

Re: schematics and PCB design software. what's best?

Re: schematics and PCB design software. what's best?

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2007-06/msg01195.html>

- *From:* Robert Baer <robertbaer@xxxxxxxxxxxxxx>
 - *Date:* Fri, 08 Jun 2007 19:09:12 GMT
-

Joerg wrote:

JeffM wrote:

Fernando Peral Pérez wrote:

i'm using OrCAD[...]I'm thinking of changing.

Joerg wrote:

cadsoftusa.com

Cadsoft used to be in the CAD business.
They are now also in the DRM business.

http://www.google.com/search?q=cache:rKVDtJhW8hgJ:en.wikipedia.org/wiki/Digital_rights_management

Before re-using anyone else's library,

http://groups.google.com/group/comp.arch.embedded/browse_frm/thread/f794e82d26b59e18/d7cf414

find out how to un-DRM the documents that are created with it.

That's about schematics generated with a cracked copy. Not an issue for me since I never copy parts of schematics from anywhere. Copying stuff and then using it for commercial purposes is, well, not quite within my understanding of ethics and etiquette.

This has generated some noise among Cadsoft users since it does inconvenience hobby users.

Re: schematics and PCB design software. what's best?

Re: schematics and PCB design software. what's best?

But ok, they've got to do something against hackers ripping them off. I doubt that their software engineers are working for free. Unlike many other CAD packages Eagle does not require those dreaded dongles. That is a serious plus in my eyes.

BTW libraries are another serious upside for Eagle. Other companies want serious Dollars for a new library. Cadsoft lets you download it for free. It's give and take, with most libraries contributed by users. Which also means you must take them with a grain of salt and thoroughly check things like footprints before using a part from a foreign library. I adopted the habit of creating my own parts.

Eagle is also remarkably inexpensive compared to products such as OrCad. That was one serious motivator for me to switch. But the main motivator was the support which is absolutely stellar.

Some free quality software?

Investigate gEDA and KiCAD.

Each was created by engineers "scratching an itch".

My experience with open source software is, to put it mildly, mixed. I haven't tried these two yet but I am not going to switch again. Support is also a dicey topic when it comes to open source products.

Over the years, i tried various versions of Eagle, and was never able to get any one of them to work; at best it was impossible to copy and/or place any simple part (spell resistor, please).

So Eagle is totally out of the picture for me *unless* someone would create from scratch, a Win2K hard drive with a completely WORKING version of Eagle, and sell that to me.

DipTrace (if i remember the name correctly) is *almost* exactly what i can use; it has some very nice features. But for small parts (SOT23-5 etc) it is not useful.