



# AUTOMATION

## DESCRIPTION OF NEW FUNCTIONS

Version 18.0

This document describes new functions in Automation version 18. Descriptions of Component Wizard, Software License, PC|SCHEMATIC Panelrouter and PC|SCHEMATIC Service are in separate booklets.  
Last revision: January 2016



## Contents

1	A little more about readability and visual updates .....	5
1.1.1	Reference points in the vertical toolbar .....	5
1.1.2	Thin lines on mechanical pages .....	5
1.2	Considerations concerning fonts .....	5
1.3	Updated symbols .....	5
1.3.1	Edit symbols .....	5
1.3.2	60617_2015 .....	6
1.3.3	MISC_2015 .....	6
1.3.4	ANSI-symbols .....	6
1.3.5	HEAD – drawing headers .....	6
1.3.6	Two small programs to update alias lists and text settings .....	7
1.3.7	Updated script that changes text font .....	7
2	Automatically generated mechanical symbols - Panelbuilder .....	8
3	Component grouping – now also on mechanical page .....	9
4	Line data fields .....	10
4.1	Create your own line data fields .....	10
4.2	How to use line data fields .....	10
5	Symbol data fields .....	11
5.1	Create your own data fields .....	11
5.2	How to use symbol data fields, part 1 .....	11
5.3	How to use symbol data fields, part 2 .....	12
5.4	Resizable component data dialog .....	12
5.5	Protection against changing names .....	12
5.6	Protection against un-intentionally changing names .....	12
6	Extended data list functions .....	13
6.1	Default values in data lists .....	13
6.2	Common use of data lists and format files .....	13
7	Object groups .....	14
7.1	Create an object group .....	14
7.2	Sub drawings convert automatically to object groups .....	15
7.3	Edit the object group .....	15
7.4	Replace object group / Sub drawing .....	16
7.5	Sub drawing or 'normal' object group .....	16
8	Reference designations .....	17
8.1	Why reference designations .....	17
8.2	Concepts .....	17
8.3	One or more ref. designations – One ref. designation set .....	17
8.3.1	Writing conventions for ref. designations in levels .....	18
8.3.2	Why more aspects .....	18
8.4	How to read reference designations on diagrams .....	18
8.4.1	One reference designation with one aspect .....	18
8.4.2	One reference designation with more aspects .....	19
8.4.3	Three reference designations .....	19
8.5	The Options module .....	19
9	How do you work with ref. designations in Automation .....	20
9.1	Format for reference designations .....	21
9.1.1	One reference designation – Combined aspects .....	21
9.1.2	More reference designations – Separate aspects .....	22
9.2	List export .....	23
9.3	List import .....	23
9.4	General settings .....	23

- 10 Open an old file ..... 24
  - 10.1 Open an old file..... 24
  - 10.2 General import formats ..... 24
    - 10.2.1 Nonconformity options with combined aspects ..... 26
    - 10.2.2 Nonconformity options with separate aspects ..... 27
  - 10.3 Create ref. designations..... 28
  - 10.4 List view – now with Edit function ..... 28
  - 10.5 Start a new project – default projects ..... 28
- 11 Twisted pair cable – how to create them and work with them ..... 29
  - 11.1 New function in Component Wizard ..... 29
- 12 List with letter codes and hints ..... 30
- 13 Export of dwg/dxf-files ..... 30
- 14 Extended installation routine ..... 31
- 15 Bits and pieces ..... 33
  - 15.1 Select list criteria..... 33
  - 15.2 IO status type is now in the PLC-liste ..... 33
  - 15.3 Crossing lines ..... 34
  - 15.4 Larger 'catch' area on small symbols ..... 34
  - 15.5 Page tabs can have colors ..... 34
  - 15.6 Easier overview in the Page menu ..... 34
  - 15.7 Settings – Special settings..... 35
  - 15.8 More functions in the printer dialog ..... 35
  - 15.9 Lines with data in the Objectlister ..... 35
  - 15.10 New icons Project data and Page data ..... 36
  - 15.11 Right-click functions are now visible ..... 36
  - 15.12 New function in the Coordinates dialog ..... 37
  - 15.13 New data fields ..... 37
  - 15.14 Project information ..... 37
  - 15.15 All demo-files in new folder ..... 37
- 16 My notes ..... 38

## 1 A little more about readability and visual updates

We have continued our work with increasing readability in the current as well as in the latest version. Last year's user meetings lead to the first two issues:

### 1.1.1 Reference points in the vertical toolbar

Reference points can be (dis)abled by using the button in to vertical toolbar.

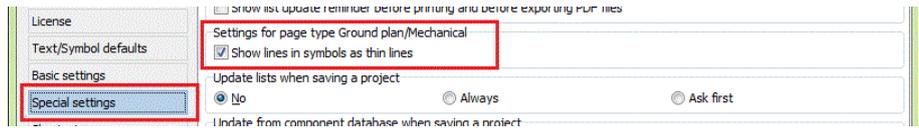


### 1.1.2 Thin lines on mechanical pages

Symbol lines are now by default thin lines on mechanical pages.

This increases

readability on the page. You can change the function for the current project in Settings|Special settings.



## 1.2 Considerations concerning fonts

During the last couple of versions, we have worked with increasing the readability of our projects. This has resulted in a couple of changes in displaying helping objects, with and without zoom, thick and thin lines in mechanical symbols. But display of texts is – still – a challenge. Our own font – the pcschematic font – needs an update, maybe mainly of its numbers, as they are difficult to read on the computer screen. Readability of numbers is best, when reference points are disabled.

The font itself could – still – need an update, but you might also go another way: we have tried to use the Arial and the OSIfonts. Both fonts are easy to read and they are very much like the CAD-font. Arial has the advantage that other programs recognize it and that is installed everywhere. The OSIfont is shareware, contains many different alphabets with many different signs – apart from European languages also Greek, (simplified) Chinese and Vietnamese.

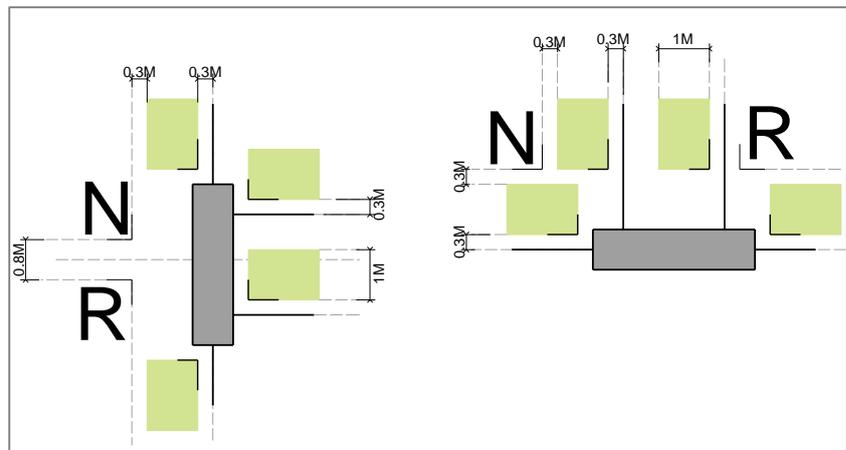
The problem is that the fonts do not contain all signs, e.g. Earth. However, you can write PE instead. If you want to try it, simply download it – we have used hikikomori82/osifont – , install it, and add one line in the Fonts section of your PCSCAD.ini file.

## 1.3 Updated symbols

All 60617-symbols have been updated so that text positions and sizes are according to standards IEC 60617 (electrical symbols) and IEC/ISO 81714-2 (design of symbols).

N is the name part, R is Reference, function, type etc.

S-numbers from IEC 60617 are also included in the symbol description.



### 1.3.1 Edit symbols

When you edit symbols, beware that the Name part (S.Name) is mirrored. That's on purpose, and necessary in order to make the Component name with ref. designations align correctly.

### 1.3.2 60617\_2015

The folder 60617\_2015 contains updated symbols from the folders

- EN 60617
- DB 60617
- IEC INST
- ALARM

When you *update* your current installation, we install the 60617\_2015 folder and we place it at the bottom of the alias-list

When you make a *new installation*, we install the 'new' 60617\_2015 folder instead of the four 'old' ones. And we place it at the top of the alias-list.

### 1.3.3 MISC\_2015

The folder MISC\_2015 contains updated symbols from the folder

- MISC

When you *update* your current installation, we install the MISC\_2015 folder and we place it at the bottom of the alias-list.

When you make a *new installation*, we install the 'new' MISC\_2015 instead of the 'old' one. And we place it at the top of the alias-list.

### 1.3.4 ANSI-symbols

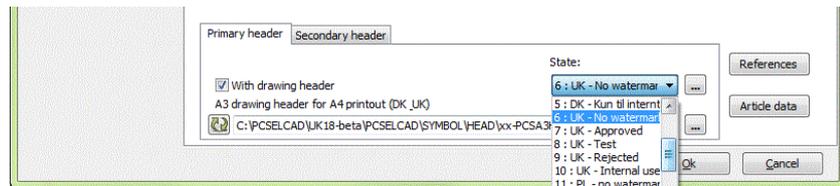
The number of symbols in the ANSI-folder is increased compared to version 17.

### 1.3.5 HEAD – drawing headers

We have updated the drawing headers with the prefix PCS so that they contain more than one language.

The state title contains language code and watermark text. The update means that you don't necessarily have to change drawing headers when you translate a project, you need only to change the state.

When you change or add a drawing header to an existing page, the program will automatically select the first state with the language code selected in Settings|System.



	<b>Project title:</b> Demo project v.18	<b>Case no.:</b> 1234
	Customer: The Factory	DCC:
	Page title: Connection diagram	Dwg. no.:
	File name: Project	Eng. (proj/page):
	Page ref.:	Appr. (date/init):

	<b>Projekttitel:</b> Demo project v.18	<b>Sagsnr.:</b> 1234
	Kunde: The Factory	DCC:
	Sidetitel: Connection diagram	Tegningsnr.:
	Filnavn: Project	Konstr. (projekt/side):
	Sideref.:	Godk. (dato/init):

### 1.3.6 Two small programs to update alias lists and text settings

In connection with the symbol updates, we have changed the alias-list and the default text settings.

The position of folders in the alias list affects database lookup of symbols: the database looks for symbols' filenames, and it looks in the symbol folders in the sequence made by the alias-list.

Text settings affect the look of all texts and symbols in a (new) project.

'Old' users who wish to have the new settings, and 'new' users who wish to use the 'old' settings, can download a small program from our homepage. The programs will only work on systems updated to version 18.

#### 1.3.6.1 Install 'New symbols'

This update makes Automation use the same symbols and text settings as a completely new Automation version 18.

It does the following:

- Installs the new folders MISC\_2015 and 60617\_2015, which become the new standard folders, i.e. that the database will select its symbols from those folders.

It doesn't delete the old folders (MISC, EN60617, DB60617, INST and ALARM), it simply moves them to a lower position in the alias-list.

The symbols in the new folders are changed:

- all texts are 2.5 mm and black
- text positions are changed, all texts are now on the left side of the symbol.
- Your text settings (under Settings|Text/Symbol defaults) are changed to 2.5 mm black, but only if you haven't altered any. If you have changed any text default, we don't touch them.

#### 1.3.6.2 Install 'Old symbols'

This update makes Automation use the same symbols and text settings as a previous installation of Automation, which is earlier than version 18.

It does the following:

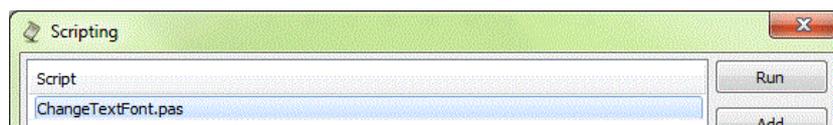
- Installs the old folders MISC, EN60617, DB60617, INST and ALARM, which become the new default symbol folders, i.e. that the database will select its symbols from these folders.

It doesn't delete the folders MISC\_2015 and 60617\_2015, it simply moves them to a lower position in the alias-list.

- Your text settings are set to all settings, as long as you haven't made any changes to them:
  - Line and symbol name: black, 2.5 mm,
  - Type, article no, function, reference: blue, 2.5 mm
  - Connection names: blue, 1.8 mm
  - Free texts and data fields: blue, 1.8 mm
  - Signal references: blue, 1.8 mm

### 1.3.7 Updated script that changes text font

We have updated the script 'ChangeTextFont' so that you can select between different fonts.



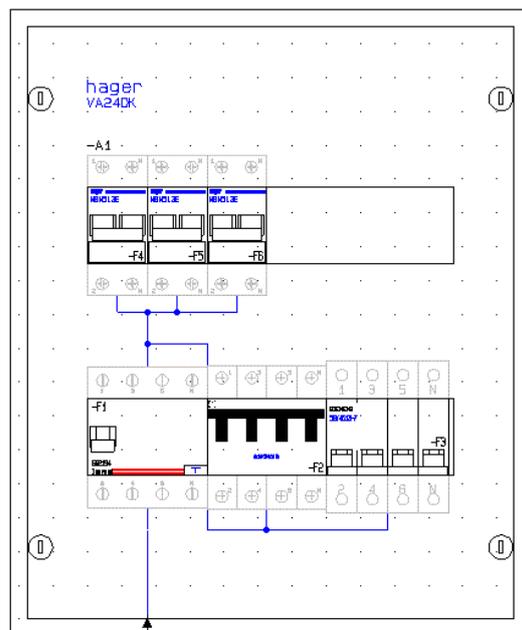
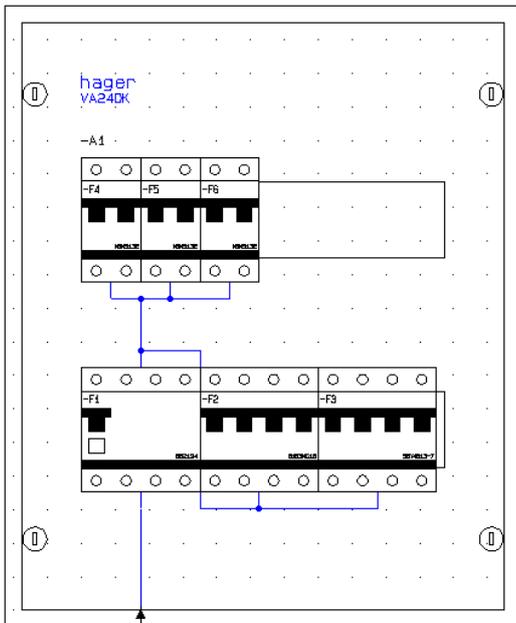
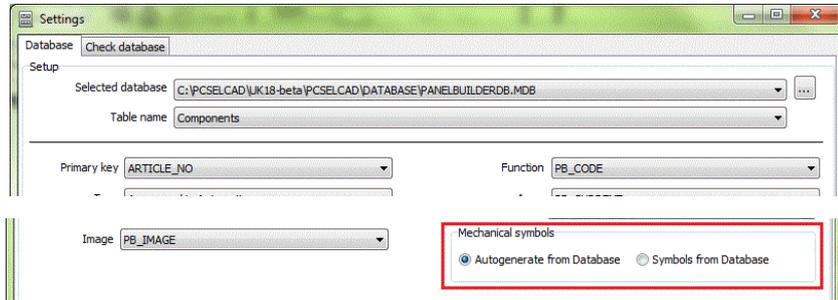
## 2 Automatically generated mechanical symbols - Panelbuilder

It takes looong time to make mechanical symbols if you want them to look like the component. And you cannot just re-use a symbol even though it looks much like another one, as the new component may have other or more or less connection points than the original component.

You will also see different 'styles' in symbol design when you look through the existing mechanical symbols, as manufacturers often have their own design of the same kind of component.

Our goal is to be able to generate mechanical symbols based on data in the database. Our first attempt has been the mechanical symbols in the Panelbuilder, as this module only contains few component types.

You can choose between the options in the module's settings.



### 3 Component grouping – now also on mechanical page

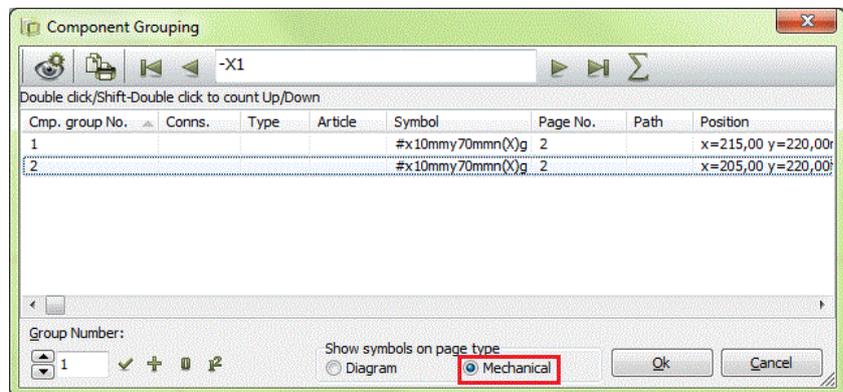
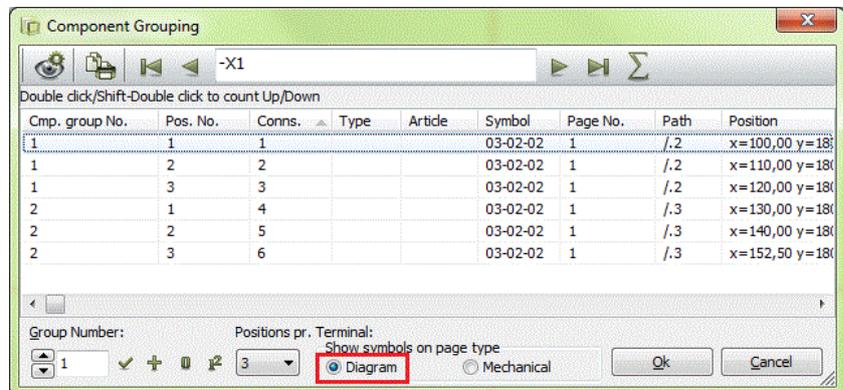
Component grouping has a new button, which can change between grouping of diagram symbols and grouping of mechanical symbols. Component has up to now only dealt with diagram symbols, now you can go directly to the mechanical page to see the same components and vice versa.

In this example, the diagram page contains a small terminal row with 6 terminal symbols on 2 triple-terminals, all without article number.

On the mechanical page, you find 2 #-symbols. When you go to the mechanical symbols, you can now assign the correct component group numbers, which in turn will make the program know, that the symbols are the same components. The same component group number means that it is the same component, i.e. parts and components lists will be correct.

You change the group number when you double-click.

You can go directly to the symbol when you press the 'eye' – or if you click once on the symbol.



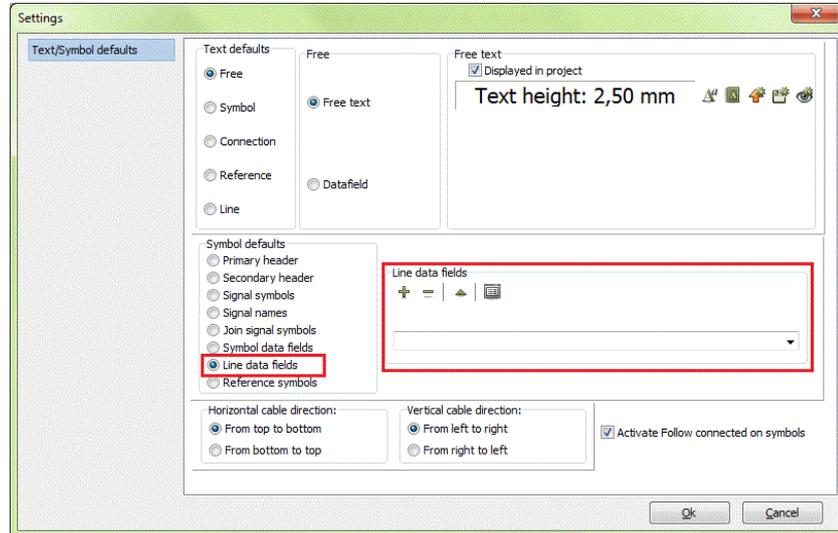
## 4 Line data fields

All lines are 'born' with the line data fields Name, Type, Article number, Function and Description. The data field 'Description', which can be mapped to a database data field, is visible since ver.17. If you haven't selected a valid article no., nothing is displayed.

### 4.1 Create your own line data fields

You can create your own line data fields: Go to Settings| Symbol/Text defaults to Line Data fields.

You can make a data list with values for each line data field, and one of the values can be a default value.



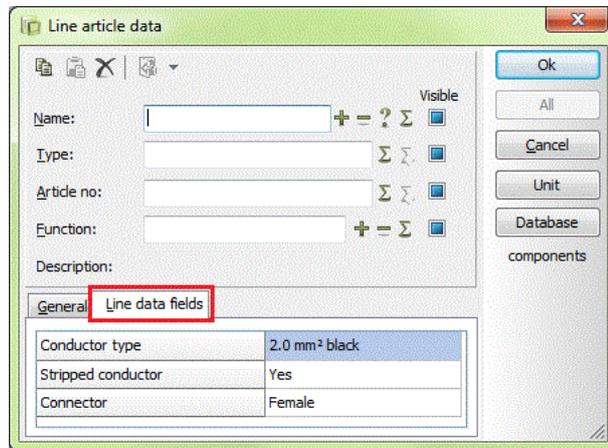
### 4.2 How to use line data fields

When you have created a line data field, you can add it to any line in the project.

If the data field has a default value, this value shows immediately when you click in the value part of the dialog. If you want another value, simply select another one from the list or type a value.

Selected values are invisible on the page, but they are selectable for various list, ie a Connection list file.

We only save Line data fields with values, which means that the new feature doesn't add a lot to the project size.



Connection list file with line data fields				
From	To	Conductor type	Stripped cond.	Connector
-X2:5	-X1:5	2.0 mm² black	Yes	Female
-X2:6	-X1:6	2.0 mm² black	Yes	Female
-X2:7	-X1:7	2.0 mm² black	Yes	Female
-X2:4	-X1:4	2.5 mm² black	Yes	No connector
-X2:1	-X1:1	2.0 mm² black	Yes	Female
-X2:2	-X1:2	2.0 mm² black	Yes	Female
-X2:3	-X1:3	2.5 mm² black	Yes	No connector

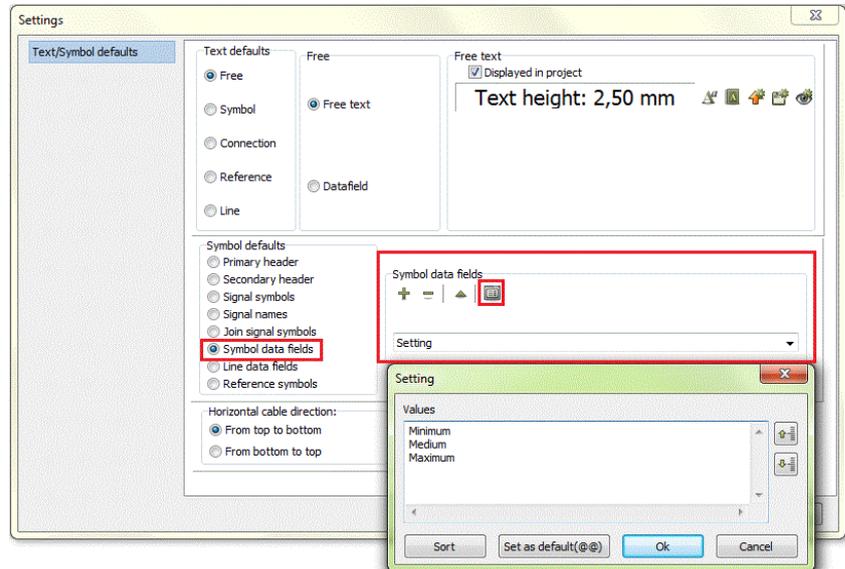
## 5 Symbol data fields

All symbols are 'born' with the data fields Name, Type, Article number, Function and Description. The data field 'Description', which can be mapped to a database data field, is visible since ver.17. If you haven't selected a valid article no. for the component, nothing is displayed.

### 5.1 Create your own data fields

You can create your own symbol data fields: Go to Settings| Symbol/Text defaults to Symbol Data fields.

You can make a data list with values for each symbol data field, and one of the values can be a default value.



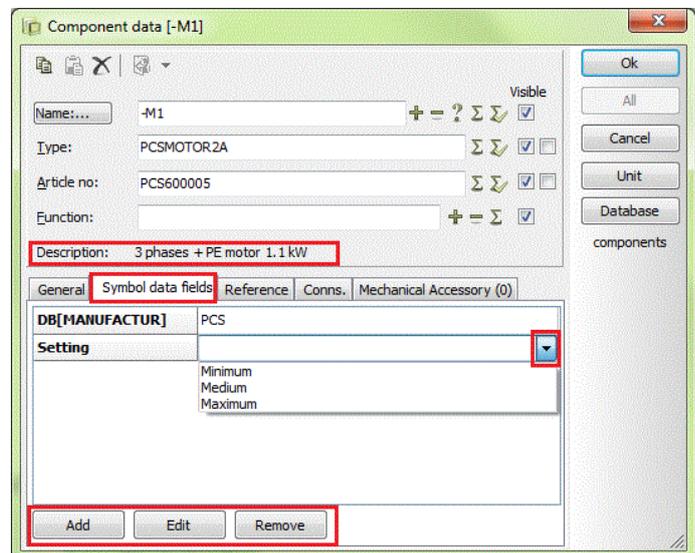
### 5.2 How to use symbol data fields, part 1

When you have created a symbol data field, you can add it to any symbol in the project.

If the data field has a default value, this value shows immediately when you click in the value part of the dialog. If you want another value, simply select another one from the list or type a value.

You can also add database data fields, so that information from here shows directly on the symbol.

All selected values are visible on the page and they are selectable for various lists.

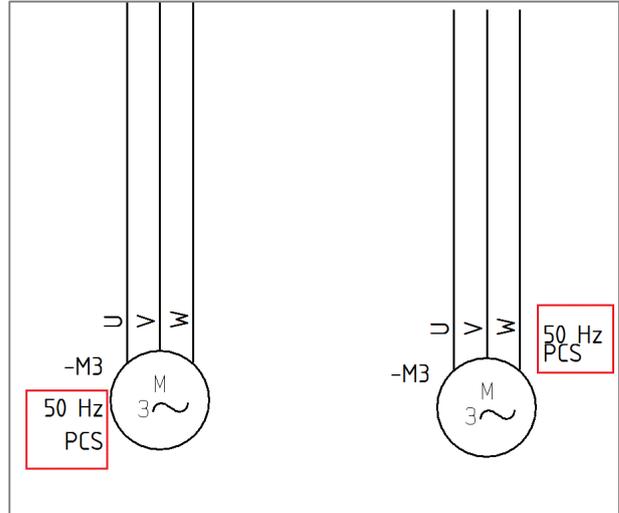


### 5.3 How to use symbol data fields, part 2

When symbols have new data fields they behave normally, which means that you can move them freely, change text settings etc.

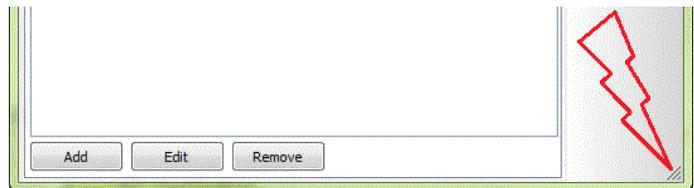
Default text settings follow the Data field setting in Tex/Symbol defaults.

You can see their default position on the symbol in the figure to the right, and you can move them to your preferred position.



### 5.4 Resizable component data dialog

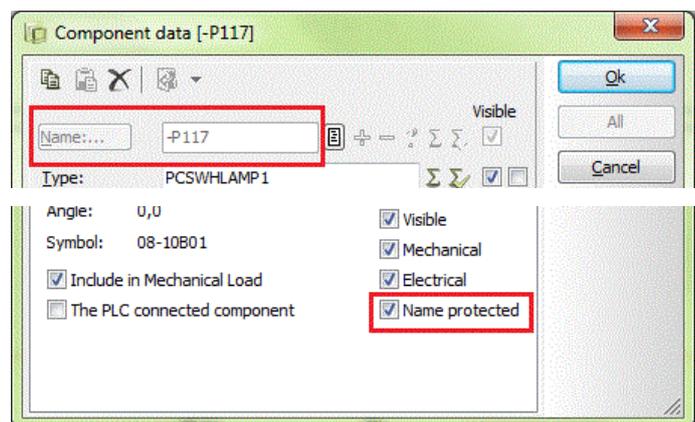
If you need – or want to – you can change the size of the Component data dialog.



### 5.5 Protection against changing names

We have added a new checkbox 'Name protected' in the Component data dialog.

If you tick it, the name is locked and dimmed.



### 5.6 Protection against un-intentionally changing names

If you double-click on a symbol name, you go directly to the Component data dialog. In this way, you don't risk changing the name of one symbol in the component.

## 6 Extended data list functions

Line and symbol data fields and all other data fields can find values in small lists. You will find small changes in the way those lists work in the program.

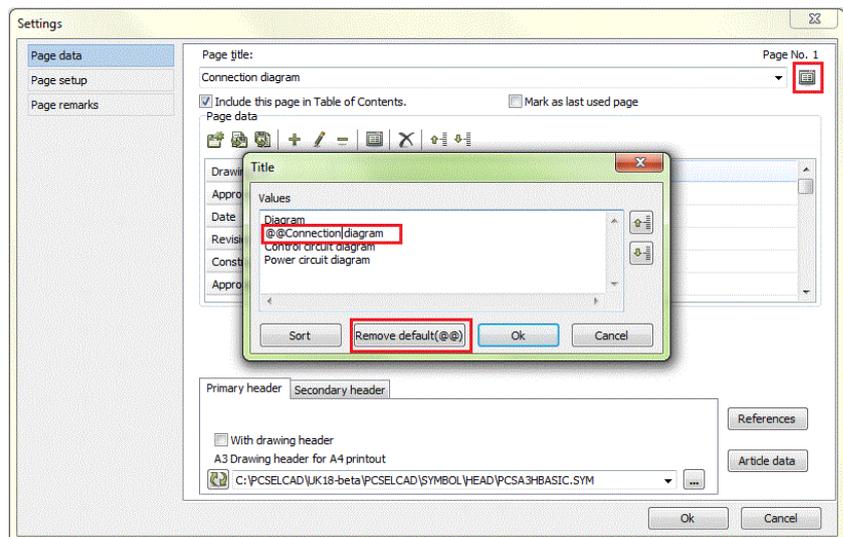
### 6.1 Default values in data lists

You can make data lists for many data fields, which will make it easy to select a value, instead of typing a value each time. Now it is possible to set a default value – and change it – in the list. When you select a value to be default, you add two @ to it in the list.

If the page has a value through a template, the program will see this value as default and add it to the page's current list.

You can make default values on the following lists

- page title
- divider title
- project title
- project data
- page data
- symbol data fields
- line data fields

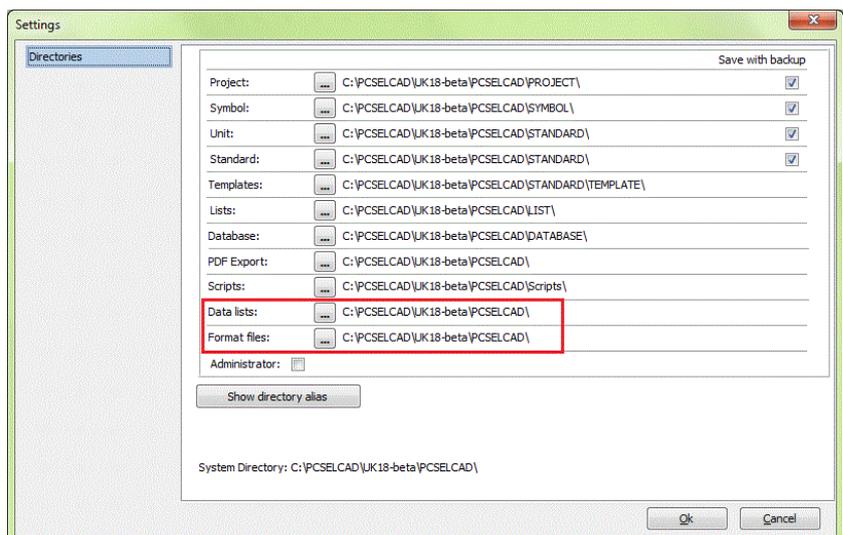


### 6.2 Common use of data lists and format files

Data lists such as page titles are now available for more than one user.

The same goes for format files, which are the files that define list file export.

Until now, those file types have only been available on the individual pc, now you can place the files on server drives.

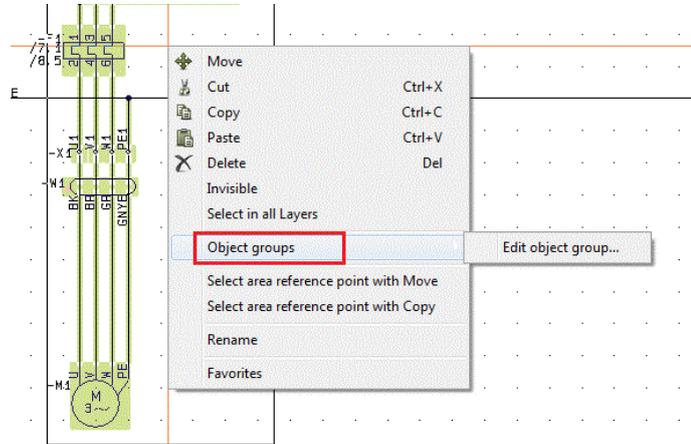


## 7 Object groups

This is a new function, which allows you to select a group of objects – symbols, lines, texts – with just one mouse click. You use this copy, move, delete or change a full group of object in one operation.

### 7.1 Create an object group

Mark an area in the project, right-click and select Object groups|Edit object groups...

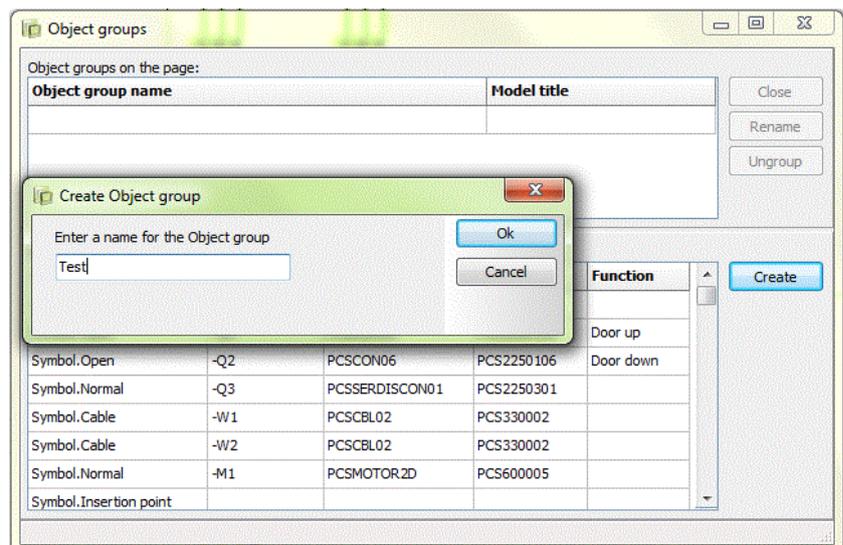


This opens the dialog where you can create the Object group and give it a name of its own.

You can see the various symbol types and data.

You can select the group, rename it and ungroup it.

When you select the group, you highlight all its objects in the project. If you select one of the objects in the group, you can remove it from the group.



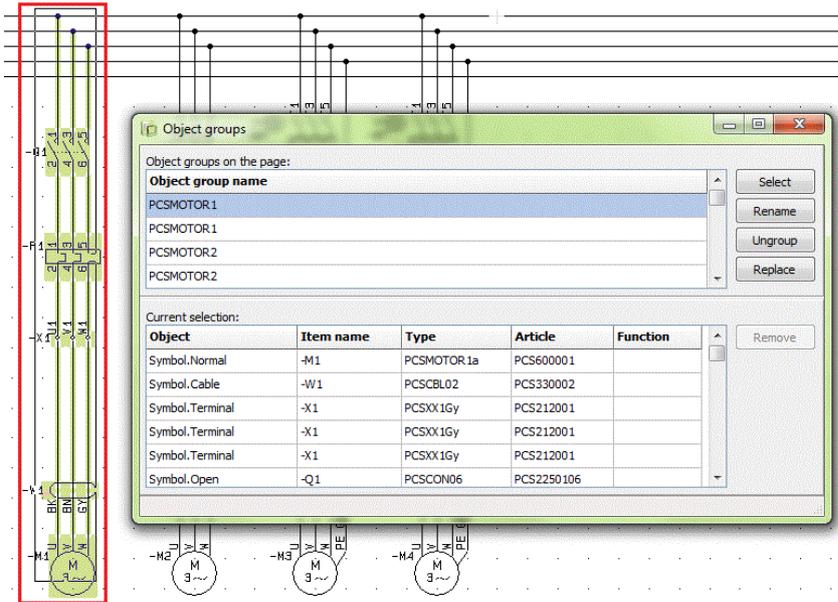
## 7.2 Sub drawings convert automatically to object groups

When you add sub drawing to a project, the program will automatically convert those to object groups. The object groups will get the same name as the sub drawing.

When you select a sub drawing / object group in the dialog, you can see all its objects. If you click a specific object in one of the groups, you can remove it.

Selected object groups can be

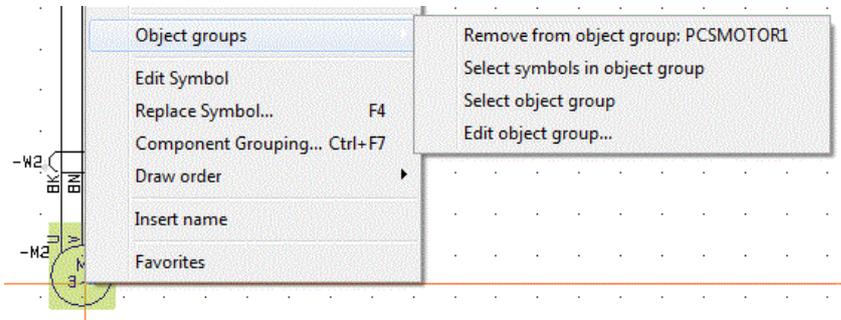
- Renamed
- Ungrouped
- Replaced



## 7.3 Edit the object group

When you select one of the drawing commands – symbol or line – you can right-click on an object. This allow you to

- Remove the object from the object group
- Select all symbols (or lines) in the object group
- Select the object group
- Edit the object group



Remove an object – happens immediately when you click the menu-item.

Select all symbols/lines will highlight them on the screen, and lets you continue to work with them.

Select the group – to be able to copy, move, and delete it and simply to make sure, that everything is included.

Edit the group – gives you a dialog in which you can delete one or more objects.

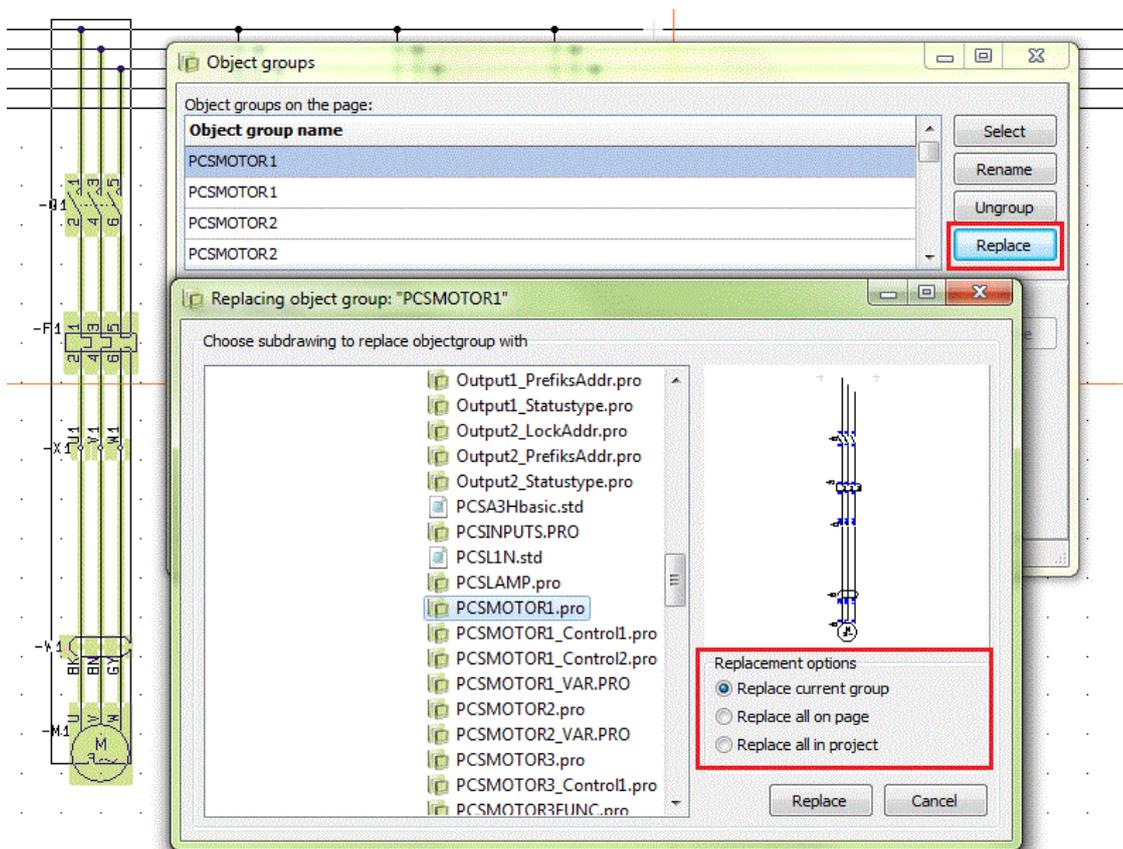
## 7.4 Replace object group / Sub drawing

You can replace object groups and/or sub drawings in a project with a sub drawing.

Right-click on the page and select 'Show object groups...'



1. Select an the object group
2. Select Replace
3. Select the sub drawing you want to replace the selected object group
4. You can now choose between the following options
  - a. Replace the current group
  - b. Replace all groups with the same name on this page
  - c. Replace all groups with the same name in the project
5. Now click Replace and all object groups are replaced



## 7.5 Sub drawing or 'normal' object group

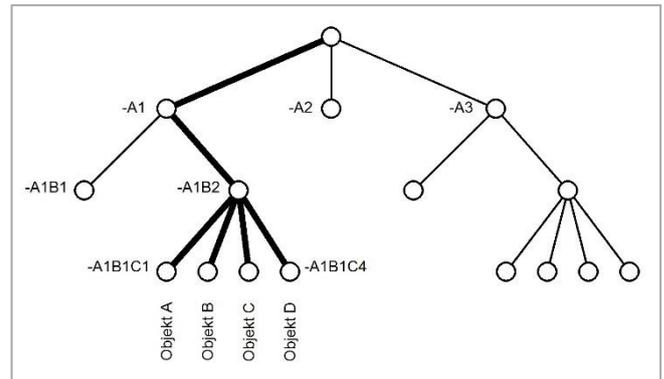
The Replace button becomes active when you select a sub drawing. That means, that it is not active when you 'only' made a grouping of symbols and lines on a page.

An object group is regarded as a replaceable sub drawing when you place and add two insertion points to the object group. The symbol INSPPOINT.SYM is in the MISC folder.

## 8 Reference designations

### 8.1 Why reference designations

- you need to have a name, that is not misunderstood
- you want to make re-use of objects easier in the project
  - ref. designations have a tree-structure, which means that all objects are constituents of higher-level objects, and thus it is easy to combine objects/groups of objects, also from different suppliers/origins.



### 8.2 Concepts

**aspect:** a pair of ‘glasses’: eg you can see a project based on its functions alone, or its locations, or its products. Sometimes you wish to see one of the aspects, sometimes more of them.

**designations:** ‘names’ in each aspect, ie -P1 is a component, +B1 is a location.

**prefix:** a sign in front of the designations, that tells which aspect the designation belongs to: = for function, + for location and - for product.

**pcs-concept:** subname: our ‘old’ way of having product designations in two levels. The database can control subnames, which means that the top level has an article number.

### 8.3 One or more ref. designations – One ref. designation set

	Reference designation	Possible graphical displays each designation	
		One line	One line per designation
Three ref. designations; one ref.design. set	=A1	=A1/-B2/+C3 <input type="checkbox"/>	=A1 <input type="checkbox"/> -B2 <input type="checkbox"/> +C3 <input type="checkbox"/>
	-B2		
	+C3		
One reference designation; one ref.design. set	=A1-B2+C3	=A1-B2+C3 <input type="checkbox"/>	=A1-B2+C3 <input type="checkbox"/>
Two reference designations; one ref.design. set	=A1-B2	=A1-B2/+C3 <input type="checkbox"/>	=A1-B2 <input type="checkbox"/> +C3 <input type="checkbox"/>
	+C3		

Our standard setup for new project is option one, which is three reference designations.

We treat old projects according to their settings. This means that some old projects will be one ref. designation set, others with three sets. Read more about this later in the section.

### 8.3.1 Writing conventions for ref. designations in levels

You can write names in each aspect in the following ways:

Use the prefix as a divider between levels:	-A1-B1-C1
Use dot as a divider between levels:	-A1.B1.C1
Use space as a divider between levels:	-A1 B1 C1
No visible divider between levels (requires capital letter followed by number):	-A1B1C1

### 8.3.2 Why more aspects

- different professionals can talk about the same object based on their individual understanding of it: some will talk about functions, others about locations, others again about products
- when different profession communicate it is important that you can focus on each aspect
- separate aspects makes it – mostly – easier to get an overview of complex projects: focus on the necessary functions or products - one is extra information to the other as in 'nice to know'
- if you let aspects depend on each other – combined aspects – will often be more complex: what depends on what? does function depend on location, always, or?
- more aspects allows for double naming, i.e. new and old tag-names on one object

## 8.4 How to read reference designations on diagrams

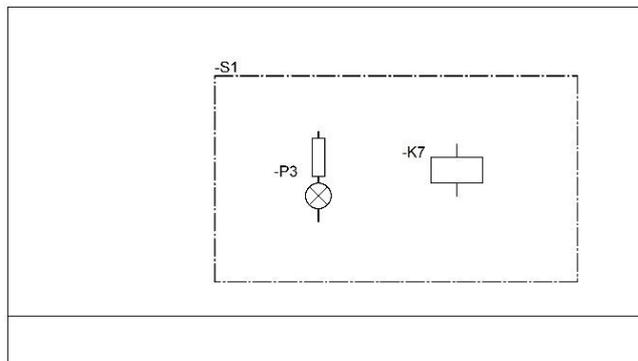
Here you see a couple of examples of how to show reference designations in diagrams. It is very common to place reference frames in the diagrams to enhance readability.

When you read the reference designations, you normally read them aspect by aspect.

### 8.4.1 One reference designation with one aspect

Object	Ref. designation
Lamp	-S1.P3
Coil	-S1.K7

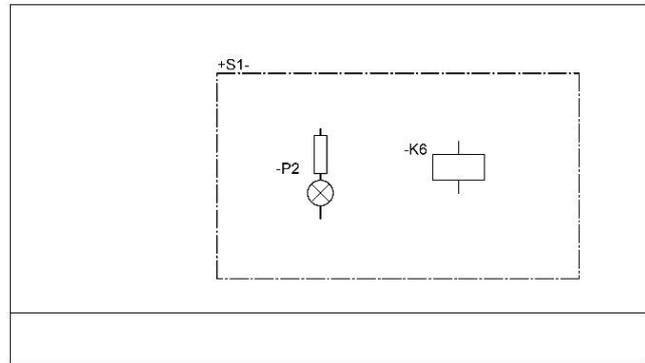
Here you simply read the frame's designation and add the symbol's designation.



### 8.4.2 One reference designation with more aspects

Object	Ref. designations
Lamp	+S1-P2
Coil	+S1-K6

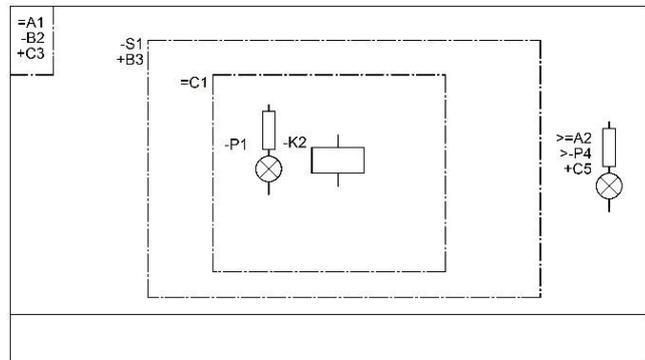
Note, that the ref. designation on the frame ends with the prefix for the next aspect (trailing prefix). This means that the designation is a combined ref. designation. This shows that the reference designation consists of more than one aspect.



### 8.4.3 Three reference designations

Object	Ref. designation
Lamp	=A1.C1/-B2.S1.P1/+C3.B3
Coil	=A1.C1/-B2.S1.K2/+C3.B3
Lamp	=A2/-P4/+C3.C5

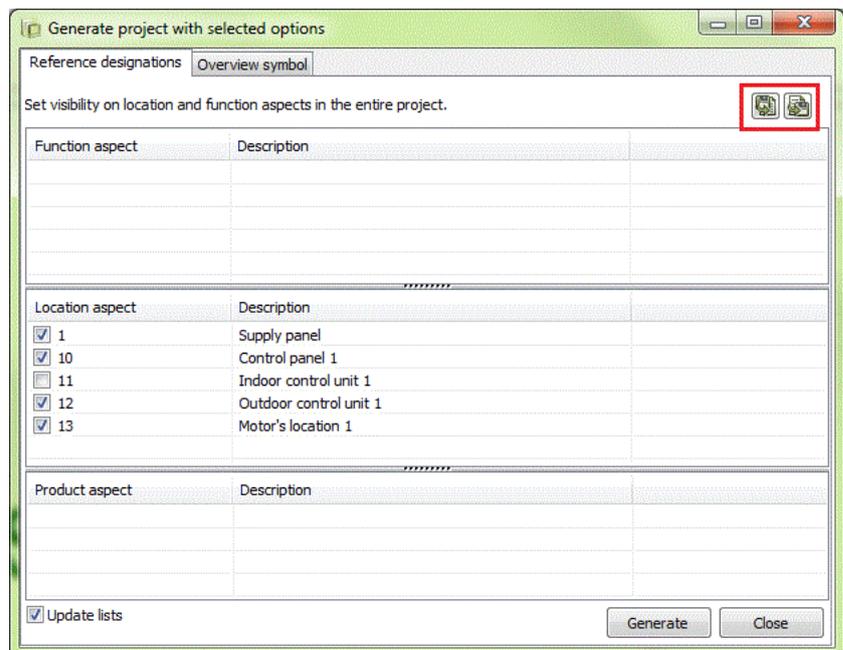
Note the lamp to the right: the > sign in connection with more aspects mean that an object doesn't belong to the page's or the frame's designation.



## 8.5 The Options module

The Options module illustrates very well how you can use reference designation to add or delete options.

The module has a new function, which allows you to import or export an Options settings, this means that you can save a setting, which contains different models. Here it could be the motor control (demo project) with all control, with indoor or outdoor control.

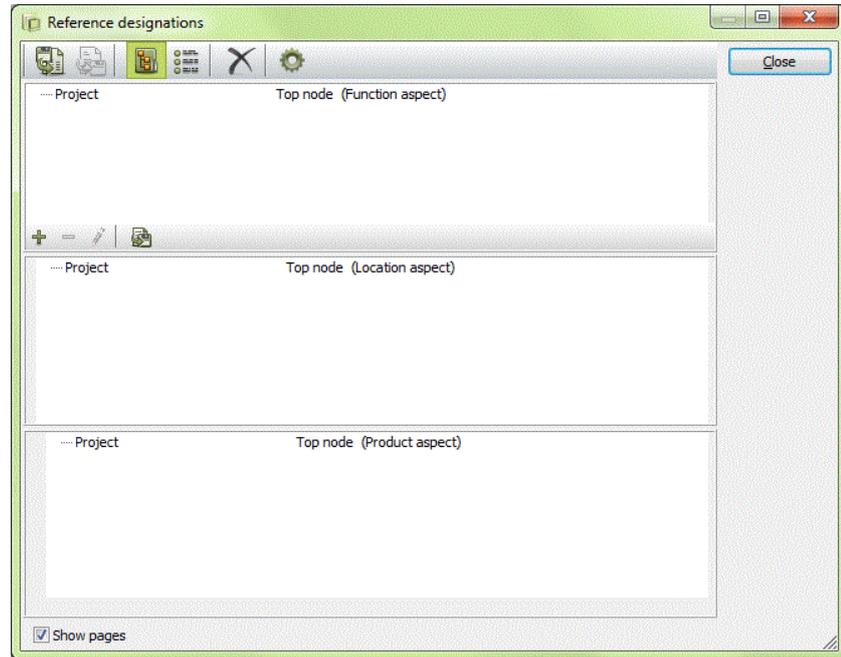


## 9 How do you work with ref. designations in Automation

When you press the icon  you will get this dialog:

Here you see the ref. designations that are present in the current project. They split into Function aspect, Location aspect and Product aspect.

You can create designations directly in each aspect, you can import a list from eg Excel, you can show all designations in a tree-structure or in a list-view, delete designations and finally, you can change settings for your reference designations in the current project.



In Automation version 18, we have made some changes to the way that you work with ref. designations in the program; one of the changes makes it possible to see whether you conform to the requirements in IEC 81346.

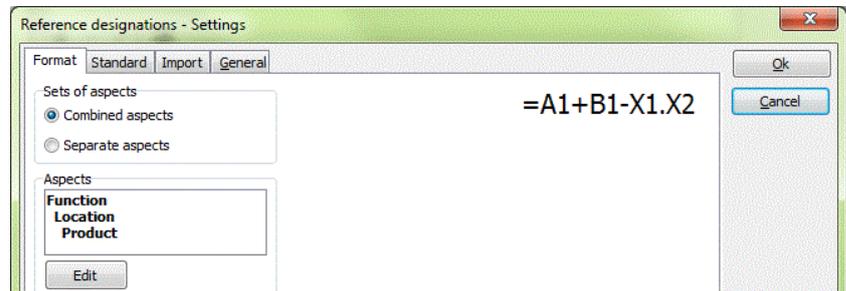
## 9.1 Format for reference designations

In this tab, you choose how to present your ref. designations.

### 9.1.1 One reference designation – Combined aspects

You can choose to have one reference designation.

When you select this setting, all designations are in one line in the diagram and in the lists.



#### 9.1.1.1 Select aspects

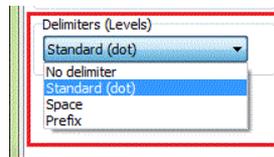
Ref. designations are in a tree-like structure. When you select one ref. designation, you have a tree with the Function at the top, Location in the middle, and Product at the bottom.

When you press the button Edit, you can select which aspects you want to use, and you can change their order.



#### 9.1.1.2 Delimiters

You can choose between different delimiters between the levels. The default choice is the dot (as earlier).



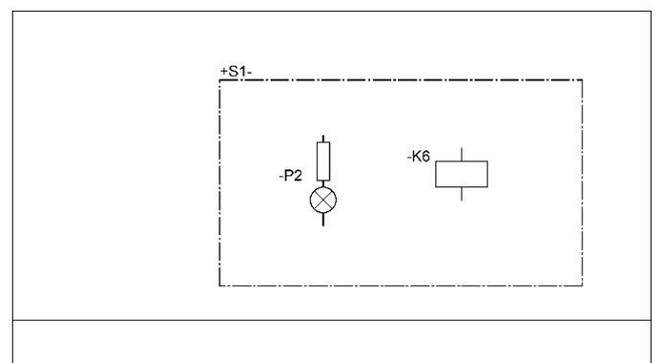
#### 9.1.1.3 Reference frames

If you have reference frames in the project or on the page, they will display like this:

Object	Ref. designations
Lamp	+S1-P2
Coil	+S1-K6

Note, that the ref. designation on the frame ends with the prefix for the next aspect. This means that the designation is a combined ref. designation. This shows that the reference designation consists of more than one aspect.

You can select *not* to conform to all display requirements in the standard, see 10.

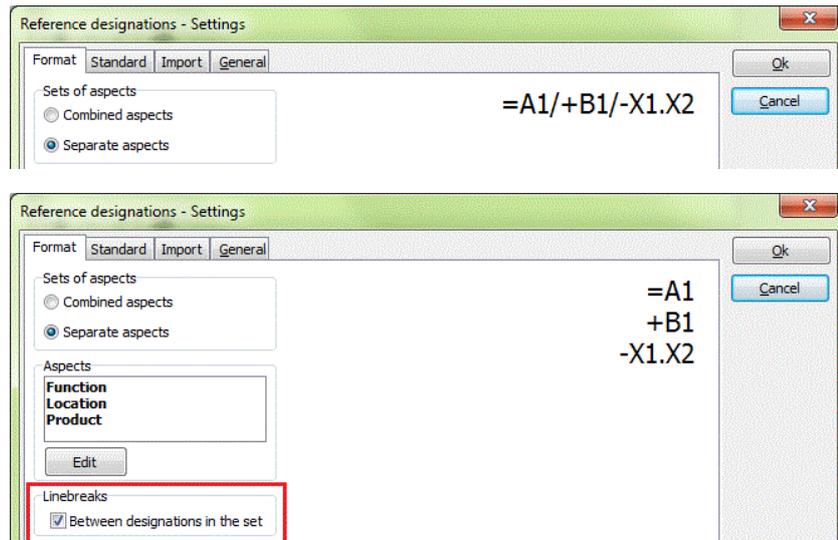


### 9.1.2 More reference designations – Separate aspects

You can choose to have more reference designations in a project.

When you have more ref. designations, you can display them *either* on separate lines or with a / between each. You select your default display type in format tab.

You can change the display type on individual symbols and reference frames. Lists will always use the /.



#### 9.1.2.1 Select aspects

When you press the button Edit, you can select which aspects you want to use, and you can change their order. See figure in 9.1.1.1.

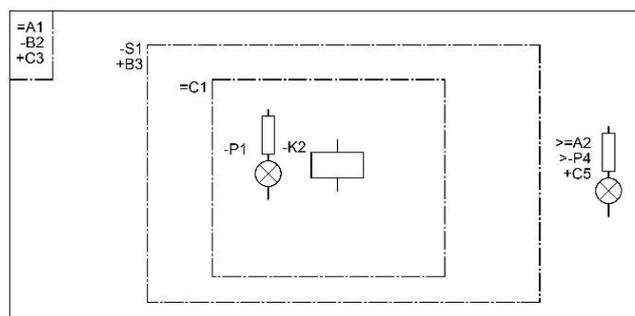
#### 9.1.2.2 Delimiters

You can choose between different delimiters between the levels. The default choice is the dot (as earlier). See figure as 9.1.1.2.

#### 9.1.2.3 Reference frames

When you have reference frames, you must know the rules for reading reference designations. Page ref. designations behave the same way.

Object	Ref. designation
Lamp	=A1.C1/-B2.S1.P1/+C3.B3
Coil	=A1.C1/-B2.S1.K2/+C3.B3
Lamp	=A2/-P4/+C3.C5



You read ref. designations aspect by aspect. That means, that you read = with =, + with + and - with -. The sign > indicate differences between the frame's (page's) designation and the symbol's designation. A symbol might have more than one >, simply because more aspects have differences.

## 9.2 List export

The format you select as display format will also become your export format for list export.

## 9.3 List import

When you import a list, it might not have the same format as your display format.

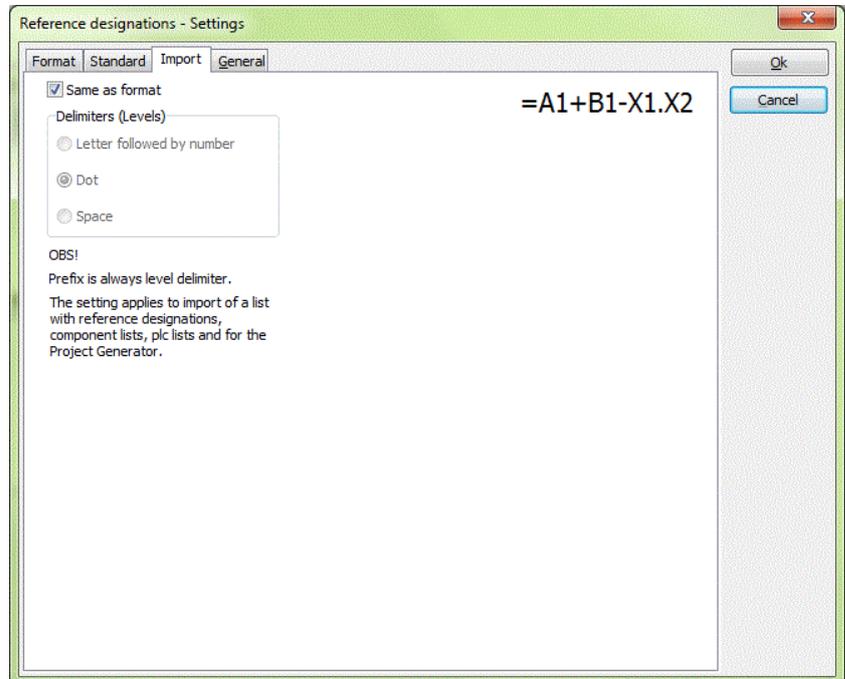
That means that you can choose a different setup for list import.

The import format covers all lists that you import to Automation:

- lists with ref. designations
- plc lists
- component lists
- files for the project generator

When you import a list, you should be aware that it must have the default prefixes on

ref. designations; otherwise, the program will not recognize them as ref. designations. That means, that even though you don't want to show the prefixes in your documentation, you must include the prefix in the list, as long as you want the designations behave correctly.

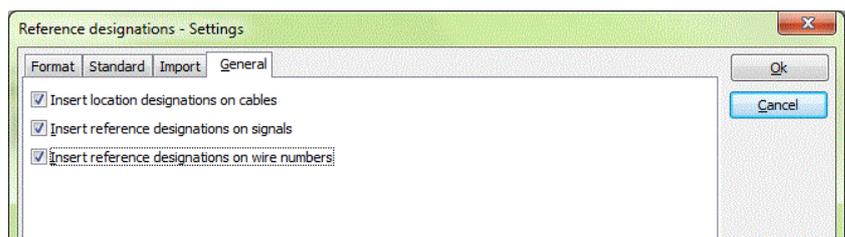


## 9.4 General settings

No changes in this tab ;-)

Here you can choose to apply location aspect to your cables.

You can also choose to apply ref.designations to signals and wire numbers.



## 10 Open an old file

In the former versions of Automation, it was possible to display ref. designations in ways that didn't conform with the requirements in IEC 81346.

This is also possible in this version. There are two reasons for this, one is that we want to be able to support other standards, the other don't want to – unintentionally – change old projects. The new thing is that now you also see how your projects don't conform to this standard, and that you can make them conform by simply selecting or deselecting an option.

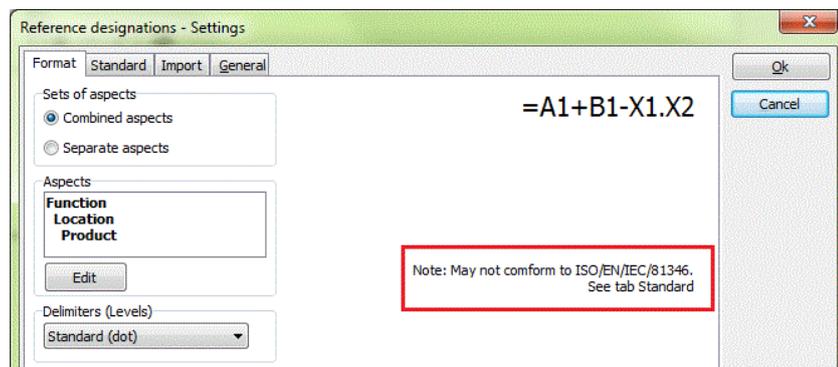
### 10.1 Open an old file

When you open an old file, the program reads the project's reference designations and it interprets the way they are used.

Generally, we assume that old projects' ref. designations are the following way:

- Delimiters are dots or Capital letter followed by a number. In ver 17 you could choose between the two options, in older projects both of them were active.
- Prefixes (=, + and -) are always level dividers.

If you have any nonconformities in the old file, you will see this warning in format tab.



Go to the Standard tab to see which you have. If you haven't selected any of the options here, your project displays reference designations in accordance with IEC 81346.

Your options for nonconformity change depending on your format – combined or separate aspects.

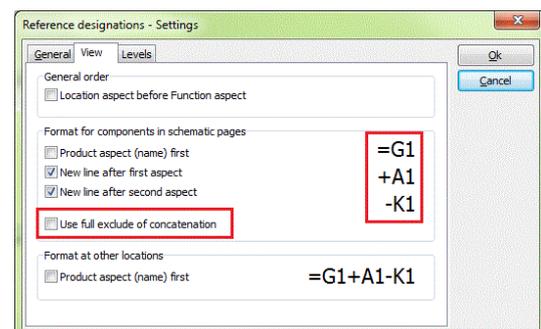
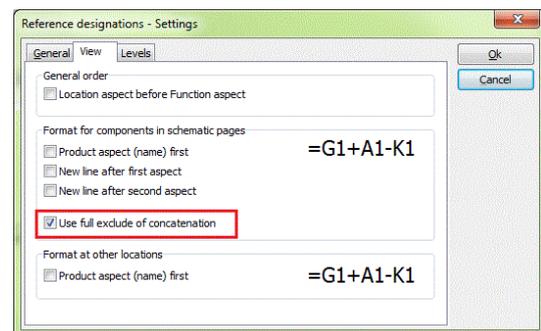
### 10.2 General import formats

Some of our import formats are easier to understand, when you see the old v.17 setting:

In version 17 the View format was the Ref. designation format, and not just a view-format, but some of the view options actually means something, according to IEC 81346: A reference designation in one line has combined aspects; a ref. designation in three lines has three separate aspects. And each method has different writing conventions.

We import projects without 'Use full exclude...' as 'Separate aspects'.

This leads to this non-conformity: Don't display slash between aspects'.

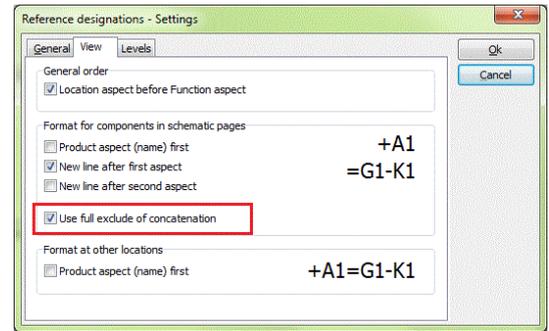
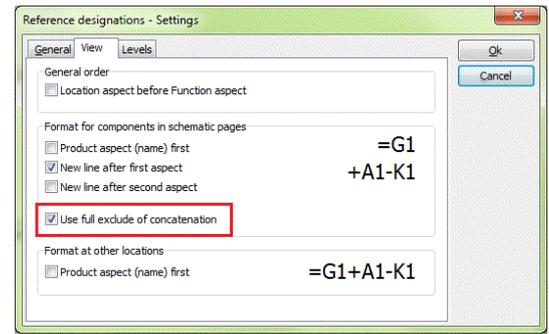


We import projects with selected 'Use full exclude of concatenation' as 'Combined aspects'.

This leads to this non-conformity: Combined aspects miss the trailing prefix.

Combined aspects with line breaks are not allowed according to IEC 81346. You can allow this in your projects: Line breaks in combined sets.

Or if your old project was without 'Full exclude ...' you could allow 'Line breaks after individual aspects'.



Shortly, no matter how you made your old projects, you will see some nonconformities in your new project; you can choose to keep them and the project will look the same, or you can choose to make your display correct. It is your choice.

If you want to KEEP those settings, simply save the file and continue your work.

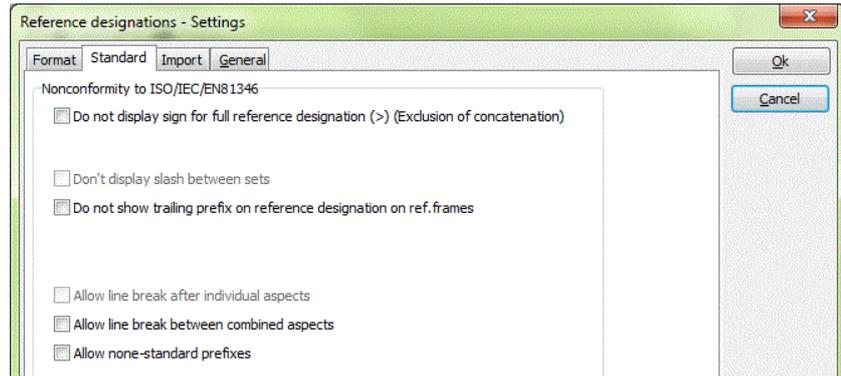
If you want to CHANGE the settings, simply delete the 'nonconformity' checks, and the program will automatically change the file. This is also true, if you change between one and more ref. designations.

Below you will find a more thorough description of the options.

### 10.2.1 Nonconformity options with combined aspects

Combined aspects, which is one unique name for objects, a name or designation that consists of all three aspects. The writing conventions are:

- ALWAYS one line, in the diagram and lists
- Trailing prefix in ref. frames on in page headers.



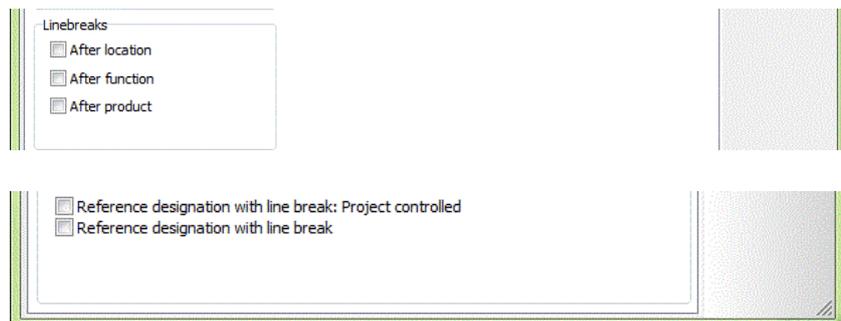
#### 10.2.1.1 Do not display sign for full reference designation (>) (Exclusion of concatenation)

If you select this option, you will not display the > sign for full ref. designation. The > in front of symbol names means that the name is the full name; that you don't concatenate (add or name together with) the designation on a reference frame or the page's ref. designation.

#### 10.2.1.2 Allow line break between combined aspects

This setting allows line breaks after one or more of the aspects on the Format tab.

No selections on the Format tab means that your default designation is without line break – as according to the standard; still you have this option on individual symbols and reference frames:



#### 10.2.1.3 Allow non-standard prefixes

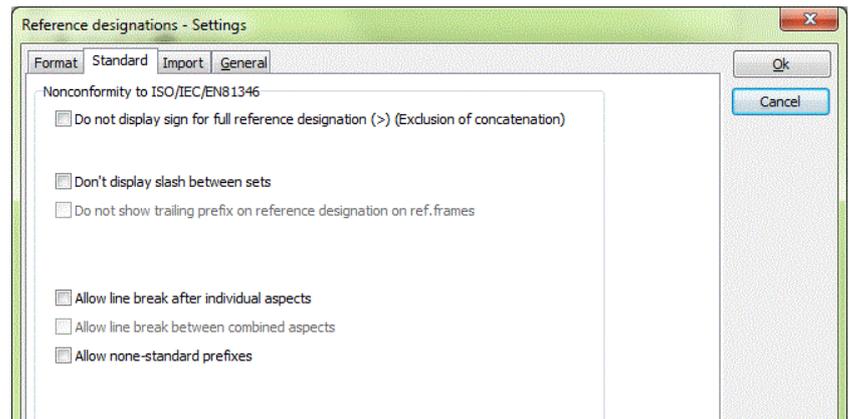
Non-standard prefixes are all prefixes different from =, + and -. No prefix is also a none-standard prefix. Standard prefixes MUST be used on ref. designations when you import them in a list (component lists, plc lists etc.).

Normally, that is without ticking this box, the program shows the standard prefixes. If you want to use other prefixes, you must type them yourself when you create a designation, alternatively when you edit imported ones.

## 10.2.2 Nonconformity options with separate aspects

Separate aspects, which means (up to) three designations for objects: a Function, a Location and a Product designation. The writing conventions are:

- Aspects separated with a / or a line break
- The > (full ref. designation) is per aspect and not for the entire set.



### 10.2.2.1 Do not display sign for full reference designation (>) (Exclusion of concatenation)

If you select this option, you will not display the > sign for full ref. designation. The > in front of symbol names means that you don't concatenate (add or name together with) the designation on a reference frame or the page's ref. designation.

When you have three separate aspects, you have three ref. designations, that means that you can have up to three > on one symbol. See 8.4.3.

### 10.2.2.2 Don't display slash between sets

This option allows you to display multiple reference designations as one single reference designation, which means that you can write it in one line. In this way, projects will have the same visual ref. designations and display of them as earlier.

### 10.2.2.3 Allow line break after individual aspects

This option is made to fit with the 'old settings' as you see them in 10.2.

### 10.2.2.4 Allow non-standard prefixes

Non-standard prefixes are all prefixes different from =, + and -. No prefix is also a none-standard prefix. Standard prefixes MUST be used on ref. designations when you import them in a list (component lists, plc lists etc.).

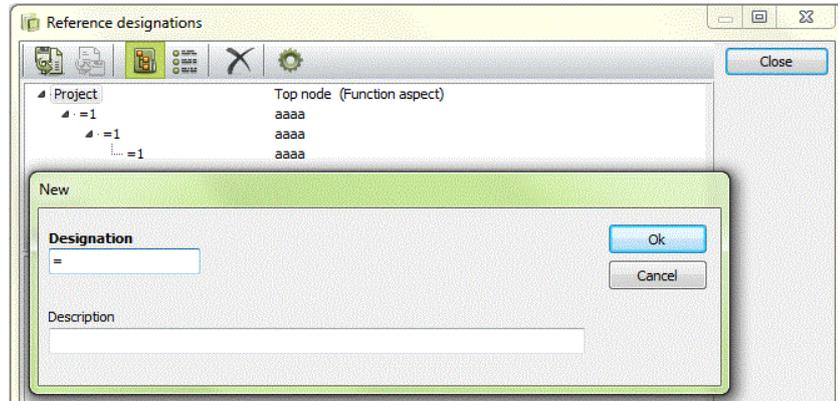
Normally, that is without ticking this box, the program shows the standard prefixes. If you want to use other prefixes, you must type them yourself when you create a designation, alternatively when you edit imported ones.

### 10.3 Create ref. designations

You can import a list from e.g. Excel. You can find an example file (ref-designations\_uk.xlsx) in the List folder. If you want to import an Excel file, it must follow the import format you defined in 9.3. Please note, that all designations MUST have a standard prefix (=, + or -). Otherwise, the program will not know which aspect they belong to.

It is also possible to create designations directly in this window:

You can type designation by designation, but it is also possible to type =2.2.2, and then the program will automatically create =2 and =2.2. This is new and could save you some time.



### 10.4 List view – now with Edit function

You can see all ref. designations in a list view – as always – but it is no longer possible to edit the designations in this view.



### 10.5 Start a new project – default projects

When you start a new project, we have decided that it has

- separate aspects
- dot is a delimiter between levels
- no nonconformities.

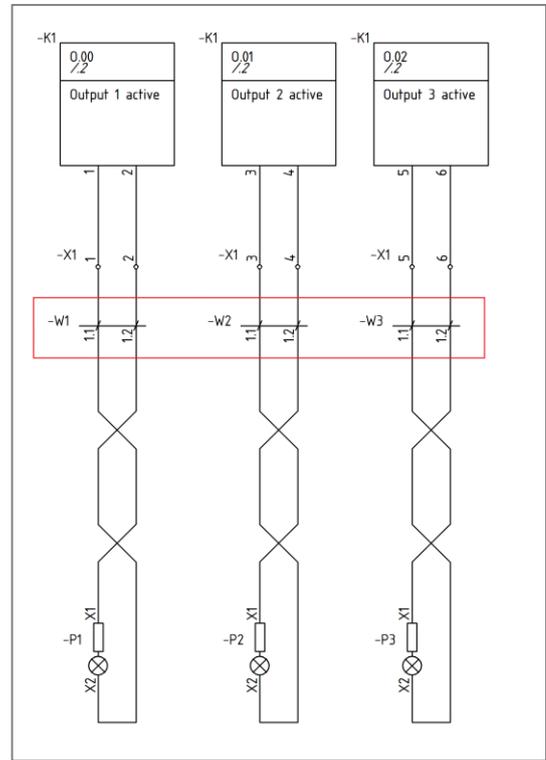
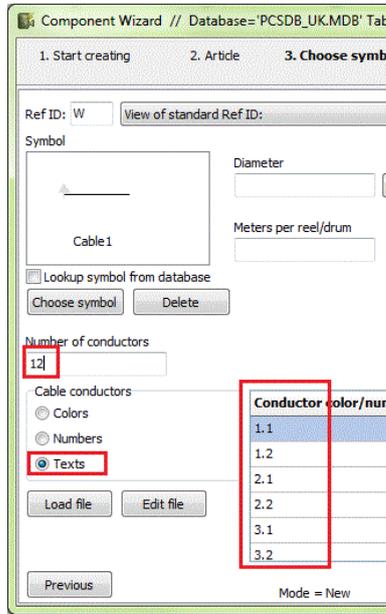
## 11 Twisted pair cable – how to create them and work with them

Here you see three pairs from the same cable:

Conductor names are Pair.Conductor.

Create this in the cable definition in the database.

When you create the cable like this, you will get a list where the conductors come in the right order.



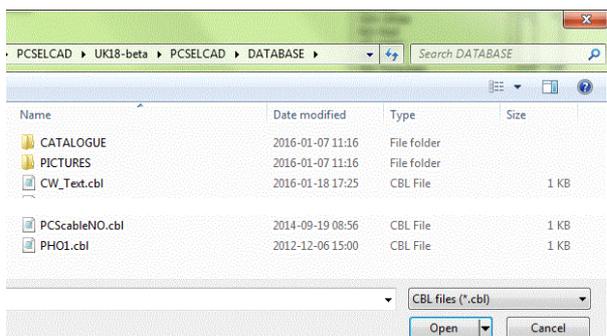
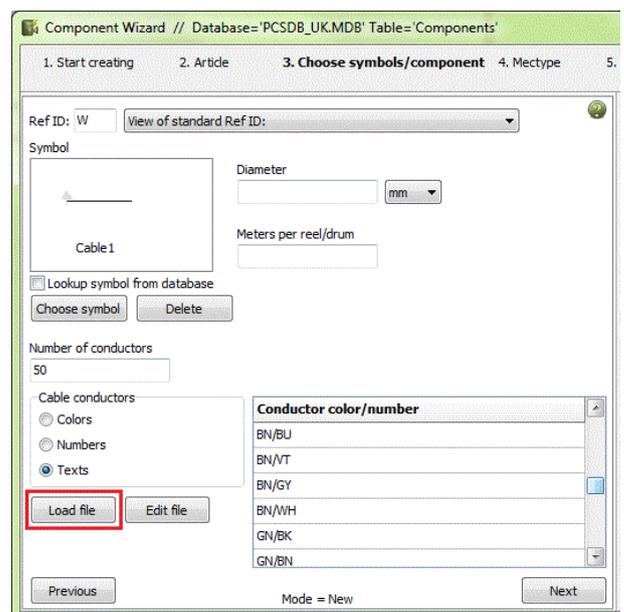
### 11.1 New function in Component Wizard

When you create a cable with the Component Wizard, you can also use the Load file option.

When you press the button, it starts looking for a \*.cbl file.

\*Cbl files contains lists with conductor names, and only a few of them allows you to have all your cable definitions in an easy way.

It is also possible to edit one of the \*.cbl-files.



## 12 List with letter codes and hints

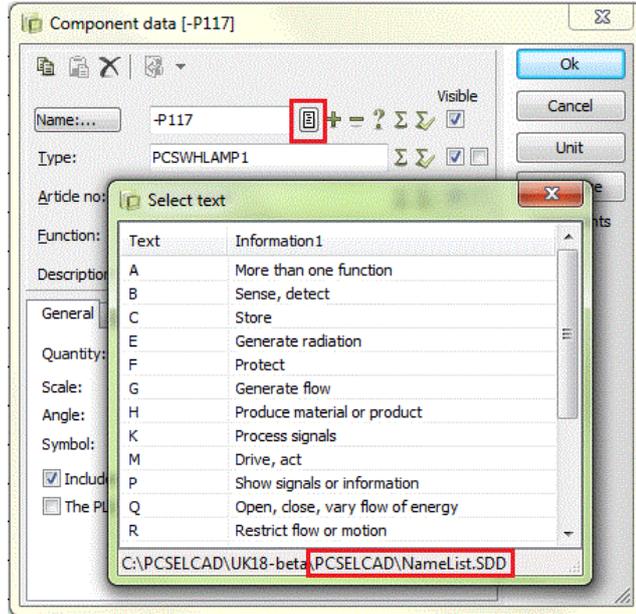
You can get a list with letter codes, which you can pick from when you place a symbol.

The symbol will still come with a letter, just like most components will, but it can give you a hint to the meaning of the letter code.

The button is active, when the file NameList.sdd is in the PCSELCAD folder.

The format for the file is:

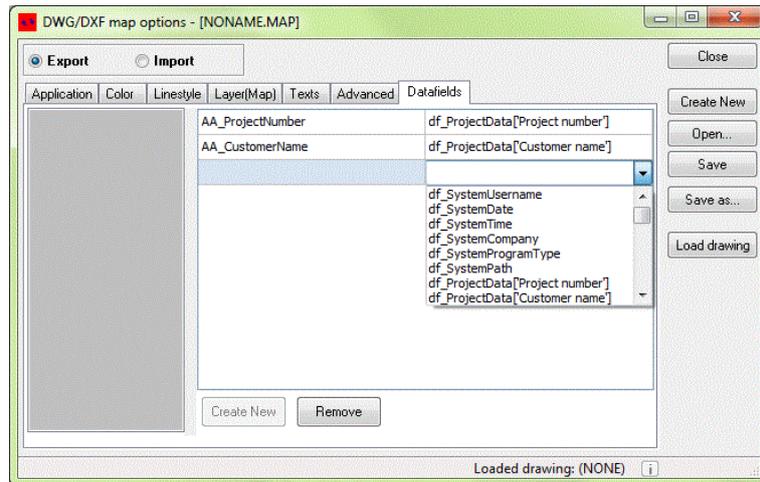
Name;Info1;Info2.



## 13 Export of dwg/dxf-files

It is now possible to export data fields from the PCSCHMATIC file to another format (dwg/dxf) to make it possible to interface to document management programs such as MDoc.

We have added a tab in MAPTOOLS where you can map data fields from the two applications.

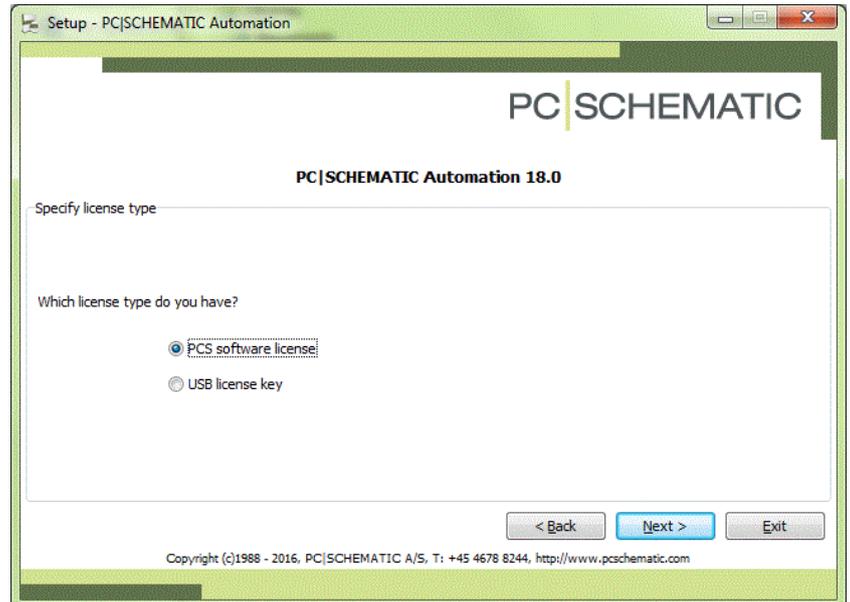


## 14 Extended installation routine

We have increased our license types, and sometimes we get questions about this issue.

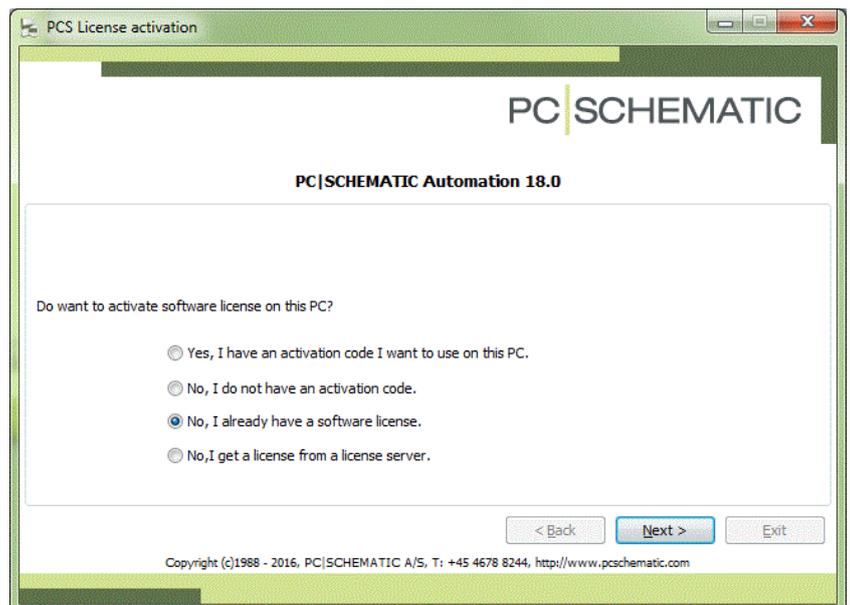
Below you can see the extra options that this has given in the installation routine.

We differ between hardware and software licenses:



If you selected the software license (default), you have these choices. The program will guide you through the rest:

1. You have or will get the activation code by email. If you want to use this option, select and click next.
2. You will receive the code later.
3. I have selected option 3, as I have a code already, and I don't need to install or activate it again.
4. This option is very similar to the usb-net license. You don't have to activate the license.

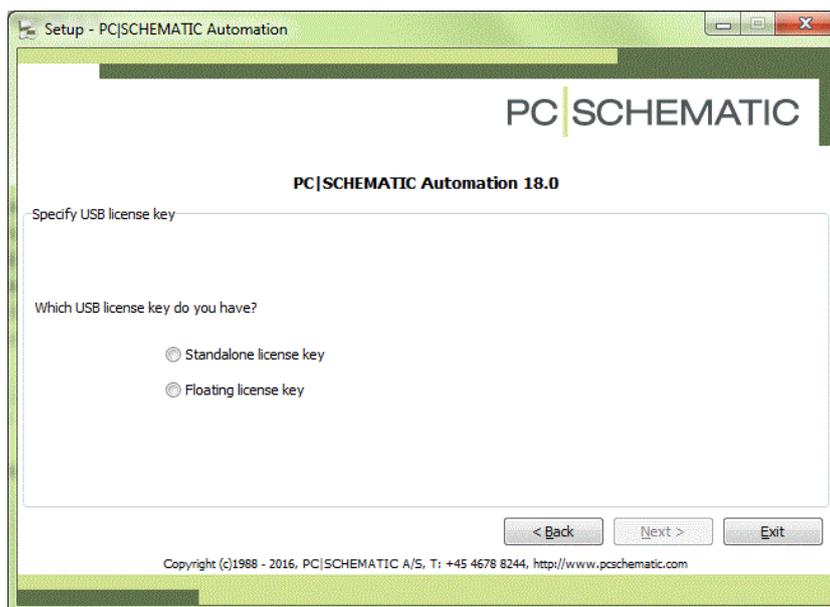


If you select the USB license key, you get those choices:

Select the standalone license key if you place the key in your own pc.

Select the floating license key if you place the key in a server.

The license keys are physically different, on the inside.



## 15 Bits and pieces

Every year we make small improvements and changes; this year's bits and pieces are in this section. Some of them may be in your program already (ver 17), in that case you can see \*\* in the headline.

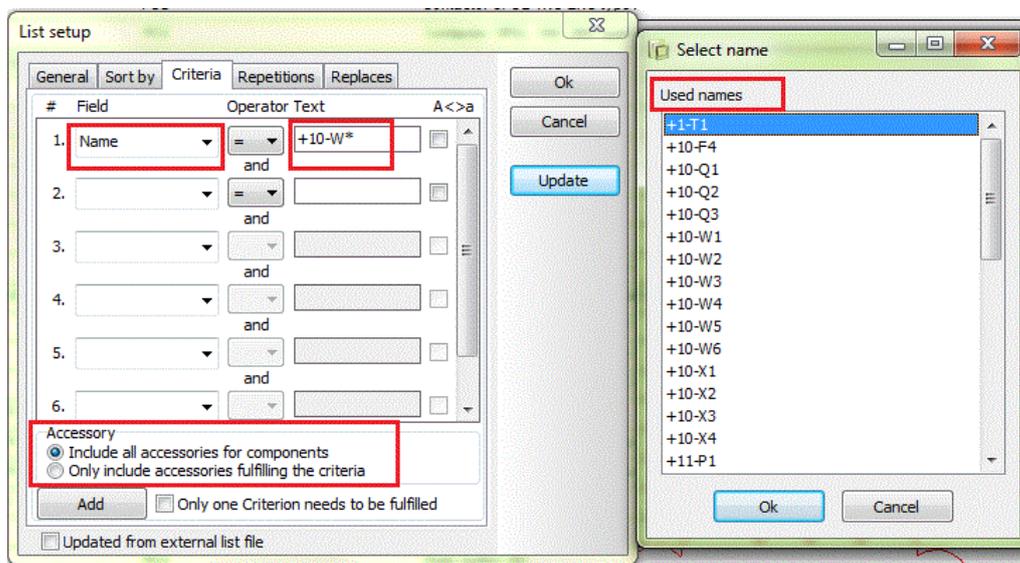
### 15.1 Select list criteria

Now you can get help, when you select criteria for a list.

When you select a data field, you will get a list with all used values in this data field in the project. Here I have chosen the data field 'Name' and I get a list of all used names in the project.

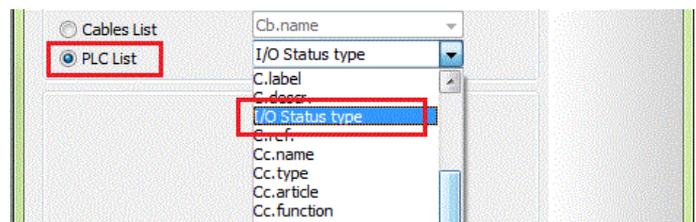
I can use this as a Help when I want to select a criteria and I can combine it with a joker (\*).

The function is valid for Name, Type, Article no, Function and Ref. designations. The function is valid in the built-in lists and in list file export. The criteria might also add to accessories.



### 15.2 IO status type is now in the PLC-liste

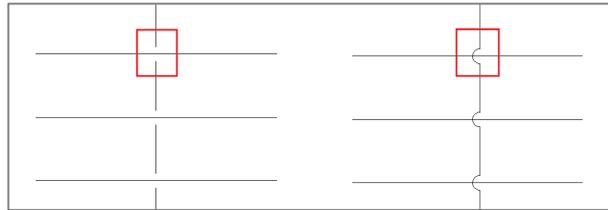
The data field I/O Status type can now be included in the PLC List.



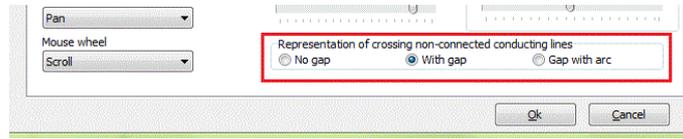
### 15.3 Crossing lines

It is now possible to show conducting lines without connections in different ways:

It can be as before, where you simply see a crossing without dot, or now you can have a small gap or a small arc.

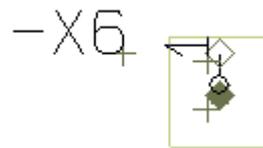


You make the setting in Pointer/Screen.



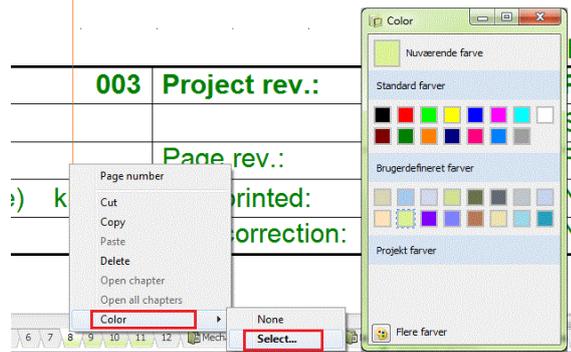
### 15.4 Larger 'catch' area on small symbols

The 'catch' area that the light green 'box' around the symbols is bigger, making it easier to catch terminals, signals and other small symbols.



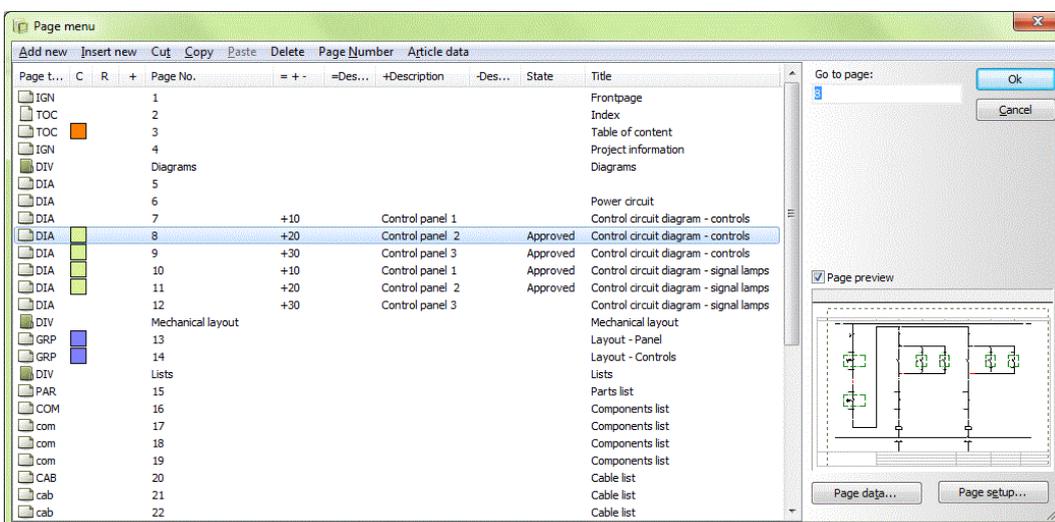
### 15.5 Page tabs can have colors

It is now possible to select a color for each page tab: Right-click on a page tab, select Color.



### 15.6 Easier overview in the Page menu

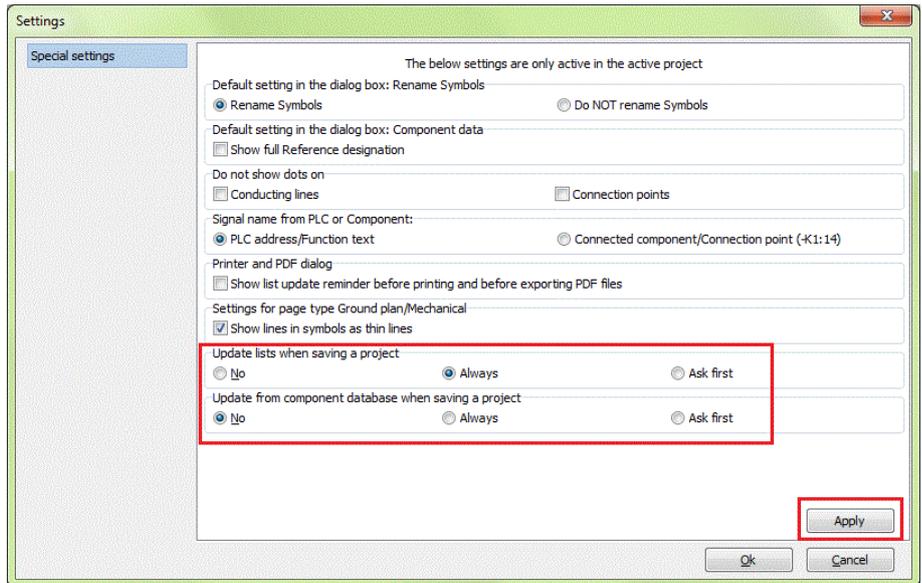
State titles have their own column in the page menu. Colors also have their own column, and you can select colors in the right click menu in the page menu.



### 15.7 Settings – Special settings

The two selections are saved in the program and not only in the current project.

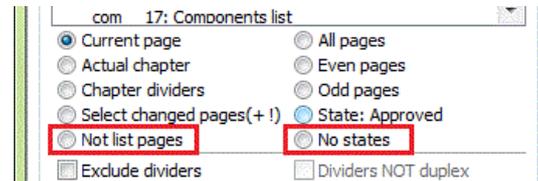
Remember press the Apply button for each selection.



### 15.8 More functions in the printer dialog

You have two more functions when you want to print from the project:

- Print all non-list pages
- Print all pages without states in the drawing header

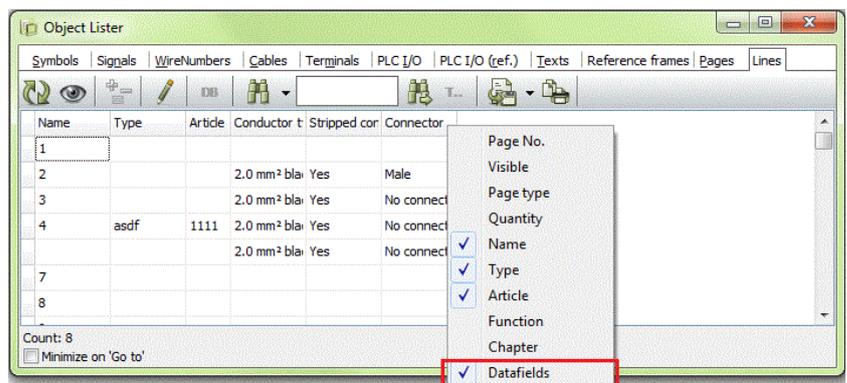


You also have a warning about list update before the printer starts.

### 15.9 Lines with data in the Objectlister

The Objectlister (F7) now includes all lines with line data.

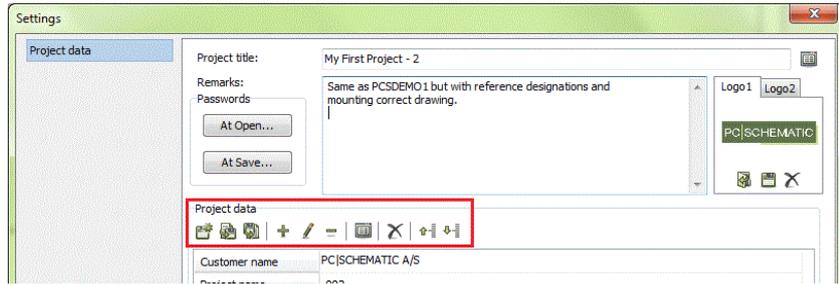
If your lines have line data fields, you can right-click in the headline section and include Datafields.



### 15.10 New icons Project data and Page data

We have made icons in Project data and Page data for the following functions:

- Insert data field
- Edit data field
- Remove data field
- Values - data list for data field
- Remove unused data fields
- Move data field up
- Move data field down



Until now, you could find the functions in the right-click menus.

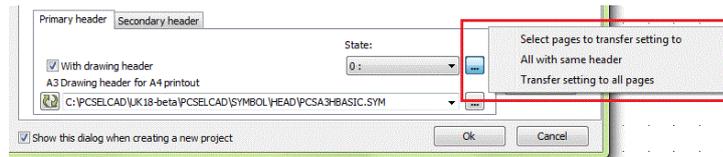
The icons for Save as default, Export to file and Import data fields from file are moved from their former position.

### 15.11 Right-click functions are now visible

The functions below have been 'secret' until this version; they were only found in the right-click menus, now they have a button.

In page data:

Transfer selection of state to other drawing headers. In primary and secondary header

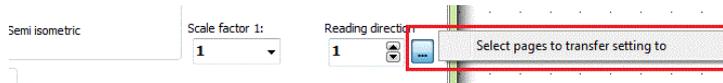


In page setup:

Transfer page function to other pages.



Transfer reading direction to other pages.

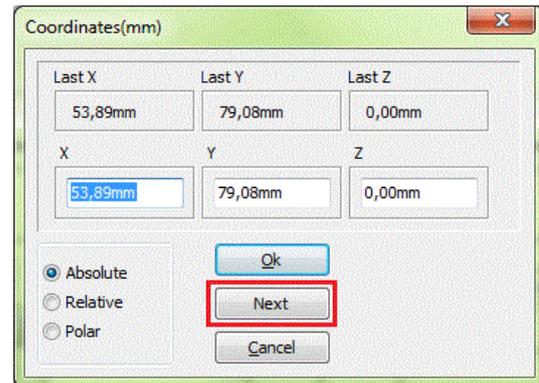


Transfer list setting to other pages.



## 15.12 New function in the Coordinates dialog

When you type coordinates in the Coordinates dialog, you can now use the Next button instead of opening the dialog again to type the next set of coordinates.



## 15.13 New data fields

Until now, the data field Description was only in Parts- and Component lists.

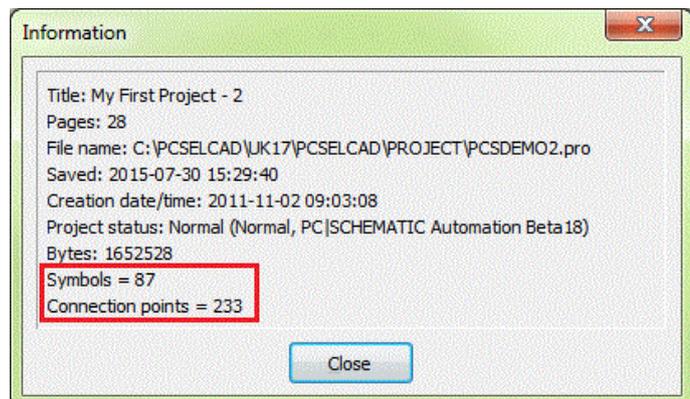
Now it is also in Terminal, Cable and PLC lists, that is, that you can select the (mapped) description for the terminal, cable or PLC.

The extra data field is in all built-in lists, list file exports, and in Insert data field.

## 15.14 Project information

Project information, which you find in the right-click menu on the page, now shows the number of used symbols and connection points.,

The count is by the same rules, as we use for the limitations in Automation 40 and Automation 200, which means that we don't count terminals, cables, signals, wire numbers and data symbols. We don't count the first mechanical page, either.



## 15.15 All demo-files in new folder

We have moved all our demo files into a new folder: Project/Demo.



