AUTOMATION

DESCRIPTION OF NEWS

Version 12.0

Developed byPC|SCHEMATIC A/S, Bygaden 7, DK-4040 Jyllinge
No copying without written consent by PC|SCHEMATIC A/S.

PC|SCHEMATIC

This document

This document describes the news in PCSCHEMATIC Automation ver. 12.

The document is revised in March 2010.

Contents

This document 2
The program supports Unicode 5
System settings for conversion into Unicode 5
Open an existing project in version 12 6
Loading ”old” symbols 6
Important 6
New functions for installation projects 7
Panel Builder 7
Step-by-step: How to test the Panel Builder 7
Step-by-step: How to do to use the Panel Builder in the future 8
Your own components for the Panel Builder 9
New drawing headers 9
”Magnetic” mechanical symbols 10
Snap and grid follow page scale 10
Drawing connections on installation drawings 10
Cables and wires can be in the pick menu 10
Elevation in the pick menu 10
Extended Conducting lines button 10
Revisions and page remarks 11
Print Page remarks 11
”Old” methods for revision or version control 11
Export to pdf-format 12
Pdf command file conversion 12
Terminal functions 13
Grouping of terminals 13
Forced dots 14
Integration with CLIP PROJECT 15
Integration with M-PRINT PRO 16
Terminals, cables and plc-references with/without current path names 16
Automatic generation of single line diagrams 17
New fields in the database dialog 18
Settings in connection with the PanelBuilder 18
Settings in connection with the generation of single line diagrams 18
Sub drawings 19
Thumbnail 19
Settings in connection with sub drawings and thumbnail 19
Tools 20
Error log 20
Symbol manipulator 20
OLE automation 20
Conversion of old projects to Unicode 20
Changes and extensions for list functions 21
Import of project data with XML-format 21
Importing a parts list in XML format 21
Create components list from diagram 21
Extended functions for pick lists 21
Data fields 22
The data field Article no. 22

Quantity and cables 22
DB fields in terminal, cable and plc lists 22
The text in front of data field contents 22
Connection lists 22
Various minor items… 23
The program has new icons 23
List format 23
System settings 23
Find/Replace 23
The Find function 23
Design check 23
Text translator 24
Insert function in the sub drawings window 24
Changes in page function 24
Text formatting 24
The program can be opened with parameter call 24
Bridges in the pick menu 25
Measuring 25
Fine snap and move with arrows 25
Moved shortcuts 25
Icons 25
Copy (identical) 25
Changing cable quantity 26
Highlighted connection points 26
Insert special characters 26
Dos projects 26
New folders 27
Pickmenu 27
Change state 27
and don't forget… 27
New material in connection with the new version 28

The program supports Unicode

Now the program can handle multiple regional character sets simultaneously. This has been a long time wish for many customers, and it is the most important reason that this version has had a long development period.

A short explanation: Up until now language settings of computers and programs have been more or less permanent. In Denmark, Germany, UK and more, our computers used a character set – WESTERN EUROPE/LATIN 1. This character set contains the letters and characters in our alphabets, including Æ, Ø, Å, Ä, Ö and Ü. Other language areas have their own character sets, as for instance China, Japan and Russia.

Earlier it was only possible to work with one regional character set at a time. You could "cheat" the program to work with a different character set (like Russian) by changing some program settings – in Automation in Settings|System. In that way, it was possible – on a Danish Windows – to work with a project, which was originally made with another character set.

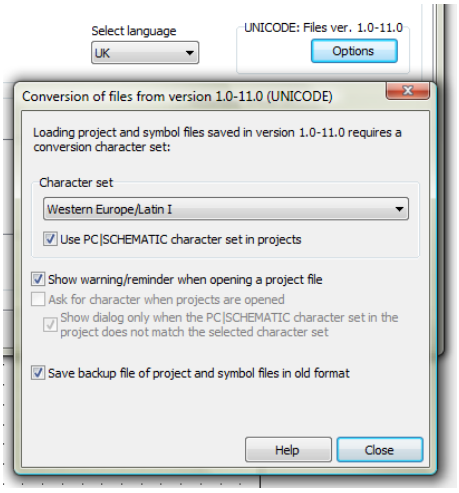
Unicode, on the contrary, contains (almost) all characters, more than 100.000, which each has its own meaning. UNI CODE = one code.

When you have a program that supports Unicode, this program can show letters/characters from all alphabets in the same file. With our program, that means that when you have a project file, that is going to be used in Russia, the file can contain a Danish/English text and Russian characters without needing to translate and thus hide/delete one of the texts. Simply, as each character has its own unique code. However, the font that is used must contain the characters that are needed, and this is not the case with all fonts.

The PC|SCHEMATIC font contains characters for Eastern and Western Europe and Russia.

System settings for conversion into Unicode

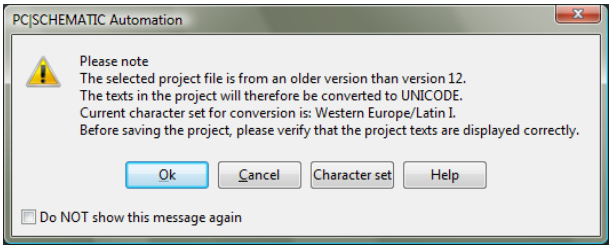
Basic setting for conversion into Unicode-format is made under Settings|System.



What happens in the program is, that all old files must be translated to the new format – that is converted from one character set to another. The program will guess the character set, see example on next page.

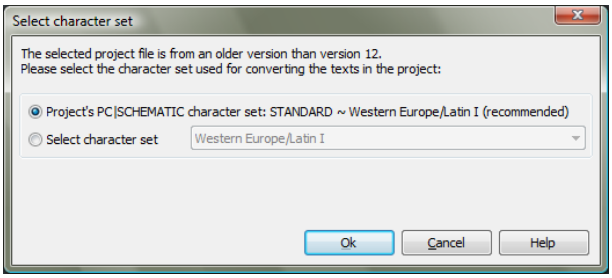
The PC|SHCEMATIC font is not a Windows font and therefore it has limited Windows functionality. However, it conforms to the standards' requirements concerning texts in technical documentation, and the font will be the standard font in most projects. It will also be converted to Unicode-format in all existing projects that you continue to work with. New projects are saved in this format automatically.

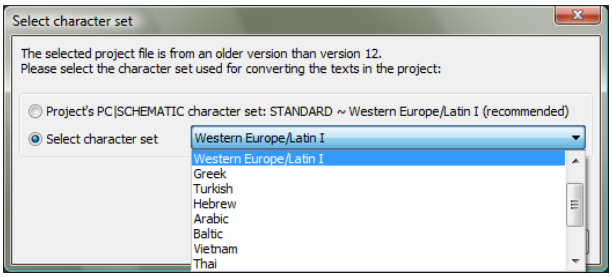
Open an existing project in version 12



When you open an old project – and in this conjunction all projects made up until now are old – the following dialog will appear. You can choose not to see the dialog again, see the section on standard settings.

If you press the button "Character set" you will enter the next dialog, where you can see the recommended character set for the actual project. If this is not the correct character set, you can press "Select character set" where you can see all supported character sets.





The following character sets are supported:

- Central Europe
- Cyrillic
- Western Europe/Latin 1
- Greek
- Turkish
- Hebrew
- Arabic
- Baltic
- Vietnam
- Thai
- Japanese Shift-JIS
- Simplified Chinese GBK
- Korean
- Traditional Chinese Big5

Loading "old" symbols

Conversion of old symbols function as according to the same principles as described above, but you will not the dialog every time you load a new symbol. However, it is important that your settings are correct, and we recommend that you keep the setting about automatic backups in old version.

Important

It is important, that you choose the right character set for old files, as you might risk saving the project in a format that makes showing or recovery of the original characters impossible.

The program makes backup of old files in the old format, if you want to. Those backups will be saves as FILENAME(ansi).~pr/~sy. This means that if you have chosen the wrong codepage for conversion, you can go back to the original format and try again.

And just to emphasize: if you haven't made projects including "foreign" alphabets/characters, you will not experience any problems, and it will be safe for you to tick off the dialog when you see it the first time.

New functions for installation projects

In this version of the program many of new functions have been added that support installation type projects, including some more mechanical drawing functions.

Panel Builder

The module is made for quick and easy design of small panels, primarily for house installations. It works in this way: start by choosing a template (type of installation), and the program will sort the components to comply with the choices you make. You can choose between manufactures and current sizes in each group, and the choices depend on the components in your database.

When all choices have been made, you will see the parts list – including prices from the database – hereafter the program can draw the panel. With the panel drawing you can generate a singleline diagram with another new function, see the paragraph

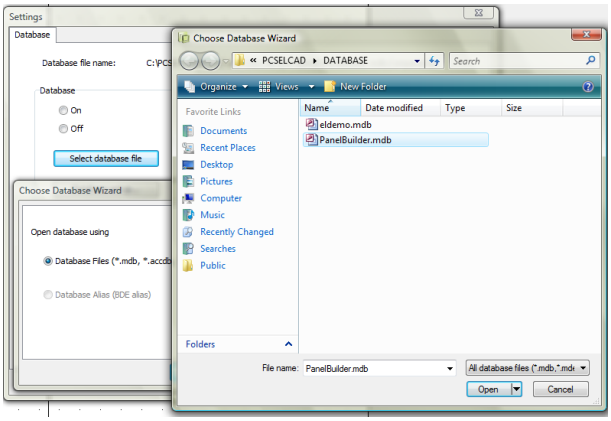
Automatic generation of single line diagrams page 17.

Because you need to make different settings before the PanelBuilder works properly, we have prepared a small test project, which is presented below.

Step-by-step: How to test the Panel Builder

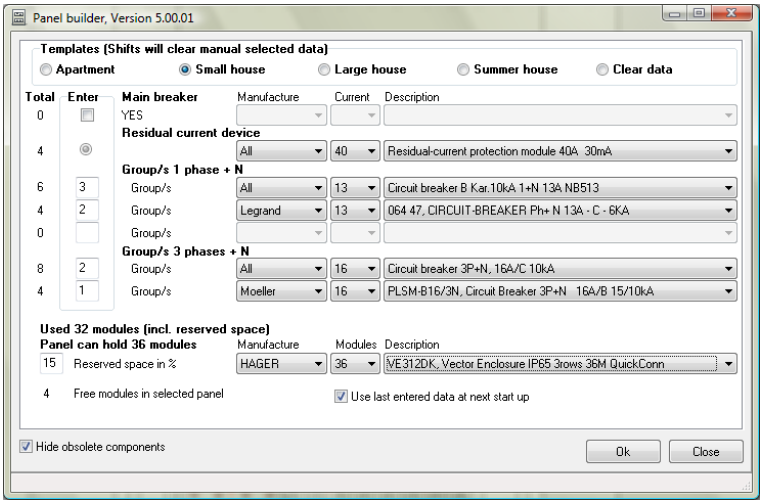
First you need change databases to the PanelBuilder database:

1. Go to the menu Settings
2. Choose Database
3. Click Select database file
4. Click Next
5. Choose PanelBuilder.mdb
6. Click Open
7. Choose the Components table



You can start the PanelBuilder from the Tools menu or click the icon:

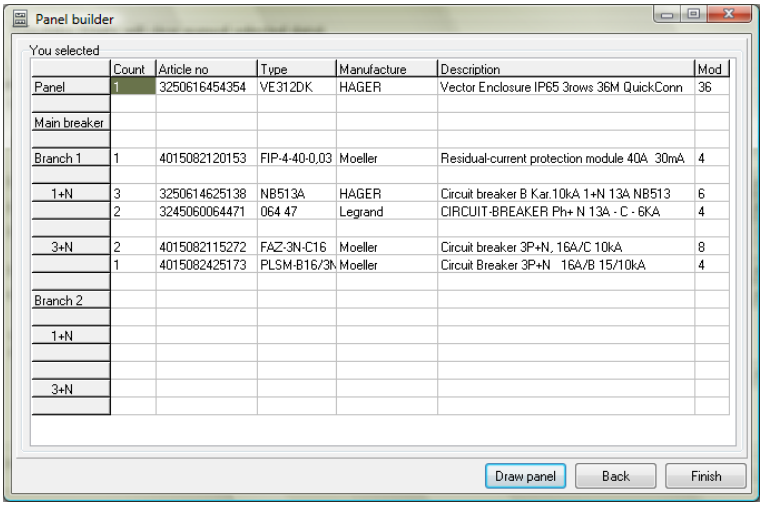
The project PanelBuilderExample.PRO contains the same data as shown below.



Select Small house, select more 1-phase and 3-phase circuit breakers, and possibly 15% reserved space in the panel.

When you are satisfied with your selections, click Ok.

The program now shows a parts list.



Now you click Draw Panel, followed by Finish.

The program will now draw the panel with your selected components, and the project also contains updated parts and component lists.

The program uses the template PanelBuilderTmp.PRO which is placed in the folder ..PCSELCAD\Standard\PanelBuilder.

Step-by-step: How to do to use the Panel Builder in the future

You need to make various settings in the database. This is explained in the paragraph Settings in connection with the PanelBuilder page 18.

If you want to use the PanelBuilder in the future, you will need to import a database that contains components for panels and you need to make fields in your existing database that are required by the PanelBuilder.

You need to make the following datafields:

PBCURRENT – contains value for ampere (13, 16 A)

PBPHASES – describes phases (1 1+N, 2, 3, 3+N)

PBMODULES – describes the size as how many modules

PBCODE – describes the type of component (FAUTO, FOR, HPFI)

PBSYMBOL – contains the file name for the panel symbol

All fields are created as WIDESTRING with a length of (at least) 16.

Your own components for the Panel Builder

If you want to create your own components for the PanelBuilder, you have to know the following:

- Symbols for PANELS need to have a marked area in which components can be placed. This area must be drawn with the colour NP. In this way the program can place the components automatically.
- Extra data fields must be made in the database for singleline symbols and for PanelBuilder components, all fields start with PB, and are meant to contain technical data for sorting in the dialog. Also see the paragraph that starts on page 18.
- If you want to make your own template, you must change the standard template called PanelbuilderTmp.pro. It is in the folder ..pcselcad\standard\panelbuilder.

New drawing headers

- New drawing headers have been made for the PanelBuilder, from A4 to A0 sizes, vertical and horizontal respectively. They are all named PCS[format]Panelbuilder.

If you wish to make an automatic single line diagram when the project is finished it can be done via the Tools menu. You need to make a project yourself, or you can use the PanelBuilderExample.PRO. Refer to the paragraph Automatic generation of single line diagrams page 17.

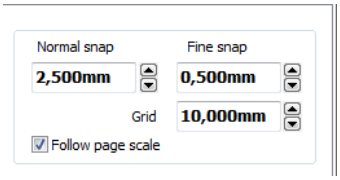
"Magnetic" mechanical symbols

This is a new function that allows easy and quick placing of components on a DIN-rail, in a certain cut-out, or just on a mechanical page. It can be difficult to place components side-by-side, especially if your scales are a little off the normal scale. This can now be done with <ctrl> + normal place or move command. Components – one at a time – can be placed precisely side-by-side in this way.

If you use <ctrl+shift> you get a quick and easy placing at either the left or right end of a rail or in a cabinet.

For symbols **without** states, the area for placing components must be drawn using the NP color in either layer 0 or layer 255. For symbols **with** states, the area must be drawn in each state using the NP color. If you want to see the area on print, you also need to draw the area with a "real" color. …

Snap and grid follow page scale



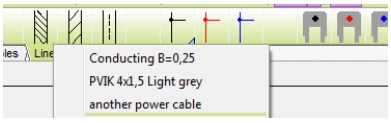
Normally, when you make mechanical drawings you work with fixed module sizes, and you wish to work with those sizes independently of the actual page scale. Now it is possible to have work with fixed module sizes and "jump" one module at the time.

Normally, the SNAP will follow paperSIZE but now you can make it follow paperSCALE instead. This is activated in Page settings|Follow page scale".

Drawing connections on installation drawings

If symbols are placed in different heights (as in house installations) drawing of connections has become easier: The program will automatically change height when the symbols are connected.

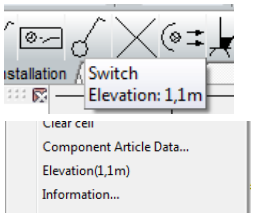
Cables and wires can be in the pick menu



You can draw with cables, wires or bridges directly from the pick menu. This means that it is possible to add these components to the pick menu with article data. Line article data can be added – as with symbols – by right-clicking the window with the line and selecting Component article data.

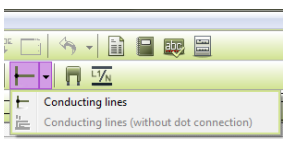
Remember to tick "Include lines with article data" in the parts list settings…

Elevation in the pick menu



You can add elevation directly in the pick menu. If you have to symbols/components that are always placed in the same elevation the elevation can be added as a property to the symbol. If you need to place it in different elevations you need to make a cell for each elevation. The elevation can be changed on placed symbols.

Extended Conducting lines button



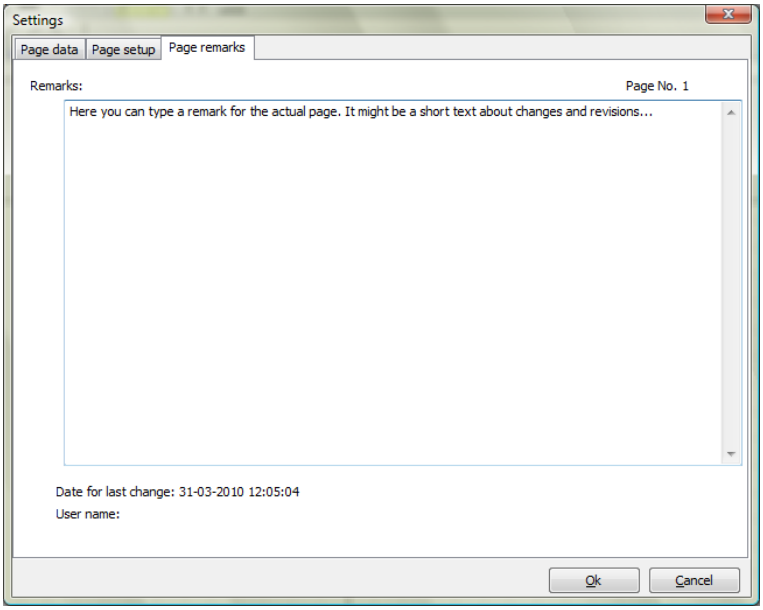
On mechanical pages you need to draw lines on top of lines on installation drawings, particularly when drawing "cables" instead of "lines". In order to avoid dots where there are no connections but only parallel cables, choose the lines without dot connection.

Revisions and page remarks

There is a new tab in Page Settings called Page remarks-

It is possible to type in free text – notes – here.

Last changed data is displayed at the bottom of the dialog, and this can be included in a list.



Print Page remarks

The new list can be found as a template – is based on a TOC.

Page no.	Page remarks	Revision	Last corrected:
1	This is a remark for page 1.		04-02-2010 11:37:46
2	This is a remark for page 2		04-02-2010 13:22:56
3	This is a very large remark for page 3, and the remark is too large for the datafield in the remark list... and I can make remarks in multiple lines		04-02-2010 13:49:06
4	and that will not look good in the list...		04-02-2010 14:01:30

In the example above, you can see page no., page remarks, page revision and last corrected date.

REMEMBER to adjust settings for the used data fields…. especially the remarks. They might easily be too long to fit in the list. Consider what to do about line breaks in the original text, you could make the text field in the list larger, or avoid line breaks, or limit the remark size, or…

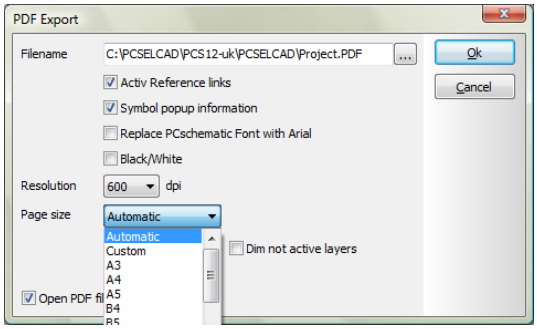
”Old” methods for revision or version control

There are multiple document handling programs on the market, that can control who, when and how you are allowed to save a file on your network. Those programs are not PC|SCHEMATIC. Apart from those methods, you have these possibilities:

- The network can be set up with read/write access levels for different profiles
- You can make ”clouds” – from version 11. They might be in different colors. In our forum you can find an OLE-module that can manipulate the clouds…
- You can copy a page, give it IGN status, and keep it in the project. The IGN-status means that all components on the page are ignored from all lists. In this way it is possible to include old versions in the “real” project.
- Drawing headers can show status for each page.
- The ”good old” way of making revision control in Automation. Here you can make revision no.s for pages and/or for projects. The numbers can be included in the drawing header. The method is described in the manual.

Export to pdf-format

You can now export to pdf-format. Earlier you could print in pdf-format, i.e. CutePdf. The difference and the settings will be shown below.



In the Files menu you will find a new item "Export" where you can select "to PDF".

In this dialog you make the wanted settings for export to PDF. You can choose to open the generated file in order to view the result immediately.

When you select the export function, it is possible to you choose a folder for the exported files under Settings|Directories.

You can also save in pdf-format in Files|Save as, where you have the pdf-format as an option in line with other formats.

Currently, only Western European texts are supported.

Pdf command file conversion

It is possible to make a command file that performs autosave in pdf-format. The function was made earlier for dxf-format.

The examples below (the green lines) are written in a SAVEASPDF-file with extension *.CMD. Automation is started with this CMD file as a parameter in the command line:

Ex.: C:\PCSELCAD\PCSELCAD.EXE C:\THE-PATH-TO-CMD-FILE\SAVEASPDF.CMD

Below you see four examples of different command lines.

FILESAVEAS .PDF C:\PCSELCAD\PROJECT*.PRO
All .pro-files in the folder C:\PCSELCAD\PROJECT\ are saved as PDF in the same folder.

FILESAVEAS .PDF C:\PCSELCAD\PROJECT*.PRO C:\PCSELCAD\PROJECT\PDFSAVE
All .pro-files in the C:\PCSELCAD\PROJECT\ folder are saved as PDF-files in C:\PCSELCAD\PROJECT\PDFSAVE.
The folder C:\PCSELCAD\PROJECT\PDFSAVE must exist in advance.

FILESAVEAS .PDF @C:\PCSELCAD\PROJECT\ProList.txt
All project files in the list file C:\PCSELCAD\PROJECT\Prolist.txt are saved in the individual PRO-file folder as PDF-file. (see Prolist.txt below).

FILESAVEAS .PDF @C:\PCSELCAD\PROJECT\ProList.txt C:\PCSELCAD\PROJECT\PDFSAVE
All project files in the list file C:\PCSELCAD\PROJECT\Prolist.txt are saved in the folder C:\PCSELCAD\PROJECT\PDFSAVE as PDF files.

The folder C:\PCSELCAD\PROJECT\PDFSAVE must exist in advance.

Example of contents of a Prolist.txt:

C:\PCSELCAD\PROJECT\PCSMOTORDEMO1.PRO
E:\PROJECTS\MYPROJECTS\PROJECT 007.PRO
C:\PCSELCAD\PROJECT\MYPROJECTS\MYNEWPROJECT.PRO

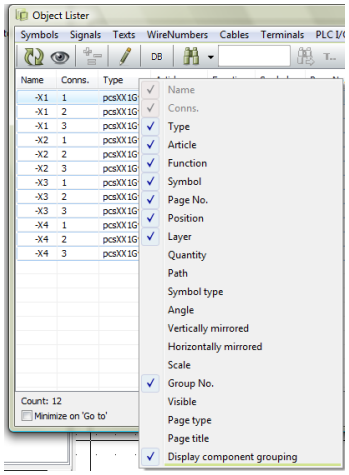
Terminal functions

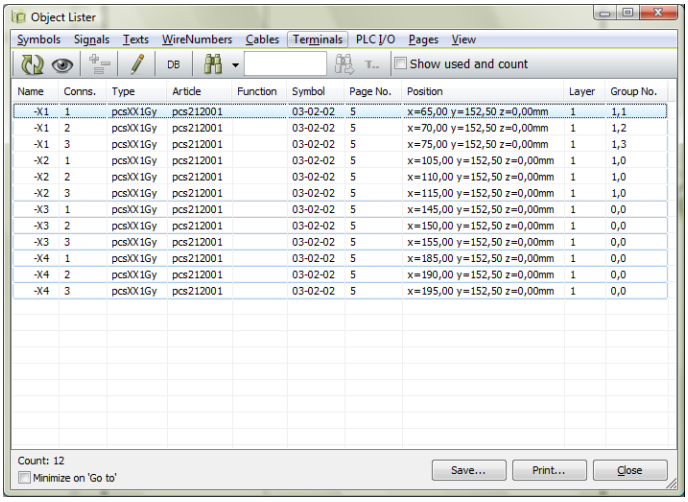
Extended functions for drawing terminals in projects.

Grouping of terminals

The Objectlister can show terminals – and other symbols – grouped by their physical positions, meaning as part of a component = group.

The View-part of the Objectlister is available by right-clicking in the topbar.





A more graphical edition of F7 has been made, and that should made component grouping more intuitive. Component grouping can be a very time consuming task, especially if you need to switch many some relations in the terminal row.

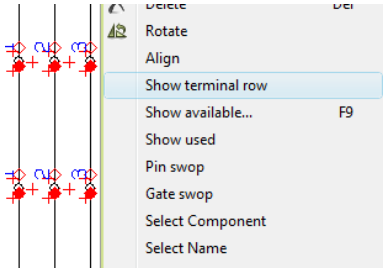
The new window can be open as you are working in the project, and in that way you can group the terminals as you work.

Control of physical components (terminal rows) that the electrical symbols represent influences the sequence and the number of components in list and graphical plans for terminals, components and parts.

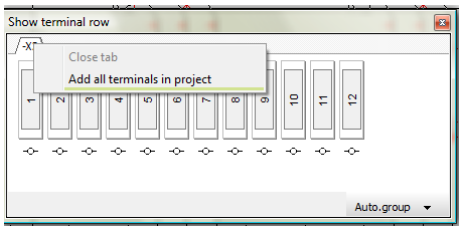
If the parts list is incorrect (to many or too few) it is probably because of component grouping. This can be caused by

- drag'n'draw can mess up group numbers
- copying of symbols can mess up group numbers

Normally group numbers will be ok, when the components come from the database.

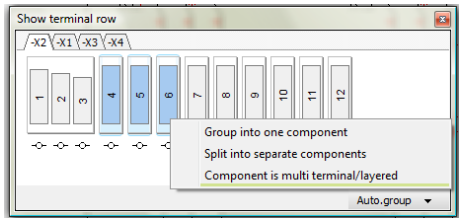


Place the terminals in the project. Select a terminal, right-click and the shown options will appear.

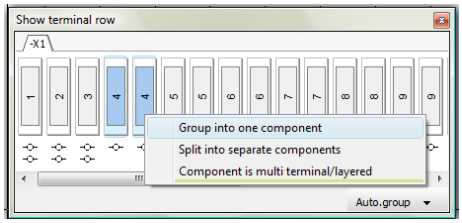


When the box appears, you will see the selected terminal row. The other terminal rows can be added if you right-click the tab followed by "Add…".

As you work with the project you can close the tabs (the terminal rows) as you proceed.



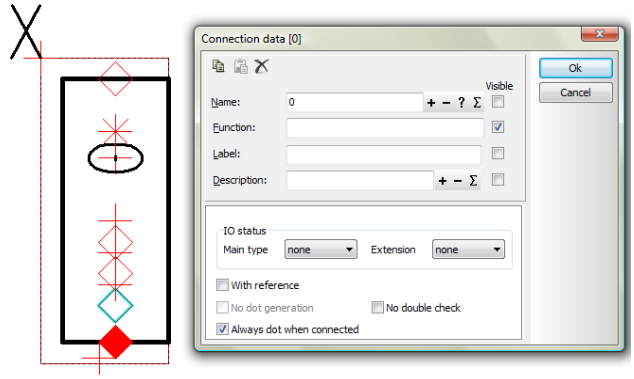
In the window you can mark more terminals, right-click and the group them into one component. When the terminal is grouped, it can be set up as multi-layer terminal, a setting that will be seen in the window.



If the same terminal appears at several positions in the project, those extra positions will be indicated as -O- below the individual terminal.

If your terminals come from the database, that means that there is a known grouping, this grouping will – of course – also appear in the project in the terminal row windows.

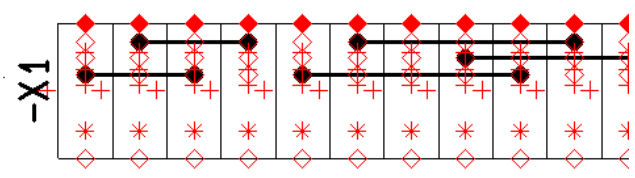
Forced dots



Dots for bridge connections will now be showed in graphical terminal plans.

The terminal symbol that is used for terminal plans must be edited in order to show the bridge connections. This is done the following way:

In Edit symbol all connection point with IO status None/None choose "Always dot when connected".

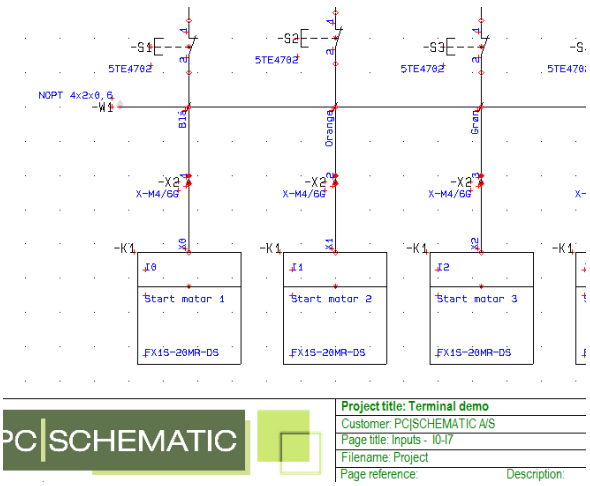


When the symbol is used in the terminal plan it looks this way. (Remember to choose the edited symbol in the terminal plan setting).

Integration with CLIP PROJECT

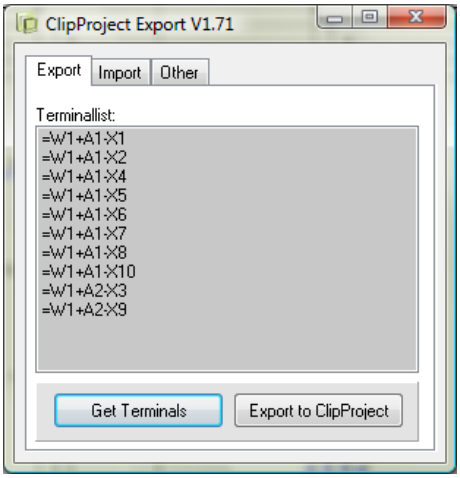
Integration to third-party programs has become easier than it used to be. The module for CLIP PROJECT shows such integration.

As not all our customers use this manufacture of terminals, this module is not installed when you receive the program (update), but it can be downloaded from our homepage, as can CLIP PROJECT from Phoenix' homepage.



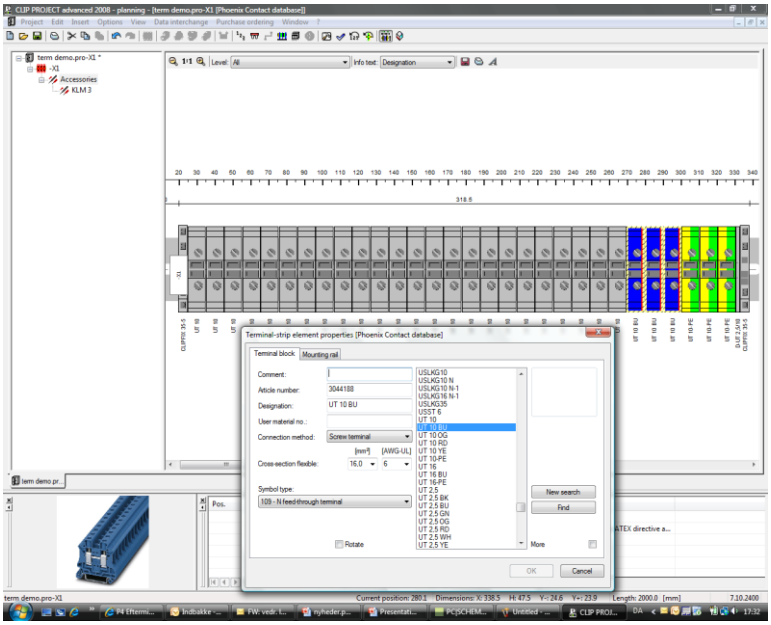
The terminals are placed in the project, without adding article data, as this will be added later in CLIP PROJECT.

Of course you can place terminals with article data, too, if you wish…



The module includes a tool for export of terminal data, and it is found in the Tools menu. Remember that you need to download from our homepage and to install the module before you have this tool.

CLIP PROJECT will then configure each terminal row: chose terminal type from the built-in catalogue, and then let the program add dividers and end pieces. See the terminal row on your screen as you pick your terminals.

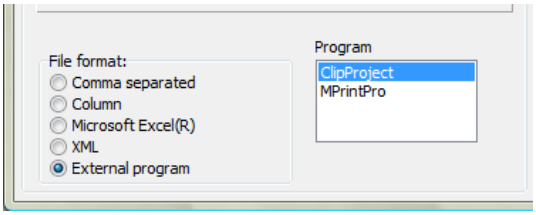


When the terminal row is fully configured, you export its to a XML-file. This XML-file is then imported into Automation where the project is updated with terminal data and a layout drawing. You can update the parts list, which now includes all parts and pieces for the terminal row.

BEWARE!! PC|SCHEMATIC does not have detailed knowledge of this program. Support on it must be obtained from Phoenix or his distributors.

Integration with M-PRINT PRO

This part of the program is not pre-installed, either. The module can be downloaded from our homepage, and the M-PRINT PRO program can be downloaded from Weidmüller's homepage, or his distributor's homepage. In Denmark, it is Wexøe.



When you want to print labels for terminals, cables or other components, this is done with the "List to file" function. This has been extended (when the module is installed), making it possible to export directly to M-Print PRO.

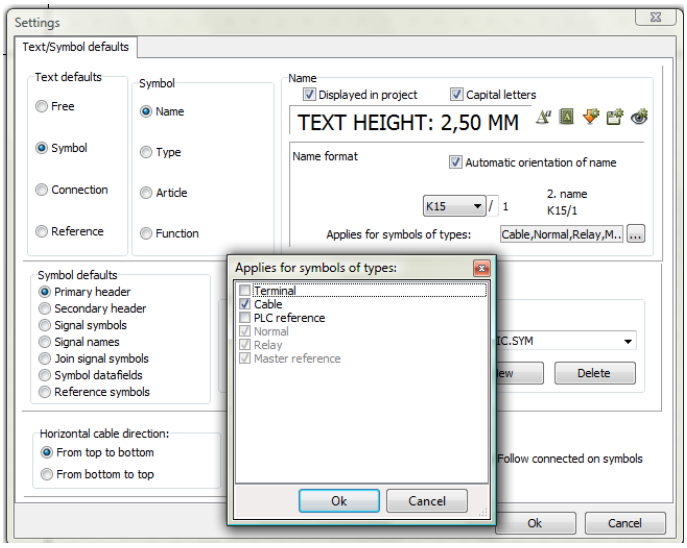
When you choose the external program is started, and the exported file is automatically loaded.

In both programs you need to make choices concerning label type. You can scroll through electronic catalogues, select your label, format the print – you can change fonts and sizes – and see a preview.

Remember to save the format files with you sorting settings, and to save it in the Tools menu.

BEWARE!! PC|SCHEMATIC does not have detailed knowledge of this program. Support on it must be obtained from Weidmüller or his distributors.

Terminals, cables and plc-references with/without current path names



It has been a long-term wish that terminal rows could be named according to page and current path.

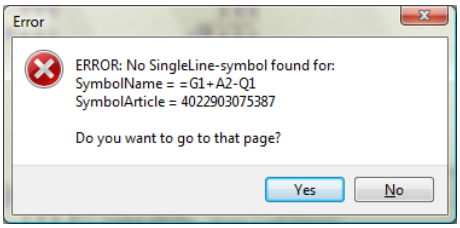
In motor controls etc. the terminal row is placed together with the engine, the circuit breaker etc. and it has been inconvenient to have manual naming or another naming system for those components, especially due to loss of automatic naming.

Naming system for cables and plc-references can be chosen in the same dialog.

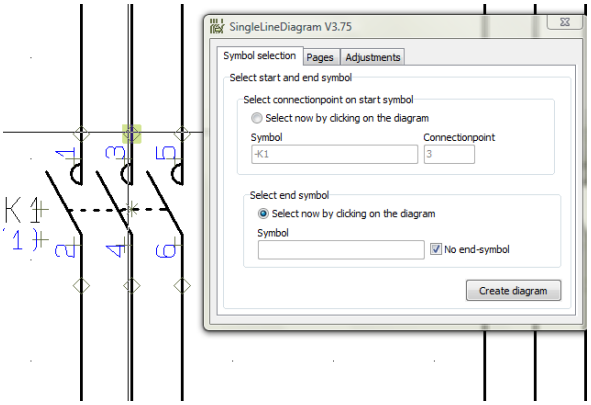
Automatic generation of single line diagrams

The program can generate automatically single line diagrams based on multi line diagrams and panel drawings. This is a module that can be installed in Files|Modules. When installed it is found in the Tools menu.

When using the module, all components in the diagrams must have valid single line symbols in the database, or the diagram cannot be generated. How to set up the data fields and connections between Automation and the database is explained in the chapter New fields in the database dialog on page 18.

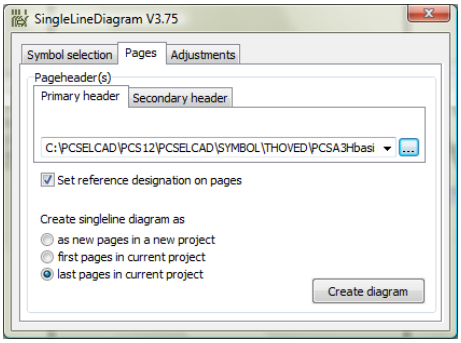


When you start the program, the following dialog will appear. Then you select and mark the connection point from which you want your single line diagram to start. If you make the diagram based a PanelDesigner file, this program identifies the start symbol itself.



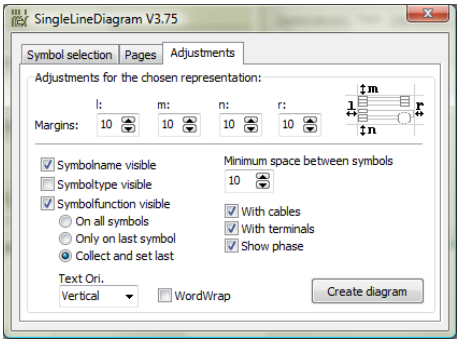
Then you choose the end symbol alternatively tick the “No end symbol” box.

The module contains more setting options on the next tabs, including drawing headers, and how the single line diagram should be generated. It can be generated as a new project or it can be included in the current project.



If you choose to include it in the current project it will be a chapter starting with a chapter divider, in the same way as the graphical plans.

The last tab gives you the option to set margins and distances between symbols, choose visibility for various texts, and to choose the components that are included in the single line diagram.



You can try the function by opening one of the demofiles (MOTORDEMO1 or MOTORDEMO2). The PCSDEMO database must be connected in order to have single line symbols for the components.

Please notice when you use the program:

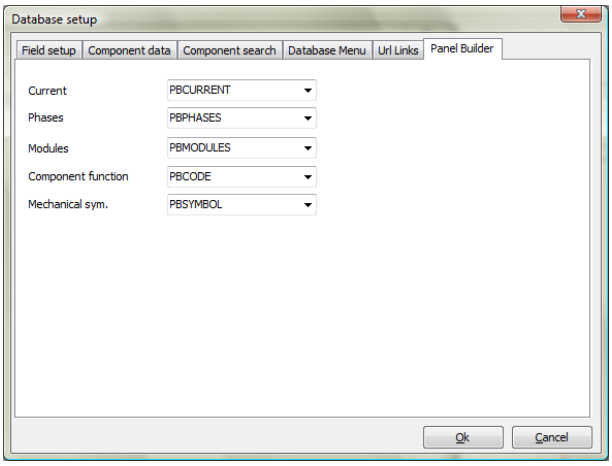
- When a component doesn’t have a single line symbol in the database, the program uses a “dummy” symbol, also for cables and terminals.
- Your symbols must be created with uneven connection points at the top or to the left (all “our” symbols are created this way).
- When you choose a starting connection point, you need to consider where the diagram continues to: if you choose to start with a PE-connection, you will show a connection that goes throughout the whole project, and that might not be, what you really intended to show.

New fields in the database dialog

In connection with the PanelDesigner and Generating single line diagrams extra data fields must be created in the database to support the extra data in existing installations.

A couple of other new functions also need new data fields. Setup of those functions is also described below.

Settings in connection with the PanelBuilder



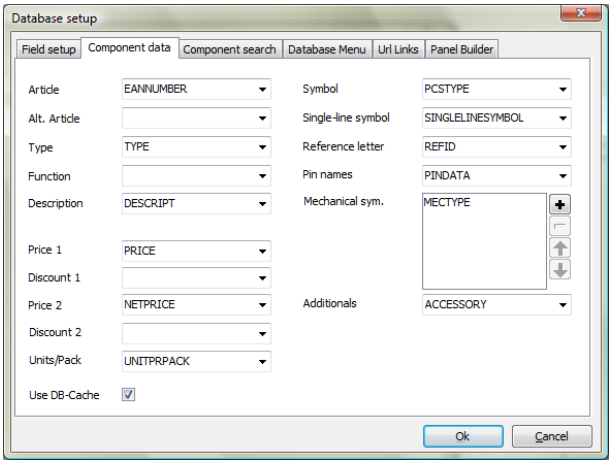
In order to make the PanelBuilder work, it needs some technical data as seen in the screen dump.

The data come from the database which then must contain the data fields.

It is up to you – or to the administrator of the database – to create those data fields, and they need not have the same names as shown here, but the mapping must, or the module cannot work.

The field PBSYMBOL contains symbols that look very much like MECTYPE symbols, but the symbols can only rarely be switched. The reason is that MECTYPE symbols either contain all connection points (when it has been made "correctly") or no connection points (when it has been made as a #xy symbol). The PBSYMBOL symbols must contain an IN and an OUT connection point in order to facilitate a correct panel drawing. All rules concerning design of symbols for the PanelBuilder is explained in the manual.

Settings in connection with the generation of single line diagrams

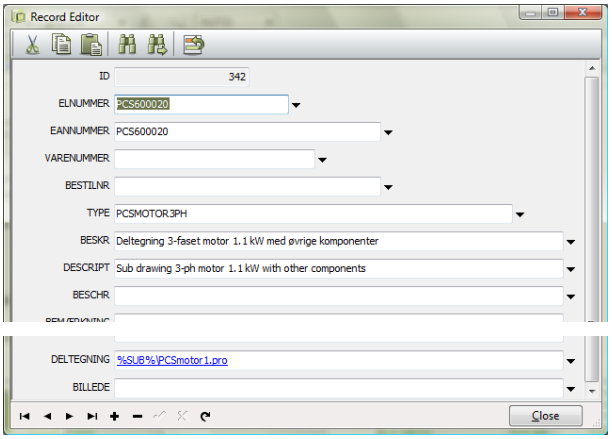


In order to make it possible to generate single line diagrams, the database must contain a single line symbol for each included component's power part.

This field must be created in the database and must be connected to Automation as shown in Database Setup|Component data.

Sub drawings

When you select in the database it has now become possible to select a sub drawing for this component and not "just" a line of symbols. This also means that the pick menu can contain sub drawings now, as the pick menu fetches it information from the database. The field that contains the sub drawing must be created in the database and subsequently linked to the program, see below.

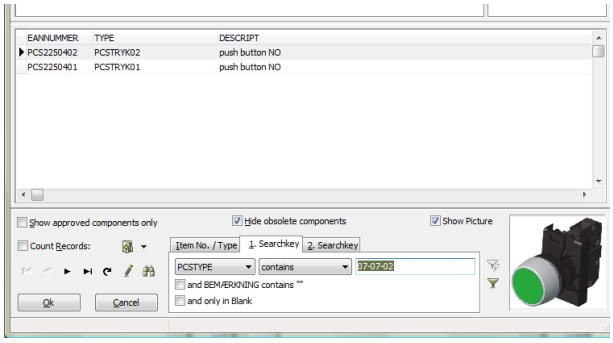


In order to have the sub drawing in the pickmenu, it must be created as a record in the database, meaning that it must have a partno, possibly a type and a description, and that the sub drawing itself is linked as shown to the data field that contains the sub drawings.

Then when you wish to place the sub drawing in the pickmenu, you first place a symbol – any symbol – and then you add component data. If the created part no. refers to a syb drawing containing model data, the model is selected when you place the sub drawing. You can place sub drawings on electrical and on mechanical pages.

Thumbnail

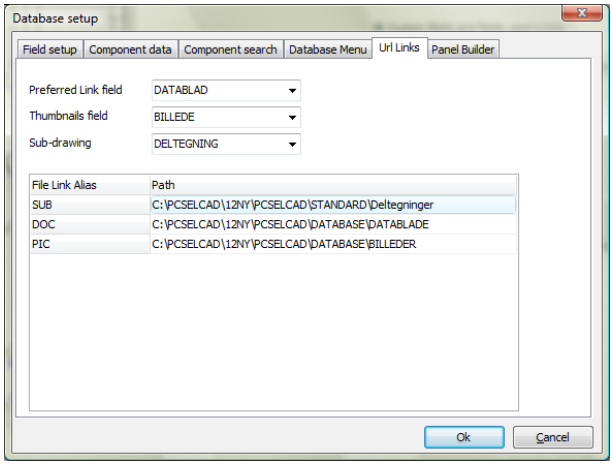
The database dialog has been extended making it possible to show a thumbnail of a component. The data field with the thumbnail link must be created in the database and linked to the program. See below.



In the THUMBNAIL data field the thumbnail is set like this: %ALIAS%/thumb.jpg.

Settings in connection with sub drawings and thumbnail

The folders that contain thumbnails and sub drawings are mapped in the Url Link tab under Settings|Database|Database setup.

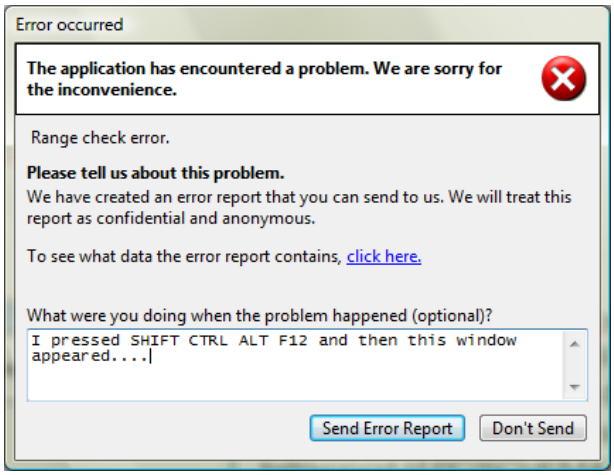


In the SUBDRAWING data field the thumbnail is set like this: %ALIAS%/subdrawing.pro.

Tools

New tools for users and our programmers.

Error log



The first tools is a "help yourself", as you now have the opportunity to send an error report to PC|SCHEMATIC when the "red bar" occurs. The error report contains data about the occurred error, and you are also encouraged to write what happened. The more info we have, the more the programmer will know about the reason for the error.

Symbol manipulator

The symbol manipulator is a small tool that can manipulate symbols in the symbol library. Manipulation in this context means changing the placing of texts, changing text fonts and sizes etc..

The program is only partly documented, and can be obtained from PC|SCHEMATIC. WE CANNOT GUARANTEE FOR ANY USE OF THIS PROGRAM (remember backup!!).

OLE automation

It will now be possible for (advanced) users to make own applications to the program.

Description of the interface can be downloaded from our homepage.

Apart from the description, you will find an example file showing the options.

We have, ourselves, used the OLE-automation for several years, as most functions in the Tools menu are actually made using the OLE-interface.

We have uploaded some example files to our user forum, among those a modulet that can manipulation clouds in a project.

Conversion of old projects to Unicode

A program module which can help you "translate" all your old projects and symbols to Unicode, if they are created with version 11 or below. Remember to make a backup!!

The module is installed under Files|Modules.

Changes and extensions for list functions

Import of project data with XML-format

We have always been able to export data with headers. The new thing is that it is also possible to import headers into the project, by utilizing the XML-format.

The import function is used already, as it is possible to get a pick list for drawing a project, either as a parts list or as a components list. Now these lists can contain project data that will be updated or created, if they are not in the project already.

Importing a parts list in XML format

When importing a parts list in XML-format a path for saving the file in *.pro or *.pdf can be included. Example:

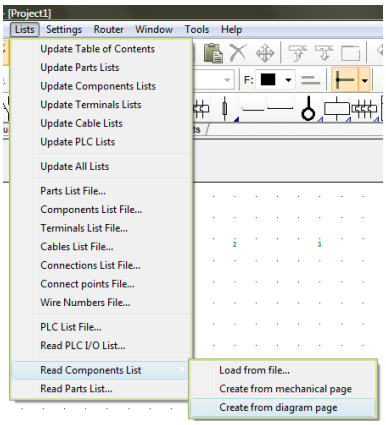
```
- <PATHSETUP>
  <Pathname="PDF" value="C:\PCsElCad\Projekt1\PDF" />
  <Path name="Project" value="C:\PCsElCad\Projekt1\Project"/>
  </PATHSETUP>
```

Create components list from diagram

You can now create components lists from diagram pages the same way as you from mechanical pages.

If the diagram is based on sub drawings with model data, the part no. for the sub drawing is loaded and it can be placed in one operation. The components that the program must recognize as part of a sub drawing must have a variable in its components data.

If not, all used components are loaded individually.



Extended functions for pick lists

In order to make the work flow better, different changes and extensions have been made especially for those that use pick lists for placing components.

Example: In the list 3 pcs, shows 3(3). Placed 3, shows 3(0). Placed 5, old version shows 3(0), new version shows 3(-2).

If you have 3 in the list, you have placed 2 and you don't want to place more, you can right-click and change the 3 to 2 (the placed number).

The component pick list can handle sub drawings. You need a database field in the database (see the chapter New fields in the database dialog from page 18). If the article number has data in the database field, the subdrawing will be loaded. The pick list will also be updated. The function works on electrical and on mechanical pages.

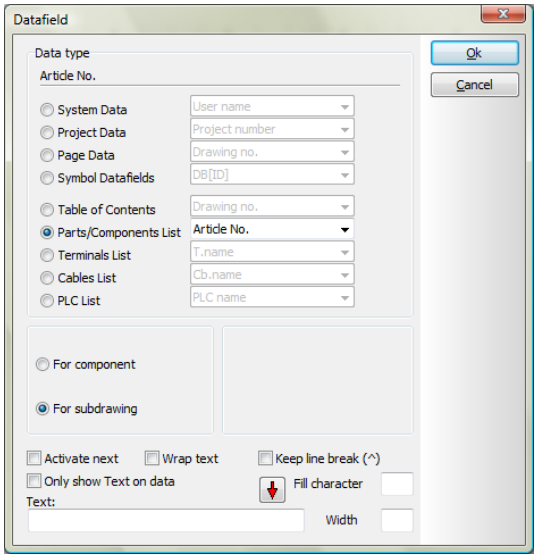
The pick window for parts list has been extended with <Ctrl-click> and a right-click menu with "-Place" as you know it from subdrawings.

The pick window for parts lists can also handles sub drawings. Subdrawings will be placed automatically if the template contains insertions points. This works on electrical and mechanical pages.

Data fields

There have been various changes and extensions regarding data fields.

The data field Article no.



The data field Article No. in parts/Components list has been added the options “For component” or “For sub drawing”. In this way it is possible to include the sub drawing article no. in components lists.

In order to show the sub drawing’s article number for the components in the components list, the component itself must have a variable in the sub drawing.

The components that the program must recognize as part of a sub drawing must have a variable in its components data.

If this is not the case, the Article No. data field will be empty.

Quantity and cables

The data field ”Quantity” is now available for cable lists and cable lists to file.

DB fields in terminal, cable and plc lists

It is now possible to include database fields – as manufacturer or units per pack – in terminal, cable and plc lists and the same lists to file.

The text in front of data field contents

In the Data field dialog the ARROW DOWN function has been changed:

It used to add a ” ” (space) to the field name. This is changed to “: “ (: and space).

Connection lists

Data fields for quantity and for bridges have been added to connection lists.

Connection lists to file will also check for correlation between line article data in the same net.

Various minor items...

The program has new icons

The program has new icons inspired by the general Windows Vista design.

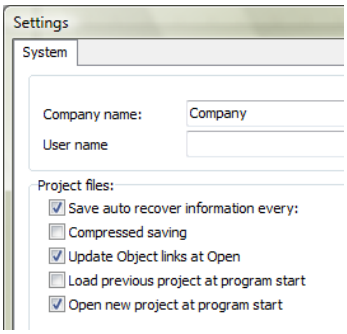
Concerning the new icons, some old icons have been deleted. They are the icons for Scroll, Zoom and PageUpPageDn buttons in the left sidebar.

List format

The new Excel format (*.xlsx) is now supported by the program. This is used for import and export of lists (lists for file, reference designations (import/export)).

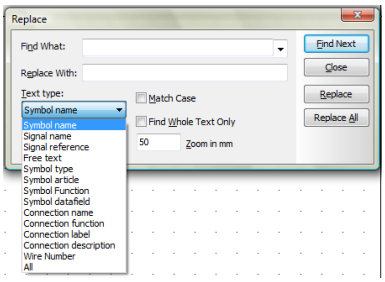
System settings

You can select whether you want to open a new project at program start.



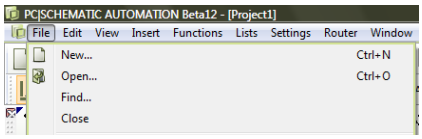
Find/Replace

The Find/Replace function now contains Signal reference texts.



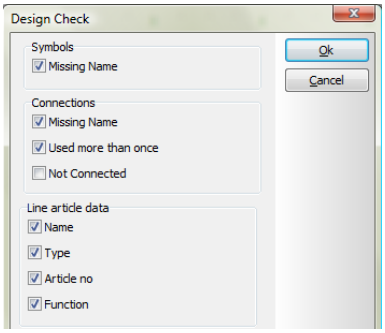
The Find function

The ”good old” search for file is now included in Files|Find…. The function works as before, but the “Remember files” is deleted from programs working on the Windows Vista and above, as the function is only relevant for the XP’s Open file dialog.



Design check

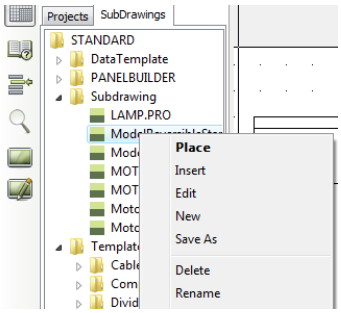
The design check function now includes an individual check for Line article data – missing name, type, article no. or function – as seen here.



Text translator

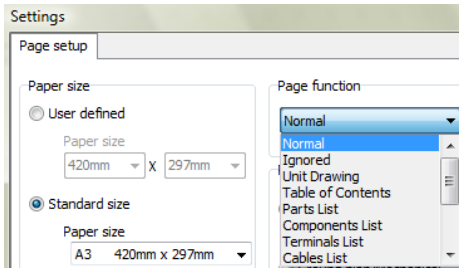
Dictionaries (text translator) can now export/import to/from Excel files (*.xls/*.xlsx).

Insert function in the sub drawings window



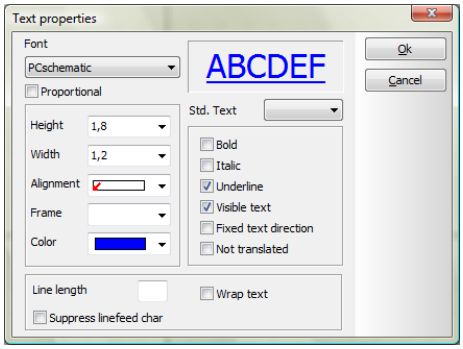
The button for inserting sub drawings in the Area toolbar is deleted and replaced by a new menu item "Insert" in the rightclick menu of the pathfinder window. The function of the menu item is equal to Ctrl-clicking or dragging.

Changes in page function



In Settings|Page settings it is now possible to change page functions, from all functions to all functions. Remember that data fields for lists always must come from the right category.

Text formatting



Underline is now an option in the Text properties dialog.

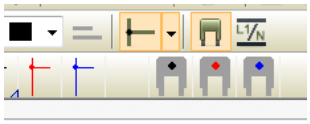
The program can be opened with parameter call

For instance: C:\PCSELCAD\STANDARD\Template.pro /IP C:\PCSELCAD\LISTS\Partslist.xls.

What happens here is that first the Template.pro file opens followed by a Parts list in Excel format. The parts list might also be in xml format. The parameter /IP means Import Parts list.

Bridges in the pick menu

Bridges can be inserted in the pick menu, with or without data. The dot in the middle of the symbol represents the line color of the bridge.



Measuring

The measuring command is now active until you press the <esc> or until you choose a new command. The command can also be used in the Edit symbol function for control purposes.

Fine snap and move with arrows

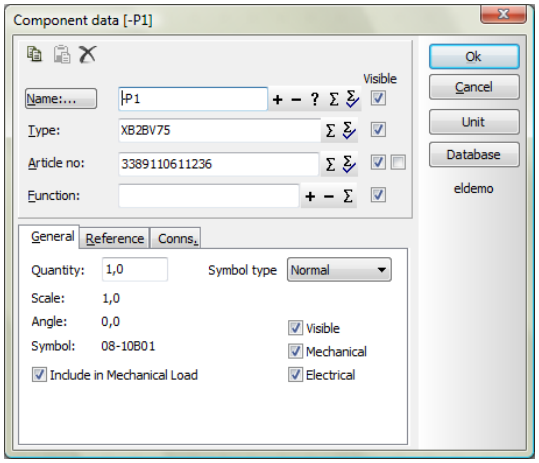
Move objects with arrow will now work together with the <Shift> button = move in fine snap.

Moved shortcuts

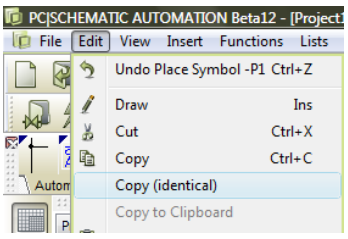
Shortcuts <Shift+arrow> have been moved to <Shift+Ctrl+arrow>.

Icons

The Edit menu in the Component data dialog has been replaced by icons, which makes the edit functions more intuitive.



Copy (identical)



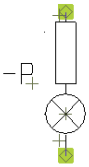
The function "Copy (identical)" has come into the Edit menu. The function makes it possible to make an identical copy of a symbol (without any intelligent automatic functionality). The primary reason for the function has been this: when you wanted to place the same terminal several places (distributed drawing) you needed to control the component grouping by ctrl+F7 and make those identical, as the copy command increases the component group. Beware, that if you copy a terminal with component group number 0 with this function, both terminals will be group 0, meaning that the program sees them as two different components.

The function is only possible when you mark a symbol, and when this symbol is not a relay.

Changing cable quantity

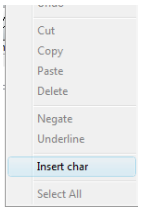
Changing of quantity for cables and number of conductors will be valid for all parts of the cable when pressing OK.

Highlighted connection points



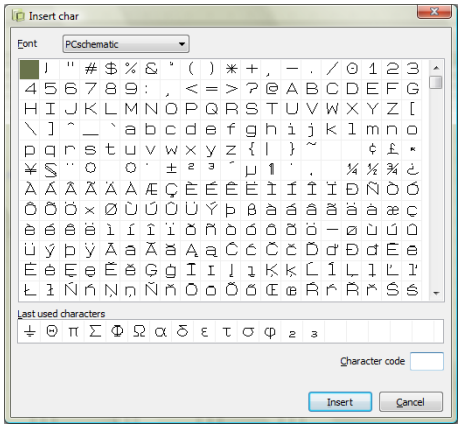
In relation to the new colour scheme connection points have been added a green box, when they are chosen.

Insert special characters



Inserting special characters by means of the Alt-button has been changed, as this cannot work together with the Unicode character sets. When you need to insert a special character in a text you right-click in the text window, and get the option “Insert char”.

When you select this option you get a dialog resembling those in other Windows programs, from which you can choose a character.



In the window you can see the last used characters, and you can select your characters from here, too.

Dos projects

From version 12 it is no longer possible to load old dos-projects. If you wish to maintain this option, you need to keep a version 11 or older installation for this purpose.

New folders

The following new folders are created in the program:

In the folder **SYMBOL**:

PCSHEAD – contains all PCS-headers. From now new headers are not included in the MISC folder. Alias is HEAD.

PBSYMBOL – contains all symbols for the PanelBuilder. Alias is PBM.

In the folder **STANDARD**:

PANELBUILDER – used for and contains the start template for the PanelBuilder. This template can be edited if you wish other lists, but it must be saved with this name.

In the folder **DATABASE**:

THUMBS – where you can save thumbnails for components.

CATALOGUE – where you can save datasheets for components.

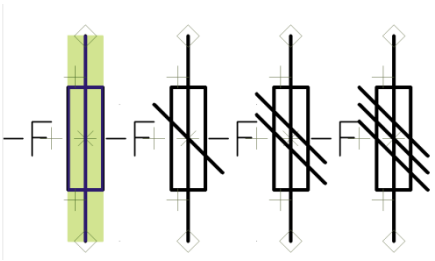
When making a new installation URL-links for those two folders are created (in Settings|Database|Database settings).

Pickmenu

When making a new installation new pickmenus will also be created, in which all new functions are shown. If you make an update of your system your will not see these. The new functions are component data on lines, bridges and sub drawings.

Change state

Change state on symbols with Shift+Space.



and don't forget....

The tab sequence has been changed in (some) dialogs to a more logical sequence.

New material in connection with the new version

When you choose to update your installation to the newest version, you only update the program itself. We never touch the other files, unless they have the same names as our files, as is the case with e.g. symbols' filenames: a coil is still 07-15-01.sym and looks the same way as always.

This means that being an existing user; you will not automatically see all new functions, as they present themselves to a new user. Therefore you will find a list below of the updated files and folders, so you can copy them to your own folders.

Projects: there are new demo projects that are based on the new sub drawings.

Sub drawings: there are new demo sub drawings for your inspiration if you are not already a user of this feature.

Templates: all templates are now Unicode format, meaning that you will not see the Unicode dialog when you select one of the templates. All lists have been updated.

Unit drawings: there are new unit drawings in the program. Apart from those included in the program, you can find a selection of new unit drawings with (Danish) installation material (OPUS and FUGA) on our website.

Demo database: the old ELDEMO database has been replaced by a new – PCSDEMO – which only contains demo-components, meaning no "real" components. The new folders described above contain projects and sub drawings with components from the new database.

Pickmenus: the new functions that are supported by the program are used in a new set of pickmenus. The used components are all from the PCSDEMO database.

Lists: in the List-folder you will see a couple of examples of lists for various purposes e.g. reference designations, IO lists, parts list etc. that can be imported into a project.

Project Generator: there are new demos in connection with the Project Generator, based on the new sub drawings and templates.

If you wish to see the new material, then make a new installation on your machine. It will unfold and it contains all files above, and you can try out the functions without interfering with your normal files.