

## Re: advice on selecting new PCB design package

---

*Source:* <http://sci.tech-archive.net/Archive/sci.electronics.cad/2006-03/msg00171.html>

---

- *From:* DJ Delorie <[dj@xxxxxxxxxxx](mailto:dj@xxxxxxxxxxx)>
  - *Date:* 23 Mar 2006 09:55:33 -0500
- 

Dave Boland <[NODARNSPAMdboland9@xxxxxxxxxxx](mailto:NODARNSPAMdboland9@xxxxxxxxxxx)> writes:

Hey, didn't we just do this last week?

Yeah. My participation is partly a "what should we work on" and partly a campaign to make sure people (1) don't underestimate what we do have, and (2) be specific about what we don't have. If I can "train" people to provide feedback that helps us make gEDA better, rather than just say "gEDA sucks!", then we won't waste as much time arguing and gEDA would improve.

At the moment, I'm filing all my replies in the "future file". I still have my day job (embedded software tools :) and my wife still wants her projects done too.

First, I do want to thank the gEDA community for all of the hard work, and to assure all of you that my comments are not meant as put-downs, but suggestions for improvement. Ready? Good!

Ready!

What the gEDA community needs to do is to make it easier to learn and use. Not that it is hard, and Stuart's Circuit Cellar articles will help — even if they are a year old (which I find hard to believe, but I'll take DJ at his word).

Publishing cycles, timing of articles to match issue themes, etc. As proof, the examples use the Xaw PCB, but the GTK pcb is almost a year old. Heck, we've already replaced the "old" GTK design with the new HID GTK design!

A specific suggestion is to copy a well know user interface such as

Re: advice on selecting new PCB design package

either Autocad or Microsoft Draw. Not that I'm in love with either, but it will reduce the learning curve.

The PCB HID project (now complete!) made some changes to the way mouse buttons work to improve usability. The lesstif HID is nearly 100% user customizable, so if you like a particular way of doing things, you can set it up that way.

I suspect a good User's Guide would make the most sense for this item, though. The GUI isn't something easily changed, because it's so subjective.

The gEDA community should always be asking itself "how can we make doing simple jobs easy, and difficult jobs not so hard."

Ok, so now we need feedback about (1) which are the simple jobs that aren't as easy as they could be, and (2) which difficult jobs should be automated most?

Most of our work is based on what we, the developers, feel we need to do our boards. Me, personally, I find the best feedback comes from people who \*don't\* know what the software's limits are, and just "expect" it to do something. I consider those expectations to be a better guide than reactions to what the tools actually do.

For example, my preference is to put effort into the trace optimizer, because our autorouter isn't that good and hand-routing doesn't always leave pretty results. I'd rather throw down ugly (perhaps drc-failing) traces, and let the optimizer clean it up. I don't want to have to measure track spacing or wade through pages of DRC logs, I want a button that says "Fix everything!".

The final aspect to productivity is feature set. After all, if it can't do the job it isn't productive. This is a marketing question more than technical or usability.

Yeah, marketing is hard to do when you don't already have people using your software. Hence my requests here.

My sense is that gEDA should aim for the market sweet spot which I suppose is something like:  
\* Cross section of 4S4P

That's an acronym I'm not familiar with.

Re: advice on selecting new PCB design package

Re: advice on selecting new PCB design package

\* Size of double EuroCard (forget these dimensions)

One thought we had was for "New layout" to offer a list of templates, pre-populated with the standard connectors, dimensions, etc. Building a PCI card? Pick one of the standard sized PCI card templates.

\* Nodes numbering into 2048, perhaps higher

I assume you mean usability in that range; we have no technical limit.

\* Schematic capture, layout, spice, electrical analysis of stripline/microstrip, 3D view of layout. Listed these in order of importance to me. Not a big fan of autorouters.

We have all but the last, and the HID interface lets us do the last with OpenGL if someone takes the time to add it.

On to a few other issues. Top of this list is `WINDOWS` version!!!

Yeah, we know. HID lets us drop in a Windows GUI for the PCB editor. Both gschem and PCB have been ported to windows before (via Cygwin) but a native (minGW perhaps) port would be best. We just need to find someone willing and able to add it.

Finally, work with someone to get a Knoppix (or other Linux LiveCD) to include the full gEDA suite. Again, more work.

We've discussed this before too.

Thanks for the feedback!

.