



# AUTOMATION

## TOOLS



# TABLE OF CONTENTS

<b>PREFACE .....</b>	<b>5</b>
<b>THE ON-LINE MANUAL .....</b>	<b>9</b>
OPENING THE ON-LINE MANUAL .....	9
THE TABLE OF CONTENTS .....	10
THE BUTTONS IN THE UPPER-RIGHT CORNER .....	10
ZOOMING AND ADAPTING TO YOUR SCREEN .....	10
DIRECT ACCESS TO THE INDEX .....	11
RETURNING TO <b>AUTOMATION</b> .....	11
PRINTING FROM THE ON-LINE MANUAL .....	11
<b>CONVERTING AUTOCAD FILES .....</b>	<b>15</b>
WHAT IS A MAP FILE? .....	15
THE WORK PROCESS WHEN CONVERTING DWG/DXF FILES .....	16
OPENING AND EDITING MAPFILES .....	18
WHICH FUNCTIONS ARE TRANSFERRED FOR DWG/DXF FILES? .....	19
IMPORTING DWG/DXF FILES .....	20
<b>DWG/DXF MAP OPTIONS .....</b>	<b>21</b>
CHOOSING IMPORT OR EXPORT .....	21
INTELLIGENT IMPORT OF DWG/DXF FILES .....	22
THE BUTTONS IN THE DIALOG BOX DWG/DXF MAP OPTIONS .....	22
MAKING MORE ROOM FOR THE LEFT-HAND SIDE OF THE TAB .....	23
THE TAB APPLICATION .....	24
THE TAB COLOR .....	27
THE TAB LINSTYLE .....	29
THE TAB LAYER(MAP) .....	31
THE TAB TEXTS .....	33
THE TAB BLOCKDEFS .....	34
THE TAB ATTDEFS .....	36
THE TAB LAYER(DEF) .....	38
THE TAB ADVANCED .....	40
IMPORTING A DWG/DXF FILE IN AN EXISTING PROJECT .....	43
IMPORTING DWG/DXF FILES AS SYMBOLS .....	44
EXPORTING DWG/DXF FILES FROM <b>AUTOMATION</b> .....	46
<b>SYMBOL DOCUMENTATION .....</b>	<b>51</b>
ON SYMBOL DOCUMENTATION .....	51
SETTINGS IN SYMBOL DOCUMENTATION .....	52
SYMBOLS .....	53
TEXTS .....	54
PAGES .....	56
ADJUSTMENTS .....	59
<b>GRAPHICAL TERMINAL PLANS .....</b>	<b>63</b>
ON GRAPHICAL TERMINAL PLANS .....	63

CREATING GRAPHICAL TERMINAL PLANS . . . . .	64
THE STRUCTURE OF THE TERMINAL PLAN . . . . .	65
SETTINGS IN CREATE GRAPHICAL TERMINAL PLAN. . . . .	66
CONNECTIONS . . . . .	67
PAGES . . . . .	69
SYMBOLS . . . . .	71
WIRES . . . . .	76
ADJUSTMENTS . . . . .	77
ADJUSTMENTS FOR TERMINAL PLANS STRUCTURED ACCORDING TO CABLES	79
UPDATING EXISTING TERMINAL PLANS . . . . .	80
<b>GRAPHICAL CABLE PLANS . . . . .</b>	<b>87</b>
ON GRAPHICAL CABLE PLANS. . . . .	87
CREATING GRAPHICAL CABLE PLANS . . . . .	87
SETTINGS IN CREATE GRAPHICAL CABLE PLAN . . . . .	88
CONNECTIONS . . . . .	89
PAGES . . . . .	90
SYMBOLS . . . . .	91
ADJUSTMENTS . . . . .	93
UPDATING EXISTING GRAPHICAL CABLE PLANS . . . . .	94
<b>PC SCHEMATIC TEXT TRANSLATOR . . . . .</b>	<b>97</b>
TRANSLATING PROJECTS . . . . .	97
THE TRANSLATIONS DIALOG BOX . . . . .	100
EDITING DICTIONARIES . . . . .	105
OPTIONS FOR THE TRANSLATION . . . . .	107
EDITING TOOL FOR EXCHANGE FILES . . . . .	111
RECOGNITION AND TRANSLATION OF TEXTS . . . . .	112
HOW TO PLACE TEXTS IN A PROJECT MEANT FOR TRANSLATION . . . . .	114
CONVERTING DICTIONARIES FROM THE PREVIOUS VERSION. . . . .	115
<b>OTHER OPTIONS IN THE TOOLS MENU. . . . .</b>	<b>117</b>
RENAME SYMBOLS . . . . .	117
DATABASE . . . . .	117
CREATE VARIABLE LIST IN (SUB)DRAWING . . . . .	117
PROJECT GENERATOR. . . . .	117
MOUSE CHASING SYSTEM. . . . .	117
<b>INDEX. . . . .</b>	<b>121</b>

## PREFACE

This manual contains descriptions of the tools which are included in the full version of **Automation**.

The manual is also accessible in an on-line version, which is described in “The On-line manual” on page 9.

For the moment these tools are:

- 1) Conversion of AutoCAD files to **Automation** projects
- 2) Automatic documentation of Symbols in Projects or Libraries
- 3) Automatic generation of Graphical Terminal Plans
- 4) Automatic generation of Graphical Cable Plans
- 5) Automatic Translation of texts in projects

Ideas for other tools are welcome. Please refer to our e-mail address on the title page.





## The On-line manual

The manual for PC|SCHEMATIC Automation both exists in a printable version and as an on-line manual.

This part describes the use of the on-line manual.





# THE ON-LINE MANUAL

When using this on-line manual, you have the benefit of reading the manual directly on the screen while working with the program. The contents is identical to the A4 printable manual, only have the pages been adapted for reading on the screen.

THIS CHAPTER DESCRIBES	
— OPENING THE ON-LINE MANUAL .....	(P. 9)
— THE TABLE OF CONTENTS.....	(P. 10)
— THE BUTTONS IN THE ON-LINE MANUAL.....	(P. 10)
— ZOOMING AND ADAPTING TO YOUR SCREEN .....	(P. 10)
— DIRECT ACCESS TO THE INDEX .....	(P. 11)
— RETURNING TO <b>Automation</b> .....	(P. 11)
— PRINTING FROM THE ON-LINE MANUAL .....	(P. 11)

## OPENING THE ON-LINE MANUAL

To open the On-line manual choose **Help** and select the manual.

**Click to view a chapter**

**Zoom**

**Fit the manual to your screen**

**About the electronic manual**

**Return**

**Up and down**

**Click directly to Index**

**Page no.**

**Click on the + to see the sub-chapters**

**Click to view the sub-chapters**

**Click to jump to other chapters**

## THE TABLE OF CONTENTS

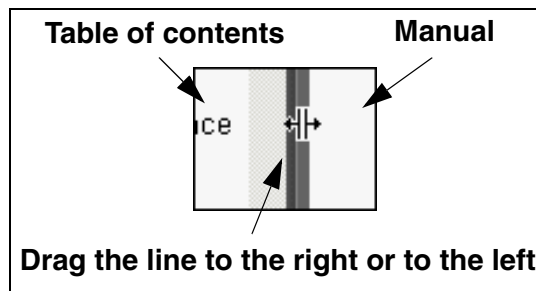
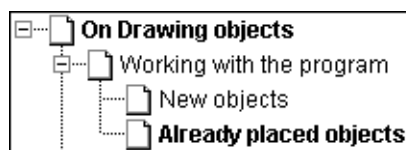
In the left-hand side of the manual, you can click directly on the section you wish to read.

A **+** in front of a section means that the chapter is divided into subsections.

When you click directly on a title of a section, the chosen section is displayed.

You can adjust the size of the table of contents on the screen. This is done by pointing on the line between the table of contents and the manual text, and then drag the line to either the left or the right.

You can turn the displaying of the table of contents on and off by pressing [F5].



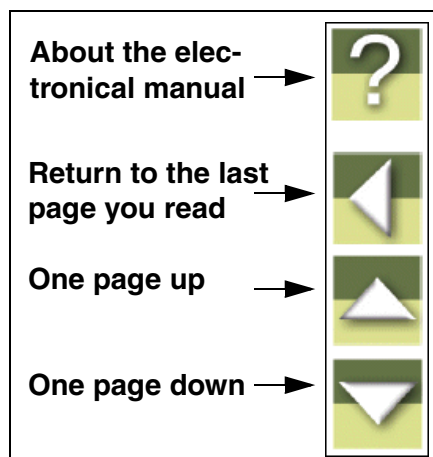
## THE BUTTONS IN THE UPPER-RIGHT CORNER

In the upper-right corner of the on-line manual, you will find these four buttons.

When you click the **?** button, you enter this chapter about using the on-line manual.

If you click the **Return** button, you jump back to the last page that you have viewed in the on-line manual.

When you click the up and down buttons, you leaf one page up or down respectively. You can also use the [PageUp] and [PageDown] buttons on the keyboard.



## ZOOMING AND ADAPTING TO YOUR SCREEN



When you open the on-line manual initially, it is opened to a size fitting your screen. If you need to zoom in on a detail in the manual, you click the **Zoom** button in the upper-left corner of your screen.



To re-size the on-line manual to fit your screen, you click the **Fit in Window** button on the upper part of the screen, or press [Ctrl+0].

DIRECT ACCESS TO THE INDEX

On the right-hand side of the on-line manual, you have direct access to the index.

button, you  
pdate.

he lower-  
g box Set-

M  
N  
O  
P  
Q  
R  
S  
T  
U  
V-W  
X-Z

Select

area in all layers . . . . . 780

different types of objects . . . . . 444

elements in lists . . . . . 61

elevated objects . . . . . 782

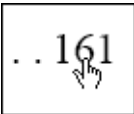
Selecting an area in all layers

It is possible to select everything in all layers, for instance *Area* and mark up an area within the page and choose **Select in all layers**. The area is now selected.

If for instance you had selected **Symbols** in the area would have been selected.

To find an explanation about fx. the **selection** function, you click **S** index letter in the right-hand side of the manual. This then looks up under **S** in the index. Now click on the dotted line after the word **Selection**, to bring up to the desired information.

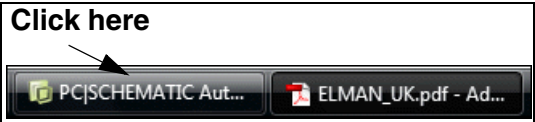
Please note, that it is important that the cursor on screen has the appearance of a hand with a pointing index-finger, when you click the dotted line — or the page number. If this is not the case, you zoom in on the text, and therefore can no longer see the full page on screen.



Click the **Fit in Window** button or press [Ctrl+0] to restore full page view.

RETURNING TO Automation

In order to return to **Automation**, you just click on its program Windows button at the bottom of your screen — or press down the <Alt> button while pressing the <Tab> button.



The reason why you have to change back and forth in this way, is because that the manual is a document in the independent program **Adobe Acrobat reader**.

PRINTING FROM THE ON-LINE MANUAL



If you wish to read the manual on print, it is recommended to see the printed manual. To print out directly from the on-line manual, just click the **Print** button on the **Acrobat** program, and specify what to print out.

Please note, that the on-line manual buttons will also be printed.





## Converting AutoCAD files

In PCISCHMATIC Automation you can automatically convert AutoCAD files — with attached intelligence — to PCISCHMATIC Automation project files.

This is the subject of this chapter.



# CONVERTING AUTOCAD FILES

In **Automation** it is possible to import AutoCAD files (DWG/DXF files), and convert these to **Automation** projects.

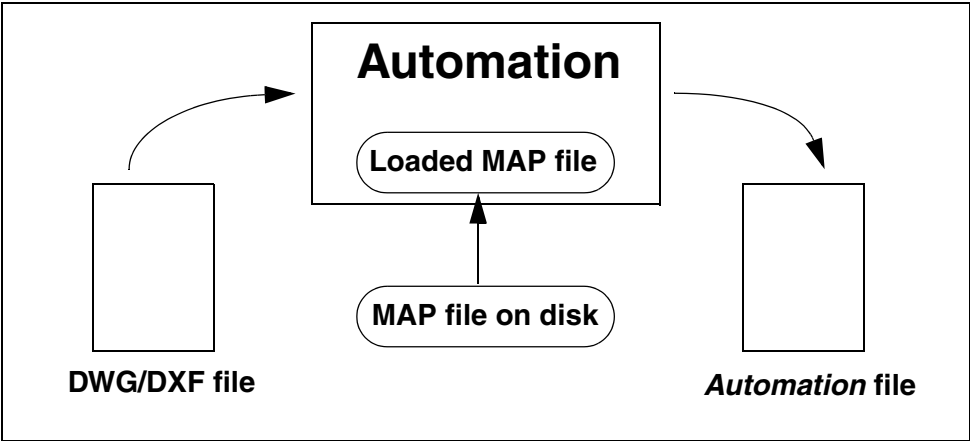
It is also possible to attach information from the original AutoCAD objects to the created **Automation** projects intelligently. Export from **Automation** to the DWG/DXF format is also possible.

The setting of import and export parameters takes place via the tool **DWG/DXF map options**, which is described in this chapter.

THIS CHAPTER DESCRIBES	
— WHAT IS A MAP FILE? .....	(P. 15)
— THE WORK PROCESS WHEN CONVERTING DWG/DXF FILES .....	(P. 16)
— OPENING AND EDITING MAPFILES .....	(P. 18)
— WHICH FUNCTIONS ARE TRANSFERRED? .....	(P. 19)
— IMPORT OF DWG/DXF FILES.....	(P. 20)
— DWG/DXF MAP SETTINGS.....	(P. 21)
— IMPORTING DWG/DXF FILES IN EXISTING PROJECTS .....	(P. 43)
— IMPORTING DWG/DXF FILES IN SYMBOLS.....	(P. 44)
— EXPORTING DWG/DXF FILES.....	(P. 46)

## WHAT IS A MAP FILE?

When you import files of another file format in **Automation**, or export files from **Automation** to another file format, this can be done via a so-called *map* file.



In this map file, it is specified how objects in the one file format, is translated/mapped to objects in the other format.

If you have detailed knowledge about **Automation**, as well as about the other program you wish to convert to/from, you can change the parameters for how the translation takes place from the one program to the other.

In the following it is showed how you can change the settings for import of DWG/DXF files.

## THE WORK PROCESS WHEN CONVERTING DWG/DXF FILES

When you convert DWG/DXF files, one part of this work takes place in **Automation**, and another takes place in the tool **DWG/DXF map options**:

*Importing the DWG/DXF file is done in **Automation***

Import of DWG/DXF files in **Automation**, takes place via the **Open** dialog box in **Automation** — via a selected map file.

*Changes in the map file are made in **DWG/DXF map options***

When you wish to make changes in the applied map file, enter the tool **DWG/DXF map options**, from where changes in map files are made.



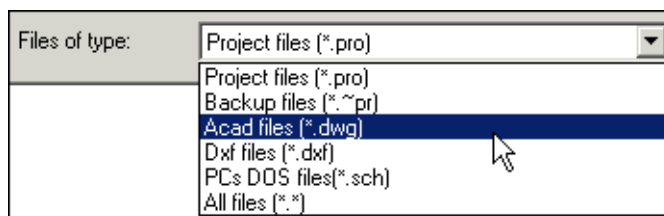
HERE IT IS DESCRIBED HOW TO IMPORT A DWG/DXF FILE IN A NEW **Automation** PROJECT.

— PLEASE ALSO SEE “IMPORTING A DWG/DXF FILE IN AN EXISTING PROJECT” ON PAGE 43 AND “IMPORTING DWG/DXF FILES AS SYMBOLS” ON PAGE 44.

## Repeated refinement of mapfiles for better import

When you import DWG/DXF files, this is a gradual refinement process:

- 1) Import the DWG/DXF file in **Automation** by choosing **File => Open**
- 2) In the dialog box **Open**: Choose the file type **\*.dwg** or the file type **\*.dxf** at the bottom of the dialog box, click on the file and click on **Open**



- 3) The program presents a new **Open** dialog box: Click on the mapfile you wish to use, and click on **Open**

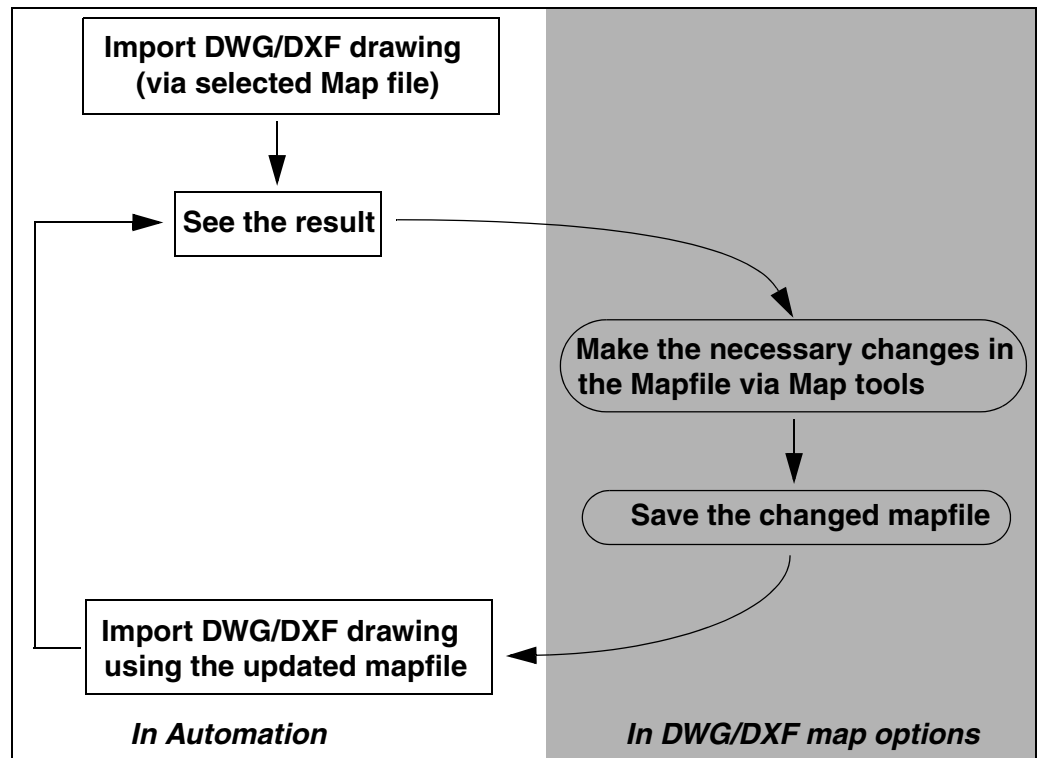


IF YOU DO NOT HAVE A MAP FILE, YOU CAN USE THE MAPFILE **ACAD.MAP** IN THE FOLDER **Pcselcad** FOR A START.

- 4) The file is now created as a **Automation** project: Analyse the created file in **Automation**, and see which blocks (symbols) and texts that could be imported more intelligently



- 5) Then open **DWG/DXF map options** — see “Opening and editing mapfiles” on page 18 —, adapt the mapfile according to your observations and save the mapfile, for instance under a new name
- 6) Now start again from step 1, and continue this repetitive process until you have made sure that you have transferred/inserted as much intelligence in the **Automation** project as you wish — please refer to the figure below



## OPENING AND EDITING MAPFILES

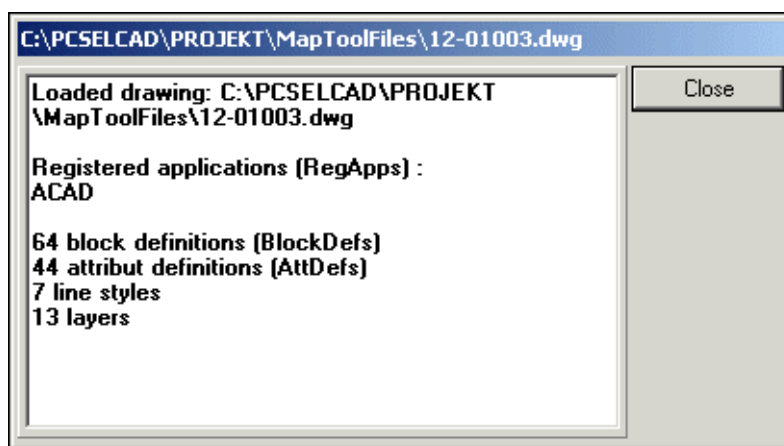
When you open the tool **DWG/DXF map options** to make changes in a mapfile based on an existing DWG/DXF file, do the following:

- 1) Choose **Tools => DWG/DXF map options** — to open a mapfile, and adjust it according to the file you wish to import
- 2) In the dialog box **DWG/DXF map options**: Click on **Open** — to open a mapfile
- 3) In the dialog box **Open**: Select a mapfile, and click on **Open**

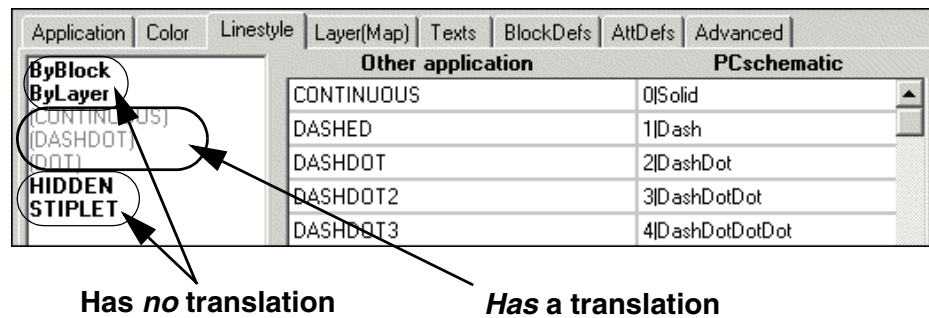
You can now edit the mapfile via the different tabs in the dialog box — as described in “DWG/DXF map options” on page 21.

When you wish to fetch information from an existing AutoCAD file, in order to adapt the mapfile more efficiently, do the following (Skip to step 9 if this is not the case):

- 4) In the dialog box **DWG/DXF map options**: Click on **Load drawing**
- 5) In the dialog box **Open file**: Select the DWG/DXF file you wish to fetch information from, and click on **Open**
- 6) You now get a status of the contents of the opened DWG/DXF file:



- 7) Click on **Close**. When you wish to see this information again, click the **Status** button at the bottom of the dialog box **DWG/DXF map options**.
- 8) In the dialog box **DWG/DXF map options**: You can now adjust the settings in the opened mapfile according to the contents of the file you wish to convert. You can do this based on the pieces of information from the selected DWG/DXF file you find in the left-hand side of the different tabs in the dialog box.



The elements written in grey in the left-hand side of the dialog box has an existing translation in the mapfile, while the elements written in black has no existing translation in the mapfile (a default parameter will be used). — Read more about the options on the individual tabs in “DWG/DXF map options” on page 21.

- 9) Click on **Save** when you have made the desired changes in the mapfile. You do not have to close the dialog box, because you probably will need to make changes in the mapfile more than once.
- 10) Now return to **Automation**, import the DWG/DXF file again, and evaluate the result.

### WHICH FUNCTIONS ARE TRANSFERRED FOR DWG/DXF FILES?

When converting, common functions for the program you import from and **Automation** will be transferred.

Furthermore a number of other functions/information can be transferred intelligently, as described in “DWG/DXF map options” on page 21.

When importing objects, which **Automation** can not handle, it is recommended to “explode” the drawing before importing. **Automation** can not handle coordinates — such as line lengths — on more than two kilometers.

#### *Importing ground plans*

When you wish to import ground plans, it is recommended to insert these as OLE objects in the **Automation** project. If the inserted OLE object is *linked* to the **Automation** project, you furthermore have the advantage that it is updated automatically in **Automation**, when it is updated in AutoCAD. Read more about this in “Inserting files from other programs / OLE” in the **Automation** manual.

If you have difficulties importing DWG or DXF files, please contact your local hot-line.

## IMPORTING DWG/DXF FILES

There are three different ways to import DWG/DXF files in **Automation**:

- a) You can start a new project by opening a DWG/DXF file — see “The work process when converting DWG/DXF files” on page 16
- b) You can import a DWG/DXF file in an existing project by importing them via the **Page menu** — see “Importing a DWG/DXF file in an existing project” on page 43
- c) You can import a DWG/DXF file as a symbol, and then work with the symbol as an ordinary symbol in your projects — see “Importing DWG/DXF files as symbols” on page 44



### PREPARING THE DWG/DXF FILE *BEFORE* IMPORTING

IT IS A GOOD IDEA TO ADAPT THE DWG/DXF FILE *BEFORE* IMPORTING IT, SO THAT IT ONLY CONTAINS EXACTLY WHAT YOU NEED.

---

# DWG/DXF MAP OPTIONS

In the dialog box **DWG/DXF map options** you can adapt mapfiles, in order to get the best possible mapping of the imported files.

This section describes the possibilities you have on the different tabs in the dialog box.

THIS CHAPTER DESCRIBES	
— CHOOSING IMPORT OR EXPORT.....	(P. 21)
— INTELLIGENT IMPORT OF DWG/DXF FILES .....	(P. 22)
— THE BUTTONS IN THE DIALOG BOX <b>DWG/DXF map options</b> .....	(P. 22)
— MAKE MORE ROOM ON THE LEFT-HAND SIDE OF THE TAB .....	(P. 23)
— THE TAB APPLICATION.....	(P. 24)
— THE TAB COLOR .....	(P. 27)
— THE TAB LINSTYLES .....	(P. 29)
— THE TAB LAYER(MAP) .....	(P. 31)
— THE TAB TEXTS .....	(P. 33)
— THE TAB BLOCKDEFS.....	(P. 34)
— THE TAB ATTDEFS .....	(P. 36)
— THE TAB LAYER(DEF) .....	(P. 38)
— THE TAB ADVANCED.....	(P. 40)

— See “The work process when converting DWG/DXF files” on page 16 and “Opening and editing mapfiles” on page 18 for general information about converting DWG/DXF files and working with mapfiles.

## CHOOSING IMPORT OR EXPORT

In the upper left-hand corner of the dialog box **DWG/DXF map options**, you choose whether to edit the settings for *Import* of files to **Automation**, or to edit the settings for *Export* from the program to the DWG/DXF format.



## Independent Import and Export settings necessary

The reason why you have to specify whether to change *import* or *export* settings, is that you can not always use the same settings both ways.

For instance you can choose to convert both of the two DWG/DXF LineStyles **DOT** and **HIDDEN** to **Automation** linestyle **2**. Therefore, when you need to convert the other way, the program does not know whether to translate the linestyle **2** from **Automation** to the linestyle **DOT** or **HIDDEN**.

Therefore independent import and export settings are needed.



### IMPORTING DWG/DXF FILES

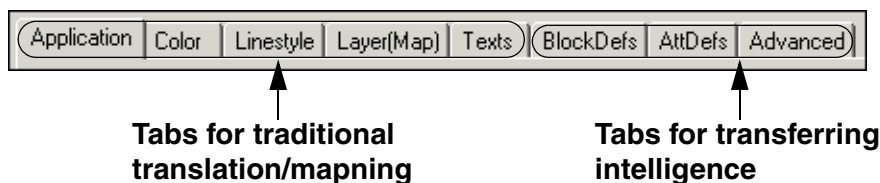
THIS CHAPTER MAINLY FOCUSES ON IMPORT OF DWG/DXF FILES.

— Read more about export from **Automation** to the DWG/DXF format in “Exporting DWG/DXF files from Automation” on page 46.

### INTELLIGENT IMPORT OF DWG/DXF FILES

In the dialog box **DWG/DXF map options**, you will find two types of tabs:

- 1) Tabs applied for traditional translation/mapping: Translation of e.g. one color in the DWG/DXF file to one color in the **Automation** project. Or mapping one linestyle to another, or one text to another, or transferring objects from one layer to another.
- 2) Tabs for transferring/creating intelligence: Here you can for instance specify which blocks that must be translated to which symboltypes in the **Automation** project. Or for instance which text attributes in the DWG/DXF file that must be translated to which symbol texts or datafields in the **Automation** project.



Read more about these options in the sections about the various tabs.

### THE BUTTONS IN THE DIALOG BOX DWG/DXF MAP OPTIONS

There are the following buttons in the dialog box:

*The button Close*

Click here and **DWG/DXF map options** is closed. The specified changes are not saved.

*The button New*

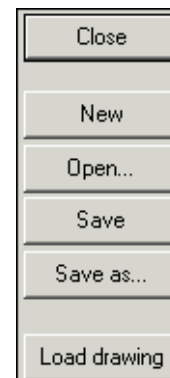
Click here to create a new mapfile from scratch.

*The button Open*

Click here to open an existing mapfile.

*The buttons Save and Save as*

Click here to save the settings in the mapfile.

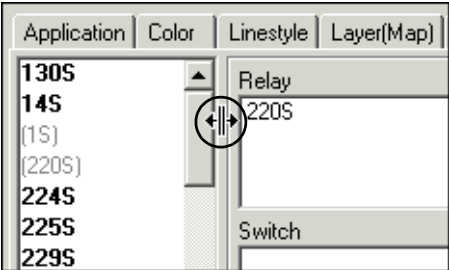


*The button Load drawing*

Click [here](#) to load data from an existing DWG/DXF file — as described in “Opening and editing mapfiles” on page 18.

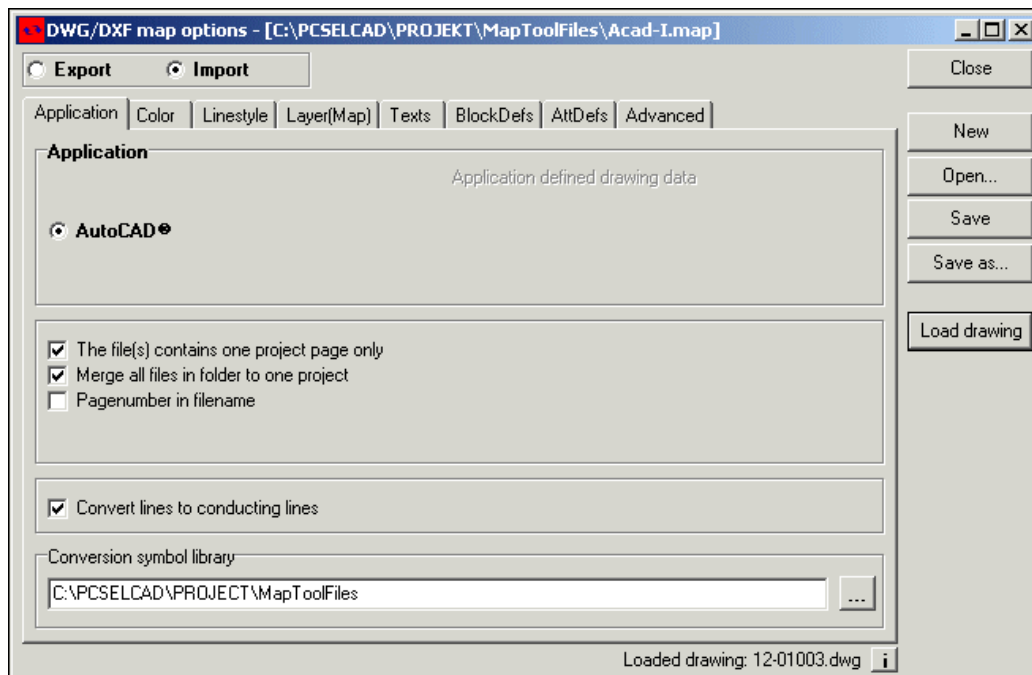
**MAKING MORE ROOM FOR THE LEFT-HAND SIDE OF THE TAB**

When you need more space for the left-hand side of the dialog box, place the mouse cursor over the vertical line between the left-hand and the right-hand side of the tab. Then drag the dividing line to the desired position.



## THE TAB APPLICATION

On the tab **Application** you specify the general settings for the mapping.



### DWG/DXF FILES FROM COMPETING ELECTRICAL APPLICATION PROGRAMS

ON THE TAB **APPLICATION** YOU CAN HAVE THE OPTION TO CHOOSE BETWEEN DIFFERENT TYPES OF AUTOCAD FILES. THIS IS BECAUSE THE INTERPRETATION OF THE AUTOCAD FILES WILL BE DIFFERENT DEPENDING ON WHICH PROGRAM THE DWG/DXF FILES ARE CREATED FROM.

IT IS POSSIBLE TO BUY ADDITIONS TO THIS CONVERSION PROGRAM, WHICH MAKES IT POSSIBLE TO CONVERT AUTOCAD FILES FROM COMPETING ELECTRICAL APPLICATIONS.

PLEASE CONTACT DPS CAD-CENTER APS FOR FURTHER INFORMATION.

### The file(s) contains one project page only

Select ***The file(s) contains one project page only***, if the DWG/DXF file only contains one single project page.

### Interpret all pages in a file catalog as one project

AutoCAD can only handle one project page in each file. **Automation** can handle all necessary project pages in one file.

Therefore, when you save all AutoCAD files attached to the same project in the same folder, you can — during the conversion to the **Automation** format — automatically place these files in the same **Automation** project.

To do this, choose ***Merge all files in folder to one project***.



### Page number in filename

During the conversion process, the file name for the DWG/DXF file is used as page number/name in the **Automation** project. When the name of the file is **ABC12**, the name of the page in the project therefore becomes **ABC12**.

If you select **Pagenumber in filename**, the program takes the last digits from the file name, and only use these as page number. When the name of the file is **ABC12**, the name of the page in the project therefore becomes **12**.

### Convert lines to electrical lines

If you choose **Convert lines to electrical lines**, all lines in the DWG/DXF file are converted to be conducting lines in the created **Automation** project.

### Applied symbol library for conversion

During the conversion it is possible to specify that a block in the AutoCAD file must be replaced with a predefined symbol in the created **Automation** project.

When the program meets a block during the conversion process, it is examined whether this block exists in the symbol library. If it exists in the library, the symbol from the library is inserted in the created **Automation** project. If it is not in the library, a new symbol is created in the project, based on the block.

In order to make this function, the symbol in the selected symbol library must have the same name as the block in the AutoCAD file.

You must also specify in which folder these special **Automation** symbols exists.

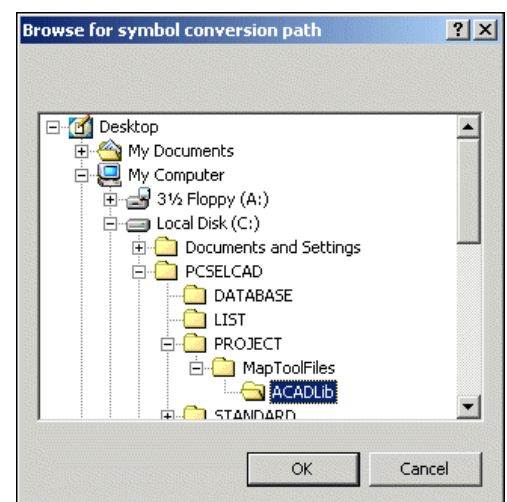
*When is this necessary?*

This can for instance be necessary if the program has not been able to insert connection points automatically on a symbol, or if you for instance wish to specify that the lower connection point on a terminal symbol has an **IO status** set to **Output**.

When you have converted a DWG/DXF file, there can be some symbols which have not been assigned connection points — or that has incorrectly placed connection points. This will be spotted quickly due to the parked lines next to these symbols.

*Specifying the location of the symbol library*

The symbol library is selected by clicking on the **Browse** button next to **Conversion symbol library**, clicking on the folder in which the symbol library is placed, and clicking **OK**.



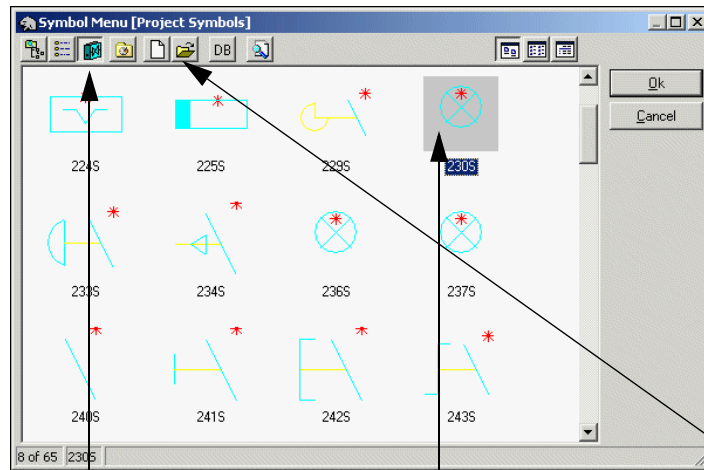
*Creating a symbol in the symbol library*

When you have translated a DWG/DXF file, and have realised that you wish to create a symbol as the translation for a specific block in the file, do the following in **Automation**:



1) Click on the **Symbols** and the **Symbol menu** button

2) In the **Symbol menu**: Click on the **Project symbols** button, click on the symbol, and click on the **Open symbol** button

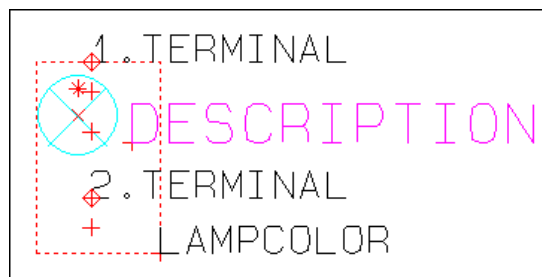


2a) Click here

2b) Click on the symbol

2c) Click here

3) In the **Symbol editor**: Adapt the symbol, save it in the symbol library folder, and choose **File => Close**

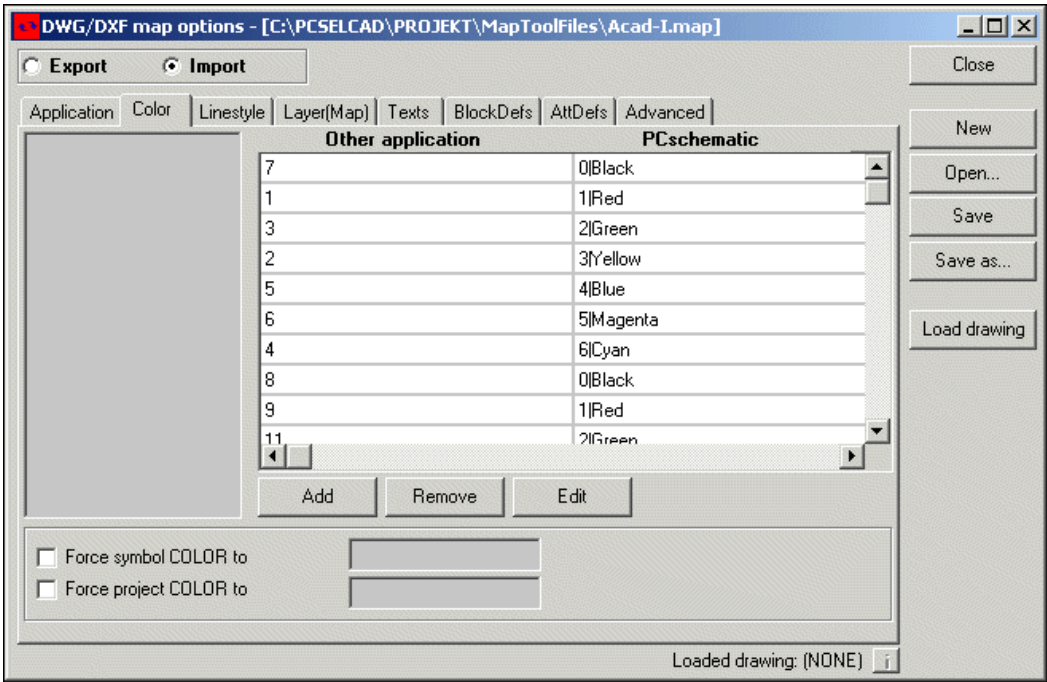


4) In the **Symbol menu**: Click **Cancel** — in order not to get the symbol in the cross-hairs

Repeat this procedure for all symbols in the project, for which this is necessary.

THE TAB COLOR

On the tab **Color** you specify how the colors in the DWG/DXF file shall be interpreted to the different colors in **Automation**.



Mapping parameters for colors in Automation

In **Automation** there are number codes for each object type, specifying for instance the color or line type of the object.

On the tab above you can specify that for instance the AutoCAD color **7** is translated to the color **0** in **Automation**.

Add, Remove and Edit

When you click on a line in the dialog box above, you can click on **Edit**, and you get the option to change the settings in the line.

If you click on **Remove**, the selected line is deleted. If you click on **Add**, a new line is added, on which you can type in new information.



Names on colors in Automation

You have the following color options in **Automation**:

Code	Color	Code	Color
0	Black	1	Red
2	Green	3	Yellow
4	Blue	5	Magenta red

CODES FOR COLORS IN Automation

Code	Color	Code	Color
6	Cyan	7	White
8	Light red	9	Light green
10	Brown	11	Light blue
12	Light magenta red	13	Light cyan
14	Not displayed on print		

CODES FOR COLORS IN **Automation****Force symbol COLOR to**

When you select **Force symbol COLOR to** you get the option to type in a color, that *all* symbols will be drawn with in the created **Automation** project — for instance the color black.

**Force project COLOR to**

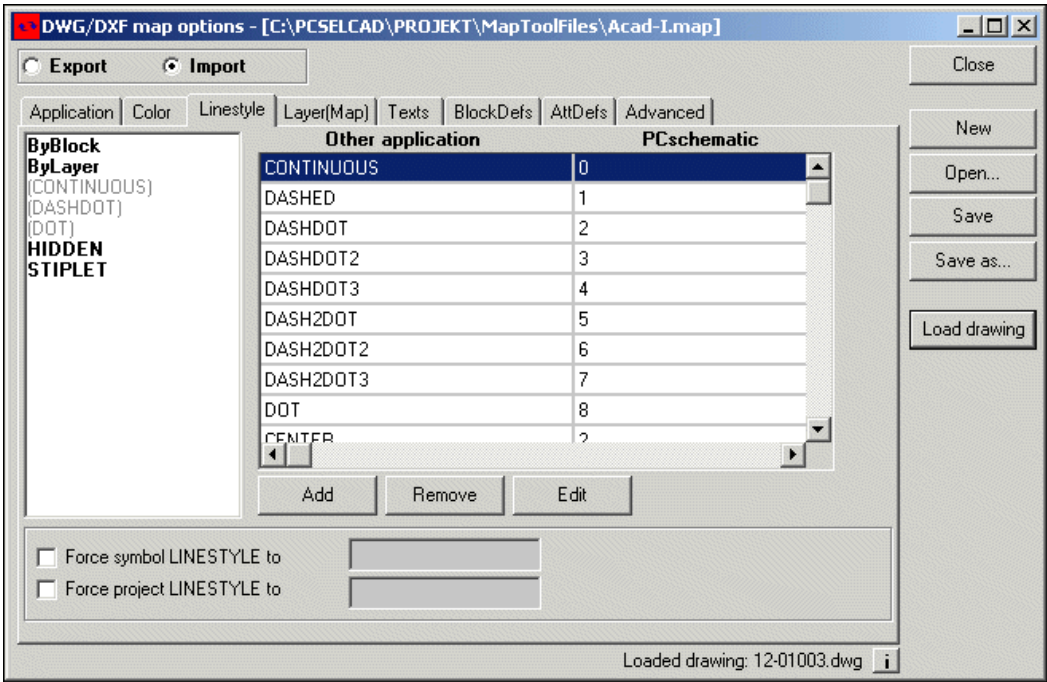
When you select **Force project COLOR to** you get the option to type in a color, which *all* lines, circles and texts in the created **Automation** project will be drawn with.

**AUTOMATIC MAPPING OF OBJECT TYPES TO FIXED VALUES**

AT THE BOTTOM OF SOME OF THE TABS YOU CAN, FOR EACH INDIVIDUAL OBJECT TYPE, CHOOSE WHETHER THE DEFINITIONS IN THE FILES AND/OR THE PROJECT DRAWINGS ALWAYS MUST BE CONVERTED TO A FIXED VALUE. THESE SETTINGS ONLY APPLIES FOR *IMPORT* to **Automation**.

THE TAB LINSTYLE

On the tab **Linestyle** you specify how the linetypes in the DWG/DXF file shall be converted to the different linetypes in **Automation**.



Changing mapping for linestyles

When you wish to change the name of a linestyle, or change what a linestyle is translated to, click on **Edit**. You then get the option to edit both fields in the line. You first get the option to edit the ***Other application*** field, then the ***PCschematic*** field.

For each object type in **Automation** there exists a number code specifying e.g. its color or linestyle. The significance of these number codes is described in “Linestyles in Automation” on page 29.

*Remove and Add*

When you wish to remove a line, click on the line, and click on the **Remove** button. Click on **Add** to add new lines.

Linestyles in Automation

**Automation** has the following linestyles:

Code	LineStyle	Code	LineStyle
0	Solid	1	Dash
2	Dot	3	DashDot
4	DashDotDot	5	DashDotDotDot
6	DashDashDot	7	DashDashDotDot
8	DashDashDotDotDot	9	FinDot

CODES FOR LINSTYLES IN Automation

Code	LineStyle	Code	LineStyle
14	SolidBox	15	EmptyBox
21	VerticalBox	22	HorizontalBox

**CODES FOR LINESTYLES IN Automation****Force symbol LINESTYLES to**

If you select **Force symbol LINESTYLES to**, you get the option to type in a line-style, which all symbols in the created **Automation** project will be drawn with — for instance the linestyle Solid.

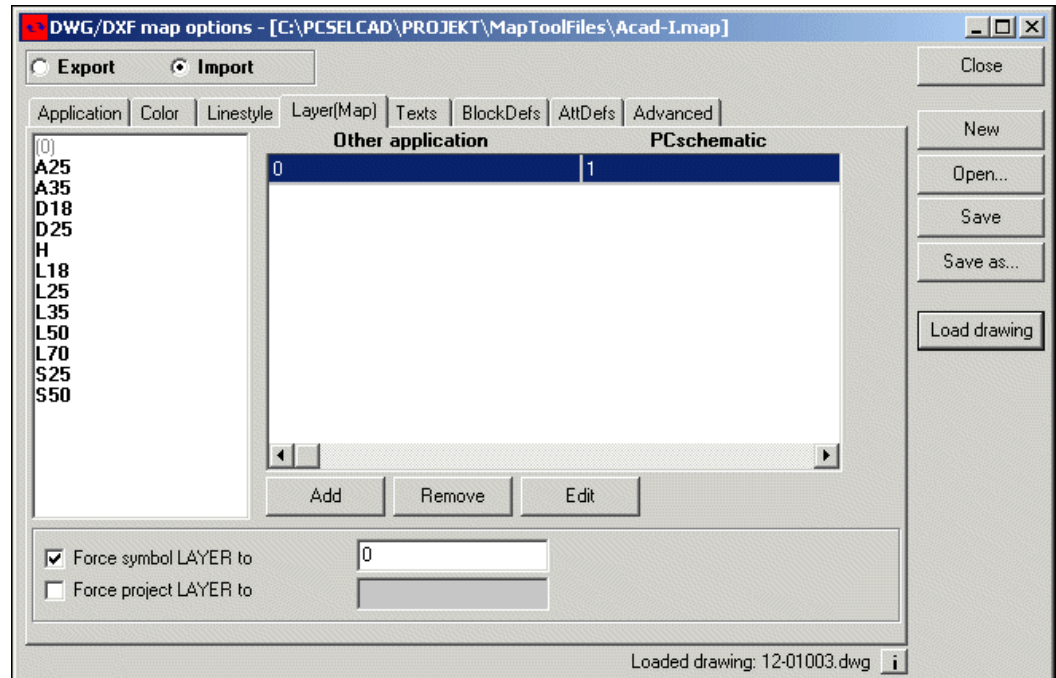
**Force project LINESTYLES to**

If you select **Force project LINESTYLES to**, you get the option to type in a line-style, which all lines in the created **Automation** project will be drawn with.

## THE TAB LAYER(MAP)

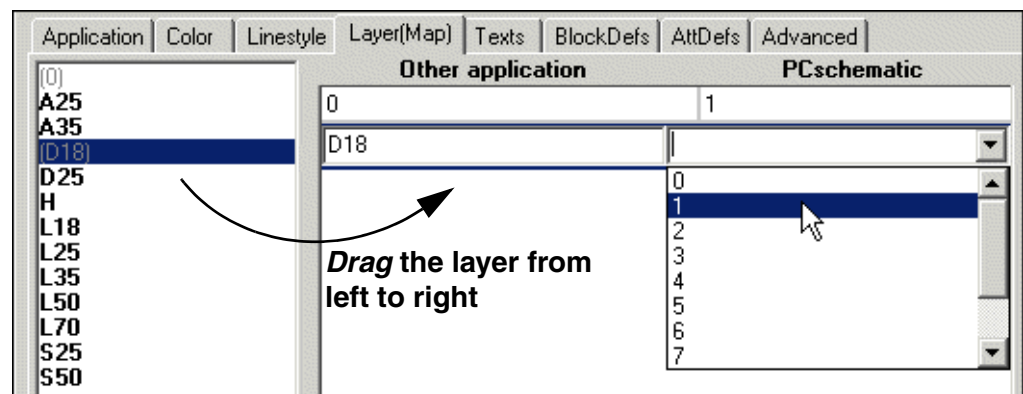
On the tab **Layer(Map)** you can specify that the contents of specified layers in the DWG/DXF file shall be placed on specified layers in the **Automation** project.

The different layers are represented by their layer numbers:



## Adding a new layer to the mapfile

To move a layer from the loaded drawing to the right side of the dialog box, just click on the layer name, and *drag* it to the right side of the dialog box. Hereby a new line is created, and you get the option to choose a layer number.



## Force symbol LAYER to

Click on **Force symbol LAYER to**, and set the value to 0.

By doing this you specify that all symbol definitions will be placed in layer 0, which *must* be the case in all **Automation** projects.

### Force project LAYER to

If you wish to place all project drawings on a specified layer in the **Automation** project, click on **Force project LAYER to**, and type in the name of the relevant layer.



#### LAYER 0 in Automation

PLEASE NOTE THAT LAYER **0** IN **Automation** IS USED FOR DEFINITIONS.

IF DRAWINGS ARE PLACED IN LAYER **0** IN **Automation**, YOU WILL THEREFORE NOT BE ABLE TO WORK ON THEM SUCCEDINGLY.

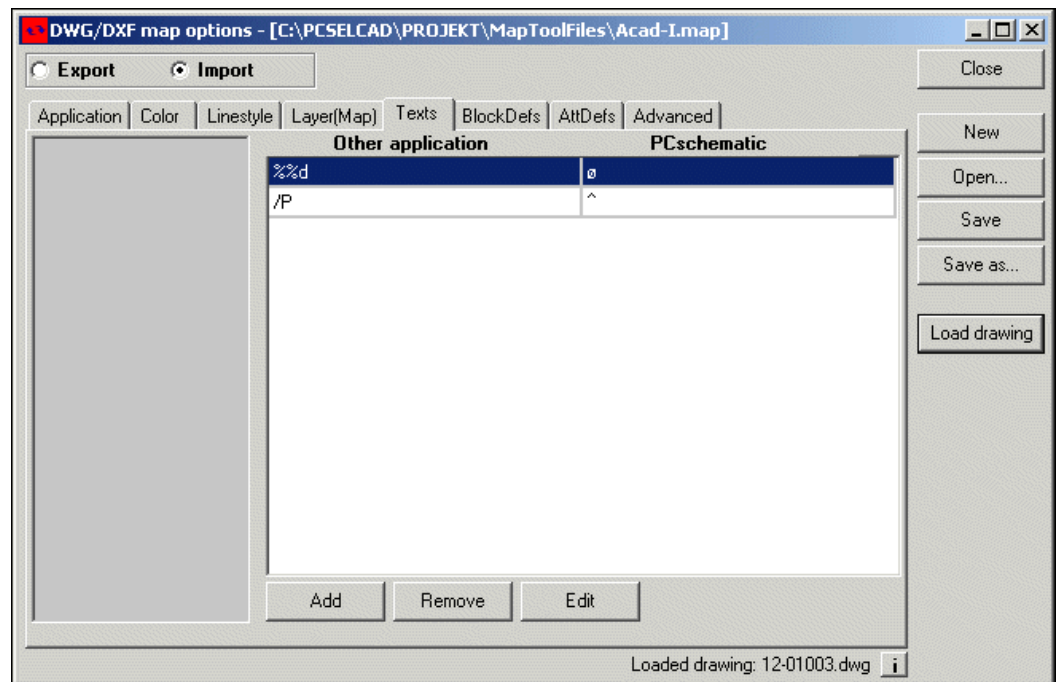
IF YOU HAVE PLACED DRAWINGS ON LAYER **0** IN THE DWG/DXF FILE, THESE MUST THEREFORE BE PLACED IN LAYER **1** (OR IN ANOTHER LAYER) IN THE **Automation** PROJECT. OTHERWISE, YOU WILL NOT BE ABLE TO CONTINUE WORKING ON THESE DRAWINGS.

---



## THE TAB TEXTS

On the tab **Texts** you specify whether certain texts in the DWG/DXF file shall be converted to other texts in the **Automation** project:



When for instance you know that **%%d** in the DWG/DXF file must be translated to the special character **ø** in the **Automation** project, you specify this on the tab **Texts** — as displayed above.

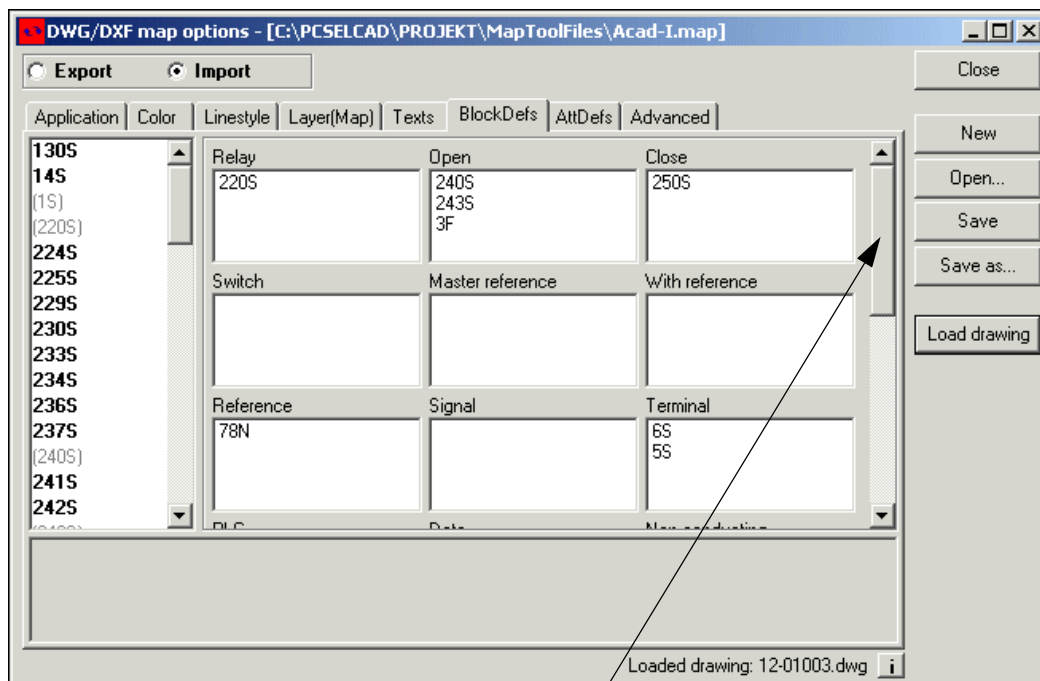
## Add, Remove and Edit

You can add, remove and edit texts as described in “Add, Remove and Edit” on page 27.

## THE TAB BLOCKDEFS

When converting DWG/DXF files, the *blocks* in the file are converted to *symbols* in the **Automation** project.

In order to be able to make use of the special options attached to the different symbol types in **Automation**, you can furthermore specify which symbol types to assign to the various symbols. This takes place on the tab **BlockDefs**:



Click here and *drag* downwards to see the rest  
— or *maximize* the dialog box

## See the blocks in the loaded drawing

When you have loaded a drawing — as described in “Opening and editing mapfiles” on page 18 — you can see which blocks exist in the loaded drawing in the left-hand side of the dialog box. You can then specify which symbol types these blocks shall be assigned in the **Automation** project during the conversion. This is done by clicking on the block name in the left-hand side of the dialog box, and then *dragging* it to the desired box in the right-hand side of the dialog box.

When a block name is dimmed in the left-hand side of the dialog box, this is because the name is already placed in one of the boxes in the right-hand side.

To get to the lower part of the boxes in the dialog box, just *drag* the elevator in the right-hand side of the tab.

### *Converting blocks in the example above*

In the displayed example above, you can for instance see:

- The block **220S** is assigned the symbol type **Relay**, and is therefore automatically assigned a reference cross / contact mirror in the **Automation** project
- The block **240S** is assigned the symbol type **Open**, and therefore automatically gets a reference to its relay in the **Automation** project

- The block **6S** is assigned the symbol type **Terminal**, and will therefore automatically be included in terminal lists and graphical terminal plans in the **Automation** project

#### *Blocks as drawing headers - or Exploded*

On the lower part of the tab, you can also specify that a block must be understood as a drawing header, or that a block must be exploded. When a block is exploded it is converted to lines, circles and texts on the page.

#### *Symbols with the status Ignored*

When you choose **Ignored**, the status for **Mechanical** and **Electrical** for the symbol is automatically **disabled** in the **Automation** project. This can be seen in the dialog box **Symbol article data** on the tab **General**. This has the consequence that the symbol will neither appear on electrical nor mechanical lists in **Automation**.

#### *Symbols which are deleted during the conversion process*

When you choose **Delete symbol**, the symbol is automatically deleted after the file has been converted.

This option can for instance be applied when the DWG/DXF file contains connection dots, which are created as symbols. Because these connection dots are created automatically in **Automation**, these should be deleted from the project file. — Please also refer to “Doughnut count” on page 40.

Ignored	Drawing header	Explode
<input checked="" type="checkbox"/>	N-SCHEME	<input type="checkbox"/>
Delete symbol		
1S		

#### **Defined text attributes in the individual blocks**

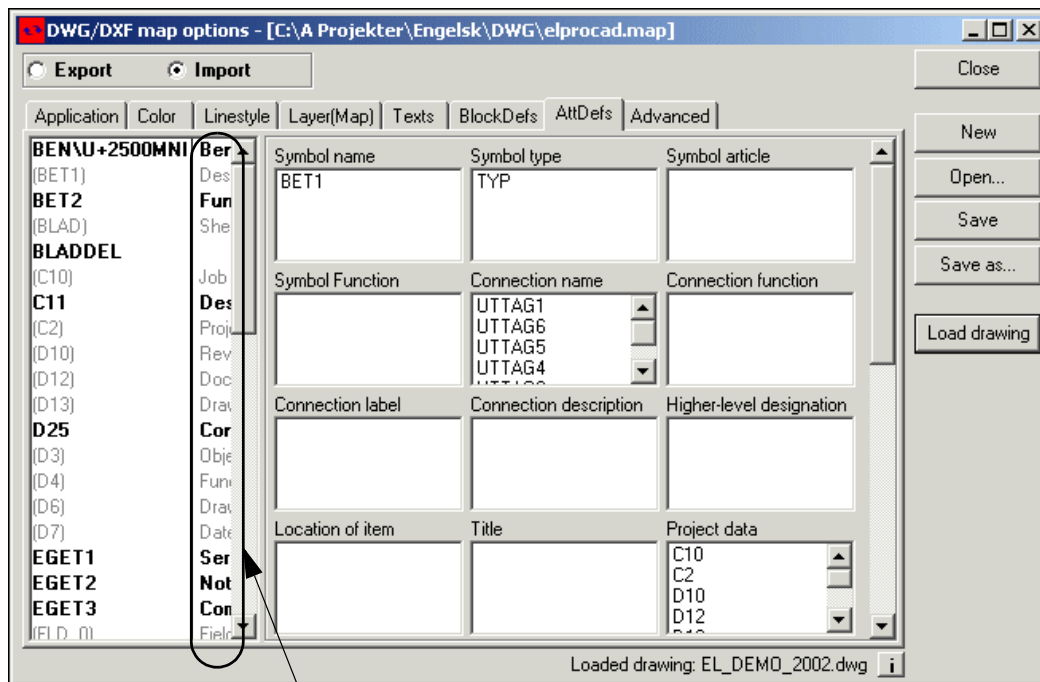
When you click on a block in the left-hand side of the tab, which has defined text attributes, these text attributes are displayed at the bottom of the dialog box.

— Read more about text attributes in “The tab Att-Defs” on page 36.

Attributes	Reference
ELPRPRI1	
ELPRY31V	
EPCB000	
EPCB001	
(EPCB001M)	
(EPCB001M)	
(EPCB001M)	
C2 Project name	
D3 Object name	
D4 Function	

## THE TAB ATTDEFS

On the tab **AttDefs** you can specify which text types or datafields in **Automation** shall contain the different text attributes in the AutoCAD file.



The prompt when placing the block in AutoCAD

This is done following the same directions as described in “The tab BlockDefs” on page 34.

When you drag in the elevator, you can furthermore see the text types displayed to the right.

### Different text attributes

The different text attributes can for instance be:

- All texts for Symbol article data
- All texts for Connection data
- System data fields

Page data	Title	Page number PAGE-NO.
Used pages	Date for last change	First used page
Last used page	Next used page	Previous used page
User name	File name	Print date
Print scale	Delete text	

### Converting the attributes in the example above

In the displayed example above, you can for instance see:

- The text attribute **BET1** will be interpreted as the **Symbol name** for a symbol in the **Automation** project.
- The text attribute **UTTAG1** will be interpreted as the **Connection name** for a symbol. For each connection name the program meets in a block in the AutoCAD file, the program automatically creates a connection point on the symbol in the **Automation** project.

- The text attribute **C10** will be interpreted as ***Project data***, and the program will therefore automatically create a project data datafield with the name **C10** in the **Automation** project. This datafield will be seen in the dialog box **Project data** for the project.

### **The Prompt from the AutoCAD file is displayed as a help**

For some of the AutoCAD text attributes, there is displayed a text, which is the prompt that appears when placing the block in AutoCAD. These is meant as a help for correct conversion of the text attributes.

## THE TAB LAYER(DEF)

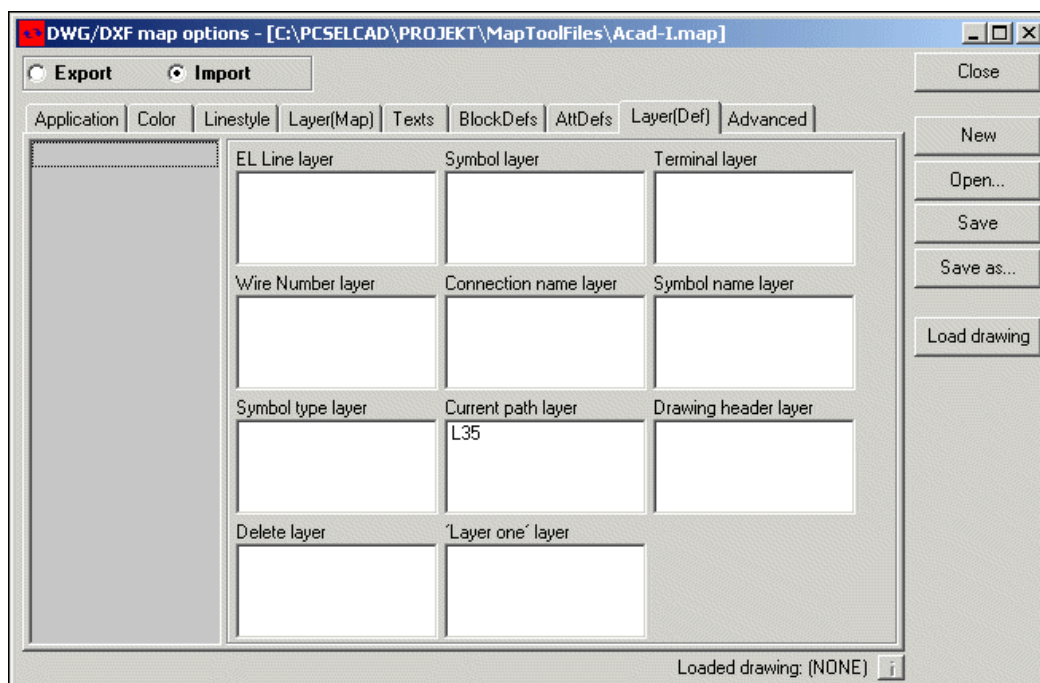
On the tab **Layer(Def)** you specify which layers in the AutoCAD file that contains the different diagram elements.



### THE TAB LAYER(DEF) IS ONLY DISPLAYED FOR SOME ELECTRICAL APPLICATION PROGRAMS

BECAUSE ONLY SOME ELECTRICAL APPLICATION PROGRAMS PLACES DIFFERENT OBJECT TYPES ON DIFFERENT LAYERS, THIS TAB IS ONLY DISPLAYED FOR AUTOCAD FILES FROM SOME ELECTRICAL APPLICATION PROGRAMS.

THIS TAB WILL NOT BE AVAILABLE IN A NORMAL INSTALLATION.



When different types of information is placed on specified layers in the DWG/DXF file, the program can fetch this information, and attach it intelligently to the relevant objects in the **Automation** project — such as e.g. symbol names, connection names or symbol types.

When wire numbers are placed on a specified layer, these are hereby created automatically as wire number symbols in the **Automation** project, and are placed on the correct line.

Or, when for instance terminals are placed on a specified layer in the DWG/DXF file, the program automatically creates all blocks on this layer as symbols with the type **Terminal** in the **Automation** project. Or, if the drawing header is placed on a specified layer, *all of the contents* on this layer is created as a drawing header in the **Automation** project.

*Current path numbers are transferred intelligently*

When there are current path numbers on a specified layer, these can be transferred to **Automation** as current path numbers — which can be adjusted from the tab **Settings => Page data => References** succeedingly, if necessary.

*Delete layer*

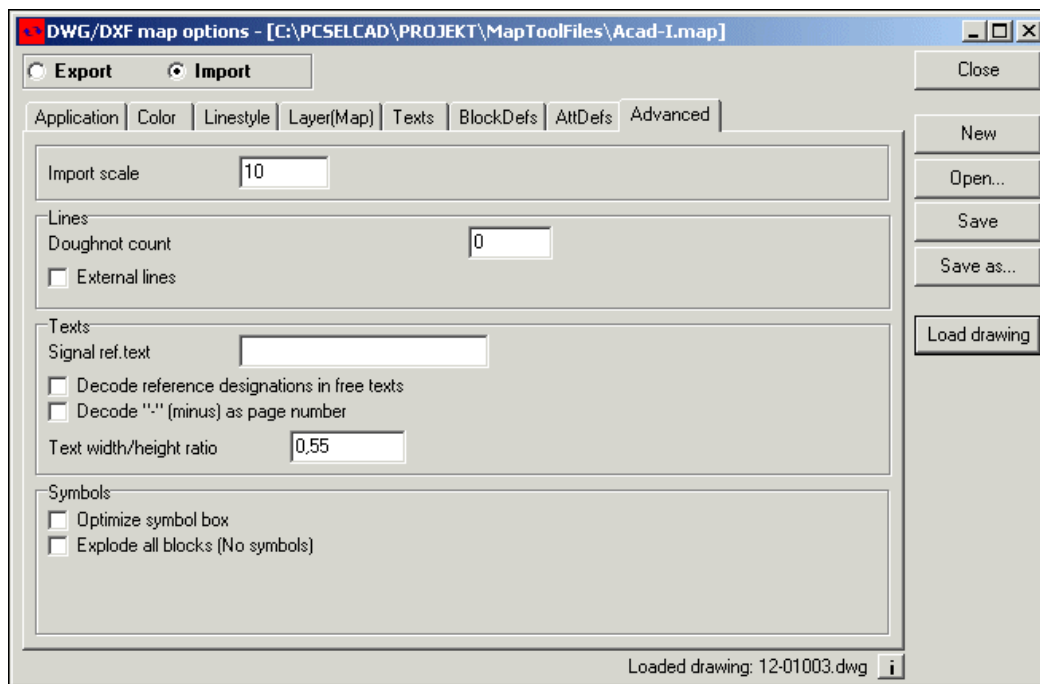
When you drag a layer name to the **Delete layer** box, none of the contents on this layer is included in the **Automation** project.

*'Layer one' layer*

All layers, which are placed here, are automatically placed on layer 1 in the **Automation** project.

## THE TAB ADVANCED

The last tab in the dialog box, is the tab **Advanced**.



Here you can make the following settings:

### Import scale

When you wish to scale the DWG/DXF file during the conversion, you can specify to scale the drawing in e.g. 1:10, in the field **Import scale**.

### Lines

Under **Lines** you have two options:

#### *Doughnut count*

Because **Automation** automatically places connection dots (doughnuts), it is a good idea to delete these from the DWG/DXF file, if the file contains doughnuts. If you know that the doughnuts are drawn as for instance 7 or 13 line segments, you can specify this here.

Hereby the program automatically deletes all POLYLINES drawn with for instance 7 or 13 line segments.

If these doughnuts are created as symbols in the project, these are deleted as described in "Symbols which are deleted during the conversion process" on page 35.

#### *Set connection points to external when dotted lines*

If you select **Set connection points to external when dotted lines**, the connection points for terminals are automatically set to IO status **Output** if these are connected to a dotted line.



## Texts

Under **Texts** you have these options:

### *Signal ref. text*

When you type in a text in this field, this text is inserted as pre-text for all signal references in the project. When for instance you write *Page*, this text is placed at the beginning of the signal reference.

### *Decode reference designations in free texts*

When you select **Decode reference designations in free texts**, all free texts in the DWG/DXF file are examined for reference designations.

When reference designations are found in texts — this is if the texts contains the characters "=" and/or "+" — these will be understood as reference designations in the **Automation** project.

In symbol names this means that the reference designation is deleted from the symbol name, and then attached intelligently to the symbol, as the reference designation for the symbol in the **Automation** project.

When there is a free text on the page containing a reference designation, this text is deleted from the page, and attached intelligently as the reference designation for the page in the **Automation** project.

### *Decode "-" (minus) as page number*

In some electrical application programs the character "-" (minus) is used for representing the name of the page in symbol names. When it for instance says **-K1** on page **10** in the file, this means that the name of the symbol is **10K1**. Correspondingly, if it says **-K1** on page **11**, this means that the name of the symbol is **11K1**.

When you choose **Decode "-" (minus) as page number**, the names of these two relay coils are changed automatically to **10K1** and **11K1** respectively during the conversion process.

### *Text width/height ratio*

The ratio between text width and text height can vary from one text font to another.

When you translate a text in a DWG/DXF file to a text in **Automation**, it can therefore be necessary to specify a ratio between text height and width. Hereby you can for instance avoid that texts during the conversion will take up too much space, and therefore overlap symbols in the project.

The ratio between text width and height is specified in the field **Text width/height ratio**.

It can be necessary to make this adjustment when the two systems does not use the same text fonts.

## Symbols

Under **Symbols** you have these options:

*Optimize symbol box*

Some blocks have their reference points in (0,0), which can be quite far from the block drawing. When you select **Optimize symbol box**, reference points are placed on the symbol drawing.

*Explode all blocks (No symbols)*

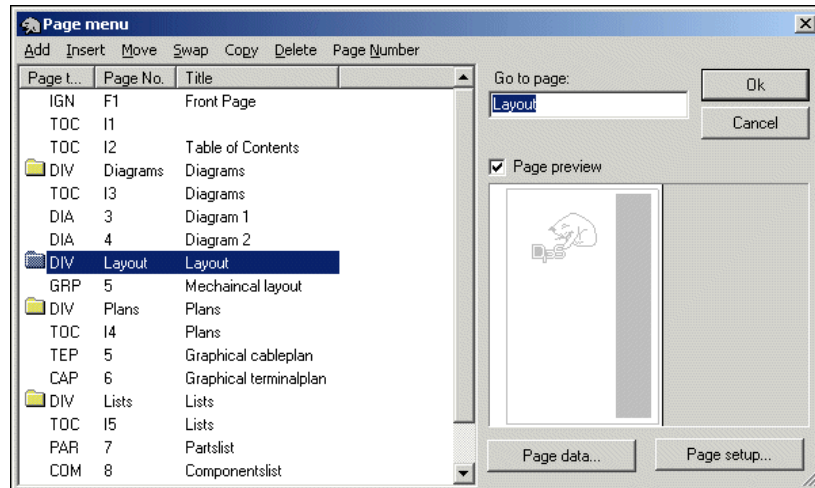
When you select **Explode all blocks (No symbols)**, all blocks in the DWG/DXF file are converted to lines, arcs and texts in the created **Automation** project. Hereby you get a project containing no symbols.

## IMPORTING A DWG/DXF FILE IN AN EXISTING PROJECT

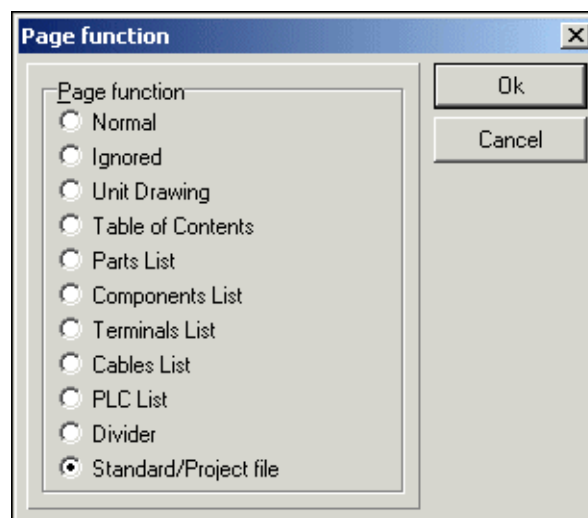
When you have a **Automation** project open and wish to import a DWG/DXF file, do the following:



- 1) Click on the **Page menu** button.

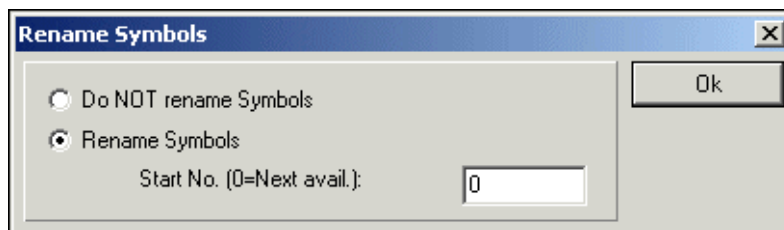


- 2) In the **Page menu**: Click where you wish to insert the file as a new page and click **Insert** — or click on **Add** to insert this page as the last page in the project. — Read more about the **Page menu** in "The Page menu" in the **Automation** manual.
- 3) In the dialog box **Page function**: Click on **Standard/Project file**, and then click on **OK**.



- 4) In the dialog box **Open**: Click on the folder where you have placed the file, and choose **Acad files (\*.dwg)** or **Dxf files (\*.dxf)** as **Filetype**, click on the file, and then on **Open**.
- 5) A new **Open** dialog box appears: Select a mapfile, and click on **Open**.

- 6) In the dialog box **Rename symbols**: Make the required settings — see "Intelligent copying of areas" in the **Automation** manual if necessary —, and click **OK**.



### *Opening more than one DWG or DXF file*

- 7) You have now loaded the DWG/DXF file, and are still in the **Page menu**. When you need to load more DWG/DXF files, repeat the same procedure for these files.
- 8) When you have finished loading files: Click **OK** to leave the **Page menu**.



### **USING DWG/DXF map options**

WHEN YOU IMPORT INTO AN EXISTING PROJECT, YOU CAN ALSO APPLY THE TOOL **DWG/DXF map options** FOR ADAPTING THE APPLIED MAPFILE, AS DESCRIBED IN "THE WORK PROCESS WHEN CONVERTING DWG/DXF FILES" ON PAGE 16.

## **IMPORTING DWG/DXF FILES AS SYMBOLS**

It is also possible to fetch a DWG/DXF file as a symbol. This has the advantage that you can move it as a symbol, and that you therefore do not risk to make changes in it by accident.

When you wish to load a DWG/DXF file as a symbol, do the following:



- 1) Click on the **Symbols** button and then on the **Symbol menu** button.
- 2) In the **Symbol menu**: Click on **Create new Symbol**.
- 3) In the **Symbol editor**: Click on the **Open** button.
- 4) In the dialog box **Open**: Choose **Acad files (\*.dwg)** or **Dxf files (\*.dxf)** in the **Filetype** field, find the file and click **Open**.
- 5) In the dialog box **Open map file**: Select the desired mapfile and click **Open**.
- 6) The DWG/DXF file is now loaded in the **Symbol editor**: Make the necessary changes — if necessary — and choose **File => Save as**.
- 7) In the dialog box **Symbol Settings**: Specify the symbol settings for the symbol — as described in "Saving the symbol" in the **Automation** manual — and click **OK**.
- 8) In the dialog box **Save as**: Give the symbol a name, choose which folder to save it in, and click **Save**.

9) In the **Symbol editor**: Choose **File => Close** — to leave the **Symbol editor**.

10) In the **Symbol menu**: The symbol is displayed.

11) Click on **OK**, and you get the loaded drawing in the cross-hairs as a symbol. Click where you wish to place the symbol in the active project.

Please note that the program automatically places a reference point in the lower left-hand corner, where all texts are placed also. The symbol is a normal symbol, with selection in the reference point. — Read more about this in "Symbol types" in the **Automation** manual.

When you wish to edit the loaded symbol — or save it in a symbol library — you can choose **Edit** in the **Symbol menu**. Here you can edit and save the symbol as for normal symbols.

— Read more about working with symbols in "Symbols" in the **Automation** manual, and about how to edit symbols in "Creating Symbols" in the **Automation** manual.

## EXPORTING DWG/DXF FILES FROM Automation

When exporting objects which the DWG/DXF format can not handle, **Automation** uses objects known by the DWG/DXF format when possible.

Electrical properties can not be transferred via the DWG/DXF format. Electrical lines will therefore be exported as "simple lines".

When you wish to export DWG/DXF files from **Automation**, you have two options:

- 1) Exporting a project as a DWG/DXF file — see “Exporting projects as DWG/DXF files” on page 46.
- 2) Exporting a symbol as a DWG/DXF file — see “Exporting symbols as DWG/DXF files” on page 47.

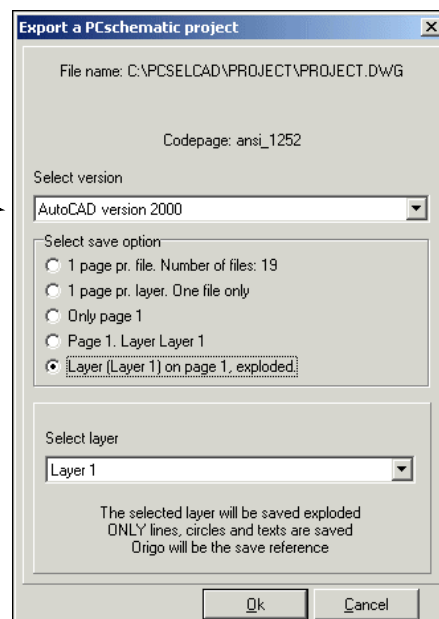
— Please also see “Map settings for Export of DWG/DXF files” on page 48.

### Exporting projects as DWG/DXF files

When you wish to export a **Automation** project as a DWG/DXF file, do the following:

- 1) Choose **File => Save as**, set the filetype to **Acad files (\*.dwg)** or **Dxf files (\*.dxf)**, type in a file name, and click on **Save**.
- 2) You now enter the dialog box **Export of a PCschematic project**:

Choose which version to export to



In addition to choosing which version to export to, you here have the follo-

wing options:

Option	Consequence
<b>1 page pr. file</b>	Each page in the project is exported as a separate file.
<b>1 page pr. layer. One file only</b>	Each page in the project is saved on a separate layer. This way you can save all of the project in the same file. If the <b>Automation</b> project contains layers, the division in layers no longer exists.
<b>Only page 1</b>	Only the active page in the project is exported.
<b>Page 1. Layer: Layer 1</b>	Only the selected layer on the active page in the project is exported. The layer is selected at the bottom of the dialog box, where you get the option to choose between the layers on the page.
<b>Layer (Layer 1) on page 1, exploded</b>	When you choose this option, you choose which layer to explode, and then export. Output can for instance be applied for a numerically controlled machine.

Select the desired settings, and click **OK**.

- 3) In the dialog box **Open mapfile**: Choose the desired mapfile, and click **Open**.

The chosen part of the project is now saved in the DWG/DXF format.



#### USING DWG/DXF map options FOR EXPORT

WHEN EXPORTING TO THE DWG/DXF FORMAT, YOU CAN ALSO USE THE TOOL **DWG/DXF map options**, AS DESCRIBED IN “THE WORK PROCESS WHEN CONVERTING DWG/DXF FILES” ON PAGE 16.

### Exporting symbols as DWG/DXF files

It is also possible to save a symbol as a DWG or a DXF file:



- 1) Click in **Symbols** and then on the **Symbol menu** button — or use the short-cut key [F8].
- 2) In the **Symbol menu**: Find the symbol you wish to save as a DWG/DXF file, click on it and click on **Edit symbol**.
- 3) In the **Symbol editor**: Choose **File => Save as**.
- 4) In the dialog box **Symbol settings**: Click **OK**.
- 5) In the dialog box **Save as**: Set the filetype to **DWG files (\*.dwg)** or **DXF files (\*.dxf)**, type in a file name, and click **Save**.
- 6) In the dialog box **Open mapfile**: Choose the mapfile and click **Open**.
- 7) In the **Symbol editor**: Choose **File => Close** — to leave the **Symbol editor**.

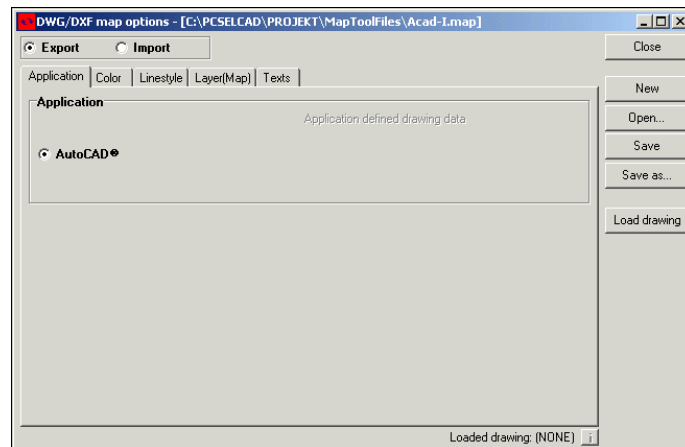
8) In the **Symbol menu**: Click **Cancel** to return to the project.

— Read more about how to edit and save symbols in "Creating Symbols" in the **Automation** manual.

### Map settings for Export of DWG/DXF files

When you export from **Automation** to the DWG/DXF format, you do not have as many options available, as when importing DWG/DXF files in **Automation**.

Therefore you only have the first part of the tabs in the tool **DWG/DXF map options** available:



— Read more about the options on the individual tabs in "DWG/DXF map options" on page 21.





## Symbol documentation

In PCISCHMATIC Automation you can automatically document symbol libraries, or document the symbols used in a project.



# SYMBOL DOCUMENTATION

THIS CHAPTER DESCRIBES	
— ON SYMBOL DOCUMENTATION .....	(P. 51)
— SETTINGS IN SYMBOL DOCUMENTATION .....	(P. 52)
— THE TAB SYMBOLS .....	(P. 53)
— THE TAB TEXTS .....	(P. 54)
— THE TAB PAGES .....	(P. 56)
— THE TAB ADJUSTMENTS.....	(P. 59)

## ON SYMBOL DOCUMENTATION

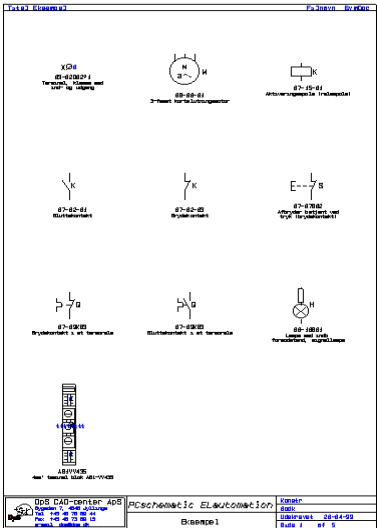
In **Automation** it is possible to create graphical symbol documentation automatically.

This can either be documentation of a symbol library, or documentation of the symbols used in a project.

You can print out a graphical survey of the symbols, as well as a list of the symbols, ordered by their symbol title. — Please refer to page 56 for further details.

The documentation is created as an independent **Automation** project, and can be printed out as a such.

To create the symbol documentation, you start by choosing **Tools => Symbol documentation**.



## SETTINGS IN SYMBOL DOCUMENTATION

You now enter the dialog box **Symbol documentation**, which contains the following four tabs:

Tab	Function	Page
<b>Symbols</b>	Here you specify what you wish to document.	(p. 53)
<b>Texts</b>	Here you specify which texts you wish to display in the graphical documentation.	(p. 54)
<b>Pages</b>	Here you specify which drawing header you wish to apply on the <i>graphical</i> documentation, as well as which drawing header you wish to apply if you print out a <i>list</i> ordered by for instance symbol names.	(p. 56)
<b>Adjustments</b>	Here you specify the guidelines for the location of the symbols in the documentation, as well as the number of symbols on each documentation page.	(p. 59)

The functions on the tabs is elaborated in the following.

## Create the documentation

When you have specified the settings you want on the tabs, you click **OK** in the right hand side of the dialog box. Then a new project is created, containing the complete symbol documentation. The project containing the symbol documentation can now be printed as an ordinary **Automation** project.

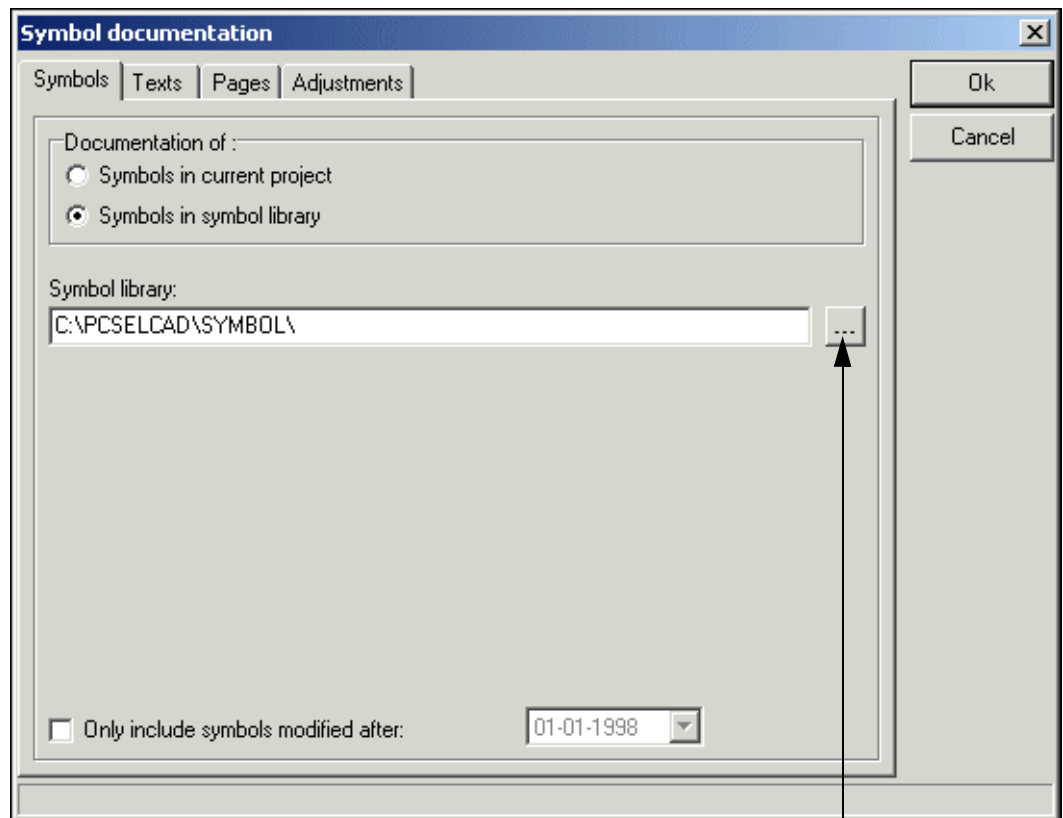
## SYMBOLS

When you wish to create a graphical survey, you specify which symbols you wish to document on the tab **Symbols**.

### Specify what you wish to document

If you have opened a project and wish to document the symbols in this project, click on ***Symbols in current project***.

If you wish to document an entire symbol library, click on ***Symbols in symbol library***, and then click your way to the desired library.



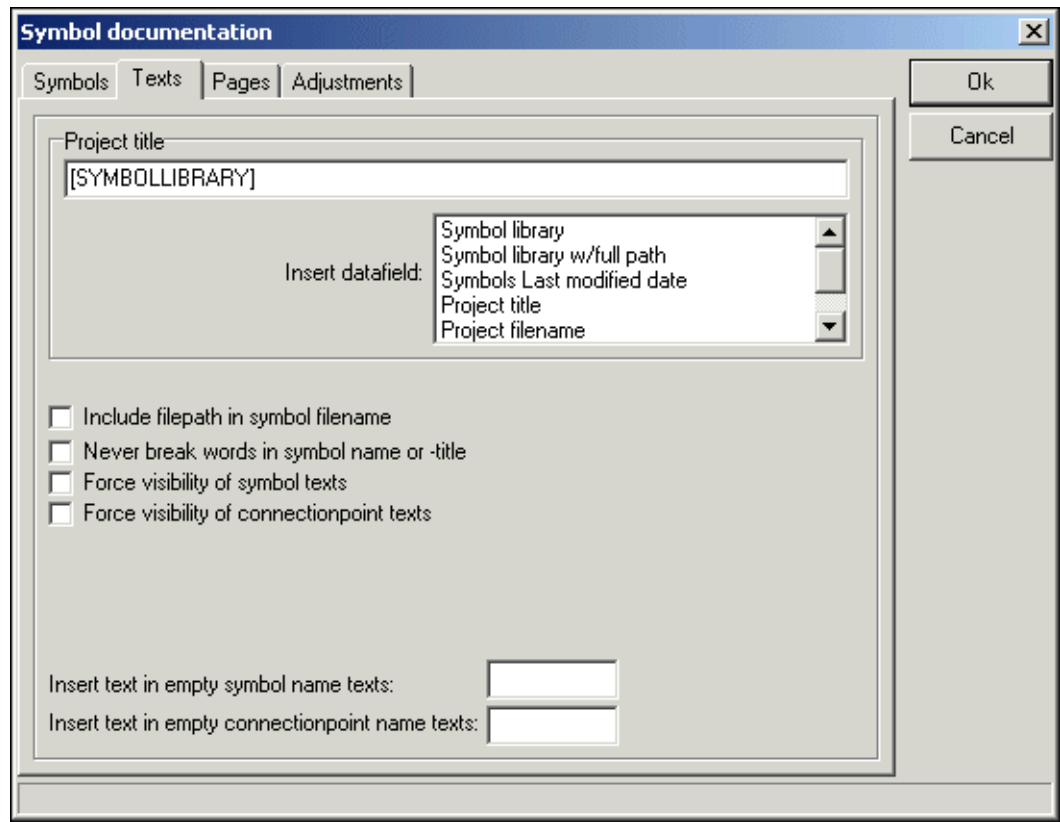
Choose library

### Other settings

If you only wish to print out symbols, which have been created/changed after a certain date, you can enable ***Only include symbols modified after*** on the lower part of the tab, and specify the date in the date field.

TEXTS

On the tab **Texts** you specify which texts you wish to include into the documentation.

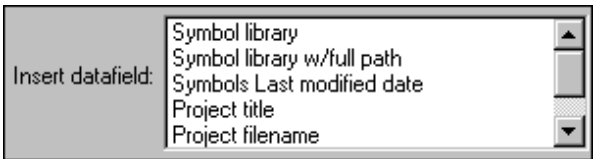


Project title in the symbol documentation

In the text field **Project title** you specify the contents of the data field **Project data** => **Title** in the documentation project. The text in this data field is inserted in the drawing headers on each page in the documentation project.

You can type in a text yourself, as well as insert information from the **Insert data field** list.

From **Insert datafield** you can choose the data fields mentioned in the table below. You can either double-click on a data field, or *drag* it to the **Project title** text field.



When you double-click on the one of the data fields, it is inserted where the cursor last was placed in the **Project title** text field. Remember to place a [space] sign when necessary.

When you let the mouse pointer rest on the **Project title** field, a hint is displayed, showing how the title will be printed in the final symbol documentation project.

Data field	States	Displayed as
Symbol library	The name of the symbol library.	[SYMBOLLIBRARY]
Symbol library w/ full path	The name of the symbol library, and the full path leading to the library.	[SYMBOLLIBRARY-PATH]

Data field	States	Displayed as
Symbols last modified date	The date specified under <b>Only include symbols modified after</b> on the tab <b>Symbols</b> .	[SYMBOLSMODIFIEDDATE]
Project title	The name from the <b>Project title</b> field in the dialog box <b>Settings =&gt; Project data</b> .	[PROJECTTITLE]
Project file name	The name of the project file.	[PROJECTFILENAME]
Project full path	The path leading to the project file, as well as the name of the project file.	[PROJECTFULLPATH]
Current date	The current date.	[DATE]

### The texts for the individual symbols

Under each symbol in the graphical survey, the name and the title of the symbol is displayed.

Furthermore, you can specify which other texts shall be placed below the individual symbols in the graphical survey.

☐ Include filepath in symbol filename  
☐ Never break words in symbol name or -title  
☐ Force visibility of symbol texts  
☐ Force visibility of connectionpoint texts

Enable	Signifies
Include file path in symbol name	The entire path is specified for each symbol file name.
Never break words in symbol name or -title	Neither the words in the symbol file name nor the title (description) must be hyphenated.
Force visibility of symbol texts	Symbol texts, which are invisible in the symbol definitions, will be visible in the documentation. If this option is disabled, only symbol texts, which are set to be visible in the symbol definition, will be printed.
Force visibility of connection point texts	Connection point texts, which are invisible in the symbol definitions, will be displayed in the documentation. If this option is disabled, only connection point texts, which are set to be visible in the symbol definition, will be printed.

### Empty texts

If there are empty texts in the symbol definitions, and you would like to see how the texts are placed, you can fill in sample texts in the text fields **Insert text in empty symbol name texts** and **Insert text in empty connection point name texts**.

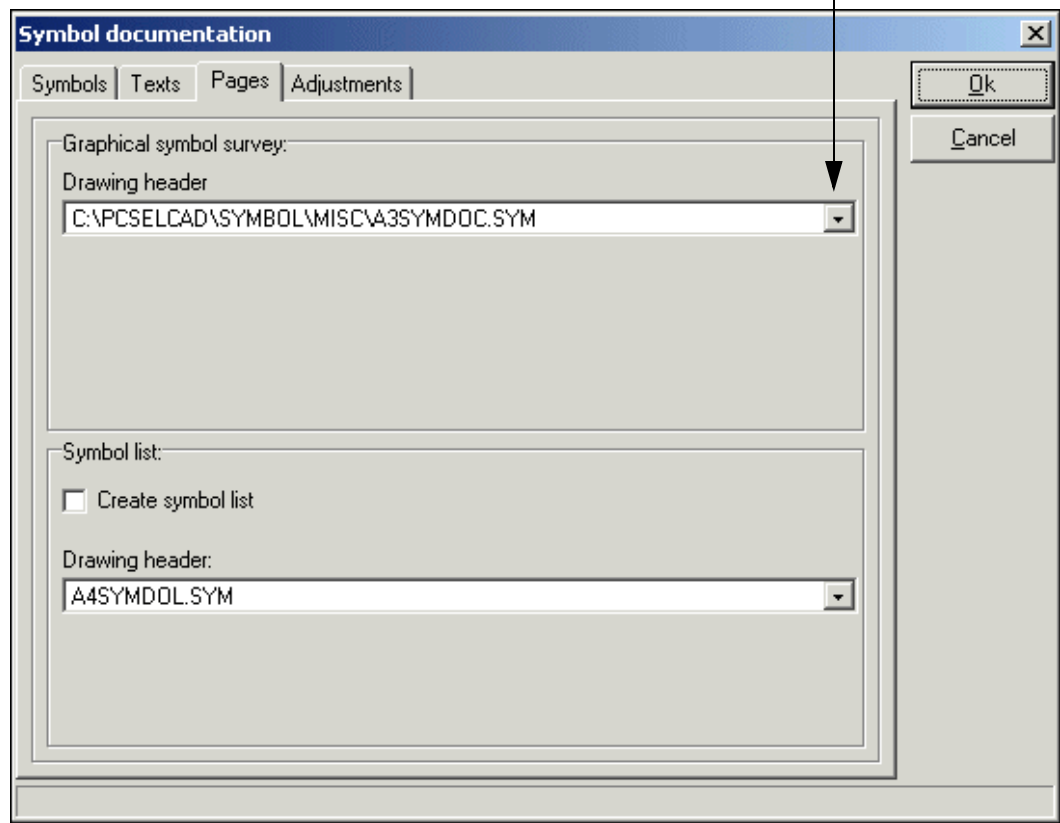
Insert text in empty symbol name texts:   
 Insert text in empty connectionpoint name texts:

For instance you could type in *NAME* in the text field for the symbol name texts, and *NNNN* in the text field for the connection point names.

PAGES

When you wish to specify which drawing headers you wish to apply in the symbol documentation, you click on the **Pages** tab. You then enter this dialog box:

Click here for a list of drawing headers



Graphical symbol survey

In the field **Drawing header** — under **Graphical symbol survey** — you choose which drawing header you wish to apply when you create the *graphical* part of the documentation.

The following drawing headers for symbol documentation can be found in the symbol library **misc**, and on the list of drawing headers in **Automation**:

Drawing header	Use
<b>A3SYMDOC</b>	A3 drawing header for documentation of symbols (On A4 paper). The symbols are listed in the documentation sorted by symbol file names.
<b>A4SYMDOC</b>	A4 drawing header for documentation of symbols. The symbols are listed in the documentation sorted by symbol file names.
<b>A4SYMDOL</b>	A4 drawing header for documentation of symbols. The symbols are listed in the documentation sorted by symbol titles.

You can apply one of these drawing headers, or a drawing header you have designed yourself. When you click on the down-arrow in the **Drawing header** field, you get the list of drawing headers in **Automation**.



— If you type in the name of the drawing header — opposed to clicking on the name of a drawing header — you have to type in the entire path leading to the drawing header.

### *User designed drawing headers*

If you have designed your own drawing header for symbol documentation purposes, you have to insert the drawing header on the list of drawing headers.

This is done by choosing **Settings => Text/Symbol defaults**, clicking on **Drawing headers**, and then on **Add**. This brings you into the **Symbol menu**, where you click on the required drawing header, and then click **OK**.

You will not be able to choose the drawing header from **SymDoc** until you have done so.

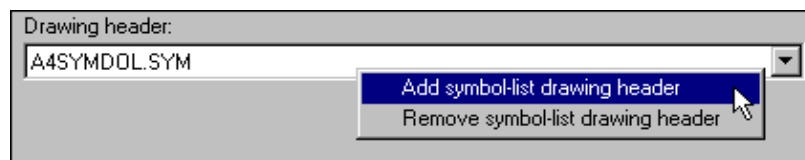
— Please refer to the **Automation** manual for further details about this, and about how to create drawing headers.

### **Drawing headers for symbol lists**

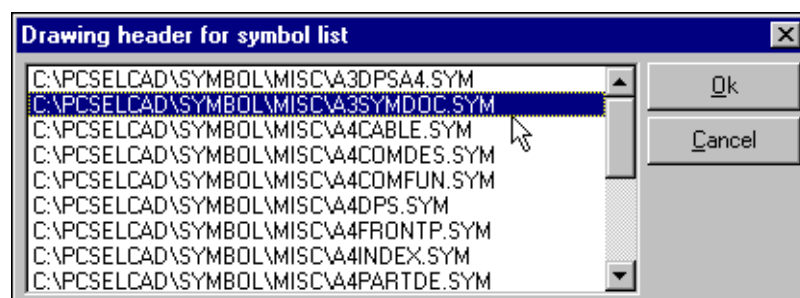
If you also want a list of the symbols, you enable **Create symbol list** in the **Symbol list** part of the tab. This list can for instance be arranged according to the symbol names, and contain references to the locations of the symbols in the graphical survey.

In the field under **Drawing header**, you can then specify which drawing header you wish to apply when you create the list.

Clicking on the down-arrow in the **Drawing header** field, gives you the list of drawing headers for symbol lists. If the required drawing header is not on the list of symbol list drawing headers, you right-click on the field, and get the following options:



Here you click on **Add symbol-list drawing header**, and enter the list of drawing headers in **Automation**.



Click on the desired drawing header, and then on **OK**.

By clicking on **Remove symbol-list drawing header** you can accordingly remove drawing headers from the list of drawing headers for symbol lists.

*Creating your own drawing headers*

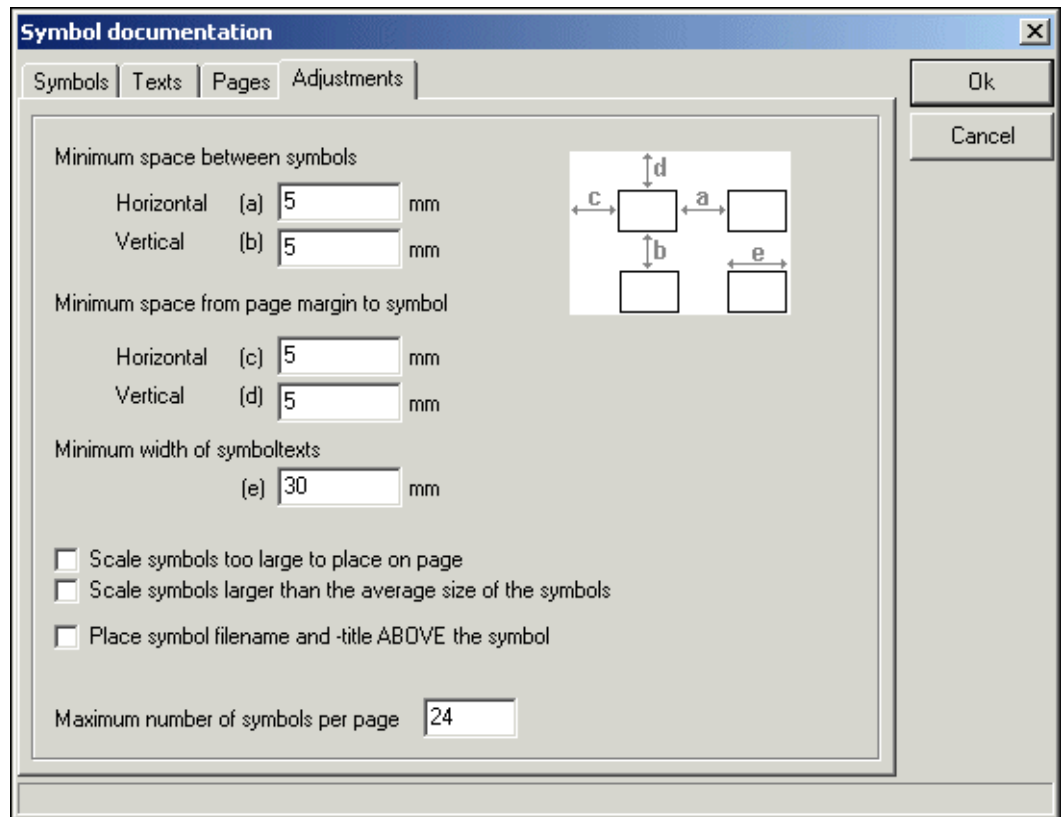
Please refer to the **Automation** manual to see how to design your own drawing headers / lists, as well as to see how to insert these on the list of drawing headers in **Automation**. — If you wish to design your own drawing headers for symbol documentation and lists, it is recommended to edit the existing drawing headers. This is because the datafields are used in a special way in these drawing headers.

*Create the documentation*

When you have specified all the settings, you click **OK** to create the symbol documentation, as described in “Create the documentation” on page 52.

## ADJUSTMENTS

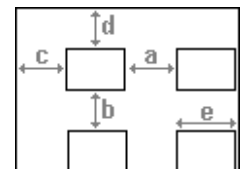
When you wish to specify the space between the symbols on the page, you click on the **Adjustments** tab:



Here you can specify the following settings, which are all illustrated on the figure on the tab.

### *Minimum space between symbols*

In the fields **Horizontal (a)** and **Vertical (b)** the space between the symbols are specified. — Please note, that the figure displays how much room the symbols *and* their texts take up.

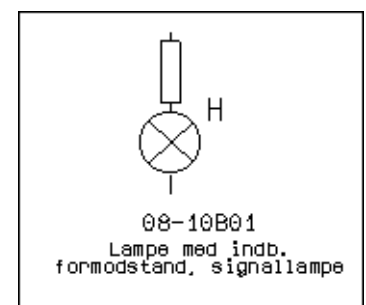


### *Minimum space form page margin to symbol*

In the fields **Horizontal (c)** and **Vertical (d)** the page margins are specified. The horizontal goes for the left- as well as the right margin on the page, while the vertical specifies the top- and bottom margin.

### *Minimum width of symbol texts (e)*

When the program calculates how much room is needed for the symbols on the page, it starts off considering the size of the symbols. If the symbols are relatively small, not much room will be left for the symbol texts. It is therefore a good idea to specify how much space the symbol texts needs as a minimum. For the displayed symbol, the text would for instance be hyphenated over several lines, if only the width of the symbol should determine the width of the text.



### *Scale symbols in the documentation project*

If you wish to downscale the largest symbols in the documentation, you have two different options at hand:

<input type="checkbox"/>	Scale symbols too large to place on page
<input type="checkbox"/>	Scale symbols larger than the average size of the symbols
<input type="checkbox"/>	Place symbol filename and -title ABOVE the symbol

If you enable **Scale symbols too large to place on page**, you downscale symbols, which are too large to be placed on a documentation page in scale 1:1.

If you enable **Scale symbols larger than the average size of the symbols**, you downscale symbols, which are larger than the average size of the symbols in the entire documentation project.

When necessary, the symbols will be downscaled to either 1:2, 1:4 or 1:8. If a symbol have been downscaled, this will be mentioned after the filename in the documentation.

### *Symbol text above the symbol*

If you enable **Place symbol filename and -title ABOVE the symbol**, the symbol filename and the symbol title will be placed *above* the symbols in the documentation. If the function is disabled, these texts will be placed *below* the symbols.

### *Maximum number of symbols per page*

On the lower part of the tab, you can specify maximum number of symbols you wish to place on a symbol documentation page.

The size of the symbols, as well as the settings you have specified on the upper part of the tab, puts a natural limit for how many symbols can be placed on the page.

Therefore it only makes sense to fill out this field if you want *less* symbols placed on a page, than the program would suggest.



## Graphical Terminal plans

In PCISCHMATIC Automation you can automatically create graphical terminal plans for the active project.



# GRAPHICAL TERMINAL PLANS

In **Automation** you can create graphical terminal plans for your projects.

THIS CHAPTER DESCRIBES	
— ON GRAPHICAL TERMINAL PLANS.....	(P. 63)
— CREATING GRAPHICAL TERMINAL PLANS .....	(P. 64)
— THE STRUCTURE OF GRAPHICAL TERMINAL PLANS .....	(P. 65)
— SETTINGS IN CREATE GRAPHICAL TERMINAL PLAN .....	(P. 66)
— THE TAB CONNECTIONS .....	(P. 67)
— THE TAB PAGES .....	(P. 69)
— THE TAB SYMBOLS .....	(P. 71)
— THE TAB WIRES .....	(P. 76)
— THE TAB ADJUSTMENTS.....	(P. 77)
— ADJUSTMENTS FOR TERMINAL PLANS ACCORDING TO CABLES .....	(P. 79)
— UPDATING EXISTING GRAPHICAL TERMINAL PLANS.....	(P. 80)

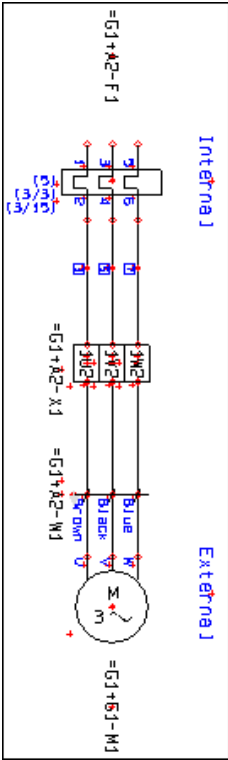
## ON GRAPHICAL TERMINAL PLANS

The graphical terminal plan for a project can either be created as an independent **Automation** project, or be inserted in the active project.

When the terminal plan is created, you can edit it by moving lines and symbols, as on any diagram page. If you, at a later stage, need an updated version of the terminal plan, you can transfer the changes you have made in the terminal plan to the new terminal plan automatically.

— When a terminal plan does not turn out the way you expected it to, you can always change the terminal plan settings, and create new terminal plan.

Graphical terminal plans can be printed just like you normally print pages in **Automation** projects.

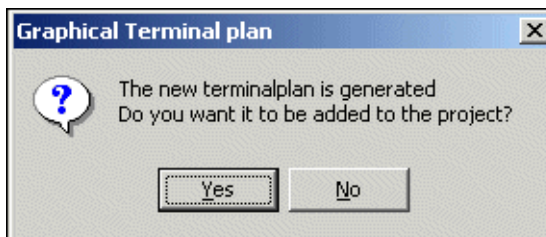


## CREATING GRAPHICAL TERMINAL PLANS

To create a graphical terminal plan for a project, open the project, and choose **Tools => Graphical terminal plan** in **Automation**. You then enter the dialog box **Create graphical terminal plan**.

If you have already created a graphical terminal plan, and have inserted it in the active project, you will enter the dialog box **Update graphical terminal plan**, as described in “Updating existing terminal plans” on page 80.

When the terminal plan is created, you will be asked whether you wish to insert it in the project — if you have not inserted it already:



### *Answering Yes - to inserting the terminal plan in the project*

If you answer **Yes**, the terminal plan is inserted in the project, along with a divider page with the title **TermPlan**. When you make manual changes in this terminal plan, these can be included when you generate a new terminal plan.

The pages in the terminal plan are assigned the page type **TEP**. In **Automation** projects they therefore get no effect on the electrical and mechanical lists in the project.

When you edit the diagram pages in the **Automation** project, the terminal plan is *not* updated automatically. To get an updated version of the terminal plan, you must therefore choose **Tools => Graphical terminal plan** again — as described in “Updating existing terminal plans” on page 80.

### *Answering No - to inserting the terminal plan in the project*

If you answer **No** in the dialog box, the terminal plan is created as an independent **Automation** project with the name **Termplan**. If you later make changes in the original project, and therefore wish to generate a new terminal plan, you will *not* have the option to include manually performed changes, from the old terminal plan, in the new terminal plan. This will not be possible until you copy the terminal plan into the original **Automation** project.

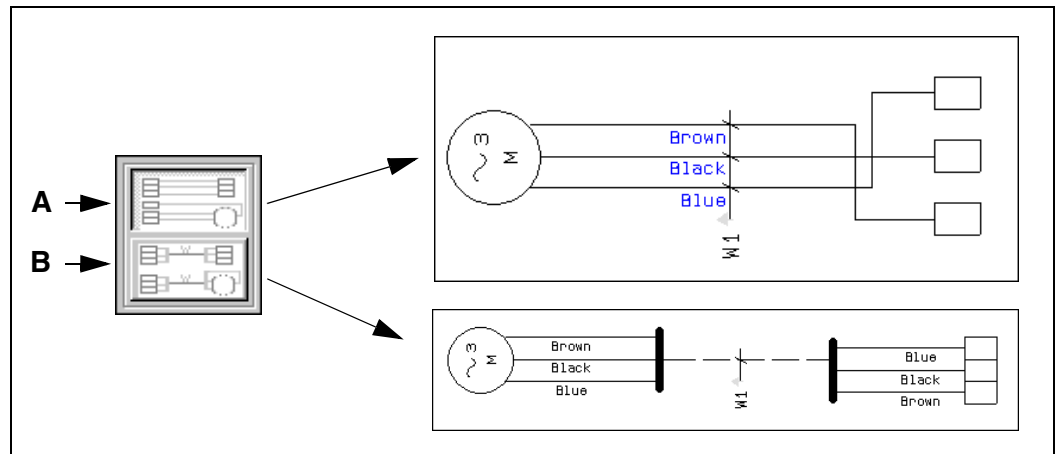


## THE STRUCTURE OF THE TERMINAL PLAN

The graphical terminal plan can be created in two ways:

- a) the *direct* connection between terminals and symbols is displayed (**A**)
- b) structured according to which *cables* the terminals are connected to (**B**)

These two ways are illustrated below:



You find the buttons **A** and **B** in the right-hand side of the dialog box.

## SETTINGS IN CREATE GRAPHICAL TERMINAL PLAN

The dialog box **Create graphical terminal plan** contains the following tabs:

Tab	Function	Page
<b>Connections</b>	Here you specify which connections you wish to document, how they are going to be placed, as well as which terminal rows you wish to document.	(p. 67)
<b>Pages</b>	Here you choose a drawing header, type in a caption for the connections, and specify whether to have page breaks between the terminal rows.	(p. 69)
<b>Symbols</b>	Here you choose which terminal symbol to apply, which symbol texts to include into the documentation, as well as whether to allow to rotate the symbols in the documentation.	(p. 71)
<b>Wires</b>	Here you specify if you want cable information or wire numbers on the cable wires in the documentation.	(p. 76)
<b>Adjustments</b>	Here you specify the minimum distance between terminals and symbols, and of the connection heading.	(p. 77)

The functions on these tabs are described in the following.

### Create the graphical terminal plan

When you have specified the settings on the tabs, you click **OK** in the right-hand side of the dialog box. Then a new **Automation** project is created, containing the graphical terminal plan for the terminal rows you have chosen to document.

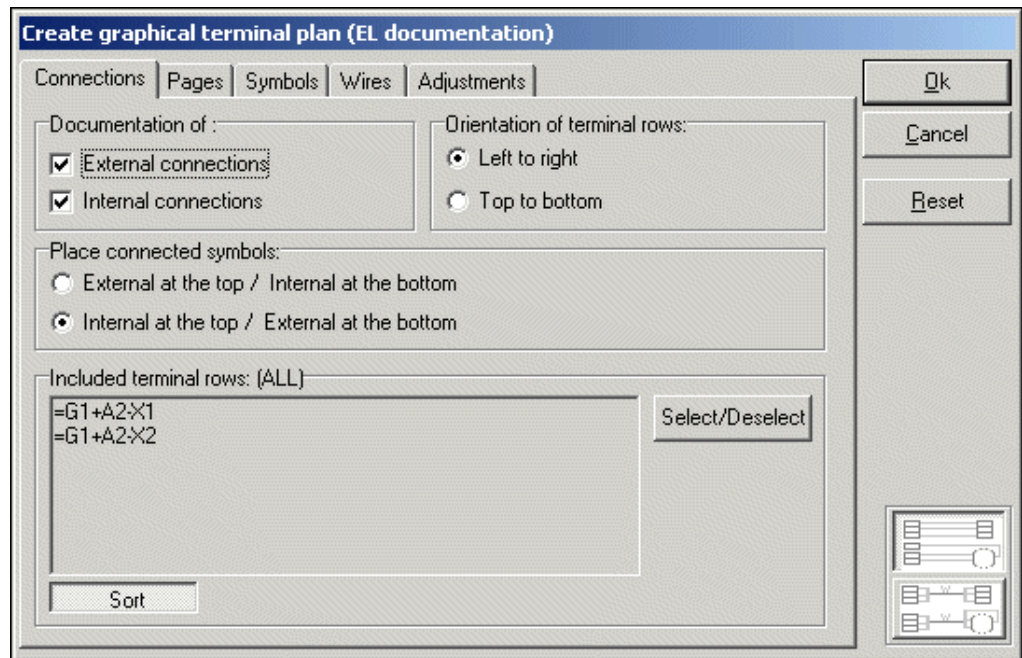
When you click on **Cancel**, you exit the dialog box, and no cable plan is created.

When you click on **Reset**, the settings in the dialog box are set to the program default settings.



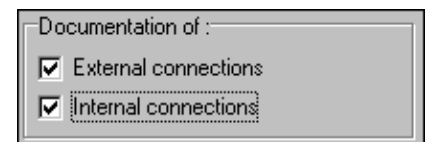
## CONNECTIONS

On this tab you for instance specify which connections you wish to document, how they are going to be placed, and which terminal rows you wish to include in the documentation.



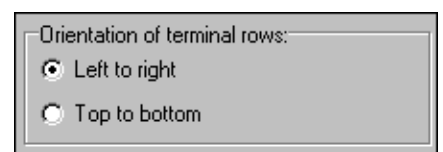
### External and/or internal connections

In the upper left-hand corner of the tab, you choose whether to document the internal and/or the external connections of the terminals in the project.



### Orientation

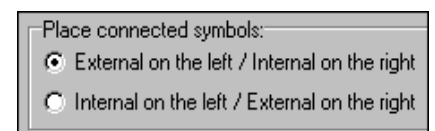
In the upper right-hand corner of the tab, you specify whether to arrange the terminal rows **Left to right** or **Top to bottom**.



To choose which drawing header — and with that which paper size — you wish to use in the documentation, please refer to “Drawing header” on page 69.

### Placing external/internal connections

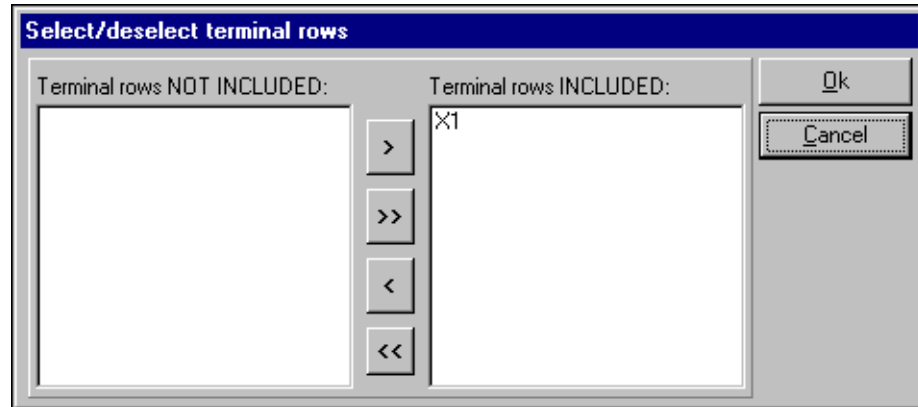
Under **Place connected symbols** you specify how you want the external and internal connections to be placed. The options depends on which connections you have chosen to document.



### Selecting terminal rows

If you do not specify otherwise, the program presumes that you wish to document all terminal rows in the project. However, if this is not the case, click **Select/**

**Deselect** in the lower right-hand corner of the dialog box. You then enter this dialog box:



If you wish to deselect a terminal row, first click on the terminal row in the column to the right, and then on the left-arrow in the dialog box.



If you wish to include the terminal row anyway, select it in the left-hand side of the dialog box, and click the right-arrow.



By clicking on the double left-arrow and the double right-arrow, you transfer all terminal rows back and forth between the columns.

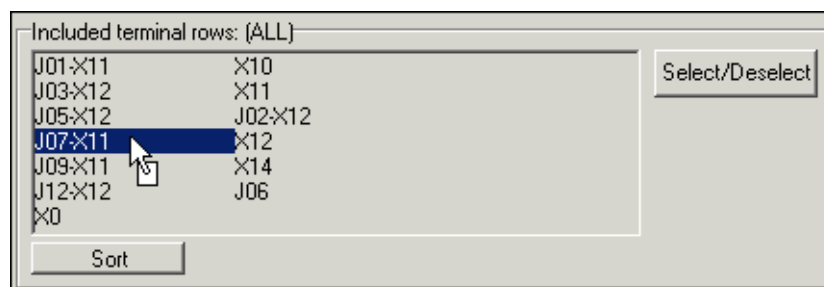
### Sorting the terminal rows

For a start, the terminal rows are sorted according to the common guidelines in **Automation** — see the manual for **Automation**.

When you want the terminal rows to appear in another order, you can click on them and *drag* them to the desired position.

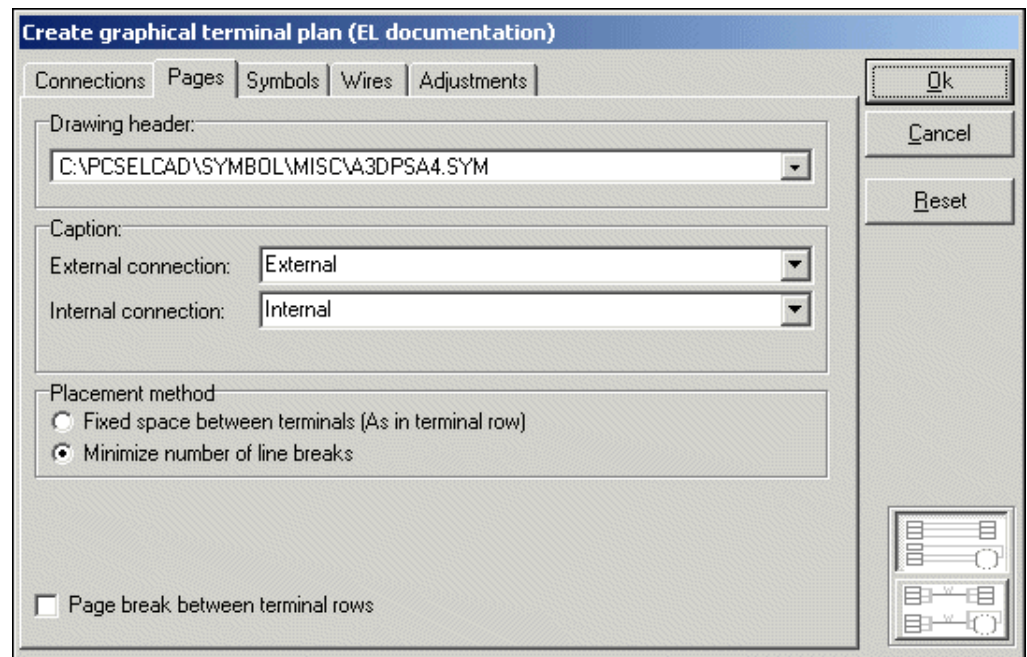
If you again want the terminal rows in sorted order in the generated terminal plan, click on **Sorted** at the bottom of the screen. When the terminal rows appears in sorted order, the button will be pressed in.

In the generated terminal plan, the terminal rows in the first column are documented first — starting with the uppermost terminal row in the column. Then the second column is documented, and so on.



## PAGES

On the **Pages** tab you for instance choose which terminal header you wish to use in the documentation project, type in a caption for the project connections, and specify whether to insert page breaks between the terminal rows.



## Drawing header

In the field **Drawing header** you specify which drawing header to use in the graphical terminal plan.

When you click on the down-arrow in the **Drawing header** field, you get the list of drawing headers in **Automation**, and click on the drawing header you wish to apply. You can only apply drawing headers which have been placed on this list.



### DRAWING HEADERS WHEN UPDATING TERMINAL PLANS

IF YOU DO NOT GET THE OPTION TO SELECT A DRAWING HEADER, THIS IS BECAUSE A DRAWING HEADER IS ALREADY PLACED ON THE CREATED TERMINAL PLAN PAGES IN THE PROJECT.

### *User defined drawing headers*

If you have designed your own drawing header for graphical terminal plans, you must insert it on the list of drawing headers in **Automation**.

You will not be able to choose the drawing header on the **Pages** tab until you have done so.

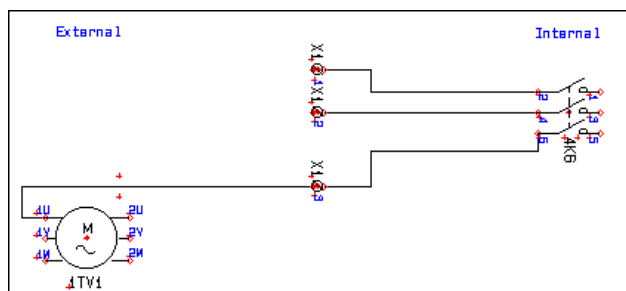
— Please refer to the **Automation** manual for further details about this, and for information about how to design your own drawing headers.

### Caption

In the fields **External connection** and **Internal connection** you specify the caption for the external- and internal connections respectively in the graphical terminal plans.

The program automatically remembers the captions you have applied on previous terminal plans.

Clicking on the down-arrows in the caption fields, gives you the option of choosing between these.



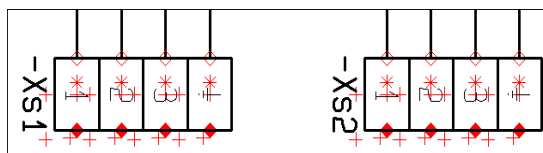
### Placing methods

When you specify according to which principle the terminals shall be placed in the terminal plan, you have two options:

#### *Fixed space between terminals*

If you enable **Fixed space between terminals**, the terminals are placed at fixed intervals.

— Please refer to “Minimum space between terminals” on page 77, to see how to specify the lower limits for the distance between the terminals.



#### *Minimize number of line breaks*

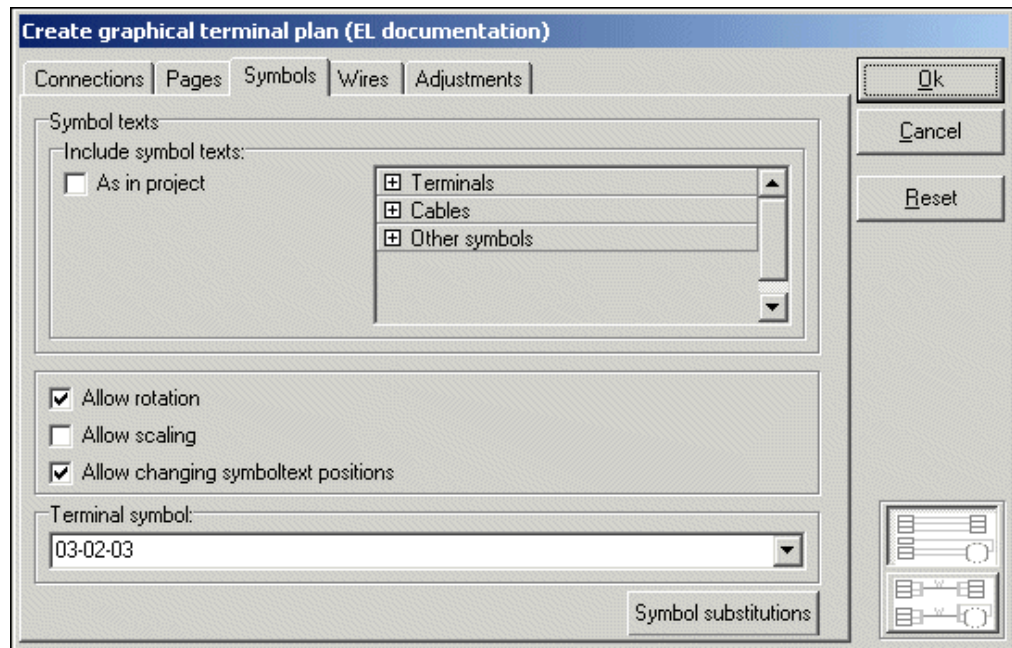
When the terminals are placed side by side, this results in more line breaks, than if it is allowed to move the terminals in order to get straight lines. This happens when you choose **Minimize number of line breaks**.

### Page break between terminal rows

When you want only one terminal row on each page, select **Page break between terminal rows** on the lower part of the tab.

## SYMBOLS

Here you for instance determine which terminal symbol to use in the graphical terminal plan, which symbol texts to include, and whether to allow the symbols to be rotated.

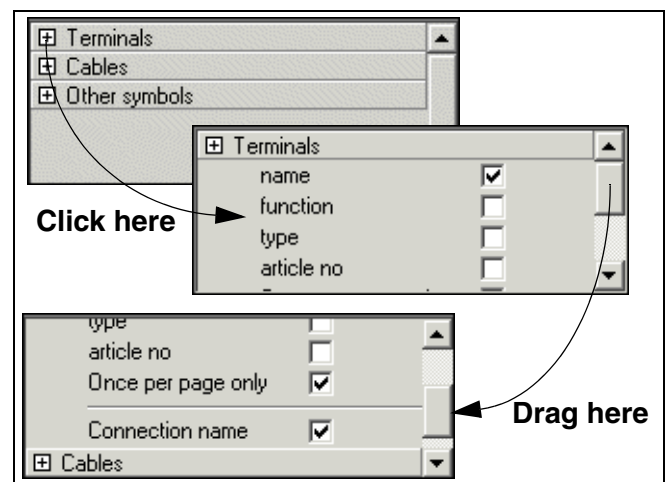


### Include symbol texts

On the upper part of the tab, you specify which symbol texts you wish to include on the graphical terminal plans.

When you select **As in project**, the texts, which are displayed in the project, are also displayed in the terminal plan.

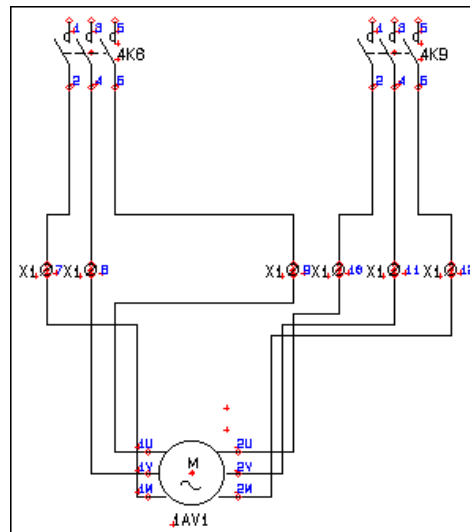
When you do *not* choose **As in project**, you can click on the + sign for **Terminals**, **Cables** and **Other symbols**. You can then select which of the texts to display in the terminal plan.



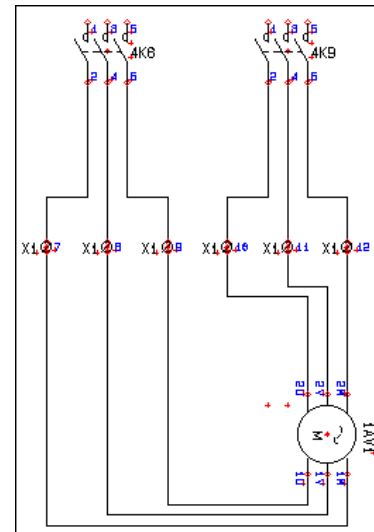
When the space between symbols and terminals is calculated in the terminal plan, the chosen texts are also taken into consideration. This can lead to room for less symbols on each page in the graphical terminal plan.

### Allow rotation

If you do not mind that the connected symbols are rotated on the terminal plan, you can enable **Allow rotation**. This can make it easier for the program to draw the lines on the terminal plan, which makes the terminal plan more clear.



Without rotation



With rotation

### Allow scaling

When you choose **Allow scaling**, you allow the program to downscale the symbol in the terminal plan, so that there can be more symbols on each page.

The symbols are only downscaled so that the connection points do not get placed too close to be readable.

### Allow changing symbol text positions

When you do *not* choose **Allow changing symbol text positions**, the symbol texts for each individual symbol are placed exactly as they are placed in the diagram pages.

Because this is not always the perfect solution in graphical terminal plans, it is recommended to let the program handle the placing of the symbol texts in terminal plans.

When this option is not selected, it will typically be necessary to use symbol substitution to avoid spending too much time editing the terminal plan after it is generated. — See “Symbol substitutions” on page 74.

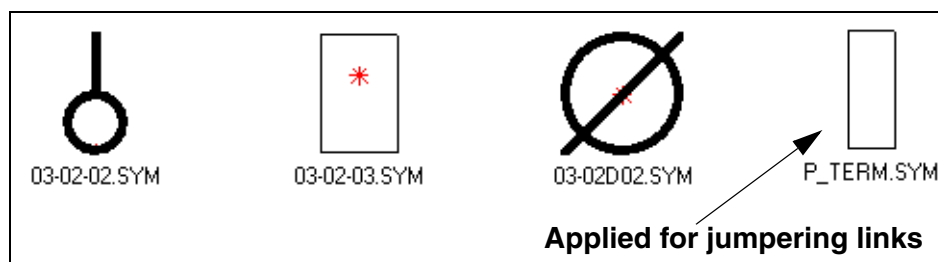
### Terminal symbol

At the bottom of the dialog box, you choose which terminal symbol to apply on the terminal plan. When you click on the down-arrow, you can choose between the previously used symbols.





The terminal symbols, you can choose between right away, look like this:



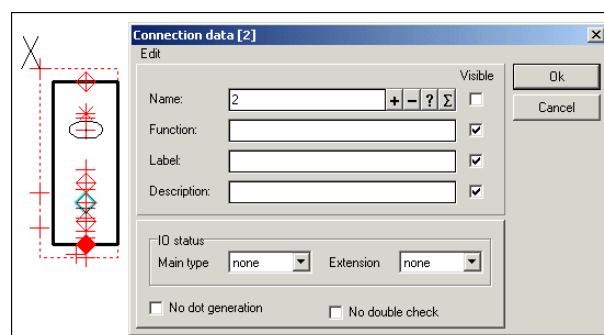
These symbols are only examples of what you might want to use for representation of the terminals in the terminal plan.

If you wish to use another terminal symbol, you can type in the name of the symbol in the field.

### *Terminal symbol for displaying jumpering links*

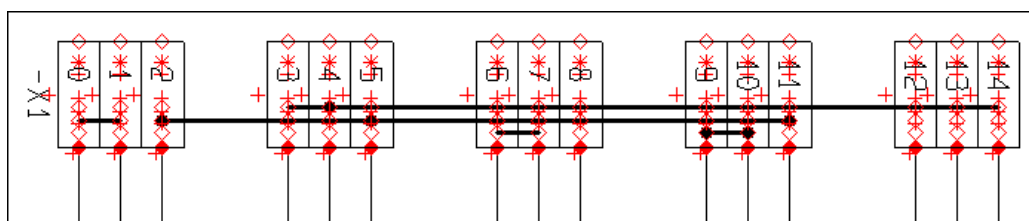
To display *jumpering links* in terminal plans, the symbol must have extra connection points. These must have **IO status Main type** set to **None** — as displayed in the figure to the right.

Read more about designing symbols in the **Automation** manual.



The symbol **P\_TERM.SYM** can be applied for displaying jumpering links in the terminal plan.

Jumpering links are displayed in the terminal plan as straight lines between the connection points in the terminal symbols.



### *Path leading to the terminal symbol*

The filename of the terminal symbol can be typed in with or without a *path* specifying in which folder the symbol is placed.

If you type in the entire path leading to the terminal symbol, the program will always be able to locate the symbol.

As you can see above, only the symbol filename is specified in the **Terminal symbol** field. This is possible because the symbols are located in a library, which has an *alias* assigned to it in the **Alias for libraries** dialog box in **Automation**.

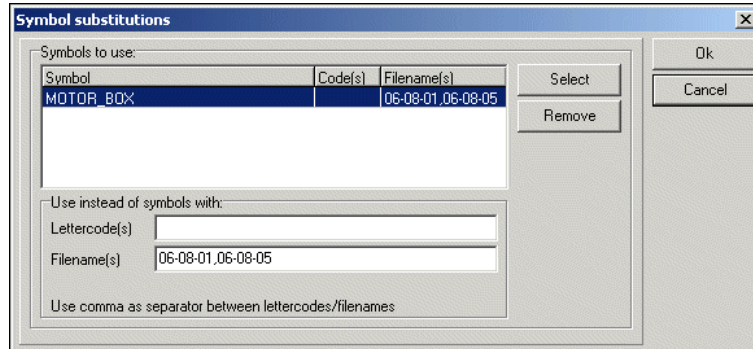
You enter this dialog box by first choosing **Settings => Libraries**, then click on **Symbols**, and on **Display library alias**.

— You can read more about *alias* in the **Automation** manual.

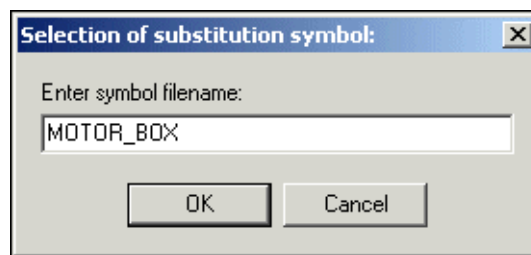
## Symbol substitutions

When you click on the button **Symbol substitution**, a dialog box is opened, which is used for identifying which project symbols, that shall be substituted by which symbols in the graphical terminal plan.

For instance you can choose to represent selected symbols — or all symbols — by simple boxes.



To specify that one symbol shall be used as a substitution for another, click on **Select**:



Type in the filename for the symbol you wish to use in the terminal plan, and click **OK**.

Then type in which symbols on the diagram pages, that shall be substituted by the selected symbol.

This can be done in two ways:

- Specifying *Lettercode(s)*: Here you can for instance specify that all symbols having the lettercode *-M* shall be replaced by the selected symbol
- Specifying *Filename(s)*: Here you can for instance specify that all symbols having the filename *06-08-01* shall be replaced by the selected symbol

When there are more project symbols that shall be substituted by the same symbol, this is specified by separating their filenames or lettercodes with a comma — as for instance *06-08-01,06-08-05* in the figure above.

To delete a line in the dialog box, click on the line, and then click **Delete**.

Click **OK** to leave the dialog box.



THE SUBSTITUTION SYMBOLS MUST HAVE AT LEAST THE SAME AMOUNT OF CONNECTION POINTS, AS THE SYMBOLS THEY REPLACE.

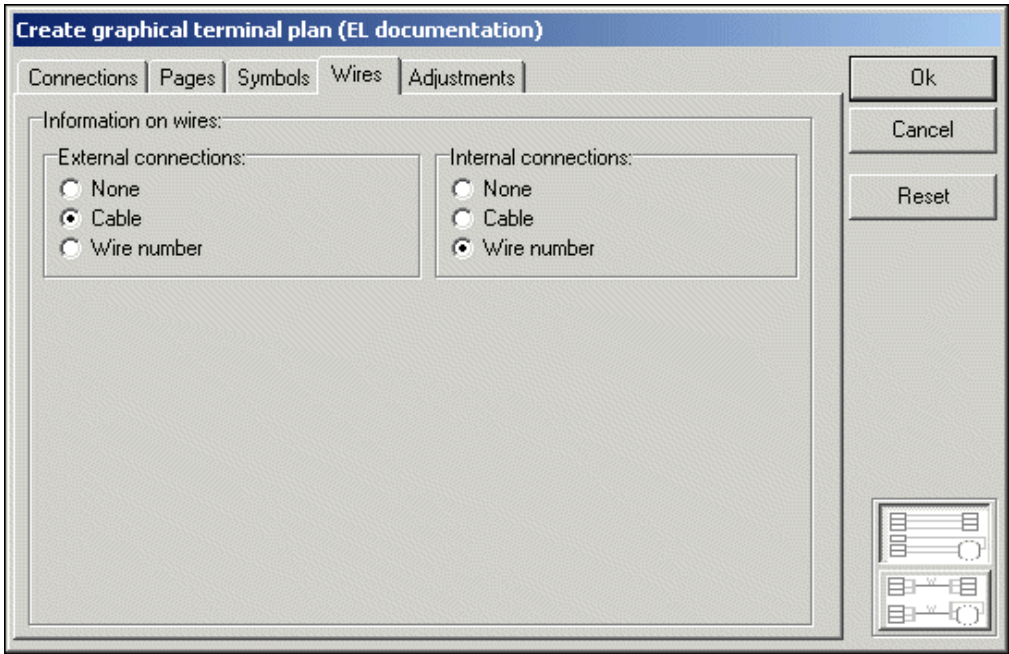
### Designing your own terminal symbols

You can also design a terminal symbol which matches the settings you have chosen for the terminal plan. How to do this is described in the **Automation** manual.

Please note that the standard terminal symbol *03-02-03* only has room for two digits for a connection name inside the box. This will make the program increase the distance between the terminals when the connection name is included/visible.

WIRES

On the **Wires** tab you for instance specify which information to attach to the cable wires on the internal and external side of the terminal.



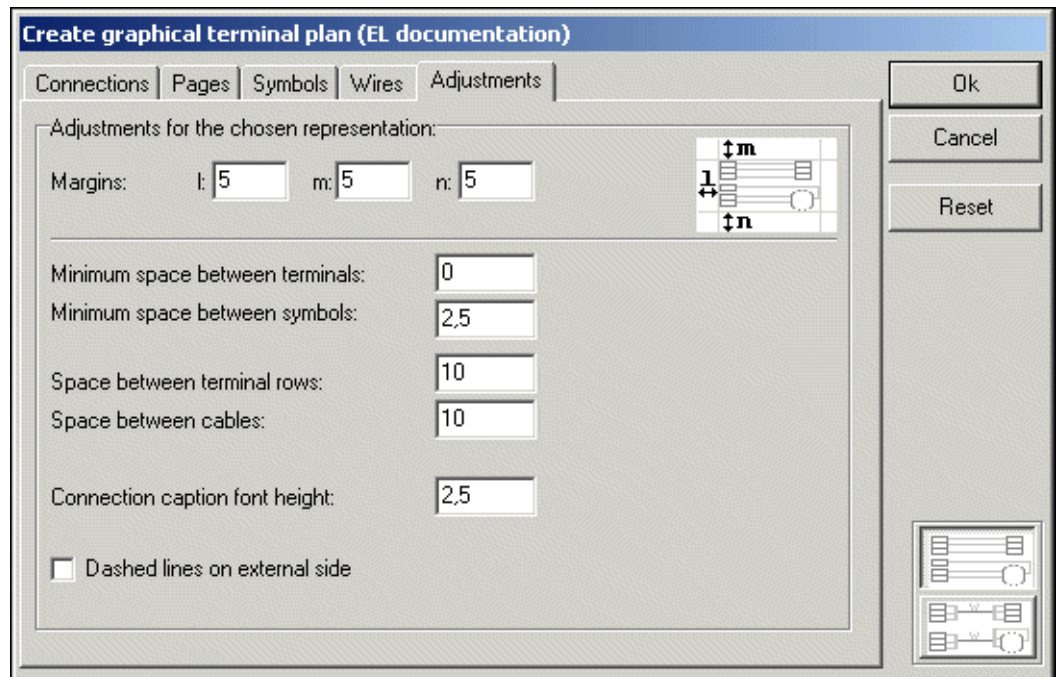
You can here choose between the following options:

Option	Function
<b>None</b>	Places no information
<b>Cable</b>	Places a cable symbol
<b>Wire number</b>	Places a wire number

This can be done at the external and the internal side of the terminal.

## ADJUSTMENTS

On the **Adjustments** tab you for instance specify the smallest allowed interval between terminals and symbols, and the size of the caption for the connections.



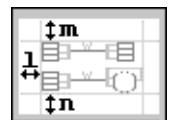
The contents of this tab depends on whether you have chosen to display the *direct* connection between terminals and symbols, or you have chosen to structure the terminal plan according to the used cables. — Please refer to “The structure of the terminal plan” on page 65 for further details.

The appearance described here is for the *direct* connection.

### Margins

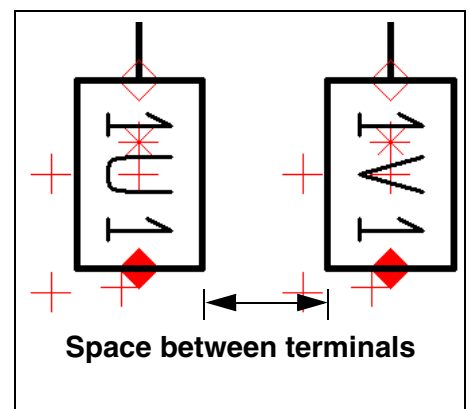
At the top of the tab you specify the left and right margin (**l**), the top margin (**m**) and the bottom margin (**n**) on the project pages.

Please note that these margins specifies the distance to the frame of the drawing header used on the pages.



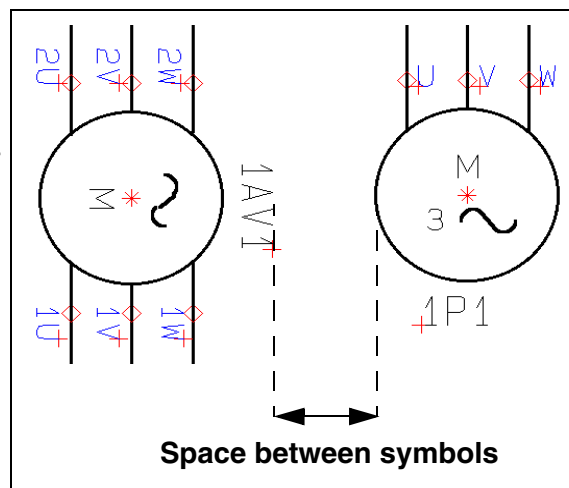
### Minimum space between terminals

In the field **Minimum space between terminals** you specify the smallest allowed distance between the terminals in the graphical terminal plan. The distance is measured from the border of the one terminal to the border of the other. However, the terminals cannot be located so close, that the texts overlap. When you set the space to 0, the terminals will be placed as close as possible.



### Minimum space between symbols

In the field **Minimum space between symbols** you specify the smallest allowed distance between the symbols in the terminal plan. Here the symbol texts are included in the calculations, and the distance is measured from the outline of the one symbol (including its texts) to the outline of the other.



### Font height of the caption

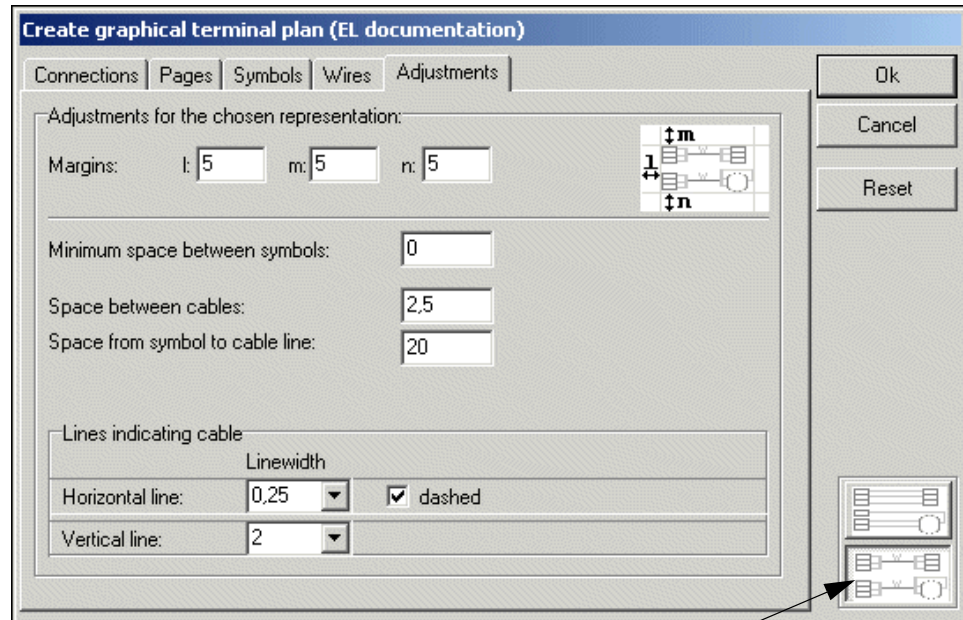
In the field **Connection caption font height** you specify the font size of the captions for the external and internal connections.

### Dashed lines on external side

If you select **Dashed lines on external side**, the lines on the external side of the terminal symbols will be drawn using a dashed linestyle.

## ADJUSTMENTS FOR TERMINAL PLANS STRUCTURED ACCORDING TO CABLES

When you have chosen to structure the terminal plan according to the used cables — please refer to “The structure of the terminal plan” on page 65 — the **Adjustments** tab looks like this:



Click here

### Lines indicating cables

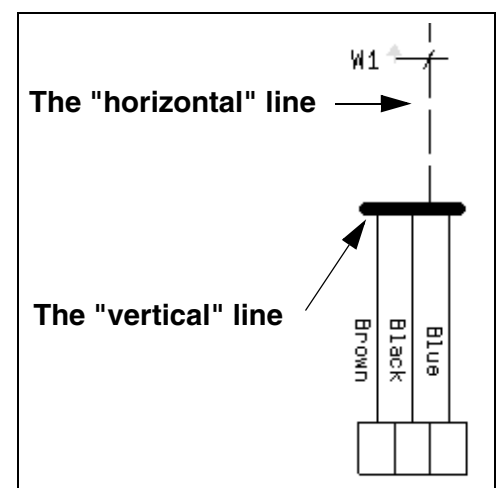
The major difference here is that you now have to specify the width of two types of lines:

- the line representing the cable (the "horizontal" line), and
- the appearance of the line used for indicating which cable wires are included in the cable (the "vertical" line)

The terms "horizontal" and "vertical" refers to the figure at the upper-right corner of the tab.

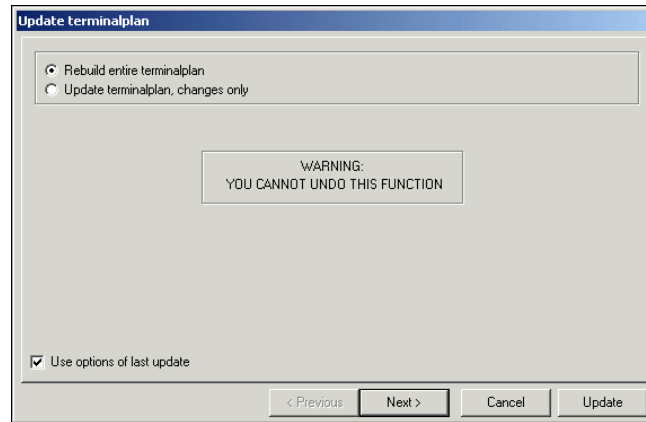
### Creating the graphical terminal plan

When you have specified the settings on the tabs, click **OK** in the right-hand side of the dialog box to create the graphical terminal plan, as described in “Create the graphical terminal plan” on page 66.



## UPDATING EXISTING TERMINAL PLANS

When you have already created a graphical terminal plan for the active project, and choose **Tools => Graphical Terminal Plan**, you enter the dialog box **Update graphical terminal plan**:



Here you have two options:

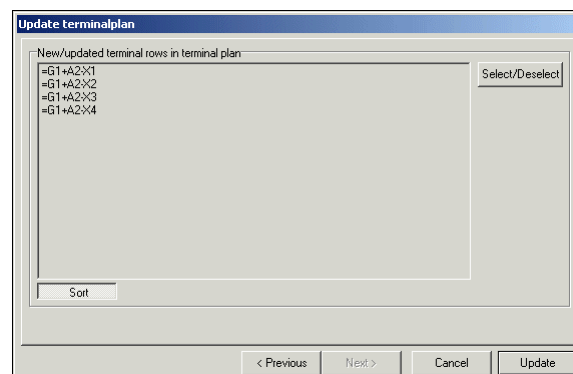
- a) **Rebuild entire terminal plan** which is described in “Rebuild entire terminal plan” on page 80: Choose this option when you have *not* made changes in the existing terminal plan, that you wish to keep
- b) **Update terminal plan, changes only** which is described in “Update terminal plan, changes only” on page 81: Choose this option when you *have* made changes in the existing terminal plan, that you do wish to keep

### Rebuild entire terminal plan

When you choose **Rebuild entire terminal plan**, the terminal plan is created from scratch.

*Rebuilding the terminal plan with Use settings from last update*

When you select **Use settings from last update** in the lower-left corner of the dialog box (hereby meaning the applied settings for connections, symbols, wires and adjustments), you enter this dialog box:



When you need to change which terminal rows to include in the terminal plan, click on **Select/Deselect**. Then select the terminal rows as described in “Selecting terminal rows” on page 67.



When you need to change the order of appearance of the documented terminal rows, this is done as described in “Sorting the terminal rows” on page 68.

When you then click on **Update**, the terminal plan is updated without entering the settings dialog box.

*Rebuilding the terminal plan Without Use settings from last update*

When you do *not* select **Use settings from last update** in the lower-left corner of the dialog box, you jump right to the update dialog box — as described in the start of this chapter — when you click on **Update**.

**Update terminal plan, changes only**

When you wish to keep the existing terminal plan — or parts of the existing terminal plan — choose **Update terminal plan, changes only** and click on **Next**:

☐ Rebuild entire terminalplan  
☒ Update terminalplan, changes only

*Specifying which terminal plans to update, add or delete*

You now enter this dialog box, in which you specify exactly which parts of the terminal plan to update, and which to keep exactly as they are now — including your possible changes:

Update terminalplan

Selection of terminal rows to update/add/remove:

Terminal rows in the termi...	Page(s)	Status	Remarks
<input type="checkbox"/> =G1+A2%1	5	Leave as is	
<input type="checkbox"/> =G1+A2%2	5 ^	Leave as is	
<input checked="" type="checkbox"/> =G1+A2%3	New	Add	
<input checked="" type="checkbox"/> =G1+A2%4	New	Add	

Page 5 will be left as is  
New pages will be added

< Previous   Next >   Cancel   Update



**DELETING A TERMINAL ROW FROM THE EXISTING TERMINAL PLAN**

WHEN YOU WISH TO DELETE A TERMINAL ROW, WHICH EXISTS IN THE CURRENT TERMINAL PLAN, SELECT THE TERMINAL ROW IN THIS DIALOG BOX. IN THE FOLLOWING DIALOG BOX YOU CAN THEN HAVE IT DELETED.

As a starting point, you get a status for the individual terminal rows, which you can change according to your needs:

**Click here to update the terminal row**

**Click here not to create terminal plan for this row**

Terminal rows in the termi...	Page(s)	Status
<input type="checkbox"/> =G1+A2×1	5	Leave as is
<input type="checkbox"/> =G1+A2×2	5 ^	Leave as is
<input checked="" type="checkbox"/> =G1+A2×3	New	Add
<input checked="" type="checkbox"/> =G1+A2×4	New	Add

**Terminal rows with created terminal plan**

**New terminal rows**

The selected terminal rows will be updated.

The program always updates *full* pages in the terminal plan.



#### SPECIFYING CHANGES IN THE EXISTING TERMINAL PLAN

THE PROGRAM AUTOMATICALLY DETECTS WHETHER THERE HAVE BEEN ADDED TERMINAL ROWS TO THE DIAGRAMS SINCE THE LAST UPDATE.

WHEN OTHER CHANGES HAVE BEEN MADE IN THE DIAGRAMS, *YOU* MUST SPECIFY THIS BY SELECTING THE CHANGED TERMINAL ROWS.

#### *Updating already documented terminal rows*

You can select a terminal row, which have already been documented in the present terminal plan. Hereby all of the page, on which the terminal plan is placed, are updated in the terminal plan.

If there is another terminal plan on this page in the existing terminal plan, this other terminal row will therefore also be updated. This is displayed by an automatically set check mark for this terminal row in the survey, when you select the first terminal row.

#### *Deselecting new terminal rows*

You can also remove the check mark from one of the new terminal rows, and the terminal row will not be documented in the terminal plan.

#### *Updates all succeeding pages in the terminal plan*

When the program updates one page in the terminal plan, all succeeding pages in the terminal plan are updated also.

*Status at the bottom of the dialog box*

At the bottom of the dialog box you see a total status of which pages remains unchanged, which existing pages are updated, and whether new pages are added to the terminal plan.

Page 5 will be left as is  
New pages will be added

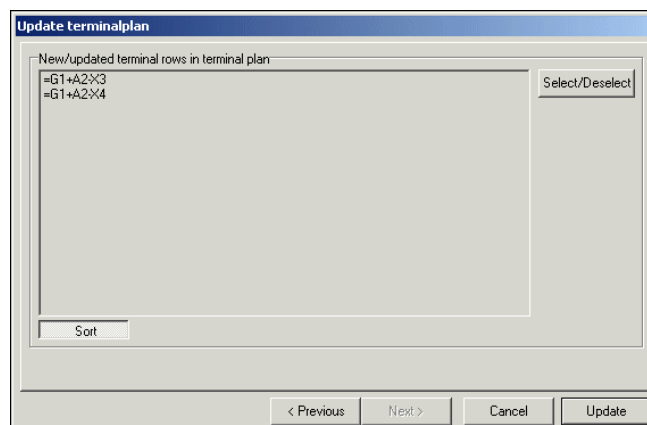
**WHEN YOU ONLY HAVE ADDED TERMINAL ROWS**

WHEN YOU KNOW THAT YOU *ONLY* HAVE ADDED NEW TERMINAL ROWS, DO NOT MAKE CHANGES IN THE DIALOG BOX. IN THIS CASE, THE PROGRAM WILL AUTOMATICALLY KEEP THE ORIGINAL TERMINAL PLAN PAGES AS THEY ARE, AND ADD THE NECESSARY PAGES TO THE TERMINAL PLAN.

Click on **Next**.

*Specifying order of appearance for new and updated terminal rows*

You now enter this dialog box, where you can specify the order of appearance for the new and for the updated terminal rows in the terminal plan:



When you wish to delete terminal rows from the terminal plan, click on **Select/Deselect**. Read more about this in “Selecting terminal rows” on page 67.

When you wish to change the order of appearance of the documented terminal rows, this is done as described in “Sorting the terminal rows” on page 68.

Click on **Update** and the terminal plan is updated.





## Graphical Cable plans

In PCISCHMATIC Automation you can automatically create graphical cable plans for the active project.



# GRAPHICAL CABLE PLANS

In **Automation** you can also create graphical cable plans for your projects.

THIS CHAPTER DESCRIBES	
— ON GRAPHICAL CABLE PLANS .....	(P. 87)
— CREATING GRAPHICAL CABLE PLANS .....	(P. 87)
— SETTINGS IN CREATE GRAPHICAL CABLE PLAN .....	(P. 88)
— THE TAB CONNECTIONS .....	(P. 89)
— THE TAB PAGES .....	(P. 90)
— THE TAB SYMBOLS .....	(P. 91)
— THE TAB ADJUSTMENTS .....	(P. 93)
— UPDATING EXISTING GRAPHICAL CABLE PLANS .....	(P. 94)

## ON GRAPHICAL CABLE PLANS

The graphical cable plan for a project can be created as an independent **Automation** project, or can be inserted directly in the active project.

When the cable plan is created, you can edit the created cable plan by moving lines and symbols, as on normal diagram pages. If you, at a later stage, need to let the program update the cable plan, you can transfer the changes you have made in the cable plan to the new cable plan.

— If a cable plan does not turn out as you want it to, you can always change the cable plan settings, and create a new cable plan.

You can print graphical cable plans as you normally print projects in **Automation**.

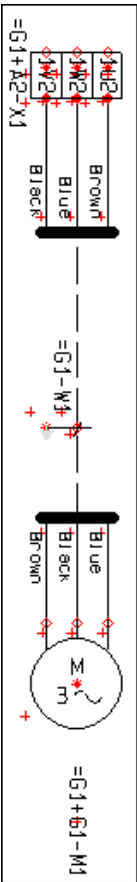
## CREATING GRAPHICAL CABLE PLANS

To create a graphical cable plan for a project, open the project, and choose **Tools => Graphical Cable Plan** in **Automation**. You then enter the dialog box **Create graphical cable plan**.

If you have already created a graphical cable plan and inserted it in the active project, you enter the dialog box **Update graphical terminal plan**, as described in “Updating existing graphical cable plans” on page 94.

When the cable plan is created, you will be asked whether you wish to insert it in the project — if you have not already inserted a cable plan.

This is done just like for graphical terminal plans. Read more about the consequences of answering **Yes** or **No** in “Answering Yes - to inserting the terminal plan in the project” on page 64 and “Answering No - to inserting the terminal plan in the project” on page 64 respectively.



The pages in the cable plan gets the page type **CAP**. In **Automation** projects their contents are not added to the electrical and mechanical lists.

### SETTINGS IN CREATE GRAPHICAL CABLE PLAN

The dialog box **Create graphical cable plan** contains the following tabs:

Tab	Function	Page
<b>Connections</b>	Here you specify which connections you wish to document, how they are going to be placed, as well as which cables you wish to document.	(p. 89)
<b>Pages</b>	Here you choose a drawing header, and specify whether to have page breaks between the cables.	(p. 90)
<b>Symbols</b>	Here you choose which terminal symbol to apply, which symbol texts to include into the documentation, as well as whether to allow to rotate the symbols in the documentation.	(p. 91)
<b>Adjustments</b>	Here you specify the cosmetic setup of the cable plan.	(p. 93)

The functions on these tabs are described in the following.

#### Create the graphical cable plan

When you have specified the settings on the tabs, click **OK** in the right-hand side of the dialog box. Then a new graphical cable plan is created for the cables you have chosen to document.

When you click on **Cancel**, you exit the dialog box, and no cable plan is created.

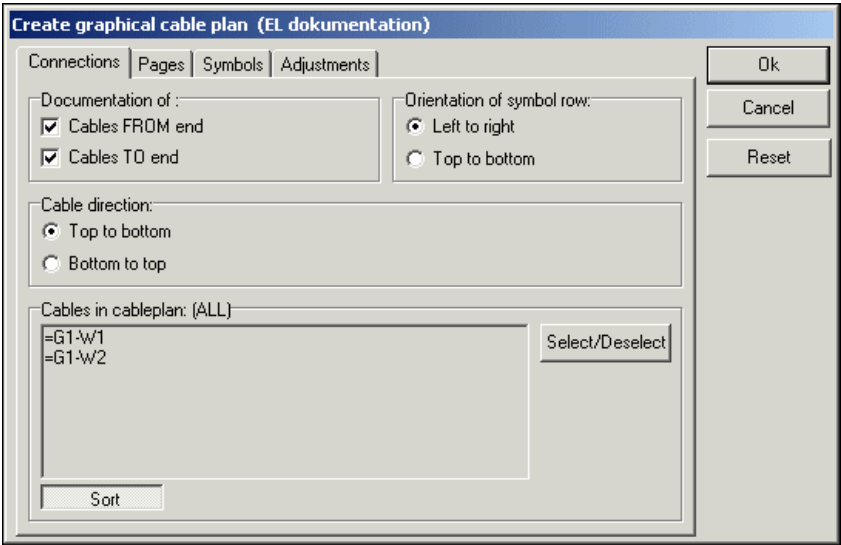
When you click on **Reset**, the settings in the dialog box are set to the program default settings.





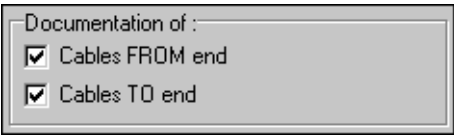
CONNECTIONS

On this tab you for instance specify which connections you wish to document, how they are going to be placed, and which cables you wish to include in the documentation.



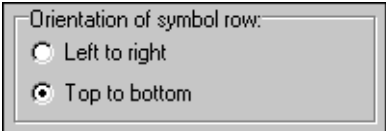
External and/or internal connections

In the upper left-hand corner of the tab, you choose whether to document the **Cables FROM end** and/or the **Cables TO end**.



Orientation

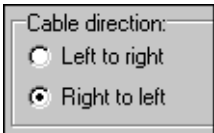
In the upper right-hand corner of the tab, you specify whether to arrange the symbol row **Left to right** or **Top to bottom**.



To choose which drawing header — and with that a paper size — you wish to apply in the documentation, please refer to “Drawing header” on page 90.

Cable direction

Under **Cable direction** you specify the cable direction in the cable plan. The options depends on the chosen orientation of the symbol row.



Selecting cables

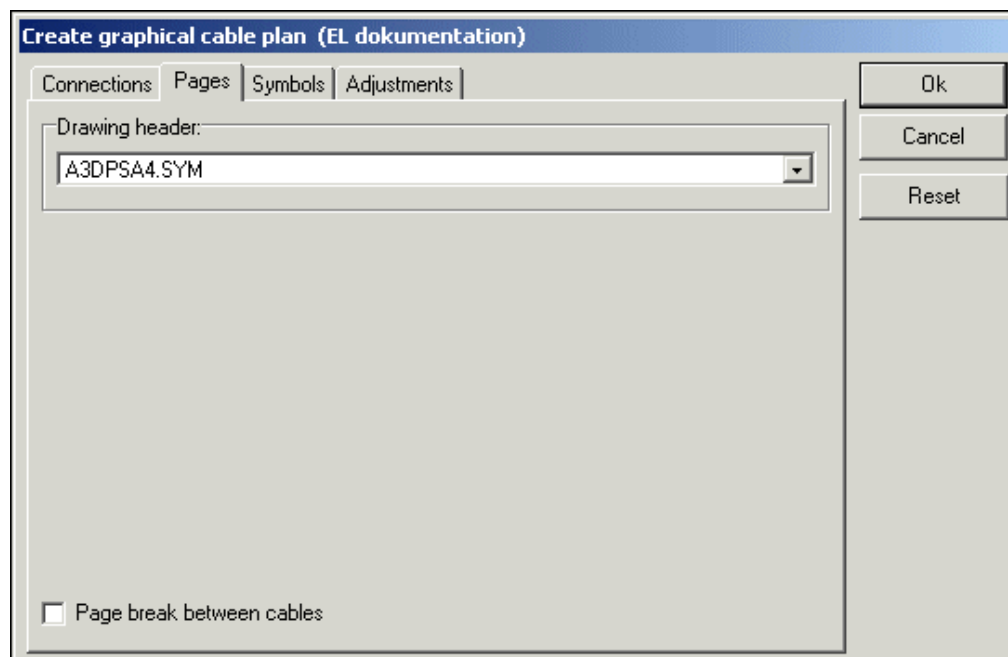
Choose which cables to document as described in “Selecting terminal rows” on page 67.

Sorting cables

Specify the sorting order of the cables as described in “Sorting the terminal rows” on page 68.

## PAGES

On the **Pages** tab you choose which drawing header you wish to use in the cable plan, and specify whether to insert a page break between the cables.



### Drawing header

In the field **Drawing header** you determine which drawing header to use in the graphical cable plan.

When you click on the down-arrow in the **Drawing header** field, you get the list of drawing headers in **Automation**. Click on the drawing header you wish to use. You can only use drawing headers which have been placed on this list in **Automation**.



#### DRAWING HEADERS WHEN UPDATING CABLE PLANS

IF YOU DO NOT GET THE OPTION TO SELECT A DRAWING HEADER, THIS IS BECAUSE A DRAWING HEADER IS ALREADY PLACED ON THE CREATED CABLE PLAN PAGES IN THE PROJECT.

#### *User defined drawing headers*

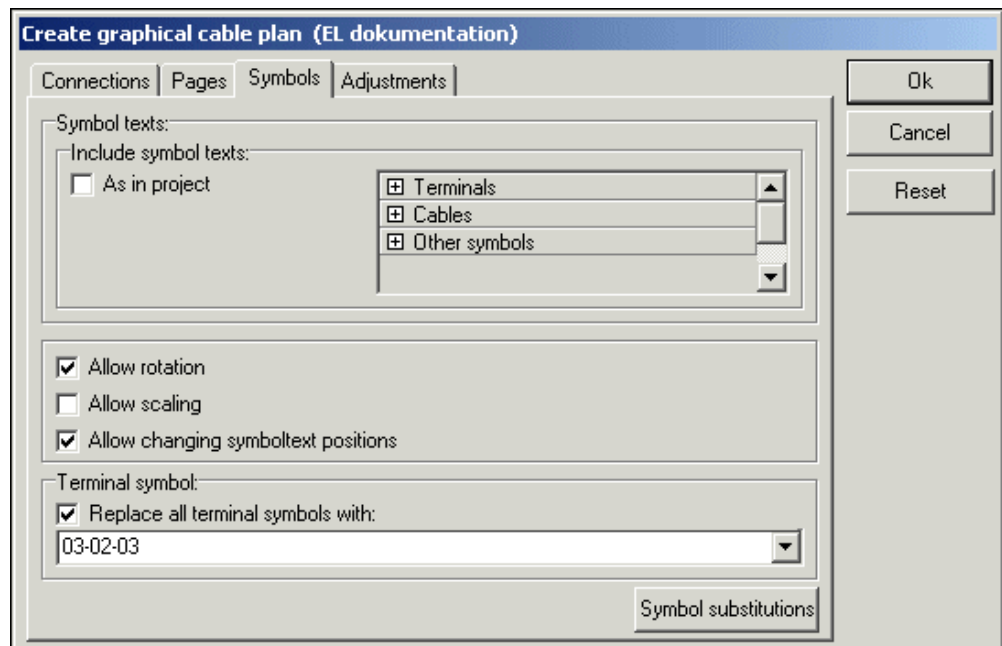
Please refer to "User defined drawing headers" on page 69 for further details about designing your own drawing headers for cable plans.

### Page break between cables

When you only want one cable on each page, select **Page break between cables** on the lower part of the tab.

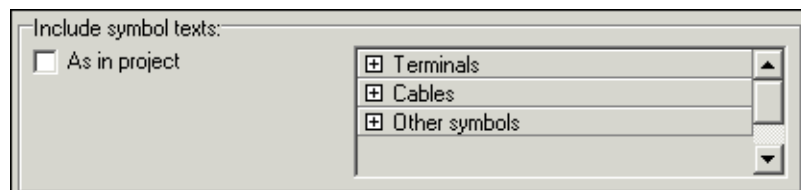
## SYMBOLS

Here you for instance determine which terminal symbol to apply in the graphical cable plan, which symbol texts to include, and whether to allow the symbols to be rotated.



### Include symbol texts

On the upper part of the tab, you specify which symbol texts you wish to include on the graphical cable plans.



Read more about this in "Include symbol texts" on page 71.

### Allow rotation

If you do not mind that the connected symbols are rotated on the cable plan, you can enable **Allow rotation**. Please refer to "Allow rotation" on page 72 for further details.

### Allow scaling

Read about this in "Allow scaling" on page 72.

### Allow changing symboltext positions

Read about this in "Allow changing symboltext positions" on page 72.

**Terminal symbol**

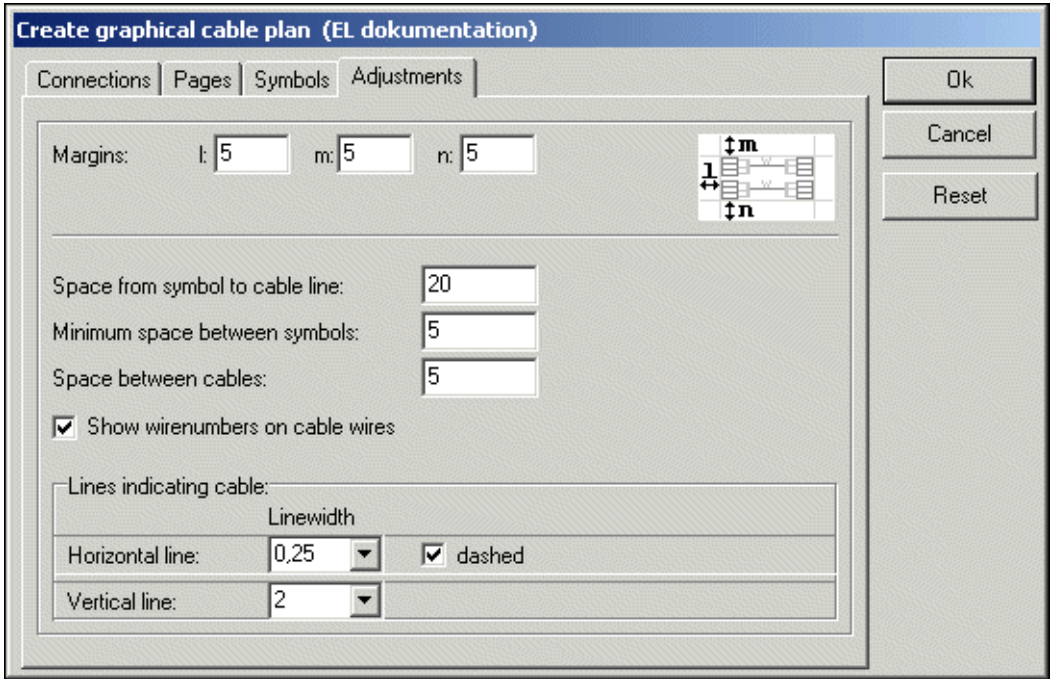
Please refer to “Terminal symbol” on page 72 for further details about choosing terminal symbols for the cable plan.

**Symbol substitutions**

Read about this in “Symbol substitutions” on page 74.

ADJUSTMENTS

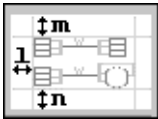
On the **Adjustments** tab you specify the cosmetic aspects of the cable plan.



Margins

At the top of the tab you specify the left margin (**l**), the top margin (**m**) and the bottom margin (**n**) on the project pages.

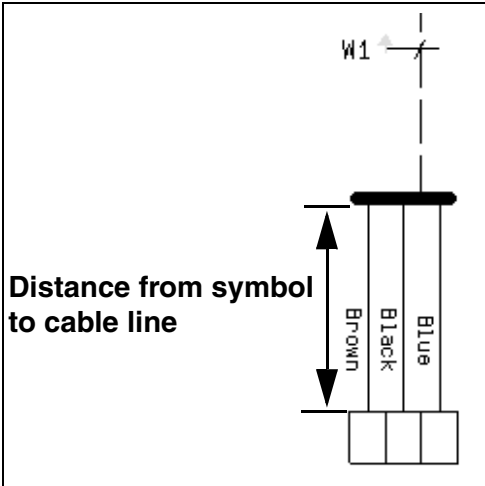
Please note that these margins are the margins between the symbols on the cable plan and the frame of the used drawing header.



Distance from cable symbol to cable line

The **Space from symbol to cable line** specifies the distance from the "vertical line" to the connected symbol — see the figure to the right.

Please refer to “Lines indicating cables” on page 79 for further details about the "vertical" line.



Minimum space between symbols

In the field **Minimum space between symbols** you specify the smallest allowed distance between the symbols in the cable plan. Please refer to “Minimum space between symbols” on page 78 for further details.

**Space between cables**

The ***Space between cables*** field sets the distance between the groups of symbols (including their texts) connected to the various cables.

**Show wire numbers on cable wires**

Select ***Show wire numbers on cable wires*** to have wire numbers placed on cable wires.

**Lines indicating cables**

Please refer to “Lines indicating cables” on page 79 for further details.

**UPDATING EXISTING GRAPHICAL CABLE PLANS**

When you have already created a graphical cable plan for the active project, and choose **Tools => Graphical cable plan**, you enter the dialog box **Update graphical cable plan**.

This takes place exactly as described for graphical terminal plans. Read more about this in “Updating existing terminal plans” on page 80.



## PCISCHEMATIC Text translator

The tool PCISCHEMATIC Text translator is used for translating project texts to other languages. This — and the other options in the tool — is described in the following.





# PC|SCHEMATIC TEXT TRANSLATOR

**PC|SCHEMATIC Text Translator** is created for translating texts from one language to another — for instance from English to German.

The program can also be used for checking e.g. an English project for spelling errors, and for checking whether project texts follow the company standard guidelines for texts in projects.

THIS CHAPTER DESCRIBES	
— TRANSLATING A PROJECT.....	(P. 97)
— THE TRANSLATIONS DIALOG BOX .....	(P. 100)
— EDITING DICTIONARIES .....	(P. 105)
— TRANSLATION OPTIONS .....	(P. 107)
— EDITING TOOL FOR TRANSLATIONS .....	(P. 111)
— RECOGNITION AND TRANSLATION OF TEXTS.....	(P. 112)
— TEXT POSITIONS IN PROJECTS FOR TRANSLATION.....	(P. 114)
— CONVERTING DICTIONARIES FROM THE PREVIOUS VERSION .....	(P. 115)

## TRANSLATING PROJECTS



To translate a project in **Automation**, you must first open the project. Then start **PC|SCHEMATIC Text Translator** by choosing **Tools => Text translator**, or by clicking the **Text translator** button in the program toolbar at the top of the screen.

You then enter the dialog box **Text translator**:





### FOR USERS OF THE PREVIOUS VERSION

IF A DICTIONARY CREATED IN THE PREVIOUS VERSION OF **PC|SCHEMATIC Text Translator** EXISTS, YOU WILL BE ASKED WHETHER TO CONVERT THIS TO THE NEW FORMAT. READ MORE ABOUT THIS IN “CONVERTING DICTIONARIES FROM THE PREVIOUS VERSION” ON PAGE 115.

## Choose which languages to translate to and from

Here you choose which language to translate *from*, and which language to translate *to*. If you select **Show languages with existing dictionaries only**, the list will only contain languages for which a dictionary exists.

### *Languages without dictionaries*

When you choose two languages, for which no dictionary exists, you will get the option of creating a dictionary based on the translations you have created. — See “The Translations dialog box” on page 100, in particular “Creating the translated project” on page 104.

## Changing options

If for instance you need to specify which project text types to translate, to change the settings for the available dictionaries, or to attach new dictionaries, click on **Options**. — Read more about this in “Options for the translation” on page 107.

## Recognition and translation of texts

You can read more about how **PC|SCHEMATIC Text Translator** recognize texts in “Recognition and Translation of texts” on page 112.



### PLACING TEXTS IN PROJECTS FOR TRANSLATION

IT IS A GOOD IDEA TO BE CAREFUL ABOUT HOW TO PLACE PROJECT TEXTS, WHEN YOU NEED TO TRANSLATE THE PROJECT TEXTS TO ANOTHER LANGUAGE LATER.

— READ MORE ABOUT THIS IN “HOW TO PLACE TEXTS IN A PROJECT MEANT FOR TRANSLATION” ON PAGE 114.

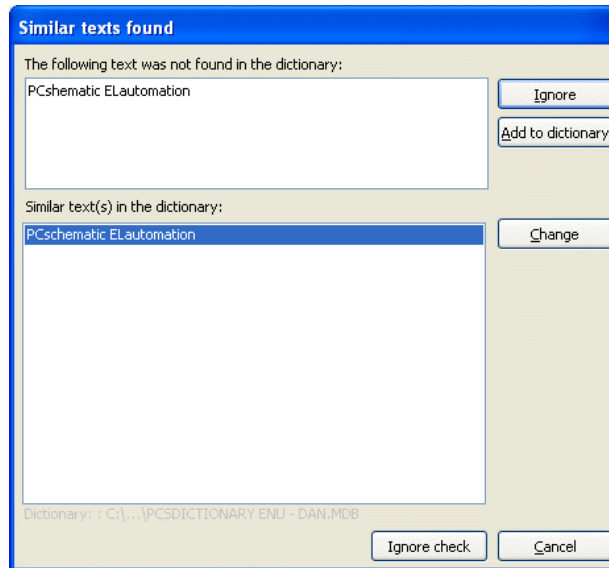
## Start translating the project

Now click on **Translate**, and the program starts translating the project.

### *Examination for Similar texts*

What happens next, is that the program runs through all the project texts, in order to translate them. If there are texts in the project, which has no translation in the

dictionary, and there exists a similar text in the dictionary, this similar text will be suggested to replace the text:



Here you can either choose to add the text to the dictionary (click **Add to dictionary**), choose to continue without taking any action (click **Ignore**), or choose to click on **Replace**.

When you click on **Replace**, the text **PCshematic ELautomation** in the figure above is replaced with the text **PCschematic ELautomation** — in the original (here: English) project. This way, you can make sure that all texts are written consequently throughout the project.

When you click on **Ignore check**, this check for similar texts is skipped, and **PC|SCHEMATIC Text Translator** continues the translation. When you click on **Cancel**, the translation is cancelled.

A status for the translation is now displayed:

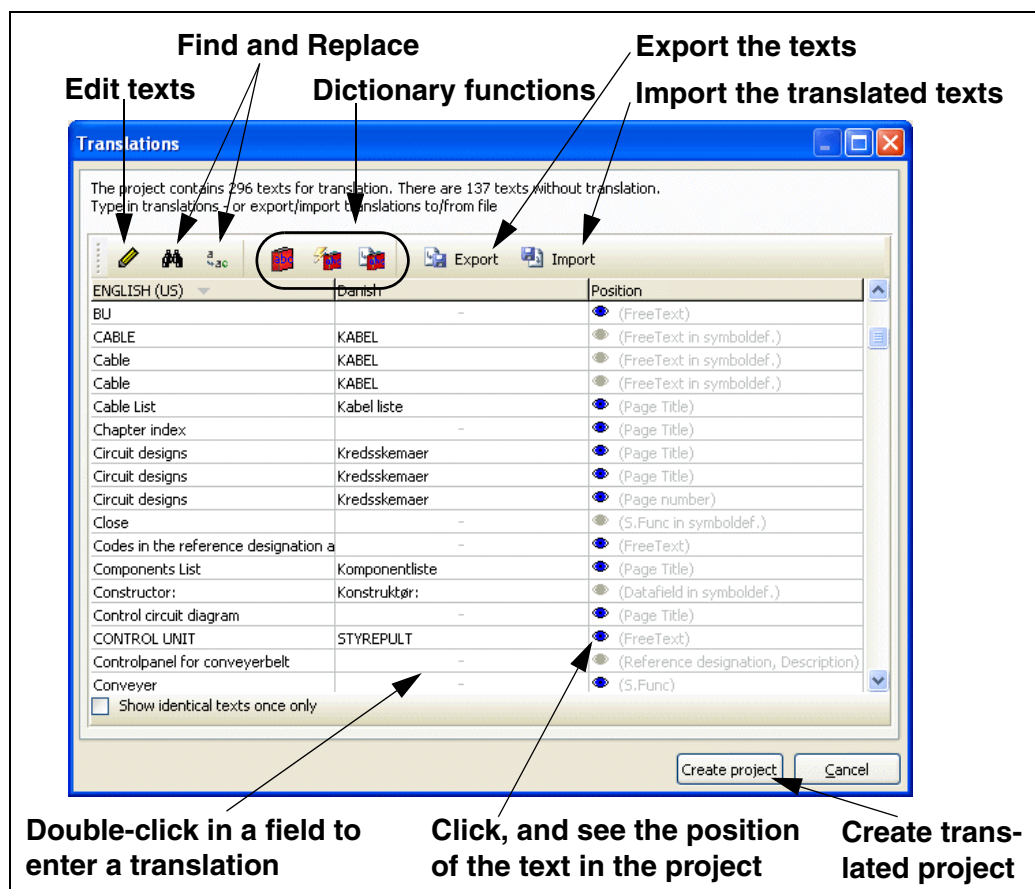


Click **OK**. You now enter the dialog box **Translations**.

— Read more about the options you have here in “The Translations dialog box” on page 100.

## THE TRANSLATIONS DIALOG BOX

In the dialog box **Translations**, you can type in the missing translations, export the missing translations and import translations:



You have the following options:

### Edit a text

You can edit a text in the **Translations** dialog box in two ways:

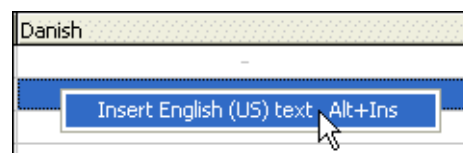


a) Activate the **Edit text** button, click on the text, and edit the text. Or:

b) Double-click on the text, and edit it.

### Insert English text

If for instance you translate a project from English to Danish, you can transfer the English text unchanged to the column with the Danish translation. To do this, right-click in the column with the translation, and then choose **Insert English text**.



Please note, that you can also select multiple fields, and then choose **Insert English text**.

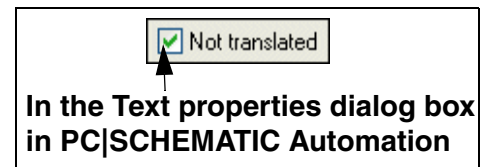
### Set to 'Not translated' status

If you wish to assign the **Not translated** status to a text, right-click in the field with the text, and choose **Set to 'Not translated' status**.



This means that the text — **L3** in the figure above — is transferred unchanged to the translated project, and that the text will not appear in the list of texts in the **Translations** dialog box.

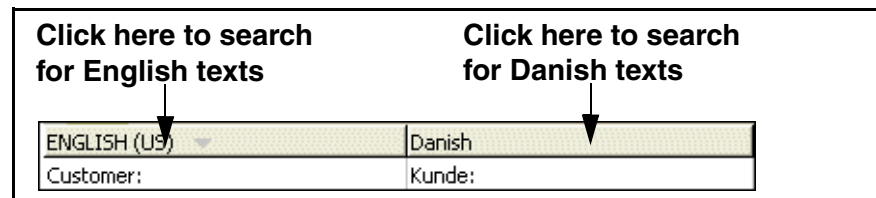
This status for the text is transferred to the original **Automation** project. In **Automation**, you can see this in the dialog box **Text properties** for the text, as displayed in the figure to the right.



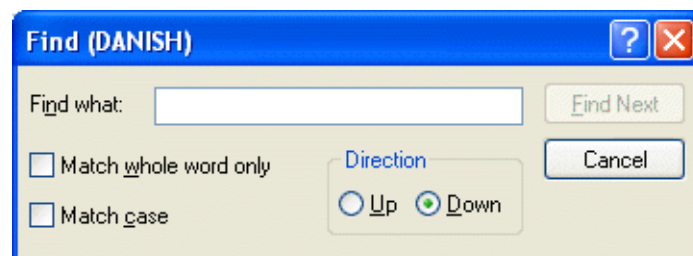
### Find and Replace

To search for a text, or a part of a text string, do this:

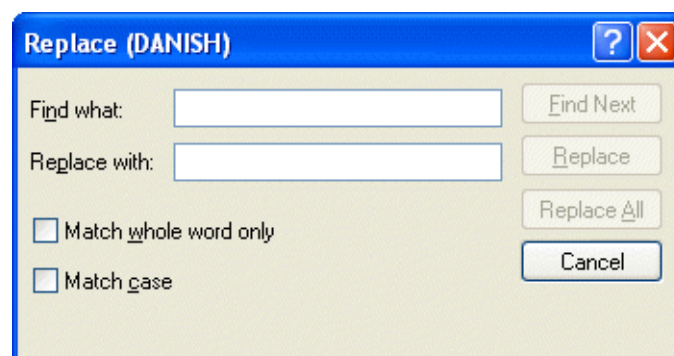
- 1) Click in the column in the dialog box, which you wish to search for texts in:



- 2) Click the **Search** button, and you enter the **Find** dialog box, where you specify what to search for, and click **Find next**:



If you instead click the **Replace** button, you enter the **Replace** dialog box, where you have the following options:



### Open the applied dictionary



To open the applied dictionary, click the **Dictionary** button.

This gives you the option of looking in the dictionary, while looking at the texts in the dialog box **Translations**. — Read more about this in “Editing Dictionaries” on page 105.



### Translation using the dictionary

When you wish to translate the project texts using the dictionary, click the **Translate using dictionary** button.

When you click this button, the texts are translated, just like what happened before you entered the **Translations** dialog box. You will therefore only need to click this button, if you previously chose to cancel the translation, or if you, or others, have updated the dictionary in the mean time. — Or if you have edited the texts in the original language.

### Adding texts to the dictionary



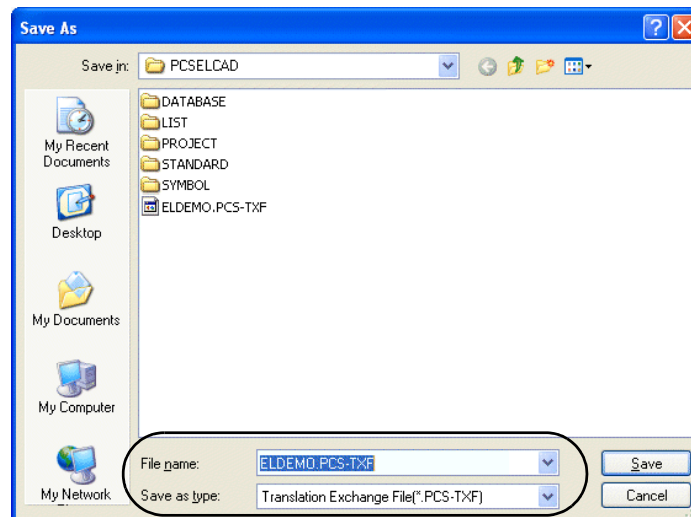
When you have entered new translations in the **Translations** dialog box, you can transfer these to the dictionary by clicking the **Add texts to dictionary** button.

### Export texts to translation

When you wish to send the texts to another person in order to have the texts translated, and this other person has no **Automation** license, do this:

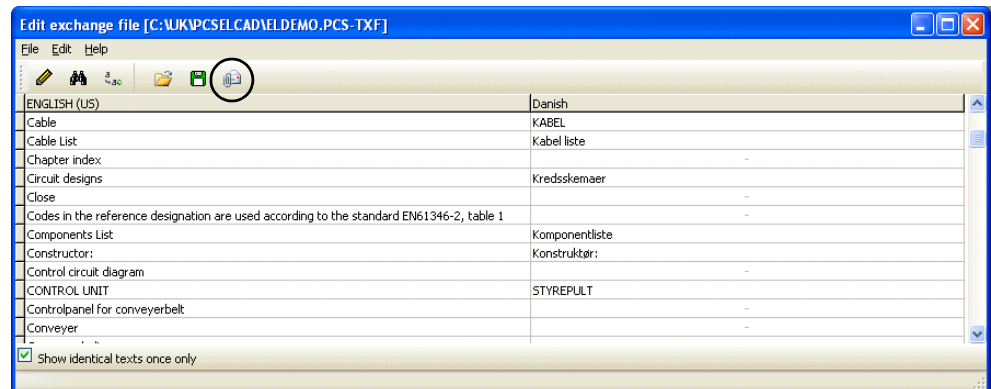


- 1) Click the **Export to translation** button.
- 2) In the dialog box **Save as**: Specify where to save the file, and click **Save**:

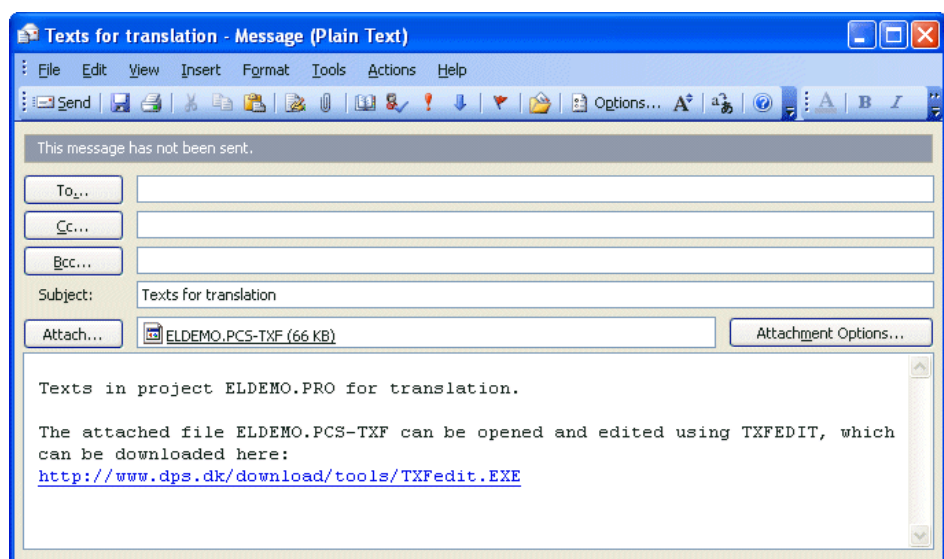


- 3) The exported file is now opened in the editing tool **Edit exchange file**. Here you can for instance edit texts and translations, and mail these to the person

that is going to translate the texts (read more in “Editing tool for exchange files” on page 111):



- 4) In the dialog box **Edit exchange file**: Click the **Send mail** button, to mail a file containing texts and translations.
- 5) Your e-mail program is now opened. Here you specify who to send the file to, and click **Send**:



When the receiver opens the attached file, he/she can open the file with the independent program **Edit exchange file** (TXFedit.exe), which is used for the translation. This program/tool can be downloaded via DpS CAD-center's website at [www.pcschematic.com](http://www.pcschematic.com), or by clicking on the link to the program in the mail.

— Read more about this in “Editing tool for exchange files” on page 111.

### Importing translated texts



When you receive a file containing the translated texts, you can open the file by clicking the **Import translated texts** button. You now enter the dialog box **Open**, where you select the file containing the translations, and click on **Open**.



### **Creating the translated project**

When all project texts are translated, click on **Create project**. Hereby a new project is created, containing the translated texts.

If not all texts in the original project have a translation, you will be informed about this, and no new project will be created.

In other words: You will not be allowed to create a new project, before all project texts have a translation.

#### *Creating a new dictionary*

If no dictionary is created for the applied languages, you will be asked whether to create a such, when you click **Create project**.



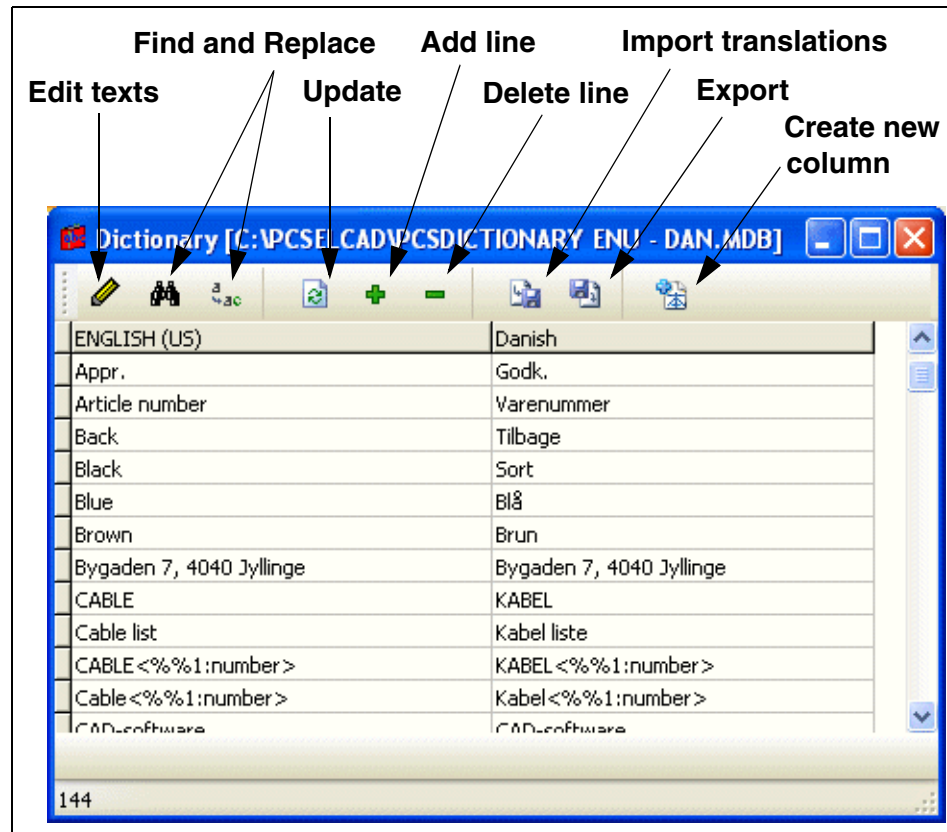
## EDITING DICTIONARIES

The dictionaries in **PC/SCHEMATIC Text Translator** can be used simultaneously by multiple users. To edit a dictionary, you must enter the **Dictionary** dialog box.



To enter the **Dictionary** dialog box, click the **Dictionary** button in the **Translations** dialog box, or select a dictionary in the **Dictionaries** dialog box, and click **Open**.

In the **Dictionary** dialog box, you have the following options for editing a dictionary:



### Edit, Find and Replace

The functions **Edit**, **Find** and **Replace** functions as described in “The Translations dialog box” on page 100.



### Updating the dictionary

When you click the **Update** button, the dictionary is updated with the changes made by other simultaneous users.



### Adding a new line in the dictionary

Click the **Add** button, and a new line is added to the dictionary. Type in a new text and a translation as well, if possible.

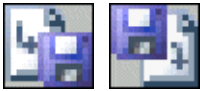


### Deleting a line from the dictionary

Click on the line you wish to translate, and click the **Delete** button.



## Export and Import



The functions **Export** and **Import** works as described in “The Translations dialog box” on page 100.

## Adding a new language to the dictionary

To add a new column for a new language, do this:

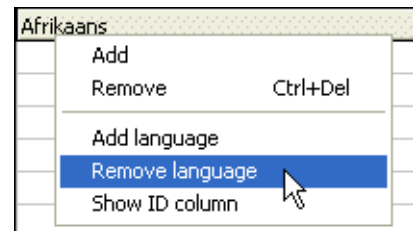


- 1) Click the **Add new language column** button.
- 2) In the dialog box **Add new language**: Select a new language in the list, and click **OK**.
- 3) The new column is now added.

## Deleting a language from the dictionary

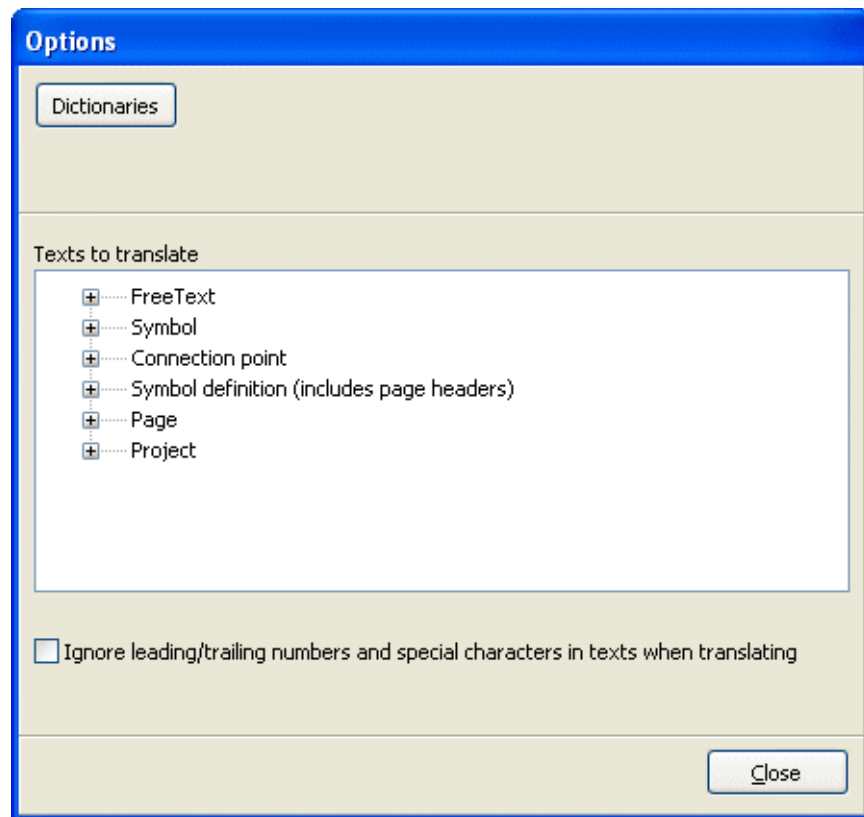
To delete a language column from the dictionary, do this:

- 1) Right-click on the header for the column you wish to delete.
- 2) Click on **Remove language**.



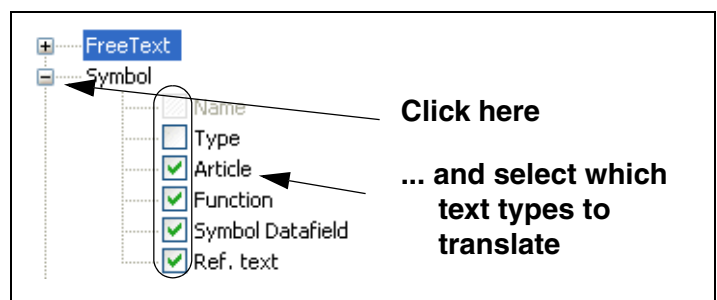
## OPTIONS FOR THE TRANSLATION

In the dialog box **Options**, you can for instance specify which type of project texts to translate, and which dictionary to use for the translation:



### Specify which texts to translate

In the field **Texts to translate**, you select which texts to translate — as displayed in the figure to the right.



### Ignore leading/trailing numbers and special characters when translating

When you select **Ignore leading/trailing numbers and special characters in texts when translating**, the **PC|SCHEMATIC Text Translator** program will behave as it did in the previous version of the program. — Read more about this in “Numbers, special characters and insertion of variables” on page 112.

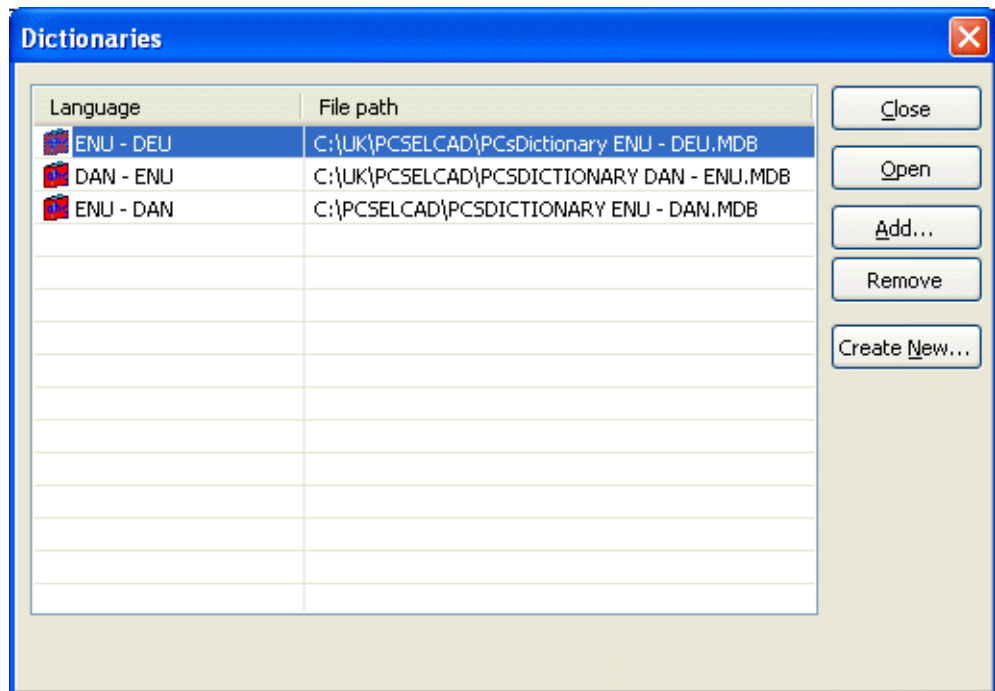
### The Dictionaries dialog box

To control which dictionary to use for the translation, click on **Dictionaries** in the **Options** dialog box.

You then enter the **Dictionaries** dialog box, where you can for instance create dictionaries, add/remove dictionaries and edit dictionaries.

If you have more than one dictionary for the same languages — for instance two English-Danish dictionaries —, **PC/SCHEMATIC Text Translator** will use the first of the dictionaries in the list.

You can change the position of a dictionary in the list, if you *click* on it, and then *drag* it to the desired position.



You have the following options:

#### *Opening a dictionary*

Click on a dictionary in the list, and click **Open**. The dictionary is opened in the dialog box **Dictionary**. — See “Editing Dictionaries” on page 105.

#### *Adding a dictionary*

Click on **Add**, and you enter the dialog box **Open**. Find the file containing the dictionary, and click on **Open**.

#### ADDING OLD DICTIONARIES

You can also add dictionaries from the previous version of **PC/SCHEMATIC Text Translator**. — Read more about this in “Converting dictionaries from the previous version” on page 115.

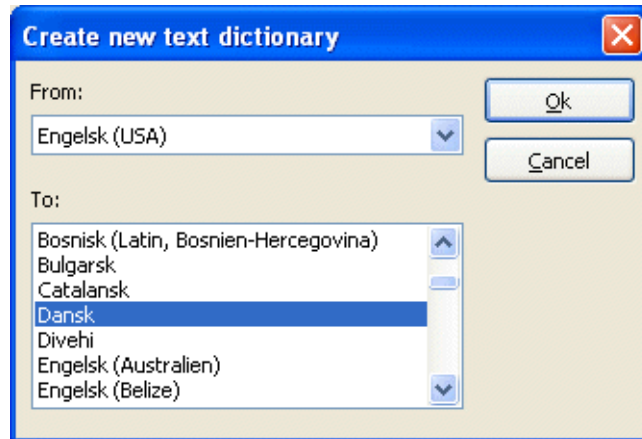
#### *Removing a dictionary*

Click on a dictionary, and click on **Remove**. Hereby the dictionary is removed from the list of dictionaries. Please note, that the dictionary itself is not deleted.

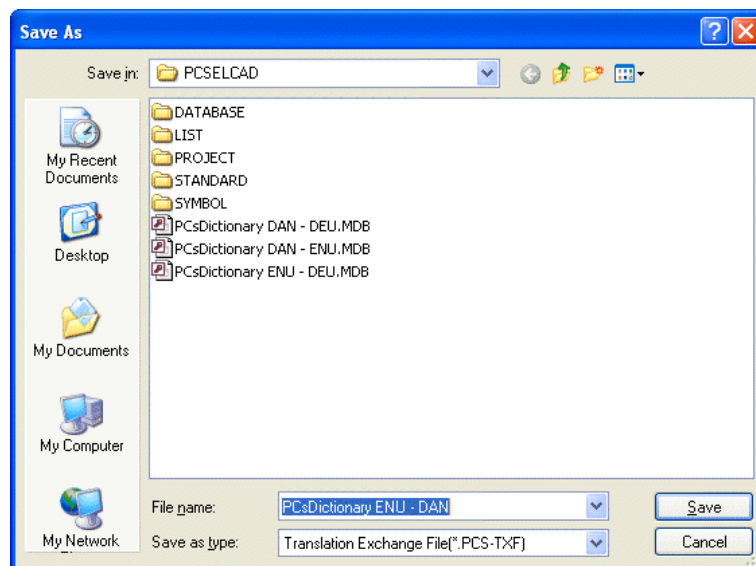
### Creating a new dictionary

To create a new dictionary, do this:

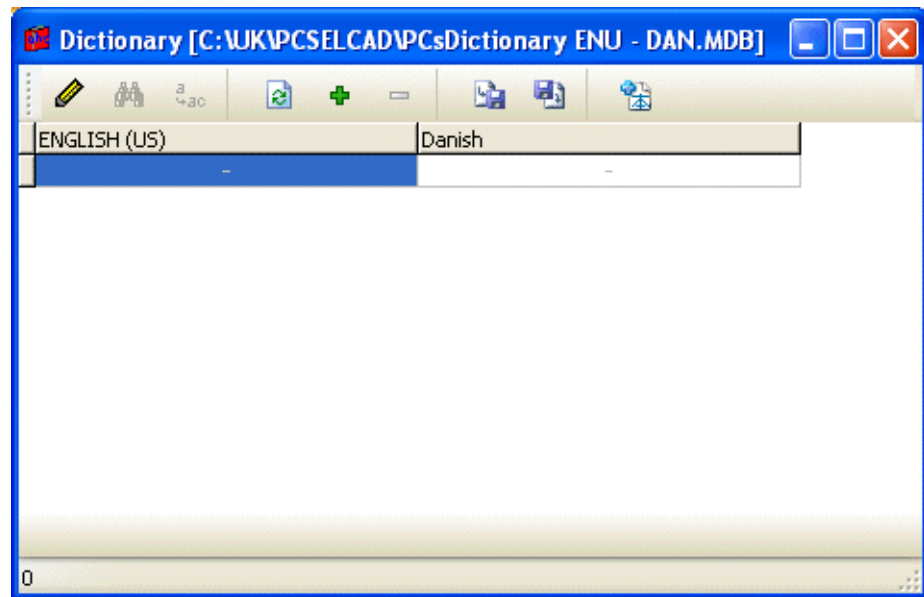
- 1) In the dialog box **Dictionaries**: Click on **Create new**, and specify the languages in the dictionary:



- 2) You can only choose one language in the **From** field, while you can choose more than one language in the **To** field. You can therefore have both an English-Danish and a English-German dictionary in the same dictionary. When you create an English-Danish dictionary, you can only use it for translating from English to Danish — and not the reverse. To translate from English to Danish, you must therefore create a Danish-English dictionary. Click **OK**, specify where to save the dictionary, and click **Save**:



3) You now enter a new dialog box:

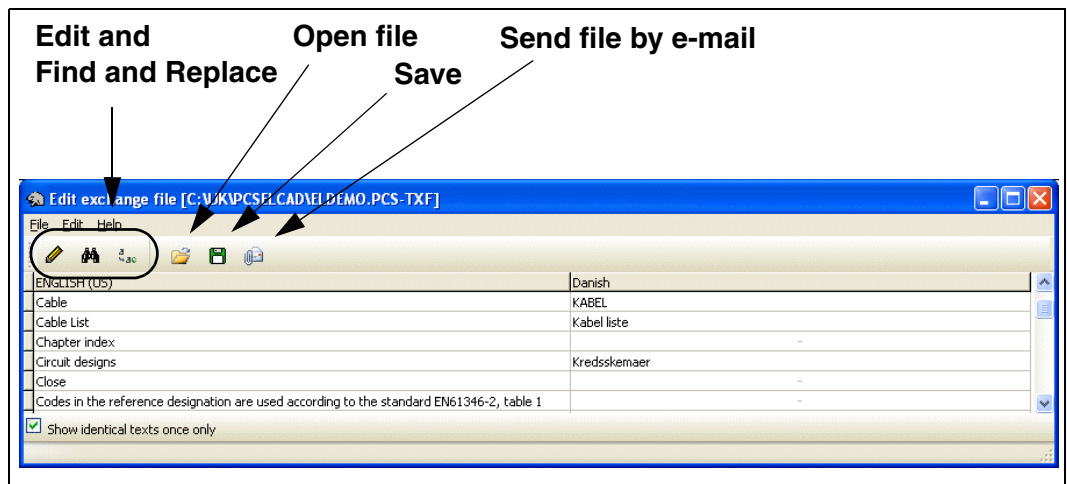


Here you can type in new texts in the dialog box, or import new texts into the dialog box, as described in “The Translations dialog box” on page 100.

## EDITING TOOL FOR EXCHANGE FILES

The editing tool **Edit exchange file** (TXFedit.exe) can be used for editing translations, originating from a project or from a dictionary.

When you send an e-mail to an external translator (see “Export texts to translation” on page 102), and he/she opens the mail containing the texts, he/she simultaneously gets the option of downloading this editing tool via a link in the mail:



### Edit, Find and Replace

The functions **Edit**, **Find** and **Replace** functions as described in “The Translations dialog box” on page 100.



### Open file

To open a new file, click the **Open** button.



### Save

To save, click the **Save** button.



### Send a file by e-mail

Click the **Send mail** to mail a file with texts and translations.

— Read more about this in “Export texts to translation” on page 102.



## RECOGNITION AND TRANSLATION OF TEXTS

To get the most benefit of **PC|SCHEMATIC Text Translator**, it is important to know something about:

- how the program recognize texts, and
- how the program translates the texts

This is described in the following.

### What does **PC|SCHEMATIC Text Translator** recognize?

First of all, it is important understand, that the program *does not recognize individual words*, but *only texts*.

If you have translated the text *Conveyorbelt Forwards*, this will not enable the program to translate the texts *Conveyorbelt* and *Forwards*, if they appear separately somewhere in the project.

This is because the program understands the text *Conveyorbelt Forwards* as a complex, and not as existing of the two individual words *Conveyorbelt* and *Forwards*.

Correspondingly, if you have translated the text *Conveyorbelt* at one place, and the text *Forwards* at another, you would still have to tell the program how to translate the text *Conveyorbelt Forwards*.

### Why does the translator not recognize a text?

If the translator does not recognize a text, it can be due to one of the following reasons:

- The text is new to the translator
- There is a spelling/typing error in the text
- The text does not follow the company standards for texts in projects
- The translator discerns between upper and lower case letters

If possible, the translator will suggest another text.

### Numbers, special characters and insertion of variables

If you in the **Options** dialog box have selected **Ignore leading/trailing numbers and special characters in texts when translating**, the **PC|SCHEMATIC Text Translator** program will behave as it did in version 9 and previous versions. This means that numbers and characters in the beginning and end of texts, will be ignored when translating. If the program can translate the text **Start motor 1**, it will therefore also be able to translate the text **Start motor 2**.

This can, however, be inexpedient if the number in the translation is placed in a different position relative to the text, compared to the original language. The English text **December 4th** should for instance be translated to the Danish text **4. december**.



You can therefore insert a variable, so that you can specify where to place the number in the translation:

DANSK	engelsk (US)
<%%1:number>. december	December <%%1:number>th
<%%1:number>/{<%%2:number>-<%%3:number>	<%%3:number>-<%%2:number>-<%%1:number>

As you can see above, you can use more than one variable in the same line. The bottom line translates the date format **27/9-2007** to the date format **2007-9-27**.

*Variables are inserted in the dictionary*

You insert these variables in the dictionary by adding a new line to the dictionary, as described in “Editing Dictionaries” on page 105.



If you are in the process of translating a project, you then click the **Translate using dictionary** button in the dialog box **Translations**, as described in “The Translations dialog box” on page 100. The project texts are then translated again, this time using the inserted variables.

### The dictionaries can only translate in one direction

In **PC|SCHEMATIC Text Translator**, the dictionaries can only translate in one direction. When you create an English-German dictionary, you can only use it for translating from English to German — and not the reverse. To translate from German to English, you must therefore create a German-English dictionary.

*A dictionary can translate To multiple languages*

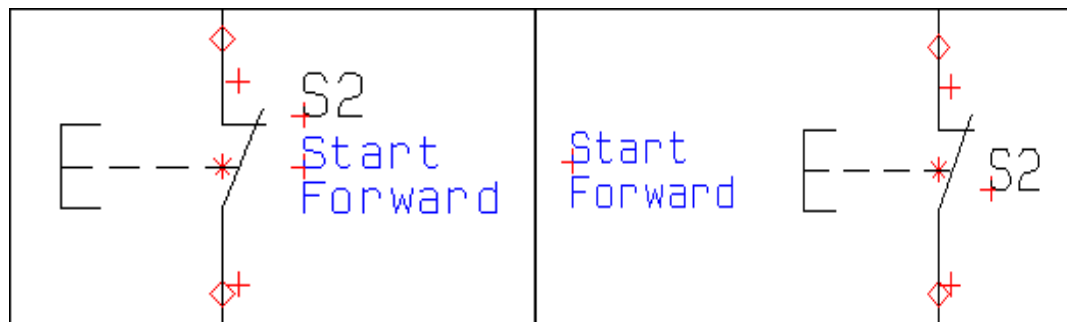
You can use a dictionary in **PC|SCHEMATIC Text Translator** to translate *to* multiple languages. This way, you can for instance have an English-German and an English-Polish dictionary in the same dictionary.

— See how to create a new dictionary in “The Dictionaries dialog box” on page 108, “Creating a new dictionary” on page 109 in particular.

## HOW TO PLACE TEXTS IN A PROJECT MEANT FOR TRANSLATION

When you are working on a project, and know that it is going to be translated at a later stage, it is a good idea to be careful about how to place the texts in relation to the lines and the symbols of the project. This is because some texts in the new language will turn out to be longer than the texts of the original language. If you have placed some of the texts inexpediently, they will overlap with symbols or lines in the project.

You specially have to be careful when you place the symbol texts. If you place them on the left-hand side of the symbol, they might overlap the symbol after the translation. If you place them on the right-hand side of the symbol, however, the texts are written in a direction going away from the symbol.



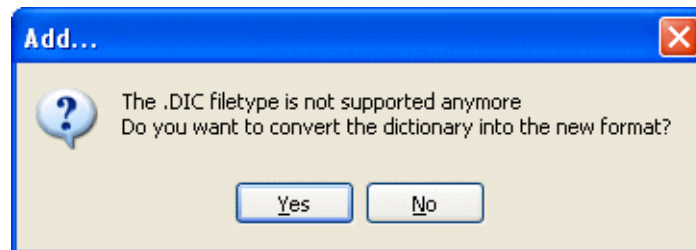
**Expedient**

**Inexpedient**

Alternatively, you can place the texts on the left-hand side of the symbols, but set the text alignment to be right justified. You can read more about text alignment in the **Automation** manual.

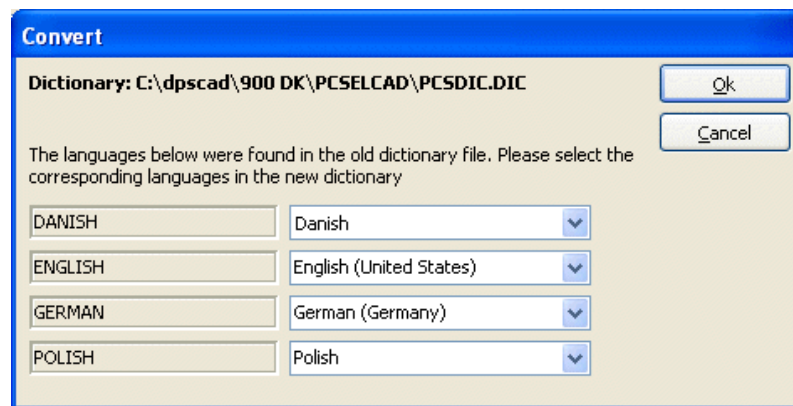
## CONVERTING DICTIONARIES FROM THE PREVIOUS VERSION

If you have used version 9 or previous versions of **PC|SCHEMATIC Text Translator**, you will be asked whether to convert the previously applied dictionaries to the new format, when you start **PC|SCHEMATIC Text Translator**.



Click **Yes** to create new dictionaries based on the old.

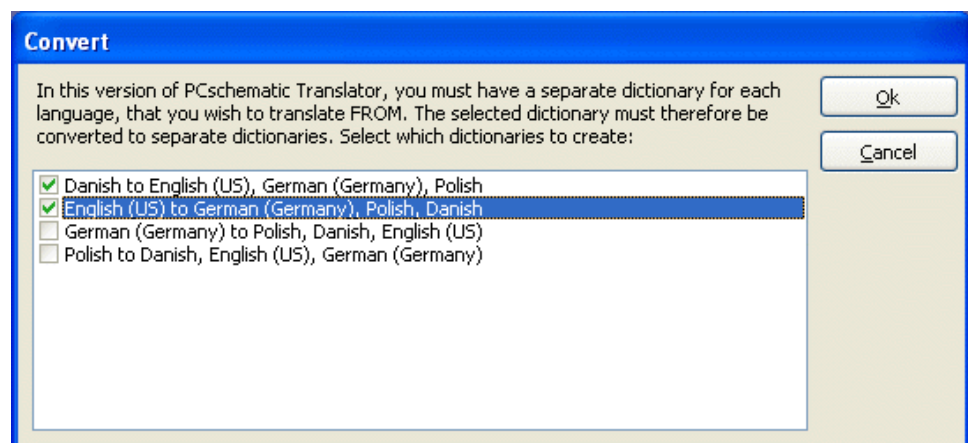
You now enter the dialog box **Convert**:



Now select the correct languages in the right side of the dialog box, and click **OK**.

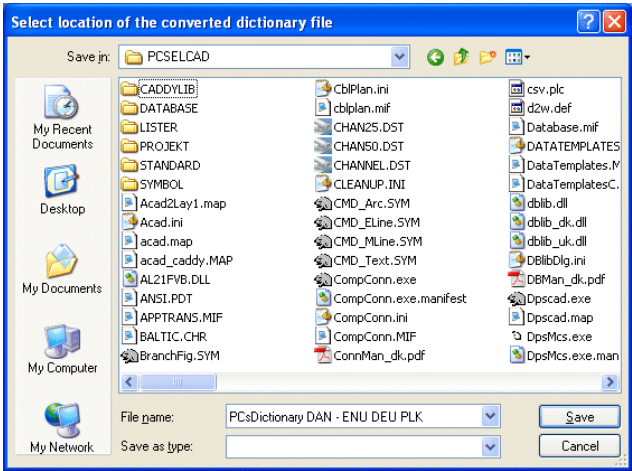
In **PC|SCHEMATIC Text Translator** you can only use the dictionaries to translate in one direction, as described in "The dictionaries can only translate in one direction" on page 113. This was not the case in the previous version.

This means that you can create several different dictionaries based on the previous version. You therefore enter this dialog box, where you specify which dictionaries to create:

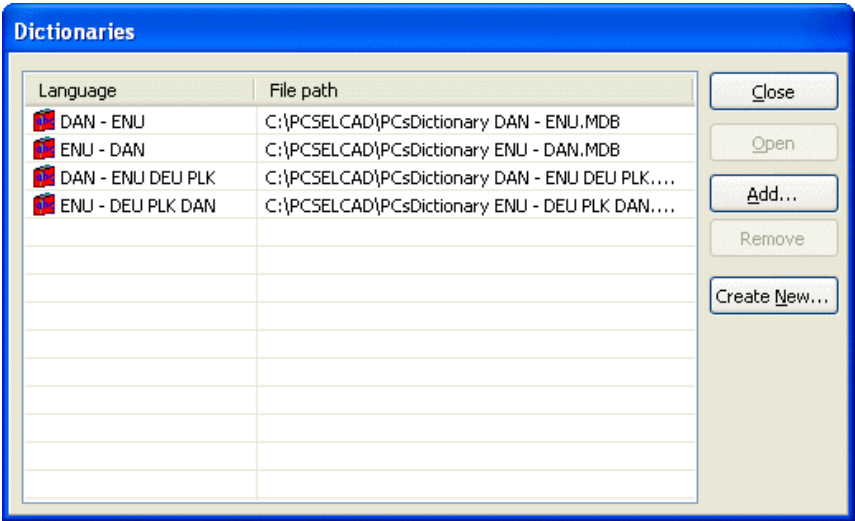


Select which dictionaries to create, and click **OK**.


You then enter the following dialog box, where you specify where to save each individual dictionary:



You can then see the new dictionaries added at the bottom of the list in the dialog box **Dictionaries**:



— Read more about the options you have here in “The Dictionaries dialog box” on page 108.



**NUMBERS AND SPECIAL CHARACTERS ARE HANDLED DIFFERENTLY NOW**

NUMBERS AND SPECIAL CHARACTERS ARE HANDLED DIFFERENTLY IN THIS VERSION OF **PC|SCHEMATIC Text Translator**, AS DESCRIBED IN “NUMBERS, SPECIAL CHARACTERS AND INSERTION OF VARIABLES” ON PAGE 112.



## Other options in the Tools menu

In the Tools menu in PCISCHEMATIC Automation, you will also find the following options:

- **Rename symbols**  
Read more about this in the PCISCHEMATIC Automation manual.
- **Database**  
Read more about this in the PCschematic Database manual.
- **Create variable list in (Sub)drawing**  
Read more about this in the PCISCHEMATIC Automation manual, and in the Project Generator manual.
- **Project generator**  
Read more about this in the Project Generator manual.
- **Mouse Chasing System**  
Read more about this in the PCISCHEMATIC Automation manual.





Index







# INDEX

## A

A3SYMDOC, drawing header	56
A4SYMDOC, drawing header	56
A4SYMDOL, drawing header	56
Add	
dictionary	108
new language to dictionary	106
Add button, translator program	105
Add symbol-list drawing header	57
Add texts to dictionary	102
Add texts to dictionary button	102
Adjustments tab	
graphical terminal plans	77, 93
when structured according to cables	79
symbol documentation	59
Adobe Acrobat Reader	11
AKIA	24
Allow rotation	72, 91
Applied symbol library for conversion	25
Attributtes	
in AutoCAD conversion	36
AutoCAD files	
converting files	15
export from PC schematic	46
Automatic	
symbol documentation	51

## B

Blocks	
delete in conversion process	35
in AutoCAD conversion	34
text attributtes for	35

## C

Cable	76
Cable direction	89
CADdy	24
Caption	70
Color, in AutoCAD conversion	27
Competing electrical applications	24
Connection caption font height	78
Connections tab	
graphical cable plans	89
graphical terminal plans	67
Convert lines to electrical lines	25
Convert, dialog box	115
Converting	
dictionaries from old version	115
objekt type	
Color	27
Layer	21
of file data during import and export	15
what is a mapfile	15
Create graphical cable plan	87

Create graphical terminal plan	64
Create new dictionary	109
Create translated project	104
Current date, data field	55

## D

Dashed lines on external side	78
Decode "-" (minus) as page number	41
Decode reference designations in free texts	41
Delete	
language in dictionary	106
Delete button, translator program	105
Dictionary	
add	108
add line	105
add new language	106
add texts	102
convert from old version	115
create new	109
delete language	106
delete line	105
edit	105
old version	108
open	102
remove	108
the dialog box Dictionaries	108
translate using dictionary	102
update	105
Dictionary button	102
Dictionaty	
translates in one direction only	113
Documentation	
of symbols	51
Doughnut count	40
Drawing headers	
for symbol lists	57
graphical cable plans	90
graphical terminal plans	69
symb.doc.	56
user defined	57
DWG files	
converting	15
export from PC schematic	46
import	20
import as new project page	43
import as symbol	44
intelligent import	22
numerically controled machine	47
save projects as	46
which functions are transferred	19
work process for converting	16
DWG/DXF map options	21
buttons in the dialog box	22
import/export	21

making room for the left-hand side of the tab	23
DXF files	
converting	15
export from PCschematic	46
import	20
import as new project page	43
import as symbol	44
intelligent import	22
numerically controled machine	47
save project as	46
which functions are transferred	19
work process for converting	16

## E

Edit	100, 105
dictionary	105
Edit exchange file tool	111
Edit text button	100
Electrical applications, competing	24
ELPROCAD	24
Empty texts in symb.doc.	55
EPLAN	24
Export	
map options for AutoCAD files	48
of AutoCAD files from PCschematic	46
texts	106
Export texts to translation	102
Export texts to translation button	102
External and/or internal connections	
graphical terminal plans	67
External translator	
tool for editing texts	111

## F

Filtype	
DWG or DXF files	44
Find	105
Find and Replace	101
Fixed space between terminals	70
Folders	
graphical survey of symbols in	51
Force project COLOR to	28
Force project LAYER to	32
Force project LINESYLES to	30
Force symbol COLOR to	28
Force symbol LAYER to	31
Force symbol LINESYLES to	30
Force visibility of connection point texts	55
Force visibility of symbol texts	55

## G

Graphical cable plans	87
Connections tab	89
create	87
document FROM and/or TO end of cable	89
drawing header	90
orientation	89
Pages tab	90
settings in dialog box	88

symbol texts	91
Graphical survey	
of symbols	51
Graphical symbol survey	56
Graphical terminal plans	63
Adjustments tab	77, 93
when structured according to cables	79
Connections tab	67
create	64, 66, 88
direct connection between terminals and symbols	65
drawing header	69
external and/or internal connections	67
orientation	67
Pages tab	69
settings in dialog box	66
structure	65
structured according to cables	65
symbol texts	71
Symbols tab	71, 91
Wires tab	76

## H

Help	9
Horizontal (a)	59
Horizontal (c)	59
Horizontal line	79

## I

Ignore leading/trailing numbers ...	107
Import	
DWG/DXF file as new project page	43
DWG/DXF file as symbol	44
texts	106
Import scale, in DWG/DXF map options	40
Import translated texts	103
Import translated texts button	103
Include file path in symbol name	55
Include symbol texts	71, 91
Index	
on-line manual	11
Insert data field	54
Insert English text	100
Insert text in empty connection point name texts	55
Insert text in empty symbol name texts	55
Interpret all pages in a file catalog as one project	24

## L

Language	
choose	98
without dictionaries	98
Layers	
delete during import, DWG/DXF map options	39
in PCschematic	32
transferring layer information intelligently	38
Left to right	
cable plans	89
terminal plans	67

**M**

Manual	9
Mapfiles	
color	27
editing	18
layer	31
linestyles	29
map options for Export of AutoCAD files	48
texts	33
Mapping	15
what is a mapfile	15
Margins	77, 93
Maximum number of symbols per page	60
Minimum space between symbols	78
Minimum space between terminals	77
Minimum width of symbol texts (e)	59

**N**

Never break words in symbol name or -title	55
None	76
Numbers and special characters	
in translation	112
Numerically controled machine	47

**O**

Old dictionaries	108
On-line manual	9
adapt to screen	10
buttons	10
index	11
index-finger	11
open	9
printing from	11
return to PCschematic	11
Table of Contents	10
zoom	10
Only include symbols modified after	53
Open	
applied dictionary	102
DWG/DXF file	43
Options	
for translation	107
Orientation	
graphical cable plans	89
graphical terminal plans	67

**P**

Page break between cables	90
Page break between terminal rows	70
Page menu	
DWG and DXF files	43
Page number in filename	25
Pages tab	
graphical cable plans	90
graphical terminal plans	69
symbol documentation	56
Path leading to terminal symbol	73
PCschematic Text Translator	97

Place connected symbol	67
Place symbol filename and -title ABOVE the symbol	60
Placing	
external/internal connections	
graphical terminal plans	67
Placing texts in projects for translation	114
Project	
translate	97
Project data =>	
Title	54
Project file name, data field	55
Project full path, data field	55
Project title	54
Project title, data field	55
Projects	
graphical survey of symbols in	51
save as DWG/DXF files	46
Prompt	37

**R**

Recognition and translation of texts	112
Remove	
dictionary	108
Remove symbol-list drawing header	57
Replace	101, 105
Replace button	101

**S**

Save	
project as DWG/DXF files	46
Scale	
symbols in the symbol documentation	60
Scale symbols larger than the average size	60
Scale symbols too large to place on page	60
Search button	101
Select	
terminal rows	67
Set connection points to external when dotted lines	40
Set to 'Not translated' status	101
Settings =>	
Libraries	73
Text/Symbol defaults	57
Show languages with existing dictionaries only	98
Signal ref. text	41
Similar texts	98
Similar texts found	99
Sorting	
terminal rows	68
Special characters and numbers	
in translation	112
Standard drawing	
for DWG/DXF files	43
Structure of	
graphical terminal plans	65
Symbol documentation	51
Adjustments tab	59

create	52
only symbols created later than	53
Pages tab	56
Symbols tab	53
Texts tab	54
Symbol library w/full path, data field	54
Symbol library, data field	54
Symbol texts	
on graphical cable plans	91
on graphical terminal plans	71
Symbols	
delete in conversion process	35
import DWG/DXF file as symbol	44
max. number per page (symb.doc)	60
minimum width of symbol texts (symb.doc.)	59
path leading to terminal symbol	73
scale in symb.doc.	60
symbol library for AutoCAD conversion	25
text above the symbol (symb.doc.)	60
Symbols in current project	53
Symbols in symbol library	53
Symbols last modified date, data field	55
Symbols tab	
graphical terminal plans	71, 91
symbol documentation	53
<b>T</b>	
Tab	
Advanced, in DWG/DXF map options	40
Application, in DWG/DXF map options	24
Attributes, in DWG/DXF map options	36
Blocks, in DWG/DXF map options	34
Color, in DWG/DXF map options	27
Layer(Def), in DWG/DXF map options	38
Layer(Map), in DWG/DXF map options	31
Linestyle, in DWG/DXF map options	29
Texts, in DWG/DXF map options	33
Table of Contents	
On-line manual	10
Terminal plans	63
Terminal rows	
select	
graphical terminal plans	67
sorting	68
Terminal symbols	
graphical cable plans	92
graphical terminal plans	72
path leading to	73
Text translator, dialog box	97
Text width/height ratio	41
Texts	
displayed for the symbols in symb.doc.	55
minimum width of symbol texts (symb.doc.)	59
placing in projects for translation	114
similar	98
specify which types to translate	107
symbol texts	
on graphical cable plans	91

on graphical terminal plans	71
text attributtes in AutoCAD conversion	36
Texts tab	
symbol documentation	54
Texts, in AutoCAD conversion	33
The file(s) contains one project page only	24
Tool	
Edit exchange file	111
Tools =>	
DWG/DXF Map options	18
Graphical Cable Plan	87
Graphical terminal plan	64
Symbol documentation	51
Text translator	97
Top to bottom	
cable plans	89
terminal plans	67
Translate	
create translated project	104
Translate a project	97
Translate using dictionary	102
Translate using dictionary button	102
Translation	
variables in texts	112
why a text is not recognized	112
Translation and recognition of texts	112
Translation program	97
Translations, dialog box	100

<b>U</b>	
Update button	105
Update dictionary	105
User defined drawing headers	69, 90
User designed drawing headers	57

<b>V</b>	
Variables	
in translation	112
Vertical (b)	59
Vertical (d)	59
Vertical line	79

<b>W</b>	
What does PCschematic Translator recognize	112
Wire number	76
Wires tab	
graphical terminal symbols	76