MANUAL

PCSCHEMATIC AUTOMATION SERVICE

This manual describes how to use the PCSCHEMATIC Automation Service program.

Developed by PCSCHEMATIC A/S.

July 2017

Last revision: Feb 2017

INTRODUCTION

This booklet describes how to use the program Automation Service.

Definitions …

PCSCHEMATIC Automation is an electrical CAD program, that you us to design electrical diagrams, general arrangement drawings, parts list etc. for documentation.

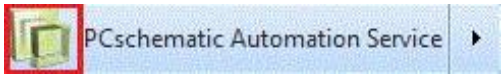
PCSCHEMATIC Automation Service is a program to show, navigate and work with the diagrams. You can use it to print the projects, navigate by using the Net Navigator and the references, you can export to 3rd party programs, e.g. from which you can print out labels. You can use the Mounting Assistant when mounting panels and plants and it helps you keep the over view and keeps track on how far you are with the mounting. But you can't make changes or corrections and save in the diagrams.

CONTENTS

INTRODUCTION 3
DEFINITIONS 3
OPEN THE PROGRAM 6
OPEN A PROJECT 6
ZOOM IN THE PROJECT 6
PANNING IN A PROJECT 6
PRINTING OPTIONS 7
PRINT A SINGLE PAGE 7
PRINT SEVERAL PAGES 7
PRINT DIALOG 7
PDF 8
EXPORT TO PDF 8
NAVIGATE IN THE PROJECT 9
PAGE MENU – OVERVIEW OF THE PROJECT PAGES 9
SELECT A PAGE 9
TABLE OF CONTENTS 10
LIST OF REFERENCE DESIGNATIONS 10
COMPONENT LIST 10
NAVIGATION USING CROSS REFERENCES 11
NAVIGATION USING SYMBOLS 11
NET NAVIGATOR 12
OBJECT LISTS (F7) 13
SEARCH IN THE OBJECT LIST 13
INFORMATION ABOUT THE DIFFERENT OBJECTS 14
COMPONENT ACCESSORIES 14
SELECT ACTIVE LAYER AND VISIBLE LAYERS 15
LIST TO FILE 16
PRINT LISTS AND LABELS 16
SETTINGS 17
PROJECT DATA 17
PAGE DATA 17
PAGE REMARKS 17
COMPONENT DATABASE 18
CHANGE LANGUAGE 18
REFERENCE POINTS 18
SHORTCUTS 19
THE MOUNTING ASSISTANT 20
START USING THE MOUNTING ASSISTANT 20
THE WINDOWS 21
SELECT REFERENCE DESIGNATIONS 22
SELECT PAGES 23
SELECT COMPONENTS 23
SELECT POTENTIALS 24
SELECT CABLES 25
SELECT JUMPERS 25
CHANGES IN THE PROJECT 26
POST-PROCESSING 28

OPEN THE PROGRAM

Open the program by clicking on the icon on your pc:

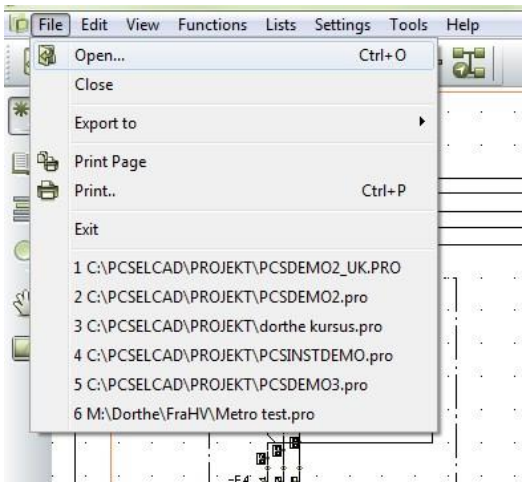


OPEN A PROJECT

Click on the Icon and find the wanted Project.

The ten latest used projects can be opened directly from the File menu.

How fast a project is opened depends on the size of the project.



ZOOM IN THE PROJECT

The zoom-icon is in the vertical toolbar.

When you click on the icon you get an oblique ↔, as you can use to frame an area you want to zoom on.

You get the whole picture back by clicking on the lower icon, press Home or ZZ.

Alternative you can zoom in/out by pressing the Ctrl-button while you are scowling the wheel on the mouse. You always zoom from the point the cursor is on.

Panning in a Project

The panning-icon is in the vertical toolbar.

If you need to move around on a page, then click on the panning-icon and start panning around the page.

PRINTING OPTIONS

Print a single page

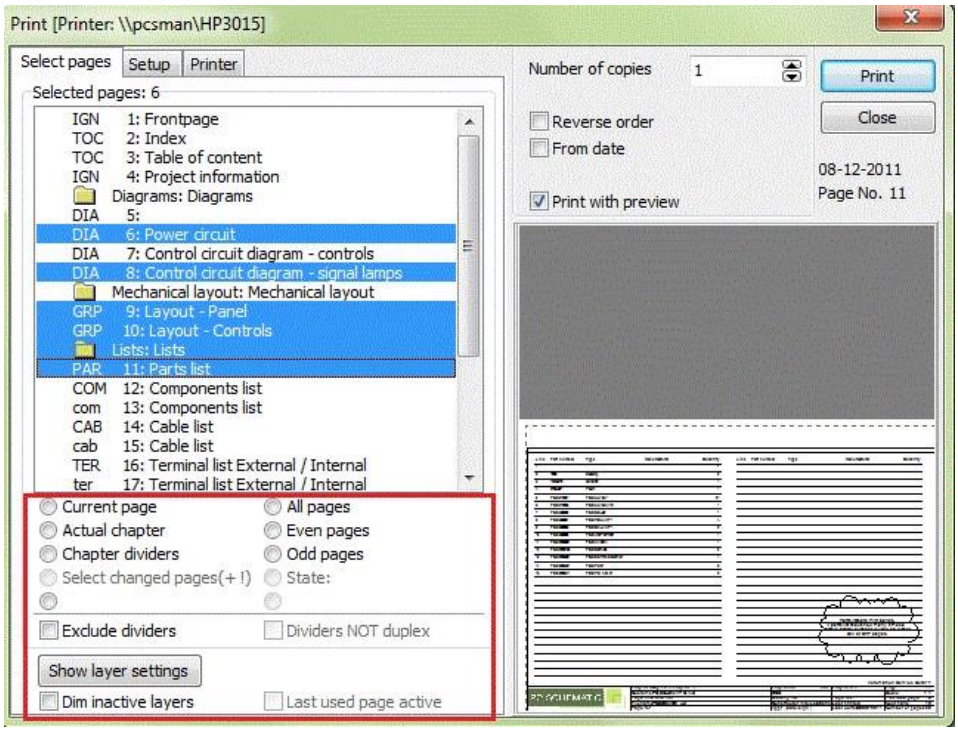
If you press on the first printing-icon, the current page is printed.

Print several pages

Press on the other printing-icon and the printing-dialog shows where you can choose several pages.

Print dialog

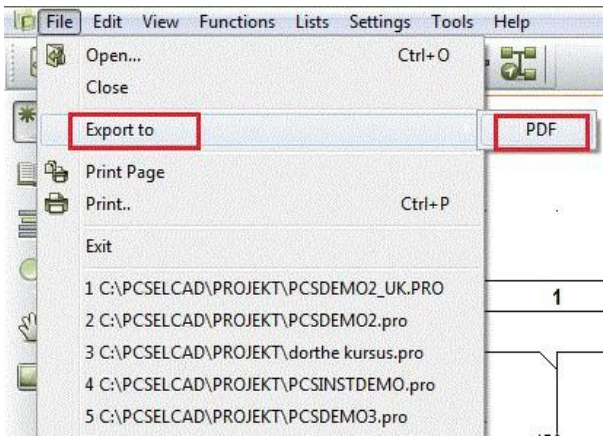
In the print dialog you can choose between different ways of selecting pages. You can also select several pages by Ctrl+click on the wanted page/pages or Shift+click for a row of pages.

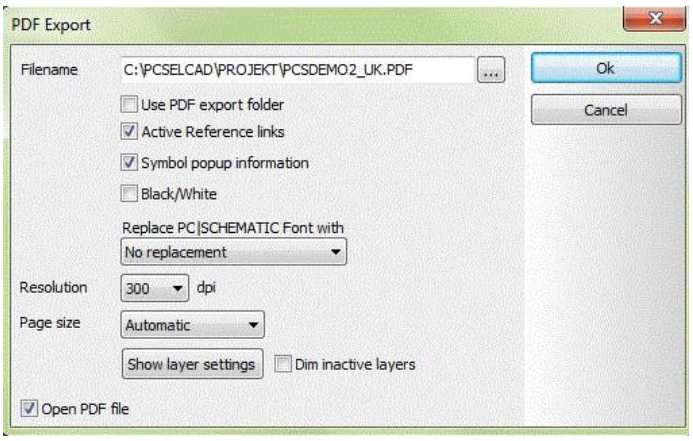


PDF

Export to PDF

You can export a project to a PDF. In the dialog box PDF export you have different opportunities for what needs to be active in the PDF file – mark and press OK.





NAVIGATE IN THE PROJECT

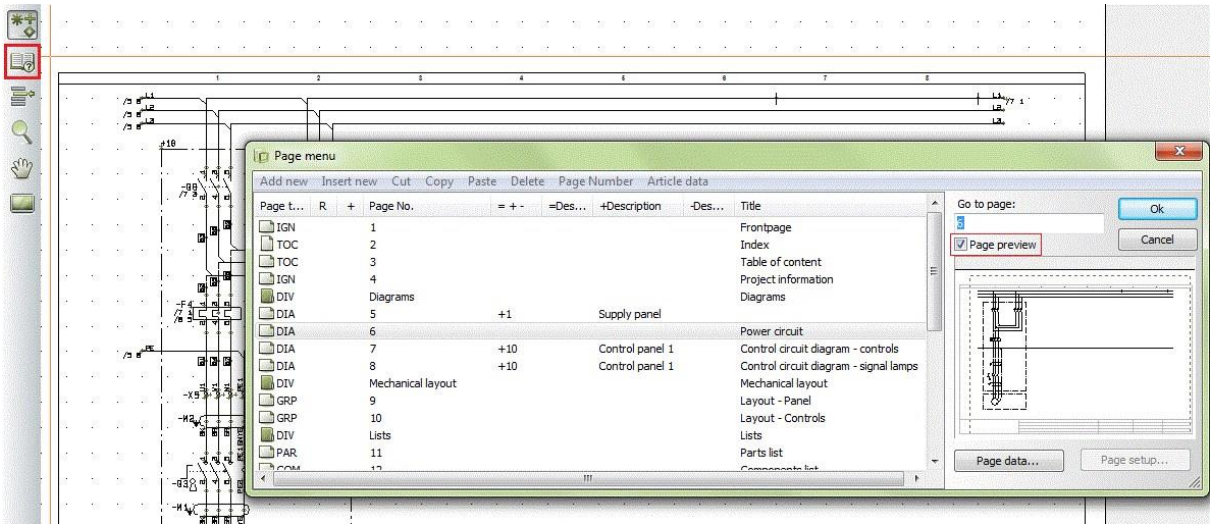
You might need to search or navigate in the project or on the pages. Depending of what you are looking for, you have several options.

You can either look for certain pages, components or reference designations etc.. Below, you can see how you get around in the project.

Finally, you can see what options you have on each page.

PAGE MENU – OVERVIEW OF THE PROJECT PAGES

You find the page menu in the vertical tool bar. The page menu shows all the pages in the project and all the data regarding these.



Page type: DIA means diagram. Other important pages can be TOC index, COMponent liste, IGNorered. You can see if the paper is standing or lying.

R means that revision is active on the page

Page no. Is obvious. ”Missing numbers” in the numbering is possible because you can number the pages as you like.

=+ - Contains the total reference designation, which is composed of Function designation (=) Location designation (+) and Product designation (-). The elements of the code refers to function, location and product, respectively.

Description All 3 aspects have separate columns.

Title on each page.

In the right part of the dialog, you have a preview of the selected page.

Select a page

It’s possible, only by using the page menu, to navigate to a certain page. Open a page by double-clicking or by marking it and then click OK.

You can flip back and forth in the project by clicking on the tabs in the bottom and by using PageUp and PageDown keys.

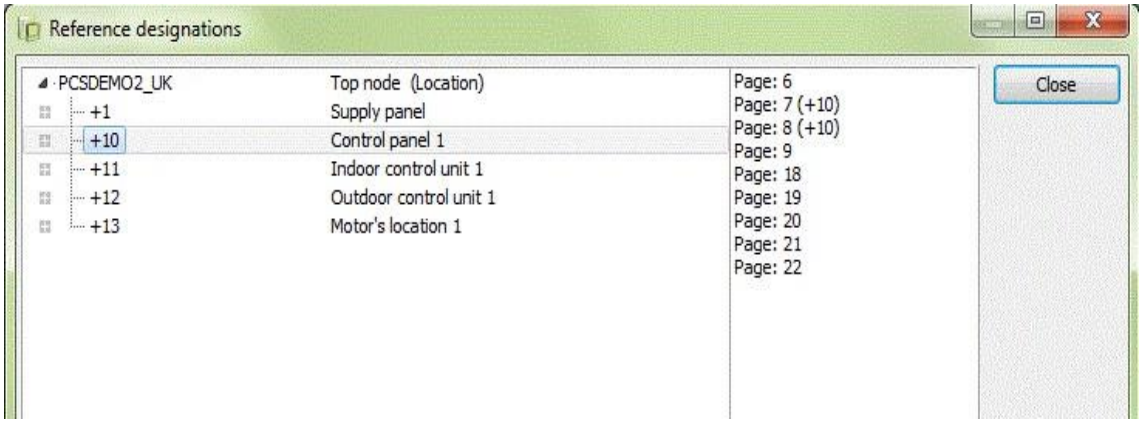
TABLE OF CONTENTS

All lists have active links. This means that you can ”jump” directly to a page from the table of contents.

LIST OF REFERENCE DESIGNATIONS

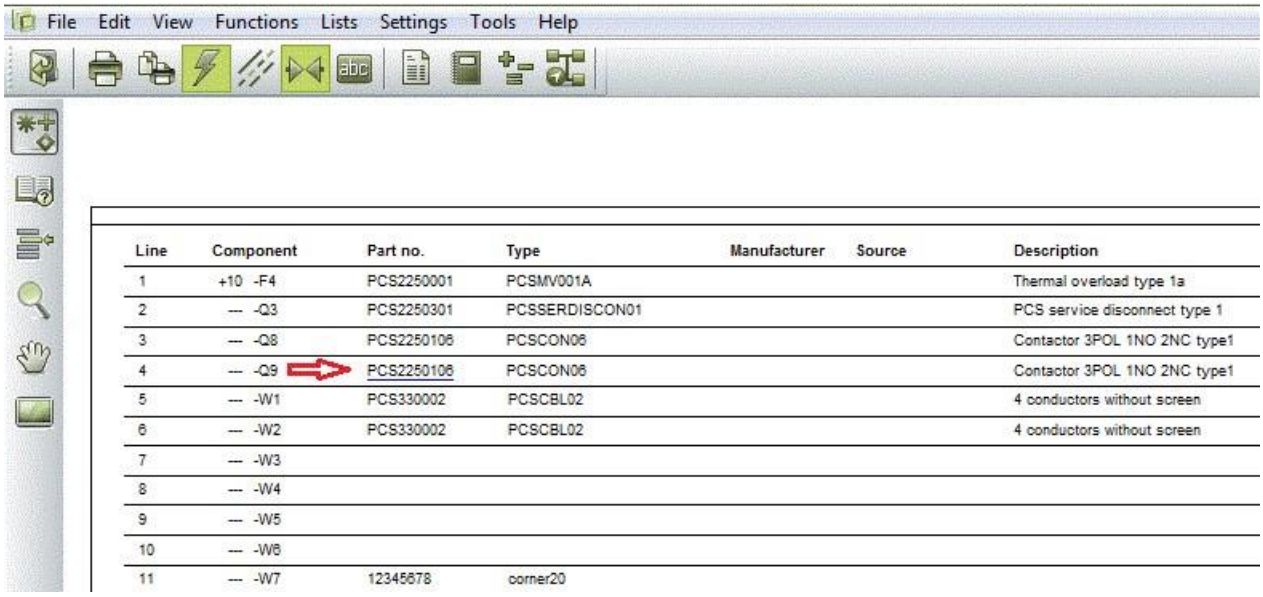
By clicking on the icon you open the list of reference designations.

When you place the cursor on a designation, you get a list of the pages where it’s used. When you double-click on a certain page you “jump” directly to it.



COMPONENT LIST

In the same way you can ”jump” directly from the Component list to a component, as all lists have active links.

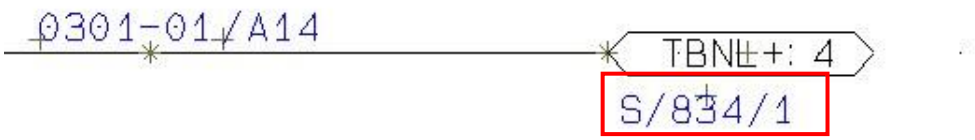


NAVIGATION USING CROSS REFERENCES

There are references between everything in a project with the same name. This means that there are references between eg coil and contacts in a relay and between signals with the same name on different pages.

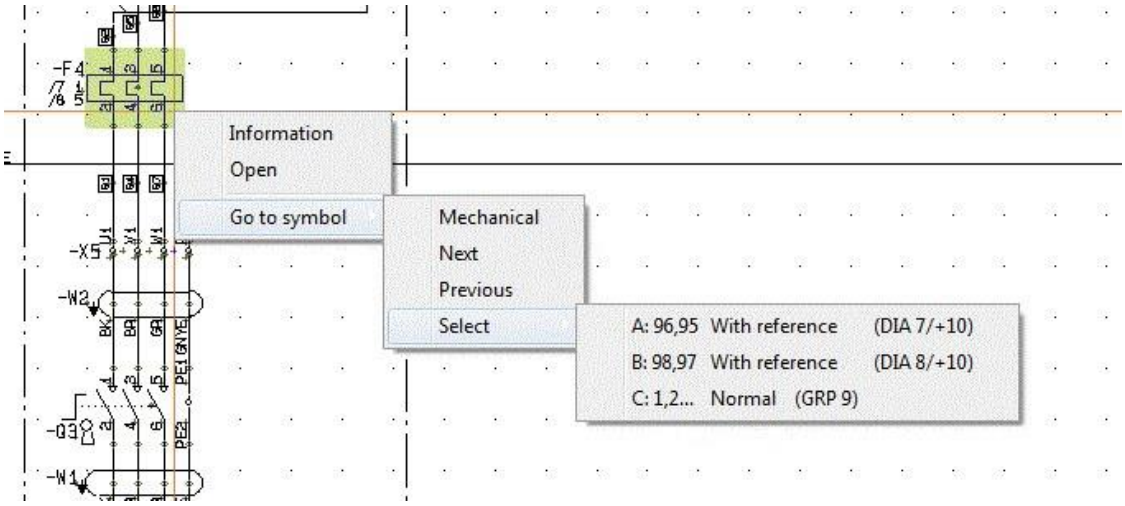
These cross references are also active links, allowing you to ”jump” directly to the current position.

Links may look like this:

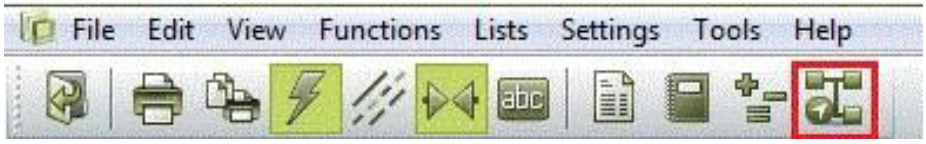


NAVIGATION USING SYMBOLS

When you right-click on a symbol and select Go to symbol – Select, you can choose among a component’s electrical or mechanical symbols. You can “jump” between them using Next and ”previous".



NET NAVIGATOR



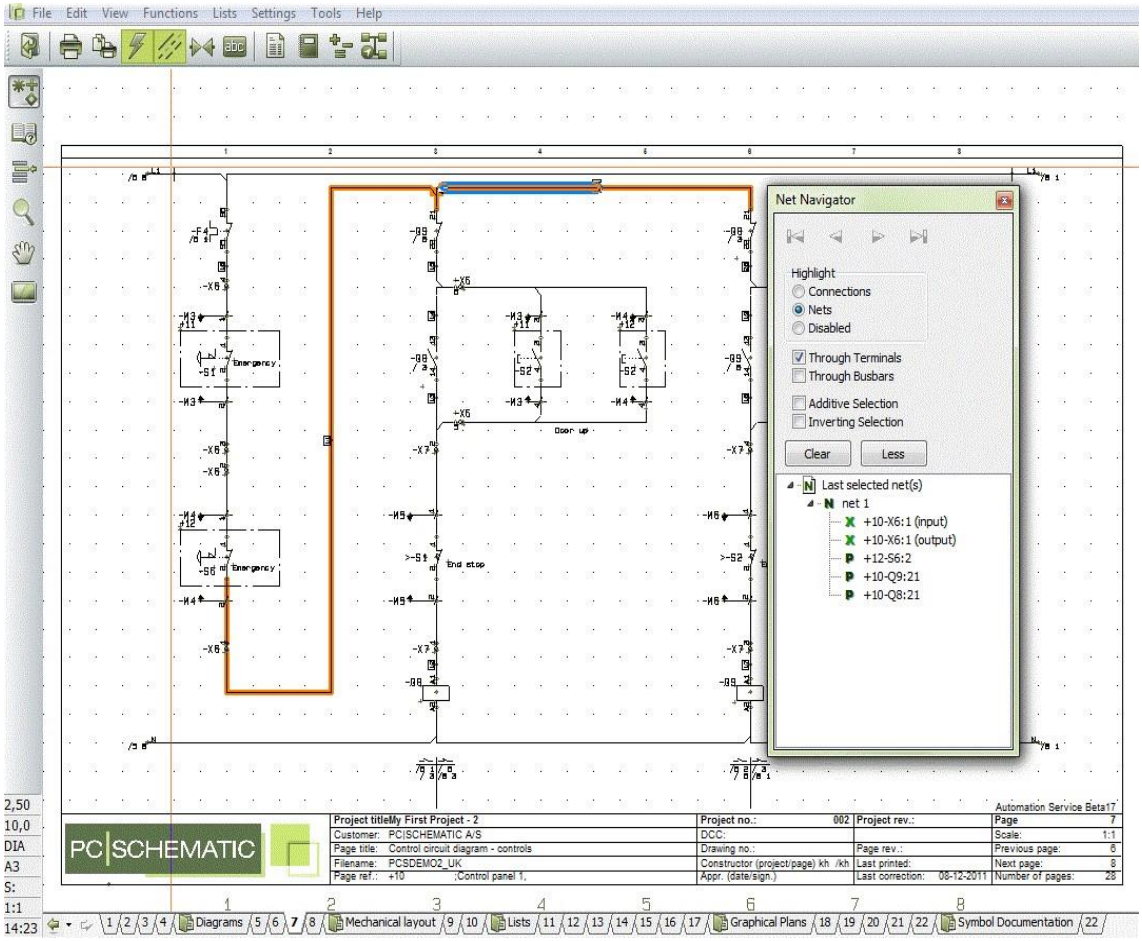
What's interesting about lines in a diagram is, that they represent potentials between components and that the potential often runs over several pages.

The function Net Navigator, which is accessed using the icon in the tool bar, makes it possible to mark one or more connections at a time and have a list of the connection points, where the potential is connected.

In the example below, you see a net (potential), which is marked in the diagram. The net is on several pages, and you can "jump" around in the net either by using references – see earlier in this booklet – or by clicking in the list that you see in the Net Navigator dialog.

In the list you can see, where the net is connected: P (pin) is a normal connection point on a component, X is a terminal. You can also see on a terminal whether the connection is on the internal/Input or the external/Output of the terminal row. If you "jump" to another page, the net is also marked there. You can either mark a single connection or a complete net, choose if the net must run through the terminals or busbars or you can choose to mark more than one net at the time (add) and remove all markings (clear).

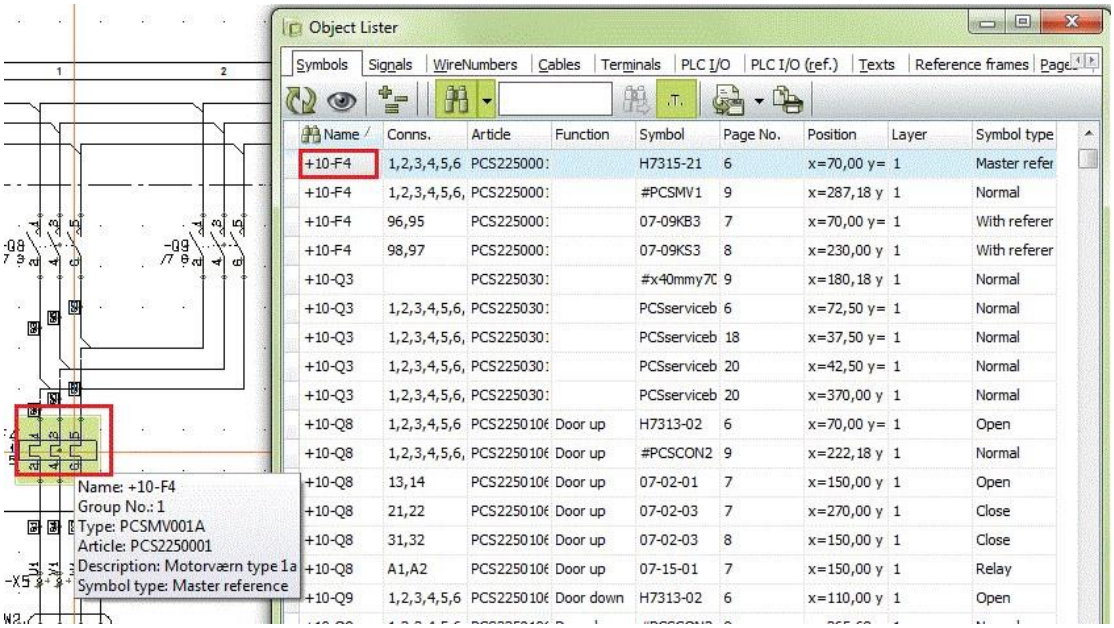
If you have marked more than one net, the nets are showed one by one in the list. Finally, you can leave the Net Navigator visible or deactivated.



OBJECT LISTS (F7)

All data in the project can be seen in the Object lists.

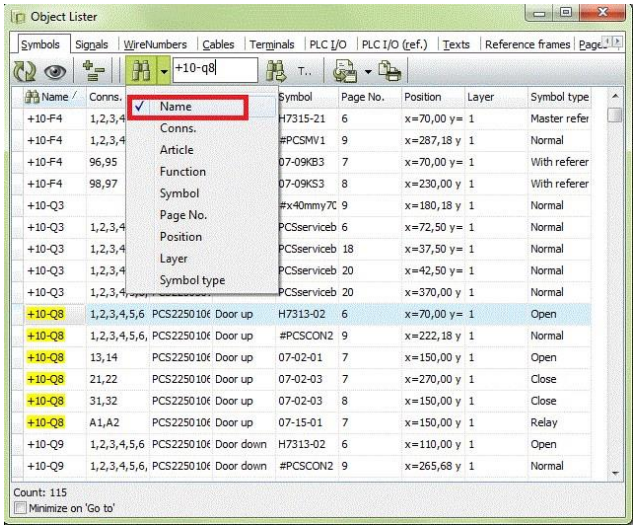
It's lists of all objects – symbols, pages, terminals etc. – existing in the project. Here you can also 'jump around' in the project, as there also are links to all objects from this window. You double-click only on what you will find.

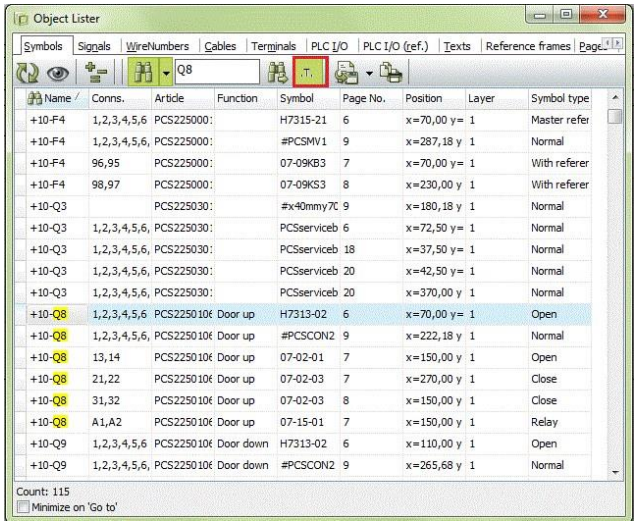


Search in the Object List

You can search in various columns – name, connection, part No. etc. in the Object lists.

E.g. choose NAME and "Begins with" (left pic.), and in the list, the names will be marked with yellow. You can also choose NAME and "Contents" (right pic.) and you can see it in the list to the right.





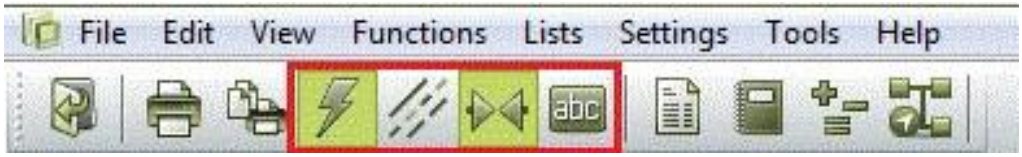
Try making your own settings by selecting fields and rearrange in order.

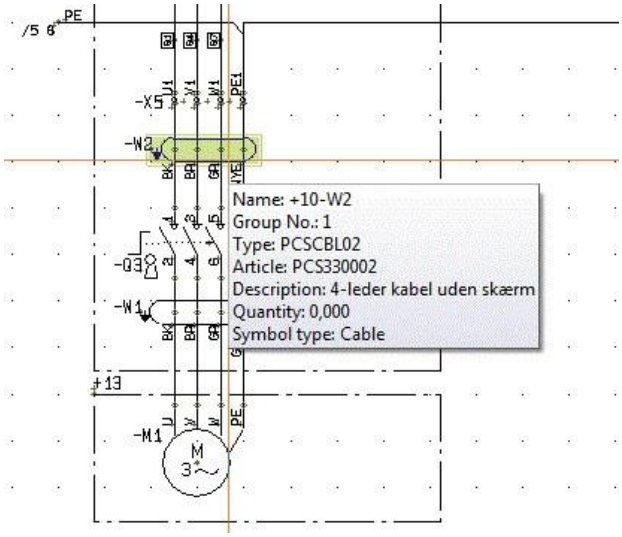
The object list may advantageously be placed in the right side of a wide-screen.

Information about the different objects

When you navigate in the project on different types of objects, you can see information about them if you click on them.

You can either turn on “Auto command change” (the lightning) or shift manually between line (L for Wire), Symbol (S) or Text (T). You click on the icon or tap the shortcut.





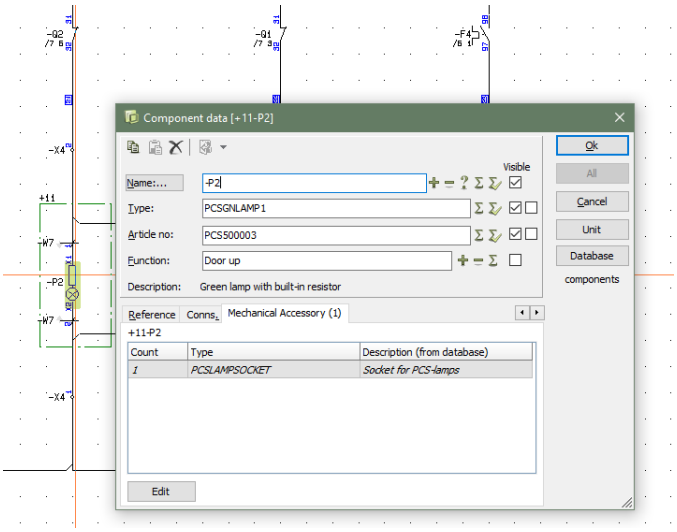
When you leave the curser on a symbol, you get information on the component in a pop-up window:

By clicking on a line you get line information if there are any. Be aware, that wirenumbers are special symbols.

You can also get information on a text by clicking on it.

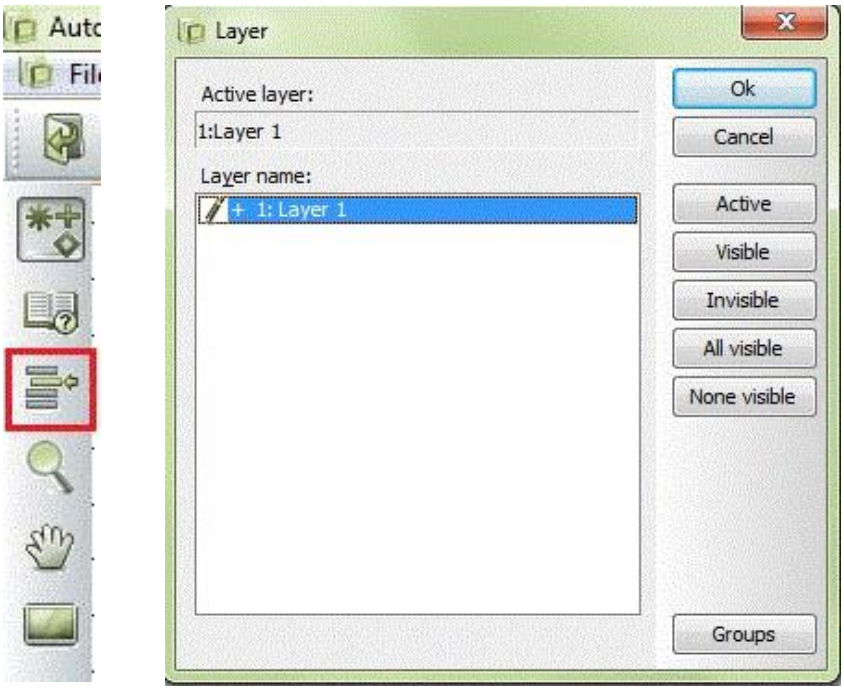
Component accessories

If a component has accessories, you can see the mechanical accessories in the Accessories tab:
Items in *Italic* are fixed accessories as defined in the database, other items are attached by the designer of the project.
When you right-click on a symbol, you can select to see Electrical accessories.



SELECT ACTIVE LAYER AND VISIBLE LAYERS

The project can be made in several layers. You find the layer button in the vertical toolbar.



When you click on the layer icon, you see a dialog box. By default, it's set to only show the active layers, so often you only see one layer.

If the page has more than one layer and you need to navigate to another layer, you can either double click on the wanted layer or click on Active followed by OK.

LIST TO FILE

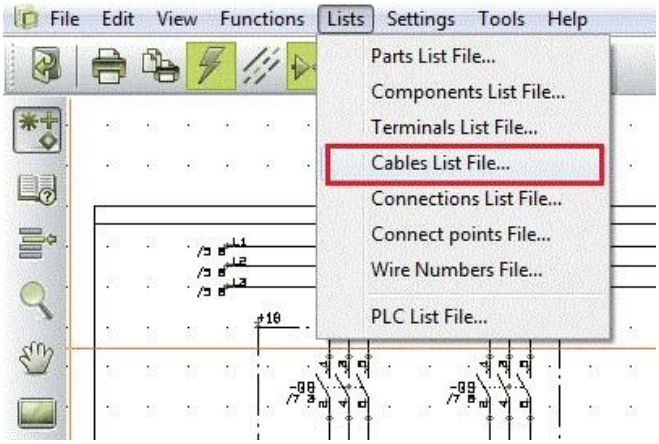
Print lists and labels

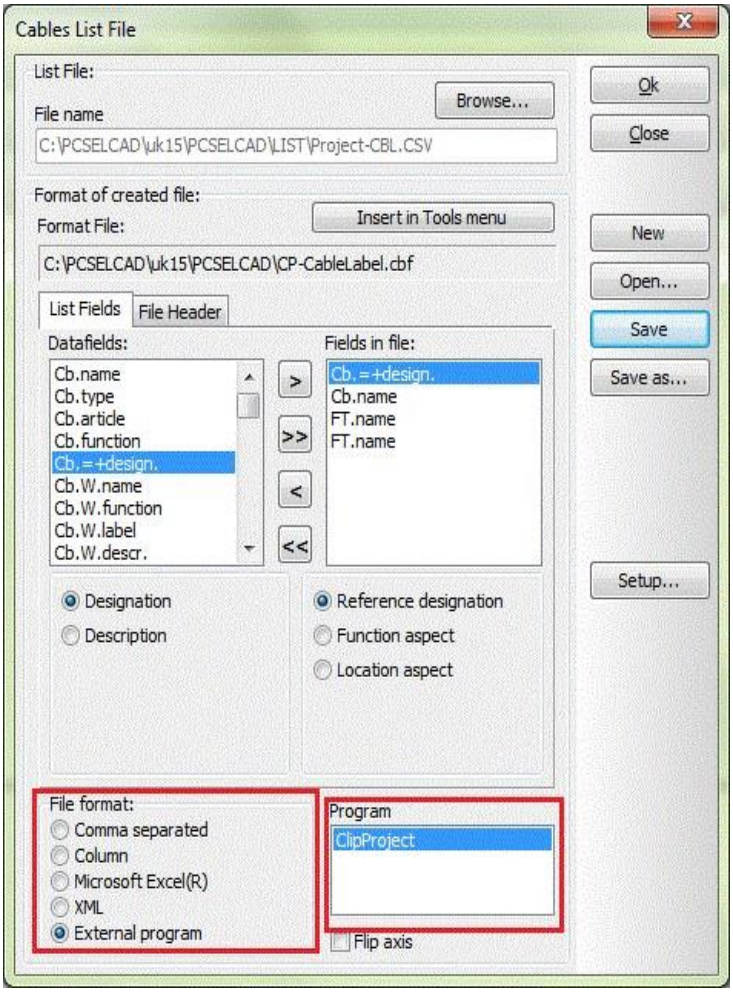
You can print all types of lists in different formats. All lists contain data based on the diagrams and labels are based on data from the lists.

To print out the different lists, you choose Lists in the main menu and then the type of list you want printed ex. a cable list.

The cable list must at least content cable name and possibly also information about from where to where the cable runs. Mark the data in the “Data field” you need to print and click > to transfer the chosen to ”Fields in file”.

Choose then the list format.





You need to do the same to print out labels.

If you install external programs to print labels it will appear in the ”Fil format” as External program.

You can see which program you have installed in ”Program”.

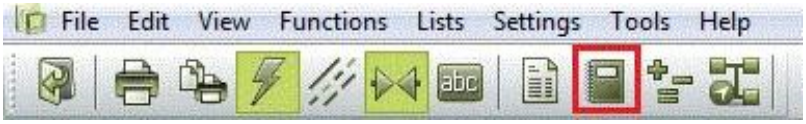
PCSCHEMATIC Automation and Automation Service can interface directly to ClipProject and M-Print PRO.

If you would like to know more about printing labels, you can find much more information in the tutorial ”Marking and Labelling”.

SETTINGS

PROJECT DATA

General data about the project can be found in Settings|Project data or by clicking on the book in the tool bar.

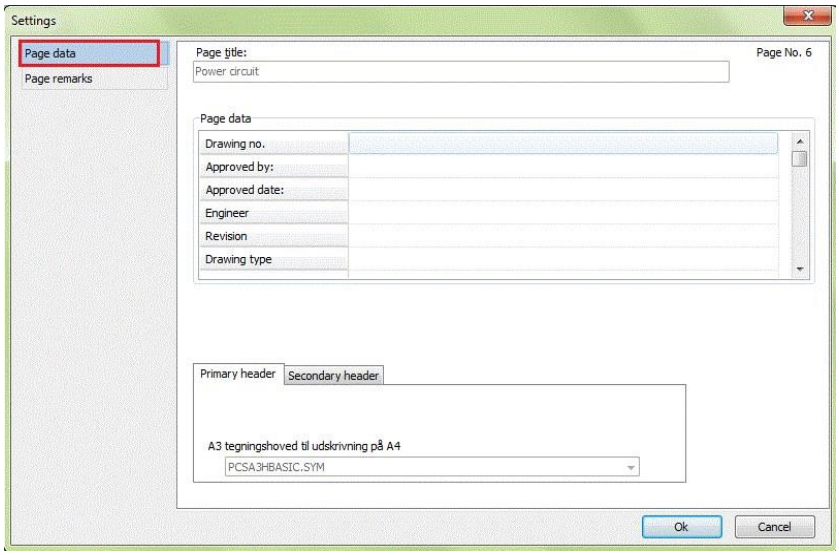


Some of this data you see in the drawing header.

PAGE DATA

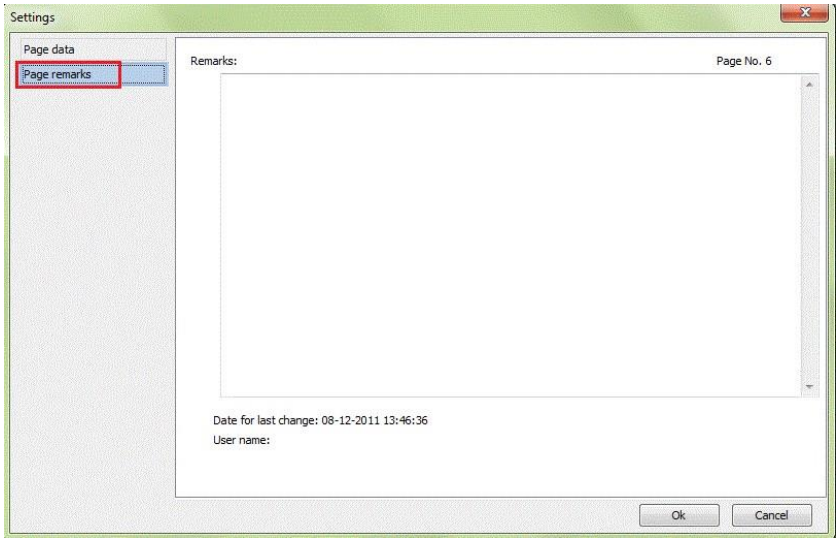
Data regarding individual pages can be found in Settings|Page data or by clicking the page icon. The data varies from page to page.

Some of this data is also in the drawing header.



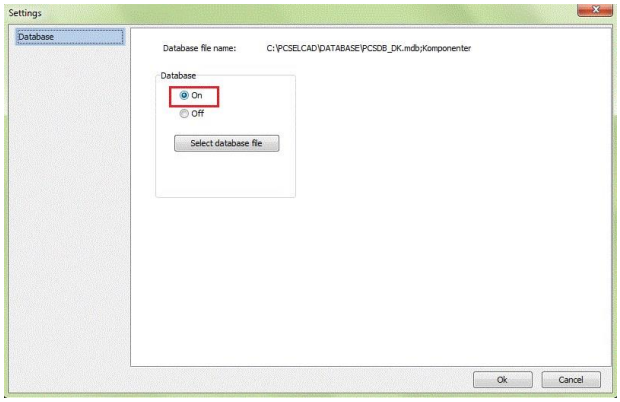
PAGE REMARKS

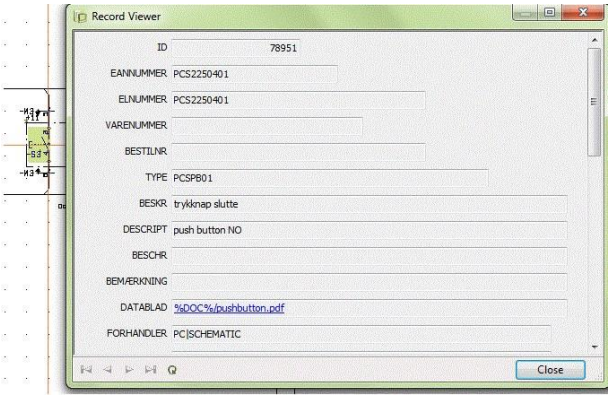
In Page Remarks you can read more detailed remarks for the selected page.



COMPONENT DATABASE

You can connect component database in Settings|Database.

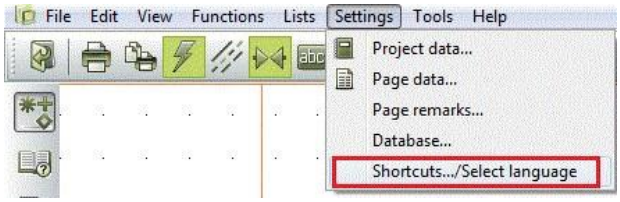


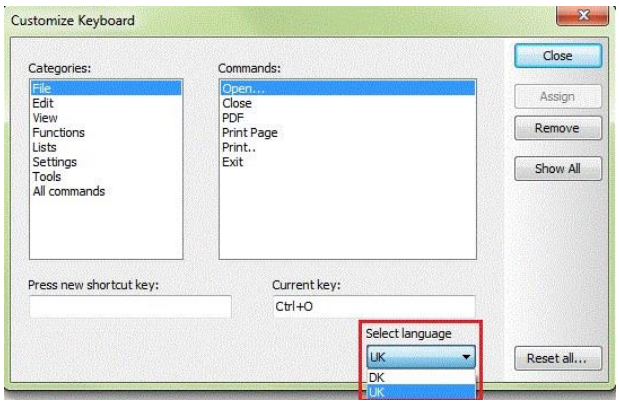


When the database is connected you can right-click on a symbol in the diagram, click Open and that way access the component database and get more information about the components.

CHANGE LANGUAGE

If you during the work with the service program, want to change language use Settings – Shortcut…/Change language.





REFERENCE POINTS

You may want to turn off the reference points, as it makes it easier to read texts in a diagram, especially around symbols.

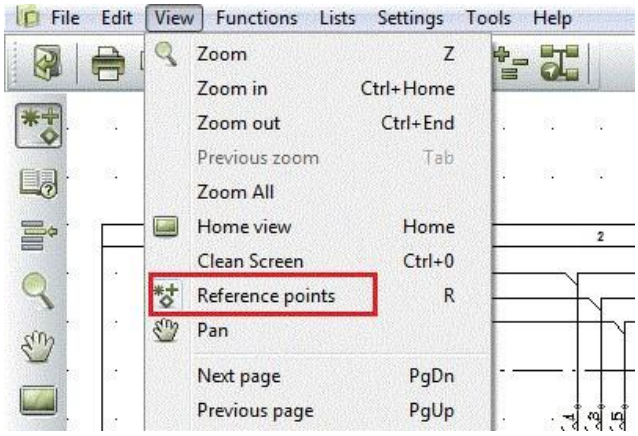
Click on the icon in the vertical toolbar to turn them either on or off. If you use it often, you could make a shortcut for the function.

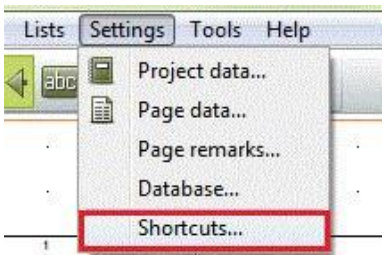
SHORTCUTS

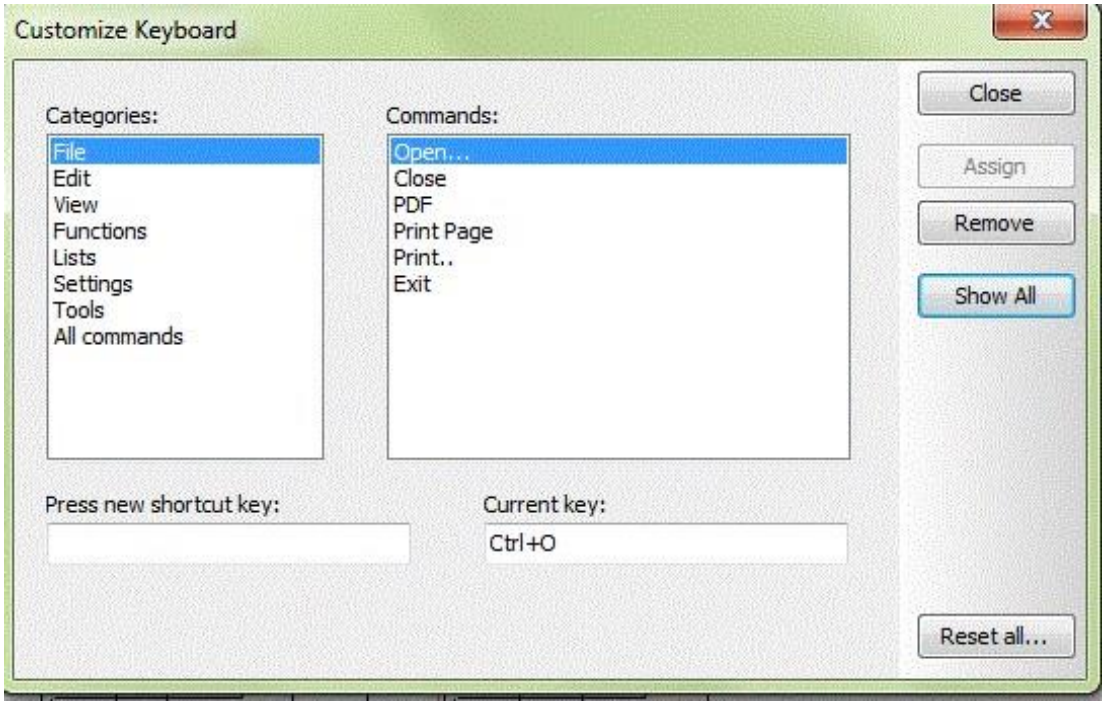
You can make your own shortcuts if you use some functions a lot.

Find the Menu where the command is located. In this case it's in the Menu View. Then click Settings and Short cuts.

Choose the category where the command is and then the command you want to give a short cut. Type your shortcut in "Press new shortcut key". If the shortcut key is already used for another comamnd, you are not allowed to use it and that will be indicated at the bottom of the dialog box.







THE MOUNTING ASSISTANT

The purpose of using the Mounting Assistant is to save time when you mount panels and control circuits.

The Assistant guides you through the mounting proces and shows automatically what to mount next, the connections are simply highlighted in the diagram.

You decide the order yourself and all the time you have an electronic overview of your progress.

If the project is changed during the mounting proces, the Assistant will tell you, so you can maintain your overview.

You can let selected connections wait and later search for them.

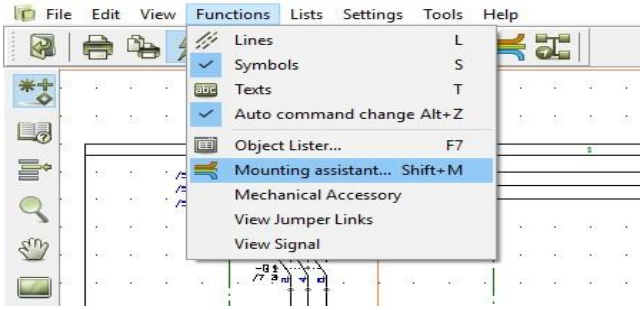
You can search for information on components and cables in the component database and the Object Lists directly from the Mounting Assistant, which is especially useful if some parts need post-processing after the mounting process.

If you are more people working in a series production you can use the same diagram for your Mounting Assistants, simply use different order numbers.

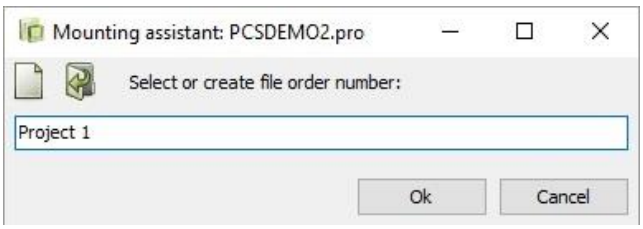
You cannot change anything in the diagram.

The Mounting Assistant is also available in PCSCHEMATIC Automation from version 19.

START USING THE MOUNTING ASSISTANT



You open the Mounting Assistant by clicking the icon in the toolbar or in the Functions menu.



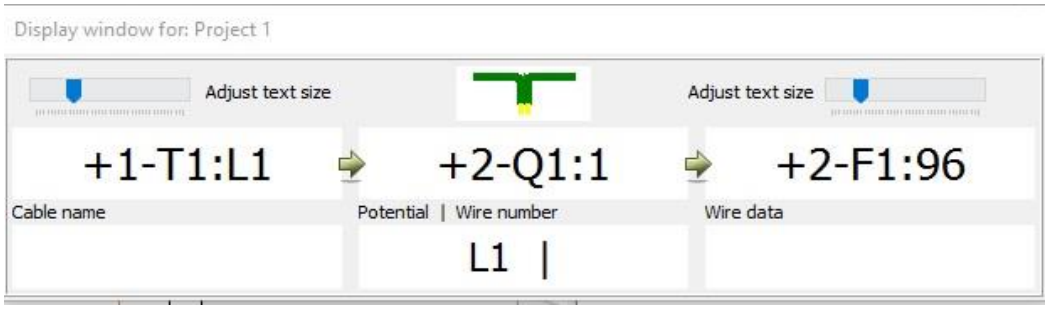
Give your job an order name or no. and you are ready to start.

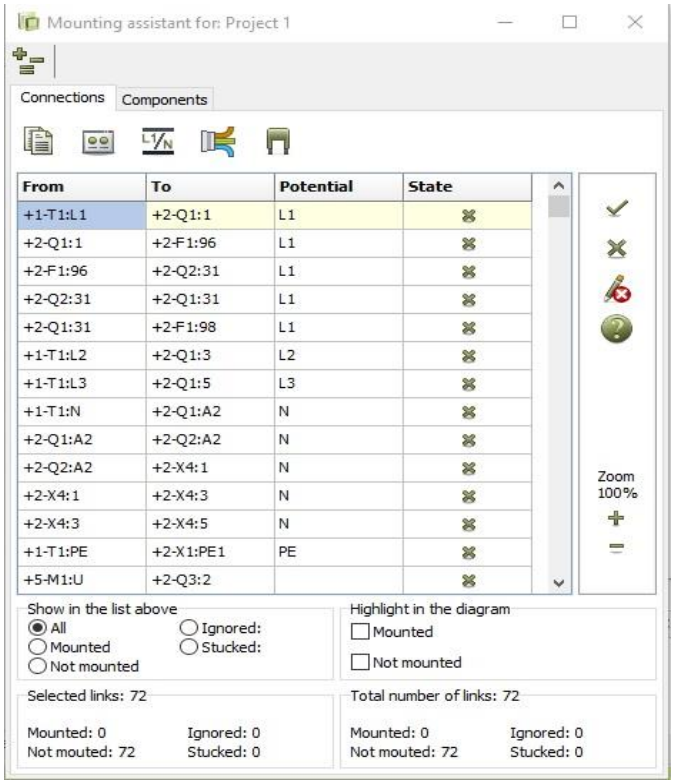
THE WINDOWS

Now you have two windows on your screen: The Display window shows a connection with information about

- where it goes from and where it goes to
- whether there are one or two connections to the terminal
- information about a possible cable name, potential name, wire number or data.

You can adjust the text sizes in both lines by moving the adjust bar.



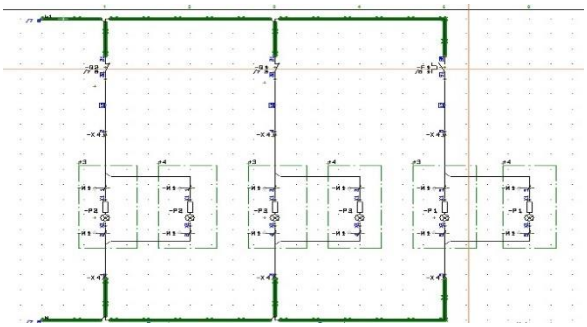


The Mounting Assistant window gives you an overview of all connections in the job. It is here you choose what you want to mount and in which order – by reference designation, phases and potentials first, jumpers next etc..

The overall order is from the first page and onwards.

It keeps track on how far you are, and in the bottom of the window it shows how many connections are mounted, waiting and ignored.

In the field “Show in the list above” you can mark what you want to be showed e.g. Not mounted.

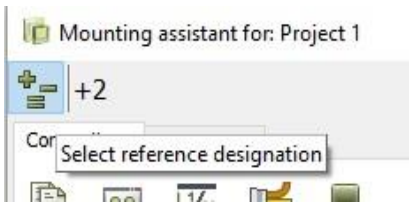


You can also choose to see in the diagram what is mounted or not or both.

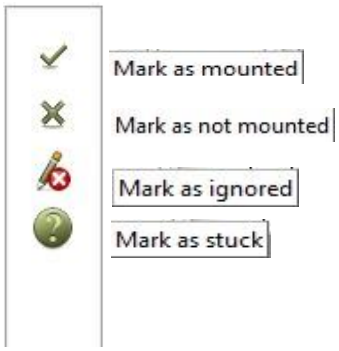
You can Zoom in the diagram on + and – in the window or on your keyboard.

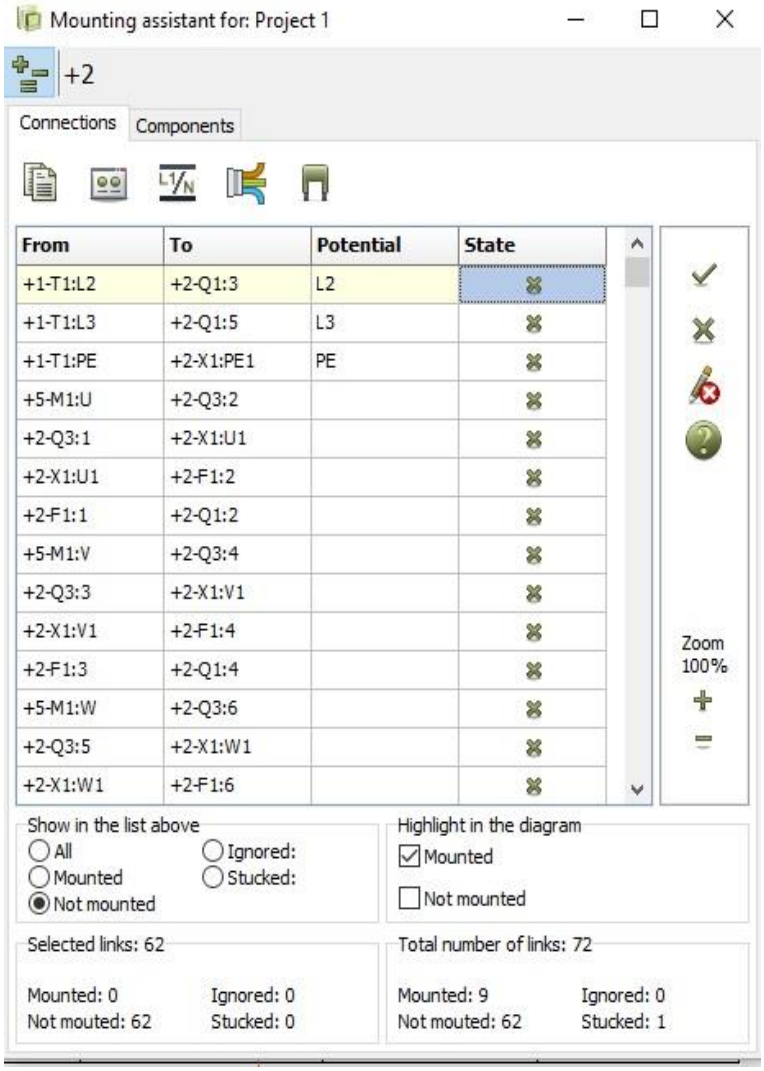
SELECT REFERENCE DESIGNATIONS

If you work with big projects with e.g. several panels and control systems in the same diagram, you may need to select which reference designation you want to mount. Click on the icon reference designation and choose one.



In the Mounting Assistant window all the not mounted connections are showed according to the selected reference designation. When you start mounting the connections, you can click on the icons in the right side to give them a status that you can use later for searching.





Mark as mounted - The connections disappear from the list as you click and they become green in the diagram. You can also mark a connection by standing on STATE and press ENTER on the keyboard.

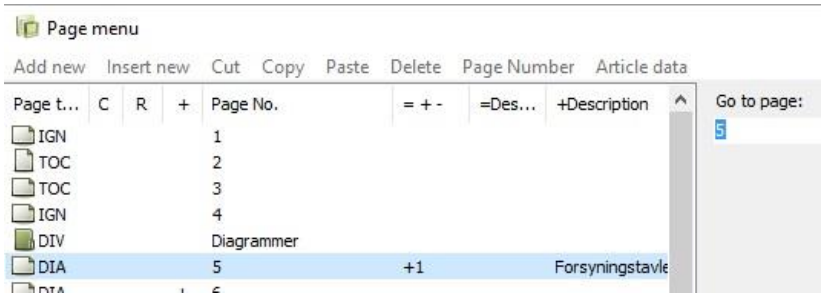
Mark as not mounted - If it was marked as mounted earliere in the process, the green marking in the diagran disappears and it gets back in the list.

Mark as ignored - The connection disappears from the list but can be showed again by choosing “Ignored” in the field “Show in the list above”.

Mark as stuck – The connection disappears from the list, but can be showed again by selecting “Stuck” in the field “Show in the list above”. The Stuck-function is useful if you need to talk to someone about some connections before mounting them.

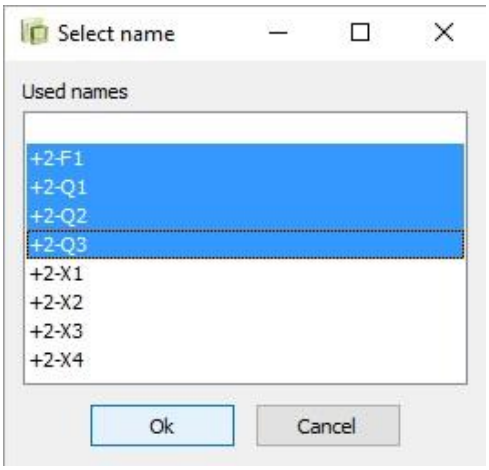
SELECT PAGES

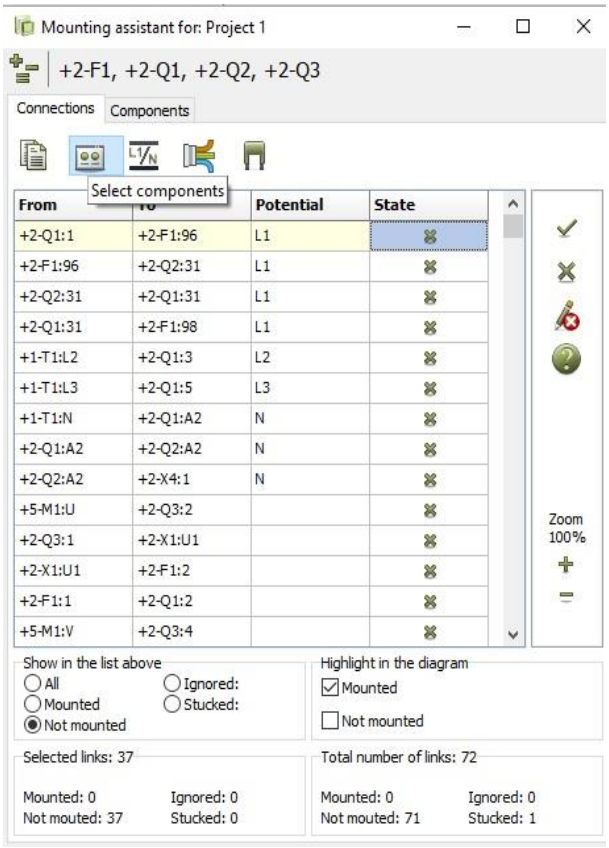
If you want to mount your connections page by page you choose pages from the Page menu in the Assistant Window and all the not mounted connections from the chosen pages will be listed in the mounting assistant window. You can follow the workflow in the bottom of the window all the time. The red line in the diagram is the selected connection in the Assistant Window, it’s the one you are about to mount.



SELECT COMPONENTS

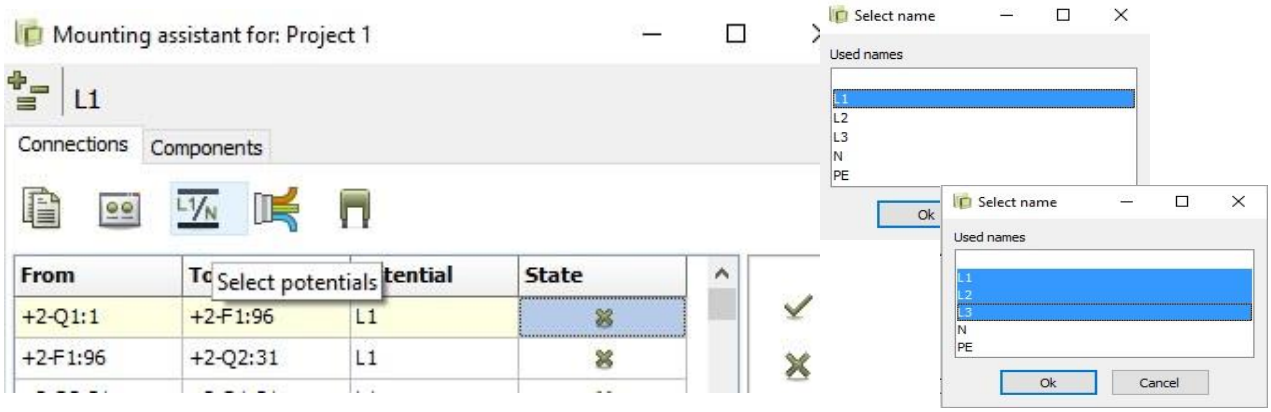
If you want to mount your panel component by component, you click on the icon Select components. In the dialog box Select name, you click on one or more components you want to mount. In the top of the Assistant Window you can always see what you have selected. In the list you now have all the connections belonging to the chosen components and you can start mounting.

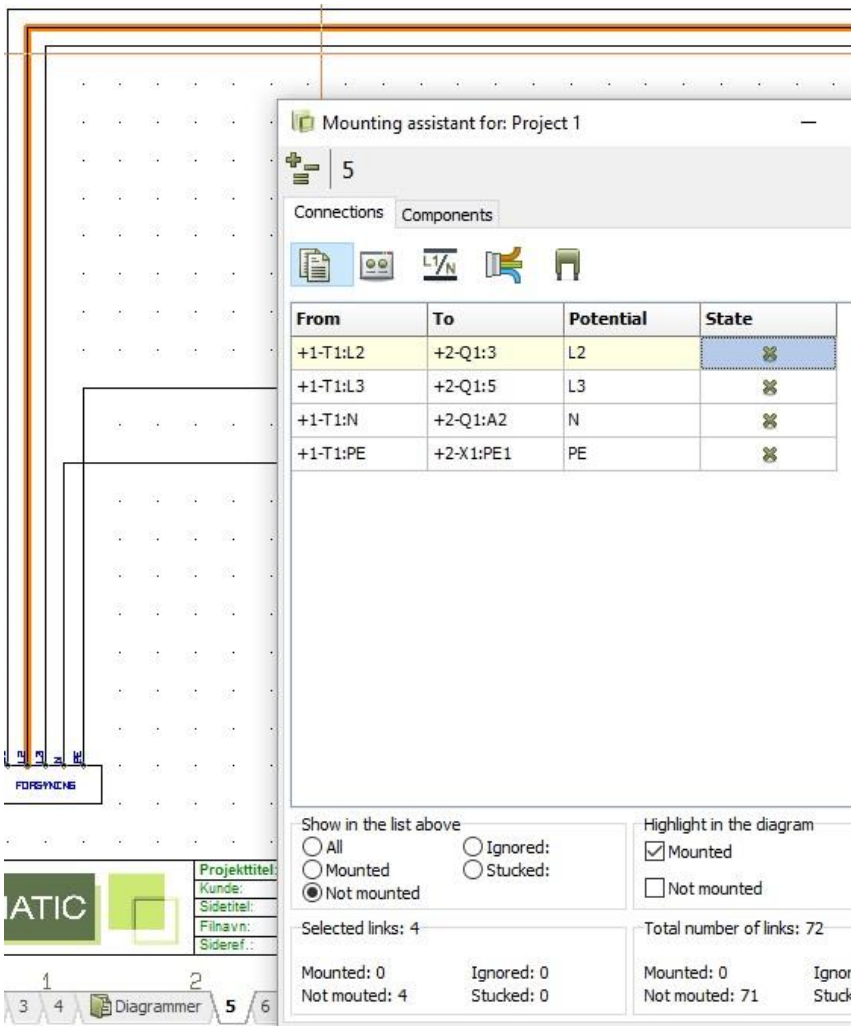




SELECT POTENTIALS

If you start mounting your potentials, you click on the button and choose which potentials you want to start with. You can choose one or more.

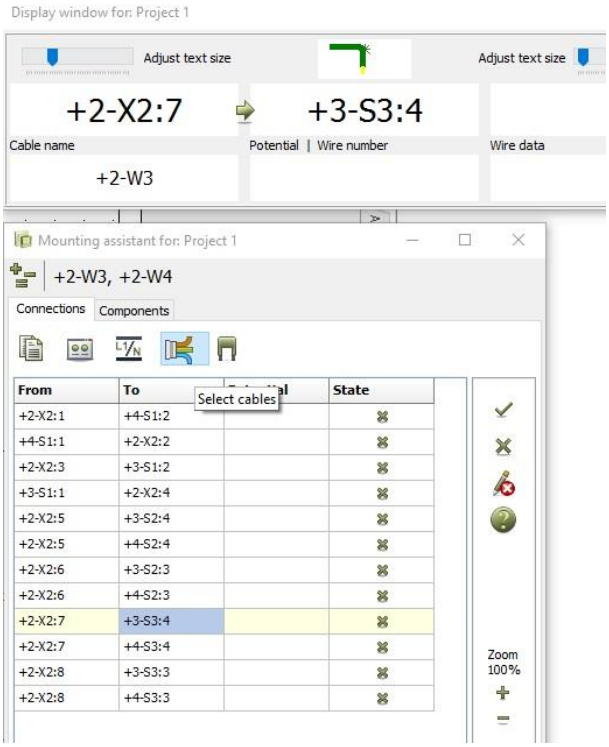


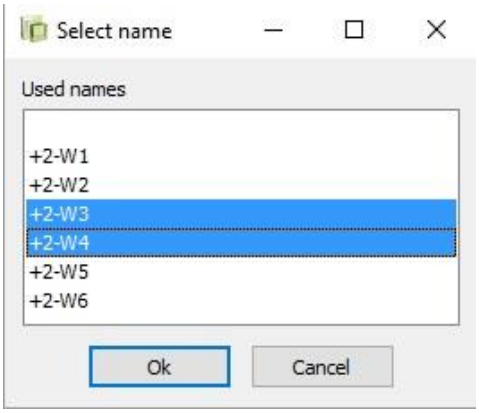


The Mounting Assistant window now contains all the connections with the selected potentials.

SELECT CABLES

Click on the icon Select Cables and mark one or more cables you want to mount and they will be showed on the mounting assistant window with cable name.

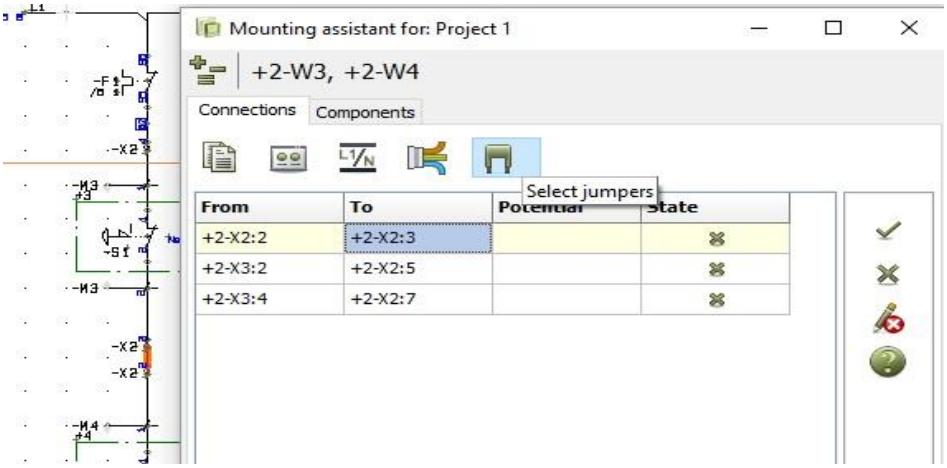




SELECT JUMPERS

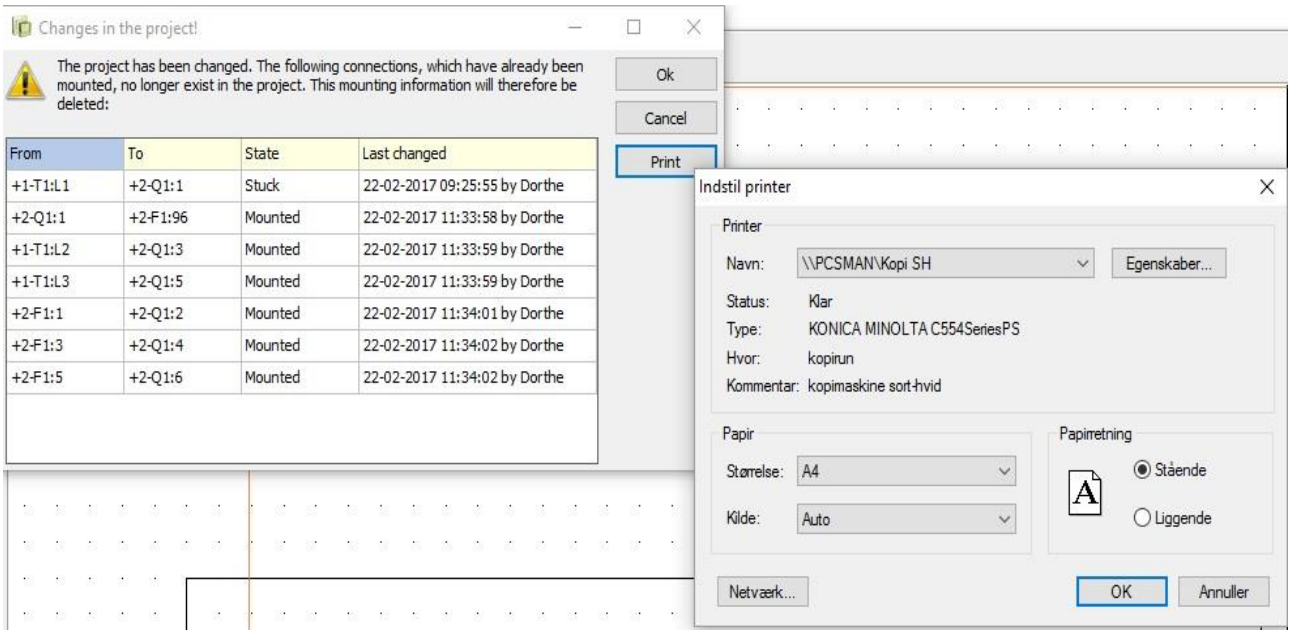
You can also select all you jumoer links.

Click on the icon Select Jumpers and they will be showed in the Mounting Assistant window and you can see them one by one in the diagram.

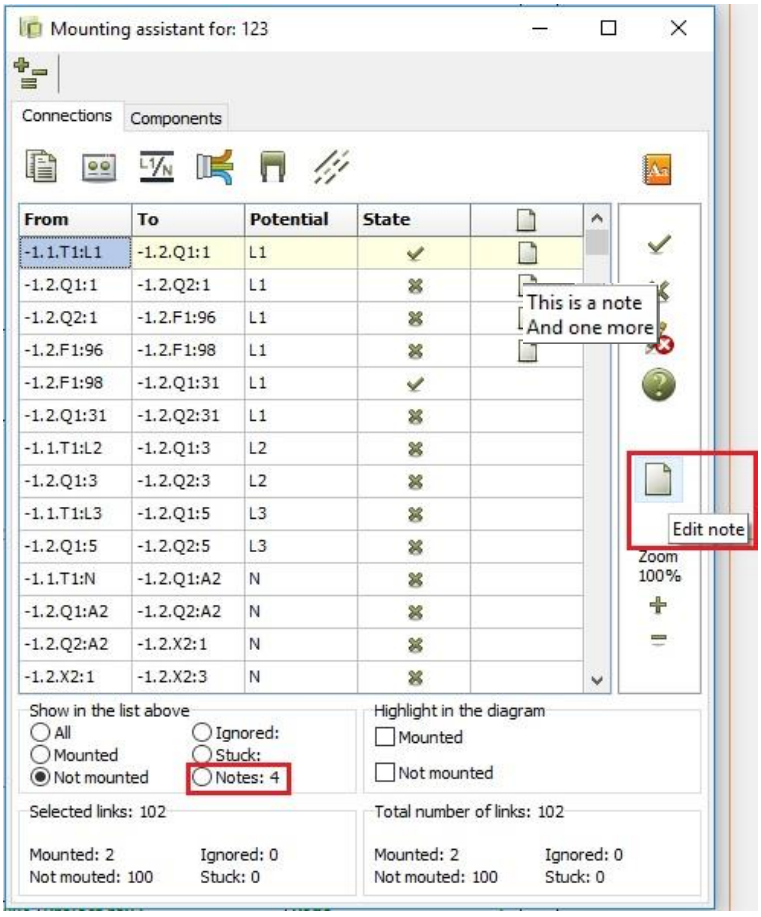


CHANGES IN THE PROJECT

If someone made changes in the project and it conflicts with some of the connections you have already mounted, a dialog box will pop up next time you open the job. You can see in the list exactly which changes have been made and by who. You can choose to print a list or just acknowledge the changes.



NOTES

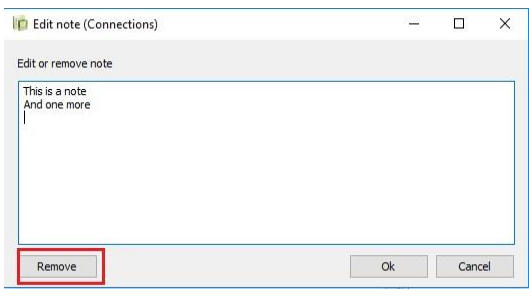


You can Write a note for every connection. The notes are locked in the project database and can be seen by everyone with access to the project.

Click Edit note, write your text and klick OK. If you place the mouse on a note, the text aperies as a hint.

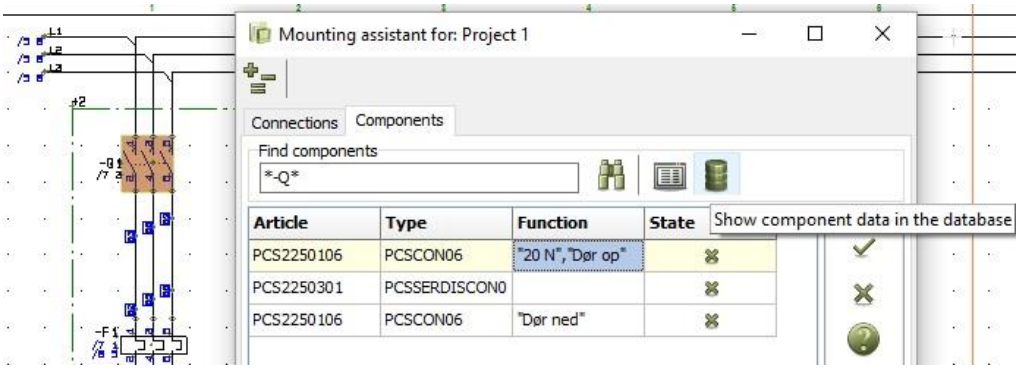
If you click Notes in the “Show in the list above” you only see the connections with notes.

Do you want to remove a note, click Edit note and then click Remove. And the note is no longer in the list



POST-PROCESSING

If you have components that need post-processing e.g. timers that need to be set or components that must be tightened with a specified torque, the necessary information can be found either in the diagram or in the component database so you can look it up.



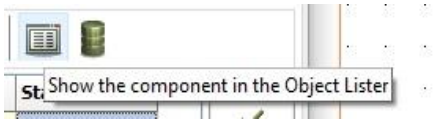
Click on the Components tab and search for the components that needs post-processing by typing *name*

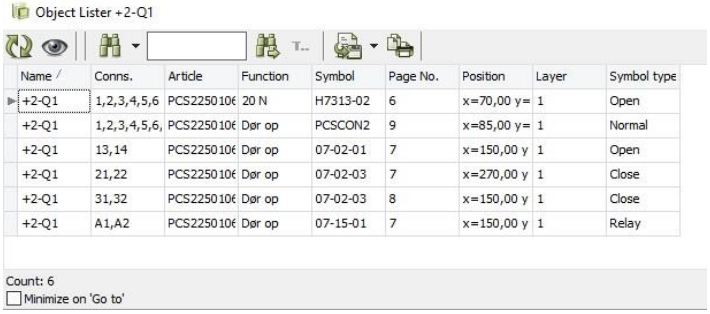
The components are showed in the window and if the component you have selected in the Mounting Assistant window is also highlighted in the diagram. If the information is added in a symbol datafield its easily found in the diagram. If not, you can look it up in the database by clicking on the database icon.

If you want to look up information in the component database, you must connect a database. The database is not connected automatically when you install the program Automation Service. See how to connect it on page 18.

You can double click on a symbol in the diagram and go directly to the data base when its assigned.

You can also get information about components by using the Object list which you can also access from the Mounting Assistant window by clicking on the icon.





Note

You can't change, correct or delete anything in the diagram.

All your connections and information about how far you are is logged in a separate database.

When you close the Mounting Assistant, it remembers how far you are and starts up just where you left it.

By giving the job a specific order number, you can be more than one person using the same pro-file at the same time. It's the unique name or number that identifies your job.