PCschematic® ELautomation

Version 10 News

PCschematic® ELautomation Version 10.0 News

List of Contents

Windows Vista 3
New Look 3
Transparent Windows 5
Reference Designations 6
- Import/Export 6
- Exclude Concatenation 8
- Reference Designations on Cables 9
- Used and Not-used Reference Designations 9
- Component Data Dialog 10

Lists 10
- Data Fields for Description of Reference Designations 10
- The Data Field Last change on Lists 10
- List Symbols on a Diagram Page 10
- Data Field for Reference to Mechanical Pages 11
- Exploding Unit Pages in a Components List 11

Wire Numbering 12
- Wire Numbering on Selected Pages 12
- Model and User Variables 12

Database 13
- The Repetition Character in the Database Field (PCSTYPE) 13
- Activating References on Connection Points 13
- Activating References on Symbols 13
- New Setting for the Description Field in the Database 14
- The Database Program 15

Symbols 16
- References on Connection Points 16
- Titles on Symbol States 16
- Data Fields Included in a State 17
- Deleting Symbols 18

Drawing Headers 19
- Using States in Drawing Headers 19
- Data Fields Showing Data from Page and Project Data 20

The Object Lister 20
The Pick Menus 21
- Keyboard Control of the Pick Menus 21
- Symbols With States 21
- Symbols With Component Data in the Pick Menu 22

Find/Replace 24
New Page Function 24
Texts 25
- Swop Texts 26

Mechanical Pages 27
- Spacing Symbols 27
- More Than One Type of Mechanical Pages 27

Measurement 30
- Lines 30
- Circles and Arcs 30
- Text Property in the Measurement Function 30

New Functions 31
- View Jumper Links and View Signal 31
- Save Logo to File 31
- The Open Dialog Box 32
- Re-addressing PLC's 32
- Text Descriptions on PLC Connection Points 33
- Space Symbols 33
- Selecting and Pointing Lines and Circles 34
- Cursor Position Returns to the Object 34
- Transfer Changes to All Symbols in the Component 34
- The SubDrawing Window 35
- Selecting Symbols Associated With Lines 35
- Symbol Data Fields in the Popup Information 35
- Start and Stop Drawing Mode 36
- New Feature for Moving a Line 36

The Layer Dialog Box 36
The Symbol Menu 37
Symbol Generator 39
Draw Order 41
Project Generator 41

Windows Vista

Version 10 is a true Windows Vista program. When running ***PCschematic® ELautomation*** on Windows Vista, you will note that the program looks differently.
screen dump

New Look

The Tool bar and the Pick menu can be set to the so-called Gradient look as known from the Microsoft Office programs.

screen dump

Or:

screen dump…

Dialog boxes do not change color, but will keep the new gray look.

Gradient look can be turned on/off in Settings | Pointer/Screen.

The color can be set in the Colors dialog box. Colors from Windows 2000, XP and Vista are pre-designed as shown in the picture below.

The colors are user defined, so you can change them as you like.

Transparent Windows

It is now possible to set transparent program windows for windows that can stay on top, ie Object Lister, Show available, Read component list, Find and replace and the Design check result window,. In this way you can see what is behind the window.

Activating a transparent window, e.g. by clicking, makes the window non-transparent. By clicking in the project, the window will return to transparent.

This function is turned on/off in Settings | Pointer/Screen. Here you also control how transparent you want these windows.
The best way to adjust this is to open e.g. the Object Lister, select symbols so you see some data. Then select Settings | Pointer/Screen. The setting button can be moved by dragging with the mouse.

Reference Designations

Import/Export

Reference designations can be exported to and imported from Excel or .csv files.

Right-click in the Function window and the menu below will appear. If you want to export Locations, you just right-click in the Location window.

If reference designations have not been defined you will see the button Import.

Below, the Excel file format is shown.Reference designation are in the first column and Descriptions are in the second column.

	A	B
1	=G1	Conveyer belt control
2	+A1	Controlpanel for conveyerbelt
3	+A2	Main switchboard for conveyer belt
4	+W1	Conveyer belt

The CSV format is shown below:

=G1;"Conveyer belt control"
+A1;"Controlpanel for conveyer belt"
+A2;"Main switchboard for conveyer belt"
+W1; Conveyer belt

Importing

When importing you will see the above dialog.
Reference designations marked **+** are new, and will be added to the project.

Reference designations with a green square are already in the project and will not be imported.

Reference designations with a yellow square are already in the project but with different descriptions. They will be imported if not removed from the list.

If you do not want to import some of the reference designations, you can delete any selected line/lines using the Remove button.

Exclude Concatenation

It is now possible to use 'full exclusion of concatenation' on reference designations on symbols. This function is selected in the Reference Designation Dialog box.

The examples below show a page with the reference designation =G1+A1.

Version 9 example: Full exclusion of concatenation NOT USED.

1. >-P1. The letter > indicates that you see the total reference designation, which is correct.
2. –P1. As the symbol name does not start with the letter > one assume that the reference designation of the page must be concatenated (added) to –P1. The symbol name should then be =G1+A1-P1. As shown above this is not correct. **=G1 is not a part of the symbol name**.
3. +W1-P1. As the symbol name does not start with the letter > one assume that the missing part (=G1) of the page reference designation must be concatenated (added) to +W1–P1. The symbol name should then be =G1+W1-P1, which is correct.

Example no. 2 will lead to a misunderstanding

Version 10 example: Full exclusion of concatenation used as described in the new standard.

1. >-P1. The letter > indicates that you see the total reference designation, which is correct.
2. >+A1–P1. The letter > indicates that you see the total reference designation, which is correct.
3. +W1-P1. As the symbol name does not start with the letter > the missing part (=G1) of the page reference designation must be concatenated (added) to +W1–P1. The symbol name should then be =G1+W1-P1, which is correct.

Using 'Full exclusion of concatenation' will not lead to misunderstandings.

Reference Designations on Cables

Location designations are not always applicable for cables as they go from one location to another location. Thus, you can exclude the use of the reference Location on cables. See below how to turn this on/off.

Used and Not-used Reference Designations

To show where reference designations are used, you can click the button Mark Used, right-click or use the <space-bar>.

Right-clicking on one of the listed pages will allow you to navigate to the page, or to see a list of the objects using the selected reference designation on the selected page.

Below you will find the list of objects using the Function aspect =G1.

Object	Name	Position
Page	2	The page
Ref.Frame	=G1+A1	x=52,50 y=220,00 z=0,00mm
Ref.Frame	=G1+A1	x=262,50 y=125,00 z=0,00mm
Ref.Frame	=G1+A1	x=122,50 y=125,00 z=0,00mm
Ref.Frame	=G1+A1	x=192,50 y=222,50 z=0,00mm
Symbol	=G1+A1-P1	x=157,50 y=100,00 z=0,00mm
Symbol	=G1+A1-P2	x=297,50 y=100,00 z=0,00mm
Symbol	=G1+A1-P3	x=332,50 y=100,00 z=0,00mm
Symbol	=G1+A1-S1	x=87,50 y=202,50 z=0,00mm
Symbol	=G1+A1-S2	x=87,50 y=177,50 z=0,00mm
Symbol	=G1+A1-S3	x=227,50 y=202,50 z=0,00mm

As shown above both the page, reference frames and symbols on page 2 are using the Function aspect =G1. The list can be printed and/or saved to a file.

Component Data Dialog

It is now possible to select the option (none) directly while changing the reference designation of a symbol.

Lists

Data Fields for Description of Reference Designations

New data fields have been created for description of reference designation for Int/Ext connections and for Int/Ext cables in the terminal list setup, as well as for description of reference designation for To/From components in cable and connection lists.

The Data Field Last change on Lists

Updating lists will only change the data field Last change if a change has been made. In version 9 the data field Last change was changed on all pages even if only one page was changed. If e.g. a part number is changed on one component it normally only changes one line on one of the pages in a list.

List Symbols on a Diagram Page

It is now possible to place a list symbol on a diagram page e.g. for displaying a components or a parts list. The rule is that the symbol must be of the symbol type **Head**. The symbol type can be set in Create new symbol, see below.

- The grid can now be turned off in Settings | Pointer/Screen | 'Grid on list pages'.

Data Field for Reference to Mechanical Pages

A new data field has been added to components and parts lists containing the page no of the page where the mechanical component is located.

Function aspect and/or the Location aspect can be printed with the page number.

Exploding Unit Pages in a Components List

When Unit drawings are exploded in a components list, the component names from the Unit drawing are shown as sub names on the component list.

The component S2 is described on the Unit drawing as the parts 1, 2 and 3.

20	A11		AIA008
21	A12		AIA007
22	S2.1		5703302007818
23	S2.2		5703302098175
24	S2.3		5703302004176

Wire Numbering

Wire Numbering on Selected Pages

The wire number function has been added the option 'Select pages…'.

The button 'Select pages…' activates the page menu where relevant pages are selected. It is only possible to select pages shown with a green icon.

Model and User Variables

When creating sub drawings with model data (variables), you can choose if the variable should be controlled by model choice. If the value in the **first** variable is set to /? the model choice will not change this variable but use the value found in the sub drawing. In this case –X1 and –M1. While placing this sub drawing you can manually change the values.

Database

The Repetition Character in the Database Field (PCSTYPE)

The PCSTYPE field in the database can now handle repetition (#n) of OR-sections. E.g.:
07-15-01;[07-02-04|07-02-03|07-02-01]#4;[H7302-01|07-02-01#3]#2
OR-sections are shown between [].

Activating References on Connection Points

References can now be turned on/off via the database. In the PINDATA field you can use '/W:1' in the connection point parameter section ('[..]').
E.g. 1[/W:1],2[10.00/I:OP/W:1],3[/W:0]
Connection '1' will have the reference activated.
Connection '2' will get the function text '10.00' (address for a PLC), the I/O status changed to 'Output PLC' and the reference activated.
Connection '3' will have the reference deactivated.

Activating References on Symbols

References on symbols can now be activated and deactivated via the database. In the PCSTYPE field you can add '*' and 'N', 'W' or 'M' to the symbol name.
E.g.:
06-05-04*N If the symbol has been created with reference, the reference will be deactivated.
06-05-04*W If the symbol has been created without reference, the reference will be activated.
06-05-04*M If the symbol has been created without reference, the reference will be activated as Master reference.
This function has no affect on symbols born as e.g. Relay, Open, Closed and so on, unless they are changed using the = sign function.
Controlling the reference can be combined with properties like symbol type and states.
E.g.: 07-02-03=T,P§3*W This symbol will be changed to the symbol type Terminal and PLC (=T,P), changed to state 3 (§3) and the reference set active to With reference (*W).

New Setting for the Description Field in the Database

In Settings | Database | Database Setup | Component data | there is a new setting for Description. If the description points at a database field like DESCRIPT, the information found in the database field DESCRIPT will be shown in symbols popup information, when the mouse rests over a symbol or in the pickmenu. It will also be displayed in the information dialog box.

The pickmenu:

A project symbol:

This Description field has a corresponding data field which can be used in components and parts lists. This can be useful instead of using the database data field DESCRIPT

A drop down button can open URL links on active records in the database dialog box.

If the record contains more than one link, the selected link in the database setup will be default.

URL Alias. In the Database Setup you can define URL alias. The purpose is to define paths to folders which contain catalogues and datasheets for the components in the database. This could be PDF files. An alias must be surrounded with % like %ABB%\ B12-22-00.230V.pdf
Aliases can be used in all database fields except where the syntax FILE= is used.
As aliases use the same syntax as Windows environment variables, the Windows environment can be overruled by this alias definition.

The Database Program

The function 'delete all records' have been added to the filter function.
A button has been added to delete all records matching the filter criteria.

The last used filter setting will be remembered by the database program.

Symbols

References on Connection Points

References on connection points can now be defined and set inactive in the function 'Create new symbol. This allows you to define the position of the reference text.

The reference can then later be activated, manually or via the database.

Titles on Symbol States

Each symbol state can now have its individual title. This is done in the function 'Create new symbol', when the button 'States to layers' is activated.

Titles are defined by selecting the 'Symbol settings' dialog box:

As states are defined in layers (Layer 5 = state 5), the title of a certain state is shown when the specific layer is selected.

In the symbol menu, the titles are displayed as popup information.

In the Component data dialog box the titles are shown when the mouse rests over a state:

State and title are also shown in the ”Information” dialog for symbols.

Data Fields Included in a State

You can also use data fields in different states. One state could be where the symbol data fields are not-visible, and next could be where they are visible.

Deleting Symbols

When you delete a symbol you can choose to delete the selected symbol, the component which includes the selected symbol, or all components with the same name. One symbol/component name can cover more than one physical component (relay with add-on or terminals).

In the example above, the symbol =G1+A2–X3:10 has been selected. The dialog box displays that the terminal =G1+A2–X3 consists of 11 components (11 physical terminals).
It also shows that the selected terminal consists of 2 symbols. The second one is found on the mechanical page.
Furthermore, the dialog box shows that there is a total of 44 symbols with the name =G1+A2–X3.
Clicking the button 'Symbol' will only delete the selected symbol.
Clicking the button 'Component' will delete the selected symbol and the mechanical component.
Clicking the button 'Name' will delete all terminals =G1+A2-X3.

Drawing Headers

Using States in Drawing Headers

Drawing headers can now be used with different states. States are selected in Settings | Page data. The different states are shown in the list.

The same drawing header can therefore be used through all phases of a project.

Right-clicking in the state combo box allows you to change state on all or selected pages.

Data Fields Showing Data from Page and Project Data

When double-clicking a data field in the drawing header the program will open the dialog box where the pointer data can be changed.

The Object Lister

The Object Lister now displays Page type and Page title. Below, see example for symbols.

As for all other information, visibility can be set via the 'View menu'.

Signals now include the signal reference.
+ indicates that this signal's reference is active, showing the from/to reference.
***** indicates that this signal's reference is active, showing All references.
- indicates that this signal's reference is deactivated.

The Pick Menus

Keyboard Control of the Pick Menus

The Database Pick menu can now be controlled by the right-arrow, left-arrow and the enter key on the keyboard. 'Show available again' and the 'Show available window' will remember the last selected position.

The Symbol Pick menu can also be controlled by the keyboard. This is activated in the View | Activate pickmenu or by <Shift+Ctrl+F12>. You can use the right-arrow, left-arrow and the enter key on the keyboard. Last position will be remembered. The Page-Up and Page-Down keys will turn to next or previous Pick menu.

Symbols With States

The Pick menu can display the different states of a symbol. The states will either be shown automatically or manually as a pop-up menu.

If the states have individual titles, these will be shown as well. Symbols with states can be recognized by the dark blue triangle in the lower right corner of the pickmenu cell.
When navigating in the Pick menu with the keyboard the down-arrow key is used to see the states.

The automatically or manually pop-up of states is controlled by the properties of the Pick menu. The Pick menu properties are found by right-clicking in the Pick menu.

Select 'Automatic popup of submenus' if you prefer automatic pop-up. The 'Popup delay' can be individually controlled. The delay time is the time from which the mouse is over the Pick menu cell and until the popup menu appears.

Deselect 'Automatic popup of sub menus' if you prefer manual pop-up. A button will now appear in the lower right corner of the Pick menu cell. To activate the popup menu you need to click this button.

Symbols With Component Data in the Pick Menu

It is now possible to place more than one component in one cell of the Pick menu. One cell can therefore contain e.g. contactors handling different power. Automatic or manual popup works the same way as for symbols with states.

Pick menu cells with more than one set of component data are displayed with a light blue colored triangle in the lower right corner.

Sets of component data are created by right-clicking in the cell and selecting 'Component data'.

The previous dialog box has been changed to the one showed below. Here you see an example with 5 sets of component data.

The + (plus) button will add an empty row, where you can enter your data or you can use the button to select a component from the database.

The – (minus) button will delete the selected row.

You can change the component order with drag and drop.

If you select a component from the database and the row contains data, you will see the below dialog box. Here you can choose to add the selected component or overwrite the selected row. This way, you can use the DB-button to quickly add new components from the database.

The Title column will automatically be filled with information from the new description field if this setting has been made. Please refer to page 14 in this document. The title can be changed at any point.

Find/Replace

The Edit | Find/Replace function now remembers the last selected text type.

The Edit | Find/Replace function can now search in symbol data fields. The function will search the value and not the name of the symbol data field.

In the Edit | Find/Replace Find What part of the function, you now can select between the existing values in the project. You do not have to type the Find text anymore, you just select.

New Page Function

If you right-click on a page tab, you will now have easy access to Page number, Cut, Copy, Paste and Delete (a page).

The Paste function still works for more than one page, if you have copied more than one page in the Page menu.

Texts

The ***PCschematic*** font can now be proportional, meaning that the PCschematic font will have the same total width as one of the Windows fonts.

Texts have been added the option to suppress forced line breaks (^). This is done in the Text properties dialog box, and can be set for each individual text. This function can be very useful for a PLC reference symbol description. On the PLC reference symbol there may be room for the text in one line, but on the specific I/O symbol the description might need to be shown in 2 lines.
The function is also very useful when using linked texts.

Example:

Below you see the individual I/O point where the forced line break is needed.

Here you see the PLC reference symbol with enough room for the text in one line.

Swop Texts

The Edit menu has been added the function 'Swop texts'. The function is also found in the right-click menu..

Mechanical Pages

Spacing Symbols

the option 'Space symbols and align' has been added to the the dialog box "Mechanical Load".

After placing the symbols on top of each other, you will be asked to select an end point for spacing the symbols.

More Than One Type of Mechanical Pages

A specific data field can be assigned to a mechanical page, allowing different views of the same component:. A component can be represented in different ways on different mechanical pages: One page could show the cabinet with open front, another page shows the front of the cabinet. In order to obtain this you need to create new fields in the database. Each database field will have different symbols. E.g. the existing MECTYPE and a new called FRONTVIEW.

Before these database fields can be used on the mechanical page, they must be selected as legal database fields for mechanical symbols. This is done in Settings | Database | Database Setup | Component data.

The plus (+) button shows the dialog below. Here you must select which database fields are valid.

The first field in the list will be used as the default field for mechanical symbols. Using the arrow buttons you can change the default setting.

The selected database fields can now be selected in the function 'Mechanical Load'

Or it can be assigned to a page in Settings | Page setup in the 'Database symbol field.

The function 'Mechanical load' will now use the symbols found in this database field

Using the graphical insertion of symbols in the database (Select component | <Ctrl-Shift-DATA>), the assigned database field will be used for the mechanical component.

(IGNORE)

The (IGNORE) description in the database field for mechanical symbols is not necessary anymore. This was indicating that no mechanical symbol was needed. The program will not check this anymore. However, it is still legal to write (IGNORE) in this field.

Measurement

Lines

The function 'Measurement' has been expanded so it can identify the edges of lines.

Circles and Arcs

The function 'Measurement' can also handle circles in 90 degrees steps.

Text Property in the Measurement Function

In the measurement dialog box you can now select individual properties for the measurement text. The program will automatically remember the last used settings.

New Functions

The lines property Jumper links can now be changed using Copy and Transfer properties.

View Jumper Links and View Signal

The function 'View Jumper links' and 'View Signal' will highlight (in green) all jumper links and a selected signal on all pages. The last selected signal name will be remembered.

Save Logo to File

In Settings | Project data it is now possible to save the used logo to a file.

The Open Dialog Box

The open dialog box will now remember the last setting for displaying the files.

Re-addressing PLC's

PLC's can now be re-addressed in the same order as shown in the project.

No matter how special the address order is in a project, the selection 'Current order' will readdress the selected PLC in the same order.

Text Descriptions on PLC Connection Points

While importing PLC lists, the description can be converted to capital letters.

Space Symbols

The function 'Space symbols' will now ask for a new start position if there is not room enough in the first pointed area. In this way symbols can be placed in several rows.

Selecting and Pointing Lines and Circles

Selecting and pointing at lines (when not in drawing mode) has been expanded to the whole graphical area of the object including line- and pattern width.

Selecting and pointing at circles and arcs (when not in drawing mode) has been expanded to the whole graphical area of the object, see below.

Cursor Position Returns to the Object

When an object is selected and the dialog box activated, the cursor will return to the position over the selected object when leaving the dialog box.

Transfer Changes to All Symbols in the Component

The checkbox '*Change all symbols for the component*' has been removed and replaced by the button 'All'. If you click OK the changes are only done on the selected symbol. If you click the All button, the changes will be done on all symbols in the selected component.

The OK or All button must be activated to close the dialog box.

The SubDrawing Window

The right-click menu in the Sub drawing window has been added the function 'Rename'. F2 can be used as short-cut.

The Sub drawing window has been expanded to show Title and project data of the pointed drawing.

Selecting Symbols Associated With Lines

Branch symbols, signal symbols and wire number symbols can be handled from the Line command.

Symbol Data Fields in the Popup Information

Symbol data fields have been added to symbol popup information.

Start and Stop Drawing Mode

The menu items 'Draw line', 'Place symbol', 'Place text', 'Place circle' or 'End line', 'End symbol' 'End text', 'End circle' have been added to the relevant right-click menus as the first menu item. The short-cut to 'Place' is <Insert> and to 'End' is <Esc>.

New Feature for Moving a Line

Marking a line and using the <Ctrl> key will release the pointed line from its connection so it be moved to another connection.

The Layer Dialog Box

The layer dialog box has been added the option 'Only show used layers'.

The Symbol Menu

The Symbol menu has been added a new option. You can now group your symbols exactly the way you want. Symbols added to this menu will still be placed in the original folder. Right-click to create the folders (Contactor, Closed and Open) and groups (1 pole, 2 pole and 3 pole).

The Right-click menus in the left and right window contain all functions to create this new symbol menu structure and for placing the symbols. The same symbol can be in as many folders as you want, as it is only a link to the original library. If you change a symbol in the original library, the change will also be in this symbol menu.

Symbols are added by right-clicking in the Symbol window (right window). Select 'Add symbol'.

Then you select the symbol library from which you want to select symbols. You can select symbols by double clicking on a symbol or by dragging the symbol from the selected library to the new menu.

The default setting for 'Select Library from alias' is Alias names and not folder names.

The setting can be changed in the Alias dialog box.

Symbol Generator

The symbol generator is a wizard to help the user create simple symbols. In earlier versions you could create simple symbols by pressing <k> followed by '#' and different syntaxes.

This is still possible, but the syntax is difficult to remember. Furthermore, the syntax has been expanded in version 10. Instead you select 'Activate symbol generator' and click OK, the below wizard will appear.

In the Symbol generator title bar the syntax is shown while creating the symbol.
Any changes in the dialog box will immediately be displayed in the upper right window. This makes it very easy just to try, as the result is shown instantly.

The Symbol generator can be accessed directly from the symbol menu.

When creating new symbols the Symbol Generator can be used to make a 'raw-symbol' that can be the start for further editing.

Draw Order

The Draw Order of objects can now be controlled. This means that it is possible to control which object should be in front or in back of the drawing, as well as the order of objects can be changed. The functions are found in the right-click menu under Draw order:

This is particularly important when using mechanical symbols. Currently, the function is only working with objects of the same type.

The program has the following (backwards) order of the different objects:
1) Drawing header and current patch
2) Pictures and OLE objects
3) Data fields containing pictures
4) Lines
5) Circles
6) Free texts
7) Data fields containing text
8) Symbols
Thus symbols will be placed on top of other objects and will cover what is below.

Send a Project with an E-mail

In the file menu the function 'Send to…' has been added which will start the e-mail program and attach the actual project to this new e-mail.

Project Generator

The Project generator can now handle connection names using the symbol name variable. E.g. write –X1:1 in the Excel sheet and the connection name of –X1 will be changed to 1. If a symbol has 3 connection points, you can write –M1:U,V,W, which will give the 3 connection points the connection names 'U', 'V' and 'W' and the symbol name will be changed to'-M1'.

As an option in the project generator, it is now possible that symbols in a sub drawing take the same reference designation as the page where it is placed.

It is now possible to select models when using the project generator. This is done by adding a variable for model selection. In the Excel sheet the variable value is written e.g. 0,06kW 0,18.

Line article data can also contain variables for Drag'n Draw and the Project generator.

The Translator

The Text Translator has been totally changed with the following benefits:

- More intuitive user interface
- Optionally, translation can be made without using text files or the dictionary, meaning that it is not necessary that all texts are found in the dictionary
- Users can share the same dictionary, as the dictionary is an Access database file
- Translations are possible between two languages with different character sets (e.g. Russian and Chinese). Windows does not have to be set to a specific set of characters
- Easier editing of the texts in the dictionary
- Texts can be exported/imported to PCS-TXF files, which e.g. can be sent to an external translator. This can be done without any conflict with wrong character sets and wrong formats
- A special editor has been developed for translation of the texts in the PCS-TXF files, and the editor is included in ***PCschematic® ELautomation***
- There is a Find/Replace function in the translator, making it easy to change words that are found in various phrases.
- Problems with double words in the dictionary do not exist anymore.
- All texts except symbol names and connection names can be translated
- Translated texts will not replace the original texts, but create a new project with the translated texts

Dictionary [C:\pcselcad\PCsDictionary DAN - ENU.MDB]

DANISH	English (United States)
Start	Start
Start motor	Start motor
Start^Frem	Start^Forward
Start^Tilbage	Start^Back
Startm	Startm
Stjerne	Star
Stm	Stm
Stop	Stop
Stop motor	Stop motor
Stopm	Stopm
Stykliste	Parts list
STYREPULT	Control Unit
test	test
Tid	Time
TIL	TO
Tilbage	Back
Titel	Title
Tlf	Phone
Total pris	Total price
Transportbånd	Conveyer
Transportbånd fejl	Conveyer error