



AUTOMATION

DESCRIPTION OF NEWS

Version 13.0

March 2011

This material

This document describes the news in Automation version 13.

Paragraphs marked with a § are NOT active when your system is updated.

Contents

Panning function.....	4
Shortcut to panning mode §	4
Easier navigation in the explorer window	4
More visibility and less nuisance	5
New buttons in the explorer window	5
New settings for Show available functions	5
Auto-activate "Show available"	5
Changes in the Reference Designations dialog	6
Tools.....	7
No reference on symbols on graphical plans	7
New settings in Symbol documentation.....	7
Measurement.....	7
O-snap function	8
Magnetic function	8
Move area.....	9
Move/copy area with new area reference point	9
Jumper links.....	9
Insert cloud	10
Change text formatting	10
Connection data dialog	10
Auto naming routine	10
Drag 'n Draw.....	11
Chapter divider rightclick menu.....	11
Active pagetab	11
State titles can be edited / renamed.....	11
Create variable list is now automatic.....	11
Insert character at different places.....	12
Shortcut to special characters	12
Select signal connections symbols	13
Mounting correct drawing.....	13
The naming field in the Component Grouping window is enlarged	13
News in relation to data fields	14
Parts and components lists	14
The field: Unit name list	14
New field: Total quantity.....	14
The data field Quantity.....	15
Cable accessories	15
Jumper links in the parts list	16
New data fields: Number of items and Length.....	16
Add-on for lines with name and article data	17
Cable list	17
New data field: Length	17
Extra option on Cable Wires	17
Terminal list	18
Extra function on wire numbers.....	20
Object lister	20
Wire number list.....	21
Las wire number	21
Connection list files.....	22
Line article data and connection lists	22

New data fields for Connections List file	22
Generally about List files	22
List setting, General.....	22
Data fields Next / Previous used page	23
Data list can now be added for Project titles and page titles	24
Export and import of project and page data.....	24
Bookmarks in symbol folders §	25
Printing functions	26
The Printer dialog	26
Select printer and printer setup	26
Check for "Ask when changing printer name" has been added	26
Printing layer groups	27
Possible to choose between several alternative PCSHEMATIC fonts	27
Export to PDF.....	27
DWG and DXF have been added to Files!Export	28
DWG import.....	28
Extra database functions	29
Index control of data fields	29
Selecting a database file	29
Saving the last used database in the project	29
Easier to make a new database.....	30
Print record.....	30
Automatic reestablishment of connection with a MSSQL-server	30
Cross references §	31
Signal reference.....	31
Symbol reference.....	31
Reference cross.....	31
Scripting Engine	32
Possibilities in the script window.....	32
Working in the text editor	33
Possibilities for scripts.....	33
Functions for installation drawing.....	34
Grupperinger af lag	34
Udskrift af laggrupper	34
Mostly for administrators.....	35
Version 13 and Windows 2000	35
Better support for Terminal server/Citrix	35
Sådan fungerer automatisk generering/opdatering af arbejdsstation	35
If you have a SuperPro net license / floating license... ..	35
Check your net license server version	35
New icons and colors	35

Panning function



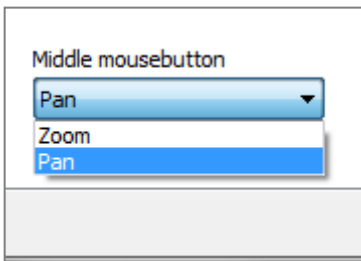
You can now use panning on your pages, that is, it is now possible to drag the diagram directly on the screen.

If you are in a zoom and panning, you can drag through the page. You can activate panning by clicking "the hand" on the vertical toolbar, and you leave it again by clicking the icon again or with Esc.

Please be aware, that hyperlinks are inactive when in panning mode.

You must also leave panning mode if you want to enter the component data window for a component.

Shortcut to panning mode §

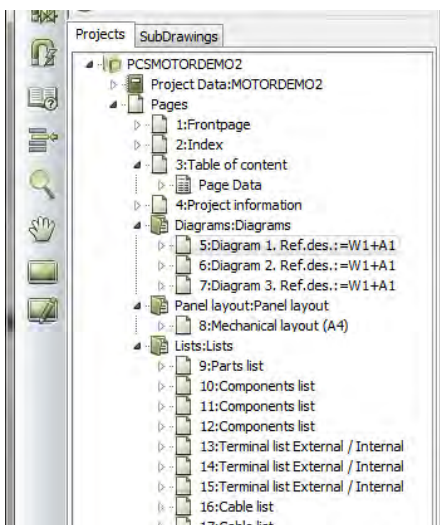


In a completely new installation the middle mouse button is set as panning shortcut. The middle mouse button used to be shortcut to the zoom function.

You can make you own selection for this button in Settings\Pointer/screen.

It is also possible – as always – to make your own shortcuts. The new panning function is found in the category View.

Easier navigation in the explorer window



Navigation with arrows has been enhanced.

This means that as you move yourself – with the arrow – to another page, focus is also moved.

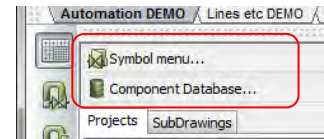
Arrow up/down navigates through open pages.

Arrow left/right will open/close folders with pages.

More visibility and less nuisance

New buttons in the explorer window

There are two new buttons in the Explorer window: One for direct access to the Symbol Menu (F8) and one for direct access to the component database (D).



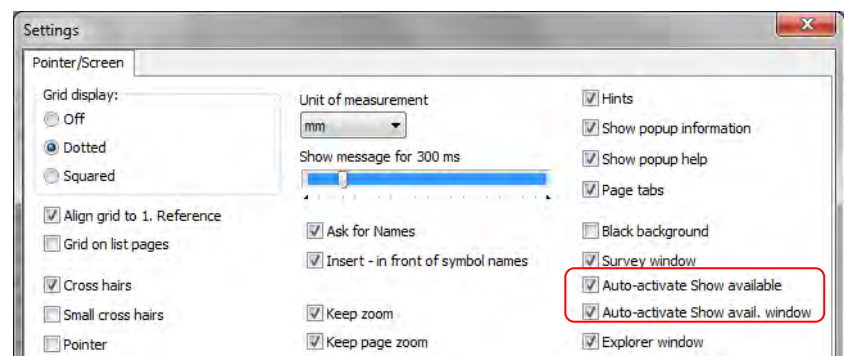
New settings for Show available functions

When you select "Settings|Pointer/screen|Auto-activate Show available window" the Show available window will be activated when you have available symbols in the project.

You will see this if you open an existing project

with available symbols, or when placing symbols from a multi-symbol component from the database (i.e. a contactor with coil and contacts).

If you close the window, the program will deactivate the setting for this session. It can be selected again with "Show available window" the rightclick-menu.

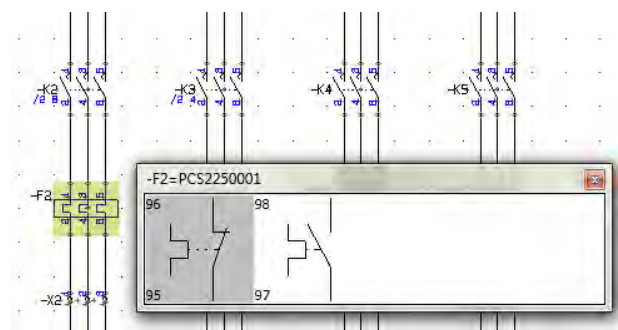


Auto-activate "Show available"

If you select "Settings|Pointer/screen|Auto-activate Show available" you will get the following: If you select a symbol in a component with more than one symbol, a pickmenu will pop up showing the available symbols for the selected component. This is the function that can be reached with "Show available" in the right-click menu. If you select another symbol from another component, the function will show available symbols for this component.

The pickmenu stays open and can be moved to another position and the size can be changed. Position and size will be remembered.

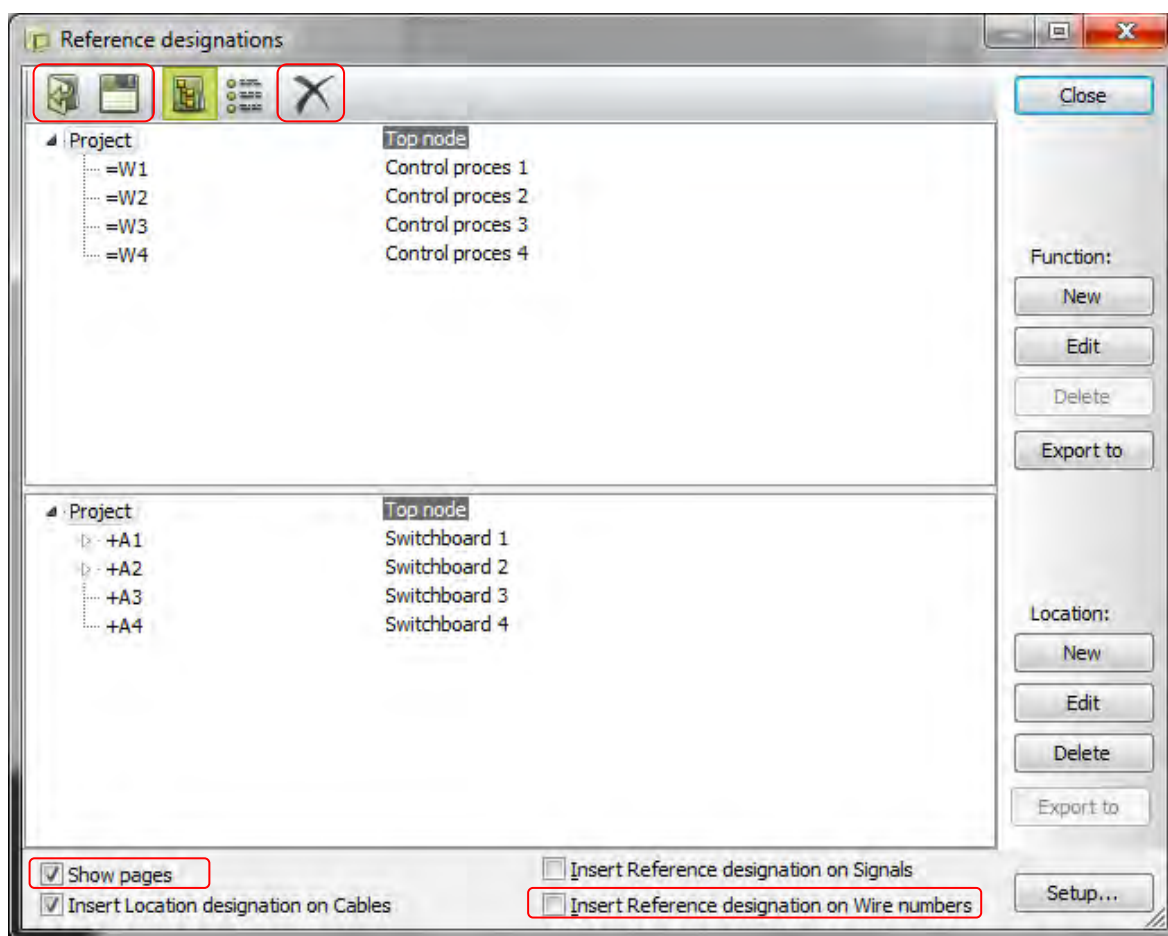
When a symbol from the pickmenu is placed in the project it will be removed from the pickmenu.



Changes in the Reference Designations dialog

The following changes have been made in order to enhance visibility for reference designation functions:

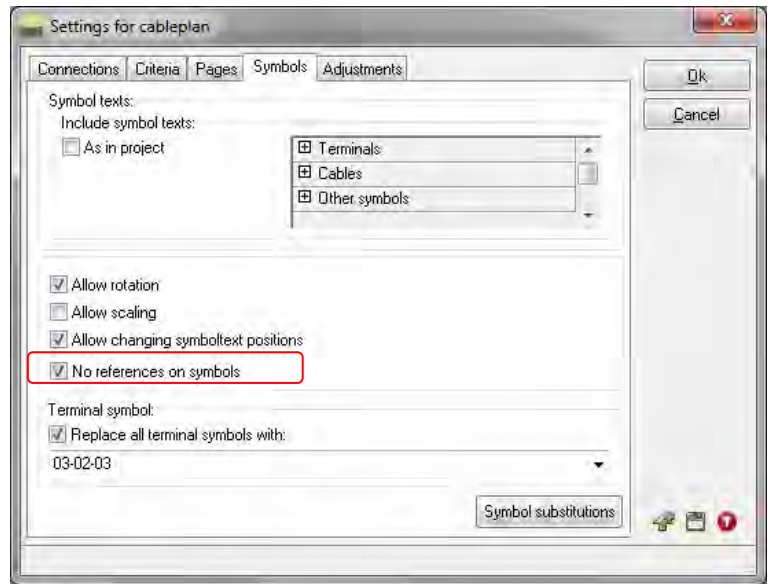
1. The button [Show used] is deleted
2. The button [Import] is deleted
3. Import and export (all) is moved from rightclick menu to visible buttons
4. The individual export from rightclick menu is moved to two visible buttons
5. Show used is now always active
6. Show pages can now be (de)selected with a checkbox below.
7. Delete unused is moved from rightclick menu to button
8. You can insert reference designations on wire numbers.



Tools

No reference on symbols on graphical plans

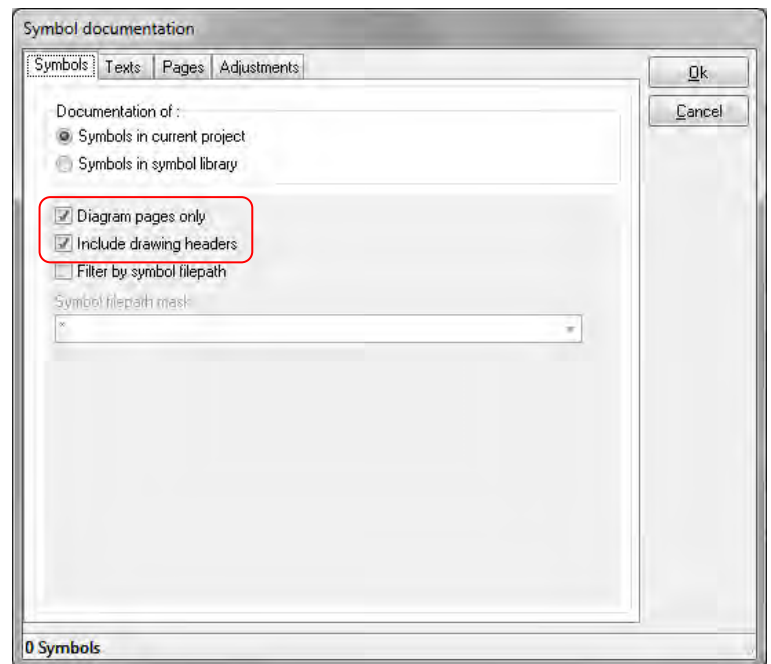
New checkbox "No references on symbols" gives you the option to deselect references on graphical plans. When deselected, the plan will show references – as before – to symbols in the diagrams. (That is, if References are selected for the diagram symbols).



New settings in Symbol documentation

New checkboxes for documentation of symbols in the current project:

- Only diagram pages: only symbols on DIA pages are included in the symbol documentation
- Includes drawing headers: The drawing headers used in the project are included in the symbol documentation.

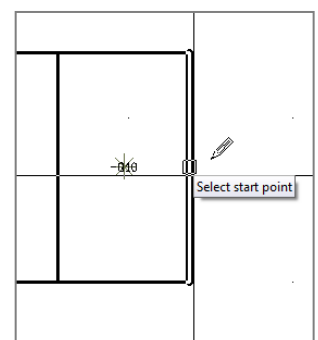


Measurement

The measurement routine can now select middle of line/object. A small square will indicate where the function "snaps".

Another change is that setting of symbol text visibility doesn't affect measurement visibility any more.

It is also possible to snap to positions outside the proposed snaps with ctrl+click...

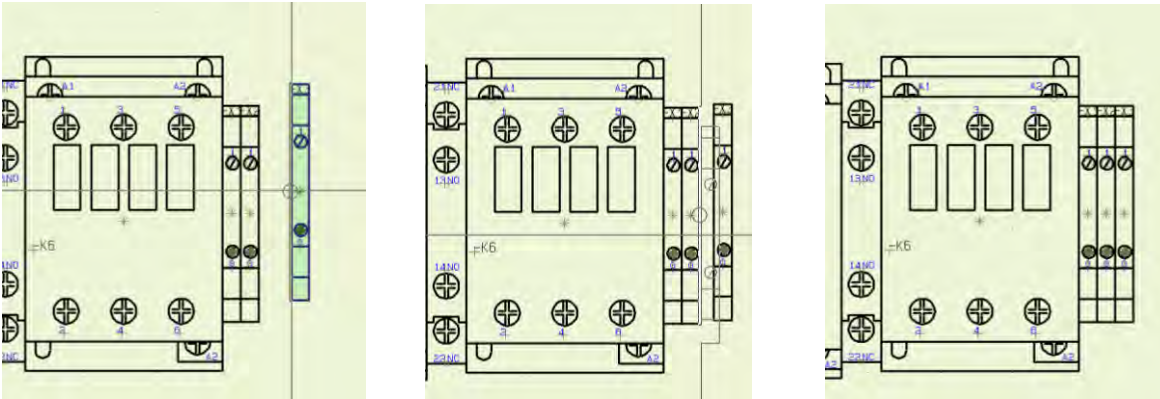


O-snap function

O-snap or Object-snap is a function that snaps to certain object points in order to have an end or middle of one object snap to end or middle of another selected object.

The O-snap object searches for end or middle in the same way as the measurement function. On the screen it is indicated with a small "o".

When mouse-clicking the two "o's" snap to each other:



1. In the first picture there is a snap to the middle of the terminal, and when you move it (with the mouse) you grab it with the "o".
2. In the second picture you point – with the first terminal on mouse-hold – to the middle of the next terminal. When the new "o" appears you release the mouse and...
3. Picture three shows the result.



The function is activated with the icon in the vertical toolbar.

O-snap is intended for easier component placement on mechanical (arrangement or layout) pages. It snaps to ends or middle of lines or symbols' line segments.

Magnetic function

A small change in how to (de)activate the "magnetic function" (which first came with version 11).



There is a new button in the vertical toolbar and a shortcut M that (de)activates the function.

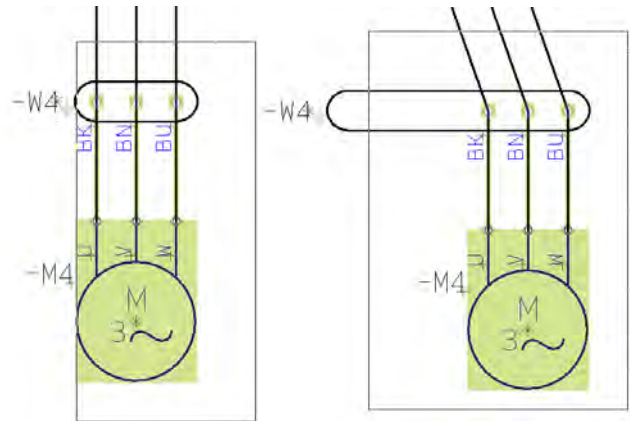
It is (still) possible to "magnetize" the internal side of a panel if you hold down the SHIFT key when Magnetize is active.

BEWARE: If you want to use the option to "magnetize" the internal side of a mounting rail the symbols (ie a panel with mounting rails or another indication of the components' physical positions) must be made according to the rules. You can read more about this in the manual.

Move area

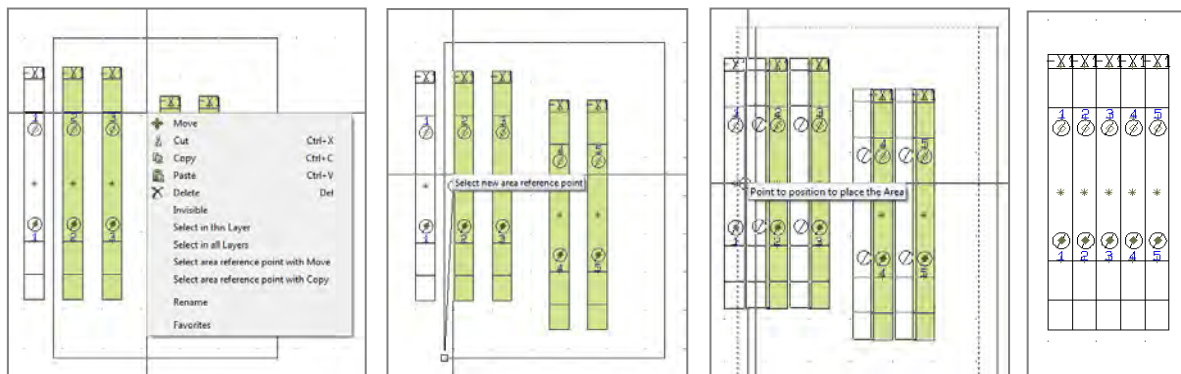
The function "Move area" will now move all cable parts without "bends". This has been a problem, when only cable conductors and NOT the cable symbol itself (*) were marked.

When you move the area you can see visible "rubber bands".



Move/copy area with new area reference point

In connection with moving or copying *areas* it has been a longtime wish that it was possible to select another reference point than the lower left corner of the selection window. The function can be used together with O-snap and is illustrated below:



1. Select the *area* that you want to move. In the rightclick menu click "Select area reference point with Move" (or copy if you want to copy the area).
2. Select a new reference point – here O-snap has selected the middle of the terminal. Release the mouse button.
3. Select the new position of the reference point. Here O-snap has selected the middle of another terminal.
4. The result of repeating the routine a couple of times can be seen here.

Jumper links

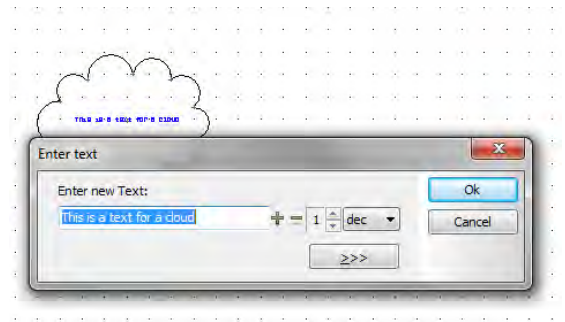
Jumper links can now be connected to all terminal points on all symbol types, except from busbars.

Insert cloud

The insert cloud routine has been enhanced:

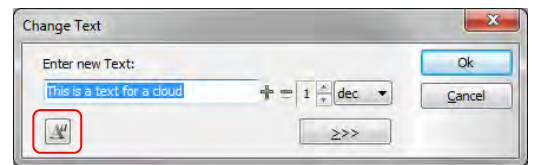
When you place the cloud you can now add text directly in a pop-up window. If you don't wish to add text, simply press Cancel or Esc.

The size of the cloud itself is very close to the size of the rectangle you when you place it.



Change text formatting

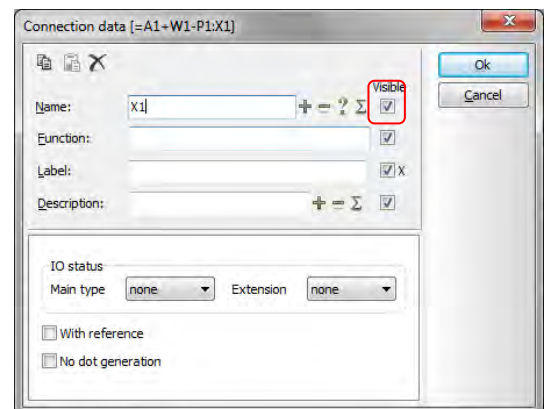
The Change Text dialog now includes a Text formatting button.



Connection data dialog

If you open the Connection data dialog, a change of name will automatically make the name visible in the project.

Connection names come with CAPITAL letters as default.

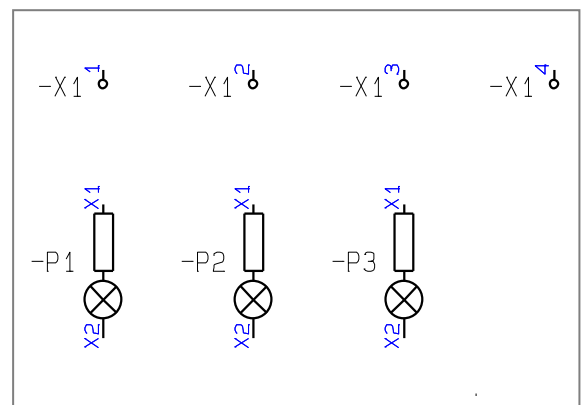


Auto naming routine

Automatic incrementation of symbol number/name and terminal row numbers has been changed to work the following way:

Terminal row connections are incremented on the last position: Type terminal row name and its first connection number [-X1:1] followed by [Ctrl and +] and the next positions are automatically incremented. This works as it has always done.

The new function is that for all other symbol types, the incrementation will be on the first position: Type first symbol name and its connections [-P1:X1,X2] followed by connection names and [Ctrl and +] and the symbol names will be incremented.



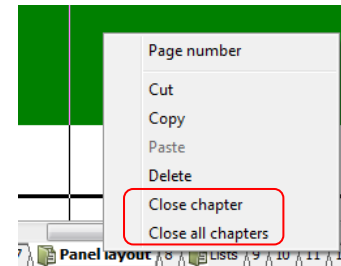
Drag 'n Draw

Drag 'n Draw of chapter dividers now opens the dialog in which you can type the chapter name.

Chapter divider rightclick menu

The following items have been added the rightclick menu for chapter dividers:

1. Close/Open chapter
2. Close/Open all chapters



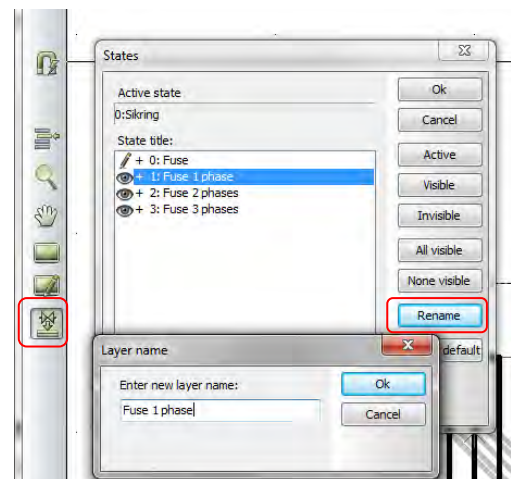
Active pagetab

The tab for the active page follows – as before – the background color of the page. The new function is that the page number turns **bold** in the same way as in e.g. Excel.

State titles can be edited / renamed

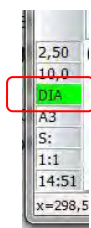
If you want to edit a symbol with states and the button “States to layers” is active, the layer dialog will show state titles.

When the symbol is saved, the changes of states' titles will be saved as well.



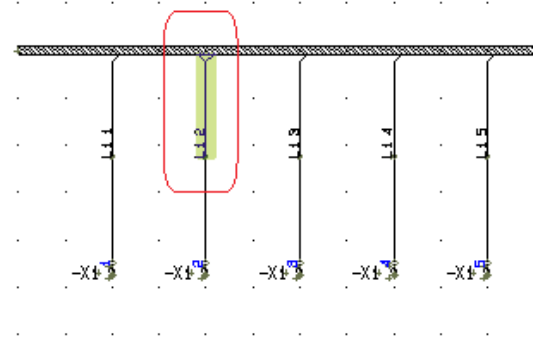
Create variable list is now automatic

Creation of variable lists is now automatic when you create or edit a subdrawing. Beware, however, that this is only valid, when DIA is blinking green.



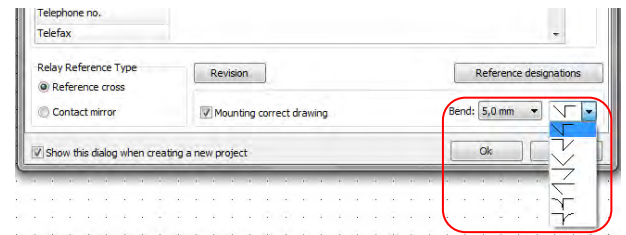
Select signal connections symbols

You can change the signal connection symbol by first selecting it and then pressing <space>. In this way you can change between the predefined types.



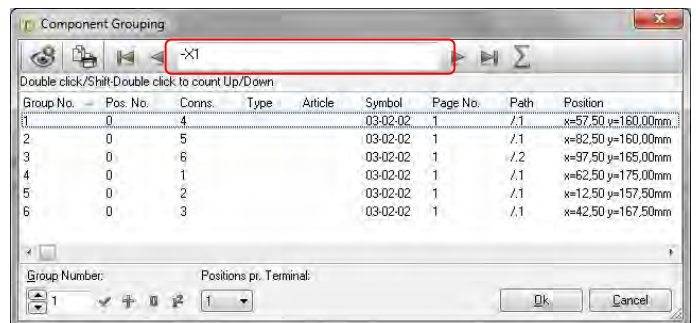
Mounting correct drawing

When selecting default bend figure, you can now choose from all seven figures.



The naming field in the Component Grouping window is enlarged

The naming field has been enlarged in order to make room for long component names.



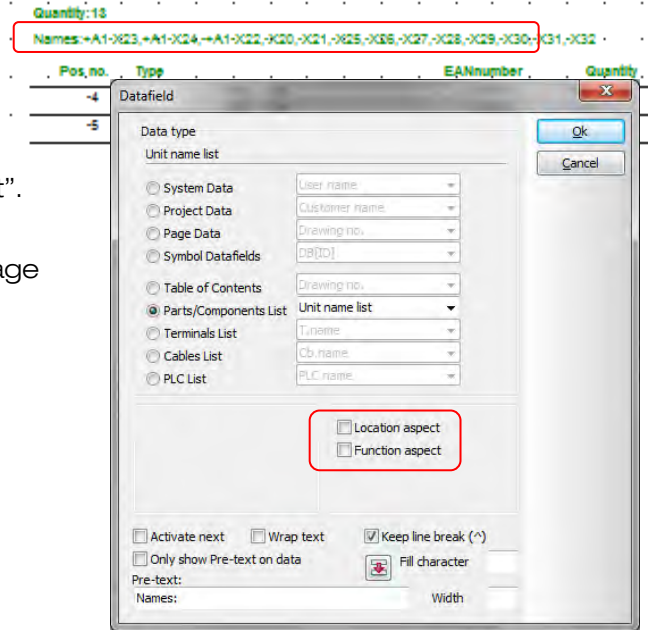
News in relation to data fields

Parts and components lists

The field: Unit name list

It is now possible to show location and/or functional aspects for the datafield "Unit name list".

This means that the full component names for components using the unit-component of this page can be seen on the unit page.

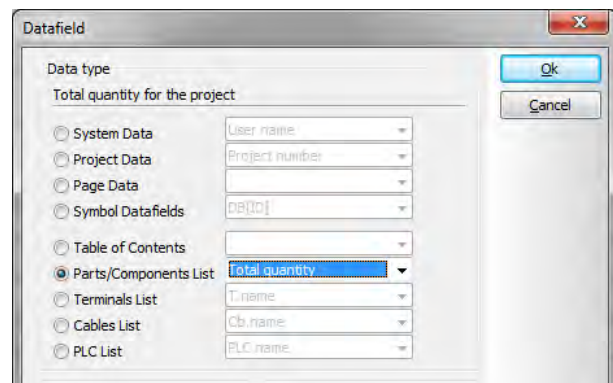


New field: Total quantity

There is a new data field – total quantity – which makes it possible to see the total quantity of an article number in the components list.

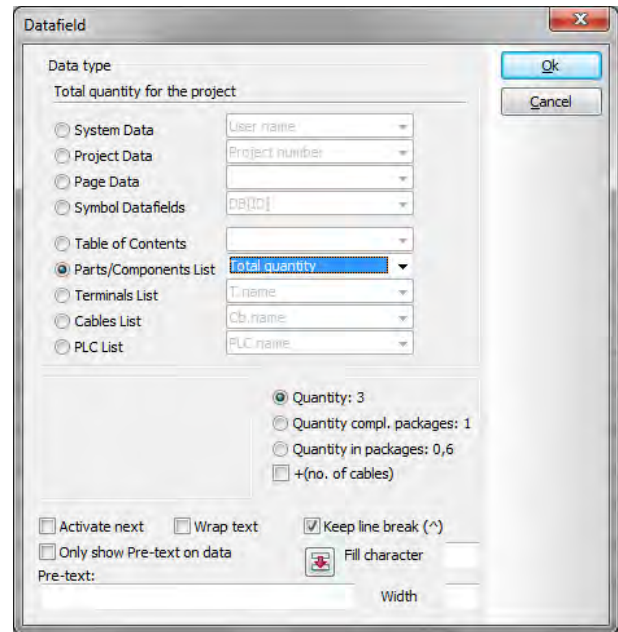
Pos.	Komponent	Antal	Samlet antal	Varenr.	Type	Fabrikat	Beskrivelse
1	-F1	1	4	pcs2250001		PCS	Motorværn type 1a
2	-F2	1	4	pcs2250001		---	Motorværn type 1a
3	-F3	1	4	pcs2250001		---	Motorværn type 1a
4	-F4	1	1	pcs2250002		---	Motorværn type 1b
5	-F5	1	4	pcs2250001		---	Motorværn type 1a
6	-K1	1	1	pcs2250106		---	Kontaktor 3POL 1NO 2NC type1
7	-K1	1	1	pcs2250107		---	Kontaktor 3POL 1NO 2NC type2

The data field "Total quantity" is added in the Parts/Components list section. It contains the same options as "Quantity" which is illustrated below.



The data field Quantity

An extraoption has been added to the Quantity data field. It makes it possible to list the number of cables in (brackets) after the quantity in parts lists.



In the **Parts list** the number of cables are shown (in brackets) adter the quantity.

Cables with a quantity of 0 are shown as - (hyphen).

Pos.	Article no.	Type	Total quantity	Quantity	Description
1	PCS212001	PCSXX1GY	12	12	Through terminal - 1 in 1 out GY
2	PCS330002	PCSCBL02	37 (4)	37 (4)	4 conductors without screen
3	PCS600001	PCSMOTOR1A	4	4	3 phase motor U,V,W 1.1 kW
4	PCS2250001	PCSMV001A	4	4	Thermal overload type 1a
5	PCS2250106	PCSCON06	5	5	Contacteur 3POL 1NO 2NC type1
6	PCS2250401	PCSPB01	4	4	push button NO
7	PCS2250411	PCSPB NC 01	4	4	push button NC

In the **Components list**, you will see a - (hyphen) if the cable quantity is 0.

Pos.	Component	Quantity	Total amount	Part no.	Type	Manufacturer	Description
5	-W1	10.75	38.25	PCS330002	PCSCBL02	PCS	4 conductors with
6	-W2	12.50	38.25	PCS330002	PCSCBL02	PCS	4 conductors with
7	-W3	15	38.25	PCS330002	PCSCBL02	PCS	4 conductors with
8	-W4	-	38.25	PCS330002	PCSCBL02	PCS	4 conductors with

Cable accessories

There are new options regarding cable accessories. This is intended for plugs, clips, connectors etc.. The options mean new rules concerning data entry in the database.

Cable connectors

Up to now, a data entry of [CONN123#2] in the ACCESSORIES data field means that you will get 2 connectors type CONN123 for every meter of the cable. And you only wanted a total of two connectors, ie one for each end. However, you still need to have other kinds of accessories to depend on the length of a cables, ie clips, bearings etc.. This is the reason for this new data entry syntax: If you type [CONN123##2] in the data field, the parts list will list 2 pcs of the connector.

Jumper links in the parts list


When you place a jumper link, the quantity is set to 1 (one).

If you place a jumper link with component data, it is now possible to have it listed in the parts list with the correct quantity, if data regarding the amount of possible connections are entered in the PINDATA data field as shown in the figure. If PINDATA is empty the quantity will be 1.


If the jumper links 2 terminals (1 connection) PINDATA must be #1. If the jumper links 3 terminals (2 connections) PINDATA must be #2.

It is also possible to enter data concerning the type of jumpers that are manually cut off: You enter the amount of possible connections in the UNITPRPACK datafield. In that way the parts list will show how many “rods” you have used. (But it cannot take excluded “teeth” into account).

Jumper links:
 When placing a JUMPER LINK quantity is default set as 1 (one)



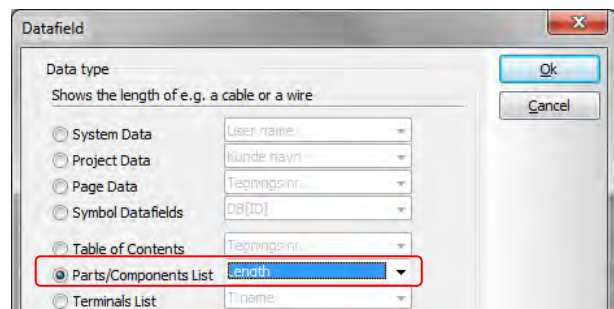
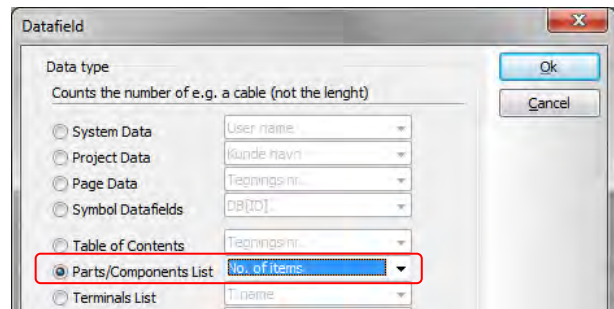
Database field
PINDATA = empty or #1



Database field
PINDATA = #2

New data fields: Number of items and Length

The data fields Number of items and Length have been added to the Parts/Components list section.



Extract from a parts list:

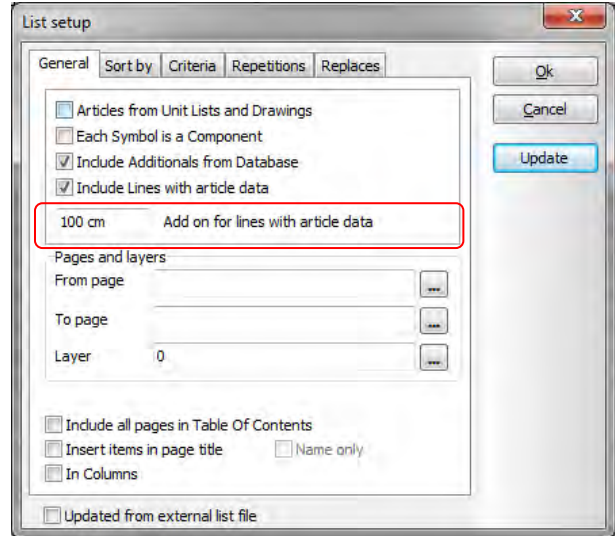
Pos.	Article no.	Type	Description	Quantity	Length	No. of items
1	PCS212001	PCSXX1Gy	Through terminal - 1 in 1 out GY	12		
2	PCS212002	PCSXX1BU	Through terminal - 1 in 1 out BU	17		
3	PCS330002	PCSCBL02	4 conductors without screen	53	53	4
4	PCS600004	PCSMOTORIA	3 phase motor U,V,W 4.4 kW	3		

Add-on for lines with name and article data

A new data field for add-ons to be included for lines in parts and components list.

The add-on will be added to “Total quantity”, “Quantity” and “Length” for LINES. That means that the data fields is mainly relevant for lines with cable data on GRP-pages.

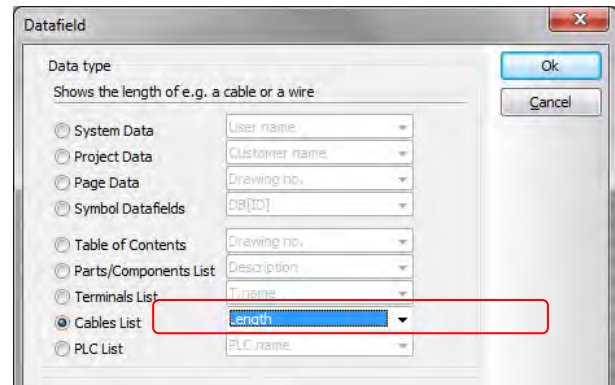
Remember to put units on the add-on. Data is remembered per list and is NOT a global project setting.



Cable list

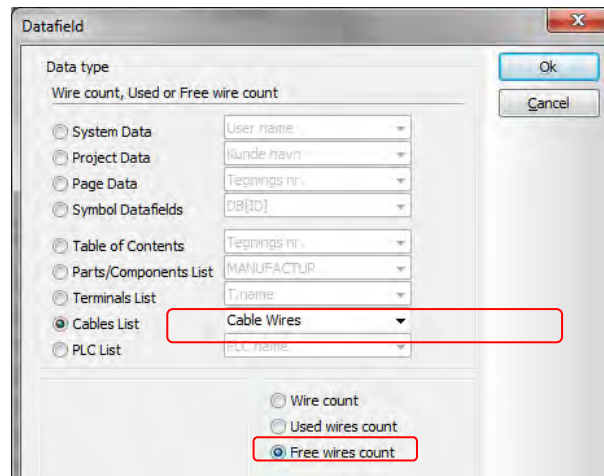
New data field: Length

A new data field LENGTH has been added in the cables list section.



Extra option on Cable Wires

This data field – cable wires – has now the option to show free available wires in a cable. (The other options were there earlier, too).



Here you can see Length and Cable wires – used/free in a cable list:

Cable	To	Type	Quantity	Length	Wires used / available
-W1 BK	-M1 U	1/1	PCSCBL02	13	13 10 3 7
-W1 BN	-M1 V	1/1	PCSCBL02	13	13 10 3 7
-W1 BU	-M1 W	1/1	PCSCBL02	13	13 10 3 7
-W2 BK	-M2 U	1/1	PCSCBL02	17	17 8 3 5
-W2 BN	-M2 V	1/1	PCSCBL02	17	17 8 3 5
-W2 BU	-M2 W	1/2	PCSCBL02	17	17 8 3 5

Terminal list

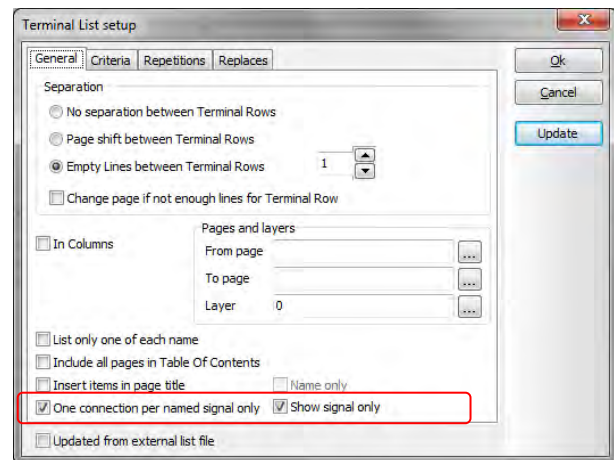
A new algorithm has been made for terminal lists, makin it more complete-

In connection to that two new check boxes has been made:

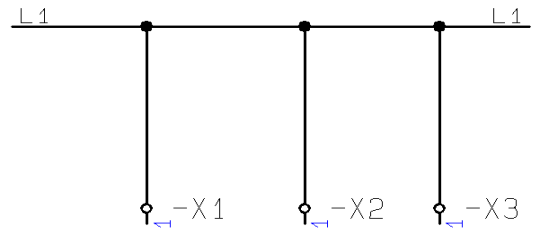
- One connection per named signal
 - o Show signal only

The function is identical for "dots" drawing and mounting correct drawing.

See next page for further explanation.



For illustration we will use this small circuit:



If "One connection per named signal only" is OFF, all connections will be listed in the terminal list:

Pos.	Cable (ext)	Connected to (ext)	Terminal	Connected to (int)
1			-X1:1	-X2:1
2				
3			-X2:1	-X3:1
4			-X2:1	-X1:1
5				
6			-X3:1	-X2:1
7				

However, this is not what is normally wished for. If all three terminals are connected to L1, then you will normally want to show this connection instead of the connections between the terminals. Typically, the terminals aren't connected directly but only through a busbar or the like.

If "One connection per named signal only" is ON, and "Show signal only" is ON, the terminal list will show that all three terminals are connected to L1. This is the default setting.

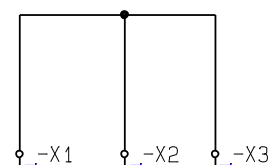
Pos.	Cable (ext)	Connected to (ext)	Terminal	Connected to (int)
1			-X1:1	L1
2				
3			-X2:1	L1
4				
5			-X3:1	L1
6				

You can also set "Show signal only" OFF. This will the terminal list show the same number of rows (that is the same amount of connections) but it will show the first connection to the terminal instead of the signal name. This list is incomplete.

This setting ("One connection per named signal only" is ON and "Show signal only" is OFF) behaves very much like the "old" terminal list.

Pos.	Cable (ext)	Connected to (ext)	Terminal	Connected to (int)
1			-X1:1	-X2:1
2				
3			-X2:1	-X3:1
4				
5			-X3:1	-X2:1
6				

The check box is called "One connection per named signal only" and this is because it only works when the connection has a signal name (here L1). That means that for this circuit ALL connections will be shown in the terminal list (as in the first example).

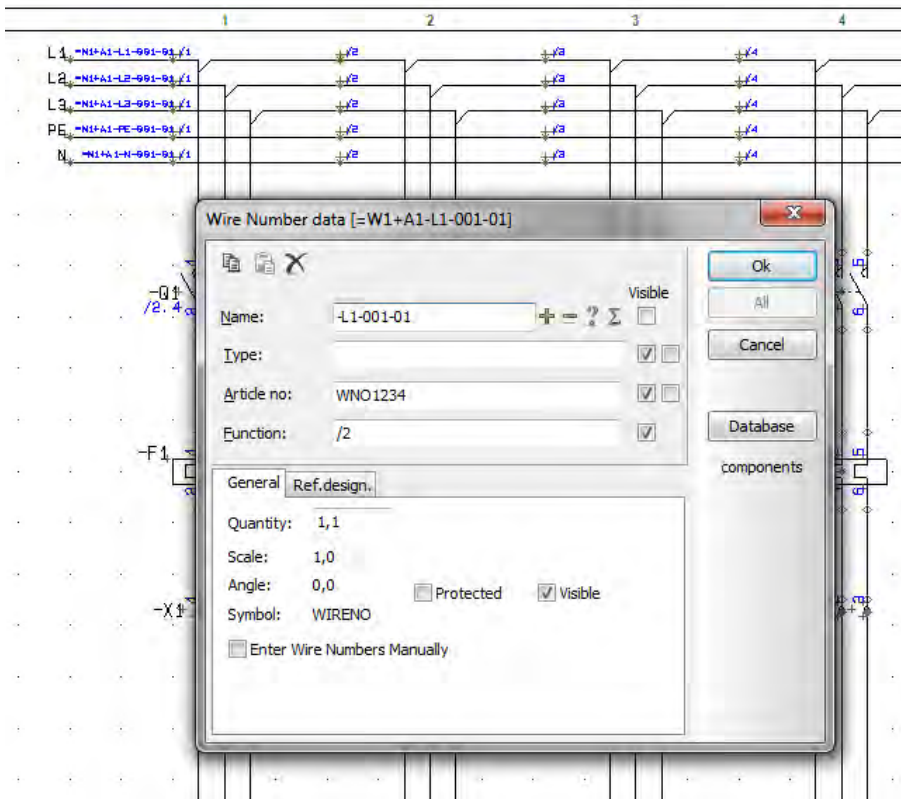


Extra function on wire numbers

It is now possible to enter TYPE, ARTICLE NO. FUNCTION and QUANTITY in a wire number symbol. The example below is numbered with the "potential numbering" method, meaning that all conductors in the same net = same potential = same number.

Because it is possible to enter data in all component data fields, you can select visibility according to your needs.

In this example it was a wish that each potential had a number and that data concerning article data and length could be identified.



Object lister

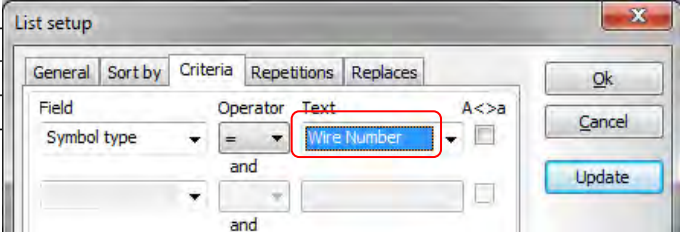
The data fields have been added in the Object lister.

Name	Symbol	Page No.	Position	Article	Type	Function	Quantity
=W1+A1-L1-001-01	WIRENO	1	x=60,00 y=260,00 z=0,00mm	WNO1234		/1	1,5
=W1+A1-L1-001-01	WIRENO	1	x=92,50 y=260,00 z=0,00mm	WNO1234		/2	1,1
=W1+A1-L1-001-01	WIRENO	1	x=132,50 y=260,00 z=0,00mm	WNO1234		/3	1,1
=W1+A1-L1-001-01	WIRENO	1	x=170,00 y=260,00 z=0,00mm	WNO1234		/4	1,1
=W1+A1-L1-001-01	WIRENO	1	x=210,00 y=260,00 z=0,00mm	WNO1234		/5	1,1
=W1+A1-L1-001-01	WIRENO	1	x=255,00 y=260,00 z=0,00mm	WNO1234		/6	1,1
=W1+A1-L2-001-01	WIRENO	1	x=60,00 y=255,00 z=0,00mm	WNO1234		/1	1,6
=W1+A1-L2-001-01	WIRENO	1	x=92,50 y=255,00 z=0,00mm	WNO1234		/2	1,2
=W1+A1-L2-001-01	WIRENO	1	x=132,50 y=255,00 z=0,00mm	WNO1234		/3	1,2

Wire number list

In order to print the wire numbers and data in a list you can make a component list with a couple of special settings:

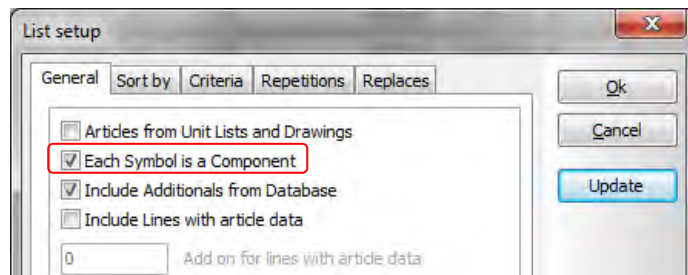
Pos.	Quantity	Wire number	Article no.	Position	Manufacturer	Descrip
1	1.50	=W1+A1 -L1-001-01 /1	WNO1234	/1.1		
2	1.10	=W1+A1 -L1-001-01 /2	WNO1234	/1.2		
3	1.10	=W1+A1 -L1-001-01 /3	WNO1234	/1.3		
4	1.10	=W1+A1 -L1-001-01 /4				
5	1.10	=W1+A1 -L1-001-01 /5				
6	1.10	=W1+A1 -L1-001-01 /6				
7	1.60	=W1+A1 -L2-001-01 /1				
8	1.20	=W1+A1 -L2-001-01 /2				



The list contains these data fields: Quantity, +=design, Name, Article No., Function, Position.

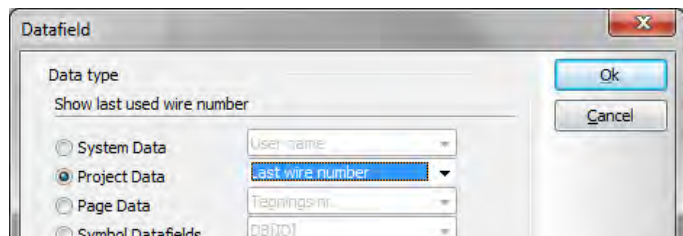
In order to make the list, you must go to the Criteria tab to select Symbol type = Wire Number. (Note: Wire number can ONLY be listed in components lists, as they are deselected in all other list types.)

You must also select "Each symbol is a component" in the General tab to make this list function.



Last wire number

There is a new (project) data field that contains the last used wire number in the project.



Connection list files

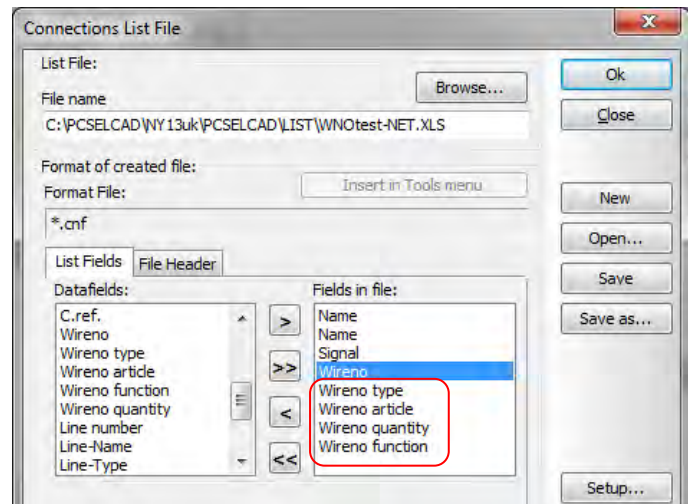
Line article data and connection lists

There are new data fields and check of consistency of data in the same net when you make connection lists to file. If there are inconsistencies they will be listed at relevant places in the file.

The new data fields are: Quantity, Jumper, Cable name and Cable conduction name.

New data fields for Connections List file

Data fields for Wireno's type, article no, quantity and function have been added to the Connections List file.

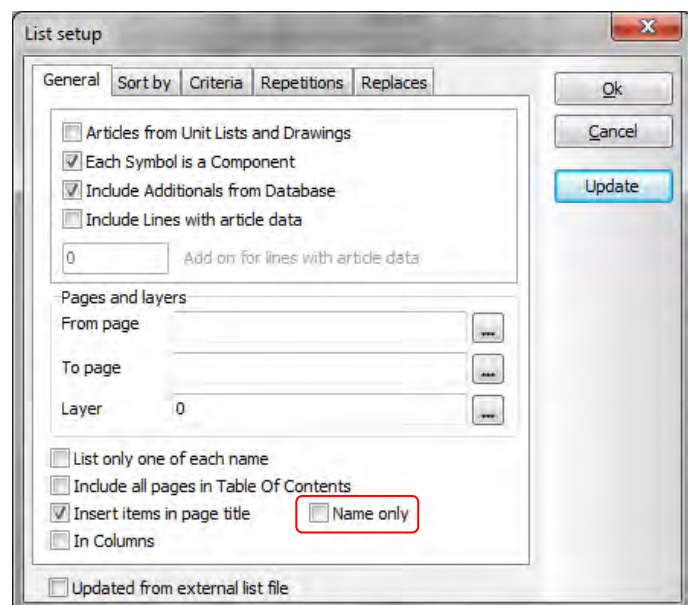


Generally about List files

When you make a "xx list file" the proposed file name will now include an extension, e.g. the parts list will have the (proposed) extension "-PRT", the Connection list file (above) will be "-NET" before the *.xls or *.csv extension.

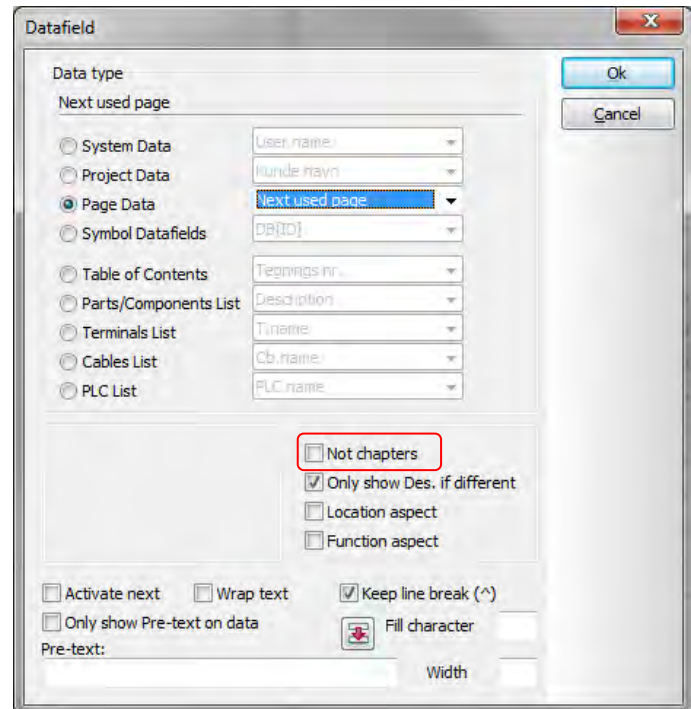
List setting, General

If you insert component names in the page title, you can now choose to show ONLY the name. Earlier, connection names (or cable conductors) would be included, too.



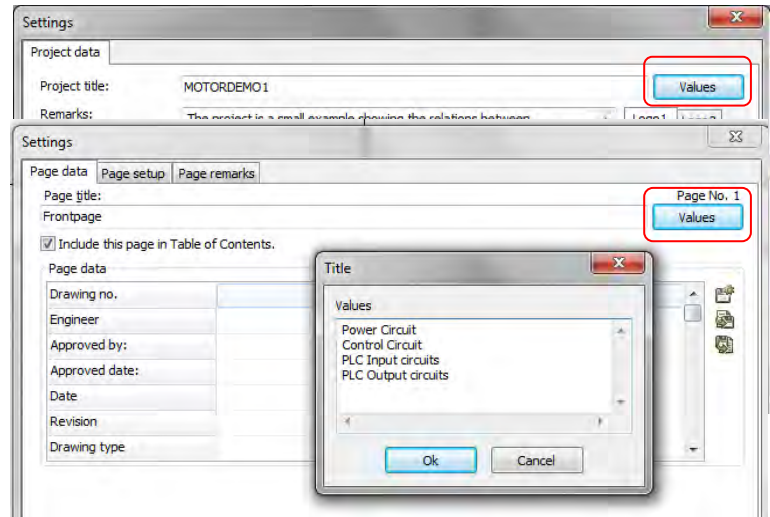
Data fields Next / Previous used page

An option has been added to exclude next/previous used page when the page number is a chapter name.



Data list can now be added for Project titles and page titles

In order to make date entry easier you can now make a data list of preferred values for Project and Page titles. You can still enter any other title manually, too.

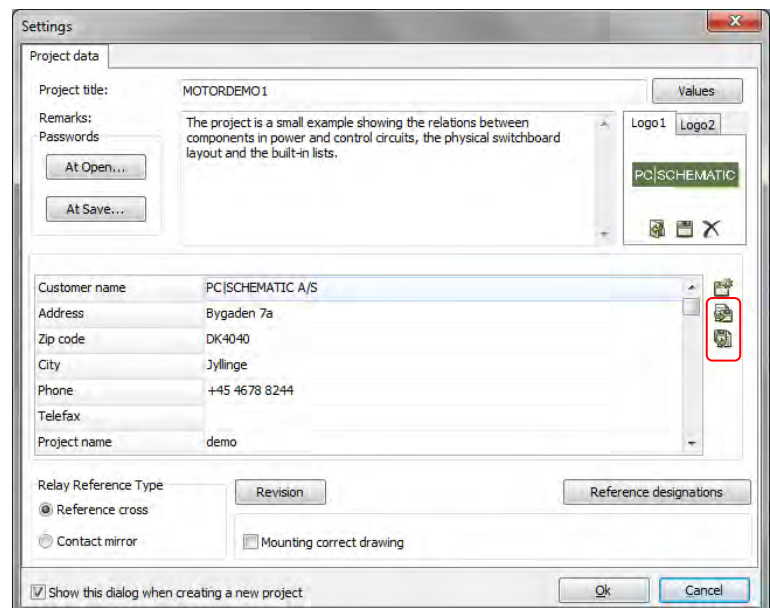


Export and import of project and page data

Project- and page data can be saved in Excel or CSV-format, and can be imported from the same two formats.

During import existing datafields are updated and new fields will be created.

Datafields that don't exist in the import file will stay as they are in the project file.



Bookmarks in symbol folders §

Bookmarks in a symbol folder are saved in the BookMark.ini file in each folder. The file can now handle several languages at once. This means that the program will automatically search for bookmarks in the language you have selected in Settings\System.

Below you see the file bookmark.ini from the EN60617 folde which contains a Danish section and an English section.

Country codes are listed below:

UK: English (default)

DE: German

CZ: Czech

PL: Polish

SE: Swedish

CH: Chinese

RU: Russian

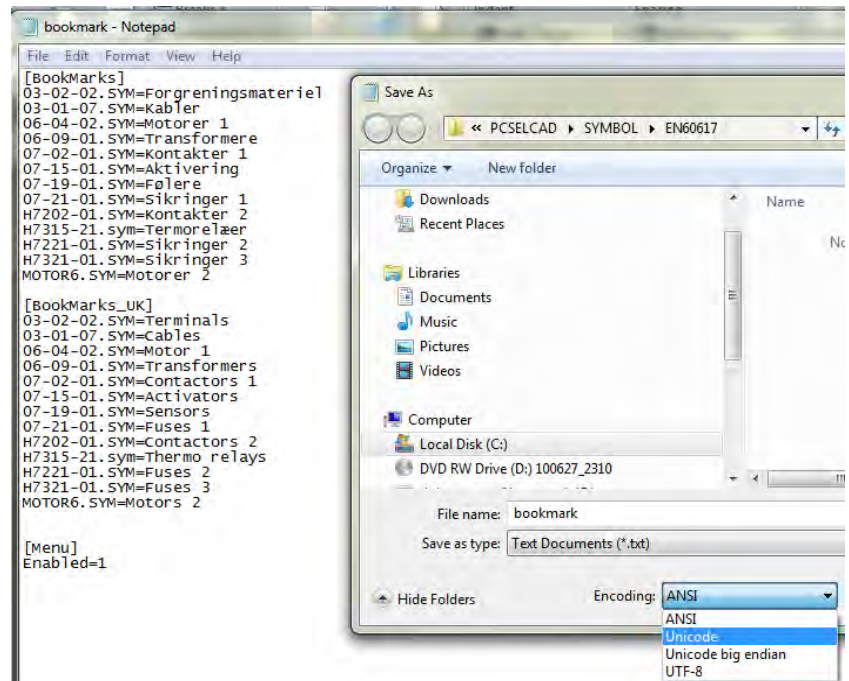
DK: Danish

The program searches for a file section called [Bookmark_xx] where xx is the country code seen above. If there is no xx it will select the bookmark without country code.

The PCSCAD.INI file now contains a section called [BookMarks].

We will not overwrite your current settings and bookmarks, so if you want to include this feature, you must do it yourself.

If you want to be able to show for instance chinese letters, your bookmark.ini must be saved as Unicode.



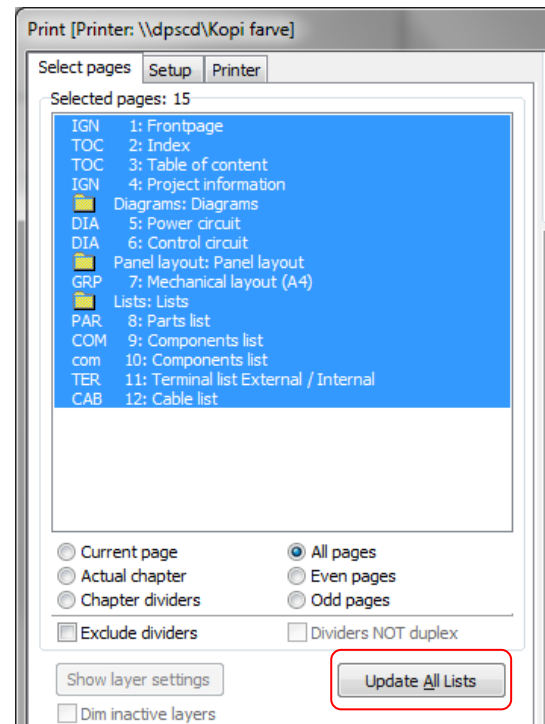
Printing functions

The Printer dialog

The button "Update all lists" has been included in the printer dialog.

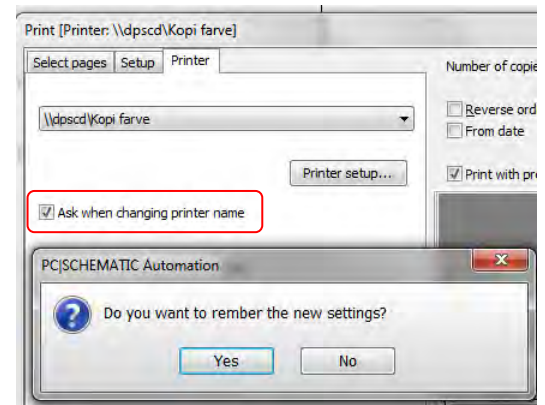
The button is only active if you have selected one or more list pages.

There is no function that checks whether the lists are updated or not.



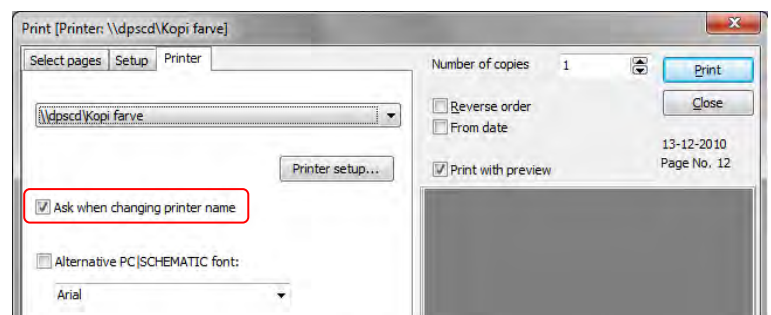
Select printer and printer setup

When changing printer or printer setup you will now be asked if you want to keep the new settings.



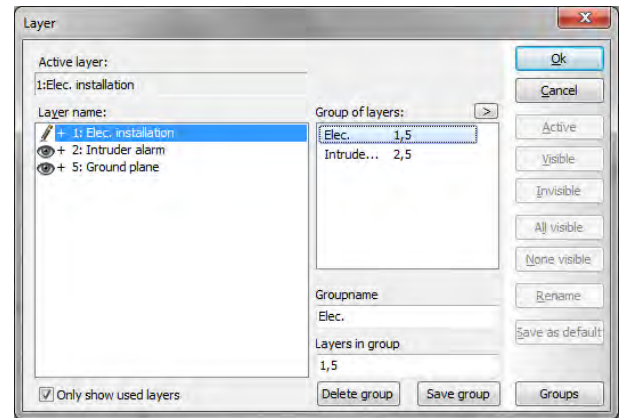
Check for "Ask when changing printer name" has been added

A checkbox has been added, that makes it possible for you NOT to see the checkbox illustrated above.



Printing layer groups

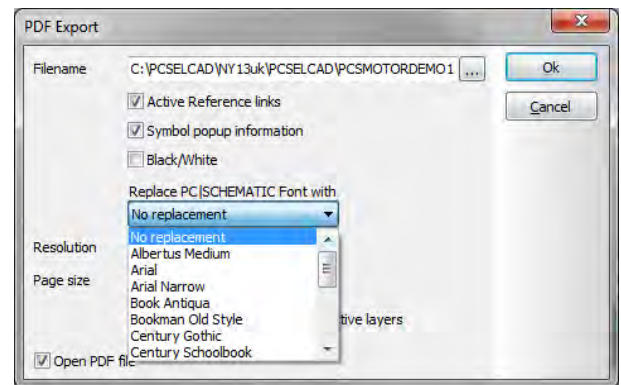
Grouping of layers is active in the printing dialog, too.



Possible to choose between several alternative PCSHEMATIC fonts

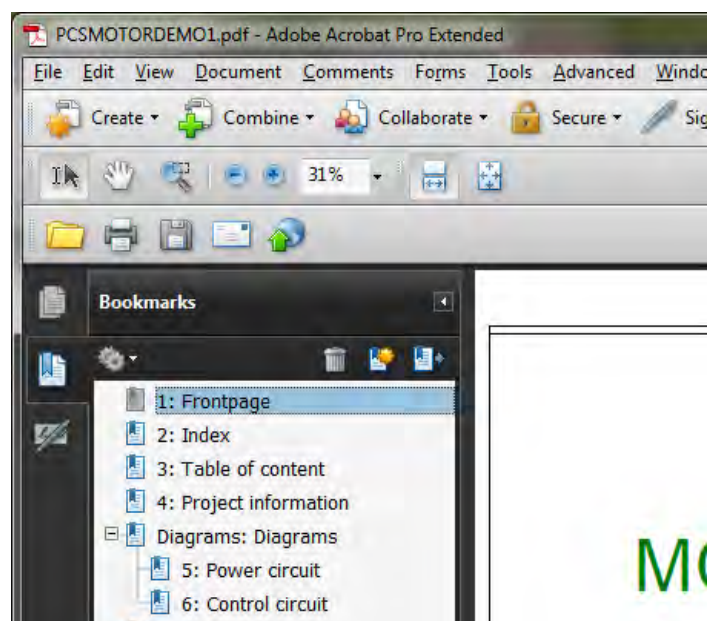
It is now possible to choose between several alternative fonts for the PCSHEMATIC font. You have the choice in the dialogs for Printing and PDF-export.

You can choose between all fonts accessible in Aumation.



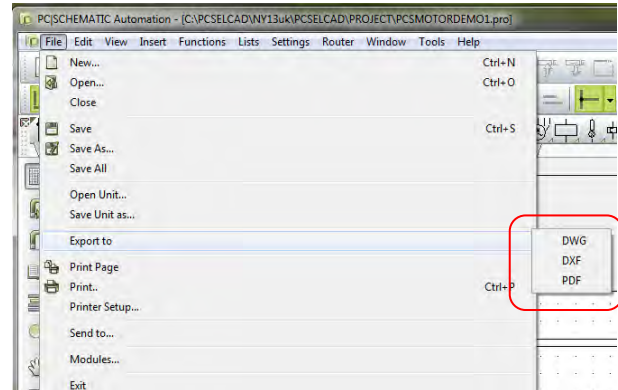
Export to PDF

Page number is now included in the PDF Table of Contents: Page number : Page title.



DWG and DXF have been added to Files|Export

The possibilities to export to other file formats have been united in Files|Export.



DWG import

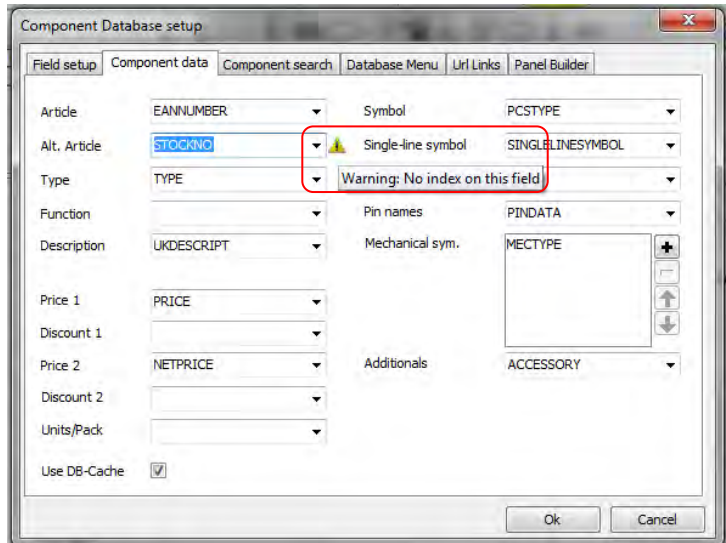
It is now possible to import dwg-files from AutoCAD 2010.

Extra database functions

Index control of data fields

In order to ensure a reasonable search time for database components, we recommend that you only index the datafields for article no. and alternative article no..

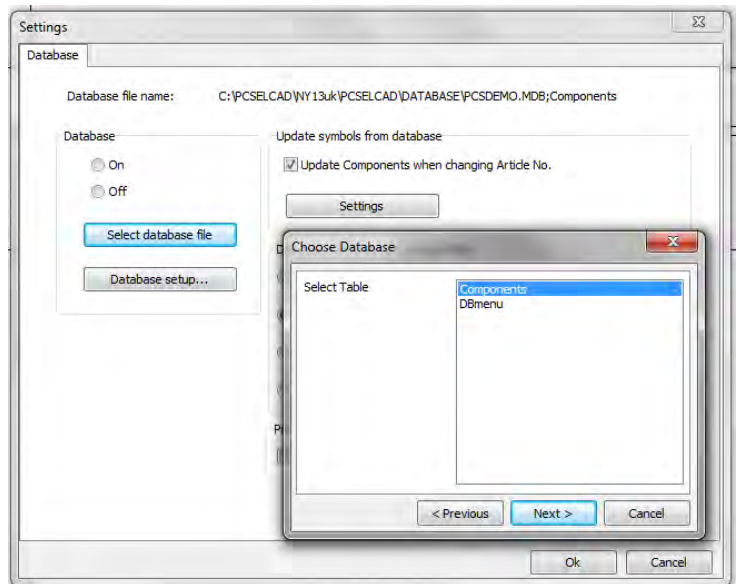
The program will now check that the data fields are actually indexed and a warning will be shown if this is not the case.



Selecting a database file

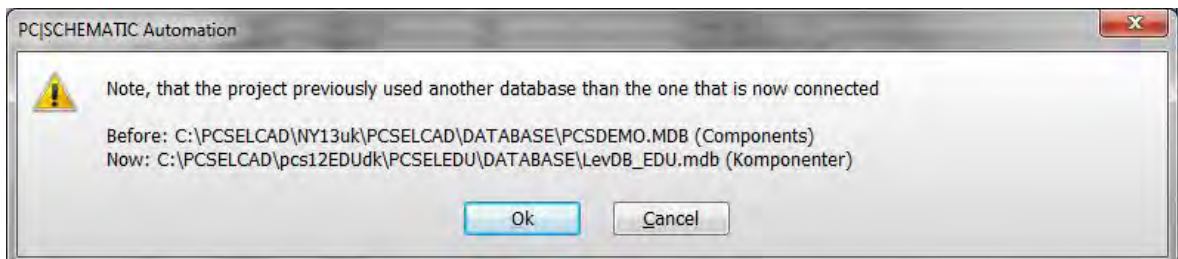
It is easier to select another database:

1. Click "Select database file"
2. The program searches the database folder
3. The program proposes the table "Components" (or what you selected the last time)
4. Finish by clicking "Next"



Saving the last used database in the project

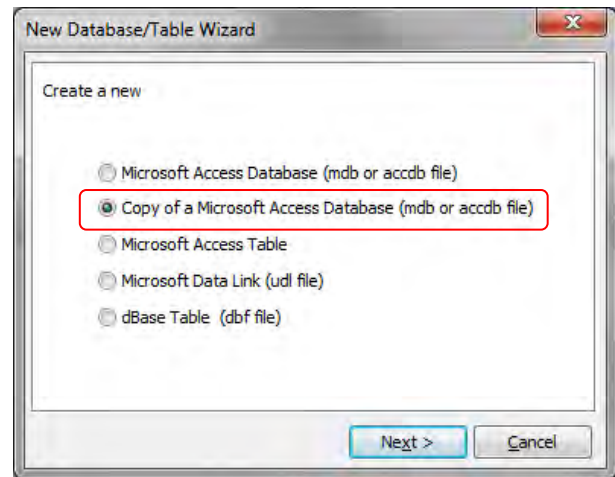
The project contains info on the last used database and the settings for that database, at the time when the project was saved. When you update a list, you will then get a warning in case your database settings have changed.



Easier to make a new database

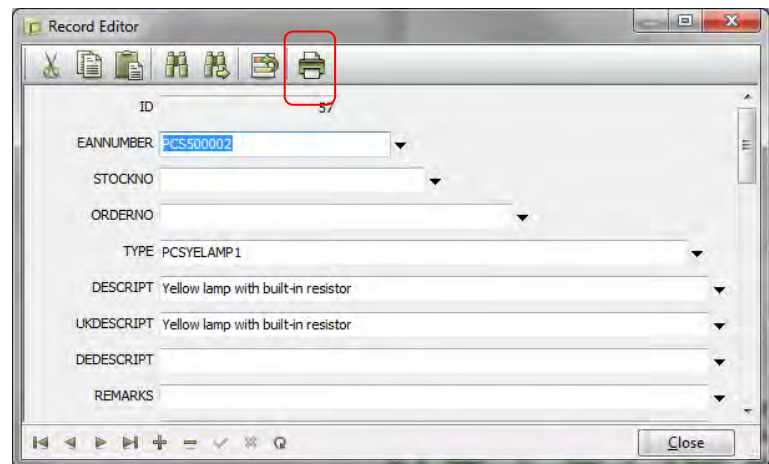
In the database program you can now make a copy of an existing database.

Especially for you who want to make your own new database, we have made an empty database of "our" database – with all datafields with their right data properties – that can be downloaded from our homepage www.pcschematic.dk.



Print record

It is now possible to print a record directly from the database Record window. You enter the Record window from the database dialog in Automation or in the database program itself.



Automatic reestablishment of connection with a MSSQL-server

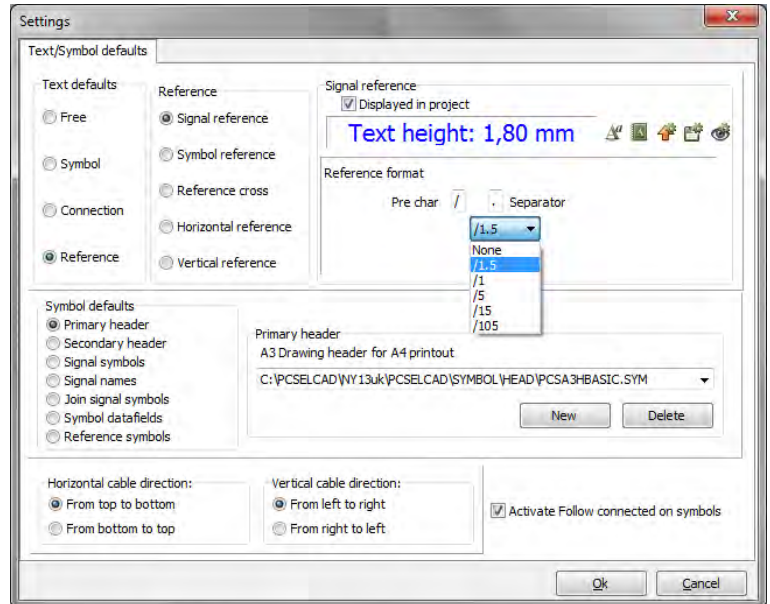
Connection to a database located on a MSSQL-server is now reestablished automatically in case it is lost in an Automation session.

Cross references §

When setting you format for cross references the following extra options have been made:

- Sign indicating "page". Default: /
- Sign indicating "current path". Default: .

Settings must be made for Signal reference, Symbol reference and Reference cross. All settings are found in Text/Symbol default settings.

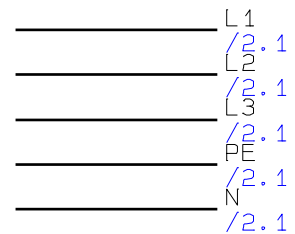


Signal reference

When there is a reference to other pages in the current project it will look this way:

/2 = page 2

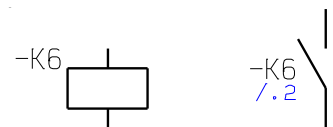
.1 = current path 1



Symbol reference

Reference between symbols, here in a contactor:

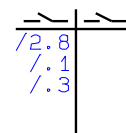
/2 means reference to SAME page, current path 2.



Reference cross

Reference between symbols in a contactor: connected contacts – NO and NC – in two columns.

There is reference to PAGE 2, current path 8 and this page's current paths 1 and 3.



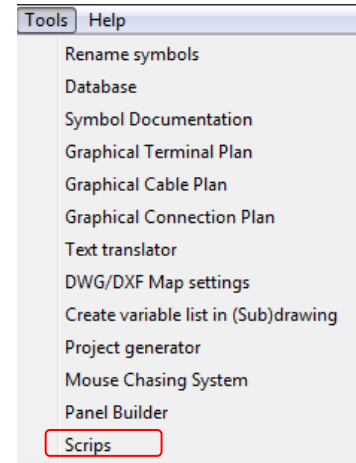
Scripting Engine

Automation now contains a Scripting Engine that can perform scripts. This is a logical extension of the OLE-interface.

It is a more user-friendly interface if you want to make your own functions, and the advantage is that all objects are already known, as the scripts are initiated directly from the program (in the same way, as the VBA-interface is used in MS Excel).

A script is a text file with a PAS or BAS extension dependent of the file format (Pascal or Basic).

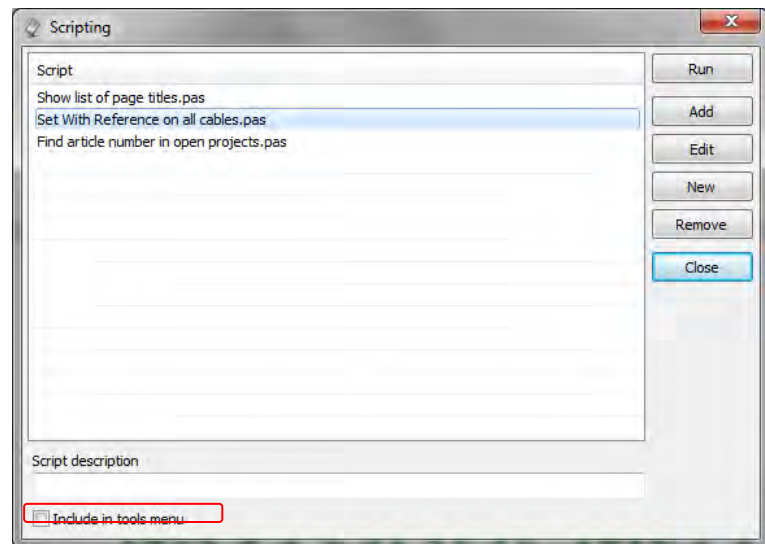
When running a script you will get a warning, as it is very easy to damage one's files, and there is **no** undo after running a script.



Select Scripts in the Tools menu to open the dialog.

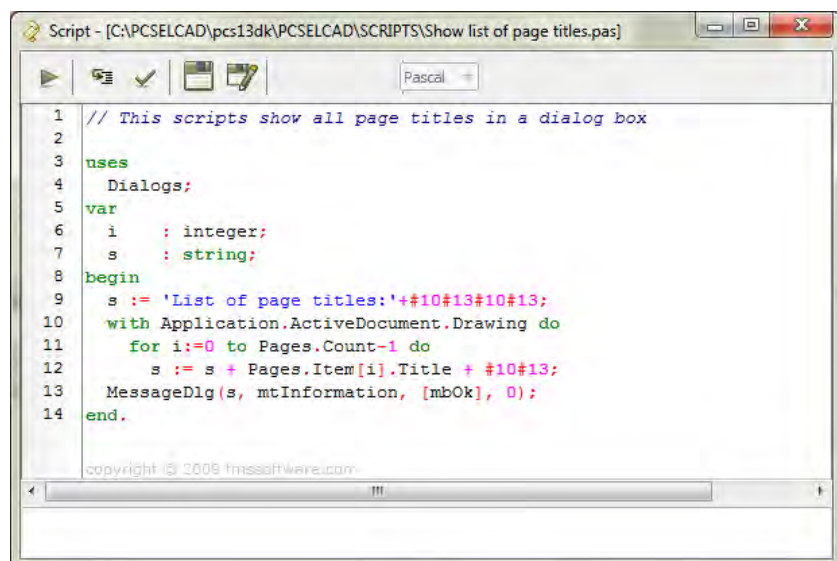
In the dialog you can select to Run, Add, Edit or delete a script, or to make your own. We have included a couple of examples in the /PCSELCAD/SCRIPTS folder.

You can also select to include a script in the Tools menu.



Possibilities in the script window

If you press **Edit** you can enter one of the existing scripts to see what they look like and contain. You can choose to add more functions, check, run it step by step and save it, when you are finished.



Working in the text editor

- There is "code completion" meaning that the program tries to finish your sentences as you write. That function allows you to write command, and let the program help you to finish it. You select a function with Ctrl+Space.
- "Syntax colouring" meaning that certain words and types are predefined to be showed in certain colors, ie text strings are shown in Blue. It makes it easier to get an overview of the code.
- Undo

Possibilities for scripts

- The programming languages Basic and Object Pascal are supported. Object Pascal follows the Delphi7 standar. Basic is very much like VB.NET (that is also very much like VisualBasic and VBA from e.g. MSWord.)
- As the script are called from Automation, the script already knows all object, without you doing anything extra (as is necessary if you use the OLE-interface from e.g. VBA in MSWord.) Here you just have to write "Application.ActiveDocument" to get access to the active document.
- You can find a lot of extra code libraries to Pascal, among them SysUtils, Dialogs, Forms, Menus, StdCtrls, Classes, that contain code to show dialogs, forms etc..

BEWARE: Scripts and OLE is not a part of our normal telephone support. The functions mentioned in this section are alone an offer to those of our customers that wish to make their own functions.

On the other hand we would refer to our user forum, where we will post various examples.

.... However, we also read questions on the forum, but what is said her, is that we cannot assist and help you as quickly with these functions as wee can with other support questions.

Functions for installation drawing

There were several new functions for installations drawing in ver 12, and a couple of other functions for this purpose has been added in ver. 13.

Grouping of layers

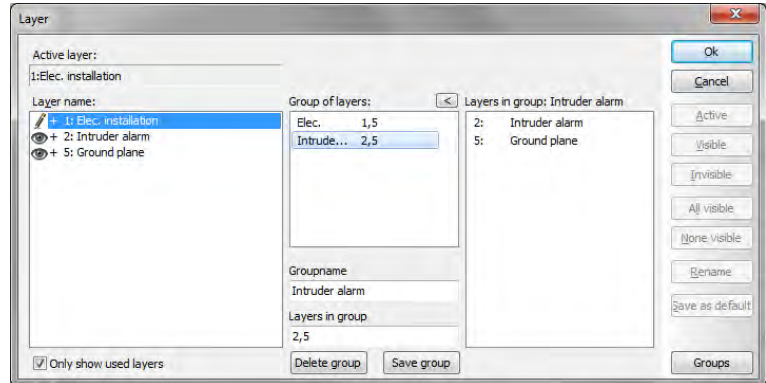
In the Layer dialog you can group a number of layers and assign them to a certain group name, here ELEC. and INTRUDER.

The groups are made this way:

1. Write a group name
2. Add layers in group – layers separated by a “,”
3. Save group

You can make as many groups as you like.

The group can be used later for prints and lists.



Extension of the function Grouping of layers

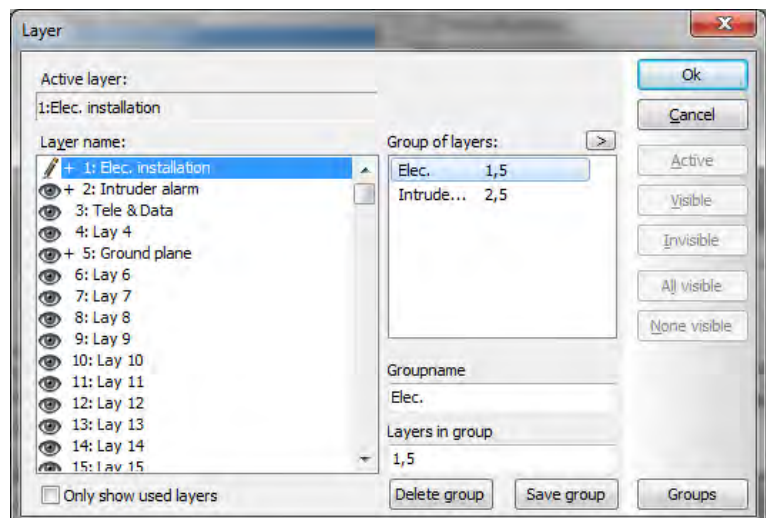
When you type "Layers in group" the first layer will be set as the active layer. The rest of layers will be visible only.

That means that in the ELEC group, layer 1 is active, layer 5 is visible.

You can press </> to see names for the layers in all groups.

Print Group of layers

Grouping of layers is also active in the printing dialog that means that you can select a layer to print.



Mostly for administrators

Version 13 and Windows 2000

Version 13 is the last Automation version to be tested on the Windows 2000 platform.

Better support for Terminal server/Citrix

Automation can now generate and update workstations on terminal servers automatically when the program starts. That means that it is no longer necessary for the administrator to manually generate workstations for the individual users on the terminal servers.

Sådan fungerer automatisk generering/opdatering af arbejdsstation

Der er automatisk generering/opdatering af arbejdsstation (Homedir) ved følgende installationstyper:

- hvis programmet kører via Remote Desktop Services (Citrix og Terminalserver)
- hvis Automation er installeret under Programfiles
- hvis Automation startes fra en lokalt installeret flerbrugerversion.

If you have a SuperPro net license / floating license...

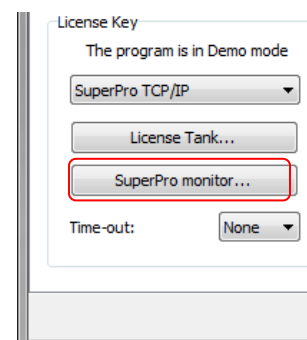
Then you must check that your SuperPro net license server software is min. ver 7.4.0 or Automation cannot find the licens server.

There has been an update of the license type, making it possible to run Remote Desktop on Windows XP, Windows Vista and Windows 7, and for this you need to run min. this version of the licens software.

The updated software for the SuperPor netlicense server can be found on the installation cd ver 12 or in the section Downloads\Other\Drivers etc. on our homepage.

Check your net license server version

In Settings\System your lower left corner contains settings of your license key. Press the SuperPro Monitor button to see your license server data.



New icons and colors

New icons have been made, some have been deleted, and everything is now in green colors, also help objects.