



AUTOMATION

DESCRIPTION OF NEW FUNCTIONS

Version 17.0

This document describes new functions in Automation version 17. Description of Component Wizard, Software License, PCISCHEMATIC Panelrouter and PCISCHEMATIC Service are in separate booklets.
Last revision: February 2015

Contents

1	The Settings menu	5
1.1	Delete unused data fields in Project data and Page data	5
1.2	Pointer/Screen – Reference points	5
1.3	Database settings, changes and new positions	5
1.4	The Basic settings menu	5
1.4.1	Colors.....	5
1.4.2	System colors.....	6
1.4.3	Custom colors.....	6
1.4.4	Using Basic colors, Custom colors or Other colors.....	6
2	Readability.....	7
2.1	Reference points and their visibility.....	7
2.1.1	Ref.points in the toolbar.....	7
2.1.2	Shortcut to view Reference points.....	7
2.1.3	Thin lines on mechanical pages	8
2.1.4	Grid dots.....	8
2.1.5	Marking of terminal's external side.....	8
3	Component wizard	9
3.1	Component kind	9
3.2	Create jumper	10
3.3	Create wire.....	10
3.4	Create wire tray.....	10
3.5	Create cable	11
3.5.1	Cable accessories	11
3.6	You can enter data/edit all selected data fields	12
3.7	You can use the Symbol generator to create diagram symbols.....	12
3.8	Access to SQL in multi-edit in database function	12
4	The Database.....	13
4.1	New data fields.....	13
4.2	New syntaxes for typing in the database.....	13
4.3	Database settings – Accessories in separate tab.....	13
4.4	Database settings – other fields in the Component Wizard.....	14
4.5	Search components in the database.....	14
4.6	Automatic import of database updates.....	14
5	Update symbols on mechanical page.....	15
6	Component accessories	16
6.1	Accessories come in three categories.....	16
6.1.1	Fixed accessories	16
6.1.2	Optional mechanical accessories	16
6.1.3	Optional electrical accessories.....	16
6.1.4	Syntax for accessories in the database.....	16
6.2	Add accessory to a component	17
6.2.1	Add mechanical accessory.....	17
6.2.2	Mechanical accessories on mechanical pages	17
6.2.3	If the accessory has accessories.....	18
6.2.4	Add electrical accessory	18
6.2.5	Electrical accessory on mechanical page	18
6.2.6	Extended list criteria	18
7	Reference designations	19

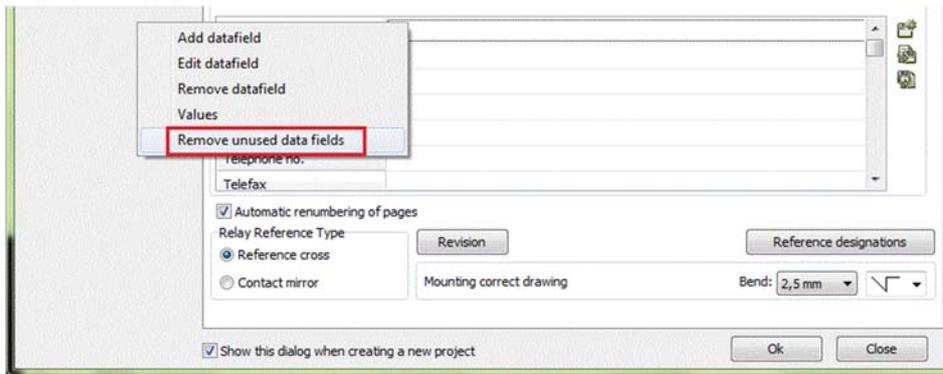
7.1	See all reference designations in the project	19
7.2	How to use reference designations	19
7.2.1	Settings	20
7.3	Create reference designations.....	21
7.3.1	Add extra aspects.....	21
7.3.2	Type new designations	21
7.3.3	Import reference designations	21
7.4	The tree structure	22
7.4.1	Show component	22
7.5	The Select Options module.....	23
8	New function in Export to dwg format.....	24
9	New Installer for installation and updates	24
10	Software license.....	24
11	Bits and pieces	25
11.1	Ver 17 is not tested on Windows XP	25
11.2	Direct access to Edit symbol from the Symbol generator.....	25
11.3	More options in printer dialog**	25
11.4	Data list is added to Divider pages.....	26
11.5	Active layer and Grouping of layers	26
11.6	Select name – now also in the Line command	27
11.7	Lines with data in the Objectlister **	27
11.8	Show current paths in Edit symbol	27
11.9	Set User Home view	28
11.10	Join signal is in the right click menu	28
11.11	Extra option for label print from Panelbuilder	28
12	New products from PCISCHEMATIC A/S.....	29
12.1	PCISCHEMATIC Service	29
12.2	PCISCHEMATIC Panelrouter.....	29
13	My notes.....	30

1 The Settings menu

This year also reveals small changes and expansions to the Settings menu. See all changes and explanations for them below.

1.1 Delete unused data fields in Project data and Page data

You can delete unused data fields in Project data and Page Data by using a new right-click command.



1.2 Pointer/Screen – Reference points

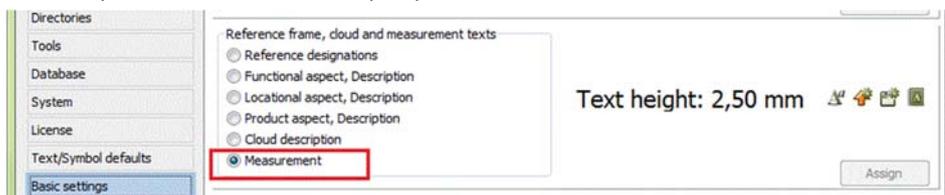
We have 'tidied up' the Pointer/Screen tab. Read more in the paragraph Readability from page 7.

1.3 Database settings, changes and new positions

See the changes in Database settings and explanations in the paragraph The Database from page 13.

1.4 The Basic settings menu

It is now possible to set Text properties for Measurements in this tab.



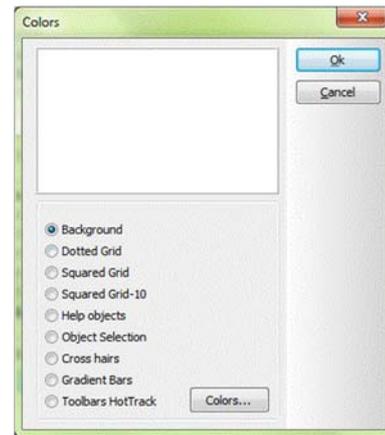
1.4.1 Colors

Color settings have moved to this menu from their previous position on the Pointer/Screen menu.



1.4.2 System colors

You find the 'old' color setting when pressing this button. This is where you decide the color for your drawing area, your pointer, cross hairs etc.. You see nothing new in this box.



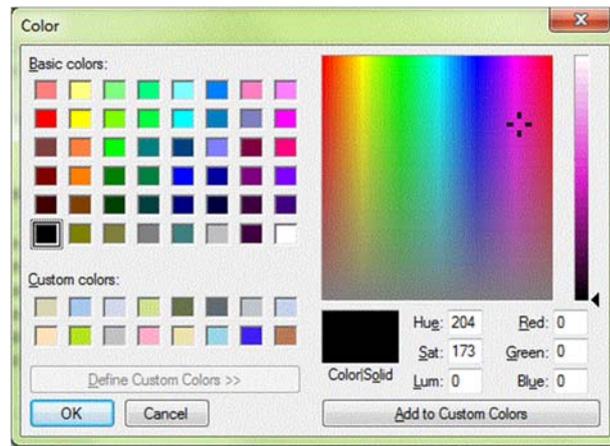
1.4.3 Custom colors

Here you define your own colors to the program.

Either, you select a new color by choosing a color in one of the two windows (remember to move the bar at the far right, if the color is too dark, ie black), *or* you type a RGB value.

Your custom colors are placed in the Custom color band and you can see them in all dialog with color options.

The color band can contain 16 colors, which means that a new color will replace an old one.



1.4.4 Using Basic colors, Custom colors or Other colors

When you choose colors from a larger selecting from version 17.

This goes for colors on lines, texts, circles/arcs and elsewhere you select colors in the program, ie symbol design and various settings.

1.4.4.1 Basic colors

The first color band contains basic colors, which are the 16 'old' colors, including the NP (not-print or invisible). This band is fixed and cannot be changed.

1.4.4.2 Custom colors

In this band, you find the custom colors that you can define. See how to do that in paragraph 1.4.3.

1.4.4.3 Other colors / More colors

At the bottom, you see 'Other colors', which are additional colors that you have chosen in the project.

If you want other colors, simply click the More colors button and select a color. You can use this color, and the selection is saved on your pc, and if other users of the project want to use the color, they can transfer settings – as always – and use it on other objects.



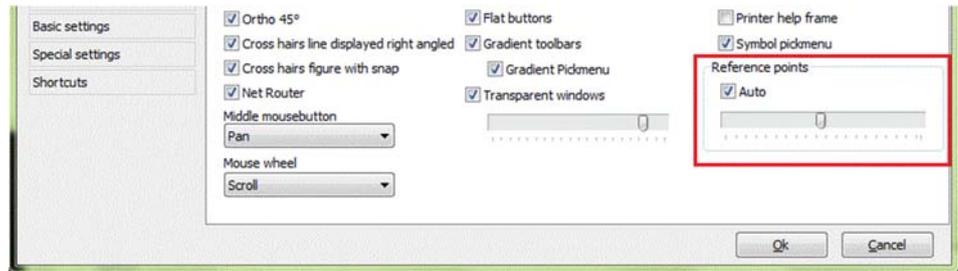
2 Readability

We have worked on making improved readability in this version. Below you can read about the measures we have made so far.

2.1 Reference points and their visibility

The intention of having the reference points is to give you a little on-line and on-screen help. However, when you have a full screen view, the same reference points are 'noisy' and make it difficult to read e.g. connection names. For that reason, it is an idea only to make the ref.points visible, when you zoom in on details on the page. This is possible when you select the 'Auto' option.

It is possible to adjust to the desired zoom level.¹



2.1.1 Ref.points in the toolbar

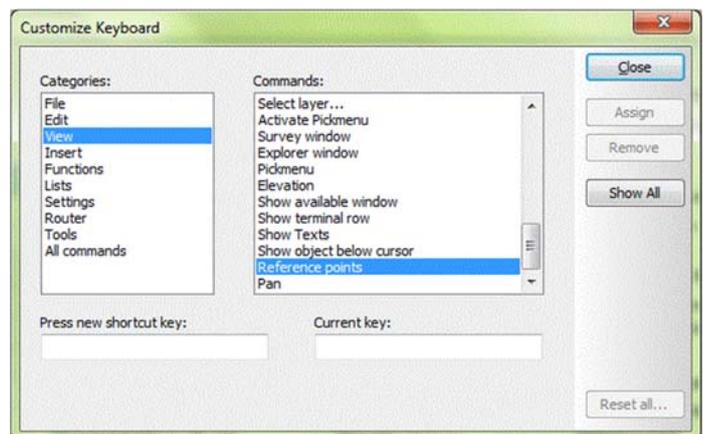
You can toggle visibility of reference points by using the new button in the vertical toolbar.



2.1.2 Shortcut to view Reference points

You can also find the command in the View menu.

You can make a short cut to the function through Settings\ Shortcuts.



2.1.3 Thin lines on mechanical pages

Symbol lines are now thin by default. This is also



part of enhancing readability. If this does not help you in the actual project, you can disable the Show lines in symbols as thin lines in Settings\Special settings.

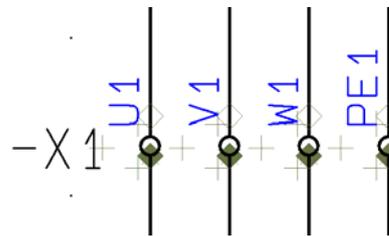
2.1.4 Grid dots

The small grid dots become almost invisible, when you have a screen with high resolution. The grid dot size now follows the zoom level.

2.1.5 Marking of terminal's external side

The external side connection point is now below the symbol itself, which enhances readability.

You can also select other colors (system colors) for ref.points etc..



3 Component wizard

The component guide's functionality has been enhanced with this version.

The guide assists you when you create components in the database, and it can now create

- Jumpers
- Wires
- Wire trays
- Cables

You can read more about the Component Wizard in a small booklet – with the same name –, which takes you through all functions. You find the booklet on our homepage, together with other manuals and booklets.

Below you see the most important window for each component type.

3.1 Component kind

When you start creating a component, you select a component kind.

As for now, you can select component kinds for some special components; all other components are set as 'normal'.

When you select a component kind, the guide goes to tabs that are special for the specific component kind.

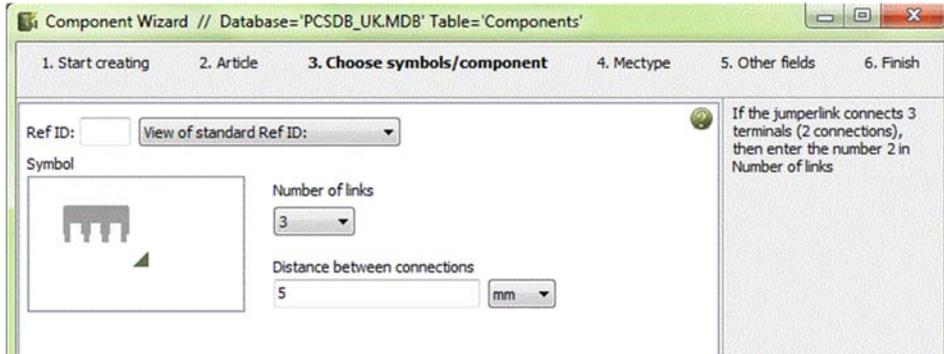
The screenshot shows the 'Component Wizard' dialog box, specifically the '2. Article' step. The title bar reads 'Component Wizard // Database='PCSDb_UK.MDB' Table='Components''. The dialog has six tabs: '1. Start creating', '2. Article', '3. Choose symbols/component', '4. Mectype', '5. Other fields', and '6. Finish'. The '2. Article' tab is active. It contains the following fields and controls:

- 'Article number (EANNUMBER)': A text box containing '1212' and a dropdown arrow.
- 'Type (TYPE)': A text box and a dropdown arrow.
- 'Table code': A text box and a dropdown arrow.
- 'Component kind': A dropdown menu with 'Normal' selected. The dropdown list is open, showing 'Normal', 'Jumper Link', 'Cable', 'Wire', and 'Wiretray'.

At the bottom left is a 'Previous' button, and at the bottom right is a 'Next' button. The status bar at the bottom center shows 'Mode = New'. On the right side of the dialog, there is a green question mark icon and a text box that says 'Enter EAN-number for the component you wish to create.'

3.2 Create jumper

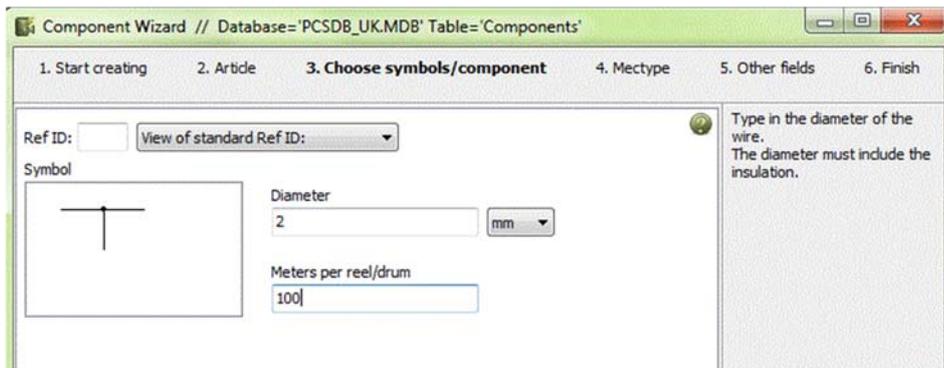
When you create jumpers in the database, you select (and show) the number of connections for the jumper as well as the distance between each connection.ⁱⁱ



3.3 Create wire

When you create wires in the database, you must type the wire's diameter (mm or ") and the number of meters per reel/drum.ⁱⁱⁱ

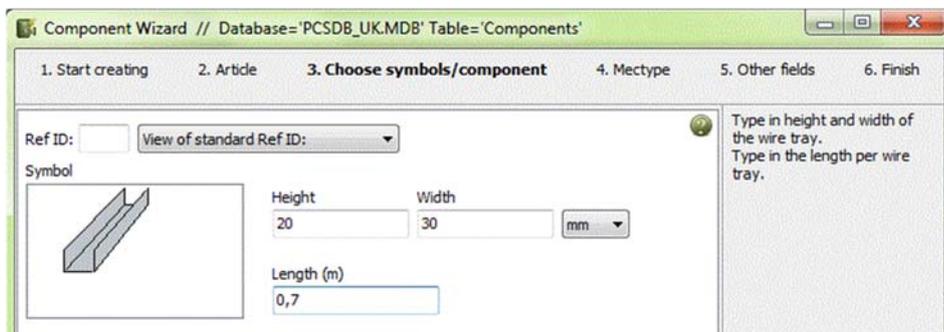
The diameter is used for calculating wire tray loads in the Panelrouter.



3.4 Create wire tray

When you create wire trays in the database, you must type the tray width and length (mm or ").

The size is used for calculating wire tray load in the Panelrouter.



3.5 Create cable

When you create cables, you must select a cable symbol, either directly in the symbol library or by using the database lookup function. The wizard will only accept a symbol of the cable type!!

You must type the cable's diameter (mm or ") and the number of meters per reel/drum. The diameter is used for calculating wire tray loads in the Panelrouter.

You must also type the number of conductors and select how you want to mark the individual conductors.

At the right side of the window, you can see standard color codes.^{iv}

Component Wizard // Database='PCSD8_UK.MDB' Table='Components'

1. Start creating 2. Article 3. Choose symbols/component 4. Mectype 5. Other fields 6. Finish

Ref ID: W View of standard Ref ID: [v]

Symbol
03-01-09

Lookup symbol from database
Choose symbol Delete

Diameter [] mm [v]

Meters per reel/drum []

Number of conductors
3

Cable conductors
 Colors
 Numbers
 Texts

Conductor color/number
 BK
 BN
 GY

Conductor colors:
 BK: Black
 BN: Brown
 GY: Grey
 BU: Blue
 WH: White
 RD: Red
 VT: Violet
 PK: Pink
 TQ: Turquoise
 GN: Green
 YE: Yellow
 OG: Orange
 GNYE: Green yellow

Type on the diameter of the cable. The diameter must include the insulation.
 Type in the number of conductors. Select colors or numbers to identify conductors.

Previous Mode = New Next

3.5.1 Cable accessories

A small repetition about cable accessories. You type accessories in the ACCESSORY datafield, which you find in the Other fields tab.

You can create accessories that come with every meter of the cable, eg 5 clips (article number CLIPS1234) per meter: Type CLIPS1234#5.

You can create accessories that come with each cable, eg 2 plugs per cable; one for each end (article number: PLUG1234): Type PLUG1234##2.

If you have two different plugs, type each article number followed by ##1 and separate with ; (semicolon).

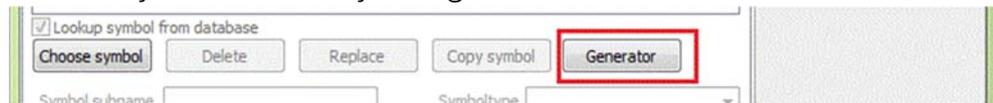
3.6 You can enter data/edit all selected data fields

When you work with the Component guide, you work with the data fields that are mapped in Database settings.

You might need to type information into other data fields than those mapped in the Component data tab. This is now possible, see how on page 14.

3.7 You can use the Symbol generator to create diagram symbols

You can make your own symbol with the symbol generator.



3.8 Access to SQL in multi-edit in database function

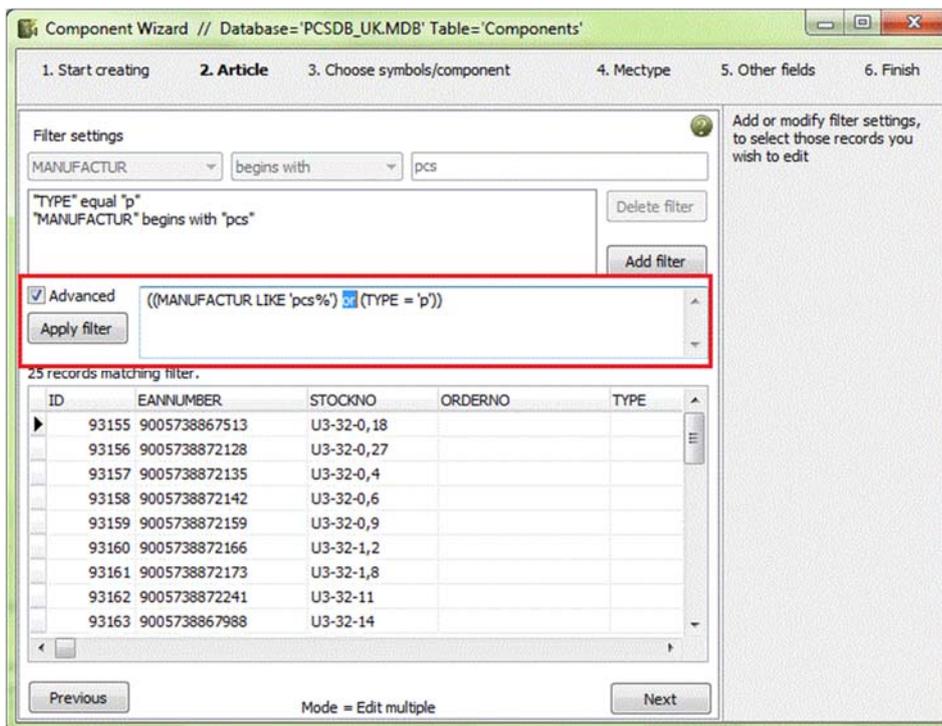
The component guide also allows multi-edit in the database.

You enter this function by clicking Ctrl+Edit.

When you select the components you want to edit, you can use the Advanced button to help you set up a filter. The normal filter that you use is a pure AND filter, which means that all components must meet Filter1 AND Filter2 AND ...

The Advanced function is a possibility to see the query as a SQL sentence, where you can write a complete query yourself or simply replace an AND with an OR thus setting up another filter. If the query is incorrect, it will not run.

Alternatively, the database program itself has a SQL function, where you can set up queries and functions using SQL.



4 The Database

The standard database contains more datafields, because new programs and functions makes it possible to use further component data.

In version 17, new functions and requirements come mainly from the Component Wizard and the Panelrouter.

4.1 New data fields

CWCODE: this is a code, which the component wizard uses to differ between cables, wires, jumper etc. See how to use it on page 9. The datafield is a system field and it is *only* used by the program.

OPT_ADDON: electrical accessories for components, ie auxilliary contacts for relais.

HEIGHT: average height for a component's connection terminals. Type value in mm, omit unit. The Panelrouter uses this datafield.

4.2 New syntaxes for typing in the database

Wire trays: Type #x30mmy40mm in MECTYPE: the tray will have a width of 30 mm (X-value) and height of 40 mm (Y-value). The two values are used to calculate wire tray load and capacity in the Panelrouter.

Wires and cables: #R0,5mm in MECTYPE means a diameter incl insulation of 1 mm.

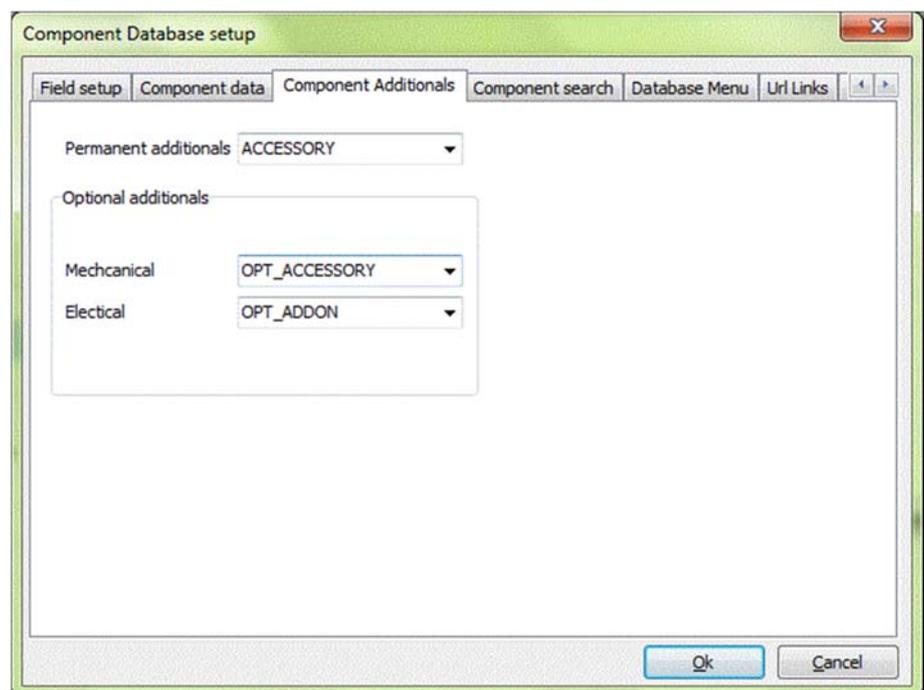
Jumpers: #X5mm in MECTYPE means 5 mm between connections. The Panelrouter checks this value with the placed jumpers' dimensions. #2 in PINDATA means that the jumper has two connections, and the value will influence the number of jumpers in the parts lists (BOM).

The Component Wizard creates the syntaxes automatically.

4.3 Database settings – Accessories in separate tab

Mapping of all accessory kinds are on a separate tab.

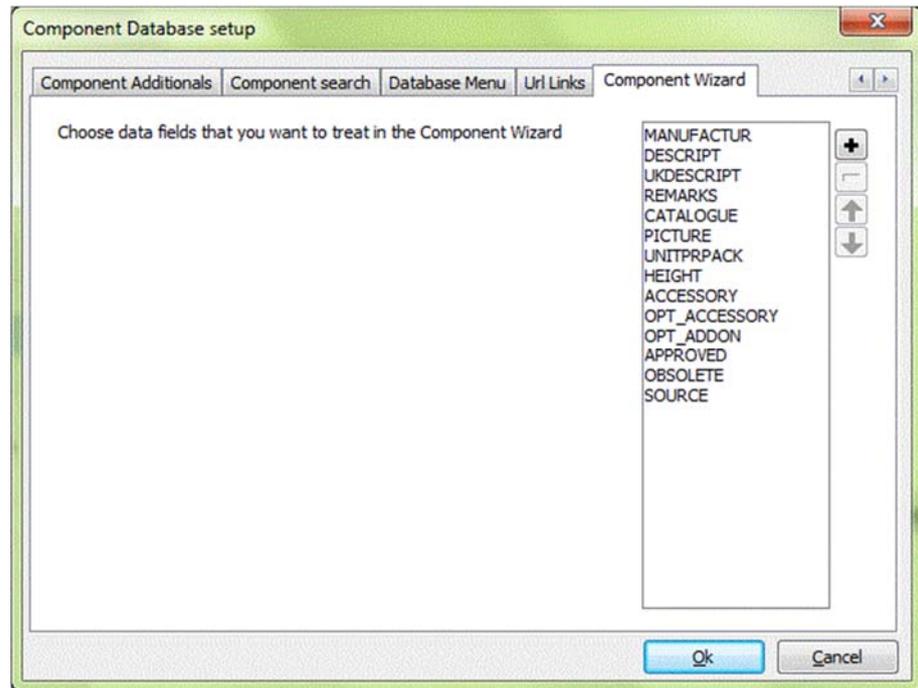
Read more about accessories from page 16.



4.4 Database settings – other fields in the Component Wizard

This tab shows all data fields that you can treat in the 'Other fields' tab in the Component Wizard.

You can add or delete database fields when you click the + key (and delete some with the - key) and change the sequence with the arrows.

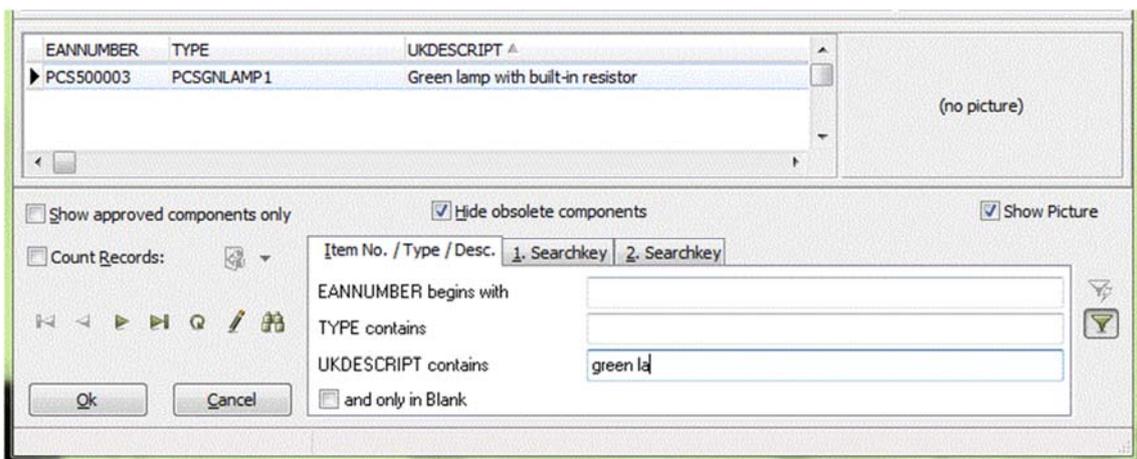


4.5 Search components in the database

The first tab in the component database has the following changes:

- TYPE is not 'contains' instead of 'begins with' and
- DESCRIPTION, which 'contains' is now placed on the first search tab.

However, it is still a good idea to make 'approved' components or to place the most commonly used components in the pickmenu ;-)



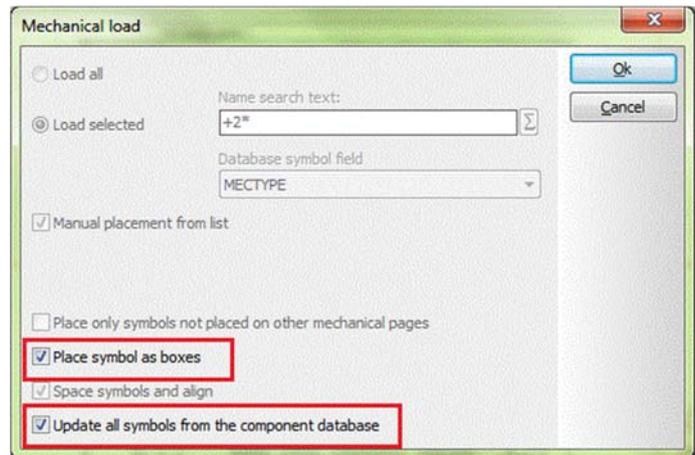
4.6 Automatic import of database updates

For future database updates, the installer automatically prompts you to import the updates.

5 Update symbols on mechanical page

It has always been possible to place either 'real' mechanical symbols or generated 'boxes' with the correct dimensions.

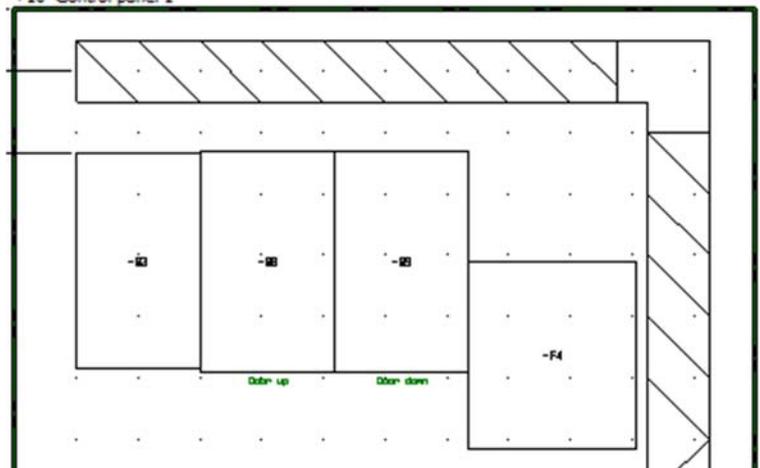
However, many 'real' mechanical symbols are #XY boxes, which means that they lack functionality in relation to the components. You could make a new 'real' mechanical symbol for the component, change it, one by one (article number) but you needed to treat each component (article number) individually.



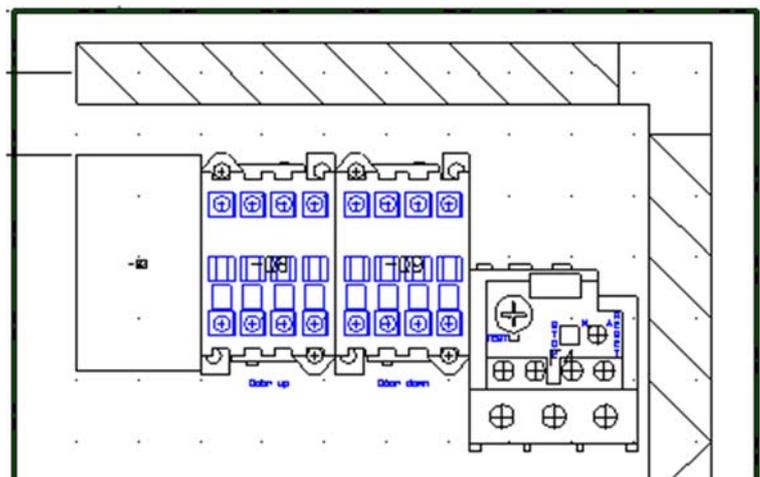
The new function 'Update all symbols from the database' enables you to update/replace all symbols on a given page in one operation.

The function also allows you to change to boxes instead; simply, make your choice in the dialog.

BEFORE:



AFTER:



or is it vice versa ☺

6 Component accessories

The component accessories function is extended compared to version 16, because it is now possible that the database contains information about 'approved' accessories for each component, fixed as well as mechanical and electrical accessories.

Below you see the options you have for accessories in the program and how to use them.

6.1 Accessories come in three categories

- Fixed accessories
- Optional mechanical accessories
- Optional electrical accessories

6.1.1 Fixed accessories

Article number(s) for this is in the ACCESSORY datafield.

You can only see this type of accessory in parts and component lists. It has the same name as the component, to which it belongs. This kind of accessory can be sockets for lamps or contactors.

In the list setting, you select to include fixed accessories.

6.1.2 Optional mechanical accessories

Article number(s) for this is in the OPT_ACCESSORY datafield.

Previously, you would add this kind of accessory by using the Support symbol, ie for end plates and dividers for terminal rows.

This kind of accessory also belongs on the mechanical pages, in the Show available list and in Mechanical placement.

6.1.3 Optional electrical accessories

Article number(s) for this is in the OPT_ADDON datafield.

Previously, you would add this kind of accessory by using separate electrical symbols, ie auxiliary contacts with the same names as the main component. In this way, all related components come together in the lists and on the mechanical pages.

You can select electrical accessory in the right click menu when you select a symbol.

This kind of accessory also is in the Show available on diagram pages and it belongs on the mechanical pages, in the Show available list and in Mechanical placement.

6.1.4 Syntax for accessories in the database

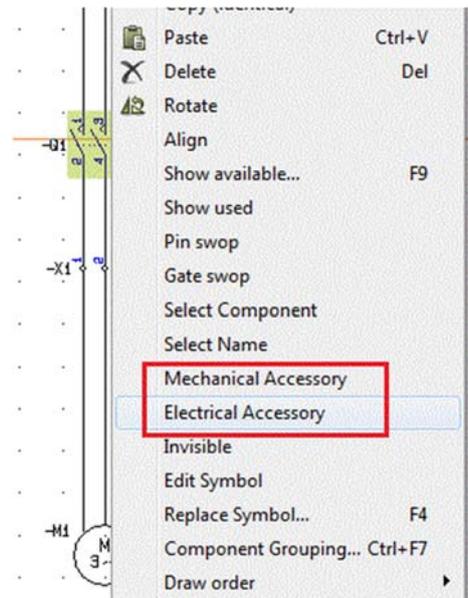
In all three datafields you type the EANNUMBER (or the number from the datafield, you mapped as primary datafield) for accessory.

More EANNUMBERS are separated by ; (semicolon).

Many *independent* EANNUMBERS can be found in an external file, which is linked with FILE=xxx.acc. In this file, linefeed means the same as semicolon. The file must be in the same folder as the database.

6.2 Add accessory to a component

Select a symbol in the component, right click and select either mechanical or electrical accessory.



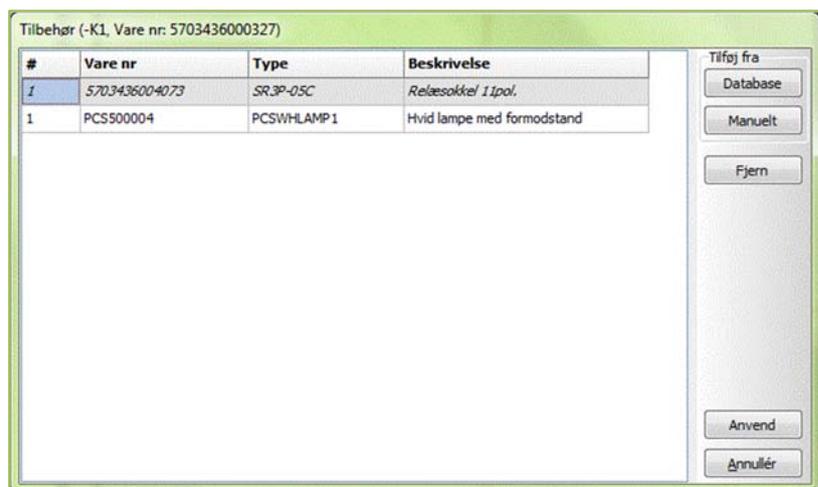
6.2.1 Add mechanical accessory

Do as in version 16:

Enter the command either via the right click menu or by using the tab Mech.accessory in the component data dialog.

In the component data dialog, you enter this window when you click the Edit button.

If the component has fixed accessory, you can see it in the top row with *italic letters*.



You add mechanical accessory in one of two ways:

Use the Database button, which takes you to the database. If there are EANNUMBERS in the OPT_ACCESSORY datafield, you can select between those EANNUMBERS/components, else you simply look up the desired component.^{vi}

Alternatively, you can type the article number and type for another component that you want to add as accessory.

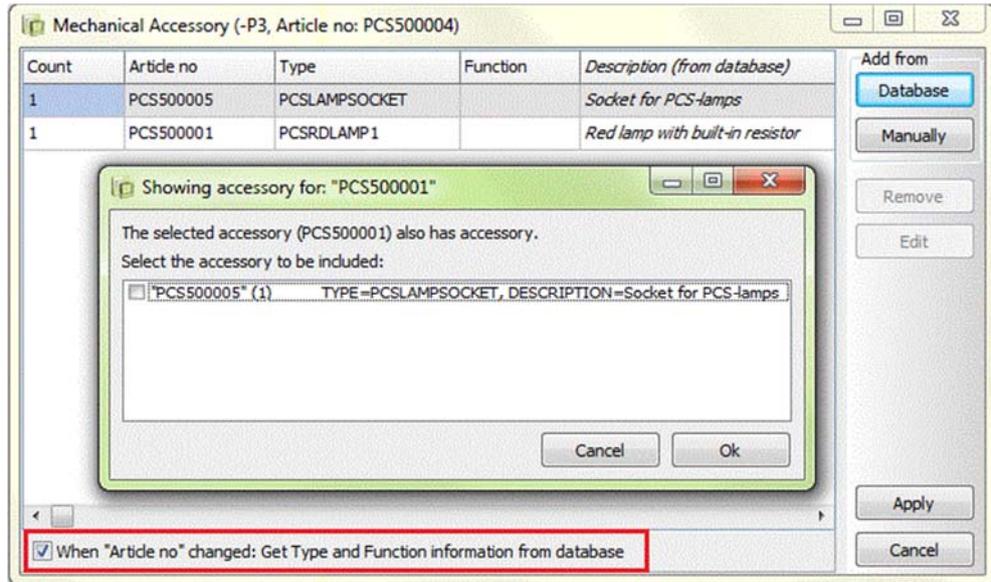
Accessories are included in the parts list with number and other data from the database and in the components list together with its main component.

6.2.2 Mechanical accessories on mechanical pages

Mechanical accessories are also available to place on the mechanical page.

6.2.3 If the accessory has accessories

On the Mech. Accessory tab, you can make automatic lookup in the database for all changes on an article number. When you tick this field, the program will automatically correct all entries for an accessory, meaning that it picks all info from the database.



It also checks for accessories on the accessory. You can include this in the lists.

6.2.4 Add electrical accessory

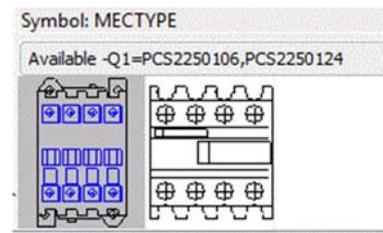
When you want to add electrical accessory you go directly to the database, where you can choose from a list of optional add-on or – if such a list isn't made – search for the accessory the same way you search for other components in the database.

If you select a component with more than one symbol, the remaining symbols are found in the 'Show available' window.

6.2.5 Electrical accessory on mechanical page

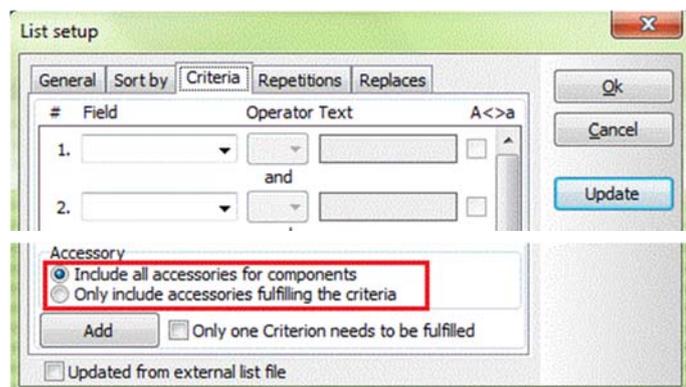
Electrical accessory, eg an auxiliary block, is also available to place on mechanical page.

If the component has a mechanical symbol, you can see it in the 'Show available' window in the mech. page or you find it with the Mechanical placement command.



6.2.6 Extended list criteria

Selection criteria on parts and components also apply for accessories.



7 Reference designations

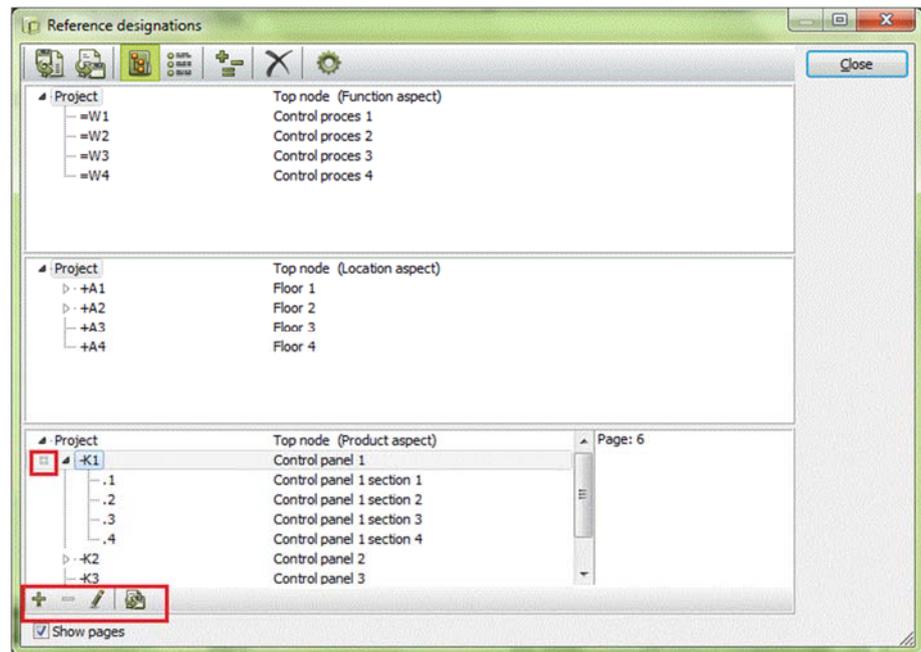
Reference designations are extended with a real Product Aspect.

Until this version, there has been a Function Aspect (=) and a Location Aspect (+); now comes the Product Aspect (-).

This makes it possible to create and use all three aspects as stated in EN81346. The aspect has been added to all functions in the program where reference designations are applied.

7.1 See all reference designations in the project

The window in which you can see reference designations in the project is extended, so it can show the product aspect designations. At the same time we have tidied up in icons and buttons. The icon for used aspects is changed.



7.2 How to use reference designations

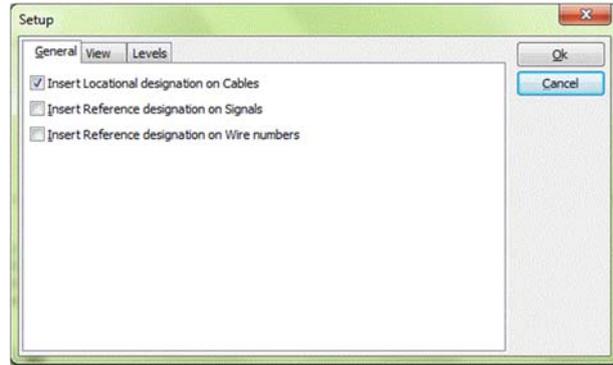
You use the new aspect in the same way as you use the 'old' aspects. That means that you still draw with symbols and give the symbol/component an aspect.

You can also import a components list into the project with full reference designations to all components, or you can make your drawing first, and add reference designations to the placed symbols, edit either individual symbols, use reference frames or use the Object Lister.

7.2.1 Settings

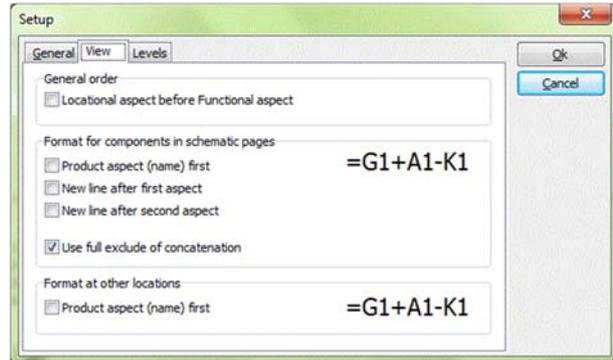
You enter the Settings when you press the button in the tool bar.

In the first tab you see the general settings. The tab is unchanged compared to earlier versions.



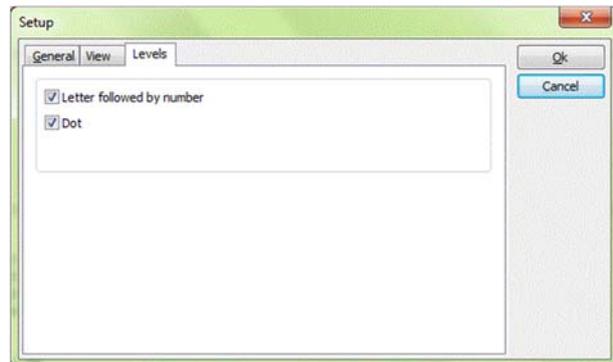
In this tab you decide – and see – how you want to show the reference designations.

This tab is also unchanged compared to earlier versions.



This tab is new.

Here you set up how the program should treat levelling when you import reference designations from an excel sheet. The set up also applies for the reference designations you create in the program.^{vii}

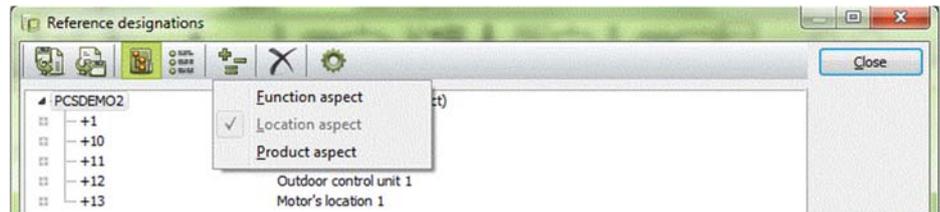


7.3 Create reference designations

You can create reference designations directly in the project by manual typing in the large window or you can import a list from eg Excel. See below.

7.3.1 Add extra aspects

The window shows only used aspects in the project. You add extra aspects by clicking the +=- button.



7.3.2 Type new designations

You can – as earlier – create new designations by typing directly in this window.

Press on of the buttons in each part of window



or use the shortcut [Insert]. You can see all short cuts when you right click.

When you create new designations, you create on a level *below* the one you are placed on.

Settings for new ref.designations are used here: if you selected that 'FULL STOP' means new level, the program adds a new level itself, if you didn't select the 'FULL STOP' the program adds the aspect's prefix.

7.3.3 Import reference designations

You can import an (Excel) list with reference designations. The various aspects must be created as texts (as in earlier versions) and they are automatically imported into the correct aspects by using their prefixes (=, +, -).

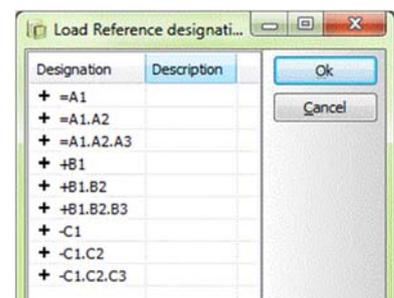
You can see an example of all aspects in the attached demo file (ref-designations_uk.xlsx).

Standard setting – see the lower picture in paragraph 7.2.1 – are, that the program sees a new designation as belonging to a new level, when the designation contains one or more (capital) letters followed by a number:

- A1, BB2, ABC3 belongs to level 1.
- A1BB2 belongs to level 2, but notice that A1 is *not* created automatically by the program; you must do that manually or in the import file.^{viii}

Standard settings are that a 'FULL STOP' is a level separator:

- 1, A, 112 are level 1
- A,B, 1.A, 1.12 are level 2. And again, notice that the program does *not* create level 1.



The prefix itself, which are =,+,- *always* means a new level, meaning that -A1-A2-A3 is level 3. But here, too, you must create -A1 and -A1-A2.

This means, that if you have a list as the one in the picture, you have three levels in A, B and C when you use the standard settings.

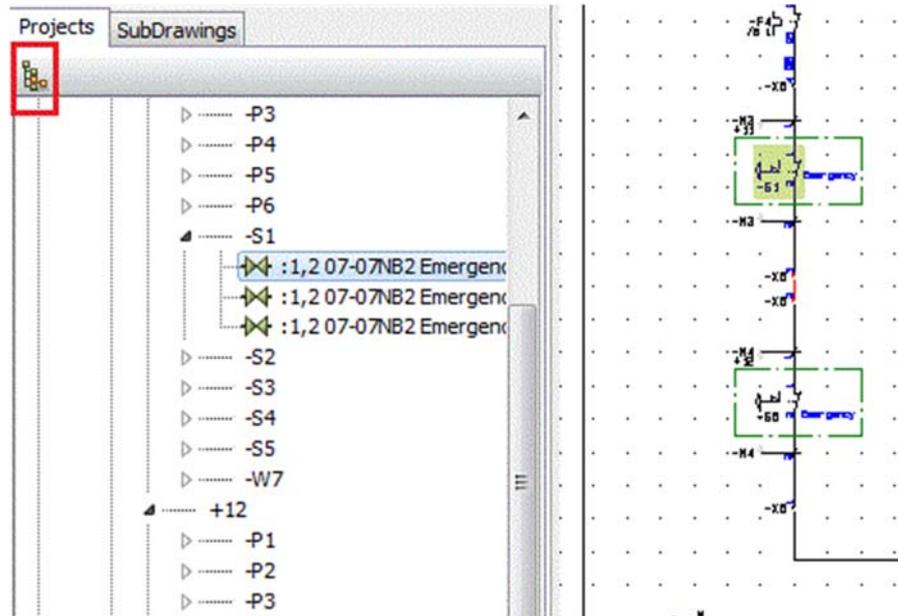
7.4 The tree structure

The program creates a tree structure based on used reference designations in the program; just as in earlier versions.

All aspects are seen in the tree.

When you click on one level, it opens to lower levels.

This means that you can see a component list of -A4 when you place your mouse on it. For each of those components you can see a list of each symbol in it and you can jump from this list directly to the symbol in the project.



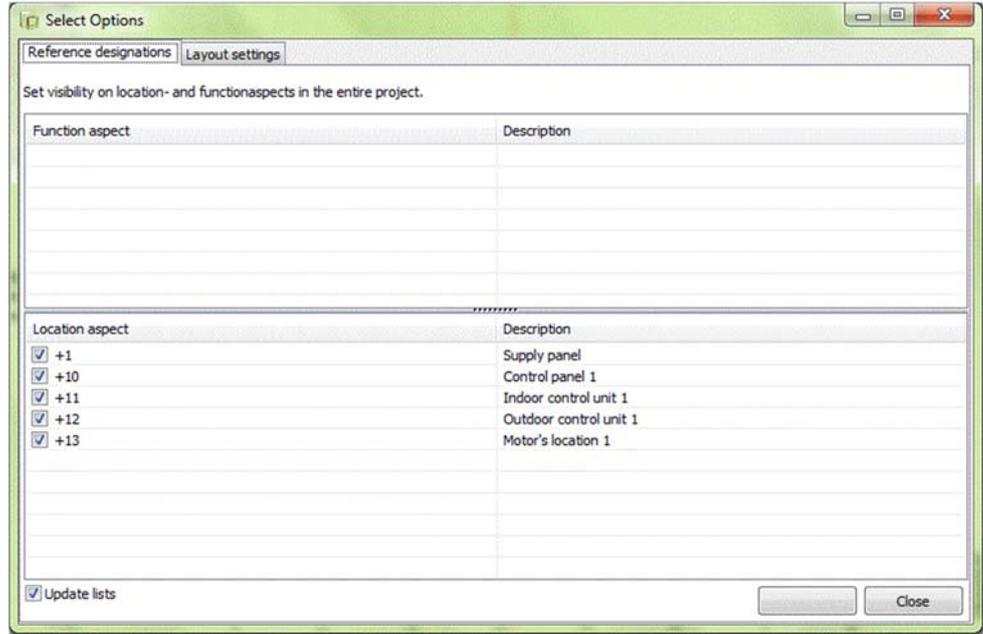
7.4.1 Show component

You can also go the other way: select a symbol in the project and click the symbol in the upper left corner. This takes you to the symbol's position in the tree structure.

7.5 The Select Options module

The Select Options module shows another way of using reference designations.

The module can generate a new project by de-/selecting options by reference designations.



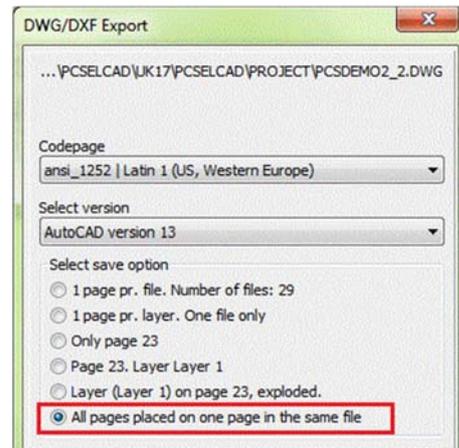
The module lists all used reference designations in a project, and you can select or deselect an option as long as you can identify the option by reference designation.

The module can generate an overview of selected options and insert a list in the project.

Enabled	Location aspect	Description
<input checked="" type="checkbox"/>	+1	Supply panel
<input checked="" type="checkbox"/>	+10	Control panel 1
<input checked="" type="checkbox"/>	+11	Indoor control unit 1
<input type="checkbox"/>	+12	Outdoor control unit 1
<input checked="" type="checkbox"/>	+13	Motor's location 1

8 New function in Export to dwg format

When you export to dwg or dxf-format, it is now possible to put all pages onto a single page in the exported file.



9 New Installer for installation and updates

We have simplified the installation and update routines. In the future, there is only one installer, which means that you need not consider making a network installation or a standard installation.

This means that regardless of the kind of installation you have, Automation can update it automatically.

The program automatically generates workstations when it starts if

- you have a terminal server session (also ver 15 and 16)
- Automation is installed in Program Files (also ver 16)
- Automation is installed on a network drive (new in ver 17)

It is still possible to make 'old-fashioned' workstation; however, this is only relevant if you want different settings for eg different customers.

10 Software license

More and more choose to have a virtual server environment. This means an increasing demand for software based license servers.

We have released a software solution in connection with version 16. We have gone further with this solution, and today it is a full solution with the same functionality as in the hardware license keys, including the borrow-functions.

You can download an English manual for the Software license server solution from our homepage.

The software license server solution is the new default license type.

11 Bits and pieces

Subjects in this chapter is inspired from input to Info@pcschematic.com.
However, we reserve our right to selectivity ;-)

11.1 Ver 17 is not tested on Windows XP

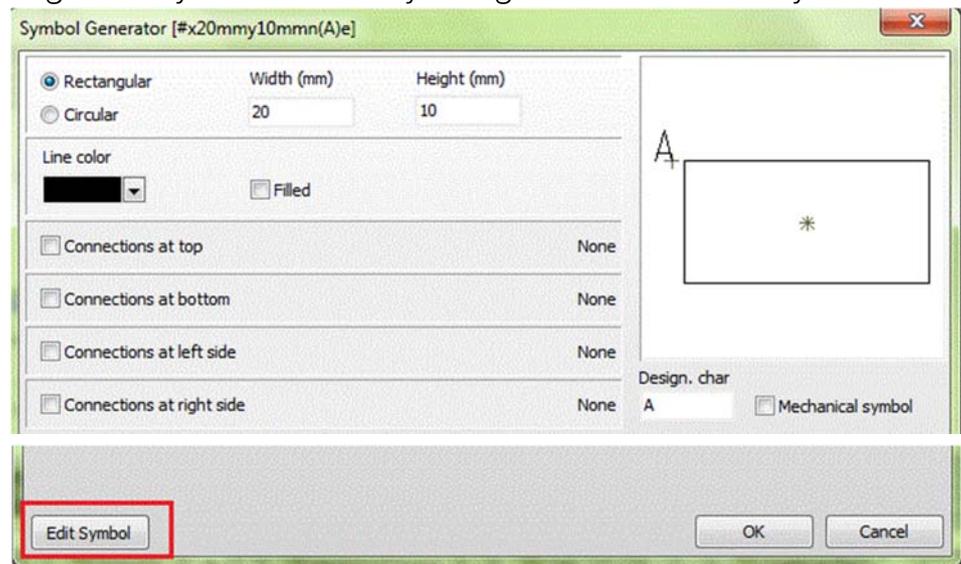
Microsoft does not support Windows XP anymore, which means that we cannot guarantee any functionality on this platform. Version 17 is *not* tested on XP.

11.2 Direct access to Edit symbol from the Symbol generator

You can start creating a new symbol from the Symbol generator: make the symbol outline here,

select connection points with names and positions and go directly to Edit Symbol where you can finish the symbol and add

auxilliary symbols.^x



11.3 More options in printer dialog**

There are two new options in the printer dialog:

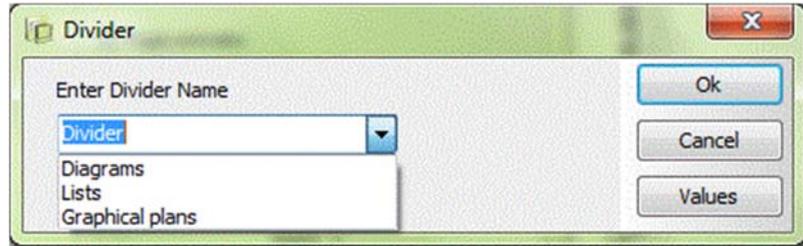
- print all non-list pages
- print all pages without states

Furthermore, you get a warning before lists are updated.



11.4 Data list is added to Divider pages

We have added a data list option on divider pages. Here you can make a list of typical divider titles.



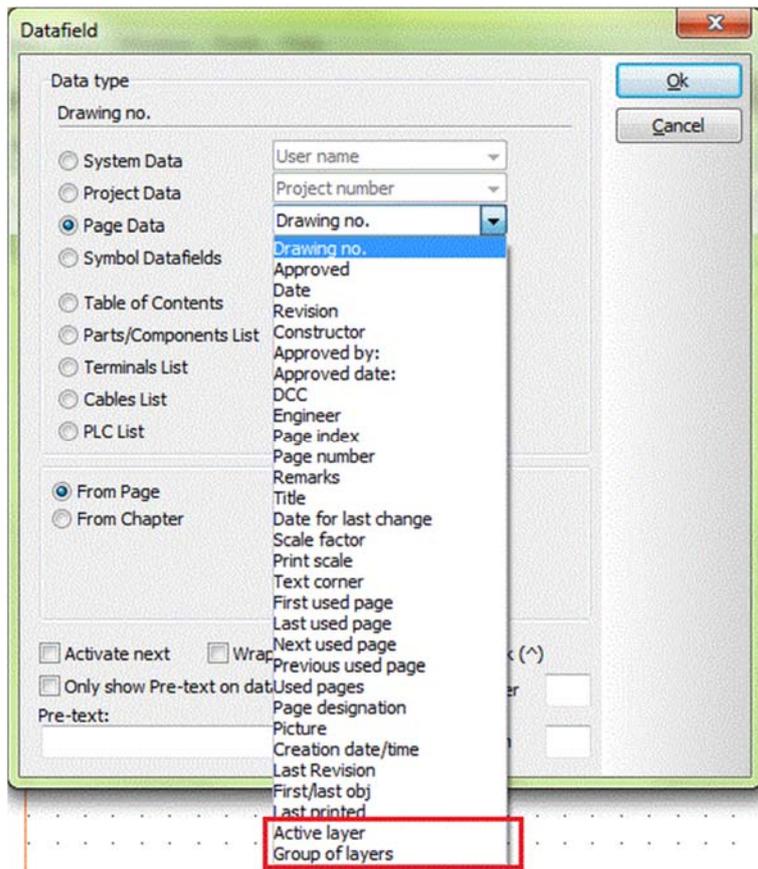
11.5 Active layer and Grouping of layers

The datafield Active layer, which is found in the Page data group, shows the current and active layer.

If you selected to show a Layer group, the datafield shows the group name here.

We have made a new datafield – Group of layers –, which shows the layers in the current group.

If you want to include this data field in the drawing header, it is a good idea to set the pretext visible, only when the datafield contains data.



Active layer: New group
Group of layers: 1,2,3

11.6 Select name – now also in the Line command

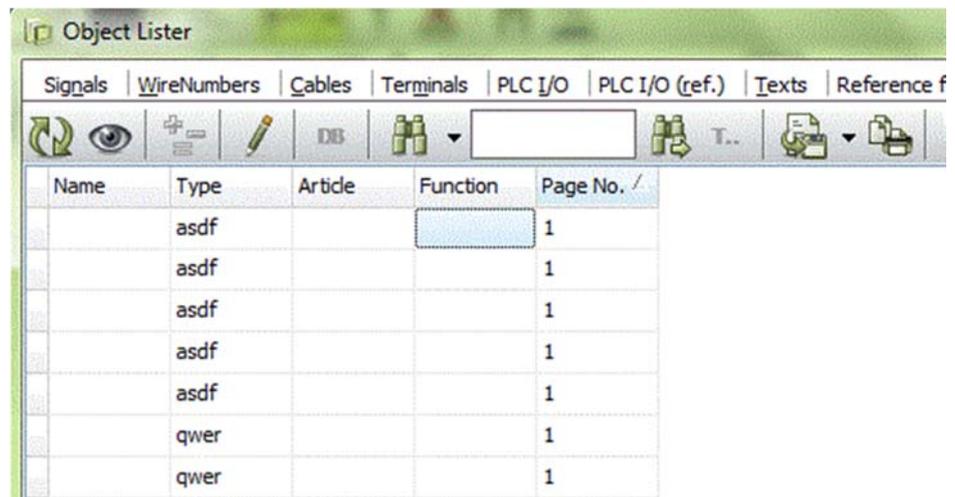
You can now use the Select Name command with lines.

The function is exactly as in with symbols, the only difference is that here you select all lines with a given name.



11.7 Lines with data in the Objectlister **

All lines with data are now in the Objektlisten (F7).^x



11.8 Show current paths in Edit symbol

It is possible to see current paths in the Symbol editor, which makes it easier to see how the symbol fits on a page.

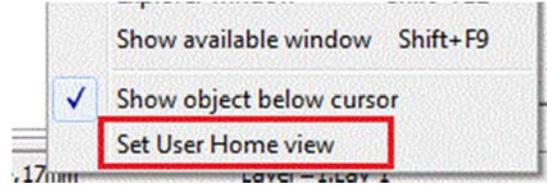
11.9 Set User Home view

This setting – found in the View menu – used to have a hard-coded shortcut:

If you clicked the <Ctrl>-button in a zoom, the selected zoom would be the User Home view.

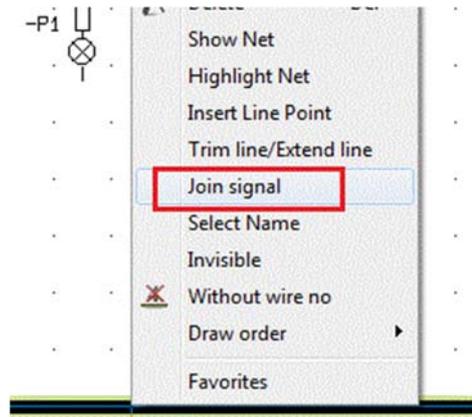
We deleted this shortcut, as we experienced user problems.

Note: you can Ctrl+0 which gives you full drawing area.



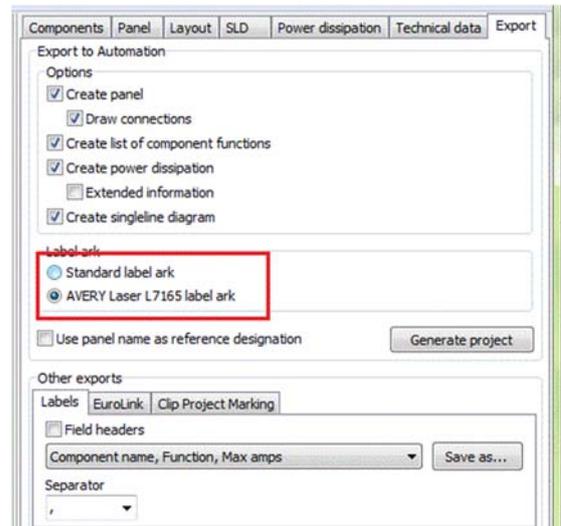
11.10 Join signal is in the right click menu

EditJoin signal (on a wide line) is now available in the right click menu.^{xi}



11.11 Extra option for label print from Panelbuilder

In relation with label print from the Panelbuilder, it is now possible to select AVERY Laser L7165 sheets. The size fits with the CE-label.^{xii}



12 New products from PCISCHMATIC A/S

12.1 PCISCHMATIC Service

Our customers have wished for a better viewer, in particular for service and maintenance purposes. This new product is called PCISCHMATIC Service.

The program has the following main features:

- Possible to look for specific components – just as in Automation and the Viewer
- Netnavigator – that is full overview of all potentials, through terminals, cables and busbars
- Print selected pages
- Export of list, eg wire numbers
- Create your own shortcuts

12.2 PCISCHMATIC Panelrouter

The PCISCHMATIC Panelrouter is a program that can suggest a routing of all wires, jumpers and cables in the mechanical layout.

The program makes intelligent use of component database data as well as data from the diagram, among it connection class options.

The program can be fully integrated with PCISCHMATIC Automation from version 17 and is released at the same time as version 17.

