

Re: Orcad vs. Open Source

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2008-03/msg00474.html>

- *From:* Anubis <anubis@xxxxxxxxxxxx>
 - *Date:* Fri, 14 Mar 2008 17:41:56 GMT
-

The Orcad file format is closed and I can't do anything to try to fix this problem. The FreePCB file format, in contrast, is open and very readable. I can view it to see what is going on although I only did that when I started using the tool and didn't understand some things about it.

By FreePCB are you referring to the "PCB" program from the geda suite?

I thought about using an open source schematic editor when I started this project, but I didn't want to take the hit on the learning curve and I did not have a lot of confidence in a new tool (to me) not screwing up. Looks like I got that anyway.

I am currently working on a hobby project and I am using that opportunity to get acquainted with gschem and pcb. I thought about using kicad for both functions (schematic capture and layout) but gschem and pcb were lighter (memory foot print wise) and I had some experience using pcb for earlier projects

That's what I love about the open source tools – the outputs are ascii files you can edit and process. Hell, I can print it out and save it as a check against losing the file on the harddrive

.