



AUTOMATION

QUICK START

1/4-2014

PREFACE

This manual gives you a quick introduction to working with the **Automation** software from PC|SCHEMATIC. You get an introduction to:

- The contents of **Automation** projects
- Inserting new project pages
- Drawing diagrams
- Useful program short-cuts

... and much more. If you need more detailed information, please refer to the main **Automation** manual.

Videos at www.pcschematic.com

At our website www.pcschematic.com, you can watch videos on how to work with the program, and read more general articles on e.g. electrical documentation standards.

PC|SCHEMATIC forum and FAQ

At www.pcschematic.com you also have access to the PC|SCHEMATIC forum, where you can exchange experiences with other PC|SCHEMATIC users.

At the forum you can also see a FAQ containing the most commonly asked questions for our support service.



INSTALLING THE PROGRAM

SEE HOW TO INSTALL THE PROGRAM IN THE INSTALLATION GUIDE.

CONTENTS

This **Quick start** manual is a quick introduction to **Automation**:

THE WORK AREA (P. 5)

Here you get a quick survey of the options in the **Automation** work area, once you have started the program.

ON PROJECTS AND PAGES (P. 9)

Here you get a survey of what an **Automation** project is, as well as how to insert, move, copy and delete project pages.

DRAWING DIAGRAMS (P. 15)

Here you get a survey of how to draw diagrams in **Automation**. For instance you can see how to draw lines and circles, and how to place texts and symbols. You also learn about the relation between components and symbols, and e.g. see how to navigate between the different symbols for a specific component.

SHORT-CUTS (P. 29)

At the last pages of the manual, you get a survey over various program short-cuts.

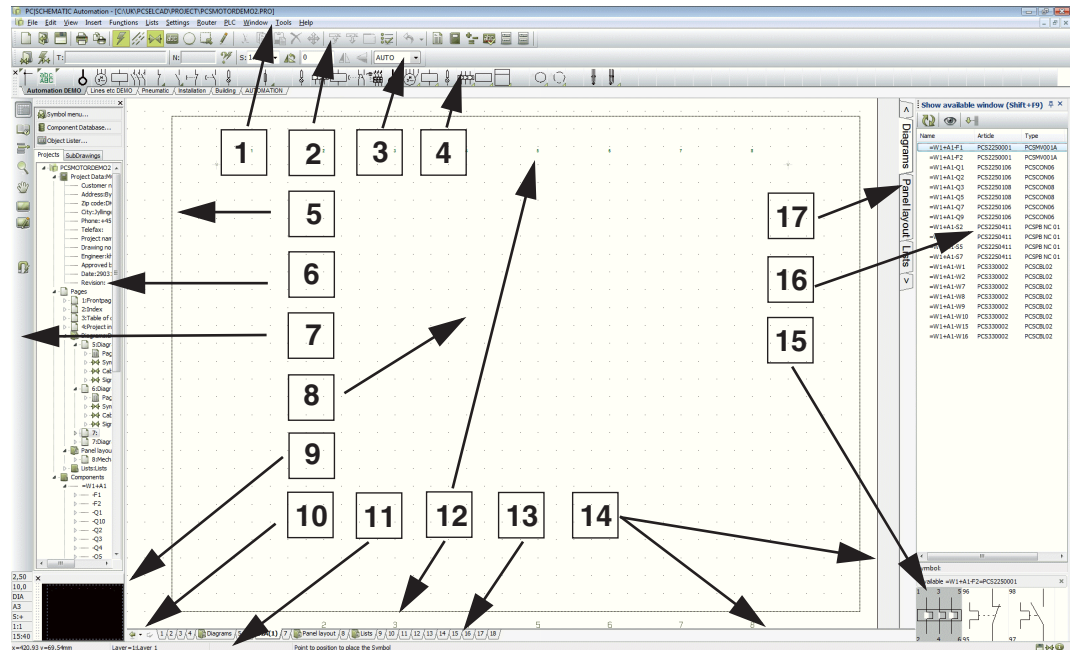


THE WORK AREA

Here you can see which options you have in the work area, once you have started the program.

THE WORK AREA

After starting **Automation**, choose **File => Open** and open the project **PCSmotordemo1.pro**. The project is placed in the folder **Pcscad\Project**. Go through the project by clicking the page tabs at the bottom of the screen (point 13 below). — The work area in **Automation** looks like this:



1. The menu bar

In the menu bar you find the menus, that contains all program functions.

2. The program toolbar

In the program toolbar you can choose the most common file and printing options, as well as general drawing and editing tools.

3. The command toolbar

The command toolbar has a varying appearance depending on which object type is selected in the program tool bar. — See “Drawing diagrams” on page 15.

4. The pickmenu

Here you place the symbols, line types, text types and arc types you use the most, so that you can fetch them right away, and place directly in the diagram pages. By clicking on the tabs at the bottom of the pickmenu, you select which of the created pickmenus to display. — See “Symbols” on page 22.

5. The help frame

The help frame is *not* activated the first time you start the program. It displays the standard margin for the selected paper size. There is also a printer help frame, displaying your printer margins. Both help frames can be activated/deactivated easily.

6. The Explorer window

In the upper part of the **Explorer window** you have direct access to the **Symbol menu**, the **Component database** and the **Object lister**.

The **Explorer window** also contains two tabs: The **Projects** tab and the **SubDrawings** tab.

The Projects tab in the Explorer window

On the **Projects** tab you can get information about all open projects. You get a survey of the project pages, and a survey of all components on these pages. When for instance you click on an page symbol for a project page, the page is displayed on the screen.

The SubDrawings tab in the Explorer window

The **SubDrawings** tab is applied for handling — e.g. placing, editing and displaying — sub drawings. For instance you can *drag* sub drawings into the active project page, or create new sub drawings based on copied areas from the active project. — See “How to insert new pages” on page 13.

7. The left toolbar

The left toolbar contains various page and zoom functions, and — at the bottom — information about the page settings and status.

8. The work area

The work area on the screen corresponds to the size of paper you have chosen to work on. The size of the paper is specified in **Settings => Page setup**.

9. The survey window

The survey window gives you a survey of the entire page. The part of the page that you currently see on the screen (the *zoom*), is represented by the black box.

10. Page history buttons

By clicking on the **Page history** buttons, you can jump to the previously active pages in the project.

11. The status line

Among others you can see here coordinates, layer titles, guiding texts as well as various messages. When you let the mouse pointer rest over for instance a button on the screen, an explanatory text will appear here.

12. Current path numbers

Current path numbers are displayed in two different places: At the location you specify in your project, and on the lower part of the screen. When you zoom in on a part of a diagram, the current path numbers will still be displayed on the lower part of the screen. Thus you will always know where you are on the diagram.

13. Page tabs

Click on the page tabs to jump back and forth between the pages. — See “How to choose pages and chapters” on page 14.

14. Scrollbars

Having zoomed in on an area, you can move the area by *dragging* the scrollbars.

15. The Show available window

In the **Show available window** you see all components in the project with available symbols (unused functions). — See “The Show available window” on page 27.

16. Available symbols for the selected component

When you have selected a component in the **Show available window**, you can see the available symbols for the component here.

17. Chapter tabs

When you click on a chapter tab, you jump to the first page in the selected chapter. — See “How to choose pages and chapters” on page 14.



PROJECTS AND PAGES

Here you see how to create new projects, and how to insert new pages in projects:

- What is a project? (p. 11)
- How to insert and select pages (p. 13)
- How to move, copy and delete pages (p. 14)

On www.pcschematic.com/videos you can also watch videos, showing you more about projects and pages.

WHAT IS A PROJECT?

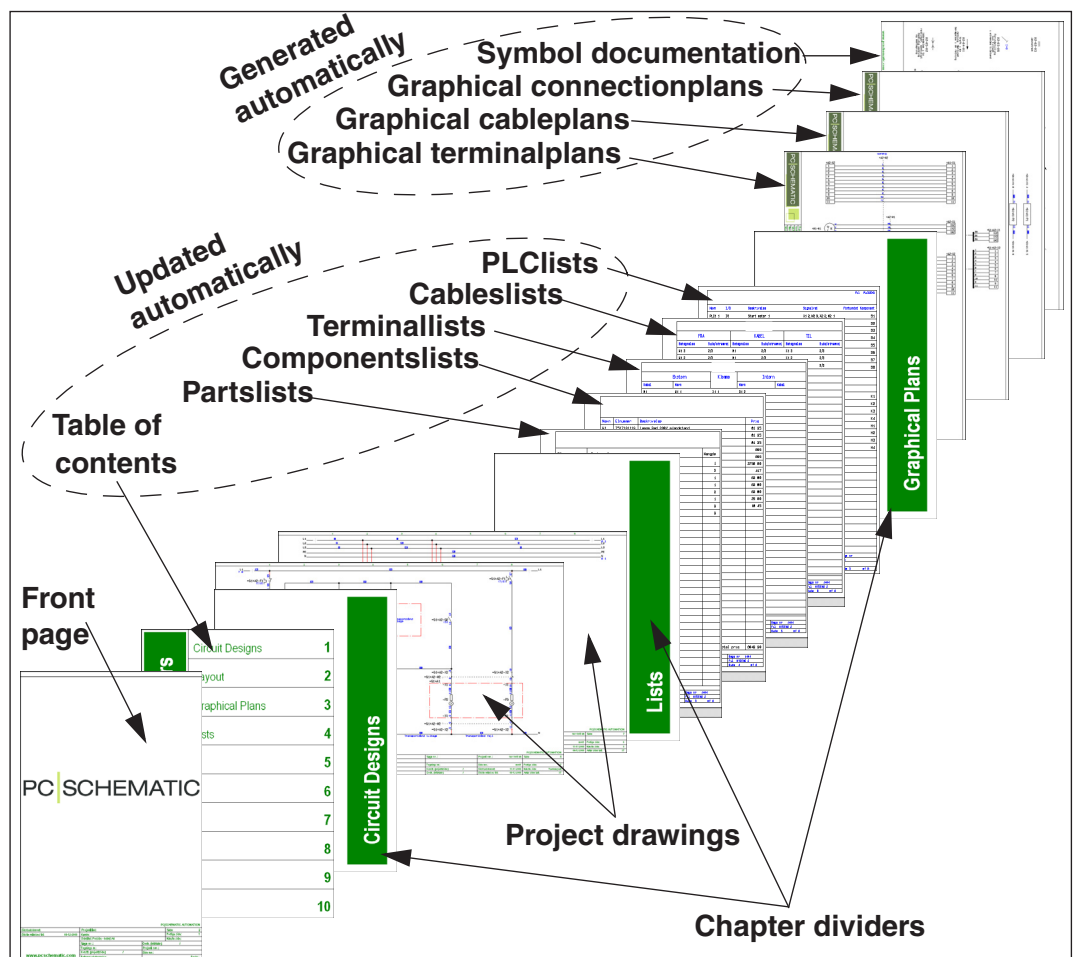
WHAT DOES "PROJECT ORIENTED" MEAN?

Automation is a *project oriented* program. This means that all the information you need in connection with a project, is gathered in one single file.

Therefore you do not need to change program, when you e.g. wish to generate bills of materials or detail drawings, because these are part of the same project file as the drawings.

WHAT DOES A PROJECT CONSIST OF?

A project can typically contain a front page, tables of contents, chapter front pages, pages with drawings, and pages containing different types of lists. Beyond this, a project can contain pages with mechanical symbols for the used components. All these parts are placed on their individual page/pages in the project.



SAMPLE PROJECTS

YOU CAN SEE SOME PROJECT EXAMPLES IN THE **PCSELCAD\PROJECT** FOLDER. IN SOME OF THE PROJECTS IN THIS FOLDER, THE PAGE TYPES ARE EXPLAINED FURTHER.

THE PROJECT DRAWINGS IS THE HEART OF THE PROJECT

The heart of your project is the *project drawings*. Here you draw the diagrams by placing symbols, lines and texts on one or more project pages. Here you also attach *article data* to the individual symbols in the diagrams. These article data can be fetched from the *component database*, and can then be transferred automatically to the project *lists*.



GRAPHICAL PLANS AND SYMBOL DOCUMENTATION

THESE PLANS CAN BE GENERATED AUTOMATICALLY:

- GRAPHICAL TERMINAL PLANS
- GRAPHICAL CABLE PLANS
- GRAPHICAL CONNECTION PLANS
- SYMBOL DOCUMENTATION

GET FURTHER DETAILS IN THE **PC|SCHEMATIC Tools** MANUAL. IN THIS MANUAL YOU CAN ALSO SEE HOW TO TRANSLATE PROJECT TEXTS USING THE **PC|SCHEMATIC TRANSLATOR** PROGRAM, AND HOW TO SET UP PARAMETERS FOR DWG/DXF IMPORT/EXPORT. THIS MANUAL CAN BE DOWNLOADED FROM WWW.PCSCHMATIC.COM/MANUALS.

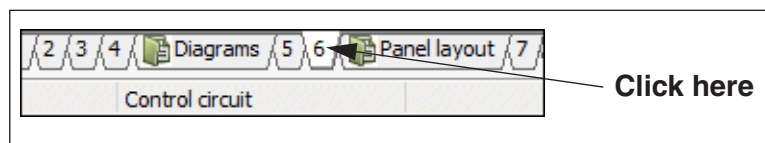
HOW TO INSERT NEW PAGES

There are many ways to insert new pages in **Automation**. Here is one of them:

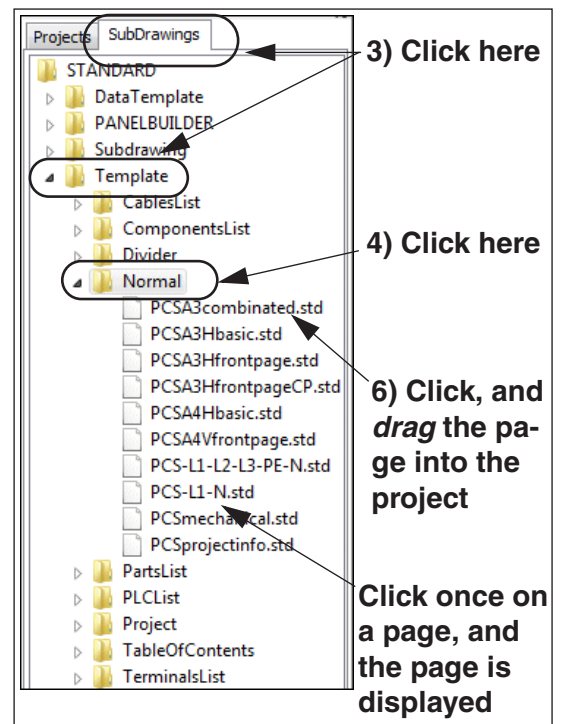
HOW TO DRAG A NEW PAGE INTO A PROJECT

Open the **PCSmotordemo1.pro** project, and insert a new diagram page after page **6**:

- 1) Choose **File => Open** and open the project **PCSmotordemo1.pro** (placed in the folder **Pcscad\Project**).
- 2) Click the pagetab for page **6** at the bottom of the screen, to specify *where* you wish to place the new project page:



- 3) In the **Explorer** window: Click the **SubDrawings** tab, and click the **Templates** folder. — Here you see all the page templates you can drag into the project. The templates are ordered in folders according to their function.



- 4) To insert a diagram page: Click the **Normal** folder, and all templates containing empty diagram pages are displayed.
- 5) Click on the pages to check what they look like. A short description is also displayed.
- 6) Click on **PCSA3Hbasic.std**, keep the mouse button pressed down, *drag* the page into the project, and release the mouse button.

- 7) The page is now inserted into the project. You can also insert a page by pressing <Ctrl>, while clicking the desired page.



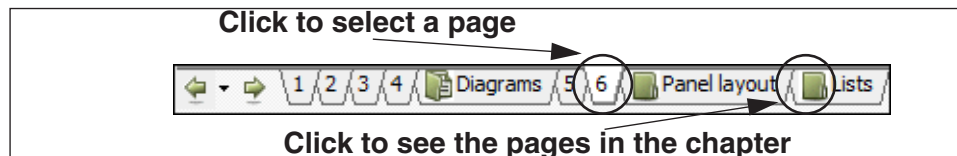
CHOOSE THE CORRECT PAGE TYPES

IF FOR INSTANCE YOU CREATE A LIST ON A DIAGRAM PAGE (FUNCTION **NORMAL**), THE LIST WILL NOT GET DATA FROM YOUR DRAWINGS AUTOMATICALLY.

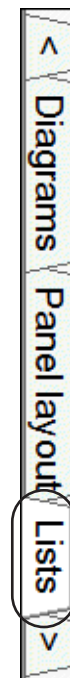
HOW TO CHOOSE PAGES AND CHAPTERS

— Open the project **Pcscad\Project\PCSmotordemo1.pro**, and do what you read, to experience it yourself.

You can change back and forth between the project pages by clicking the tabs on the lower part of the screen:



When you click the chapter tabs in the right-hand side of the screen, the first page in the selected chapter is displayed. You can also use the [Page Up] and the [Page Down] keys to jump back and forth between project pages. Also, you can choose **View => Go to page**, and type in a page number.

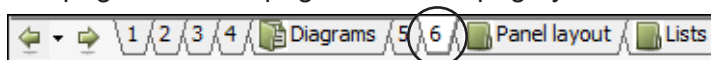


HOW TO MOVE, COPY AND DELETE PAGES

Below is just described one among many ways to move, delete and copy pages.

How to move pages using the page tabs

- 1) Select the page: Click the page tab for the page you wish to move.



- 2) Click the page tab again, keep the mouse button pressed down, and *drag* the page to the desired position. Release the mouse button.
- 3) Click **OK** to move the page.

If you have moved a chapter divider, you also are asked whether to move all pages in the chapter.

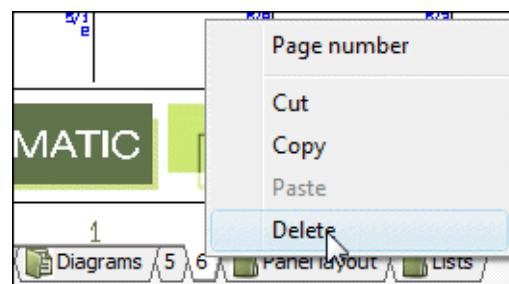
Copy pages and chapters using the page tabs

Pages and chapters are copied just like described above; the only difference is that you press the <Ctrl> key while doing it. When you copy pages/chapters, a small + is displayed in the cursor.

When you copy pages, you are asked whether/how to rename the symbols on the page — and specify reference designations (if any).

Delete pages via the page tabs

- 1) Right-click the page tab.
- 2) Choose **Delete**.





DRAWING DIAGRAMS

Here you can see how to draw diagrams.

You can read about:

- What are drawing objects (p. 17)
- Selecting, moving and deleting objects (p. 18)
- Drawing lines..... (p. 19)
- Drawing circles (p. 20)
- Placing texts (p. 21)
- Placing symbols (p. 22)
- Article data on symbols (p. 23)
- Deleting symbols..... (p. 25)
- Components with multiple symbols (p. 26)
- Available symbols for a component (p. 27)
- Navigating between symbols (p. 28)

DRAWING OBJECTS

WHAT IS A DRAWING OBJECT?

In **Automation** there are four different types of drawing objects: **Lines**, **Symbols**, **Texts** and **Circles**. Any object you place in a drawing, will always be one of these four types of drawing objects.

There is also the **Area** command, which makes it possible to work with all object types at the same time — e.g. when copying, deleting, moving or rotating objects.

HOW TO CHANGE OBJECT TYPE

When you wish to work with a specific type of drawing objects, start by clicking one of these buttons (short-cut keys in brackets):

Symbols [s]



Texts [t]



Lines [l]



Circles [c]



Areas [a]



WHEN YOU CLICK ONE OF THESE BUTTONS YOUR OPTIONS CHANGE:

- THE PROGRAM TOOLBAR CHANGES
- THE OPTIONS IN THE PROGRAM MENUS CHANGE
- YOU ARE ONLY ALLOWED TO WORK WITH OBJECTS OF THE CHOSEN TYPE

Auto command change



If you want **Automation** to change command for you automatically, click the **Auto command change** button, located on the left-hand side of the **Lines** button.

When this function is activated, **Automation** automatically switch to the object type of the selected object — e.g. **Lines**. The function is deactivated by clicking the **Auto command change** button again.

HOW TO WORK WITH DRAWING OBJECTS



In **Automation** there are two different modes to work in. Which mode you work in is determined by the **Draw** button — also called the **Pencil**. You either:

- a) Draw/place *new objects* (activated/highlighted **Pencil**), or you work on
- b) *Already placed objects* (deactivated Pencil) — e.g. when moving, copying, deleting or editing objects. The objects are selected by clicking them, or by clicking and dragging the mouse over the selected objects. — See “Selecting multiple objects” on page 18.

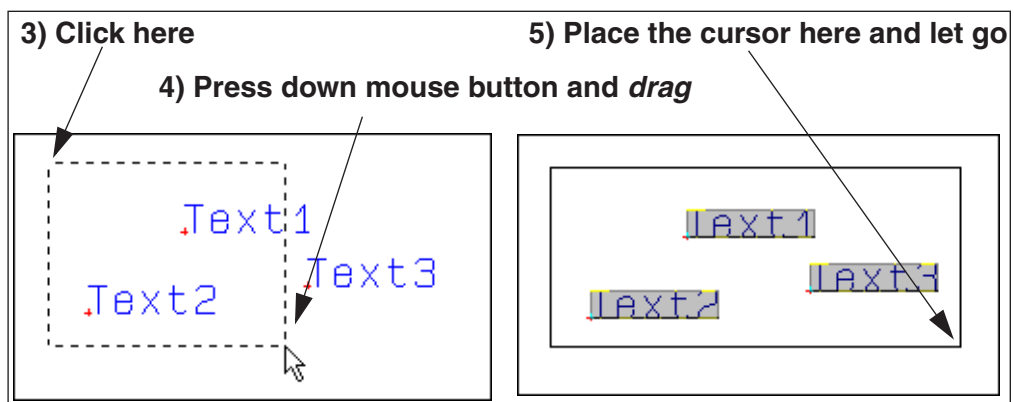
SELECTING MULTIPLE OBJECTS

— Start by opening the project **Pcselcad\Project\PCSmotordemo1.pro**, and try for yourself how to select objects etc. as described below.

When you work with already placed objects, start by selecting the objects:



- 1) Click the object type button — e.g. **Text**.
- 2) Make sure that the Pencil is not selected: Press <Esc>.
- 3) Click in one corner of the area you wish to select (see the figure below).
- 4) Still pressing down the mouse button, *drag* the mouse to the opposite corner of the area, until a rectangle is drawn around the objects you wish to select.
- 5) Release the mouse button. The texts in the area are now selected:



If you wish to *deselect* one or more objects inside the window, press the <Ctrl> key and click on the object(s). If you also wish to select objects outside the area, also press the <Ctrl> key, and click on these objects.

How to move, copy and edit objects



Having selected the objects, click the **Copy**, **Move** or **Delete** button. When you copy symbols, you will be asked whether to rename them, when placing the symbols.

Press <Esc> to remove the objects from the cross hairs.

You can undo your actions by clicking the **Undo** button, and redraw the page by clicking the **Redraw** button.

— See also “How to delete a symbol” on page 25.

HOW TO SELECT DIFFERENT OBJECT TYPES IN ONE OPERATION



To select multiple object types in a single operation, first click the **Area** button, and then continue as described above.

LINES

CONDUCTING AND NON CONDUCTING LINES

There are two different types of lines in **Automation**:

- a) Conducting (electrical) lines
- b) Non conducting / free lines

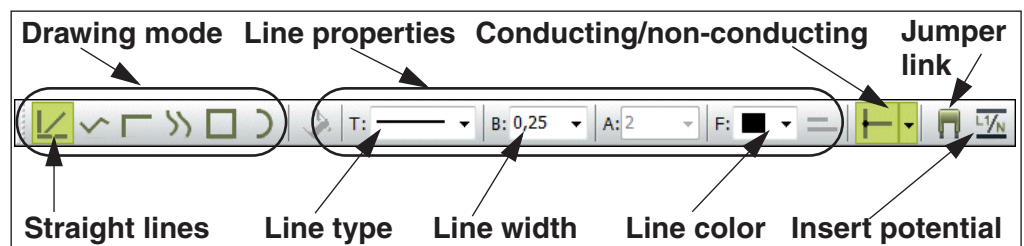


When **Conducting lines** is selected, the line will be understood as an electrical connection.

When it is *not* selected, the line will *not* be understood as an electrical connection.

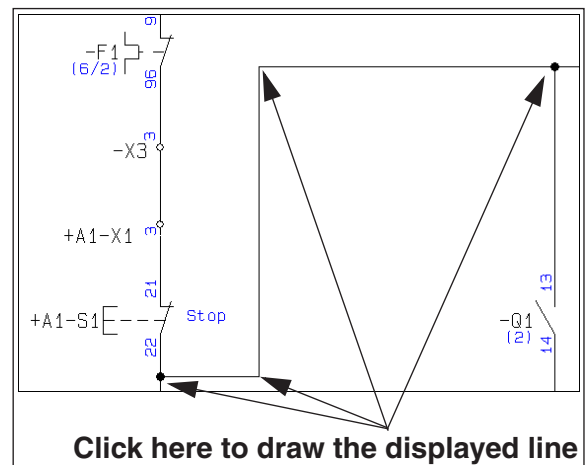
HOW TO DRAW LINES

- 1) Click the **Lines** button, or press the short-cut key [I].
- 2) The toolbar for lines appears:



- 3) To draw lines: Click the **Pencil** to select it.
- 4) Select the drawing mode— e.g. **Straight lines**.
- 5) Click where you want to start the line, and click once at each position where you want the line to change direction.
— Get more information about drawing conducting lines on the following page.

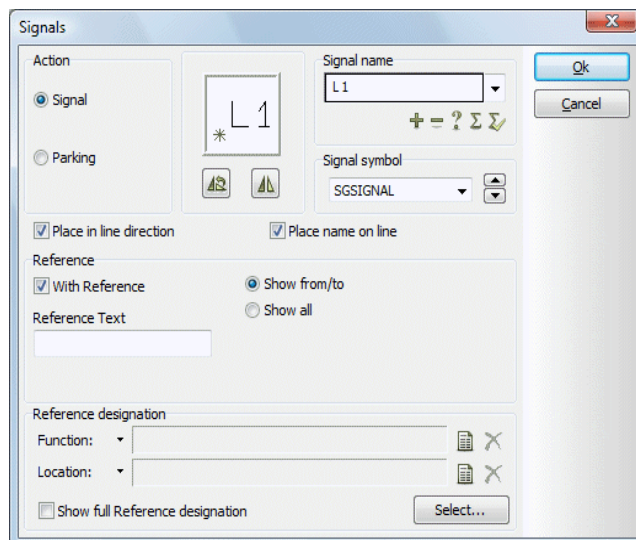
To stop drawing the line, press <Esc> to deactivate the Pencil.



SIGNAL NAMES AND CONDUCTING (ELECTRICAL) LINES



The first time you click on a page to draw **Conducting lines**, you enter the **Signals** dialog box:



This is because **electrical lines can only start and end in electrical points**. — An electrical point can be: Another electrical line, a connection point on a symbol or a *signal*.

When you start or end an electrical line without an electrical connection, you must therefore specify a *signal name*, or choose to *park* the line (no signal name).

When you do this, you can draw the line.

What are signal names used for?

The *signal name* for a *signal symbol* is used to specify an electrical connection to another signal symbol with the same signal name. This means that all electrical points with identical signal *names*, are the same electrical potential. — Signals are typically used for continuing an electrical potential on another project page.

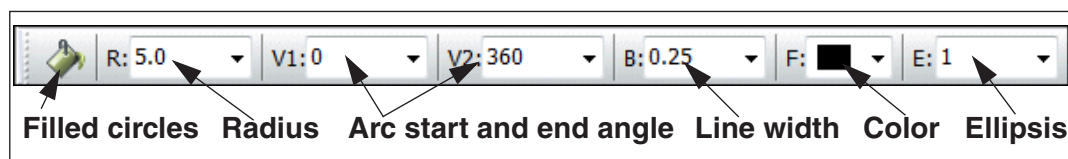
What is the meaning of parking a line?

If you want to wait to decide what to connect the conducting line to, choose **Parking**. Thus the line gets no electrical connection. However, this is just a temporary solution, and there should never be parked lines in a completed project.

ARCS/CIRCLES



To draw circles, click the **Circles** button and activate the **Pencil**. Circles have the short-cut key [c]. In the toolbar for circles, you have these options:

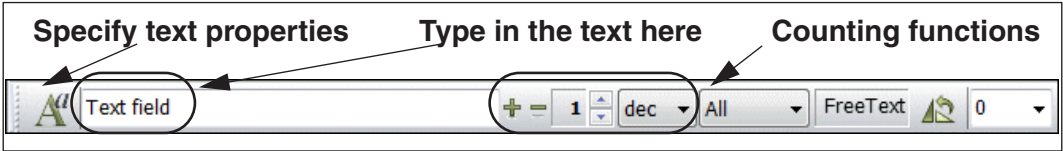


TEXTS

HOW TO PLACE A TEXT



- 1) Click the **Texts** button — or click the short-cut key [t].
- 2) The toolbar for texts is displayed:



- 3) Click in the text field, type in the text as displayed above and press <Enter>.
- 4) The text is now in the cross hairs. Click where you wish to place it.

HOW TO EDIT A TEXT



- 1) Click the **Texts** button, and disable the Pencil by pressing <Esc>.
- 2) Double-click on the text.
- 3) Edit the text and press <Enter> — or click **OK**.

TEXT TYPES IN Automation

There are various types of texts in **Automation**:

TEXT TYPE	DESCRIPTION
FREE TEXTS	TEXTS, THAT CAN BE USED ANYWHERE IN THE PROJECT.
SYMBOL TEXTS	TEXTS, THAT ARE ATTACHED TO EACH SYMBOL, CONTAINING INFORMATION ON THE COMPONENT THAT IS REPRESENTED BY THE SYMBOL. (NAME , TYPE , ARTICLE NO. AND FUNCTION).
CONNECTION POINT TEXTS	TEXTS, THAT ARE ATTACHED TO EACH OF A SYMBOLS CONNECTION POINTS. (NAME , FUNCTION , LABEL , DESCRIPTION).
DATAFIELDS	TEXT FIELDS THAT ARE FILLED OUT AUTOMATICALLY.

SYMBOLS

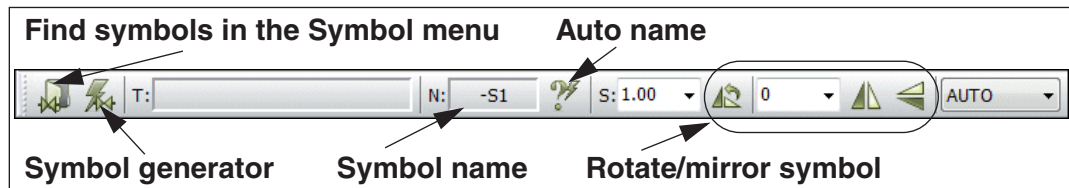
ON SYMBOLS

In **Automation** you use *symbols*, when you wish to represent a specific electrical function in a diagram. When for instance you wish to represent a lamp, you find the *symbol* for a lamp, and place it in the diagram.
Components with multiple electrical functions can have more than one symbols attached — see “Components” on page 26.

THE SYMBOLS TOOLBAR



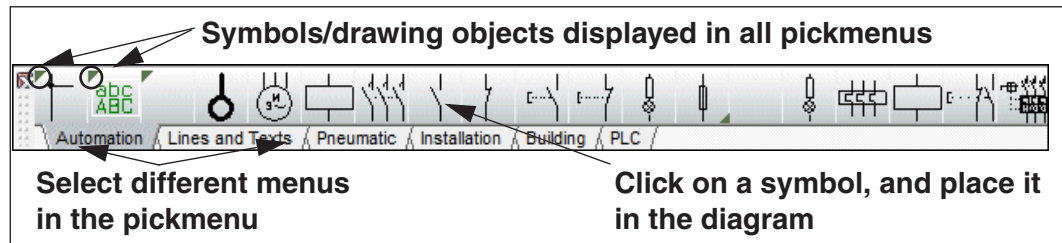
When you work with **Symbols**, you have the Symbols toolbar on the screen:



In the **Symbol menu** you can find the electrical symbols ordered in folders.
In the **Symbol generator** you can generate rectangular and circular symbols on the fly.

USING THE PICKMENU

When working in **Automation**, a **Pickmenu** is displayed on the upper part of the screen. Here you place the symbols — and other drawing objects — you use the most.

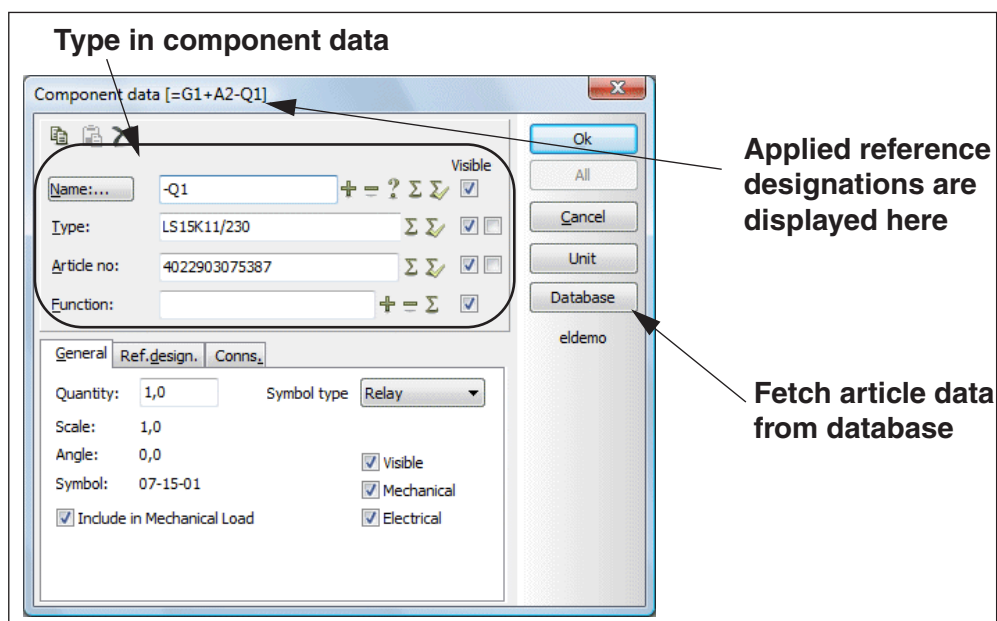


HOW TO PLACE A SYMBOL FROM THE PICKMENU



- 1) Click on the desired symbol in the pickmenu. The program changes automatically to work with **Symbols**, and the **Pencil** is activated.
- 2) The symbol is now in the cross hairs: Click where you wish to place the symbol.

3) You now enter the dialog box **Component data**:



4) Type in component name, type, article number and/or function description, and click **OK**.

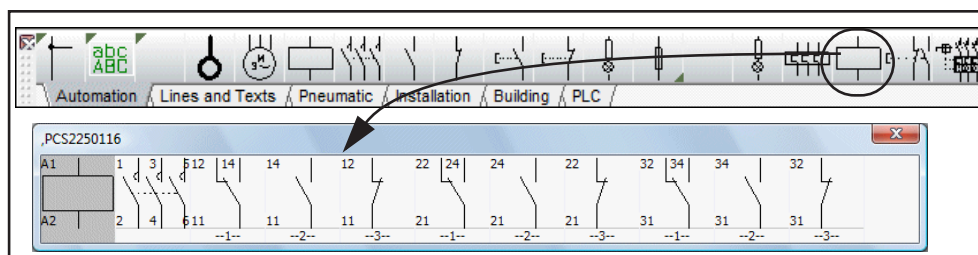
5) The symbol is now placed in the diagram, and have attached article data.

Press <Esc> to remove the symbol from the cross hairs.

SYMBOLS WITH COMPONENT DATA IN THE PICKMENU

The symbols in the pickmenu can be attached to components in the database. When this is the case, it is important to know:

- 1) When you place a such symbol in a project page, component data is automatically attached to the symbol.
- 2) When you click on a symbol/component that contains more than one function in the database, you get a component pickmenu containing the electrical symbols for the component.



3) When you click on one of the symbols and place it, the pickmenu disappears. When you press [Ctrl+F9] the component pickmenu is displayed again.

SELECTING COMPONENTS IN THE DATABASE VIA THE PICKMENU

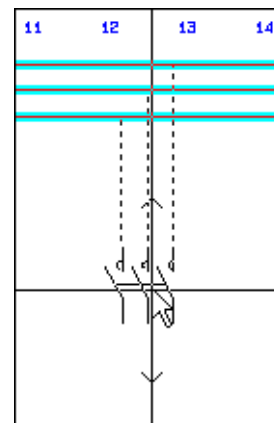
- 1) Press <Ctrl> while clicking on a specific symbol in the pickmenu.
- 2) You enter the **Database** menu: Here you see all components in the database, that contains this symbol.
- 3) Click on the desired component, click **OK**, and place the component symbol(s) in the project.

HOW TO USE THE ROUTER FOR SYMBOLS

When you place a symbol you can use the **Router** function, which draws the lines connecting the symbol to the lines automatically.

To use the router choose **Router => Active**, to set a check mark in front of **Active**. To disable the router choose **Router => Active** again, and the check mark disappears.

Please note, that pressing the spacebar changes the direction of the router.

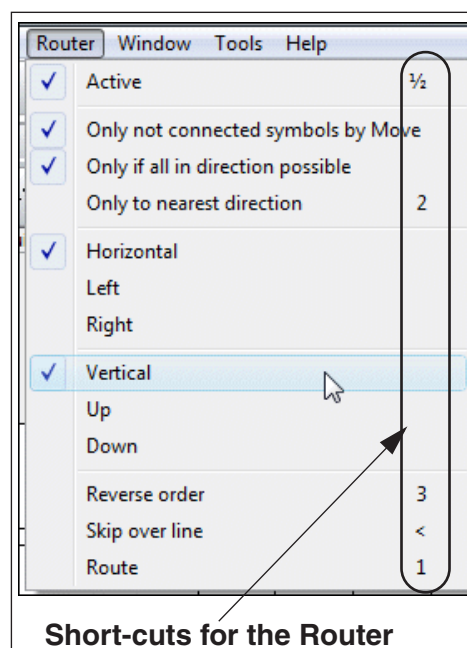


See the short-cuts in the Router menu

In the **Router** menu (see the figure), you can see which options you have in the Router, and see the short-cuts for the Router function.

When you become familiar with the short-cuts in the Router function, it becomes a truly powerful tool. You can define your own short-cuts in **Settings => Pointer/Screen => Shortcuts**.

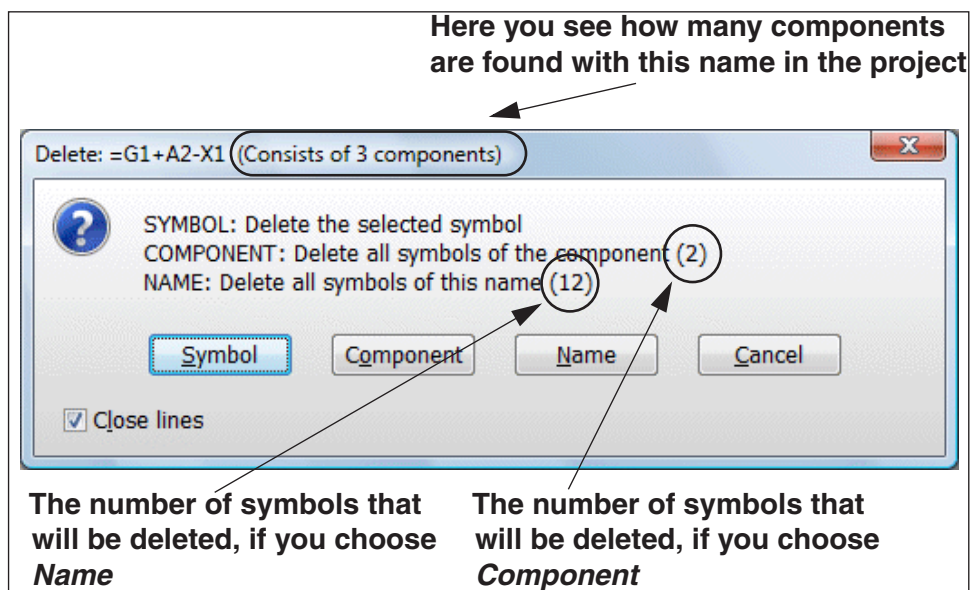
The Router can both be used when working with symbols and when working with lines.



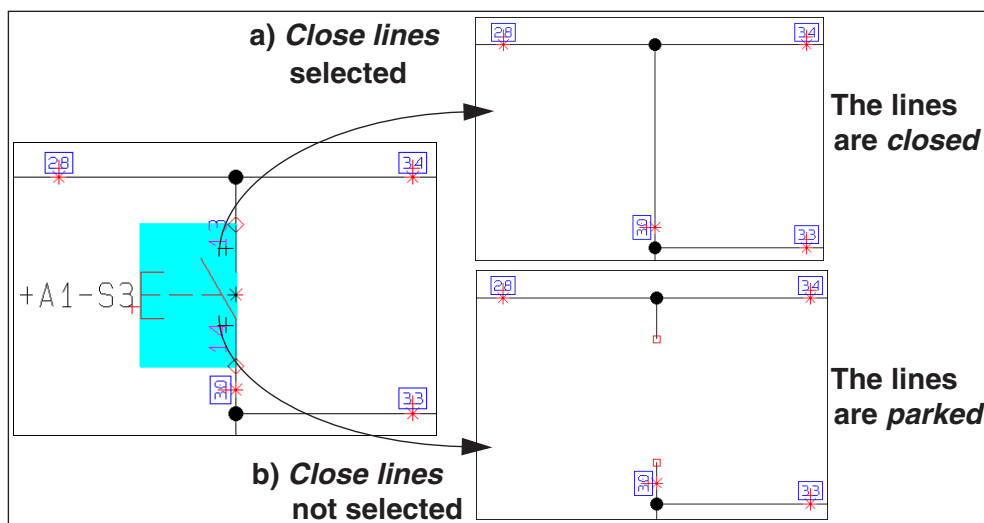
HOW TO DELETE A SYMBOL



- 1) Click the **Symbols** button, and press <Esc> to deselect the Pencil.
- 2) Click the **Delete** button, and the dialog box **Delete** is displayed:



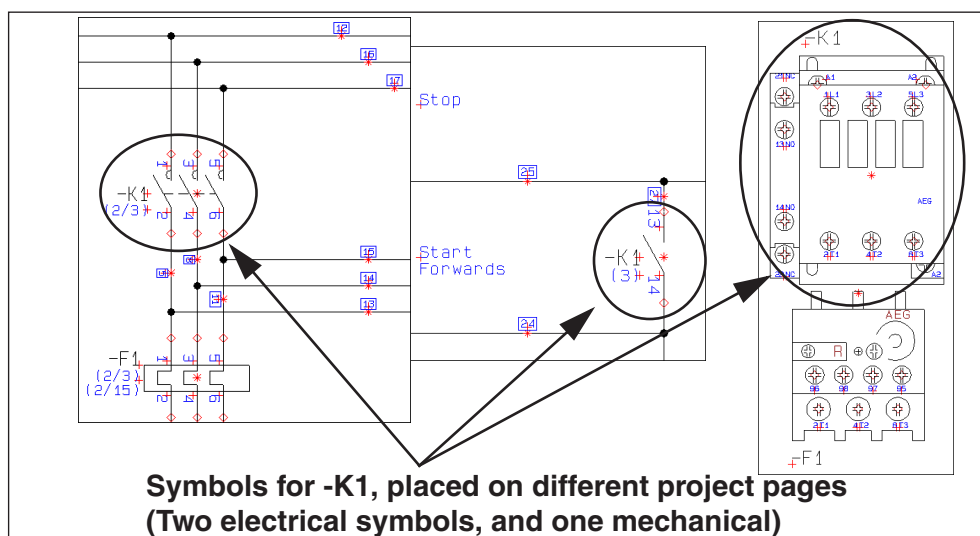
- 3) You now have the following options:
 - Click on **Symbol**: Only the selected symbol is deleted.
 - Click on **Component**: All symbols for the selected component is deleted (in the figure above, you can see that the physical terminal, from which the symbol is selected, consists of two symbols).
 - Click on **Name**: All symbols with this name are deleted (the terminal row in the figure above consists of 3 physical components, represented by 12 symbols in the project).
 - Click **Cancel**: No action is taken.
- If you select **Close lines**, the lines are closed for all deleted symbols — whenever this is possible. If you do not select **Close lines**, the lines are parked.



COMPONENTS

THE RELATIONSHIP BETWEEN SYMBOLS AND COMPONENTS

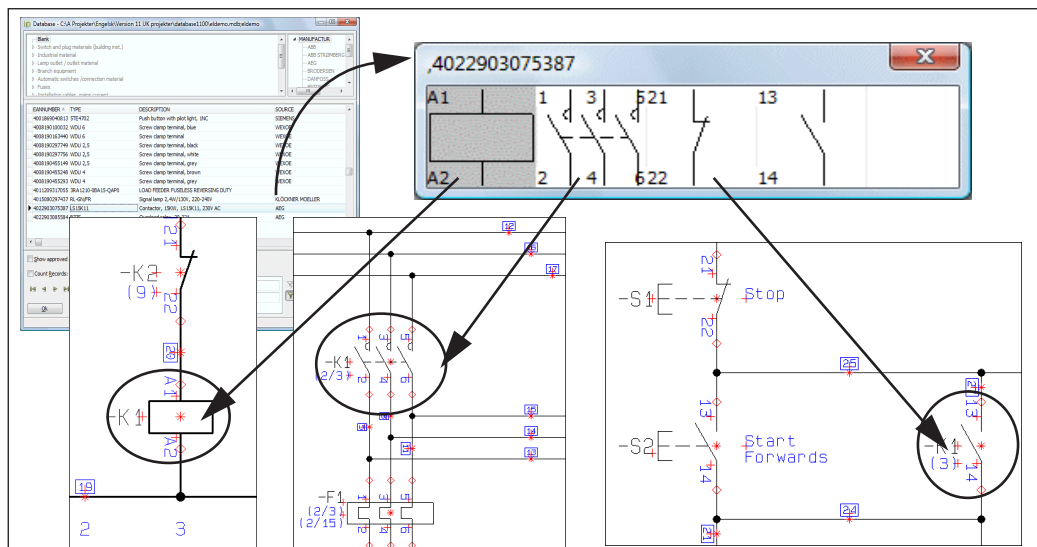
- 1) A *symbol* represents an electrical function in a diagram.
- 2) A *component* is one or more symbols, having a specific type and article number. When a component has multiple electrical functions, the component has an attached symbol for each function. To specify that two symbols are part of the same component, the symbols must have the same *component name*.



- 3) When you type in the same symbol name for two electrical symbols in a project, this specifies that they are part of the same component.

When choosing components via the database

When you choose a component via the database, you automatically get its electrical symbols on the screen, and can place these in the diagrams.

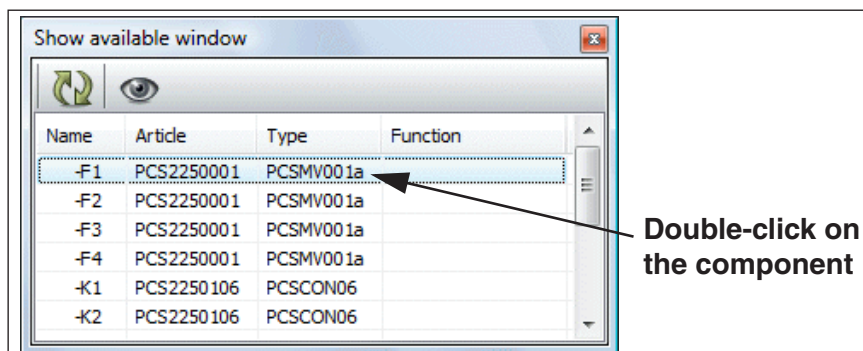


HOW TO SELECT AN AVAILABLE SYMBOL FROM A COMPONENT

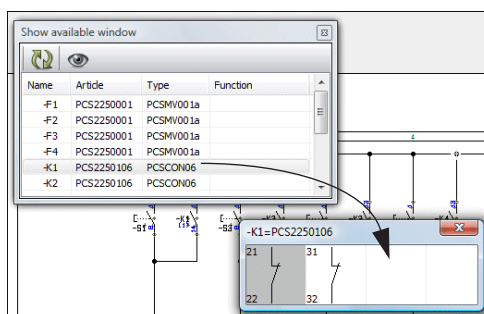
Because **Automation** intelligently understands which symbols are parts of which components, you get a lot of benefits in your daily work.

When you use components from the database, you can for instance easily get a survey over all project components still having unused/available symbols:

- 1) Choose **View => Show available window** to display all components in the project having available functions/symbols:



- 2) Double-click on a component in the window, and you get a pickmenu containing the available symbols for the component:



- 3) Click on the symbol you need, and place it in the diagram.

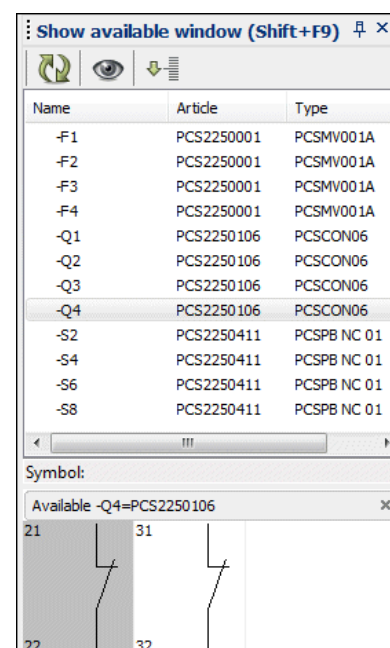
The Show available window

The **Show available window** is displayed permanently when you in **Settings => Pointer/Screen** have selected **Auto-activate Show avail. window**.

In the right-hand side of the work area, you then get an overview over all of the used components in the project, that have available symbols/functions.

When you click on a component in the list, you get access to its available functions/symbols at the bottom of the window.

When you are working on a **Diagram** page, all components with available *electrical* symbols are displayed. When you are working on a **Mechanical** page, all components with available *mechanical* symbols are displayed.



NAVIGATING BETWEEN SYMBOLS FOR THE SAME COMPONENT

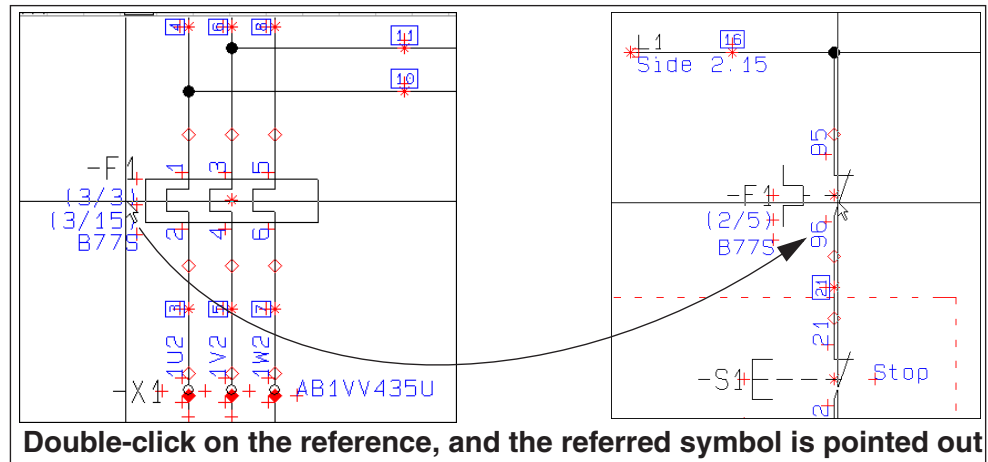
When you place more than one symbol for the same component in a project, the program automatically creates references between these symbols in the project pages. Next to each symbol you can therefore see references to the other symbols for the same component.

Lists can also be created so that you can jump directly from a component in a list, to the component in the diagram pages.

How to jump to another symbol for the component

To jump to a symbol which is referred to, do this:

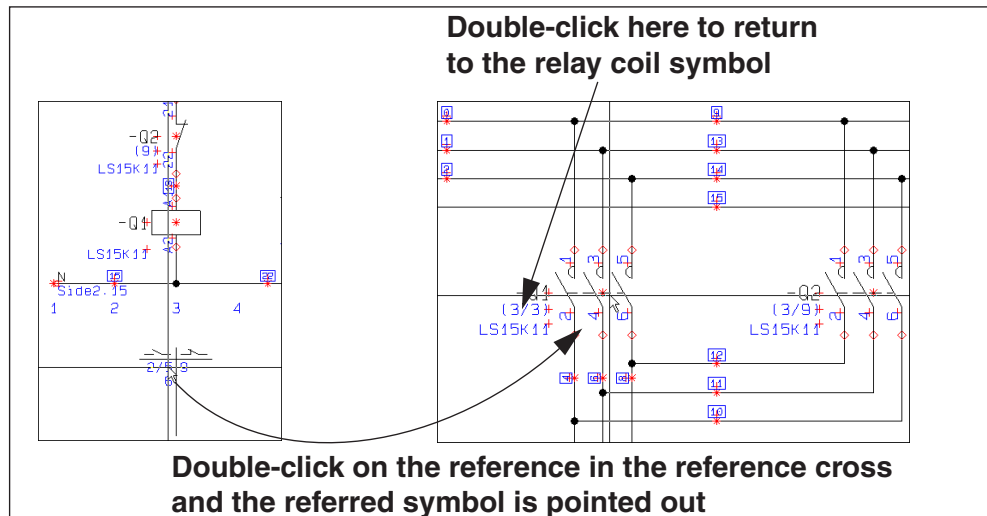
- 1) Double-click on the reference.



- 2) The program now automatically jumps to the project page, where the referred to symbol is placed. The referred symbol is pointed out by the cross hairs.

Automatic jump to reference from reference cross

You can also double-click on the reference in a reference cross/contact mirror:



When you double-click on the reference for a symbol in the reference cross/contact mirror, you return to the symbol having the symbol type **Relay**.



SHORT-CUTS

Here you can see which short-cut options you have in the program.

SHORT-CUT KEYS

In **Automation** you will find most commands accessible through short-cut keys. It is also possible to define your own short-cut keys in the program. To do this, choose **Settings => Pointer/screen => Shortcuts**.

FUNCTION	SHORT-CUT KEY	FUNCTION	SHORT-CUT KEY
AREAS	[a]	PRINT	[Ctrl+p]
CIRCLES / ARCS	[c]	REDRAW	[Ctrl+g]
COMPONENT GROUPING	[Ctrl+F7]	REPLACE	[Ctrl+h]
COORDINATES	[Ctrl+ i]	REPLACE SYMBOL	[F4]
COPY	[Ctrl+c]	ROTATE 90°	[spacebar]
DATABASE	[d]	SAVE PROJECT	[Ctrl+s]
DELETE		SELECT ALL	[Ctrl+a]
DRAW ON/OFF	<Insert>	SHOW AVAILABLE	[F9]
ELEVATION ON/OFF	[F11]	SHOW AVAILABLE AGAIN	[Ctrl+F9]
FIND	[Ctrl+f]	SLANTED LINES	[Ctrl+k]
FIND ARTICLE	[n]	SNAP, FINE	<Shift>
FIND NEXT	[F3]	STRAIGHT LINES	[Ctrl+r]
FIND TYPE	[v]	SURVEY WINDOW ON/OFF	[F12]
INSERT POTENTIAL	[p]	SYMBOL MENU	[F8]
KEYBOARD ENTRY	[k]	SYMBOLS	[s]
LINES	[l]	TEXTS	[t]
NEXT	[F5]	UNDO	[Ctrl+z]
NEXT PAGE	<PageDown>	VIEW ARTICLE DATA	[Ctrl+F2]
NEW DOCUMENT	[Ctrl+n]	VIEW CONDUCTING	[Shift+F2]
OBJECT LISTER	[F7]	WRITE TEXT	[k]
ON-LINE MANUAL	[F1]	ZOOM	[z]
OPEN	[Ctrl+o]	ZOOM IN	[Ctrl+Home]
PRECEDING	[F6]	ZOOM, MOVE / PAN	<Ctrl>+Arrow keys
PREVIOUS PAGE	<PageUp>	ZOOM OUT	[Ctrl+End]
PREVIOUS SELECTED PAGE	[F10]	ZOOM TO PAGE	<Home>

THE SHORT-CUT KEYS

Please note that using lower or upper case letters effects the short-cut function. Thus [Shift+c] will not activate the Circles command. Whether <CapsLock> is active makes no difference.

Right-clicking

To get further functions/options when working with the program, use right-click. For example you can right-click on a symbol, a line, a text, a circle, the pickmenu or on the entire page. Right-clicking in many cases causes a menu to appear, a menu containing functions and/or data dealing with whatever you are clicking on.

When in doubt, you can always try right-clicking.