AUTOMATION

DESCRIPTION OF NEW FUNCTIONS

Version 14.0

January 2012

This material

This document describes the new functions in Automation version 14.

PC | SCHEMATIC

Contents

Plc-functions

Re-addressing of plc-projects can be a long and tedious task. When designing a machine the focus is on the functions and tasks that the machine is going to carry out and not on the hardware that you end up with,

That means that when the design is finished you have a lot of I/O's that "only" need the right address. And the only way to do this has been to open each I/O symbol to match it to a valid address on the reference symbol.

That is the reason for the new functions concerning handling of the re-addressing issues. Briefly, the new functions mean that you can re-address a complete project with just a few mouse-clicks, as addresses from the reference symbols can be transferred to the rest of the project.

It is also desirable, when talking (very) large plc-projects, that the reference symbols could be addressed sequentially. This is also possible, and the function will also differentiate between analog and digital addresses.

New PLC menu

All plc commands are grouped in a new menu.

The commands "Transfer PLC adresses", "Re-address PLC reference symbols" and "Define PLC I/O status types" are new and will be explained below.

The other commands are "old" plc-commands that have been moved to this new menu.

PLC	Window	Tools	Help
Transfer PLC adresses
Readdress PLC reference symbols
Define PLC I/O status types
List PLC I/O symbols
List PLC reference symbols
Read PLC I/O List...
PLC List File...
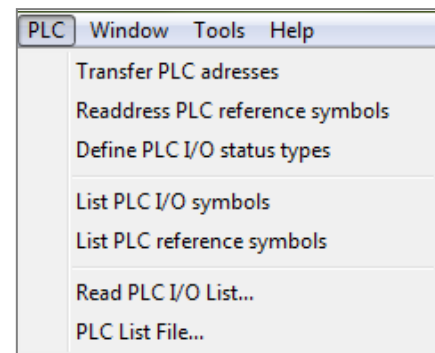
The Transfer PLC addresses dialog

Re-addressing of plc's are carried out from this dialog.

Input/output symbols or I/O-symbols are re-addressed in the same sequence as they are found in the project, this means from page 1 and forward, and from left to right.

You can make various selections for I/Os, which will ensure that not all I/Os are automatically addressed.

See below for a further description.

Section 1

In this section you have the following methods to make the address transfer:

- I/O status (e.g. digital or analog status on the I/Os)
- Address prefix (e.g I.xx, O.xxx, PIW.xxx or the like)
- Functional aspect
- Locational aspect

The program finds the data used for filtering in the project. For each data type you can select a filter or method, e.g. re-address only analog addresses in the locational aspect +B7. The new item I/O status is described below.
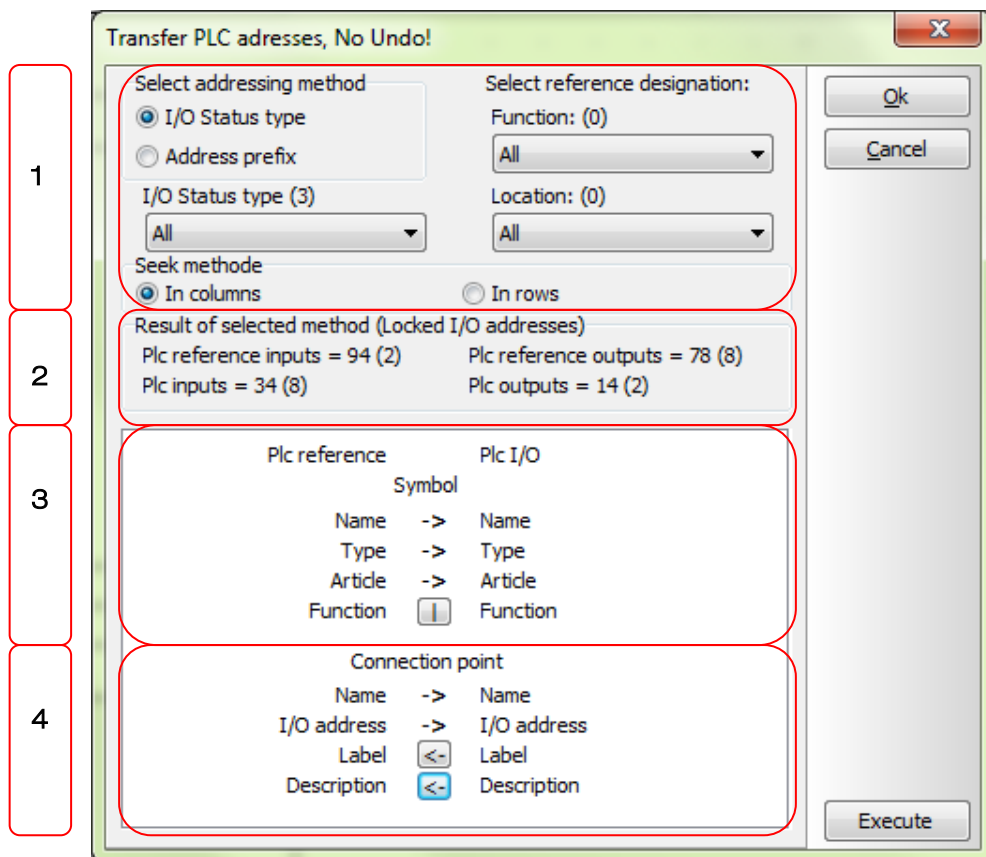
The address transfer can be vertical (columns) or horizontal (rows) in the project.

Section 2

After selecting a method you can see how many inputs and outputs that are going to be re-addressed in section 2. If you lack addresses on the reference symbols there will be a red line telling you how many. If some of the addresses are locked, you can see how many in brackets (). Below, you can see how to lock an address. Locked addresses are omitted from the address transfer. Invisible I/Os cannot be seen and therefore they are omitted, too.

Section 3

Section 3 shows the information that is transferred from the reference symbol, i.e. the plc name, type and article number. Transfer direction or ignore transfer can be selected for the function text by pressing the button.

Section 4

Section 4 is about the connection points. In order to "pair" the reference symbol and the I/Os connection names must be identical. Connection names and addresses are transferred from the reference symbols, but labels and descriptions can be transferred the other way or can be ignored.

Above you can see the defaultsettings for the dialog box, and here labels and descriptions will be transferred from the I/Os to the reference symbol.

The Readdress PLC reference symbols dialog

With a large plc project you have a large number of reference symbols as well. Here it is convenient if you can re-address the reference symbols automatically. Especially when one plc may consist of several reference symbols.
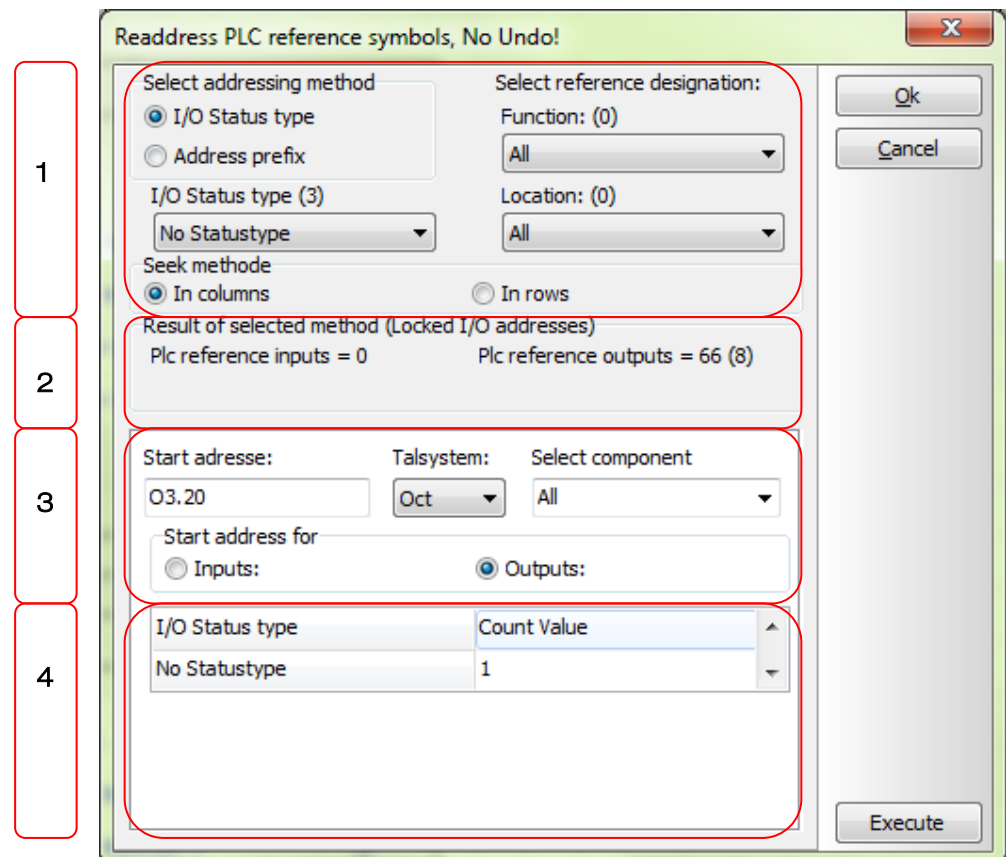
Section 1

Similar to the transfer dialog, this dialog allows filtering the addresses according to the existing pre-addressing of the reference cards:

- I/O status (e.g. digital or analog status of the I/Os)
- Address prefix (e.g. I.xx, O.xxx, PIW.xxx or the like)
- Functional aspect
- Locational aspect
  - The address transfer can be vertical (columns) or horizontal (rows) in the project.

Section 2

When you have chosen your filter, this section shows how many inputs or outputs are affected. If some of the addresses are locked, you can see the amount in brackets (). Below, you can see how to lock an address. Locked addresses are omitted from the address transfer. Invisible I/Os cannot be seen and therefore they are omitted, too.

Section 3

In section 3 you can select start address, number system and a certain component, as well as you can choose to re-address inputs or outputs.

Section 4

Section 4 is about connection points, i.e. I/O addresses. Here you can type a count value for each card type as selected above.

Inputs and outputs in the project

In order to be able to make automatic address transfer or re-addressing of reference symbols, you need to be able to control the inputs and outputs in the project. There are a variety of "rules" to fulfil; some of them are product related, others are more "we use to" specific. Therefore you need to prepare the inputs and outputs in the project.

Remember to make necessary preparations in subdrawings and templates as well.

Define PLC I/O status types

As there are many different types of PLC I/Os and as they are going to be treated in different ways, this dialog lets you define the status types that are relevant for you.

You can create new ones, edit or delete existing types, and finally save the list.
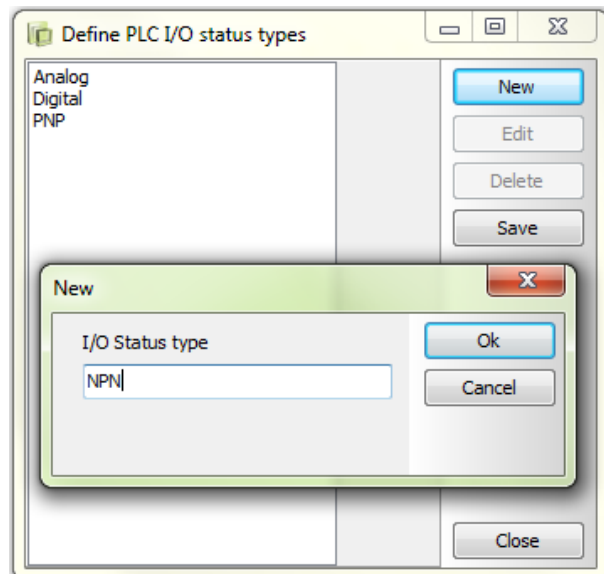
Define I/O status type for the individual input or output

When the I/O status types are created, you can use them on the individual inputs or outputs.

Selection of status type is available only when I/O status extension is PLC or PLC/TERM.
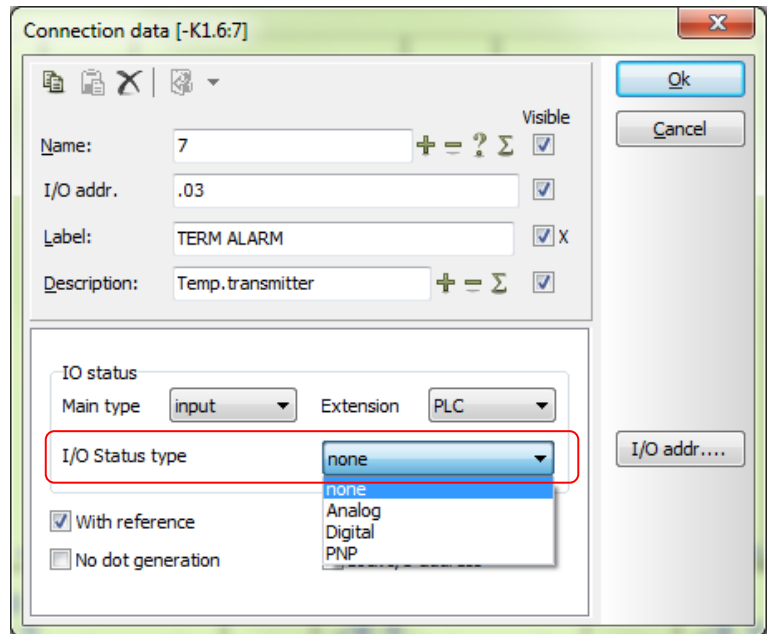
Exceptions from re-addressing

You might need to have fixed addresses for certain in- and outputs, e.g. for safety circuits, where you have fixed functions and locations, that is that the same connection point always has the same address and the same purpose.

Lock I/O address

Start by locking the address on the reference symbol:

Open the connection point, and check √ the Lock I/O address.

Do the same in the I/O symbol.

The two locked addresses are then paired: Find available addresses by pressing the button [I/O addr…], select the proper address, and all data related to this address is now transferred to the I/O symbol.
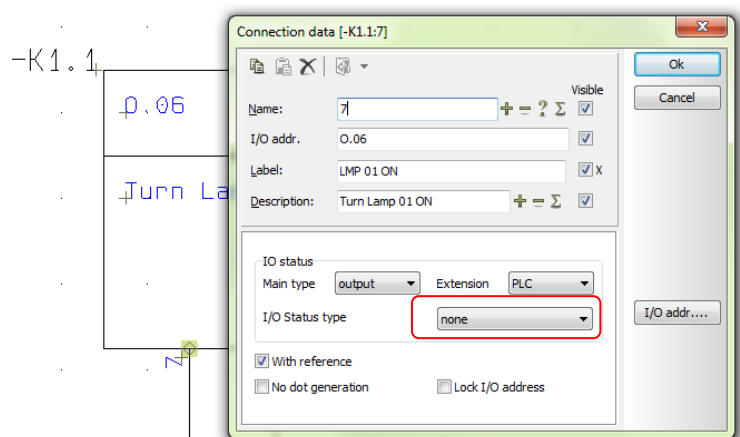
Invisible inputs and outputs

You can – as always – place invisible symbols in the projects. Also in plc-projects.

You would do this in projects where you want to place optional inputs/outputs or on "gross" pages used as templates.

Invisible symbols can be seen on the screen but are not printed, are not included in any lists and they will be omitted from the re-addressing functions as described here.
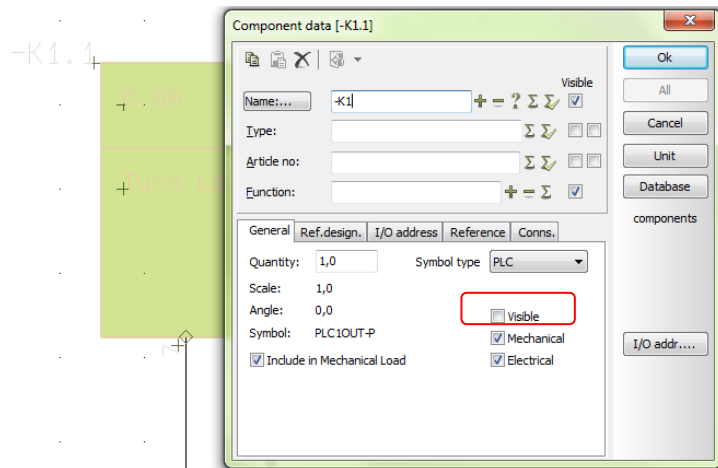
List PLC I/O symbols and List PLC reference symbols

Selecting one of these menu items will open the Objectlister and show either I/O symbols placed in the project or the I/O addresses defined in the reference symbols in the project.

At the bottom of each list you can see the number of inputs and outputs respectively.

Read PLC I/O list and PLC List file…

The two menu itmes for reading and listing I/O lists have been moved to this menu, too, as they are PLC-related commands.

A new shortcut

When in the dialog Component data Ctrl+I opens the [I/O addr…].

Design check

Design check of a project means that the program can detect some of the more obvious errors in the design.

If, for instance, you have chosen to make a mounting correct drawing, you can let the program check that all connections are indeed made with bends and not with dots; if you need a very detailed connection list you can check for consistent line data, or you could – very simply – check the project for short circuits, not connected terminals or.....

The function has its own icon, which is located in the upper tool bar:

Pressing this icon, the "result" will pop up in a new window

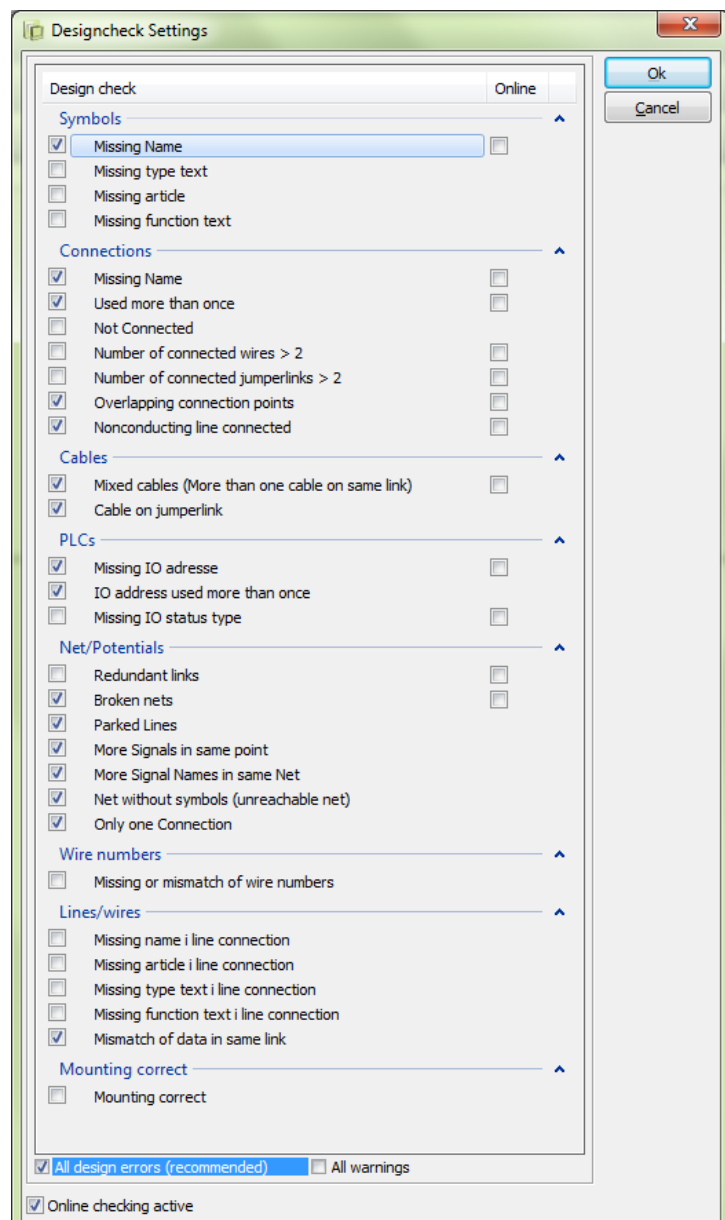As before, pressing this icon enters the Designcheck Settings box

Designcheck Settings

The box only shows groups when first opened. Unfold the groups to see the settings.

From ver 14, you will see different types of designchecks and furthermore, some of them can be online checks.

You can select or deselect all errors and warnings with one click, so that you only need to open the group with the desired check o select it. Online check can be opened at the bottom of the box.

We have distributed a PCSDEMO-designcheck project with version 14, which shows examples of the available checks.

Designcheck Settings		
Design check	Online	
Symbols		^
☑ Missing Name	☐	
☐ Missing type text		
☐ Missing article		
☐ Missing function text		
Connections		^
☑ Missing Name	☐	
☑ Used more than once	☐	
☐ Not Connected		
☐ Number of connected wires > 2	☐	
☐ Number of connected jumperlinks > 2	☐	
☑ Overlapping connection points	☐	
☑ Nonconducting line connected	☐	
Cables		^
☑ Mixed cables (More than one cable on same link)	☐	
☑ Cable on jumperlink		
PLCs		^
☑ Missing IO adresse	☐	
☑ IO address used more than once		
☐ Missing IO status type	☐	
Net/Potentials		^
☐ Redundant links	☐	
☑ Broken nets	☐	
☑ Parked Lines		
☑ More Signals in same point		
☑ More Signal Names in same Net		
☑ Net without symbols (unreachable net)		
☑ Only one Connection		
Wire numbers		^
☐ Missing or mismatch of wire numbers		
Lines/wires		^
☐ Missing name i line connection		
☐ Missing article i line connection		
☐ Missing type text i line connection		
☐ Missing function text i line connection		
☑ Mismatch of data in same link		
Mounting correct		^
☐ Mounting correct		

☑ All design errors (recommended) ☐ All warnings

☑ Online checking active

Ok
Cancel

Online check

When you have enabled Online check you can see small yellow warnings on the drawing.
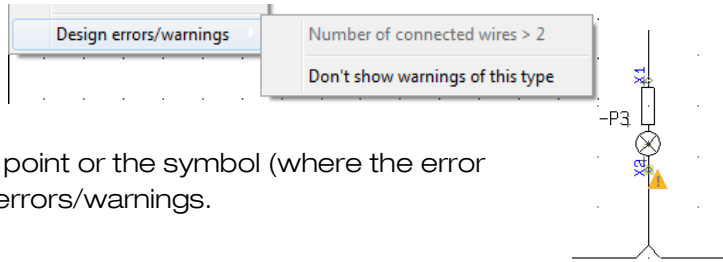
If you right-click on the line, the connection point or the symbol (where the error occurs) you get a new menu item; Design errors/warnings.

The individual checks

| Group	Checks	Meaning…
Symbols	Missing Name, Type, Article number, Function	Symbols without name/type/article number/function in component data
Connections	Missing name	Connections without names do not make sense in electrical list, such as connection lists, terminal lists, cabel lists and graphical plans. Beware, that all PCS symbols have names, even when they are invisible.
	Used more than once	It is an error if a component's terminal is connected to two or more different nets/potentials.
	Not connected	Non-connected terminals are allowed by the program, also when generating lists. However, you can check for non-connected terminals. Not connected connections are also found, when a jumper/line passes terminals WITHOUT actually connecting. Graphically, they LOOK connected.
	Number of connected wires >2	More than 2 wires are connected to the connection point. The number can be changed.
	Number of connected jumperlinks >2	More than 2 jumpers are connected to the connection point. The number can be changed.
	Overlapping connection points	When two connection points are on top of each other
	Non-conducting line connected	You get a warning when you try to connect a con-conducting line to a terminal, but you can ignore the warning. This option can search for non-conducting connections in the project.
Cables	Mixed cables	Check for mixed cables meaning more than one cable on same link. Earlier the check was only available in cable lists.
	Cable on jumperlink	Jumperlinks cannot be part of a cable
PLCs	Missing IO address	Missing IO address on a plc-symbol is an error
	IO address used more than once	Check for the same address more than once in the project
	Missing IO status type	If you decide to use this function you can check for plc symbols without status type.
Net/ Potentials	Redundant lines	Does the connection exist when a conductor is removed?
	Broken nets	When mounting correct is in-correct, you can end up with nets without connection. With this function you can check (old) projects.
	Parked lines	Lines without connection to signals or other symbols or lines.
	More signals in same point	More signal symbols in the same point
	More signals in same net	Short circuit – more signal names in the same net.
	Net without symbols (unreachable net)	Boxes and the like made with conducting lines. It is considered a net by the system, but will be unreachable
	Only one connection	A net with only one connection cannot have electrical functions
Wire numbers	Missing or mismatch	Links without wirenumber or different wirenumbers on the same link.
Lines/ wires	Missing Name, Type, Article number, Function	Lines without name/type/article number/function in component data
	Mismatch of data in same link	When you use detailed connection lists, that contain line data, the program can check for different data (mismatch) on two or more segments in the same link.
Mounting correct	Mounting correct	Are all connections made with bends and not dots

Show available functions

Due to the feedback we received to version 13, the Show available functions have been further improved in this version.

Most of the improvements are about showing, sorting and filtering the lists.

Docking the Show Available Window

The Show Available window can be docked at the right side of the screen, and the available symbols can be docked inside the Show Available list. The line between the two windows can be adjusted.
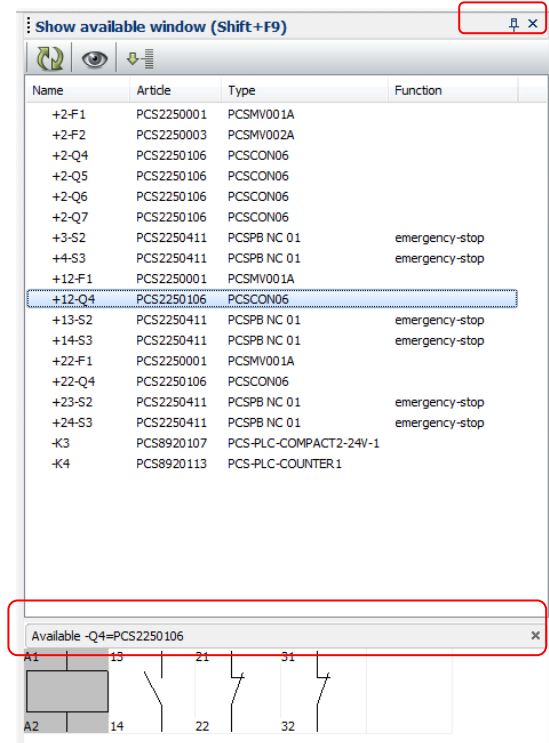
If you don't want to have the window present constantly, you can make the window "slide" on command: Change the setting of the pin by clicking it. Now you can slide the window open by clicking the vertical buttom, or by pressing F9 on a component with available symbols. The window will slide back again shortly after clicking the diagram page.
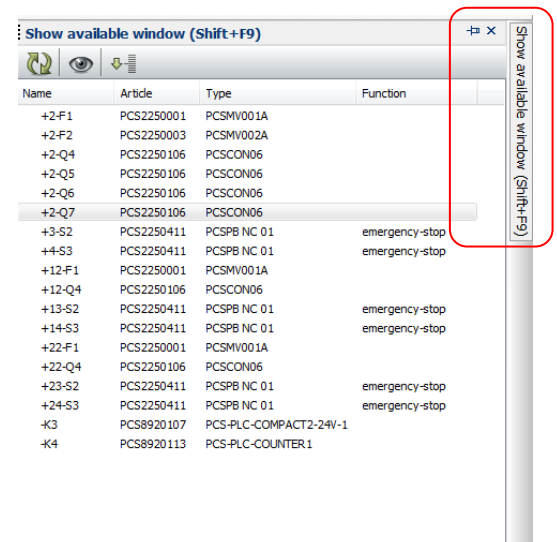
Finally, you can un-dock the window by dragging the window's title line. In that way you can move the window on your screen or move it to a second screen.

Filtering options

You can now sort and filter in the Show available window:

You can (still) sort A-Z or Z-A in all columns by clicking the titel row.

If you rigth-click in the name column you can see the A-Z filters also in all reference aspects, as well as you can see a text filter which contains various criteria.

When you activate the filter, the components selected by the filter are at the top in the Show available window.

The other components can be found under "Filtered out".
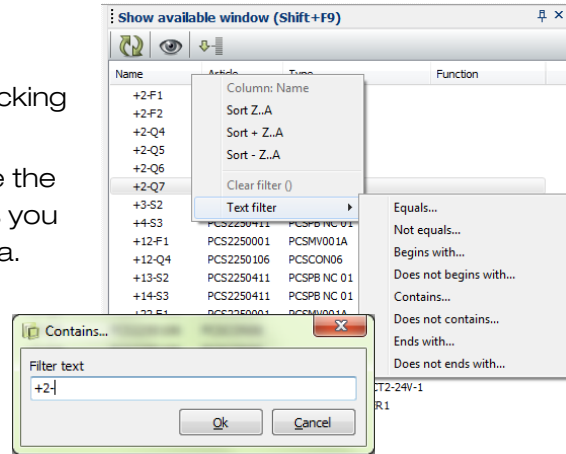
The Hide option can be found in the right-click menu or as a button in the toolbar. Components can be selected with Ctrl + or Shift +.

The hidden components can be seen by clicking the arrow. You can place symbols directly from this part and you can move one or more of them back to the ordinary list.
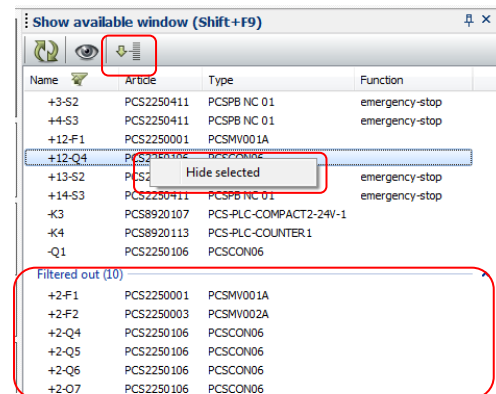
To help select the right symbol thea vailable symbol window has been extended with tabs for inputs, outputs and reference symbols.
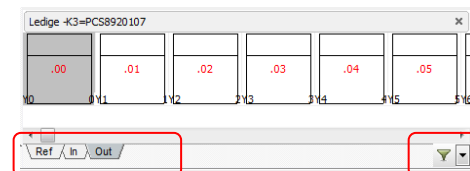
For plc's you also have an address filter that can be activated by the "funnel". The filtering values can be selected by the "arrow down" button. You can find the address filters on inputs and outputs.
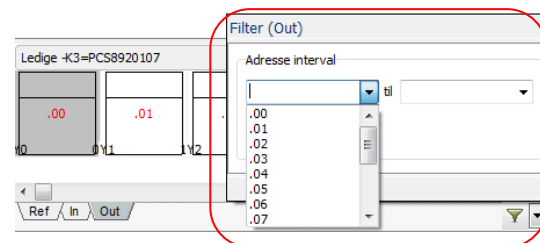
PanelBuilder

PanelBuilder is a new module for making documentation of small panels according to the requirements stated in EN60439-3 concerning power dissipation calculations.

The module has the following functions:

- Easy user interface
- Drag'n drop placement of components in a tree structure
- Select components based on function, manufacture, amps and phases
- Function labels on outputs
- Online phase balancing
- Online overview of phase distribution
- Online overview of load per phase
- Power dissipation calculation with selectable ambient temperature
- Automatic placement of components in the panel
- Multiple panels in the same project
- Single-line drawings of the complete project

The program window

When you open the module you will see a window divided into two: the first half shows the distribution tree for the project, the other half contains a tab for each work process in the project. Components placed in a panel are dark in the tree, while components not yet placed are light.
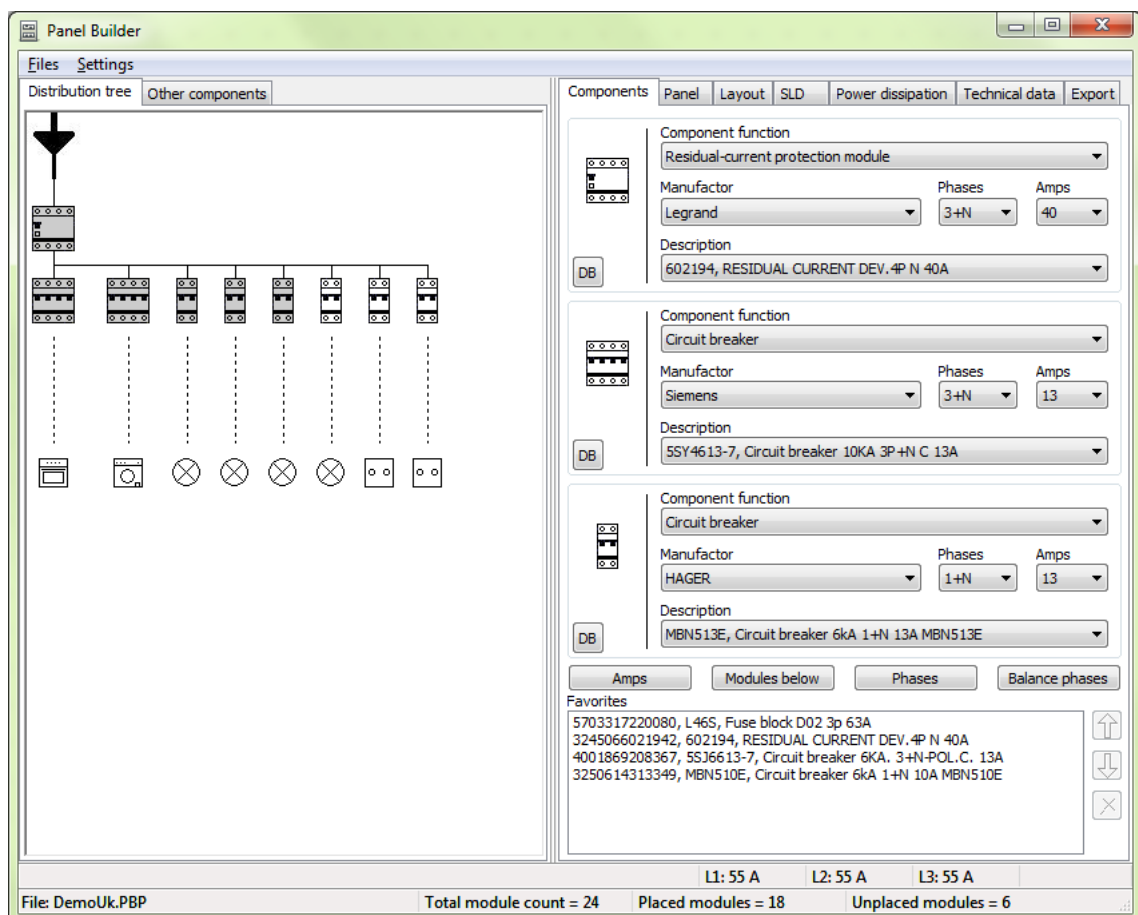
Selecting components

You select components based on the data that can be selected in the window: component function, manufacture, number of phases and size in Amps. The selections are filtered according to previous choises thus narrowing the selection.

You have three component boxes making it possible to have your favorites present, and their values are saved when you close the module.

If you choose a component that needs accessories – as a fuse block – the possible accessories are listed in the database and they can be selected in the dialog that pops up when the component is placed in the project.

You can save more favorite components in the box "Favorites" from which you can drag them directly into the project.

Drag'n Drop placement of components in the tree structure

All components have an icon that is shown in the project. This icon can be dragged into the tree structure by using the mouse.

Components can be placed in the tree, when the small "square" pops up. The first component must be placed on top of the ARROW that indicates the tree structure's supply.

A tree structure will appear as you place the components as the component are placed "below" the component they are dragged onto.

Components can be moved by dragging them with the mouse, any components connected below the selected one are moved as well.

A component can be changed by holding down the ctrl-button when you drop and place a new component (instead of just dropping and placing it).

Apart from the components placed in the tree structure you can select other components, i.e. push buttons, surge protection and signal lamps that are not part of the tree strcture itself, but placed in a panel.

You can place function labels with a text below the buttom component. It is possible to make your own labels (*bmp eller *jpg).

The loads are continuosly balanced across the phases; however, it is possible to force a component to a phase. The phase setup naturally also concerns the underlying components.

If you place the mouse on top of one of the placed components you will get a hint pop-up with component data.
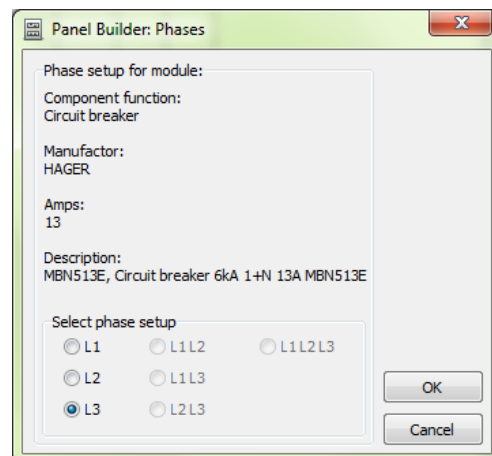
Online status overviews

At the buttom of the window you can see online overviews of how many modules there are in the project and how many modules are placed in the panel(s).

You have an online overview of load per phase.

If you want to see values for each component you can press and hold down one of the buttons [Amps], [Modules below] or [Phases].

You can make an automatic balancing of phases by pressing the button [Balance phases].

Automatic placing of components

When you have made the tree structure for the project the module can make a panel drawing.

Start by choosing a panel – the necessary number of modules can be seen in the status line at the buttom.

The selected panel is shown in the tab. If you want multiple panels, simply choose the next panel which is then automatically added to the project.

Simply delete the panel if you made a wrong choice.

Placing components in a panel

Components can be placed automatically from one of the panel's corners. The panel positions are based on the positions in the tree structure.

If you want to control the positions in order to make space for other components, you can manually place empty modules on the rails, and you can drag components from the tree structure into the panel and place it on the rails. The rest of the components can then be placed automatically. If some components have wrong positions in the panel simply right-click to delete them one by one, or click the button to delete all components in the panel.

Components that have been placed in the panel are marked as grey in the tree structure.

In the Panel tab you can see how the panels are placed on the panel drawing that you export to Automation. You can drag the panels to another position on the page. What you see here is equivalent to Print Preview.

Power dissipation

The module calculates the power dissipation when all components have been placed in the panel(s). You will get a message if there are problems with making the calculation.

To make the calculation you start by selecting an ambient temperature and then choose between your selected operating current or let the module make a calculated one.

The result is shown as collected data for the project – calculated internal temperature, operationg current, simultaneity factor and finally a list of all components and their calculation data. You can select to have extended information in the list by ticking the box at the buttom of the tab.

```
POWER DISSIPATION
  Operating current for panel(s): Calculated    -    Panel(s) calculated at: 44,99 'C
  Calculated internal temperature: (20 + 24,51 ) = 44,51 'C    -    Ambient temperature: 20 'C
  Operating current: 32,5 A    -    Simultaneity factor (SF): 0,95
  Power dissipation inside panels:
    For components: 23,18 W    -    15% dissipation for internal wires: 3,48 W
    Total power dissipation: 26,66 W
```

Comp.	Type	Description	In	Pn	Int	L1	L2	L3	L0	Imax	Pv	Pv x SF2	Pconst	Imax x SF
	Input		34,1			34,1	34,1	34,1	0					32,5
-F1	602194	RESIDUAL CURRENT DEV.4P N 40A	40	12	32,5	34,1	34,1	34,1	0	34,1	8,72	7,92	0	32,5
-F2	G103NC16	Circuit breaker C 16A 3P+N 10KA	16	7,71	13	13	13	13	0	13	5,09	4,62	0	12,39
-F3	5SY4613-7	Circuit breaker 10KA 3P+N C 13A	13	5,6	10,56	10,6	10,6	10,6	0	10,6	3,72	3,38	0	10,1
-F4	MBN513E	Circuit breaker 6kA 1+N 13A MBN513E	13	4	10,56	10,6	0	0	10,6	10,6	2,66	2,42	0	10,1
-F5	MBN513E	Circuit breaker 6kA 1+N 13A MBN513E	13	4	10,56	0	10,6	0	10,6	10,6	2,66	2,42	0	10,1
-F6	MBN513E	Circuit breaker 6kA 1+N 13A MBN513E	13	4	10,56	0	0	10,6	10,6	10,6	2,66	2,42	0	10,1

Single line diagrams of the project

You can generate a single-line diagram of the project. In the tab you can select different text settings, among them whether you want to print label texts and/or component descriptions from the database.

Export the file

Finally, the project file can be exported to Automation, where you can choose to elaborate more on the project or simply just print it.

You can export the panel drawing with or without the power dissipation and/or the single-line diagrams.

The export is made with a standard template containing pages for panel drawing, single-line diagram, parts list, component list, power dissipation calculation, technical data, CE marking and EC Declaration of conformity.

In the full version it is possible to change the template that is used for export to Automation.

Save the project

You can save the project for later use in Files|Save. The files have their own format: *.PB2.

Database

The module has its own database, which is an extended edition of the Automation database in the sense that it contains different extra data fields for necessary data for calculating the power dissipation.

Bits and pieces...

As usual, there are various smaller improvements that are not part of a larger theme.

Some of – most of – these smaller news are picked from the mail sent to proposals@pcschematic.com. Note, though, that we have the right to choose between the proposals....

The Component data dialog

The dialog contains a button that opens linked files, e.g. data sheets.

The dialog also shows a picture if it has been specified in the database.

The following formats are supported: bmp, jpg, png, gif – which are the same formats that can be used for logo and inserted pictures.
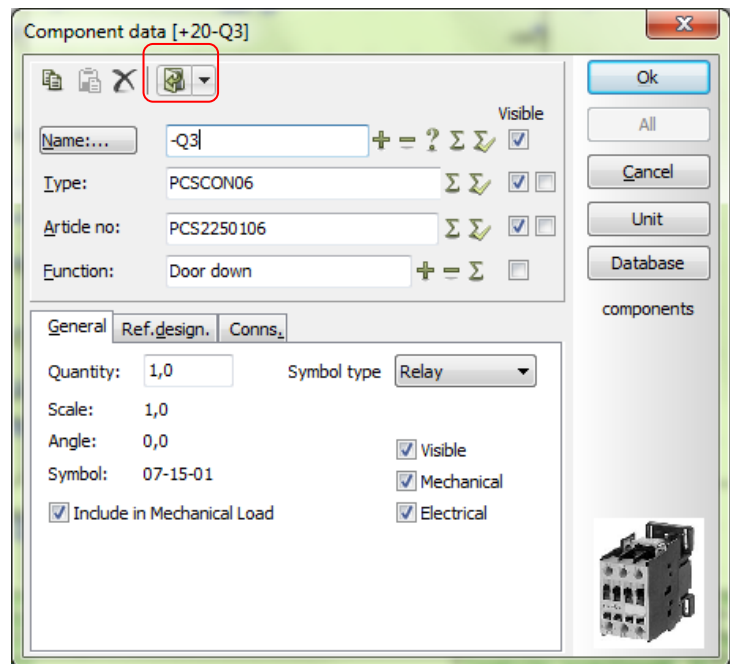
Insert picture and Insert object

You can now insert *png and *gif files via the menu Insert|Insert picture.

Pickmenus are active in Symbol editor

You can now have pickmenus in the Symbol editor. You use them the same way as in the program itself, that is, you can give them names; you might have more groups of symbols each with a set of auxilliary symbols. The pick menus can contain symbols, lines and text formats, just as the normal pickmenus.
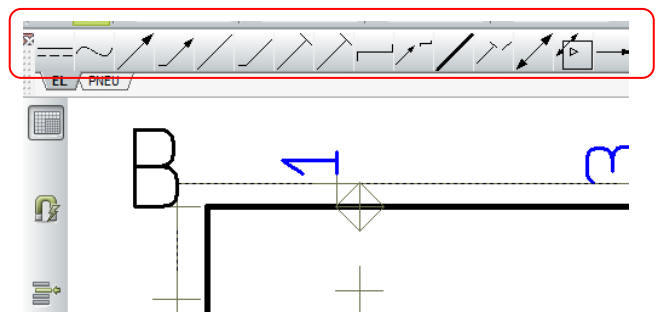
Various "line items"

Lines without wire number

When you mark a line you can select or de-select wirenumbering for the actual line. You can find the command in the right-click menu, in the Edit menu and in the Line toolbar.
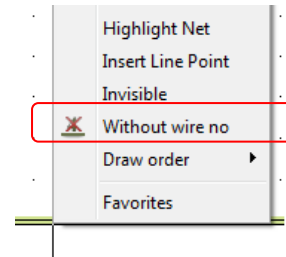
Mouning correct drawing ON/OFF

Mounting correct drawing is not (anymore) a project setting, as the program allows mixed drawings, actually from version 12.

This button is now in the lines toolbar and is used to de-/activate mounting correct drawing. Setup of bend size and type is still found in Settings|Project data.

The designs check function can now check the project to see if all connections are made mounting correct, that is with bends and not dots.

Rotation of line types

Choose between line types:

Mark the line, press <space> and the next line type is selected. Press <shift + space> to select the former line type.

You can only change between "valid" line types, i.e. the types intented for conducting or non-conducting lines, respectively.

Settings|Text/symbol standard

It is possible to make a standard setting for line texts.
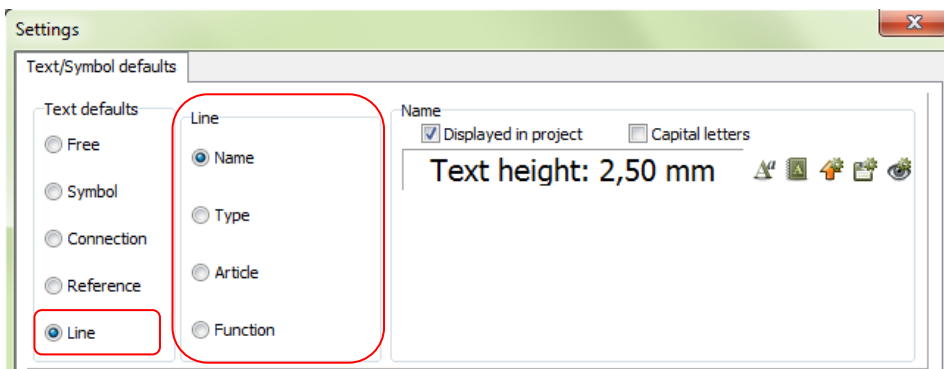
In the dialog you choose to display a given text in the project or not, and the text font and size.

The texts that have been selected here are shown with a √ in the dialog below.

When the settings have been changed they can be saved as standard settings, similar to symbols' standard text settings.

Display of line texts

Line texts can be displayed in two ways in the diagram. In the dialog you select to display them either aligned or above each other.

Which texts to display and the text formats are selected as shown above.

Texts can be selected or de-selected for each line.
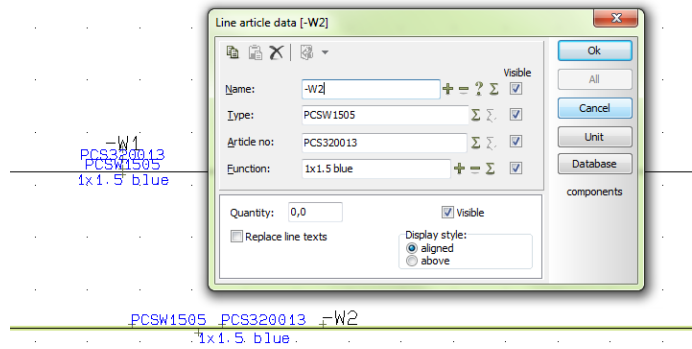
How to control LineData display and position with PCSCAD.ini

In the PCSCAD.ini file you can add the section [LineData] if you want to change/control the sequence for the four line texts and their default positions.

If you open your PCSCAD.ini file you will not see this section, as the default values are chosen, but the lines can be added if desired and if you wish to change the values.

PSCCAD.ini	How does the setting work when I choose	
	Above	**Aligned**
[LineData] *NamePos=3 (default)* *TypePos=1 (default)* *ArticlePos=2 (default)* *FunctionPos=-1 (default)*	-1 is below the line, 1 is the first above the line etc. With vertical lines -1 is at the right side of the line.	The number is the text's position in a reading direction from the left. Negative number is below the liene.
PosFactor=500 (default)	This is where the texts are displayet, if the line data is valid for more than one line segment, it will be on the longest line segment. 500 equals 0.5, which is the middle of the line, 250 equals 0.25, that is ¼ from the left. With vertical lines it is ¼ from the lowest point.	
LineTextDisplayStyle= 1 (default)	0 (null) means aligned and 1 (one) means above each other. This is the only setting saved by the program.	
LineTextDistance=3500 (default)	Default distance between the texts when they are aligned (LineTextDisplayStyle=0).	

Join signals

Joining of signals to a signal bus can now be made with more connections in one operation:

The bus is drawn as a non-conducting line.

1. Mark the bus.
2. Go to Edit|Join signal.
3. Mark all connection points at once by dragging the mouse over them.
4. The dialog pops up.
5. The connection function i.e. the IO address for plc-symbols is fetched automatically.
6. Click the IO address if you want to use it as signal name.
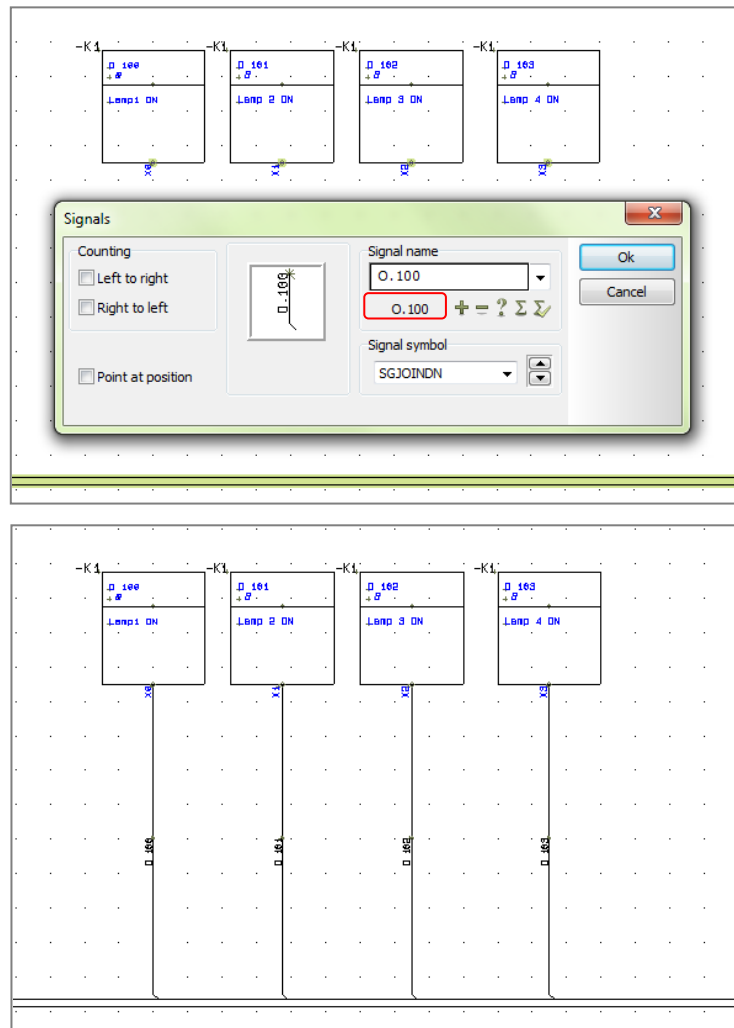
Jumper links in parts and component lists

Lines can be included in parts and component lists by ticking the "Include lines with article data".

If you only want jumper links, you need to tick the "Only jumper links" box.
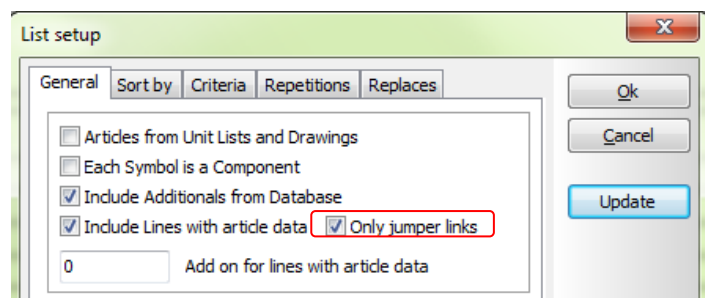
Connection list to file

In the list design dialog you can now find database fields.

These data fields can be selected for the list and you can choose to have the value from the TO or FROM component or from the connecting line.
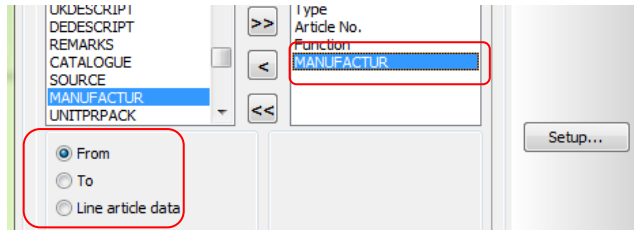
Extra option for time and date fields

The following data fields have been extended with the Year option:

- Project data: File date/time and Creation date/time
- Page data: Last revision, Date for last change, Creation date/time and Last printed
- Table of contents: Date for last change and Creation date/time
- Parts and component lists: Symbol date

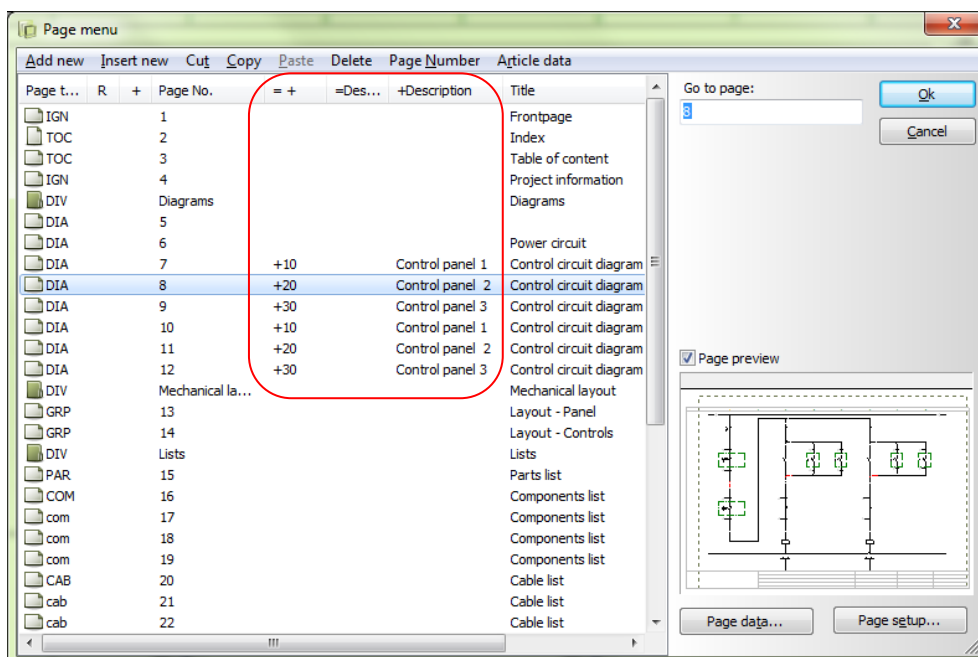The date format is picked from the pc's Windows setup.
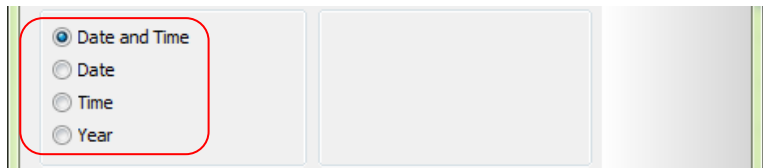
Table of contents

The table of contents can now list its own pages.

Extended Page menu

The page menu shows pages' reference designations and descriptions:

Optimized save routine

Inserted pictures and OLE-objects in projects are always compressed during the save routine, meaning that PRO-files will be smaller, also when you save the project file without compression.

Two screens

The program now works for users working sometimes with one screen and sometimes with two screens. During start-up the program will check for both screens, if they are not found, the program selects screen setup for the connected screen. This means that dialogs will not suddenly "disappear" anymore, because of a missing screen.

Terminals with different connector names

Up to now it has been necessary to have a dedicated terminal symbol if you wanted to have different connector names on a terminal symbol. This option is now available directly on the terminal symbol:

On terminal symbols you can select symbol type 2= None. Then the symbol is as the "normal" terminal symbol, meaning the same connection name on both sides. If you select Symbol type2=Normal you will get the option to have different connection names on the two sides.

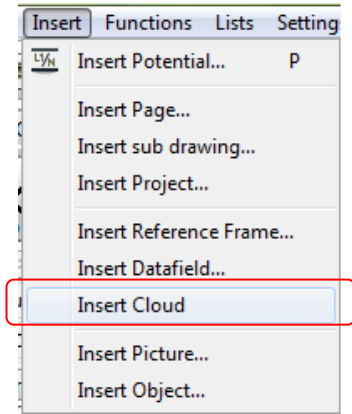You can name the connections 21 and 22 as in "position 2, internal and position 2, external".

At the moment auto-numbering with Ctrl+ and Multiplacement don't work with this option.
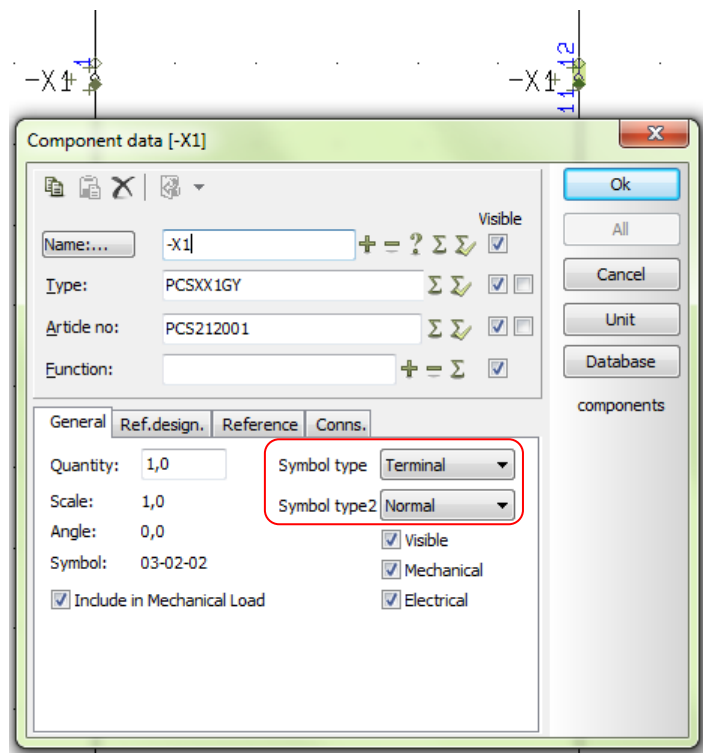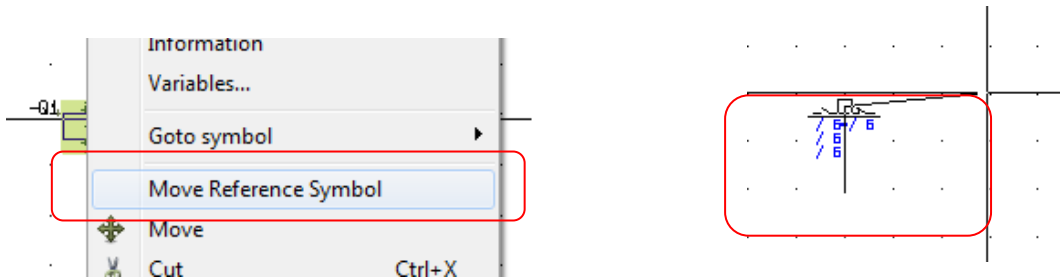
Insert Cloud

The function will now continue until you press <Esc>.

Move reference symbols

You can now move your reference symbol freely.

Mark the coil in the relay, and when right-clicking you will find the menu item "Move Reference symbol". Now you will get a "rubber band" on the reference symbol. Use it to move it to the desired position.

Program in demo state

If the program is in demo state and you select Save, you will get the following dialog.
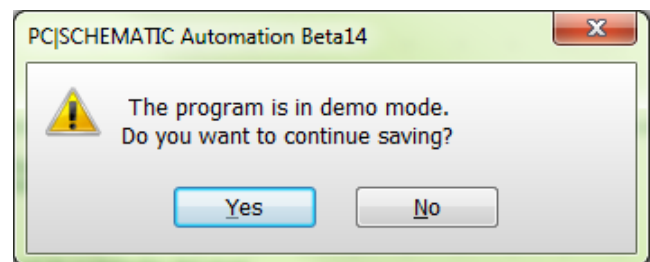
Windows2000

As stated at the release of version 13, Windows 2000 is no longer supported by Microsoft. Automation version 14 can NOT be installed on this operating system.

Open file and Open page functions

In all Automation versions it has been possible to open another file by giving a (dummy) symbol a filename as article number.

Open PCSDEMO_overview to try the function "live":

1. Mark one of the filenames (with text command).
2. Right-click and select Open.
3. The file will open – here PCSDEMO1.pro.

The command opens a file in the same folder as your current file. If the file is in another folder you must include the pathname as well.
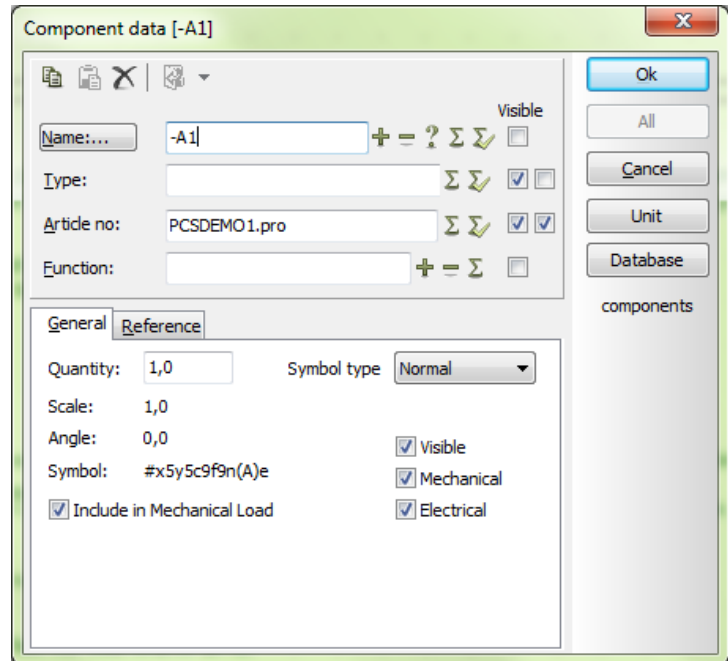
The function has now been extended to work with reference designations:

If you place reference frames in a project, the Open command will work this way:

If you have pages with the selected reference designation, the cursor will move to the first page with this designation.

Otherwise, the program will search for a project with the same filename as the reference designation, and will open this file.

My notes

My notes