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

■ Laplot a function Rep Met My LA Ad (20) An in the matching of the second in	
Button bar	-
Absolute position (mm)	
Absolute grid position (mils/mm)	o
Relative (grid)position (mils/mm)	
lasi 4 222,4800 vv lagessi 2 100,100 v silasi 2 100,100 v silasi XI	 기

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 (inktjet- 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





Press Print button

Sub menu File menu item Print screen

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 pole 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 $(0,0 = \text{left top})$
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)

BackGroundColor 24 bit RGB color (Stored as 32 bit) ViewLayer1Color 24 bit RGB color (Stored as 32 bit) ViewLayer2Color 24 bit RGB color (Stored as 32 bit) 24 bit RGB color (Stored as 32 bit) ViewLayer3Color 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) 24 bit RGB color (Stored as 32 bit) CompValueColor 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) 24 bit RGB color (Stored as 32 bit) ShapePinsBottomColor 24 bit RGB color (Stored as 32 bit) ViaPinsColor 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



Press u



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
Zoom out
Window based Zooming
Hide/view layers
Hide/view layers
Pan window
Pan window
Window based panning
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 fille	ed with the background color.
Corbor fill		The enceful will be a simulated work on a constant

The areafill will be a simulated gerber/penplot output. Gerber fill .

Polygon surround : Areafill displayed with polygon surrounding

: Areafill displayed with a horizontal line Horizontal fill

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 ⇐,⇒,¶,IJ
»	Press x
>>	Press Shift and move the mouse the window border
Window	Use the scrollbars

When pressing the \mathbf{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, than 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 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 drawed 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 mutiple 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



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
 Activate components
 Select component menu
 Activate component menu
 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 textheight component reference/value
- Change text line width component reference/value

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

>>

Component values

- Edit Hide
- Visible
- On top layer
- On bottom layer
- Height

Activate component menu
 Select components

Protect components

>>

Protect

The **Selected** components will be protected. Protected components can **not** be selected. To unprotect components use <u>Component protection</u>.

Component info

»	
»	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

>>

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 one a trace/via/pin/connection and press the **left mouse button** to activate the trace drawing. For every trace segment to be placed press the **left mouse button**. To end with trace drawing, place the mouse cursor in the neigbourhood 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 to 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

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 appriopiate, 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 appriopiate, 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

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.

Diselect 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 DelDelete

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 <u>Hide/view layers</u> 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 powernets (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
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

Remove powerplane

AREAFILL MENU

Remove powerplane -> Select layer

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

>>

>>

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

Change powerplane

AREAFILL MENU

>>

Change powerplane -> Select layer

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

Activate areafills/powerplanes menu
Image: 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 desciption 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

>>

>>

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.

Change hatch areafill

>>

>>

Change hatch areafill -> select hatch

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

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.

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 Dal

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

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 textheight or pen thickness of the text use the <u>Change component parameters</u> 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 textheight or pen thickness of the text use the <u>Change component parameters</u> functions.

Special objects

07	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 :

<u>Lines</u>

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, an 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).

<u>Circles</u>

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).

<u>Arcs</u>

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).

<u>Texts</u>

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

Change special objects

SPECIALS MENU

Select objecs
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.

<u>Copy</u>

» Сору

» 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.

<u>Rotate</u>

» Rotate

» Press R

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

<u>Delete</u>

>>

Delete button

» Delete

» Press Del

Delete **selected** objects.

Change line thickness

Change line thickness

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

<u>Change text</u>

» 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 swappeble 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 de layout editor has no effect on the schematic editor(s) selections.

The following objects selected in the schematic editor will be reflected in the layout editor :

<u>Bus</u>

All the connections of the bus netnames will made visible

<u>Wire</u>

The connections of the wires netname will made visible

Components

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