

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

---

*Source:* <http://sci.tech-archive.net/Archive/sci.electronics.cad/2005-04/msg00252.html>

---

- *From:* Stuart Brorson <sdb@xxxxxxxxxx>
  - *Date:* Thu, 28 Apr 2005 11:44:36 -0000
- 

Hi DJ --

DJ Delorie <dj@xxxxxxxxxx> wrote:

: Stuart Brorson <sdb@xxxxxxxxxx> writes:

:> PCB tops out at 6 layers,

: 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?

: My recent flag changes "open the door" to supporting up to 32 copper  
: layers with a trivial change, or more with a slightly bigger change.

Awesome! Thank you!

Note to the OP: A new, GTK-based version of PCB is slated to be  
released any day now, so some of the usability issues which others  
have mentioned will be reduced.

:> OTOH, it does lack a couple of features present in commercial tools.

: Like?

The biggies are: 1. no hierarchical busses in gschem, and 2. no  
easy backannotation between PCB and gschem.

Last year somebody submitted patches which implemented hierarichal  
busses. However, the code was apparently incomplete & buggy, and  
never made it into the main development branch.

As for PCB -> gschem backannotation, we have discussed this in some of  
our Free EDA meetings, and I have an idea of how to implement this in  
the gEDA/gaf file format. The project just awaits a developer with  
time to tackle the job. (I don't right now.)

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

There are also some little issues, like gattrib doesn't print, and won't handle net or pin attribs yet. Missing features in gattrib are my fault. Also, the project manager "geda" is still buggy. As far as I can tell, most power users bypass the project manager and run the tools individually from the command line. The project manager doesn't seem to be actively supported right now; it's stuck using GTK-1.2.

In conclusion, lots of folks use gEDA/PCB for real work. It works fine for small-to-mid-sized projects. It has several features which make it better than comparable commercial tools, e.g. Linux based flow, less buggy than Protel, all ASCII file formats, rapid bugfixes, not crippled, etc. However, it isn't perfect (although it's close!), and lacks a couple of features (hierarchical busses & backanno). If you absolutely need those features, gEDA isn't ready for you -- please check back later. But if your boards are simple enough that you don't need these power features, then check out the gEDA Suite -- it's available for free download, so you can always try it and make your own decision!

<http://www.geda.seul.org/download.html>

Stuart

---

- **Follow-Ups:**

- ◆ **Re: gEDA suite vs my creaky old Protel Client 3.5?**  
◇ From: DJ Delorie

- **References:**

- ◆ **gEDA suite vs my creaky old Protel Client 3.5?**  
◇ From: Mike Rocket J. Squirrel Elliott
- ◆ **Re: gEDA suite vs my creaky old Protel Client 3.5?**  
◇ From: Stuart Brorson
- ◆ **Re: gEDA suite vs my creaky old Protel Client 3.5?**  
◇ From: DJ Delorie

- Prev by Date: **Re: PCB holes overlapping**
- Next by Date: **Parametric sweep of model implementation**
- Previous by thread: **Re: gEDA suite vs my creaky old Protel Client 3.5?**
- Next by thread: **Re: gEDA suite vs my creaky old Protel Client 3.5?**
- Index(es):
  - ◆ **Date**
  - ◆ **Thread**