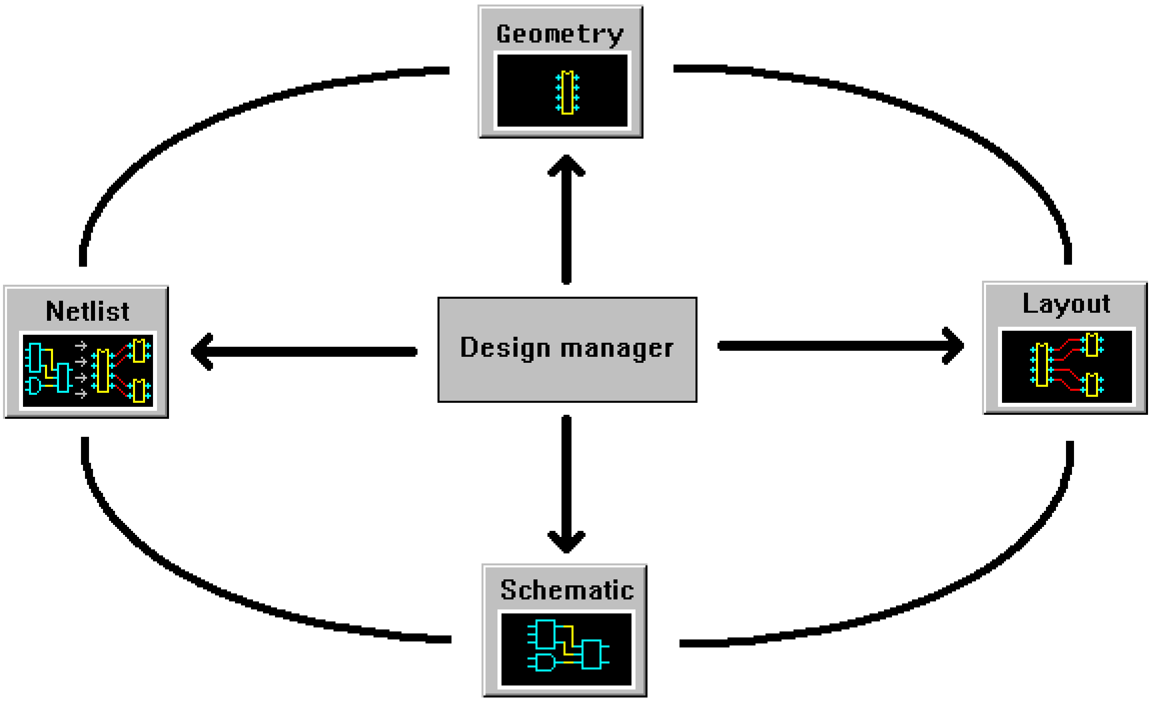
PCB elegance 2.0



Merco Electronics

PCB elegance

Version 2.0

© Copyright Merco Electronics

Information in this document is subject to change without notice. Companies, names, and data used in examples herein are fictitious unless otherwise noted.

No part of this document may be reproduced or transmitted in any or by any means, electronics or mechanical, for any purpose, without the express written permission of Merco Electronics

Merco electronics makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

Printing history

Version 1.2 January 1999
Version 1.3 January 2000

© Copyright January 2000 by

Merco Electronics
The Netherlands
http://www.merco.nl/

Table of contents

Introduction

With this software package it is possible to design a PCB (Printed Circuit Board). After designing the PCB output files can be generated, and a PCB manufacturer can make a PCB. The development of a PCB is divided into a number of steps. The first step is the creation of schematics. After the schematics are ready, annotation will follow. After annotation a netlist and components list will be made from the schematics. With this netlist and components list the Layout phase can be started. After the layout is ready, output files (gerber, drill data) can be generated. With these output files a PCB manufacturer can make a PCB.

Installation

Requirements

- Standard PC with a mouse
- CDROM drive
- Processor 486 or higher
- 16 Mb RAM (32 Mb preferred)
- 30 Mb harddisk space
- Operating system
    - Windows 95
    - Windows 98
    - Windows NT4.0
    - Windows 2000

Install

To install this software package on your harddisk, place the CD into the CDROM drive. After a few seconds installation starts. Normally the program will be installed in the directory **c:\pcb_elegance**, but this can be changed. After the install directory has been chosen, installation will continue and all files will be copied. All the files and directories need by this program are stored into the directory **c:\pcb_elegance** or the renamed directory. The **registry** of windows 95/98 or windows NT4.0, will not be used. Besides the install directory, no other directory is being used. There are also no DLL files copied to the windows directories, because DLL files are not used.

Installation on a network

Normally the executables directory is the same as the project directory containing the projects. However the project directory can be different. When executing the design manager (design.exe) a parameter (/pdirectory) can be specified for the project directory.

Example 1:

```
design.exe /pd:\projects
```

The project directory will be d:\projects

Example 2:

Set a environment variable **PCB_ELEG_ENVIRONMENT** to **d:\projects**.

```
design.exe
```

The project directory will be d:\projects

Example 3:

Set a environment variable **PCB_ELEG_ENVIRONMENT** to **d:\projects**.

```
design.exe /p%PCB_ELEG_ENVIRONMENT%\local
```

The project directory will be d:\projects\local

Example 4:

Set a environment variable **PCB_ELEG_ENVIRONMENT** to **d:\projects**.
The user directories environment variable **USER** is equal to **harry.**

```
design.exe /p%PCB_ELEG_ENVIRONMENT%\%USER%
```

The project directory will be d:\projects\harry

Deinstall

To deinstall this software package, run **uninstall.exe**. During uninstall all the directories/files in the directory **c:\pcb_elegance** or user defined directory will be deleted. Also the links in the **Start** menu will be deleted.

Design manager

Introduction

The design manager of PCB elegance is the central tool to start the schematic editor, geometry editor, and the layout editor.

How to make a simple PCB

In this chapter the making of a simple PCB will be described.

The making of the PCB will be divided into a number of steps.

Create a new project

Create a new project:

Action: Use the menu item **New design** from the design manager **File menu** to create a new project.

In the next dialogbox (window) some parameters must be entered.

Action: Fill in the first editbox (Design directory) your new project directory. For example **c:\pcb_elegance\simple**.
Action: Fill in the second editbox (Design name) the new of your project. This name can be the same name as the project directory. In this case **simple**.
Action: Fill in the third editbox (Top sheet name) the new of the schematic of this project. This name can be the same name as the project directory. In this case **simple**.

After clicking **OK** the new project will be created. In the directory **c:\pcb_elegance\simple** a number of files and directories will be created.

After creation of the project the schematic must be drawn.

Action: Click on the button **Schematic**.

After clicking this button the **Schematic editor** will be started with the schematic **simple.sch**. Inside this schematic we will import some symbols, and connect them with wires.

In this **simple** example we will use the following symbols/components:

- 74HCT14

- Resistor
- Capacitor
- Capacitor (Electrolytic)
- Two pins header
- Power terminal
- GND
- VCC

There are two methods to import symbols/components:

The first and direct method will import a component. A component is a symbol with all the required parameters (The required parameters are reference name, value name and the geometry).

The second method is to import symbols. After importing such a symbol, the value name and the geometry are empty and must be filled in later.

To demonstrate the two methods we import the TTL device, resistor, capacitor and the two pins header via the first method, and import the capacitor (Electrolytic), power terminal, GND and VCC via the second method.

Import the TTL symbol/component 74HCT14:

Action: Open the right mouse button by clicking on the right mouse button. A menu will be visible. Now select menu item
Add database component -> **IC** -> **7400 series** -> **74HCTxx**
In the next dialogbox (window) select the item 74HCT14.
After clicking on **OK** the symbol can be placed.

Action: The other symbols (resistor 10k, capacitor 100n and the two pins header) can be added similar.

Resistor 10k:

Action: **Add database component** -> **Resistor** -> **Through hole** -> **Pitch 5**
Select 10k

Capacitor 100n:

Action: **Add database component** -> **Capacitor** -> **Through hole** -> **Pitch 5**
Select 100n

Two pins header:

Action: **Add database component** -> **Passive** -> **Connectors** -> **Headers**
Select Header2

Importing symbols by using the second method

Action: Open the right mouse button by clicking on the right mouse button. A menu will be visible. Now select menu item

Add symbol. In the next dialogbox (window) select the item C:\pcb_elegance\sym in the top listbox.

All the symbols available in the directory C:\pcb_elegance\sym will
Be listed in the bottom listbox.

Action: Select the symbol ELCO (capacitor electrolytic). This symbol can now be placed.

Import the symbols GND,VCC,Power terminals (CON1) similar.

After placing the symbols, some parameters must be edited.

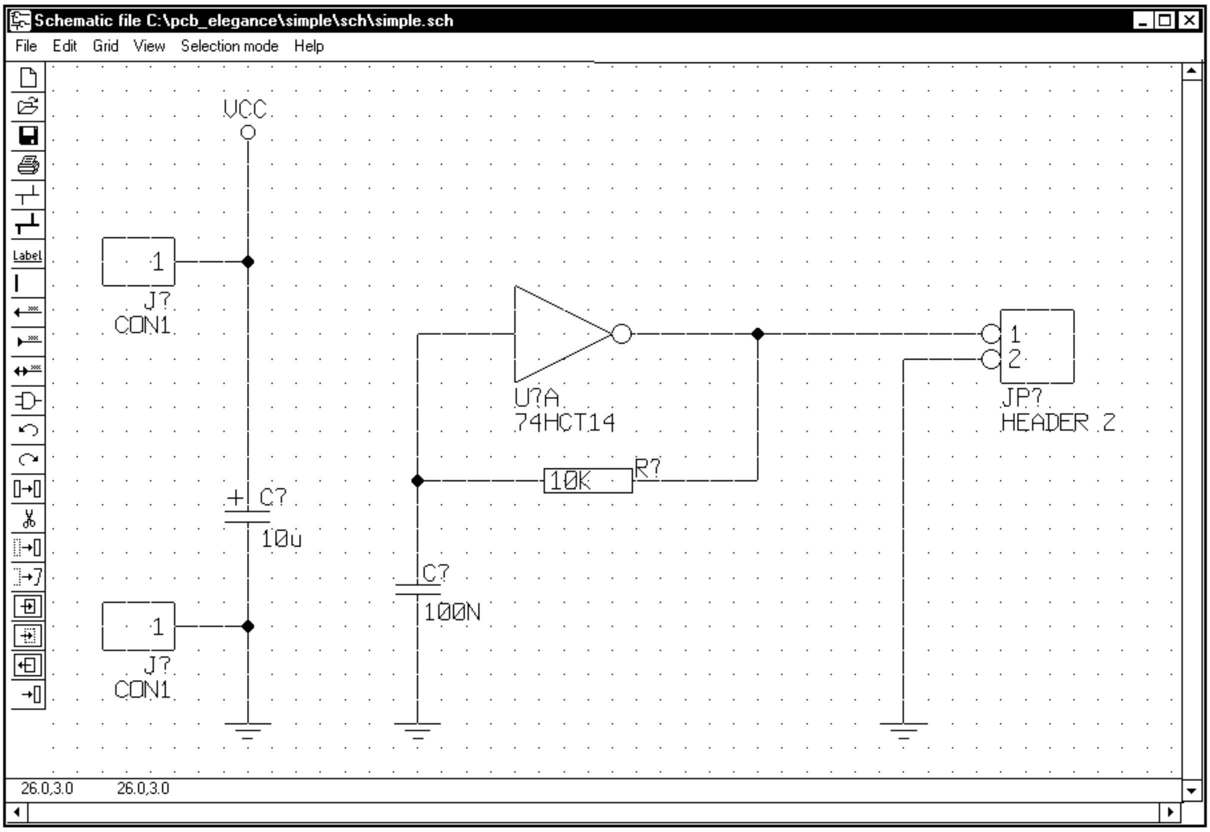
Action: Select the ELCO symbol.
Open the right mouse button by clicking on the right mouse button. A menu will be visible. Now select menu item
Edit text.

Action: In the next dialogbox (window) edit the value and geometry field. Fill the value with 10u, and to fill in the geometry field, click on **select geometry** button. There will be another dialogbox visible with all available geometries. Now select the geometry **elco1_rad6mm**.
A new window will popup which show this geometry.
Click on the **OK** button and the geometry will be placed into the geometry field.
Click on the **OK** button and the symbol parameters will be activated.

Action: Modify the parameters of the two power terminals similar, and use the geometry **point_1_1.**

The GND,VCC symbol do not need a geometry. After placing those symbols on the sheet, wires needs to be drawn to connect the symbols.

Action: Click on the wire button or press the key **w** to start a wire.
Draw the wires as shown in the figure below.



Action: The schematic is now ready, and can be saved by clicking on the save button.

Action: Exit the schematic drawing.

Annotate schematic

The schematic needs to be annotated.

Action: Click on the **Annotate** button in the design manager, and
Click on the **Restart annotation** button. Now click the **OK** button in the messagebox.

The schematic will now be annotated. Before annotation the reference of the resistor was **R?**, and after annotation **R100**.

Create netlist

The schematic is now ready, and the netlist should be made.

Action: Click on the **Netlist** button in the design manager.

Create layout

The netlist will be created, and will be used by layout editor.

Action: Click on the **Layout** button in the design manager.

The layout editor will be started. In the next dialogbox (window) the parameters of the new PCB should be filled in.

Change the following parameters:

Action:	PCB size	Width	3000 mil
		Height	3000 mil
	Design rules	Trace width	12 mil
		Clearance	12 mil
		Silkscreen	12 mil
	Via definition	Pad size	60 mil
		Drill diameter	40 mil
		Clearance	12 mil
		Solder mask	70 mil

Action: Click on the **OK** button, and the PCB will be made visible.

Import netlist

After creating the PCB the netlist must be imported.

Action: Use the menu item **Import components/netlist** from the **File** menu and choose **simple.net** in the open window to import the netlist.

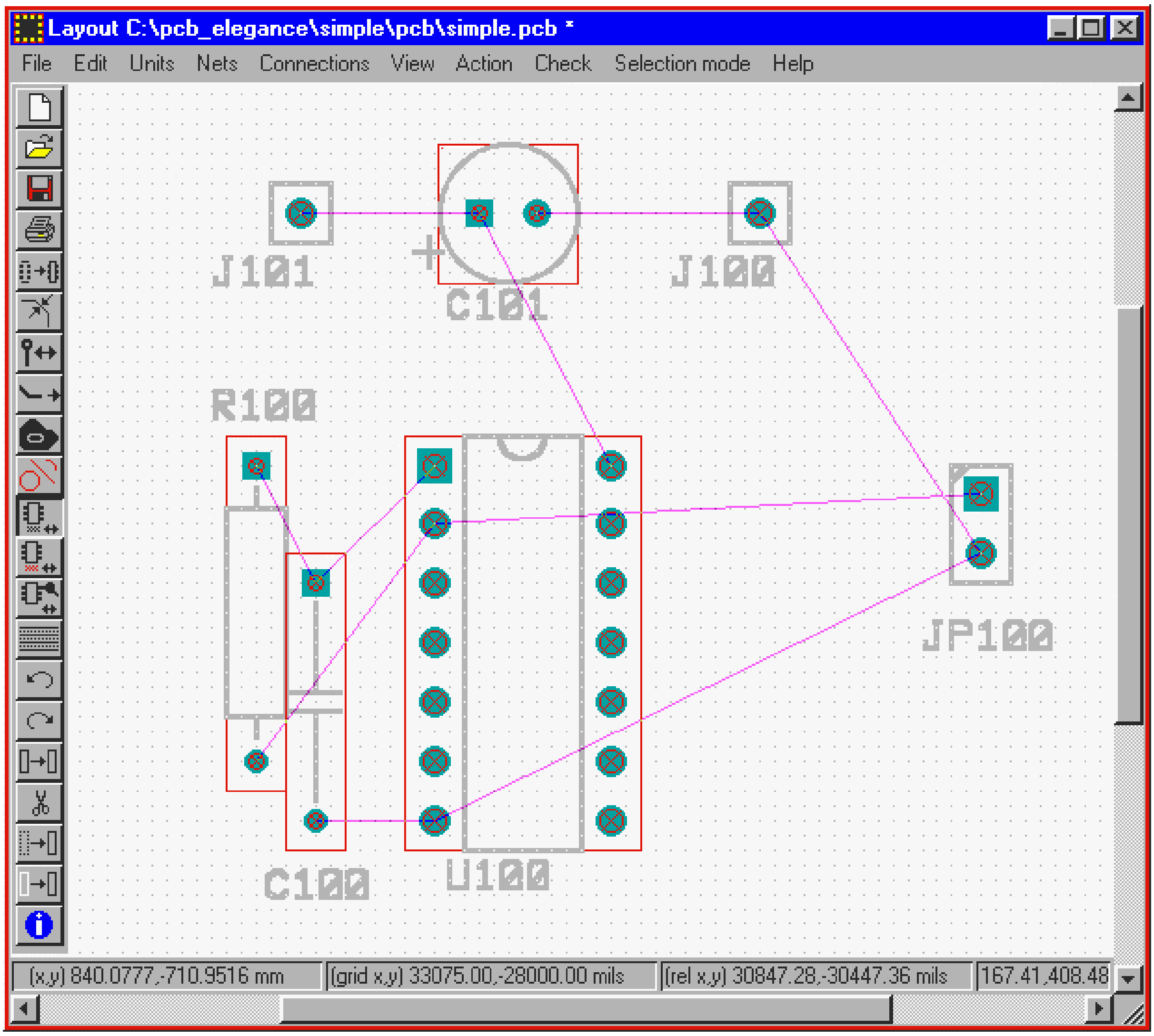
Place components

After importing the netlist a number of components should be placed around the PCB. The components should be moved inside the PCB.

Action: Select the component to be moved. To select a component place the mouse on a component and press the left mouse button. The component will be selected (White) and by pressing the key **m** or right mouse buttons menu item **Move** the component can be moved. When moving is active, and the right mouse button is pressed the component will rotate.

The green wires indicates which pins should be connected to each other.

Action: Move the components as shown in the figure below.



Route traces

After the components have been placed, traces must be routed (drawn). Before traces can be drawn the Trace menu should activated.

Action: The Trace menu can be activated by using the right mouse button menu (**Other menus** -> **Routing menu**), or by pressing the key **s**.

To start drawing a trace, click on a pad or green wire. When trace drawing is active, the current net pins will be marked. To end a trace place the mouse cursor in the neighborhood of a pin, and the trace will snap to that endpoint.
During trace drawing switching to the other can be done by using the right mouse button menu **Select layer** -> **Choose layer**.

Check PCB

After all traces have been drawn (No green lines visible anymore) the design must be checked for (design rule) errors, and connectivity errors.

Action: Use the menu item **Connectivity f**rom the **Check** menu to check the connectivity. If there are no connectivity errors a messagebox will be shown.

Action: Use the menu item **Design rule** -> **All layers f**rom the **Check** menu to check the design rules. If there are no design rule errors a messagebox will be shown.

The PCB is now ready and should be saved, by clicking on the save button.

Create output files

After finishing the PCB the output files should generated. There are three output options:

- Gerber output plots
- Penplot output
- Output to a printer

Generate gerber output plots:

Action: Use the menu item **Output gerber/drill** from the **File** menu to generate the gerber output plots. In the next dialogbox the layers can be selected, the gerber output format (RS274D or RS274X), X mirroring, plotting board outline and the gerber output number format can be selected. There are also two editboxes available. In those two editboxes (each four lines) some information about the PCB can be stored. This information will then be plotted additionally for each layer. After clicking the **OK** button the gerber files and drill file will be generated.

Generate penplot output:

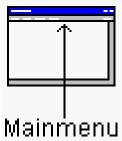
Action: Use the menu item **Output penplot** from the **File** menu to generate the penplot output. In the next dialogbox the layers, scale factor, pensize(s), origin, plotting board outline and mirror X can be selected. After pressing the **OK** button the plotfiles will be plotted to the files penplot.*. If possible drill holes will be left open, by shorten traces. After clicking the **OK** button the penplot files will be generated.

Output to a printer

Action: Use the menu item **Output penplot** from the **File** menu to generate the penplot output. In the next dialogbox the layers, scale factor, drawing board outline and mirror X can be selected. After clicking the **OK** button the plotfiles will be printed. (All the drill holes will be open)

File

Make new design

	Sub menu **File** menu item **New design**

In the next dialogbox the parameters of the new design can be edited.

In the **Design directory**, the new directory has to be filled in. In the **Design name** editbox the name of the design should be edited. Usually it is not necessary to change the next two editboxes: **Schematic symbol library** directory and **Geometry library directory**. In the **Top sheet name** editbox the first sheetname should be filled in. When all parameters are filled a new design will be created. The following directories/files in the **Design directory** are created.

File	<Design name>.dsn	Settings design
File	geom.ini	Settings geometry editor

The geom.ini file will be copied from the executables directory. (If exists)

File	sch.ini	Settings schematic editor

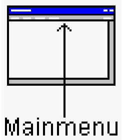
The sch.ini file will be copied from the executables directory. (If exists)

Directory	backup	
Directory	pcb	Layout file (*.pcb)
Directory	pcb\gerber	Gerber/drill output files (*.ger)
Directory	pcb\hpgl	HPGL output file (*.hgl)
Directory	pcb\backup	Backup layout files (*.pcb)
File	pcb\pcb.ini	Settings layout editor

The pcb.ini file will be copied from the executables directory. (If exists)

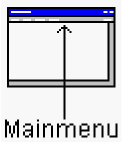
Directory	pcb\shapes	Local geometries file (*.shp)
Directory	pcb\shapes\backup	Backup local geometries files
Directory	sch	Schematic files (*.sch)
Directory	sch\backup	Backup schematic files
File	sch\<top sheet name>.sch	
Directory	sym	Local (sheet)symbol files
Directory	sym\backup	Backup sym files

Open

	Sub menu **File** menu item **Open design**

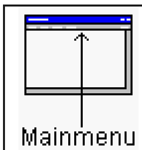
Opens a design (.dsn). If there was already a design open, this design will be closed. The name of the design will be visible in the window titlebar.

Close

	Sub menu **File** menu item **Close design**

Closes the current design.

Copy symbols/geometries locally

	Sub menu **File** menu item **Copy symbols/geometries locally**

All the symbols and geometries used in this project will be copied to the local directories in the project. This can be useful to interchange a project with someone else, because the whole directory can be copied, inclusive all symbols and geometries.

Directory structure design

Root directory design

<Design name>.dsn	Design settings
sch.ini	Settings schematic editor
net.nr	Net numbers
component.txt	Bill Of Materials output file
<Design name>.bom	Bill Of Materials output file

Subdirectory **Backup**

Previous Bill Of Materials output files.

Subdirectory **pcb** (layout subdirectory)

Subdirectory backup

- Backup previous version layout file
- Backup layout file before today (<Design name>.1)
- Backup previous netlist file

<Design name>.pcb	Layout file
<Design name>.net	Netlist file
<Design name>.neu	Neutral file (PCB testing)
pcb.ini	Settings layout editor
gatepin.swp	Gate/pin swap file
gatepin.ban	Gate/pin swap back annotation file
pos_mils.txt	Component position file (mils)
pos_inch.txt	Component position file (inch)
pos_mm.txt	Component position file (mm)

Subdirectory **pcb\gerber** (gerber output subdirectory)

<Design name>.drl	Drill output file
drills.txt	Drill tool file
drill.rck	Drill tool file (binary)
gerber.txt	Aperture file gerber output files
layers.txt	Layer info file
Top.ger	Gerber output file top (Component side)

Bottom.ger	Gerber output file bottom (Solder side)
Inner1.ger	Gerber output file inner layer 1
Inner2.ger	Gerber output file inner layer 2
SolderMaskTop.ger	Gerber output file solder mask top
SolderMaskBottom.ger	Gerber output file solder mask bottom
PasteMaskTop.ger	Gerber output file paste mask top
PasteMaskBottom.ger	Gerber output file paste mask bottom
SilkScreenTop.ger	Gerber output file silkscreen top
SilkScreenBottom.ger	Gerber output file silkscreen bottom
BoardOutline.ger	Gerber output file board outline
Info.ger	Gerber output file info layer
Info2.ger	Gerber output file info layer2

Subdirectory **pcb\hpgl** (hpgl subdirectory)

Top.hgl	Penplot output file top (Component side)
Bottom.hgl	Penplot output file bottom (Solder side)
Inner1.hgl	Penplot output file inner layer 1
Inner2.hgl	Penplot output file inner layer 2
SolderMaskTop.hgl	Penplot output file solder mask top
SolderMaskBottom.hgl	Penplot output file solder mask bottom
PasteMaskTop.hgl	Penplot output file paste mask top
PasteMaskBottom.hgl	Penplot output file paste mask bottom
SilkScreenTop.hgl	Penplot output file silkscreen top
SilkScreenBottom.hgl	Penplot output file silkscreen bottom
BoardOutline.hgl	Penplot output file board outline
Info.hgl	Penplot output file info layer
Info2.hgl	Penplot output file info layer2

Subdirectory **pcb\shapes** (local geometries subdirectory)

Subdirectory backup
Backup local geometry files

geom.ini Settings geometry editor

Subdirectory **sch** (schematic subdirectory)

Subdirectory backup
Backup previous version schematic files

*.sch files	Schematics
*.wir files	Link files for the layout editor

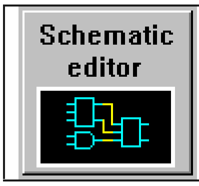
Subdirectory	**sym**	(local symbols subdirectory)

Subdirectory backup
Backup previous version local symbols files

*.sym Local (sheet)symbol files

Edit

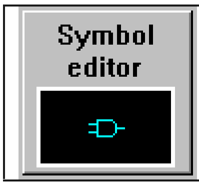
Open sheet

	Press **Schematic editor** button

Press this button with the **left mouse button**, and the **Schematic editor** will be executed with the topsheet

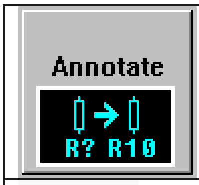
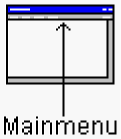
Press this button with the **right mouse button**, and the **Schematic editor** can be loaded with a sheet, selected from the pulldown menu or with a sheet containing a reference.

Edit symbol

	Press **Symbol editor** button

Press this button with the **left mouse button**, and the **Symbol editor** will be started.

Annotation

	Press **Annotation** button
	Sub menu **Edit** menu item **Annotation**

Annotation means numbering the component references who are named like (R?,C?,U?) automatically. After annotation this number replaced the quotation character of the component references. The numbering of component references is done per sheet. The Top sheet will start with the number 100. For example resistors will start with R100 etc. The next (sub)sheet will start with the number 200. If the top sheet contains for example more then 100 resistors, the next (sub)sheet will start with the number 300. If possible do not place more than 100 component references of each family (resistors, capacitors) on each sheet. Component references that do not have this quotation character will not change.

If there has been a small modification to a sheet, for example one resistor added annotation will proceed as follows: This new resistor with the component reference R? will be renamed. The number that will replace the quotation character will be the highest not used number on this sheet. This means that resistors (numbers) that have been deleted in an earlier stage will not be used again.

In the next dialogbox four annotation methods can be selected

Restart annotation (Standard numbering)

All the component references will be renumbered starting with one.

Restart annotation (Numbering per sheet)

All the component references will be renumbered, starting on a hundred per sheet. The resistors on the first sheet start with 100, and resistors on the seconds will start with 200, etc.

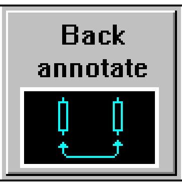
Appending annotation (Standard numbering)

In a existing design this annotation form will be used. Only the component references who are not numbered will get a new number. A new number means a number that has not be used before. Usually this will be a number greater than the highest number used.

Appending annotation (Numbering per sheet)

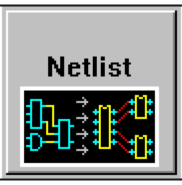
In a existing design this annotation form will be used. Only the component references who are not numbered will get a new number. A new number means a number that has not be used before. Usually this will be a number greater than the highest number used. On every sheet this new number will be the last number used plus one.

Back annotation

	Press **Back annotation** button

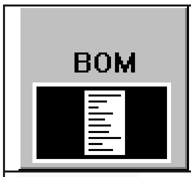
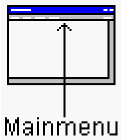
Back annotation means the changes made with the gate and pin swaps will be reflected into the schematics. When the **Back annotation** function is executed, the file **pcb\gatepin.ban** will be red, and the necessary schematics will be modified.
For example: The two pins of a 7400 ttl device have been swapped by using the layout editor. After **Back annotation** the wires connected to those two 7400 pins in the schematics will be switched.

Create netlist

	Press **Netlist** button

When this button is pressed the netlist will be calculated. This netlist consists of components, and the actual netlist. This netlist (.net) will be placed in the designs **pcb** subdirectory. Before making the netlist, the previous netlist file will copied to the **pcb\backup** subdirectory. Also the gate/pin swap info file **pcb\gatepin.swp** will be generated.

Bill of materials

	Press **BOM** button
	Sub menu **Edit** menu item **Bill Of Materials**

In the next dialogbox three Bill Of Materials methods can be selected The previous Bill Of Materials file will be copied to the **backup** subdirectory.

List of components

Every component will be listed on a line. Components in the sheets which are marked as a placing option, will marked with an asterisk '*' in this Bill Of Materials. The filename of this Bill Of Materials is **component.txt**

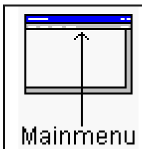
Bill of materials with component references listed

In this Bill Of Materials components are summed and listed. Components in the sheets, which are marked as a placing option, will not be included in this Bill Of Materials. The filename of this Bill Of Materials is **<design>.bom**

Bill of materials without component references

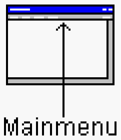
In this Bill Of Materials components are summed and listed, but the component references are not listed. Components in the sheets, which are marked as a placing option, will not be included in this Bill Of Materials. The filename of this Bill Of Materials is **<design>.bom**

Check

	Sub menu **Edit** menu item **Check schematics**

Checks the schematics of the design for errors.

Conversion ORCAD schematic/libraries

	Sub menu **File** menu item **Convert ORCAD schematic** **Convert ORCAD library**

With these two functions ORCAD schematics and libraries can be converted to PCB elegance. ORCAD schematics and libraries being used in the windows version are not supported.

Conversion ORCAD schematic:

1. After selection of the menu item the program continues with a dialogbox for the ORCAD schematic filename. Always select the top sheet, if there is a hierarchical structure for the schematics. The other sheets will be converted automatically.
2. In the next dialogbox some parameters can be entered. If the project directory does not exist it will be created. The schematic(s) and symbols used will be copied inside this project directory (**sym** directory).
3. A provision has been made to use the geometries/shapes names attached to the components. ORCAD uses partfields for specifying such geometries/shapes attributes. The partfield nummer can be selected in the dialogbox.
4. In most cases the geometries/shapes of ORCAD and PCB elegance do not match. In the “Geometry conversion file” field a filename can be placed, which will be used to translate the geometries.
5. This file consists of a number of lines. Each line contains two strings. The first string is the geometry name used in the ORCAD file, and the second string is the PCB elegance geometry.
6. Empty lines or lines starting with a ‘;’ will be ignored.
7. Now the conversion starts with reading the ORCAD configuration file **sdt.cfg**. This configuration file (Which libraries to be used) should be in the same directory as the schematic file. If the sdt.cfg file does not exist you should create this file, otherwise no symbol can be found.

Conversion ORCAD library:

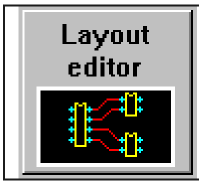
After selection of the menu item the program continues with a dialogbox for the ORCAD library filename. In the next dialogbox the directory of the converted library can be entered.

Orcad sdt.cfg example

```
{ OrCAD/SDT IV  Configuration File }
PDRV    = 'D:\ORCADESP\DRV\'
PSCH    = ''
PLIB    = 'D:\ORCADESP\SDT\LIBRARY\'
DD      = 'VGA640.DRV'
PRD     = ''
PLD     = 'HP.DRV'
LIB     = 'TTL.LIB'
LIB     = 'CMOS.LIB'
LIB     = 'DEVICE.LIB'
```

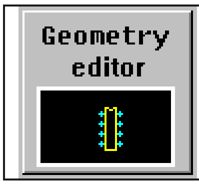
If necessary such a sdt.cfg file can be created. The lines containing the strings PLIB and LIB are important, and the first line for identification.
The PLIB string will contain the library directory, and the LIB strings the individual libraries in the library directory.

Start layout editor

	Press **Layout editor** button

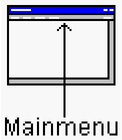
When this button is pressed the layout editor will be started with the layout of the current design.

Start geometry editor

	Press **Geometry editor** button

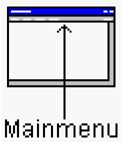
When this button is pressed the geometry editor will be started.

Start library manager symbols

	Sub menu **Library manager symbols**

Start the library manager for the schematic symbols.

Start library manager geometries

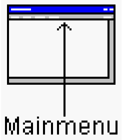
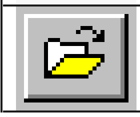
	Sub menu **Library manager geometries**

Start the library manager for the geometries.

Layout editor

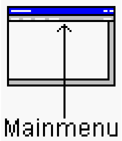
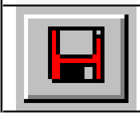
File

Open

	Sub menu **File** menu item **Open**
	Press **Open** button

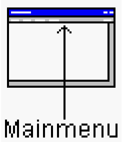
Opens a layout file (.pcb) from the current design directory. The geometries used in the layout file will be loaded first from the local pcb\shapes directory, the global geometries directory **shapes** or from the geometry libraries in directory **shplib**.

Save

	Sub menu **File** menu item **Save**
	Press **Save** button

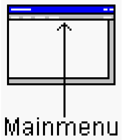
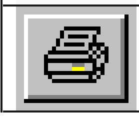
Saves the current layout file (.pcb) in the current **pcb** subdirectory.

Save as

	Sub menu **File** menu item **Save as**

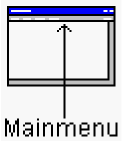
Saves the current layout file (.pcb) under another name.

Print screen

	Sub menu **File** menu item **Print screen**
	Press **Print** button

The current file will be printed. The scale will be adjusted to fit the page. The background color on paper will be white.

Make new layout

	Sub menu **File** menu item **New**

In the next dialogbox the parameters for a new design can be entered.
The width, height and origin of the PCB can be entered. The number of layers (Trace layer + powerplanes) can be entered. The standard via definition parameters can be entered. The standard design rules for trace width, clearance, linewidth silkscreen can be entered. Also the board outline keep out can be entered. If the board outline keepout is greater then zero, the design rule check will be increased with the board outline keepout check. This board outline keepout check will only be executed when the board outlines consists of closed objects.
When the **mils/mm** button is pressed the dimension (units) will switch between mils and mm, also every parameter will be recalculated for the new dimension.

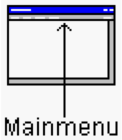
Design rules when using a printer for the plot outputs

When a standard printer (inkjet- or laserprinter) is being used, there are optimum trace/clearance widths for the most used printer resolutions.

Printer resolution	Trace width
300	12 mil
360	10 mil,14 mil

600	6 mil, 10 mil
720	6 mil, 8 mil, 10 mil
1200	5 mil, 8 mil, 10 mil

Importing components/netlist

	Sub menu **File** menu item **Importing components/netlist**

In the **File** menu the **Importing components/netlist**, function can be used to import components and the netlist in the design. Before importing the components, the whole design will be deleted (This can not be **undone**). After importing the netlist, the connections (Air lines, service lines, guide wires) of each net will be calculated (ratsnest).

Importing components/netlist

All the components starting with a reference name R (resistors), will be placed below the PCB. The R components with the smallest geometry will be placed first, just below the PCB. The R components with the next smallest geometry will be placed under the previous ones, etc.

All the components with starting with a reference name C (capacitors), will be placed right to the PCB. The C components with the smallest geometry will be placed first, just at the right of the PCB. The C components with the next smallest geometry will be placed to the right of the previous ones, etc.

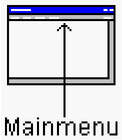
All the other components will be placed on top of the PCB, the smallest geometries first, and the greater the component geometries the higher they will be placed.

After all components are placed, the netlist will be read, and processed.

Calculating connections (ratsnest)

Calculating connections means, find the shortest connections between the pads of a net. Every found connection is visible by a line. When the net contains many pads, (>200) usually the power nets, a different calculation will be used. For those power nets, every connection will go to a central point below the PCB. The reason for doing this, is speed up calculations for those nets.

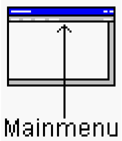
Updating components/netlist

	Sub menu **File** menu item **Updating components/netlist**

In the **File** menu the **Updating components/netlist**, function can be used to update components and netlist in the design. All Undo/Redo information will be lost, and also this update can not be **undone**.

For updating new components the same rules are used, as for importing component/netlist for a new design.

Plot output to gerber format

	Sub menu **File** menu item **Output gerber/drill**

Plot selected layers in the gerber format. In the next dialogbox the layers can be selected, the gerber output format (RS274D or RS274X), X mirroring, plotting board outline and the gerber output number format can be selected. There are also two editboxes available. In those two editboxes (each four lines) some information about the PCB can be stored. This information will then be plotted additionally for each layer. Initial three macros are stored into the first editbox.

$DesignName	Current design name
$Layer	Current layer
$Date	Current date

After pressing the **OK** button the plotfiles will be generated.
The possible plotfile names:

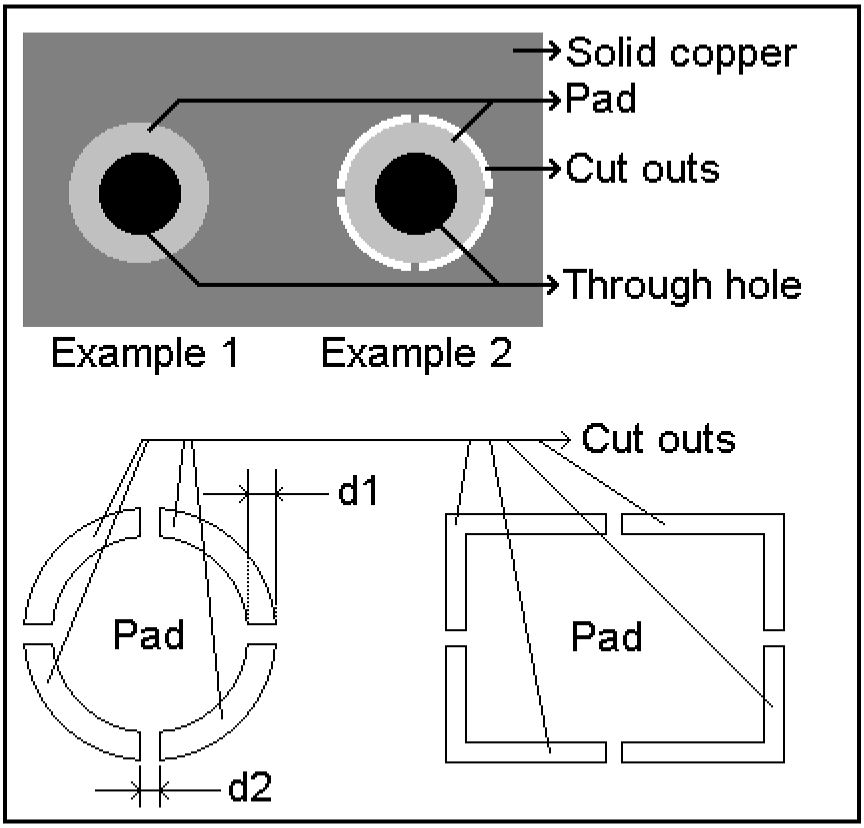
Top.ger	**Gerber output file top (Component side)**
Bottom.ger	**Gerber output file bottom (Solder side)**
Inner1.ger	**Gerber output file inner layer 1**
Inner2.ger	**Gerber output file inner layer 2**
SolderMaskTop.ger	**Gerber output file solder mask top**
SolderMaskBottom.ger	**Gerber output file solder mask bottom**
PasteMaskTop.ger	**Gerber output file paste mask top**
PasteMaskBottom.ger	**Gerber output file paste mask bottom**

SilkScreenTop.ger	**Gerber output file silkscreen top**
SilkScreenBottom.ger	**Gerber output file silkscreen bottom**
BoardOutline.ger	**Gerber output file board outline**
Info.ger	**Gerber output file info layer**
Info2.ger	**Gerber output file info layer2**

The aperture file (generated automatically) will have the name **gerber.txt.** The drill file (Excellon format) will have the name **<design>.drl** and the drill tool file the name **drills.txt**. The PCB information is stored in the **layers.txt** file.

All the files will be generated in the **pcb\gerber** subdirectory.

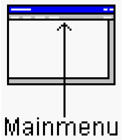
Thermal relief



In the above figure there are two examples. The first example is a through pole pin without thermal relief's, and the second example a through hole pin with thermal relief's. In the first example there will be problems when the through hole is soldered. Because the through hole pin and pad are fully surrounded with copper, and copper is a good thermal conductor, all the heat needed to solder the through hole properly will directly flow to the surrounded copper. In the second example there are four cut outs around the solder pad. Because the pad is not fully surrounded with copper, the through hole pin will solder properly. Thermal relief's are necessary when the diameter of the through hole is greater than 0.7 mm.

Normally thermal relief's are not necessary for vias, because the via hole will fill with solder without those thermal relief's.

Penplot output

	Sub menu **File** menu item Output **penplot**

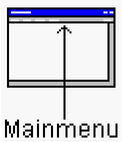
Penplots the **selected** layers to one or more files. In the next dialogbox the layers, scale factor, pensize(s), origin, plotting board outline and mirror X can be selected. After pressing the **OK** button the plotfiles will be plotted.

The possible plotfile names:

Top.hgl	**Penplot output file top (Component side)**
Bottom.hgl	**Penplot output file bottom (Solder side)**
Inner1.hgl	**Penplot output file inner layer 1**
Inner2.hgl	**Penplot output file inner layer 2**
SolderMaskTop.hgl	**Penplot output file solder mask top**
SolderMaskBottom.hgl	**Penplot output file solder mask bottom**
PasteMaskTop.hgl	**Penplot output file paste mask top**
PasteMaskBottom.hgl	**Penplot output file paste mask bottom**
SilkScreenTop.hgl	**Penplot output file silkscreen top**
SilkScreenBottom.hgl	**Penplot output file silkscreen bottom**
BoardOutline.hgl	**Penplot output file board outline**
Info.hgl	**Penplot output file info layer**
Info2.hgl	**Penplot output file info layer2**

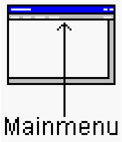
If possible drill holes will be left open, by shorten traces. All the files will be generated in the **pcb\hpgl** subdirectory.

Plot output to printer

	Sub menu **File** menu item **Print plotfiles**

Plots the **selected** layers to the printer. In the next dialogbox the layers, scale factor, drawing board outline and mirror X can be selected. After pressing the **OK** button the plotfiles will be printed.

Output component position

	Sub menu **File** menu item **Output component position** **Output component position (mils)** **Output component position (mm)** **Output component position (inch)**

Output component position (mils)

A file **pos_mils.txt** will be made, which contains the component position, rotation and layer. The component positions will be in **mils**.

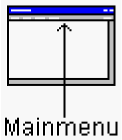
Output component position (inch)

A file **pos_inch.txt** will be made, which contains the component position, rotation and layer. The component positions will be in **inch**.

Output component position (mm)

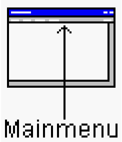
A file **pos_mm.txt** will be made, which contains the component position, rotation and layer. The component positions will be in **mm**.

Output netlist

	Sub menu **File** menu item **Output netlist**

A file **design.net** will be made, which contains the component- and netlist.

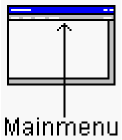
Reload geometries

	Sub menu **File** menu item **Reload geometries**

When geometries, used by the design has been changed by the geometry editor, the design geometries can be reloaded with this function.

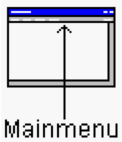
Edit

Move entire PCB

	Sub menu **Edit** menu item **Move entire PCB**

After pressing the **OK** button the entire PCB will moved with coordinates typed. **This operation can not be undone.**

Change design rules

	Sub menu **Edit** menu item **Change design rules**

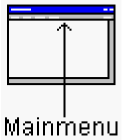
In the next dialogbox the design rules settings can be modified. **This operation can not be undone.**

Zero relative cursor

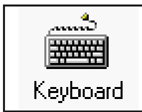
	Press **Ctrl z**

The relative cursor will be set to zero. On the window a white cross will mark the zero point.

With the next menu item function the relative position will be on the grid or not.

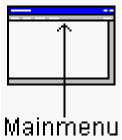
	Sub menu **View** menu item **Relative position on grid**

Center view on component

	Press **Ctrl c**

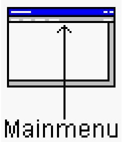
In the next dialogbox a component reference can be entered. After pressing **OK** the window will center on this component.

Via definition

	Sub menu **Edit** menu item **Via definition**

In the next dialogbox four via definitions can be entered. With the **Get via definitions** buttons the parameters will be loaded into the edit fields, and with **Set via definitions** buttons the parameters of the edit fields will be put into the via definition.

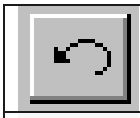
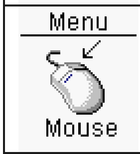
Component protection

	Sub menu **Edit** menu item **Component protection**

In the next dialogbox, all the components are displayed. The protected components are selected. By selecting or deselecting components can be protected/unprotected.

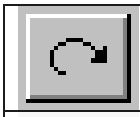
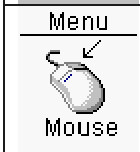
See also Protect components

Undo

	Press **Select layers** button
	Press **u**
	**Undo**

This function will undo almost all-previous actions.

Redo

	Press **Select layers** button
	**Redo**

This function will redo previous undo actions.

Selection/deselection objects

To select an object, place the mouse cursor above the object, and press and hold the **left mouse button**. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **Replacement mode**, and the second selection mode is the **Adding selection mode**.

The **Replacement selection mode** means, every time a new selection rectangle is drawed the previous objects selected will be unselected. When pressing down the **shift** key together with the **left mouse button** it is possible to use more than one selection at a time.

The other selection mode is the **Adding selection mode**. In this mode every object which is selected stays selected, until the deselect all function is executed. To deselect an object press the **left mouse button** and place the selection rectangle around this object again.

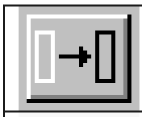
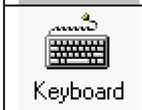
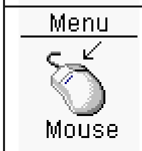
To change the selection mode use the **Replacement** or **Appending** in the **Selection mode** section of the menu.

Make selections in dialog listboxes

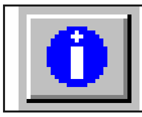
In dialogboxes with listboxes designed for multiple selections, are some features to easily select a huge number of items.

- By pressing and keeping down the **left mouse button** and moving the mouse cursor down or up, items can be selected. When more than one big selection is necessary, press and hold down the **Ctrl** key for every new selection range.
- To select/deselect an item press the **Ctrl** key and the **left mouse button**.
- For very large selections (>100 items) in series, select the first item with the **left** mouse button. Now scroll with the scrollbars or the **Page up/down** keys to the last item to be selected. Press the **shift** key and the **left mouse button**. All items between the first item and last item will be selected.

Deselect all

	**Unselect all** button
	Press **F2**
	**Deselect all**

Info on selected objects

	**Info selected objects** button

Keyboard	Press **i**
Menu Mouse	**Info**

Displays some information about **selected** objects.

View

Hide/view layers

	Select layers button
Keyboard	Press **Ctrl a**
Mainmenu	Sub menu **View** menu item **Layers**

Change visibility layers.

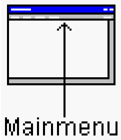
Change visibility layers. When the button **Areafill mode** is pressed a dialogbox will be visible. In this dialogbox the fill mode of areafills can be modified. The four options:

None : Only the areafill surround will be displayed
Hatch : The areafill will be hatched.
Solid : The areafill will be filled with layer color. The cut outs will be filled with the background color.
Gerber fill : The areafill will be a simulated gerber/penplot output.
Polygon surround : Areafill displayed with polygon surrounding
Horizontal fill : Areafill displayed with a horizontal line

The **hide/view layers** function can be used in every possible drawing/moving function.

See also Change grid

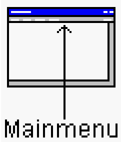
Zoom in

	Press **z**
	Sub menu **View** menu item **Zoom in**

The **zoom in** function can be used in every possible drawing/moving function.

See also Window based zooming

Zoom out

	Press **Z**
	Sub menu **View** menu item **Zoom out**

The **zoom out** function can be used in every possible drawing/moving function.

See also Window based zooming

Window based Zooming

To zoom in on a window, place the mouse cursor to the left top place of the window. Hold down the **Ctrl** key, than press and hold down the **left mouse button**. Move the mouse cursor in the right bottom direction of your window. After releasing the **Ctrl** key and the **left mouse button** zooming in will take place.
To zoom out, use the previous function, but now move the mouse cursor in the left top direction. The non-changing rectangle visible is the border of your design. The changing rectangle is the zoom out window. After releasing the **Ctrl** key and the **left mouse button**, zooming out will take place.

The **window based zooming** function can be used in every possible drawing/moving function.

Pan window

Keyboard	Press **⇐,⇒,⇑,⇓**
Keyboard	Press **x**
Keyboard	Press **Shift** and move the mouse the window border
Window	Use the scrollbars

When pressing the **x** key, the window will be panned around the current mouse position, and the mouse position will be moved to the window center.

The **pan window** function can be used in every possible drawing/moving function.

Window based panning

There is a function available to view a different part of your design (special window for panning). To enter this function, hold down the **Ctrl** key, than press and hold down the **right mouse button**. The non-moving rectangle visible is the border of your design. The moving rectangle is the viewable window. After releasing the **Ctrl** key and the **right mouse button** panning will take place.

The **Window based panning** function can be used in every possible drawing/moving function.

Return to previous view window

Keyboard	Press **v**
Menu Mouse	**Previous view**

Return to a previous view.

The **Previous view** function can be used in every possible drawing/moving function.

Repaint

Keyboard	Press **F5**
Menu Mouse	**Repaint**

The whole window will be repainted.

The **Repaint** function can be used in every possible drawing/moving function.

View whole design

Keyboard	Press **Shift F8**
Menu Mouse	**View whole design**

The window view will be scaled that the whole design will fit.

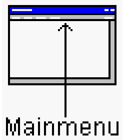
The **View whole design** function can be used in every possible drawing/moving function.

Change colors

Mainmenu	Sub menu **View** menu item **Change colors**

The color settings can be modified in the next dialogbox. The color settings will be copied into the **pcb.ini** initialization file. This file is stored into the **pcb** subdirectory of the project. To use those pcb colors for new designs, copy this **pcb.ini** file to main directory. Whenever a new design is created this **pcb.ini** file in the main directory will be copied to the **pcb** subdirectory of the new design.

Load default colors

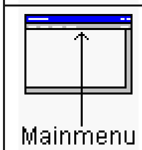
	Sub menu **View** menu item **Load default colors**

The default color settings will be loaded.

Programmable keys

The most important functions of the layout editor have a short cut key (Accelerator). Those keys can be modified by editing the **pcb.ini** file, section **[Keys]**.

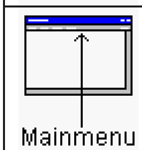
Change units

	Press **Ctrl u**
	Sub menu **units** menu item **Mm** **mils**

Changing the units (between mils/mm) is possible in every drawing/moving function.

See also Initialization file pcb.ini

Change grid

	Press **Ctrl g**
	Sub menu **View** menu item **Change grid**

In the dialogbox the grid settings can be modified. There are three main grid settings.
The **default grid** setting will be used normally.
The **grid when moving components** will be used when moving components is active, and this value is not zero.
The **grid when drawing traces** will be used when drawing traces is active, and this value is not zero.

When using the two above grid settings, it is possible to use different grid settings for moving components and drawing traces.

Changing the grid is possible in every drawing/moving function.
The grid settings in the dialogbox can be modified by changing the **pcb.ini** settings.

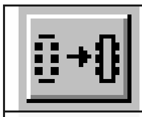
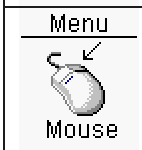
See also Initialization file pcb.ini

View/hide grid

	Press **g**

View/hide grid.

Components

	Press **Info selected objects** button
	Press **c**
	**Other menus** -> **Components menu**

The **Components** menu can be activated by one of three above actions. Activation of the **Components** menu is made visible on the info bar at the bottom right of the window, **Move/rotate/change components** is now visible. Also the **Move/rotate/change components** button is visible pressed.

Move components

Keyboard	Press **m**
Menu Mouse	**Move**

Move **selected** components. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Mouse	Press the **right mouse button**

Rotate **selected** components 90 degrees counter clock wise

Keyboard	Press **Space bar**

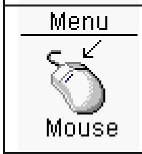
When the spacebar is pressed, the **selected** components can be moved to fixed or relative position, by typing the coordinates. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Recalculate ratsnets after move

Mainmenu	Sub menu **Connections** menu item **Recalculate ratsnets after move**

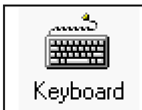
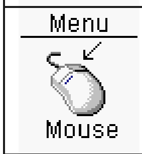
When moving/rotating components/traces/vias and this menu item is checked all the net connections involved in the current move will be recalculated.

Move components by reference

	Press **r**
	**Move component by reference**

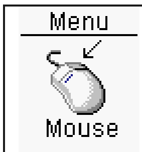
In the next dialogbox the component reference can be entered. After clicking on the **OK** button the component can be moved.

Rotate components

	Press **R**
	**Rotate**

Rotate **selected** components 90 degrees counter clockwise.

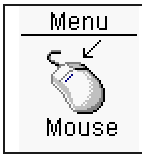
Move component to top/bottom layer

	**Move components to top layer** **Move components to bottom layer**

Move **selected** components to top/bottom layer.

Components on the bottom layer will have reversed component reference text.

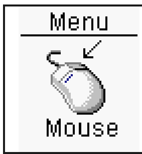
Regroup components

	**Regroup**

Regroup **Selected** components.

Regroup means, components moved close to each other. Close to each other means occupation of the smallest possible area.

Edit geometry

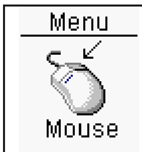
	**Edit geometry component**

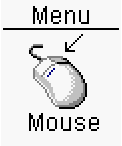
The geometry of the selected component can be edited with this function. After saving the geometry, the layout editor will automatically execute the **Reload geometries** function to update the PCB.

Change component parameters

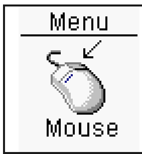
If the component references/values are visible, some parameters can be changed. The parameters are:

- Edit component value text
- Hide component reference/value text
- Unhide component reference/value text
- Move component reference/value text to top layer
- Move component reference/value text to bottom layer (Reversed text)
- Change textheight component reference/value
- Change text line width component reference/value

	**Component references** **Hide** **Visible** **On top layer** **On bottom layer** **Height** **Line width**

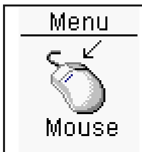
	**Component values** **Edit** **Hide** **Visible** **On top layer** **On bottom layer** **Height** **Line width**

Protect components

	**Protect**

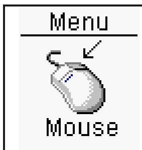
The **Selected** components will be protected. Protected components can **not** be selected. To unprotect components use the Component Protection function from the **Edit** menu.

Copy component placement outline to a different layer

	**Copy component placement outline to** -> **Layer**

Copies the component placement outlines of the selected components to the selected layer. The thickness will be the designs silkscreen width.

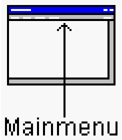
Copy component outline to a different layer

	**Copy component outline to** -> **Layer**

Copies the component outlines of the selected components to the selected layer.

Nets

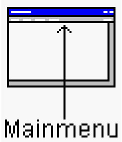
Change design rules net

	Sub menu **Nets** menu item **Design rules nets**

The design rules of one or more nets can be modified by this function. In the next dialogbox nets names can be selected, and in two edit boxes the trace width and clearance width can be edited. After pressing **OK** button the design rules for the selected nets will be modified. Already existing traces/vias of the selected nets will not be modified. The design rules will only be applied for new traces

See also Make selections in dialog listboxes

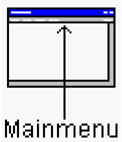
Highlight/unhighlight nets

	Sub menu **Nets** menu item **Highlight/unhighlight nets**

In the next dialogbox, all the nets are displayed. The highlighted nets are selected in the listbox. By selecting or deselecting, net traces/vias/connections can be highlighted or unhighlighted.

See also Make selections in dialog listboxes

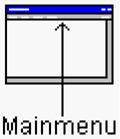
Disable connections nets

	Sub menu **Nets** menu item **Disable connections nets**

In the next dialogbox, all the nets are displayed. The disabled nets are selected in the listbox. By selecting or deselecting net connections can be disabled or activated. Disabled net connections are useful for the power nets, in combination with powerplanes, because when using powerplanes for power nets, there are many connections, and connections are not very useful.

See also Make selections in dialog listboxes

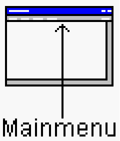
Hide connections nets

	Sub menu **Nets** menu item **Hide connections nets**

In the next dialogbox, all the nets are displayed. The hidden nets are selected in the listbox. By selecting or deselecting net connections can be hidden or made visible.

See also Make selections in dialog listboxes

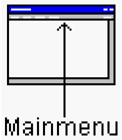
Highlight visible connections

	Sub menu **Nets** menu item **Highlight visible connections**

In the next dialogbox, all the nets are displayed. The hidden nets are selected in the listbox. By selecting or deselecting net connections can be highlighted or displayed normal.

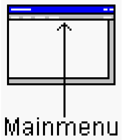
See also Make selections in dialog listboxes

Recalculate ratsnets after move

	Sub menu **Connection** menu item **Recalculate ratsnets after move**

When moving/rotating components/traces/vias and this menu item is checked all the net connections involved in the current move will be recalculated.

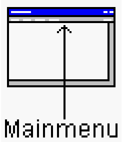
Unselect traces/vias nets

	Sub menu **Nets** menu item **Unselect traces/vias nets**

In the next dialogbox, all the nets are displayed. The nets of selected traces/vias are selected in the listbox. Traces/vias of the selected nets in the dialogbox will be deselected.

See also Make selections in dialog listboxes

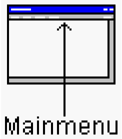
Delete traces/vias nets

	Sub menu **Nets** menu item **Delete traces/vias nets**

In the next dialogbox, all the nets are displayed. Traces/vias of the selected nets in the listbox will be deleted and connections will be recalculated.

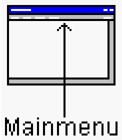
See also Make selections in dialog listboxes

Unhighlight all

	Sub menu **View** menu item **Unhighlight all**

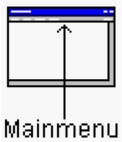
Traces/vias/connections will be unhighlighted.

View all connections

	Sub menu **Connections** menu item **View all connections**

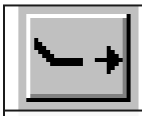
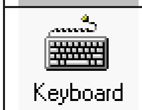
Hidden connections will be made visible.

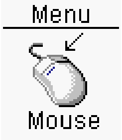
Hide all connections

	Sub menu **Connections** menu item **Hide all connections**

Visible connections will be hidden.

Routing

	**Routing traces** button
	Press **s** (For a maximum of three times)

	**Other menus** -> **Routing menu**

The **Routing** menu can be activated by one of three above actions.

When pressing the key **s** the default menu will switch between:

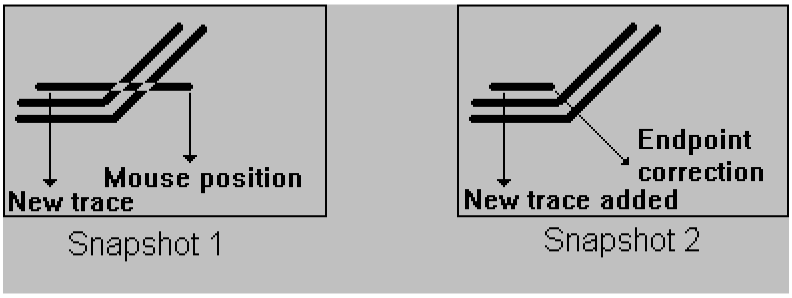
- Routing traces
- Drag one trace
- Change traces/vias

Activation of the **Routing** menu is made visible on the info bar at the bottom right of the window. **Routing traces** is now visible. Also the **Routing traces** button is visible pressed.

Add trace

When the **Routing** menu is active, new traces can be added, and existing traces modified. Place the mouse cursor one a trace/via/pin/connection and press the **left mouse button** to activate the trace drawing. For every trace segment to be placed press the **left mouse button**. To end with trace drawing, place the mouse cursor in the neighborhood of another trace/via/pin, and the new trace will automatically centered and added (This is only when drawing traces with two trying traces). After adding the trace(s) drawing will stop. Another possibility to stop the trace drawing is to press the **ESC** key. When a new trace segment is added inside an areafill, the areafill is adjusted. The current netname and trace width/clearance will be visible in the info bar at the bottom right of the window.

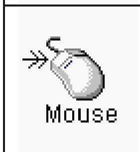
Trace drawing feature



In the above example a new trace is drawn. In snapshot 1 the new trace is to long because it overlaps two other traces. The special feature of the trace drawing is the ability to adjust the length of the new trace. The new added trace will not overlap other traces/vias/pins. This is visible in snapshot 2. The new added trace will be adjusted to the nearest valid grid position.

Another feature is moving one other trace during drawing of traces. Place the mouse cursor on the trace to be moved (Shoved) and press the **Shift** button. This one trace can now be moved. By releasing the **Shift** button the trace will be placed, and routing will be activated again.

Add via

	Press **.**
	Press the left mouse button twice

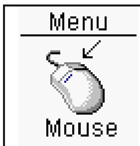
When trace drawing is active, a **via** will be inserted at the current mouse position. If necessary one or two traces will be added first. After inserting the **via**, the current drawing layer will be switched to a previous used and/or different layer.

Trace popup menu

When trace drawing is active the following functions (in the popup menu) are available when the **right mouse button** is pressed:

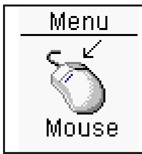
- Display clearance
- Display two trying traces
- Display via option
- Finish trace
- Highlight/unhighlight net
- Switch to another layer
- Delete trace
- Goto previous trace segment
- Change trace width
- Change clearance
- Change cross hair of the mouse cursor

Display clearance

	**Display clearance** -> **on** **Display clearance** -> **off**

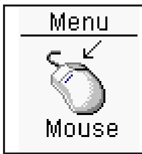
With this function the clearance of the one or two trying traces will be toggled on or off.

Display two trying traces

	**Display two trying traces**

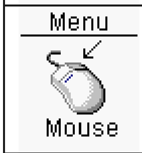
Displays one or two trying traces.

Display via option

	**Display via option**

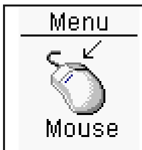
With this function a circle will be drawn at the current mouse cursor indicating the current via + clearance size. With this function exact via placement is simple.

Finish trace

	Press **Space bar**
	**Finish trace**

When trace drawing is activated by clicking on a connection line, and the space bar is pressed, the trace will be finished to the opposite point of the connection (if possible).

Highlight/unhighlight net

	**Highlight net** **Unhighlight net**

During drawing a trace (routing), traces/vias/connections of a net can be highlighted or unhighlighted.

Switch to another layer

Menu Mouse	**Select layer**
Keyboard	Press **F4**

When the starting point of the current trying trace is connected to a through hole pin or a via, switching to a different layer is possible. In the submenu of **Select layer** the layer can be chosen.

P.S. It is possible to switch to a powerplane layer, however drawing traces on a powerplane layer is not possible.

Delete trace

Menu Mouse	**Delete trace**

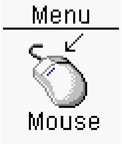
The current drawing trace will be deleted, and trace drawing stopped.

Goto previous trace segment

Keyboard	Press **b**
Menu Mouse	**Trace backwards**

When **Trace backwards** is executed, the current drawing trace will be deleted, and tracing drawing will continue with trace, which was connected at the start point of the current trace.

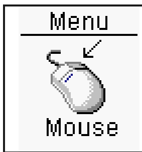
Change trace width

	**Change Trace width**

In the submenu of **Change trace width**, the trace width for the current trace can be modified. After trace drawing the new trace width will not be active anymore. In the submenu is the current trace width marked (If available). If none of the trace widths is appropriate, use the last item of the submenu. If the last menu item is selected, the trace width can be typed in the following dialogbox.

The trace settings in the pull down menu can be modified by changing the **pcb.ini** settings.

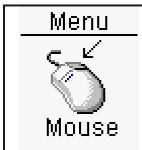
Change clearance

	**Change clearance**

In the submenu of **Change clearance**, the clearance for the current trace can be modified. After trace drawing the new clearance width will not be active anymore. In the submenu is the current clearance width marked (If available). If none of the clearance widths is appropriate, use the last item of the submenu. If the last menu item is selected, the clearance width can be typed in the following dialogbox.

The clearance settings in the pull down menu can be modified by changing the **pcb.ini** settings.

Change cross hair of the mouse cursor

	**Cross hair type**

When drawing traces a cross hair is visible at the current mouse position. With this function this crosshair can be switched between (x,+)

Add extra trace

Menu Mouse	**Add extra trace**

Adding an extra trace can be useful when a trace should be added from the middle of another trace.

Start routing with the shortest net

Menu Mouse	**Start routing with the shortest net**
Keyboard	Press **f**

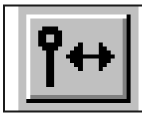
This function will search for the shortest net, center the display on the first pad of the net, and start the routing function.

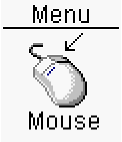
Add extra objects on top/bottom layer

Menu Mouse	**Add copper on top layer** **Add copper on bottom layer**

Lines and text (copper) can be added on top/bottom layer with these functions. These added objects will not be involved in the design rule check.

Change traces/vias

	Press **Change traces/vias** button

	Press **s** (For a maximum of three times)
	**Other menus** -> **Change traces/vias menu**

The **Change traces/vias** menu can be activated by one of three above actions.

When pressing the key **s** the default menu will switch between:

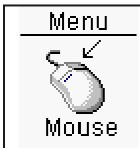
- Routing traces
- Drag one trace
- Change traces/vias

Activation of the **Change traces/vias** menu is made visible on the info bar at the bottom right of the window. **Change traces/vias** is now visible. Also the **Change traces/vias** button is now visible pressed.

When the right mouse button is pressed the following functions are available:

- Move traces/vias
- Copy traces/vias
- Select only
- Change trace width
- Change clearance traces/vias
- Change via
- Calculate length trace
- Swap traces/vias two nets
- Delete traces/vias net selected trace
- Delete

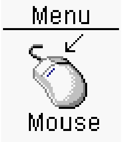
Move traces/vias

	**Move traces/vias**

With this menu function **selected** traces/vias can be moved (dragged) to a new position. Traces/vias will be moved to their new position, when they do not occupy other traces/vias/pins/areafills, otherwise vias will be deleted and traces replaced by a connection.

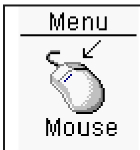
By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.
The connections of the nets involved are being recalculated.

Copy traces/vias

	**Copy traces/vias**

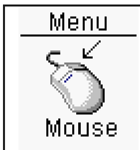
The **selected** traces/vias of one net will be copied to the desired location. The center of the selected objects is a pin/via. Traces/vias will be moved to their new position, when they do not occupy other traces/vias, otherwise vias will be deleted and traces replaced by a connection. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Select only

	**Select only** -> **Traces** **Select only** -> **Vias**

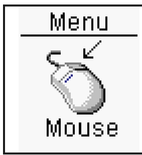
Traces or vias will be selected only with this function.

Change trace width

	**Change trace width**

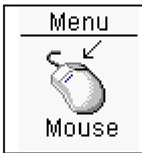
The trace width of **selected** traces can be changed into a new value typed in the following dialogbox.
The trace width of selected traces will be changed, when they do not occupy other traces/vias/pins/areafills.
The connections of the nets involved will **not** being recalculated, because it is time consuming.

Change clearance traces/vias

	**Change clearance**

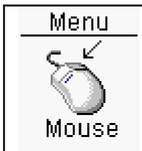
The clearance of **selected** traces/vias can be changed into a new value typed in the following dialogbox.
The clearance of selected traces/vias will be changed, when they do not occupy other traces/vias/pins/areafills.
The connections of the nets involved will **not** being recalculated, because it is time consuming.

Change via

	**Change via**

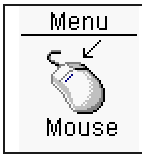
Selected vias can be changed with the next dialogbox. After pressing the **OK** button selected vias will be changed.
The selected vias will be changed, when they do not occupy other traces/vias/pins/areafills.
The connections of the nets involved will **not** being recalculated, because it is time consuming.

Calculate length trace

	**Calculate length trace**

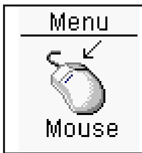
With this menu function, the length of all the traces from a net with a **selected** trace, will be summed and displayed. Only when all the traces are chained, the result is the summed length of the traces, otherwise the result could be wrong.

Swap traces/vias two nets

	**Swap nets**

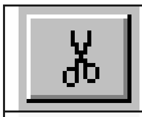
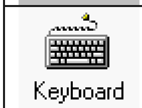
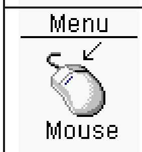
With this menu function, traces/vias of two different nets will be swapped. By selecting one trace for each net, this function can be activated.

Delete traces/vias net selected trace

	**Delete traces/vias net selected trace**

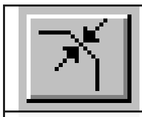
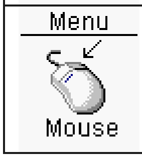
All traces and vias of the net specified by one **selected** trace will be deleted.

Delete

	Press **Delete** button
	Press **Del**
	**Delete traces/vias net selected trace**

Selected traces/vias will be deleted, and the connections of the nets involved are being recalculated.

Drag one trace

	Press **Drag one trace** button
	Press **s** (For a maximum of three times)
	**Other menus** -> **Drag one trace menu**

The **Drag one trace** menu can be activated by one of three above actions.

When pressing the key **s** the default menu will switch between:

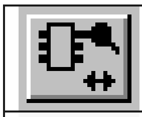
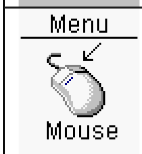
- Routing traces
- Drag one trace
- Change traces/vias

Activation of the **Drag one trace** menu is made visible on the info bar at the bottom right of the window, **Drag one trace** is now visible. Also **Drag one trace** button is visible pressed.

When a trace is selected dragging will be activated The dragging of this trace will be displayed in real time. Any collisions with others traces/vias/pins/areafills will be avoided. **Only traces/vias/pins/areafills in the current view will be used in the collision detection**.

See also Trace drawing feature

Dragging traces/vias/components

	Press **Drag traces/vias/components** button
	**Other menus** -> **Drag traces/vias/components menu**

Activation of the **Drag traces/vias/components** menu is made visible on the info bar at the bottom right of the window **Drag traces/vias/components** is now visible. Also **Drag traces/vias/components** button is visible pressed.

Select the traces/vias/components, and use the function **Drag traces/vias/components** to drag/rotate the traces/vias/components.

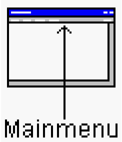
After pressing the **left mouse button** traces/vias/components will be placed on their new positions. If components are placed on traces/vias, the traces/vias under component pins will be deleted. If the dragging is in vertical or horizontal or diagonal direction traces will be extended (if possible). By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

By pressing the **right mouse button** during dragging the traces/vias/components will be rotated.

The connections of the nets involved are being recalculated.

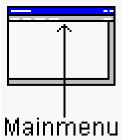
Check

Check connectivity

	Sub menu **Check** menu item **Connectivity**

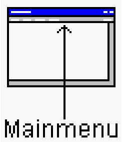
The connectivity against the netlist can be checked for all nets, or for one net. The connectivity check for all nets means that all nets (inclusive hidden or disabled) will be checked. When the check is completed all the new calculated connections except for the disabled nets will be made visible. The nets with connectivity errors will be put into a dialogbox.
Also the connectivity of one net can be checked. In the next dialogbox the net can be selected. After pressing the **OK** button the connectivity for that net will be checked.

Check design rules

	Sub menu **Check** menu item **Layers**

The design rules can be checked for all layers, or for one layer. A design rule check means, to check that traces/vias/pins/areafills from different nets do not overlap each other. When a design rules check has been executed for one or all layers, all design rule errors will be made visible.
If the board outlines consists of closed segments, and the board outline keepout parameter is greater than zero, all the pads/traces/vias will be checked if within these board outlines.

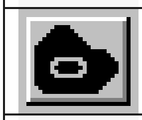
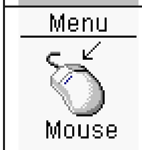
View design rule errors

	Sub menu **View** menu item **Select error**

Displays a specific design rule error or all design rule errors. In the next dialogbox the errors will be listed. When selecting one of the lines, that error will be displayed centered.

The design rule errors can be hidden with Hide/view layers function

Powerplanes

	Press **a**
	Press **Areafills/Powerplanes** button
	**Other menus ->Areafills/powerplanes menu**

The **Areafills/powerplanes** menu can be activated by one of two above actions.

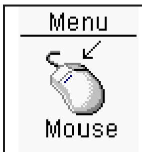
Activation of the **Areafills/powerplanes** menu is made visible on the info bar at the bottom right of the window. **Add/change areafills/powerplanes** is now visible. Also the **Areafills/powerplanes** button is visible pressed.

A powerplane is one the layers coupled to one net, and is almost fully filled with copper. For example powerplanes are used for the powernets (VCC, 3V3,GND). For those nets it is necessary to have low impedance anywhere on the PCB.

When the right mouse button is pressed the following functions are available:

- Add powerplane
- Delete powerplane
- Cut from powerplane
- Change powerplane

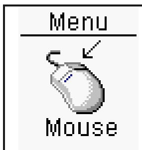
Add powerplane

	**Add powerplane** -> **Select layer**

In the next popup menu a number of layers is visible. If all the layers are already occupied with either traces or other powerplanes, no layers are visible. In the next dialogbox the net can be selected, the powerplane clearance and the distance to the PCB border can be specified. Also some parameters for thermal reliefs can be specified. After pressing the **OK** button the powerplane will be added. If there are closed board outlines, the powerplane will be limited by those board outlines.

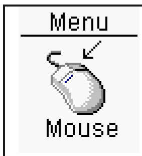
See also Thermal relief

Remove powerplane

	**Remove powerplane** -> **Select layer**

The (existing) powerplane of the selected layer will be deleted.

Cut from powerplane

	**Cut from powerplane** -> **Polyline** **Rectangle** **Circle** **Horizontal trace** **Vertical trace** -> **Select layer**

With this function, the selected powerplane can be changed by cutting pieces of copper. There are three cutout possibilities, with a **circle**, **rectangle, horizontal trace, vertical trace** and a **polyline**.

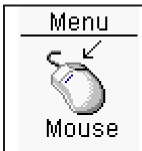
Polyline : When drawing this polyline use the **right mouse button** menu to change the drawing direction, goto the previous polyline point (Backwards) and to finish the polyline drawing. After finishing the polyline drawing and the polyline does not contain any crossings of lines, the area enclosed by the polyline will be cut from the powerplane.

Rectangle : A rectangle will be visible. The rectangle can be changed by pressing the **shift** key. When the **spacebar** has been pressed a dialogbox will be visible. The width, height parameters of the rectangle can be entered. Every time the **left mouse button** is pressed the rectangle area will be cutout from the powerplane. To leave this function by the **ESC** key or use the **right mouse button** menu.

Circle : Same as the rectangle cutout.
Horizontal trace : Same as the rectangle cutout.
Vertical trace : Same as the rectangle cutout.

When the cutout function is active all the pins of the powerplane net will be highlighted

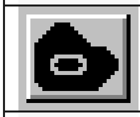
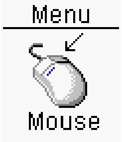
Change powerplane

	**Change powerplane** -> **Select layer**

The thermal relief definition of the powerplane can be changed with this function.

See also Thermal relief

Areafills

	Press **a**
	Press **Areafills/Powerplanes** button
	**Other menus** ->**Areafills/powerplanes menu**

The **Areafills/powerplanes** menu can be activated by one of two above actions.

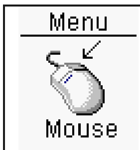
Activation of the **Areafills/powerplanes** menu is made visible on the info bar at the bottom right of the window. **Add/change areafills/powerplanes** is now visible. Also the **Areafills/powerplanes** button is visible pressed.

An areafill is a piece of copper and can have almost any form. An areafill can be used if a large piece of copper is needed on some layer (for example a low impedance path)

When the right mouse button is pressed the following functions are available:

- Add areafills
- Add areafill inside a powerplane
- Cut from areafill
- Change areafill
- Change hatch areafill
- Delete areafill

Add areafill

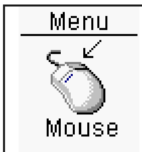
	**Add areafill** -> **Select layer**

In the next dialogbox the net can be selected, and the areafill clearance can be specified. Also some parameters for thermal relief’s can be specified.

After pressing the **OK** button a polyline must be drawn. When drawing this polyline all the pins of the areafill net will be highlighted (yellow). When drawing this polyline use the **right mouse button** menu to change the drawing direction, goto the previous polyline point (Backwards) and to finish the polyline drawing. After finishing the polyline drawing and the polyline does not contain any crossings of lines, the area enclosed by the polyline will be calculated and added as areafill. The areafill will only be added if the areafill does not overlap other areafills. During calculation of the areafill, any objects (traces/vias/pins) which do not belong to the areafill net will be cut out from the areafill. For large areafill this can take a while. If the calculation time is to long press the **ESC** key to abort, and the areafill will not be added to the design.

See also Thermal relief

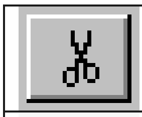
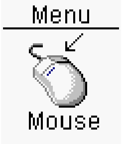
Add areafill inside a powerplane

	**Add areafill** -> **Select (powerplane) layer**

An example:

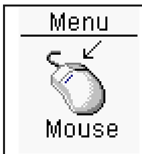
In a 5V powerplane is another smaller powerplane for +3.3 V needed. It is possible to use a small area of this powerplane for an areafill. The procedure to do is almost the same as for adding an areafill, only use the 5V powerplane as layer. The next dialogbox is almost the same as for adding a normal areafill. In this dialogbox there is a new item ‘Areafill inside powerplane’. The **Clearance** has to be specified. This clearance will be the distance between the 5V powerplane and the 3.3V areafill.
After pressing the **OK** button a polyline must be drawn. When drawing this polyline all the pins of the areafill net will be highlighted in yellow, and the pins of the powerplane are highlighted with red. All the pins highlighted in yellow must be included, and the pins highlighted in red must be excluded. When drawing of the polyline finished an area will be cut out from the powerplane, and the new areafill will be included. The cut out area in the powerplane will be a little bit greater.
In this new areafill there will be not cut outs (from vias/pins) calculated, so are not visible. In a later stage when the output films are generated, the cut outs will be calculated.

Delete areafill

	Press **Delete** button
	Press **Del**
	**Delete**

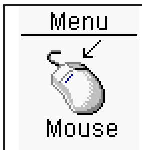
The **selected** areafill or deletion polygon inside the areafill will be deleted.

Cut from areafill

	**Cut from areafill** -> **Polyline** **Rectangle** **Circle** **Horizontal trace** **Vertical trace** -> **Select layer**

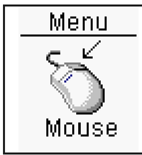
With this function areafills can be made smaller by cutting pieces of copper. The procedure is the same as for **Cut from powerplane**.

Change hatch areafill

	**Change hatch areafill**

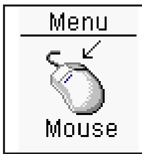
The hatch pattern of the selected areafill can be changed with this function.

Merge areafills

	**Merge**

Selected areafills with the same net, layer and thermal relief can be merged into a new areafill.

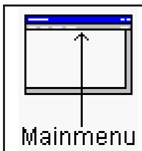
Change areafill

	**Change areafill**

The thermal relief definition of the **selected** areafill can be changed with this function. If the thermal relief has been changed the areafill will be recalculated.

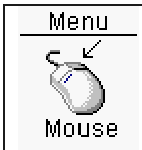
See also Thermal relief

Recalculate areafill after inserting an object

	Sub menu **Edit** menu item **Recalc areafill after inserting an object**

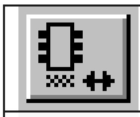
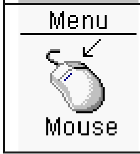
When this menu item is checked areafills will be recalculated if an object is inserted/moved/changed inside the areafill.

Rebuild areafill

	**Rebuild areafill**

With this function an areafill can be rebuild.

Moving component references

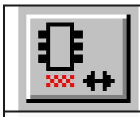
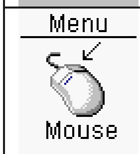
	Press **Move component references** button
	**Other menus** -> **Component references menu**

The **Component references menu** can be activated by one of two above actions.

Activation of the **Move component references** menu is made visible on the info bar at the bottom right of the window. **Move component references** is now visible. Also the **Move component references** button is visible pressed.

Selected component reference text can be moved to a new location. Pressing the **right mouse button** will rotate the component reference text.
To change the textheight or pen thickness of the text use the Change component parameters functions.

Moving component values

	Press **Move component values** button
	**Other menus** -> **Component values menu**

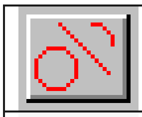
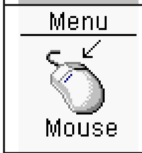
The **Component values menu** can be activated by one of two above actions.

Activation of the **Move component value** menu is made visible on the info bar at the bottom right of the window, **Move component values** is now visible. Also the **Move component values** button is visible pressed.

Selected component value text can be moved to a new location. Pressing the **right mouse button** will rotate the component value text.

To change the textheight or pen thickness of the text or the value text use the Change component parameters functions.

Special objects

	Press **Draw/change objects other layers** button
	**Other menus** -> **Specials menu**

Special object means extra objects on the top/bottom **silkscreen** layer, or objects on an **info** layer.
The **Specials** menu can be activated by one of two above actions.

Activation of the **Specials** menu is made visible on the info bar at the bottom right of the window. **Draw/change objects other layers** is now visible. Also the **Draw/change objects other layers** button is visible pressed.

The following actions can be done

- Add special objects
- Change special objects

Add objects on board outline

With the next Add special objects function objects can be added on the board outline.

Add special objects

Objects can be added on the following layers:

- Top layer (Copper objects)
- Bottom layer (Copper objects)
- Board outline layer
- Info layer

- Info layer 2
- Solder mask top layer
- Solder mask bottom layer
- Paste mask top layer
- Paste mask bottom layer
- Drills (plated) layer
- Drills (unplated) layer

The following objects can be added/changed:

Lines

A line object will be added. When the spacebar is pressed, a dialogbox will popup, and the line parameters can be edited by hand. As many as 16 points (15 lines) can be edited. In addition, one point can be edited for the starting point of the line. When the first character typed is a @ the coordinates will be relative against the Relative (grid)position. The coordinates typed in will be used with the current units (dimension).

Rectangles

A rectangle object will be added. When the spacebar is pressed, a dialogbox will popup, and the rectangle parameters can be edited by hand. The first two parameters are the width, and height. The optional third and fourth parameter is the rectangle center. When the first character typed is a @ the coordinates will be relative against the Relative (grid)position. The coordinates typed in will be used with the current units (dimension).

Circles

A circle object will be added. When the spacebar is pressed, a dialogbox will popup, and the circle parameters can be edited by hand. The first parameter is the diameter. The optional second and third parameter is the circle center. When the first character typed is a @ the coordinates will be relative against the Relative (grid)position. The coordinates typed in will be used with the current units (dimension).

Arcs

An arc object will be added. When the spacebar is pressed, a dialogbox will popup, and the arc parameters can be edited by hand. The first parameters are the diameter. The optional second and third parameter is the arc center. The optional fourth and fifth parameter is the first radial ending point. The optional sixth and seventh parameter is the second radial ending point. When the first character typed is a @ the coordinates will be relative against the Relative (grid)position. The coordinates typed in will be used with the current units (dimension).

Texts

A text object will be added. In the next dialogbox the text can be entered. In addition the textheight can be edited. After pressing the OK button the text can be placed. When the spacebar is pressed, a dialogbox will popup, and the text placement point can be edited by hand. When the first character typed is a @ the coordinates will be relative against the

Relative (grid)position. The coordinates typed in will be used with the current units (dimension).

Change objects on board outline

With the next Change special objects function objects can be changed on the board outline.

Change special objects

Move

Keyboard	Press **m**
Menu Mouse	**Move**

By **selecting** objects, those objects can be moved, copied or changed otherwise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Move to another layer

Keyboard	**Move objects to** -> **Select layer**

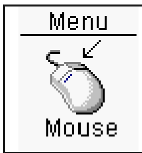
The **selected** objects will be moved to the selected layer.

Copy

Menu Mouse	**Copy**

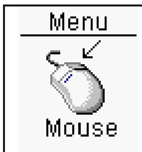
Copy **selected** objects.
By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Copy to another layer

	**Copy objects to** -> **Select layer**

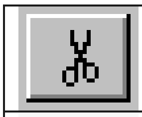
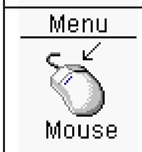
The **selected** objects will be copied to the selected layer.

Rotate

	**Rotate**

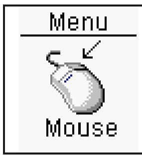
Rotate **selected** objects 90 degrees counter clockwise.

Delete

	**Delete** button
	Press **Del**
	**Delete**

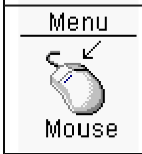
Delete **selected** objects.

Change line thickness

	**Change line thickness**

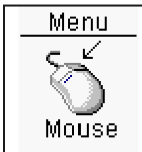
Change the line width of **selected** objects.

Change text

	Press **e**
	**Change text**

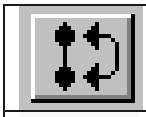
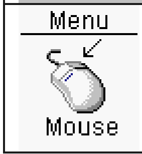
In the next dialogbox the **selected** text can be changed.

Change text height

	**Change text height**

In the next dialogbox the textheight of **selected** text objects can be changed.

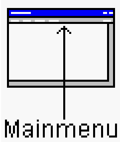
Gate/pin swap

	Press **Gate/pin swap** button
	**Other menus** -> **Gate/pin swap menu**

The **Gate/pin swap menu** can be activated by one of two above actions.
Activation of the **Gate/pin swap menu** menu is made visible on the info bar at the bottom right of the window. **Gate/pin swap** is now visible. Also **Gate/pin swap** button is visible pressed.
When the mouse cursor is placed on a pin, clicking on the **left mouse button** will display the gate/pin swap information for that pin. Swappable pins of gates will be highlighted, and also a number is visible in the center of the pad. Swappable pins are highlighted in a different color.

After selecting a pin, move the mouse cursor to the swappable pin or swappable gate pin, and by clicking on the **left mouse button** the gates/pins will be swapped.
All the gate/pin swap changes will be recorded in the file **pcb\gatepin.ban**. After all gate/pin swaps are done, the schematics should be updated with the **Back annotation** function of the design manager.

Schematic link

	Sub menu **Action** menu item **Active schematic select**

When placing components there is a special feature available. This feature is a link with the schematic editor. When this function is activated and the schematic editor is opened with a schematic file from the current design, selections made in the schematic editor will be reflected in the layout editor. When activating this function, all unhighlighted connections will be made invisible.
During the time this function is active the schematic editor(s) will be the master(s). This means selecting/deselecting in de layout editor has no effect on the schematic editor(s) selections.
The following objects selected in the schematic editor will be reflected in the layout editor:

Bus

All the connections of the bus netnames will made visible

Wire

The connections of the wires netname will made visible

Components

The component will be selected, and all connections, connected to a component pin will be made visible.

Initialization file pcb.ini

The initialization file **pcb.ini** is used to save some designs parameters. The file will be in the same directory as the design **.pcb** file.

The following parameters are used:

[Settings]

WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
Units	(0 = mils,1 = mm)
GridSize	The gridsize (10nm units)
DrawGrid	0 = FALSE,1 = TRUE
DrawAreaFills	0 = FALSE,1 = TRUE
DrawAreaFillMode	0 = None 4 = With hatch 5 = Gerber emulation plot surroundings 6 = Gerber emulation plot horizontal fill 7 = Both 8 = Solid
DrawClearances	0 = FALSE,1 = TRUE
DrawCompOutline	0 = FALSE,1 = TRUE
DrawConnections	0 = FALSE,1 = TRUE
DrawDrills	0 = FALSE,1 = TRUE
DrawInnerPads	0 = FALSE,1 = TRUE
DrawTopPads	0 = FALSE,1 = TRUE
DrawBottomPads	0 = FALSE,1 = TRUE
DrawCompPlacement	0 = FALSE,1 = TRUE
DrawSilkScreen	0 = FALSE,1 = TRUE
DrawObjects	0 = FALSE,1 = TRUE
DrawVias	0 = FALSE,1 = TRUE
DrawViaClearances	0 = FALSE,1 = TRUE
DrawCompReference	0 = FALSE,1 = TRUE
DrawCompValue	0 = FALSE,1 = TRUE
DrawTwoTryingTraces	0 = FALSE,1 = TRUE
SelectionMode	0 = replacement, 1= appending
Layer0	Draw bottom layer 0 (0 = FALSE,1 = TRUE)
Layer1	Draw layer 1 (0 = FALSE,1 = TRUE)
Layer2	Draw layer 2 (0 = FALSE,1 = TRUE)
…	…
Layer31	Draw layer 31 (0 = FALSE,1 = TRUE)
Grid0	Gridsize definition 0 (10nm units)

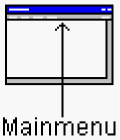
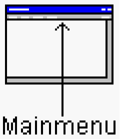
Grid1	Gridsize definition 1 (10nm units)
…	…
Grid29	Gridsize definition 29 (10nm units)
TraceWidth0	Trace width definition 0 (10nm units)
TraceWidth1	Trace width definition 1 (10nm units)
…	…
TraceWidth29	Trace width definition 29 (10nm units)
ClearanceWidth0	Clearance width definition 0 (10nm units)
ClearanceWidth1	Clearance width definition 1 (10nm units)
…	…
ClearanceWidth29	Clearance width definition 29 (10nm units)
BackGroundColor	24 bit RGB color (Stored as 32 bit)
ViewLayer1Color	24 bit RGB color (Stored as 32 bit)
ViewLayer2Color	24 bit RGB color (Stored as 32 bit)
ViewLayer3Color	24 bit RGB color (Stored as 32 bit)
ViewLayer4Color	24 bit RGB color (Stored as 32 bit)
ViewLayer5Color	24 bit RGB color (Stored as 32 bit)
ViewLayer6Color	24 bit RGB color (Stored as 32 bit)
ViewLayer1HilitedColor	24 bit RGB color (Stored as 32 bit)
ViewLayer2HilitedColor	24 bit RGB color (Stored as 32 bit)
ViewLayer3HilitedColor	24 bit RGB color (Stored as 32 bit)
ViewLayer4HilitedColor	24 bit RGB color (Stored as 32 bit)
ViewLayer5HilitedColor	24 bit RGB color (Stored as 32 bit)
ViewLayer6HilitedColor	24 bit RGB color (Stored as 32 bit)
ConnectionsColor	24 bit RGB color (Stored as 32 bit)
ConnectionsHilitedColor	24 bit RGB color (Stored as 32 bit)
NetPinsColor	24 bit RGB color (Stored as 32 bit)
NetPinsColor2	24 bit RGB color (Stored as 32 bit)
SilkScreenTopColor	24 bit RGB color (Stored as 32 bit)
SilkScreenBottomColor	24 bit RGB color (Stored as 32 bit)
ReferenceColor	24 bit RGB color (Stored as 32 bit)
CompValueColor	24 bit RGB color (Stored as 32 bit)
ShapePlacementOutLineColor	24 bit RGB color (Stored as 32 bit)
ShapeCompOutLineColor	24 bit RGB color (Stored as 32 bit)
ShapePinsTopColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillUnplatedColor	24 bit RGB color (Stored as 32 bit)
ShapePinsBottomColor	24 bit RGB color (Stored as 32 bit)
ViaPinsColor	24 bit RGB color (Stored as 32 bit)
ViaPinsHilitedColor	24 bit RGB color (Stored as 32 bit)
ViaPinsDrillColor	24 bit RGB color (Stored as 32 bit)
ObjectsInfoColor	24 bit RGB color (Stored as 32 bit)
ClearanceColor	24 bit RGB color (Stored as 32 bit)
ErrorColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)

BoardOutlineColor	24 bit RGB color (Stored as 32 bit)
SwappablePinsGateColor	24 bit RGB color (Stored as 32 bit)
SwappableGatePinsColor	24 bit RGB color (Stored as 32 bit)

Schematic editor

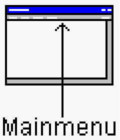
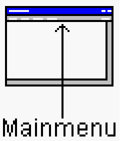
File

New sheet

	Sub menu **File** menu item **New sheet**
	Sub menu **File** menu item **New sheet in new window**

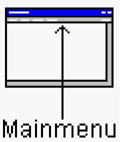
Creates a new sheet file (.sch) in the **sch** subdirectory of the current design.

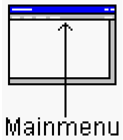
New symbol

	Sub menu **File** menu item **New symbol**
	Sub menu **File** menu item **New symbol in new window**

Creates a new symbol file (.sym) in the **sym** subdirectory of the current design.

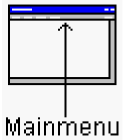
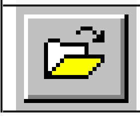
New sheetsymbol

	Sub menu **File** menu item **New sheetsymbol**

	Sub menu **File** menu item **New sheetsymbol in new window**

Creates a new sheetsymbol file (.sym) in the **sym** subdirectory of the current design.

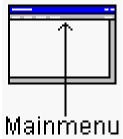
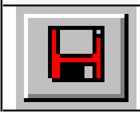
Open

	Sub menu **File** menu item **Open**
	Press **Open** button

Opens a sheet/symbol file (.sch/.sym) from the current design directory. When a sheet

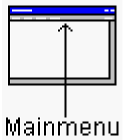
The symbols used in the schematic file will be loaded first from the local **sym** directory, the global symbols directory **sym** or from the symbol libraries in directory **lib**.

Save

	Sub menu **File** menu item **Save**
	Press **Save** button

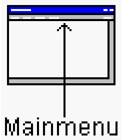
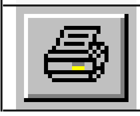
Saves the current file in the current (design) directory

Save as

	Sub menu **File** menu item **Save as**

Saves the current file under another name, in the current (design) directory

Print

	Sub menu **File** menu item **Print**
	Press **Print** button

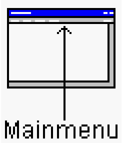
Prints the current file to the printer.

View

The following functions are the same as for the layout editor:

- Zoom in
- Zoom out
- Window based zooming
- Pan window
- Window based panning
- Return to previous view window
- Repaint
- View whole design
- Deselect all
- Undo
- Redo
- View/hide grid

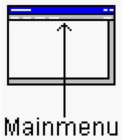
Change colors

	Sub menu **View** menu item **Change colors**

The color settings can be modified in the next dialogbox. The color settings will be copied into the s**ch.ini** initialization file. This file is stored into the directory of the project.

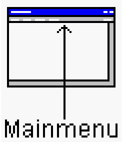
To use those sch colors for new designs, copy this s**ch.ini** file to main directory. Whenever a new design is created this s**ch.ini** file in the main directory will be copied to the directory of the new design.

Load default colors (Black background)

	Sub menu **View** menu item **Load default colors (Black background)**

The default color settings with a black background will be loaded.

Load default colors (Grey background)

	Sub menu **View** menu item **Load default colors (Grey background)**

The default color settings with a grey background will be loaded.

Programmable keys

The most important functions of the schematic/symbol editor have a short cut key (Accelerator). Those keys can be modified by editing the **sch.ini** file, section **[Keys]**.

Selection/deselecting objects

To select an object, place the mouse cursor above the object, and press and hold the left mouse button. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **Replacement mode**, and the second selection mode is the **Adding selection mode**.

The **Replacement selection mode** means, every time a new selection rectangle is drawed the previous objects selected will be unselected. When pressing down the **shift** key together with the **left mouse button** it is possible to use more than one selection at a time.

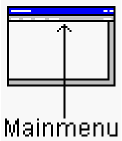
The other selection mode is the **Adding selection mode**. In this mode every object which is selected stays selected, until the deselect all function is executed. To deselect an object press the **left mouse button** and place the selection rectangle around this object again.

To change the selection mode use the **Replacement** or **Appending** in the **Selection mode** section of the menu.

Deselect all

The **Deselect all** function is the same as for the layout editor.

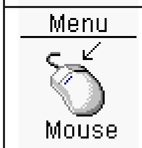
Line select mode

	Sub menu **Edit** menu item **Line select mode (normal)** **Line select mode (extended)**

Line select mode can be switched between **normal** and **extended** mode. In **normal** mode a line (wire/bus/line) will be selected if one or two line endpoints are in the selection window. In **Extended** mode a line (wire/bus/line) will be selected if whatever piece of the line is in the selection window.

Edit

Edit component parameters

	Press **e**
	**Edit component parameters**

In the next dialogbox the parameters of the **selected** component can be edited.
The following parameters can be changed:

- Reference
- Value
- Part nr
- Geometry
- Part description
- Package part nr
- Placing option

Reference

The reference of the component.

Value

The value of the component.

Part nr

The part nr of the component

Geometry

With the **Select geometry** button a geometry can be selected. In the top window the geometry directories and libraries are visible, and in the bottom window the geometries stored in that directory or library. By clicking on a directory/library item geometries in the directory/library will be listed in the bottom window. By clicking on a geometry the geometry will be visible in a new window to the right. By clicking on the **OK** button the geometry will be selected.

Part description

The part description is initially copied from the symbol description, but can be modified.

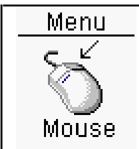
Package part nr

The package partnumber can be selected.

Placing option

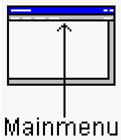
When the **Placing option** checkbox has been marked, the component will be marked as a placing option. Behind the component value a string **(*)** will be added to indicate the placing option.

Edit symbol

	**Edit <symbol>**

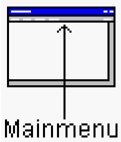
The symbol of the **selected** component can be edited with this function. After closing the symbol editor, the Schematic editor will reload the symbols.

Protect symbols

	Sub menu **Edit** menu item **Protect symbols**

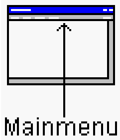
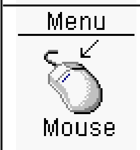
Selected symbols will be protected. Protected symbols can not be selected.

Unprotect symbols

	Sub menu **Edit** menu item **Unprotect symbols**

All protected symbols will be unprotected.

Edit gate/pin swap

	Sub menu **Edit** menu item **Edit gate/pin swap**
	**Edit gate/pin swap**

In the next dialogbox the gate/pin swap parameters of the symbol can be edited.
Editing gate swap info for devices like 7400 (Four nand gates) is not necessary.
In the first small editbox a group code (1 to 15) should be edited. In the right large editbox the pinnames of a gate should be edited. Every pinname will be separated by a comma, pin(s)/pinbus(ses) which are swappable should be enclosed by parentheses.

An example:

(3,4,5),(A[0:7]) : Pins 3,4 and 5 can be swapped against each other, and the eight pins inside the pinbus A[0:7] can be swapped against each other.

A few examples:

Pin swap example

7400 TTL device with two pins: IN1 and IN2 which should be swappable:
The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	(IN1,IN2)

Gate swap example1

74245 TTL device with eight inputs and outputs. Every Input/Output combination can be swapped against any other seven Input/Outputs. This means there are eight gates, each with two pins.

The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	2,18
Line 2	1	3,17
Line 3	1	4,16
Line 4	1	5,15
Line 5	1	6,14
Line 6	1	7,13
Line 7	1	8,12
Line 8	1	9,11

All the pins in lines with the same **code** numbers can be swapped. This means the number of pins in each line should be the same, and the numbers of pins enclosed by parentheses should be the same.

Gate swap example2

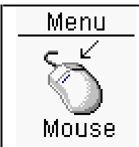
74244 TTL device with two sets of four inputs and outputs. Every Input/Output combination of a set can be swapped against any other three Input/Output. This means there are two sets of four gates, each with two pins.

The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	2,18
Line 2	1	4,16
Line 3	1	6,14
Line 4	1	8,12
Line 5	2	11,9
Line 6	2	13,7
Line 7	2	15,5
Line 8	2	17,3

The four gates in lines 1 to 4 have **code** number one, which means they are swappable. Also the four gates in lines 5 to 8 with **code** two are swappable.

Edit pinbus reorder

	**Add pinbus reorder -> Pinbus** **Delete pinbus reorder -> Pinbus** **Edit pinbus reorder -> Pinbus**

In the next dialogbox the pinbus reorder parameters of a **selected** components pinbus can be edited.
Pinbus reorder means the sequence of pins will be reordered. Such a pinbus reordering of pins is necessary when pins in this pinbus are swapped. The numbers in a pinbus reorder are index numbers.

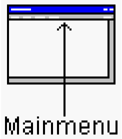
For example:

Pinbus: 3,9,45,12,41,89,23,63 with pinbus reorder 0,1,2,3,4,5,6,7

Pins 12 and 23 should be swapped.

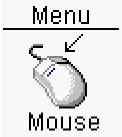
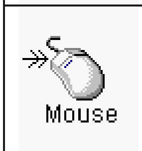
The pinbus reorder would be 0,1,2,6,4,5,3,7 (Index 3 and 6 are swapped)

Export text

	Sub menu **Edit** menu item **Export text**

The text of **selected** pins, powerpins, pinbusses or standard text, will be made visible in a dialogbox. This text can be used to import into a wordprocessor.

Edit any text

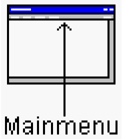
	Press **e**
	**Edit any text**
	Press the left mouse button twice

In the next dialogbox **selected** text can be edited.

The **selected** text can be:

- Reference name
- Value name (Only for editing sheets)
- Powerpin
- External connection
- Netlabel
- Normal text

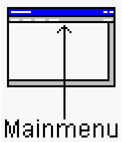
Search for any text

	Sub menu **Edit** menu **Search for any text**

In the next dialogbox the search text can be edited. After pressing the **OK** button the schematic editor tries to find this text, and will move the view window around this text. The text itself will be selected. The text can be:

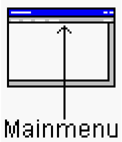
- Symbol name
- Value name (Only for editing sheets)
- Reference name
- Pinnumber
- Any pinnumber in a pinbus
- Netname powerpin
- Any pinnumber in a powerpin
- External connection
- Netlabel
- Normal text

Edit number of parts per package

	Sub menu **Edit** menu item **parts** -> **count**

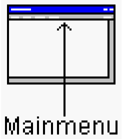
Edit the number of symbols in a device.

Edit pinnumbers package parts

	Sub menu **Edit** menu item **parts** -> **pins**

In the next dialogbox there is a listbox and editbox. The listbox shows the pinnames of the symbol. In the right editbox the pin numbers for each part in the device should be edited, starting with the first part. The pin numbers should be separated by commas. Spaces are not allowed.

Edit symbolnames

	Sub menu **Edit** menu item **Symbolnames**

When editing a symbol, the symbol properties can be changed with this function. The top editbox represents the symbolname. This symbolname can not be changed, because the symbolname and the filename of the symbol are the same. If the symbolname should be changed, save the symbol under another name. The checkbox to the right of the editbox defines if the symbolname is visible when the symbol is used in a sheet.
The second editbox contains the interface name. Normally the interface name is the same as the symbolname. When the symbol is a part of a bigger symbol (see Multiple symbols), the interface name will be different. The checkbox to the right of the editbox defines if the symbol is part of a multiple symbol.
The third editbox defines the initial reference name. This initial reference name should always end with a quotation mark **?**. This quotation mark is necessary for annotation. The checkbox to the right of the editbox defines if the initial reference name is visible when the symbol is used in a sheet. In the fourth editbox a description (help) can be edited.

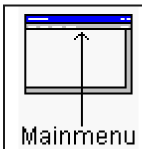
See also Annotation

The checkbox at the bottom defines if the symbol is protected in a sheet. Protected symbols in a sheet can not be selected.

Multiple symbols

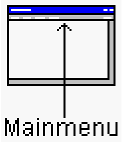
When the pincount of a symbol is very high, there is a possibility to split the symbol into two or more separate symbols.
Every symbol contains a certain amount of pins. For every symbol some properties must be edited. To edit the properties of symbol use the **Symbol names** of the **Edit** menu. The **Interface name** should be the same for all the symbols, and the **multiple symbols** checkbox should be marked.

Clear references

	Sub menu **Edit** menu item **Clear references**

All the **selected** symbols with numbered references will be reset to xxx?. For example reference **ABC234** will be reset to **ABC?**. References names that have a quotation mark, as last character will not be changed.

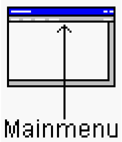
Check sheet

	Sub menu **Edit** menu item **Check**

The current sheet will be checked for errors. Possible errors are:

- Not connected busconnections.
- Not connected external connections.
- External connections must be unique.
- External connections directly connected to a busconnection.
- Wires directly connected to a bus.
- Busconnections not connected properly to a wire/bus.
- In a sheet the pins of a symbol are not connected to a wire/bus endpoint.
- Netlabel is not connected to a wire or bus.

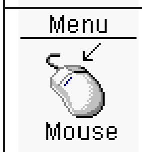
Check symbol

	Sub menu **Edit** menu item **Check**

The current symbol will be checked for errors. Possible errors are:

- Double pinnumbers

Edit pin normal symbol

	Press **e**
	**Edit text**

In the next dialogbox parameters of **selected** pins can be edited. Connection, type, pinname, labelname and visibility can be modified. After clicking the **OK** button the pin will be changed.

Edit sheet symbol pin

Keyboard	Press **e**
Menu / Mouse	**Edit text**

In the next dialogbox parameters of **selected** sheetsymbol pins can be edited. Connection type and labelname can be modified. After clicking the **OK** button the pin will be changed.

Edit pinbus

Keyboard	Press **e**
Menu / Mouse	**Edit text**

In the next dialogbox parameters of **selected** pinbusses can be edited. Connection, type, pinname, nr pins and labelname can be modified. After clicking the **OK** button the pinbus will be changed.

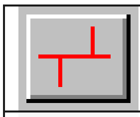
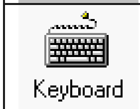
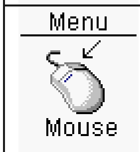
Change grid

Keyboard	Press **ctrl g**
Mainmenu	Sub menu **Grid** menu item (**0.1,0.2,1.0)**

Pressing **Ctrl g** will switch the grid **0.1** and **1.0**.
Changing the grid is possible in every drawing/moving function.

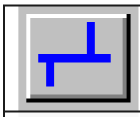
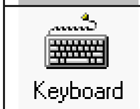
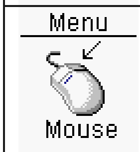
Add objects

Add wire

	Press **Add wire** button
	Press **w**
	**Add wire**

Add wire.

Add bus

	Press **Add bus** button
	Press **b**
	**Add bus**

A bus is a collection of a number of signals. The amount of signals in a bus depends on how the bus is named. For instance if the bus is a databus with the signals AD0, AD1, AD2, AD3, AD4, AD5, AD6, and AD7 the name of the bus would be AD[0:7]. When a bus has such a name the amount is signals in a bus is limited to eight is this case. A bus can also contain an unspecified number of signals; for example a possible name could be SYSCON.
Via busconnections signals (wires) are connected to a bus. The signal names that are connected to a bus must be unique. When a bus is directly connected to a pinbus, the pinbus should have a name with a number range. The amount of signals within this number range should be the same as the pinbus signal count.

It is not allowed to connect a signal to a bus once (unused signal). For example if the signal MEMR is connected via a busconnection to the bus MEMCON, the same signal should be connected to this bus on another place. The reason for this is to avoid some mistakes, when connecting signals to a bus. In the previous example signal MEMR is used to connect to a bus. Suppose you have typed MEMR wrong (MEM), for the first connection, and the second signal connection name is typed with MEMR. The result of this is that the two signals are not connected to each other. When building the netlist these errors will be reported.

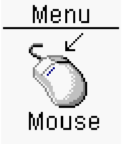
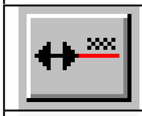
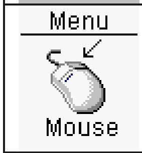
Add busconnection

	Press **Add busconnection** button
Keyboard	Press **B**
Menu Mouse	**Add busconnection**

A busconnection is a small symbol that is used to connect a bus with a wire or pinbus. After the function is activated the busconnection can be placed. The thick part of the busconnection should be placed against the bus, and the smallest part to the wire or pinbus end point.
By pressing the right mouse button the busconnection can be mirrored.

Add external connection

	Press **Add external input connection** button
Menu Mouse	**Add external connections** -> **Input**
	Press **Add external output connection** button

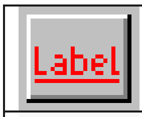
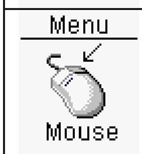
	**Add external connections** -> **Output**
	Press **Add external I/O connection** button
	**Add external connections** -> **I/O**

An External connection is used to connect signals or busses from different sheets in a hierachical design. An external connection is an Input, Output or I/O. The name used for the External connection should be the same, as the pin name in the sheetsymbol in the above sheet.

After the function is activated a dialogbox is visible. You have to fill in a name for the external connection. After the name has been typed and the **ok** button is pressed, the external connection can be placed. The external connection should be placed at the endpoint of a wire or bus.
By pressing the **right mouse button** the external connection can be mirrored.

See also Hierarchical designs

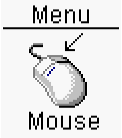
Add netlabel

	Press **Add label to wire/bus** button
	Press **n**
	**Add label to wire/bus**

A netlabel is a text string which names the wire or bus. When a wire is connected to a bus via a busconnection, the wire needs a netlabel. A netlabel is always required for a bus. When wires or busses are drawn with two or more lines, only one line of them needs this netlabel.
To add a netlabel to a wire or bus, **select** the wire or bus first. After the wire or busses has been selected use this function. After the function is activated a dialogbox is visible. You have to fill in a name for the netlabel. After pressing the **OK** button, the netlabel can be

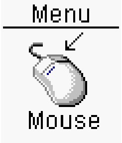
placed. There is a help line visible; to show at which endpoint of the wire or bus the netlabel is connected.

Add incremental netlabels to wires

	**Add incremental netlabels to wires**
	Press **N**

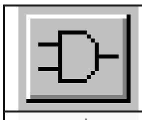
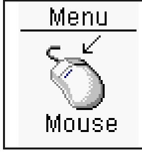
When incremental netlabels are required for a number of wires, this function can be used. First select the wires, and then use this fnuction. In the dialogbox the name of the netlabel should be filled in.

Add netlabel + wire

	**Add netlabel + wire**

In the next dialogbox the names of the netlabels should be filled in. For every line in the dialogbox a wire and netlabel will be created.

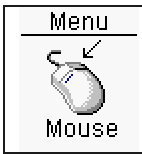
Add symbol

	Press **Add label to wire/bus** button
	Press **I**
	**Add label to wire/bus**

A symbol is a graphical representation of a component (Resistor, capacitor, IC).
A symbol can be imported from a library or a symbol directory.

In the top window the symbol directories and libraries are visible, and in the bottom window the symbols stored in that directory or library. By clicking on a directory/library item symbols in the directory/library will be listed in the bottom window. In the edit box at the bottom of the dialogbox a symbol description is shown for the selected symbol. By clicking on a symbol the symbol can be placed on the schematic. By pressing the **right mouse button** a number of timer the symbol will rotate/mirror. By pressing the **left mouse button** the symbol will be placed. The dialogbox remains visible, and will disappear by clicking on the **Cancel** button.

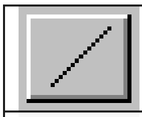
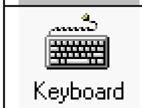
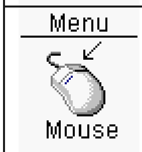
Add component

	**Add database component** -> S**elect component**

Adding components can be a lot quicker than adding symbols; because after a symbol has been added some parameters have to be changed. Those parameters are the **geometry** and **value**.
The standard components like resistors, capacitors, 74xx range of IC and some other components can be added directly.
In the series of pulldown menus the component family can be selected. In the next dialogbox the component can be selected and by pressing the **OK** button the symbol can be placed.

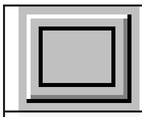
Add other objects

Line

	Press **Add line** button (Only when editing symbols)
	Press **l**
	**Add special** -> **Line**

Add a line.

Rect

	Press **Add rect** button (Only when editing symbols)
Keyboard	Press **r**
Menu Mouse	**Add special** -> **Rect (4 lines)**

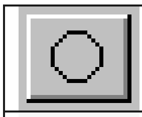
Add a rectangle. This rectangle is build up with four lines. This can be useful when the rectangle should be changed, because lines can be dragged.

Rect (normal)

Menu Mouse	**Add special** -> **Rect**

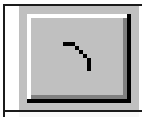
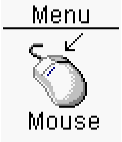
Add rectangle.

Circle

	Press **Add circle** button (Only when editing symbols)
Keyboard	Press **C**
Menu Mouse	**Add special** -> **Circle** -> **Select circle**

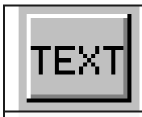
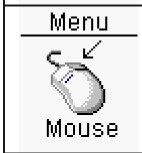
Add a (partial) circle.

Arc

	Press **Add arc** button (Only when editing symbols)
	Press **a**
	**Add special** -> **Arc**

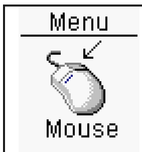
Add an arc.

Text

	Press **Add text** button (Only when editing symbols)
	Press **t**
	**Add special** -> **Text**

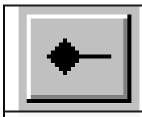
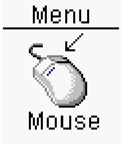
Add text

Numbers incremental

	**Add special** -> **Numbers incremental**

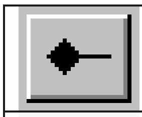
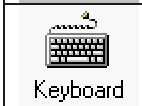
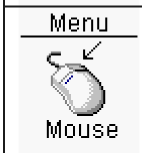
Add a range of numbers.

Add pin

	Press **Add pin** button
	Press **a**
	**Add pin**

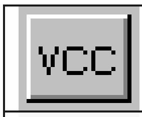
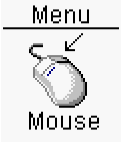
After the function is activated a dialogbox is visible. In this dialogbox one or more pins can be added to the symbol. The dialogbox consists of three edit boxes and two series of radiobuttons. The first editbox is the edit box for the pin numbers (pinnames). The second editbox is the edit box for the pin labels. This editbox is only relevant when adding pins for a sheet symbol. When adding pins for a normal symbol you do not have to fill this editbox. (It will be filled automatically) The third editbox is the editbox for the pin text. This pintext will be added as normal text. With the two series radiobuttons, the electrical direction and electrical type can be selected, for all pins that will be added. After filling the dialogbox click **OK**, and the pin(s) can be placed. To mirror or rotate the pin press the **right mouse button**.

Add sheetsymbol pin

	Press **Add pin** button
	Press **a**
	**Add pin**

After the function is activated a dialogbox is visible. In this dialogbox one or more pins can be added to the sheetsymbol. The dialogbox consists of three edit boxes and a serie of radiobuttons. The first editbox is the edit box for the labelnames. The second and third editbox is for the incremental placement of the pins. With the serie of radiobuttons the electrical direction can be selected, for all pins that will be added. After filling the dialogbox click **OK**, and the pin(s) can be placed. To mirror or rotate the pin press the **right mouse button**.

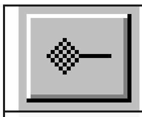
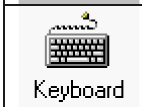
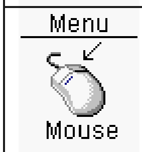
Add powerpin

	Press **Add powerpin** button
	Press **p**
	**Add powerpin**

When adding pins to a symbol who are connected to power (+5V,GND), there is another option adding these pins, the so-called powerpins (power pin text). For example for a 74LS00 device pin 7 should be connected to ground, and pin 14 to the VCC. Instead of adding two pins, two powerpins can be added.
When adding powerpins two items should be edited, the netname and the powerpin number(s). The netname specified will be used as a standard net for the whole design. The powerpin number(s) specified consists of one or more pinnumbers separated by commas. If necessary two or more powerpins with same nets and different pinnumbers can be added.

To add a powerpin use one of the above three actions. After the function is activated a dialogbox is visible. In this dialogbox two editboxes are visible. In the first editbox the netname has to be specified, and in the second editbox the pinnumbers separated by commas. After filling the dialogbox click **OK**, and the powerpin can be placed. To rotate the powerpin text press the **right mouse button**.

Add pinbus

	Press **Add pin** button
	Press **P**
	**Add pinbus**

A pinbus is special pin definition, to replace a series of standard pins. For example if a numbers of pins for a databus should added to the symbol, a pinbus can be used instead of adding every pin of the databus separately. In digital designs with a CPU and memory devices pinbusses are very useful.

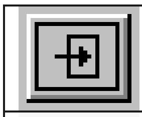
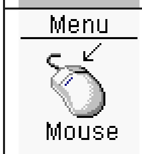
To add a pinbus use one of the above three actions. After the function is activated a dialogbox is visible. The dialogbox consists of four edit boxes, and two series of radiobuttons. The first editbox is the edit box for the pin numbers (pinnames) separated by commas. The second editbox is the edit box for the pin label. The third editbox is the editbox for the pin text. This pintext will be added as normal text. In the fourth editbox the amount of pins has to be filled in. With the two series radiobuttons the electrical direction and electrical type can be selected, for the pinbus that will be added. After clicking **OK**, the pinbus can be placed. To mirror or rotate the pinbus press the **right mouse button**.

The amount of pinnumbers separated by commas and the value in the pin count editbox has to be the same. There is a maximum of **64** pins for a pinbus.

Hierarchical designs

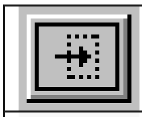
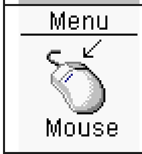
A hierarchical design is a design with more that one sheet. To be able to use more than one sheet an interface has to been used to connect the different sheets with each other. The interface used consists of sheet symbols in one sheet, and external connections on other sheets. To make this more understandable an example will be used.
In a hierarchical design there is always a start sheet, the so-called top sheet. In this top sheet, sheet symbols are included. Every sheetsymbol represents a (sub)sheet. The pins defined in the sheetsymbol represent the external connections of the subsheet. The label name of a pin in the sheet symbol must be the same, as the name of the external connection in the subsheet. Also the pin type (Input,Output,I/O) must be the same. Every subsheet can also contain sheetsymbols, which represents further subsheets

Open subsheet

	Press **Open subsheet** button
	**Edit sheet selected symbol**

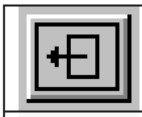
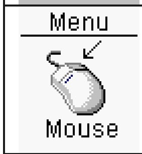
When a sheetsymbol is **selected** and this function is executed, the current design will be left, and the sheet related to this sheetsymbol will be opened.

Open sheetsymbol

	Press **Open subsheet** button
	**Edit selected sheetsymbol**

When a sheetsymbol is **selected** and this function is executed, the current design will be left, and the sheetsymbol will be opened.

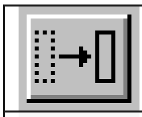
Goto higher sheet

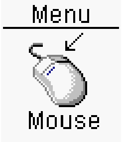
	Press **Goto higher sheet** button
	**Goto higher sheet**

Go back to a higher sheet. When the **Open subsheet** or **Open sheetsymbol** function was used to open the current sheet or sheetsymbol, **Goto higher sheet** will return to the previous sheet

Change objects

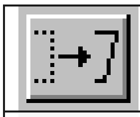
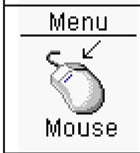
Move objects

	Press **Move** button
	Press **m**

	**Move**

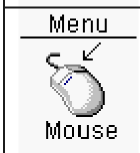
Move **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Drag objects

	Press **Drag** button
	Press **d**
	**Drag**

Drag **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Rotate objects

	Press **R**
	**Rotate**

Rotate **selected** objects 90 degrees counter clockwise.

Mirror objects

Menu Mouse	**Mirror X** **Mirror Y**

Mirror **selected** objects in X or Y direction.

Copy objects

	Press **Copy** button
Keyboard	Press **c**
Menu Mouse	**Copy**
Menu Mouse	**Copy multiple -> Select number (2..10)**

With this function **selected** objects can be copied once, or 2..10 times. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Copy objects to clipboard

Mainmenu	Sub menu **Edit** menu item **Copy objects to clipboard**
Keyboard	Press **Ctrl ins**

Selected objects will be copied to the clipboard. This function will work when editing symbols and sheets.

Paste objects from clipboard

Mainmenu	Sub menu **Edit** menu item **Paste objects from clipboard**
Menu Mouse	**Paste objects from clipboard**
Keyboard	Press **Shift ins**

Objects that previously had been copied to the clipboard will be pasted in the current design. This function will work when editing symbols and sheets.

Delete objects

	Press **Delete** button
Keyboard	Press **Del**
Menu Mouse	**Delete**

Delete **selected** objects.

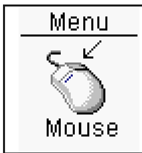
Netlabels connected to selected wire(s) or bus(ses), will also be deleted.

Unselect objects

Menu Mouse	**Unselect** -> **Objects**

Unselect objects

Select only

	**Select only** -> **Objects**

Select only one object type.

Initialization file sch.ini

The initialization file **sch.ini** is used to save some designs parameters. The file will be stored in the design directory, if there is a design active.
If the schematic editor has been started as stand alone, the search rules for **sch.ini** will be as follows. When the last subdirectory name of the full edit path is **sch** or **sym**, the schematic editor will use the one level up directory. An example:

Symbols pull path: c:\design\test\sym\testsymbol.sym

The schematic editor is looking for the **sch.ini** in the c:\design\test directory. This situation is the case when a symbol or schematic from a design will be opened.

When there is no **sch** or **sym** directory, or the **sch.ini** does not exist, the schematic editor tries to find the **sch.ini** in the same directory as the symbol or sheet.

The following parameters in the **sch.ini** are used:

[Settings]

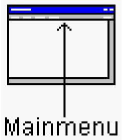
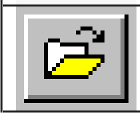
WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
SymbolDialogStartX	Origin X of the add symbol dialog window
SymbolDialogStartY	Origin Y of the add symbol dialog window
Repeat mode	(0 = off,1 = on)
LineSelectMode	(0 = normal, 1 = extended)
SelectMode	(0 = Appending,1 = Replacement)
BackGroundColor	24 bit RGB color (Stored as 32 bit)
WireColor	24 bit RGB color (Stored as 32 bit)
BusColor	24 bit RGB color (Stored as 32 bit)
BusConnectionColor	24 bit RGB color (Stored as 32 bit)
GlobalConnectionColor	24 bit RGB color (Stored as 32 bit)
JunctionColor	24 bit RGB color (Stored as 32 bit)
NetLabelColor	24 bit RGB color (Stored as 32 bit)
InstanceRefTextColor	24 bit RGB color (Stored as 32 bit)
InstanceValueTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPinColor	24 bit RGB color (Stored as 32 bit)
SymbolPinBusColor	24 bit RGB color (Stored as 32 bit)
SymbolPinTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPowerPinTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPinBusTextColor	24 bit RGB color (Stored as 32 bit)
SymbolLineColor	24 bit RGB color (Stored as 32 bit)

SymbolRectColor	24 bit RGB color (Stored as 32 bit)
SymbolTextColor	24 bit RGB color (Stored as 32 bit)
SymbolArcColor	24 bit RGB color (Stored as 32 bit)
SymbolCircleColor	24 bit RGB color (Stored as 32 bit)
ObjectLineColor	24 bit RGB color (Stored as 32 bit)
ObjectRectColor	24 bit RGB color (Stored as 32 bit)
ObjectCircleColor	24 bit RGB color (Stored as 32 bit)
ObjectArcColor	24 bit RGB color (Stored as 32 bit)
ObjectTextColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)

Geometry editor

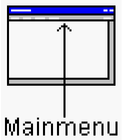
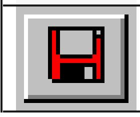
File

Open

	Sub menu **File** menu item **Open**
	Press **Open** button

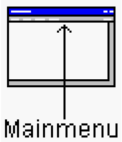
Opens a geometry file (.shp) from the standard geometry directory.

Save

	Sub menu **File** menu item **Save**
	Press **Save** button

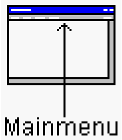
Saves the current geometry file (.shp) in the standard geometry directory

Save as

	Sub menu **File** menu item **Save as**

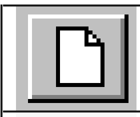
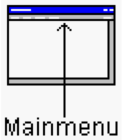
Saves the current geometry file (.shp) under a new name, in the standard geometry directory

Print

	Sub menu **File** menu item **Print**

Prints the current file to the printer.

Make new geometry

	Press **New geometry** button
	Sub menu **File** menu item **New**

In the next dialogbox there is a listbox with six items.

- A new (empty) geometry will be made.
- DIL
- QUAD flatpack
- BGA
- PGA
- SOIC

New DIL geometry

In the next dialogbox the parameters for a new DIL (Dual In Line) geometry can be entered. The DIL geometry is a device based on through holes.

The following parameters can be changed:

Nr pins

If the total amount of pins is two times the NrPins entered. The pinnumber counting starts with one and increments with one for the following pins. The pin counting direction for the left column with pads is downwards, and for the right column with pads upwards.

Pad

Pad size solder mask

If the solder mask pad is not necessary, fill the parameter with zero.

Diameter anti power pad

The anti power pad will always be a circle.
If the anti power pad is not necessary, fill the parameter with zero.

Diameter inner pad

The inner pad will always be a circle.
If the inner pad is not necessary, fill the parameter with zero.

Drill hole

Distance

Clearance

The initial clearance is the clearance used for this geometry.

Pin 1 type

The first pin (1) can be a square or a circle. All the next pins will be circles.

New Quad flatpack geometry

In the next dialogbox the parameters for a QUAD flatpack geometry can be entered. The QUAD flatpack geometry is a SMD based device.

The following parameters can be changed:

Nr pins X,Nr pins Y

If the total amount of pins is two times the (Nr pins X + Nr pins Y) entered. The pinnumber counting starts with an optional string and a number, and increments with one for the following pins. The pin counting direction is counter clockwise.

Pad size X,Y

Pitch

Pad size solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad size paste mask

If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Distance X,Y

Starting pin nr

The starting pin nr consists of two editboxes. The first editbox (optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with one for the next pads.

New BGA geometry

In the next dialogbox the parameters for a BGA (Ball Grid Array) geometry can be entered. The BGA geometry is a SMD based device.

The following parameters can be changed:

Nr pins

If the total amount of pins is (Nr pins X * Nr pins Y). The pinnumber counting starts with **A1** and increments with one for the following pins in the horizontal direction. In the vertical direction, counting is based on letter increments. For the first 23 rows the following letters will be used: **A,B,C,D,E,F,G,H,J,K,L,M,N,P,R,S,T,U,V,W,X,Y,Z**. Only the letters **I,O,Q** will not be used because of similarity with other characters. When more then 23 rows are necessary, two letters will be used. The 24th row will use the letters **AA**. The following rows will use the letters **AB,AC,AD, … AZ,BA,BB** …**BZ**, etc.

Pad

Pitch

Pad solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad paste mask

If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Starting pin nr

The starting pin nr is **A1**, and the pad can be a circle or a square.

New PGA geometry

In the next dialogbox the parameters for a BGA (Ball Grid Array) geometry can be entered. The BGA geometry is a device based on through holes.

The following parameters can be changed:

Nr pins

If the total amount of pins is (Nr pins X * Nr pins Y). The pinnumber counting starts with **A1** and increments with one for the following pins in the horizontal direction. In the vertical direction, counting is based on letter increments. For the first 23 rows the following letters will be used: **A,B,C,D,E,F,G,H,J,K,L,M,N,P,R,S,T,U,V,W,X,Y,Z**. Only the letters **I,O,Q** will not be used because of similarity with other characters. When more then 23 rows are necessary, two letters will be used. The 24th row will use the letters **AA**. The following rows will use the letters **AB,AC,AD, … AZ,BA,BB** …**BZ**, etc.

Pad

Diameter anti power pad

The anti power pad will always be a circle.
If the anti power pad is not necessary, fill the parameter with zero.

Diameter inner pad

The inner pad will always be a circle.
If the inner pad is not necessary, fill the parameter with zero.

Pitch

Drill

Pad solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

Starting pin nr

The starting pin nr is **A1**, and the pad can be a circle or a square.

New SOIC geometry

In the next dialogbox the parameters for a SOIC geometry can be entered. The SOIC geometry is a SMD based device.

The following parameters can be changed:

Nr pins

If the total amount of pins is two times the NrPins entered. The pinnumber counting starts with one and increments with one for the following pins. The pin counting direction for the left column with pads is downwards, and for the right column with pads upwards.

Pad size X,Y

Pitch

Pad size solder mask

If the solder mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Pad size paste mask

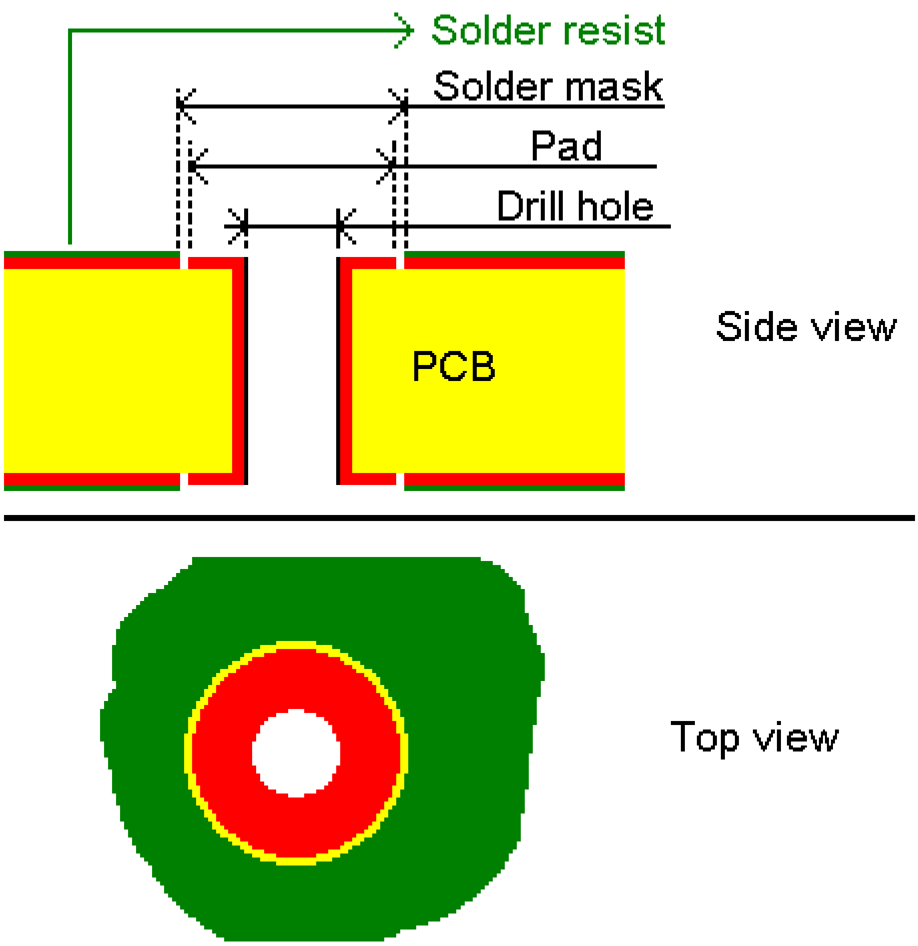
If the paste mask pad is not necessary, fill one of the parameters (X,Y) with zero.

Clearance

The initial clearance is the clearance used for this geometry.

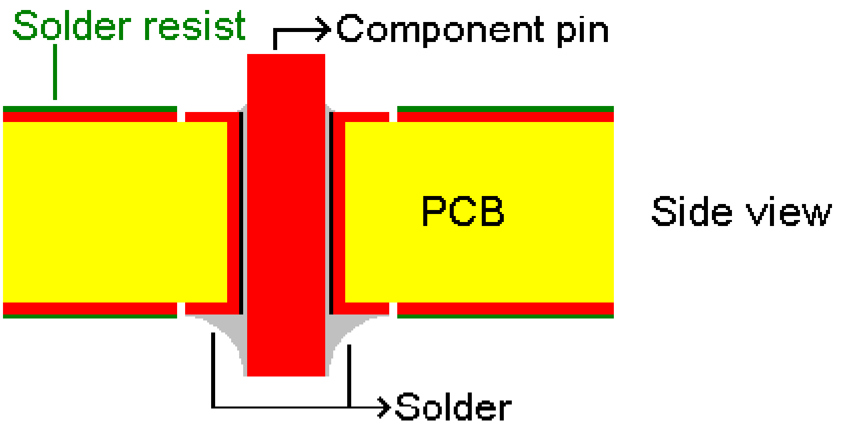
Distance

Through hole pin



In the above figure a through hole pin is shown.

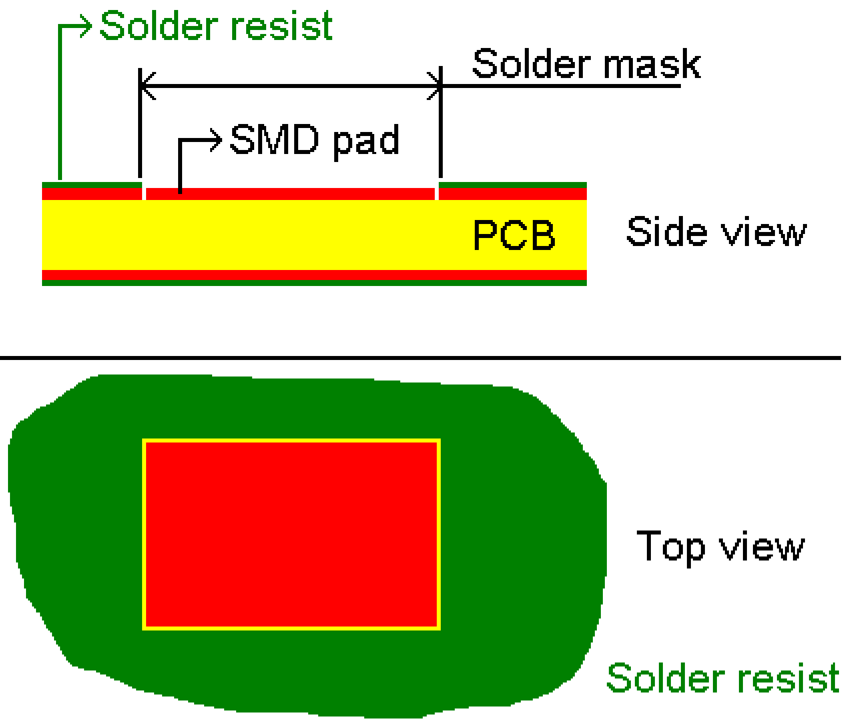
After soldering this through hole pin will look like



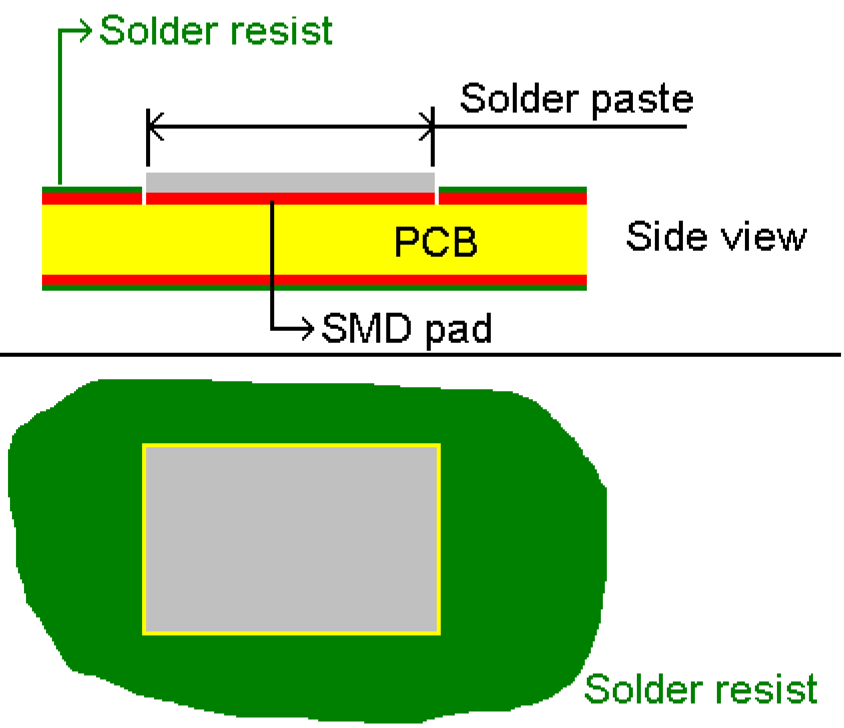
After the bottom side of the PCB has gone through a solder bath, all copper areas at the bottom of the PCB are soldered. The copper areas on the bottom PCB side that are covered with solder resist, are not soldered.

Usually the anti pad for the solder mask is 8 mil greater than the copper pad. The size of +8 mil for the solder mask, is because of tolerances.

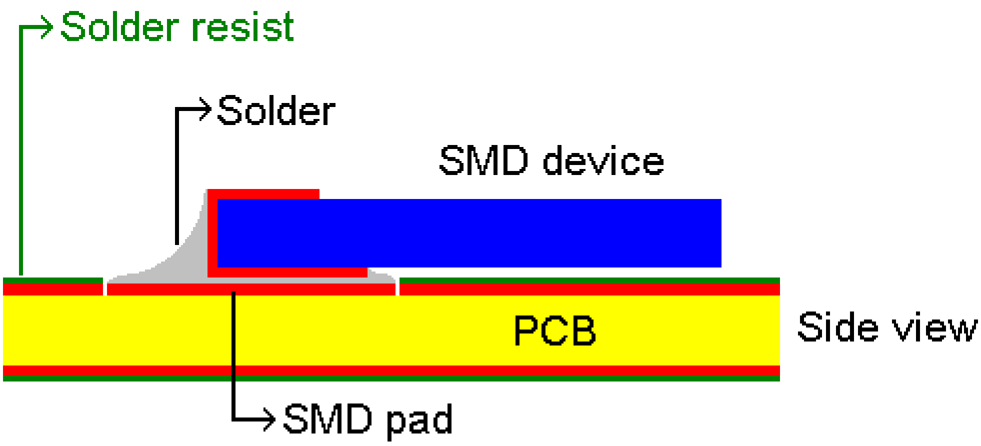
SMD pad



In the above figure a SMD pad is shown.



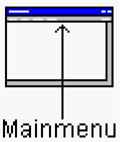
In the above figure a SMD pad with solder paste is shown.



In the above figure a SMD pad is shown after soldering. By applying heat on top of the PCB, SMD devices will be soldered using the solder paste on the pad as the solder.
Usually the anti pad for the solder mask is 8 mil greater than the copper pad, and the paste pad is the same size. The size of +8 mil for the solder mask, is because of tolerances.

Edit

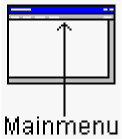
Thickness line/clearance

	Sub menu **Edit** menu item **Thickness line/clearance**

In the next dialogbox there are four items which can be changed:

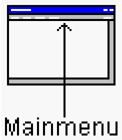
- Trace thickness
- Clearance
- Line thickness component outline
- Line thickness silkscreen

Set origin point geometry

	Sub menu **Edit** menu item **Move origin**

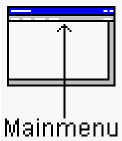
The origin of the geometry will be moved to the mouse position after pressing the **left mouse button**.

Set origin point geometry to center selected objects

	Sub menu **Edit** menu item **Set origin to center selected objects**

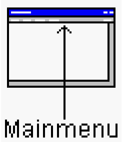
The origin of the geometry will be set to the center of the selected objects.

Set insertion point geometry

	Sub menu **Edit** menu item **Set insertion point**

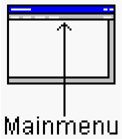
The insertion point of the geometry will be moved to the mouse position after pressing the **left mouse button**.

Set insertion point geometry to center selected objects

	Sub menu **Edit** menu item **Set insertion point to center selected objects**

The insertion point of the geometry will be set to the center of the selected objects.

Change geometry name

	Sub menu **Edit** menu item **Change geometry name**

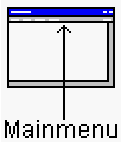
In the next dialogbox the geometry name (And also the filename) can be changed.

View

The following are the same as for the layout editor:

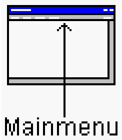
- Zoom in
- Zoom out
- Window based zooming
- Pan window
- Window based panning
- Return to previous view window
- Repaint
- Hide/view layers
- View whole design
- Change grid
- View/hide grid
- Zero relative cursor
- Deselect all
- Undo
- Redo

Change colors

	Sub menu **View** menu item **Change colors**

The color settings can be modified in the next dialogbox. The color settings will be copied into the **geom.ini** initialization file. This file is stored into the current geometries directory. To use those colors for new designs, copy this **geom.ini** file to main directory. Whenever a new design is created this **geom.ini** file in the main directory will be copied to the **pcb\shapes** subdirectory of the new design.

Load default colors

	Sub menu **View** menu item **Load default colors**

The default color settings will be loaded.

Programmable keys

The most important functions of the geometry editor have a short cut key (Accelerator). Those keys can be modified by editing the **geom.ini** file, section **[Keys]**.

Selection/deselection objects

To select an object, place the mouse cursor above the object, and press and hold the left mouse button. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **Replacement mode**, and the second selection mode is the **Adding selection mode**.

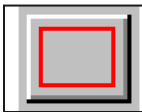
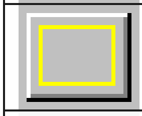
The **Replacement selection mode** means, every time a new selection rectangle is drawed the previous objects selected will be unselected. When pressing down the **shift** key together with the **left mouse button** it is possible to use more than one selection at a time.

The other selection mode is the **Adding selection mode**. In this mode every object which is selected stays selected, until the deselect all function is executed. To deselect an object press the **left mouse button** and place the selection rectangle around this object again.

To change the selection mode use the **Replacement** or **Appending** in the **Selection mode** section of the menu.

Add objects

Add rectangle objects

	Press **Add rectangle component outline** button
	Press **Add rectangle placement outline** button
 Keyboard	Press **r**
Menu  Mouse	**Add pad** -> **Rectangle** -> **Select layer** **Add rectangle** -> **Select layer**

A rectangle object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the rectangle parameters can be edited by hand. The first two parameters are the width, and height. The optional third and fourth parameter is the rectangle center. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

See also Add rectangle SMD pads with solder and paste mask
See also Add through hole pads with solder mask and drill hole

Add pad -> rectangle

A rectangle (solid) pad can be added on the following layers:

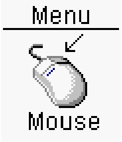
- Pad top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

Add rectangle

A rectangle (open) can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

Add circle objects

	**Add pad** -> **circle** -> **select layer** **Add circle** -> **select layer**

A circle object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the circle parameters can be edited by hand. The first parameter is the diameter. The optional second and third parameter is the circle center. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

See also Add circle SMD pads with solder and paste mask
See also Add through hole pads with solder mask and drill hole

Add pad -> circle

A circle (solid) pad can be added on the following layers:

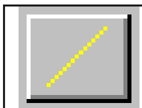
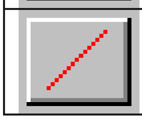
- Pad top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

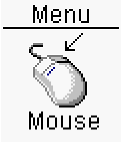
Add circle

A circle (open) can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

Add line objects

	Press **Add line component outline** button
	Press **Add line placement outline** button

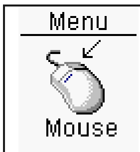
	Press **l**
	**Add line** -> **select layer**

A line object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the line parameters can be edited by hand. As many as 16 points (15 lines) can be edited. In addition, one point can be edited for the starting point of the line. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position.** The coordinates typed in will be used with the current units (dimension).

A line can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

Add arc objects

	**Add arc** -> **select layer**

An arc object will be added. When the **spacebar** is pressed, a dialogbox will popup, and the arc parameters can be edited by hand. The first parameters are the diameter. The optional second and third parameter is the arc center. The optional fourth and fifth parameter is the first radial ending point. The optional sixth and seventh parameter is the second radial ending point. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position.** The coordinates typed in will be used with the current units (dimension).

An arc can be added on the following layers:

- Silkscreenlayer
- Component outline layer
- Placement outline layer

Add text objects

Keyboard	Press **t** (Text on component outline layer)
Keyboard	Press **T** (Text on the silkscreen layer)
Menu Mouse	**Add text** -> **select layer**

A text object will be added. In the next dialogbox the text can be entered. In addition the textheight can be edited. After pressing the **OK** button the text can be placed. When the **spacebar** is pressed, a dialogbox will popup, and the text placement point can be edited by hand. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position.** The coordinates typed in will be used with the current units (dimension).

Text can be added on the following layers:

- Silkscreenlayer
- Component outline layer

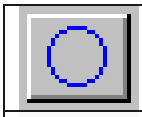
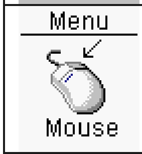
Add trace

Menu Mouse	**Add trace top layer** -> **Copper** **Solder mask** **Paste mask** **Add trace bottom layer** -> **Copper** **Solder mask** **Paste mask**

A trace with the current trace width and clearance and layer will be added.

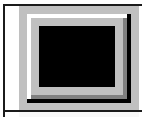
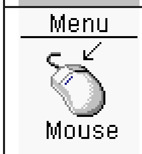
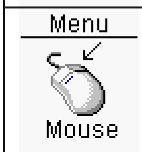
See also Thickness line/clearance

Add drill

	Press **Add unplated drill hole** button
	**Add drill** -> **plated** **Add drill** -> **unplated**

A drill hole (plated/unplated) will be added. When the **spacebar** is pressed, a dialogbox will popup, and the drill hole parameters can be edited by hand. The first parameter is the diameter. The optional second and third parameter is the drill hole center. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Add rectangle SMD pads with solder and paste mask

	Press **Add rectangle SMD pads** button
	**Add pad** -> **SIL SMD rectangle pads**
	**Copy special** -> **Add SIL SMD based on selected objects**

If a number of rectangular SMD pads (Pads,paste mask and solder mask) on an equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clockwise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pad can be edited by hand. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

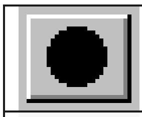
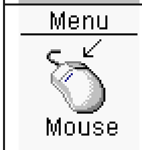
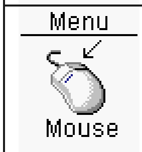
The function **Add SIL SMD based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

The following parameters can be changed:

Pad width and height
Pitch
Nr pads
Pad width and height solder paste
    If the solder paste pad is not necessary, fill the X or Y parameter with zero.
Pad width and height solder mask
    If the solder mask pad is not necessary, fill the X or Y parameter with zero.
Top/bottom layer
    The pads will be placed on the top or bottom layer
Clearance
    The initial clearance is the clearance used for this geometry.
Startpin
    The startpin consists of two editboxes. The first editbox (optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.

See also SMD pad

Add circle SMD pads with solder and paste mask

	Press **Add round SMD pads** button
	**Add pad** -> **SIL SMD rectangle pads**
	**Copy special** -> **Add SIL SMD based on selected objects**

If a number of circular SMD pads (Pads,paste mask and solder mask) on an equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clockwise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pad can be edited by hand. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

The function **Add SIL SMD based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

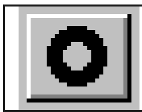
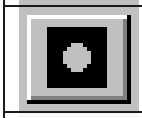
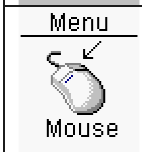
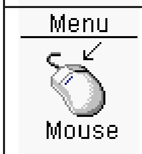
The following parameters can be changed:

Pad diameter

Pitch
Nr pads
Pad diameter solder paste
    If the solder paste pad is not necessary, fill the parameter with zero.
Pad diameter solder mask
    If the solder mask pad is not necessary, fill the parameter with zero.
Top/bottom layer
    The pads will be placed on the top or bottom layer
Clearance
    The initial clearance is the clearance used for this geometry.
Startpin
    The startpin consists of two editboxes. The first editbox (optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.

See also SMD pad

Add through hole pads with solder mask and drill hole

	Press **Add round through hole** button
	Press **Add square through hole** button
	**Add pad** -> **SIL through hole pads**
	**Copy special** -> **Add SIL SMD based on selected objects**

If a number of circular through hole pads (Pads,paste mask) on an equal distance needs to be included, this function can do the job. In the next dialogbox all the necessary parameters can be entered. After pressing the **OK** button the pads can be placed. When pressing the **right mouse button** the pads will rotate 90 degrees counter clockwise. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the position of the first pin can be edited by hand. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

The function **Add SIL based on selected objects** will do the same, but the dialogbox parameters will already be filled, with the parameters of a **selected** pad.

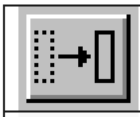
The following parameters can be changed:

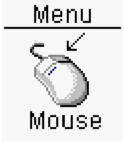
Pad size
Pitch
Nr pins
Pad size solder mask
    If the solder mask pad is not necessary, fill the parameter with zero.
Diameter anti power pad
    The anti power pad will always be a circle.
    If the anti power pad is not necessary, fill the parameter with zero.
Diameter inner pad
    The inner pad will always be a circle.
    If the inner pad is not necessary, fill the parameter with zero.
Drill hole
Clearance
    The initial clearance is the clearance used for this geometry.
Pintype
    The pin can be a square or a circle.
Startpin
    The startpin consists of two editboxes. The first editbox (optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.

See also Through hole pin

Change objects

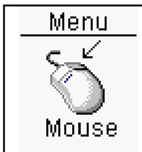
Move objects

	Press **Move** button
	Press **m**

	**Move**

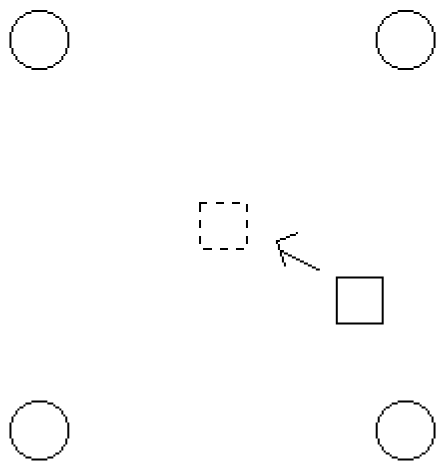
Move **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the endpoint parameters can be edited by hand. The endpoint coordinates will be the center of the selected objects. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Move objects (special)

	**Special move/centering** -> **Mark center selected objects** **Special move/centering** -> **Move objects centered to previous selected objects**

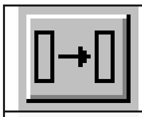
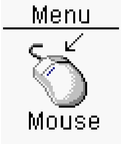
Move **selected** objects on a special way.

An example:



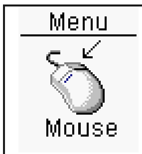
Suppose the square has to be moved to the center of the four corner circles. This can be achieved by selecting the four circles. When the four circles have been selected use the function **Mark center selected objects**. After this execution of this function select the rectangle, and use the function **Move objects centered to previous selected objects**.

Copy objects

	Press **Copy** button
	Press **c**
	**Copy**

With this function **selected** objects can be copied to a new location. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. When the **spacebar** is pressed, a dialogbox will popup, and the endpoint parameters can be edited by hand. The endpoint coordinates will be the center of the selected objects. When the first character typed is a @ the coordinates will be relative against the **Relative (grid)position**. The coordinates typed in will be used with the current units (dimension).

Copy objects to a different layer

	**Copy to other layer** -> **Copper top (Only traces/rectangles/circles)** **Copper bottom (Only traces/rectangles/circles)** **Silkscreen (Only lines/circles/arcs/texts)** **Component outline (Only lines/circles/arcs/texts)** **Paste mask top (Only traces/circles/rectangles)** **Paste mask bottom (Only traces/circles/rectangles)** **Solder mask top (Only traces/circles/rectangles)** **Solder mask bottom (Only traces/circles/rectangles)** **Drill plated (Only circles)** **Drill unplated (Only circles)** **Inner pad (Only circles)** **Power pad (Only circles)**

With this function **selected** objects can be copied to the specified layer. Selected objects on the same layer as the specified layer will not be copied.

Copy on multiple coordinates

Menu Mouse	Copy on multiple coordinates

In the next dialogbox a maximum of 16 coordinates (x,y) can be typed. At every coordinate the **selected** objects will be copied. This can be handy when a range of the same pins should be added, on many different coordinates.

Delete objects

	Press **Delete** button
Keyboard	Press **Del**
Menu Mouse	**Delete**

Delete **selected** objects.

Rotate objects

Keyboard	Press **R**
Menu Mouse	**Rotate**

Rotate **selected** objects 90 degrees counter clockwise.

Mirror objects

Menu Mouse	**Mirror X** **Mirror Y**

Mirror **selected** objects in X or Y direction.

Change circle objects

Menu Mouse	**Change diameter circles**

The diameter of **selected** circles can be changed into a new value typed in the following dialogbox.

Change rectangle objects

Menu Mouse	**Change width/height rectangles**

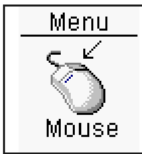
The width and height of **selected** rectangles can be changed into a new value typed in the following dialogbox.

Change text

Menu Mouse	**Change text**
Keyboard	Press **e**

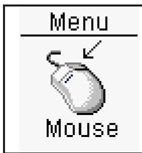
The **selected** text can be changed in the following dialogbox.

Change text height

	**Change textheight**

The textheight of **selected** texts can be changed into a new value typed in the following dialogbox.

Change trace width

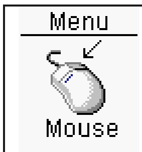
	**Change trace width**

The line width of **selected** objects can be changed into a new value typed in the following dialogbox.

Objects:

- Copper top/bottom layer
- Solder mask top/bottom layer
- Paste mask top/bottom layer

Change line width

	**Change line width**

The line width of **selected** objects can be changed into a new value typed in the following dialogbox.

Objects:

- component outline layer
- silkscreen layer

- geometry name

Change clearance

Menu Mouse	**Change clearance**

The clearance of **selected** objects can be changed into a new value typed in the following dialogbox.

Unselect objects

Menu Mouse	**Unselect** -> **Select object layer**

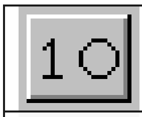
Unselect objects.

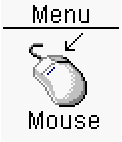
Select only

Menu Mouse	**Select only** -> **Select object layer**

Select only objects.

Assign objects to pin

	Press **Assign objects to pin** button
Keyboard	Press **a**

	**Assign objects to pin**

Select objects will be assigned to a pinnumber (pinname). In the next dialogbox the pinnumber (name) can be selected or edited.
It is possible to assign as many objects as necessary to a pinnumber (name).
The maximum length of a pinnumber (name) is 9 characters.

Initialization file geom.ini

The initialization file **geom.ini** is used to save some designs parameters. The file will be in the same directory as the geometries.

[Settings]

WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
Units	(0 = mils,1 = mm)
GridSize	The gridsize (10nm units)
DrawGrid	0 = FALSE,1 = TRUE
DrawAreaFills	0 = FALSE,1 = TRUE
DrawAreaFillWithHatches	0 = FALSE,1 = TRUE
DrawClearances	0 = FALSE,1 = TRUE
DrawCompOutline	0 = FALSE,1 = TRUE
DrawConnections	0 = FALSE,1 = TRUE
DrawDrills	0 = FALSE,1 = TRUE
DrawInnerPads	0 = FALSE,1 = TRUE
DrawTopPads	0 = FALSE,1 = TRUE
DrawBottomPads	0 = FALSE,1 = TRUE
DrawCompPlacement	0 = FALSE,1 = TRUE
DrawSilkScreen	0 = FALSE,1 = TRUE
DrawObjects	0 = FALSE,1 = TRUE
DrawVias	0 = FALSE,1 = TRUE
DrawViaClearances	0 = FALSE,1 = TRUE
DrawCompReference	0 = FALSE,1 = TRUE
DrawCompValue	0 = FALSE,1 = TRUE
DrawTwoTryingTraces	0 = FALSE,1 = TRUE
SelectionMode	0 = replacement, 1= appending
Layer0	Draw bottom layer 0 (0 = FALSE,1 = TRUE)
Layer1	Draw layer 1 (0 = FALSE,1 = TRUE)
Layer2	Draw layer 2 (0 = FALSE,1 = TRUE)
…	…
Layer31	Draw layer 31 (0 = FALSE,1 = TRUE)
Grid0	Gridsize definition 0 (10nm units)
Grid1	Gridsize definition 1 (10nm units)
…	…
Grid29	Gridsize definition 29 (10nm units)
TraceWidth0	Trace width definition 0 (10nm units)
TraceWidth1	Trace width definition 1 (10nm units)

…	…
TraceWidth29	Trace width definition 29 (10nm units)
ClearanceWidth0	Clearance width definition 0 (10nm units)
ClearanceWidth1	Clearance width definition 1 (10nm units)
…	…
ClearanceWidth29	Clearance width definition 29 (10nm units)
BackGroundColor	24 bit RGB color (Stored as 32 bit)
SilkScreenColor	24 bit RGB color (Stored as 32 bit)
CompOutlineColor	24 bit RGB color (Stored as 32 bit)
ShapePlacementOutLineColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillColor	24 bit RGB color (Stored as 32 bit)
ShapePinsDrillUnplatedColor	24 bit RGB color (Stored as 32 bit)
ShapePinsCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapePinsSoldSideColor	24 bit RGB color (Stored as 32 bit)
ShapeInnerPadColor	24 bit RGB color (Stored as 32 bit)
ShapePasteMaskCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapePasteMaskSoldSideColor	24 bit RGB color (Stored as 32 bit)
ShapeSoldMaskCompSideColor	24 bit RGB color (Stored as 32 bit)
ShapeSoldMaskSoldSideColor	24 bit RGB color (Stored as 32 bit)
GeomNameColor	24 bit RGB color (Stored as 32 bit)
PowerPadColor	24 bit RGB color (Stored as 32 bit)
ClearanceColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)

Index

I

L

M

N

O

P

R

S