

PCB Layout Designers

Source: <http://sci.tech-archive.net/Archive/sci.electronics.design/2007-05/msg02787.html>

- *From:* Tom <tom@xxxxxxxxxxxx>
 - *Date:* 21 May 2007 23:32:55 -0700
-

Morning All

Guess this is really 2 questions, first up.....

I am still using an old DOS version of EasyPC Pro for PCB layout, mainly because they want a stack load of money to upgrade. I did a long time ago post a question about free PCB packages, and found one, but the problem was that the package really relies on entering the schematic first, creating the nets the polishing the layout afterwards in PCB mode.

I don't actually dot he circuit designs, the company is my fathers, he designs them, I translate his PCB designs on paper to designs in EasyPC for printing out and either home manufacture or for sending of for bulk manufacture.

Does anyone know of an existing package that allows for this approach, ie not bothering with the schematic?

The second question....

This is kind of related to the first, I'd be interested to know if everyone designs their PCB's from the schematic stage upwards on computers, or whether there are many people creating the PCB layout only?

Tom

.