



Pcbnew

Reference manual

Copyright

This document is Copyright © 2010–2011 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (<http://www.gnu.org/licenses/gpl.html>), version 3 or later, or the Creative Commons Attribution License (<http://creativecommons.org/licenses/by/3.0/>), version 3.0 or later.

All trademarks within this guide belong to their legitimate owners.

Contributors

Jean-Pierre Charras, Fabrizio Tappero.

Feedback

Please direct any comments or suggestions about this document to the kicad mailing list:
<https://launchpad.net/~kicad-developers>

Acknowledgments

None

Publication date and software version

Published on October 19, 2011. Based on LibreOffice 3.3.2.

Note for Mac users

The kicad support for the Apple OS X operating system is experimental.

Table of Contents

1 - Introduction to Pcbnew.....	7
Description.....	7
Principal design features.....	8
General remarks.....	8
2 - Installation.....	9
Installation of the software.....	9
Modifying the default configuration.....	9
3 - General operations.....	1
Toolbars and commands.....	1
Mouse commands.....	2
Basic commands.....	2
Operations on blocks.....	3
Selection of grid size.....	3
Adjustment of the zoom level.....	3
Displaying cursor coordinates.....	4
Keyboard commands - hot keys.....	4
Operation on blocks.....	4
Units used in dialogs.....	5
Top menu bar.....	5
The File menu.....	6
The Preferences menu.....	6
Other preferences and dimensions.....	6
The Design Rules menu.....	7
The Display 3D model menu.....	7
The Help menu.....	8
Using icons on the top toolbar	8
Right hand-side toolbar.....	10
Left hand-side toolbar	11
Pop-up windows and fast editing.....	12
Available modes.....	12
Normal mode.....	12
Footprint mode.....	13
Tracks mode.....	14
4 - Schematic Implementation.....	1
Linking a schematic to a printed circuit board.....	1
Procedure for creating a printed circuit board.....	3
Procedure for updating a printed circuit board.....	3
Reading netlist file - loading footprints.....	3
Dialog box.....	3
Available options.....	4
Loading new footprints.....	4
5 - Setting up working layers.....	1
Select copper layers.....	1
Introduction.....	1
Select number of layers.....	1

Copper layers.....	3
Auxiliary technical layers.....	3
Paired layers.....	3
Layers for general use.....	3
Special layer.....	3
Selection of the active Layer.....	3
Selection using the layer manager.....	4
Selection using the upper toolbar.....	4
Selection using the pop-up window.....	5
Selection of the Layers for Vias:.....	5
Using the high-contrast mode.....	6
Copper layers in high contrast mode.....	6
Technical layers.....	7
6 - Create and modify a board.....	1
Creating a board.....	1
Drawing the board outline.....	1
Reading the netlist generated from the schematic.....	2
Correcting a board.....	3
Steps to follow.....	3
Deleting incorrect tracks.....	4
Deleted components.....	4
Modified modules.....	5
Advanced options - selection using time stamps.....	5
Direct exchange for footprints already placed on board.....	5
7 - Modules placement.....	1
Assisted placement.....	1
Manual placement.....	1
General re-orientation of modules.....	3
Automatic Module Distribution.....	3
Automatic placement of modules.....	4
Characteristics of the automatic placer.....	4
Preparation.....	5
Interactive auto-placement.....	5
Additional note.....	5
8 - Setting routing parameters.....	1
Current settings.....	1
Accessing the main dialog.....	1
Current settings.....	1
General options.....	2
Netclasses.....	2
Setting routing parameters.....	3
Netclass editor.....	3
Global Design Rules.....	4
Via parameters.....	6
Track parameters.....	6
Specific sizes.....	6
Examples and typical dimensions.....	6
Track width.....	6
Insulation (clearance).....	6
Examples.....	7
Manual routing.....	8

Help when creating tracks.....	8
Creating tracks.....	8
Moving and dragging tracks.....	9
Via Insertion.....	9
Select/edit the track width and via size.....	10
Using the horizontal toolbar.....	10
Using the pop-up menu.....	11
Editing and changing tracks.....	11
Change a track.....	11
Global changes.....	12
9 - Creating copper zones.....	1
Creating zones on copper layers.....	1
Creating a zone.....	2
Creating the limits of a zone.....	2
Filling the zone.....	3
Filling options.....	5
Filling mode.....	5
Clearance and minimum copper thickness.....	5
Pad options.....	6
Thermal reliefs parameters.....	7
Parameters choice.....	7
Adding a cutout area inside a zone.....	7
Outlines editing.....	8
Editing zone: parameters.....	10
Final zone filling.....	10
Change zones net names.....	10
Creating zones on technical layers.....	11
Creating zone limits.....	11
10 - Files for circuit fabrication.....	1
Final preparations.....	1
Final DRC test.....	2
Setting coordinates origin.....	3
Generating files for photo-tracing.....	3
GERBER format.....	5
HPGL format.....	6
POSTSCRIPT format.....	6
Plot options.....	6
Global clearance settings for the solder stop and the solder paste mask.....	7
Solder mask clearance.....	7
Solder paste clearance.....	7
Generating drill files.....	8
Generating cabling documentation.....	9
Generation of files for automatic component insertion.....	9
Advanced tracing options.....	9
11 - ModEdit - Managing Libraries.....	1
Overview of ModEdit.....	1
ModEdit.....	2
ModEdit user interface.....	2
Top toolbar in Modedit.....	3

Creating a new module.....	4
Creating a new library.....	5
Saving a module in the active library.....	5
Transferring a module from one library to another.....	5
Saving all modules of your board in the active library.....	5
Documentation for library modules.....	5
Documenting libraries – recommended practice.....	6
12 - ModEdit - Creating and Editing Modules.....	8
ModEdit overview.....	8
Module elements.....	9
Pads.....	9
Contours.....	9
Fields.....	9
Starting ModEdit and selecting a module to edit.....	9
Module Editor Toolbars.....	9
Edit toolbar.....	10
Display toolbar.....	11
Context Menus.....	11
Module properties dialog.....	13
Creating a new module.....	13
Adding and editing pads.....	14
Adding pads.....	14
Setting pad properties.....	14
Rectangular pads	15
Rotate pads.....	15
Not plated through hole pads.....	15
Pads not on copper layers.....	15
Offset Parameter.....	15
Delta Parameter (trapezoidal pads).....	16
Setting clearance for solder mask and solder paste mask layers.....	16
Remarks.....	16
Solder paste mask parameters.....	16
Footprint level settings.....	17
Fields Properties.....	17
Automatic placement of a module.....	18
Attributes.....	18
Documenting modules in a library.....	18
3-dimensional visualisation.....	19
Saving a module into the active library.....	20
Saving a module to the board.....	21

1 - Introduction to Pcbnew

Table of Contents

<i>1 - Introduction to Pcbnew.....</i>	<i>1</i>
<i>Description.....</i>	<i>1</i>
<i>Principal design features.....</i>	<i>1</i>
<i>General remarks.....</i>	<i>2</i>

Description

Pcbnew is a powerful printed circuit board software tool available for the Linux, Microsoft Windows and Apple OS X operating systems.

Pcbnew is used in association with the schematic capture software program Eeschema, which provides the Netlist file - this describes the electrical connections of the PCB to design.

A second program CvPcb is used to assign each component in the Netlist produced by Eeschema, to a module that is used by Pcbnew. This can be done either interactively or automatically using equivalence files.

Pcbnew manages libraries of modules. Each module is a drawing of the physical component including its footprint - the layout of pads providing connections to the component. The required modules are automatically loaded during the reading of the Netlist produced by CvPcb.

Pcbnew integrates, automatically and immediately, any circuit modification, by removal of any erroneous tracks, addition of the new components, or by modifying any value (and under certain conditions any reference) of the old or new modules, according to the electrical connections appearing in the scheme.

Pcbnew provides a rats nest display, a hairline connecting the pads of modules which are connected on the schematic. These connections move dynamically as track and module movements are made.

Pcbnew has an active Design Rules Check (DRC) which automatically indicates any error of track layout in real time.

Pcbnew can automatically generate a copper plane, with or without thermal breaks on the pads.

Pcbnew has a simple but effective autorouter to assist in the production of the circuit. An Export/Import in SPECCTRA dsn format allows to use more advanced auto-routers.

Pcbnew provides options specifically for the production of ultra high frequency circuits (such as pads of trapezoidal and complex form, automatic layout of coils on the printed circuit...).

Pcbnew displays the elements (tracks, pads, texts, drawings and more) as actual size and according to personal preferences:

- display in full or outline.
- display of the track/pad clearance.

Principal design features

Pcbnew has an internal resolution of 1/10000 inch.

Pcbnew works on 16 layers of copper, plus 12 technical layers (silk screen, solder mask, component adhesive, solder paste, drawings and comments...) and manages in real time the hairline indication (rats nest) of missing tracks.

The display of the PCB elements (tracks, pads, text, drawings...) can be customizable:

- In full or outline.
- With or without track clearance.
- by hiding certain elements (copper layers, technical layers, zones of copper, modules...), which is useful for high density multi-layer circuits.

For the complex circuits, the display of layers, zones, components can be removed in a selective way for a better legibility of the screen.

Modules can be rotated to any angle, with a step of 0,1 degree.

Pads can be round, rectangular, oval or trapezoidal (the latter is necessary for the production of ultra high frequency circuits). In addition several basic pads can be grouped.

Both the size of each pad, and the layers where they appear, can be adjusted.

The drilling of holes can be offset.

Pcbnew can automatically generate copper planes, with automatic generation of thermal breaks around the pads concerned.

The Module Editor can be accessed from the Pcbnew toolbar. The Editor allows creation or modification of a module from the PCB or a library and then saved to either. A module saved to the PCB can be subsequently saved to a library. In addition all modules on the PCB can be saved to a library by creating a footprint archive.

Pcbnew generates in an extremely simple way all the documents necessary:

- Fabrication outputs:
 - Files for Photoplotters in GERBER RS274X format.
 - Files for drilling in EXCELLON format.
- Plot files in HPGL, SVG and DXF format.
- Plot and drilling maps in POSTSCRIPT format.
- Local Printout.

General remarks

Pcbnew requires a 3 button mouse. The 3rd button is mandatory.

Finally it should be noted that the diagrammatic tool Eeschema and CvPcb are needed to create the required netlists.

2 - Installation

Table of Contents

<i>2 - Installation.....</i>	<i>1</i>
<i> Installation of the software.....</i>	<i>1</i>
<i> Modifying the default configuration.....</i>	<i>1</i>

Installation of the software

The installation procedure is described in the kicad documentation.

Modifying the default configuration

A default configuration file kicad.pro is provided in kicad/share/template. This file is used as the initial configuration for all new projects.

This configuration file can be modified to change libraries to be loaded

To do this:

- Launch pcbnew using kicad or directly. On Windows is in c:\kicad\bin\pcbnew.exe and on Linux you can run /usr/local/kicad/bin/kicad or /usr/local/kicad/bin/pcbnew if the binaries are located in /usr/local/kicad/bin.
- Select Preferences - Libs and Dir.
- Edit as required.
- Save the modified configuration (Save Cfg) to kicad/share/template/kicad.pro.

3 - General operations

Table of Contents

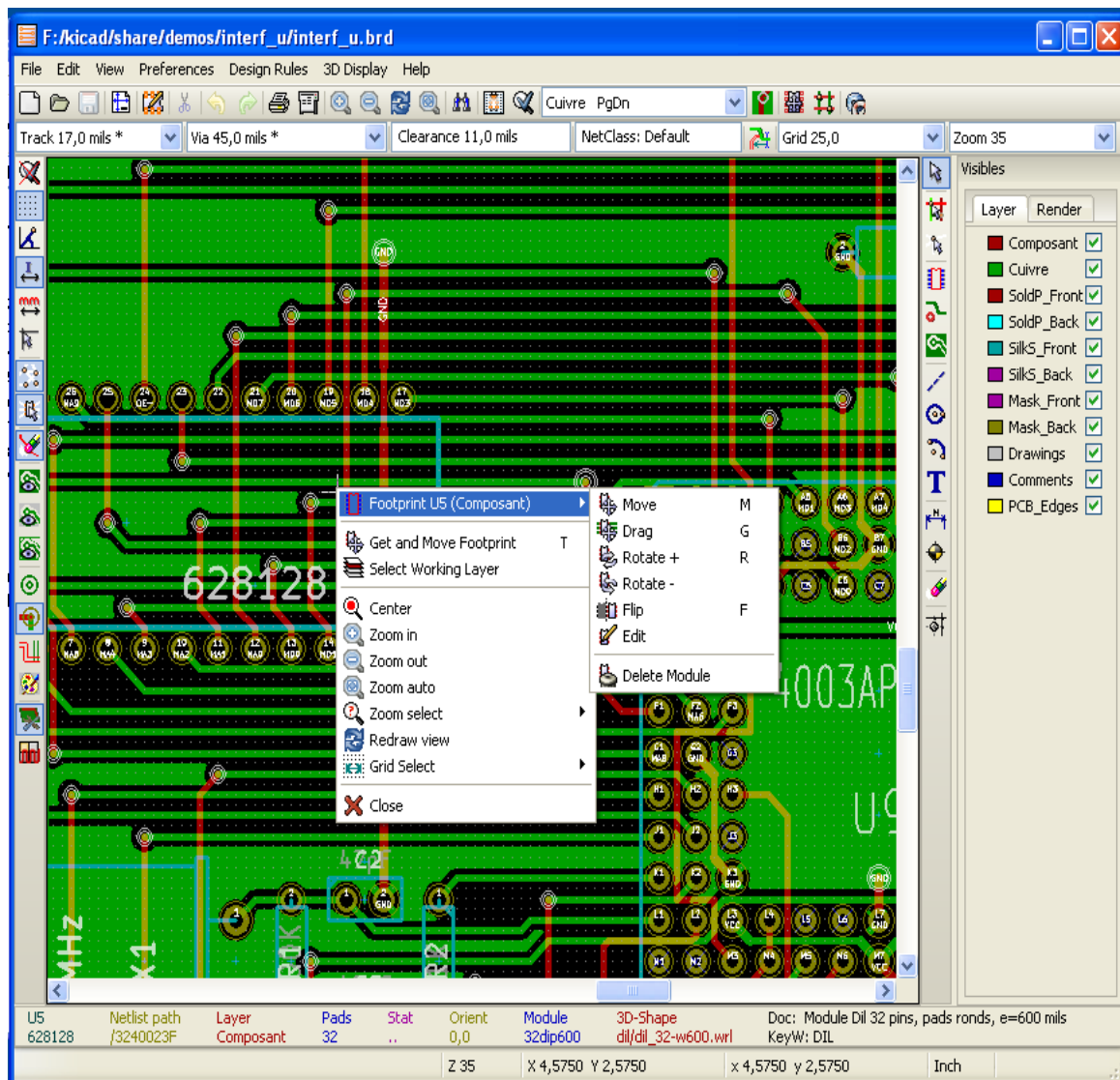
3 - General operations.....	1
Toolbars and commands.....	1
Mouse commands.....	2
Basic commands.....	2
Operations on blocks.....	2
Selection of grid size.....	3
Adjustment of the zoom level.....	3
Displaying cursor coordinates.....	3
Keyboard commands - hot keys.....	4
Operation on blocks.....	4
Units used in dialogs.....	5
Top menu bar.....	5
The File menu.....	6
The Preferences menu.....	6
Other preferences and dimensions.....	6
The Design Rules menu.....	7
The Display 3D model menu.....	7
The Help menu.....	8
Using icons on the top toolbar	8
Right hand-side toolbar.....	9
Left hand-side toolbar	11
Pop-up windows and fast editing.....	12
Available modes.....	12
Normal mode.....	12
Footprint mode.....	13
Tracks mode.....	14

Toolbars and commands

In Pcbnew it is possible to execute command via various means:

- text-based menu at the top of the main window.
- top toolbar menu.
- right toolbar menu.
- left toolbar menu.
- mouse buttons (menu options). Specifically:
 - The right-hand mouse button reveals a Pop up menu the content of which depends on the element under the mouse arrow.
- keyboard (Function keys F1, F2, F3, F4, Shift, Delete, +, - Page Up, Page Down and "space" bar). The "Escape" key generally cancels an operation in progress.

The screen-shot below illustrates some of the possible accesses to the operations:



Mouse commands

Basic commands

- Left button
 - Single click displays the characteristics of the module or text under the cursor to the lower status bar.
 - Double click displays the editor (if the element is editable) of the element under the cursor.
- Centre button/wheel
 - Rapid zoom and some commands in layer manager. A 2 button mouse is undesirable. Hold down the centre button and draw a rectangle to zoom to the described area. The rotation of the mouse wheel will allow you to zoom in and zoom out.
- Right button
 - Displays a pop-up menu

Operations on blocks

Operations to move, invert (mirror), copy, rotate and delete a block are all available via the pop-up menu. In addition the view can zoom to the area described by the block.

The framework of the block is traced by moving the mouse whilst holding down the left mouse button. The operation is executed when the button is released.

By holding down one of the hotkeys “Shift” or “Ctrl”, or both keys “Shift and Ctrl” together, whilst the block is drawn the operation invert, rotate or delete is automatically selected as shown in the table below:

Summarised orders:

Left mouse button held down	Trace framework to move block
Shift + Left mouse button held down	Trace framework for invert block
Ctrl + Left mouse button held down	Trace framework for rotating block 90°
Shift+Ctrl + Left mouse button held down	Trace framework to delete the block
Centre mouse button held down	Trace framework to zoom to block

When moving a block:

- Move block to new position and operate left mouse button to place the elements.
- To cancel the operation use the right mouse button and select Cancel Block from the menu (or press the Esc key).

Alternatively if no key is pressed when drawing the block use the right mouse button to display the pop-up menu and select the required operation.

For each block operation a selection window enables the action to be limited to only some elements.

Selection of grid size

During elements layout the cursor moves on a grid. The grid can be turned on or off using the icon on the left toolbar.

Any of the pre-defined grid sizes, or a User Defined grid, can be chosen using the pop-up window, or the drop-down selector on the toolbar at the top of the screen. The size of the User Defined grid is set using the menu bar option Dimensions - User Grid Size.

Adjustment of the zoom level

To change the zoom level:

- Open the pop-up window (using the right mouse button) and to select the desired zoom.
- Or use the function keys:
 - F1**: Enlarge (zoom in)
 - F2**: Reduce (zoom out)
 - F3**: Redraw the display
 - F4**: Centre view at the current cursor position
- Or rotate the mouse wheel.
- Or hold down the middle mouse button and draw a rectangle to zoom to the described area.

• Displaying cursor coordinates

The cursor coordinates are displayed in inches or millimetres as selected using the 'In' or 'mm' icons on the left hand side toolbar.

Whichever unit is selected Pcbnew always works to a precision of 1/10,000 of inch.

The status bar at the bottom of the screen gives:

- The current zoom setting.
- The absolute position of the cursor.
- The relative position of the cursor. Note the relative coordinates (x,y) can be set to (0,0) at any position by pressing the space bar. The cursor position is then displayed relative to this new datum.

In addition the relative position of the cursor can be displayed using its polar co-ordinates (ray + angle). This can be turned on and off using the icon on the left hand side toolbar.



Keyboard commands - hot keys

Many commands are accessible directly with the keyboard. Selection can be either upper or lower case. Most hot keys are shown in menus. Some hot keys that do not appears are:

- Key Delete (or Del): deletes a module or a track. Available only if the Module tool or the track tool is active.
- Key V: if the track tool is active switches working layer or place via, if a track is in progress.
- Key + and -: active layer = next or previous layer.
- Key ?: display the list off all hot keys.
- Key space: reset relative coordinates.

Operation on blocks

Operations to move, invert (mirror), copy, rotate and delete a block are all available from the pop-up menu. In addition the view can zoom to that described by the block.

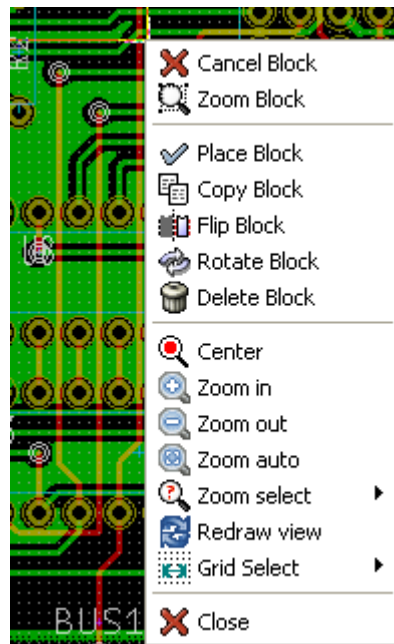
The framework of the block is traced by moving the mouse whilst holding down the left mouse button. The operation is carried out on releasing the button.

By holding down one of the keys “Shift” or “Ctrl” or both “Shflt and Ctrl” together or “Alt”, whilst the block is drawn the operation invert, rotate,delete or copy is automatically selected as shown in the table below:

Left mouse button held down	move block
Shift + Left mouse button held down	invert (mirror) block
Ctrl + Left mouse button held down	rotate block 90°
Shft+Ctrl + Left mouse button held down	delete the block
Alt + Left mouse button held down	copy the block

If move block is initially selected (no keys held down), one of the alternative options can be chosen by displaying the pop-up menu using the right mouse button.

Any of the commands above can be canceled via the same pop-up menu or pressing the Escape key (Esc).



Units used in dialogs

Units used to display dimensions values are inch and mm. The desired unit can be selected by



pressing the icon located in left toolbar. However one can enter the unit used to define a value, when entering a new value.

Accepted units are:

1in	1 inch
1"	1 inch
25th	25 thou
25mi	25 mils, same as thou
6mm	6 mm

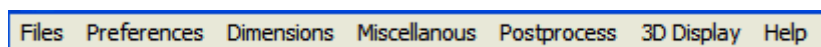
The rules are:

- Spaces between the number and the unit are accepted.
- Only the first two letters are significant.

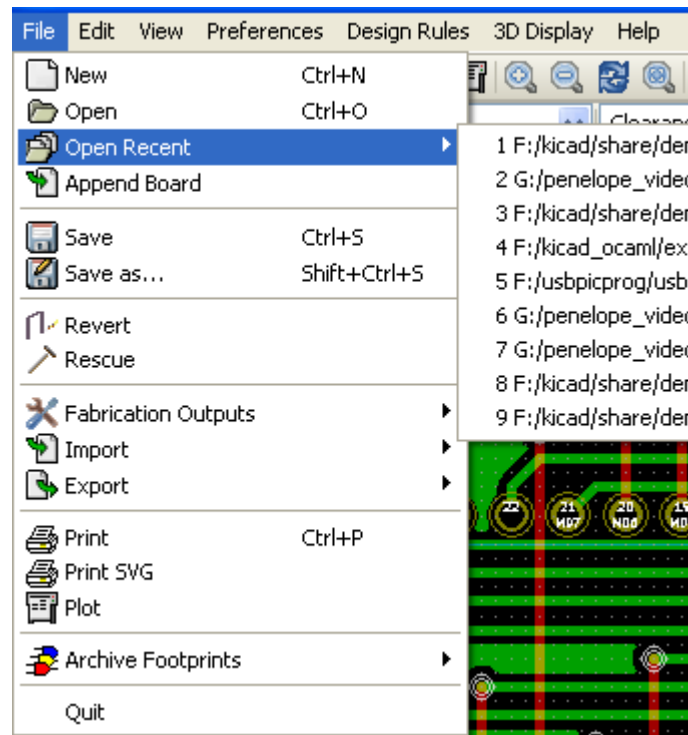
In countries using an alternative decimal separator than the period, the period (.) can be used as well. Therefore 1,5 and 1.5 are the same in French.

Top menu bar

The top menu bar provides access to the files (loading and saving), configuration options, printing, plotting and the help files.

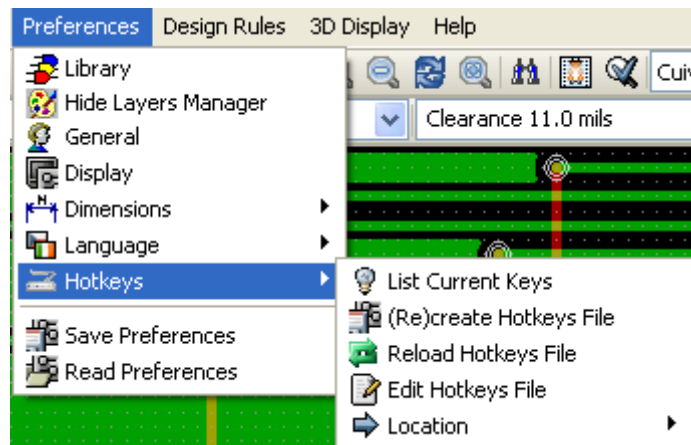


The File menu



The File menu allows the loading and saving of printed circuits files, as well as printing and plotting the circuit board. It enables the export (with the format GenCAD 1.4) of the circuit for use with automatic testers.

The Preferences menu

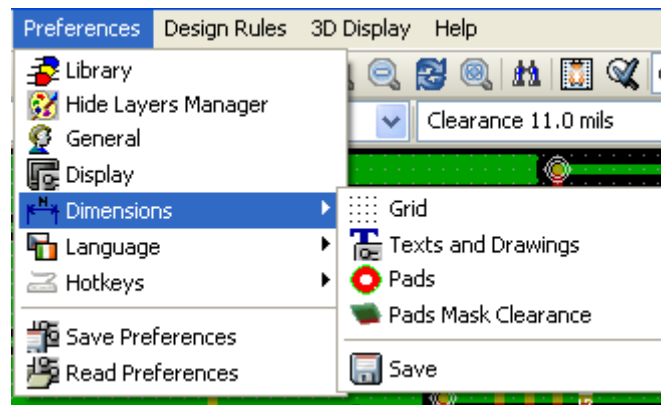


Allows:

- Selection of the module libraries.
- Hide/Show the Layers manager(colors selection for displaying layers and other elements. Also enables the display of elements to be turned on and off.)
- Management of general options (units, etc.).
- The management of other display options.
- Creation, edition (and reread) of the hot keys file.

Other preferences and dimensions

An important sub menu.



Allows adjustment of:

- User grid size.
- Size of texts and the line width for drawings.
- Dimensions and characteristic of pads.
- Setting the global values for solder mask and solder paste layers

The Design Rules menu

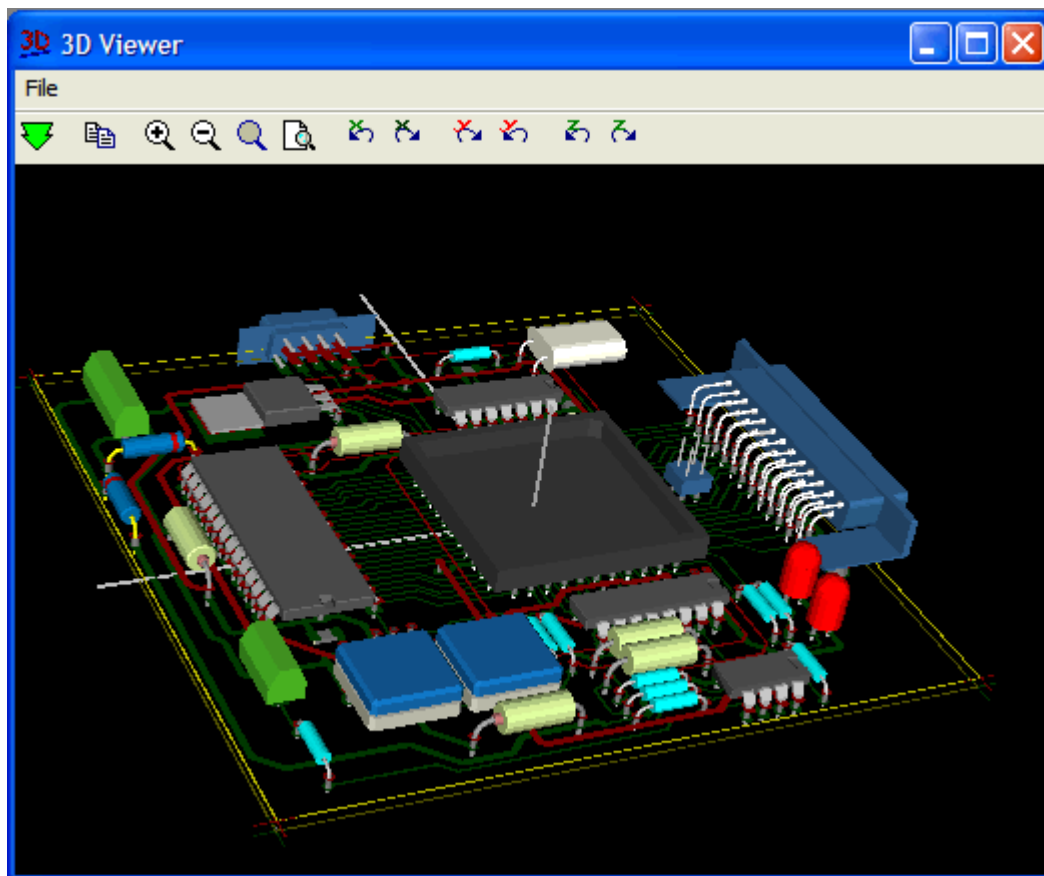


Provides access to 2 dialogs:

- Setting the Design rules (tracks and vias sizes, clearances).
- Setting layers (Number, enabled and layers names)

The Display 3D model menu

Brings up the 3D viewer used to display the circuit board in 3 dimensions.

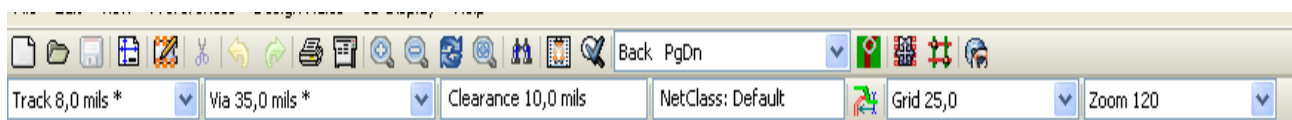











The Help menu









Provides access to the user manuals and to the version information menu (Pcbnew About).

Using icons on the top toolbar

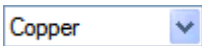

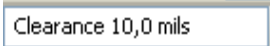
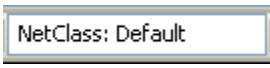


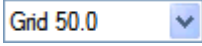
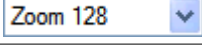
This toolbar gives access to the principal functions of Pcbnew.




	Creation of a new printed circuit.
	Opening of an old printed circuit.
	Save printed circuit.
	Selection of the page size and modification of the file properties.
	Opens module editor (<u>M</u> odedit) to display/edit library or pcb modules.
	Delete the elements selected during a move block operation.
	Undo Redo last commands (10 levels)
	Display print menu.
	Display plot menu.














	Zoom in and Zoom out (relative to the centre of screen).
	Redraw the screen and Auto Zoom.
	Find module or text.
	Netlist options (selection, reading, testing and compiling).
	DRC (Design Rule Check): Automatic check of the tracks.
	<i>Footprint mode</i> : when active this enables module options in the pop-up window.
	<i>Routing mode</i> : when active this enables routing options in the pop-up window
	Direct access to the web router FreeRoute



Auxiliary toolbar:

	Selection of the work layer.
	Selection a thickness of track already in use.
	Current track and vias clearance (That is set in the Design rules dialog)
	Display the current selected netclass (That is set in the Design rules dialog) A netclass defines the current tracks and vias parameters.
	Automatic track width: if enabled when creating a new track, when starting on an existing track, the width of the new track is set to the width of the existing track.
	Selection of a dimension of via already in use.
	Selection of the grid size.
	Selection of the zoom.


Right hand-side toolbar









	<p>This toolbar gives access to the tools for:</p> <ul style="list-style-type: none"> • Placement of modules, tracks, zones of copper, texts, etc. • Net Highlighting. • Creating notes, graphic elements, etc. • Deleting elements.
---	--










	Select the standard mouse mode.
	Highlight net selected by clicking on a track or pad.
	Display local ratsnest (Pad or Module).
	Add a module from a library.
	Placement of tracks and vias.
	Placement of zones (copper planes).
	Draw Lines on technical layers (i.e. not a copper layer).
	Draw Circles on technical layers (i.e. not a copper layer).
	Draw Arcs on technical layers (i.e. not a copper layer).
	Placement of text.
	Draw Dimensions on technical layers (i.e. not the copper layer).
	Draw Alignment Marks (appearing on all layers).
	<p>Delete element pointed to by the cursor (see note below)</p> <p>Note:</p> <p>when Deleting if several superimposed elements are pointed to priority is given to the smallest (in the decreasing set of priorities tracks, text, module).</p> <p>the function “Undelete” of the upper toolbar allows the cancellation of the last item deleted.</p>

	Offset adjust for drilling and place files.
	Grid origin. (grid offset). Useful mainly for edition and placement of footprints. Can also be set in Dimensions/Grid menu.

Left hand-side toolbar

	<p>This left hand-side toolbar provides display and control options</p>
--	---

	Turns DRC (Design Rule Checking) on/off. Caution: when DRC is off incorrect connections can be made.
	Turn grid display on/off (Note: a small grid may not be displayable).
	Polar display of the relative co-ordinates on the status bar on/off.
	Display/entry of co-ordinates in inches or millimeters.
	Change cursor display.
	Display general rats nest (incomplete connections between modules).
	Display module rats nest dynamically as it is moved.
	Enable/Disable automatic deletion of a track when it is redrawn.

	<p>Display mode for copper zones.</p> <p> = Shows all (outlines + filled areas)</p> <p> = Shows outlines only (no filled areas)</p> <p> = Shows all outlines (zone outlines + filled areas outlines) Filling itself is not shown</p>
	Display of pads in sketch mode on/off.
	Display of tracks and vias in sketch mode on/off.
	High contrast display mode on/off. In this mode the active layer is displayed normally, all the other layers are displayed in gray. Useful for working on multi-layer circuits.
	Hide/Show the Layers manager
	Access to microwaves tools. Under development

Pop-up windows and fast editing

A right click of the mouse open a pop-up window. Its contents depends on the element pointed at by the cursor.





This gives immediate access to:

- Changing the display (centre display on cursor, zoom in or out or selecting the zoom).
- Setting the grid size.
- Additionally a right click on an element enables editing of the most usually modified element parameters.

The screenshot below shows what the pop-up window looks like.

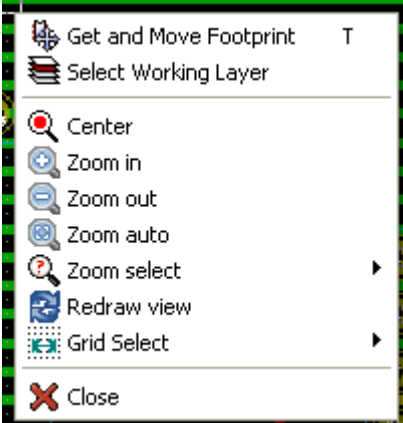
Available modes

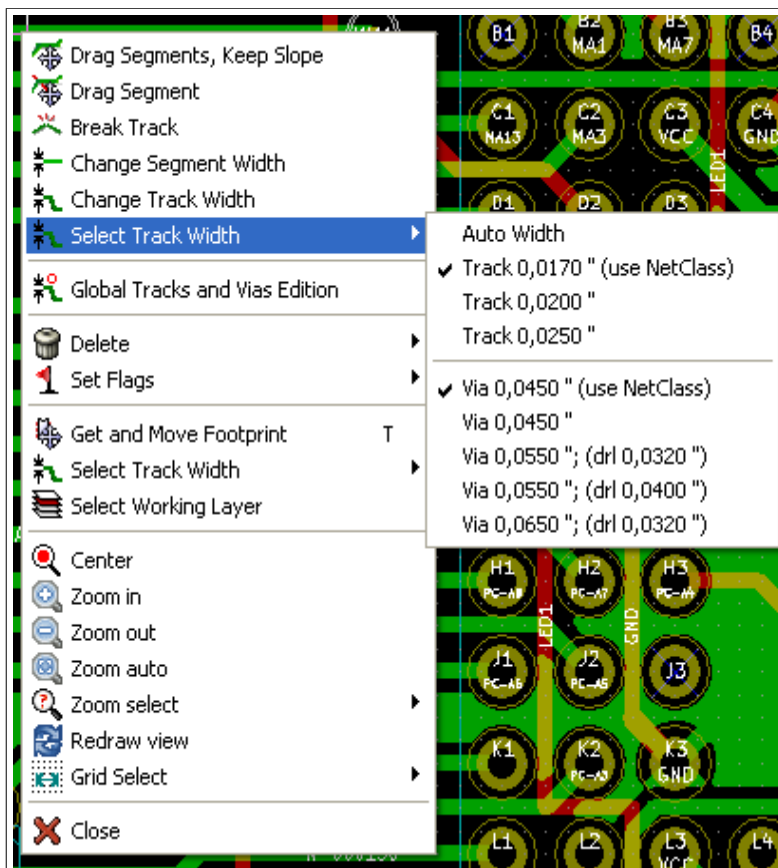
There are 3 modes when using pop up menus.

	and  disabled	normal mode
	enabled	footprint mode
	enabled	tracks mode

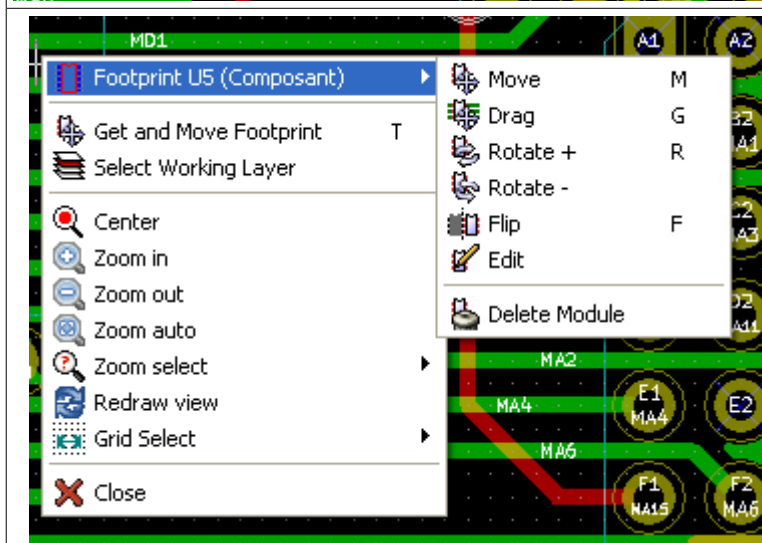
In the pop-up menus, these modes add or remove some specific commands.

Normal mode

	Pop up with no selected item.
---	-------------------------------



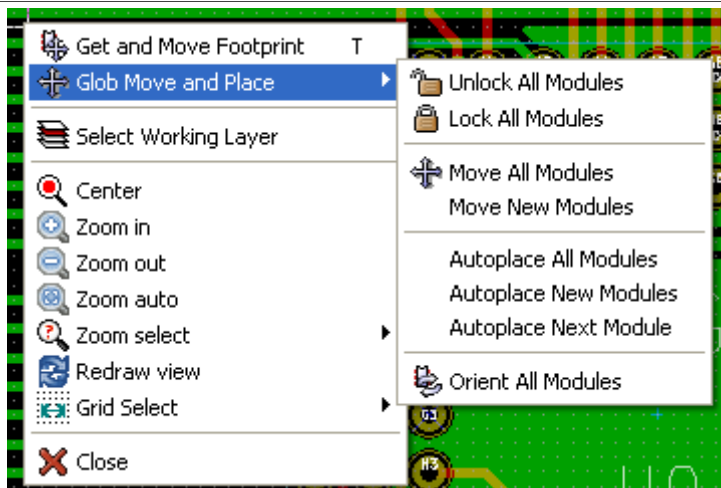
Pop Up on a track.



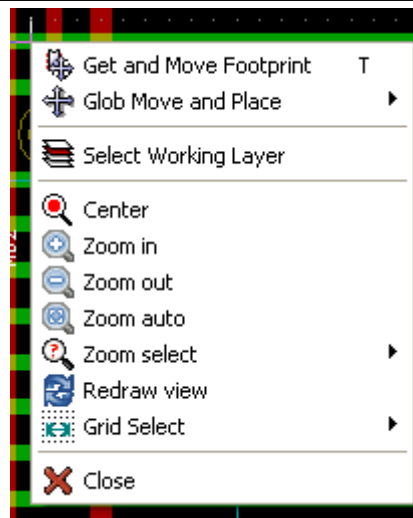
Pop up on a footprint.

Footprint mode

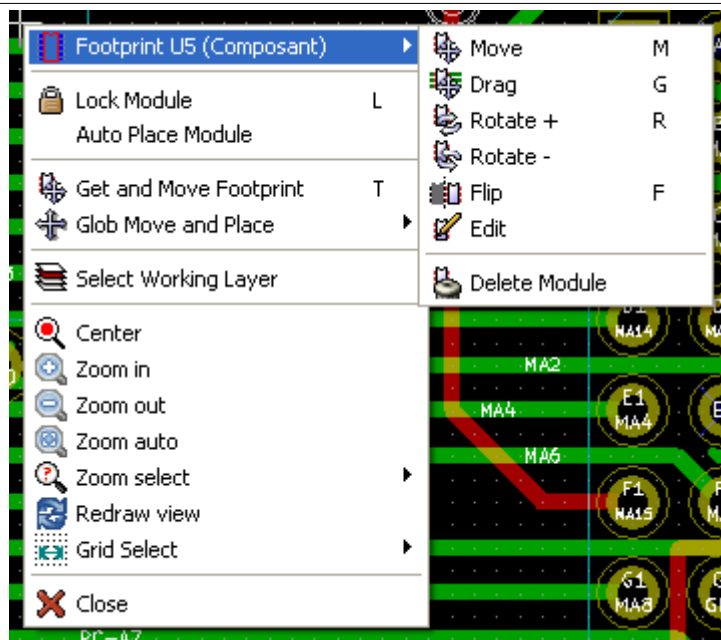
Same cases in Footprint mode ( on)



Pop up with no selected item.



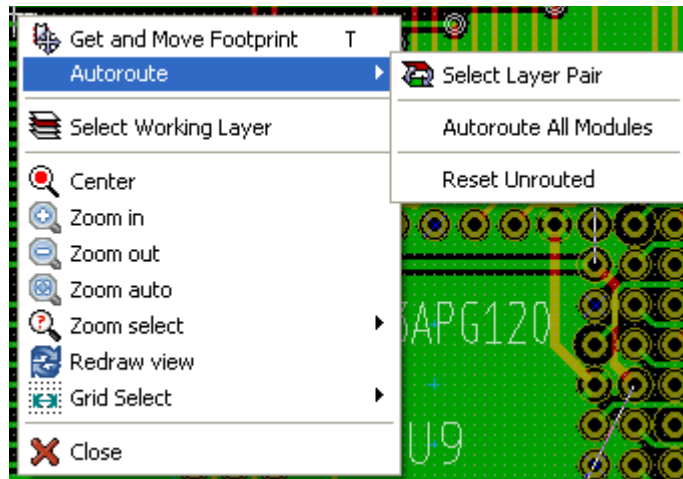
Pop up on a track.



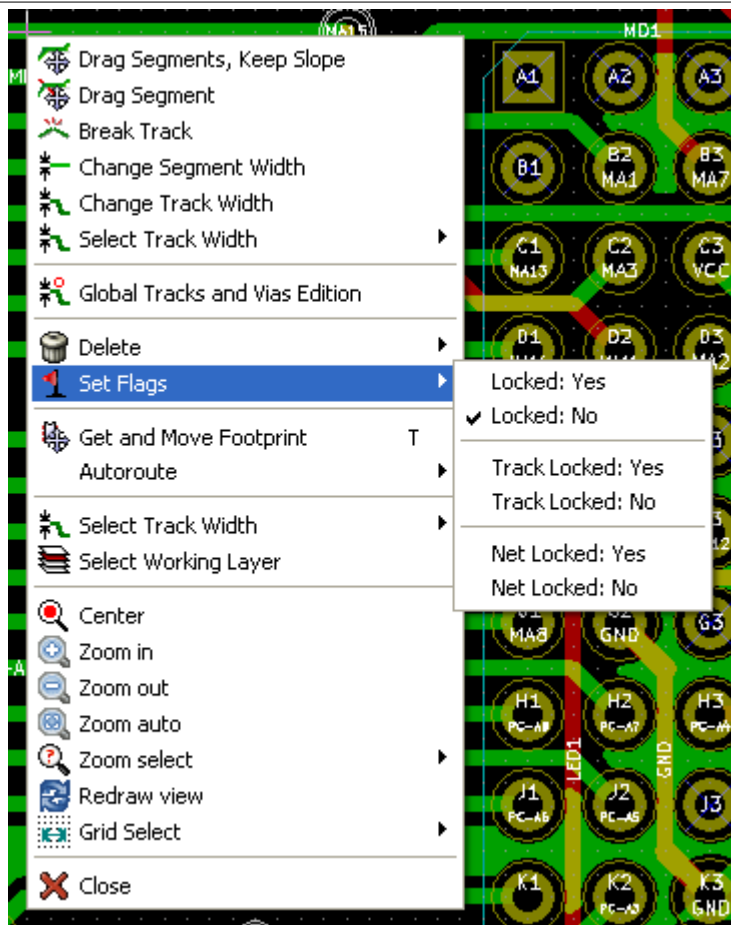
Pop up on a footprint.

Tracks mode

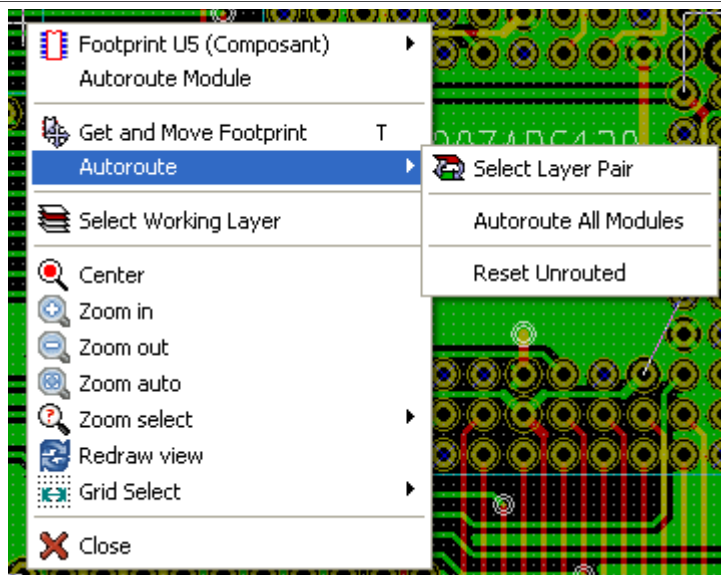
Same cases in Tracks mode ( on)



Pop up with no selected item.



Pop up on a track.



Pop up on a footprint.

4 - Schematic Implementation

Table of Contents

<i>4 - Schematic Implementation.....</i>	<i>1</i>
<i> Linking a schematic to a printed circuit board.....</i>	<i>1</i>
<i> Procedure for creating a printed circuit board.....</i>	<i>3</i>
<i> Procedure for updating a printed circuit board.....</i>	<i>3</i>
<i> Reading netlist file - loading footprints.....</i>	<i>3</i>
<i> Dialog box.....</i>	<i>3</i>
<i> Available options.....</i>	<i>4</i>
<i> Loading new footprints.....</i>	<i>4</i>

Linking a schematic to a printed circuit board

Generally speaking, a schematic sheet is linked to its printed circuit board by means of the netlist file, which is normally generated by the schematic editor used to make the schematic. Pcbnew accepts netlist files made with Eeschema or Orcad PCB 2.

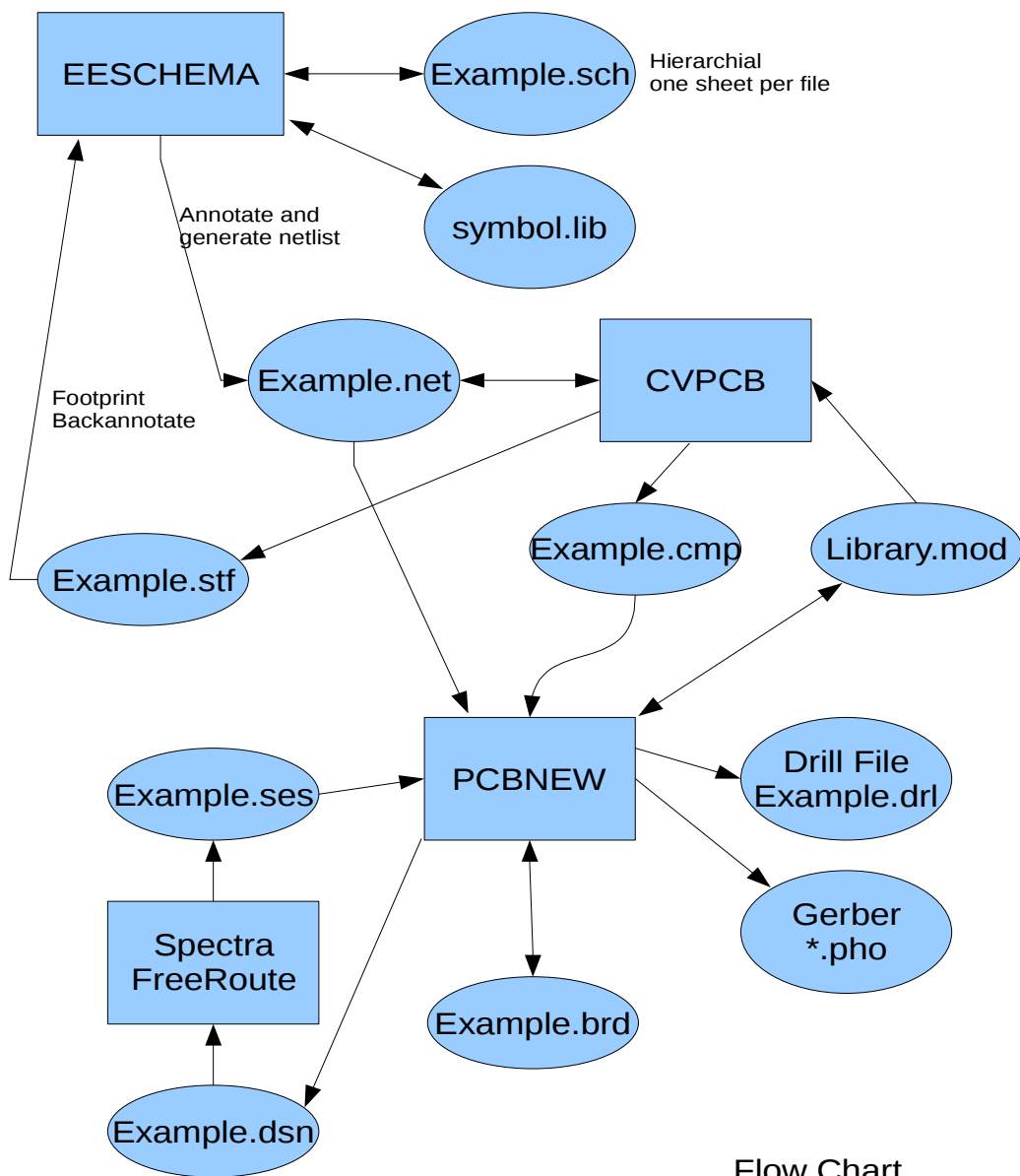
The netlist file, generated from the schematic is usually missing the footprint modules that correspond to the various components. Consequently an intermediate stage is necessary. During this intermediate process the association components/modules is performed. In KiCad, CvPcb is used to create this association and a file named *.cmp is produced. CvPcb also updates the netlist file using this information.

Cvppcb can also output a "stuff file" *.stf which can be back annotated into the schematic file as the F2 field for each component, saving the task of re-assigning module footprints in each schematic edit pass. In Eeschema copying a component will also copy the footprint assignment and set the reference designator as unassigned for later auto-incremental-annotation.

Pcbnew reads the modified netlist file .net and, if it exists, the .cmp file

In the event of a module being changed directly in Pcbnew the .cmp file is automatically updated avoiding in this way the requirement to run CvPcb again.

The following figure illustrate the whole work-flow of Kicad and how intermediate files are obtained and used by the different software tools that made Kicad.



Flow Chart

Procedure for creating a printed circuit board

After having created your schematic in Eeschema:

- Generate the netlist using Eeschema.
- Assign each component in your netlist file to the corresponding module (often called footprint) used on the printed circuit using Cvpcb.
- Launch Pcbnew and read the modified Netlist, this will also read the file with the module selections.

Pcbnew will then load automatically all the necessary modules. Modules can now be placed manually or automatically on the board and tracks can be routed.

Procedure for updating a printed circuit board

If the schematic is modified (after a printed circuit board has been generated), the following steps must be repeated:

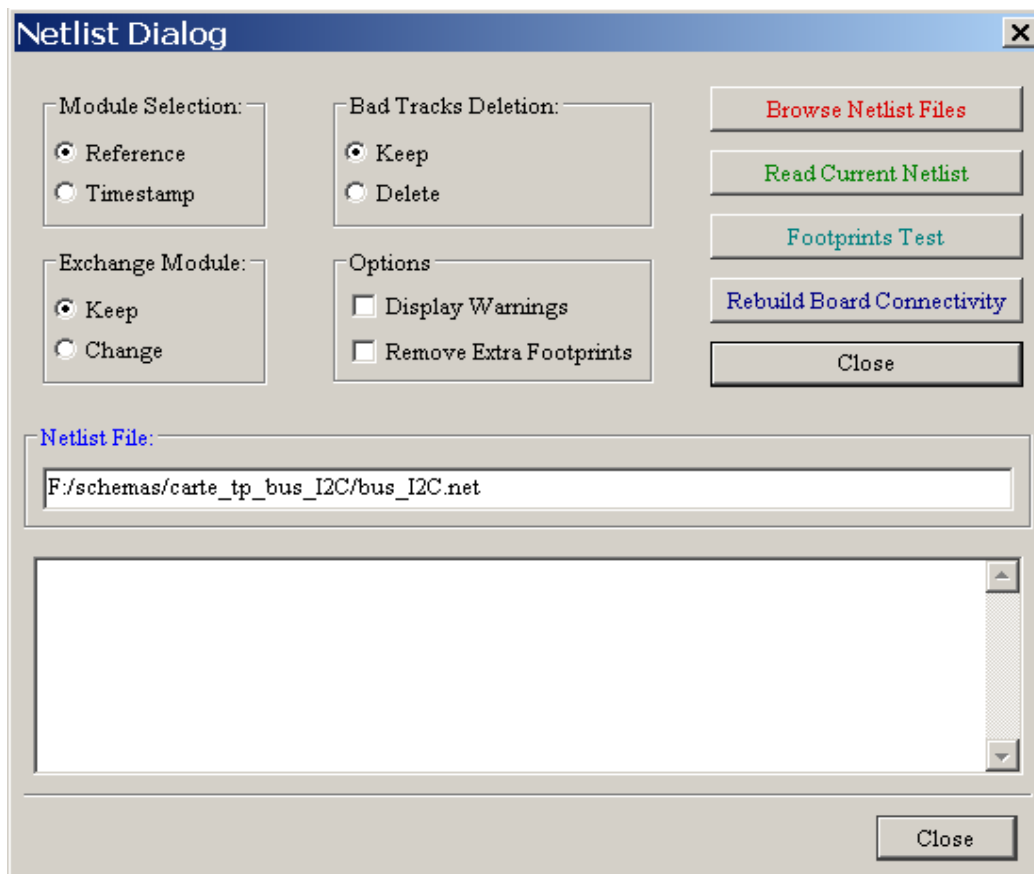
- Generate a new netlist file using Eeschema.
- If the changes to the schematic involve new components, the corresponding modules must be assigned using Cvpcb.
- Launch Pcbnew and re-read the modified Netlist (this will also re-read the file with the module selections).

Pcbnew will then load automatically any new module, add the new connections and remove redundant connections. This process is called forward annotation and is a very common procedure when a PCB is made and updated.

Reading netlist file - loading footprints

Dialog box

Accessible from the icon 

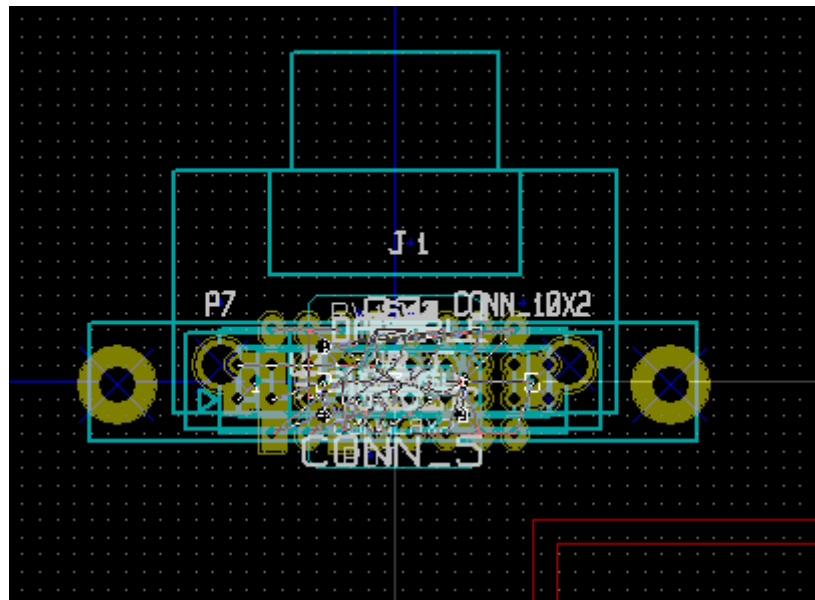


Available options

Exchange Module:	If a footprint has changed in the netlist: keep old footprint or change to the new one.
Bad Tracks Deletion	Keep all existing tracks, or delete erroneous tracks
Options : (on/off)	Display or hide all messages. Remove footprints which are on board but not in the netlist. Footprint with attribute "Locked" will not be removed.

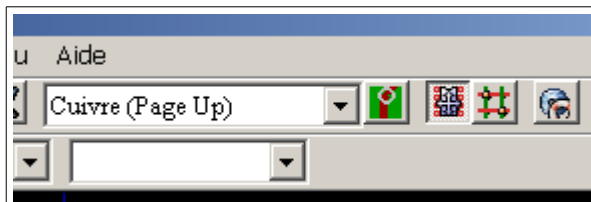
Loading new footprints

When new footprints are found in the netlist file, they will be automatically loaded and placed at coordinate (0,0).



New footprints can be moved and arranged one by one. A better way is to automatically move (unstack) them:

- Activate footprint mode

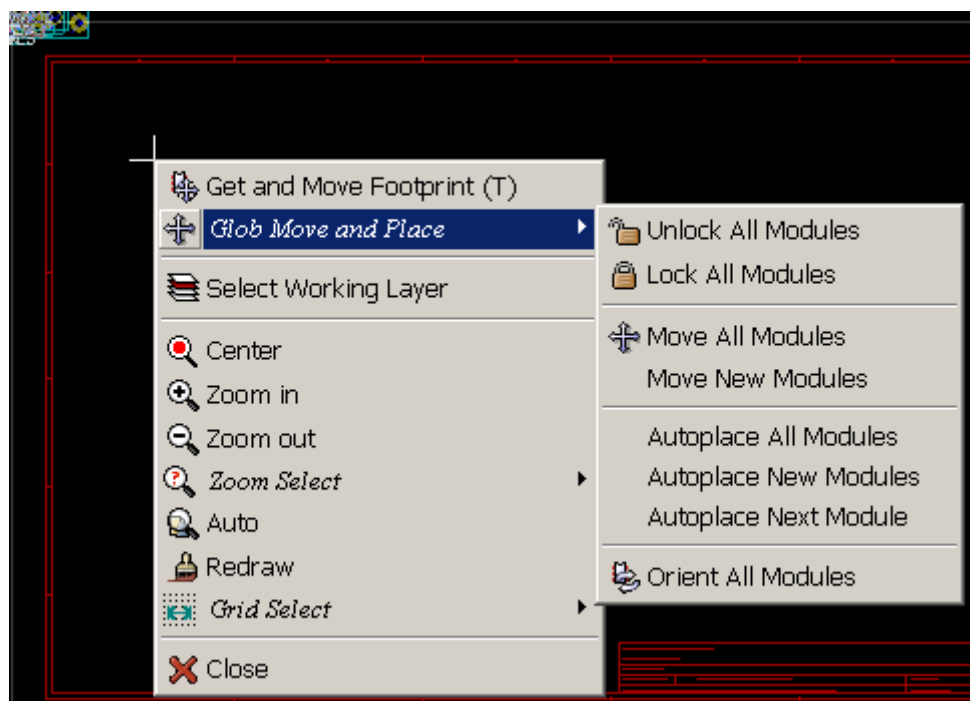


activate the "footprint mode".



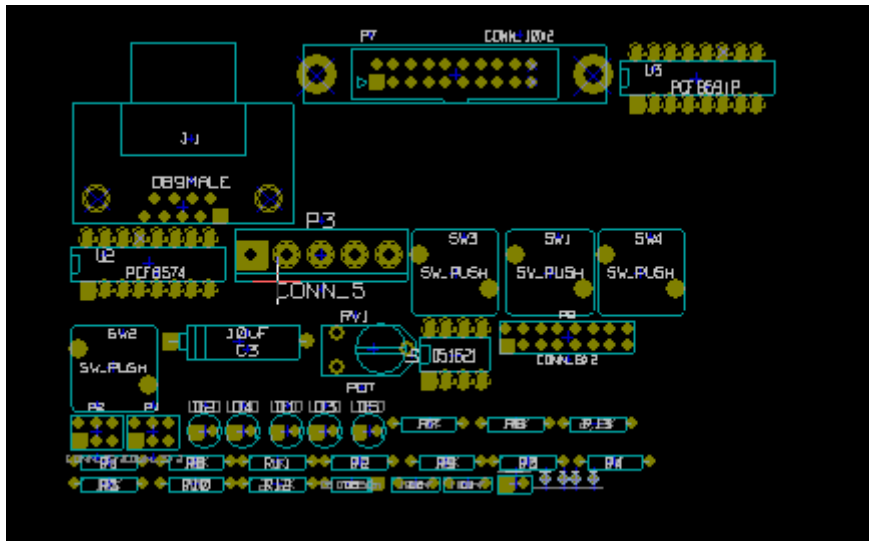
footprint mode activated.

Move the mouse cursor to a suitable (free of component) area, and click on the right button



- Move New Modules if there is already a board with existing footprints.
- Move All Modules, the first time (when creating a board).

The following screenshot shows the results.



5 - Setting up working layers

Table of Contents

5 - Setting up working layers.....	1
Select copper layers.....	1
Introduction.....	1
Select number of layers.....	1
Copper layers.....	2
Auxiliary technical layers.....	3
Paired layers.....	3
Layers for general use.....	3
Special layer.....	3
Selection of the active Layer.....	3
Selection using the layer manager.....	4
Selection using the upper toolbar.....	4
Selection using the pop-up window.....	5
Selection of the Layers for Vias:.....	5
Using the high-contrast mode.....	6
Copper layers in high contrast mode.....	6
Technical layers.....	7

Pcbnew can work on 28 different layers:

- 16 layers of copper (or of routing of tracks)
- 12 auxiliary technical layers.

The number of copper layer, and, if needed, their names and attributes should be set.

Unused technical layers can be disabled.

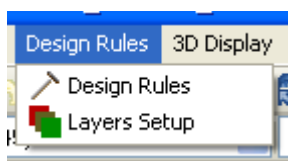
Select copper layers

Introduction

Copper layers are the usual working layers used by the automatic router to place and re-arrange tracks. Layer 1 is the copper (solder) layer. Layer 16 is the component layer. Other layers are the inner layers, from L2 to L15.

Select number of layers

To enable navigation between layers, it is necessary to select the number of working layers. To do this you can use the menu bar and select Preferences - Layers Setup.



Then select the number of layers wanted, from 2 to 16.

Layer Setup ✕

Preset Layer Groupings: Custom

Copper Layers: 2

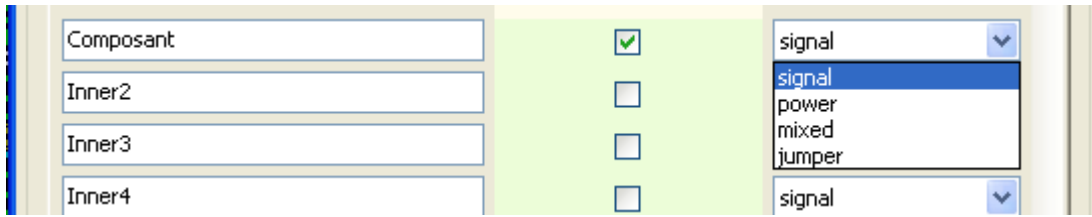
Layers

Name	Enabled	Type
Adhes_Front	<input type="checkbox"/>	Off-board, manufacturing
SoldP_Front	<input checked="" type="checkbox"/>	Off-board, manufacturing
SilkS_Front	<input checked="" type="checkbox"/>	On-board, non-copper
Mask_Front	<input checked="" type="checkbox"/>	On-board, non-copper
Composant	<input checked="" type="checkbox"/>	signal
Inner2	<input type="checkbox"/>	signal
Inner3	<input type="checkbox"/>	signal
Inner4	<input type="checkbox"/>	signal
Inner5	<input type="checkbox"/>	signal
Inner6	<input type="checkbox"/>	signal
Inner7	<input type="checkbox"/>	signal
Inner8	<input type="checkbox"/>	signal
Inner9	<input type="checkbox"/>	signal
Inner10	<input type="checkbox"/>	signal
Inner11	<input type="checkbox"/>	signal
Inner12	<input type="checkbox"/>	signal
Inner13	<input type="checkbox"/>	signal
Inner14	<input type="checkbox"/>	signal
Inner15	<input type="checkbox"/>	signal
Cuivre	<input checked="" type="checkbox"/>	signal
Mask_Back	<input checked="" type="checkbox"/>	On-board, non-copper
SilkS_Back	<input checked="" type="checkbox"/>	On-board, non-copper
SoldP_Back	<input checked="" type="checkbox"/>	Off-board, manufacturing
Adhes_Back	<input type="checkbox"/>	Off-board, manufacturing
PCB_Edges	<input checked="" type="checkbox"/>	Board contour
Eco2	<input type="checkbox"/>	Auxiliary
Eco1	<input type="checkbox"/>	Auxiliary
Comments	<input checked="" type="checkbox"/>	Auxiliary
Drawings	<input checked="" type="checkbox"/>	Auxiliary

OK Cancel

Copper layers

The name of any copper layer is editable. Copper layers have attributes useful when using the external router *FreeRouter*.



Auxiliary technical layers

Some are associated in pairs, others not. When they appear as a pair this affects the behaviour of modules. The elements making up a module (pads, drawing and text) appearing on a layer (solder or component), appear on the other complementary layer when the module is inverted (mirrored).

The technical layers are:

Paired layers

- **The Adhesives layers (Copper and Component):**
These are used in the application of adhesive to stick SMD components to the circuit board, generally before wave soldering.
- **The Solder Paste layers paste SMD (Copper and Component):**
Used to produce a masks to allow solder paste to be placed on the pads of surface mount components, generally before reflow soldering. In theory only surface mount pads occupy these layers.
- **The Silk Screen layers (Copper and Component):**
They are the layers where the drawings of the components appear.
- **The Solder Mask layers (Copper and Component):**
These define the solder masks. Normally all the pads appear on one or the other of these layers (or both for through pads) to prevent the varnish covering the pads.

Layers for general use

- Comments
- E.C.O. 1
- E.C.O. 2
- Drawings

These layers are for any use. They can be used for text such as instructions for assembly or wiring, or construction drawings, to be used to create a file for assembly or machining.

Special layer

Layer Edges PCB:

this layer is reserved for the drawing of circuit board outline. Any element (graphic, texts...) placed on this layer appears on all the other layers.

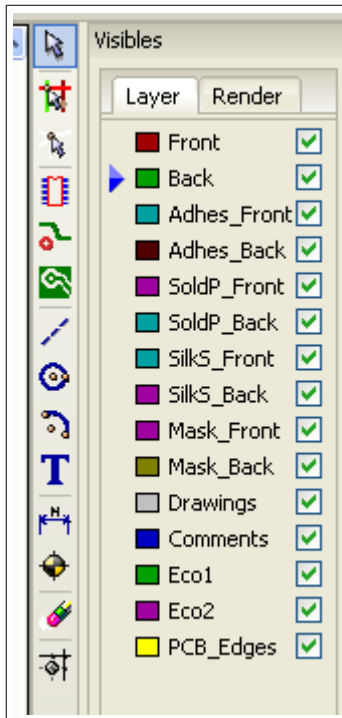
Use this layer only to draw board outlines.

Selection of the active Layer

The selection of the active working layer can be done in several ways:

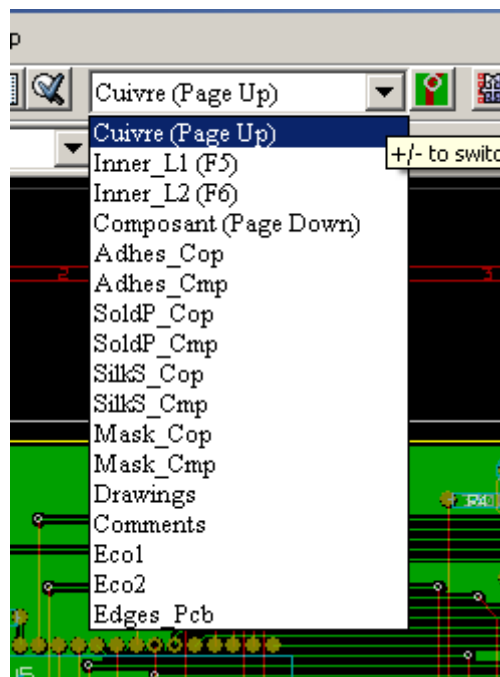
- Using the right toolbar (Layer manager).
- Using the upper toolbar.
- With the Pop-Up window (activated with the right mouse button).
- Using the + and – keys (works on copper layers only).
- By hot keys.

Selection using the layer manager



The layer manager also allows you to change color layers and their visibility

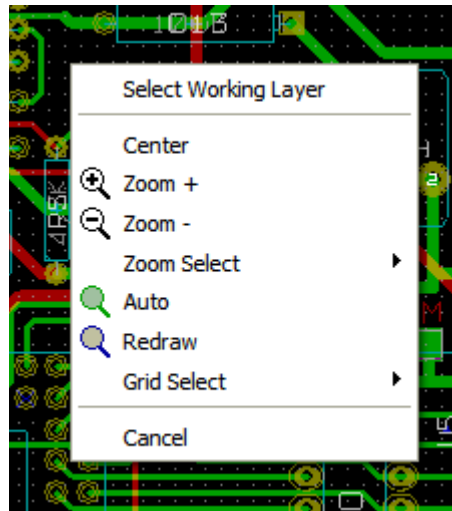
Selection using the upper toolbar



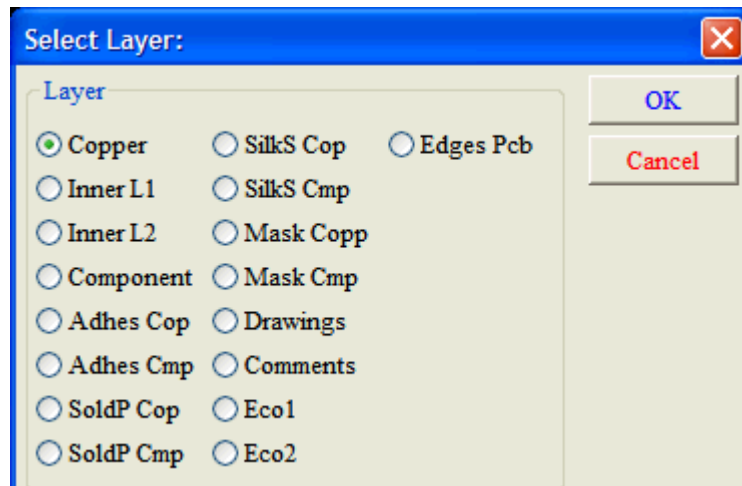
This directly selects the working layer.

Hot keys to select the working layer are displayed.

Selection using the pop-up window

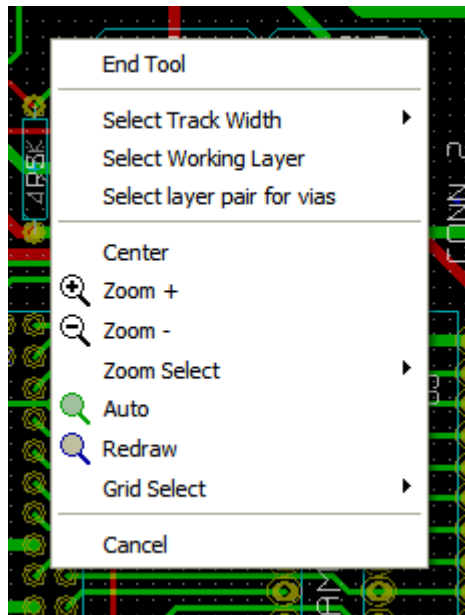


The Pop-up window opens a menu window which provides a choice for the working layer

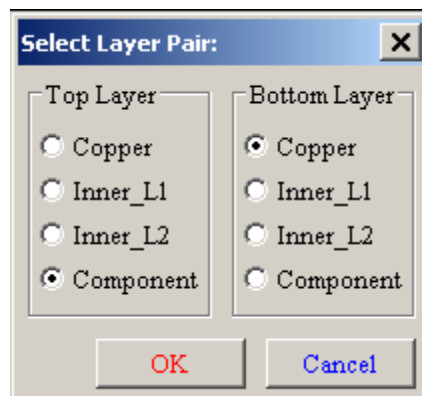


Selection of the Layers for Vias:

If the **Add Tracks and Vias** icon is selected on the right hand toolbar, the Pop-Up window provides the option to change the layer pair used for vias:




This selection opens a menu window - which provides choice of the layers used for vias.



When a via is placed the working (active) layer is automatically switched to the alternate layer of the layer pair used for the vias.

One can also switch to an other active layer by hot keys, and if a track is in progress, a via will be inserted.

Using the high-contrast mode

This mode is entered when the tool  (left toolbar) is activated.

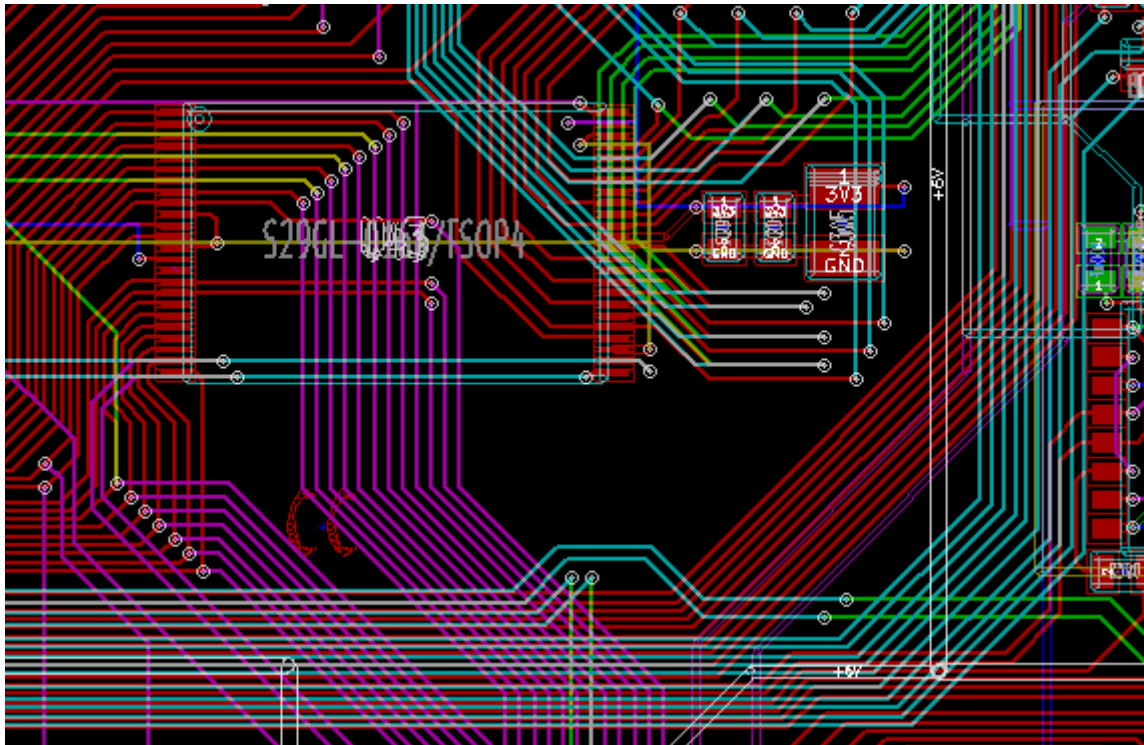
When using this mode, the active layer is displayed like in the normal mode, but all others layers are displayed in gray color.

There is two useful cases:

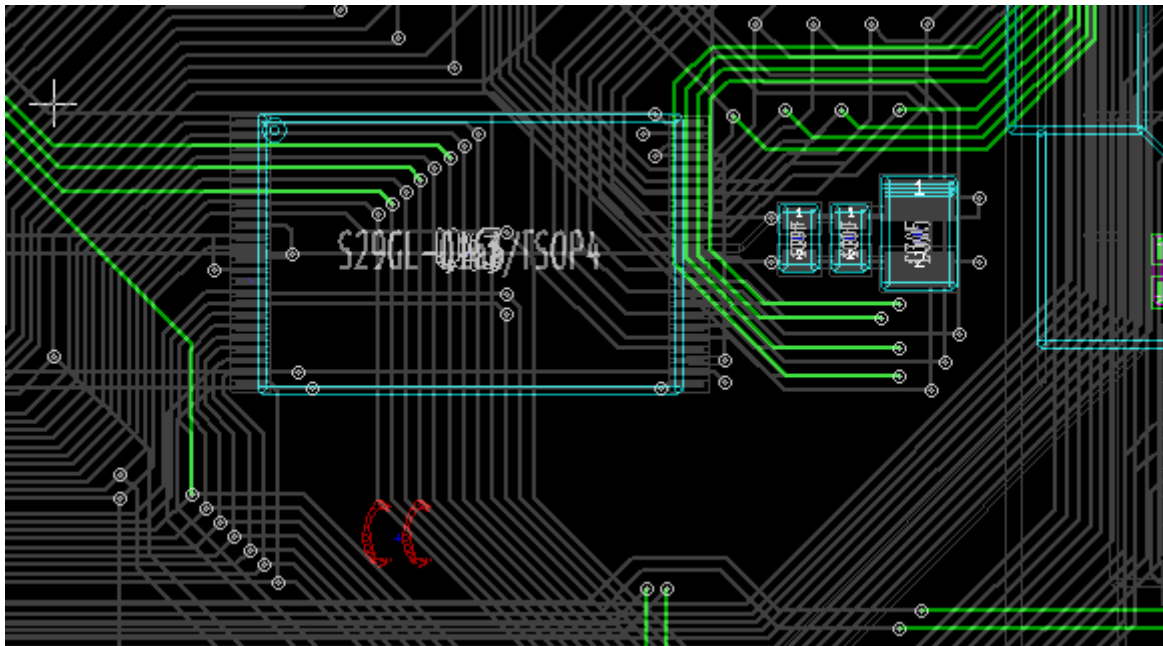
Copper layers in high contrast mode

When a board uses more than four layers, this option allows the design to be seen more easily the active copper layer:

- **Normal mode normal (back side copper layer active)**



- **High Contrast mode (back side copper layer active):**



Technical layers

The other case is when it is necessary to examine solder paste layers and solder mask layers, that are usually not displayed.

Masks on pads are displayed if this mode is active:

- **Normal mode (front side solder mask layer active):**

6 - Create and modify a board

Table of Contents

6 - Create and modify a board.....	1
Creating a board.....	1
Drawing the board outline.....	1
Reading the netlist generated from the schematic.....	2
Correcting a board.....	3
Steps to follow.....	3
Deleting incorrect tracks.....	4
Deleted components.....	4
Modified modules.....	4
Advanced options - selection using time stamps.....	5
Direct exchange for footprints already placed on board.....	5

Creating a board

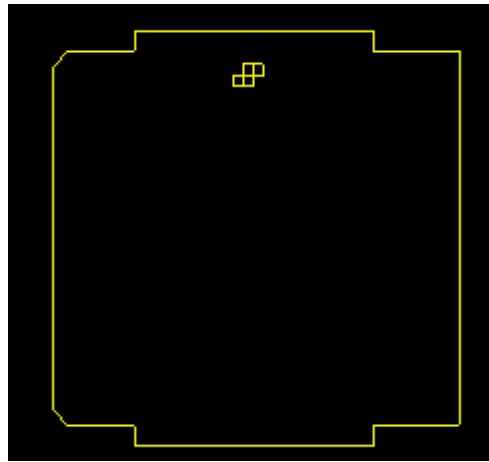
Drawing the board outline

It is usually a good idea to define the outline of the board first. The outline is drawn as a sequence of line segments. Select 'Edges pcb' as the active layer and use the 'Add graphic line or polygon' tool to trace the edge, clicking at the position of each vertex and double-clicking to finish the outline. Boards usually have very precise dimensions, so it may be necessary to use the displayed cursor coordinates while tracing the outline. Remember that the relative coordinates can be zeroed at any time using the space bar, and that the display units can also be toggled using 'Alt-U'. Relative coordinates enable very precise dimensions to be drawn. It is possible to draw a circular (or arc) outline:

- Select the 'Add graphic circle' or 'Add graphic arc' tool
- Click to fix the circle centre
- Adjust the radius by moving the mouse
- Finish by clicking again.

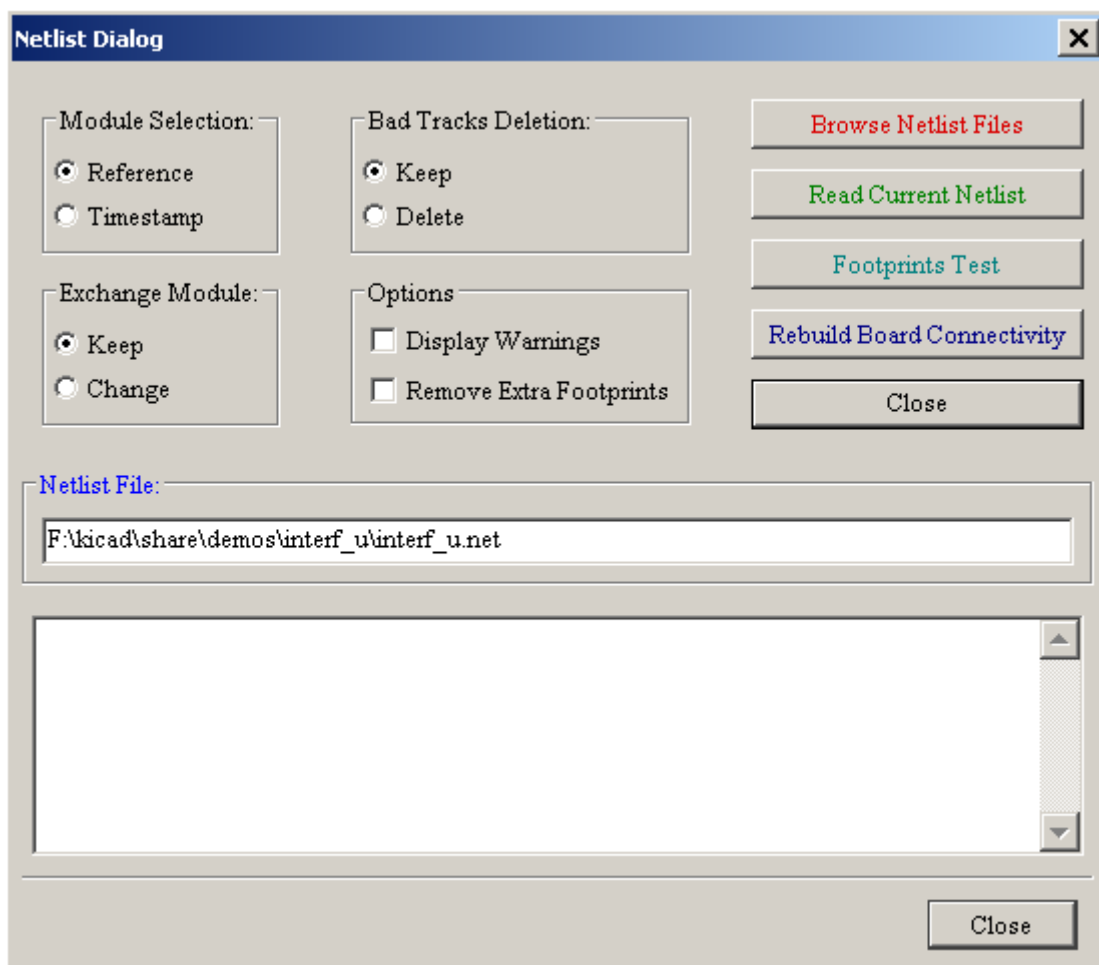
Note that the width of the outline can be adjusted, in the Parameters menu (recommended width = 150 in 1/10 mils) or via the Options, but this will not be visible unless the graphics are displayed in other than outline mode.

The resulting outline might look something like this:

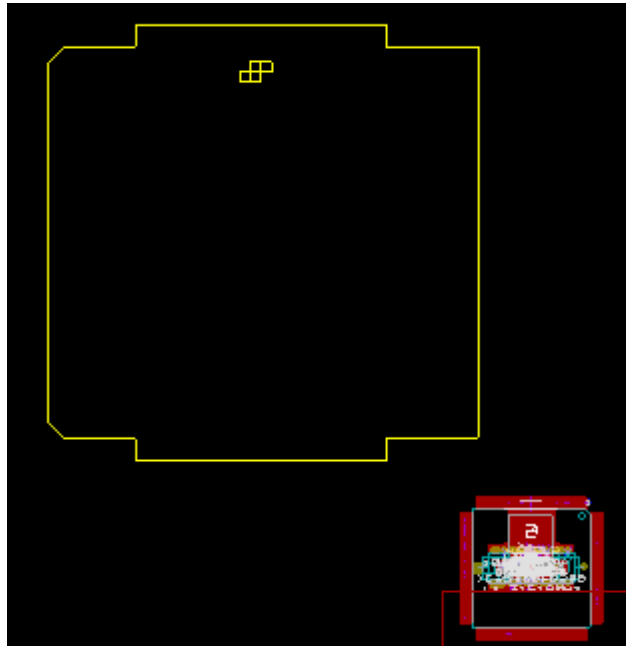


Reading the netlist generated from the schematic

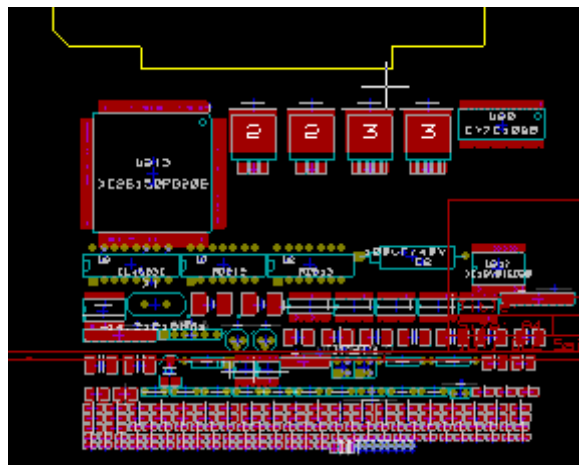
Activate the  icon to display the netlist dialog window:



If the name (path) of the netlist in the window title is incorrect, use the 'Select' button to browse to the desired netlist. Then 'Read' the netlist. Any modules not already loaded will appear, superimposed one upon another (we shall see below how to move them automatically).



If none of the modules have been placed, all of the modules will appear on the board in the same place, making them difficult to recognize. It is possible to arrange them automatically (using the command 'Global Place/Move module' accessed via the right mouse button). Here is the result of such automatic arrangement:



Important note:

If a board is modified by replacing an existing module with a new one (for example changing a 1/8W resistance to 1/2W) in CVPCB, it will be necessary to delete the existing component before PCBNEW will load the replacement module. However, if a module is to be replaced by an existing module, this is easier to do using the module dialog accessed by clicking the right mouse button over the module in question.

Correcting a board

It is very often necessary to correct a board following the corresponding change in the schematic.

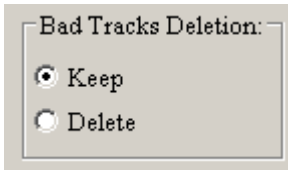
Steps to follow

- Create a new netlist from the modified schematic.
- If new components have been added, link these to their corresponding modules in cvpcb.

- Read the new netlist in Pcbnew.

Deleting incorrect tracks

Pcbnew is able to delete automatically tracks that have become incorrect as a result of modifications. To do this, check the 'Delete' option in the 'Bad tracks deletion' box of the netlist dialog:



However, it is often quicker to modify such tracks by hand (the DRC function allows their identification).

Deleted components

Pcbnew can delete modules corresponding to components that have been removed from the schematic.

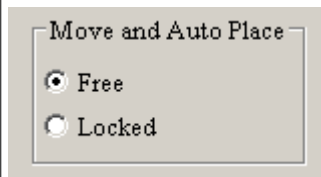
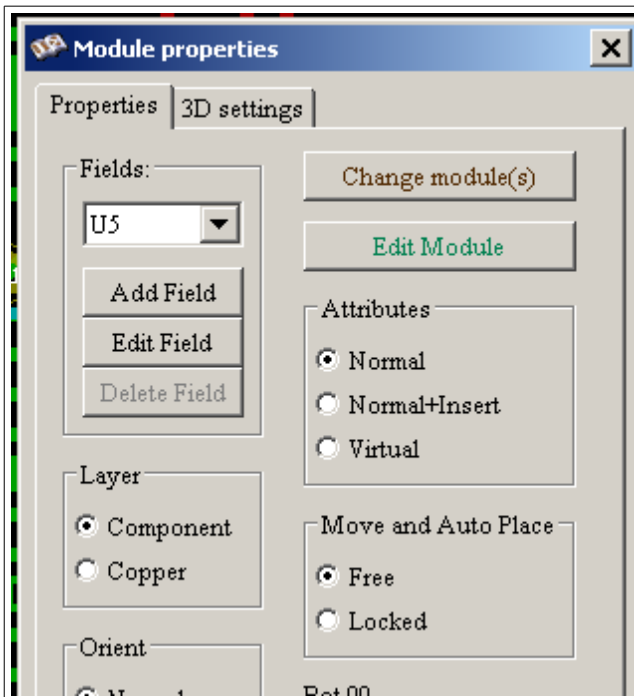
This is optional.

This is necessary because there are often modules (holes for fixation screws, for instance) that are added to the PCB that never appear in the schematic.



If option Remove Extra Footprints is checked, a footprint corresponding to a component not found in netlist will be deleted, unless they have the option "Locked" active.

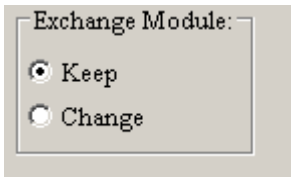
This is a good idea to activate this option for "mechanical" footprints.



Option to lock/unlock a footprint.

Modified modules

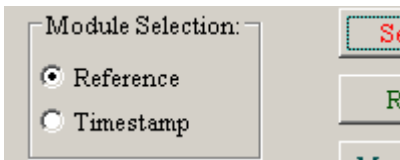
If a module is modified in the netlist (using *Cvpcb*), but the module has already been placed, it will not be modified by *Pcbnew*, unless the corresponding option of the 'Exchange module' box of the netlist dialog is checked:



Changing a module (replacing a resistance with one of a different size, for instance) can be effected directly by editing the module.

Advanced options - selection using time stamps

Sometimes the notation of the schematic is changed, without any material changes in the circuit (this would concern the references - like R5, U4...). The PCB is therefore unchanged (except possibly for the silkscreen markings). Nevertheless, internally, components and modules are represented by their reference. In this situation, the 'Timestamp' option of the netlist dialog may be selected before rereading the netlist:



With this option, *Pcbnew* no longer identifies modules by their reference, but by their time stamp instead. The time stamp is automatically generated by *Eeschema* (it is the time and date when the component was placed in the schematic).

Great care should be exercised when using this option (save the file first!)

This is because the technique is complicated in the case of components containing multiple parts (e.g. a 7400 has 4 parts and one case). In this situation, the time stamp is not uniquely defined (for the 7400 there would be up to four – one for each part). Nevertheless, the time stamp option usually resolves re annotation problems.

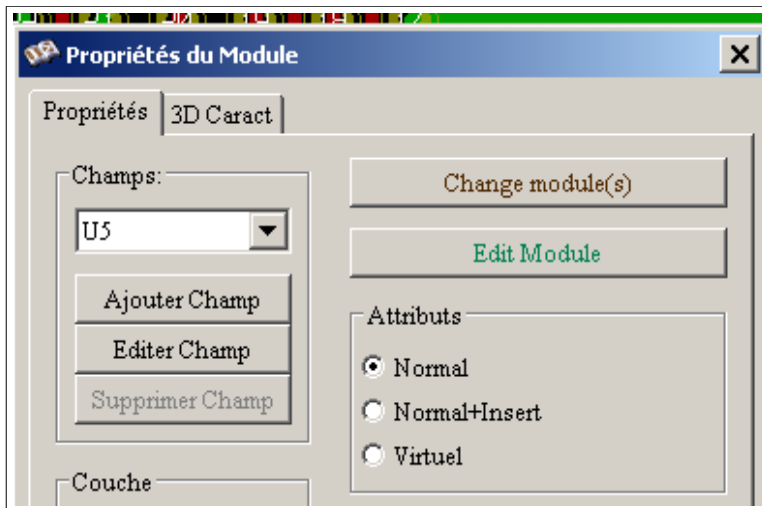
Direct exchange for footprints already placed on board

Changing a footprint (or some identical footprints) to an other footprint is very usefull.

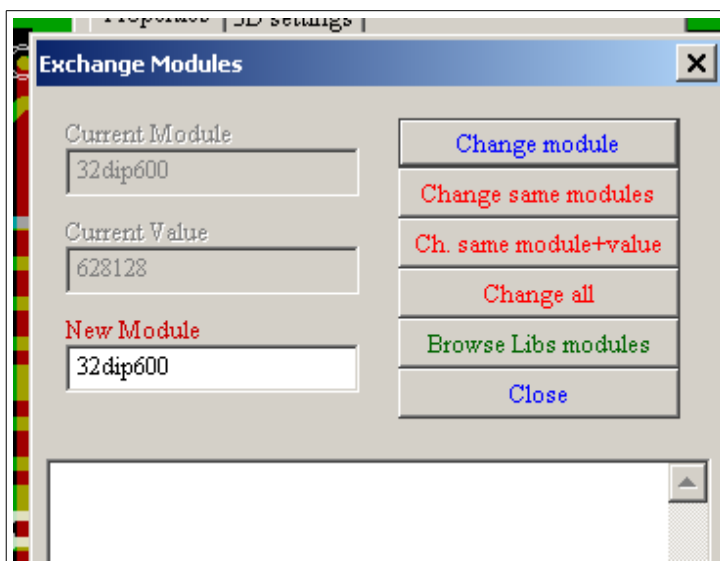
This is very easy:

Cick on a footprint to open the Edit dialog box.

Activate Change Modules.



access to Change Modules



Options for footprints exchange:

One must choose a new footprint name and use:

- **Change Module** for the current footprint
- **Change same modules** for all footprints like the current footprint.
- **Change same module+value** for all footprints like the current footprint, restricted to components which have the same value.

Note:


- **Change all** reload all footprints on board.

7 - Modules placement

Table of Contents

7 - Modules placement.....	1
Assisted placement.....	1
Manual placement.....	1
General re-orientation of modules.....	2
Automatic Module Distribution.....	3
Automatic placement of modules.....	4
Characteristics of the automatic placer.....	4
Preparation.....	4
Interactive auto-placement.....	5
Additional note.....	5

Assisted placement

Whilst moving modules the module ratsnest (the net connections) can be displayed to assist the placement. To enable this the icon  of the left toolbar must be activated.

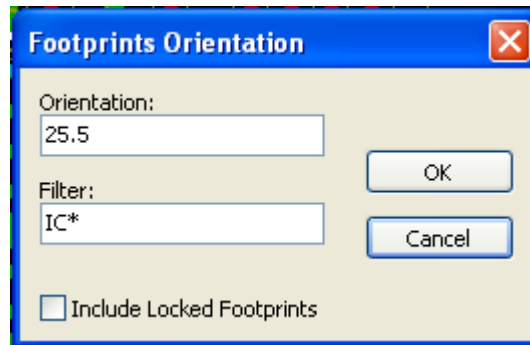
Manual placement

Select the module with the right mouse button then chose the Move command from the menu. Move the module to the required position and place it with the left mouse button. If required the selected module can also be rotated, inverted or edited. Select Cancel from the menu (or press the Esc key) to abort.

General re-orientation of modules

Initially all modules inherit the same orientation that they had in the library (normally 0).


If an alternative orientation is required for an individual module, or all modules (for example all vertical) use the menu option AutoPlace/Orient All Modules. This orientation can be selective (for example to relate only to the modules whose reference starts with "IC").



Automatic Module Distribution

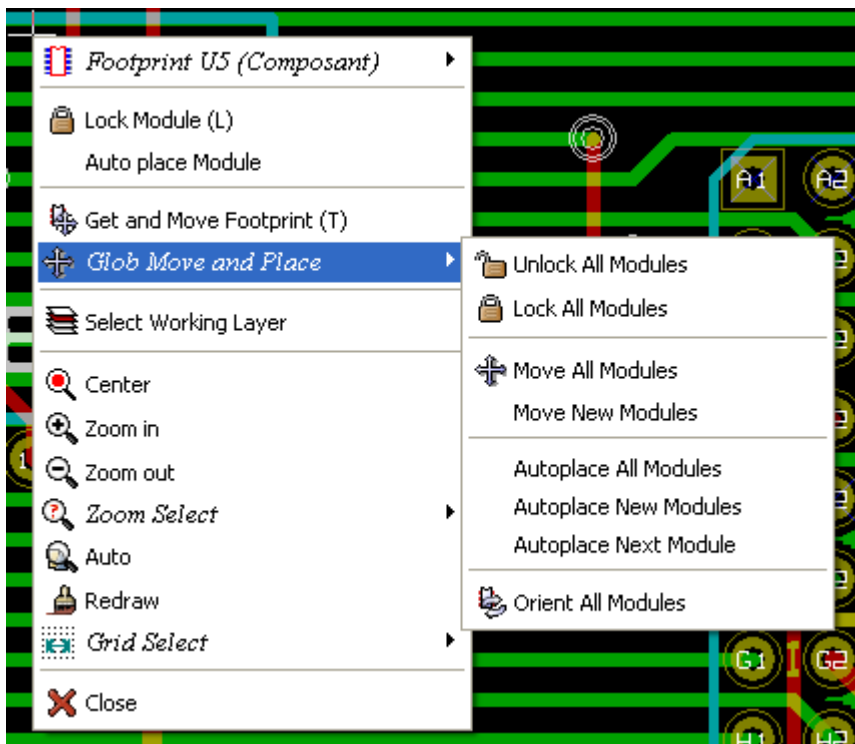
Generally speaking, modules can only be moved if they have not been "Fixed". This attribute can be turned on and off from the pop-up window (click right mouse button over module) whilst in Module Mode, or through the Edit Module Menu.

As stated in the last chapter, new modules loaded during the reading of the netlist appear piled up at a single location on the board. PCBNEW allows an automatic distribution of the modules to make manual selection and placement easier.

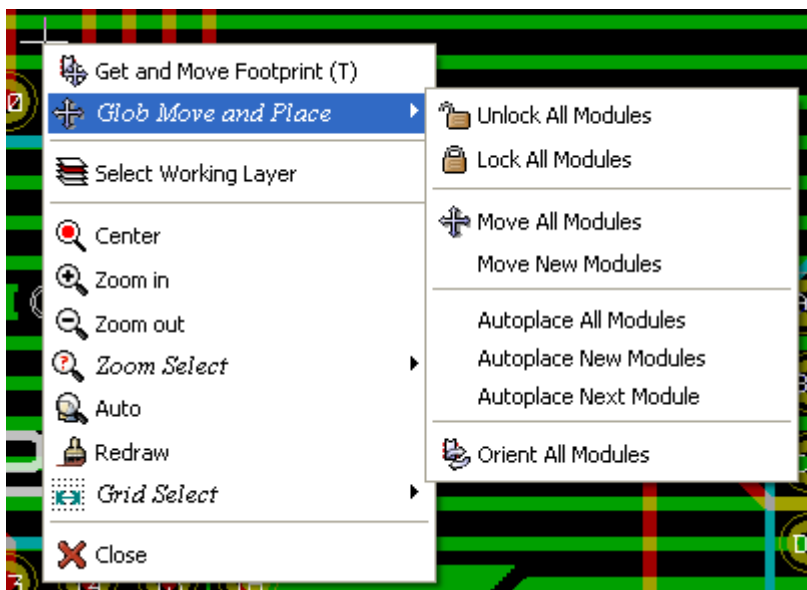
Select the option "Module Mode" (Icon  on the upper toolbar).

The PopUp window activated by the right mouse button becomes:

If there is a module under the cursor:



If there is nothing under the cursor:



In both cases the following commands are available:

- **Move all Modules** allows the automatic distribution of all the modules not Fixed. This is generally used after the first reading of a netlist.
- **Move new Modules** allows the automatic distribution of the modules which have not been placed already within the PCB outline. This commands requires that an outline of the board has been drawn to determine which modules can be automatically distributed.

Automatic placement of modules

Characteristics of the automatic placer

The automatic module of placement allows the placement of the modules onto the 2 faces of the circuit board (however switching a module onto the copper layer is not automatic).



It also seeks the best orientation (0, 90, - 90, 180 degrees) of the module.

The placement is made according to an optimization algorithm, which seeks to minimize the length of the ratsnest, and which seeks to create space between the larger modules with many pads. The order of placement is optimized to initially place these larger modules with many pads.

Preparation

PCBNEW can thus place the modules automatically, however it is necessary to guide this placement, because no software can guess what the user wants to achieve.

Before an automatic placement is carried out one must:

- Create the outline of the board (It can be complex, but it must be closed if the form is not rectangular).
- Manually place the components whose positions are imposed (Connectors, clamp holes...).
- Similarly certain SMD modules and critical components (large modules for example) must be on a specific side or position on the board and this must be done manually.
- Having completed any manual placement these modules must be "Fixed" to prevent them being moved. With the Module Mode icon  selected right click on the module and pick "Fix Module" on the Pop-up menu. This can also be done through the Edit/Module Pop-up menu.
- Automatic placement can then be carried out. With the Module Mode icon  selected, right click and select Glob(al) Move and Place – then Autoplace All Modules.

During automatic placement, if required, PCBNEW can optimize the orientation of the modules. However rotation will only be attempted if this has been authorized for the module (see Edit Module Options).

Usually resistances and non-polarized capacitors are authorized for 180 degrees rotation. Some modules (small transistors for example) can be authorized for +/- 90 and 180 degrees rotation.

For each module one slider authorizes 90 degree Rot(ation) and a second slider authorizes 180 degree Rot(ation). A setting of 0 prevents rotation, a setting of 10 authorizes it, and an intermediate value indicates a preference for/against rotation.

The rotation authorization can be done by editing the module once it is placed on the board. However it is preferable to set the required options to the module in the library as these settings will then be inherited each time the module is used.

Interactive auto-placement

It may be necessary during automatic placement to stop (press Esc key) and manually re-position a module. Using the command Autoplace Next Module will restart the autoplacement from the point at which it was stopped.

The command Autoplace new modules allows the automatic placement of the modules which have not been placed already within the PCB outline. It will not move those within the PCB outline even if they are not 'fixed'.

The command Autoplace Module makes it possible to re-place the module pointed to by the mouse, even if its 'fixed' attribute is active.

Additional note

PCBNEW automatically determines the possible zone of placement of the modules by respecting the shape of the board outline, which is not necessarily rectangular (It can be round, or have cutouts...).

If the board is not rectangular, the outline must be closed, so that PCBNEW can determine what is inside and what is outside the outline. In the same way, if there are internal cutouts, their outline will have to be closed.

PCBNEW calculates the possible zone of placement of the modules using the outline of the board, then passes each module in turn over this area in order to determine the optimum position at which to place it.

8 - Setting routing parameters

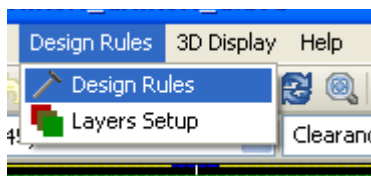
Table of Contents

8 - Setting routing parameters.....	1
Current settings.....	1
Accessing the main dialog.....	1
Current settings.....	1
General options.....	2
Netclasses.....	2
Setting routing parameters.....	3
Netclass editor.....	3
Global Design Rules.....	4
Via parameters.....	5
Track parameters.....	6
Specific sizes.....	6
Examples and typical dimensions.....	6
Track width.....	6
Insulation (clearance).....	6
Examples.....	6
Manual routing.....	8
Help when creating tracks.....	8
Creating tracks.....	8
Moving and dragging tracks.....	9
Via Insertion.....	9
Select/edit the track width and via size.....	10
Using the horizontal toolbar.....	10
Using the pop-up menu.....	10
Editing and changing tracks.....	11
Change a track.....	11
Global changes.....	12

Current settings

Accessing the main dialog

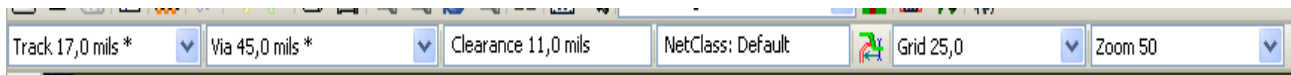
The most important parameters are accessed from the following drop-down menu.



and are set in the Design Rules dialog.

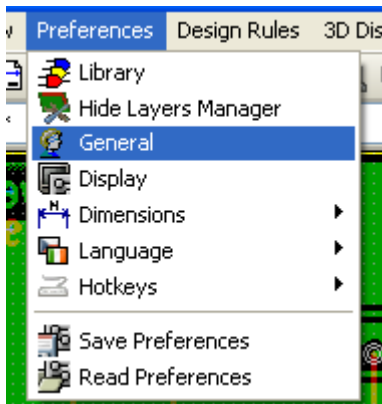
Current settings

Current settings are displayed by the top toolbar.

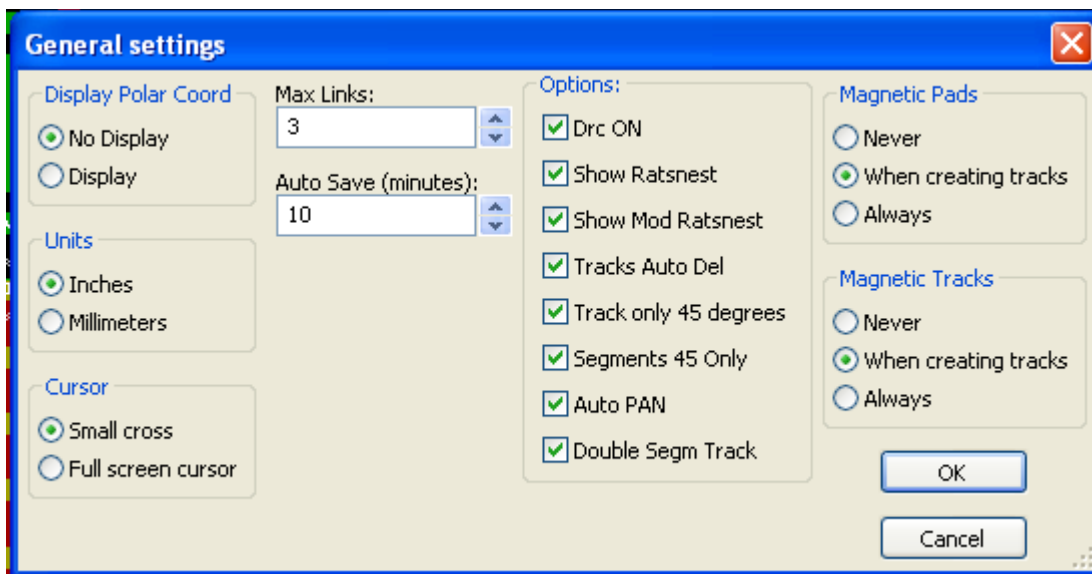


General options

The General options menu is available via the top toolbar link Preferences → General dialog.



The dialog menu looks like the following.



For the creation of tracks the necessary parameters are:

- Tracks 45 Only: Directions allowed for track segments are 0, 45 or 90 degrees.
- Double Segm Track: When creating tracks, 2 segments will be displayed.
- Tracks Auto Del: When recreating tracks, the old one will be automatically delete if considered redundant.
- Magnetic Pads: The graphic cursor becomes a pad, centered in the pad area.
- Magnetic Tracks: The graphic cursor becomes the track axis.

Netclasses

Pcbnew allows you to define different routing parameters for each net. Parameters are defined by a group of nets.

- A group of nets is called a Netclass.
- There is always a netclass called default.

- Users can add others Netclasses.

A netclass specifies:

- The width of tracks, via diameters and drills.
- The clearance between pads and tracks (or vias).

When routing, Pcbnew selects automatically the netclass corresponding to the net of the track to create or edit, and therefore the routing parameters.

Setting routing parameters

The choice is made in the menu: Design Rules → Design Rules.

Netclass editor

The Netclass editor allows you to:

- add or delete Netclasses.
- set routing parameters values : clearance, track width, via sizes.
- group nets in netclasses.

Design Rules Editor

Net Classes Editor Global Design Rules

Net Classes:

	Clearance	Track Width	Via Dia	Via Drill	uVia Dia	uVia Drill
Default	0,0110	0,0170	0,0450	0,0250	0,0200	0,0080
auxrule	0,0150	0,0200	0,0450	0,0250	0,0200	0,0080
power	0,0150	0,0200	0,0450	0,0250	0,0200	0,0080

Add Remove Move Up

Membership:

Default power

	Default
/8MH-OUT	Default
/ACK	Default
/AUTOFD-	Default
/BIT0	Default
/BIT1	Default
/BIT2	Default
/BIT3	Default
/BIT4	Default
/BIT5	Default
/BIT6	Default
/BIT7	Default
/BUST+	Default
/CLKLCA	Default
/CS1-	Default
/D0	Default
/D1	Default
/D2	Default
/D3	Default
/D4	Default

<<< >>> << Select All Select All >>

	power
GND	power
VCC	power

Messages:

Current general settings:
 Minimum value for tracks width: **0,0080 "**
 Minimum value for vias diameter: **0,0350 "**
 Minimum value for microvias diameter: **0,0200 "**

OK Cancel

Global Design Rules

The global design rules are:

- Via type.
- Enabling/disabling micro-via use.
- Minimum clearance (minimum distance between tracks, vias and pads).

- Minimum tracks and vias sizes.

A DRC error is rose when a value smaller than the minimum value specified is encountered. The second dialog panel is:

Design Rules Editor

Net Classes Editor | Global Design Rules

Via Options:

Default Via Type

☒ Through via
☐ Blind or buried via

Min via diameter ("): 0,0350
 Min via drill dia ("): 0,0190

Micro Via Options:

Micro Vias:

☒ Do not allow micro vias
☐ Allow micro vias

Min micro via drill dia ("): 0,0070

Minimum Allowed Values:

Min track width ("): 0,0080

Select the current via type. Through via is the usual selection

Specific via diameters and track widths, which can be used to replace default Netclass values on demand, for arbitrary via or track segments.

Custom Via Sizes:

Drill value: a blank or 0 => default Netclass value

	Diameter	Drill
Via 1	0,0450	
Via 2	0,0550	0,0320
Via 3	0,0550	0,0400
Via 4	0,0650	0,0320
Via 5		
Via 6		
Via 7		

Custom Track Widths:

	Width
Track 1	0,0200
Track 2	0,0250
Track 3	
Track 4	
Track 5	
Track 6	
Track 7	

Messages:

Current general settings:
 Minimum value for tracks width: **0,0080 "**
 Minimum value for vias diameter: **0,0350 "**
 Minimum value for microvias diameter: **0,0200 "**

OK Cancel

This dialog also allows to enter a "stock" of tracks and vias sizes.

When routing, one can select one of these values to create a track of via, instead of using the default netclasses values ones.

Useful in critical cases when a small track segment must have a specific size.

Via parameters

Pcbnew handles 3 types of vias:

- The through via (usual vias).
- Blind or buried vias.
- Micro Vias, like buried vias but restricted to an external layer to its nearest neighbor.

They are intended to connect BGA pins to the nearest inner layer. Their diameter is usually very small and they are drilled by laser.

By default, all vias have the same drill value.

This dialog specifies the smallest acceptable values for vias parameters. On a board, a via smaller than specified here generates a DRC error.

Track parameters

Specify the minimum acceptable track width. On a board, a track width smaller than specified here generates a DRC error.

Specific sizes

Specific via diameters and track widths, which can be used to replace default Netclass values on demand, for arbitrary via or track segments.

Custom Via Sizes:

Drill value: a blank or 0 => default Netclass value

	Diameter	Drill
Via 1	0,0450	
Via 2	0,0550	0,0320
Via 3	0,0550	0,0400
Via 4	0,0650	0,0320
Via 5		
Via 6		
Via 7		

Custom Track Widths:

	Width
Track 1	0,0200
Track 2	0,0250
Track 3	
Track 4	
Track 5	
Track 6	
Track 7	

One can enter a set of extra tracks and/or vias sizes. While routing a track, these values can be used on demand instead of the values from the current netclass values.

Examples and typical dimensions

Track width

Use the largest possible value and conform to the minimum sizes given here.

Units	CLASS 1	CLASS 2	CLASS 3	CLASS 4	CLASS 5
mm	0,8	0,5	0,4	0,25	0,15
mils	31	20	16	10	6

Insulation (clearance)

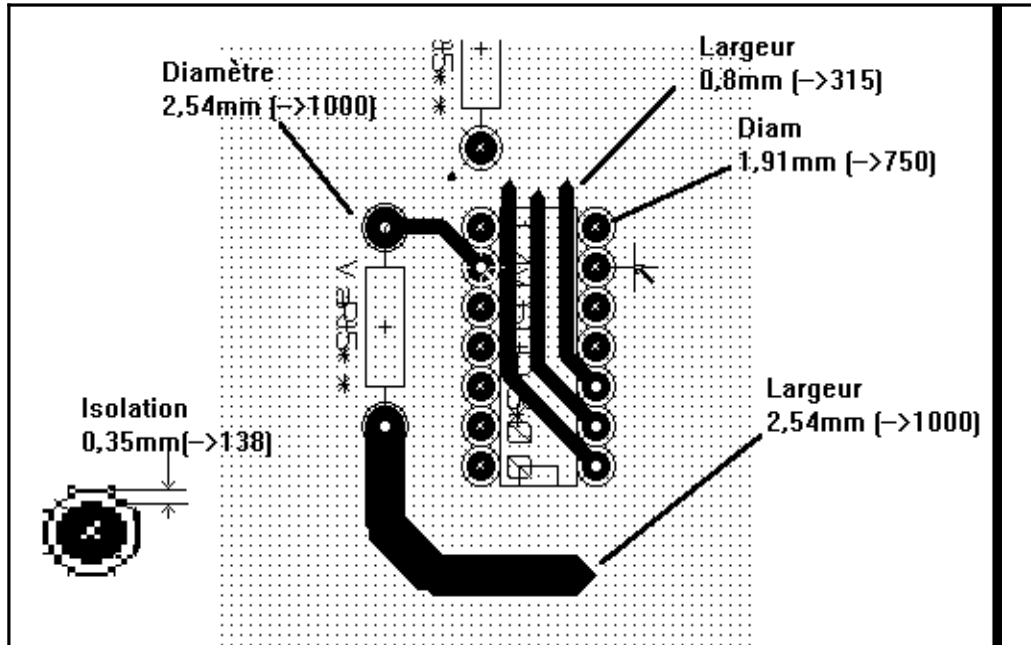
Unité	CLASS 1	CLASS 2	CLASSE3	CLASS 4	CLASS 5
mm	0,70	0,5	0,35	0,23	0.15
mils	27	20	14	9	6

Usually, the minimum clearance is very similar to the minimum track width.

Examples

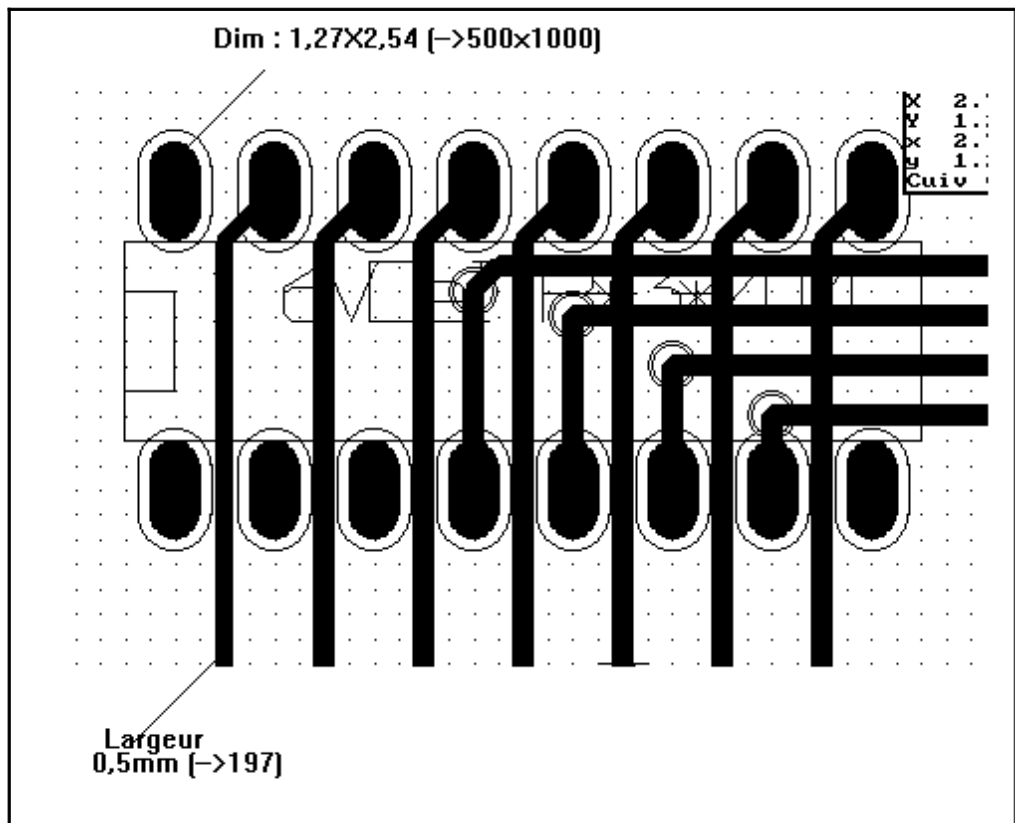
'Rustic'

- Clearance: 0.35mm (0.0138 inches).
- Track width: 0.8mm (0.0315 inches).
- Pad diameter for ICs and vias: 1.91mm (0.0750 inches).
- Pad diameter for discrete components: 2.54mm (0.1 inches).
- Ground track width: 2.54mm (0.1 inches).



'Standard'

- Clearance: 0.35mm (0.0138 inches).
- Track width: 0.5mm (0.0127 inches).
- Pad diameter for ICs: make them elongated in order to allow tracks to pass between IC pads and yet have the pads offer a sufficient adhesive surface (1.27 x 2.54 mm -->0.05x 0.1 inches).
- Vias : 1.27mm (0.0500 inches).





Manual routing

Manual routing is often recommended, because it is the only method offering control over routing priorities. For example, it is preferable to start by routing power tracks, making them wide and short and keeping analog and digital supplies well separated. Later, sensitive signal tracks should be routed. Amongst other problems, automatic routing often requires many vias. However, automatic routing can offer useful insight into the positioning of modules. With experience, you will probably find that the automatic router is useful for quickly routing the 'obvious' tracks, but the remaining tracks will best be routed by hand.


Help when creating tracks

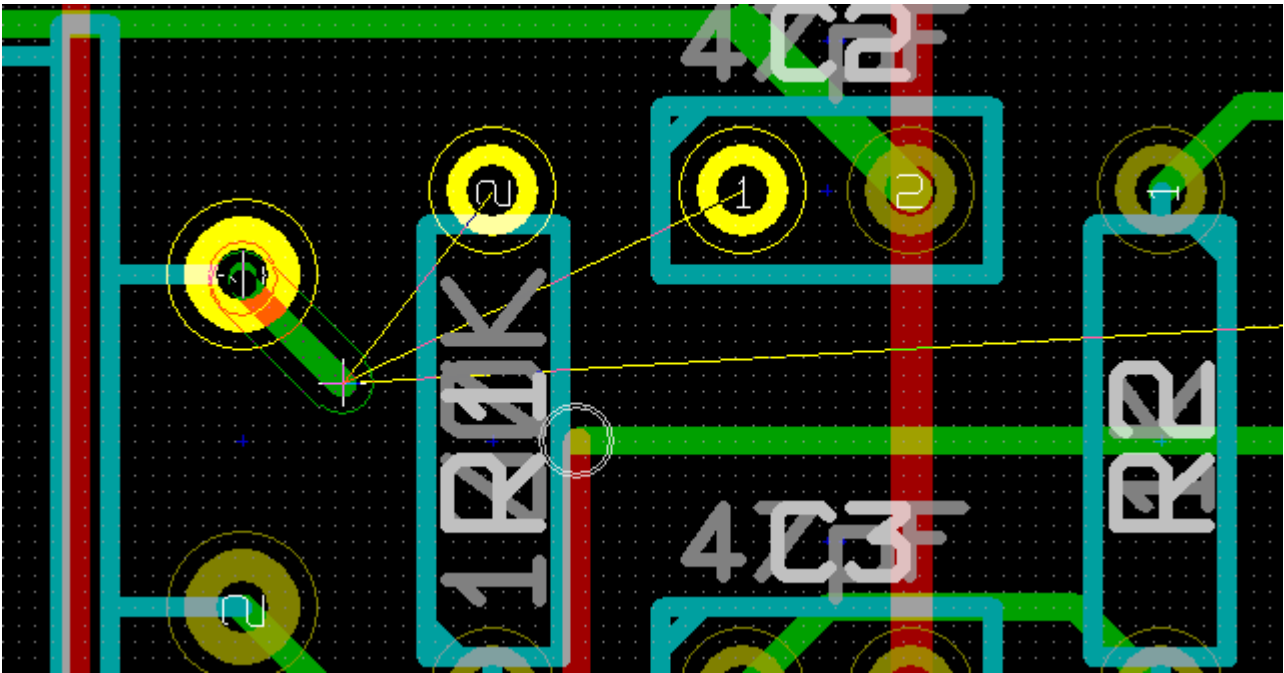
Pcbnew can display the full ratsnest, if the button  is activated.

The button  allows to highlight a net (click to a pad or an existing track to highlight the corresponding net).

The DRC checks in real time tracks when creating them. One cannot create a track which does not match the DRC rules. It is possible to disable the DRC by clicking on the button . This is however not recommended, use it only in specific cases.

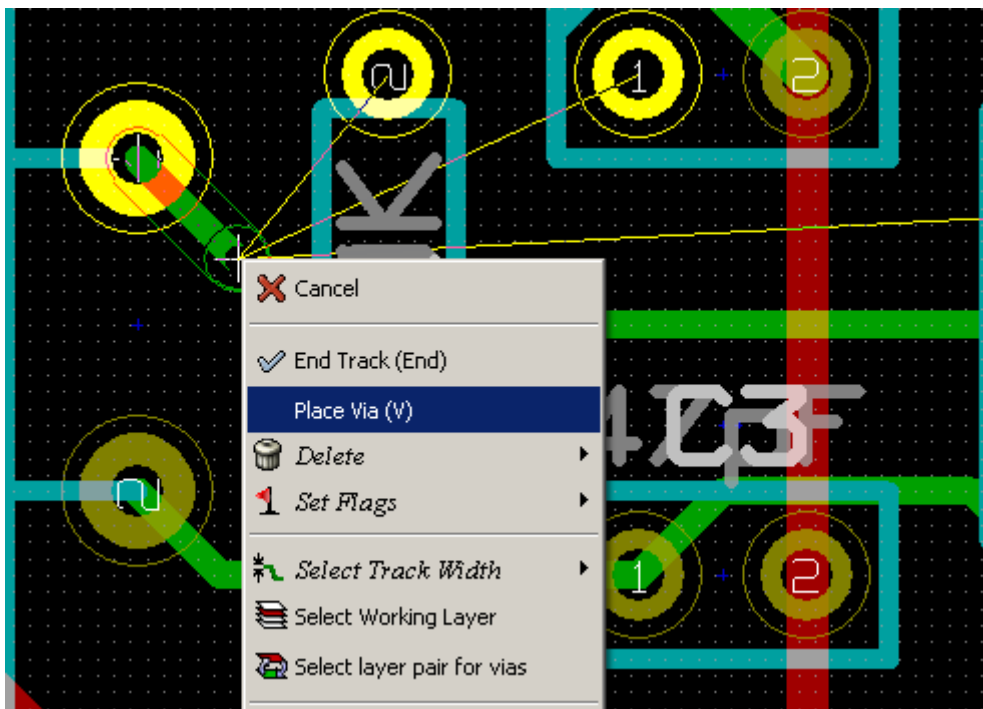
Creating tracks

A track can be created by clicking on the button . A new track must start on a pad or on another track, because Pcbnew must know the net used for the new track (in order to match the DRC rules).




When creating a new track, Pcbnew shows links to nearest not connected pads, link number set in option "Max. Links" in General Options.

End the track by a double click, by the pop-up menu or by its hot key.



Moving and dragging tracks

When the button  is active, the track where the cursor is positioned can be moved with the hotkey 'm'. If you want to drag the track you can use the hotkey 'g'.

Via Insertion

A via can be inserted only when a track is in progress:


- By the pop-up menu.

- By the hotkey 'v'.
- By switching to a new copper layer using the appropriate hotkey.

Select/edit the track width and via size

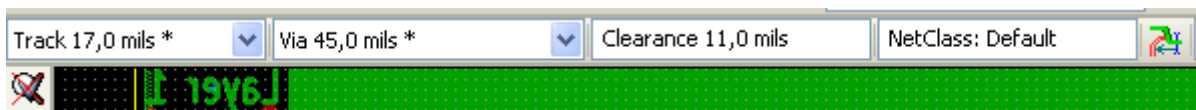
When clicking on a track or a pad, Pcbnew automatically selects the corresponding Netclass, and the track size and vias dimensions from this netclass.

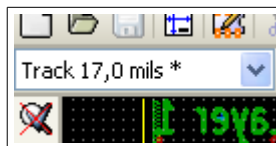
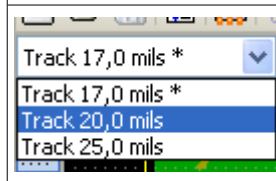
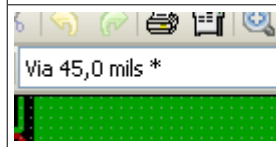
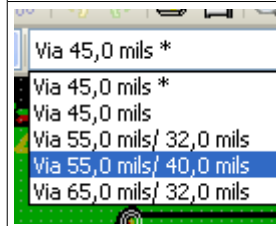
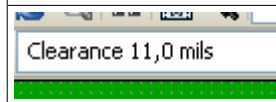
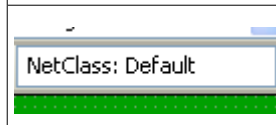

As previously seen, the Global Design Rules editor has a tool to insert extra tracks and vias sizes.

- The horizontal toolbar can be used to select a size.
- When the button  is active, the current track width can be selected from the pop-up menu (accessible as well when creating a track).

The user can utilize the default Netclasses values or a specified value.

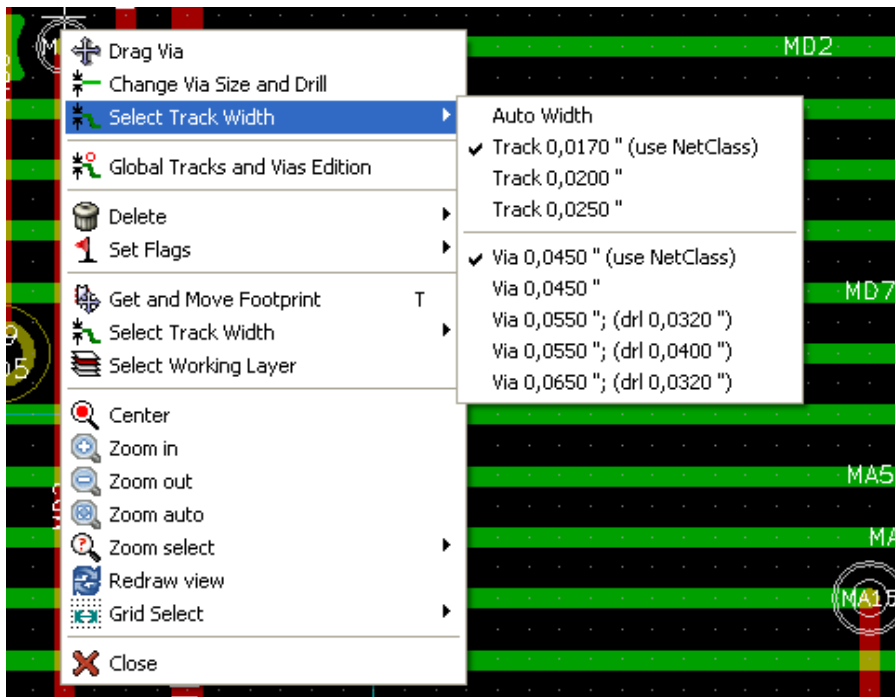
Using the horizontal toolbar



	<p>Track width selection.</p> <p>The symbol * is a mark for default Netclass value selection.</p>
	<p>Selecting a specific track width value.</p> <p>The first value in list is always the netclass value.</p> <p>Others values are tracks widths entered from the Global Design Rules editor.</p>
	<p>Via size selection.</p> <p>The symbol * is a mark for default Netclass value selection.</p>
	<p>Selecting a specific via dimension value.</p> <p>The first value in list is always the netclass value.</p> <p>Others values are vias dimensions entered from the Global Design Rules editor.</p>
	<p>Display the current clearance value.</p> <p>This is the clearance value set in the current selected Netclass.</p>
	<p>Current selected Netclass.</p> <p>When clicking on a track or a pad, Pcbnew automatically selects the corresponding Netclass, and displays its name.</p>
	<p>When enabled: Automatic track width selection.</p> <p>When starting a track on an existing track, the new track has the same width as the existing track.</p>

Using the pop-up menu

One can select a new size for routing, or to change a previously created via or track segment.

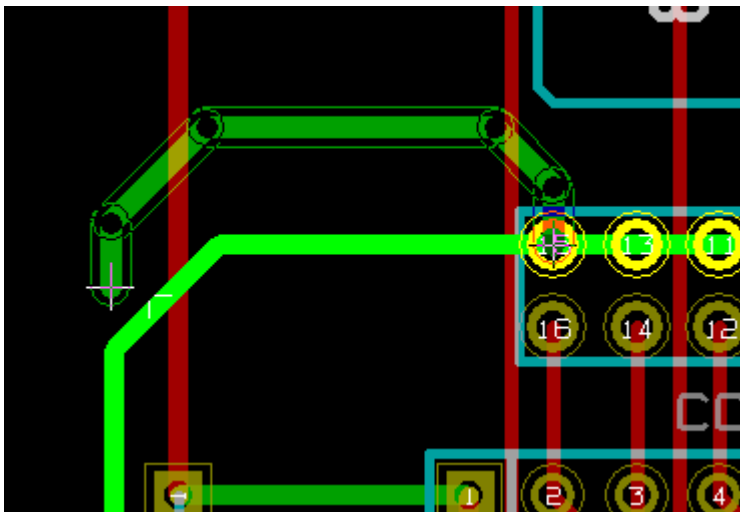


If you want to change many vias (or tracks) size, the best way is to use a specific Netclass to for the net(s) that must be edited (see global changes).

Editing and changing tracks

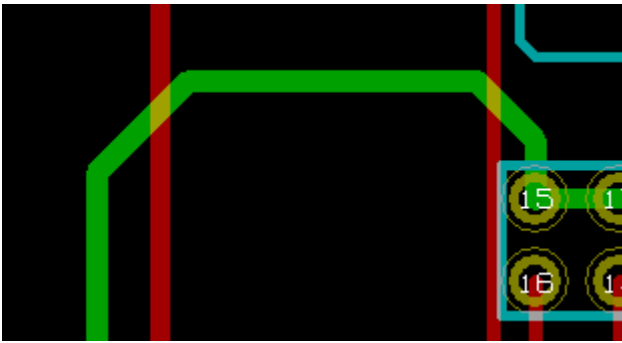
Change a track

In many cases redrawing a track is recommendable.



new track (in progress).

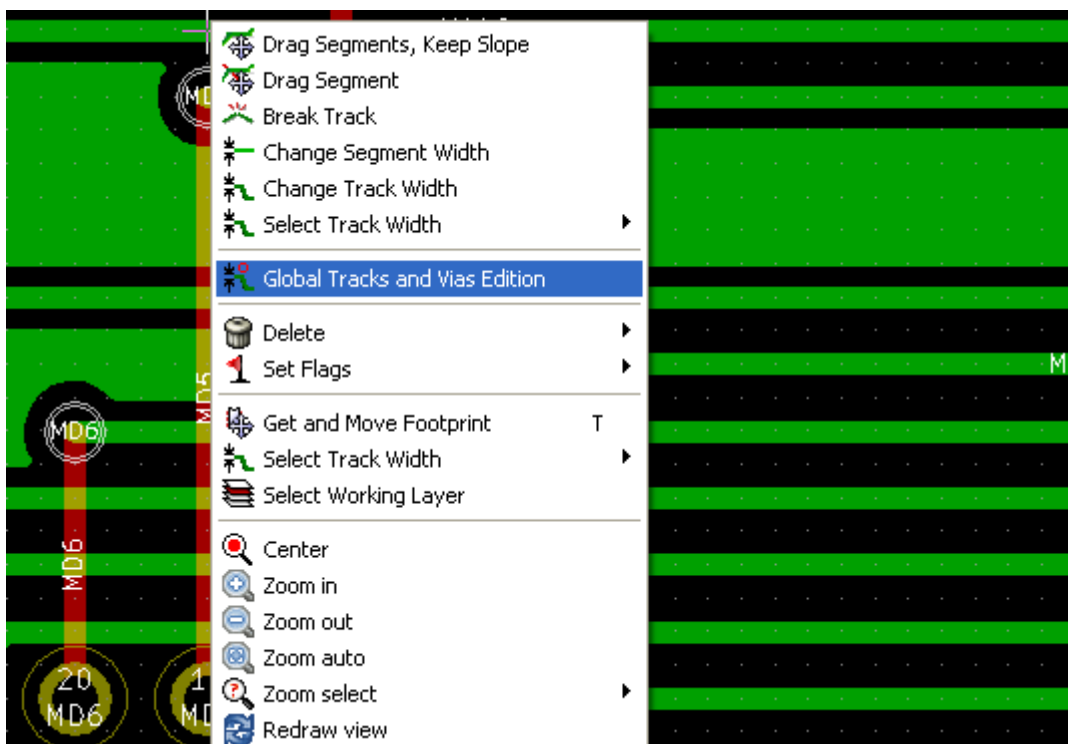
When finished:



Pcbnew remove automatically the old track if it is redundant.

Global changes

Global tracks and via sizes dialog editor is accessible via the pop-up window by right clicking on a track.



The dialog editor allows global changes of tracks and/or vias for:

- The current net.
- The whole board.

Global Edition of Tracks and Vias

Current Settings:

Current Net: /MD5
Current NetClass: Default

	Track size	Via diameter	Via drill	uVia size	uVia Drill
Netclass value	0,0170 "	0,0450 "	0,0250 "	0,0200 "	0,0080 "
Current value	Default	Default	Default	Default	Default

Global Edition Option:

☐ Set tracks and vias of the current Net to the current value

☒ Set tracks and vias of the current Net to the Netclass value

☐ Set all tracks and vias to their Netclass value

☐ Set all vias (no track) to their Netclass value

☐ Set all tracks (no via) to their Netclass value

OK

Cancel

9 - Creating copper zones

Table of Contents

9 - Creating copper zones.....	1
Creating zones on copper layers.....	1
Creating a zone.....	2
Creating the limits of a zone.....	2
Filling the zone.....	3
Filling options.....	5
Filling mode.....	5
Clearance and minimum copper thickness.....	5
Pad options.....	6
Thermal reliefs parameters.....	7
Parameters choice.....	7
Adding a cutout area inside a zone.....	7
Outlines editing.....	8
Editing zone: parameters.....	10
Final zone filling.....	10
Change zones net names.....	10
Creating zones on technical layers.....	11
Creating zone limits.....	11

Copper zones are defined by an outline (closed polygon), and can include holes (closed polygons inside the outline). A zone can be drawn on a copper layer or alternatively on a technical layer.

Creating zones on copper layers

Pad (and tracks) connections by filled copper areas are checked by the DRC engine. A zone must be filled (not just created) to connect pads. Pcbnew uses currently track segments or polygons to fill copper areas.

Each option has its advantages and its disadvantages, mainly when redraw screen. The final result is however the same.

For calculation time reasons, the zone filling is not remade after each change, but only:

- If a filling zone command is executed.
- When a DRC test is made.

Copper zones must be filled or refilled after changes in tracks or pads. Copper zones (Usually ground and power planes) are usually attached to a net.

In order to create a copper zone you should:

- Select parameters (net name, layer ...). To switch on the layer and highlight this net is not mandatory but is a good practice.
- Create the zone limit (If not, all the board will be filled.).

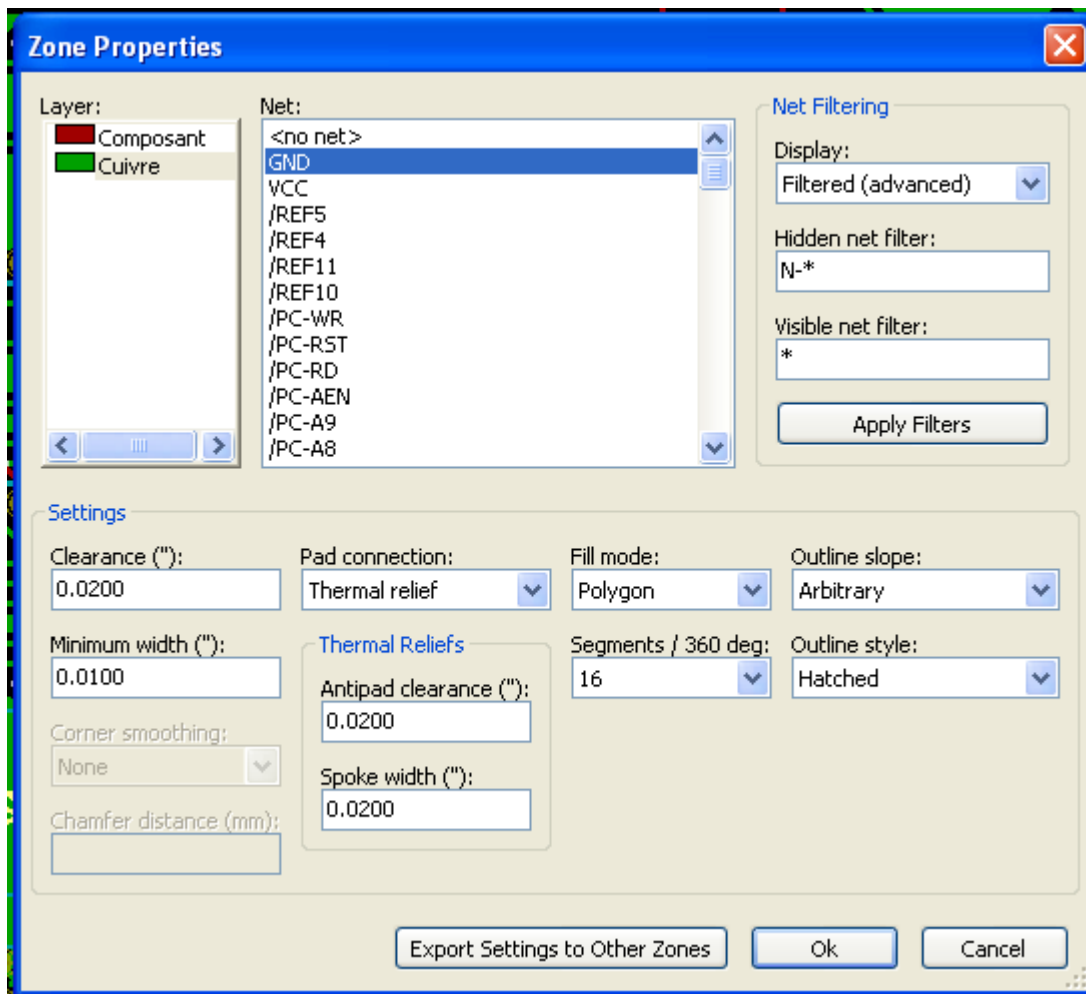
- Fill the zone.

Pcbnew tries to fill all zones in one piece, and usually, there will be no unconnected copper blocks. So it can happens that some areas remain not filled. Zones having no net are not cleaned and can have insulated areas.

Creating a zone

Creating the limits of a zone

Use the tool . The active layer must be a copper layer. When clicking to start the zone outline, the following dialog box will be opened.



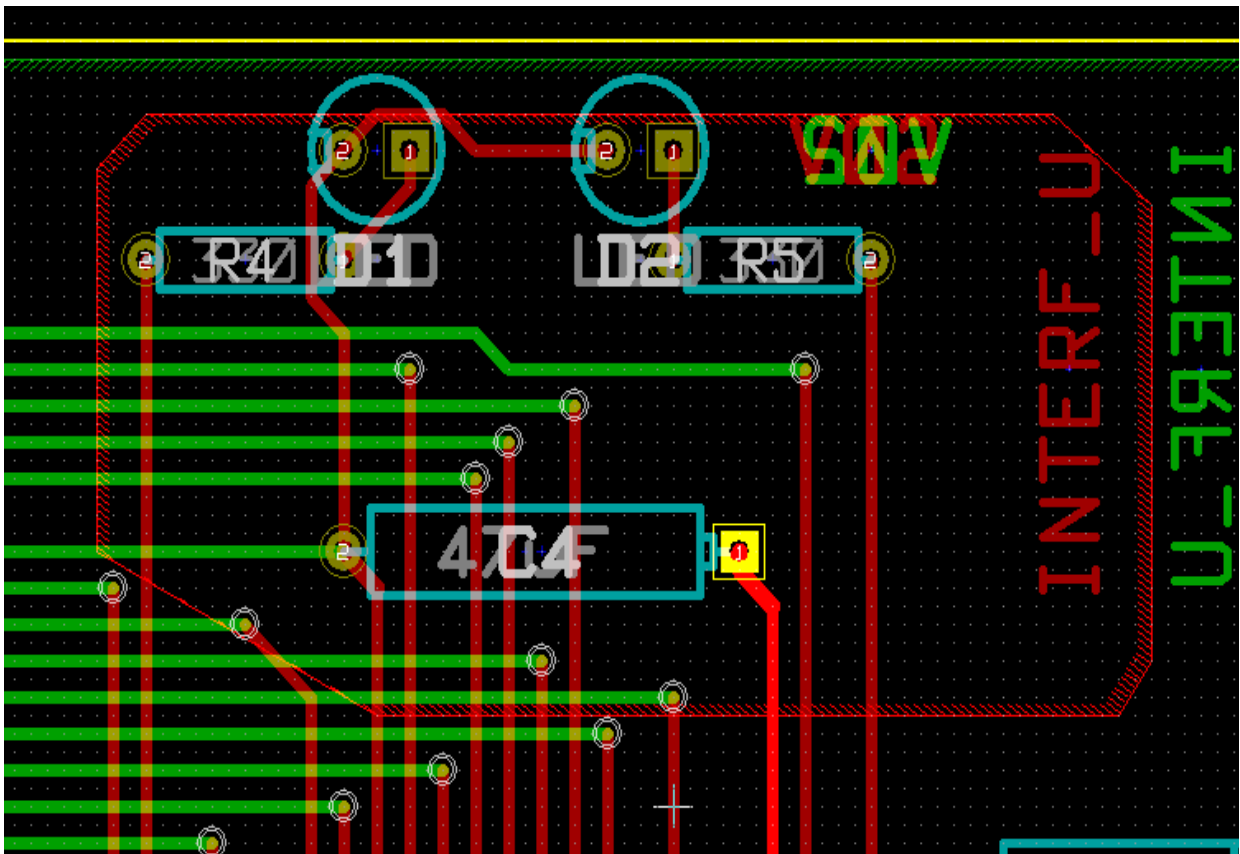
You can specify all parameters for this zone (net, layer, filling options, pad options ...). Draw the zone limit, on this layer. This zone limit is a polygon, created by a left clicking at each corner. A double click will end the polygon.

The polygon will be automatically closed. If the starting point and the ending point are not at the same coordinate, Pcbnew will add a segment from the end point to the start point.

Remarks:

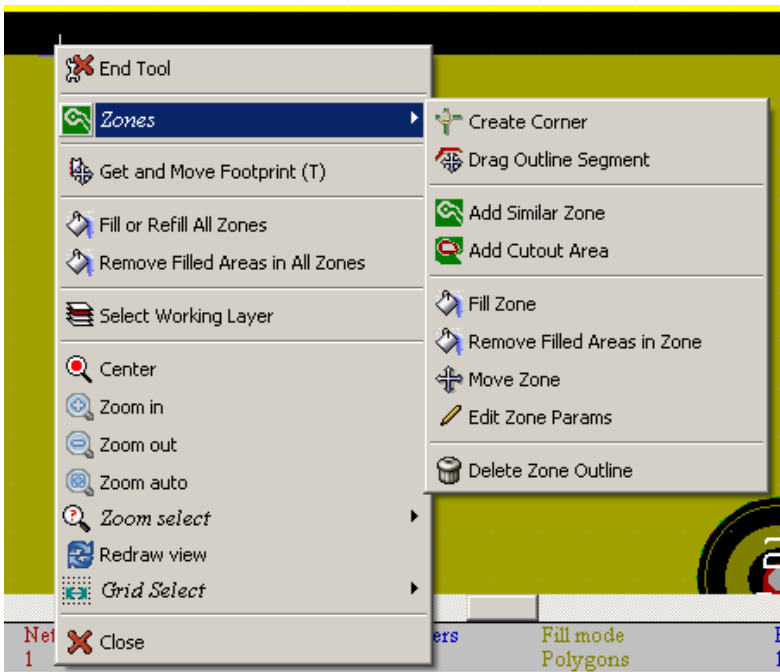
- The DRC control is active when creating zone outlines.
- A corner which creates a DRC error will not be accepted by Pcbnew.

Here=follow you can see and example of a zone limit (polygon in thin hatched line)

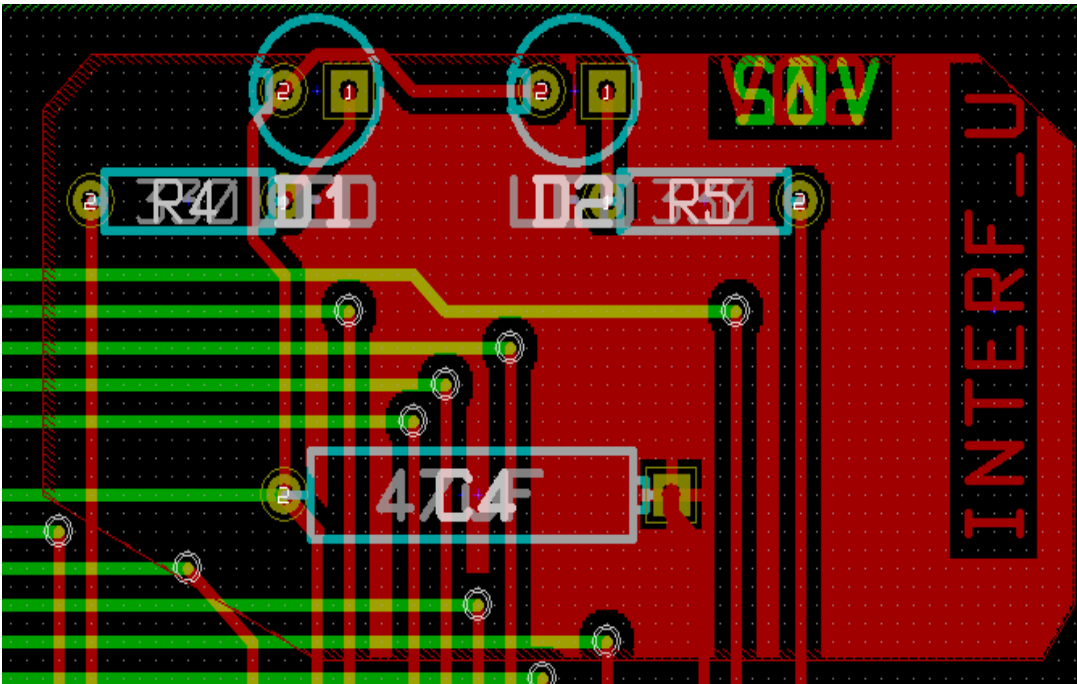


Filling the zone

When filling a zone, Pcbnew remove all not connected copper islands. To access the zone filling command, right click on the edge zone.



Activate the "Fill Zone" command. Here-follow is the filling result for a starting point inside the polygon.

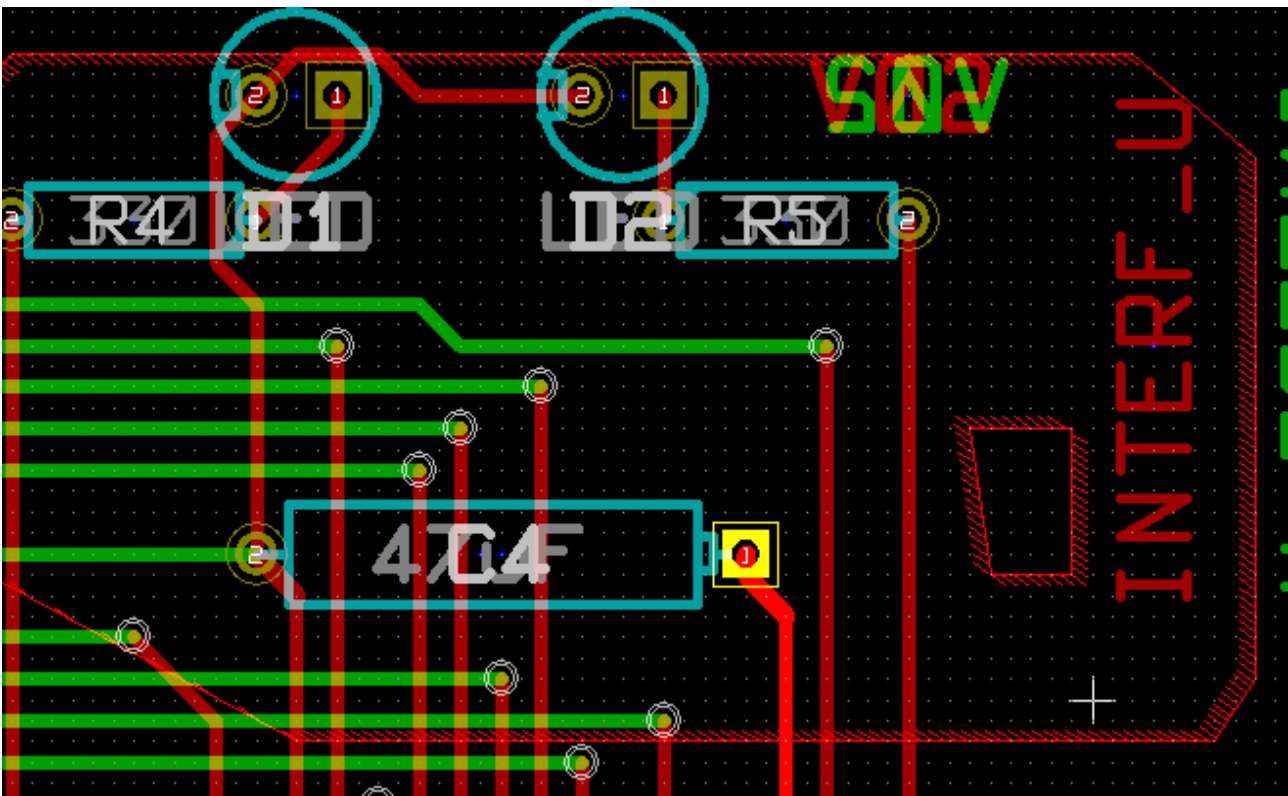


The polygon is the border of the filling area. You can see a non-filled area inside the zone, because this area is not accessible:

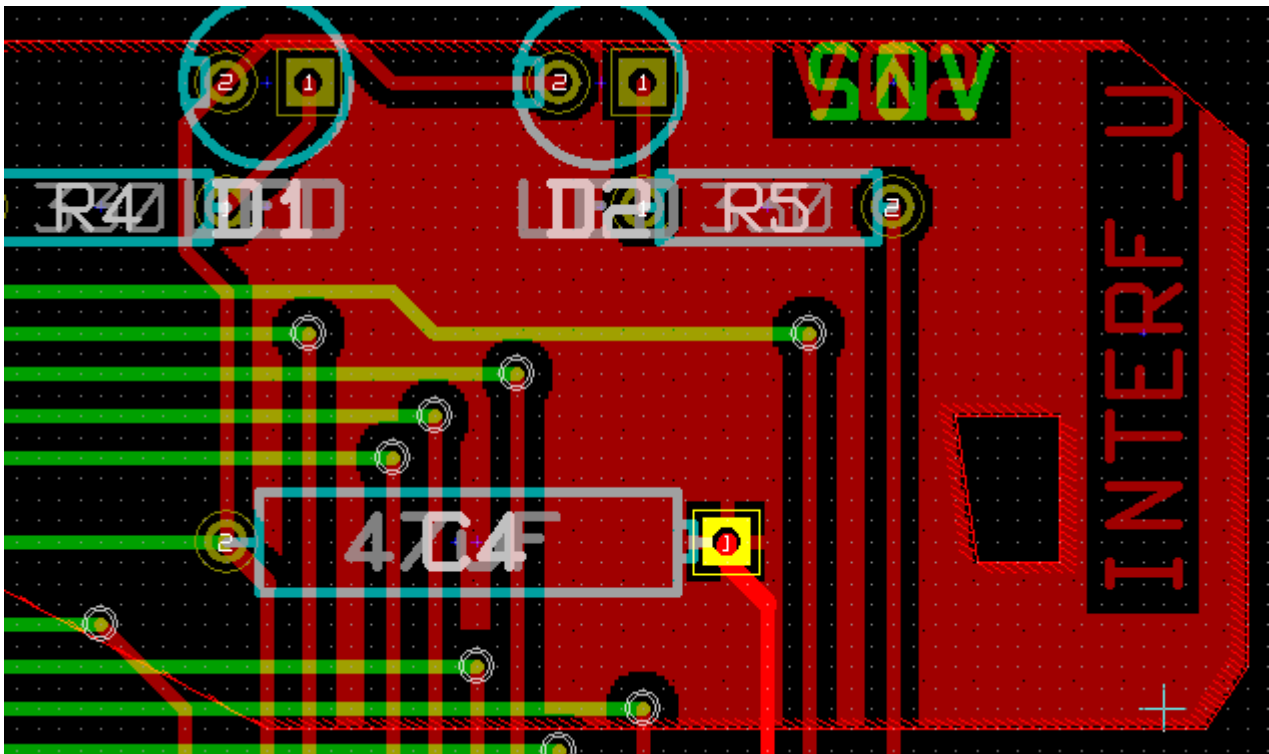
- A track creates a border and
- There is no starting point for filling in this area.

Note:

you can use many polygons to create cutout areas. Here-follow you can see an example.



And here is the result.



Filling options

When you fill and area, you must choose:

- The mode for filling.
- The clearance and minimum copper thickness.
- How pads are drawn inside the zone (or connected to this zone).
- Thermal reliefs parameters.

Filling mode

Zones can be filled using polygons or segments. The result is the same. If you have problems with polygon mode (slow screen refresh) you should use segments.

Clearance and minimum copper thickness

A good choice for clearance is a grid that is a bit bigger than the routing grid. Minimum copper thickness value ensures that there are no too small copper areas.

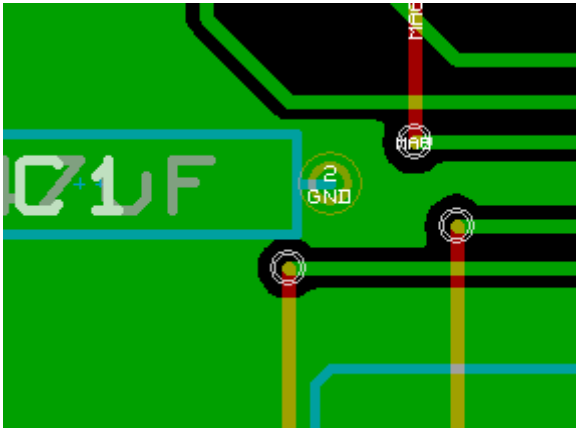
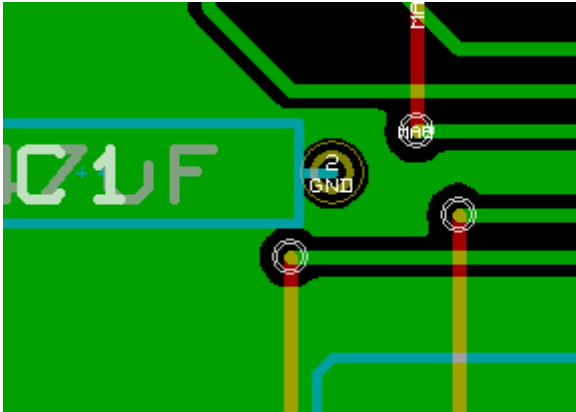
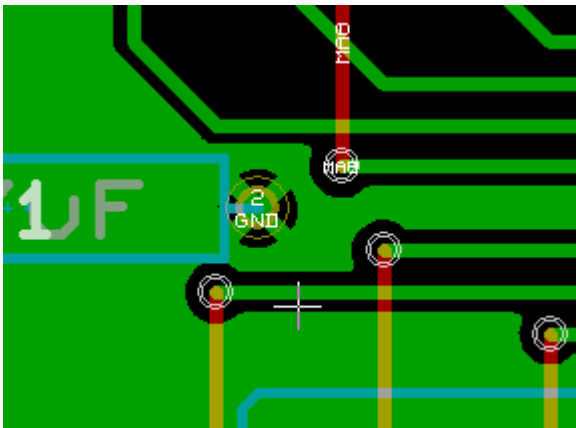
Warning: if this value is too large, small shapes like thermal stubs in thermal reliefs cannot be drawn.

Pad options

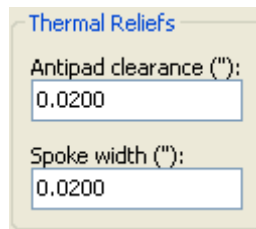
Pads of the net can either be included or excluded from the zone, or connected by thermal reliefs.

- If included, soldering and un-soldering can be very difficult.
- If excluded, the connection to the zone will not be very good.
- A thermal relief is a good compromise.

Here is the result for the 3 options.

	Include Pads
	Exclude Pads Note: <ul style="list-style-type: none">• The zone can be filled only if tracks exists to connect zones areas.• Pads must be connected by tracks.
	Thermal relief. Pad is connected by 4 track segments. The segment width is the current value used for the track width.

Thermal reliefs parameters

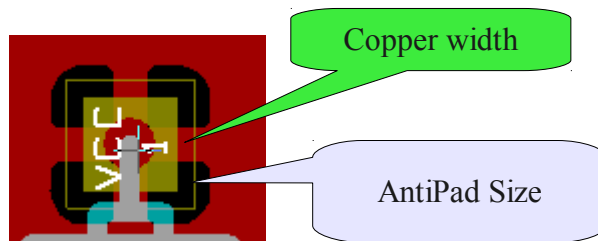


Thermal Reliefs

Antipad clearance ("):
0.0200

Spoke width ("):
0.0200

You can set two parameters for thermal reliefs.



Parameters choice

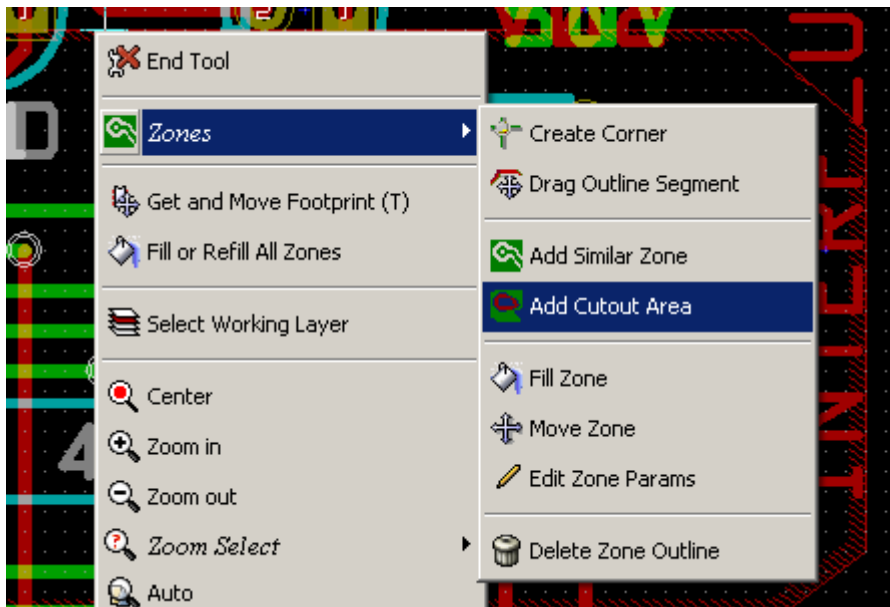
The copper width value for thermal reliefs must be bigger than the minimum thickness value for copper zone. If not, they cannot be drawn.

Additionally, a too large value for this parameter or for antipad size does not allow to create a thermal relief for small pads (like pads sizes used for SMD components).

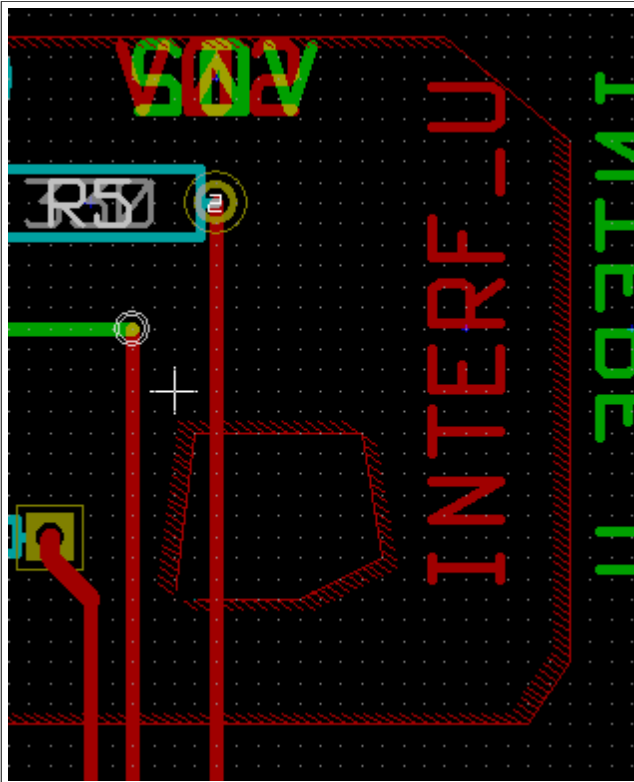
Adding a cutout area inside a zone

A zone must already exist. To add a cutout area (a non filled area inside the zone):

- Right click on an existing edge outline.
- Select Add Cutout Area.
- Creates the new outline.



After creating the outline.



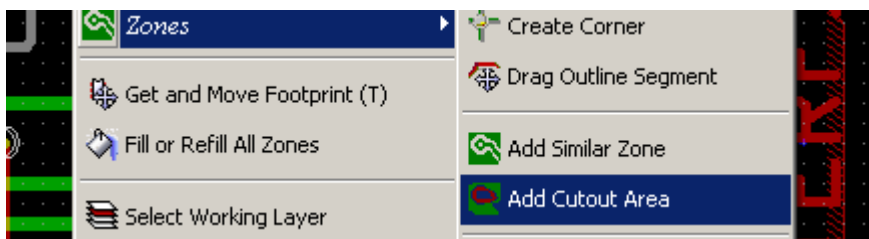
See the cutout outline.

Outlines editing

An outline can be modified by:

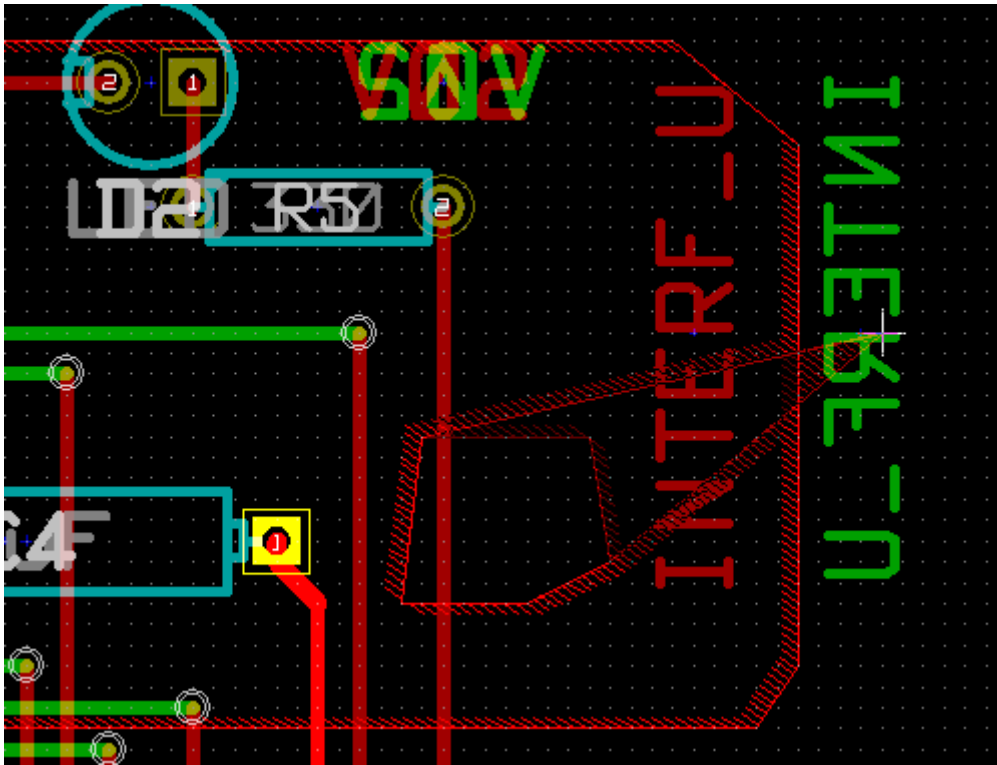
- Moving a corner or an edge.
- Deleting or adding a corner.
- Adding a similar zone, or a cutout area.

If polygons are overlapping they will be combined.

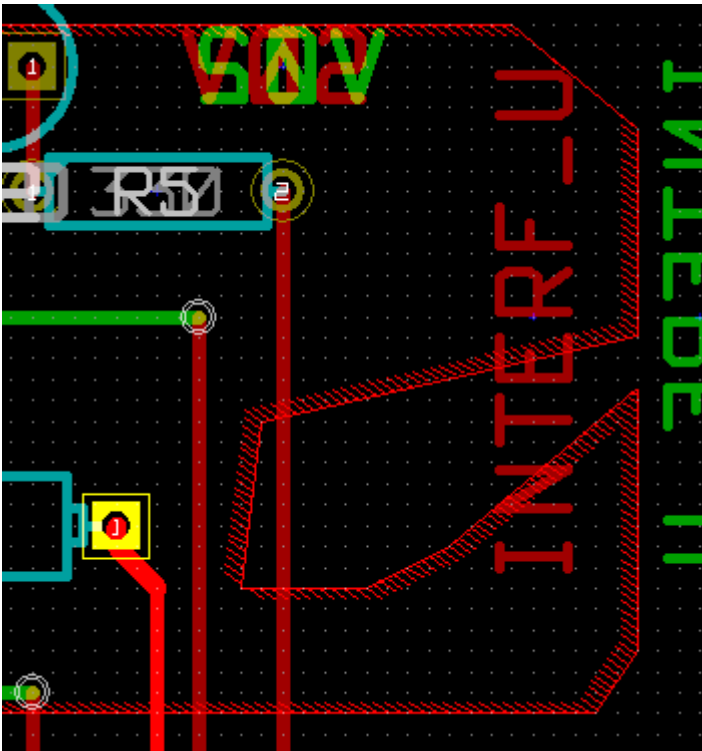


To do that, right click on a corner or on an edge, en select the proper command.

Here is a corner (from a cutout) that has been moved.



Here is the final result:



Polygons are combined. Adding a similar zone:


	<p>Adding the zone</p>
	<p>Final result</p>

Editing zone: parameters

When right clicking on an outline, and using 'Edit Zone Params' the Zone params Dialog box will open. Initial parameters can be imputed . If the zone is already filled, refilling it will be necessary.

Final zone filling

When the board is finished, one must fill or refill all zones. To do that:

- Activate the tool zones via the button .
- Right click to display the pop-up menu.

- Use Fill or Refill All Zones .

Warning, calculations can take some time, if the filling grid is small.

Change zones net names


After editing a schematic, you can change the name of any net. For instance VCC can be changed to +5V.

When a global DRC control is made Pcbnew checks if the zone net name exists, and displays an error if not.

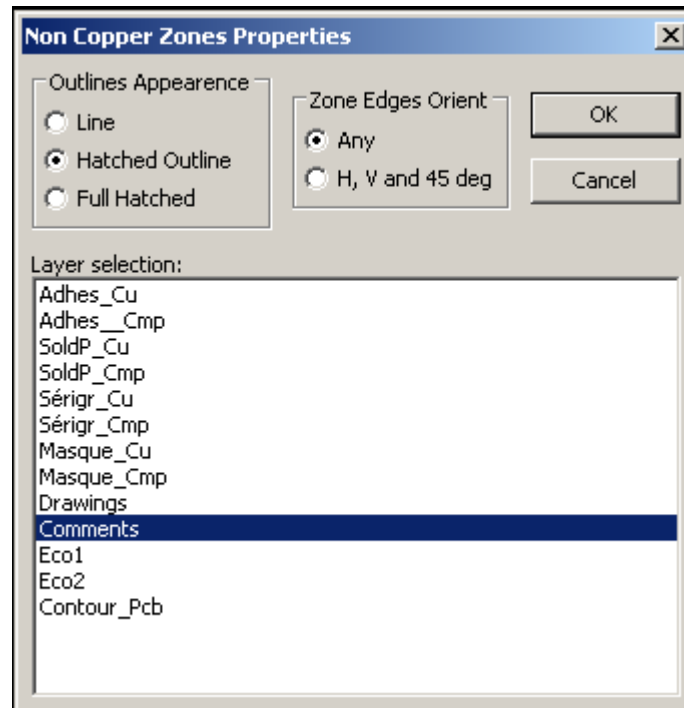
A "by hand" parameter zone edition will be necessary to change the old name to the new one.

Creating zones on technical layers

Creating zone limits

This is done using the button . The active layer must be a technical layer.

When clicking to start the zone outline, this dialog box is opened.



Select the technical layer to place the zone and draw the zone outline like explained previously for copper layers.

Notes:

- For editing outlines use the same way as for copper zones.
- In necessary, cutout areas can be added.

10 - Files for circuit fabrication

Table of Contents

<i>10 - Files for circuit fabrication.....</i>	<i>1</i>
<i> Final preparations.....</i>	<i>1</i>
<i> Final DRC test.....</i>	<i>2</i>
<i> Setting coordinates origin.....</i>	<i>3</i>
<i> Generating files for photo-tracing.....</i>	<i>3</i>
<i> GERBER format.....</i>	<i>5</i>
<i> HPGL format.....</i>	<i>6</i>
<i> POSTSCRIPT format.....</i>	<i>6</i>
<i> Plot options.....</i>	<i>6</i>
<i> Global clearance settings for the solder stop and the solder paste mask.....</i>	<i>6</i>
<i> Solder mask clearance.....</i>	<i>7</i>
<i> Solder paste clearance.....</i>	<i>7</i>
<i> Generating drill files.....</i>	<i>7</i>
<i> Generating cabling documentation.....</i>	<i>9</i>
<i> Generation of files for automatic component insertion.....</i>	<i>9</i>
<i> Advanced tracing options.....</i>	<i>9</i>

Let us see now what are the step necessary for the creations of the necessary files for the production of your printed circuit board.

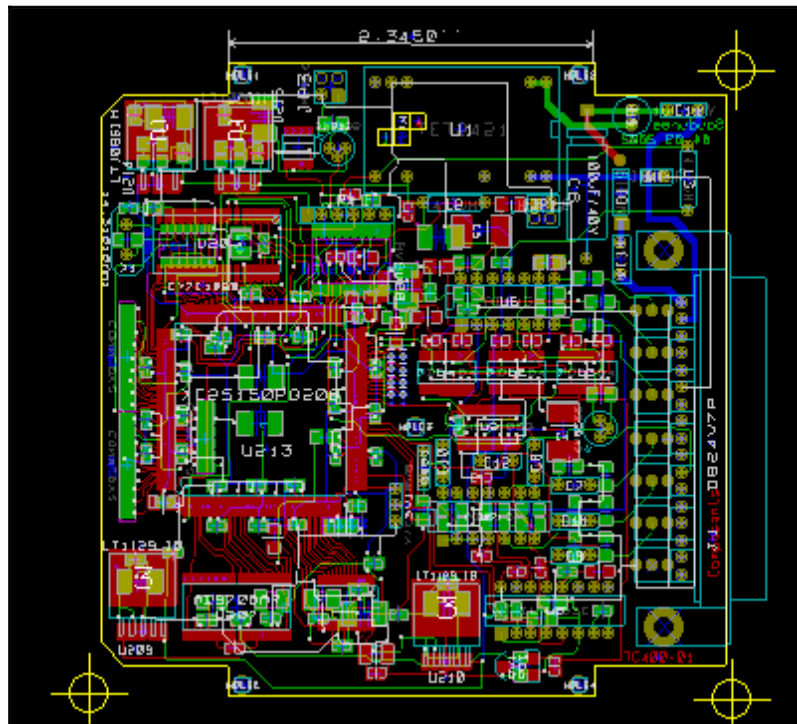
All files generated by Kicad are placed in the working directory which is the same directory that contains the file xxxxxx.brd for the printed circuit board.

Final preparations

The generation of the necessary files for the production of your printed circuit board includes the following preparatory steps.

- Mark any layer (e.g., 'top or front' and 'bottom or back') with the project name by placing appropriate text upon each of the layers.
- All text on copper layers (sometimes called 'solder' or 'bottom') must be mirrored.
- Create any ground planes, modifying traces as required to ensure they are contiguous.
- Place alignment crosshairs and possibly the dimensions of the board outline (these are usually placed on one of the general purpose layers).

Here is an example showing all these elements, except the ground planes, which have been omitted for better visibility.




A color key for the 4 copper layers has also been included:

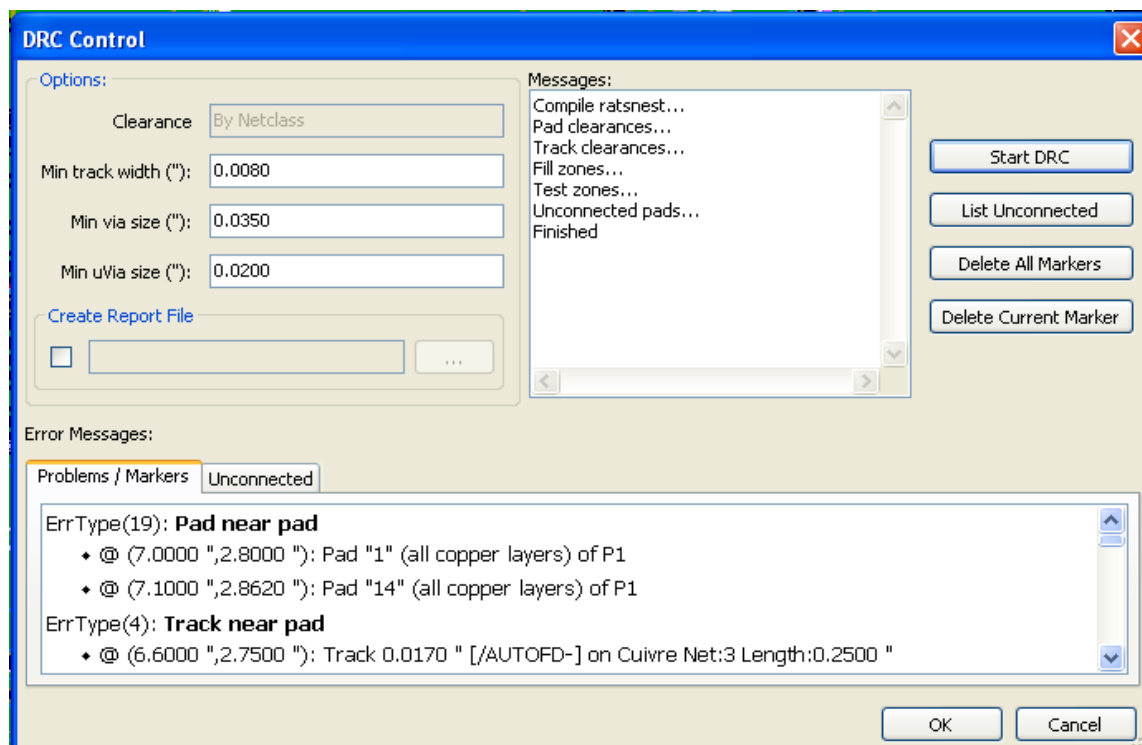


Final DRC test

Before generating the output files, a global DRC test is very strongly recommended.

Note:

Zones are filled or refilled when starting a DRC. Press the button  to launch the following DRC dialog.




Adjust parameters accordingly and then press

Start DRC

This final check will prevent any unpleasant surprise.

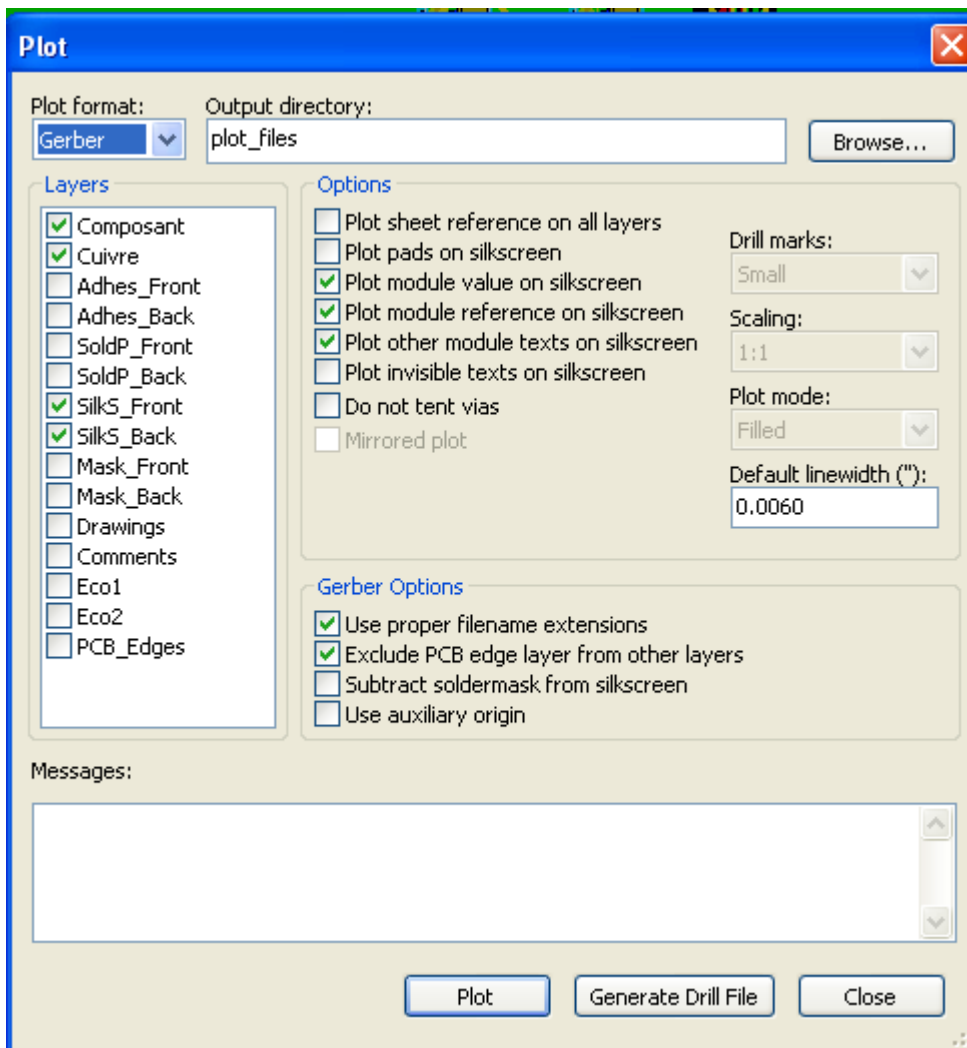
Setting coordinates origin

Set the coordinates origin for the photo plot and drill files, one must place the auxiliary axis on this origin. Activate the icon . Move the auxiliary axis to the chosen location by clicking for instance on this location.

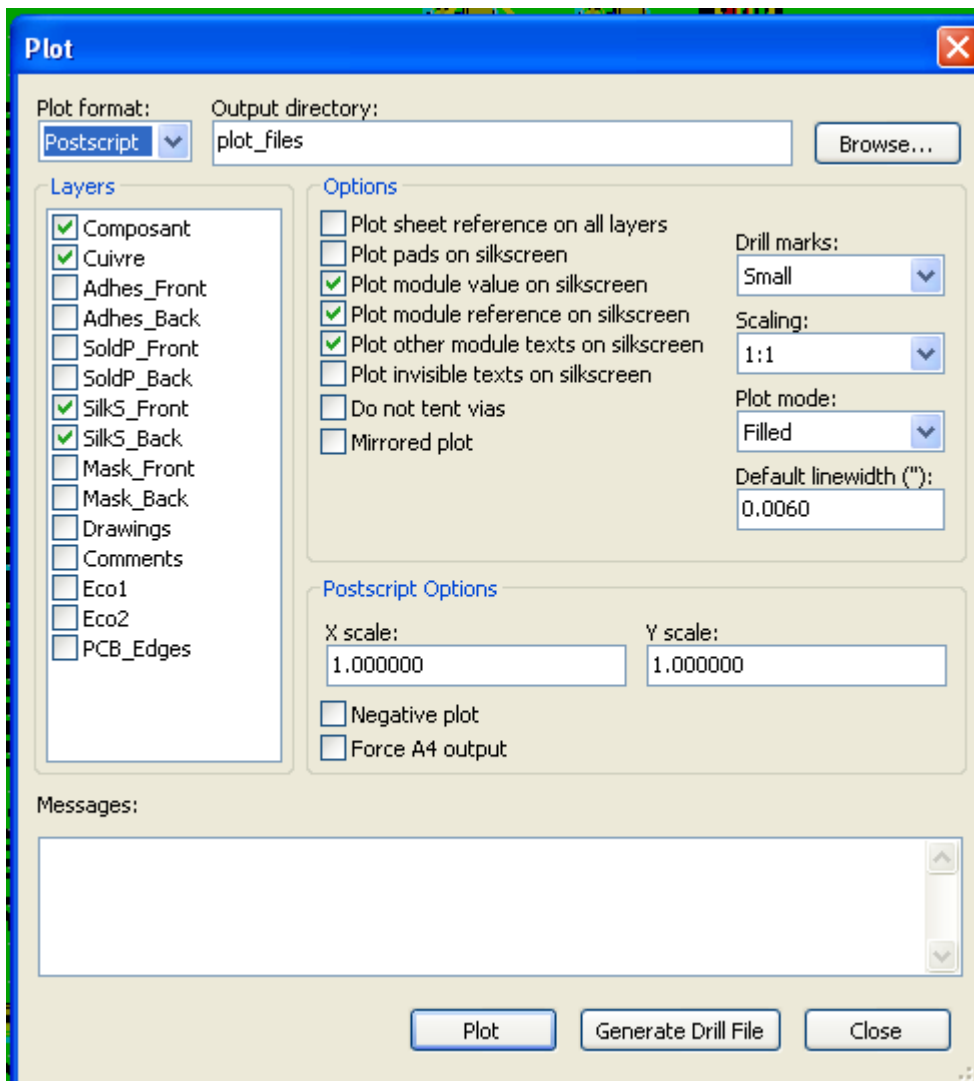


Generating files for photo-tracing

This is done via the Files/Plot menu option.



Usually, the files are in the GERBER format. Nevertheless, it is possible to produce output in both HPGL and POSTSCRIPT formats. When Postscript format is selected, this dialog will appear.



In these formats, a fine scale adjust can be used to compensate for the plotter accuracy and to have a true scale 1 for the output:

X scale:	Y scale:
1.000000	1.000000

GERBER format

For each layer, Pcbnew generates a separate file following the GERBER 274X standard, by default in 3.4 format (each coordinate in the file is represented by 7 digits, of which 3 are before the decimal point and 4 follow it; the units are inches). The tracing is always drawn to scale 1)

It is normally necessary to create files for all of the copper layers and, depending on the type of circuit, for the solder stop, solder mask, and silkscreen (component markings) layers. All of these files can be produced in one go, by selecting the appropriate check boxes.

For example, for a double-sided circuit with solder stop, silkscreen and solder mask (for CMS components), 8 files should be generated ('xxxx' represents the name of the .brd file).

- xxxx.copper.pho for the copper side.
- xxxx.cmp.pho for the component side.
- xxxx.silkscomp.pho for the component-side silkscreen markings.

- xxxx.silkscu.pho for the copper-side silkscreen markings.
- xxxx.soldpcmp.pho for the component-side solder mask.
- xxxx.soldpcu.pho for the copper-side solder mask.
- xxxx.maskcmp.pho for the component-side solder stop mask.
- xxxx.maskcu.pho for the copper-side solder stop mask.

GERBER files format:

The format used by Pcbnew is RS274X format 3.4, Imperial, Leading zero omitted, Abs format. This is very usual settings.

HPGL format

The standard extension for the output files is .plt.

The tracing can be done at user-selected scales and can be mirrored. The Print Drill Opt list offers the option of pads that are filled, drilled to the correct diameter or drilled with a small hole (to guide hand drilling).

If the Print Sheet Ref option is active, the sheet cartridge is traced.

POSTSCRIPT format

The standard extension for the output files is .ps in the case of postscript output.

As for HPGL output, the tracing can be at user-selected scales and can be mirrored.

If the Org = Centre option is active, the origin for the coordinates of the tracing table is assumed to be in the centre of the drawing.

If the Print Sheet Ref option is active, the sheet cartridge is traced.

Plot options

<input type="checkbox"/> Use Proper Gerber Extensions <input type="checkbox"/> Exclude pcb edge layer <input type="checkbox"/> Subtract Mask from Silk <input type="checkbox"/> Print sheet reference <input checked="" type="checkbox"/> Print pads on silkscreen <input checked="" type="checkbox"/> Print module value <input checked="" type="checkbox"/> Print module reference <input checked="" type="checkbox"/> Print other module texts <input type="checkbox"/> Force print invisible texts	<input type="checkbox"/> Use Proper Gerber Extensions <input type="checkbox"/> Exclude pcb edge layer <input type="checkbox"/> Subtract Mask from Silk <input type="checkbox"/> Print sheet reference <input checked="" type="checkbox"/> Print pads on silkscreen <input checked="" type="checkbox"/> Print module value <input checked="" type="checkbox"/> Print module reference <input checked="" type="checkbox"/> Print other module texts <input type="checkbox"/> Force print invisible texts
Gerber format	other formats

GERBER format specific options:

Use Proper Gerber Extensions	use .gbl .gtl instead of .pho for file name extensions.
Exclude Pcb Edge Layer	Do no draw items on Edge layer onto other layers
Subtract Mask from Silk	Remove all Silk from solder paste areas.

Global clearance settings for the solder stop and the solder paste mask

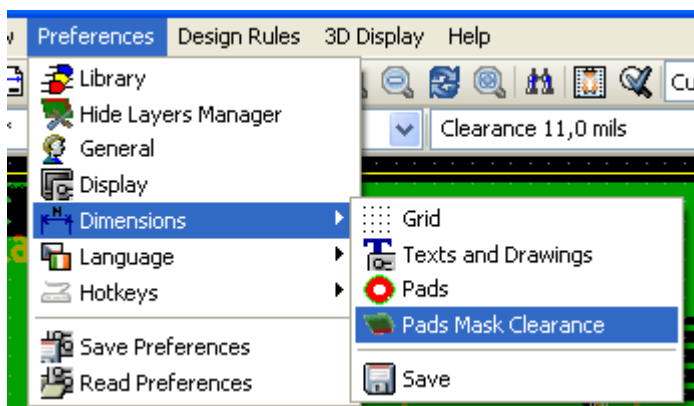
Masks clearances values can be set globally for the solder mask layers and the solder paste layers. These clearances can be set following these steps.

- At pads level.
- At footprint level.
- Or globally.

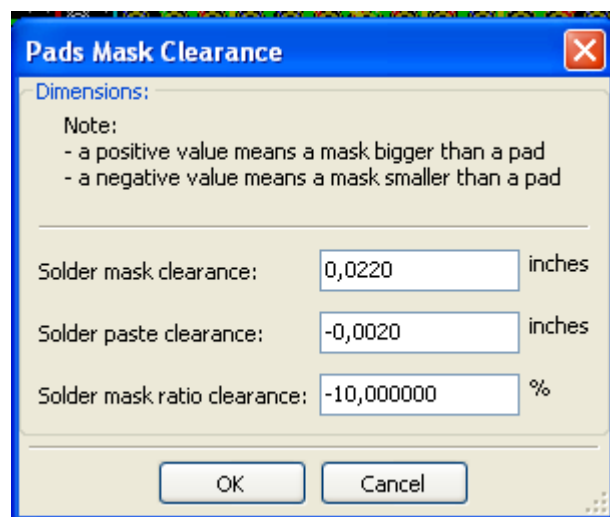
And Pcbnew uses by priority order.

- Pad values. If null:
- Footprint values. If null:
- Global values.

The menu option for this is available via the Preferences/Dimensions link.



The dialog box is the following.



Solder mask clearance

A value near to 10 mils is usually good. This value is positive because the mask usually is bigger than the pad.

Solder paste clearance

The final clearance is the sum of the solder paste clearance and a percentage of the pad size.

This value is negative because the mask usually is smaller than the pad.

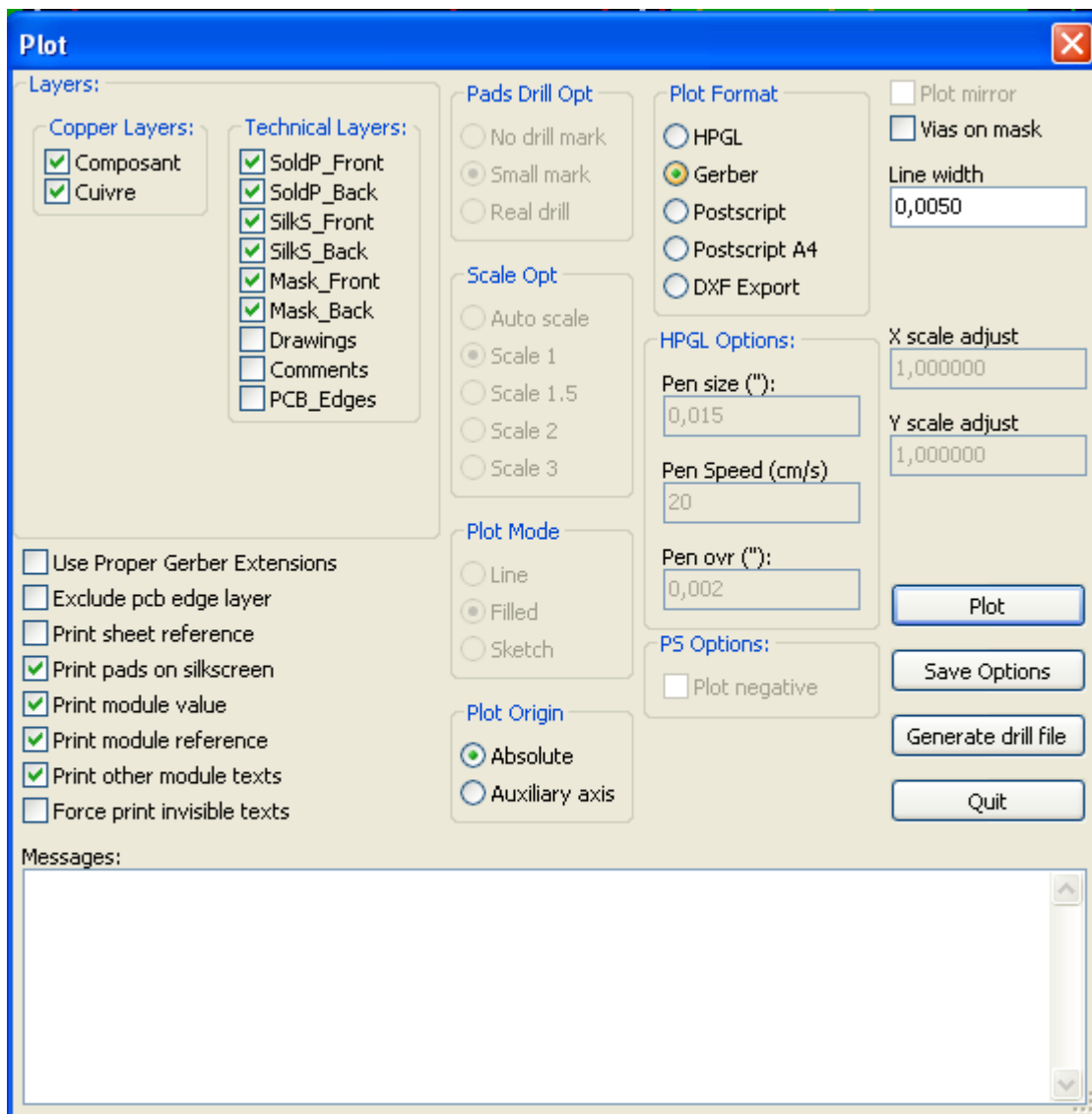
Generating drill files

The creation of a drill file xxxxxx.drl following the EXCELLON standard is always necessary.

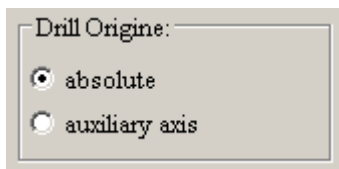
One can also produce an optional drill plan, which will be in HPGL (xxxxxx.plt) or POSTSCRIPT (xxxxxx.ps) format, and/or an optional drill report (a plain text file). However, this is only occasionally useful, as an additional check. The generation of these files is controlled via:


- the **Create Drill File** button.
- or the Files/Fabrication Outputs/Drill file menu selection.

The Drill tools dialog box will be the following.



For the HPGL tracing of the drill plan, it is possible to define the n°. and speed of the pen used. For setting the coordinate origin, the following dialog box is used.



- Absolute: absolute coordinate system is used.
- Auxiliary axis: coordinates are relative to the auxiliary axis (use the icon  (right toolbar) to set it.

Generating cabling documentation

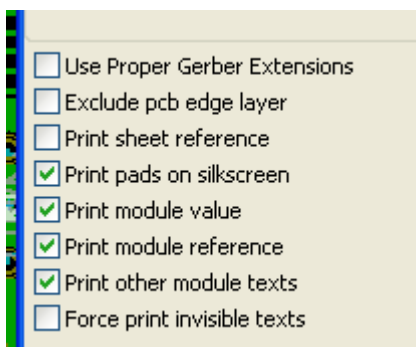
To produce cabling documentation files, the component and copper silkscreen layers can be traced. Usually, just the component-side silkscreen markings are sufficient for cabling a PCB. If the copper-side silkscreen is used, the text it contains should be mirrored in order to be readable.

Generation of files for automatic component insertion

This option is accessed via the Postprocess/Create Cmp file menu option. However, no file will be generated unless at least one module has the Normal+Insert attribute activated (see Editing Modules). One or two files will be produced, depending upon whether insertable components are present on one or both sides of the PCB. A dialogue box will display the names of the file(s) created.

Advanced tracing options

The options described below (part of the Files/Plot dialogue) allow for fine-grained control of the tracing process. They are particularly useful when printing the silkscreen markings for cabling documentation.



The available options are:

Use Proper Gerber Extensions	GERBER format specific. When creating files, use specific extensions for each file. If disabled the Gerber file extension is .pho
Exclude pcb edge layer	GERBER format specific. Do not plot graphic items on edge layer.

Print Sheet Ref	Trace sheet outline and the cartridge.
Print Pads on Silkscreen	Enables/disables printing of pad outlines on the silkscreen layers (if the pads have already been declared to appear on these layers). In fact useful for preventing any pads from being printed, in the disabled mode.
Print Module Value	Enables printing of VALUE text on the silkscreen.
Print Module Reference	Enables printing of the REFERENCE text on the silkscreen.
Print other module texts	Enables the printing of other text fields on the silkscreen.
Force Print Invisible Texts	Forces printing of fields (reference, value) declared as invisible. In combination with Print Module Reference and Print Module Value, this option enables production of documents for guiding cabling and repair. These options have proven necessary for circuits using components that are too small (CMS) to allow readable placement of two separate text fields.

11 - ModEdit - Managing Libraries

Table of Contents

<i>11 - ModEdit - Managing Libraries.....</i>	<i>1</i>
<i> Overview of ModEdit.....</i>	<i>1</i>
<i> ModEdit.....</i>	<i>2</i>
<i> ModEdit user interface.....</i>	<i>2</i>
<i> Top toolbar in Modedit.....</i>	<i>3</i>
<i> Creating a new module.....</i>	<i>4</i>
<i> Creating a new library.....</i>	<i>5</i>
<i> Saving a module in the active library.....</i>	<i>5</i>
<i> Transferring a module from one library to another.....</i>	<i>5</i>
<i> Saving all modules of your board in the active library.....</i>	<i>5</i>
<i> Documentation for library modules.....</i>	<i>5</i>
<i> Documenting libraries – recommended practice.....</i>	<i>6</i>

Overview of ModEdit

Pcbnew can simultaneously maintain several libraries. Thus, when a module is loaded, all libraries that appear in the library list are searched until the first instance of the module is found. In what follows, note that the active library is the library selected within the Module Editor (ModEdit), the program we be now described

ModEdit enables the creation and the editing of modules:

- Adding and removing pads.
- Changing pad properties (shape, layer) for individual pads or globally for all pads of a module.
- Editing graphic elements (lines, text).
- Editing information fields (value, reference, etc.).
- Editing the associated documentation (description, keywords).

ModEdit allows the maintenance of the active library as well by:

- Listing the modules in the active library.
- Deletion of a module from the active library.
- Saving a module to the active library.
- Saving all of the modules contained by a printed circuit.


It is also possible to create new libraries. A library is in fact made up of two files:

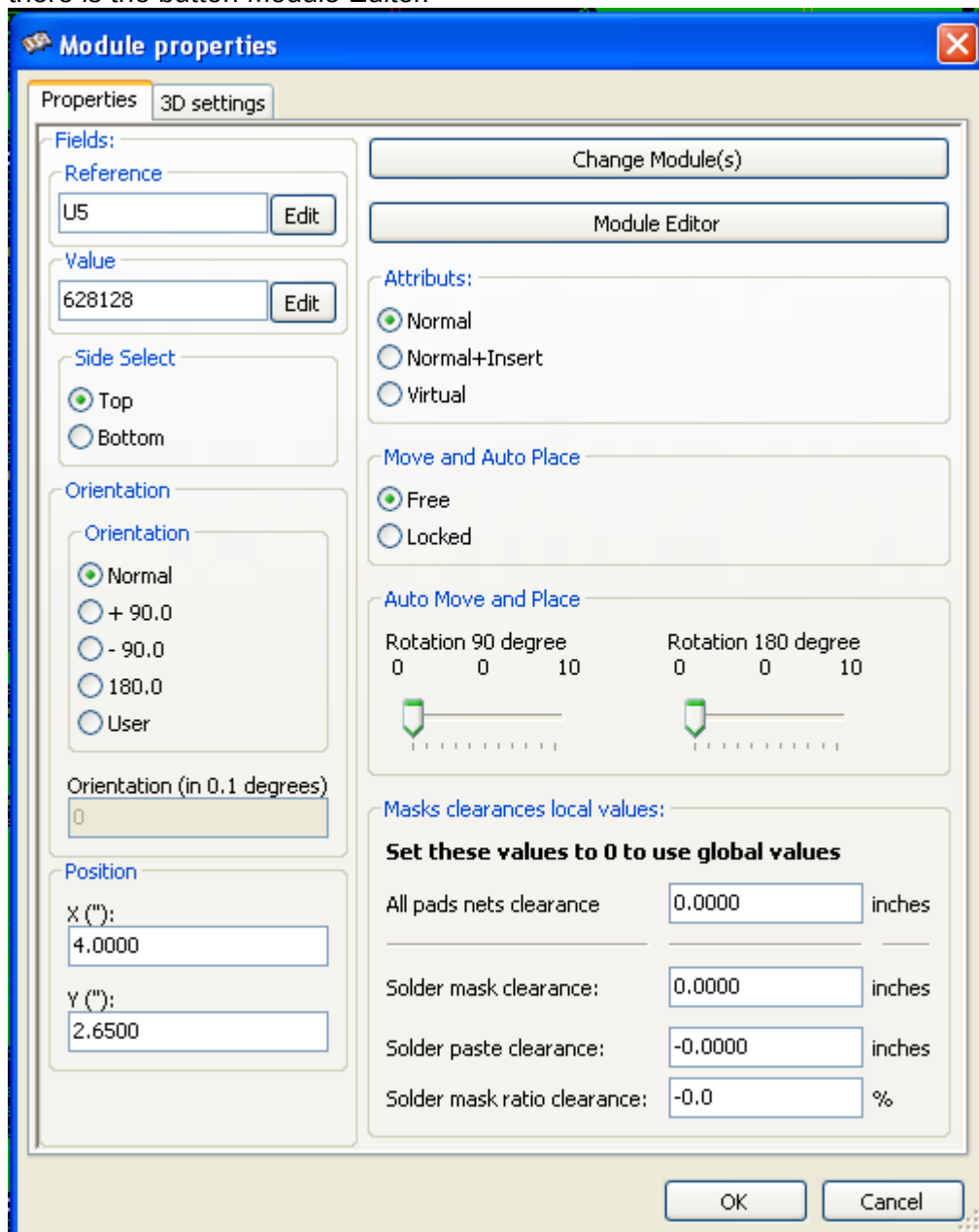
- The library itself (file extension .lib).
- The associated documentation (file extension .dcm).

The documentation file is systematically regenerated after the modification of the corresponding .lib file. In this way the file can easily be recovered if lost. The documentation is used to accelerate the access to the module documentation.

ModEdit

The Module Editor can be accessed in two different ways:

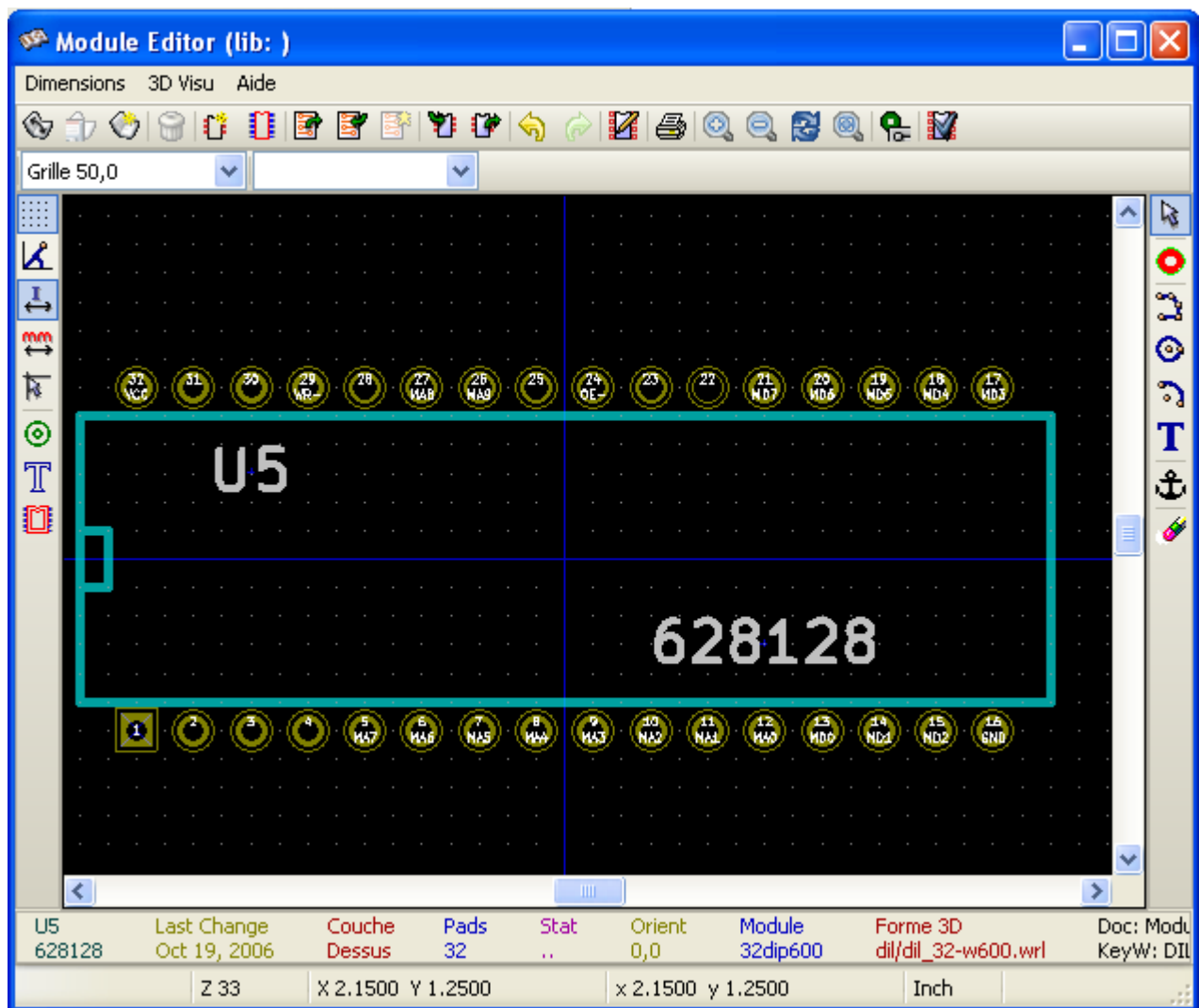
- Directly, via the icon  in the main toolbar of Pcbnew.
- In the edit dialog for the active module (see figure below: accessed via the context menu), there is the button Module Editor.



In this case, the active module of the board will be loaded automatically in ModEdit, enabling immediate editing or archiving.

ModEdit user interface

















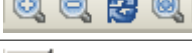


By calling ModEdit the following window will appear.




Top toolbar in Modedit



From this toolbar, the following functions are available:

	Select the active library.
	Save the current module to the active library.
	Create a new library and save the current module in it.
	Access a dialog for deleting a module from the active library.
	Create a new module.
	Load a module from the active library.
	Load (import) a module from the printed circuit board.
	Export the current module to the printed circuit board. If the module was previously imported from the current board, it will replace the corresponding module on the board (i.e., respecting position and orientation). If the module was loaded from a library, it will be copied on to the printed circuit board at position 0.
	Import a module from a file created by the Export command ()
	Export a module. This command is essentially identical to that for creating a library, the only difference being that  creates a library in the user directory, while  creates a library in the standard library directory (usually <i>kicad/modules</i>).
	Undo - Redo
	Not used.
	Call the print dialog.
	Standard zoom commands.
	Call the pad editor.
	Not used.




Creating a new module

The button  allows the creation of a new module. You will be asked for the name by which the module will be identified in the library. This text also serves as the module reference, which will be replaced by the final reference on the printed circuit board (U1, IC3, etc.)



It will be necessary to add the following to the new module:

- Outlines (and possibly text).
- The pads.
- A value (place-holding text that will subsequently be replaced by the true value).

When a new module is similar to an existing module in a library or board, this alternative and often quicker method should be used:

1. Load the similar module via the buttons ,  and .
2. Modify the reference field to the name of the new module.
3. Edit and save the new module.

Creating a new library


The creation of a new library is done via the button , in this case the file is created by default in the library directory or via the button , in which case the file is created by default in your working directory.

A file-choosing dialog allows the name of the library to be specified and its directory to be changed. In both cases, the library will contain the module being edited.

WARNING







If an old library of the same name exists, it will be overwritten without warning.

Saving a module in the active library

The action of saving a module (thereby modifying the file of the active library) is performed using this button . If a module of the same name already exists, it will be replaced. Since you will depend upon the accuracy of the library modules, it is worth double-checking the module before saving.

It is recommended to edit either the reference or value field text to the name of the module as identified in the library.

Transferring a module from one library to another

Select the source library via the button . Load the module via the button . Select the destination library via the button . Save the module via the button . You may also wish to delete the source module. Reselect the source library then delete the old module via the button  and the button .

Saving all modules of your board in the active library

It is possible to copy all of the modules of a given board design to the active library. These modules will keep their current library names.

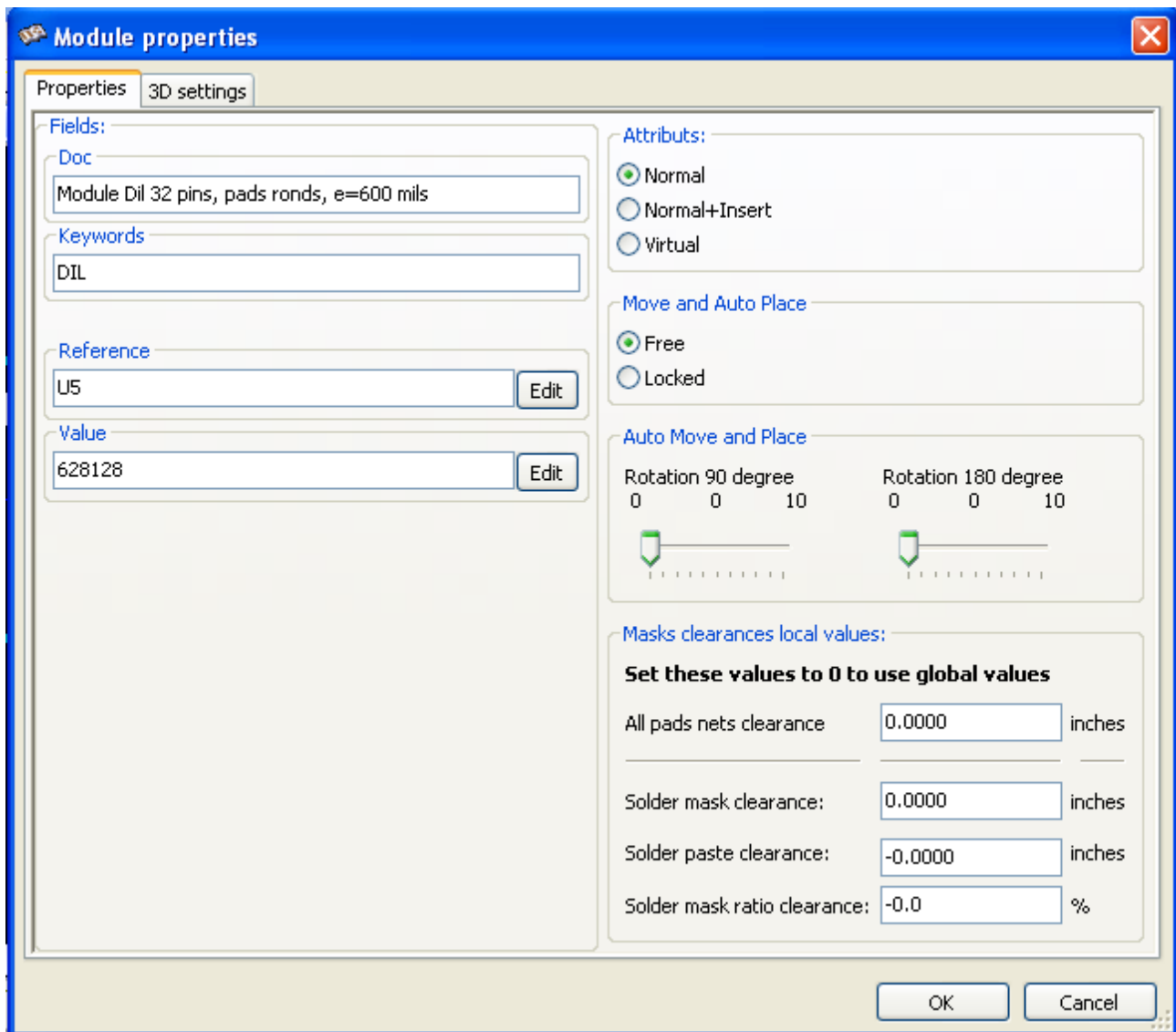
This command has two uses.

- To create an archive or complete a library with the modules from a board, in the event of the loss of a library.
- More importantly, it facilitates library maintenance by enabling the production of documentation for the library, as below.

Documentation for library modules

It is strongly recommended to document the modules you create, in order to enable rapid and error-free searching.

For example, who is able to remember all of the multiple pin-out variants of a TO92 package? The Module Properties dialog offers a simple solution to this problem.




This dialog accepts:

- A one-line comment/description.
- Multiple keywords.

The description is displayed with the component list in Cypcb and, in Pcbnew, it is used in the module selection dialogs.

The keywords enable searches to be restricted to those modules corresponding to particular keywords.

When directly loading a module (the icon  of the right-hand Pcbnew toolbar), keywords may be entered in the dialog box. Thus, entering the text "=CONN" will cause the display of the list of modules whose keyword lists contain the word CONN.

Documenting libraries – recommended practice

It is recommended to create libraries indirectly, by creating one or more auxiliary circuit boards that constitute the source of (part of) the library, as follows:

- Create a circuit board in A4 format, in order to be able to print easily to scale (scale = 1).
- Create the modules that the library will contain on this circuit board.

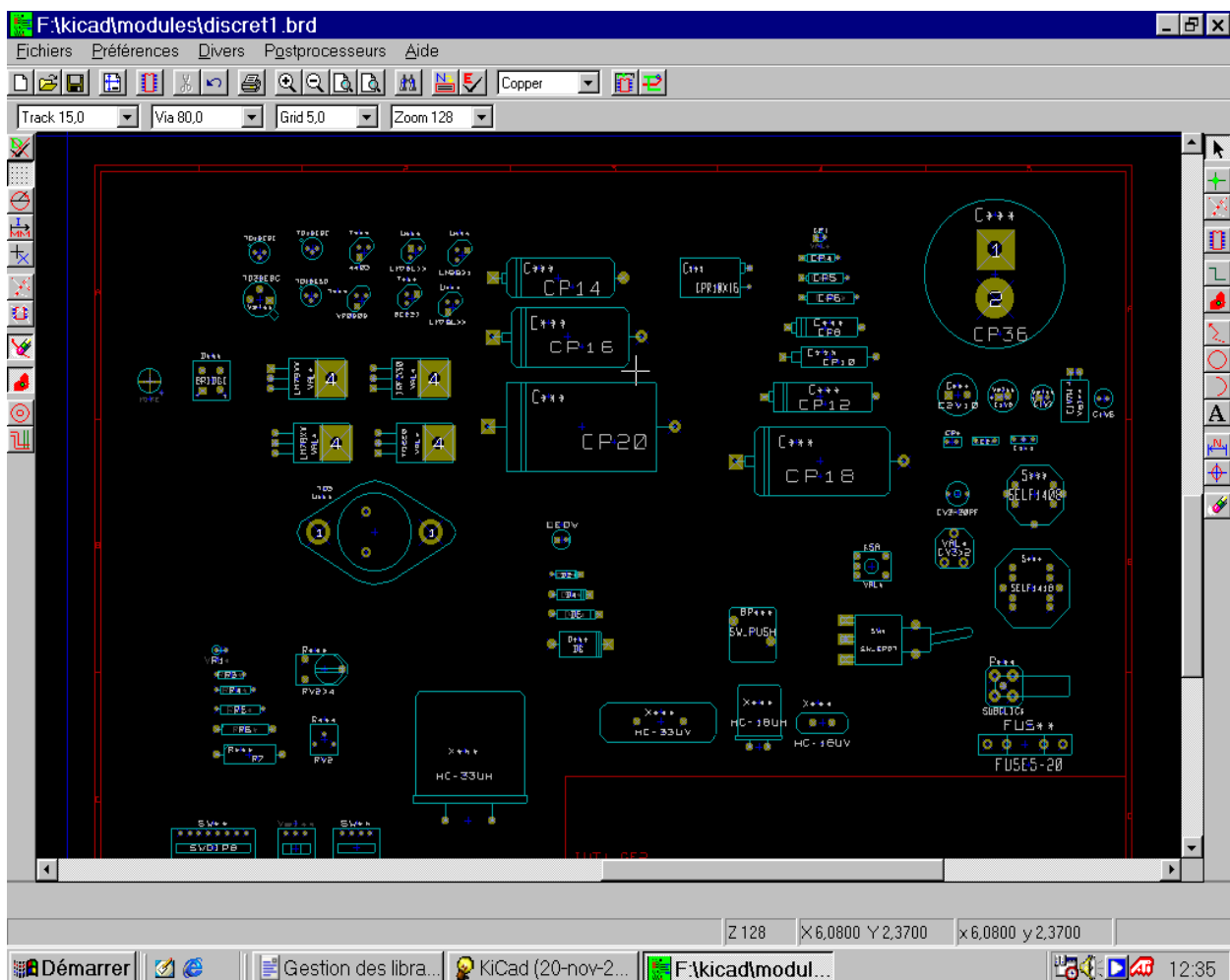
- The library itself will be created with the File/Archive footprints/Create footprint archive command.



The "true source" of the library will thus be the auxiliary circuit board, and it is on this circuit that any subsequent alterations of modules will be made. Naturally, several circuit boards can be saved in the same library.

It is generally a good idea to make different libraries for different kinds of components (connectors, discretés,...), since Pcbnew is able to search many libraries when loading modules.

Here is an example of such a library source:



This technique has several advantages:

1. The circuit can be printed to scale and serve as documentation for the library with no further effort.
2. Future changes of Pcbnew may require regeneration of the libraries, something that can be done very quickly if circuit-board sources of this type have been used. This is important, because the circuit board file formats are guaranteed to remain compatible during future development, but this is not the case for the library file format.

12 - ModEdit - Creating and Editing Modules

Table of Contents

12 - ModEdit - Creating and Editing Modules.....	1
ModEdit overview.....	1
Module elements.....	2
Pads.....	2
Contours.....	2
Fields.....	2
Starting ModEdit and selecting a module to edit.....	2
Module Editor Toolbars.....	2
Edit toolbar.....	3
Display toolbar.....	4
Context Menus.....	4
Module properties dialog.....	6
Creating a new module.....	6
Adding and editing pads.....	7
Adding pads.....	7
Setting pad properties.....	7
Rectangular pads	8
Rotate pads.....	8
Not plated through hole pads.....	8
Pads not on copper layers.....	8
Offset Parameter.....	8
Delta Parameter (trapezoidal pads).....	9
Setting clearance for solder mask and solder paste mask layers.....	9
Remarks.....	9
Solder paste mask parameters.....	9
Footprint level settings.....	10
Fields Properties.....	10
Automatic placement of a module.....	11
Attributes.....	11
Documenting modules in a library.....	11
3-dimensional visualisation.....	12
Saving a module into the active library.....	13
Saving a module to the board.....	14

ModEdit overview

ModEdit is used for editing and creating PCB modules. This includes:

- Adding and removing pads.
- Changing pad properties (shape, layer), for individual pads or for all pads in a module.
- Adding and editing graphic elements (contours, text).

- Editing fields (value, reference, etc.).
- Editing the associated documentation (description, keywords).

Module elements

A module is the physical representation (footprint) of the part to be inserted in the PCB and it must be linked to the relative component in your schematic. Each module includes three different elements:

- The pads.
- Graphical contours and text.
- Fields.

In addition, a number of other parameters must be correctly defined if the auto-placement function will be used. The same holds for the generation of auto-insertion files.

Pads

Two pad properties are important:

- Geometry (shape, layers, drill holes).
- The pad number, which is constituted by up to four alphanumeric characters. Thus, the following are all valid pad numbers: 1, 45 and 9999, but also AA56 and ANOD. The pad number must be identical to that of the corresponding pin number in the schematic, because it defines the matching pin and pad numbers that Pcbnew links pins and pads with.

Contours


Graphical contours are used to draw the physical shape of the module. Several different types of contour are available: lines, circles, arcs, and text. Contours have no electrical significance, they are simply graphical aids.

Fields

These are text elements associated with a module. Two are obligatory and always present: the **reference field** and the **value field**. These are automatically read and updated by Pcbnew when a netlist is read during the loading of modules into your board. The reference is replaced by the appropriate schematic reference (U1, IC3, etc.). The value is replaced by the value of the corresponding part in the schematic (47K, 74LS02, etc.). Other fields can be added and these will behave like graphical text.

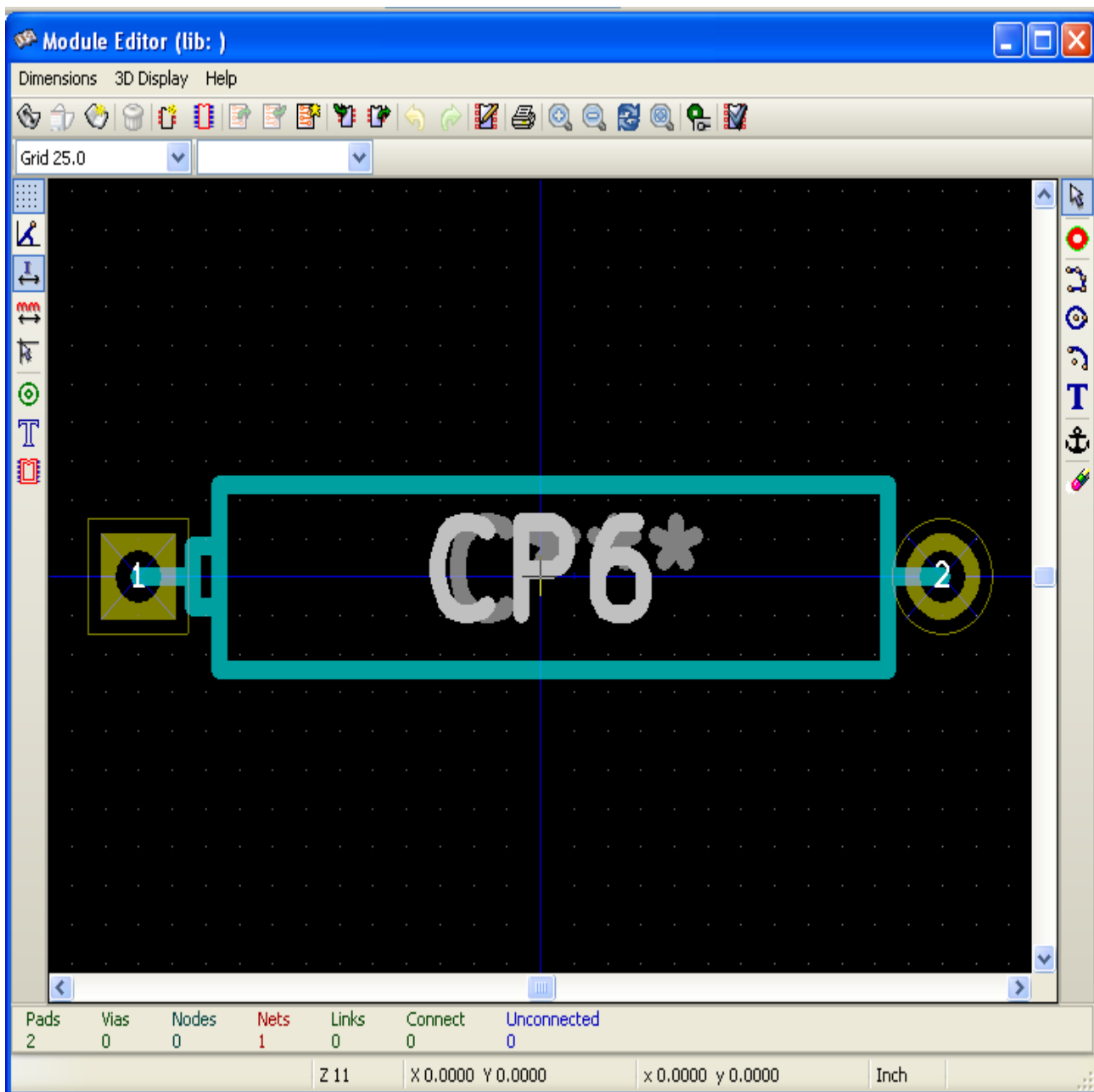
Starting ModEdit and selecting a module to edit

ModEdit can be started in two ways:

- Directly via the  icon from the main toolbar of Pcbnew. This allows creation or modification of a module in the library.
- Double-clicking a module will launch the 'Module Properties' menu, which offers a 'Go to Module Editor' button. If this option is used, the module from the board will be loaded into the editor, for modification or for saving.

Module Editor Toolbars









Calling ModEdit will launch a new window that looks like this.




Edit toolbar

	<p>This toolbar contains tools for:</p> <ul style="list-style-type: none"> • Placing pads. • Adding graphic elements (contours, text). • Positioning the anchor. • Deleting elements.
--	---








The specific functions are the following:

	Add a pad.
	Draw line segments and polygons.
	Draw circles.
	Draw circular arcs.
	Add graphical text (fields are not managed by this tool).
	Position the module anchor.
	Delete elements.
	Grid origin. (grid offset). Useful for placement of pads. The grid origin can be put on a given location (the first pad to place), and after the grid size can be set to the pad pitch. Placing pads is therefore very easy

Display toolbar

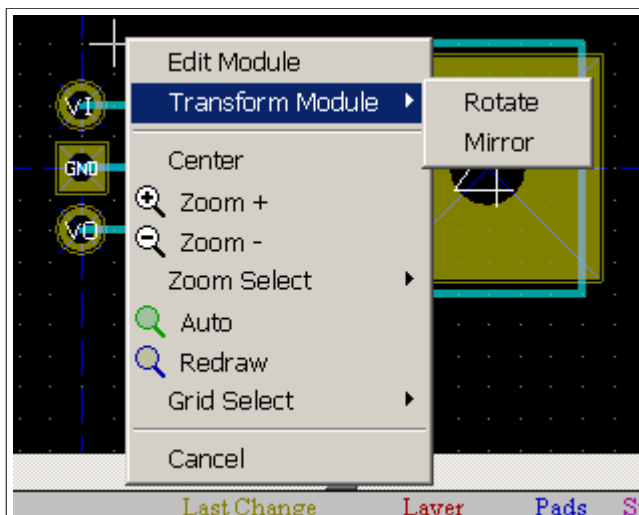
	These tools manage the display options in ModEdit
--	---

These options are active when the button is pressed:

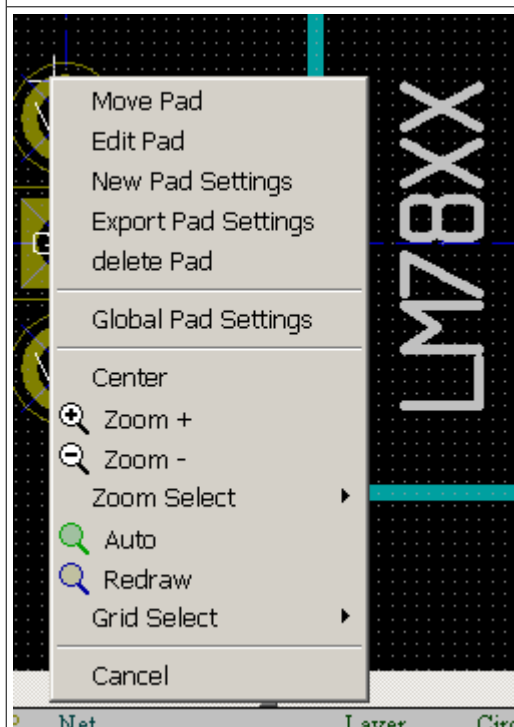
	Display the grid.
	Display polar coordinates.
	Use units of mm (update: now mm/inches are toggled via two buttons).
	Crosshair cursor.
	Display pad in outline mode.
	Display text in outline mode.
	Display contours in outline mode.

Context Menus

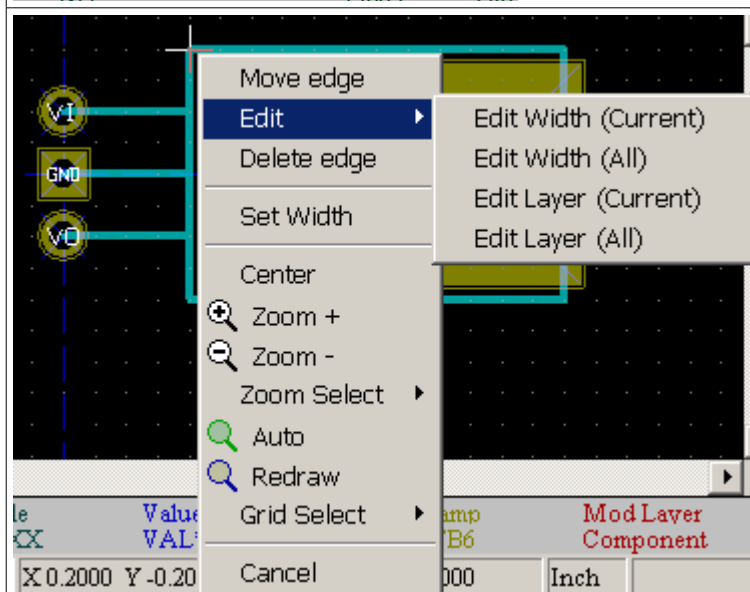
The right mouse button calls up menus that depend upon the element beneath the cursor.



The context menu for editing module parameters.



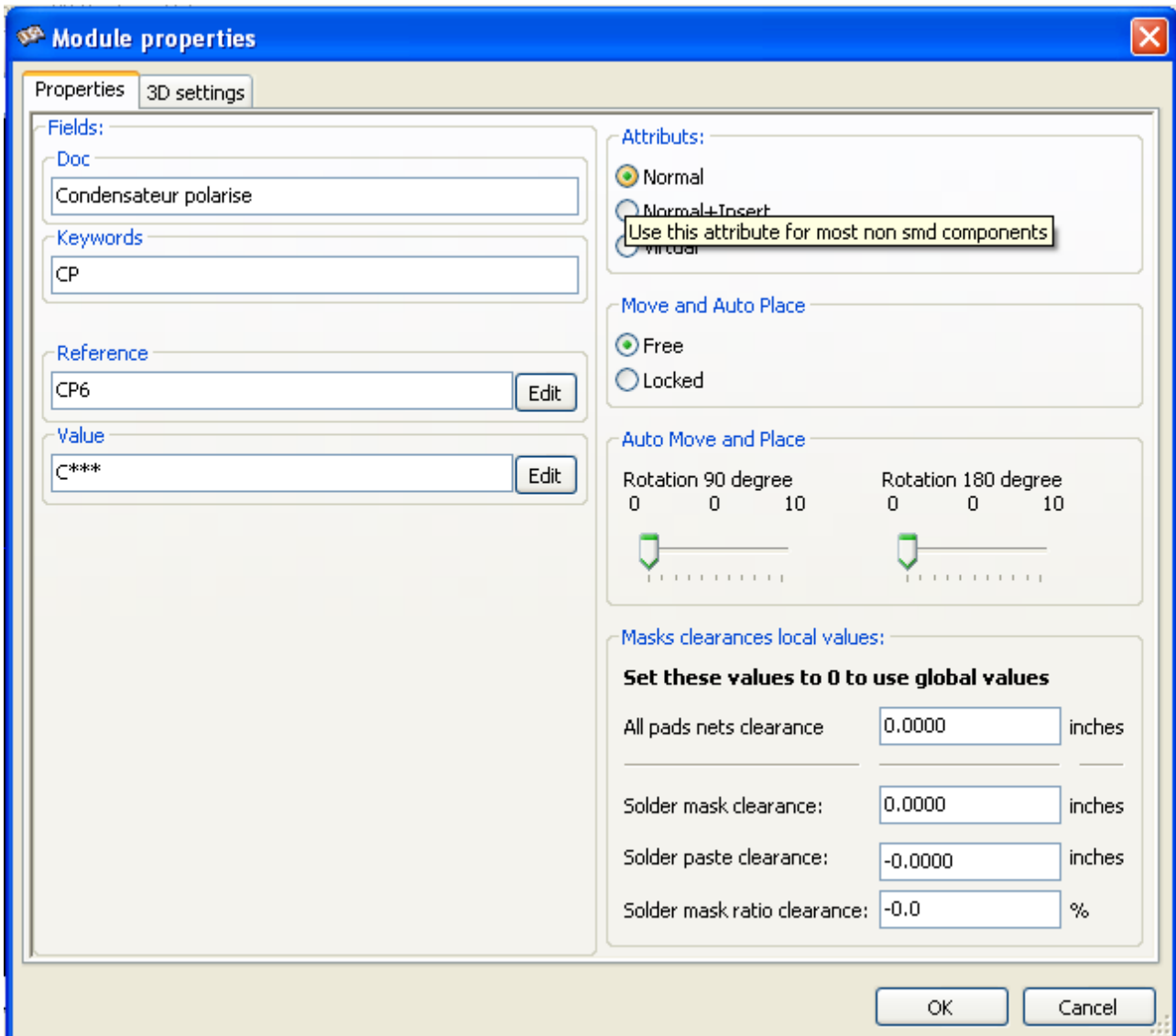
The context menu for editing pads.



The context menu for editing graphic elements.


Module properties dialog

This dialog can be launched when the cursor is over a module by clicking on the right mouse button and then selecting 'Edit Module'.



The dialog can be used to define the main module parameters.

Creating a new module

A new module can be created via the button . The name of the new module will be requested. This will be the name by which the module will be identified in the library.




This text also serves as the module reference, which is ultimately replaced by the real reference (U1, IC3...).

The new module will require:

4. Contours (and possibly graphic text).
5. Pads.
6. A value (hidden text that is replaced by the true value when used).

Alternative method:


When a new module is similar to an existing module in a library or a circuit board, an alternative and quicker method of creating the new module is as follows:

- Load the similar module (, , or )
- Modify the reference field in order to generate a new identifier (name).
- Edit and save the new module.

Adding and editing pads

Once a module has been created, pads can be added, deleted or modified. Modification of pads can be local, affecting only the pad under the cursor, or global, affecting all pads of the module.


Adding pads

Select the  icon from the right hand toolbar. Pads can be added by clicking in the desired position with the left mouse button. Pad properties are predefined in the pad properties menu.

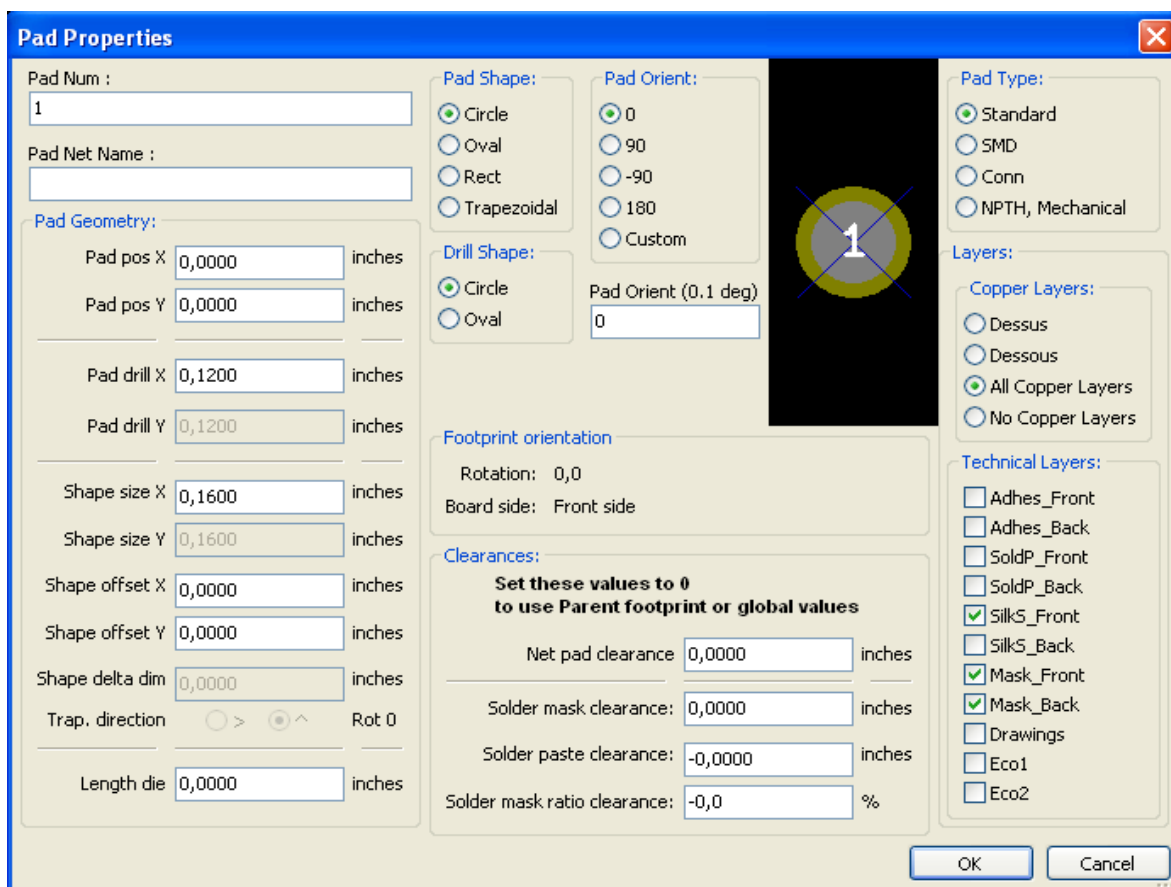
Do not forget to enter the pad number.

Setting pad properties

This can be done in three different ways:

- Selecting the  icon from the horizontal toolbar.
- Clicking on an existing pad and selecting 'Edit Pad'. The pad's settings can then be edited.
- Clicking on an existing pad and selecting 'Export Pad Settings'. In this case, the geometrical properties of the selected pad will become the default pad properties.

In the first two cases, the following dialog window will be displayed:



The **Pad Properties** dialog window is used to configure the properties of a specific pad. It includes the following sections:

- Pad Num :** A text field containing the value '1'.
- Pad Net Name :** An empty text field.
- Pad Geometry:**
 - Pad pos X :** 0,0000 inches
 - Pad pos Y :** 0,0000 inches
 - Pad drill X :** 0,1200 inches
 - Pad drill Y :** 0,1200 inches
 - Shape size X :** 0,1600 inches
 - Shape size Y :** 0,1600 inches
 - Shape offset X :** 0,0000 inches
 - Shape offset Y :** 0,0000 inches
 - Shape delta dim :** 0,0000 inches
 - Trap. direction :** Radio buttons for >, •, and ^, with 'Rot 0' selected.
 - Length die :** 0,0000 inches
- Pad Shape:**
 - ☒ Circle
 - ☐ Oval
 - ☐ Rect
 - ☐ Trapezoidal
- Drill Shape:**
 - ☒ Circle
 - ☐ Oval
- Pad Orient:**
 - ☒ 0
 - ☐ 90
 - ☐ -90
 - ☐ 180
 - ☐ Custom
- Pad Orient (0.1 deg):** A text field containing the value '0'.
- Footprint orientation:**
 - Rotation:** 0,0
 - Board side:** Front side
- Clearances:**

Set these values to 0 to use Parent footprint or global values

 - Net pad clearance:** 0,0000 inches
 - Solder mask clearance:** 0,0000 inches
 - Solder paste clearance:** -0,0000 inches
 - Solder mask ratio clearance:** -0,0 %
- Pad Type:**
 - ☒ Standard
 - ☐ SMD
 - ☐ Conn
 - ☐ NPTH, Mechanical
- Layers:**
 - Copper Layers:**
 - ☐ Dessus
 - ☐ Dessous
 - ☒ All Copper Layers
 - ☐ No Copper Layers
 - Technical Layers:**
 - ☐ Adhes_Front
 - ☐ Adhes_Back
 - ☐ SoldP_Front
 - ☐ SoldP_Back
 - ☒ SilkS_Front
 - ☐ SilkS_Back
 - ☒ Mask_Front
 - ☒ Mask_Back
 - ☐ Drawings
 - ☐ Eco1
 - ☐ Eco2

The dialog also features a central preview window showing a circular pad with the number '1' and a blue crosshair. At the bottom right are **OK** and **Cancel** buttons.

Care should be taken to define correctly the layers to which the pad will belong. In particular, although copper layers are easy to define, the management of non-copper layers (solder mask, solder pads...) is equally important for circuit manufacture and documentation.

The Pad Type selector triggers an automatic selection of layers that is generally sufficient.

Rectangular pads

For SMD modules of the VQFP/PQFP type which have rectangular pads on all four sides (both horizontal and vertical) it is recommended to use just one shape (for example, a horizontal rectangle) and to place it with different orientations (0 for horizontal and 90 degrees for vertical). Global resizing of pads can then be done in a single operation.

Rotate pads

Rotations of -90 or -180 are only required for trapezoidal pads used in microwave modules.

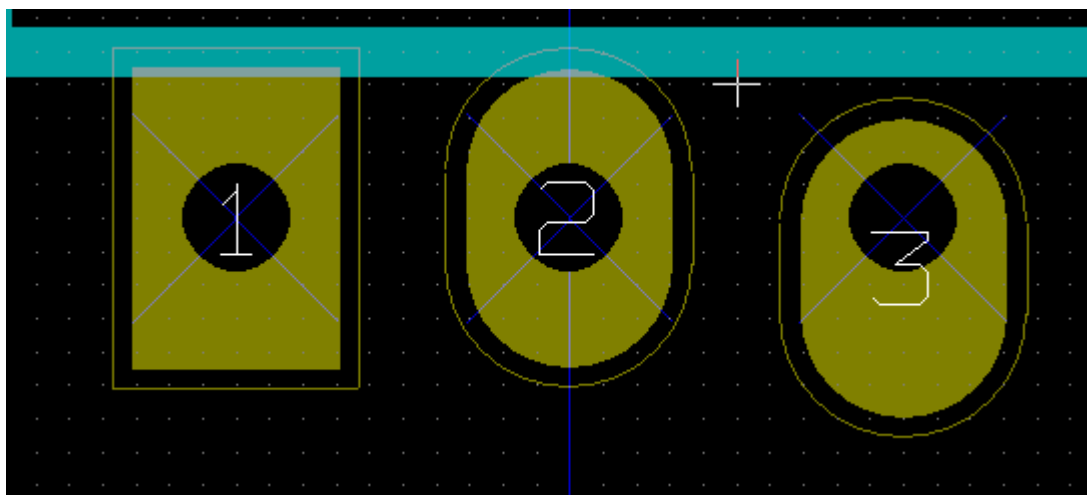
Not plated through hole pads

- Pads can be defined as Not Plated Through Hole pads (NPTH pads).
- These pads must be defined on one or all copper layers (obviously, the hole exists on all copper layers).
- This requirement allows you to define specific clearance parameters (for instance clearance for a screw).
- When the pad hole size is the same as the pad size, for a round or oval pad, this pad is NOT plotted on copper layers in GERBER files.
- These pads are used for mechanical purposes, therefore no pad name or net name is allowed. A connection to a net is not possible.

Pads not on copper layers

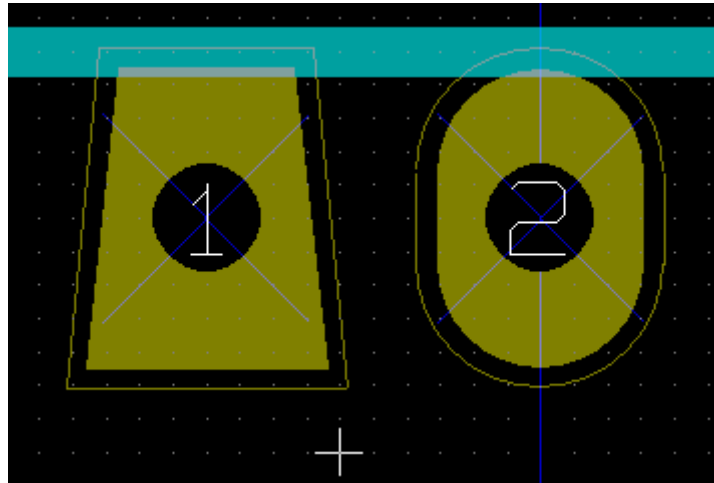
These are unusual pads. This option can be used to create fiducials or masks on technical layers.

Offset Parameter



Pad 3 has an offset Y = 15 mils.

Delta Parameter (trapezoidal pads)



Pad 1 has its parameter Delta X = 10 mils

Setting clearance for solder mask and solder paste mask layers

Setting clearance can be made at 3 levels:

3. Global level.
4. Footprint level.
5. Pad level.

Pcbnew uses to calculate clearance:

- Pad settings.
If null
- Footprint settings.
If null
- Global settings.

Remarks

The solder mask pad shape is usually bigger than the pad itself. So the clearance value is positive. The solder paste mask pad shape is usually smaller than the pad itself. So the clearance value is negative.

Solder paste mask parameters

For solder paste mask there are two parameters:

- A fixed value.
- A percentage of the pad size.

The real value is the sum of these 2 values.

Footprint level settings

Masks clearances local values:

Set these values to 0 to use global values

All pads nets clearance	0.0000	inches
Solder mask clearance:	0.0000	inches
Solder paste clearance:	-0.0000	inches
Solder mask ratio clearance:	-0.0	%

Pad level settings:

Clearances:

Set these values to 0 to use Parent footprint or global values

Net pad clearance	0.0000	inches
Solder mask clearance:	0.0000	inches
Solder paste clearance:	-0.0000	inches
Solder mask ratio clearance:	-0.0	%

Fields Properties

There are at least two fields: reference and value.

Their parameters (attribute, size, width) must be updated. You can access the dialog box from the pop-up menu, by double clicking on the champ, or by the footprint properties dialog box.

Footprint text properties [X]

Module CP6 (C***) orient 0.0

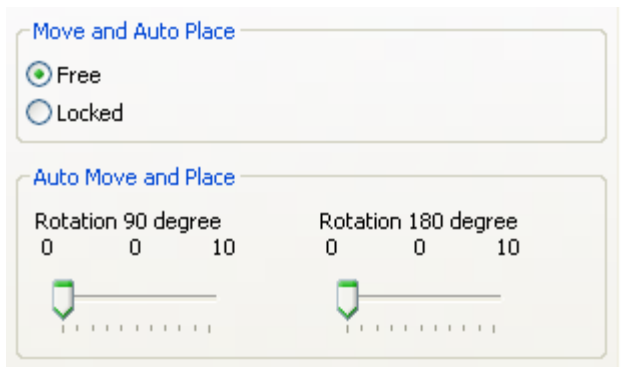
Reference:
CP6

Size X ("): 0.0600	Thickness ("): 0.0120	Style <input checked="" type="radio"/> Normal <input type="radio"/> Italic
Size Y ("): 0.0600	Orientation <input checked="" type="radio"/> horizontal <input type="radio"/> vertical	
Offset X ("): 0.0000	Display <input checked="" type="radio"/> Visible <input type="radio"/> Invisible	
Offset Y ("): 0.0000		

OK Cancel

Automatic placement of a module

If the user wishes to exploit the the full capabilities of the auto-placement functions, it is necessary to define the allowed orientations of the module (Module Properties dialog).



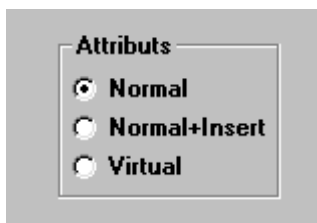
Usually, rotation of 180 degrees is permitted for resistors, non-polarized capacitors and other symmetrical elements.

Some modules (small transistors, for example) are often permitted to rotate by +/- 90 or 180 degrees. By default, a new module will have its rotation permissions set to zero. This can be adjusted according to the following rule:

A value of 0 makes rotation impossible, 10 allows it completely, and any intermediate value represents a limited rotation. For example, a resistor might have a permission of 10 to rotate 180 degrees (unrestrained) and a permission of 5 for a +/- 90 degree rotation (allowed, but discouraged).

Attributes

The attributes window is the following:

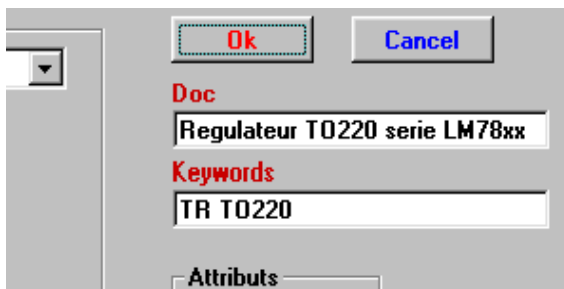


- Normal is the standard attribute.
- Normal+Insert indicates that the module must appear in the automatic insertion file (for automatic insertion machines). This attribute is most useful for surface mount components (SMDs).
- Virtual indicates that a component is directly formed by the circuit board. Examples would be edge connectors or inductors created by a particular track shape (as sometimes seen in microwave modules).

Documenting modules in a library

It is strongly recommended to document newly created modules, in order to facilitate their rapid and accurate retrieval. Who is able to recall the multiple pin-out variants of a TO92 module?


The Module Properties dialog offers a simple and yet powerful mean for documentation generation.



This menu allows:

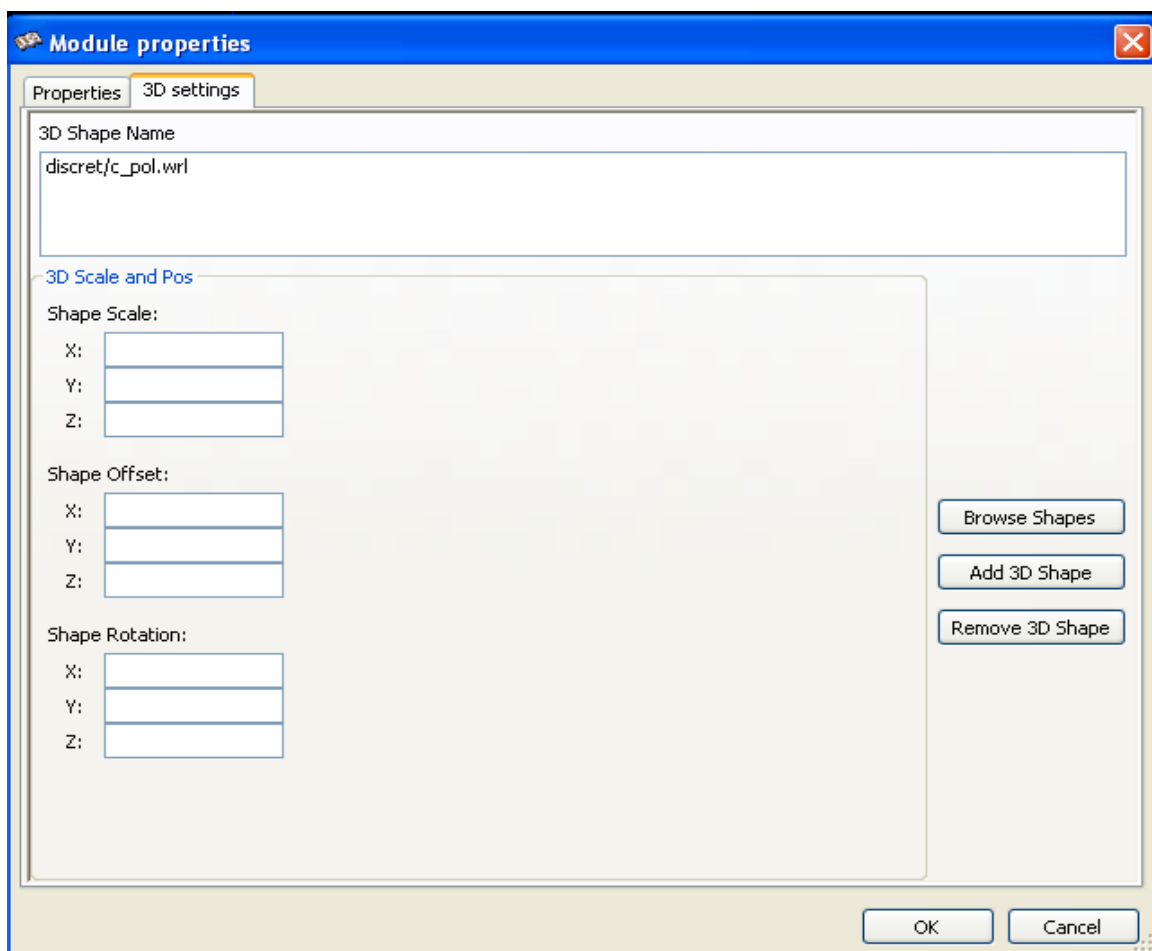
- The entry of a comment line (description).
- Multiple keywords.

The comment line is displayed with the component list in CVPCB and in the module selection menus in PCBNEW. The keywords can be used to restrict searches to those parts possessing the given keywords.

Thus, while using the load module command (icon  in the right-hand toolbar in Pcbnew), it is possible to type the text "=TO220" into the dialog box to have PCBNEW display a list of the modules possessing the keyword "TO220".

3-dimensional visualisation

A module may have been associated with a file containing a three-dimensional representation of itself. In order to associate such a file with a module, select the 3D Settings tab. The options panel is the following:



The data information should be provided:

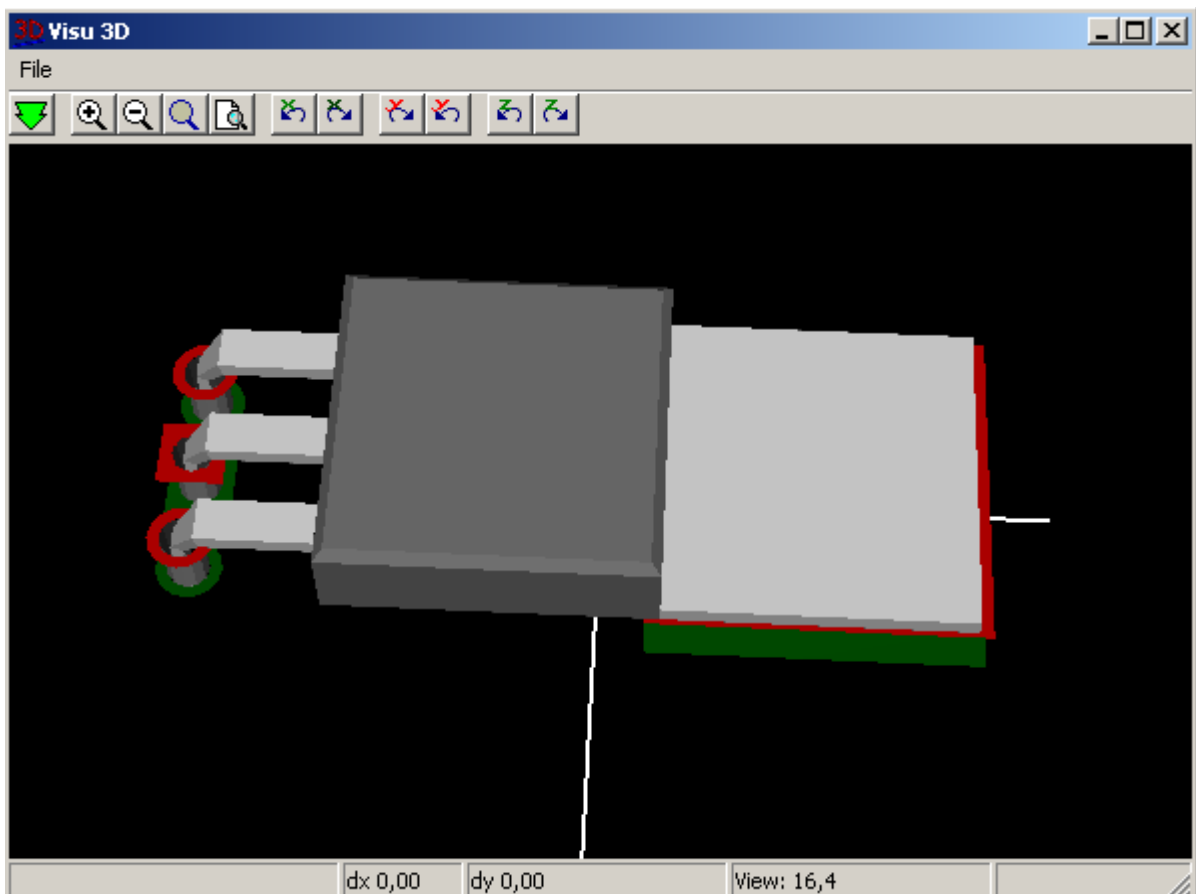
- The file containing the 3D representation (created by the 3D modeler wings3d, in vrml format, via the export to vrml command).
The default path is kicad/modules/package3d. In the example, the file name is discret/to_220horiz.vrl, using the default path)
- The x, y and z scales.
- The offset with respect to the anchor point of the module (usually zero).
- The initial rotation in degrees about each axis (usually zero).

Setting scale allows:

- To use the same 3D file for footprints which have similar shapes but different sizes (resistors, capacitors, SMD components...)
- For small (or very large) packages, a better use of the wings3D grid.


Scale 1 -> 0.1 inch in Pcbnew = 1 grid unit in wings3D
--

If such a file has been specified, it is possible to view the component in 3D.



The 3D model will automatically appear in the 3D representation of the printed circuit board.


Saving a module into the active library

The save command (modification of the file of the active library) is activated by the  button.

If a module of the same name exists (an older version), it will be overwritten. Because it is important to be able to have confidence in the library modules, it is worth double-checking the module for errors before saving.

Before saving, it is also recommended to change the reference or value of the module to be equal to the library name of the module.

Saving a module to the board

If the edited footprint comes from the current board, the button  will update this footprint on the board.