

Re: What is your favourite PCB software?

Re: What is your favourite PCB software?

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2008-04/msg00838.html>

- *From:* Robert Adsett <sub2@xxxxxxxxxxxxxxxxxxxxxxxx>
 - *Date:* Mon, 21 Apr 2008 19:06:27 -0400
-

In article <36270c91-22f2-4be2-8324-5d613ab1d0a1@a23g2000hsc.googlegroups.com>, rickman says...

On Apr 20, 9:53 pm, Robert Adsett <s...@xxxxxxxxxxxxxxxxxxxxxxxx> wrote:

In article <2c0987d9-e17d-4797-9adf-4960fd3f3b6b@8g2000hse.googlegroups.com>, rickman says...

On Apr 10, 9:49 pm, James Morrison
<sp...@xxxxxxxxxxxxxxxxxxxxxxxx> wrote:

The real reason I like it is that the schematic and PCB are coming from the same database (other s/w has this to, PCB123 from Sunstone is one example). Thus there is no forward or back annotation—all modifications are applied to both simultaneously. This is a big bonus and seriously cuts down on chaos when things change.

Is it an open database? Can I write tools to pull data out of it and update it without using Eagle? For example, if I design in a part and in procurement the part is substituted, do I have to go into Eagle to make the change or can I change the database externally?

I don't follow Rick. If the part is form and function compatible then there is no need to update the PCB or schematic. If they are not then you need to go into the programs to changes the schematic and/or PCB. Surely you're not thinking of keeping approved source in the PCB/Schematic? (I know people who object to keeping resistor values in the schematic, preferring to keep them separately for configuration

Re: What is your favourite PCB software?

flexibility)

I put part numbers in my schematic. I then generate a BOM from the schematic using those part numbers. I don't have any sort of data base to track component changes, so I update the schematic when I change components. Not including resistor values would certainly make reading a schematic difficult without the lookup table for the values.

It has occurred to me that this is a good reason to use company part numbers. A company part number can be equated to multiple qualified parts for that socket. But this is another level of complexity that I am not currently prepared to support.

I'd bet you don't put resistor part numbers on your schematic (just resistance values) so you are already part way there :)

And how do you deal with variant stuffing?

Seriously, if you are producing more than one or two boards just for yourself you need to do this, it'll keep you sane as part supplies vary. Part specs can be as simple as listing approved manufacturers and their associated part number up to a full set of critical to quality parameters to make it easier for a contract manufacturer to suggest alternates. Something like Parts and Vendors helps a lot with the first.

There are still plenty of situations where I would want to edit the parts information in the schematic like it was a spread sheet or externally update it. Orcad actually provides this capability and would only be better if they did a more complete job of it.

It doesn't even matter if you agree with my example. The point is that open formats are better for many reasons.

That I won't disagree with.

I don't want to work with closes proprietary design files anymore and I won't. For my next design I will not use Orcad any more and I won't be using Eagle if their file formats are closed.

For myself, I find copy protection far more disagreeable and potential dangerous. Eagle does have that drawback as well even if it is one of the lighter forms. Eagle does have provision for getting at their

Re: What is your favourite PCB software?

Re: What is your favourite PCB software?

database programmatically and writing it out so it's not fully closed.

OTOH, I wouldn't want to give up the tight coupling between schematic capture and layout that Eagle provides. It's not impossible to get the schematic and PCB out of sync but you have to work at it a bit.

The last time I checked the open alternatives they were pretty much in their infancy, little coupling between PCB and layout, and difficult to install.

Robert

** Posted from <http://www.teranews.com> **