

Re: Low cost PCB layout software

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2004-01/0163.html>

From: Ville Voipio (vvoipio_at_kosh.hut.fi)

Date: 01/03/04

Date: 03 Jan 2004 12:27:18 +0200

Leon Heller <aqzf13@dsl.pipex.com> writes:

> *EasyPC is *very* much easier to use than Eagle (it's a proper Windows
> application), has fewer bugs and is cheaper.*

I've been using Eagle for several medium-complexity designs (two layers, TQFP/MLF chips, around 100 components), and have not found many bugs. There were some GUI crashes (not data-corrupting) in earlier versions, but haven't seen them in 4.11.

Eagle is not perfect. But it seems that it gets most jobs done. There are a few complaints I have:

- making new components is clumsy, as copy&pasting from one library to another requires opening the libraries several times. Separating the component layouts and symbols to different places would simplify the process significantly.
- the cut&paste method works is different from the "standard" cut&paste.
- only the vector font can be used on silkscreen layers. Not a very beautiful one.
- the tutorial is not a good one. Most of the learning process needs to be done through trial and error, especially with libraries.

Other than these, I have been rather satisfied.

- Ville

--

Ville Voipio, Dr.Tech., M.Sc. (EE)