

AUTOMATION

WHITEPAPER



AUTOMATION

PCSCHEMATIC Automation is a CAD solution for documentation of automation, installation, pneumatics and hydraulics. The software enables you to use your resources as effectively as possible, which allows extra energy for other important tasks. In this folder you get an introduction to Automation, see which functions are included, and learn how the specially designed PCSCHEMATIC databases save many work hours when performing your daily tasks.

Focus on the electrical aspects

PCSCHEMATIC Automation is designed from the basic concept of setting you free to focus on the electrical aspects, while the program handles the practical side. Beyond the basic drawing functions, Automation offers you a comprehensive series of functions specially designed for electrical projects.

Customized for electrical projects

Among these functions you will find automatically updated references between electrical symbols for the same component, automatic PLC functions, survey symbols for PLCs, automatic line drawing (router), intelligent renaming of components when copying, support of mounting correct drawing and single-line diagrams, design check, control of color codes for cable wires, automatic replacing of symbols, automatic wire numbering, automatic generation of graphical terminal, cable and connection plans, as well as automatic updating of project and page information.

You soon become confident with the programme, and the programme buttons and menus follows common Windows guidelines.

A project file can contain an infinite number of pages, each page containing up to 255 layers. You can also work in different

for PLCs, sensors and transducers, intelligent building installation, computer and telecommunication, alarm installations, as well as symbols for ground plan drawings. You can easily create your own symbols, and have your symbol libraries documented automatically.

48 component databases

In Automation you can create your own component databases. As an extra advantage, several leading component manufacturers have created their own component databases for the programme.

Beyond common component information - such as part number, descriptions, component vendor and prices - these databases contain both electrical and mechanical symbols for the individual components. Using these databases, you can therefore easily and safely create diagrams and mechanical drawings, and generate lists.

Along with the program you also get PCSCHEMATICs own database program. Automation can use Access databases directly, and is also open to other database systems supported by MDAC or ODBC. You will therefore be able to use all major database systems in conjunction with Automation - such as for instance MS-SQL and Oracle. Therefore, the databases can become an integrated part of the database



minerals lists, PLC lists, cables lists, labels, connections lists and tables of contents. These lists are updated automatically by the programme. Unit drawings can also be a part of the projects.

The lists can both be created as pages in the projects, and as user-defined files, which can easily be imported into other systems - for instance as order files.

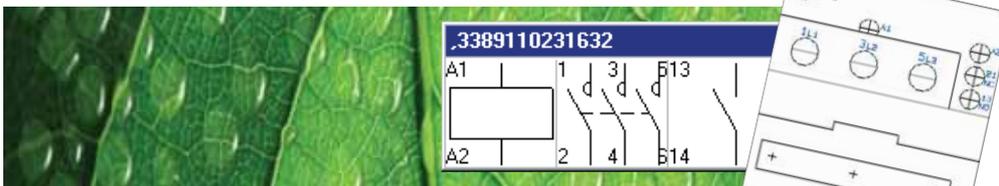
Sub drawings with article data and automatic project generation

You can easily create sub drawings with article data for each individual symbol in the sub drawing. You can then *drag* the sub drawings directly into any of your projects. When dragging a sub drawing into a project, you can furthermore select which set of model data to attach to the symbols in the sub drawing. This is called Module and Model Based drag'n draw. Based on these sub drawings, you can generate projects containing thousands of pages automatically.

Intelligent reference designations

In Automation you can automatically assign reference designations on project, page, area and symbol level.

When placing a symbol on a page or in an area with reference designations, the symbol is automatically assigned the ref-



elevations, and draw isometrically.

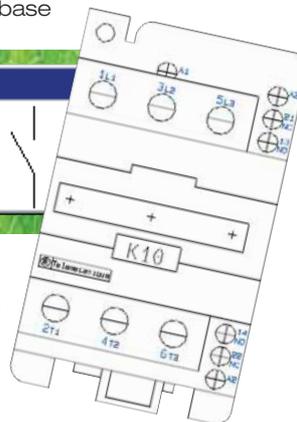
Plenty of Standard IEC symbols

With the programme you get the standard IEC symbols for automation, installation, hydraulics/pneumatics and flow diagrams. Furthermore, you will find symbols

system of the company.

Automatic update of all Lists

In Automation you can design your own parts lists (BOM), components lists, ter-



reference designations of the page or area (IEC 61346). Succeedingly, you can have lists filled out according to the applied reference designations automatically. When you change a reference designation, the program can alter this reference designation in the full project accordingly.

Intelligent chapters

You can further structure your projects by applying Automation functions for creating chapters. Tables of contents and lists can be created for both each individual chapter, and for the full project.

IEC standards supported intelligently

In all parts of the program, structuring the documentation has been made as easy as possible. In Automation you can follow the present electrical standards, hardly without noticing it.

Import/export of DWG/DXF files

You can import DWG/DXF files, and export DWG/DXF files. Naturally, only elements known by both programs are exchanged.

Inserting OLE objects

Documents from for instance Word or Excel can be inserted directly on the project pages, because the program handles so-called OLE objects. If for instance you insert an AutoCAD document, you can draw on top of the inserted document in Automation.

Interfacing to PLC tools

You can interface to PLC tools and to dimensioning tools. During import from PLC tools, the connected PLC data are corrected directly in the diagrams - of course under strict control.

networks facilities, such as extended file security and a flexible sharing of floating licences and administrator control.

Create your own applications

Automation is designed so openly, that you have the option of creating your own applications. If you miss a specific function in the program, you therefore have the option of creating it yourself. Please contact PCSCHMATIC A/S for further information.

Flexible licenses

When you purchase a floating license, you get the option of allowing others to tank/borrow licenses via the network. If a colleague needs to update an electrical documentation abroad, you can also mail a time limited license to your colleague.

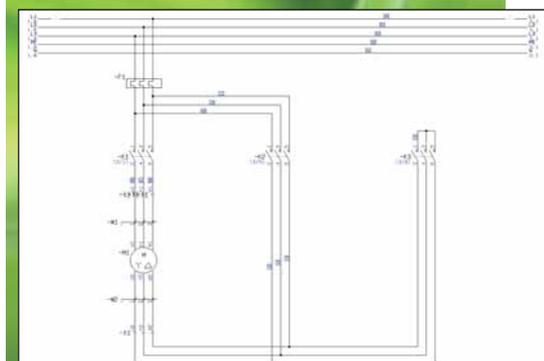
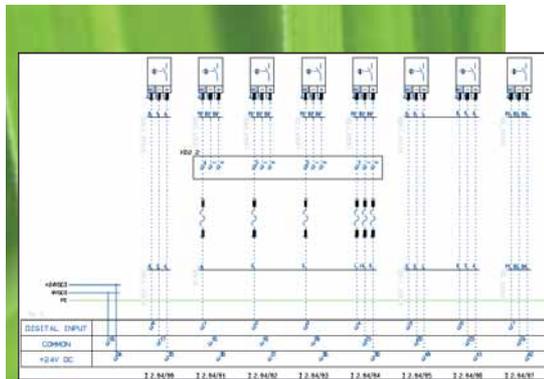
Repetitive strain Injuries

To reduce the use of the mouse, Automation has included the program Mouse Chasing System, for learning program short-cuts while working. Short-cuts in the program are predominantly single keys - such as „s“ for the „Symbols“ command.

Languages and character sets

You can easily install the English version of the program, including online manuals. Danish, German, Czech, Polish, Hungarian, Russian, Swedish, Norwegian, Portuguese, Turkish and Chinese versions are also available.

The program includes a number of character sets - for instance Russian, Polish, Czech and Baltic characters. Furthermore, a program for translation of project texts is included. In this program you can add new languages when necessary.

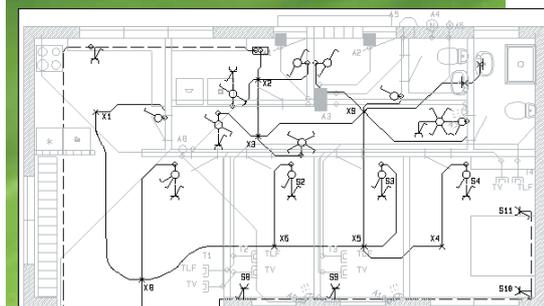


SEE THE GET STARTED VIDEOS AT
[YOUTUBE.COM/PCSCHMATIC](https://www.youtube.com/PCSCHMATIC)

Networks

As a consequence of the many major customers, Automation has extended

Automation was originally created more than 25 years ago, and is now used worldwide.



FUNCTIONS IN AUTOMATION

	Automation 40	Automation Flex	Automation Smart	Automation Advanced
Overview and navigation				
Project oriented: All parts of a project gathered as pages in a single file – including diagrams, mechanical pages, lists, tables of contents, chapter dividers etc.	✓	✓	✓	✓
Enhanced Explorer window: For easy navigation between projects and project pages, and for finding symbols etc. in project pages	✓	✓	✓	✓
Net navigator for easy navigation through electrical potentials	✓	✓	✓	✓
Cross references are created and updated automatically for all types of components: Click and jump between the symbols for a component, or click in lists and jump directly to the component	✓	✓	✓	✓
Object Lister for overview over all objects in the documentation – and for editing their attached texts/ data. (Texts can also be edited by clicking on the individual texts in the project pages)	✓	✓	✓	✓
Show available window: Easy access to all components with unused symbols/functions	✓	✓	✓	✓
Reference designations, templates, sub drawings, project generation ...				
Creation and handling of reference designations at component, area, page and project level (function, location and product aspects) – according to IEC 81346, KKS or user defined standards	✓	✓	✓	✓
Reference designations handled intelligently when changing, copying and merging pages and projects	✓	✓	✓	✓
Project, page and list templates – just drag them into your projects	✓	✓	✓	✓
Sub drawings / sub diagrams with attached article data – just drag them into your projects. Possible to select different sets of model data for each individual sub drawing	✓	✓	✓	✓
Module and Model Based Drag'n Draw: Create full projects just by dragging in pages, sub drawings, lists and sub projects	✓	✓	✓	✓
Automatic project generation via project definition file – e.g. via Excel ("Autodiagramming")	✓	✓	✓	✓
Automatic project generation based on selected options	✓	✓	✓	✓
Panel router for automatic routing in conduits	✓	✓	✓	✓
Mounting assistant for overview over and control of mounted components and wires	✓	✓	✓	✓
Databases, components and symbols				
Databases from 48 component manufacturers created designed specially for PCSCHEMATIC. All components have attached both electrical and mechanical symbols	✓	✓	✓	✓
Component wizard for quick and effective creation of own components with attached symbols	✓	✓	✓	✓
Unique timesaving database workflow: Symbol pickmenus for all multi-function components in the database – just click and place the symbols in the diagrams and article data are attached automatically	✓	✓	✓	✓
Advanced copying: All symbols and their reference designations are renamed intelligently – e.g. when copying symbols and areas, or when inserting subdrawings, subprojects and projects. Attached article data are included when copying	✓	✓	✓	✓
Rename symbols and Insert name at project level	✓	✓	✓	✓
Automatic load of mechanical symbols for the components used – handled by the database	✓	✓	✓	✓
Number of cable wires and colour codes controlled by the database	✓	✓	✓	✓
Easy search for / editing of data for symbols and components	✓	✓	✓	✓
Automatic documentation of symbol libraries and applied symbols in projects	✓	✓	✓	✓
Comprehensive IEC 60617 symbol libraries	✓	✓	✓	✓
Design your own symbols as well as Symbol generator for quick generation of simple symbols	✓	✓	✓	✓
PCSCHEMATIC Database program Included for handling component data in e.g. Access	✓	✓	✓	✓
ODBC and MDAC interface for communication with database systems	✓	✓	✓	✓
Drawing functions				
Easy to get familiar with the program – and easy to create and maintain documentation	✓	✓	✓	✓
Reuse existing documentation intelligently	✓	✓	✓	✓
Automatic naming of all types of symbols	✓	✓	✓	✓
Handling of subnames on symbols – including individual references	✓	✓	✓	✓
Automatic naming and update of symbol names according to current path numbers possible	✓	✓	✓	✓
Reference crosses or contact mirrors with cross references – click and jump to the individual symbols for the component	✓	✓	✓	✓
Replace symbol function: On project or page level, and for individual symbols	✓	✓	✓	✓
Handling of multi-layer terminals – electrically, mechanically and in lists	✓	✓	✓	✓
Mounting correct drawing supported – with option to display in "dot" drawing mode	✓	✓	✓	✓
Counting functions in numerous numbering systems	✓	✓	✓	✓

	Automation 40	Automation Flex	Automation Smart	Automation Advanced
Single-line and multi-line diagram features	✓	✓	✓	✓
Lines are closed automatically when symbols are deleted	✓	✓	✓	✓
Router: Automatic drawing of connecting lines – for instance when placing symbols	✓	✓	✓	✓
Aligning, spacing and trimming functions	✓	✓	✓	✓
Handling of jumper links and busbars	✓	✓	✓	✓
Automatic wire numbering function	✓	✓	✓	✓
Design check	✓	✓	✓	✓
Single-key keyboard short cuts – define them easily	✓	✓	✓	✓
Drawing headers, lists and import/export				
Automatic list update: Of tables of contents, parts lists (BOM), component lists, terminal lists, cable lists, connection lists, labels and PLC lists. Any of these lists can be defined freely by you	✓	✓	✓	✓
Excel/XML/text files can be exported for all of the mentioned list types	✓	✓	✓	✓
Export of wire numbers	✓	✓	✓	✓
Drawing headers for any type of project page can be defined freely – and can be replaced on project level	✓	✓	✓	✓
Automatic generation of graphical terminal, cable and connection plans	✓	✓	✓	✓
Print labels via Excel or via e.g. CLIP PROJECT Marking or M-Print PRO	✓	✓	✓	✓
Export of PDF files containing active links in the documentation	✓	✓	✓	✓
PLC				
PLC survey symbols	✓	✓	✓	✓
Automatic PLC I/O addressing	✓	✓	✓	✓
Automatically updated cross references between PLC symbols	✓	✓	✓	✓
Import/export of PLC I/O files to/from diagrams – e.g. via spreadsheet/Microsoft Excel	✓	✓	✓	✓
Open for integration with PLC configuration tools – e.g. Simatic TIA configurator	✓	✓	✓	✓
House installation				
Import/export of DWG/DXF files (Member of Open Design Alliance)	✓	✓	✓	✓
Drawing with elevation displayed in 2D or isometrically	✓	✓	✓	✓
Unit drawings	✓	✓	✓	✓
Pages for mechanical layouts and ground plan drawings as well as drawing in layers	✓	✓	✓	✓
Measurement objects	✓	✓	✓	✓
Design a panel, and the panel layout is generated automatically. Single line diagrams are drawn automatically based on the panel design	✓	✓	✓	✓
Document small switchboards and panels according to EN61439-1 and 3 concerning power dissipation calculation and more	✓	✓	✓	✓
Revision control, passwords etc.				
Revision control and revision clouds	✓	✓	✓	✓
Password protection of projects	✓	✓	✓	✓
Translation of texts on project pages	✓	✓	✓	✓
Insert OLE objects and images	✓	✓	✓	✓
OLE automation: Create your own and scripts applications for Automation (full documentation available)	✓	✓	✓	✓
Net versions only				
Flexible sharing of net licenses and license borrowing		✓	✓	✓
Administrator control		✓	✓	✓

Automation variants	Available functions	Allowed number of symbols / connections	Max number of diagram + mechanical pages	Also available as net version
Automation 40 – free version	All	40/200	10	No
Automation Flex versions	All	150/750 or 350/1750	Unlimited	Yes
Automation Smart versions	All	Unlimited	20 or 40	Yes
Automation Advanced versions	All	Unlimited	Unlimited	Yes

Automation 40 can be downloaded for free at www.pcschematic.com/demo • Automation Flex & Automation Smart are economy versions

Let Automation revolutionize your job

PCSCHMATIC Automation is revolution in electrical CAD software. No less. You can now arrange your workflow, so that you practically speaking have created 80% of the documentation before you get started.

When creating electrical documentation, you typically apply the same 30-40 sub drawings, which you use in different combinations. These sub circuits / sub drawings can furthermore have different sets of article data attached, depending on which physical components the symbols in the drawings represent.

In PCSCHMATIC Automation you can therefore *drag* these sub drawings directly into the project pages, and at the same time select which *models* of the sub drawings you wish to use - and hereby which article data to attach automatically to the individual symbols in the selected sub drawing.

Just drag in the sub drawings

In Automation, this is called "Module and Model Based Drag'n Draw". Applying this function, you *drag* sub drawings into the project from a window in the right-hand side of the screen, and Automation places the selected sub drawing on the active project page.

At the same time, the symbols in the sub drawing are assigned article data, depending on which model of the sub drawing you choose. Furthermore, the

sub drawing is automatically connected to the drawn diagram - e.g. to the electrical potentials on the page - and the applied symbols are named intelligently by the programme. (The free tutorial videos at www.pcschematic.com shows this workflow visually).

No need to draw any more

The revolutionary aspect is therefore that you can now create a complete electrical documentation by just *dragging* all of the desired documentation parts/elements directly into the project, and the program draws them for you. Practically speaking, you hardly need to draw any longer.

And it is not just sub drawings you can drag into the active project. You can also drag full pages into the project, such as chapter front pages, tables of contents, all types of lists and complete diagram pages. Furthermore, you can even drag other projects into the project.

Workflow revolutionized

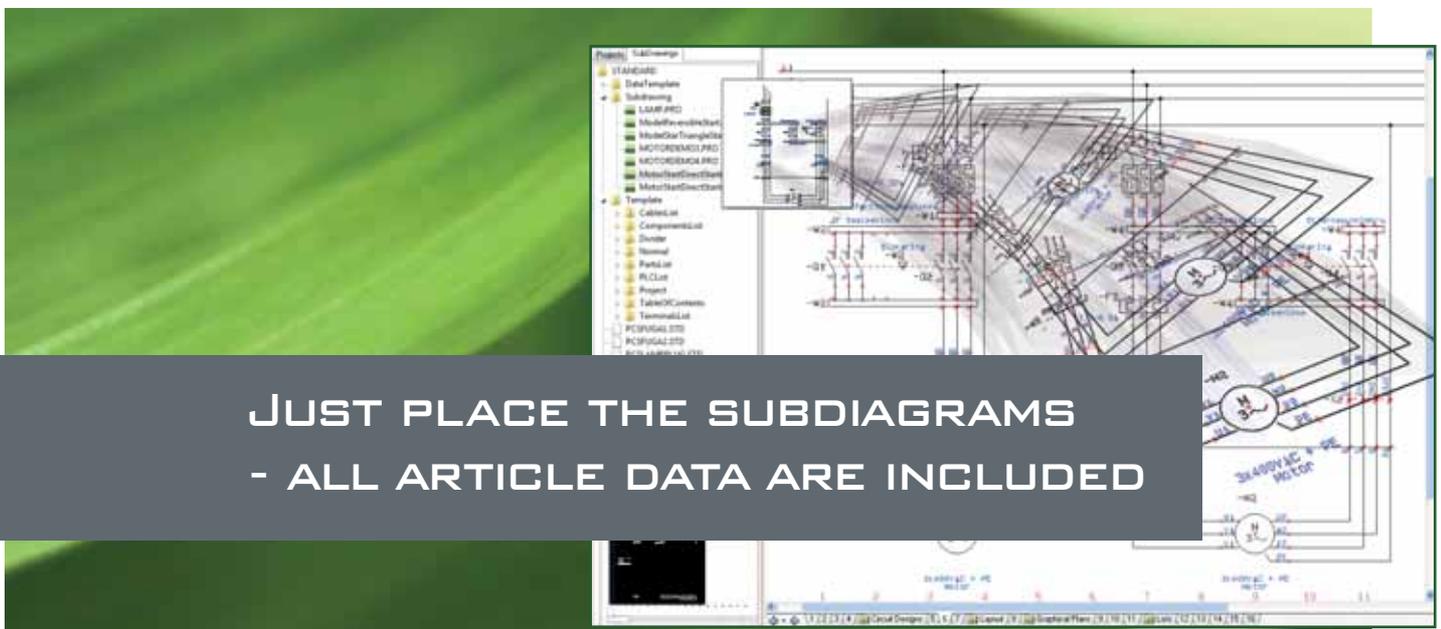
To truly grasp the full extent of this, it is important to understand that it opens for a revolutionary fast and simple workflow.

You can therefore start a new project by dragging a project template into Automation. This project can contain the project pages you always need when starting a project, such as tables of contents, chapter front pages, schematics pages and selected list pages. When you subsequently need for instance more diagram pages with already inserted electrical potentials, you also drag these into the project. Then you drag the desired sub drawings directly into the project pages, where they connect to e.g. the electrical potentials on the page, and the symbols are renamed intelligently.

Article data included automatically

Because the sub drawings have attached article data, you then just update the project lists automatically, and the documentation is now complete. All lists are now filled with article data etc., and you can send the order requisitions to the component vendors.

During the tendering phase of a project, you can therefore quickly get a survey of the total cost of the components, because it can be calculated automatically in the lists.



This is how fast and simple it is.

Concurrent creation of libraries of sub drawings

In order to make this possible, you must, quite obviously, first have created a library containing the set of sub drawings, which your company applies in its electrical documentation projects.

This could sound like a bit of a mouthful, but this has been taken into consideration in the development of the program functions. It is therefore easy to create sub drawings as they should have been created in the documentation anyway. Having drawn them once, you can subsequently apply them with different sets of model data in all future projects. Furthermore, it is really simple to copy sub-diagrams from existing Automation projects.

Save 80% time

To make it possible for existing and new users to get started quickly, PCSCHMATIC - and its distributors - offers a consultation service, consisting of an analysis of the already existing electrical documentation. Based on this analysis, the optimal set of sub drawings, models and project templates can be effectively created.

This also has the consequence that new users can get started creating complete electrical documentation projects really quickly.

In larger companies, this way of working might result in that one(!) single person is pointed out as the main responsible for creating the company sub drawings, while the others in future almost only will need to create documentation by dragging templates and sub drawings into projects.

Experiences from the industry suggests that the documentation time can be reduced by up to 80 percent.

Mitsubishi and Siemens sub drawings

At present 48 component manufacturers have created component databases for PCSCHMATIC, including article data as well as electrical and mechanical symbols for all components in the databases.

Furthermore, Siemens and Mitsubishi have created selected sub drawings, including attached article data, ready to be dragged directly into your diagrams. - So, with a little luck, you might not even have to create some of the sub drawings yourself.

MODULES

ALL PARTS OF THE ELECTRICAL DOCUMENTATION CAN NOW BE CREATED AS *MODULES*:

- SUB DRAWINGS / SUB DIAGRAMS
- PROJECT PAGES FOR INSERTING SUB DRAWINGS
- CHAPTER FRONT PAGES AND CHAPTER TABLES OF CONTENTS
- LISTS AND INFORMATION PAGES
- PROJECT TEMPLATES / SETS OF PROJECT PAGES

MODELS

WHEN YOU *DRAG* IN A SUB DRAWING, YOU ALSO SELECT THE SUB DRAWING *MODEL*:

- THE SYMBOLS IN THE SUB DRAWING GETS ARTICLE DATA ACCORDING TO THE MODEL
- THE SYMBOLS IN THE SUB DRAWING ARE RENAMED
- WIRE NUMBERS IN THE SUB DRAWING ARE RENAMED
- REFERENCE DESIGNATIONS ARE UPDATED
- THE SUB DRAWING IS AUTOMATICALLY CONNECTED TO THE DIAGRAM ON THE PAGE

MMB DRAG'N DRAW

CREATE FULL ELECTRICAL DOCUMENTATION PROJECTS BY *DRAGGING* IN MODULES AND MODELS:

- DRAG IN A TEMPLATE PROJECT CONTAINING THE PAGES YOU TYPICALLY USE
- DRAG IN E.G. SOME EXTRA SCHEMATICS PAGES WITH ELECTRICAL POTENTIALS, WHICH YOU NEED FOR THIS PARTICULAR PROJECT
- DRAG THE DESIRED SUB DRAWINGS INTO THE PAGES, AND SELECT WHICH MODELS TO APPLY
- UPDATE THE PROJECT LISTS AUTOMATICALLY. (POSSIBLE BECAUSE THE SUB DRAWINGS CONTAINS ARTICLE DATA FROM THE DATABASE)

THE DOCUMENTATION IS NOW COMPLETE.

Support, training and webinars

Support and upgrade

When you purchase an Automation license, you can also purchase an Automation maintenance agreement. With the maintenance agreement, you secure the latest versions and access to our unique support.

The telephone, e-mail and web based support ensures you, that you never grind to a halt. Instead, you are quickly and effectively assisted with the best imaginable solutions to your challenges.

The Automation software is updated every year, so that you can work competently and comply with the latest standards.

Training and webinars

Automation training is arranged at a regular basis. We also arrange custom made webinars for companies all over the world. Please contact your local PCSCHMATIC agent, or contact PCSCHMATIC A/S, for further details.



Our databases are different. This saves you a lot of time!

Automation gives you a unique workflow, which saves you lots of time every single day. The unique databases plays a major role. 48 leading component manufacturers have created them specially for PCSCHEMATIC.

Automation has a unique workflow, which automatically handles all article data. This takes effect from the moment you select the first electrical symbol via the component database, to the moment where all of your project lists are updated automatically with component and article data.

Major timesaver

Three main factors make it possible to save so much time:

- 1) The comprehensive manufacturer databases for Automation, containing both article data and electrical symbols for all components
- 2) Easy access to finding these components, and placing their symbols in the

schematic diagrams - so that the article data from the database remains attached to the symbols

3) Attached article data etc. for the symbols are automatically transferred to the project lists, and to list files which can be forwarded to e.g. purchasing and storage management systems.

Handles all trivial tasks

When you place symbols in the schematic diagrams, the program automatically keeps track of which of the symbols for the component are used in the project, updates references between symbols for the components, creates a survey of which components still have available

symbols in the project, suggests the next available symbol name etc.

Because the program handles all these trivial details, it sets you free to focus on the electrical aspects of your job.

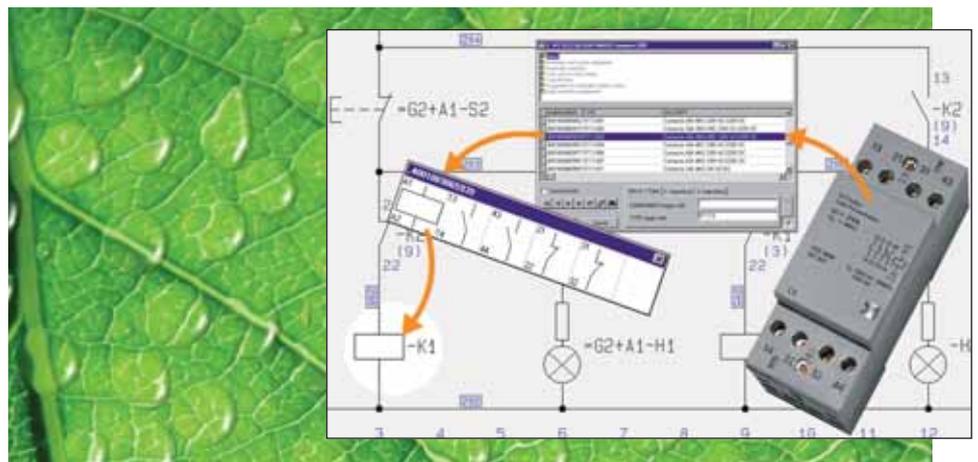
Mechanical symbols for mechanical panel layouts

The databases for Automation also contain mechanical symbols for the components. The program can therefore automatically supply mechanical symbols for the components you have used in your project - to insert directly in a mechanical panel layout.

The 48 component manufacturers have included ordering data, as well as electrical and mechanical symbols for the components in their databases for PCSCHEMATIC.

When you click on a component in the database, you automatically get a menu containing all of the electrical symbols for this specific component.

When you subsequently place the symbols in your diagrams, the program memorizes which components these symbols represent.



**JUST PLACE THE SYMBOLS
- ALL ARTICLE DATA ARE INCLUDED**

When you have completed your electrical diagrams, the program can therefore automatically fill out components lists and parts lists etc. - and need not type in any component information. Thus, you can focus on the electrical aspects, and let the program deal with the practicalities.

Originating from the electrical symbols in the diagrams, the program can also fetch the mechanical symbols for the components automatically. These symbols can for instance be used for mechanical layouts.

