



AUTOMATION

Description of new functions

Version 20.0

This document describes the new functions in Automation version 20 in connection with the program release early 2018.

PCSCHEMATIC Automation has its own manual, which is included in the program.

The Panelrouter, the Component Wizard and PCSCHEMATIC Automation Service (the Mounting Assistant) have their own dedicated manuals describing their functions, and those manuals are also included in the Automation program.

Last edit: May 2018

Contents

1	'Show available' has extended functionality	5
1.1	Generally	5
1.1.1	The window consists of two parts, and it can be moved to display 2.....	5
1.1.2	New icons	5
1.1.3	Filtering functions	6
1.1.4	The page's database field	6
1.2	Placing components in a diagram page	7
1.2.1	Placing a component with 1 diagram symbol.....	7
1.2.2	Placing a component with more diagram symbols.....	7
1.2.3	Placing plcs	7
1.2.4	Optional electrical accessory.....	8
1.2.5	Placing terminals	9
1.2.6	Component with alternatives	9
1.2.7	Component with 'alternative in alternative'	10
1.3	Placing components on the layout page	11
1.3.1	Placing one component.....	11
1.3.2	Placing a terminal row	11
1.3.3	Optional electrical accessories / add-on	11
1.3.4	Mechanical accessories	11
1.4	Shortcuts	12
1.4.1	F9 – Show available symbols.....	12
1.4.2	Shift + F9 – Show components	12
1.4.3	Ctrl + F9 – Show available symbols again	12
1.4.4	Alt + F9 – Show symbols for component (new shortcut).....	12
1.5	If you work with a display with low resolution	13
1.6	Deactivating the Components window	13
2	The Mounting Assistant	14
3	Arrangement drawing or not ... A case about creating output for wire assembly machines	15
3.1	Repetition af arbejde på mekaniske sider	16
3.1.1	Use the layers for different panel parts, ie mounting plate and rails	16
3.1.2	Use panel module sizes on the page	16
3.1.3	Modularized symbols.....	17
3.1.4	Mechanical load	17
3.1.5	Placing terminals	17
3.1.6	Move area with new reference point	17
3.1.7	Import from dwg/dxf.....	17
3.1.8	Export to dwg/dxf.....	17
4	Mechanical components	18
4.1	The future and more (available) production data	19
4.2	Create a mechanical component symbol ion the database	19
5	The Panelrouter	20
5.1	The database and mechanical symbols.....	20
5.2	Line data fields	20
5.3	The Panelrouter menu.....	20
5.4	The Panelrouter output.....	20
5.5	A little repetition and 'Nice to know' about diagrams and mech pages.....	21
5.6	Where to place objects on the mechanical page	22
5.6.1	Routing classes	22
5.6.2	Dedicated wire trays	22
5.6.3	Blocked routes.....	22
5.6.4	Overview of Routing Classes	22
6	Line issues	23
6.1	Line data fields	23

6.2	Line data can be included in the Components list	23
6.3	Insert signal symbol on parked line	23
6.4	Change from Dots to bend and from Bends to dots	24
6.5	Text leader	24
7	Smaller bits and pieces	25
7.1	Change drawing header keeps state	25
7.2	Format files are language independent	25
7.3	Possible to select visibility on Symbol data fields	25
7.4	New default shortcuts	26
7.5	The OSI font is NOT included	26
7.5.1	You can find the OSI font here	26
7.5.2	Add the font to your setup	26
7.5.3	How to use the font in your projects	27
7.6	New data field	28
7.7	New special setting	28
7.8	Replace symbol	28
7.9	Symbols in states – insert new state	29
7.10	New page type – Connection list	30
8	My notes	31

1 'Show available' has extended functionality

The Show available function – which is called the Component Window from now on – has been extended so that it has more functions included.

The main purpose for this change has been to move focus to the fact that we work with Components and not mere symbols, and this improve the overview of components and their add-ons. We also wanted to tidy up in the different list functions in the program; functions that have been very like, but still different. Finally, we have a wish to improve operation speed in the program.

1.1 Generally

The Components window is open when you start the program. You can close it – and open it again – by clicking the double arrows in the top of the window. The window and the project are always synchronized: this means that when you select a component in the project, the cursor will be moved to the same component in the window. Add-ons for components are independent components and thus they have their own lines in the list. The symbol is marked in the tree-structure – where it is dimmed in order to prevent you from selecting it again.

The synchronization between the windows also means, that naming of eg terminals is clear, so that you can name the terminal on the mechanical page and have the same name values on the diagram pages.

The data that is used in the Components window is primarily project data – which is a great improvement regarding speed – and connection to the external database is only made, when the button 'Update from database' is activated.

1.1.1 The window consists of two parts, and it can be moved to display 2

The window consists of two parts:

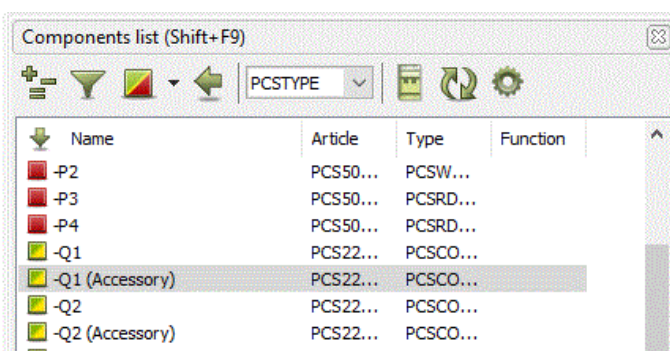
- The first window shows a list of components and the window contains variable new functions that are explained below
- The second window contains a tree structure that shows all component symbols and possible electrical accessories, partly an overview of all symbols (as in the 'old' Show available symbols)

The window can be undocked and subsequently placed on another display.

1.1.2 New icons

The window and its icons look like the old window, yet it looks differently:

- Filter for RDS has its own icon
- Text filter has its own icon – and works exactly as the currently (hidden) right-click function
- Show components: Red/yellow /green markings as in other 'old' lists
- Return to last used component
- The page's database field can be changed directly from here
- Show the component's symbols and their location (new icon instead of the 'eye')
- Update from database, meaning that you create connection to the external database
- Settings (wheel): select columns (and change their size and positions later)



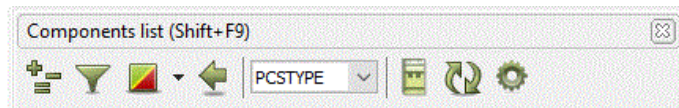
The symbol window also contains new icons:

- Show symbols 'stacked' ie 12 x plc input or show all 12
- Select symbol size (the wheel)
- Use filter

Read more about the icons and how to use them in the section about plcs on page 7.

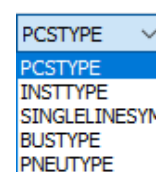
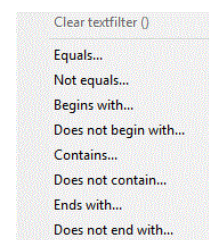
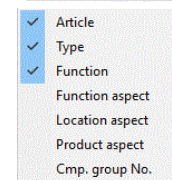
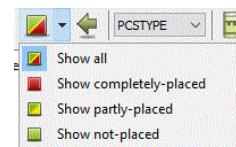
1.1.3 Filtering functions

The window can be sorted A-Z / Z-A on all used columns, ie type or function. Name is default.



Apart from that you have the following filters:

- Show components: Red/yellow/green markings as in other 'old' lists
- Settings (wheel): select columns (and change size and sequence later)
- Text filter
- RDS – reference designations – have their own icon
- The page's database field



1.1.4 The page's database field

The page's database field is also visible, and this is also where you change the database field for the page, instead of entering the current page's Page settings. This means that you don't need as many mouse clicks. The selectable fields are set up – as earlier – under Database settings.

If you have 'old' page templates that are set up to a non-default database field (ie other than PCSTYPE or MECTYPE) then the database field will automatically be seen here.

1.2 Placing components in a diagram page

When you select components in the database – possibly by using the pickmenu – different things happen, depending on the type of component:

1.2.1 Placing a component with 1 diagram symbol

When you select a component with only one electrical symbol, this symbol will be visible in the cross hairs and can be placed directly.

The symbol is also showed in Available symbols.

If the component has optional electrical accessories, they are seen directly in the pickmenu.

The symbol sticks to the cross hairs, making it possible to place more components with the same article number immediately; also, automatic numbering using Ctrl++ or Ctrl+? is possible.

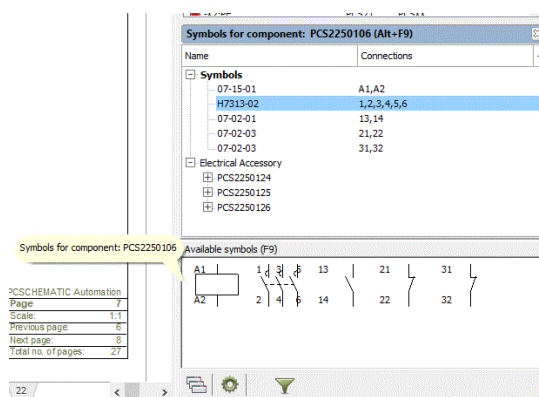
When the symbols are placed and named, they turn up in the list in the Components window.

1.2.2 Placing a component with more diagram symbols

When you place a symbol with more than one electrical symbol, all symbols are placed in the symbol window:

The component is not on the components list – it hasn't been placed yet – but it is ready, and you can see what it consists of, in the tree structure and in the symbol window.

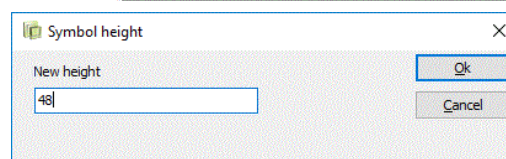
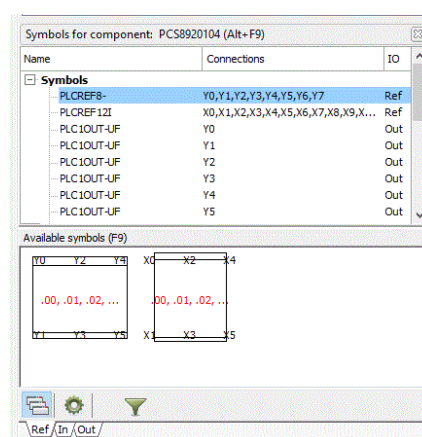
Select one of the symbols and place it in the diagram, where you also name it, possibly also with reference designations.



1.2.3 Placing plcs

When you select a plc in the database, the symbols are also seen in the bottom window. As earlier, the symbols are sorted as ref. symbols, inputs, outputs, other symbols, respectively.

In the tree-structure, you can see all connection names and if you wish to see more details, it is possible to adjust the symbol size when you click the wheel.



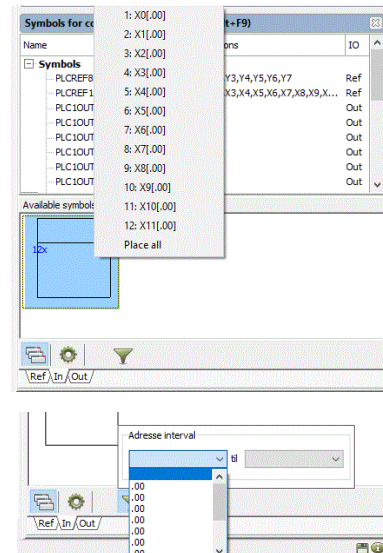
1.2.3.1 Placing IO's

On the tabs for inputs and outputs, it is possible to see the symbols stacked or one by one.

When you want to place the symbols, you can select one symbol at a time, either in the tree structure or from the list, which appears on mouse clicks.

When you select 'Place all', you get one symbol per current path.

As earlier, you also have a filter function in the window, which is very useful when working with large plcs.

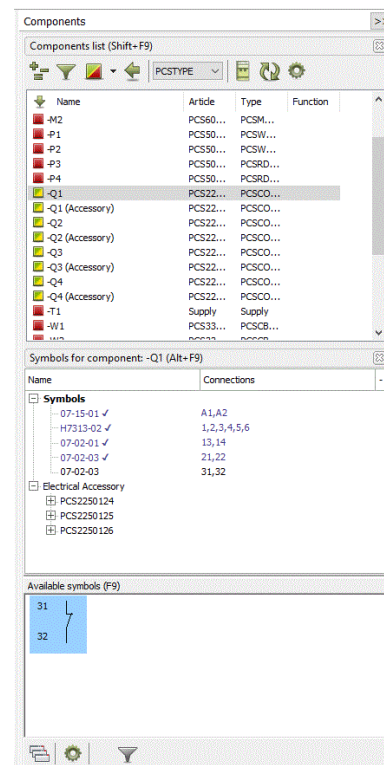


1.2.4 Optional electrical accessory

For components with defined electrical accessories in the database, you can see the article numbers for the accessories in the tree structure.

In this way, it is possible to get a quick overview for optional accessories for the components.

When you select an accessory, you can see its symbols in the symbol window, and when you place one of its symbols, you get a new line in the Components window with the accessory, named after the main component and (accessory).

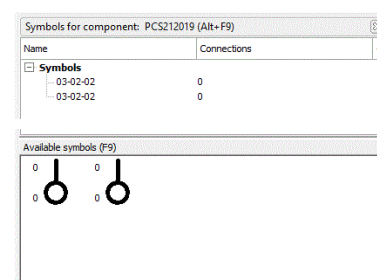


1.2.5 Placing terminals

Placing of terminals, particularly multi-layer terminals, has always been a challenge.

When you select a 2-layer terminal in the database, it pops up with 2 terminal symbols in the Available symbols window.

The first one (from the left) is understood as layer 1, the next as layer 2 etc, no matter how and when you place them.

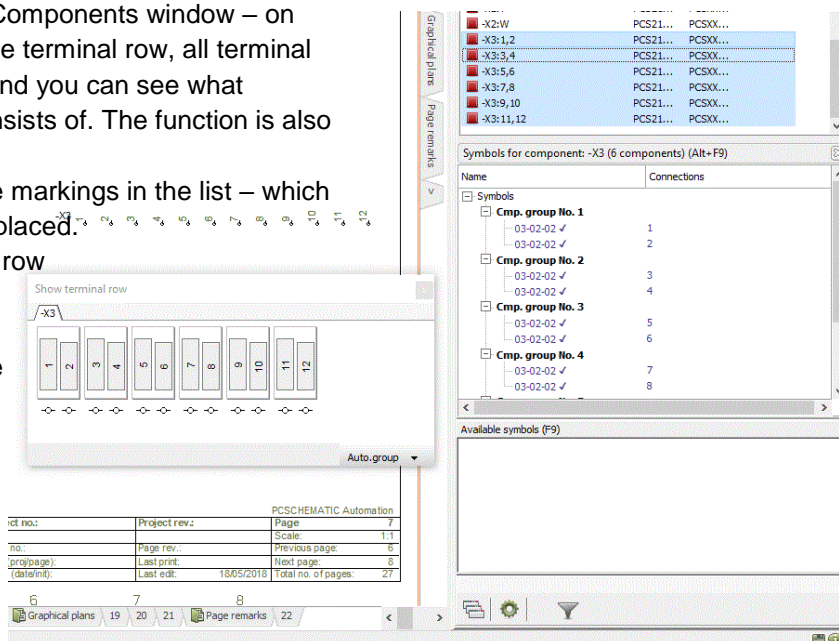


If you double-click – in the Components window – on one of the components in the terminal row, all terminal components are selected, and you can see what components the full row consists of. The function is also in the right-click menu.

You can also see – from the markings in the list – which parts of the terminal that is placed.

The function Show terminal row also shows the component grouping.

This should help to increase the overview of terminals.



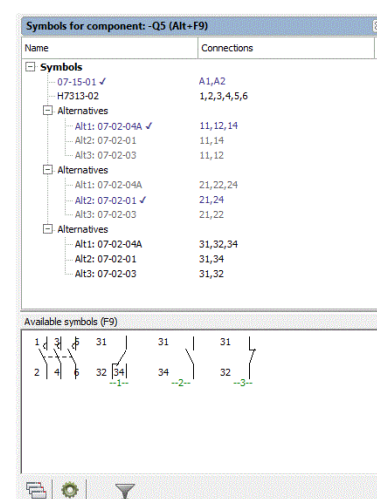
1.2.6 Component with alternatives

Components with alternatives, that is that one or more electrical functions have different symbol options, have been shown as a long series of symbols, which has made it hard to grasp all options.

Now you can see the symbol selections in the tree-structure, which makes it easy to get an overview.

Furthermore, it is also clear which symbols you selected, and which symbols you made the selection from.

Once again, you can make the selection either in the tree-structure or in the symbol window.



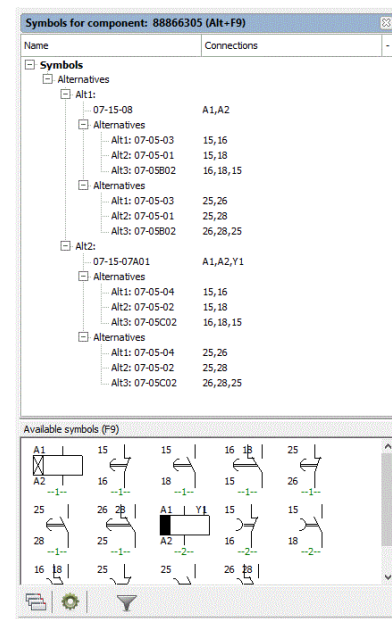
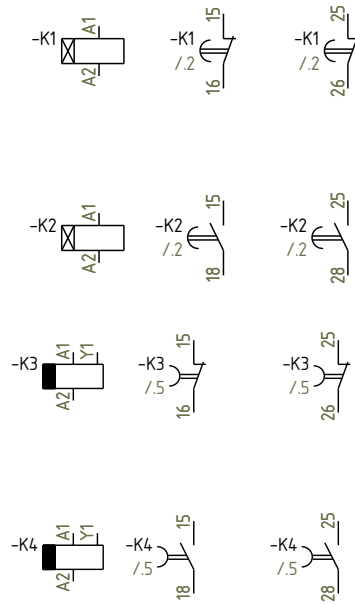
1.2.7 Component with 'alternative in alternative'

A new function in version 20 allows you to have 'alternatives in alternatives'. The example here is that first you select a coil – there are two different ones – and depending on your coil selection you can select contacts.

That means that Alternative1 is the Coil, Alternative2 is the contacts.

Until now, it was necessary to 'multiply the brackets', which resulted in a long list of alternatives and a lacking overview.

Here you can see how the component is presented in the tree-structure and four examples of how you could present it in the diagram.



The function is not yet supported by the Component Wizard.

1.3 Placing components on the layout page

The Arrangement page, the layout page, the mechanical page ... When you use the mechanical page, you get a mechanical view of the project, where you can see whether you can fit all components into the available space.

1.3.1 Placing one component

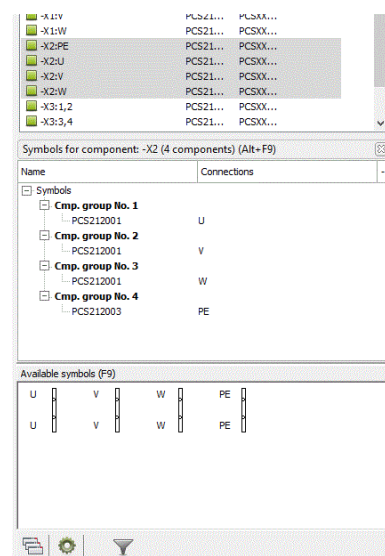
When you place components on the mechanical page, you use the function Mechanical Placement in the right-click menu or you use the Show available function.

When you use the Components window, you select the mechanical symbol in the Symbol window. If you have made a setup to use the new Automec symbols, you will see these symbols in the window.

1.3.2 Placing a terminal row

In the list you can see all individual components in the terminal row. You select the complete terminal row by either double-clicking on one of the terminals or by selecting the Select name option in the right-click menu.

When the terminal row is selected, right-click in the Symbol window and use the Place all function to place the full terminal row.



1.3.3 Optional electrical accessories / add-on

If you selected electrical accessories in the diagram, the Components list has one line for each component – the main component and the accessory – as both components have the same name but different article numbers.

You can also use the double-click or Select name function here, and subsequently use the Place all function.

1.3.4 Mechanical accessories

Fixed mechanical accessories, that is the accessories defined in the database field Accessory, is now seen as a mechanical symbol that can be placed on the layout page.

You select and place Optional mechanical accessories on this page, eg end and divider plates for terminals. Please note, that if you need to place 2 dividers – one in both ends of the terminal row – the database field must define 2 pieces of the article number.

1.4 Shortcuts

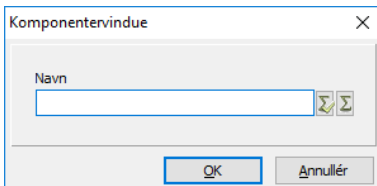
All shortcuts for Show available functions are still on the F9-key. And if possible, the same function has the same key as before. In the picture below, we have highlighted the three windows' names, and in the section below, you can read more about each shortcut.

1.4.1 F9 – Show available symbols

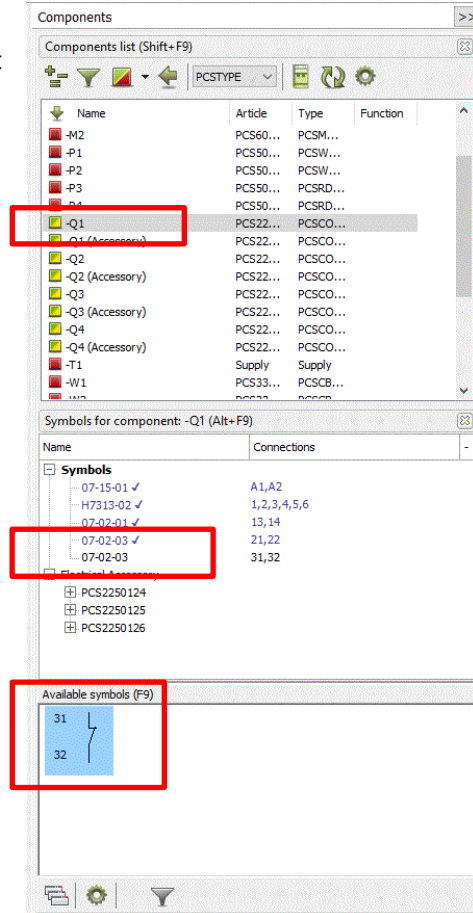
If you select a symbol belonging to a component with available symbols, focus goes to the symbol window., so that you can – also using the keypad – select another symbol.

If there are no available symbols in the component, focus still goes to the symbol window, but you only see the component if you Show all components (red/yellow/green) is active.

If the Components window has been closed, it opens automatically, when you press F9, still what you actually see depends on your settings.



If you haven't selected any symbols, a small window pops up in which you can select a component among the project's components.



1.4.2 Shift + F9 – Show components

When you press this shortcut, focus goes to the Components list, that is the list of all components in the project.

1.4.3 Ctrl + F9 – Show available symbols again

When you press this shortcut, the focus goes back to the symbol grid for the component, that you just placed symbols for.

If there are no available symbols, nothing happens.

1.4.4 Alt + F9 – Show symbols for component (new shortcut)

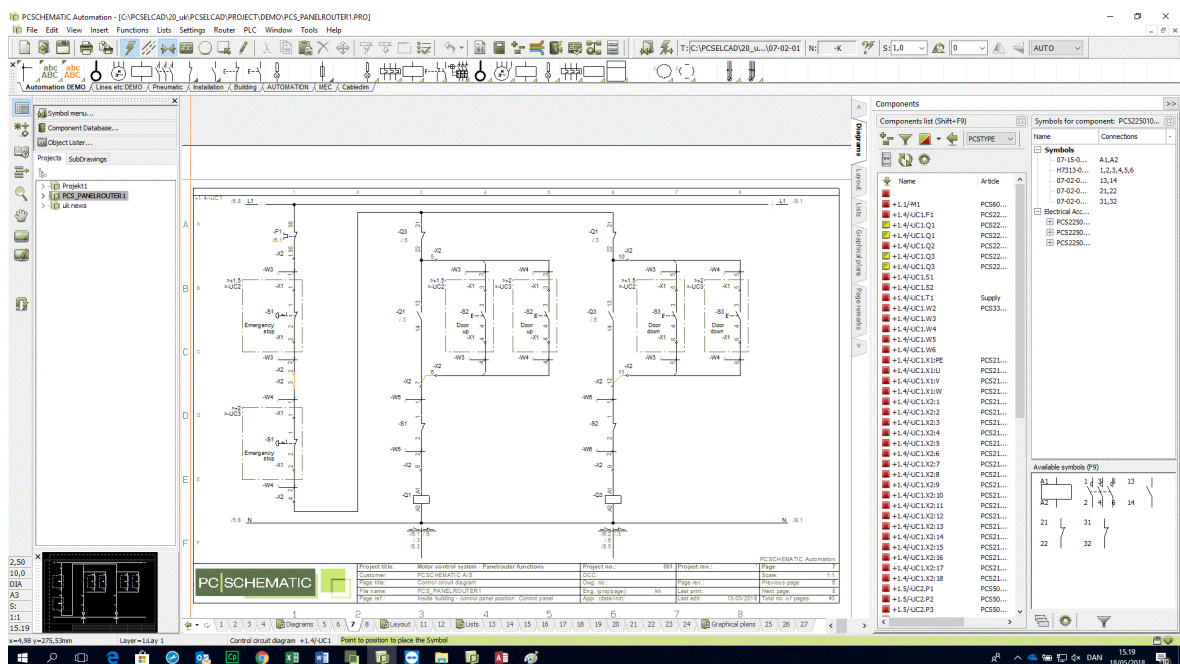
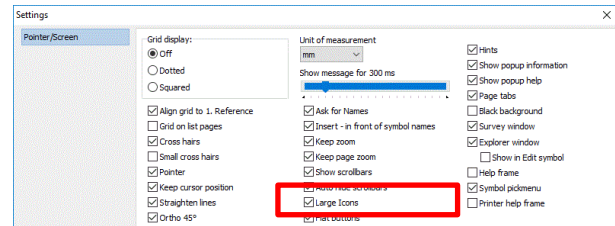
When you press this (new) shortcut, you go to the tree structure, from which you can also select symbols.

1.5 If you work with a display with low resolution

When you work with a display with low resolution – maybe your portable pc – it may be a challenge to fit everything on the display.

You can find a few tricks here:

- All icons can be Small icons.
- Maybe you can put your tool bars on one line.
- You can reduce the height of the pickmenu
- You can close or just reduce the size of the explorer window
- Finally, you can choose to split the new Components window vertically instead of horizontally – which also works quite ok, also when the resolution is high.

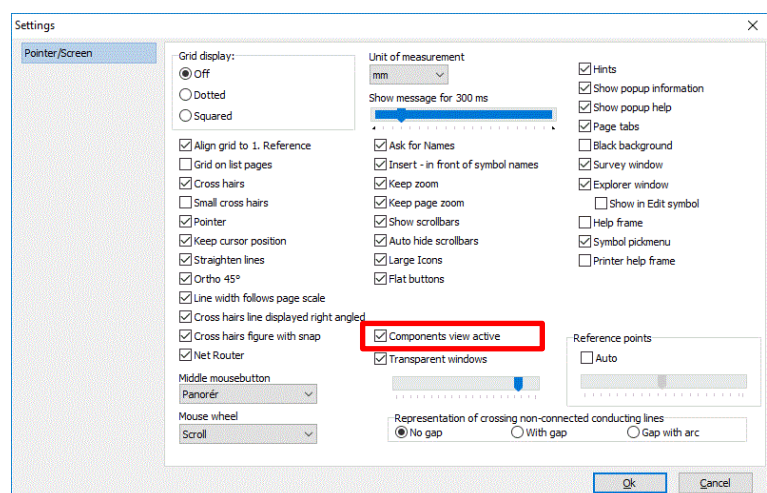


1.6 Deactivating the Components window

You can deactivate the Components window in Settings|Pointer/screen.

The setting is not active before you restart the program.

If you deactivate the function, it also means that you cannot search components with available symbols using the F9 key.

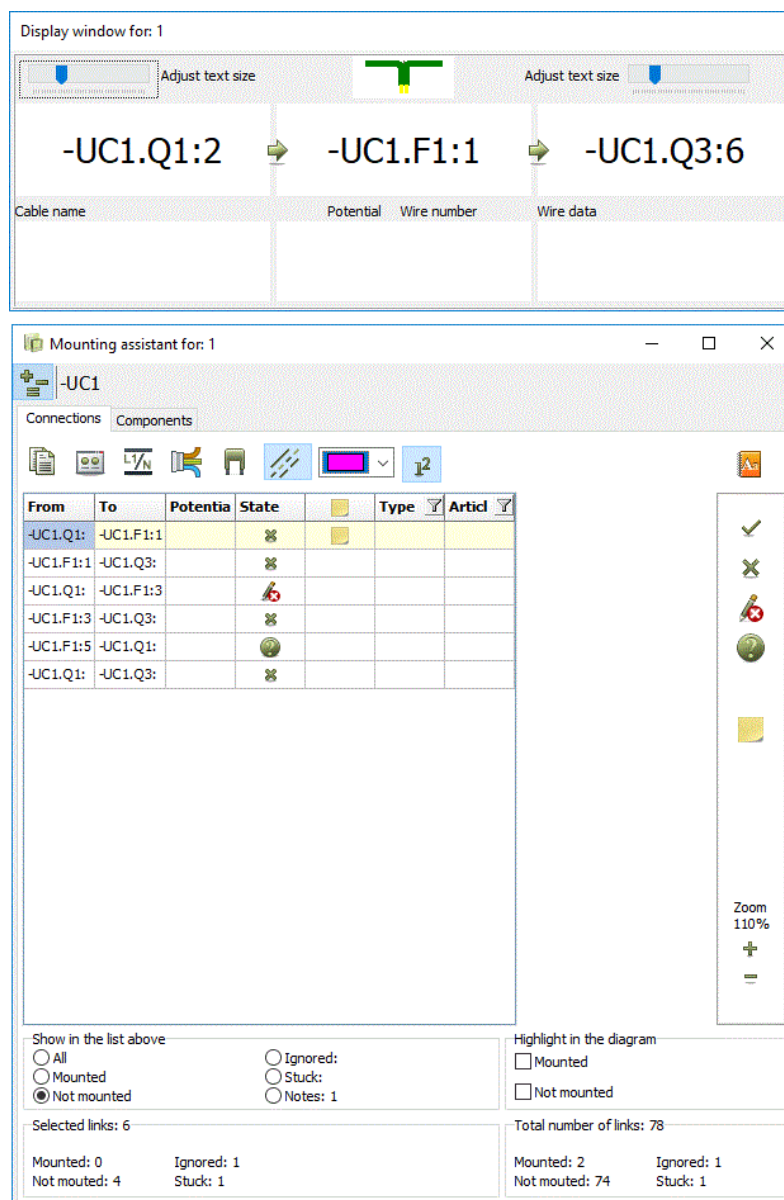


2 The Mounting Assistant

The Mounting Assistant, which is developed for – and is a part of – the Service program, is also part of the Automation program from version 19. The Service program has its own manual, in which you can see detailed description of all functions. The manual is also included in the Automation program.

Since the introduction last year, we have made further enhancements to the program:

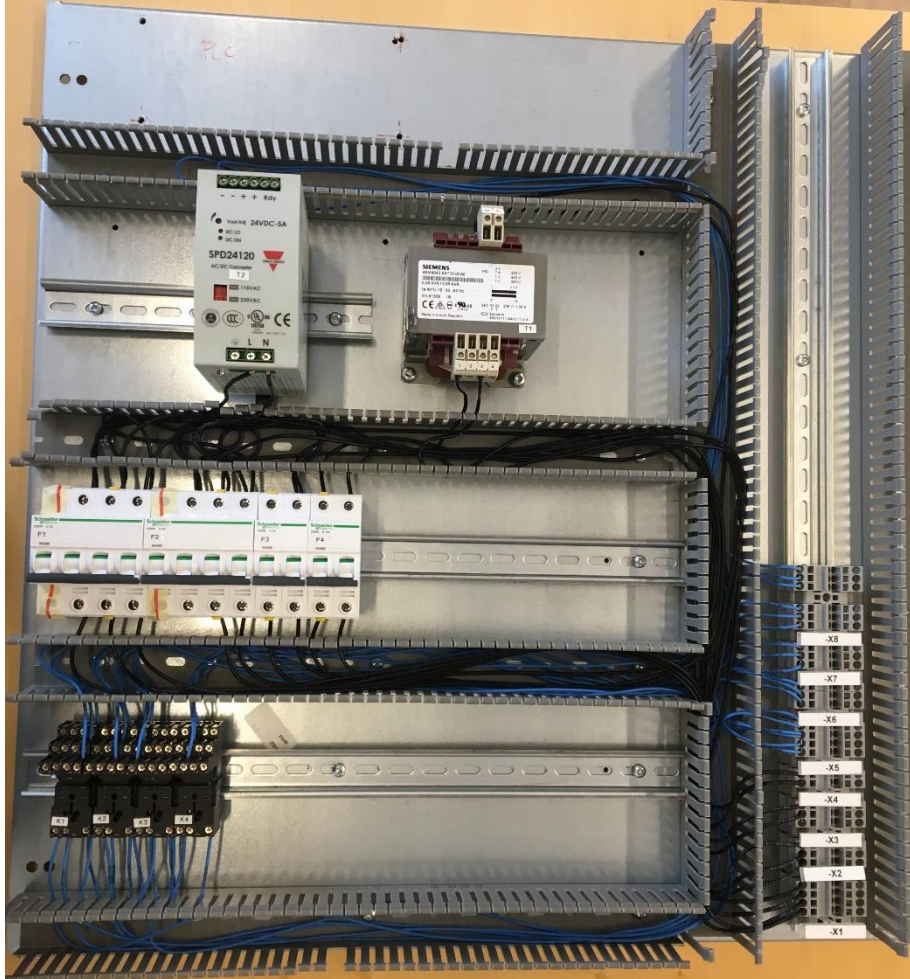
- Both parts of the overview window can now be resized
- Possible to add notes to all connections
- Filter options when selecting component connections – type/article no/function/note
- Filter options when selecting wires – type/article no/color/wire no
- Load company guide lines
- The Component tab also includes filters for components – type/article no/function/note
- And access to company guidelines, Object lister and Database



3 Arrangement drawing or not ... A case about creating output for wire assembly machines

One of the comments that came over and over, when we presented the Mounting Assistant last year was 'what about wire lengths?'

This goes through a case where you work on the mechanical page – and below you can see physical mounting plate with real components on. And here follows the long answer to the short question 'What about wire lengths?'



The chapter is structured according to a number, that indicates where you are in the process.

We know that a lot of our customers don't use the mechanical page for various reasons. The chapter here guides you through the options that you have and how you carry them out practically.

Electrical diagrams with components

Placing components on the mechanical page

- Modul sizes
- Export to punching machinery
- Parts lists including mounting rails and wire ducts
- Real mechanical symbols
- Wire lengths
- Production of wires with the correct finishing – wire assembly

3.1 Repetition af arbejde på mekaniske sider

The short version about working on the mechanical page is that either you use the Show available function – now the Components window – or you use the Mechanical load function from the right-click menu.

In the Mechanical load function, you can place all components or a selection of components by using the Σ , maybe in combination with * for a larger selection.

You can place the components' mechanical symbol (from MECTYPE) or you can choose to generate boxes.

Everything on the mechanical pages is with true measures, and you can add measurements to the drawing by using the Measurement command. That means that it is easy to get an overview of whether all components fit into the selected cabinet. It is also possible to place wire trays, and if you have article data for them, they can also be on the parts list with the length used on the mechanical drawing.

But if you wish more than 'just' an overview, it can be a tedious task. However, all data is in the same file, with all the functionality that comes with easily updated lists and overview.

Below you can see and read about a couple of tricks to easier creation of an even better arrangement page.

3.1.1 Use the layers for different panel parts, ie mounting plate and rails

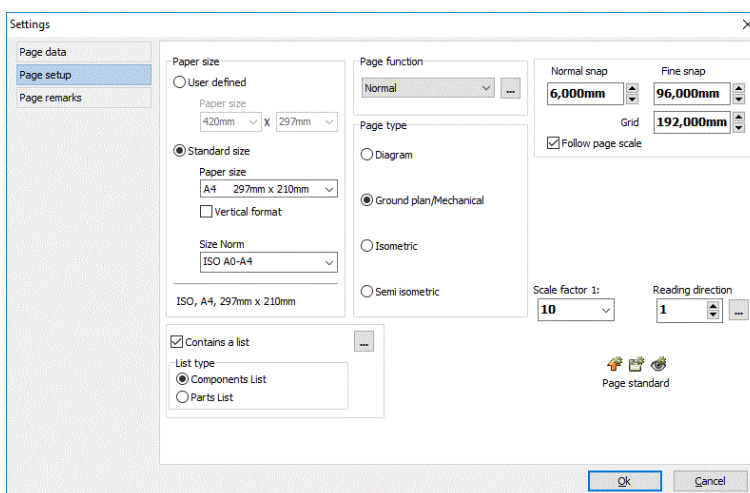
If you need to change parts in the panel – and you often do – it can be an advantage to use the layers. In that way the mounting plate can be placed in one layer, the mounting rails in another, wire trays in a third layer, components in a fourth etc..

3.1.2 Use panel module sizes on the page

When you work with normal panel types, it can be a good idea to create templates that fit when the various types:

Here you can see a setup for one type of panel (Tabula (dk?)), and yes – we have made a logical switch between the fine and the normal snap, but that is made on purpose.

We have selected to follow the page scale, which means that the dots refer to the 'real' 192 mm distance.



The finest snap (6 mm) – and the explanation for switching the snaps – is the most commonly used and that's why it is set up as normal snap. The fine snap (96 mm) is only used sometimes, so therefore it is defined as the exception = fine snap. The shortcut to use the fine snap is Shift+mouse.

3.1.3 Modularized symbols

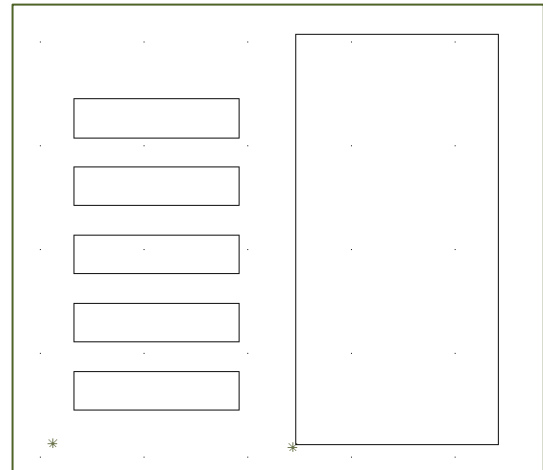
It is possible to create mechanical symbols that fit perfectly with the module sizes.

Here is some inspiration:

The symbol to the left is a set of mounting rails, made to fit to a one module-wide panel. It is made as a symbol in states, where the states refer to modules that are 1, 2, 3 etc modules high.

The symbol to the right is the related mounting plate, also made as a symbol in states, where the states is 1, 2, 3 etc modules.

The trick here is the states and that the reference point (*) is positioned in the modules corner, which is outside of the symbol.



The symbols are placed in the correct module position by using the fine snap, see paragraph 3.1.2.

3.1.4 Mechanical load

When you place individual components on the arrangement page, you often want them to be placed side by side. When you select them with the Mechanical load function, you can do this automatically, when you select them from the Components window, you need to do it manually, or use the Magnetic snap function from the vertical menu.

3.1.4.1 Snap function for mounting rails

There is a trick to make the first component snap to a mounting rail:

On the symbol for mounting rails (see 3.1.3) there is a *RECTANGLE* made with a NP-colored line in layer 255. You can read more about this function and symbol design in the manual.

3.1.5 Placing terminals

It is ok with one more repetition of this subject: when the terminals are grouped correctly – and have article numbers – you can select a complete terminal row by using the Components window, and the order is correct.

If Magnetic snap is on, the group will snap as one entity.

3.1.6 Move area with new reference point

When you move something, you use the Area command. Here you can also – in the right-click menu – select an area reference point.

3.1.7 Import from dwg/dxf

It is also possible to start the arrangement drawing by importing a dwg/dxf-file.

There are different tools, with which you start making the mounting plate, all of them with preparation for drilling or punching holes.

3.1.8 Export to dwg/dxf

When you finish designing the mounting plate, you can export it to dxf. The export function allows you to export one layer per drawing thus making it possible to have mounting holes in specific layers, which can be 'printed' to the dxf-file and used in the production, eg by punching machines.

4 Mechanical components

The largest job when talking about making the arrangement pages work, is making the mechanical symbols for the components.

When we started making mechanical symbols in the program, they were made from dwg-drawings, that had added (electrical) connection points. The drawing was flat as in 2D. The fact that the symbols have had the electrical connection points from the very beginning, means that it has always been possible to see the electrical connections on the mechanical pages. For many, many years ...

Making those symbols is a tedious work, which has led to a high amount of not-existing mechanical symbols and to many simplified symbols which are empty boxes with the right measures, eg #x40mmx70mm for a component of 40 mm wide and 70 mm high.

This way of making mechanical symbols has been ok if the only requirement to the arrangement drawing was to see whether all components could fit into the box. And if you could live with variable graphical quality.

It has also been possible to generate boxes based on the actual sizes of the detailed symbols.

A very useful solution has been to create a number of 'standard symbols' for eg MCBs; easy because they come in modular sizes:

- Create a box with the right measures
- If you like, you can 'decorate' it, so that you can easily differ between different basic component types
- Place connection points with the true names – the photo shows a GE-component. In the panel itself, we have placed a Schneider component with the same function, but physically the N terminals have a different position. That means that they need two different mechanical symbols
- Remember, that it is also necessary to differ between two connection terminals with the same name: The actual component has 2 x N: one is N, the other N'. They could also be N1 and N2, thus using the convention about 'uneven connections at the top and even ones at the bottom'
- Place desired symbol data fields on the symbol
- Save it with where you can find it.

In relation to the Panelrouter, we use the database datafield HEIGHT to indicated the physical height of (all) connection points on the component.



4.1 The future and more (available) production data

In relation with customers wanting more and more exact production data, it is necessary to have more information about the individual connection points. Not only the names, but very much also exact position and rules about mounting the individual terminal.

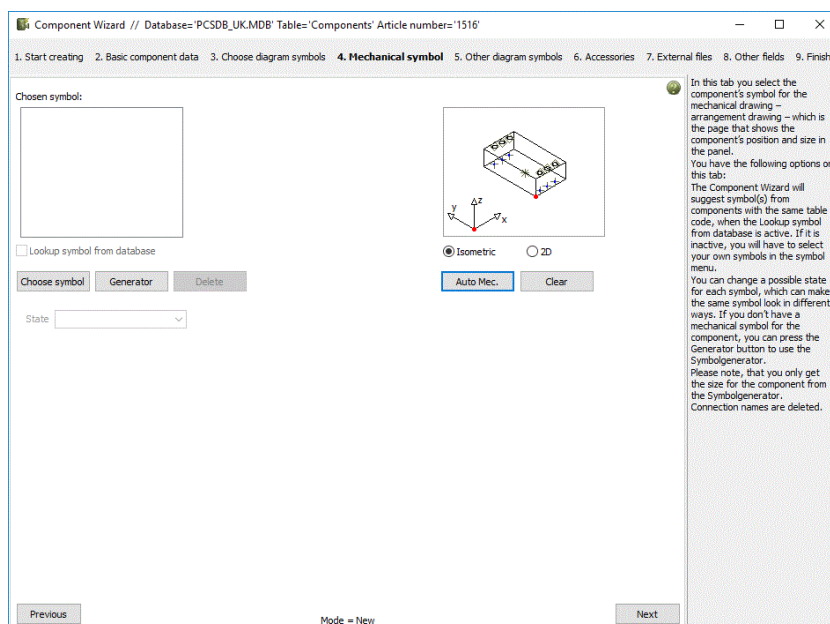
In the future, the database will also contain the possibility to either import this data from other databases (ecl@ss) or directly type it into by using the Component Wizard.

The database comes with a new data field for those mechanical components, and the field is of a new type – Widememo – and data in this field can only be added and edited by using the Component Wizard, and not by editing directly in the database.

4.2 Create a mechanical component symbol ion the database

You use the Component Wizard to create mechanical components.

The Mechanical symbol tab contains a new section in which you can create and have a preview of the Automec symbol.

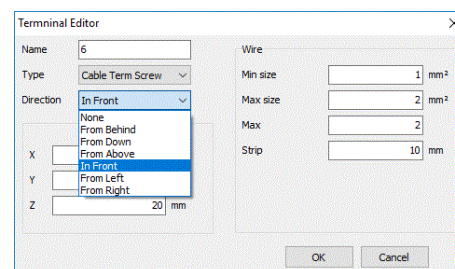
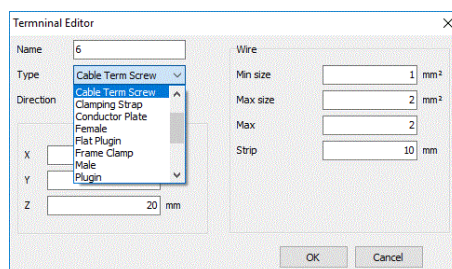
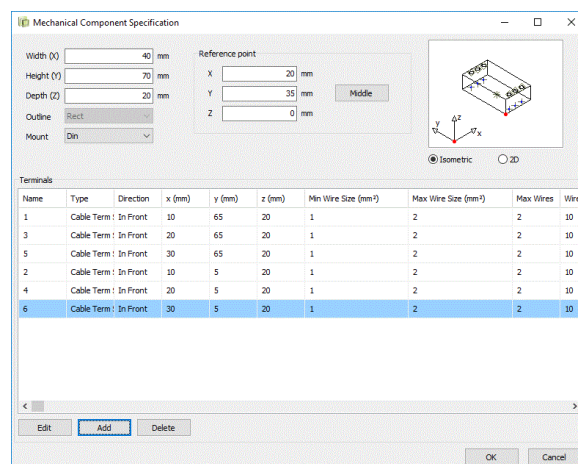


The mechanical symbol is created by using its physical measurements and its mounting method.

As you type the data, the program generates a preview that shows the component.

An important part of the information for the mechanical component is about each connection point, its type and mounting direction as well as wire size and number.

The data is used when we generate wire lists for eg wire cutting and assembly machines.



5 The Panelrouter

The integrated tool the Panelrouter which can find wire lengths on the mechanical page, has also been improved in this version.

5.1 The database and mechanical symbols

The functionality depends on the availability of connection terminals on the mechanical symbol. You can make flat (2D) symbols as previously, but you need connection points to make it work, not the #xy-symbols.

2D-symbols in connection with a HEIGHT value in the database is sufficient for most components. Which is also the reason why there is a long explanation about how to make them (see page 18).

5.2 Line data fields

We have created a dedicated line data field for Route classes. In earlier versions of the Panelrouter, we have used the Function text for this, which is not an optimal solution.

Regarding line data fields, it is now also possible to select visibility for them; up till now they are always invisible. See more about lines on page 23.

5.3 The Panelrouter menu

The Panelrouter menu has more tabs, in which it is possible to examine the project even further than earlier before the project is released for production.

There is a new tab which shows all nets in the project, and you can investigate each net individually.

There is a new tab which shows all components, which makes it possible to investigate individual (critical) components' connections.

And the 'old' tab showing Route classes is still there, making it possible to investigate eg all Power connections easily or critical Signal connections. In the demo example, we have used the class Power, XXX and Signal. The Route class function lets you get a quick overview of the critical connections, mainly to prevent them from being routed in the same wire trays.

Finally, there is a tab showing the actual settings for the Panelrouter.

5.4 The Panelrouter output

The Panelrouter Output menu – which is found in the right click menu on the mechanical page, has new tabs for output directly to two different Komax wire assembly machines.

Any other interesting output formats? Input is very welcome at info@pcschematic.com.

5.5 A little repetition and 'Nice to know' about diagrams and mech pages

... to make the Panelrouter and Designcheck work correctly.

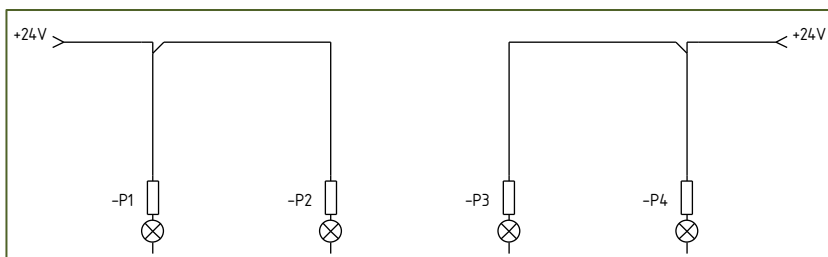
The Panelrouter places the connections between components as they are understood based on the diagrams. That means, that it is not required to make a mounting correct drawing, as dots are interpreted correctly.

You can alt+click on connections in the Panelrouter window when you want to examine the diagram to see whether the routing has been as expected. If not, you can go back to the diagram and make one or several connections mounting correct – see page 24.

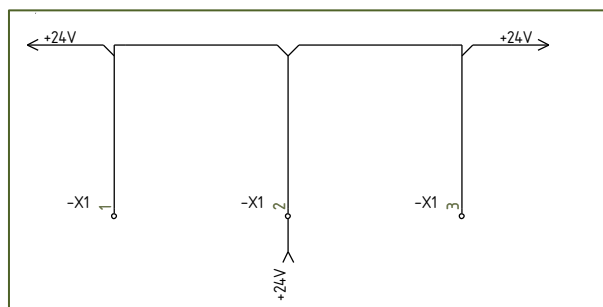
To make the change function correctly, you should understand what the various bends mean. If not, you risk that Designcheck finds errors 😊.

The direction of the bend shows where the supply/signal comes from:

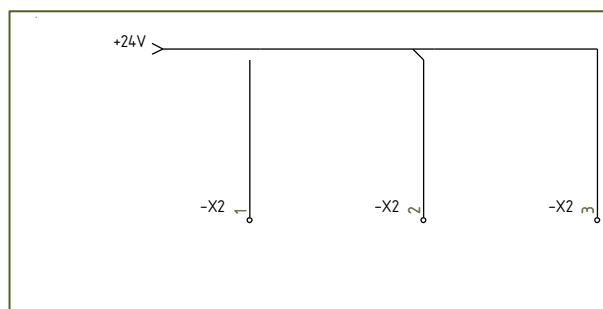
All bends here mean – mountingwise – that you loop on to the next connection. The direction is about showing the origin of the signal/supply.



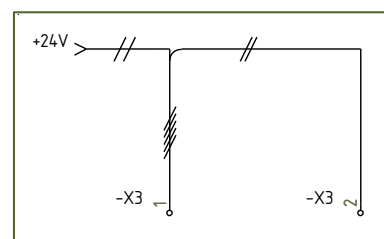
The same situation here, but the middle bend shows, that the signals goes to both directions.



Here the signal comes from the left and there is a wire to each of the terminals.



And the 'round' bends are for single line diagrams, showing wire bundles.



And just to make sure: there is no 'directional' functionality in the signal symbols with arrows; it is only a graphical symbol.

5.6 Where to place objects on the mechanical page

When you want to use the arrangement page to find the correct wire lengths, it is necessary to have a few rules for the wires' routes in the wire trays.

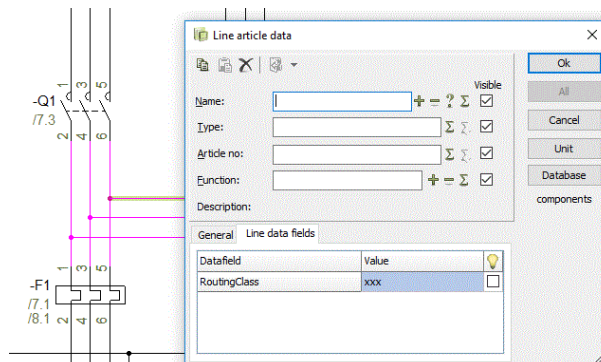
How to decide the actual connections between components is described above. But the physical position of the connections can be done in the diagram by using what we call Routing Classes.

5.6.1 Routing classes

In the diagram you can assign a Routing Class to a line.

RoutingClass is a predefined Line data field from version 20.

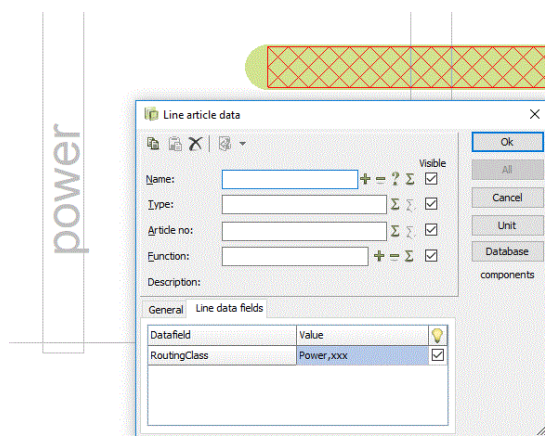
Routing Classes can be controlled on the mechanical page.



5.6.2 Dedicated wire trays

When you place your wire trays, you can assign one or more Routing Classes to it. Routing Classes are separated with commas – see picture.

When you assign Routing Classes to the tray, only wires with the same classes are allowed in the tray.

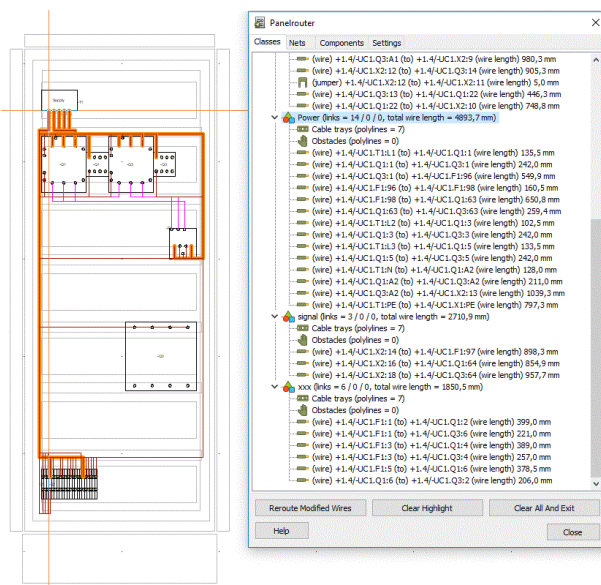


5.6.3 Blocked routes

You can also create obstacles. Obstacles are **red** lines with Routing Classes, and they will prevent lines with the selected Routing classes from passing the route.

5.6.4 Overview of Routing Classes

When your wires are routed, you can mark the Routing Classes one by one and visually ensure that critical connections are routed in separate trays.



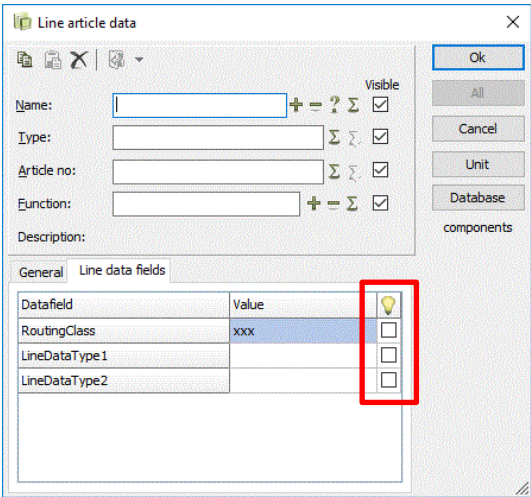
6 Line issues

Traditionally, lines have been 'simple' connection lines without any special data. It is possible to assign special properties – jumper, cable, no wire number – but mostly we don't assign line article data.

In relation to developing the Panelrouter and the Mounting Assistant, we get feedback about the need to have more line data, some data should preferably be visible in the project, other data should 'only' make it possible to control some outputs. All this makes us develop some 'line issues'.

6.1 Line data fields

Line data fields have become 'alive' and movable, and it is also possible to select visibility for each data field.



6.2 Line data can be included in the Components list

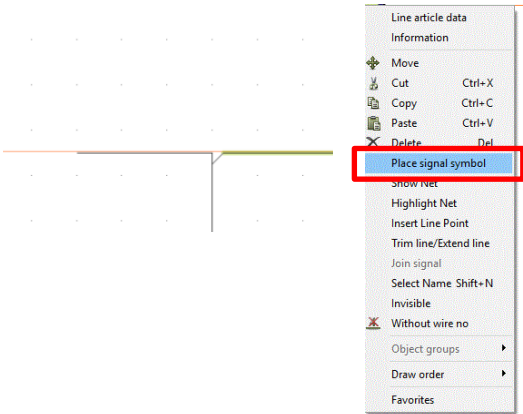
Line data fields can be included in the Components list. The created Line data fields are found in the Parts / Components list section in the Insert Datafield dialog.

The list can also be exported to eg Excel.

	A	B	C	D
1	Wire	RoutingClass	LineDataType1	LineDataType2
2	1	Power	aaa	bbb
3	2	Power	ddd	rrr
4	3	Signal	zzz	qqq
5				

6.3 Insert signal symbol on parked line

When you have a parked line in a project, it has been quite tedious to add a signal symbol to it. Up to now. With version 20 you get the option to add a Signal symbol directly through the right-click menu.



6.4 Change from Dots to bend and from Bends to dots

Until now, you can change between dots and bends either by redrawing single connections or by changing the full project to either dot or bend by using Functions|Special Functions|Convert dot to bend (and vice versa).

From version 20 you get the option to change

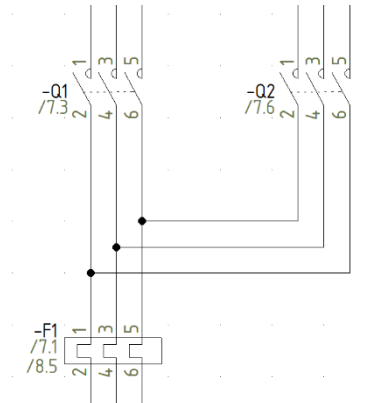
- A single connection
- All connections on the actual page
- All connection within a selected (line) area

You cannot Undo the function with Ctrl+Z, but you can change both ways, so simply select again to undo.

The function is found in the right-click menu.

Please note, that you need to mark the line when you change from Dot to bend; and you need to mark the bend when you want to change from Bend to dot.

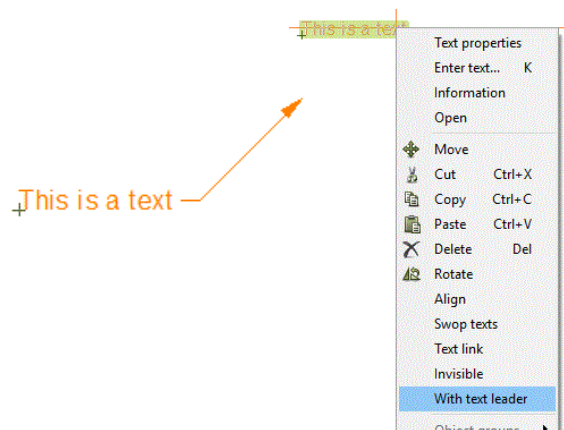
When you change to bends, the function automatically uses the standard bend as selected in Project data.



6.5 Text leader

You can add a text leader to all texts.

The function is in the right-click menu when you mark a text. You can also deselect the leader in the right-click menu.



7 Smaller bits and pieces

Also, this year – a selection of smaller functions and extension that have been made in the program.

7.1 Change drawing header keeps state

When you change a drawing header, the new one has been in state 0. From version 20 the new drawing header will have the same state as the one it replaced; if it has the same states (or more).

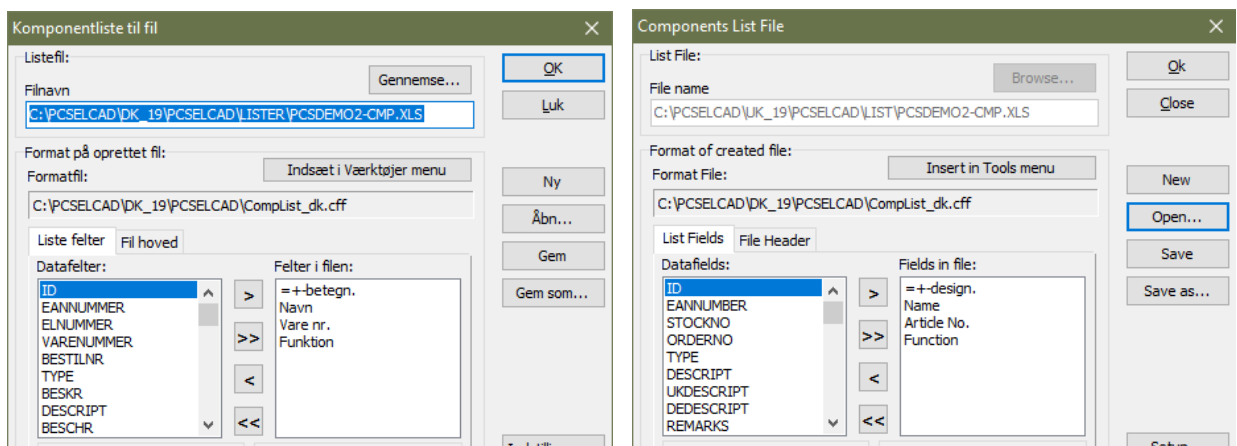
7.2 Format files are language independent

When you make a 'list export to file', you use a format file to decide which data you want in the list.

The data fields are internal data fields that have different names depending on the language you have chosen for your program in Settings|System.

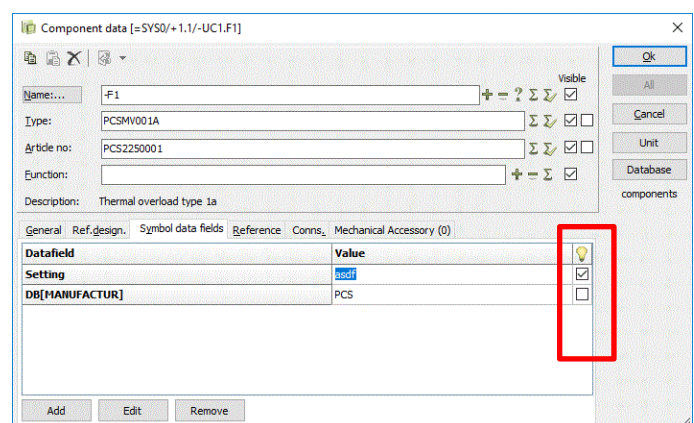
Note, that your own symbol and line data fields as well as your database fields have the names that you see in the list.

(In the pictures below, I have used two different program: a DK version and a UK version.)



7.3 Possible to select visibility on Symbol data fields

It is now possible to select visibility on individual symbol data fields.



7.4 New default shortcuts

When we released version 19, we created new default shortcuts. We have also added a few more from version 20.

All shortcuts can be seen in Settings|Shortcuts.

7.5 The OSI font is NOT included

The OSI font is NOT included, which means that if you want to use this font, you need to install it and add it to your PCSCHEMATIC installation.

Then it will work as all other (included) fonts.

7.5.1 You can find the OSI font here

I have used Google to find it, and used this link from Hikikomori82:

[GitHub - hikikomori82/osifont: Free TrueType font for CAD projects](https://github.com/hikikomori82/osifont)
<https://github.com/hikikomori82/osifont> ▼ Oversæt denne side
 Free TrueType font for CAD projects. Contribute to osifont development by creating an account on GitHub.
 Du har besøgt denne side den 30-11-17.

7.5.2 Add the font to your setup

Open your PCSCAD.ini file.

In the section [TextFonts] you add the line with OSIfont:

```
[TextFonts]
FontNo0=PCSCHEMATIC
FontNo1=Albertus Medium
FontNo2=Arial
FontNo3=Arial Narrow
FontNo4=Book Antiqua
FontNo5=Bookman Old Style
FontNo6=Century Gothic
FontNo7=Century Schoolbook
FontNo8=Comic Sans MS
FontNo9=Courier New
FontNo10=Garamond
FontNo11=Georgia
FontNo12=Lucida Console
FontNo13=Lucida Sans Unicode
FontNo14=Tahoma
FontNo15=Times New Roman
FontNo16=Trebuchet MS
FontNo17=Verdana
FontNo18=OSIfont
```

Save the file and restart the program ☺

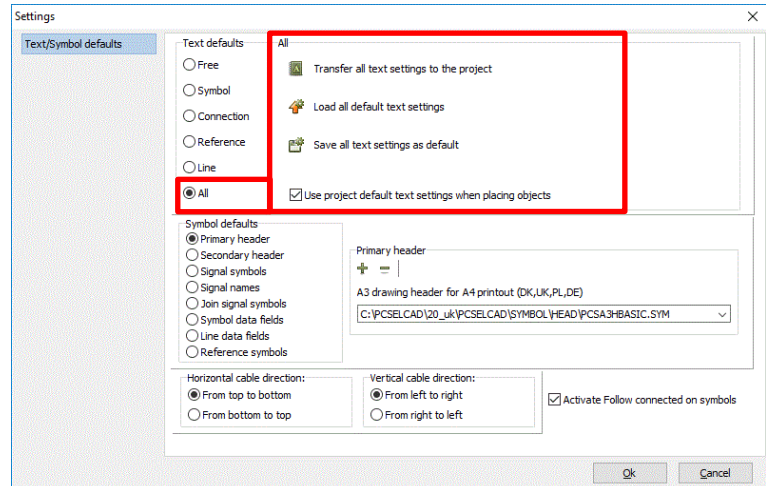
7.5.3 How to use the font in your projects

In the demo-folder, we have added to files in which all texts have been changed to Arial and OSIfont, respectively. In that way, it is easy to change between fonts.

We have also done a little with colors in the files ...

Follow this instruction to change text settings in the project:

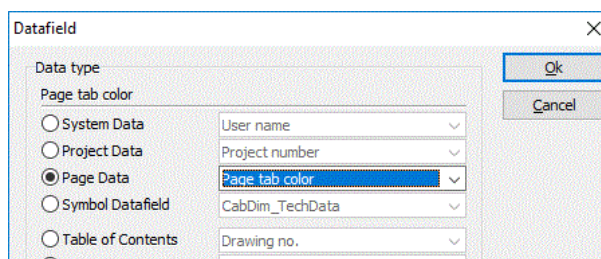
1. Open the file with the desired text settings
2. Press the Save icon to save the file's settings
3. Open the 'old' project, to which you want to transfer the settings
4. Press the Load icon
5. Press the Transfer icon
6. Check the 'Use project default ...' to use the new defaults in the project



7.6 New data field

There is a new data field: Page color. The data field is available in Page data and Table of Contents.

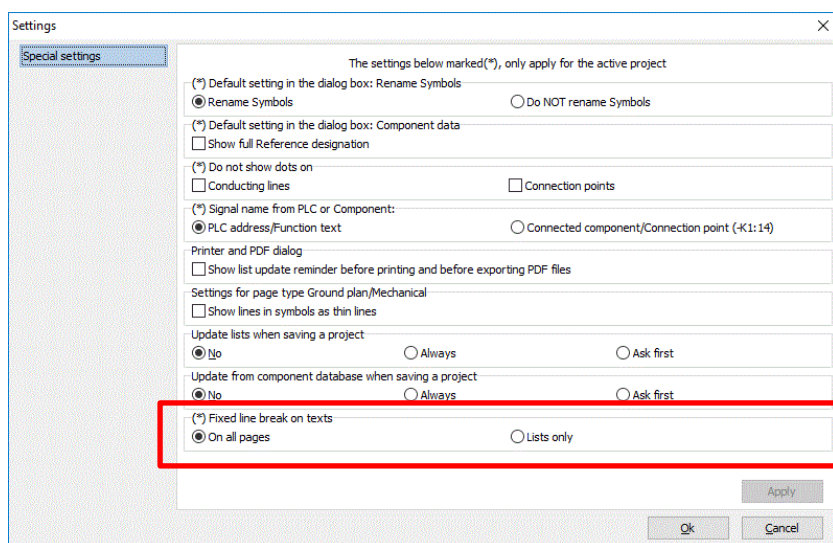
The data field cannot have any pre-text and it is always 3 characters long.



7.7 New special setting

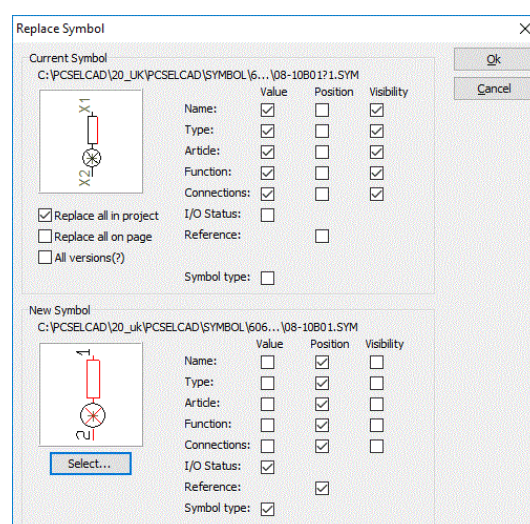
It is now possible to force a line break after an exact number of characters, which is selected as 'Text width'.

The setting can be selected for all pages or list pages only.



7.8 Replace symbol

It is now possible to replace ?-versions. ? is a generation mark, and you can replace 'all generations/versions' either in the full project or on the actual page.

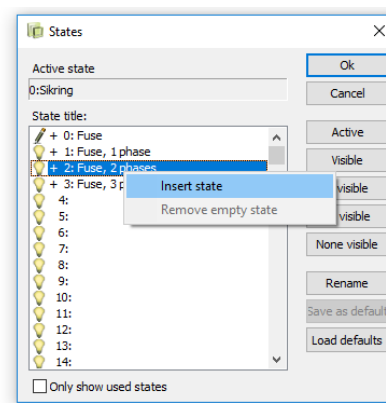
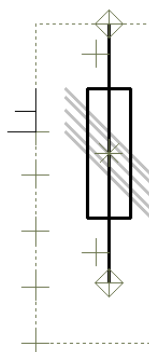


7.9 Symbols in states – insert new state

If you have symbols with many states, you might want to insert a new state, and logically, this state is between two existing states.

Now you can right-click and Insert a state.

It is also possible to remove an empty state.

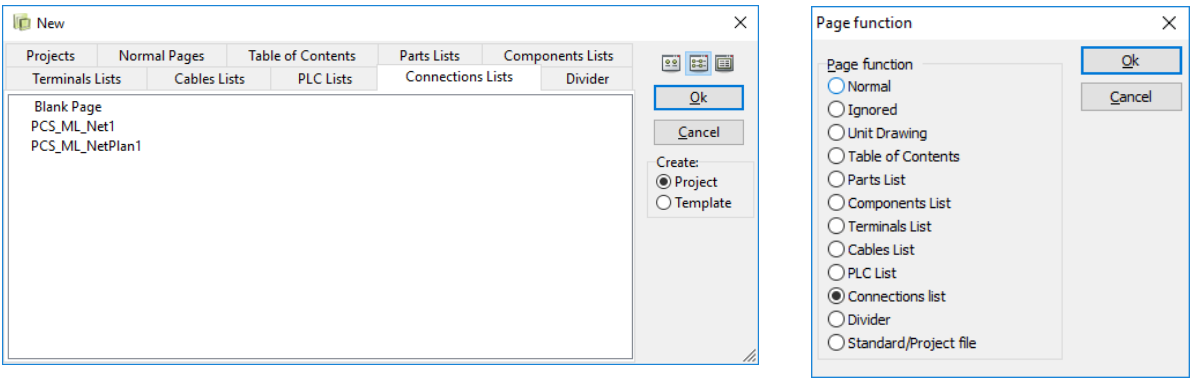


7.10 New page type – Connection list

The connection list can now be included in projects.

We have created a new page type – NET – that can be selected in the program.

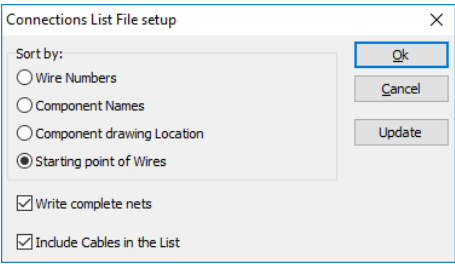
There will also be a new tab in the New folder, that contains the template(s) for this type of list. For those who don't have the tab, the templates are also found the Normal tab.



The list functions are the same as the 'old' connection list file, meaning that you can have a list with information about all connections in the diagrams, sorted in multiple ways.

FROM	TO	Wire no	Signal	Cable	Jumper	Line name	Line type	Routing class
+1.4/-UC1.T1:L1	+1.4/-UC1.Q1:1		L1:L1					Power
+1.4/-UC1.Q1:1	+1.4/-UC1.Q3:1		L1					Power
+1.4/-UC1.F1:98	+1.4/-UC1.F1:98		L1:L1					Power
+1.4/-UC1.F1:98	+1.4/-UC1.Q1:83		L1					Power
+1.4/-UC1.Q3:1	+1.4/-UC1.F1:98		L1:L1					Power
+1.4/-UC1.Q1:83	+1.4/-UC1.Q3:83		L1					Power
+1.4/-UC1.F1:1	+1.4/-UC1.Q1:2	08						xxx
+1.4/-UC1.F1:1	+1.4/-UC1.Q3:8	08						xxx
+1.4/-UC1.X1:U	+1.4/-UC1.F1:2	02						
+1.4/-UC1.Q2:1	+1.4/-UC1.X1:U	01						
+1.4/-UC1.T1:L2	+1.4/-UC1.Q1:3		L2:L2					Power
+1.4/-UC1.Q1:3	+1.4/-UC1.Q3:3		L2					Power
+1.1/-M1:U	+1.4/-UC1.Q2:2				-W2:BK			

In the enclosed list example, there are columns for all connections types: wire number, signals, cables, jumper links and information about possible line data, such as routing class (can be seen in the Panelrouter demo project).



8 My notes

