

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/msg01307.html>

- *From:* Robert Baer <robertbaer@xxxxxxxxxxxxxx>
 - *Date:* Sat, 09 Jun 2007 03:55:21 GMT
-

Joerg wrote:

Robert Baer wrote:

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_DRM

Before re-using anyone else's library,

http://groups.google.com/group/comp.arch.embedded/browse_frm/thread/f794e82d26

find out how to un-DRM the documents that

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

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. 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.

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

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).

It just takes a little practice. For example, in Eagle–speak cut means copy and so on. For me the switch from OrCad to Eagle was similar to learning to drive a car in Scotland, with right–hand steering. After a couple of days it felt comfortable.

Placing is easy. Click ADD, then pick a resistor you like from the discretes lib or from the SMT lib. If you have the German edition it would be called Widerstand. You can select EU style or US style, and all kinds of packages.

** *None* of those supposed functions worked in any shape, size or form. Most of the time i was lucky that the damn program installed inthe first place.

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.

C'mon, it ain't that hard. Mine does run on the Win2K PC but I can't sell you my PC because I need it. And I am not going to retire anytime soon ;–)

** You do not have to sell the PC; just clone the HD and remove what you do not want me to see, and send me that.

I will put it in a removeable HD kit carrier so i can swap it in when i need to run Eagle. What would be your asking price?

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.

Don't know that one. Eagle can do pretty much anything, even the teeny tiny SC–75 packages. But I usually don't do layouts myself.

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