

geda-warts-n-wishes

These notes were assembled since 2007 until now from top to bottom. Some (many?) items may have been resolved. I did not systematically check. I did not check for double entries, either.

The list is “raw” in the sense that it was meant as a personal notebook – not polished, well worded feature requests and bug reports. — [Kai-Martin Knaak](#) 2012/10/30 12:39

A LaTeX version of the notes: [warts-n-wishes.pdf](#)

<markdown> spring 2007

* gschem: I miss search and replace * gschem: There should be a way to preview footprints * gschem: There should be a modifier for the move action, so that blue lines rip off. * pcb: Pads should be colored according to their layer. (top/bottom) * pcb: Please allow text in footprints * PCB: Change of background color needs a restart to apply. * gschem: Please make more obvious the difference of an open end, and a successfully connected node. * gschem: Symbols should automatically update after “symbol-up”. * gschem: netnames should by default show value only * gschem: By default netname attributes touch the corresponding net. They should be placed at a distance (e.g. 50 units). * gschem: Please add an option to restrict nets to manhattan style. Add a perpendicular net if necessary. * gschem : Please add a net if a connection threatens to break during move. * Feature-request: A keyboard compatibility mode for gschem and pcb * pcb: <ctrl-f> on a component selects only connections of one pad. It would be more usefull to select all connections to the component. * pcb feature request: move and rotate without a switch of modes —> rotate on keypress during select mode * pcb feature request: History in select dialog. * Bug: Regexp does not work as expected

—————> “U1” selects U11, U12, U13... too. Looks like a “*” is added to the string. This is also true for lists like “R1|U1|L12”. “R?8” selects R8, R18, R28, C18, C8, C38, U8... * pcb-cvs: annoying decimals in coordinates when in metric grid mode * pcb feature request: The ability to move a single object from within the current selection * pcb feature request: select objects should optionally zoom to the found object. * Sometimes change of gridsize fails with “Don't combine metric/English grids like that!” —> Seperate shortcuts for metric and imperial increase are annoying. A three finger short cut reminds me of my first guitar lessons. * pcb feature request: A way to make silk screen objects visible but insensitive to seperate selection. Alternatively: A way to suppress seperate selection of parts of components. * pcb feature request: “Select all locked objects” * An improved placement workflow:n improved placement workflow:

- 0) Get list of refdeses from gschem
- 1) List of Refdeses in Select-Dialog
- 2) Pull all selected components to mouse
- 3) place group of components
- 4) disperse components

* gschem usability: Some dialogs, that don't have a default button defined, that will be accessed by the return key:

- a) "Single Attribute Editor"

b) "Edit Attributes"

* pcb feature request: Improved update workflow. Please add an option to look for foo.new.pcb and foo.net on start-up and load them by default. * gschem feature request: Add menu items to run DRC or gschem2pcb * The command line should receive keyboard focus right after it was called with ":". * Why is the history limited to 25? * pcb: Is there a shortcut to select the route style? * pcb feature request: More complex modes of track manipulation:

a) Insert an elbow: `"` _ "` --> "` _/_ "``
b) Insert a plateau : `"` _ "` --> "` _/---_ "``

c) Push tracks out of the way, while drawing other tracks. (Like in protel99)

d) Vias should be placed, if a track gets flipped to the other side with the m-key.

* pcb: Delete does not work on tracks if the mouse is in "endpoint mode". * Rubber band mode still chooses the wrong nodes to move if short tracks are involved. * Missing feature: DRC clearance is neither shown, nor enforced during manipulation of tracks. * DRC clearance is not enforced when drawing vias. * pcb bug: Undo while routing focus often jumps to the wrong nodes. * GTK HID, layer buttons: Please exchange the roles of radio button and colored button. * pcb feature request: A command to update the selected polygo * pcb feature request: It should be possible to print an A3 sized schematic to two A4 sheets. (Work-around: the poster utility). * pcb feature request: Insert should insert a node, even if the resulting track won't change. * pcb feature request: remove nodes (rather than lines) * pcb: frontassembly.ps should contain the lineout layer, too. * pcb feature request: The application should remember postscript print parameters * pcb feature request: assembly with refdes and assembly with values * text in gschem postscript output needs a fudge factor 1.4. This looks suspiciously like $\sqrt{2}$... * gschem: The current state of options should show in the menu. * pcb feature request: Transparente Leiterbahn-Darstellung, bitte. * gschem bug: rotation of a pin renders an incorrect alignment of the pinnumber attribute. (Alignment changes from lower left to lower right) * pcb bug: polygon tool sometimes does not snap to start point.

How to reproduce:

- 1) set grid to 100 mil
- 2) select polygon tool
- 3) start polygon at (100/100)
- 4) set polygon corner at (100/300)
- 5) set polygopn corner at (200/300)
- 6) try to close the triangle with a click at (100/100). pcb will set a corner point at (100/200) instead.

Oktober2007, PCB v20070912

* Non-metal-holes don't convert to elements * Lines drawn on silk keep the "found" flag after drawing action ended. Consequently, they stay drawn in turquoise. Only way to put them into normal state seems to be copy to buffer and paste back into place. * Please include the layer lineout to frontassembly.ps and backassembly.ps . * values and refdes in printout * Sometimes I can't close the preference dialog. The terminal prints an error:(pcb:1184): Gtk-CRITICAL **: gtk_widget_destroy: assertion `GTK_IS_WIDGET (widget)' failed * Where are the defaults for ps output set? (I always need a4 rather than letter, multipage output, etc.) * default postscript output name is "foo.ps". With multi page output this results in strange filenames like "foo.ps.frontassembly.ps". * gerber output should

produce a file dedicated to the lineout layer exclusively. This file should get a telling name. At the moment, gerber output produces a file PD-ac.group3.gbr which, in addition to the lines on lineout, contains all the vias. * Convert selection to elements does not work. The selection just vanishes. * refdes/label should be visible if on a pad on the bottom/solder layer * I get lots of vias if I move tracks to other layers (e.g. outline, or silk) * tab is missing in the list of key bindings. * sometimes tracks on silk are not handles properly * Please add a big, fat warning in the install READE that old pcb versions will fail to render files produced by this version. * rubberband mode is (still) far too sloppy. Only endpoints closer than the current cursor snap diameter should be moved in unison. * Please include "lock text" and "text only" in the menu of GTK-HID. Currently, it is possible to lock the text with the lesstif-HID only. If the layout is opened with GTK-HID there is no way to unlock the text. I provided a (rather trivial) patch in the bug reports some months ago. * gsch2rc should look into .pcb/preferences for the default layer stack. * If two footprints of the same kind are stacked, they vanish when moved. This happens when adding two elements by load layout data to buffer. * gschem bug: Alignment inconsistency with rotated text. * gschem bug: New attributes should be placed on grid, even when attribute placement hooks are not activated. * Bug pcb: [g] and [shift-g] ignore preference setting * Non-feature pcb: [ctrl-g] and [ctrl-shift-g] to increase grid size if metric. * Grid-Geschichten in pcb gefallen mir nicht * feature request pcb: Please allow metric sizes in pcb format * pcb bug: A click on pads with onsolder flag selects the whole element rather than just the pad. A [q] key on the pad toggles the square flag of all pads of the element. This bug does not show if the layout is viewed from the far side. * bug gschem: system-gschemrc overwrites the setting of default-titleblock in \$HOME/.gEDA/gafrc . I suspect, this will happen to other settings too. The user gafrc should be loaded after system-gschemrc. I'd propose the following rc file load sequence:

```
(.)/share/gEDA/system-gschemrc
(.)/share/gEDA/gafrc
$HOME/.gEDA/gschemrc
$HOME/.gEDA/gafrc
$PWD/gschemrc
$PWD/gafrc
```

* pcb feature request: Add lineout to the list of default layers produced by gsch2pcb * gschem: Is there a way to exchange a (referenced) symbol via the GUI? * pcb bug: save-buffer-to-file brings up "file exists" warning twice. * gschem non-feaure: autonumber on placement does not check for existing numbers * gschem feature request: Put buttons for frequently used symbols in the task bar. * gschem non feature: autonumber does not work with "Include component as individual objects". * gschem bug: Nets are not connected after "Include component as individual objects". * geschem bug:

- 1) copy an object with [CTRL-c]
- 2) start paste mode with [CTRL-v]. Contents of the paste buffer appear at the mouse cursor.
- 3) abort paste operation with [esc]. The contents of the paste cursor remain on screen until the next redraw event.

* gschem bug: Symbol translate ([e-t]) disconnects nets from symbols. A workaround to reconnect nets to symbols is to select all and pretend to move with the mouse. * gschem feature request: modifier [shift] to do a tear-off-move (in addition to [o-r] * feature request: Make [u] a shortcut for undo rather than for drawing busses. * pcb bug: If preferences specify "grid-units-mm = 1" pcb starts up with grid size[shift- 0.254 mm. Attempts to change grid size via the keyboard with [ctrl-g] yield "Don't combine metric/English grids like that!" in the status window. Workaround: After a switch to imperial and back to metric [ctrl-g] and [ctrl-shift-g] will work. * pcb feature request: Add keyboard shortcuts to manipulate gridsize according to a configurable list of grids. Suggestion: [alt-g], [alt-shift-

g] * pcb wart: The units setting mangles two distinct tasks – How to display and how to act. It makes sense to differentiate these. E.g. it is useful to work on an imperial grid but show sizes in metric units. Suggestion: Add a preference parameter “display-units”. Toggle between “mm/mil” and “mil/mm” with a right mouse click on the gridsize button on the upper right corner of the canvas. * pcb non feature: The preferences dialog contains parameters of the current layout. Suggestion: Add a dialog “settings”. * pcb feature request: Access “remove selected” with [backspace]. * pcb bug: [ctrl-x] should cut selection to buffer but currently only copies selection to buffer and fails to remove the selection from canvas. * pcb bug: Some arcs don't transform correctly during left-right flip. To reproduce:

- 1) Draw an 90° arc east to south.
- 2) Do a left-right-flip ([shift-tab]). The arc transforms into south-east to south-west instead of south to east.

* pcb feature request: Improved usability for the lock tool. * pcb non-feature: Why is track width limited to 250 mil? Suggestion: Issue a warning if tracks get larger than 250 mil, but allow to override. * pcb feature request: A (powerful) find and replace dialog. * gsch2pcb bug: If a component is represented by two symbols with the same refdes gsch2pcb checks only * the first for a footprint. Consequently, the component is missing in the pcb file. Workaround: Cut the footprintless symbols to buffer and paste them back into place. This will put them at the end of the file. So gsch2pcb will happily find the sub symbol with footprint. * pcb bug: After edit name of layout the layout is zoomed to full extents. * pcb non-feature: Autoroute draws tracks on lineout layer. * pcb feature request: show only unrouted nets in netlist * pcb bug: “align-marks = 0” and “outline = 0” in settings file are ignored. * pcb bug: lines on silk don't print in frontassembly.ps and backassembly.ps . * pcb feature request: read local settings and preferences like gschem does. * pcb bug: convert buffer to element does not care for the layer group to decide on which side a pad will go. It uses first and second layer instead. * pcb non-feature: Dense rats nests obscure refdeses. Suggestion: Text should be rendered above rats. * pcb suggestion: “lock names” → “ignore names” * pcb feature request:

“ignore rats”, “ignore pads and pins”, “ignore layer”, → A whole ignore menu 😊 * pcb feature request: “grab only at handle” * pcb bug: add layer does not work if the current layer is silk or rat lines * pcb quirk: rats don't start at the end of tracks * pcb quirk: load-layout-data-to-paste-buffer places the elements contained in foo.new.pcb with an offset to the mouse. (5mm down, 20 mm to the right) * pcb quirk: minimum size of pcb for no apparant reason. (Enforced in GTK-HID only) * pcb-quirk: Maximum track size is 250 pt * pcb feature request: Set DRC-clearance per route style. * pcb feature request: Add keyboard shortcut for different route styles * pcb quirk: Sizes in the preference dialog should increment by mils not by 0.1 mil * pcb quirk: pads should only snap on the side they are on. * pcb bug: [M] and [shift-M] are not in the list of accel keys * pcb quirk: update rats while drawing a track with the line tool does not affect the current action. * pcb quirk: “Only names” should be reset at the end of a session. * pcb bug: component name cannot be moved when selected * gschem wart: Please introduce a decent origin in gschem symbols * gschem bug: [ctrl-c] and [ctrl-v] do not work in the value fields of the Edit-Attributs dialog. * gschem feature request: a convenient way to select the symbol, but not the attributes.

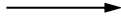
März 2008 v20080127, git

* Howto configure gschem to not rotate Attributes while rotating a symbol? * xgsch2pcb feature request: The list of schematic pages should be sorted according to name. Manual sorting would be even better (right-click menu for the entries with move-u/move-down action) * xgsch2pcb feature request: right-click menu for the entries in the list of schematics * xgsch2pcb feature request: edit

schematics or layout with external ascii editor * gschem feature request: Add some of the most important one-key-accelers of pcb

- [s]/[shift-s] to increase size
- [n] to edit text
- [esc] to get into arrow-mode

* gschem wart: On [tab] the edit attribute dialog should switch to the next item rather than insert a tab. Add a tab with [ctrl-tab] * gschem wart: The single attribute editor should behave like the general edit attribute dialog. That is, stay on the screen after update of the attribute. * gschem and pcb feature request: An action to move the selected objects on grid. This would provide a means to alleviate some of the most frustrating experiences with both applications. * gschem feature request: issue a big, fat warning, if snap to grid is turned off. * gschem bug: copy-paste of a pin does not maintain the color of the pin label. * gschem feature request. Make the text alignment dialog more GUI-friendly — provide a graphic rather than a list of values. * gschem wart: Dialogs “show-text” and “hide-text” don't show up in window list. * gschem feature request: The print dialog should remember the path of the last print-to-file * pcb bug: Dieresis don't work in pcb. * gschem/feature request: Some form of netclass support * gschem feature request: Add the notion of an origin to the file format * gschem feature request: Down-schematic with [double click] on the subsheet symbol. Down-symbol with [ctrl-double-click]]. * gschem feature request: A more powerful matching syntax for the

autonumber routine.  Please stick to some standard: regexp, globbing ... * gschem rant: magnetic mode is sometimes just too magnetic * gschem suggestion: In magnetic mode do “Z” nets rather than “L”. * gschem feature request: A dialog to view (and edit) the source of a symbol. * gschem feature request: An object properties report * pcb feature request: The ability to directly edit the properties of an object. Suggestion: A properties report on steroids. * pcb feature request: If started with a new filename, the new design should be named like the filename rather than unnamed.pcb. * pcb bug: Add layer does fails if the current layer is silk. * gsch2pcb feature request: Set the name of the layout to some reasonable value, if it is still unnamed. * gsch2pcb bug: Ignores the default layer stack given in \$HOME/.pcb * gschem wart: Sometimes “symbols” are “components” * gschem feature request: Add graphical hints whether or not a symbol is embedded. * gschem feature request: Optionally save symbols locally on unembed. —> Build a local library. * gschem feature request: Make symbols and schematics better distinguishable in the list of the page manager. Suggestion: put icons in front of the items. Different color scheme. * gschem/pcb feature request: More accel key compatibility

- Edit text: [ee] - [n]
- Move selected: [m] - [?]
- Delete selected:
- Copy selected:
- Scale text [s]

* pcb feature request: attach default layer stacks to color scheme. * pcb wart: Currently, the color scheme is handled independently from the layer stack. Suggestion: The color scheme should be coupled to the layer stack. * pcb bug: The preference dialog mixes preferences with properties of the current board. Suggestion: Add a layout properties dialog. * gschem feature request: renumber only non slotted symbols * pcb wart: not all actions are accessibel via the the menu —> Interferes with the gnome way of accel key configuration on the fly. * pcb feature request: 180° rotation of footprints without rotating the text. This might be associated with a modified mouse click while using the rotate tool. * pcb feature request: Add rotate actions to select mode. Suggestion: rotate the currently

focussed object by 90° if the space key is hit. * pcb feature request: Remember the position and size of ripped off menus. * xgsch2pcb feature request: Add DRC check of schematic * gschem feature request: Add netlister options to the menu. * pcb feature request: pcb should check (and warn) if another instance of pcb is working on the same layout. * pcb feature request SMD pad color should hint for the layer they are on. * gschem usability request: When going up from symbol the option "show hidden" should be set off. * pcb feature request: If the buffer contains just one footprint, save-buffer to-file should set the default file name to the name of the footprint. * pcb-GL wart: If I move the mouse very fast, the cross follows slowly. * pcb wart: layout data in paste buffer is not at the mouse, but a few centimeters to the right and to the bottom. * pcb bug: Layout data preview of identical footprints is invisible if their number is even. * gnetlist feature request: if no -o option is given, output of the bom backends should be sent to standard out. This would allow for pipelines. * pcb feature request: show (and enforce) DRC limits while moving tracks, options and vias. * gschem feature request: a mode to add all slots of a symbol at once. * pcb feature request: Add the ability to do complex searches like "Select all 15 mil tracks that have the join flag set" * pcb feature request: optionally zip the gerber files. * pcb feature request: DRC should check for accidentally exposed copper. (Leiterbahnen in der Nähe von Pins mit großer Clearance. * gEDA feature request: Down to schematic with double click to ease navigation of hierarchical designs. Up schematic with double click somewhere on the canvas. * pcb usability request: ESC while in Line tool mode should end stop drawing and return to the state before starting the track. * pcb usability request: Panning speed should scale with the deviation of the mouse from the viewport. * pcb wart: rats don't go to the nearest end point of a track * pcb feature request: Add an option to show the current DRC clearance of objects already on the canvas. * pcb usability request: Do restricted move when shift modifier is pressed during the action. * pcb feature request: Add setflag/clrflag(...,join) to the menu * gsch2pcb/gnetlist feature request: Tell the user about the status of \$NAME.net * pcb usability suggestion: Replace [j] with [SHIFT-j] to set [CTRL-j] to clear the join flag. Rationale: A simple [j] is too easily pressed by accident. There is no visual feedback about the join flag. This makes the toggle action a stab in the dark. Better explicitly set or clear the flag. * pcb usability request: A simple way to detect the net of a copper object. (Currently, I use the line tool and pretend to draw a track) * pcb feature request: Remember the position of ripped off menus across sessions. * minor pcb bug: If a file exists, Save-buffer-elements-to-file sends two warning dialogs. * pcb bug: The update button of the library is not always available. * pcb feature request: When asked to export pcb should check for shortened nets and warn accordingly. (Add an option to never warn again) * pcb feature request: allow for different connected color according to layer. * pcb none feature: Why do the grid shortcuts depend on the units setting? * pcb bug: lines on silk do not print on assembly page * gnetlist feature request: add *.g2p to the list of known applications * pcb wart: The default extension for postscript export is "\$NAME.ps" This is not useful for multi-file output. It produces file names like "pidpeltier.ps.fab.ps". * pcb feature request: put all postscript and png exports into a dedicated directory "print". * pcb mini bug: [n] is missing in the list of accel keys * gschem feature request: Add-to-number in auto number dialog * pcb feature request: A functional man page. * gschem improvement: An option to not mix the refdes of slotted symbols during renumber. * gschem improvement: An option to not mix with multi part symbols during renumber. * gschem incomplete feature: Right-mouse-Click fails to scroll during placement * gschem-wart: No %Title statement in the PS file → cups- pdf prints to job_foo-untitled_document.pdf * gschem-wart: Some gschem GUI dialogs are missing in the gnome window list. (e.g. Autonumber_Text)

* gschem bug: The find dialog does not find anything after the find string was edited for the second time. * gnetlist -g drc2 bug: DRC complains about duplicated slot with multi part components. * gnetlist -g drc2 wart: There is no way to configure the Tsts applied to the schematic * pcb wart: For every project there are two groups of windows in the window manager → should be one. *

pcb-GL: Two instances of pcb with GL get glacially slow. xscreensaver is affected too. * gschem feature request: select all symbols directly connected to a specific (selected?) symbol. * gschem feature request: Output a |-seperated list of the refdeses of a selection * gschem feature request: An advanced find dialog * pcb feature request: An advanced find dialog * gschem wart: Open file should shows only *.sch. Should show *.sch and *.sym by default. * xgsch2pcb feature request: Make BOM * xgsch2pcb feature request: issue a redraw rats event every n seconds. * xgsch2pcb wart: update layout forgets about hidden nets * xgsch2pcb wart: update layout switches off visible grid * gschem feature request: The add text dialog should allow to set the font size * pcb wart: layers should keep their color, when moved up, or down in the stack. * gschem wart: handles should not scale with the zoom. * gschem wart: attributes should not automatically detach when copied to paste buffer * gschem feature request: add redo accel SHIFT-R to duplicate the behaviour of pcb * gschem wart: Why is eps print in "file → write"? * gschem feature request: The write dialog should ask for a conformation if the file extension does not match the chose file format. * xgsch2pcb wart: Log window is blocking * pcb feature request: add setting "confine to grid" * pcb feature request: Allow to add to the set of found objects with shift modifier * pcb feature request: Add a way to convert the set of found object to selection. * pcb feature request: Show DRC restrictions while mopving tracks * pcb feature request: auto enforce DRC while moving tracks * pcb feature request: show new tracks in a color derived from the current layer color. * pcb feature quirk: insert mode adds a new node only after successfull drag. Suggestion: add the node no matter what. * pcb feature request: A mode to remove nodes * pcb bug: undo/redo does not return to the same state during line draw. * pcb feature request: An all-direction-modifier-key while routing * pcb room-for-improvement: Umlauts should be supported by the default font. * pcb room-for-improvement: Window→Library should bring a hidden library window to the front. * pcb room-for-improvement: Polygons should be recalculated before gerber export. * pcb feature request: add a shortcut for "select connected" * gschem feature request: unembedding a symbol that is not available should produce and save a sym file. * pcb bug: (Peter C) pin numbers hide behind annular ring. * pcb room for improvement: pin names should match pin nummbers by default on convert to component. * pcb room for improvement: The warning "file exists (...)" should contain the name of the offending file. * pcb wart: The dialog "file exists (...)" appears twice. * pcb room for improvement: There is no way to print the top silk in bootom view (see-through..) * pcb room for improvement: update rats should not complain about connection to pins not in the netlist. * pcb room for improvement: route styles should be available via shortcuts * pcb room for improvement: pcb should honor gnome style shortcut localization * gschem room for improvement: The print dialog should present the printers known to the system. * gschem room for improvement: When printing to CUPS PDF printer, the resulting file should be named properly . *

gschem wart: tries to write backups of symbols even when not changing anything. → Lots of warnings when no write permissions. * pcb room for improvement: Footprints should be able to contain silk on both sides of the board * gschem wart: ever growing pile of logs in \$HOME/.geda/logs * pcb wart: the cross hair does not snap to silk * pcb room for improvement: More control on snapping. A way to not snap to the grid but only to features. * pcb feature request: Copy with refdes generation

→ Prefix the refdes with incrementing integer. → plug-in?us * pcb room for improvement: Prest the name of save buffer to element according to component name * pcb feature request: Export all components in the selection to different footprint files. * pcb feature request: set join / unset join flag of selection should be available via shortcut * pcb room for improvement: set all the flags throug the menu * pcb wart: ctrl-right-click should map to something more important than crosshair settings * pcb room for improvement: locked position and ignored element should be independent attributes. * pcb feature request: Add a toggle to activate snapping to lines on silk. * pcb room for improvement: Save to buffer warning "file exists" should mention the actual file name. * pcb wart: Save to buffer offers the layout name as default file name. Should be the component name if the buffer contains just one component. Else, no default name should be given. * pcb wart: Save to buffer warning "file exists" is presented twice. * gschem bug: The find tool finds only once. * pcb wart:

auto-pan is always on on start-up * pcb-wart: grid is always invisible on start-up * eagle-like: A straight line that hints towards a DRC issue * eagle-like "split line" * gschem wart: The open dialog in gschem should show symbols and schematics by default. * gschem-pcb interaction: Use the same

shortcuts to scale the grid \longrightarrow bracket-left/bracket-right ? * gschem feature request: show the grid in the status line * gschem wart: The page manager does not show in the window list. * gschem missing feature: gnetlist -g drc2 does not complain about two different power symbols on the same net. * pcb regression: Convert to component puts all pads on the same layer. * pcb missing feature: The local origin should be saved * pcb wart: Metric coordinates should be rounded to a

reasonable precision. (1.9999 \longrightarrow 2.0000) * pcb-wart: .pcb/preference stores the absolute path of the color file * pcb-wart: color-file variable is not read from .pcb/settings * gschem usability-request: pins should have pin number and pin sequence set to the same value by default. * gschem usability wart: Sometimes [shift-click] on selected sometimes does not work as expected (selects the whole connected net rather than unselect the segment) * pcb feature request: Drill holes in the fab layer should be given in mm, too. * pcb feature request: A modifier to not snap to the grid (but only to pins and tracks) * pcb wart: pcb library ignores sub directories * gschem usability improvement: If down_symbol does not find a fitting schematic file, it should offer to open a new schematic with the name taken from the symbol in the current working directory. * gschem usability request: Add a window menu for the various log and library windows. (Currently, they reside in "Options") * gschem usability improvement: When opening a *.sym file, automatically show hidden text, or at least ask to show hidden text. When opening a *.sch file don't show hidden text. * gschem usability improvement: Add rotation property to Edit-Text-Properties dialog * gschem wart: rename "Edit Text Properties" dialog to "gschem: Text Properties" * gschem wart: the window title of the attribute editor dialog is simply "gschem". Should be "gschem: Attributes" * gschem window handling: [i], [ex], [pm], [ol] should bring the text properties dialog to front if it exists hidden behind other windows. * gschem window handling: There are still some modal dialogs in gschem: The translate dialog ([et]), * gschem window handling: The window title of page manager and log window is just "gschem" * gschem window handling: The window title of page manager and log window should be set to some descriptive value. Currently, the window title is just "gschem" * gschem window handling wart: Select components does not show in the window list. * gschem window handling: mark not saved changes of the currently shown schematic with an asterisk in the title * gschem usability improvement: There should be just one edit properties action that covers, attributes and text properties, color, line width, fill type, etc. The properties dialog should morph according to whatever is currently selected. * gschem usability improvement: Apply the properties dialog to multiple objects in a selection in a

decent way \longrightarrow Show only properties shared by all objects in the selection. Change them all on exit. * gschem usability improvement: Optionally unset "show hidden text" when going up in hierarchy. * Warning: argument passing may have been confused by

```
a comma in a component value:  
Check 1D1 LED_3MM LED  
Maybe just use a space instead of a comma?  
---> Why not handle commas properly in the first place?
```

* gschem usability request: Sort the order of attributes in the attribute editor * pcb usability

improvement: Add a one button/one key way to rotate while the select mode is active. \longrightarrow [space]? * pcb window management: indicate a changed layout in the title of the main window. * pcb GUI improvement: add an accel key for the net list window. * pcb window management: bring child windows that already exist to front. * pcb usability wart: Select by name should find components

according to refdes even if the refdes is not shown * pcb usability wart: The select by name dialog should stay open after application. * pcb GUI wart: Find is hidden unnecessarily deep in the menu system. The scope of the find action is quite limited, too.

Suggestions:

- 1) rename "Select_by_name" to "find"
- 2) move the "find" item to the "edit" menu
- 3) unite the "select_by_name" with its sub dialogs
- 4) add a help button to explain the regexp syntax
- 5) alternatively "select" or "find" components
- 6) add the ability to search for properties other than name and object type: flags, width, layer, orientation
- 7) add the ability to search for nets
- 8) add the ability to search for combined criteria
- 9) optionally move the found objects to the mouse
- 10) optionally disperse the found objects
- 11) optionally do a substring search by default
- 12) optionall do a globbing search

* gschem GUI wart: The find dialog is pretty limited.

Suggestion: upgrade to the same functionality as suggested for pcb

* gschem/pcb interaction: Add the ability to do synchronized select in pcb and gschem * gschem misfeature: renumber divorces symbols with the same refdes. Should keep them to the same number. * pcb wart: drag-move does not work * pcb bug: update-schematics offsets the name to a few centimeters from the component * pcb usability request: Save the size and position of all dialogs on exit. * pcb feature request: add a second silk layer to the footprint format (just like *.pcb format) * pcb wart: rats don't print in the eps HID * gschem room for improvement: If an autosave backup is found, the dialog should offer to show the diff of the two files. * gschem usability improvement: If a symbol contains slots, insert should optionally add all slots at once with slot numbers automatically incremented. * gschem usability: The add text dialog should automatically apply the current text whe the mouse leaves the dialog. * gschem usability: The add text dialog should contain widgets to set text attributes like size, color, orientation,... * gschem usability: Accel to increment/decrement text size on [s]([shift-s] similar to pcb * gschem feature request: It should be possible to give different

values in components contained in ha hierarchy.

Variables that get evaluated while

netlisting? * gschem feature request: Alternative positions for visible attributes. foobar=, foobar-altpos1=, foobar-altpos2= ... * gschem feature request: A preferences and a settings dialog. * gschem feature request: A tabs-panel as alternative to the page manager * pcb feature request:

convert text to lines → useful for footprint creation * pcb feature request: convert pol to

lines → useful for footprint creation * pcb feature request: a dedicated move command for selected objects. The absence of such a command is almost a bug. * pcb bug: Option -only-visible does not work if the layer stack contains "solderside" * pcb bug: the eps HID produces invalid bounding bos syntax.

\$ epstool -bbox -copy infile.eps outfile.eps DSC Warning At line 2:

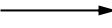
```
%%BoundingBox: 0 0 721.000000 721.000000
```

This line is incorrect. The bounding box must be in integer coordinates, not floating point. A floating point bounding box should be specified using HiResBoundingBox: A BoundingBox: should still be provided with integer values.

* pcb feature request: add a display action to set the mask visibility (rather than just toggle it) * pcb feature request: add a no-GUI HID to allow for non-interactive action scripting * pcb feature request: add an option to ChangeClearSize to explicitly refer to mask or polygon clearance. * pcb wart: Inconsistent syntax in *.pcb files – Element* are rendered with space between keyword and “[”. Pins, Pads and Lines written without a space at this point.

Januar 2011

* # gschem feature request: Add a modified move that temporarily disables rubberbanding. * # gschem feature request: Add a way to connect and disconnect crossing nets. * # gschem2pcb non-feature: gschem2pcb strips lower case letters from refdeses. This magic may surprise users. * ~ ~ gEDA documentation: A wiki page that states what gEDA/gaf is. ~ ~ * # gEDA documentation: There is no wiki page for gschem2pcb. Best bet currently is <<http://geda.seul.org/wiki/geda:faq-gschem2pcb>> * # gschem2pcb wart: The net of input/output symbols is evaluated from its refdes. Consequently, the restrictions to refdeses apply: No uncapitalized suffix, no hyphen, etc. Proposal: evaluate the net from dedicated attributes. E.g. “in-net”, “out-net” * # gschem feature request: An accel key that selects the “next” symbol. The next symbol should be chosen according to the algorithms used by autonumber. This would be handy when mass editing footprints, input nets, or values. * # gschem feature request: On selection, In the multiple attributes dialog automatically focus on and open the same attribute as was last edited. * gschem feature request: Add pin should take the properties of the last created pins as a default. pinseq and pin number should be incremented in case they are purely numeric. * gschem usability request: integrate the functionality of edit text attributes in the edit attributes editor * gschem usability request: unify single / multi attribute editor * gschem usability: EE should bring an existing attribute editor to the front of all windows, including non gschem windows. * gschem feature request: Add an action to update embedded symbols. This action should try to find embedded symbols in the available library. If there is a difference, it should display the difference and ask the user whether or not to update the symbol in the schematic. An option should allow to skip the questions and just update all. Crank out a note on what has been done in the log window. * # gschem bug: The multi attribute editor gets insensitive if it is open during switch between schematics. Work-around: Close and open the multi attribute editor. * gschem feature request: “Down


Schematics” should assist to maintain inter sheet connections.  Collect all pin labels of the sub-sheet symbol, compare them to the refdeses of input/output symbols and warn if there is a mismatch. * gschem feature request: “Down Schematics” on a sub sheet symbol that does not yet correspond to a schematic should open a dedicated sub sheet template. Pin labels of the sub sheet symbol should be collected and turned into input-output symbols * gschem feature request: add a way to extract pin names from subsheet symbols as named nets. * gaf/pcb feature request: add the notion of global variables. Some global variables may be defined by default. E.g. \$TITLE, \$FILENAME, \$DRAWN_BY, \$REVISION, \$DATE_OF_LAST_EDIT, \$PAGE_NUMBER, \$NUMBER_OF_PAGES, \$GEDA_VERSION. Gschem may fill these variables on session start-up, or on demand. The value of the variables may be updated from a dedicated file in the project directory. Alternative: Read the values from gafrc. * gEDA feature request: The print dialog should remember its options. At least, while the session lasts. * gEDA wart: The text entry in the attribute editor behaves in a non standard way. On selection the text is not highlighted and the cursor is at the end of the line rather than where the mouse cursor was during click.

The value of attributes in the attribute editor need two clicks to get

focussed for keyboard edit. Should be one click

Right click on the value area does nothing useful - just unfocusses the entry area.

* gschem feature request: A way to read the file name of a symbol - Suggestion: A line in the multi attribute editor * gschem feature request: some feedback on the embedded status of symbols. * pcb feature request: "Down footprint" similar to "Down symbol" * pcb feature request: A simple way to show the file name of a footprint * pcb bug: If "Save buffer elements to file" hits an existing file, it asks for confirmation twice. * pcb wart: Different accel keys for metric and imperial grid. * pcb feature request: The same accel keys to go up/down a predefined sequence of grid sizes * pcb-gschem feature request: cross probing - Select components and symbols in unison * pcb feature request: Add an option "Displayed element name = none" * pcb feature request: Don't rubberband tracks on rotate. * pcb feature request: Optional minimum copper spacing associated with route style * pcb feature request: More route styles, please. * gschem feature request: Back and forward buttons like in www browsers. * gschem bug: If a promoted attribute is moved somewhere else, the boundingbox still refers to the original position * gschem wart: autonumber warns about missing numslots with subsheets "numslots attribute missing. Slotting not allowed for this component. New slot number out of range." * gschem wart: the grid color of the light theme is next to invisible on some screens. * gnetlist wart: A refdes may not start with (contain?) a hyphen - yields invalid netlist * gschem mini improvement: autonumber should be clear that hierarchy refers to the hierarchy below current sheet * gnetlist wart: gnetlist does not report, if it changed the netlist * pcb wart: the preference dialog does

not update completely on ok. change size, go to some other tab click ok  size is changed but has no effect. * gschem: "Warning - There is an autosave file (...)" should inform about possible explanations. Another gschem running, previous gschem session died, etc. * gschem should check if another instance is working on the same sheet. /usr/local/bin/pcb * pcb: idea for search dialog: Add a user editable prefix * pcb feature request: sort and filter options for the netlist window - sort alphabetic/length of rat line, filter for substrings, regexp * pcb usability improvement: Left-mouse click on empty canvas should empty the find buffer. (Like it does with the select buffer) * pcb routing improvement: Show a circle of the CRC clearance of the current via at the head of a drawn trace * pcb routing improvement: Show DRC clearance while moving objects * pcb routing wart: Inconsistent DRC interpretation of clearance at rounded end caps of tracks * pcb routing wart: The draw line tool can (and will) select rats * pcb routing wart: inconsistent rubberband mode - Does not respect inner layers * pcb routing bug: undo while routing breaks if segments have been fused * pcb routing bug: update rats (sometimes?) ignores the end of a track * pcb routing wart: move track snaps to pins and pads * pcb routing wart: rubber band move of tracks does not care for vias * pcb routing mode idea: include a via to the list of currently found objects * pcb routing feature request: zoom to the next shortest/longest rat * gnetlist wart: The bom2 backend produces a file called "output.net" by default. Should be output.bom2 * gnetlist wart: The bom2 backend fails if no attrib file is supplied. Should use some reasonable default. * gschem feature request: The ability to turn on/off autonumber in the GUI * gschem bug: open-file presents recently-used rather than the PWD. * gschem feature request: default filter should be "schematics&symbols" rather than just "schematics". * gschem feature request: The filter in the component chooser should also search in attribute values. * gschem bug: strange unstable interpretation of "current working directory" when called via nautilus. * pcb bug: rubberband mode does not work with silk * pcb feature request: Make polygons aware of the net they are supposed to connect to. E.g with an attribute. * gnetlist wart: Lower case letters at the end of a refdes are ignored by gnetlist. This leads to unexpected errors on import to pcb. A better way to achieve the desired letters would be to append the value of the slot attribute to the refdes as shown in the schematic. This also avoids redundancy. * gattrib wart: The cursor keys do not act as expected. When in cell edit mode, the right key does not move the cursor to the right inside the cell. Instead, it warps the focus to the next cell to the right. * pcb feature request: More control on the units in the file. E.g.

a preference to enforce metric units. * pcb feature request: More control on tenting of vias. E.g. A toggle in route style to make vias non-tented. Or a menu item to make a

For large layouts:

* pcb wart: Menues are sluggish if FPS is low. * pcb feature request: A cancel button for DRC * pcb feature request: Write DRC result to file (only) * pcb feature request: Load DRC result * pcb feature request: Filter for DRC result * pcb feature request: More information in the DRC window - Affected layers, violating distance, type of object, current number of violations * pcb feature request: Put a user configurable limit on the number of rendered DRC violations * pcb feature improvement: A click on a node in the netlist warps the view so the node is in the center of the canvas. Please do the same for a click on DRC violation preview. * pcb feature request: delegate update symbols to a different processor * pcb feature request: make the preferences dialog persistent * pcb bug: auto-DRC is too conservative when going diagonal between round vias * PCB wart: Crosshair snas to pads on layers that are not current. * PCB wart: The curso icon of the insert tool is identical to the node-icon. * PCB wart: The arc icon looks like a question mark * PCB-bug: Regression: The grid is not set correctly on start-up * PCB wart: Obnoxious warning during DRC: "9 pads have the no paste flag set" * PCB bug: DRC loses the last move (of a via). * PCB wart: There is no obvious start of a list of shorts

→ Add a summary line * PCB feature request: Issue the number of shorts on update-rats. * PCB wart: The size for auto-DRC during manual routing should not interfere rules for DRC. This is an invitation to error. * PCB feature request: Only components (like only names) * PCB wart: design rules at the head of the DRC summary in the log window are always given in imperial units * PCB wart: [shift-f] does not remove the found flag set by click on DRC-preview * PCB feature request: Add the ability to ignore specific DRC-violations on subsequent DRC runs. * PCB feature request: A detachable toolbox rather than a rigidly placed side bar. * PCB feature request: A search utility for the netlist window * gnetlist wart: False positive warnings for attribute conflict for multi part symbols. E.g:

```
Possible attribute conflict for refdes: M1U1
```

```
name: value
```

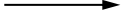
```
values: (OPA2244 #f OPA2244)
```

The "#f" is from a part that does not have a value attribute included in the first place. Proposal: Skip undefined values when checking for inconsistencies.

* gschem feature request: "add-last" in the add-symbol dialog. An accel key for this would be nice, too. * gschem feature request: A list of recently used symbols. * gschem wart: There is no scrollbar with the text fields in the single attribute editor * gschem wart: There is no scroll bar with the text field in the multi attribute editor * gschem wart: selected attributes in the attribute editor ignore the delete key * gschem missing feature: If down-schematic cannot find the schematic given by the source attribute, it should open a new sheet. The name should be preset to the string given by the source attribute. * gschem feature request: draw a line from a selected attribute to the corresponding symbol. (Much like pcb does) * gschem wart: After paste the pasted objects should be selected, not the original selection. * gschem bug: The autonumber on copy hook rennumbers refdeses even when they did originally not contain a number * gschem feature request: Do not show the whole path in the page manager. → Skip the common part.


Oktober 2011

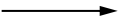
* PCB feature request: "first" should be default in the GUI of gerber export. (The gerber HID may go on to use the traditional naming scheme) * PCB trick: Temporarily delete the rat lines to find only connections that are actually connected. * PCB request: An action to find unconnected copper * PCB wart: Control modifier is used to add something to the set of found objects – Should be shift, for

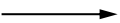
similarity with select. * PCB wart: ctrl-RightMouseClicked cycles the cross hair style.  This a waste of UI resources. Should be bound to something more useful. * gschem bug: It is impossible to paste into the value field of the attribute editor. The field immediately loses focus on ctrl. * gschem feature request: Let the attribute editor dialog act on multiple symbols. If more than one symbol is selected, treat symbols differently depending on whether they exist in every symbol.

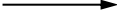
- a) attributes that are in all symbols and have the same value --> change in all selected symbols in unison
- b) attributes that are in all selected symbols and have different values --> Put values in drop-down list. Offer an option to make the value the same in all selected symbols
- c) attributes that are only in some selected symbols --> Same as b plus the option to add the attribute to all symbols.

* gschem feature request: Attribute lines in the attribute editor sorted according to the default attribute list. Plus alphabetically for attributes not included in the default. * gschem feature request: Put an asterisk in the title bar if the current sheet has changed (like pcb does). * gschem feature request: Offer to reload the current sheet if the file was changed by some other application. (like pcb does) * gp geda.org feature request: Offer a dump of the dokuwiki content for download. * pcb feature request: Try to guess the correct orientation of the current footprint and rotate an updated footprint accordingly. * pcb wart: Rubberband mode does not show in the preview, when vias are moved. * pcb feature request: The shrinking circle animation should stop on ESC * pcb wart: "Import schematics"

 "Import from schematic" * pcb wart: selected text does not move on drag * pcb/gschem harmony: [m] to move the current selection * pcb feature request: show the current layer in the status line (like the lesstif GUI does) * pcb missing feature: limit the boundary of eps export to the current viewport * gschem feature request: Add a way to change the symbol name in the GUI * pcb feature request: A read-only mode * gschem bug: gschem barfs on empty trailing lines. * pcb bug: unjaggy removes pre-routed vias (seen in monitor.pcb) * pcb/gschem compatibility: Bind rotate-current-object to middle mouse click – In select mode and in buffer mode. * pcb bug: Crosshair snaps much too strongly to diamond when crosshair-snaps-to-pins-and-pads is not activated. * gschem feature request: Add a standard attribute: obsolete=<date> . These symbols should pop up a warning when selected for instantiation. They should load fine for existing schematics. * pcb wart: If a pin of a footprint is selected, the footprint cannot be moved. * pcb bug: The number of pads on the bottom layer gets masked by the pad * pcb bug: The numbers of lines are not shown in the GUI * pcb bug: A selected footprint name cannot be moved by dragging with the mouse * pcb graphics artifact: Selected text leaves a non functional copy when moved. * pcb bug: The crosshair snaps to the middle of pads no matter what the setting "snaps to pins and pads" is set to. * gschem bug: gschem does not check for major updates of symbols in sub sheets in a hierarchy. * pcb wart: pcb does not play nice

with file managers.  import schematics does not work. * pcb wart: there is no intuitive way to set the corresponding schematic in the GUI. * gschem wart: The list of symbols with major version

can make the dialog outgrow the monitor height  add a scroll bar. * gschem wart: The "Text-Content" dialog is modal. Gnome 3 insists to place modal dialogs in the top middle of the

application window. So the dialog blocks the view on the schematic.  Please make this dialog non-modal. * gschem bug: [ctrl-s] does not work when in down-symbol * pcb feature improvement: edit all three text fields of a component on accel key [n] - description, reference designator, value. Add a button to make all of them the same. * pcb bug: rotate buffer via accel key [shift-F7] does not work * pcb wart: move to bottom rotates the component in a weird way, when the orientation of the component was cd Por non-90° * pcb feature request: add the ability to set the angle to absolute values to FreeRotateBuffer() * pcb wart: The GUI restricts placements of components to the area specified by "size"? Please remove this restriction. * gschem wish: copy a placed symbol to the buffer. That way, [ctrl-v] provides an intuitive way to throw some more of the same. * pcb wart: default_font should use the new format with units. * pcb feature request: Add an action to increase/decrease/set the width of strokes in the font. * pcb wart: Import schematics does not find newly created footprints. Work-around: Restart pcb. Suggestion: Add an action "reload library". * gschem feature usability: By default include attributes when drag-selecting a symbol. * gschem usability suggestion: Modifiers for drag selection

Inspired by Quartus

* A checkbox "Repeat-insert-mode" in the library dialog </markdown>

From:
<https://elektroniq.iqo.uni-hannover.de/> - **ElektronIQ**

Permanent link:
<https://elektroniq.iqo.uni-hannover.de/doku.php?id=wiki:geda-wartsnbugs&rev=1351597580>

Last update: **2012/10/30 11:46**

