

FreePCB User Guide

Supplement #1

Covering Versions
1.328 Through 1.354

Contents

Introduction.....	5	DRC Error X,Y Location Listed.....	14
V1.328 – May 9, 2007.....	6	Run DRC from Context Menu.....	14
Parts List Sort.....	6	V1.339 – Oct 15, 2007.....	15
Trace Selection and Length Display.....	6	Report File.....	15
Net Selection.....	7	Delete Empty Nets.....	15
Panelized Drill File.....	7	V1.344a – May 28, 2008.....	16
Reference Highlight.....	7	Part Value.....	16
Project Save on Router Import.....	7	Footprint Angle.....	16
V1.330 – May 19, 2007.....	8	Solder Mask Pads.....	16
FP Ed. Move Origin.....	8	Paste Mask Pads.....	16
Library Name.....	8	Adhesive Dots.....	17
V1.331 – May 30, 2007.....	9	Inner Pads.....	17
Redo.....	9	All Pad Shapes Flashed.....	17
Group Rotate.....	9	1.348a – Jun 25, 2008.....	18
FP Move Origin Cancellation.....	9	DRC and CAM Settings Saved.....	18
V1.332 – Jun 12, 2007.....	10	Settings Saved in Registry.....	18
Measure Tool.....	10	Pin 1 Position in Report.....	18
Negative Text.....	10	Tab Order.....	18
PNG Render.....	11	1.353 – Aug 13, 2008.....	19
Arrow Cursor.....	11	Combine Nets.....	19
V1.333 – Jun 23, 2007.....	12	Cu Areas Added and Highlighted with Net Select ..	20
Pin # Increment.....	12	Default Library Path.....	20
Redo in FP Ed.....	12	Add Pin Positioning.....	20
PADS2000 Netlist.....	12	Header Warning Message.....	20
V1.335 – Jul 15, 2007.....	13	Cu Layer Change Keystroke Shortcut.....	20
Cu Area Net Reassignment.....	13	1.354 – Oct 13, 2008.....	21
Show Part with Ref Selected.....	13	"Slide" Trace Segment Move.....	21
Set Side Style.....	13	DRC/CAM Annular Width Warning.....	21
V1.337 – Aug 9, 2007.....	14	Project Options Menu.....	22
Full Support for 16 Cu Layers.....	14	FreePCB Revision History.....	23

Introduction

This supplement is an outgrowth of another project I did a few weeks ago where I chronicled, in outline form, FreePCB's revision history based on Allan's *Latest News* forum postings. Soon after finishing that history, I noticed that a high percentage of forum posts were from new users inquiring about or requesting features that exist but were undocumented in the User Guide.

FreePCB has grown and matured a great deal since the last User Guide, PDF version 1.4, was released almost two years ago. It was based on FreePCB version 1.327. Many new features have been added and refined that, until now, have only been documented in the *User Forum* and, as such, remain relatively unknown, especially to new users.

With this supplement, I've attempted to list all of the items that have been added or changes since version 1.327. The changes and additions are listed by revision and only those revisions that involve new or changed features are included; any gaps in the sequence are due to bug fix releases.

Bruce Parham
25 Jan 2009

V1.328 – May 9, 2007

Major new features:

(None Listed)

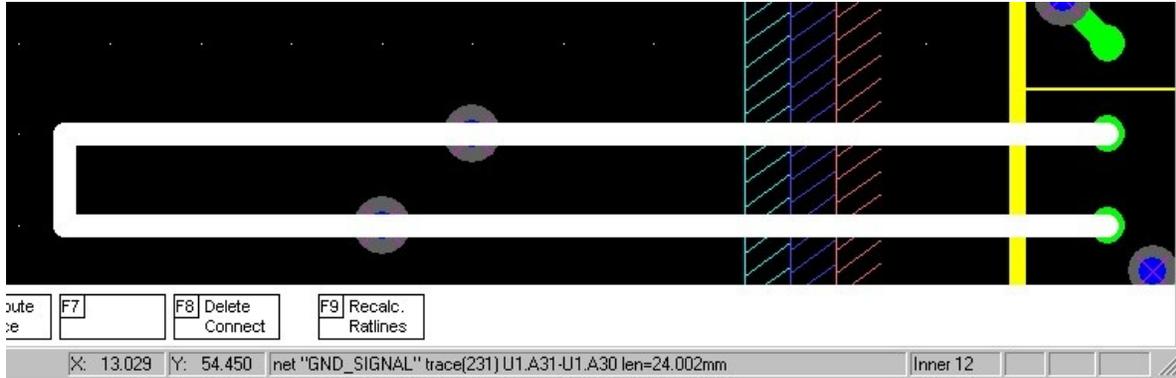
Minor changes:

Parts List Sort

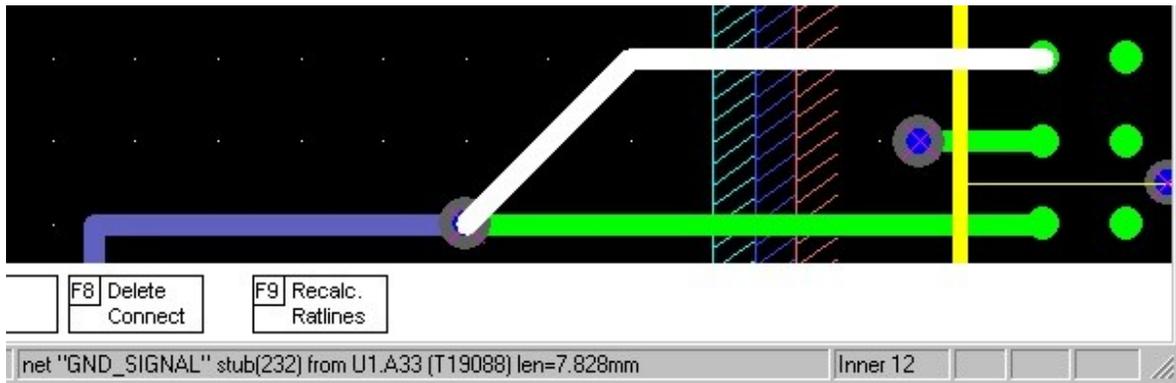
The Project→Parts list can be sorted by clicking on the column headings.

Trace Selection and Length Display

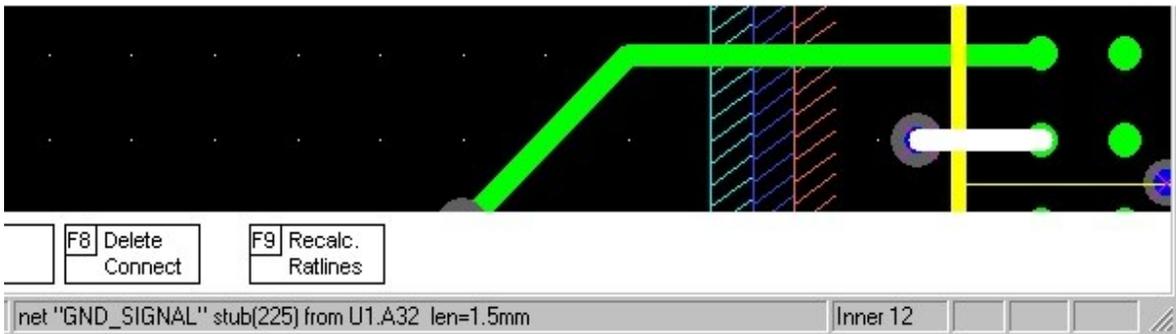
With a trace vertex or segment selected, hitting the **T** key will highlight the entire pin to pin, branch or stub trace as well as display its end points and overall length in the status line.



a: Pin-to-Pin Trace



b: Branch Trace

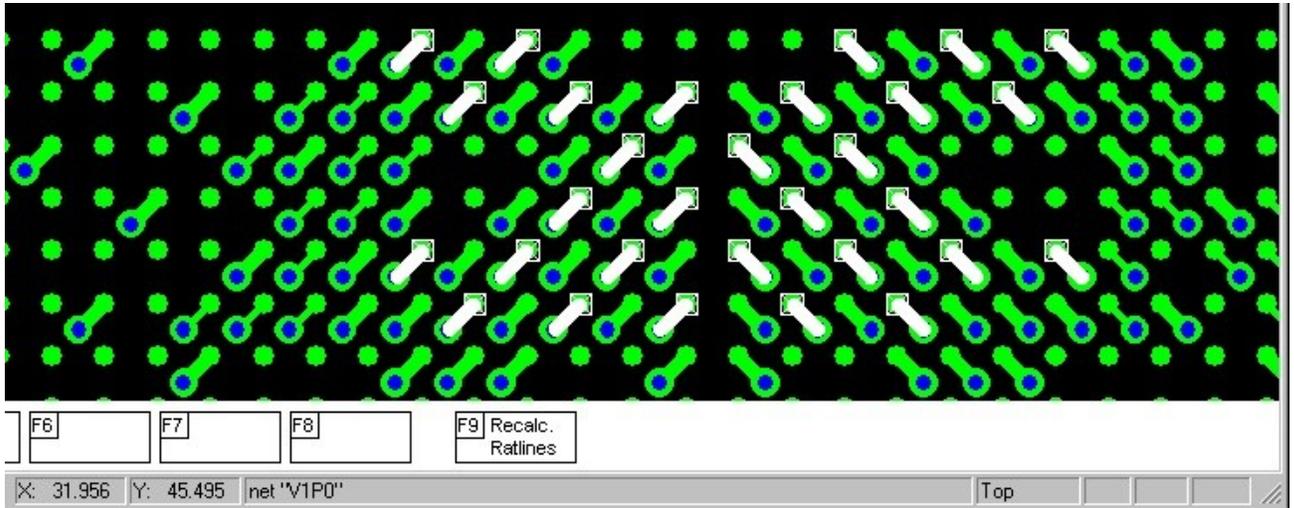


c: Stub Trace

Sad note: as shown above in frame b, selecting a branch trace reports that it is a stub trace in the status line with version 1.354.

Net Selection

With a pin, trace vertex or trace segment selected, hitting the **N** key will highlight the entire net.



Panelized Drill File

Drill files, for panelized projects, are optimized to minimize drill changes,

Reference Highlight

Selecting a part highlights its reference too.



Project Save on Router Import

The currently loaded project is saved to a temporary file when a router session file is imported.

V1.330 – May 19, 2007

Major new features:

FP Ed. Move Origin

The Move Origin command was added to the footprint editor default context menu which allows the origin to be dragged to a new position or set relative to its current position.



Minor changes:

Library Name

Library files use file names only not full path. (It is not clear if this applies to the Parts menu or the Footprint Editor)

V1.331 – May 30, 2007

Major new features:

Redo

In the layout editor, most operations can use **Redo** (Ctl-Y) after an **Undo** (Ctl-Z).

Group Rotate

A selected group of objects can be rotated, in 90° increments, with the **F3** key.



Minor changes:

FP Move Origin Cancellation

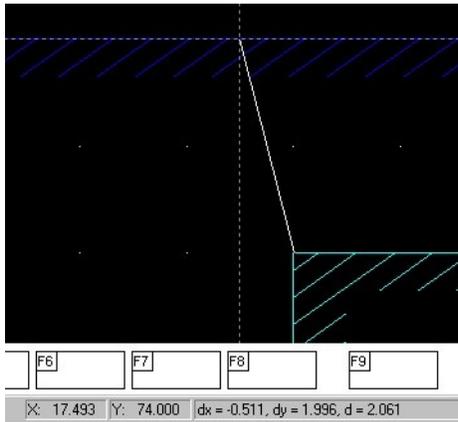
In the footprint editor, while dragging the origin, the operation can be canceled by clicking the Right Mouse Button (RMB).

V1.332 – Jun 12, 2007

Major new features:

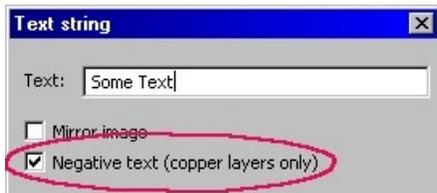
Measure Tool

In the layout editor, the **M** key will start the measurement tool. Place the cross-hairs over the starting point and click the mouse to begin measuring with the elastic ruler. As the ruler end is moved about with the mouse, the status line will show ΔX , ΔY and total distance in the current units. A second mouse click exits the measurement tool.



Negative Text

Text objects placed on signal (Cu) layers can be specified to be negative or positive.

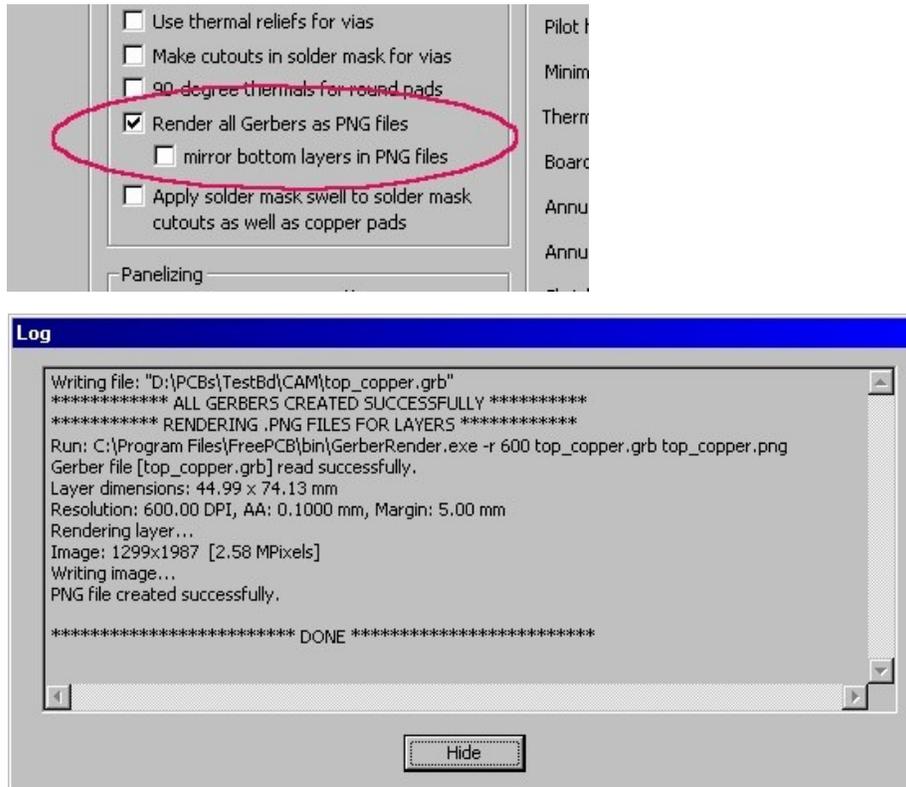


As shown in the Gerber file view below, for visibility, positive text, within a Cu area, is isolated from the surrounding Cu by the current CAM *Cu to Cu-fill Clearance* value. This can produce a large number of small isolated areas and may reduce the effectiveness of the signal plane. Negative text will minimize these issues and the final results will be easier to read.



PNG Render

A check-box has been added to the CAM Files menu that enables generating .PNG graphic files. When used, each newly created Gerber file is, in turn, rendered to a .PNG file using Merlin's single file rendering tool *GerberRender* at a fixed resolution of 600 DPI with 5 mm margins.



See Merlin's (Guillaume Rosanis) **PCB-Tools** documentation for more details about Gerber file rendering. (Available on the FreePCB website Downloads page.)

Minor changes:

Arrow Cursor

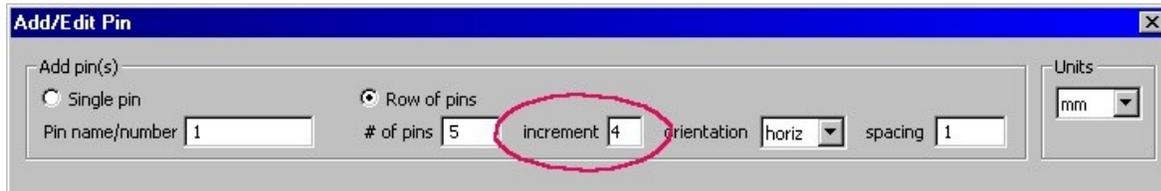
The default arrow cursor is suppressed when cross-hairs are used.

V1.333 – Jun 23, 2007

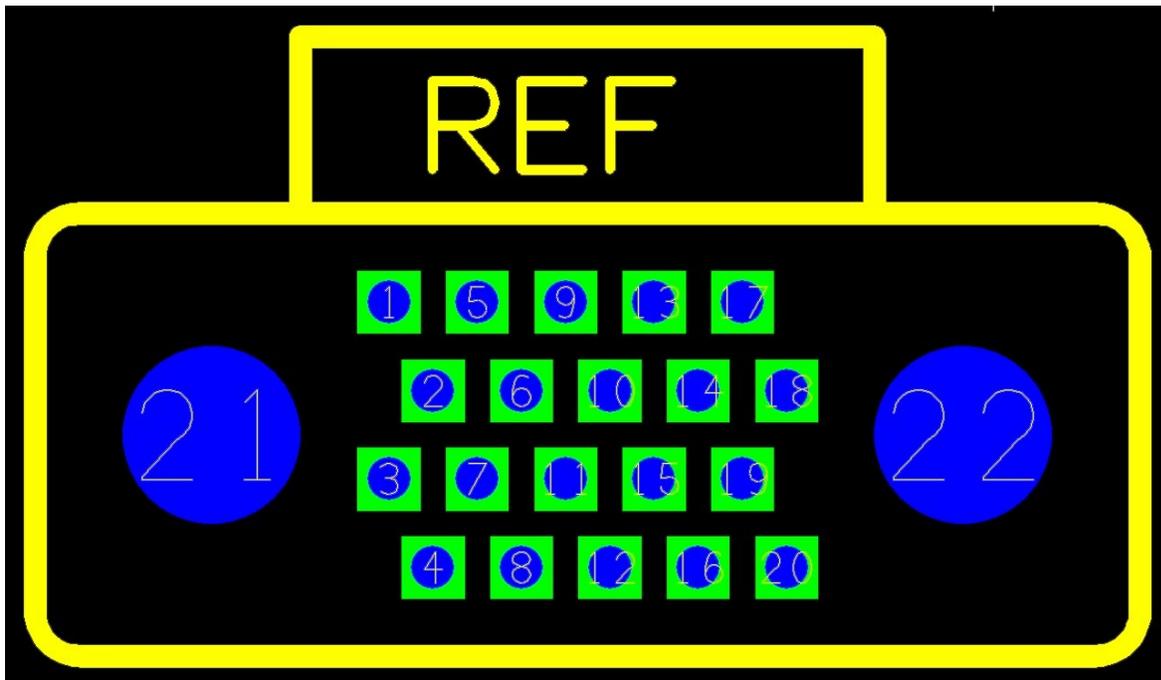
Major new features:

Pin # Increment

A pin increment field was added to the Footprint editor **Add/Edit Pin** menu.



This addition makes it easy to create footprints with staggered pin rows or columns.



Redo in FP Ed.

In the footprint editor, most operations can use **Redo** (Ctl-Y) after an **Undo** (Ctl-Z).

Minor changes:

PADS2000 Netlist

Pads2000 as well as PADS-PCB netlists can be imported. This is considered a minor change because the two file formats are identical with the exception of the first line:

PADS-PCB

vs

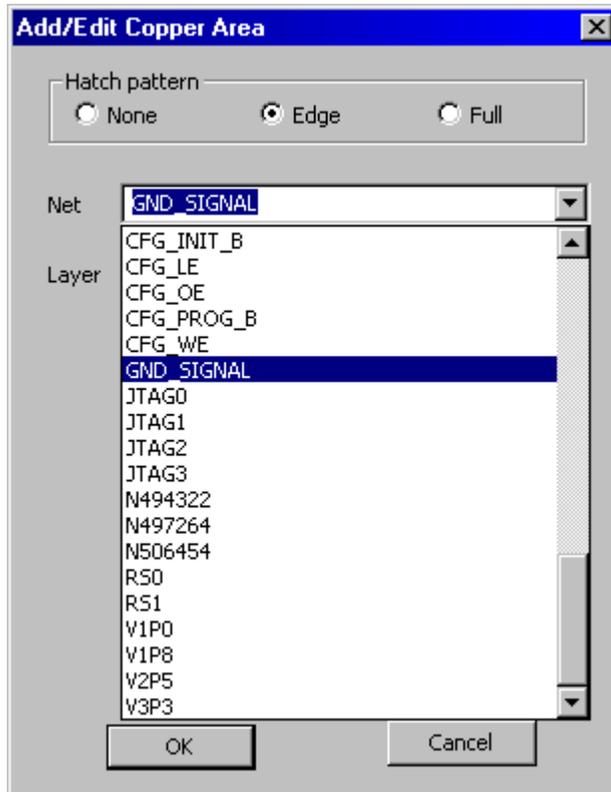
PADS2000

V1.335 – Jul 15, 2007

Major new features:

Cu Area Net Reassignment

In the Add/Edit Cu Areas menu, the net assigned to an area can be reassigned to any existing net. (This menu is accessed through the context menu Edit Area Params command or **F2** key when a Cu area edge is selected.)



Show Part with Ref Selected

When a part reference is selected, the context menu contains the command **Show Part**. This command will select the corresponding part and pan the display to center on it. (The zoom level is not changed so the selected reference may go off screen.)

Minor changes:

Set Side Style

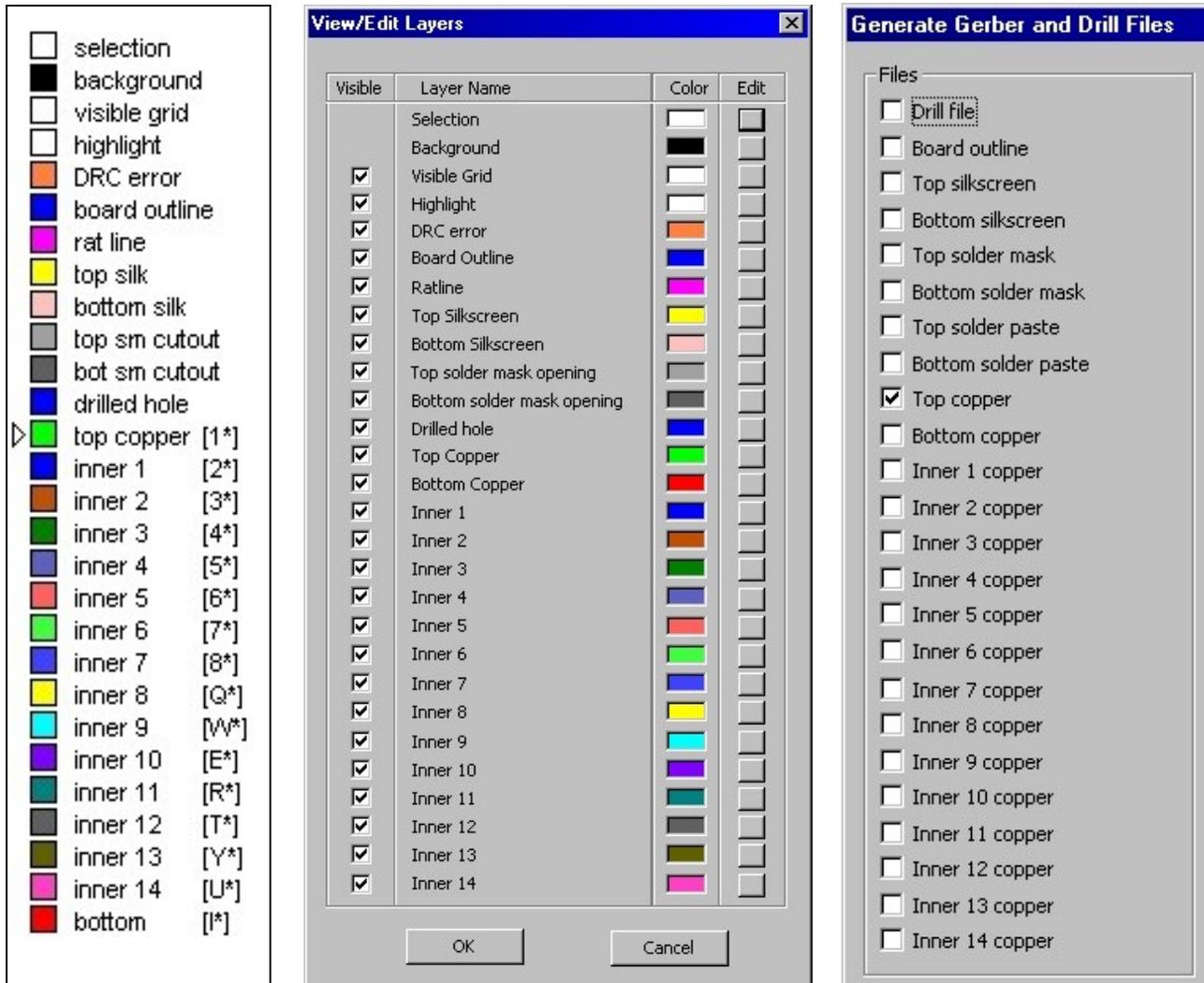
The context menu, as well as the **F1** key, can open the Side Style menu when a Cu area edge is selected.

V1.337 – Aug 9, 2007

Major new features:

Full Support for 16 Cu Layers

Full support for 16 Cu layers added to the layout editor, the View/Edit Layers menu and the Gerber output menu.



Minor changes:

DRC Error X,Y Location Listed

The X,Y location of each DRC error is listed in the DRC log.

Run DRC from Context Menu

Run and Rerun DRC added to the default context menu.

V1.339 – Oct 15, 2007

Major new features:

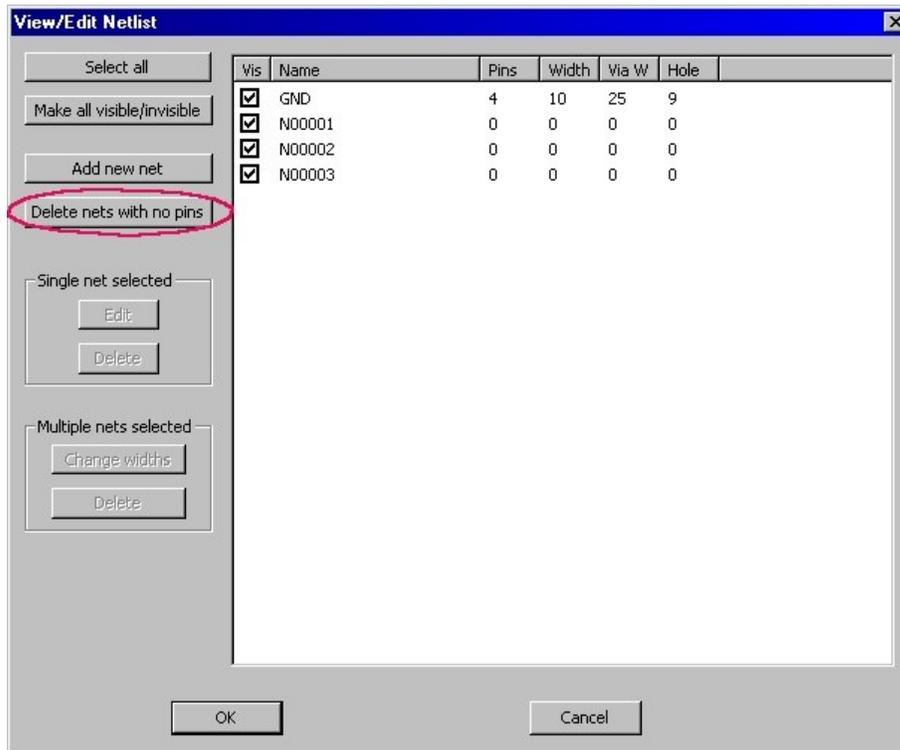
Report File

The **Generate report file** function was added to the **File** drop down menu.



Delete Empty Nets

"Delete nets with no pins" button added to **View/Edit Netlist** menu.



Minor changes:

(None Listed)

V1.344a – May 28, 2008

Major new features:

Part Value

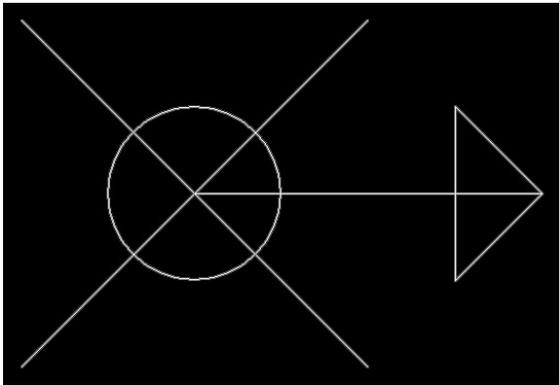
Support for a "Value" text string added to footprints and parts, The value text is sized and positioned much like the part's reference text. An imported netlist can include the part value, in the *PARTS* section, between the part reference and the footprint with the value text delimited on the left by one or more space characters and on the right by the @ at sign. The value text may include embedded spaces:

U123 The Value String@Footprint

(Note that the netlist import function is not working correctly in version 1.354.)

Footprint Angle

The ability to offset a parts reported rotation angle has been added to the footprint centroid. The offset, as represented by the CCW angle between the arrowhead and the +X axis, can be added to the parts actual rotation, in the report file, as needed for use with pick-n-place machines.



Solder Mask Pads

Paste Mask Pads

Two new pad types have been added to the outer layers of footprint pins to override the system default behavior for solder mask cutouts and solder paste application. While a shape value of <default> will continue to use system settings, any other shape will be used exactly as specified including "NONE" which deletes the feature.

Top copper and mask pads				
	Shape	Width	Length	Corner radius
Copper pad	rect	2.2	1.2	0
Solder mask cutout	rounded-rect	2.7	1.7	0,25
Paste mask cutout	oval	2.0	1.0	0

Copper area connection default no connect thermal no thermal

Adhesive Dots

One or more adhesive spots, of user specified size and location, may be added to a footprint and will be listed in the report file.



Minor changes:

Inner Pads

Generated Gerbers now use the inner pad shape and size specified in a footprint. The previous behavior was to ignore the inner pad values and add round pads, as needed, based on the CAM pad minimum annular size value. These "virtual" pads were only added to the Gerbers if connected to a trace, otherwise they were omitted. The new behavior is to use the pad as specified whether it connects to a trace or not. The old, virtual pad behavior, will still be used if the pad shape is specified as "NONE".

All Pad Shapes Flashed

Prior to this release, some pad shapes were not flashed but were drawn as polygons in the Gerber files. Now, all non-basic shapes are defined with aperture macros and flashed as needed. This should simplify netlist extraction, by board houses, for bare board testing.

1.348a – Jun 25, 2008

Major new features:

DRC and CAM Settings Saved

New values entered into the DRC and CAD menus can be save with the "DONE" button without executing there respective functions. This was done because many of the value on these menus are also used by the autorouter export tool, *FpcROUTE*, and this provides a simple means of updating the values without wasting time.

Minor changes:

Settings Saved in Registry

Some basic window settings for state, size and position are saved in the registry for use with subsequent restarts.

Pin 1 Position in Report

The X,Y position of each part's pin 1 is included in the report file. Note that this only works if the part has a pin named named "1". If the part is a BGA, which usually starts with pin "A1", or a diode with pins labeled "C" and "A", this will fail and the report column will be empty.

Tab Order

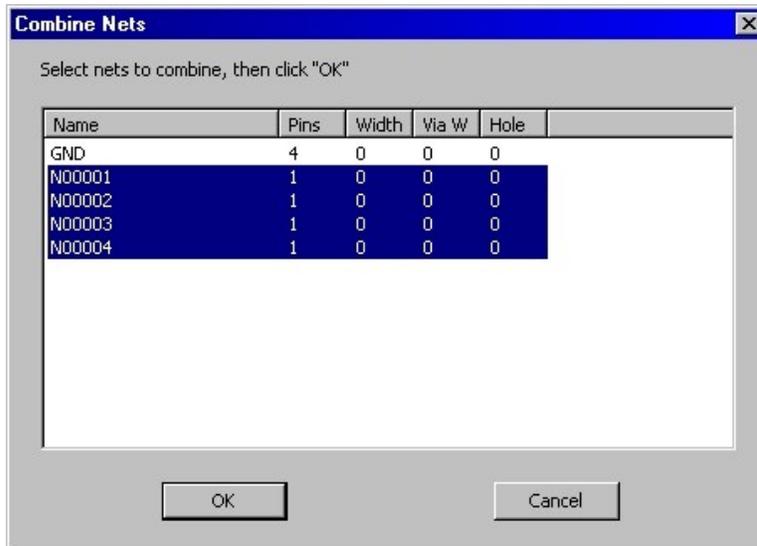
Many of the menus have, over time, been modified by adding new features. This has lead to the tab order being less than intuitive and somewhat scrambled. With this release, the tab order, in all menus, has been restored. (For the benefit of those who *insist* on using the keyboard.)

1.353 – Aug 13, 2008

Major new features:

Combine Nets

The "Combine nets..." command was added to the **Project** pull down menu. This command will merge two or more nets into a single net.



The new, combined net may inherit the name of any of its component nets.



Minor changes:

Cu Areas Added and Highlighted with Net Select

When a complete net is selected and highlighted with the **N** key, any Cu areas assigned to the net are also highlighted. (Sad note: this feature does not work in version 1.354.)

Default Library Path

Changes in the default library path specified in the **Project Options** menu are also applied to the Add/Edit Part menu.

Add Pin Positioning

In the footprint editor, the position of a pin may be specified relative to edges and corners as well as its center.

Header Warning Message

When a project containing possibly invalid header footprints is loaded, a warning message is displayed. (This applies to a specific set of 100 mil headers known to have undersized, 28 mil, holes.)

Cu Layer Change Keystroke Shortcut

With a trace, trace segment, Cu area edge or corner selected, the Shift-n key combination will move the selected Cu to layer n. This is a handy shortcut for changing layers but should be used with caution. Changing the layer of a trace segment connected to a SMT pin will physically break the connection but is still considered connected by the **Check Connectivity** tool.

1.354 – Oct 13, 2008

Major new features:

"Slide" Trace Segment Move

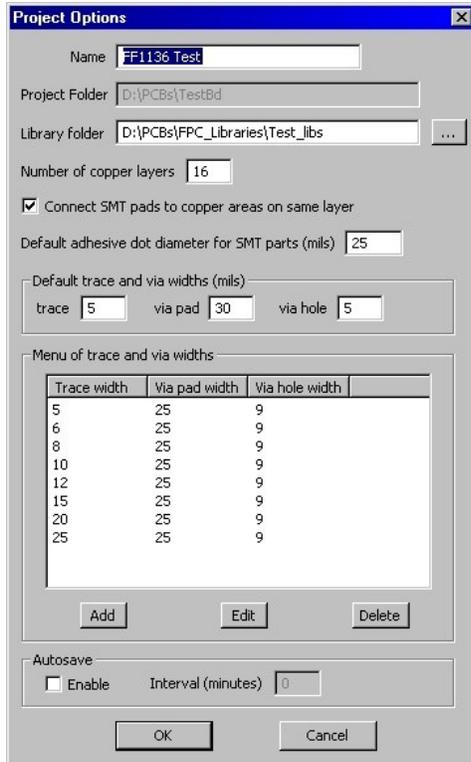
A selected trace segment can, with some limitations, be moved by hitting the F4 key. The moving trace and its adjoining segments will maintain their relative angles while their length vary. Segment movement is limited such that no segments length may become or pass through zero. Also, segments attached to branch Tees or with a vertex inside a pin may not be moved.

Minor changes:

DRC/CAM Annular Width Warning

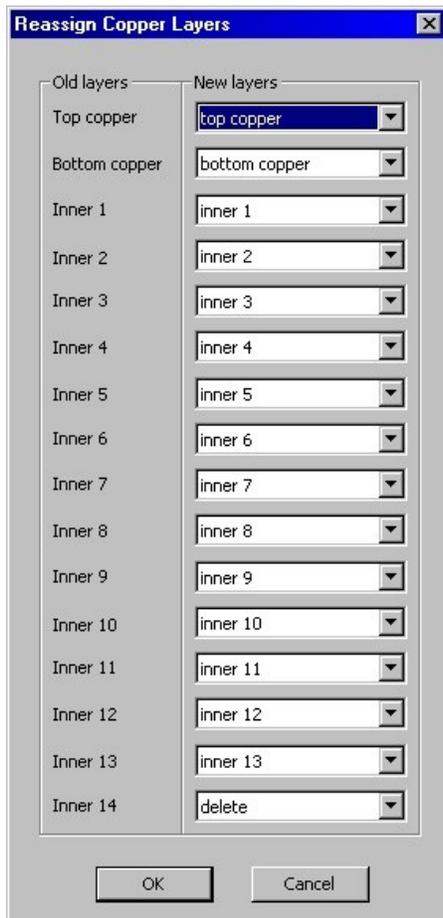
When executing a DRC, a warning message is displayed if either DRC annular ring limit exceeds its equivalent CAM value.

Project Options Menu



Although not changed since version 1.324, the Project Options needs to be reexamined. As mentioned on page 20 of the User Guide, any field that is not grayed out, including the number of layers used, can be edited.

Adding layers is simply a matter of increasing the layer count and clicking the **OK** button to add new, empty layers.



Removing layers, on the other hand, can be more involved. For that reason, if the layer count is decreased, the **Reassign Copper Layers** menu is invoked where existing traces and areas can, by layer, be deleted or merged with other layers. Note that layer reassignment only applies to traces and areas; text objects do not change.

FreePCB Revision History

(As documented in the FreePCB forum [Latest News](#) section.)

Vers	Date	Bug Fixes	New Features
1.001	Mar 10, 2005	Crash on stub trace unrout	Show Part function added
1.002	Mar 15, 2005	View Part crash with missing footprint	Recalc Ratlines with F8
1.003	Mar 19, 2005	Routing crash when net has bad pin	
1.004	Mar 20, 2005	Text display with Win Use Large Font	
1.005	Apr 5, 2005	Misc.	
1.1	Apr 12, 2005		<ul style="list-style-type: none"> - Text in footprints - Oval, Octagonal and RndRec pads - Function key display clickable - Layer List display clickable - Default context menu expanded
1.101	Apr 16, 2005	Gerber generation of new pad shapes	
1.102	Apr 17, 2005	Cu area clearance of new pad shapes	Footprint SMT pads on bottom side
1.103	Apr 20, 2005	Library display of through hole pins	
1.104	Apr 25, 2005	Footprint Ed. Add pin dialog	Files menu Recent Files works
1.105	Apr 28, 2005		<ul style="list-style-type: none"> - Netlist import options menu - File Open remembers last folder used
1.106	May 6, 2005	Round pads in the Footprint Wiz.	
1.107	May 12, 2005	<ul style="list-style-type: none"> - Starting trace on bottom layer edge conn pin - Recent Files list 	
1.108	May 21, 2005	Routing to edge connectors	<ul style="list-style-type: none"> - Solder mask cutouts - Check Connectivity tool
1.109	May 24, 2005	Solder mask cutouts	
1.110	May 25, 2005	Misc	
1.111	May 26, 2005		Cu area cutouts (no arcs)
1.112	Jun 1, 2005		<ul style="list-style-type: none"> - Move Origin - Trace and Net display highlight - Change Layer function - Update existing traces added to net trace width

Vers	Date	Bug Fixes	New Features
1.113	Jul 4, 2005	- DRC on pins with different size pads - Editing pin in the Footprint Ed. - Misc. (same version reposted 15 hrs later)	- Improved through hole pin display - Solder mask via cutouts controllable - Thermals on pads and vias - Group Move - Arrow keys move selected object(s) - Selection Mask - Shift key changes snap size - Status bar <i>Distance Moved</i> - Help menu Keyboard Shortcuts list
1.114	Jul 6, 2005	DBGHELP.DLL dependency removed	
1.115	Jul 6, 2005	Group select with missing footprint(s)	
1.116	Jul 8, 2005	Misc	
1.117	Jul 9, 2005	Cu area cutouts on wrong layer	
1.118	Jul 25, 2005	Pin/via connection to Cu areas with cutouts	
1.119	Jul 27, 2005	<i>Change Layer</i> function	
1.120	Aug 7, 2005	Changing footprints and importing netlists	
1.2	Aug 14, 2005		(Stable Version)
1.201	Sep 14, 2005	Netlist pin assignment undo	
1.202	Oct 13, 2005	- Crash moving origin without an outline - Add part-drag-cancel-undo crash - Add pin display glitch in footprint ed. - Layer (>4) colors in new projects	
1.203	Oct 23, 2005	Footprint ed. pin edits	Apply new pin setting to existing pins
1.204	Oct 25, 2005	- Default library folder on startup - Cu area edit undo - Footprint ed add pin undo	
1.205	Nov 18, 2005	- Gerber gen Cu area arc crash - Footprint units	Backspace key unroutes last trace segment
1.3	Dec 9, 2005	Netlist import	- Cutouts in Cu areas with arcs - Combine Cu areas tool
1.301	Dec 12, 2005	Pin names not ending with a digit	Combine Cu Areas renamed Check Cu Areas
1.302	Mar 6, 2006	- Cu area DRC - Misc	- Some Cu area ops faster - Via size adjustable - Trace/Via size adjustment independent - Vertex add/delete - Part reference editable - Cu area Change Layer - Debug and Release versions available

Vers	Date	Bug Fixes	New Features
1.303	Mar 16, 2006	Overlapping Cu areas DRC	Off-grid starts snap to grid line/point and maintain the snap angle
1.304	Mar 27, 2007	- File ops crashes due to Win settings - Stub trace visibility after group move	
1.305	Apr 2, 2006	- Manual ratline routing - Ratlines to moved parts - Cursor 45° ref lines when adding a vertex - Delete vertex crash with unrouted segments	Executable type (release/debug) listed in Help→About
1.306	Apr 3, 2006	- Netlist import checks for PADS-PCB format - Parts mask ignored when a group selected - Zero-length segments removed on save	
1.307	Apr 5, 2006	File save with single zero-length segment	Check Traces tool
1.308	Jul 1, 2006		Copy/Paste groups of parts and traces
1.309	Jul 2, 2006	Group Copy/Paste part reference position	
1.310	Jul 3, 2006	Shorts between traces and Cu areas with cutouts	
1.311	Jul 15, 2006	Save deletes zero-length stubs	Cu areas within cutouts
1.312	Aug 7, 2006	Misc	- Multiple Board Outlines - Group Copy/Paste includes all elements - Branch traces
1.313	Aug 7, 2006	Misc in 1.312	
1.314	Aug 11, 2006	Misc	- Recalc ratlines moved to F9 - Gerber format changed to 2.4
1.315	Aug 27, 2006	- Polyline creation undo - Multiple board outline save - Clicking on F9 - CAM folder not remembered - Solder mask on padless holes - Pads shorting to adjacent Cu areas - Group ops crashes	
1.316	Aug 27, 2006	Critical bug in 1.315	
1.317	Aug 28, 2006	- Part move ratline drag - Release version Add/Edit Part	
1.318	Aug 30, 2006	- Pad clearance in Cu area/cutout edges - Cu areas within cutouts	

Vers	Date	Bug Fixes	New Features
1.319	Sep 1, 2006	- Several re nested Cu areas and traces - SMT pad shape NONE crash in FP Ed - SMT pad layer preserved in FP Ed Apply to others	
1.320	Oct 15, 2006	Misc	- Off-grid initial segments snap to grid using dogleg segments - Source file renamed on save - Autosave uses sub-folder
1.321	Oct 22, 2006	Several re traces to bottom layer edge conn pads	Traces and Cu areas preserved if net name changed with netlist import
1.322	Jan 17, 2007	Several mainly re netlist re-import with isolated Cu areas	
1.323	Feb 12, 2007	Several mainly re trace/via clearance within Cu areas	(Code development moved to VC2005)
1.324	Feb 19, 2007	- Group paste - Netlist import into an existing project	- Group delete - Project file paste as a group - Project name, default library and number of layer editable in the Project Options menu
1.325	Mar 6, 2007	Misc + Win98 incompatibility	- Autorouter file export/import - Autorouter start - Vias always connect to same net Cu areas - Forced vias
1.326	Mar 6, 2007	Router file names with embedded spaces	Project file save prompt on router export
1.327	Apr 7, 2007	- Selection rectangle size update in FP Ed - Via size change effecting trace width - Router import topology rerouted	- Panelization - Gerber Paste Mask file generation - Gerber Board Outline file generation
1.328	May 9, 2007	- Polyline edit in FP Ed - Add parts tree control memory - Space character width - Add pin to a net dialog closes on Enter	- Project parts list sortable by reference - Trace selectable with segment or vertex selected - Net selectable with segment, vertex or pin selected - Trace length in status bar when trace selected - Panelized drill file minimizes tool changes - Selecting a part highlights reference - Project save to temporary file on router import

Vers	Date	Bug Fixes	New Features
1.329	May 11, 2007	- Pin to Cu area clearance - Router import dialog	
1.330	May 19, 2007	- Silkscreen minimum width on Cu layers - Vias missing from some tee vertexes - Crash when saving to non-existent device	- Move Origin in FP Ed - Library file uses name not full path
1.331	May 30, 2007	- Undo - Group paste required a group be selected	- Redo for most ops - Move Origin, in FP Ed, RMB cancel - Group Rotate
1.332	Jun 12, 2007	Group copy of text only objects	- Measure tool - Negative Text in Cu areas - PNG Render in CAM dialog - Arrow cursor hidden with cross-hairs
1.333	Jun 23, 2007	- Save Footprint crash if log uninitialized - PNG file options - Undo/Redo of text ops	- PADS2000 netlist import - Pin number increment in FP Ed add row of pins - Redo in FP Ed
1.334	Jul 4, 2007	- FP text wrong width - Cursor position dump in log - Position display when moving items with arrow keys	0.1 mil display resolution
1.335	Jul 15, 2007	- F1 move of vertex not undoable - Via size change on nearby vertex deletion - Folder change when project name changed - Footprint names with trailing spaces crash - Menus grayed when no project loaded	- Cu area net reassignment - Show Part in context menu when reference selected - Set Side Style in Cu area side context menu
1.336	Jul 20, 2007	- Open File dialog - Crash on closing	Spaces in router file path
1.337	Aug 9, 2007		- 16 Cu layers - X,Y location listed for each DRC error - Run DRC without opening dialog
1.338	Aug 15, 2007	Crash on close immediately after open	
1.339	Oct 15, 2007	- Duplicate names in library - Footprint select rectangle not including text - Change trace width effects existing traces - Crash when moving first corner of outline canceled - DRC hole-trace violation	- Report File generation - Delete nets with no Pins in nets menu
1.340	Nov 30, 2007	RMB click crashes	

Vers	Date	Bug Fixes	New Features
1.341	Dec 18, 2007	- Group paste with Cu area cutouts crash - Group paste with pin/trace mismatch crash - Random crashes with LMB clicks (mostly with a group selected)	
1.342	Apr 5, 2008	Rounding error in panelization	
1.343	Apr 14, 2008	Centroid location in report file	
1.344 a	May 28, 2008		- Value in parts, footprints and netlist (in netlists use: "REF VALUE@FPNAME") - Footprint axis - Inner layer pads honor footprint settings - Solder mask pads - Solder paste pads - Adhesive dots - All pad shapes flashed in Gerber files
1.345 a	May 28, 2008	- Values not on silkscreen layers - Text after *END* in netlist	

1.347 a	Jun 4, 2008	Project parts list import	
1.348 a	Jun 25, 2008	- Undo disabled after drag ops - Drag part with ref visibility off crash - Pin edit, in FP Ed, undo	- DRC and CAM dialogs save setting with Done button - Window settings save in registry - Pin 1 position in report file - Tab order in all dialogs redone
1.349 a	Jul 2, 2008	Crashes when generating Gerber files (mostly inner Cu layers)	
1.350 a	Jul 3, 2008	Through hole clearance in inner Cu layers	
1.351 a	Jul 6, 2008	Part edit with no footprint assigned crash	
1.352 a	Jul 7, 2008	Edited footprint applied to multiple parts crash	

Vers	Date	Bug Fixes	New Features
1.353	Aug 13, 2008	<ul style="list-style-type: none"> - View All - View Board 	<ul style="list-style-type: none"> - Net select highlights Cu areas - Combine Nets function - Cu areas added to Select Net - Default library folder change takes effect immediately - Add Pin can position by edge or corner - Headers, with 28 mil holes, warning - Shift-n moves selected Cu to layer n
1.354	Oct 13, 2008	<ul style="list-style-type: none"> - Report generator hang with no footprint - Solder mask shape of NONE with holes - Solder mask pads - Add pin position by edge - Centroid angle in report - Crash when opening project with zero-length last segment 	<ul style="list-style-type: none"> - Slide trace segment - Warning message if DRC annular widths exceeds CAM annular widths