

Getting started



Select objects



Viewable window



Zoom in



Zoom out



Pan window



Schematic link



Activate component menu



Activate routing menu



Activate change traces/vias menu



Drag one trace



Activate areafills/powerplanes menu



Component references menu



Component values menu



Add solder mask pads



Add paste mask pads

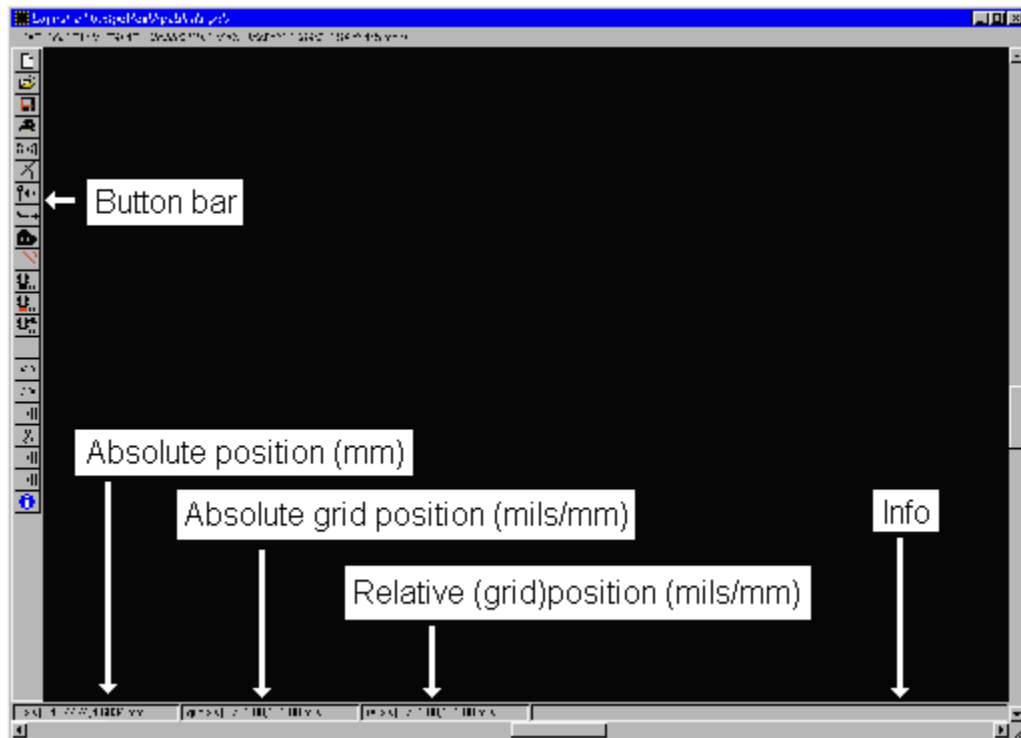


Add drill holes

Layout editor PCB elegance

The layout tool of PCB elegance is a tool to design a PCB (Printed circuit board). PCBs are being used in the electronics industrie and consumer electronics.

Viewable window



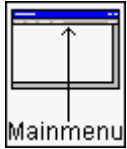
In the viewable window there are four bars visible who have a function. Those four bars are the **Button bar**, **Absolute position (mm)**, **Absolute grid position (mils/mm)**, **Relative (grid)position (mils/mm)** and **Info**.

The **Absolute grid position (mils/mm)** displays the current gridposition of the mouse cursor. The dimension depends on the current units, and is in mils or mm. The **Relative (grid)position (mils/mm)**



Zero relative (grid)position

Make new layout



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

In the next dialogbox the parameters for a new design can be entered.

The width,height and origin of the PCB can be entered. The number of layers (Trace layer + powerplanes) can be entered. The standard via definition parameters can be entered. The standard design rules for trace width, clearance, linewidth silkscreen can be entered. Also the board outline keep out can be entered. If the board outline keepout is greater then zero, the design rule check will be increased with the board outline keepout check. This board outline keepout check will only be executed when the board outlines consists of closed objects.

When the **mils/mm** button is pressed the dimension (units) will switch between mils and mm, also every parameter will be recalculated for the new dimension.



Design rules when using a printer for the plot outputs



Importing components/netlist

Backup



Backup

Design rules when using a printer for the plot outputs

When a standard printer (inkjet- or laserprinter) is being used, there is a optimum trace/clearance width 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

Print screen



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



Press **Print** button

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

Importing components/netlist



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

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

Importing components

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

Calculating connections (ratsnest)

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

Updating components/netlist



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

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

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

Plot output to gerber format



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

Plot **selected** layers in the gerber format. In the next dialogbox the layers, the gerber output format (RS274D or RS274X), X mirroring, plotting board outline and the gerber output number format can be selected. For the silkscreen layers there is an option to plot the component references or the component values. It is also possible to modify the pen sizes which will be used to fill an areafill. The small pen will be used to draw the surrounding of the polygons, and the thick pen will be used to fill the rest of the areafill with horizontal strokes. There are also two editboxes available. In those two editboxes (each four lines) some information about the PCB can be stored. This information will then be plotted additionally for each layer. Initial three macros are stored into the first editbox.

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

After pressing the **OK** button the plotfiles will be generated.

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

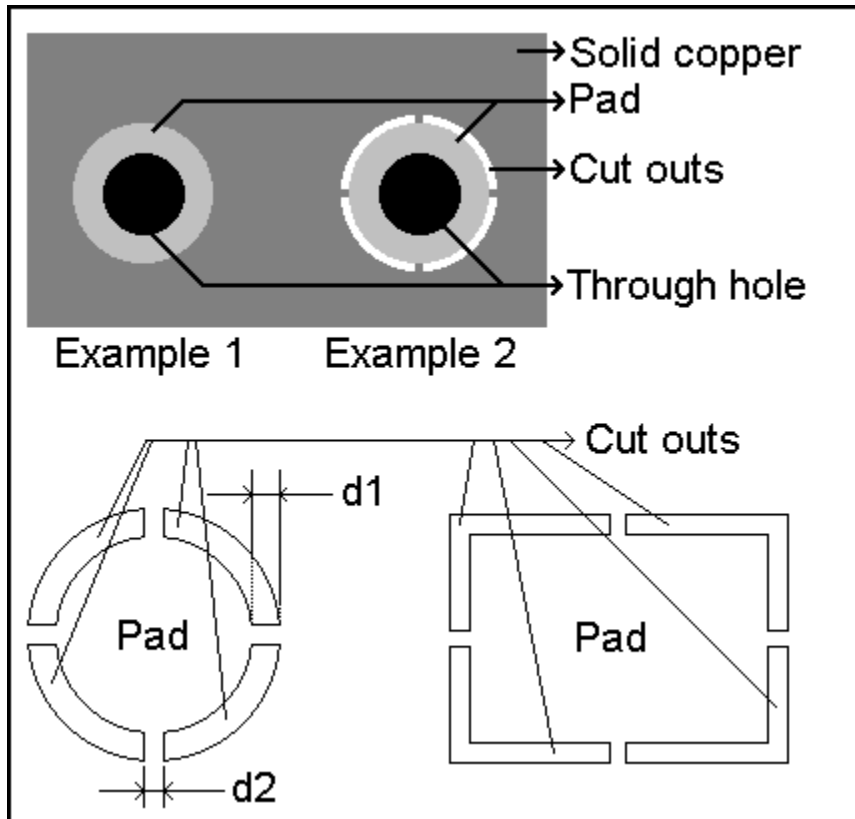
The aperture file (generated automatically) will have the name **gerber.txt**. The drill file (Excellon format) will have the name **<design>.drl** and the drill tool file the name **drills.txt**. The PCB information is stored in the **layers.txt** file.

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



Thermal relief

Thermal relief



In the above figure there are two examples. The first example is a through hole pin without thermal reliefs, and the second example a through hole pin with thermal reliefs. 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.

Penplot output



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

Penplots the **selected** layers to one or more files. In the next dialogbox the layers, scale factor, pensize(s), origin, plotting board outline and mirror X can be selected. For the silkscreen layers there is an option to plot the component references or the component values. After pressing the **OK** button the plotfiles will be plotted.

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

If possible drill holes will be left open, by shorten traces.

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



Plot output to gerber format



Thermal_relief

Plot output to printer



Sub menu **File** menu item **Print plot files**

Plots the **selected** layers to the printer. In the next dialogbox the layers, scale factor, drawing board outline and mirror X can be selected. For the silkscreen layers there is an option to plot the component references or the component values. After pressing the **OK** button the plotfiles will be printed.



Design rules when using a printer for the plot outputs



Plot output to gerber format



Thermal_relief

Output component positions



Sub menu **File** menu item **Output component position**

Output component position (mils)

Output component position (mm)

Output component position (inch)

Output component position (mils)

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

Output component position (inch)

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

Output component position (mm)

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

Output netlist



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

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

Reload geometries



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

When a geometry used in the design has being changed by the geometry editor, the designs geometries can be reloaded with this function.

Initialisation file pcb.ini

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

The following paramaters are used:

[Settings]

WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
GeometryDialogInitialX	Origin X of the geometry selection dialog window
GeometryDialogInitialY	Origin Y of the geometry selection dialog window
GeomStartX	Origin X of the geometry window
GeomStartY	Origin Y of the geometry window
GeomScreenWidth	The width of the geometry window
GeomScreenHeight	The height of the geometry window
Units	(0 = mils, 1 = mm)
GridSize	The gridsize (10nm units)
CompGridSize	The comp gridsize (10nm units)
TraceGridSize	The trace gridsize (10nm units)
DrawGrid	0 = FALSE, 1 = TRUE
DrawAreaFills	0 = FALSE, 1 = TRUE
DrawAreaFillMode	0 = None 4 = With hatch 5 = Gerber emulation plot surroundings 6 = Gerber emulation plot horizontal fill 7 = Both

	8 = Solid
DrawClearances	0 = FALSE,1 = TRUE
DrawCompOutline	0 = FALSE,1 = TRUE
DrawConnections	0 = FALSE,1 = TRUE
DrawDrills	0 = FALSE,1 = TRUE
DrawInnerPads	0 = FALSE,1 = TRUE
DrawTopPads	0 = FALSE,1 = TRUE
DrawBottomPads	0 = FALSE,1 = TRUE
DrawDrillType	0 = FALSE
	1 = Cross type 1
	2 = Cross type 2
	3 = Drill hole
DrawSoldMaskBottomMode	0 = FALSE
	1 = Only extra pads
	2 = All pads
DrawSoldMaskTopMode	0 = FALSE
	1 = Only extra pads
	2 = All pads
DrawPasteMaskBottomMode	0 = FALSE
	1 = Only extra pads
	2 = All pads
DrawPasteMaskTopMode	0 = FALSE
	1 = Only extra pads
	2 = All pads
DrawCompPlacement	0 = FALSE,1 = TRUE
DrawSilkScreen	0 = FALSE,1 = TRUE
DrawObjects	0 = FALSE,1 = TRUE
DrawVias	0 = FALSE,1 = TRUE
DrawViaClearances	0 = FALSE,1 = TRUE
DrawCompReference	0 = FALSE,1 = TRUE
DrawCompValue	0 = FALSE,1 = TRUE
DrawTwoTryingTraces	0 = FALSE,1 = TRUE

DrawBoardOutline	0 = FALSE, 1 = TRUE
SelectionMode	0 = replacement, 1 = appending
RecalcNetsAfterMove	0 = FALSE, 1 = TRUE
RecalcAreafillAfterInsert	0 = FALSE, 1 = TRUE
MouseCursorOnGrid	0 = FALSE, 1 = TRUE
Layer0	Draw bottom layer 0 (0 = FALSE, 1 = TRUE)
Layer1	Draw layer 1 (0 = FALSE, 1 = TRUE)
Layer2	Draw layer 2 (0 = FALSE, 1 = TRUE)
...	...
Layer31	Draw layer 31 (0 = FALSE, 1 = TRUE)
AreafillPen1	Gerber plot pen1 (Thick pen) used to fill an areafill
AreafillPen2	Gerber plot pen2 (Small pen) used to fill an areafill
PlotPen1	Penplot pen1
PlotPen2	Penplot pen2
Grid0	Gridsize definition 0 (10nm units)
Grid1	Gridsize definition 1 (10nm units)
...	...
Grid29	Gridsize definition 29 (10nm units)
TraceWidth0	Trace width definition 0 (10nm units)
TraceWidth1	Trace width definition 1 (10nm units)
...	...
TraceWidth29	Trace width definition 29 (10nm units)
ClearanceWidth0	Clearance width definition 0 (10nm units)
ClearanceWidth1	Clearance width definition 1 (10nm units)

...

ClearanceWidth29

...

Clearance width definition 29 (10nm units)

BackColor

24 bit RGB color (Stored as 32 bit)

ViewLayer1Color

24 bit RGB color (Stored as 32 bit)

ViewLayer2Color

24 bit RGB color (Stored as 32 bit)

ViewLayer3Color

24 bit RGB color (Stored as 32 bit)

ViewLayer4Color

24 bit RGB color (Stored as 32 bit)

ViewLayer5Color

24 bit RGB color (Stored as 32 bit)

ViewLayer6Color

24 bit RGB color (Stored as 32 bit)

ViewLayer1HilitedColor

24 bit RGB color (Stored as 32 bit)

ViewLayer2HilitedColor

24 bit RGB color (Stored as 32 bit)

ViewLayer3HilitedColor

24 bit RGB color (Stored as 32 bit)

ViewLayer4HilitedColor

24 bit RGB color (Stored as 32 bit)

ViewLayer5HilitedColor

24 bit RGB color (Stored as 32 bit)

ViewLayer6HilitedColor

24 bit RGB color (Stored as 32 bit)

ConnectionsColor

24 bit RGB color (Stored as 32 bit)

ConnectionsHilitedColor

24 bit RGB color (Stored as 32 bit)

NetPinsColor

24 bit RGB color (Stored as 32 bit)

NetPinsColor2

24 bit RGB color (Stored as 32 bit)

SilkScreenTopColor

24 bit RGB color (Stored as 32 bit)

SilkScreenBottomColor

24 bit RGB color (Stored as 32 bit)

ReferenceColor

24 bit RGB color (Stored as 32 bit)

CompValueColor

24 bit RGB color (Stored as 32 bit)

ShapePlacementOutLineColor

24 bit RGB color (Stored as 32 bit)

ShapeCompOutLineColor

24 bit RGB color (Stored as 32 bit)

ShapePinsTopColor

24 bit RGB color (Stored as 32 bit)

ShapePinsDrillColor

24 bit RGB color (Stored as 32 bit)

ShapePinsDrillUnplatedColor

24 bit RGB color (Stored as 32 bit)

ShapePinsBottomColor

24 bit RGB color (Stored as 32 bit)

ViaPinsColor

24 bit RGB color (Stored as 32 bit)

ViaPinsHilitedColor

24 bit RGB color (Stored as 32 bit)

ViaPinsDrillColor	24 bit RGB color (Stored as 32 bit)
ObjectsInfoColor	24 bit RGB color (Stored as 32 bit)
ClearanceColor	24 bit RGB color (Stored as 32 bit)
ErrorColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)
BoardOutlineColor	24 bit RGB color (Stored as 32 bit)
ObjectsInfoColor	24 bit RGB color (Stored as 32 bit)
ObjectsInfo2Color	24 bit RGB color (Stored as 32 bit)
PasteMaskTopColor	24 bit RGB color (Stored as 32 bit)
PasteMaskBottomColor	24 bit RGB color (Stored as 32 bit)
SoldMaskTopColor	24 bit RGB color (Stored as 32 bit)
SoldMaskBottomColor	24 bit RGB color (Stored as 32 bit)
SwappablePinsGateColor	24 bit RGB color (Stored as 32 bit)
SwappableGatePinsColor	24 bit RGB color (Stored as 32 bit)

Move entire PCB



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

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

Change design rules



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

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

Zero relative cursor



Press **Ctrl z**

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

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



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

Center view on component



Press **Ctrl c**

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



Activate component menu

Via definition



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

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

Component protection



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

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



Make selections in dialog listboxes



Protect selected components

Undo

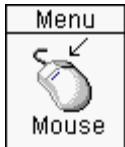


Press **Undo** button



Keyboard

Press **u**



Menu

Mouse

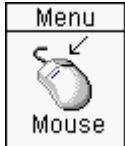
Undo

This function will undo almost all previous actions. The undo function will not undo **Importing components/netlist**, and **Updating components/netlist**.

Redo



Press **Redo** button



Redo

This function will redo previous undo actions.

View topics



[Zoom in](#)



[Zoom out](#)



[Window based Zooming](#)



[Hide/view layers](#)



[Pan window](#)



[Window based panning](#)



[Return to previous view window](#)



[Repaint](#)



[View whole design](#)



[Change grid](#)

Hide/view layers



Press **Select layers** button



Press **Ctrl a**



Sub menu **View** menu item **Layers**

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

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

When the button **Drill type** is pressed a dialogbox will be visible. In this dialogbox the drill type can be modified. When one of the first two options is active, the drills will be made visible by means of a cross. When the third option is active the actual drill hole will be displayed, in the current background color.

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.



Change grid



Change hatch areafill

Zoom in



Press **z**



Sub menu **View** menu item **Zoom in**

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



Window based zooming

Zoom out



Press **Z**



Sub menu **View** menu item **Zoom out**

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



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



Press $\leftarrow, \rightarrow, \uparrow, \downarrow$



Press **x**



Press **Shift** and move the mouse the window border

Window

Use the scrollbars

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

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



Window based panning

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, then press and hold down the **right** mouse button. The non changing rectangle visible is the border of your design. The changing rectangle is the viewable window. After releasing the **Ctrl** key and the **right mouse button** panning will take place.

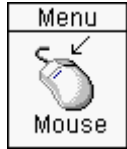
The **Window based panning** function can be used in every possible drawing/moving function.

During normal operation and by pressing the **Shift** key, moving the mouse cursor towards the end of the window, the window will pan.

Return to previous view window



Press **v**



Previous view

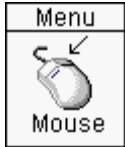
Return to a previous view.

The **Return to previous view window** function can be used in every possible drawing/moving function.

Repaint



Press **F5**



Repaint

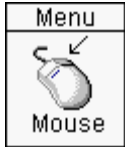
The whole window will be repainted.

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

View whole design



Press **Shift F8**



View whole design

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

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

Change colors



Sub menu **View** menu item
Change colors

The color settings can be modified in the next dialogbox. The color settings will be copied into the **pcb.ini** 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 main directory will be copied to the **pcb** subdirectory of the new design.



Initialisation file **pcb.ini**



Load default colors

Load default colors



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

The default color settings will be loaded.



Initialisation file pcb.ini

Short cuts

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

Change grid



Press **Ctrl g**



Sub menu **View** menu item
Change grid

In the dialogbox the grid settings can be modified. There are three main grid settings.

The **default grid** setting will be used normally.

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

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

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

Changing the grid is possible in every drawing/moving function.

The grid settings in the dialogbox can be modified by changing the **pcb.ini** settings.



[Initialisation file pcb.ini](#)

View/hide grid



Press **g** (View/hide grid)

View/hide grid

Change units



Press **Ctrl u**



Sub menu **Units** item

Mm

mils

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



[Initialisation file pcb.ini](#)

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 left 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 and place the selection rectangle around this object again.

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



Deselect all

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



Press **Deselect all** button



Press **F2**



Deselect all

Deselect all function.

Info on selected objects



Press **Info selected objects** button



Press **i**



Info

Displays some information about **selected** objects.

Insert layer



Sub menu **Edit** menu item **Insert layer**

A new layer will be inserted. This operation can not be undone.

Remove layer



Sub menu **Edit** menu item **Remove layer**

A layer will be removed. This operation can not be undone.

Swap layers



Sub menu **Edit** menu item **Swap two layers**

Two layers will be swapped. This operation can not be undone.

Components



Press **Move/rotate/change components** button



Press **c**



Other menus -> Components menu

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

Activation of the **Components** menu is made visible on the info bar at the bottom right of the window. **Move/rotate/change components** is now visible. Also the **Move/rotate/change components** button is visible pressed.



Move components



Move component by reference



Rotate components



Edit geometry component



Change geometry component



Regroup components



Protect components



Component protection



Move components to top/bottom layer



Change component parameters



Schematic link



Info on selected components



Copy placement outline to a different layer



Copy component outline to a different layer

Move component by reference

COMPONENT MENU



Press r



Move component by reference

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



Move component

Move components

COMPONENT MENU



Press **m**



Move

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



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).



Press the right mouse button

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



Activate component menu



Select components



Schematic link



Zero relative (grid)position



Relative (grid)position



Recalculate ratsnets after moving components



Recalculate areafill after moving components

Rotate components



Press **R**



Rotate

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



Activate component menu



Select components



Activate component menu



Schematic link



Zero relative (grid)position



Relative (grid)position



Recalculate ratsnets after rotating components



Recalculate areafill after rotating components

Move components to top/bottom layer

COMPONENT MENU



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.



Activate component menu



Select components

Regroup components

COMPONENT MENU



Regroup

Regroup **Selected** components.

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



Activate component menu



Select components

Edit geometry

COMPONENT MENU



Edit geometry component

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



Activate component menu



Reload geometries



Select components

Change geometry

COMPONENT MENU



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.



Activate component menu



Reload geometries



Select components

Change component parameters

COMPONENT MENU

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 text height component reference/value
 - Change text line width component reference/value
-



Component references

Hide

Visible

On top layer

On bottom layer

Height

Line width



Component values

Edit

Hide

Visible

On top layer

On bottom layer

Height

Line width



Activate component menu



Select components

Protect components



Protect

The **Selected** components will be protected. Protected components can **not** be selected. To unprotect components use [Component protection](#).

Component info

COMPONENT MENU



Press **Info** button



Press **I**



Component info

Parameters of **selected** components will be shown, using this function.



Activate component menu



Select components

Copy component placement outline to a different layer

COMPONENT MENU



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

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

Copy component outline to a different layer



**Copy component outline to ->
Layer**

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

Delete components

Components can not be deleted in the layout editor, instead components should be deleted in the schematic editor. After **Update components/netlist** components are deleted.

Change design rules nets



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

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



Make selections in dialog listboxes

Highlight/unhighlight nets



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

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



Highlight/unhighlight net during trace drawing



Make selections in dialog listboxes



Unhighlight all

Disable connections nets



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

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

Disabled net connections are useful for the power nets, in combination with powerplanes, because when using powerplanes for power nets, connections are confusing.



Make selections in dialog listboxes

Hide connections nets



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

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



Make selections in dialog listboxes



Hide all connections



View all connections

Highlight visible connections



Sub menu **Connections** menu item **Highlight visible connections**

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



Make selections in dialog listboxes

Unselect traces/vias nets



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

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



Make selections in dialog listboxes

Recalculate ratsnets after move



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

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

Delete traces/vias nets



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

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



Make selections in dialog listboxes

Unhighlight all



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

Traces/vias/connections will be unhighlighted.

[View all connections](#)



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

Hidden connections will be made visible.



[Hide all connections](#)

[Hide all connections](#)



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

Visible connections will be hidden.



[View all connections](#)

Traces/vias



Press **Routing traces** button



Press **s** (For a maximum of three times)



Other menus -> Routing menu

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

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

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

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



Add trace



Add extra trace



Start routing with the shortest net

Add trace

ROUTING MENU

When the **Routing** menu is active, new traces can be added, and existing traces modified. 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 neighbourhood 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 trace drawing will stop. Another possibility to stop the trace drawing is to press the **ESC** key. When a new trace segment is added inside a 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.



Right mouse button menu

A via can be added with the function Add via



Activate routing menu



Trace drawing feature



Recalculate areafill after inserting traces

Trace popup menu

When trace drawing is active the following functions are available when the right mouse button is pressed.:



Display clearance



Display two trying traces



Display via option



Highlight/unhighlight net



Switch to another layer



Delete trace



Goto previous trace segment



Change trace width

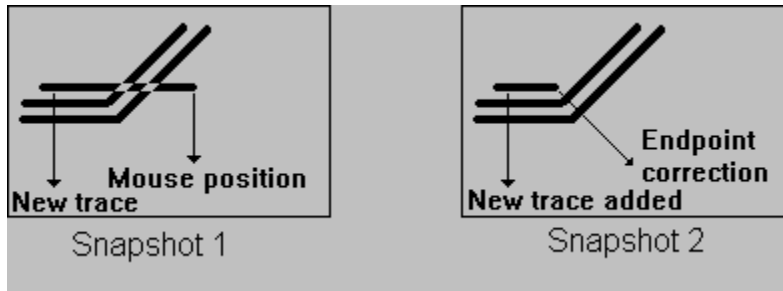


Change clearance



Change cross hair of the mouse cursor

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, to 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

ROUTING MENU



Press .



Press the left mouse button twice (double click)

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



Recalculate areafill after inserting a via

Display clearance

ROUTING MENU



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

ROUTING MENU



Display two trying traces -> on

Display two trying traces -> off

With this function trace drawing is taking place with one or with two trying traces.

Display via option

ROUTING MENU



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.



Add via

Finish trace

ROUTING MENU



Press **Space bar**



Finish trace

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

Highlight/unhighlight net

ROUTING MENU



Highlight net

Unhighlight net

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



Highlight/unhighlight nets

Switch to another layer

ROUTING MENU



Press **F4**



Select layer

When the start point of the current trying trace is connected to a through hole pin, or a via switching to a layer different 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



Delete trace

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

Goto previous trace segment



Press **b**



Trace backwards

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

Change trace width



Change trace width

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

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



[Initialisation file pcb.ini](#)

Change clearance



Change clearance

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

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



Initialisation file pcb.ini

Change cross hair of the mouse cursor



Cross hair type

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

Add extra trace



Add extra trace

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



Recalculate areafill after inserting traces

Start routing with the shortest net



Start routing with the shortest net



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.



Recalculate areafill after inserting traces

Change traces/vias



Press **Change traces/vias** button



Press **s** (For a maximum of three times)



Other menus -> **Change traces/vias** menu

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

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

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

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

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



Move traces/vias



Copy traces/vias



Select only



Change trace width



Change clearance traces/vias



Change via



Calculate length trace



Swap traces/vias two nets



Delete traces/vias net selected trace



Delete

Move traces/vias

Change traces/vias menu



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.



Activate change traces/vias menu



Recalculate ratsnets after moving traces/vias



Recalculate areafill after moving traces/vias

Copy traces/vias

Change traces/vias menu



Copy traces/vias

The **selected** traces/vias of one net will be copied to the desired location. This location will be a pin. 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.



Activate change traces/vias menu



Recalculate areafill after copying traces/vias

Select only

Change traces/vias menu



Select only -> Traces

Select only -> Vias

Traces or vias will be selected only with this function.



Unselect traces/vias net

Change trace width

Change traces/vias menu



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.



[Activate change traces/vias menu](#)



[Recalculate areafill after changing trace widths](#)

Change clearance traces/vias

Change traces/vias menu



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.



[Activate change traces/vias menu](#)



[Recalculate areafill after changing clearance traces/vias](#)

Change via

Change traces/vias menu



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.



Activate change traces/vias menu



Recalculate areafill after changing vias

Calculate length trace

Change traces/vias menu



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.



Activate change traces/vias menu

Swap traces/vias two nets

Change traces/vias menu



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.



Activate change traces/vias menu



Select traces

Delete traces/vias net selected trace

Change traces/vias menu



Delete traces/vias net selected trace

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



Activate change traces/vias menu



Select traces

Delete



Press **Delete** button



Press **Del**



Delete

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



Activate change traces/vias menu

Drag one trace



Press **Drag one trace** button



Press **s** (For a maximum of three times)



Other menus -> Drag one trace menu

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

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

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

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

When a trace is selected dragging will be activated The dragging of this trace will be real time displayed. 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.**



Recalculate areafill after dragging one trace



Trace drawing feature

Dragging traces/vias/components



Drag traces/vias/components button



Other menus -> Drag traces/vias/components
menu

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

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

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

The connections of the nets involved are being recalculated.



Recalculate ratsnets after deleting traces/vias



Recalculate areafill after dragging component/traces/vias

Check connectivity



Sub menu **Check** -> **Connectivity**

The connectivity can be checked for all nets, or for one net. The connectivity check for all nets means that all nets (inclusive hidden or disabled) will be checked. When the check is completed all the new calculated connections except for the disabled nets will be made visible. The nets with connectivity errors will be put into a dialogbox.

Also the connectivity of one net can be checked. In the next dialogbox the net can be selected. After pressing the **OK** button the connectivity for that net will be checked.

Check design rules



Sub menu **Check** -> **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/warning will be made visible.

If the board outlines consists of closed segments, and the board outline keepout parameter is greater than zero, all the pads/traces/vias will be checked if within these board outlines.



[Change design rules](#)

View design rule errors



Sub menu **View** -> **Select error/warning**

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

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

Powerplanes



Press **Areafills/Powerplanes** button



Press **a**



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 pownets (VCC,3V3,GND). For those nets it is necessary to have a 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

AREAFILL MENU



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.

A description for thermal reliefs can be found at [Thermal relief](#)



[Activate areafills/powerplanes menu](#)

Remove powerplane

AREAFILL MENU



Remove powerplane -> Select layer

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



Activate areafills/powerplanes menu

Cut from powerplane

AREAFILL MENU



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 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



[Activate areafills/powerplanes menu](#)

Change powerplane

AREAFILL MENU



Change powerplane -> Select layer

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



Activate areafills/powerplanes menu



Thermal relief

Areafills



Press **Areafills/Powerplanes** button



Press **a**



Other menus ->Areafills/powerplanes menu

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

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

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

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



Add areafills



Add areafill inside a powerplane



Cut from areafill



Change areafill



Change hatch areafill



Delete areafill



Merge areafills



Rebuild areafill



Powerplanes



Recalculate areafill after inserting objects

Add areafill

AREAFILL MENU



Add areafill -> Select layer

In the next dialogbox the net can be selected, and the areafill clearance can be specified. Also some parameters for thermal reliefs can be specified. A description for thermal reliefs can be found at ...

After pressing the **OK** button a polyline must be drawn. When drawing this polyline all the pins of the areafill net will be highlighted (yellow). When drawing this polyline use the 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 64 points can be edited. The coordinates typed in will be used with the current units (dimension). 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 of other nets. During calculation of the areafill, any object (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.



Thermal relief



Activate areafills/powerplanes menu

Add areafill inside a powerplane

AREAFILL MENU



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 the drawing of the polyline is 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.



Thermal relief



Activate areafills/powerplanes menu

Cut from areafill



Cut from areafill ->

Polyline

Rectangle

Circle

Horizontal trace

Vertical trace ->

Select layer

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



[Activate areafills/powerplanes menu](#)



[Cut from powerplane](#)

Rebuild areafill



Rebuild areafill

With this function an areafill can be rebuild.



[Activate areafills/powerplanes menu](#)

Change hatch areafill



Change hatch areafill -> select hatch

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



Activate areafills/powerplanes menu

Change areafill



Change areafill

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



Thermal relief



Activate areafills/powerplanes menu

Merge areafills



Merge

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



[Activate areafills/powerplanes menu](#)

Recalc areafill after inserting an object



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

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

Delete areafill



Press **Delete** button



Delete



Press **Del**

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



Activate areafills/powerplanes menu

Moving component reference



Press **Move component references** button



Other menus -> Component references menu

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

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

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

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

Moving component values



Press **Move component values** button



Other menus -> Component values menu

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

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

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



Edit component value functions

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

Special objects



Press **Draw/change objects other layers** button



Press **o**



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 executed:

- Add special objects
- Change special objects
- Select only objects on a layer
- Select only line/rectangle/circle/arc/text objects
- Copy objects from one layer to another
- Move objects from one layer to another
- Mirror text



Add special objects



Change special objects

Add objects on board outline

SPECIALS MENU



Add special objects



Activate special objects menu

Change objects on board outline

SPECIALS MENU



Change special objects



Activate special objects menu

Add special objects

SPECIALS MENU



Activate special objects menu

Objects can be added on the following layers:

- Top layer (Copper objects)
- Bottom layer (Copper objects)
- Board outline layer
- Info layer
- Info layer 2
- Solder mask top layer
- Solder mask bottom layer
- Paste mask top layer
- Paste mask bottom layer
- Drills (plated) layer
- Drills (unplated) layer

The following objects can be added/changed :

Lines

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

Rectangles

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

Circles

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

Arcs

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

Texts

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

Change special objects

SPECIALS MENU



Select objects



Activate special objects menu

The following actions can be done on **selected** objects :

Move



Move



Press **m**

Move **selected** objects.

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

Move to another layer



Move objects to -> Select layer

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

Copy



Copy



Press **C**

Copy **selected** objects.

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

Copy to another layer



Copy objects to -> **Select layer**

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

Rotate



Rotate



Press **R**

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

Delete



Delete button



Delete



Press **Del**

Delete **selected** objects.

Change line thickness



Change line thickness

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

Change text



Change text



Press **e**

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

Change text height



Change text height

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

Gate/pin swap



Press **Gate/pin swap** button



Other menus -> Gate/pin swap menu

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

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

When the mouse cursor is placed on a pin, clicking on the **left mouse button** will display the gate/pin swap information for that pin. Swappable pins of gates will be highlighted, and also a number is visible in the center of the pad. Swappable pins are highlighted in a different color.

After selecting a pin, move the mouse cursor to the swappable pin or swappable gate pin, and by clicking on the **left mouse button** the gates/pins will be swapped.

All the gate/pin swap changes will be recorded in the file **pcb\gatepin.ban**. After all gate/pin swaps are done, the schematics should be updated with the **Back annotation** function of the design manager.



Back annotation



Edit gate/pin swap in symbols

Schematic link



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

When placing components there is a special feature available. This feature is a link with the schematic editor. When this function is activated and the schematic editor is opened with a schematic file from the current design, selections made in the schematic editor will be reflected in the layout editor. When activating this function, all unhighlighted connections will be made invisible.

During the time this function is active the schematic editor(s) will be the master(s). This means selecting/deselecting in 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's pin will be made visible.

