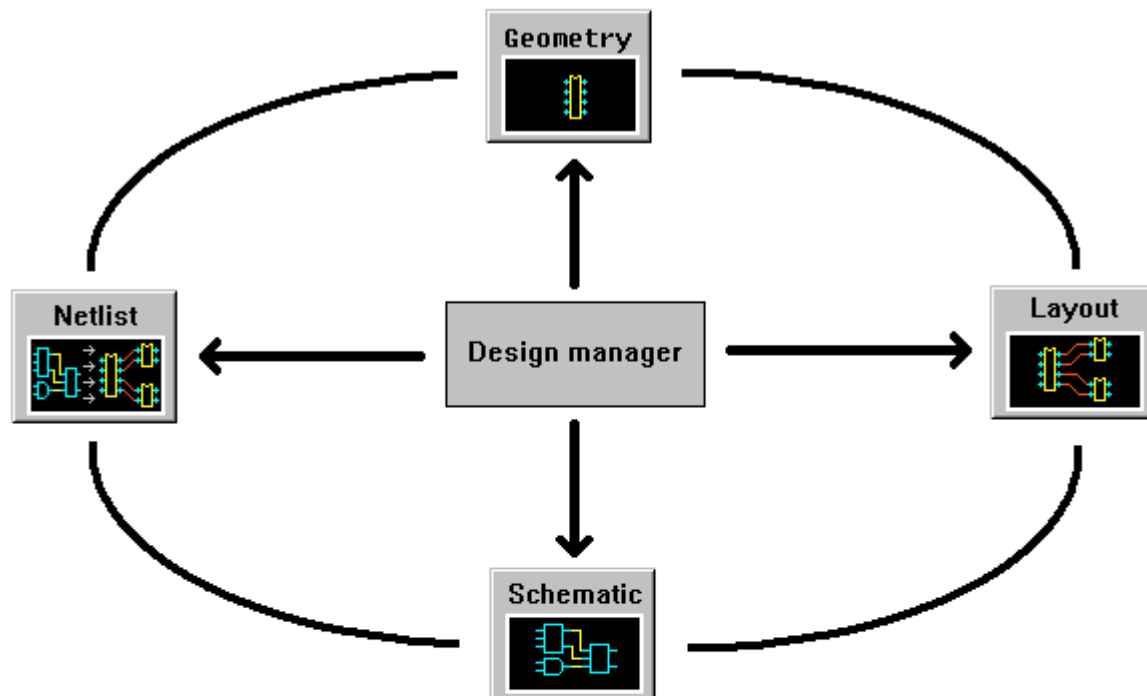


PCB elegance 3.5



PCB elegance

Version 3.5

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

Printing history

Version 1.2 January 1999
Version 1.3 January 2000
Version 1.4 October 2001
Version 1.5 December 2003
Version 2.0 September 2005
Version 2.1 May 2006
Version 2.2 February 2008
Version 2.3 May 2010
Version 2.4 May 2012

<http://www.pcbelegance.com>

Table of contents

Introduction	1
Installation	2
Requirements	2
Install.....	2
Installation on a network.....	2
Deinstall	3
Design manager	5
Introduction	6
How to make a simple PCB.....	6
Create a new project	6
Annotate schematic.....	8
Create netlist.....	9
Create layout.....	9
Import netlist	9
Place components.....	9
Route traces	10
Check PCB	11
Create output files	11
Generate gerber output plots:	11
Generate penplot output:	11
Output to a printer	11
File	13
Make new design	13
Open	14
Close.....	14
Copy symbols/geometries locally	14
Directory structure design	14
Edit.....	17
Open sheet	17
Edit symbol	17
Design settings.....	17
Annotation.....	18
Back annotation	19
Create netlist.....	19
Bill of materials.....	20
Check.....	21
Print all sheets	21
Print all sheets to PDF.....	21
Edit design settings	22
Change symbols	22
Conversion ORCAD schematic/libraries	22
Conversion ORCAD schematic:	23
Conversion ORCAD library:	23
Orcad sdt.cfg example	23
Start layout editor	24
Gerber viewer.....	24
Start geometry editor.....	24
Start library manager symbols.....	25
Start library manager geometries	25
Layout editor	27
File	28

Open	28
Save.....	28
Save as	28
Print screen	29
Make new layout	29
Design rules when using a printer for the plot outputs	29
Importing components/netlist	30
Updating components/netlist	30
Plot output to gerber format.....	31
Thermal relief	31
Export to PDF	32
Export to bitmap	33
Save layout as a geometry	33
Penplot output.....	33
Plot output to printer	34
Export component position	34
Output netlist.....	34
Reload geometries	35
Import DXF files	35
Export to DXF file	35
Import a bitmap	36
Import a gerber file	36
Import component positions	36
Edit.....	37
Move entire PCB	37
Change design rules	37
Zero relative cursor	37
Center view on component.....	38
Goto xy	38
Via definition	38
Component protection	39
Undo	39
Redo	39
Selection/deselection objects	40
Make selections in dialog listboxes.....	40
Deselect all	40
Info on selected objects.....	41
View	41
Hide/view layers	41
Zoom in.....	42
Zoom out.....	42
Window based Zooming.....	43
Pan window.....	43
Window based panning.....	43
Return to previous view window	43
Repaint	44
View whole design	44
Measurement	45
Change colors	45
Load default colors.....	45
Programmable keys	45
Options	45
Change units.....	47
View/hide grid	47
Components.....	47
Move components.....	48
Move components by reference	48
Rotate components	49

Move component to top/bottom layer	49
Regroup components	50
Align components	50
Edit geometry	50
Change geometry	50
Change component parameters	51
Protect components	51
Copy component layer objects to the objects layer	52
Edit schematic containing reference	52
Component selections	52
Component selections by list	52
Move multiple components	53
Nets	53
Change design rules net	53
Highlight/unhighlight nets	54
Disable connections nets	54
Hide connections nets	54
Highlight visible connections	55
Unselect traces/vias nets	55
Delete traces/vias nets	55
Unhighlight all	55
View all connections	56
Hide all connections	56
Routing	56
Add trace	57
Trace drawing feature	57
Add via	58
Trace popup menu	58
Trace drawing all angle	58
Arc trace drawing (90 degrees)	58
Arc trace drawing (45 degrees)	59
Display clearance	59
Display two trying traces	59
Display via option	59
Finish trace	60
Highlight/unhighlight net	60
Switch to another layer	60
Delete trace	60
Goto previous trace segment	61
Change trace width	61
Change clearance	61
Change cross hair of the mouse cursor	62
Change design rules net	62
Add extra trace	62
Start routing with the shortest net	62
Show next connection	63
Change traces/vias	63
Move traces/vias	64
Copy traces/vias	64
Copy traces/vias to clipboard	65
Copy traces/vias from clipboard	65
Select only	65
Change trace width	65
Change clearance traces/vias	66
Change via	66
Change design rules net	66
Calculate length trace	67
Swap traces/vias two nets	67

Delete traces/vias net selected trace	67
Delete	67
Drag one trace	68
Dragging traces/vias/components	68
Check.....	69
Check connectivity	69
Check design rules.....	70
View design rule errors.....	70
Show next design rule error/warning	70
Powerplanes	70
Add powerplane	71
Remove powerplane	71
Cut from powerplane	72
Change powerplane	72
Areafills	73
Add areafill	73
Add areafill inside a powerplane.....	74
Delete areafill	75
Copy areafill	75
Move areafill.....	75
Mirror X areafill.....	76
Mirror Y areafill.....	76
Stretch areafill	76
Change areafill	76
Change clearance areafill.....	76
Cut from areafill.....	77
Merge areafills.....	77
Change areafill	77
Add to areafill	78
Rebuild areafill	78
View vertices areafill.....	78
Copy start polygon to info4 layer	78
Modify component references	79
Modify component values.....	79
Special objects	80
Add special objects	81
Lines	81
Rectangles.....	81
Circles.....	81
Arcs	81
Texts.....	82
Polyline	82
Polygons.....	82
Arrows.....	82
Dimension.....	82
Change special objects	82
Move.....	82
Move to another layer	83
Copy	83
Copy to another layer.....	83
Copy objects array	83
Copy objects polar	84
Rotate	84
Delete	84
Scale.....	85
Change circle diameter	85
Change arc width/height	85
Change arc angles.....	85

Change rectangle width/height.....	85
Change line thickness	86
Change text	86
Change text height.....	86
Mirror X	86
Mirror Y	87
Cut from polygon.....	87
Calculate area polygon	87
Assign net to objects	88
Assign no net to objects	88
Gate/pin swap	88
Schematic link	89
Schematic editor	90
File	91
New sheet	91
New symbol	91
New sheetsymbol	91
Open	92
Save.....	92
Save as	92
Print	93
Export to BMP	93
Export to PDF	93
Export to DXF	93
Import from DXF.....	93
View	94
Change colors	94
Load default colors (Black background)	94
Load default colors (Grey background)	94
Programmable keys	95
Selection/deselecting objects	95
Component/net info popup display	95
Deselect all	96
View sheet/symbol options.....	96
Edit.....	96
Edit symbol parameters.....	97
Reference	97
Value	97
Part nr.....	97
Geometry	97
Part description	98
Package part nr.....	98
Placing option	98
Properties	98
Edit symbol	98
Protect symbols.....	98
Unprotect symbols	99
Edit gate/pin swap.....	99
Pin swap example	99
Gate swap example1	100
Gate swap example2	100
Edit pinbus reorder.....	100
Export text.....	101
Edit any text	101
Search for any text	102
Edit number of parts per package	102
Edit pinnumbers package parts	102
Edit symbolnames.....	103

Edit layout containing reference	103
Component selections by list	103
Multiple symbols	104
Clear references	104
Check sheet	104
Check symbol	104
Edit pin normal symbol	105
Edit sheet symbol pin	105
Edit pinbus	105
Change grid	106
Add objects	107
Add wire	107
Add bus	107
Add busconnection	108
Add external connection	108
Add netlabel	109
Add netlabel properties	109
Add incremental netlabels to wires	110
Add netlabel + wire	110
Add one pinnet mark	111
Add symbol	111
Add symbol on shortcut	112
Add component from database	112
Compmenu.txt	112
Comp.txt	113
Add other objects	113
Line	113
Add right pointed arrow	113
Add left pointed arrow	114
Add left/right pointed arrow	114
Add dimension	114
Rect	114
Rect (normal)	115
Circle	115
Arc	115
Text	116
Numbers incremental	116
Add pin	116
Add sheetsymbol pin	117
Add powerpin	117
Add pinbus	118
Hierarchical designs	118
Open subsheet	119
Open sheetsymbol	119
Goto higher sheet	119
Change objects	120
Move objects	120
Drag objects	120
Rotate objects	120
Scale objects	121
Mirror objects	121
Copy objects	121
Align text objects left/right	122
Change text height	122
Change line thickness	122
Copy objects to clipboard	122
Paste objects from clipboard	123
Delete objects	123

Unselect objects	123
Select only	124
Edit symbol	124
Reload symbols.....	124
Goto x,y	124
Geometry editor	126
File	127
Open	127
Save.....	127
Save as.....	127
Print	127
Import DXF files	128
Export to DXF file	128
Import a bitmap	128
Make new geometry	129
New DIP geometry.....	129
New Quad flatpack geometry	130
New BGA geometry	130
New PGA geometry	131
New SOIC geometry	131
Through hole pin	133
SMD pad	135
Design rules pad	136
Edit.....	136
Thickness line/clearance	137
Set origin point geometry	137
Set origin point geometry to center selected objects	137
Set insertion point geometry.....	137
Set insertion point geometry to center selected objects.....	138
Change geometry name.....	138
Number of copper layers	138
Check geometry	138
View	139
Change colors	139
Load default colors	139
Programmable keys	139
Selection/deselection objects	140
Info on selected objects.....	140
View vertices polygon.....	140
Measure distance.....	141
Measurement	141
Add objects	141
Add rectangle objects.....	141
Add circle objects	142
Add line objects.....	142
Add arrow/dimension objects	143
Add arc objects	143
Add text objects.....	144
Add polyline	144
Add polygon	144
Add drill	145
Add rectangle SMD pads with solder and paste mask.....	145
Add circle SMD pads with solder and paste mask	146
Add through hole pads with solder mask and drill hole	147
Change objects	148
Move objects.....	148
Move objects (special).....	149
Copy objects	149

Copy objects to a different layer	150
Move objects to a different layer.....	150
Copy on multiple coordinates	150
Delete objects	151
Rotate objects	151
Mirror objects	151
Scale objects.....	152
Change circle objects	152
Change rectangle objects.....	152
Change diameter arc objects.....	152
Change angle arc objects.....	153
Change text.....	153
Change text height	153
Change line width.....	153
Change clearance	153
Convert lines into polygon	154
Cut from object.....	154
Merge objects to polygon	154
Unselect objects.....	154
Select only	155
Assign objects to pin	155
Index.....	156

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
- Processor with SSE2 instruction set (pentium 4 or higher,AMD althon 64 or higher)
- 128 Mb preferred
- 60 Mb haddisk space
- Operating system
 - Windows 2000
 - Windows XP
 - Windows Vista (32/64-bit)
 - Windiws 7 (32/64-bit)

Install

To install this software package run the executable pcb_eleg35.exe. This executable can be downloaded from the website (www.pcbelegance.com). 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.

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 (/p directory) can be specified for the project directory.

Example 1:

```
design.exe /p d:\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 name 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 name 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 menu 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

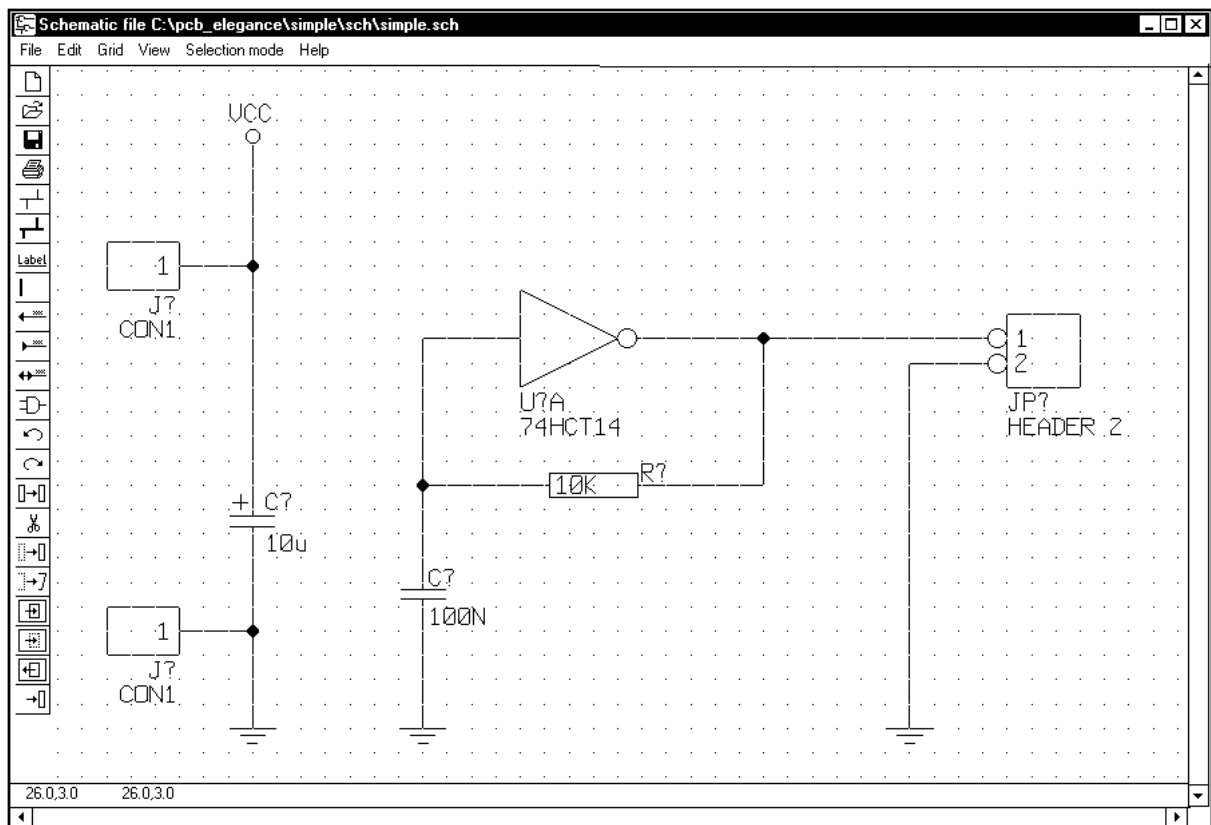
Edit text.

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.

Design manager

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

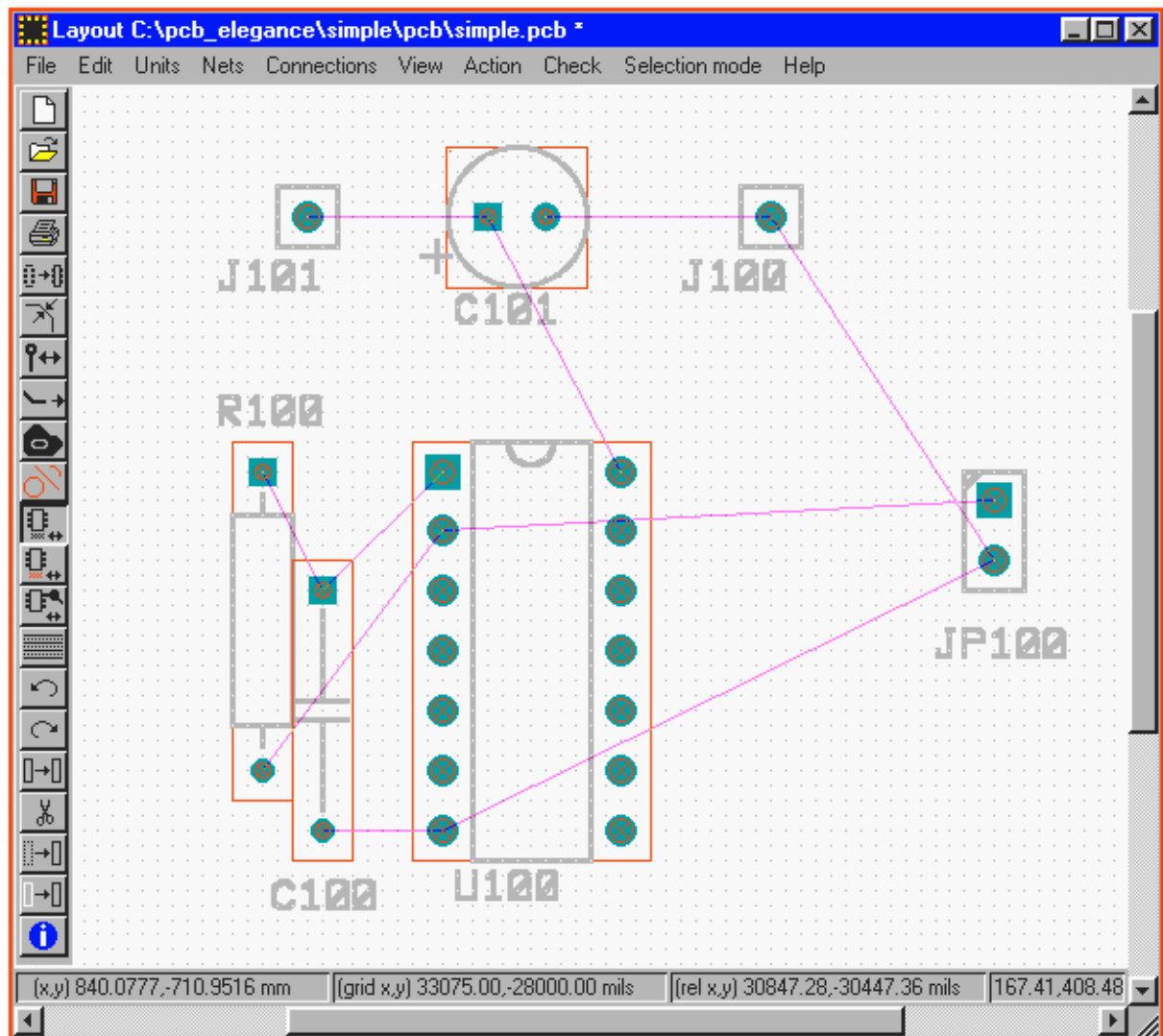
After importing the netlist a number of components should be placed around the PCB. The components should be moved inside the PCB.

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 be 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** from 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** from 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.

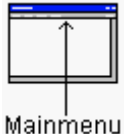
Output to a printer

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

 Mainmenu	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. . In the **Top sheet name** editbox the first sheetname should be filled in. In the editbox **User symbol libraries** own symbol libraries can be put in, if required. In the editbox **User geometry libraries** own geometry libraries can be put in, if required. There is also an option for disabling one pinnet checks and for saving symbols/geometries locally. Locally means that the symbols used will be stored into the designs **sym** directory, and the geometries ised will be stored into the designs **pcb\shapes** directory. 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 projects/executables directory. (If exists)

File	sch.ini	Settings schematic editor
File	viewplot.ini	Settings gerber viewer

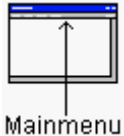
The file sch.ini and viewplot.ini will be copied from the projects/executables directory. (If exists)

Directory	backup	
Directory	pcb	Layout file (*.pcb)
Directory	pcb\dxg	DXF output files (*.dxf)
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 projects/executables directory. (If exists)

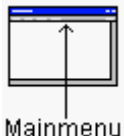
Directory	pcb\shapes	Local geometries file (*.shp)
Directory	pcb\shapes\backup	Backup local geometries files
Directory	sch	Schematic files (*.sch)
Directory	sch\backup	Backup schematic files
File	sch\<top sheet name>.sch	
Directory	sym	Local (sheet)symbol files
Directory	sym\backup	Backup sym files

Open

 Mainmenu	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

 Mainmenu	Sub menu File menu item Close design
---	--

Closes the current design.

Copy symbols/geometries locally

 Mainmenu	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

Design manager

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\dxs	(DXF subdirectory)
--------------	---------	--------------------

<Design name>.dxf	Exported DXF file
-------------------	-------------------

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
Info3.ger	Gerber output file info layer3

Design manager

Info4.ger	Gerber output file info layer4
-----------	--------------------------------

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
Info3.hgl	Penplot output file info layer3
Info4.hgl	Penplot output file info layer4

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. In this pulldown menu there are also options to open a sheet with the view centered on a reference, or to open the layout with the view centered on a reference.

Edit symbol

	Press Symbol editor button
---	-----------------------------------

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

Design settings


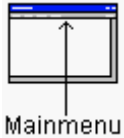
	Sub menu Edit menu item Design settings
---	---

Modification of :

- Layout name
- Top sheet name
- User symbol libraries
- User geometry libraries
- Usage of partnumbers

- Disable one pinnet check
- Save symbols/geometries locally

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.

It is also possible that a sheet has a start number for annotation of the references.

See also [View sheet/symbol options](#) when editing schematics.

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, and the maximum of references per sheet.

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. If there is a need for more than 100 references for a group (resistors/capacitors), increase the number of references per sheet.

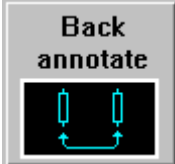
Appending annotation (Standard numbering)

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.

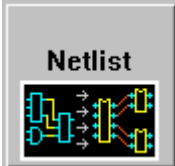
Back annotation

	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 will be placed in the designs **pcb** subdirectory. Also the gate/pin swap info file **pcb\gatepin.swp** will be generated. Also saved in the netlist are component and net properties.

The format of a component property is:
(property_name,"property_value")

When a component should not be placed (Not placed marked in the schematic) in the BOM, the following property will be used:
(MULTI_ASSY,"NP")

The format of a net property is:
(net_property_name,"net_property_value")

For the layout editor the TRACEWIDTH and CLEARANCE net property can be used. A few examples:
(TRACEWIDTH,"8 mil")
(CLEARANCE,"8 mil")

The standard tracewidth and clearance will be 8 mil in this example. For the units the following strings can be used:

8 mil

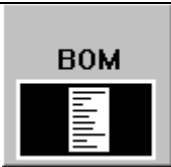
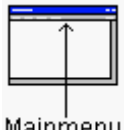
8 mils

8 th

2.5 mm

0.008 inch

Bill of materials

	Press BOM button
 Mainmenu	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. The columns in each line can also be selected. Possible columns are:

- Position
- Reference
- Value
- Geometry
- PartNr
- Description
- Extra component properties (optional)

The columns can be shifted horizontal using the two arrow buttons. With the **delete** button a column can be removed, and with the **Add** button a new column can be inserted. If there components marked

with a placing option a component property **MULTI_ASSY = NP** will be used. This MULTI_ASSY property will be added to the possible selectable columns.
The filename of this Bill Of Materials is **component.txt** and **component.tdl**. The **component.tdl** file can be used to import in a spreadsheet.

Bill of materials

In this Bill Of Materials components are summed and listed. The filename of this Bill Of Materials is **<design-name>.bom** and **<design-name>.tdl**. The **<design-name>.tdl** file can be used to import in a spreadsheet. There are three buttons which control which columns are to be used.
pos,value,#,geometrie

The columns used are Position,value,nr components and geometrie.

pos,value,#,geometrie,part nr

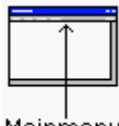
The columns used are Position,value,nr components, geometrie and part nr.

pos,value,#,geometrie,part nr,description

The columns used are Position,value,nr components, geometrie, part nr and description.


There is also an option for sorting on value or part nr if applicable.

Check

 Mainmenu	Sub menu Edit menu item Check schematics
---	--


Checks the schematics of the design for errors.

Print all sheets

 Mainmenu	Sub menu File menu item Print all sheets
---	--

All the schematics of the current design will be printed. The top sheet will be printed first.

Print all sheets to PDF

 Mainmenu	Sub menu File menu item Print all sheets to PDF
---	---

Print all the schematics to one PDF file.


Edit design settings

 Mainmenu	Sub menu Edit menu item Design settings
---	---

Modification of :

- Layout name
- Top sheet name
- User symbol libraries
- User geometry libraries
- Disable one pinnet check
- Save symbols/geometries locally

Change symbols

 Mainmenu	Sub menu Edit menu item Change components
---	---

In the next dialogbox values,geometry,description,properties,part number from components can be changed for all sheets at once.

For the search there are the following options:

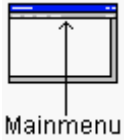
- Symbol name
- Value
- Geometry
- Part number
- Add property
- Change property

For the replace every item can be replaced with the entered string if the edit box has been changed.

There will be a warning whenever symbolname(s) change.

This operation can not be undone.

Conversion ORCAD schematic/libraries

 Mainmenu	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 number 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'
```


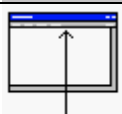
If necessary such a sdt.cfg file can be created. The lines containing the strings PLIB and LIB are important, and the first line for identification.
The PLIB string will contain the library directory, and the LIB strings the individual libraries in the library directory.

Start layout editor

 The icon for the Layout editor button, showing a circuit board layout with yellow and green components and traces.	Press Layout editor button
--	-----------------------------------

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

Gerber viewer

 The icon for the View gerber button, showing a blue and orange circuit board layout.	Press View gerber button
 The icon for the Main menu, showing a window with an upward arrow. Mainmenu	Sub menu File menu item Gerber viewer


The gerber viewer VIEWPlot will be started, and loaded with the latest generated gerber/drill files.

Start geometry editor

 The icon for the Geometry editor button, showing a circuit board layout with yellow and green components and traces.	Press Geometry editor button
--	-------------------------------------

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

Start library manager symbols

 Mainmenu	Sub menu Library manager symbols
---	---

Start the library manager for the schematic symbols.

Start library manager geometries



 Mainmenu	Sub menu Library manager geometries
--	--

Start the library manager for the geometries.

Layout editor



File

Open

 Mainmenu	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

 Mainmenu	Sub menu File menu item Save
	Press Save button

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

Save as

 Mainmenu	Sub menu File menu item Save as Sub menu File menu item Save as version 2.5
---	--

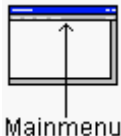

Save as:

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

Save as version 2.5:

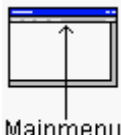
Saves the current layout file in the version 2.5 format.

Print screen

 Mainmenu	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

 Mainmenu	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.

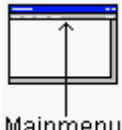
Design rules when using a printer for the plot outputs

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

 <p>Mainmenu</p>	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.

The layout editor also supports net properties. There are two properties which can be used by the layout editor: TRACEWIDTH and CLEARANCE. The properties have the following format:
(TRACEWIDTH,"<value>")
(CLEARANCE,"<value>")

Examples of the value are: "8 mil" or "0.2 mm".

The TRACEWIDTH and CLEARANCE values will overrule the standard value for the net.

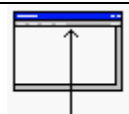
Updating components/netlist

 <p>Mainmenu</p>	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

 <p>Mainmenu</p>	Sub menu File menu item Output gerber/drill
---	---

Plot selected layers in the gerber format. In the next dialogbox the following items can be selected:

- Layers
- X mirroring
- Plotting board outline
- Layer numbering
- Plotting the drills as gerber
- Neutral file output for PCB testing
- Gerber output (RS274X) number format
- PCB specification (-> layers.txt file)

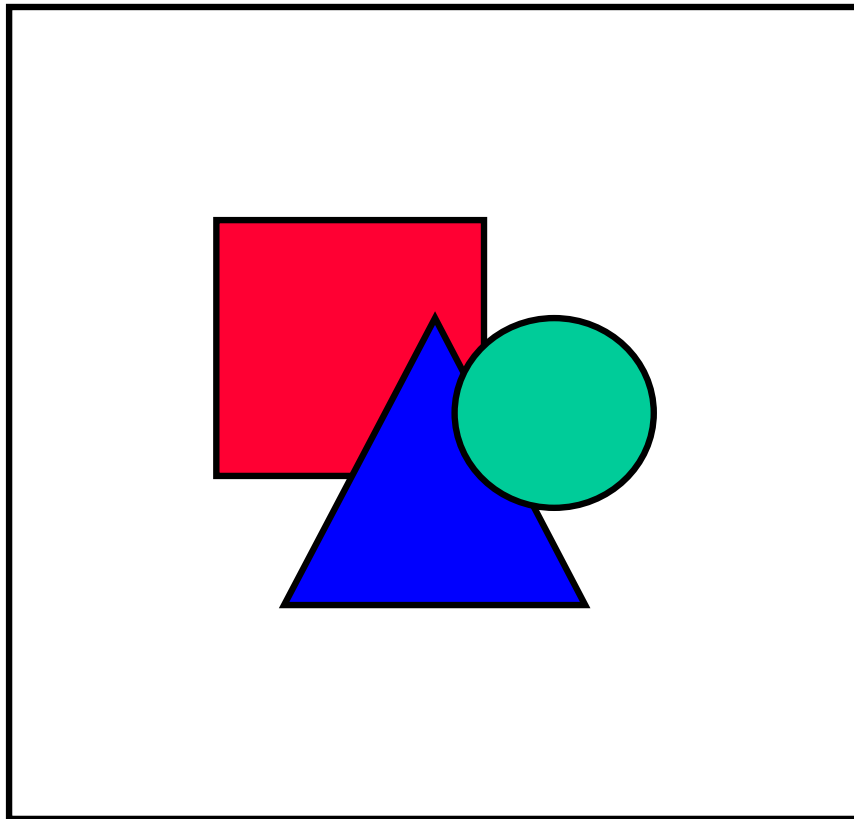
There are also two editboxes available. In those two editboxes (each four lines) some information about the PCB can be stored. This information will then be plotted additionally for each layer. Initial three macros are stored into the first editbox.

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

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 hole 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.

Export to PDF

 <p>Mainmenu</p>	Sub menu File item Export item Plotfiles as PDF
---	--

Plot **selected** layers in the PDF format. In the next dialogbox the layers, papersize, paper orientation and plotting board outline 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

The PDF file <design>.pdf will be generated in the **pcb** subdirectory.

!!! This feature depends on the current license.

Export to bitmap

 Mainmenu	Sub menu File item Export item Plotfiles as bitmap
--	---

Plot **selected** layers in the bitmap format. Every layer will be plotted in a separate bitmap file. In the next dialogbox the layers, bitmap compression, bitmap resolution and plotting board outline 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

For the bitmap resolution are two possibilities. The first is selection of dpi (Dots per inch), and the second is specification of the pixel (dot) size.

!!! Warning: If the files are saved in the “Non compressed” mode, the file size will be big.

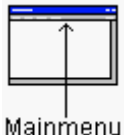
All the files will be generated in the **pcb\gerber** subdirectory.

Save layout as a geometry

 Mainmenu	Sub menu File menu item Save layout as a geomtry
--	--

Save the layout as a geometry. When multiple smaller and same layouts are necessary in a bigger layout this method can be used.


Penplot output

 Mainmenu	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.

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

 Mainmenu	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.

Export component position

 Mainmenu	Sub menu File item Export item Export component positions (Pick and place)
---	---

In the next dialogbox there are some options for export component positions:


- Units (mils/mm/inch)
- Either printing the component value or partnumber
- Printing only SMD components, or all components
- Printing a Not placed marker at the end of the text output line

First the top components will be printed, and than the bottom components.

There will be two files generated:

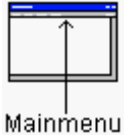
- Comp_pos.txt containing the output component positions as text lines
- Comp_pos.tdl containing the output component positions to be used in a spread sheet (Tab delimited list)

Output netlist

 Mainmenu	Sub menu File item Export item Netlist
---	---

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

Reload geometries

 Mainmenu	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.

Import DXF files

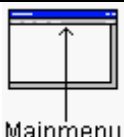
 Mainmenu	Sub menu File item Import item from DXF file Info layer 1 Info layer 2 Info layer 3 Info layer 4
---	---

Import DXF file into Info layer 1..4.

After selecting a DXF file from the next dialogbox, this DXF file will be scanned for its layers. In the following dialogbox one or more DXF layers can be selected. After including the objects into the design, a messagebox with the min/max values of the selected objects will be displayed. By using the scale function of the right mouse button menu, de scale of the objects can be adjusted.

!!! Dimensions are not supported.

Export to DXF file

 Mainmenu	Sub menu File menu item Export to DXF file
---	--

Exports the **selected** layers to a DXF file. In the next dialogbox the layers, X mirroring, filled objects and drawing board outline can be selected. For the silkscreen layers there is an option to plot the component references or the component values. Also the DXF export filename can be modified if necessary.

If the option "Objects filled" is not marked lines/traces/circles/arcs will be hairlines (Thickness is zero), circle/rectangle pads will exported with a contour.

Regardless of the "Objects filled" option, polygon/areafills will always be exported with a contour.

The DXF file will be generated in the **pcb\dxs** subdirectory.

Import a bitmap

 Mainmenu	Sub menu File item Import item Import bitmap
---	---

In the next dialogbox a window bitmap file (monochrome) will be requested. After **OK** the pixel size is requested in the next dialogbox. After pressing **OK** the (valid) windows bitmap will be imported on Info layer1.

Import a gerber file

 Mainmenu	Sub menu File item Import item Import gerber file
---	--

In the next dialogbox select a layer where the gerber file should be imported. Next a gerber file is requested in the following dialogbox. After selecting a file the gerber file will be imported on the selected layer. If the file is not a valid gerber file, an error will be shown. If the gerber file is ok, the objects found in the gerber file should be placed on the right position. For easier placing gerber objects on copper layer, snap functionality has been included. After clicking on the left mouse button, the mouse cursor will snap to the object closest to the cursor. If necessary the moving center can be changed by holding the shift key. After release the moving center will snap to the moving object closest to the mouse cursor. There are a number of rules how traces/pads are converted. First objects who after placement are not connected to any net, will be skipped. If necessary vias will be created from circle pads, if this circle pad does not overlap an already existing pad/via. Polygons in the gerber file are not supported.

!!! This feature depends on the current license.

If the gerber polygon are necessary there is a workaround. This workaround is loading the gerber file in the gerber viewer VIEWPlot. In VIEWPlot the vertices of polygon can be copied on the clipboard. Next the vertices on the clipboard can be used, when adding an areafill.

Import component positions

 Mainmenu	Sub menu File item Import item Import component positions
---	--

In the next dialogbox a textfile containing the component positions should be specified. Every textline should contain the name, location, layer and rotation of a component. In the next dialogbox the columns containing name, location, layer and rotation of a component can be modified. Also the units and how many lines should be skipped can be modified. The units can be modified using the units button or by specifying a value. (For example 0.012 mm or 0.04 inch or 400 mils) There are two

predefined buttons for the column settings. The strings in each textline may be separated by spaces, commas or tabs. The rotation can be in degrees CCW (Counter clock wise), in degrees CW, in quadrant nr CCW or in quadrant nr CW.

Quadrant CCW: 0 = 0 degrees, 1 = 90 degrees CCW, etc


Quadrant CW: 0 = 0 degrees, 1 = 90 degrees CW, etc

The layer string maybe specified as top/bottom or as 0,1. The bottom layer should 0.

After **OK** the components in the textfile will mapped on the components in the design.


Edit

Move entire PCB

 Mainmenu	Sub menu Edit menu item Move entire PCB
---	---

After pressing the **OK** button the entire PCB will moved to coordinates typed. If the coordinates are started with @, the pcb will be moved relative of the original position. **This operation can not be undone.**

Change design rules

 Mainmenu	Sub menu Edit menu item Change design rules
---	---

In the next dialogbox the design rules settings can be modified. If required the design rule modification will modify the line/trace/clearance width on the whole design.

This operation can not be undone.


Zero relative cursor

 Keyboard	Press Ctrl z
---	---------------------

Layout editor

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.



 Mainmenu	Sub menu View menu item Relative position on grid
---	---

Center view on component

 Keyboard	Press Ctrl c
---	---------------------


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

Goto xy

 Mainmenu	Sub menu Edit menu item Goto xy
Menu  Mouse	Goto xy


In the next dialogbox a x,y location can be entered. After pressing **OK** the window will center on this location.

Via definition

 <p>Mainmenu</p>	Sub menu Edit menu item Via definition
---	--

In the next dialogbox ten 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

 <p>Mainmenu</p>	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
 <p>Keyboard</p>	Press u
<p>Menu</p>  <p>Mouse</p>	Undo

This function will undo almost all-previous actions.

Redo

	Press Select layers button
---	-----------------------------------

<div data-bbox="180 228 311 378"> <div data-bbox="215 235 279 257">Menu</div>  <div data-bbox="215 347 279 369">Mouse</div> </div>	<div data-bbox="478 235 550 257">Redo</div>
---	---

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 drawn 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
 Keyboard	Press F2
Menu  Mouse	Deselect all

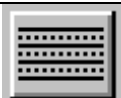


Info on selected objects

	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. Changes made will be immediately visible.




When the button **Solder/paste mask pads** is pressed a dialogbox will be visible. In this dialogbox the visibility of the solder/paste on top or bottom layer can be modified.

When the second option is active, only the extra added pads will be visible. When the third option is active all pads will be visible.

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

See also [Change grid](#)


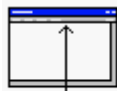

Zoom in

 Keyboard	Press z
 Mainmenu	Sub menu View menu item Zoom in
	Hold the Alt key and use the scroll wheel of the mouse

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

See also [Window based zooming](#)

Zoom out

 Keyboard	Press Z
 Mainmenu	Sub menu View menu item Zoom out
	Hold the Alt key and use the scroll wheel of the mouse

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
	For vertical movement use the scroll wheel of the mouse For horizontal movement use the scroll wheel of the mouse and hold the Shift key
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
<div>Menu</div>  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
<div>Menu</div>  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
<div>Menu</div>  Mouse	View whole design

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

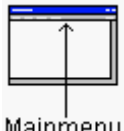
The **View whole design** function can be used in every possible drawing/moving function.

Measurement

	Measurement
---	--------------------

When activated and after clicking on the left mouse button an arrow with its length will be shown on the screen.

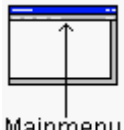
Change colors

	Sub menu View menu item Change colors
---	---

The color settings, brush and brush second color can be modified in the next dialogbox. The color settings will be copied into the **pcb.ini** initialisation 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 project 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]**.

Options

 Keyboard	Press Ctrl g
 Mainmenu	Sub menu Edit menu item Options

In the dialogbox a number of settings can be modified:

Intelligent ratsnets dragging:

When moving components with attached connections (air wires/guides), this option will be used for whether a connection stays attached or connections will shift depending on a closer location of a certain net.

Recalc areafill after inserting an object:

When this option is on and a trace is drawn on an areafill on the same layer, a cutout will be made in the areafill. Also when moving components on an areafill and this option is on, the areafill will get additional cutouts for the pads of the components.

It might be usefull to switch off this option when moving a lot of components on areafills. After all component movement is ready, the areafills should be rebuild, otherwise design rule errors will occur.

Repaint after every trace draw:

When on their will be a repaint of the whole screen after drawing of a trace segment.

Repaint after every component move:

When on their will be a repaint of the whole screen after moving components.

Do not show areafill thin lines errors:

When on the thin lines (smaller than default trace width) created during the making of an areafill, will not result in an error during the design rule check.

Start with maximum view:

When on the next time the layout editor editor will startup with maximum view of the design.

Snap modes:

Snap on for special objects is being used for moving special objects. When snap is on, objects can placed exactly on a objects endpoint.

With **Snap on first component location** components being moved having the cross hair center around the zero point of the first component.

Repeat :

When repeat is on the previous action will be repeated after selection.

For example when moving components: The first time a component is selected the move action has to be initiated from the pulldown menu. The next time a selection of components will trigger the move action immediately.

Mouse cursor info display:

When on and the mouse cursor on a copper objects, some info will be displayed in a popup window. The start en duration time can be modified.

There are four 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.



The **grid when drawing areafills** will be used when drawing areafills is active, and this value is not zero.

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.

With **View relative position on grid** the coordinates of the relative mouse cursor as displayed in the third box from the left of the status bar, will follow the current grid or not.

Change units

 Keyboard	Press Ctrl u
 Mainmenu	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](#)

View/hide grid

 Keyboard	Press g
---	----------------

View/hide grid.




Components

	Press Info selected objects button
 Keyboard	Press c


  Mouse	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
  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.
When the **Alt** key is pressed **selected** components will be rotated by 45 degrees.

 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).




Move components by reference

 Keyboard	Press r
---	----------------

<div>Menu</div>  <div>Mouse</div>	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

<div>Keyboard</div> 	Press R
<div>Menu</div>  <div>Mouse</div>	Rotate
<div>Menu</div>  <div>Mouse</div>	Rotate at any angle


Rotate:

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

Rotate at any angle:

Rotate **selected** components at any angle counter clock wise. The angle should be filled in the next dialogbox.

Move component to top/bottom layer

<div>Menu</div>  <div>Mouse</div>	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

<div>Menu</div>  <div>Mouse</div>	Regroup
--	----------------

Regroup **Selected** components.



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

Align components

<div>Menu</div>  <div>Mouse</div>	Component alignment/spacing
--	------------------------------------


In the next dialogbox the alignment/spacing method can be selected. Alignment can be based on left/right/top and bottom of pins, or horizontally/vertically based on the component centers. Component spacing evenly can be horizontal or vertical based. The component center will be the center of the pins.

Edit geometry

<div>Menu</div>  <div>Mouse</div>	Edit geometry component
 <div>Keyboard</div>	Press E

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 geometry


<div>Menu</div>  <div>Mouse</div>	Change geometry component
--	----------------------------------


The geometry of the selected component can be changed with this function. After selecting the geometry, the Layout editor will check whether the new geometry pin names are ok. The geometry change will also be recorder in the backannotation file. Via the “Back annotation” button of the design manager the geometry change can be copied in the schematics.

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

<div>Menu</div>  <div>Mouse</div>	Component references Hide Visible On top layer On bottom layer Height Line width
--	---

<div>Menu</div>  <div>Mouse</div>	Component values Hide Visible On top layer On bottom layer Height Line width
--	---

Protect components

<div>Menu</div>  <div>Mouse</div>	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 layer objects to the objects layer

<div>Menu</div>  <div>Mouse</div>	Copy objects component layer -> Component layer -> objects layer
--	---


Copies the component objects from the selected component layer to the objects layer selected.

Edit schematic containing reference

<div>Menu</div>  <div>Mouse</div>	Edit schematic containing reference xxx
--	--


The schematic editor with the sheet containing reference xxx , and will be centered around this reference.

Component selections

<div>Menu</div>  <div>Mouse</div>	Component selections -> Select all components with partnr xxx Select all components with geometry xxx Select all components with value xxx Deselect all components with partnr xxx Deselect all components with geometry xxx Deselect all components with value
--	---

Based on one selected components, all component having the same partnr, geometry or value can be (de)selected.

Component selections by list

 <p>Menu Mouse</p>	Component selections by list
---	-------------------------------------

A dialogbox will be made visible with all components listed. The list can be sorted on value, geometry, partnr and reference. Every component listed can be (de)selected.

When only one component is selected the button "Center screen on component" can be used. There is also a button ("Move components") to move the selected components immediatly.

Move multiple components

 <p>Menu Mouse</p>	Move multiple components
---	---------------------------------


A dialogbox will be made visible. In this dialogbox components that should be placed to a certain position should be listed. The format of each (text)line is the same as with info on components. The format of each text line is:

<Reference> <top/bottom> <Geometry> <Part nr> <Value> <Position X> <Position Y>
<mm/mils/inch> <Rotation>

Each string should be separated by a space or a tab. The <Geometry>, <Part nr> or <Value> strings can start/end with double quotes. The <Part nr> is optional, and the rotation is in degrees CCW.

Nets

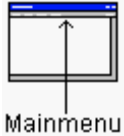
Change design rules net

 <p>Mainmenu</p>	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. The via used can also be modified. After pressing **OK** button the design rules for the selected nets will be modified, and additionally a messagebox will be shown with the question to change the already existing traces/vias to the new design rules.

See also [Make selections in dialog listboxes](#)


Highlight/unhighlight nets

 Mainmenu	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](#)

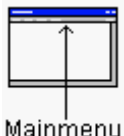
Disable connections nets

 Mainmenu	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

 Mainmenu	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

 <p>Mainmenu</p>	<p>Sub menu Nets menu item Highlight visible connections</p>
---	--

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](#)


Unselect traces/vias nets

 <p>Mainmenu</p>	<p>Sub menu Nets menu item Unselect traces/vias nets</p>
--	--

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

 <p>Mainmenu</p>	<p>Sub menu Nets menu item Delete traces/vias nets</p>
---	--

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

 <p>Mainmenu</p>	Sub menu View menu item Unhighlight all
---	---


Traces/vias/connections will be unhighlighted.

View all connections

 <p>Mainmenu</p>	Sub menu Connections menu item View all connections
---	---




Hidden connections will be made visible.

Hide all connections

 <p>Mainmenu</p>	Sub menu Connections menu item Hide all connections
---	---

Visible connections will be hidden.

Routing

	Routing traces button
 <p>Keyboard</p>	Press s (For a maximum of three times)
<p>Menu</p>  <p>Mouse</p>	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.

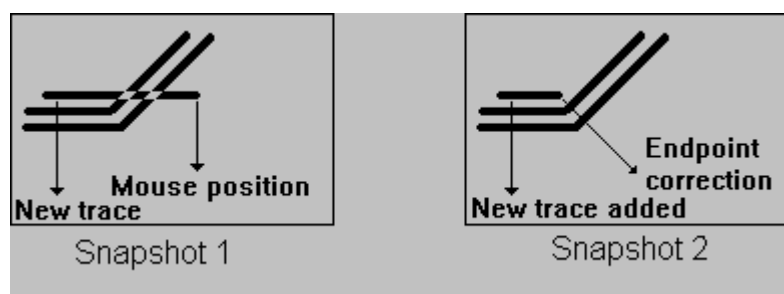
Add trace

When the **Routing** menu is active, new traces can be added, and existing traces modified. The traces that can be placed are:

- 45/90 degrees angle traces
- All angle traces
- Arc traces

Place the mouse cursor on 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 too 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

 Keyboard	Press .
 Mouse	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:



- Trace drawing all angle
- Arc trace drawing (90 degrees)
- Arc trace drawing (45 degrees)
- 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
- Change design rules net

Trace drawing all angle

 Keyboard	Press a (For a maximum of four times)
<div>Menu</div>  Mouse	Trace drawing all angle



When activated, all angle traces can be placed.

Arc trace drawing (90 degrees)

 Keyboard	Press a (For a maximum of four times)
<div>Menu</div>  Mouse	Arc trace drawing (90 degrees)


When activated, a 90 degree arc trace and an additional trace can be placed.
!!! This feature depends on the current license.

Arc trace drawing (45 degrees)

 Keyboard	Press a (For a maximum of four times)
<div>Menu</div>  Mouse	Arc trace drawing (45 degrees)


When activated, a 45 degree arc trace and an additional trace can be placed.
!!! This feature depends on the current license.

Display clearance

<div>Menu</div>  Mouse	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

<div>Menu</div>  Mouse	Display two trying traces
--	----------------------------------



Displays one or two trying traces.

Display via option

<div>Menu</div>  Mouse	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

 Keyboard	Press Space bar
Menu  Mouse	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

Menu  Mouse	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

<div>Menu</div>  <div>Mouse</div>	Delete trace
--	---------------------



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

Goto previous trace segment

 <div>Keyboard</div>	Press b
<div>Menu</div>  <div>Mouse</div>	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

 <div>Keyboard</div>	Press w
<div>Menu</div>  <div>Mouse</div>	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

<div>Menu</div>  <div>Mouse</div>	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

<div>Menu</div>  <div>Mouse</div>	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,+)

Change design rules net

<div>Menu</div>  <div>Mouse</div>	Change design rules net
--	--------------------------------



The design rules of the current net can be modified by this function. In the next dialogbox with the current net already been preselected, the trace width and clearance width can be edited. The via used can also be modified. After pressing **OK** button the design rules for the selected nets will be modified, and additionally a messagebox will be shown with the question to change the already existing traces/vias to the new design rules.

Add extra trace

<div>Menu</div>  <div>Mouse</div>	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

<div>Menu</div>  <div>Mouse</div>	Start routing with the shortest net
 <div>Keyboard</div>	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.

Show next connection

<div>Menu</div>  <div>Mouse</div>	Center view on next connection
 <div>Keyboard</div>	Press n

This function will search the next available connection and center the view around it. By using this function you can cycle through the available connections.

Change traces/vias

	Press Change traces/vias button
 <div>Keyboard</div>	Press s (For a maximum of three times)
<div>Menu</div>  <div>Mouse</div>	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
- Change design rules net
- Calculate length trace
- Swap traces/vias two nets
- Delete traces/vias net selected trace
- Delete
- Copy traces/vias to clipboard
- Copy traces/vias from clipboard

Move traces/vias

<div>Menu</div>  <div>Mouse</div>	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

<div>Menu</div>  <div>Mouse</div>	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.

Copy traces/vias to clipboard

<div>Menu</div>  <div>Mouse</div>	Copy traces/vias to clipboard
--	--------------------------------------


The **selected** traces of the current drawing layer and vias will be copied to the clipboard.

Copy traces/vias from clipboard

<div>Menu</div>  <div>Mouse</div>	Copy traces/vias from clipboard
--	--


If there are traces/vias stored on the clipboard, they will be imported and displayed. The traces/vias can now be moved to the correct position. To make this more easy, snap functionality has been included. After clicking on the left mouse button, the mouse cursor will snap to the object closest to the cursor. If necessary the moving center can be changed by holding the shift key. After release the moving center will snap to the moving object closest to the mouse cursor.

Select only

<div>Menu</div>  <div>Mouse</div>	Select only -> Traces Select only -> Vias
--	--

Traces or vias will be selected only with this function.


Change trace width

<div>Menu</div>  <div>Mouse</div>	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

<div data-bbox="178 376 311 515"> <div>Menu</div>  <div>Mouse</div> </div>	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


<div data-bbox="178 974 311 1113"> <div>Menu</div>  <div>Mouse</div> </div>	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.

Change design rules net

<div data-bbox="178 1509 311 1648"> <div>Menu</div>  <div>Mouse</div> </div>	Change design rules net
---	--------------------------------

The design rules of the net of the selected trace can be modified by this function. In the next dialogbox with the current net already been preselected, the trace width and clearance width can be edited. The via used can also be modified. After pressing **OK** button the design rules for the selected nets will be modified, and additionally a messagebox will be shown with the question to change the already existing traces/vias to the new design rules.

Calculate length trace

<div>Menu</div> <div></div> <div>Mouse</div>	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

<div>Menu</div> <div></div> <div>Mouse</div>	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

<div>Menu</div> <div></div> <div>Mouse</div>	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
 Keyboard	Press Del

<div>Menu</div>  <div>Mouse</div>	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
<div>Keyboard</div> 	Press s (For a maximum of three times)
<div>Menu</div>  <div>Mouse</div>	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
---	---

 <p>Menu Mouse</p>	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/areafills, and use the function **Drag traces/vias/components** to drag/rotate the traces/vias/components.

After pressing the **left mouse button** traces/vias/components/areafills 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/areafills will be rotated by 90 degrees, or when the **Alt** key is pressed 45 degrees.
The connections of the nets involved are being recalculated.

Check


Check connectivity

 <p>Mainmenu</p>	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

 Mainmenu	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/areafills will be checked if within these board outlines.



View design rule errors

 Mainmenu	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

Show next design rule error/warning

 Keyboard	Press e
<div>Menu</div>  Mouse	Cycle through design rule errors/warnings

If there are design rule errors/warnings and also visible, the view will be centered around the next error/warning.

Powerplanes

 Keyboard	Press a
	Press Areafills/Powerplanes button
Menu  Mouse	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

Menu  Mouse	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

<div data-bbox="183 235 311 369"> <div>Menu</div>  <div>Mouse</div> </div>	Remove powerplane -> Select layer
---	---

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

Cut from powerplane

<div data-bbox="183 591 311 725"> <div>Menu</div>  <div>Mouse</div> </div>	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

<div data-bbox="183 1686 311 1821"> <div>Menu</div>  <div>Mouse</div> </div>	Change powerplane -> Select layer
---	---

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

See also [Thermal relief](#)

Areafills

 Keyboard	Press a
	Press Areafills/Powerplanes button
<div>Menu</div>  Mouse	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
- Copy areafill
- Cut from areafill
- Change areafill
- Change clearance areafill
- Delete areafill
- Add to areafill
- Mirror areafill over the X-axes
- Mirror areafill over the Y-axes
- Merge areafill
- Rebuild areafill
- Move areafill
- Stretch areafill
- View vertices areafill
- Copy start polygon to info4 layer

Add areafill

<div data-bbox="180 228 311 378"> <div>Menu</div>  <div>Mouse</div> </div>	Add areafill -> Select layer Add areafill with no net -> Select layer
---	--

Add areafill:

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.


Add areafill with no net:

A polyline must 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

<div data-bbox="180 1281 311 1431"> <div>Menu</div>  <div>Mouse</div> </div>	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
 Keyboard	Press Del
Menu  Mouse	Delete



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

Copy areafill

 Keyboard	Press C
Menu  Mouse	Copy

The **selected** areafill will be copied by this function.

Move areafill

 Keyboard	Press m
Menu  Mouse	Move

The **selected** areafill will be moveable by this function.

Mirror X areafill

<div data-bbox="178 315 309 452"> <div>Menu</div>  <div>Mouse</div> </div>	Mirror X areafill
---	--------------------------

The **selected** areafill will be X mirrored.

Mirror Y areafill

<div data-bbox="178 698 309 835"> <div>Menu</div>  <div>Mouse</div> </div>	Mirror Y areafill
---	--------------------------

The **selected** areafill will be Y mirrored.

Stretch areafill

<div data-bbox="178 1081 309 1218"> <div>Menu</div>  <div>Mouse</div> </div>	Stretch areafill
---	-------------------------


With this function the **selected** areafill can be stretched. After activation of this function the user can put a rectangle on the segment points to be moved.

Change areafill

<div data-bbox="178 1529 309 1666"> <div>Menu</div>  <div>Mouse</div> </div>	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.

Change clearance areafill

<div data-bbox="188 232 309 367"> <div>Menu</div>  <div>Mouse</div> </div>	Change clearance
---	-------------------------

The clearance of the **selected** areafill can be changed with this function.

Cut from areafill

<div data-bbox="188 586 309 721"> <div>Menu</div>  <div>Mouse</div> </div>	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**.

!!! Every cutout will be stored, so when you do a rebuild all the cutouts will be applied again.

Merge areafills

<div data-bbox="188 1285 309 1420"> <div>Menu</div>  <div>Mouse</div> </div>	Merge
---	--------------

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

Change areafill

<div data-bbox="188 1671 309 1805"> <div>Menu</div>  <div>Mouse</div> </div>	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](#)

Add to areafill

<div>Menu</div> <div></div> <div>Mouse</div>	Add to areafill
---	------------------------


With this function a polygon area can be added to the **selected** areafill. Next this polygon should be drawn, and when finished the polygon area will be added to the areafill.

Rebuild areafill

<div></div> <div>Keyboard</div>	Press r
<div>Menu</div> <div></div> <div>Mouse</div>	Rebuild areafill


With this function an areafill can be rebuild. If there were cutouts for this areafills, then the rebuild will apply these cutouts again.

View vertices areafill

<div>Menu</div> <div></div> <div>Mouse</div>	View vertices areafill
---	-------------------------------



After a areafill is selected, the vertices (points) of the areafill can be shown with this function. By copy/paste (Add areafill) these points an areafill can be used again.

Copy start polygon to info4 layer

<div>Menu</div> <div></div> <div>Mouse</div>	Copy start polygon to info4 layer
---	--

The start polygon of the selected areafill will be copied to the info4 layer as a polyline.

Modify component references

	Press Modify component references button
<div>Menu</div>  <div>Mouse</div>	Other menus -> Component references menu



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

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

Based on the selected component references the following actions can be applied:

- Move references
- Rotate references
- Change reference text heights
- Change reference text line widths
- Change visibility
- Change top/bottom layer

Modify component values

	Press Modify component values button
<div>Menu</div>  <div>Mouse</div>	Other menus -> Component values menu



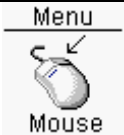
The **Component values menu** can be activated by one of two above actions.

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

Based on the selected component values the following actions can be applied:

- Move values
- Rotate values
- Change value text heights
- Change value text line widths
- Change visibility
- Change top/bottom layer

Special objects

	Press Draw/change objects other layers button
 Keyboard	Press o
 Menu Mouse	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
- Select only objects on a layer
- Select only line/rectangle/polygon/arc/text objects
- Copy objects from one layer to another
- Move objects from one layer to another

- Mirror text
- Assign a net to objects on the copper layers
- Assign no net to objects on the copper layers

Add special objects

Objects can be added on the following layers:

- Copper layers
- Board outline layer
- Info layer
- Info layer 2
- Info layer 3
- Info layer 4
- Solder mask top layer
- Solder mask bottom layer
- Paste mask top layer
- Paste mask bottom layer
- Drills (plated) layer
- Drills (unplated) layer
- Routing keepout layers

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.

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.

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.

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.

Texts

A text object will be added. In the next dialogbox the (Multiline) text can be entered. In addition the text height and font can be edited. Upto 256 characters including line feeds can be entered. 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.

Polyline

When drawing this polyline use the the **right mouse button** menu to change the drawing direction, goto the previous polyline point (Backwards) and to finish the polyline drawing. When the spacebar is pressed, a dialogbox will popup, and the polyline parameters can be edited by hand. As many as 256 points can be edited. The coordinates typed in will be used with the current units.

Polygons

When drawing this polygon use the the **right mouse button** menu to change the drawing direction, goto the previous polyline point (Backwards) and to finish the polygon drawing. When the spacebar is pressed, a dialogbox will popup, and the polygon parameters can be edited by hand. As many as 256 points can be edited. The coordinates typed in will be used with the current units.

Arrows



A left pointed, right pointed or both pointed arrows will be added.

Dimension

A axial or radial dimension including the value will be added.


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	Copy/move special -> Move objects to -> Select layer
---	---

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


Copy

 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

 Mouse	Copy/move special -> Copy objects to -> Select layer
--	---


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

Copy objects array

 Mouse	Copy/move special -> Copy objects array
--	---

The **selected** objects will be copied via an x,y array. In the next dialogbox the distance in X and Y direction can be entered, and the number of copies in X and Y direction. After pressing the **OK** button the selected will be copied.

Copy objects polar



<div>Menu</div>  <div>Mouse</div>	Copy/move special -> Copy objects polar
--	---

The **selected** objects will be copied rotational. In the next dialogbox the rotation angle, the number of copies and some options can be entered. The options:

- The centre of rotated objects can be selected from **Rotate around user centre** and **Rotate around coordinate centre**. When **Rotate around user centre** has been selected the user can place the mouse cursor on the centre. When **Rotate around coordinate centre** has been selected the user can enter the coordinates of centre in the dialogbox.
- The objects can be rotated individually




After pressing the **OK** button the selected will be copied.

Rotate

 <div>Keyboard</div>	Press R
<div>Menu</div>  <div>Mouse</div>	Modify object -> Rotate objects


In the next dialogbox the rotation angle should be filled in. In this dialogbox are some options for rotation centers. After **OK** the **selected** objects will be rotated.

Delete

	Delete button
 <div>Keyboard</div>	Press Del
<div>Menu</div>  <div>Mouse</div>	Delete


Delete **selected** objects.

Scale

<div>Menu</div>  <div>Mouse</div>	Modify object -> Scale objects
--	--


In the next dialogbox the scale factor should be filled in. After **OK** the **selected** objects will be scaled.

Change circle diameter

<div>Menu</div>  <div>Mouse</div>	Modify object -> Change circle diameter
--	---


Change the diameter of the **selected** circle objects.

Change arc width/height

<div>Menu</div>  <div>Mouse</div>	Modify object -> Change arc width/height
--	--


Change the width/height of the **selected** arc objects.

Change arc angles

<div>Menu</div>  <div>Mouse</div>	Modify object -> Change arc angles
--	--


Change the start/end angle of the **selected** arc objects.

Change rectangle width/height

<div>Menu</div>  <div>Mouse</div>	Modify object -> Change rectangle width/height
--	--



Change the width/height of the **selected** rectangle objects.

Change line thickness

<div>Menu</div> <div></div> <div>Mouse</div>	Modify object -> Change line thickness
---	--


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

Change text

<div></div> <div>Keyboard</div>	Press e
<div>Menu</div> <div></div> <div>Mouse</div>	Modify object -> Edit


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

Change text height

<div>Menu</div> <div></div> <div>Mouse</div>	Modify object -> Change text height
---	-------------------------------------


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

Mirror X

<div>Menu</div>  <div>Mouse</div>	Modify object -> Mirror objects X
--	---


The **selected** objects will be X mirrored.

Mirror Y

<div>Menu</div>  <div>Mouse</div>	Modify object -> Mirror objects Y
--	---

The **selected** objects will be Y mirrored.

Cut from polygon

<div>Menu</div>  <div>Mouse</div>	Cut from polygon -> Polyline Rectangle Circle Horizontal trace Vertical trace
--	---

With this function, the selected polygon can be changed by cutting with an object. There are three cutout possibilities, with a **circle**, **rectangle**, **horizontal trace**, **vertical trace** and a **polyline**.

Polyline : When drawing this polyline use the 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.

Calculate area polygon

<div>Menu</div>  <div>Mouse</div>	Calculate area polygon
--	-------------------------------


The vertices of the **selected** polygon will be shown.

Assign net to objects

<div>Menu</div>  <div>Mouse</div>	Assign objects to a net
--	--------------------------------



In the next dialogbox a net should be selected. After **OK** the **selected** objects on the copper/drill layers will be assigned to this net. The next time a netlist check or a design rule check is performed these new objects will be taken into account.

Assign no net to objects

<div>Menu</div>  <div>Mouse</div>	Assign objects to a not used net
--	---

The **selected** objectson the copper/drill layers will be assigned to a not used net. These objects will not be used anymore when a netlist check or a design rule check is performed.

Gate/pin swap

	Press Gate/pin swap button
<div>Menu</div>  <div>Mouse</div>	Other menus -> Gate/pin swap menu

The **Gate/pin swap menu** can be activated by one of two above actions.

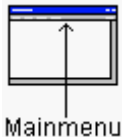
Activation of the **Gate/pin swap menu** is made visible on the info bar at the bottom right of the window. **Gate/pin swap** is now visible. Also **Gate/pin swap** button is visible pressed.

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

 Mainmenu	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 the 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 be made visible

Wire

The connections of the wires netname will be made visible



Components

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

Schematic editor



File

New sheet

 Mainmenu	Sub menu File menu item New sheet
 Mainmenu	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

 Mainmenu	Sub menu File menu item New symbol
 Mainmenu	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

 Mainmenu	Sub menu File menu item New sheetsymbol
---	---

 <p>Mainmenu</p>	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

 <p>Mainmenu</p>	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

 <p>Mainmenu</p>	Sub menu File menu item Save
	Press Save button



Saves the current file in the current (design) directory

Save as

 <p>Mainmenu</p>	Sub menu File menu item Save as
---	---

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

Print

 Mainmenu	Sub menu File menu item Print
	Press Print button

Prints the current file to the printer.

In the design manager all sheets can be printed at once.

Export to BMP

 Mainmenu	Sub menu File menu item Export to bmp
---	---


Export to PDF

 Mainmenu	Sub menu File menu item Export to pdf
---	---

Export to DXF

 Mainmenu	Sub menu File menu item Export to dxf
---	---

Import from DXF

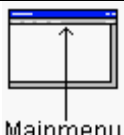
 Mainmenu	Sub menu File menu item Import from dxf
---	---

View

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

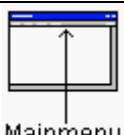
- Zoom in
- Zoom out
- Window based zooming
- Pan window
- Window based panning
- Return to previous view window
- Repaint
- View whole design
- Deselect all
- Undo
- Redo
- View/hide grid

Change colors

 Mainmenu	Sub menu View menu item Change colors
---	---

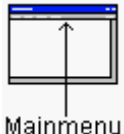
The color settings can be modified in the next dialogbox. The color settings will be copied into the **sch.ini** initialization file. This file is stored into the directory of the project.
To use those sch colors for new designs, copy this **sch.ini** file to main directory. Whenever a new design is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

Load default colors (Black background)

 Mainmenu	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

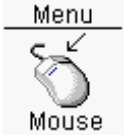
To select an object, place the mouse cursor above the object, and press and hold the left mouse button. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **Replacement mode**, and the second selection mode is the **Adding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

The **Replacement selection mode** means, every time a new selection rectangle is drawn 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.

Component/net info popup display

	Mouse cursor on component reference/value Or on a netlabel with properties
---	---

When the mouse cursor is placed on a symbol reference/value or on a netlabel with properties a popup display is made visible. In this popup display are all symbol parameters and optional properties visible.

In case of a netlabel with properties the popup display shown the netlabel and its properties. A netlabel with properties is shown as text with additional three dots.

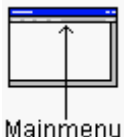
A symbol with additional properties will be made visible through the reference with added three dots.

See also [View sheet/symbol options](#)

Deselect all

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

View sheet/symbol options

 Mainmenu	Sub menu Edit menu item Sheet options Symbol options
---	---

In the next dialogbox some sheet/symbol options and features can be changed.

For a sheet these options/features are:

- Cross hair
- Disable one pinnet check on/off (Will be overruled by the same setting in the project settings of the design)
- Mouse cursor info display
- Grid on/off
- Grid size
- Repeat on/off (for continuously selecting and moving objects)
- Selection mode replacement/appending
- Wire/line selection mode (normal,extended)
- Sheet start number for annotation

For a symbol these options/features are:



- Cross hair
- Grid on/off
- Grid size
- Repeat on/off (for continuously selecting and moving objects)
- Selection mode replacement/appending
- Wire/line selection mode (normal,extended)

Wire/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 symbol parameters

 Keyboard	Press e
 Mouse	Edit symbol parameters

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

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

A component with additional properties will be made visible through the reference with added three dots.

See also [Component/net info popup display](#)

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.

Properties

Symbol properties can be used to add specific features to this symbol. When the netlist is created properties for these symbol will be added to the netlist. The geometry and partnr of a symbol are already (hidden) properties.

Another special symbol properties are powerpin netname overrules.

For example if want to use a symbol with 2V5 powerpins, but you the symbol has powerpins of 3V3 you can add a powerpin netname overrule.

When the netlist is created the 3V3 netname will be replaced by 2V5 for this symbol.

A symbol with additional properties will be made visible through the reference with added three dots.


See also [Symbol/net info popup display](#)

Edit symbol

<div>Menu</div>  <div>Mouse</div>	Edit <symbol>
--	----------------------------


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

Protect symbols

 <div>Mainmenu</div>	Sub menu Edit menu item Protect symbols
---	---


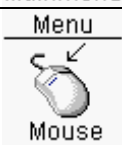
Selected symbols will be protected. Protected symbols can not be selected.

Unprotect symbols

 Mainmenu	Sub menu Edit menu item Unprotect symbols
---	---

All protected symbols will be unprotected.

Edit gate/pin swap

 Mainmenu	Sub menu Edit menu item Edit gate/pin swap
 Menu Mouse	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(es) 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

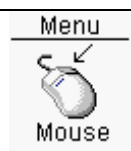
74244 TTL device with two sets of four inputs and outputs. Every Input/Output combination of a set can be swapped against any other three Input/Output. This means there are two sets of four gates, each with two pins.

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

 Mainmenu	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

 Keyboard	Press e
 Menu	Edit any text
 Mouse	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 and additional properties
- Normal text

See also [Netlabel properties](#)


Search for any text

 Mainmenu	Sub menu Edit menu Search for any text
 Keyboard	Press f

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

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

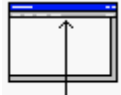
Edit the number of symbols in a device.

Edit pinnumbers package parts

 Mainmenu	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

 <p>Mainmenu</p>	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.

Edit layout containing reference

 <p>Menu Mouse</p>	Open layout centered on reference xxx
---	---------------------------------------

After a reference text has been typed into the dialogbox the layout editor will be started and centered around this reference.

Component selections by list

 <p>Menu Mouse</p>	Component selections by list
---	-------------------------------------

A dialogbox will be made visible with all components listed. The list can be sorted on value, geometry, partnr and reference. Every component listed can be (de)selected.


When only one component is selected the button "Center screen on component" can be used. There is also a button ("Move components") to move the selected components immediatly.

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

 Mainmenu	Sub menu Edit menu item Clear references
---	--

All 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

 Mainmenu	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.
- Symbol pins without a one pinnet and without any other connection
- Overlapping wires/busses
- Lose wires



Check symbol

 Mainmenu	Sub menu Edit menu item Check
---	---

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



- Double pinnumbers

Edit pin normal symbol

 Keyboard	Press e
Menu  Mouse	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


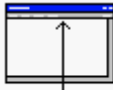
In the next dialogbox parameters of **selected** sheetsymbol pins can be edited. Connection type and labelname can be modified. After clicking the **OK** button the pin will be changed.

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 (Special drawing mode)
 Keyboard	Press w
 Keyboard	Press W (Special drawing mode)
Menu  Mouse	Add wire

Add wire. When pressing the button or the key 'W' a special mode of drawing wires is activated. When the mouse is placed on a existing wire, this wire can be redrawn.

Add bus

	Press Add bus button (Special drawing mode)
 Keyboard	Press b
 Keyboard	Press B (Special drawing mode)
Menu  Mouse	Add bus

Add a bus. When pressing the button or the key 'B' a special mode of drawing busses is activated. When the mouse is placed on a existing bus, this bus can be redrawn.




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 in 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 q
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




<div>Menu</div>  <div>Mouse</div>	Add external connections -> Output
	Press Add external I/O connection button
<div>Menu</div>  <div>Mouse</div>	Add external connections -> I/O

An External connection is used to connect signals or busses from different sheets in a hierarchical 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
 <div>Keyboard</div>	Press n
<div>Menu</div>  <div>Mouse</div>	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 netlabel properties

<div>Menu</div>  <div>Mouse</div>	Add/modify net properties
<div>Menu</div>  <div>Mouse</div>	Edit text
 <div>Keyboard</div>	Press e



For a netlabel additional properties for this net can be added. There are already two fixed properties in use: TRACEWIDTH and CLEARANCE. When the netlist is created, than those properties will be added to the netitems. The layout will use than those TRACEWIDTH and CLEARANCE properties when importing/updating the netlist.

A netlabel property can be added after selecting a netlabel or a wire where a netlabel is attached to. In the next dialogbox property IDs and Values can be typed in. There are already two buttons for TRACEWIDTH and CLEARANCE. If one of them is pressed a property ID (TRACEWIDTH is added, and a property value (8 mil) is added. For TRACEWIDTH and CLEARANCE values **mil** or **mm** can be used as a unit.

After **OK** the property(ies) are added to this netlabel. This will be made visible by the addition of three dots to the netlabel name.


See also [Component/net info popup display](#)

Add incremental netlabels to wires

<div>Menu</div>  <div>Mouse</div>	Add incremental netlabels to wires
 <div>Keyboard</div>	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 function. In the dialogbox the name of the netlabel should be filled in.

Add netlabel + wire

<div>Menu</div>  <div>Mouse</div>	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 one pinnet mark




<div>Menu</div>  <div>Mouse</div>	Add one pinnet mark
 <div>Keyboard</div>	Press o

With the one pinnet mark special symbol, component pins can be marked as one pin nets. Component pins with a one pinnet mark will show up in the netlist as a net with only one pin.

Component pins which are not marked as one pinnets should be connected to something else (wire,bus,global connection). When the schematic is saved or when using the special check, component pins are being scanned for having either a one pinnet check mark, or are being connected to something else.

With such a check unconnected component pins, which should be connected to something else will be automatically detected.

Add symbol



	Press Add label to wire/bus button
 <div>Keyboard</div>	Press i
<div>Menu</div>  <div>Mouse</div>	Add symbol on shortcut

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. The search order for a symbol will be based on the top window. First the symbol will be searched in the "Local symbol directory". If not found the "Global symbol directory" will be searched, etc. 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 times 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 symbol on shortcut

 Keyboard	Press s
Menu  Mouse	Add label to wire/bus

In the next dialogbox a symbolname can be entered. This symbolname may contain wildcards. For example when the string ***Is00** has been entered, all the symbol names containing **Is00** will be listed. After searching, all the found symbol names will be listed in a dialogbox. In the edit box at the bottom of the dialogbox a symbol description is shown for the selected symbol. After selecting the symbolname and clicking **OK**, 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.

Add component from database

Menu  Mouse	Add database component
 Keyboard	Press Ctrl d

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 next dialogbox the symbol with the right value and geometry can be selected from the database and inserted into the sheet.

There are also five search possibilities for geometry, symbol name, value, description and partnr. A search text can be entered into on the edit boxes and after pressing the corresponding search button, the database will be searched. The found component will be listed in the list box above.

The information for this component database is put into two files in the base directory of PCB elegance. The names for those files are **compmenu.txt** and **comp.txt**.

Compmenu.txt

This is a text file containing menu items. The first character of each line will be used for decoding.

First character:

; Line will be ignored (Also empty lines)

\$	The number directly after \$ (Code1 in the comp.txt file) is the main index in the component pulldown menu. The next string will be visible in the menu.
#	The next number (Code2 in the comp.txt file) will not be used if there are further pull down menus. If there are no further pulldown menus the number will be used. The next string will be visible in the second pulldown menu.
^	The next number is the Code2 in the comp.txt file The next string will be visible in the second pulldown menu.

There are two options to edit this file. The first option is to add at the bottom of the file the new components. The second option is insert lines between the standard menu items. The **Code1** numbers 1 to 8 are reserved, and the **Code2** numbers used by the PCB elegance are ending on a zero. The user can insert other lines, but the **code2** numbers must end on 1 to 9. Do not enter new **Code2** members, because there are reserved for future implementations.

Comp.txt



The comp.txt is a text database file. Empty lines, or lines starting with a ; are ignored. Every line contains seven items.

Item1	Symbolname
Item2	Code1 definition
Item3	Code2 definition
Item4	Partnumber (Optional)
Item5	Value
Item6	Geometry
Item7	Description

When pulldown menu item has been selected with corresponding Code1 and Code2 numbers, the program makes a subselection from the comp.txt file. This subselection will be displayed into a dialogbox, and the component can be selected and included into the schematic.



Add other objects

Line

 Keyboard	Press I
 Menu Mouse	Add special -> Line


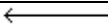
Add a line.

Add right pointed arrow

<div data-bbox="178 232 309 360"> <div>Menu</div>  <div>Mouse</div> </div>	<div data-bbox="478 232 810 259">Add special -> </div>
---	--


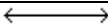
Add a right pointed arrow.

Add left pointed arrow

<div data-bbox="178 636 309 763"> <div>Menu</div>  <div>Mouse</div> </div>	<div data-bbox="478 636 810 663">Add special -> </div>
---	--


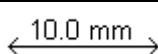
Add a left pointed arrow.

Add left/right pointed arrow

<div data-bbox="178 1043 309 1171"> <div>Menu</div>  <div>Mouse</div> </div>	<div data-bbox="478 1043 810 1070">Add special -> </div>
---	--

Add a left/right pointed arrow.


Add dimension

<div data-bbox="178 1420 309 1547"> <div>Menu</div>  <div>Mouse</div> </div>	<div data-bbox="478 1420 861 1473">Add special -> </div>
---	--

Add a dimension.


Rect

<div data-bbox="178 1796 309 1883"> <div>Menu</div>  <div>Keyboard</div> </div>	<div data-bbox="478 1796 569 1823">Press r</div>
--	--

<div>Menu</div>  <div>Mouse</div>	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)

<div>Menu</div>  <div>Mouse</div>	Add special -> Rect
--	-------------------------------



Add rectangle.

Circle

 <div>Keyboard</div>	Press C
<div>Menu</div>  <div>Mouse</div>	Add special -> Circle -> Select circle



Add a (partial) circle.

Arc

 <div>Keyboard</div>	Press a
<div>Menu</div>  <div>Mouse</div>	Add special -> Arc


Add an arc.

Text

 Keyboard	Press t
<div>Menu</div>  Mouse	Add special -> Text




Add text

Numbers incremental

<div>Menu</div>  Mouse	Add special -> Numbers incremental
--	--

Add a range of numbers.

Add pin

	Press Add pin button
 Keyboard	Press a
<div>Menu</div>  Mouse	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.

To add a powerpin click the 'Pinname is powernet' checkbox. The pinname edited will be the net in the schematic. (These powerpins can be used for GND symbols).

To add a series of pins with incrementing pinnames the **Auto numbering** options can be used.




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
 Keyboard	Press a
Menu  Mouse	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

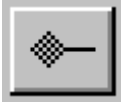

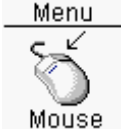
	Press Add powerpin button
 Keyboard	Press p
Menu  Mouse	Add powerpin

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**.

For overruling powernet names see [Symbol properties](#)

Add pinbus

	Press Add pin button
 Keyboard	Press P
 Menu Mouse	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 name of the sheetsymbol represent a sheet with the same name. 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
Menu  Mouse	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
Menu  Mouse	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
Menu  Mouse	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
 Keyboard	Press m
Menu  Mouse	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
 Keyboard	Press d
Menu  Mouse	Drag


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

Rotate objects

 Keyboard	Press R
---	----------------

<div>Menu</div>  <div>Mouse</div>	Rotate
--	---------------

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

<div>Menu</div>  <div>Mouse</div>	Rotate at any angle
--	----------------------------

Rotate **selected** objects at any angle. In the next dialogbox the angle in degrees can be filled in.

Scale objects

<div>Menu</div>  <div>Mouse</div>	Scale
--	--------------




Scale **selected** object. In the next dialogbox the scale factor can be filled in.


Mirror objects

<div>Menu</div>  <div>Mouse</div>	Mirror X Mirror Y
--	------------------------------------

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


Copy objects

	Press Copy button
 Keyboard	Press c
<div>Menu</div>  <div>Mouse</div>	Copy

<div>Menu</div>  <div>Mouse</div>	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.

Align text objects left/right

<div>Menu</div>  <div>Mouse</div>	Align text objects left Align text objects right
--	---


The selected text objects (Netlabels/text) will be aligned left or right.

Change text height

<div>Menu</div>  <div>Mouse</div>	Change text height
--	---------------------------


Change text height **selected** text objects. In the next dialogbox the text height can be filled in.

Change line thickness

<div>Menu</div>  <div>Mouse</div>	Change line width
--	--------------------------

Change line width **selected** objects. In the next dialogbox the line width can be filled in.





Copy objects to clipboard

 <div>Mainmenu</div>	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	Paste objects from clipboard
 Mouse	
 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	Delete
 Mouse	

Delete **selected** objects.


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

Unselect objects

<div>Menu</div>  <div>Mouse</div>	Unselect -> Objects
--	-------------------------------


Unselect objects

Select only

<div>Menu</div>  <div>Mouse</div>	Select only -> Objects
--	----------------------------------


Select only one object type.

Edit symbol

<div>Menu</div>  <div>Mouse</div>	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.

Reload symbols

 <div>Mainmenu</div>	Sub menu File menu item Reload symbols
---	--

If one of the symbols in the schematic has been changed, this function can be used to reload the all the symbols used in the schematic.

Goto x,y

 <div>Mainmenu</div>	Sub menu Edit menu item Goto x,y
<div>Menu</div>  <div>Mouse</div>	Goto x,y

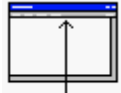

 Keyboard	Press Ctrl g
---	---------------------

In the dialogbox a screen coordinate can be typed in, and after **OK** the screen will center around this coordinate.

Geometry editor

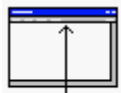

File

Open

 Mainmenu	Sub menu File menu item Open
	Press Open button

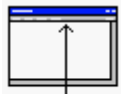
Opens a geometry file (.shp) from the standard geometry directory.

Save

 Mainmenu	Sub menu File menu item Save
	Press Save button


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

Save as

 Mainmenu	Sub menu File menu item Save as
---	---


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

Print

 Mainmenu	Sub menu File menu item Print
---	---

Prints the current file to the printer.

Import DXF files

 Mainmenu	Sub menu File menu item Import from DXF file Info layer 1 Info layer 2 Info layer 3 Info layer 4
---	--

Import DXF file into Info layer 1..4.

After selecting a DXF file from the next dialogbox, this DXF file will be scanned for its layers. In the following dialogbox one or more DXF layers can be selected. After including the objects into the design, a messagebox with the min/max values of the selected objects will be displayed. By using the scale function of the right mouse button menu, the scale of the objects can be adjusted.

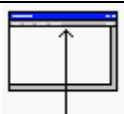
!!! Dimensions are not supported.

Export to DXF file

 Mainmenu	Sub menu File menu item Export to DXF file
---	--

Exports the **selected** layers to a DXF file. In the next dialogbox the layers and drawing board outline can be selected. There are two options available: Filled objects means the objects will be exactly represented in the DXF file. If this option is not marked, only the hairlines will be put into the dxf file. The second option is a mirror in the X direction. After pressing the **OK** button the DXF file will be generated.



Import a bitmap

 Mainmenu	Sub menu File menu item Import bitmap
---	---

In the next dialogbox a window bitmap file (monochrome) will be requested. After **OK** the pixel size is requested in the next dialogbox. After pressing **OK** the (valid) windows bitmap will be imported and

should be placed on the right position. After clicking on the left mouse button the bitmap will be placed as pixel lines on the bottom layer.

Make new geometry

	Press New geometry button
 Mainmenu	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 DIP 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.

The **Use default rules for solder paste/mask, inner/anti power pad and clearance** button can be used the fill in values based on the drill size.

See also [Design rules pad](#)

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.

The **Use default rules for solder paste/mask and clearance** button can be used the fill in values based on the pad size.

See also [Design rules pad](#)

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.

The **Use default rules for solder paste/mask and clearance** button can be used the fill in values based on the pad size.

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.

The **Use default rules for solder paste/mask, inner/anti power pad and clearance** button can be used the fill in values based on the drill size.

See also [Design rules pad](#)

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

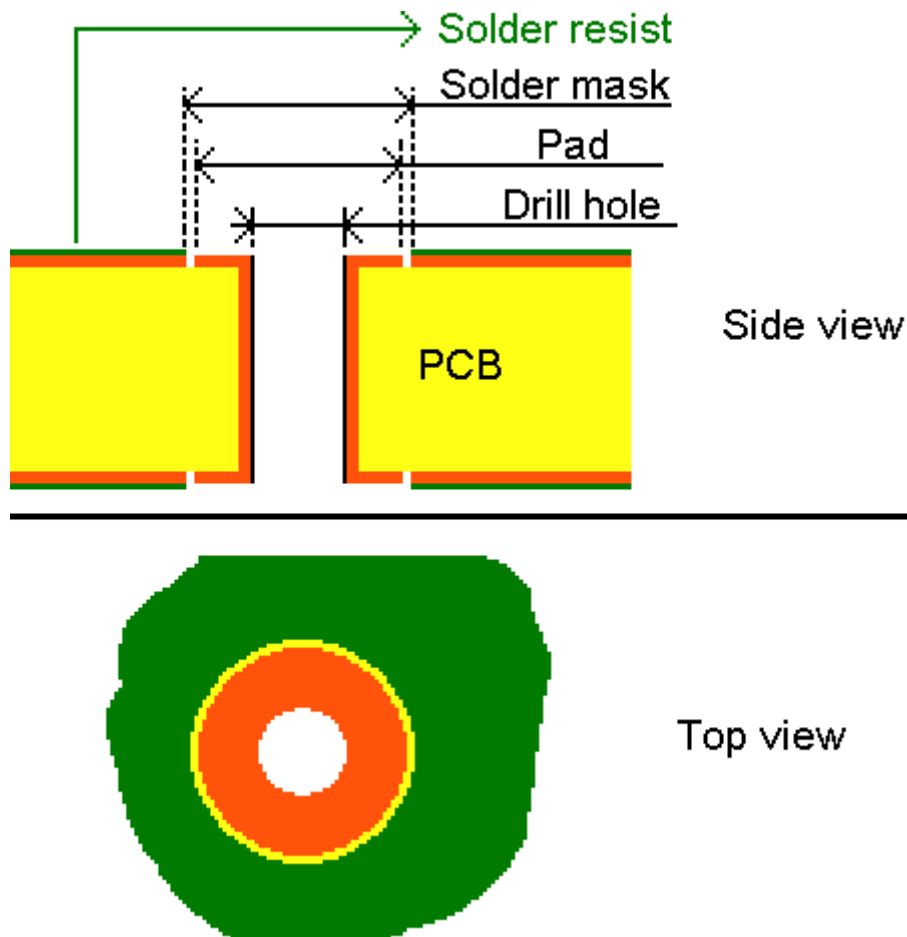
The initial clearance is the clearance used for this geometry.

Distance

The **Use default rules for solder paste/mask and clearance** button can be used the fill in values based on the pad size.

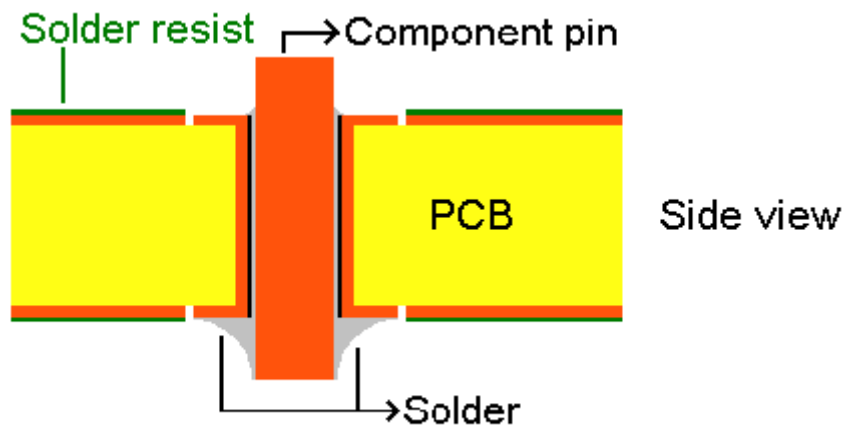
See also [Design rules pad](#)

Through hole pin



In the above figure a through hole pin is shown.

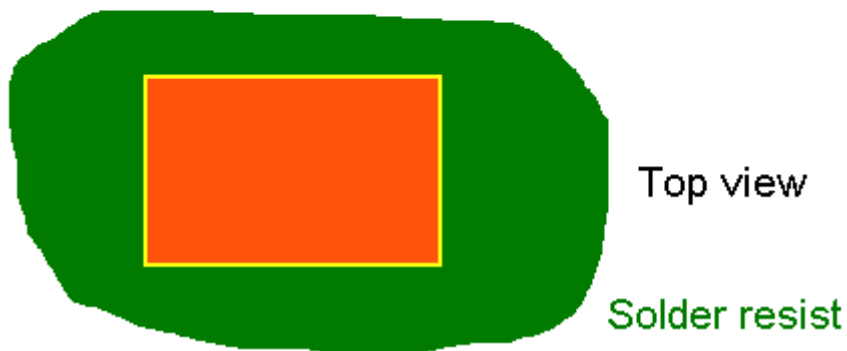
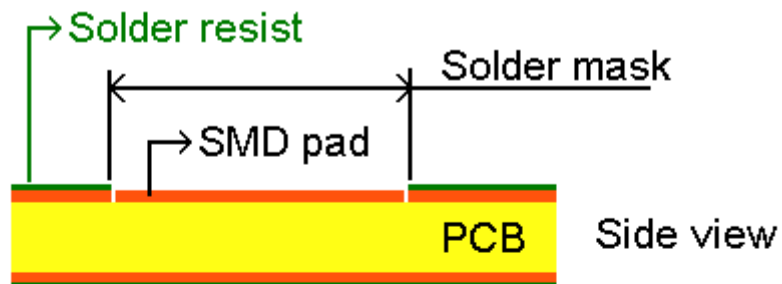
After soldering this through hole pin will look like



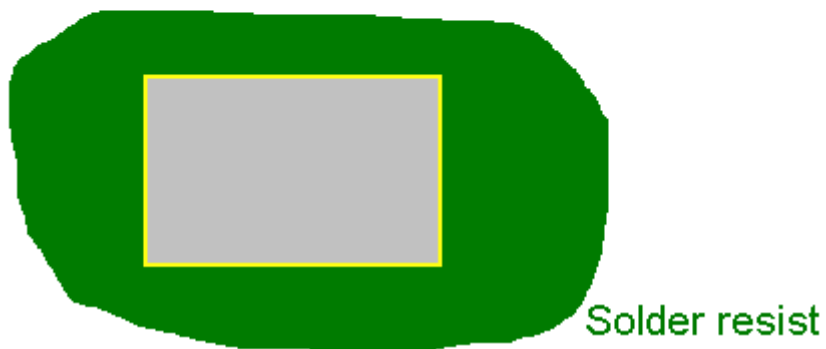
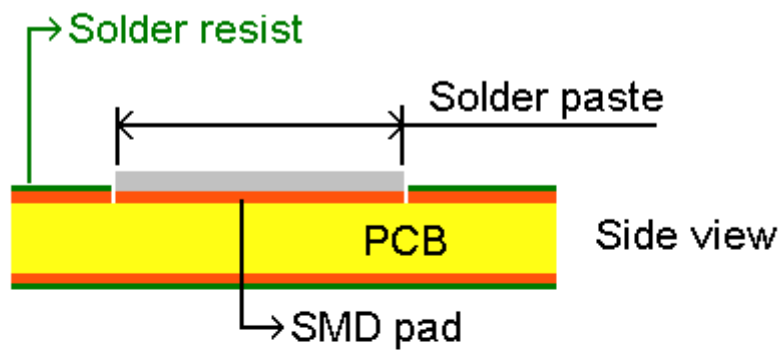
After the bottom side of the PCB has gone through a solder bath, all copper areas at the bottom of the PCB are soldered. The copper areas on the bottom PCB side that are covered with solder resist, are not soldered.

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.

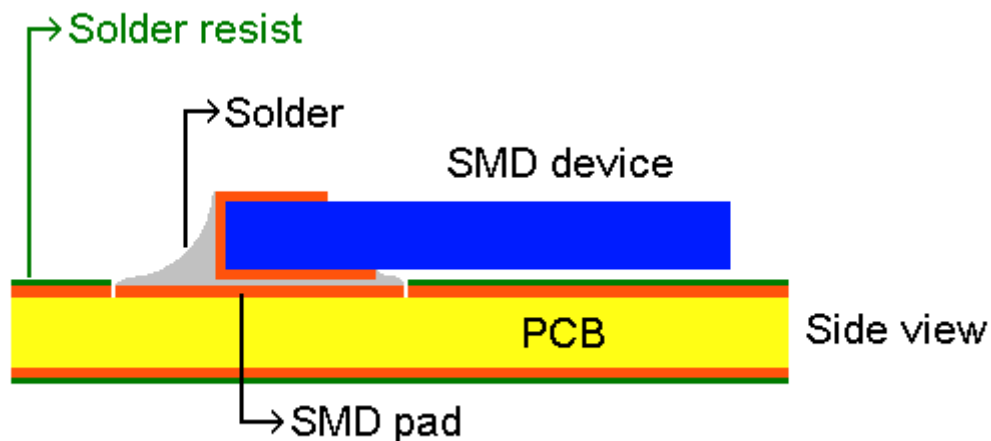
SMD pad



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.

Design rules pad

When creating a standard geometry (DIP,SOIC,BGA,PGA,QFP) a set of rules can be used for creating pads with the right padsize, soldermask/paste mask size.

For DIP/PGA the drill size will be used as a reference for the padsize, soldermask size, anti powerpad, inner padsize.

For SOIC/BGA/QFP the padsize will be used as a reference for the pastemask size, soldermask size.

In the geom.ini file there are seven rule name to define the additions to the drill or padsize.

Pad size = Drill size + DefaultRulePad (through hole)

Soldermask = Drill size + DefaultRuleSolderMask_TH (through hole)

Anti power pad = Drill size + DefaultRuleAntiPowerPad (through hole)

Inner pad = Drill size + DefaultRuleInnerPad (through hole)

Soldermask = Pad size + DefaultRuleSolderMask_SMD (SMD)

Pastemask = Pad size + DefaultRulePasteMask_SMD (SMD)

There is also a rule for the standard clearance: DefaultRuleClearance

See also [Initialisation file geom.ini](#)

Edit


Thickness line/clearance

 Mainmenu	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
- Line width info layers
- Line width board outline layer

Set origin point geometry

 Mainmenu	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

 Mainmenu	Sub menu Edit menu item Set origin to center selected objects
---	---

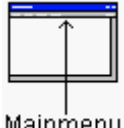
The origin of the geometry will be set to the center of the selected objects.

Set insertion point geometry

 Mainmenu	Sub menu Edit menu item Set insertion point
---	---

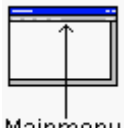
The insertion point of the geometry will be moved to the mouse position after pressing the **left mouse button**.

Set insertion point geometry to center selected objects

 Mainmenu	Sub menu Edit menu item Set insertion point to center selected objects
---	--

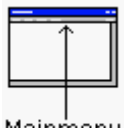
The insertion point of the geometry will be set to the center of the selected objects.

Change geometry name

 Mainmenu	Sub menu Edit menu item Change geometry name
---	--

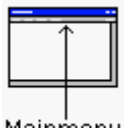
In the next dialogbox the geometry name (And also the filename) can be changed.

Number of copper layers

 Mainmenu	Sub menu Edit menu item Nr layers
---	---

In the next dialogbox the number of copper layers can be changed. Default are two layers.

Check geometry

 Mainmenu	Sub menu Edit menu item Check geometry
---	--

The current geometry will be checked for:

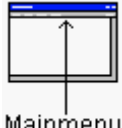
- Double pinnames
- Two copper objects overlap
- Copper object without a pinname
- Through hole pin has upper/bottom pad/soldermask or anti powerpad or inner pad
- Existing placement outline
- Existing component outline.

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

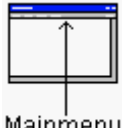
Change colors

 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 **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

 Mainmenu	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 drawn 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.

Info on selected objects

	Press Info selected objects button
 Keyboard	Press I
Menu  Mouse	Info


Displays some information about selected objects.

View vertices polygon

Menu  Mouse	View vertices polygon
--	------------------------------

After a polygon is selected, the vertices (points) of the areafill can be shown with this function. By copy/paste (Add polygon/polyline) these points a polygon can be used again.

Measure distance

<div>Menu</div>  <div>Mouse</div>	Calculate distance between objects
--	---

Measures the distance between two objects. There will be a minimum distance calculated, and a center distance.


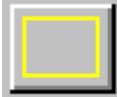


Measurement

<div>Menu</div>  <div>Mouse</div>	Measurement
---	--------------------

When activated and after clicking on the left mouse button an arrow with its length will be shown on the screen.

Add objects

Add rectangle objects

	Press Add rectangle component outline button
	Press Add rectangle placement outline button
 <div>Keyboard</div>	Press r
<div>Menu</div>  <div>Mouse</div>	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](#)

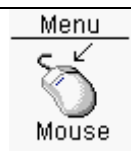
Add pad -> rectangle

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

Add rectangle

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

Add circle objects

 <p>Menu Mouse</p>	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

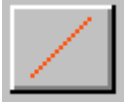



A circle (solid) pad can be added on the following layers:

Add circle

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


Add line objects

	Press Add line component outline button
---	--

	Press Add line placement outline button
 Keyboard	Press I (Placement outline layer)
 Keyboard	Press L (Component outline layer)
 Mouse	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).

Add arrow/dimension objects

 Mouse	Add arrow/dimension -> Left pointed arrow Right pointed arrow Left/Right pointed arrow Axial dimension Radial dimension
--	---




With this function three sorts of arrow objects can be added, a axial dimension object and a radial dimension object can be added.

Add arc objects

 Mouse	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).


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:


Add polyline

Menu  Mouse	Add polyline -> select layer
--	--

A polyline must be drawn. 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. When the spacebar is pressed, a dialogbox will popup, and the polyline parameters can be edited by hand. As many as 200 points can be edited. The coordinates typed in will be used with the current units (dimension).



See also [Thickness line/clearance](#)

Add polygon

Menu  Mouse	Add polygon -> select layer
--	---------------------------------------




A polyline must be drawn. 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. When the spacebar is pressed, a dialogbox will popup, and the polyline parameters can be edited by hand. As many as 200 points can be edited. The coordinates typed in will be used with the current units (dimension).

Add drill

	Press Add unplated drill hole button
<div>Menu</div>  <div>Mouse</div>	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
<div>Menu</div>  <div>Mouse</div>	Add pad -> SIL SMD rectangle pads
<div>Menu</div>  <div>Mouse</div>	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.

The **Use default rules for solder paste/mask and clearance** button can be used the fill in values based on the pad size.

See also SMD pad

See also Design rules pad

Add circle SMD pads with solder and paste mask

	Press Add round SMD pads button
<div>Menu</div>  <div>Mouse</div>	Add pad -> SIL SMD rectangle pads
<div>Menu</div>  <div>Mouse</div>	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



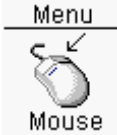
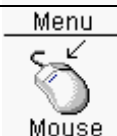
The startpin consists of two editboxes. The first editbox (optional) contains text or a number, and will not be changed. The second editbox contains a start number. This startnumber will be increased with **Increment** for the next pads.

The **Use default rules for solder paste/mask and clearance** button can be used the fill in values based on the pad size.

See also SMD pad

See also Design rules pad

Add through hole pads with solder mask and drill hole

	Press Add round through hole button
	Press Add square through hole button
 Menu Mouse	Add pad -> SIL through hole pads
 Menu Mouse	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.

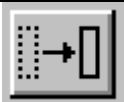


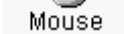
The **Use default rules for solder paste/mask, inner/anti power pad and clearance** button can be used the fill in values based on the drill size.

See also Through hole pin

See also Design rules pad


Change objects

Move objects

	Press Move button
 Keyboard	Press m
 Menu  Mouse	Move

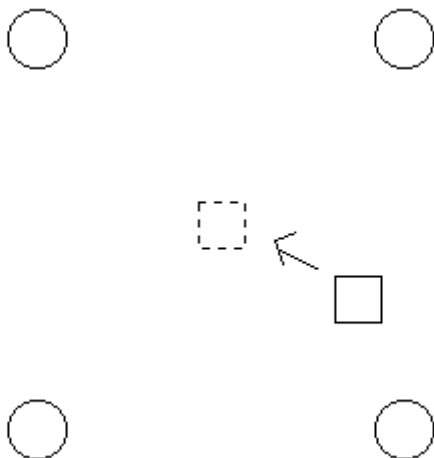
Move **selected** objects. Normally after the left button the objects will be moved to the object under the mouse cursor using a snap function. This snap function can be disabled via the right mouse button menu. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. The snap function will also be used for the moving center. 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)

<div data-bbox="180 678 311 817"> <div>Menu</div>  <div>Mouse</div> </div>	<p>Special move/centering -> Mark center selected objects Special move/centering -> Move objects centered to previous selected objects</p>
---	---

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

<div data-bbox="180 1848 311 1935">  </div>	<p>Press Copy button</p>
--	---------------------------------

 Keyboard	Press c
 Mouse	Copy


With this function **selected** objects can be copied to a new location. Normally after the left button the objects will be copied to the object under the mouse cursor using a snap function. This snap function can be disabled via the right mouse button menu. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change. The snap function will also be used for the moving center. 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

 Mouse	Copy to other layer -> Select layer
---	---


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. Not all objects can be copied to the desired layer.

Move objects to a different layer

 Mouse	Move to other layer -> Select layer
--	---




With this function **selected** objects can be moved to the specified layer. Selected objects on the same layer as the specified layer will not be moved. Not all objects can be moved to the desired layer.

Copy on multiple coordinates

 Mouse	Modify objects -> 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	Modify objects -> Rotate objects Modify objects -> Rotate objects at any angle

Rotate:

Rotate **selected** objects 90 degrees counter clock wise.

Rotate objects at any angle:


In the next dialogbox the angle (counter clock wise) can be put in. Selected text will be rotated in 45 degrees increments.

Mirror objects

Menu  Mouse	Mirror X Mirror Y
--	------------------------------------


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

Scale objects

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Scale
--	-----------------------------------


Selected objects can be scaled by a value entered by the user. There is an option for “scale per object”. If this option is marked all objects will be scaled individually and their place remains the same. If this option is not marked, all objects will be scale and moved.

Change circle objects

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change diameter circles
--	---


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

Change rectangle objects

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change width/height rectangles
--	--


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

Change diameter arc objects

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change diameter arc
--	---



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

Change angle arc objects

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change angles arc
--	---


The start and end of **selected** arcs can be changed into the values typed in the following dialogbox.

Change text

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change text
 <div>Keyboard</div>	Press e


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

Change text height

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change textheight
--	---


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

Change line width

<div>Menu</div>  <div>Mouse</div>	Modify objects -> Change line width
--	---


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

Change clearance

<div>Menu</div> <div></div> <div>Mouse</div>	Modify objects -> Change clearance
---	--


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

Convert lines into polygon

<div>Menu</div> <div></div> <div>Mouse</div>	Modify objects -> Convert lines into polygon
---	--


If the **selected** line objects form a polyline, this polyline will be converted into a polygon.

Cut from object

<div>Menu</div> <div></div> <div>Mouse</div>	Modify objects -> Cut from object
---	---


Cut a polygon area from an object. The required polygon must now be drawn.

Merge objects to polygon

<div>Menu</div> <div></div> <div>Mouse</div>	Modify objects -> Merge objects to polygon
---	--


Multiple selected objects which overlaps each other can be combined to one polygon with this function.

Unselect objects

<div>Menu</div> <div></div> <div>Mouse</div>	Unselect -> Select object layer
---	---



Unselect objects.

Select only

<div>Menu</div> <div></div> <div>Mouse</div>	Select only -> Select object layer
---	--

Select only objects.

Assign objects to pin

<div>10</div>	Press Assign objects to pin button
<div></div> <div>Keyboard</div>	Press a
<div>Menu</div> <div></div> <div>Mouse</div>	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.

Index

A

Accelerator, 45, 95, 140
Add drill holes, 81
Add on silkscreen, 80
Add paste mask pads, 81
Add solder mask pads, 81
Air-lines, 30
Annotation, 18
Areafill, 73
Areafill merging, 77

B

Back annotation, 19
Bill Of Materials, 20

C

Change height component reference, 51
Change height component value, 51
Change line width component references, 51
Change line width component value, 51
Change units, 47
Change visibility component reference, 51
Change visibility component value, 51
Check, 21
Component list, 20
Copper pour, 73

D

Deinstall, 2
Deselect all, 41
Deselect objects, 40

G

Gate/pin swap, 88, 99
Gerber output, 31
Guide wires, 30

H

Hide component references, 51
Hide component values, 51
Hide layers, 42
Hierarchy, 118
HPGL, 33

I

Importing components/netlist, 30
Info layer, 81
Info layer 2, 81
Info layer 3, 81
Info layer 4, 81
Install, 2

L

Library manager, 25
List of components, 20
Loading geometries in layout file, 28
Loading symbols in schematic file, 92

M

Move component reference text to bottom layer, 51
Move component reference text to top layer, 51
Move component to top layer, 49

N

Netlist, 30

O

ORCAD, 23
ORCAD libraries, 23
ORCAD schematics, 23

P

Pan window, 43
Paste mask, 81
Penplot output, 33
Pinbus, 100
Power text, 117
Powerplane, 71
Previous view, 44
Print, 29, 34
Programmable keys, 45, 95, 140
Protect components, 51, 52, 53, 103

R

Redo, 40
Repaint, 44
Restart annotation, 18

S

Select component by reference, 38
Select objects, 40
Service lines, 30
Short cuts, 45, 95, 140
SMD pad, 135
Solder mask, 81
Subsheets, 118

T

Thermal relief, 32
Through hole pin, 133

U

Undo, 39

V

Via definitions, 39
View whole design, 44
View/hide grid, 47

W

Window based panning, 43

Window based zooming, 43

Z

Zoom in, 42
Zoom out, 42

