# Headings

1 - Presentation
- 1.1 - Description
- 1.2 - Principal Design features
- 1.3 - Remark

2 - Installation
- 2.1 - Installation of the software
- 2.2 - Modifying the default configuration

3 - General Operations
- 3.1 - Access to the operations
- 3.2 - Commands with the Mouse
  - 3.2.1 - Basic commands
  - 3.2.2 - Operations on blocks
- 3.3 - Selection of grid size
- 3.4 - Adjustment of the ZOOM
- 3.5 - Displaying the Cursor Coordinates
- 3.6 - Quick Commands using the keyboard (“Hot Keys”)
- 3.7 - Operation on blocks
- 3.8 - Menu Bar Options
  - 3.8.1 - Files
  - 3.8.2 - Preferences
  - 3.8.3 - Dimensions
  - 3.8.4 - Miscellaneous
  - 3.8.5 - Postprocess
  - 3.8.6 - 3D Display
  - 3.8.7 - Help
- 3.9 - Commands using icons on the upper toolbar
- 3.10 - Commands using icons on the right hand toolbar
- 3.11 - Commands using icons on the left hand toolbar
- 3.12 - POP Up Windows and fast editing of elements

4 - Schematic Implementation
- 4.1 - Linking the Schematic to the Printed Circuit
- 4.2 - Procedure for Creating the Printed Circuit
- 4.3 - Procedure for Updating the Printed Circuit
- 4.4 - Reading netlist - Loading footprints - Options:
  - 4.4.1 - Dialog box:
  - 4.4.2 - Options:
  - 4.4.3 - Loading new footprints:

5 - The Working layers
- 5.1 - Layers of copper
  - 5.1.1 - General information:
  - 5.1.2 - Selection of the number of layers:
- 5.2 - Auxiliary technical layers
- 5.3 - Selection of the Active Layer:
  - 5.3.1 - Selection using the upper toolbar:
  - 5.3.2 - Selection Using the Pop-Up Window:
- 5.4 - Selection of the Layers for Vias:

6 - Creation/correction of a board
- 6.1 - Creating a board
  - 6.1.1 - Drawing the board outline
  - 6.1.2 - Reading the netlist generated from the schematic
- 6.2 - Correcting a board
  - 6.2.1 - Steps to follow:
  - 6.2.2 - Deleting incorrect tracks:
  - 6.2.3 - Deleted components:
  - 6.2.4 - Modified modules:
  - 6.2.5 - Advanced options - selection using time stamps:

7 - Placement of the modules
- 7.1 - Assisting the Placement
- 7.2 - Manual placement
- 7.3 - General Reorientation of the modules
- 7.4 - Automatic Module Distribution

- 7.5 - Automatic placement of the modules
  - 7.5.1 - Characteristics of the automatic placer
  - 7.5.2 - Preparation
  - 7.5.3 - Interactive Autoplacement
  - 7.5.4 - Note
- 8 - Routing PCBs
  - 8.1 - Choosing routing parameters and routing a PCB
  - 8.2 - Typical dimensions for different PCB classes
    - 8.2.1 - Track width
    - 8.2.2 - Insulation (clearance)
  - 8.3 - Some typical combinations
    - 8.3.1 - 'Rustic'
    - 8.3.2 - 'Standard'
  - 8.4 - Manual routing
  - 8.5 - Creating the copper zones
    - 8.5.1 - Net selection:
    - 8.5.2 - Creating a zone:
      - 8.5.2.1 - Creating the zone limits:
      - 8.5.2.2 - Filling the zone:
    - 8.5.3 - Fill Options:
      - 8.5.3.1 - Working grid for filling.
      - 8.5.3.2 - Clearance
      - 8.5.3.3 - Pad options
- 9 - Preparation of files for circuit fabrication
  - 9.1 - Final preparations
  - 9.2 - Final DRC test:
  - 9.3 - Generating files for phototracing
    - 9.3.1 - GERBER format:
    - 9.3.2 - HPGL Format:
    - 9.3.3 - POSTSCRIPT Format:
  - 9.4 - Adjusting the clearance for the solder stop mask:
  - 9.5 - Generating the drill file(s)
  - 9.6 - Generating the cabling documentation:
  - 9.7 - Generation of file(s) for automatic component insertion:
  - 9.8 - Advanced tracing options:
- 10 - ModEdit: Managing LIBRARIES
  - 10.1 - Overview of ModEdit
  - 10.2 - ModEdit:
  - 10.3 - ModEdit user interface:
  - 10.4 - Main toolbar in Modedit:
  - 10.5 - Creating a new module:
  - 10.6 - Creating a new library:
  - 10.7 - Saving a module to the active library:
  - 10.8 - Transferring a module from one library to another:
  - 10.9 - Saving all the modules of a circuit in the active library:
  - 10.10 - Documentation for library modules:
  - 10.11 - Documenting libraries – recommended practice:
- 11 - ModEdit: editing modules
  - 11.1 - Overview.
  - 11.2 - Module elements.
    - 11.2.1 - Pads.
    - 11.2.2 - Contours.
    - 11.2.3 - Fields.
  - 11.3 - Starting ModEdit and selecting a module to edit.
  - 11.4 - Module Editor Toolbars:
    - 11.4.1 - Righthand Toolbar - editing
    - 11.4.2 - Lefthand Toolbar – display options
  - 11.5 - Context Menus.
  - 11.6 - The Module Properties dialog.
  - 11.7 - Creating a new module.
  - 11.8 - Adding and editing pads.
    - 11.8.1 - Adding a pad.
    - 11.8.2 - Setting pad properties.

11.9 - Information about automatic placement for a module.
11.10 - Attributes.
11.11 - Documenting modules in a library.
11.12 - Managing 3-dimensional visualisation
11.13 - Saving a module to the active library

# Headings:

1 - Presentation
    1.1 - Description
    1.2 - Principal Design features
    1.3 - Remark

## 1 - Presentation

### 1.1 - Description

PCBNEW is a powerful printed circuit board program, available for both the LINUX and WINDOWS operating systems.
It 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 advanced autorouters.
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…) as actual size and according to personal preferences:

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

### 1.2 - 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 customised:

- 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:

- Files for Photoplotters in GERBER format
- Files for drilling in EXCELLON format and plans of drilling
- Files for tracing and drilling in HPGL format
- Files for tracing and drilling in POSTSCRIPT format
- Local Printout.

## 1.3 - Remark

PCBNEW requires a **3 button mouse** (the 3rd button is not strictly necessary, but extremely useful for many commands).
Finally it should be noted that the diagrammatic tool Eeschema and CVPCB are needed to create the required netlists.
r

# Headings:

2 - Installation
  2.1 - Installation of the software
  2.2 - Modifying the default configuration

## 2 - Installation

### 2.1 - Installation of the software

The installation procedure is described in the **kicad** documentation.

### 2.2 - Modifying the default configuration

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

This configuration file can be modified, generally to change the list of libraries.

To do this:

- Launch pcbnew using kicad or directly (smethink like c:\kicad\bin\pcbnew.exe).
  (Linux: run **/usr/local/kicad/bin/kicad** or **/usr/local/kicad/bin/pcbnew** if binaries are 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.**

**PCBNEW:** General Operations.

# Headings:

3 - General Operations
- 3.1 - Access to the operations
- 3.2 - Commands with the Mouse
  - 3.2.1 - Basic commands
  - 3.2.2 - Operations on blocks
- 3.3 - Selection of grid size
- 3.4 - Adjustment of the ZOOM
- 3.5 - Displaying the Cursor Coordinates
- 3.6 - Quick Commands using the keyboard (“Hot Keys”)
- 3.7 - Operation on blocks
- 3.8 - Menu Bar Options
  - 3.8.1 - Files
  - 3.8.2 - Preferences
  - 3.8.3 - Dimensions
  - 3.8.4 - Miscellaneous
  - 3.8.5 - Postprocess
  - 3.8.6 - 3D Display
  - 3.8.7 - Help
- 3.9 - Commands using icons on the upper toolbar
- 3.10 - Commands using icons on the right hand toolbar
- 3.11 - Commands using icons on the left hand toolbar
- 3.12 - POP Up Windows and fast editing of elements

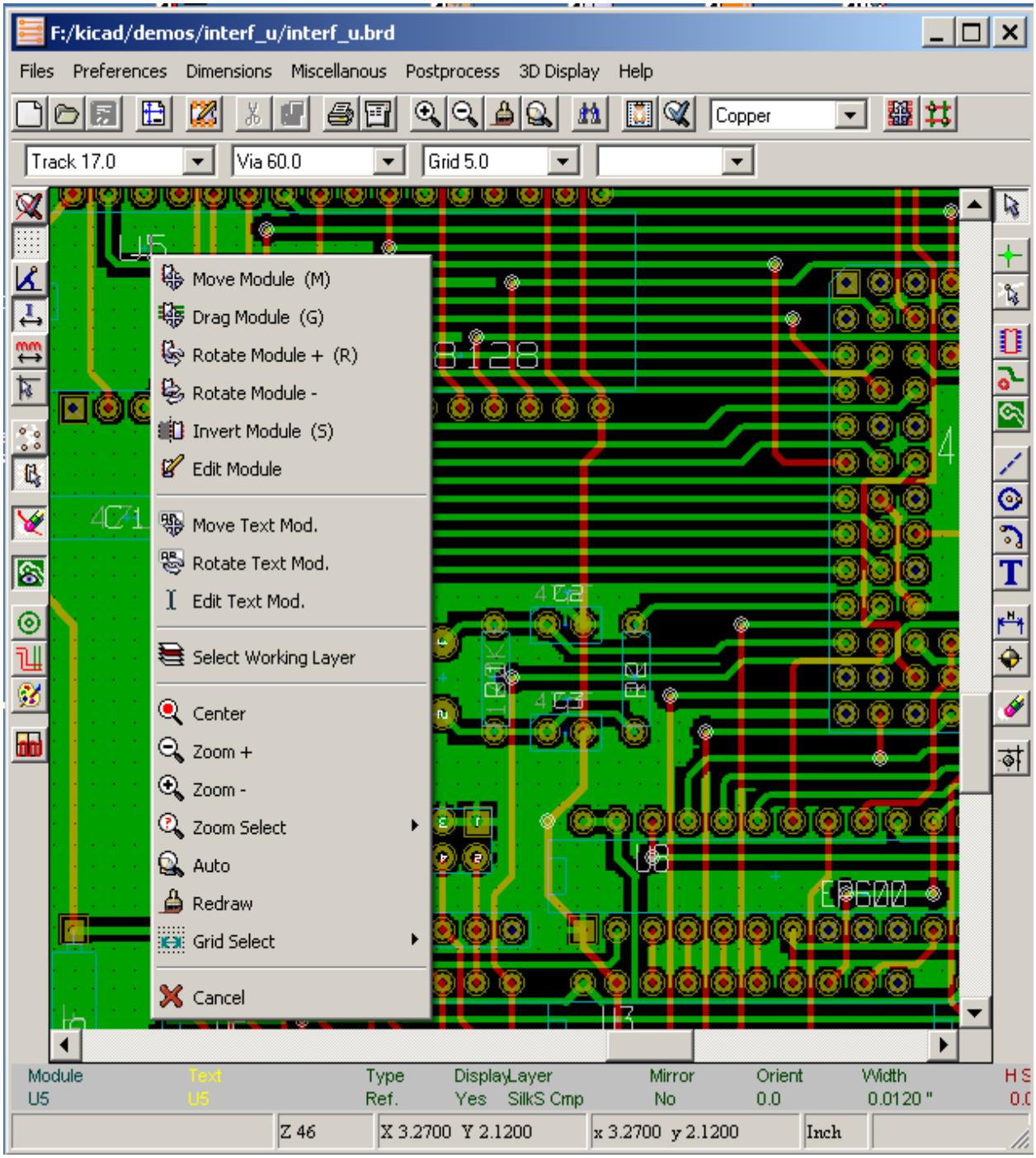
## 3 - General Operations

### 3.1 - Access to the operations

One reaches the various operations by using:

- the menu bar (top of screen).
- the icons at the top of the screen (general orders)
- the icons on the right of the screen (specific orders or “**tools**”)
- the icons on the left of the screen (Display Options)
- the mouse buttons (provides menu options).
  **In particular:**
  *The right-hand mouse button reveals a POP Up Menu the contents of which depend on the element under the cursor (Zoom, grid and editing the element).*
- the keyboard (Function keys **F1**, **F2**, **F3**, **F4**, **Shift**, **Delete, +, - Page Up, Page Down** and “**space**” bar).
  **In particular:**
  *the “Escape” key generally cancels an operation in progress*.

The screenshot below illustrates some of the possible accesses to the operations:




## 3.2 - Commands with the Mouse

### 3.2.1 - 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.
  The rapid zoom operation is only available on the 3 button mouse consequently it has the advantage over a 2 button mouse. Hold down the centre button and draw a rectangle to zoom to the described area.
  Rotation of a mouse wheel zooms in and out.

**- Right Button:**

- Displays a Pop-up Menu

### 3.2.2 - Operations on blocks

Operations to move, invert (mirror), copy, rotate and delete a block are all available on 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 carried out on releasing the button.
By holding down one of the keys “Shift” or “Ctrl”, or both keys “Shflt 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° |
| Shft+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 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.

## 3.3 - Selection of grid size

The cursor during layout of elements moves on a grid, the display of the grid can be turned on or off using the icon on the left hand side 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.

## 3.4 - Adjustment of the ZOOM

To change the “ZOOM”:

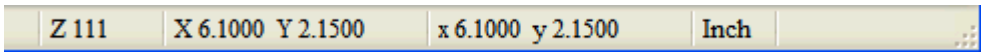
- Open the POP-Up window (using the right mouse button) and to select the desired zoom.
- Or use the function keys:
  F1: Enlarge (zoom in)
  F2: Reduce (zoom out)
  F3: Redraw the display
  F4: Centre view at the current cursor position
- Or rotate the mouse wheel.
- Or hold down the centre mouse button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

## 3.5 - Displaying the Cursor Coordinates

The cursor co-ordinates are displayed in inches (inch or “) or millimetres (mm) as selected using the 'I' 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 co-ordinates (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.




## 3.6 - Quick Commands using the keyboard (“Hot Keys”)

Certain commands are accessible directly with the keyboard (selection can be either upper or lower case). The options are:

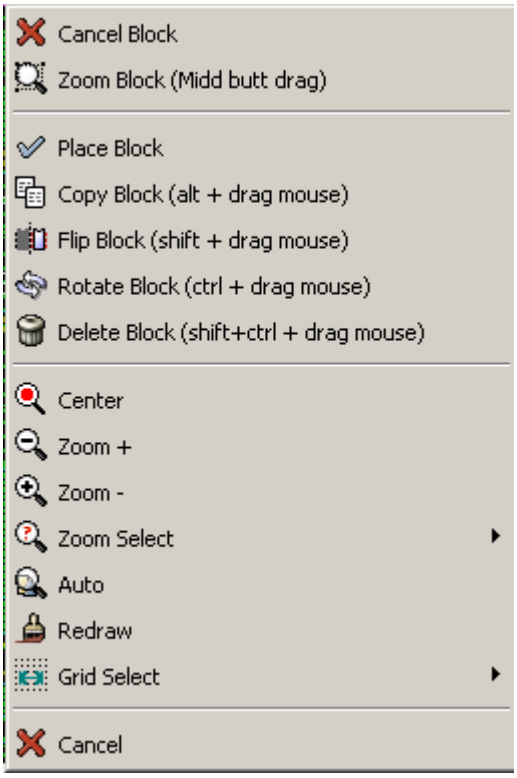
- **Delete** (or Del): Deletes a Module or track (only if the Module tool or the track tool is active)
- **V,** if the track tool is active Switches working layer (Component <> Copper) or place via, if a track is in progress.
- **R**: Rotates a module
- **S**: Switches module to opposite layer (Component <> Copper)
- **M**: Move module (placement by click of left mouse button).
- **G**: Drag module (placement by click of left mouse button).
- **+** et **-**: Active layer = next or previous layer.
- « **Page Up** » Active layer = component.
- « **Page Down** » Active layer = cppper.

## 3.7 - Operation on blocks

Operations to move, invert (mirror), copy, rotate and delete a block are all available on 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:

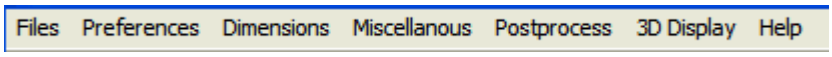
| | |
|---|---|
| Left mouse button held down | move block |
| Shift + Left mouse button held down | invert (mirror) block |
| Ctrl + Left mouse button held down | rotate block 90° |
| Shft+Ctrl + Left mouse button held down | delete the block |
| Alt + Left mouse button held down | copy the block |

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 cancelled via the same pop-up menu or pressing the Escape key (Esc).

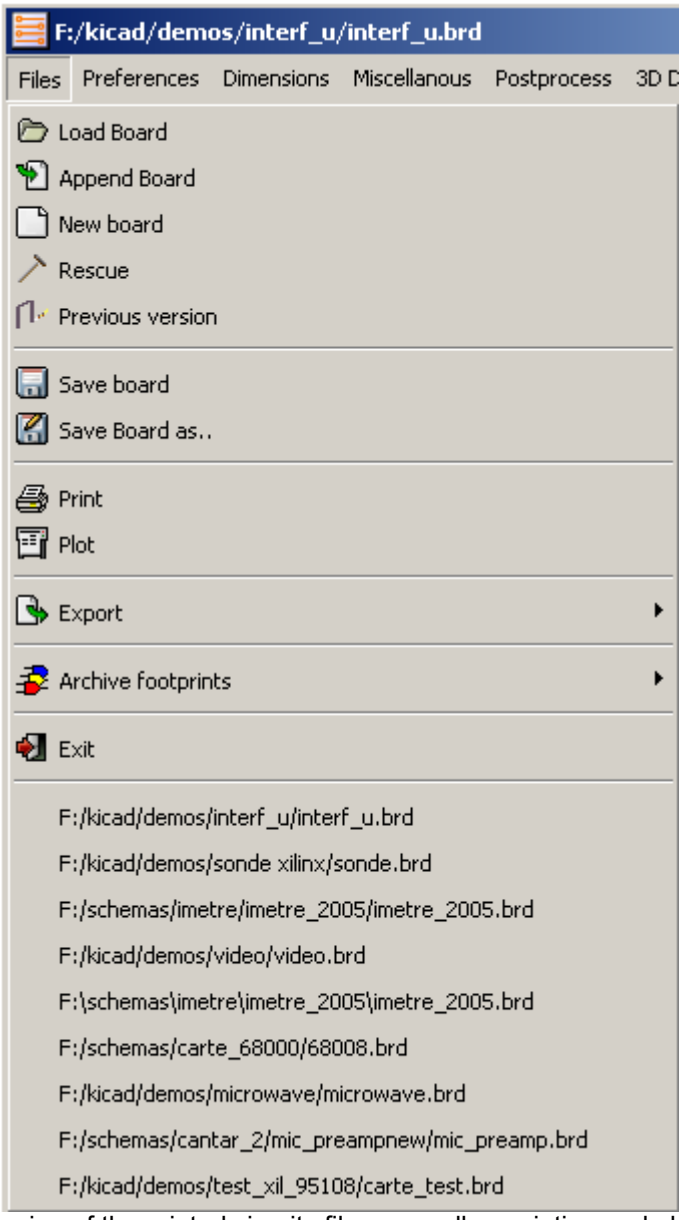



## 3.8 - Menu Bar Options

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

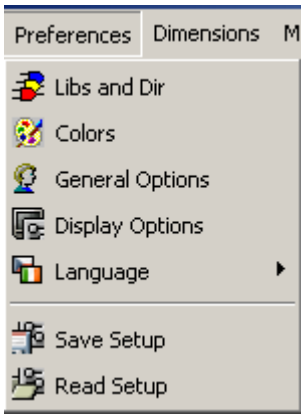


### 3.8.1 - Files




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

### 3.8.2 - Preferences

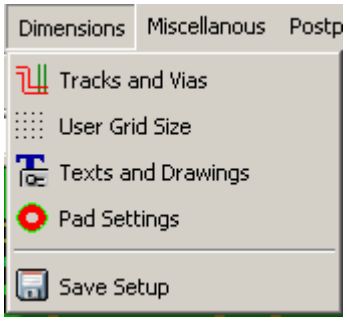



Allows:

- Selection of the module libraries.

- The choice of the colours for displaying layers and other elements. Also enables the display of elements to be turned on and off.
- Management of general options (units, number of layers, etc.).
- The management of other display options.

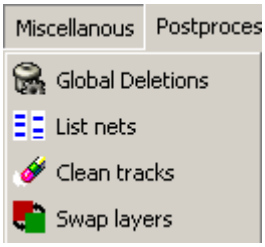
### 3.8.3 - Dimensions




Allows adjustment of:

- Width of tracks and the type and size of vias.
- User grid size.
- Size of texts and the line width for drawings.
- Dimensions and characteristic of pads.

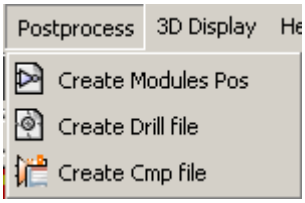
### 3.8.4 - Miscellaneous




Provides access to:

- Global deletion of elements.
- Listing of Nets.
- Deletion of redundant segments of track.
- Switching elements between layers.

### 3.8.5 - Postprocess

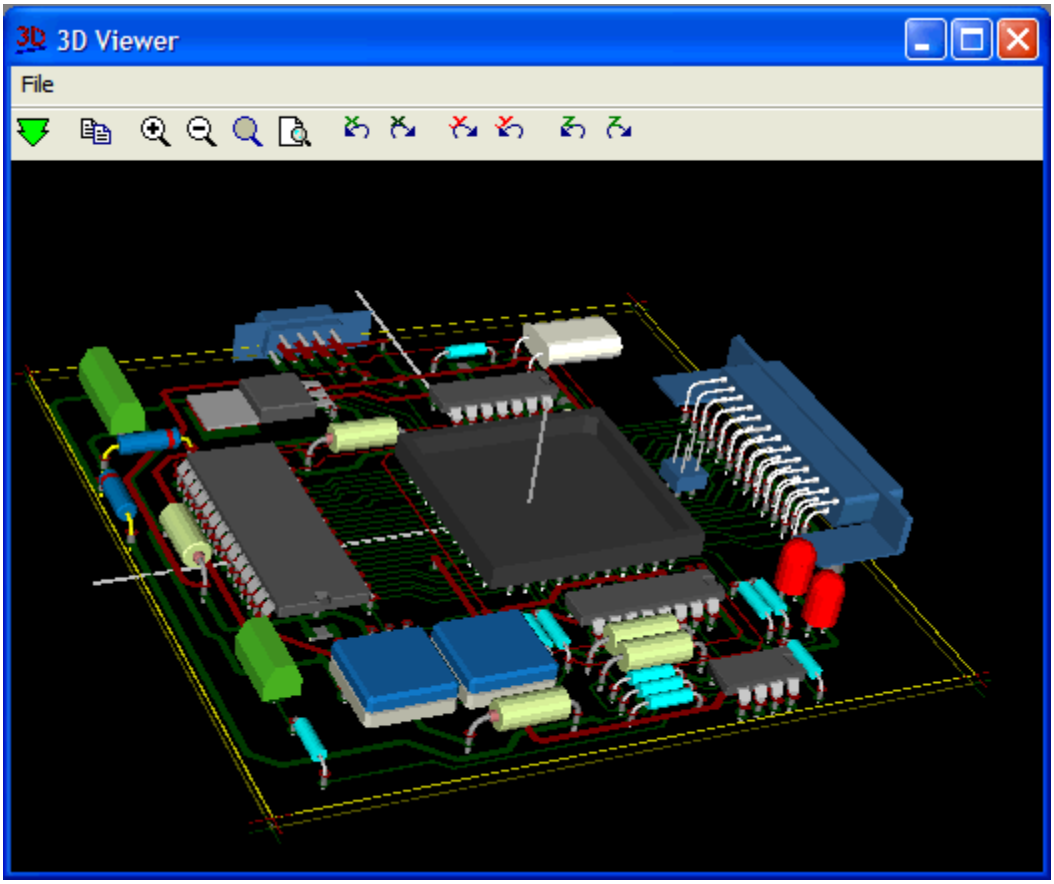



Allows the creation of the following files:

- Components placement (for automatic placement).
- Drilling.
- Association of components/modules (normally generated by CVPCB).

### 3.8.6 - 3D Display

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




### 3.8.7 - Help

Provides access to the help file and version information (Pcbnew About).

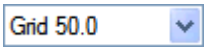
## 3.9 - Commands using icons on the upper toolbar

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 (**Modedit**) to display/edit library or pcb modules. |
| | Delete the elements selected during a move block operation. |
| | Undo the last deletion |
| | 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. |
| | Module mode: when active this enables module options in the pop-up window. |
| | Routing mode: 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. |
| | Selection of a dimension of via already in use. |
|  | Selection of the grid size. |
| | Selection of the zoom. |

## 3.10 - Commands using icons on the right hand toolbar

| | |
|---|---|
| | This toolbar gives access to the **tools** for:<br>- Placement of modules, tracks, zones of copper, texts…<br>- Net Highlighting.<br>- Creating notes, graphic elements….<br>- Deleting elements. |
| | |
| | Cancel the tool in progress. |
| | 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). |
| | Delete element pointed to by the cursor (see note below)<br>Note:<br>● when Deleting if several superimposed elements are pointed to priority is given to the smallest (in the decreasing set of priorities tracks, text, module).<br>● the function “Undelete” of the upper toolbar allows the cancellation of the last item deleted. |
| | Offset adjust for drilling and place files. |

## 3.11 - Commands using icons on the left hand toolbar

| | |
|---|---|
| | This left hand toolbar provides display and control options |

| | |
|---|---|
| | Turns DRC (Design Rule Checking) on/off. Caution: when DRC is off incorrect connections can be made. |
| | 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 millimetres. |
| | Change cursor display. |
| | Display general ratsnest (incomplete connections between modules). |
| | Dislay module ratsnest dynamically as it is moved. |
| | Enable/Disable automatic deletion of a track when it is redrawn. |
| | Display of the copper zones on/off. |
| | 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 grey. Useful for working on multi-layer circuits. |
| | Under development |

## 3.12 - POP Up Windows and fast editing of elements

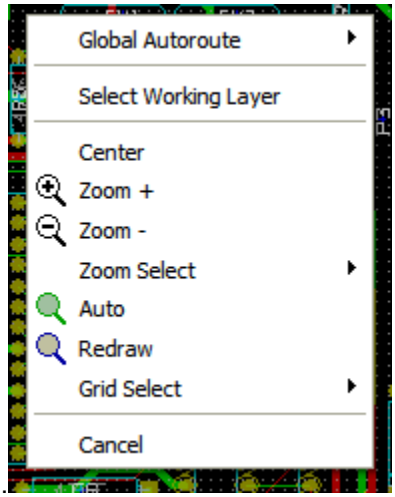
A click of the right mouse button reveals a Pop-Up Window the contents of which depend on the element pointed at by the cursor.
This gives immediate access to:

- 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.

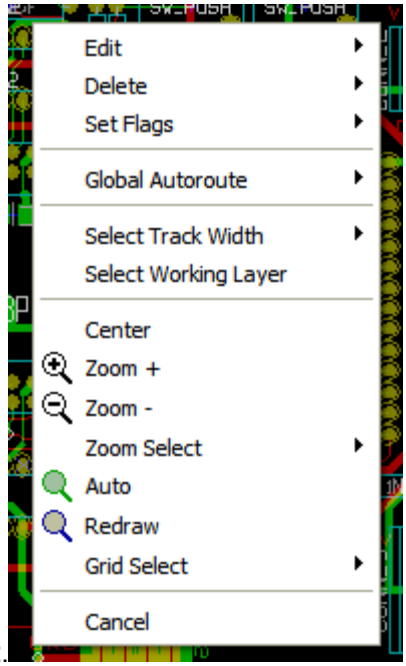
The views below show the pop-up windows:

1. with no element selected
2. with a track selected.
3. with a module selected.

1.

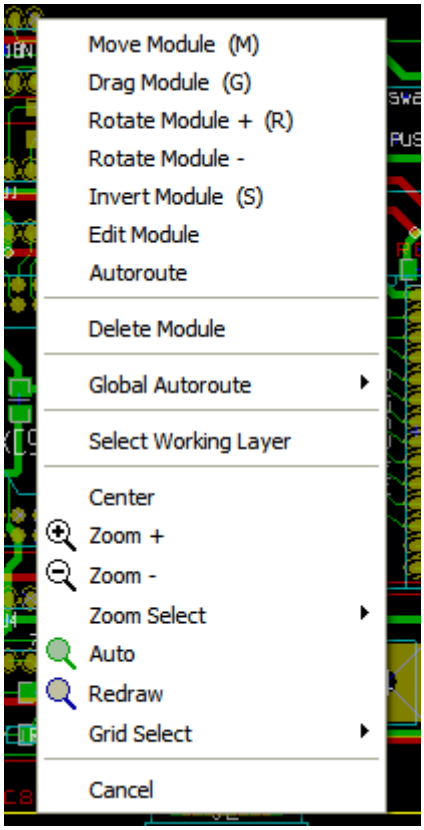



2.




3.

**Pcbnew**

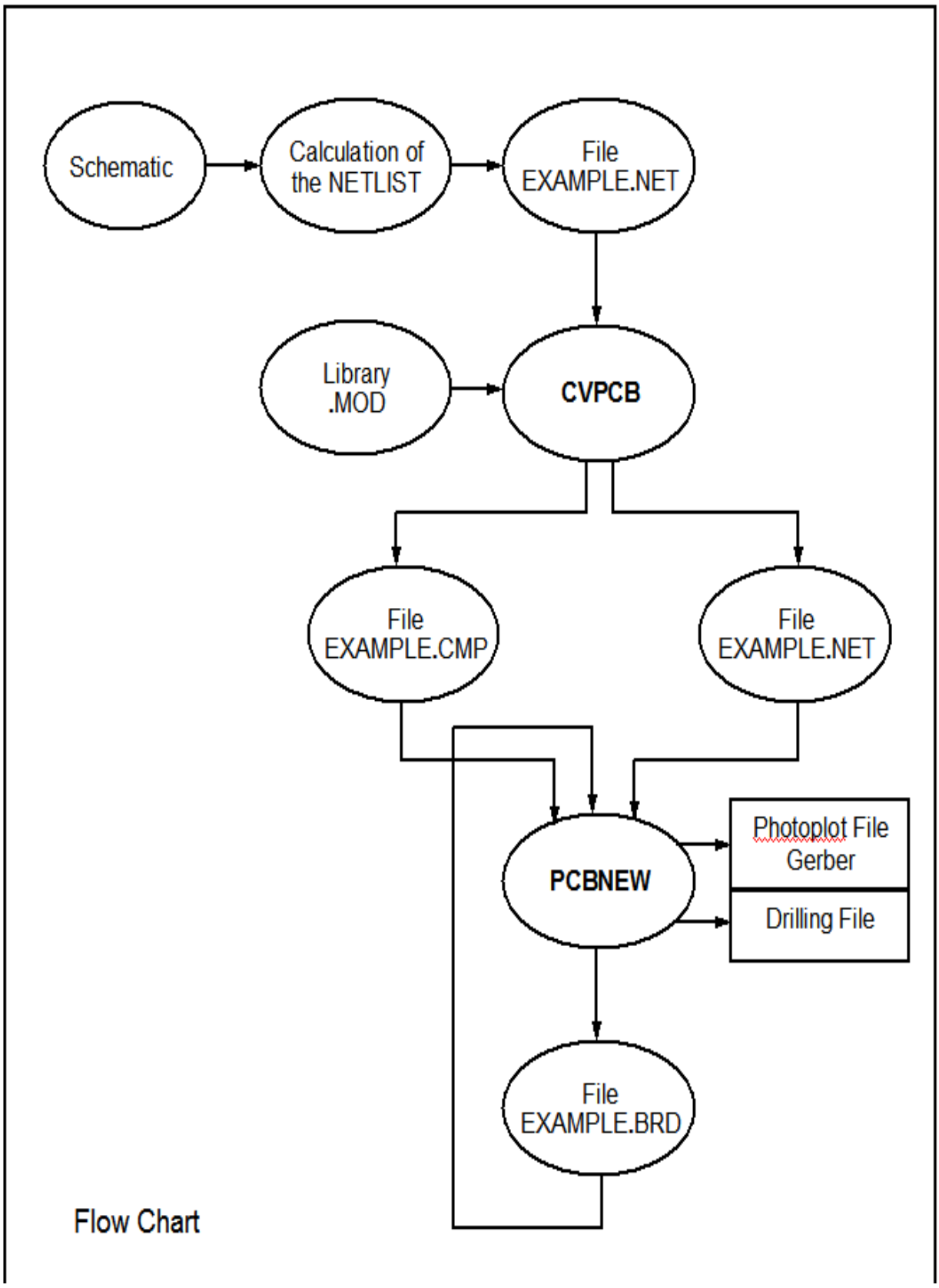

Move Module (M)
Drag Module (G)
Rotate Module + (R)
Rotate Module -
Invert Module (S)
Edit Module
Autoroute
Delete Module
Global Autoroute
Select Working Layer
Center
Zoom +
Zoom -
Zoom Select
Auto
Redraw
Grid Select
Cancel

# Headings:

4 - Schematic Implementation
- 4.1 - Linking the Schematic to the Printed Circuit
- 4.2 - Procedure for Creating the Printed Circuit
- 4.3 - Procedure for Updating the Printed Circuit
- 4.4 - Reading netlist - Loading footprints - Options:
  - 4.4.1 - Dialog box:
  - 4.4.2 - Options:
  - 4.4.3 - Loading new footprints:

## 4 - Schematic Implementation

### 4.1 - Linking the Schematic to the Printed Circuit


Schematic
Calculation of the NETLIST
File EXAMPLE.NET
Library .MOD
CVPCB
File EXAMPLE.CMP
File EXAMPLE.NET
PCBNEW
Photoplot File Gerber
Drilling File
File EXAMPLE.BRD
Flow Chart

The schematic is linked to PCBNEW by means of the Netlist file, which is normally generated by the schematic program used.
Note: PCBNEW accepts Netlist files with the **EESchema** or **ORCAD PCB 2** formats.
The Netlist file initially generated is usually incomplete as there has been no assignment of the modules that correspond to the various components used in the schematic. Consequently an intermediate stage is necessary, the generation of the file of association components/modules.
CVPCB is used to create this file which is named * .CMP.
CVPCB also updates the Netlist file using this information.
PCBNEW reads the modified Netlist file * .NET and, if it exists, the file * .CMP.
In the event of a module being changed directly in PCBNEW the * .CMP file is automatically updated so avoiding the requirement to run CVPCB again.

## 4.2 - Procedure for Creating the Printed Circuit

After having created the required schematic:

- Generate the netlist using Eeschema.
- Assign each components in the Netlist produced by Eeschema to the corresponding module 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 modules.
The modules can now be placed manually or automatically on the board and and the tracks routed.

## 4.3 - Procedure for Updating the Printed Circuit

If the schematic is modified, the following steps must be repeated:

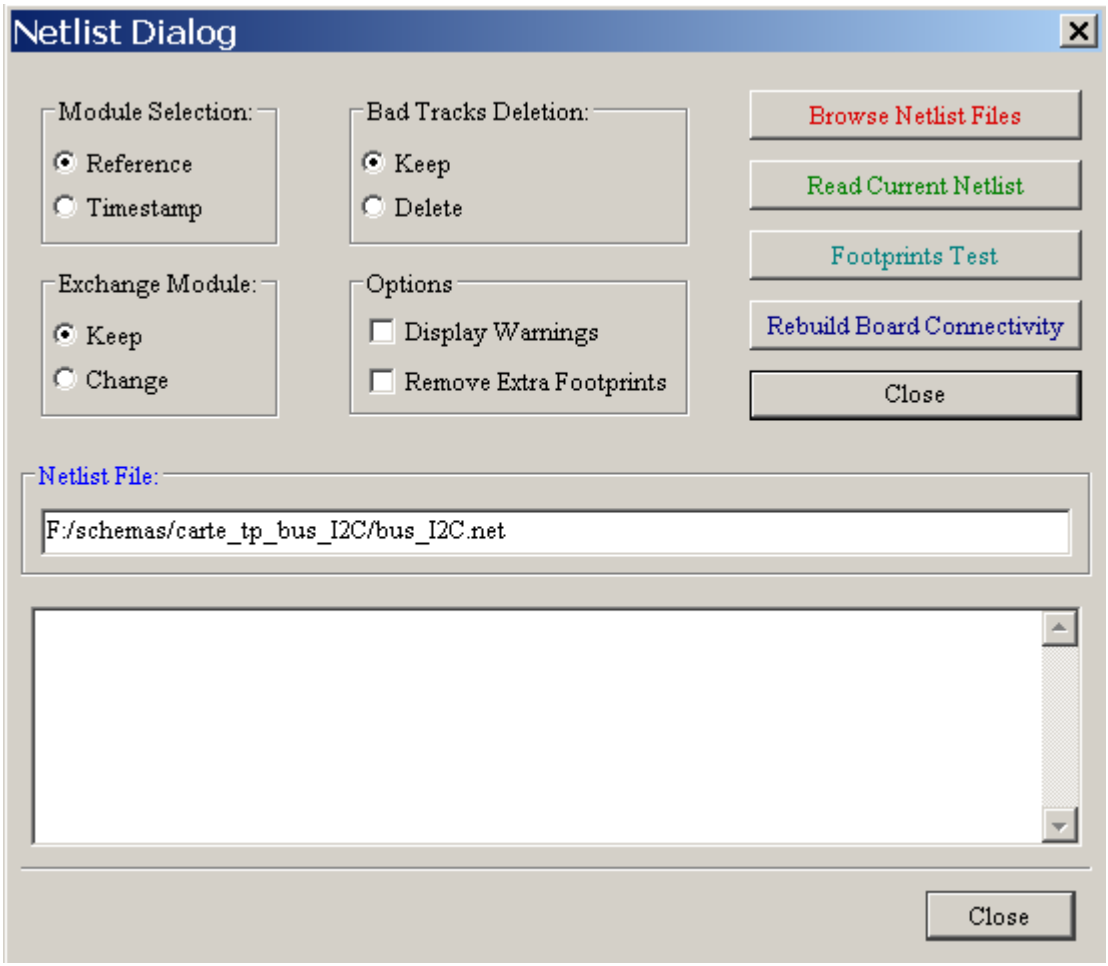
- Generate a new netlist using Eeschema.
- If the changes to the schematic involve new components, the corresponding modules must be assigned using CVPCB.
- Launch PCBNEW and re-read the modified Netlist (this will also re-read the file with the module selections).

PCBNEW will then load automatically any new modules, add the new connections and remove redundant connections.

## 4.4 - Reading netlist - Loading footprints - Options:

### 4.4.1 - Dialog box:

Access by the tool:

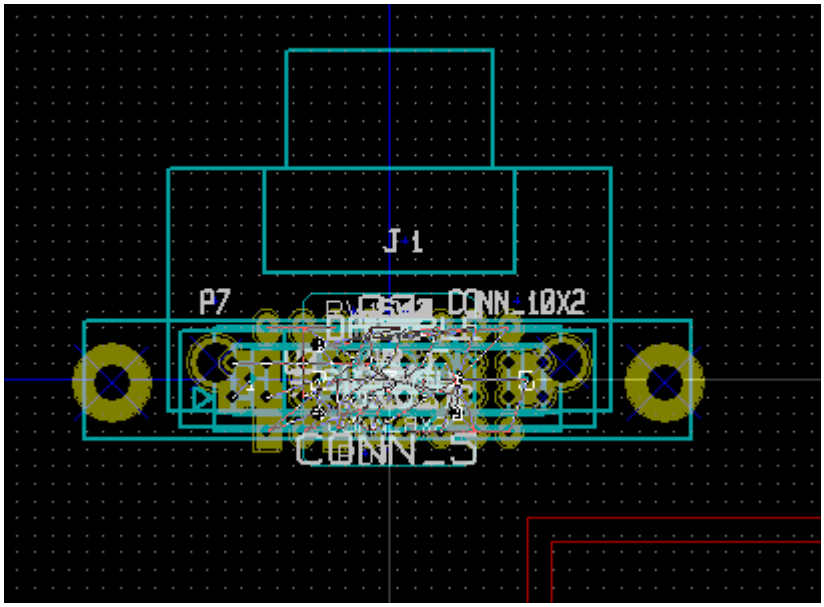



### 4.4.2 - Options:

| | |
|---|---|
| **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 :**<br>**(on/off)** | Display all messages (or not)<br>Remove footprints which are on board but not in netlist.<br>Footprint with attribute "Locked" will be not removed |

### 4.4.3 - Loading new footprints:

When new footprints are found in netlist, they are automatically loaded:

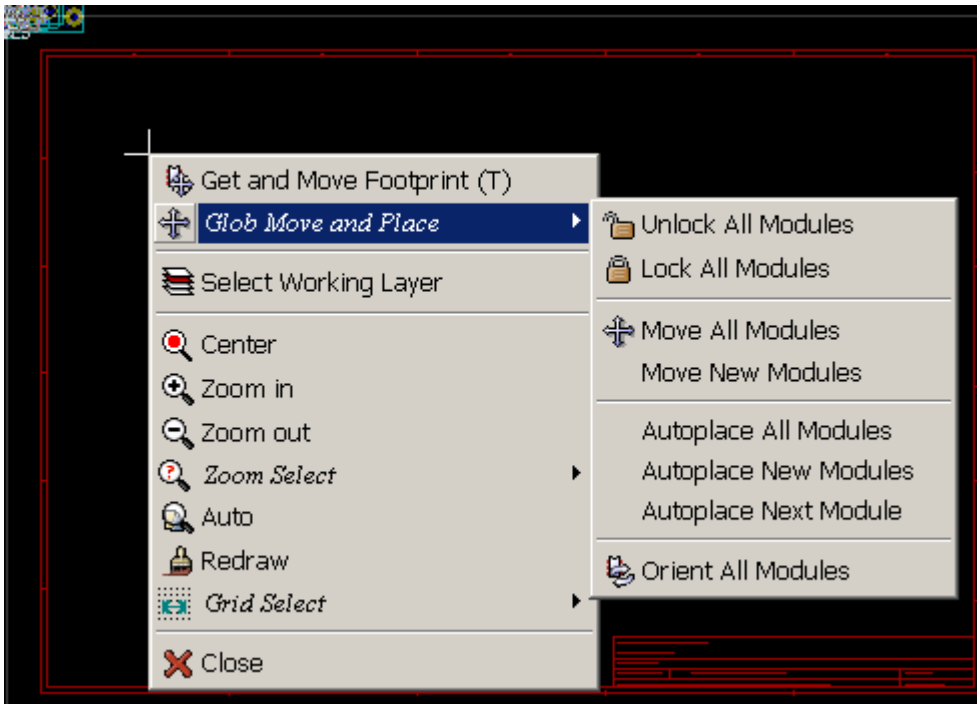



They are stacked at coordinate 0,0.
One can move footprints one by one.
But a better way is to automatically move (unstack) them:

- Activate footprint mode"

| | |
|---|---|
| | activate the "footprint mode" |
| | footprint mode activated |

Move the mouse cursor to a suitable (free) area, and activate the right button:




***Use***:

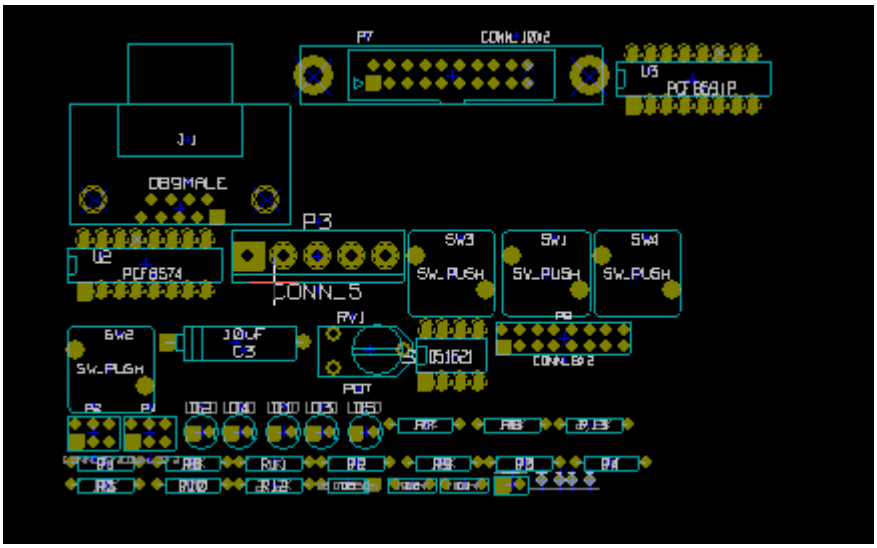
**Pcbnew**

- ***Move New Modules*** if there is already a board with existing footprints.

or

- ***Move All Modules***, the first time (when creating a board)

Here is result:



# Headings:

5 - The Working layers
5.1 - Layers of copper
5.1.1 - General information:
5.1.2 - Selection of the number of layers:
5.2 - Auxiliary technical layers
5.3 - Selection of the Active Layer:
5.3.1 - Selection using the upper toolbar:
5.3.2 - Selection Using the Pop-Up Window:
5.4 - Selection of the Layers for Vias:

## 5 - The Working layers

PCBNEW works on 28 different layers:

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

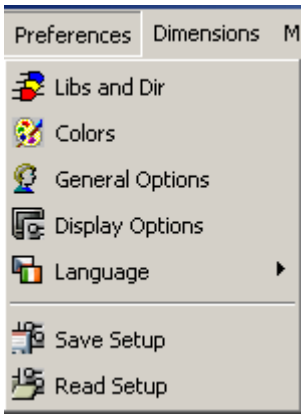
### 5.1 - Layers of copper

#### 5.1.1 - General information:

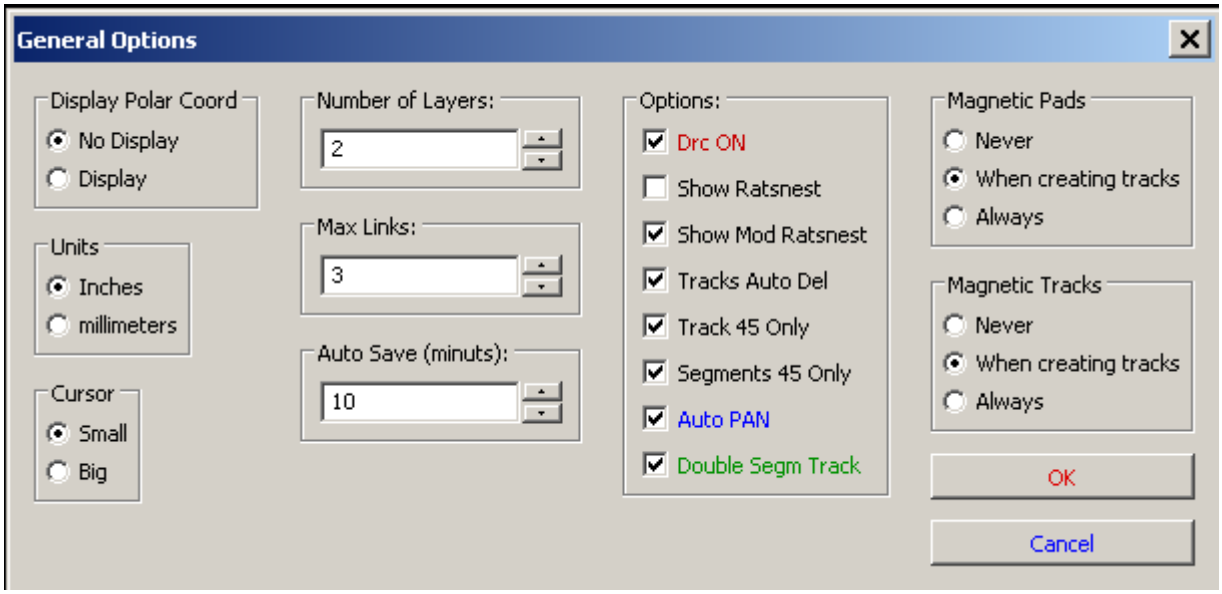
They are the usual layers of work, used by the automatic router, on which tracks can be placed. Layer 1 is the copper (solder) layer. Layer 16 is the component layer. The other layers are the internal layers (L2 to L15).

#### 5.1.2 - Selection of the number of layers:

To aid navigation between layers, it is necessary to select the number of working layers.
To do this from the menu bar select **Preferences – General Options**.




Then enter the required number of layers (1 to 16).




## 5.2 - 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:

- **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.

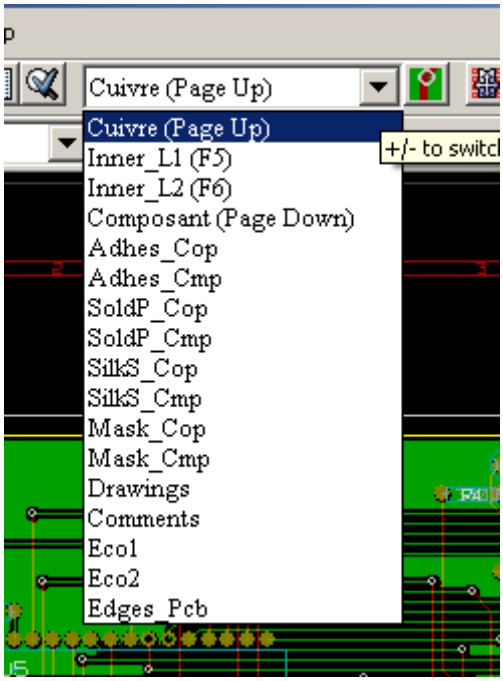
- **Layer Edges PCB**: this layer is reserved for the drawing of circuit board outline. Any element (graphic, texts…) placed on this layer appears on all the other layers.

## 5.3 - Selection of the Active Layer:

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

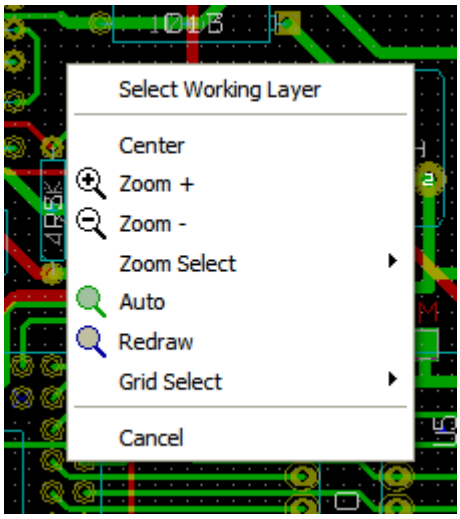
- Using the upper toolbar.
- With the Pop-Up window (activated with the right mouse button).
- Using the + and – keys (works on copper layers only).
- By hot keys.

### 5.3.1 - Selection using the upper toolbar:

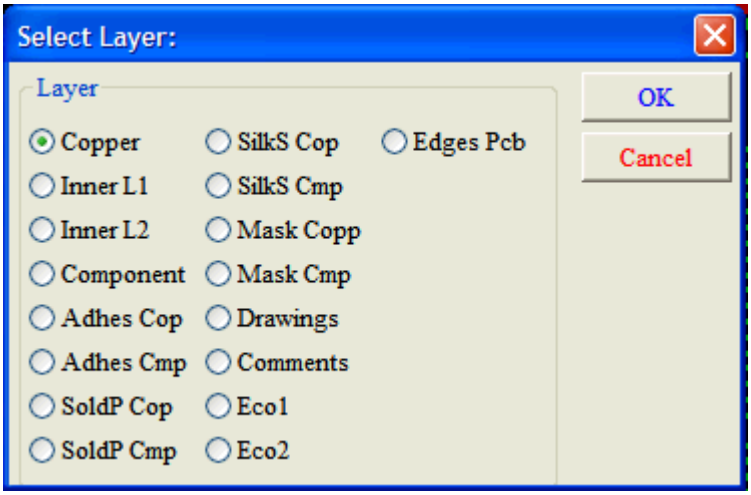



This directly selects the working layer.
Hot keys to select the working layer are displayed.

### 5.3.2 - Selection Using the Pop-Up Window:

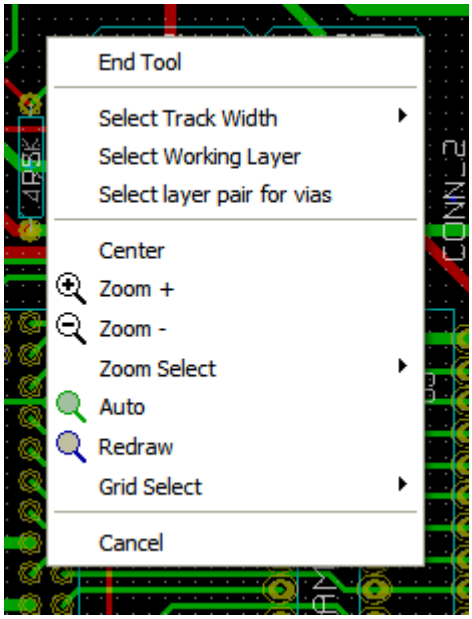



The Pop-up window opens a menu window - which provides choice of the working layer:

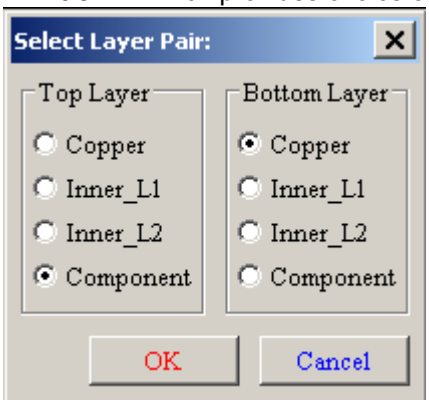



## 5.4 - Selection of the Layers for Vias:

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.

# Headings:

6 - Creation/correction of a board
- 6.1 - Creating a board
  - 6.1.1 - Drawing the board outline
  - 6.1.2 - Reading the netlist generated from the schematic
- 6.2 - Correcting a board
  - 6.2.1 - Steps to follow:
  - 6.2.2 - Deleting incorrect tracks:
  - 6.2.3 - Deleted components:
  - 6.2.4 - Modified modules:
  - 6.2.5 - Advanced options - selection using time stamps:

## 6 - Creation/correction of a board

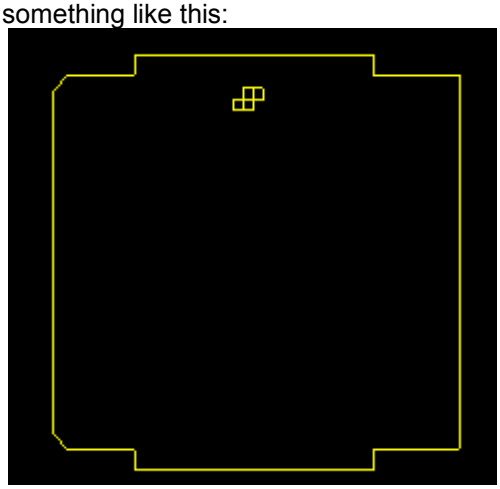
### 6.1 - Creating a board

#### 6.1.1 - 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:

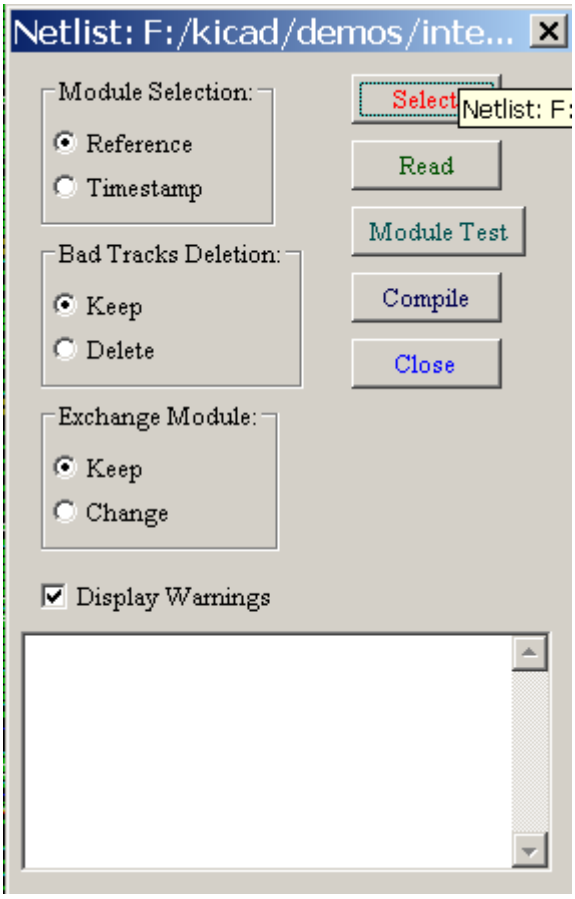
1. Select the 'Add graphic circle' or 'Add graphic arc' tool
2. Click to fix the circle centre
3. Adjust the radius by moving the mouse
4. Finish by clicking again.

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:

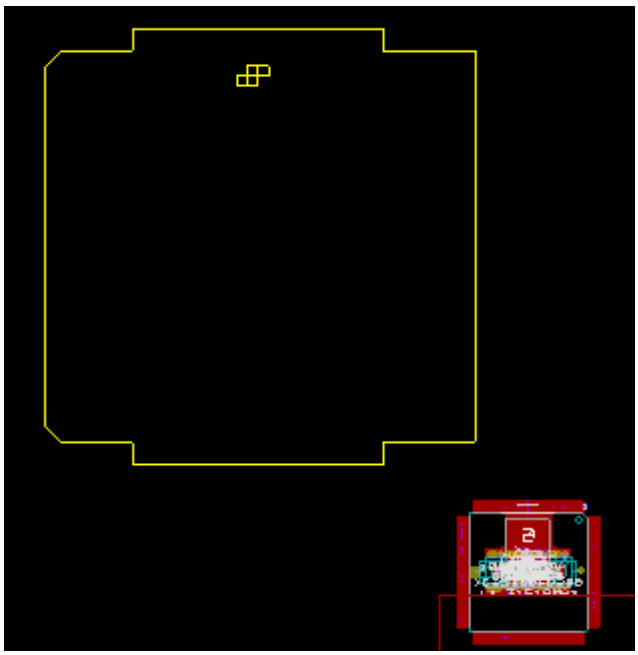


#### 6.1.2 - Reading the netlist generated from the schematic

Activate the icon to display the netlist dialog window:

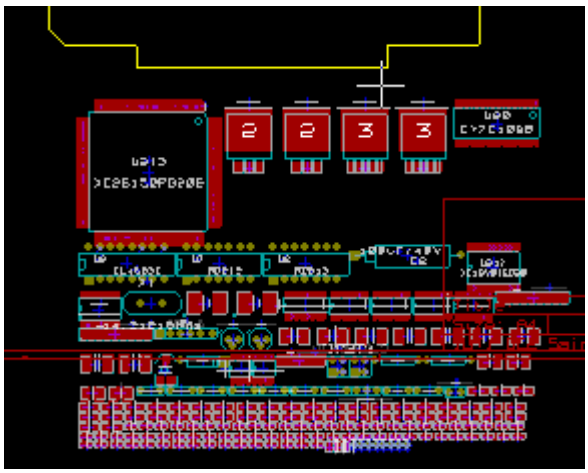



If the name (path) of the netlist in the window title is incorrect, use the 'Select' button to browse to the desired netlist. Then 'Read' the netlist. Any modules not already loaded will appear, superimposed one upon another (we shall see below how to move them automatically).



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 recognise. 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.

## 6.2 - Correcting a board

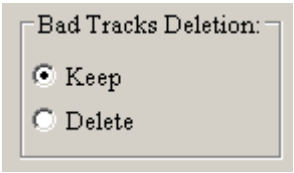
It is very often necessary to correct a board following the corresponding change in the schematic.

### 6.2.1 - Steps to follow:

1. Create a new netlist from the modified schematic.
2. If new components have been added, link these to their corresponding modules in ***cvpcb***.
3. Read the new netlist in ***pcbnew***.

### 6.2.2 - 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:



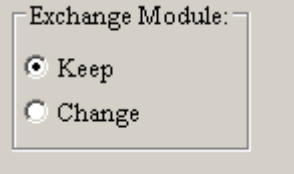

However, it is often quicker to modify such tracks by hand (the DRC function allows their identification).

### 6.2.3 - Deleted components:

Pcbnew **does not automatically delete** modules corresponding to components that have been removed from the schematic – these must be deleted by hand. This convention is necesary because there are often modules (holes for fixation screws, for instance) that are added to the PCB that never appear in the schematic.

### 6.2.4 - Modified modules:

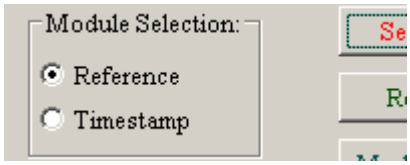
If a module is modified in the netlist (using C*vpcb*), but the module has already been placed, it will not be modified by Pcbnew, unless the corresponding option of the 'Exchange module' box of the netlist dialog is checked:




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

### 6.2.5 - 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 reannotation problems.

*PCBNEW*: Placement of the modules

# Headings:


7 - Placement of the modules
- 7.1 - Assisting the Placement
- 7.2 - Manual placement
- 7.3 - General Reorientation of the modules
- 7.4 - Automatic Module Distribution
- 7.5 - Automatic placement of the modules
  - 7.5.1 - Characteristics of the automatic placer
  - 7.5.2 - Preparation
  - 7.5.3 - Interactive Autoplacement
  - 7.5.4 - Note

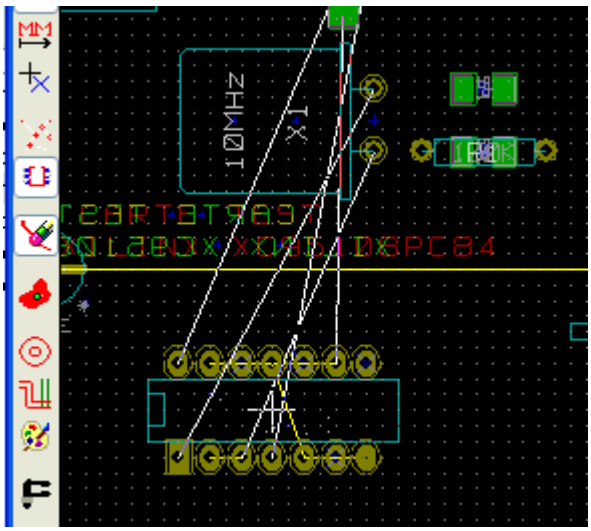

## 7 - Placement of the modules

### 7.1 - Assisting the Placement

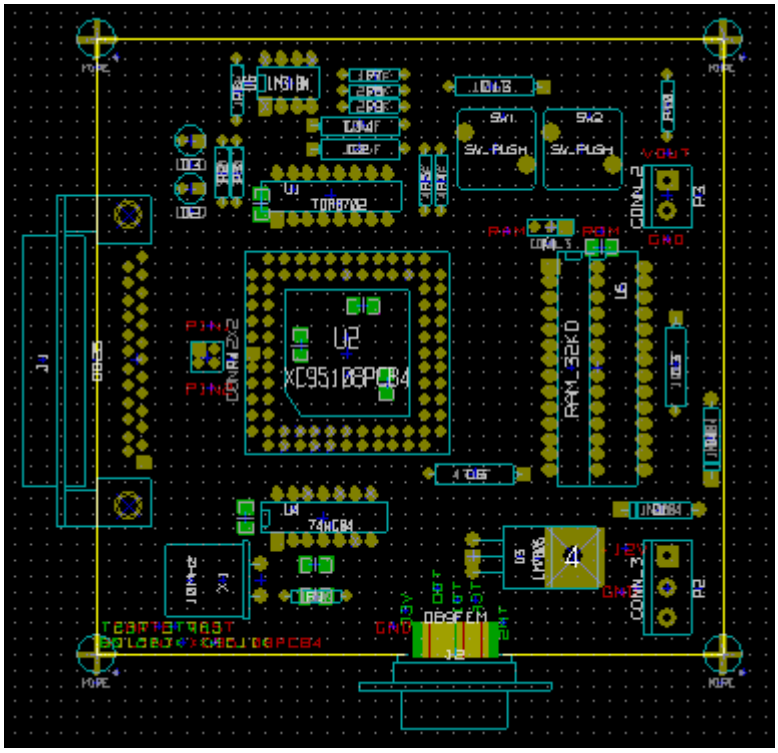
Whilst moving modules the module ratsnest (the net connections) can be displayed to assist the placement. To enable this the icon of the left toolbar must be activated.

### 7.2 - 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.



Here you can see the display of the module ratsnest during a move.
The circuit once all the modules are placed may be as shown:




## 7.3 - General Reorientation of the 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**". (Translators Note: This does not seem to be working.)

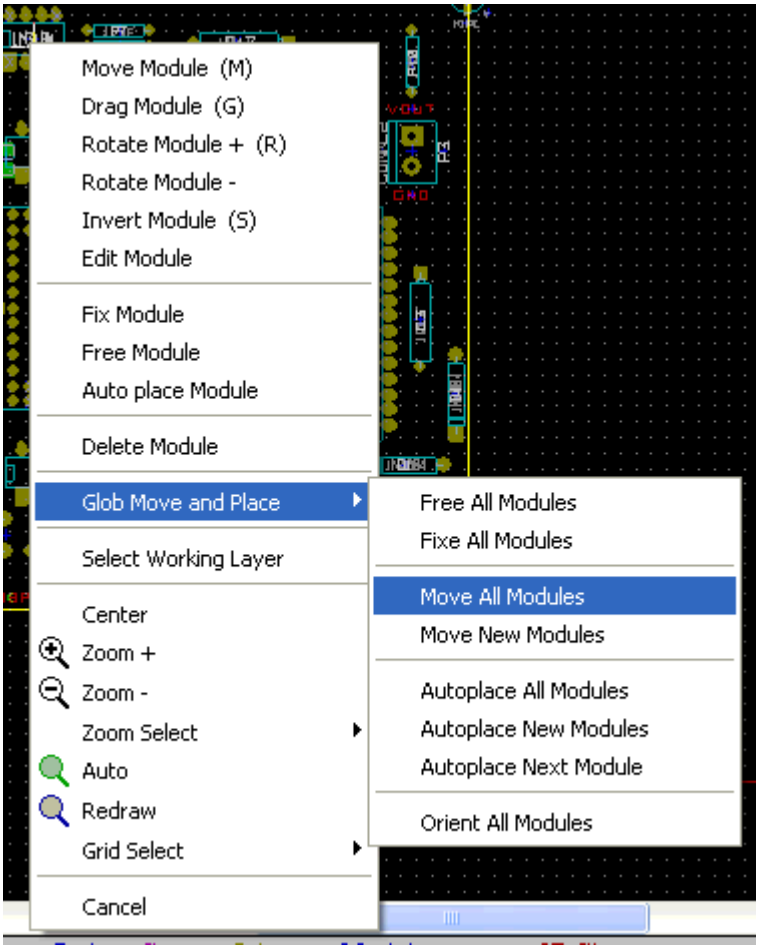
## 7.4 - Automatic Module Distribution

Note:
Generally 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.

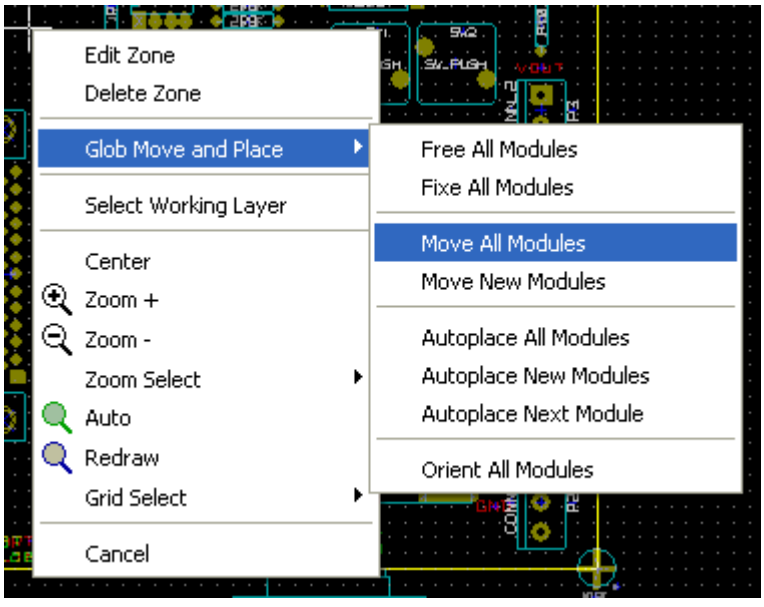
As stated in the last chapter, new modules loaded during the reading of the netlist appear piled up at a single location on the board. PCBNEW allows an automatic distribution of the modules to make manual selection and placement easier.

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.

## 7.5 - Automatic placement of the modules

### 7.5.1 - 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 with many pads. The order of placement is optimized to initially place these larger modules with many pads.

### 7.5.2 - 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..

### 7.5.3 - Interactive Autoplacement

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.

### 7.5.4 - 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**

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.

# Heading:

8 - Routing PCBs
- 8.1 - Choosing routing parameters and routing a PCB
- 8.2 - Typical dimensions for different PCB classes
  - 8.2.1 - Track width
  - 8.2.2 - Insulation (clearance)
- 8.3 - Some typical combinations
  - 8.3.1 - 'Rustic'
  - 8.3.2 - 'Standard'
- 8.4 - Manual routing
- 8.5 - Creating the copper zones
  - 8.5.1 - Net selection:
  - 8.5.2 - Creating a zone:
    - 8.5.2.1 - Creating the zone limits:
    - 8.5.2.2 - Filling the zone:
  - 8.5.3 - Fill Options:
    - 8.5.3.1 - Working grid for filling.
    - 8.5.3.2 - Clearance
    - 8.5.3.3 - Pad options

## 8 - Routing PCBs

### 8.1 - Choosing routing parameters and routing a PCB

The choice is made in the menu: **Dimensions ->Tracks and Vias**.
The dimensions are given in inches or mm, depending upon the active units.
Reminder: 2.54 cm = 1 inch (or " ) = 1000 mils = 10000 tenths of mils.

### 8.2 - Typical dimensions for different PCB classes

#### 8.2.1 - 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 |
| 1/10mils | 310 | 200 | 160 | 100 | 60 |

#### 8.2.2 - Insulation (clearance)

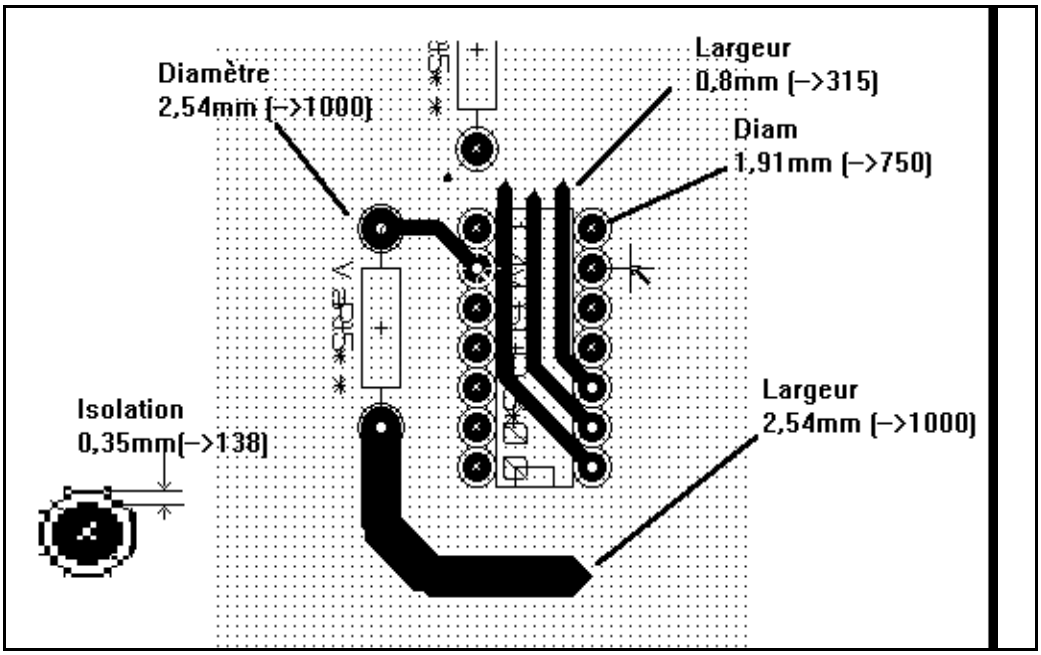
| Unité | CLASS 1 | CLASS 2 | CLASSE3 | CLASS 4 | CLASS 5 |
|---|---|---|---|---|---|
| mm | 0,70 | 0,5 | 0,35 | 0,23 | 0,20 |
| 1/10mils | 270 | 200 | 140 | 90 | 80 |

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

### 8.3 - Some typical combinations

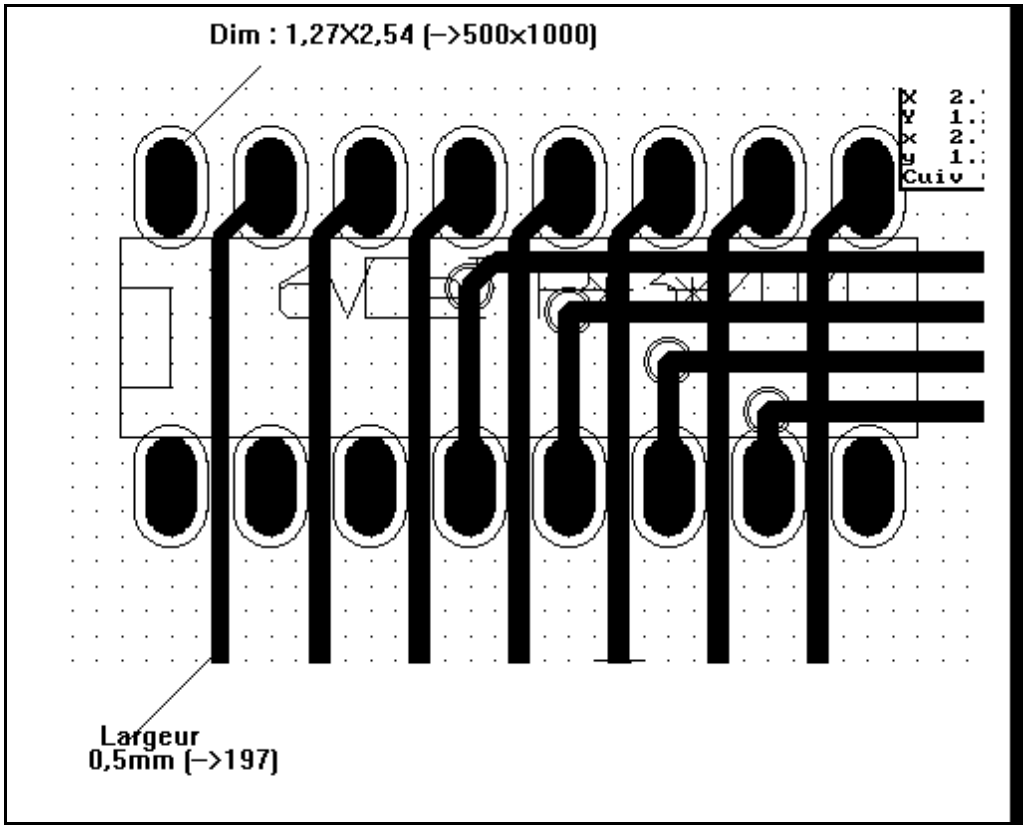
#### 8.3.1 - 'Rustic'

- 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).




### 8.3.2 - '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).




## 8.4 - Manual routing

Manual routing is recommended, because it is the only method offering control of routing priorities. For example, is is preferable to start by routing power tracks, making them wide and short and keeping analog and digital supplies well separated. Then 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.

## 8.5 - Creating the copper zones

Copper zones must be created when all the routing is finished.
***After a route modification, zones must be deleted and recreated***.
Pads of the net must be already connected by standard tracks. Do not expect connecting pads through the zone.
This is because:

- All board caracteristics (pads, board edges) must be known.
- D.R.C. does not test zones (because of time computation problems).

Copper zones (Usually ground and power planes) are usually attached to a net.
In order to create a copper zone one must:

- Sélect the net (Hightlight this net), to attach the zone to this net.
- Create the zone limit (If no, All the board will be filled.)
- Fill the zone from a starting point.

A zone is always created allof a piece, to insure there is not any unconnected copper block.

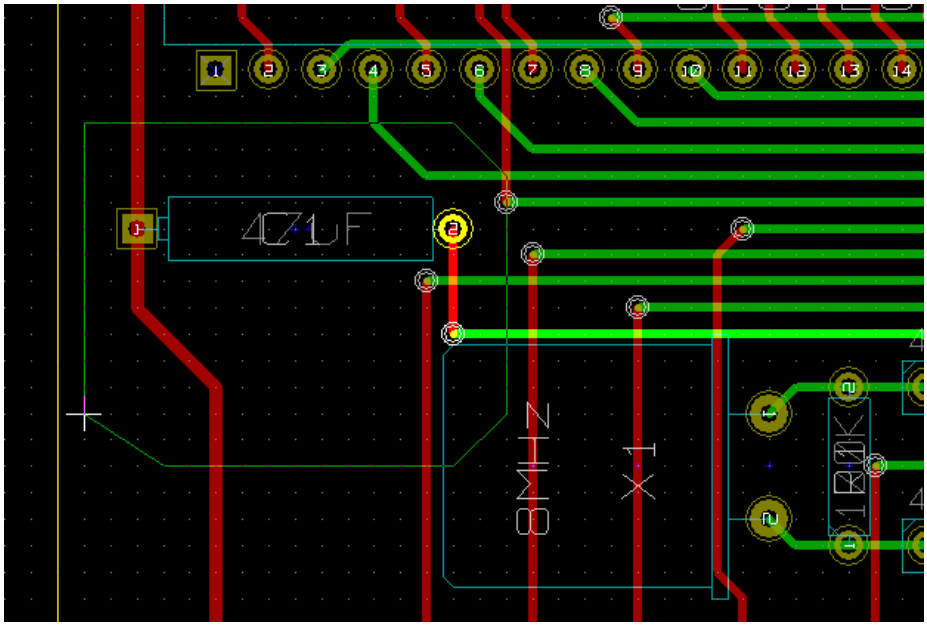
### 8.5.1 - Net selection:

Use the tool , and click on a pad connected to the net, which must be highligted.

### 8.5.2 - Creating a zone:

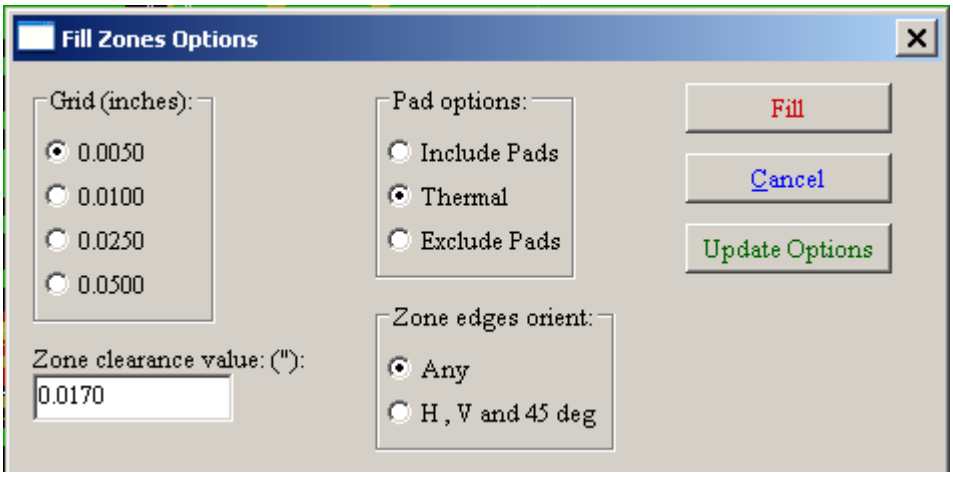
#### *8.5.2.1 - Creating the zone limits:*

Use the tool .
Choose the layer for the zone.
Draw the zone limit, on this layer.
This zone limit is a polygon, created by a click (left button) for each corner.
A double click ends the polygon.
The polygon will be automaticaly 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.
Here is a zone limit created (polygon in thin line):



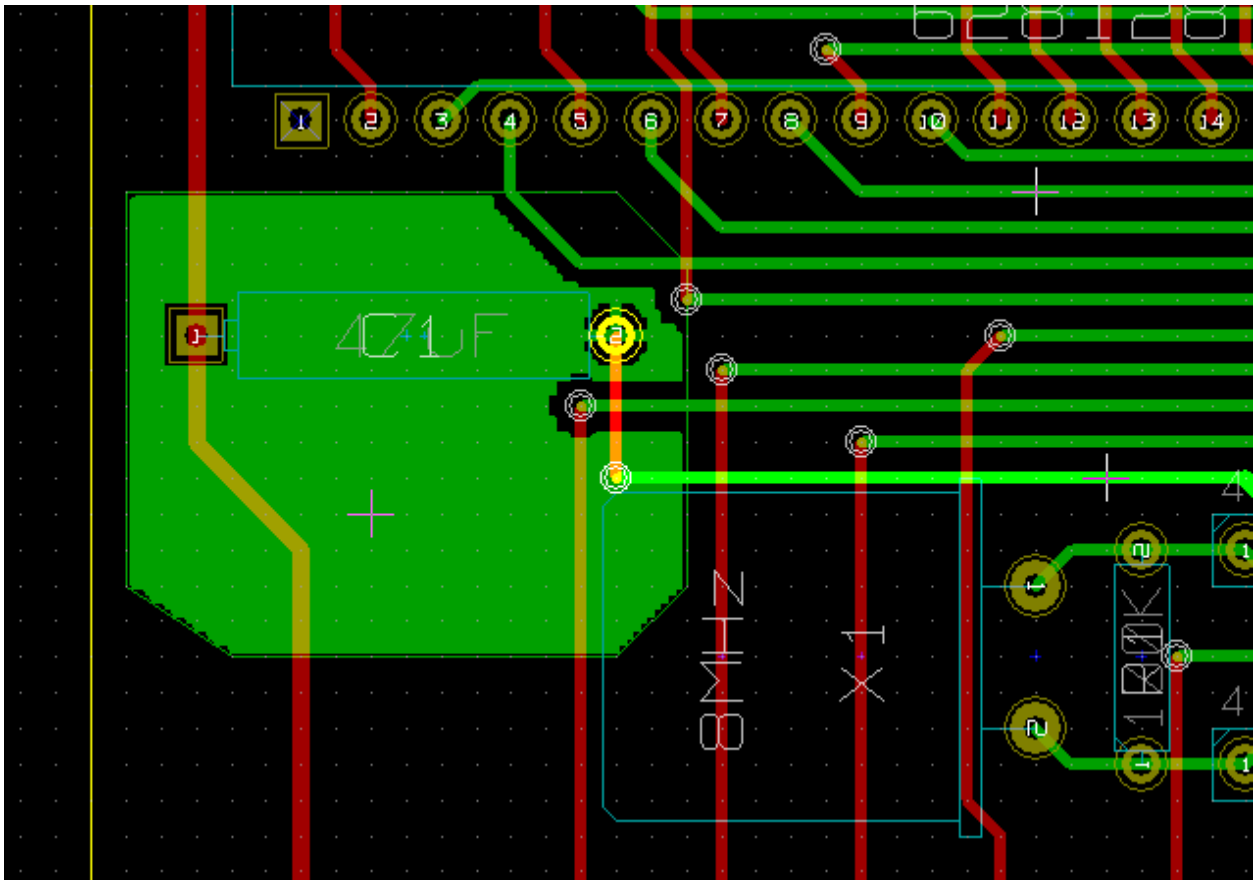

### *8.5.2.2 - Filling the zone:*

Tje filling zone is initiated from a starting point (Mouse cursor).
In order to do it, when the edge zone is finished, put the mouse cursor on a starting point  for the filling. This point can be inside or outside the polygon, and on a free spot. Click the right mouse button. This menu will be showed:

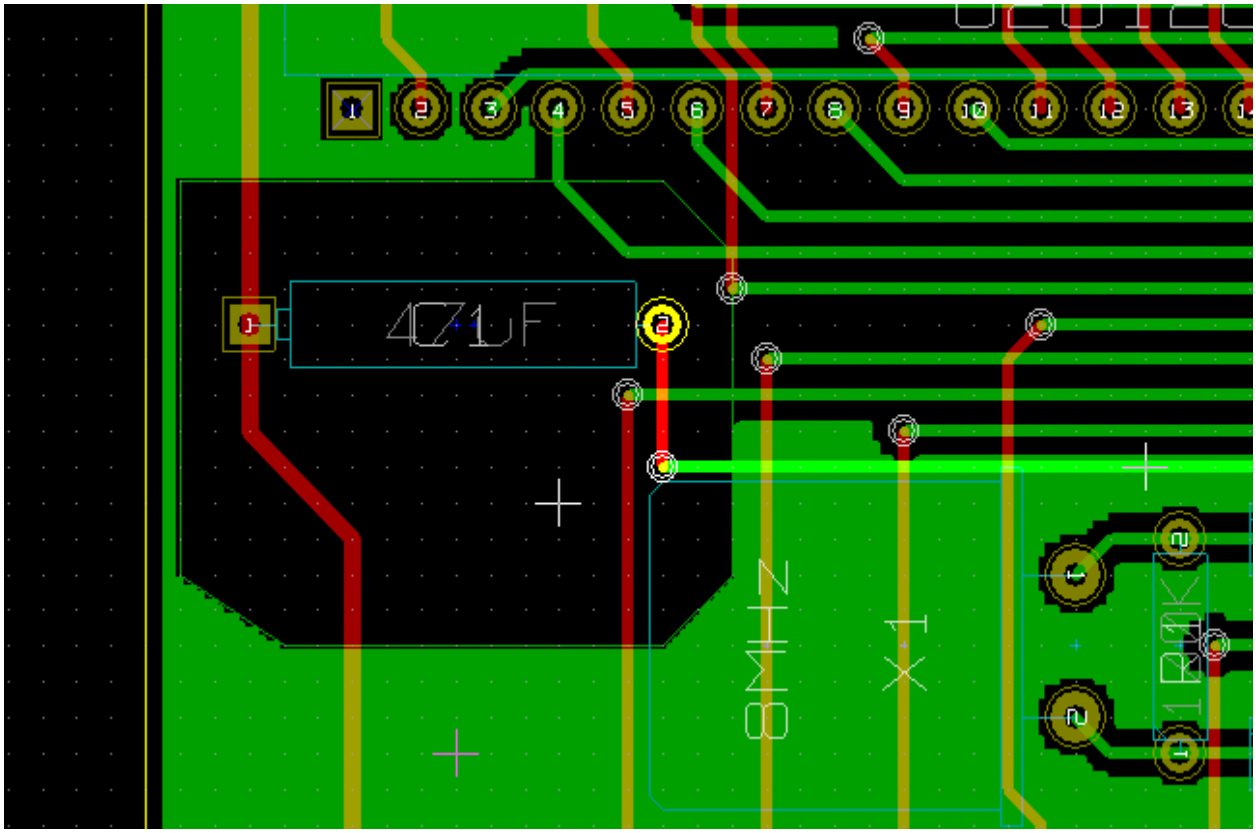



Activate the "**Fill**" button

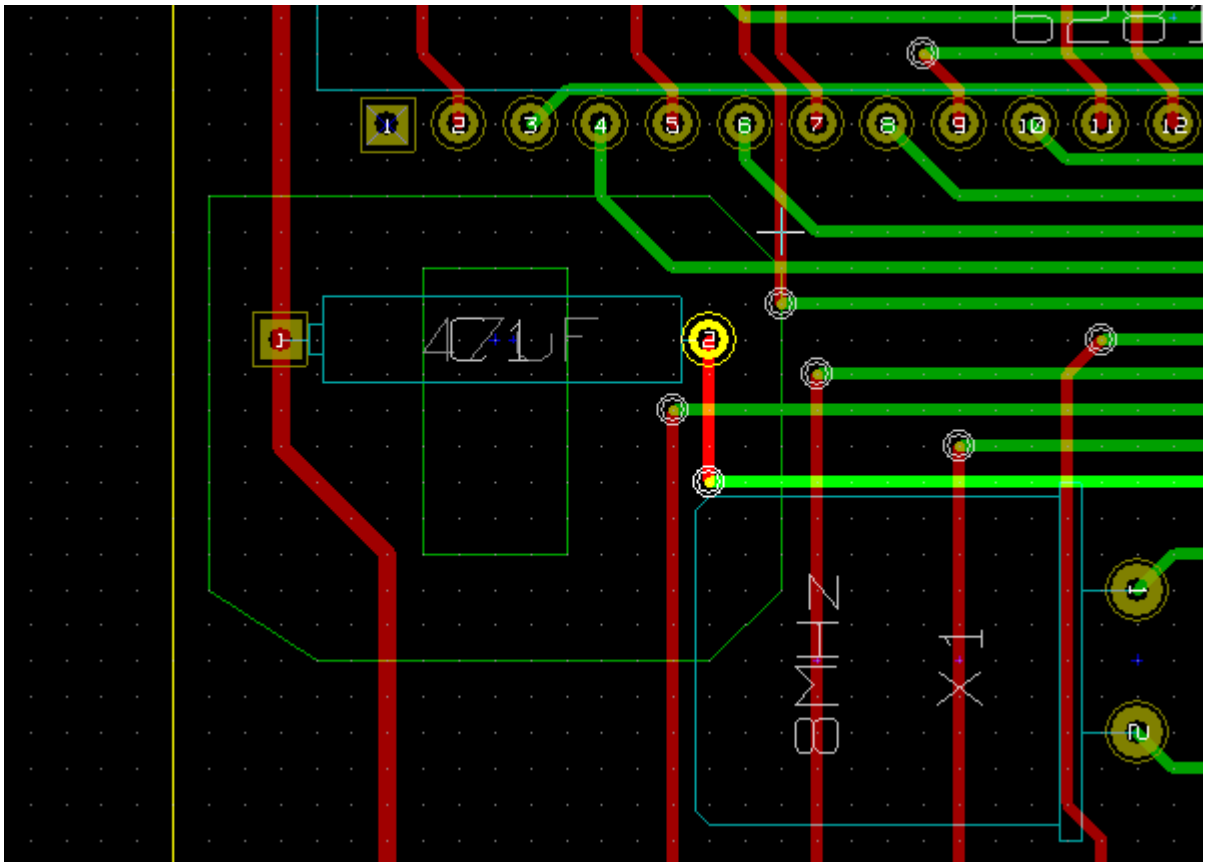
Here is the filling result for a starting point ***inside*** the polygon:



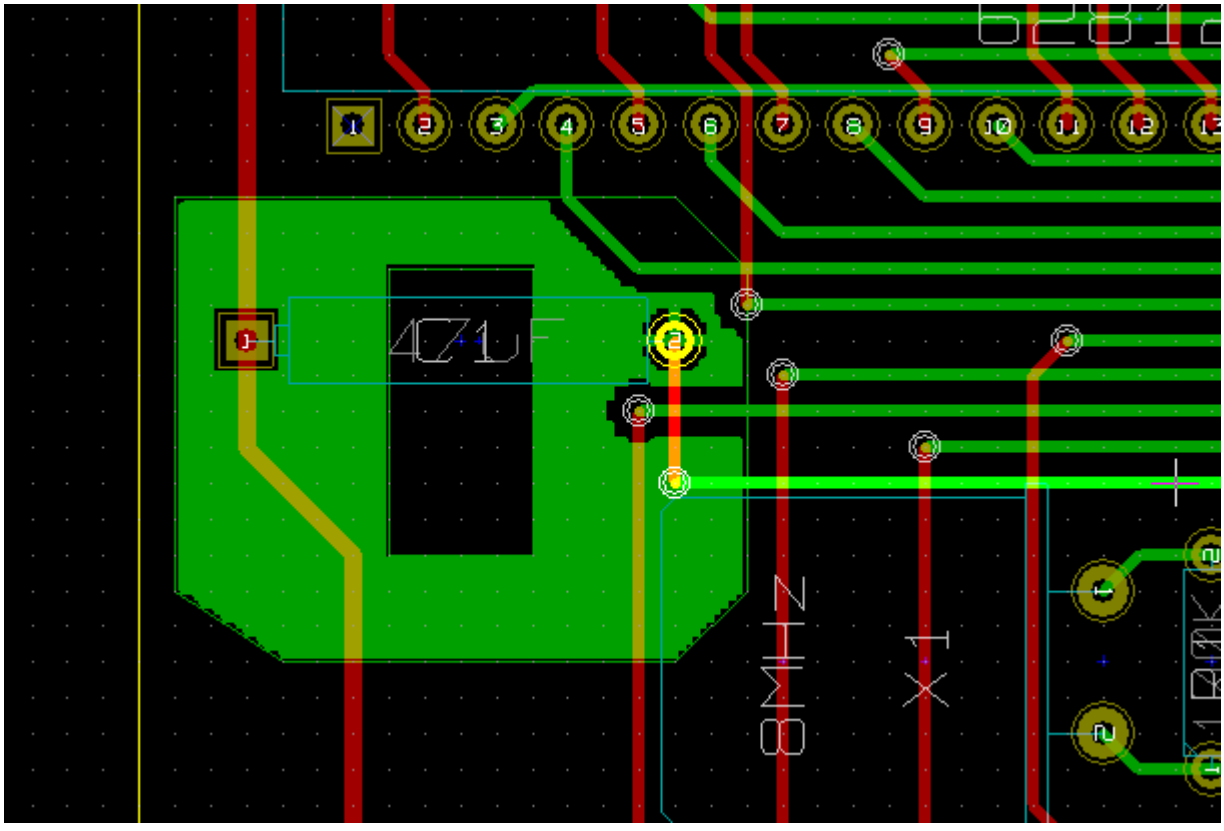

Here is the filling result for a starting point ***outside*** the polygon:



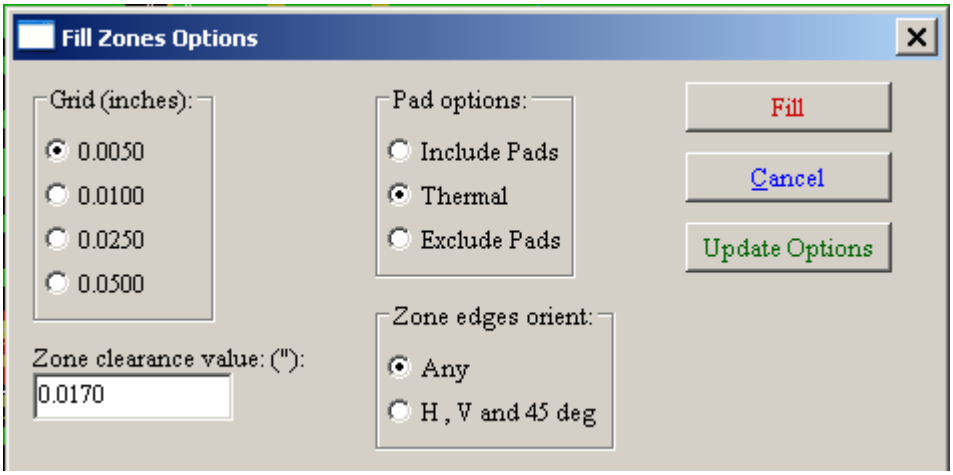
The polygon is a frontier for filling.
***Note:***
you can use many polygons to create a complex limit. Here is an example:




Here is the filling result for a starting point inside the large polygon and ***outside*** the small polygon:



## 8.5.3 - Fill Options:




One must choose:

- The working grid for filling.
- The clearance.
- How the pads are drawn inside the zone (or connected to this zone).

### *8.5.3.1 - Working grid for filling.*

The smaller the filling grid is, the better the filling is.
However, the filling is made with horizontal and vertical track segments, if the grid is small, board files can be very larges (several Mo.).
A 0.01 inch grid is a good compromise.

### *8.5.3.2 - Clearance*

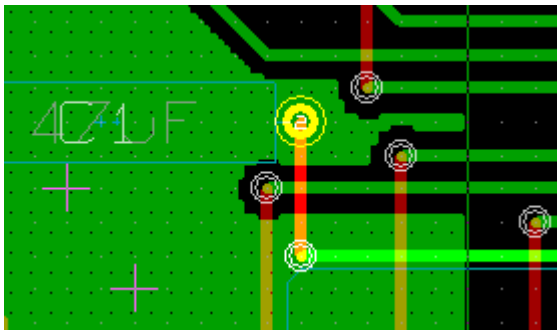
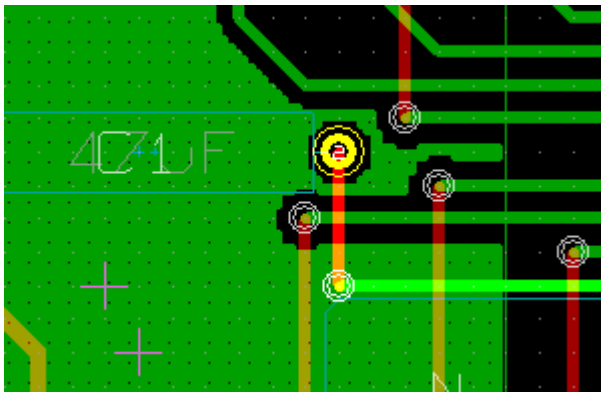
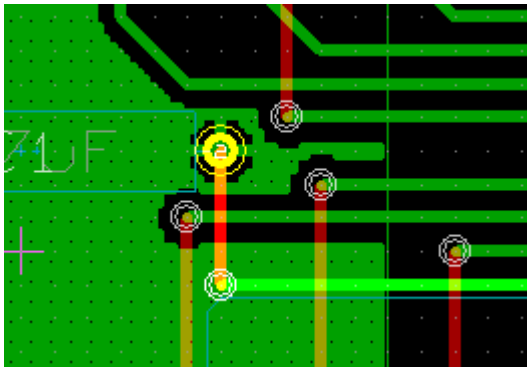
A good choice is a grid a bit bigger than the routing grid.

### *8.5.3.3 - Pad options*

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

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

Here is the result for the 3 options:

| | |
|---|---|
|  | Include Pads |
|  | Exclude Pads |
|  | Thermal relief.<br>Pad is connected by 4 track segments.<br>The segment width is the current value used for the track width. |

# Headings:

9 - Preparation of files for circuit fabrication
9.1 - Final preparations
9.2 - Final DRC test:
9.3 - Generating files for phototracing
9.3.1 - GERBER format:
9.3.2 - HPGL Format:
9.3.3 - POSTSCRIPT Format:
9.4 - Adjusting the clearance for the solder stop mask:
9.5 - Generating the drill file(s)
9.6 - Generating the cabling documentation:
9.7 - Generation of file(s) for automatic component insertion:
9.8 - Advanced tracing options:

## 9 - Preparation of files for circuit fabrication

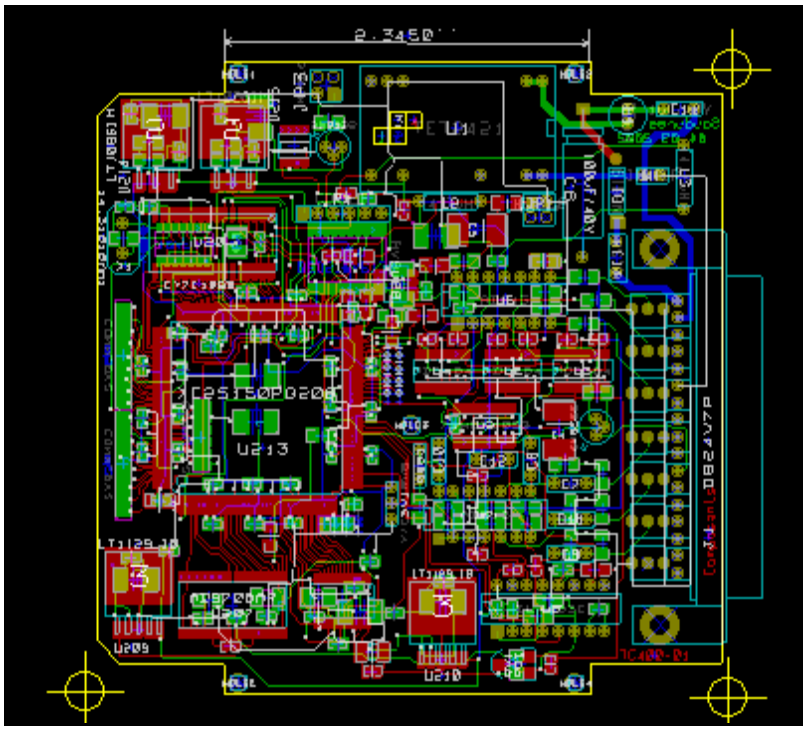
***Note:***
All of the files generated are placed in the working directory, i.e., in the same directory as the file **xxxxxx.brd** of the printed circuit board.

### 9.1 - Final preparations

**It is necessary to:**

- Indicate the layer (e.g., '*component*' and '*copper*') and project names by placing appropriate text upon each of the layers.
- All text on the '*copper*' (sometimes called '*solder*' or '*bottom*') side 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 of these elements, except the ground planes, which have been omitted for better visibility:

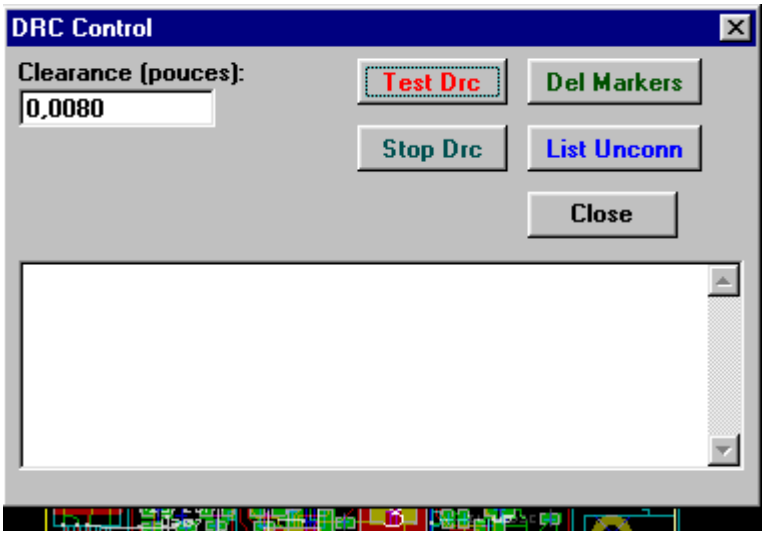


A colour key for the 4 copper layers has also been included:

## 9.2 - Final DRC test:

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

Press the button to launch the DRC dialogue:

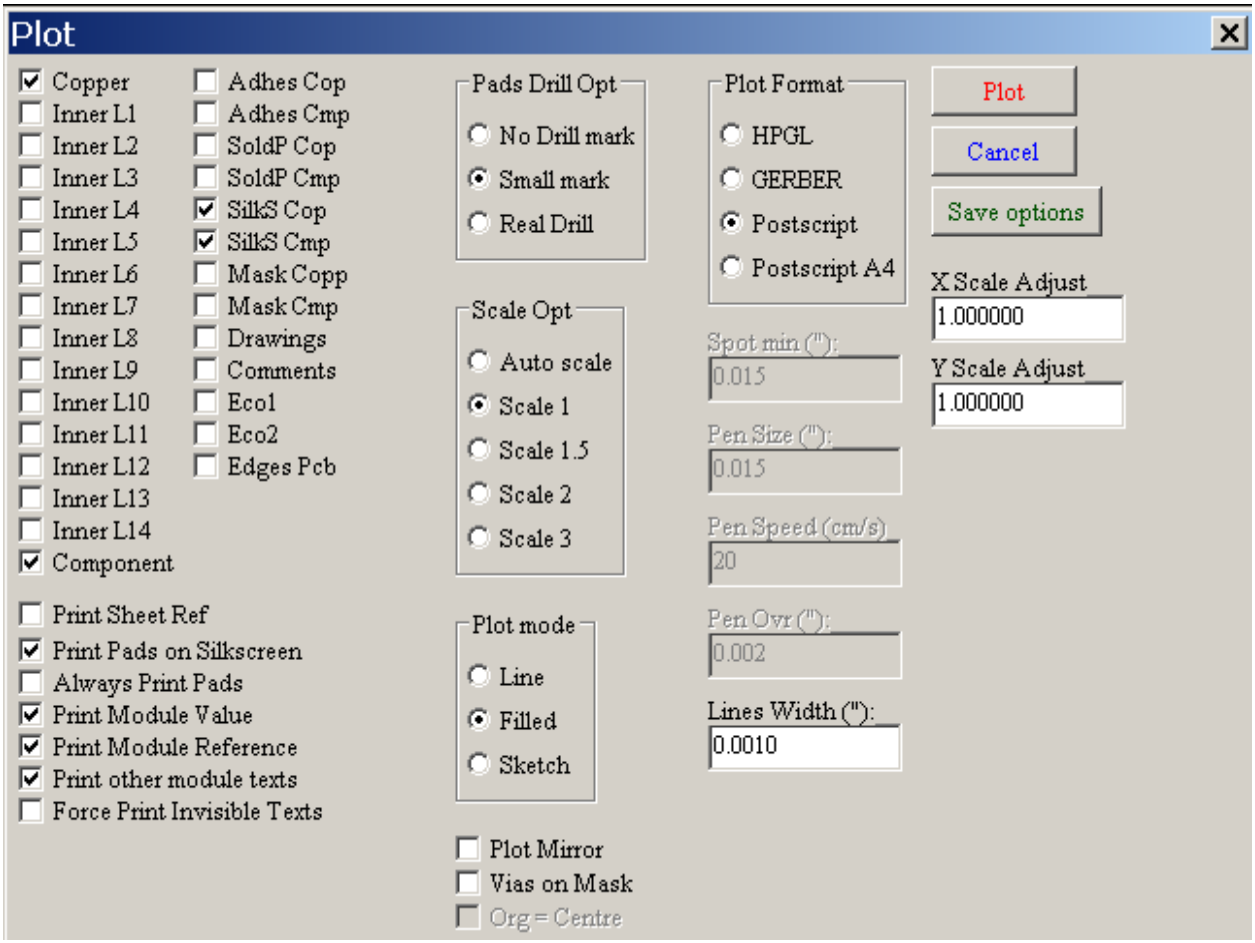



and then press .

This final check will prevent any unpleasant surprises...

## 9.3 - Generating files for phototracing

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.

### 9.3.1 - 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 (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 checkboxes.

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

- **xxxx.copper.pho** for the copper side.
- **xxxx.cmp.pho** for the component side.
- **xxxx.silkscmp.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.

### 9.3.2 - HPGL Format:

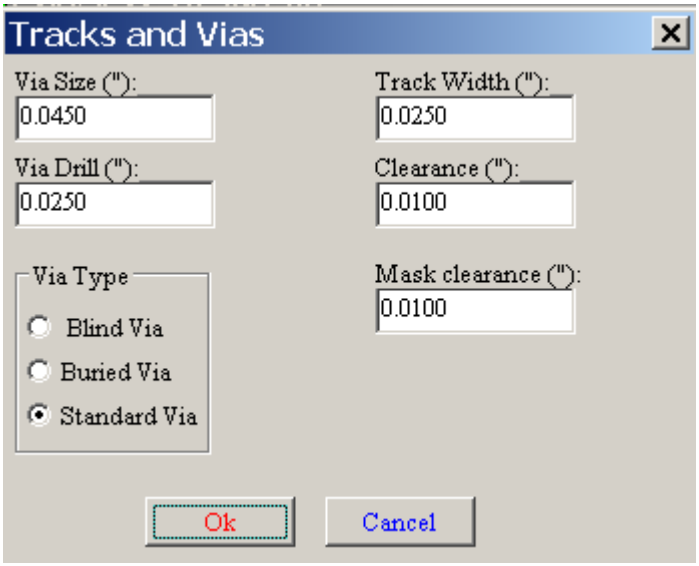
The standard extension for the output files is **.plt**.
The tracing can be done at user-selected scales and can be mirrored.
The **Print Drill Opt** list offers the option of pads that are filled, drilled to the correct diameter or drilled with a small hole (to guide hand drilling).
If the **Print Sheet Ref** option is active, the sheet cartridge is traced.

### 9.3.3 - 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.

## 9.4 - Adjusting the clearance for the solder stop mask:

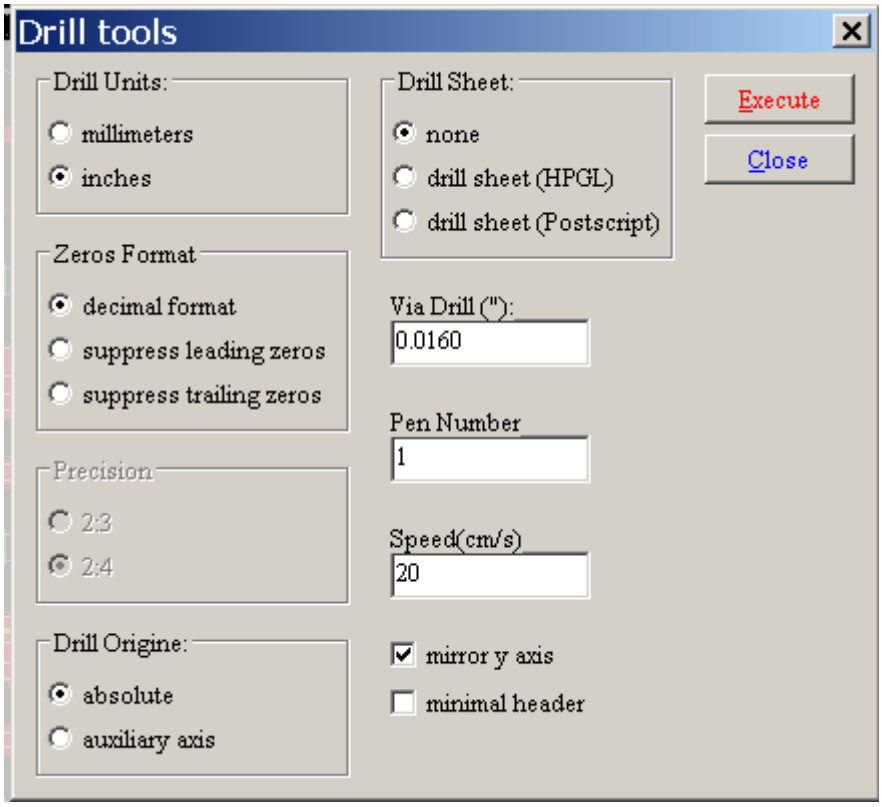
The menu option for this is accessed via **Dimensions/Tracks and Vias**:




and entering the desired setting in the bottom-right option (**Mask clearance**); the default value is 0.01 inches.

## 9.5 - Generating the drill file(s)

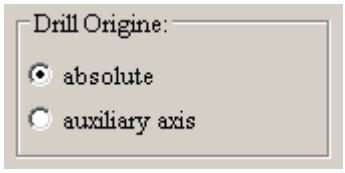
The creation of a drill file xxxxxx.drl following the **EXCELLON** standard is always necessary.
One can also produce an optional drill plan, which will be in HPGL (xxxxxx.plt) or POSTSCRIPT (xxxxxx.ps) format. However, this is only occasionally useful, as an additional check.
The generation of these files is controlled via the **Postprocess/Create Drill file** menu selection**:**




The default drill diameter for vias (For vias without a special drill value) should be defined here. For the HPGL tracing of the drill plan, it is possible to define the no. and speed of the pen used.

***Coordinate origin:***

The dialog box is:




- **Absolute**: absolue coordinates are used
- **Auxiliary axis**: coordinates are relative to the auxiliary axis (use the tool (right toolbar) to put it in good place.

## 9.6 - Generating the cabling documentation:

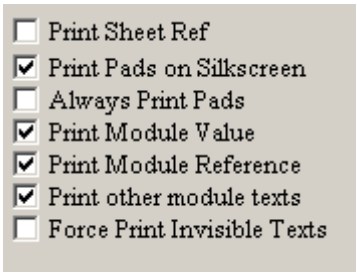
To produce these files, the component and copper silkscreen layers can be traced. Usually, just the component-side silkscreen markings are sufficient for cabling a PCB. If the copper-side silkscreen is used, the text it contains should be mirrored in order to be readable.

## 9.7 - Generation of file(s) for automatic component insertion:

This option is accessed via the **Postprocess/Create Cmp file** menu option. However, no file will be generated unless at lesat 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.

## 9.8 - 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 options are:

| | |
|---|---|
| ***Print Sheet Ref*** | Trace sheet outline and the cartridge. |
| ***Print Pads on Silkscreen*** | Enables/disables printing of pad outlines on the silkecreen 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. |
| ***Always Print Pads*** | Forces tracing of all pads on ALL layers. |
| ***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. |

# Headings:

10 - ModEdit: Managing LIBRARIES
10.1 - Overview of ModEdit
10.2 - ModEdit:
10.3 - ModEdit user interface:
10.4 - Main toolbar in Modedit:
10.5 - Creating a new module:
10.6 - Creating a new library:
10.7 - Saving a module to the active library:
10.8 - Transferring a module from one library to another:
10.9 - Saving all the modules of a circuit in the active library:
10.10 - Documentation for library modules:
10.11 - Documenting libraries – recommended practice:

## 10 - ModEdit: Managing LIBRARIES

### 10.1 - Overview of ModEdit

PCBNEW simultaneously maintains several libraries. Thus, when a module is loaded, all of the 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 shall now describe.

**ModEdit** enables the **creation and 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, ...);
- Editing the associated documentation (description, keywords).;

as well as the **maintenance of the active library**:

- 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:

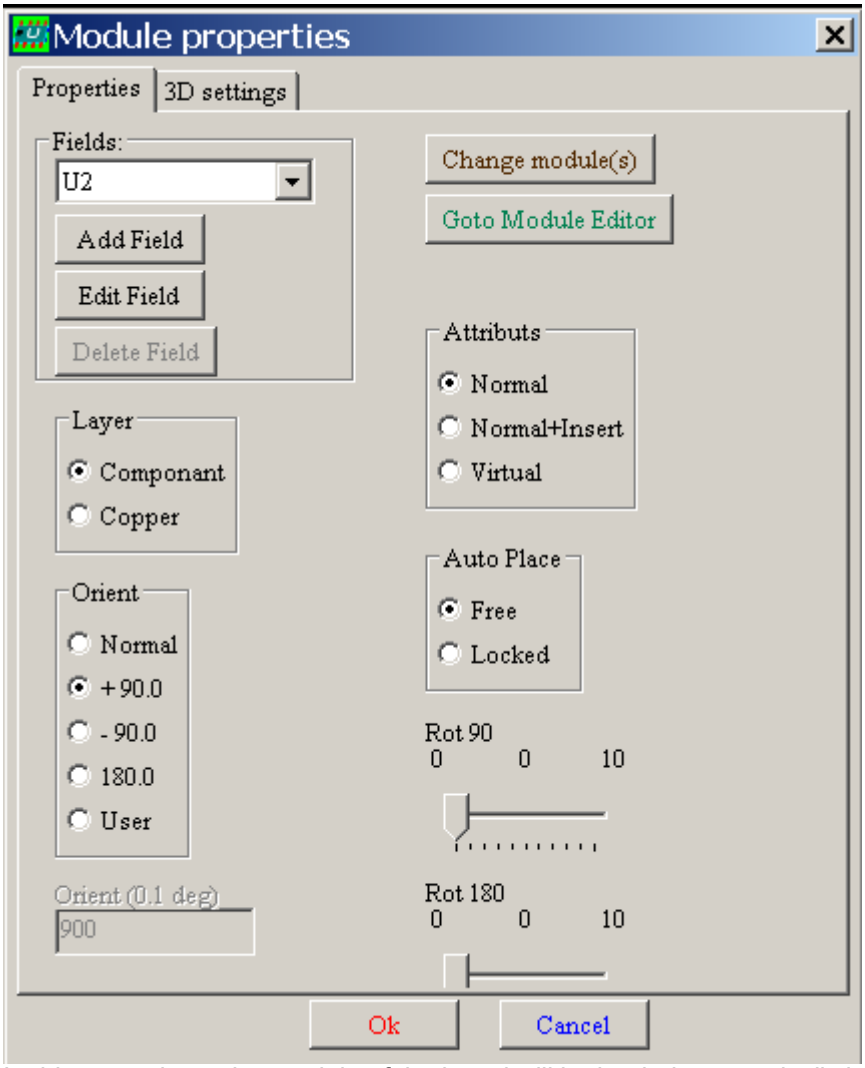
- The library itself (file extension **.lib**)
- The associated documentation (file extension **.dcm**)

The documentation file is systematically regenerated after each modication of the corresponding .lib file; in this way it can easily be recovered if lost. The documentation file serves to accelerate access to module documentation.

### 10.2 - ModEdit:

The Module Editor can be accessed in two ways:

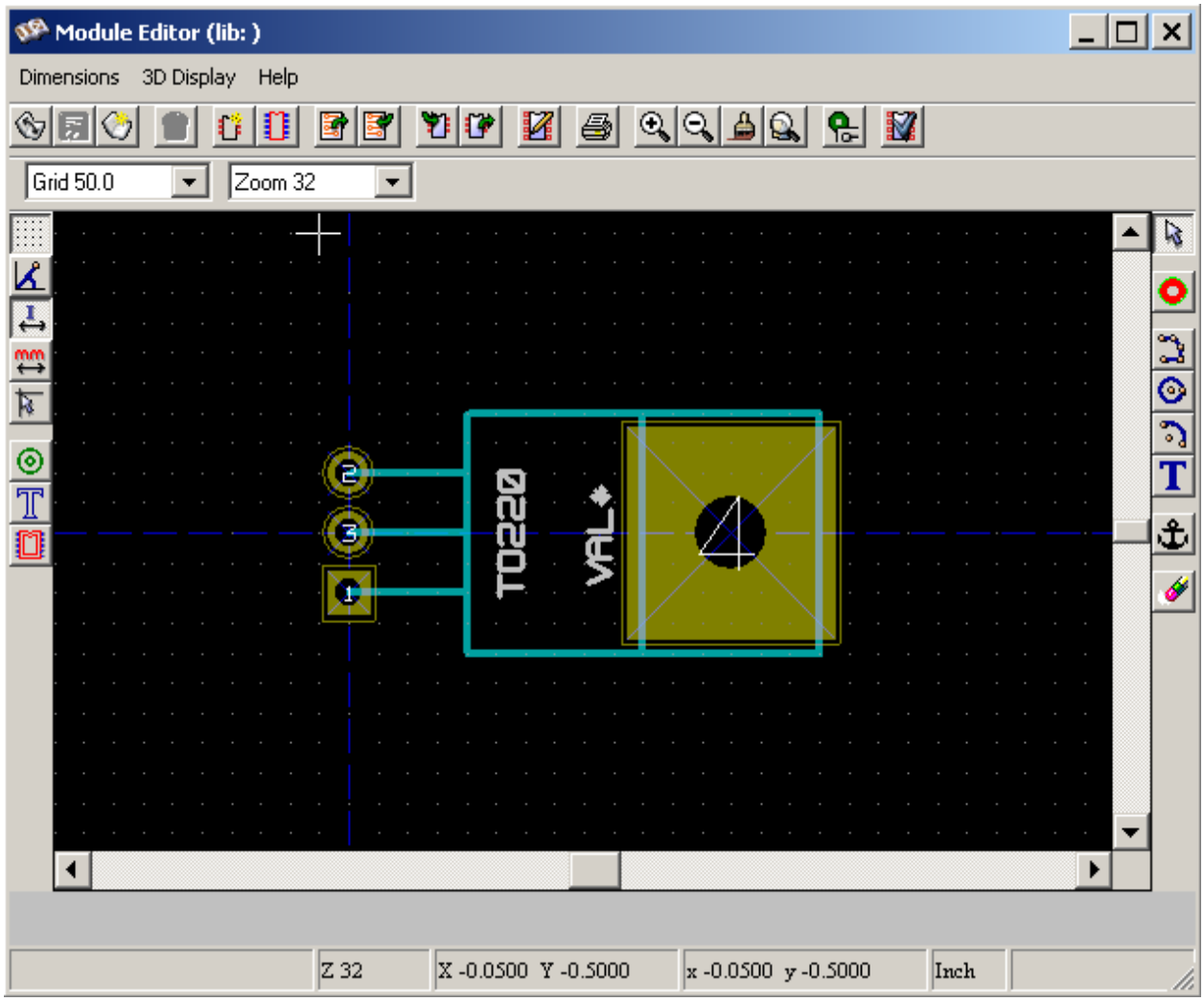
- Directly, via the icon in the main toolbar of Pcbnew;
- In the edit dialogue for the active module (see figure below; accessed via the context menu), there is a button (***Goto Module Editor***).




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

## 10.3 - ModEdit user interface:

Calling ModEdit causes the following window to appear:




## 10.4 - Main toolbar in Modedit:

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 dialogue 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, it will replace the corresponding module on the printed circuit 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 and with orientation 0. (***Translator's note***: I do not find this clear: previously imported from the pcb? or from a library (see below)? So what's the difference with loading it from a library?) |
| | 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 *kicad/modules*). |
| | Not used. |
| | Call the print dialogue. |
| | Standard zoom commands. |
| | Call the pad editor. |
| | Not used. |

## 10.5 - Creating a new module:

Allows 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...)

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, an alternative and often quicker method is the following:

1. Load the similar module ( , , ou )
2. **Modify the reference field** to the name of the new module.
3. Edit and save the new module.

## 10.6 - Creating a new library:

The creation of a new library is done using:

, in which case the file is created by default in the library directory;
or by

, in which case the file is created by default in the working directory.
A file-choosing dialogue 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.

## 10.7 - Saving a module to the active library:

The action of saving a module (thereby modifying the file of the active library) is performed using this icon .
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 also recommended to edit either the reference or value field text to the name of the module as identified in the library.

## 10.8 - Transferring a module from one library to another:

Select the source library( ).
Load the module ( ).
Select the destination library ( ).
Save the module ( ).
You may also wish to delete the source module: reselect the source library then delete the old module ( then .).

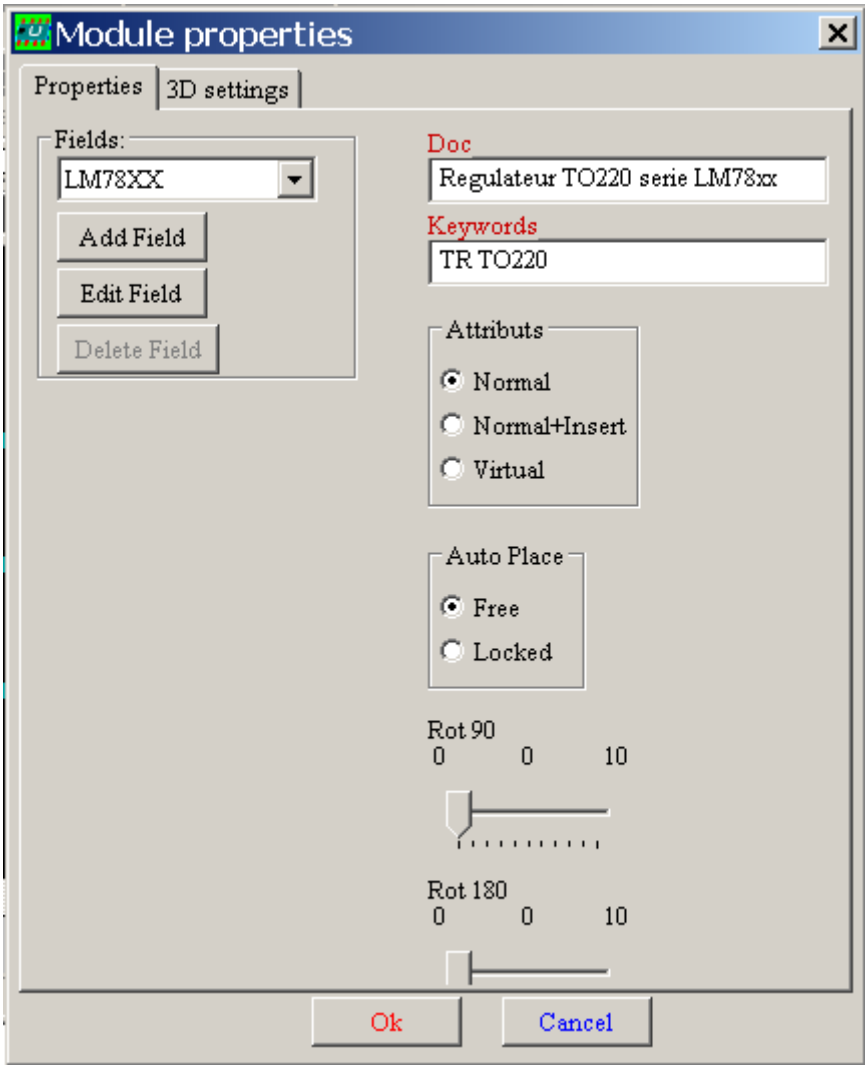
## 10.9 - Saving all the modules of a circuit in the active library:

It is posible 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 circuit, 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 explained below.

## 10.10 - 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 pinout variants of a TO92 package? The **Module Properties** dialogue offers a simple solution to this problem.




This dialogue accepts:

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

The description is displayed with the component list in CVPCB and, in PCBNEW, it is used in the module selection dialogues.
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 dialogue box. Thus, entering the text "=CONN" will cause the display of the list of modules whose keyword lists contain the word CONN.

## 10.11 - Documenting libraries – recommended practice:

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

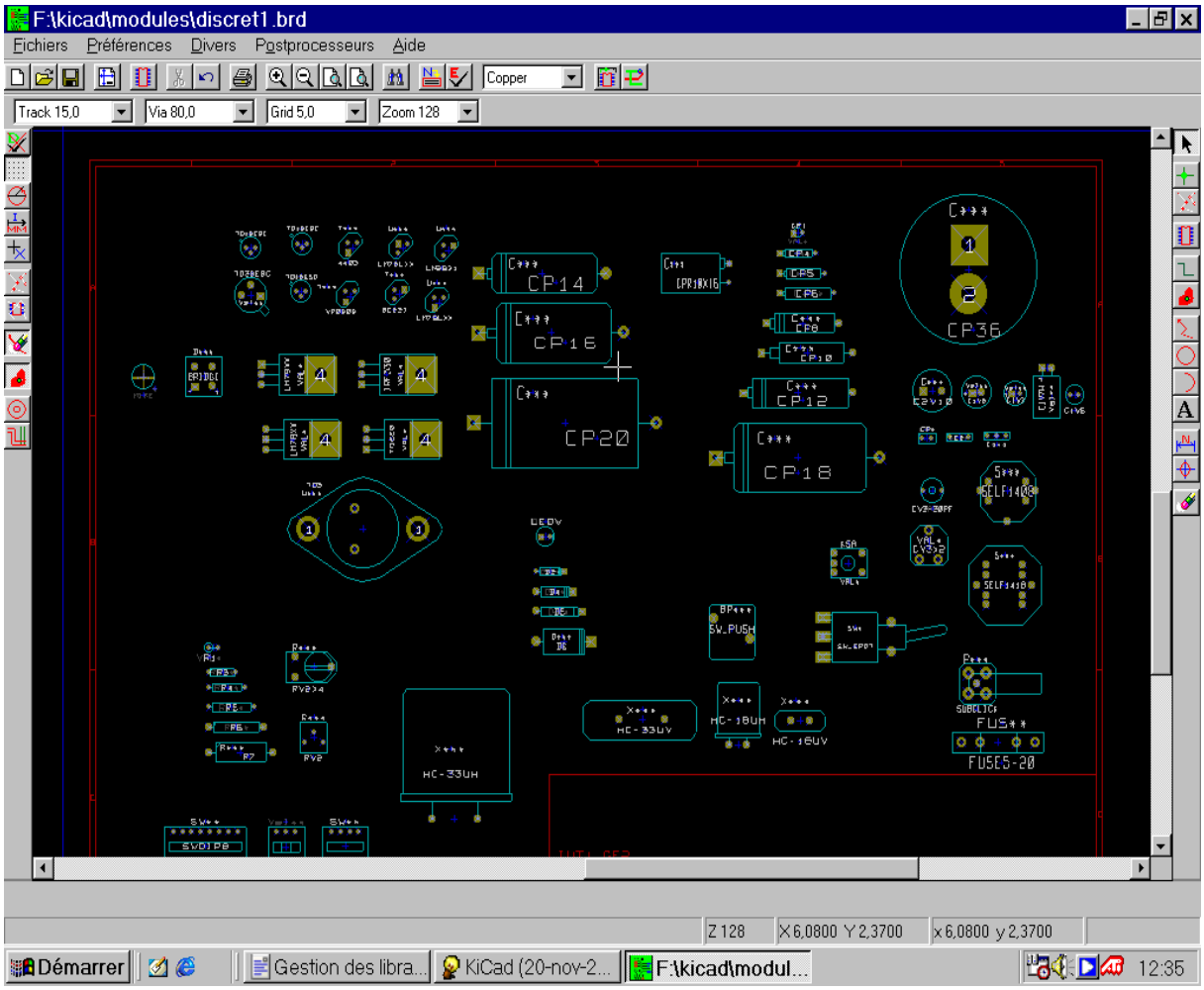
- Create a circuit board in A4 format, in order to be able to print easily to scale (scale = 1).
- Create the modules that the library will contain on this circuit board.
- The library itself will be created with the ***File/Archive footprints/Create footprint archive*** command.

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.

**Pcbnew**

It is generally a good idea to make different libraries for different kinds of components (connectors, discretes,...), 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.

# Headings:

11 - ModEdit: editing modules
11.1 - Overview.
11.2 - Module elements.
11.2.1 - Pads.
11.2.2 - Contours.
11.2.3 - Fields.
11.3 - Starting ModEdit and selecting a module to edit.
11.4 - Module Editor Toolbars:
11.4.1 - Righthand Toolbar - editing
11.4.2 - Lefthand Toolbar – display options
11.5 - Context Menus.
11.6 - The Module Properties dialog.
11.7 - Creating a new module.
11.8 - Adding and editing pads.
11.8.1 - Adding a pad.
11.8.2 - Setting pad properties.
11.9 - Information about automatic placement for a module.
11.10 - Attributes.
11.11 - Documenting modules in a library.
11.12 - Managing 3-dimensional visualisation
11.13 - Saving a module to the active library

## 11 - ModEdit: editing modules

### 11.1 - Overview.

**ModEdit** is used for **editing and creating 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,...)
- Editing the associated documentation (description, keywords).

### 11.2 - Module elements.

A module is the physical representation of the part to be inserted, but it must also link to the schematic. Each module comprises three different elements:

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

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

#### 11.2.1 - 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 is by matching pin and pad numbers that Pcbnew links pins and pads for the module.

#### 11.2.2 - 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.

### 11.2.3 - Fields.

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

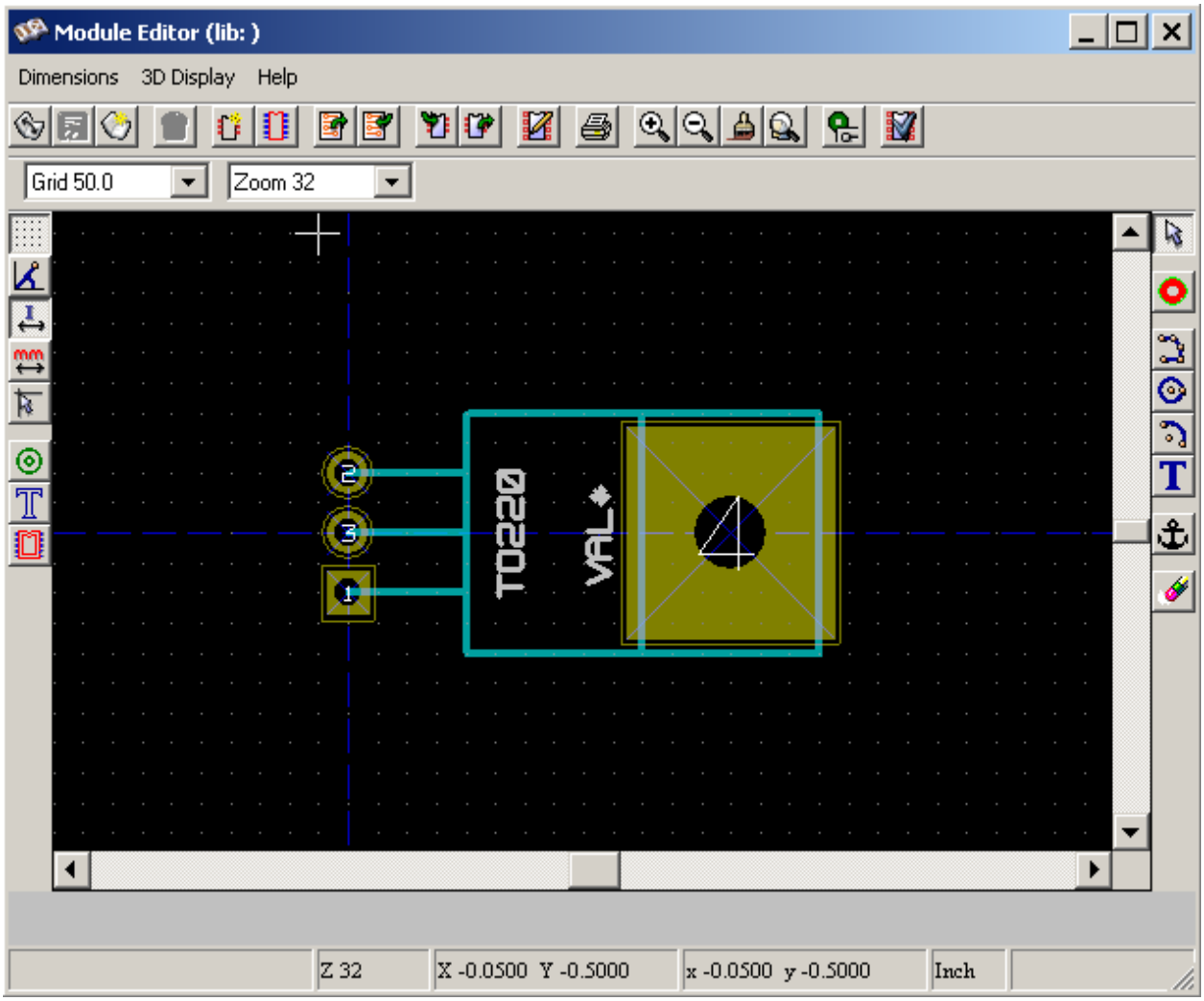
## 11.3 - Starting ModEdit and selecting a module to edit.

ModEdit can be started in two ways:

- Directly via the icon of 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 'Goto Module Editor' button. If this option is used, the module from the board will be loaded into the editor, for modification (or saving).

## 11.4 - Module Editor Toolbars:

Calling ModEdit will launch a window like this:



### 11.4.1 - Righthand Toolbar - editing

| | |
|---|---|
| | This toolbar contains tools for:<br>- Placing pads.<br>- Adding graphic elements (contours, text).<br>- Positioning the anchor.<br>- 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. |

### 11.4.2 - Lefthand Toolbar – display options

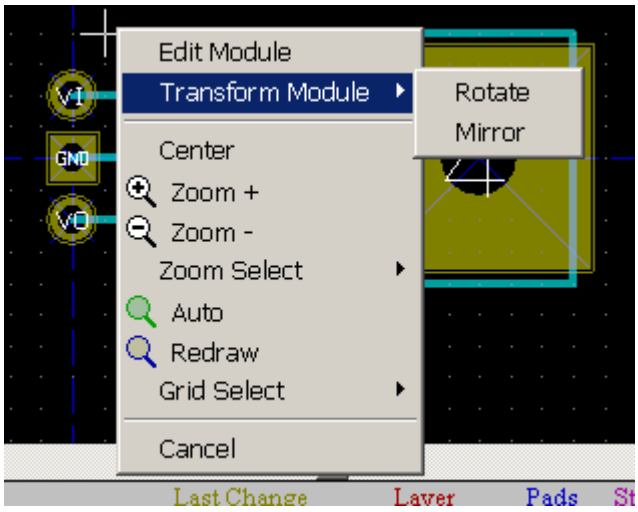
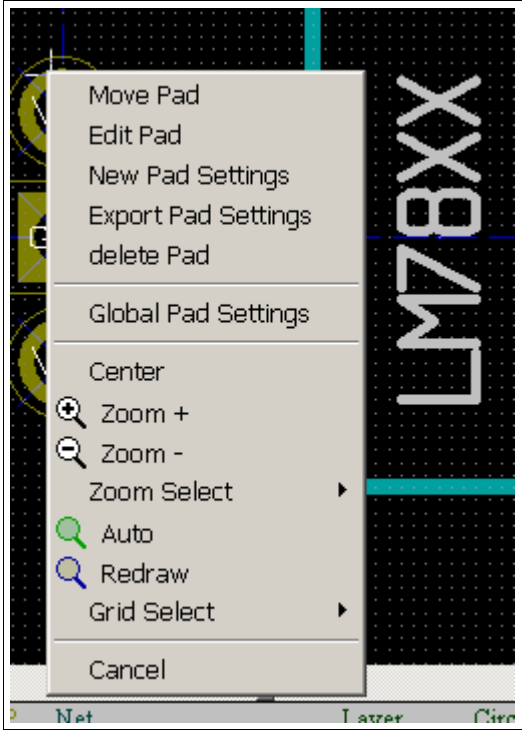
| | |
|---|---|
| | These tools manage the display options in ModEdit |

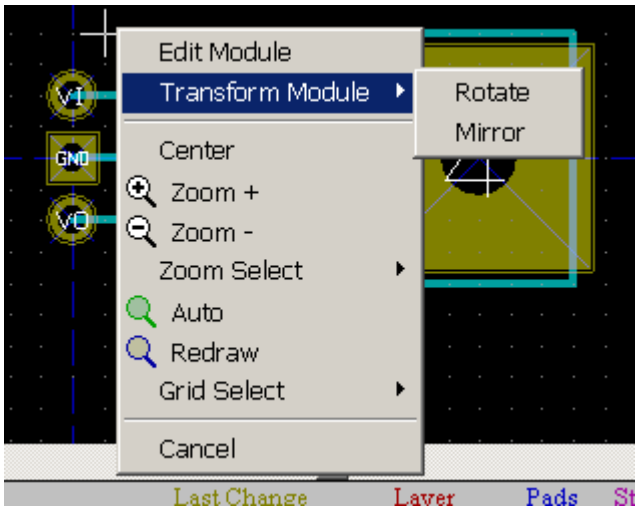
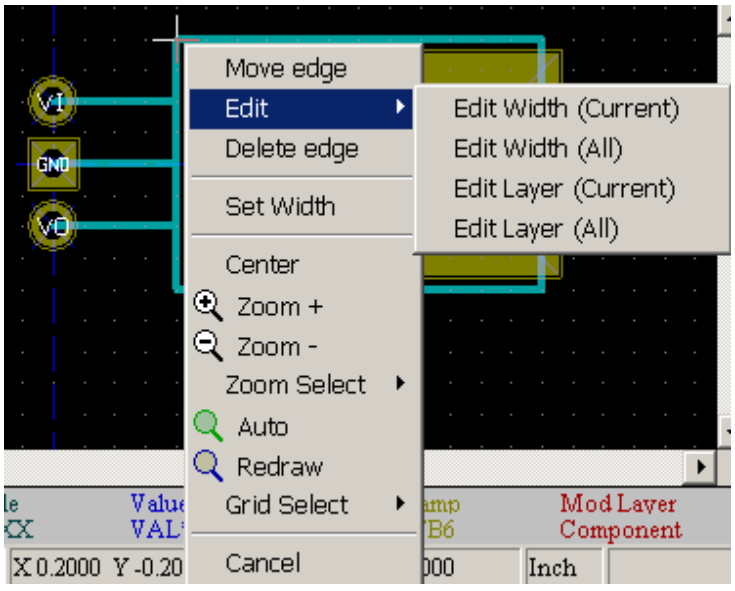
These options are active when the button is pressed:

| | |
|---|---|
| | 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. |

## 11.5 - Context Menus.

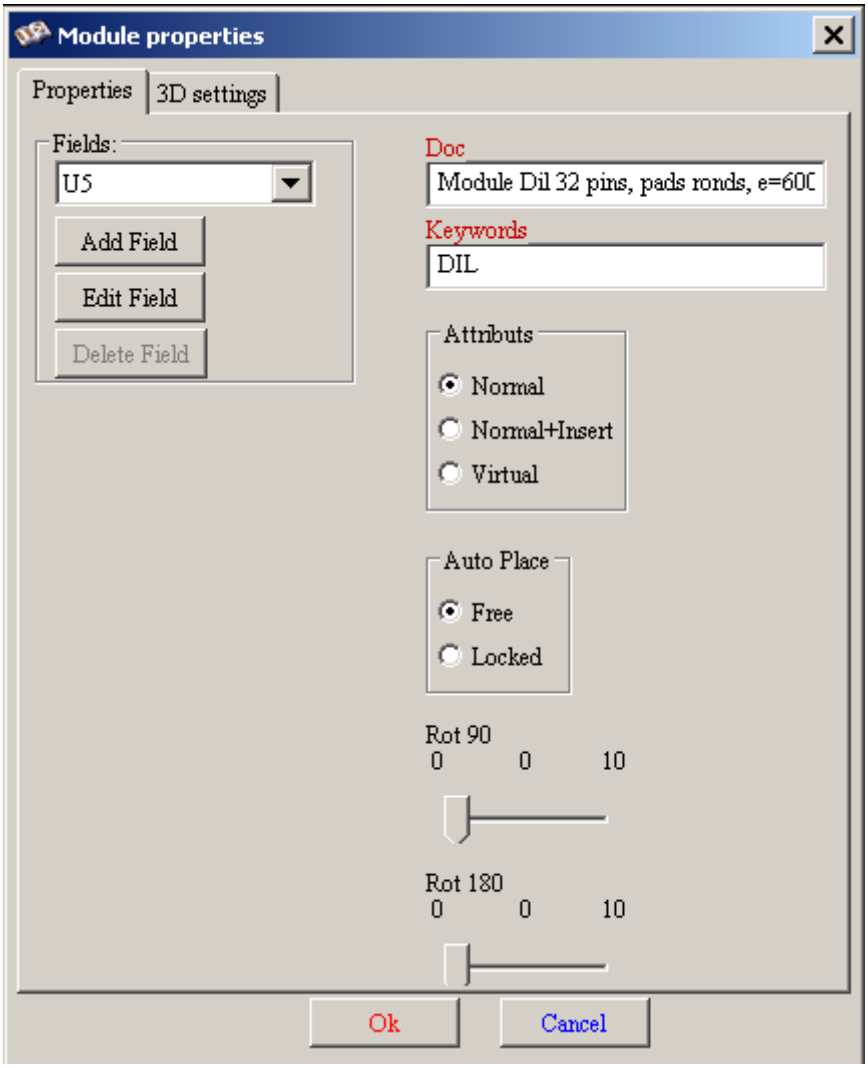
The right mouse button calls up menus that depend upon the element beneath the cursor:

| | |
|---|---|
|  | The context menu for editing module parameters. |
|  | The context menu for editing pads. |

| | |
|---|---|
|  | The context menu for editing module parameters. |
|  | The context menu for editing graphic elements. |

## 11.6 - The 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.

## 11.7 - Creating a new module.

This icon is used to create a new module.
The name of the new module will be requisted (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:

- Contours (and possbily graphic text).
- Pads.
- 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:

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

## 11.8 - 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.

### 11.8.1 - Adding a pad.

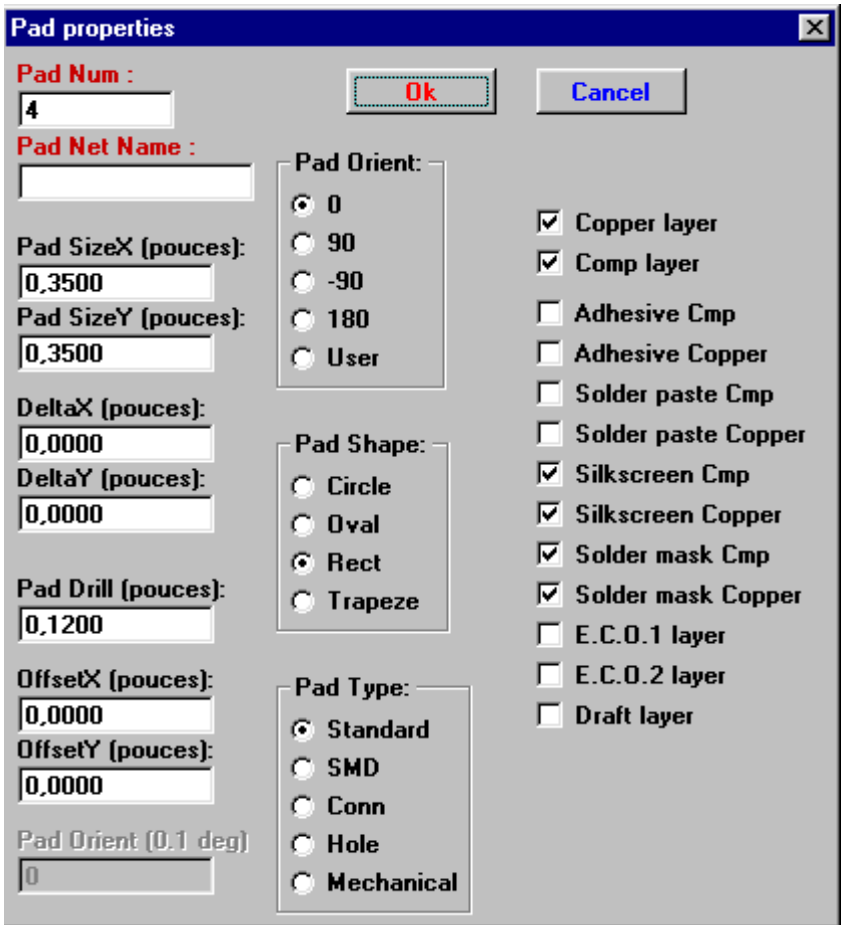
Select the tool in the right-hand toolbar.
Pads can be added by clicking in the desired position with the left mouse button.
Their properties are predefined in the *Pad properties* menu.
Don't forget to enter the **pad number**.

### 11.8.2 - Setting pad properties.

This can be done in three ways.

1. Selecting the tool from the horizontal toolbar.
2. Clicking on an existing pad and selecting '**Edit Pad**'. The pad's settings can then be edited.
3. 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.

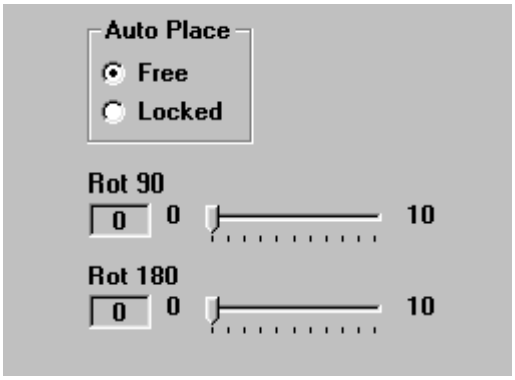
In the first two cases, the following dialog window will be displayed:




Care should be taken to define correctly the layers to which the pad will belong.
In particular, although the 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.
***Note 1:***
For smd modules of the VQFP/PQFP type, which have rectangular pads on all four sides, i.e., 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.
***Note 2:***
Rotations of -90 or -180 are only required for trapezoidal pads used in microwave modules.

## 11.9 - Information about automatic placement for a module.

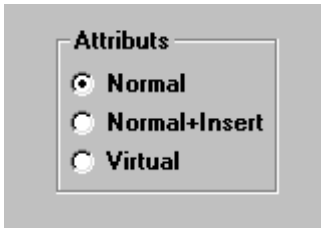
If the user wishes to exploit the the full capabilities of the autoplacement functions, it is necessary to define the allowed orientations of the module (**Module Properties** dialog).




Usually, rotation of 180 degrees **(Translators note: French said 190 degrees...)** is permitted for resistors, non-polarised 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 vallue of 0 makes rotation impossible, 10 allows it completely, and any intermediate value represents a penalty for 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).

## 11.10 - Attributes.

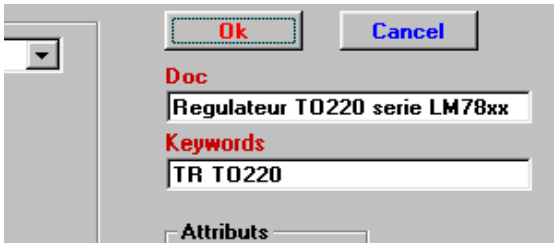
The attributes section is the following:




- **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 inductances created by a particular track shape (as sometimes seen in microwave modules).

## 11.11 - 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 pinout variants of a TO92 module?
The **Module Properties** dialog offers a simple solution to this problem.




It allows:

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

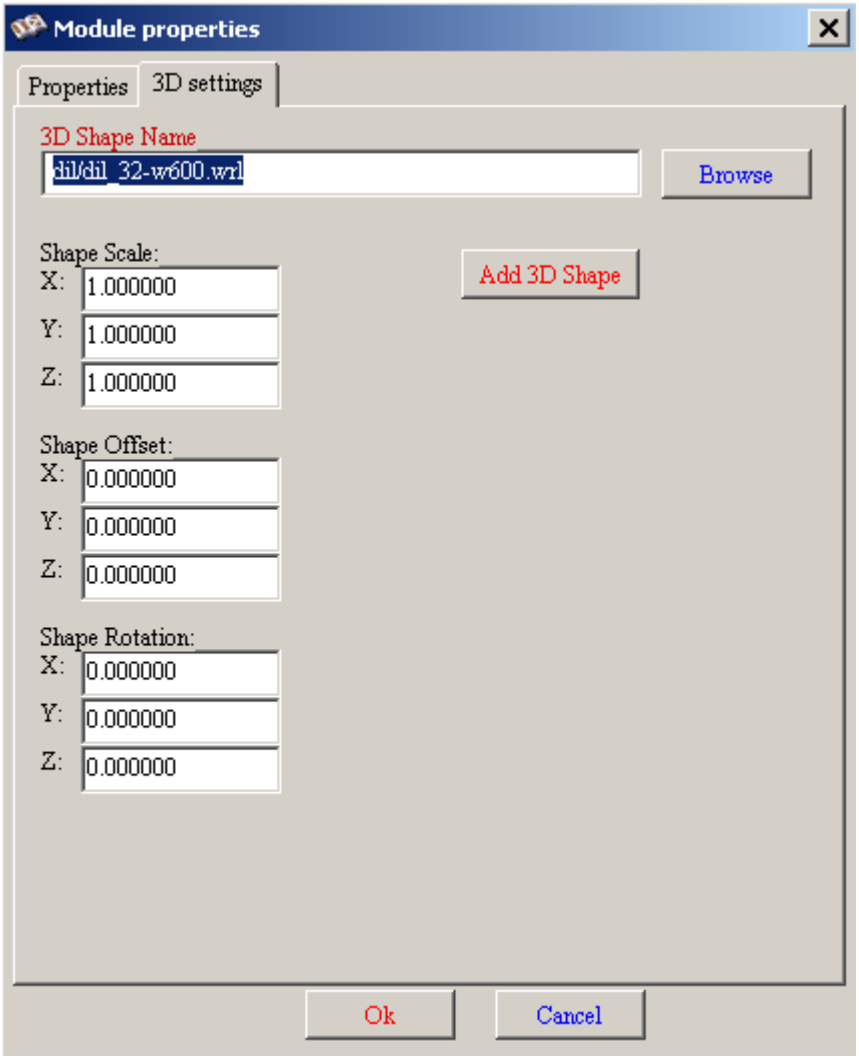
The comment line is displayed with the component list in CVPCB and in the module selection menus in PCBNEW.
The keywords can be used to restrict searches to those parts possessing the given keywords.

Thus, while using the load module command ( 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".

## 11.12 - Managing 3-dimensional visualisation

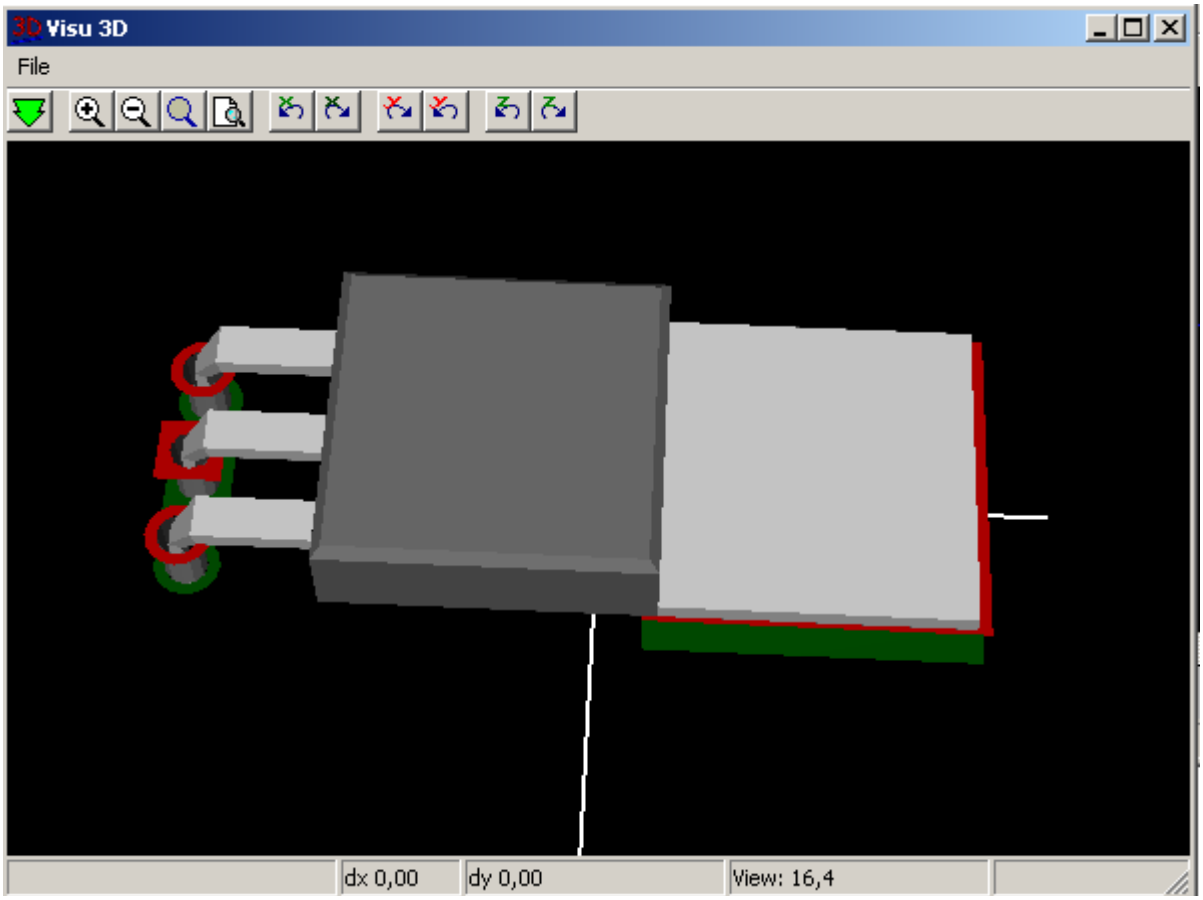
A module may have associated with it a file containing a three-dimensional representation of the component. In order to assocate such a file with a module, select the **3D Settings** tab.
The options panel is the following:




The follwing information should be specified:

- The file containing the 3D representation (created by the 3D modeller **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.wrl, 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).

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.

## 11.13 - Saving a module to the active library

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