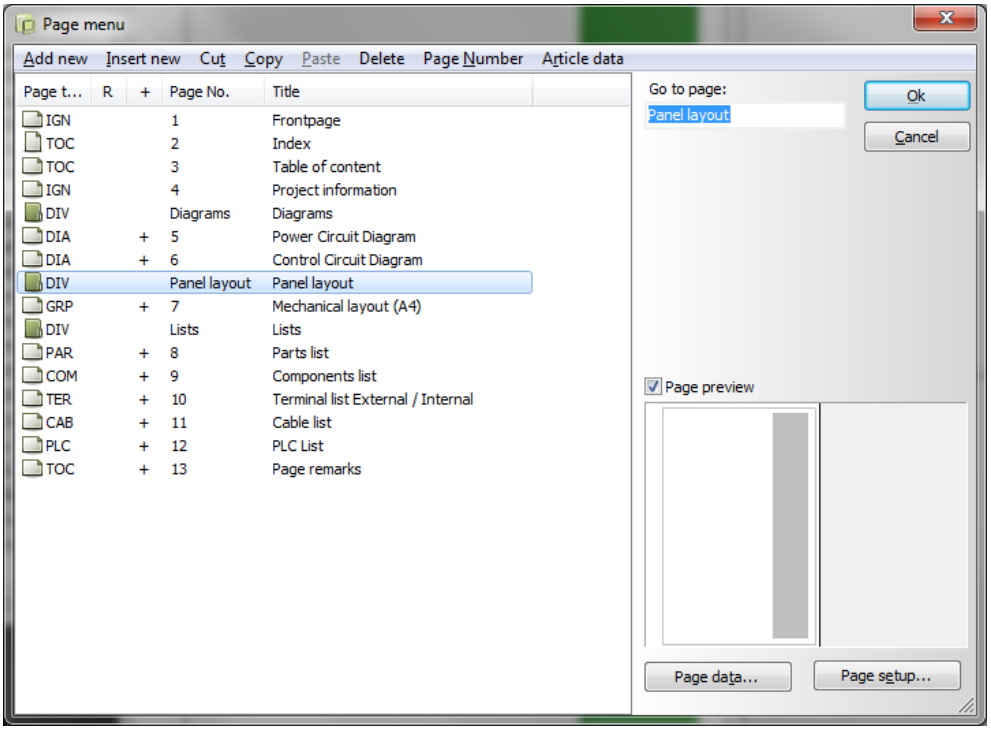
You can change a lot of data with the Object lister. Try it....

If you change anything with the Object lister you change it simultaniously in the project – it is a list view of the objects in the project. So you should not save or anything – it is there already.

THE LAST FINISH...

You have started by using a template. That means that you might have to add extra pages or that you might have to delete some other pages. After that you will have to renumber all pages in the project.

Those tasks are done in the Page menu. Press the icon in the vertical toolbar.



Add/Insert new pages

You can insert more pages here. The difference is that “Add...” adds pages at the end of the project, whereas “Insert.” insert in front of the current page.

Delete excess pages

You can delete excess pages with Cut or Delete.

New page numbers

If you mark one or more pages, you can change page numbers on those.

All commands are found in the top line of the dialog.