

Re: Low cost PCB layout software

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2003-12/0169.html>

From: Richard (*rh86_at_azglobal.com*)

Date: 12/27/03

Date: Sat, 27 Dec 2003 12:22:23 -0700

Ralph Malph wrote:

- > *The Eagle software from CadSoft seemed like it had*
- > *the most promise, but I tried to download the older*
- > *version 4.09 as someone recommended and could not*
- > *get either release to install.*

Ralph,

Eagle really is a good package. It'd be worth working through your install errors, if you've got the patience.

Cadsoft hosts newsgroups for product support, and I've been pleased with the user community as well. Before I bought, I scoured both to look for the chronic complaints and found there were no serious issues, and a lot of compliments – a nice side-effect of an open support forum. Here are their English newsgroups:

<news://news.cadsoft.de/eagle.userchat.eng>

<news://news.cadsoft.de/eagle.support.eng>

If you're like me, you'll dislike Eagle's mouse UI, but once you learn the keyboard syntax things go much faster (it also has some potent scripting capabilities). It does take a little training; the UI is not intuitive to newcomers. There's a tutorial you can download from Cadsoft's site – I highly recommend it.

The demo version is full-featured, but limited to ~4x6" board and 2 layers. (The board, traces, and PTH can exceed this dimension, but not component pads/holes.) The demo sadly does not include the PDF manual, which makes eval quite a bit harder.

And as someone else commented – when it comes to libraries, "Trust but verify" against datasheets. I've been very pleased with the extensive Eagle library, though I have found a few errors (like pad spacing on an SMT package). The good thing is... it's fairly easy to publish the corrections to the user community.