

Re: gEDA suite vs my creaky old Protel Client 3.5?

## Re: gEDA suite vs my creaky old Protel Client 3.5?

---

*Source:* <http://sci.tech-archive.net/Archive/sci.electronics.cad/2005-04/msg00260.html>

---

- *From:* "Mike Rocket J. Squirrel Elliott" <[j.michael.elliottAT@xxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxx](mailto:j.michael.elliottAT@xxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxxx)>
  - *Date:* Thu, 28 Apr 2005 08:37:23 -0700
- 

On 4/28/2005 6:01 AM Dave Boland wrote:

DJ Delorie wrote:

Stuart Brorson <[sdb@xxxxxxxxxxx](mailto:sdb@xxxxxxxxxxx)> writes:

: 8 layers, or 6 plus two power planes, although if you want a complex  
: outline you use one of the layers for that.

How do you get 8? Do you have to redefine the layer buttons on  
the left? If so, how do you do that?

In the default pcb-gtk settings, you've got component, solder, gnd,  
power, signal1, signal2, unused, unused. Eight. You can rename them  
or group them, but as far as the internal are concerned, they're just  
eight copper layers.

The Xaw version of pcb used to default to grouping some of the layers  
together to form logical layers, reducing the number of gerber plots,  
but you could always break the groupings to get back to eight  
individual layers. It's always been that way.

Note to the OP: A new, GTK-based version of PCB is slated to be  
released any day now,

## Re: gEDA suite vs my creaky old Protel Client 3.5?

Too late ;-)

Although I don't know if there's a snapshot of it yet, but it is what you get if you check it out of CVS. The Xaw version has been relegated to a branch.

2. no easy backannotation between PCB and gschem.

Yeah, we talked about that a bit. I suspect the hard parts are agreeing on a file format and redrawing the schematics if needed. Spitting out the difference between the netlist and the current connectivity shouldn't be *that* hard for pcb to do.

There was an article about gEDA in Circuit Cellar, March issue. I read it because I was considering using it and wanted to reduce the learning curve (never enough time and no one will pay me to learn something). I likely won't be using it now, but I'd like to share my thoughts (sent to the article authors) and all of you can comment as you see fit.

1. The article was a nice introduction, but not enough detail for me to feel comfortable doing a project yet. Hope they will do a series of articles on using all of the gEDA facilities, which are impressive.

2. What would be nice is a book similar to "Build Your Own Printed Circuit Board" (by Al Williams), which uses Eagle. This is a very good hand-holder, but not a great reference text. Still, it serves the purpose of getting people started with Eagle. I'm using Eagle because a free version came with the book. The problem is that who has the time to write a book!?! Doing a book is really a project in its self. An example of a good e-book (on Knoppix) is at:

<http://www.pjls16812.pwp.blueyonder.co.uk/knowning-knoppix/index.html>

Re: gEDA suite vs my creaky old Protel Client 3.5?

3. May as well go whole hog (so to speak) and then do a livedcd with the complete gEDA system on it. This is another bit of work, but I seem to use livedcd's more than I ever thought I would, and it would serve as a bridge to Windows users.

Hi -- OP here. I am a Windows user, so aside from the original curiosity about what gEDA has to offer to me vis a vis Protel Client 3.5, I know that it doesn't run under WinXP, but that is not a s.e.cad matter so I have set that little hurdle aside. So . . . whatever the heck a livedcd is, it sounds like it might let me eval gEDA. Count me in if you want to make a few.

--

Mike "Rocket J Squirrel" Elliott  
71 Type 2: the Wonderbus  
84 Westfalia: "Mellow Yellow (The Electrical Banana)"  
KG6RCR  
.