

Re: PCB Layout Designers

Source: <http://sci.tech--archive.net/Archive/sci.electronics.design/2007-05/msg03044.html>

- *From:* D from BC <myrealaddress@xxxxxxxxxx>
 - *Date:* Wed, 23 May 2007 18:57:34 GMT
-

On Wed, 23 May 2007 11:26:42 -0700, John Larkin
<jjlarkin@xx> wrote:

On Wed, 23 May 2007 09:23:55 +0800, budgie <me@xxxxxxxxxxxxx> wrote:

On 22 May 2007 15:27:32 -0700, john@xxxxxxxxxxxxxxxxxxxxxxxxxx wrote:

I've come across a number of commercial designers that still route manually. They'll go the circuit diagram route only if the board complexity demands it (say 20 or more packages). They say the main reason is time saving from not having to fart about creating unique library components and the ensuing struggle with third rate diagram editors.

I'm a one-man band when it comes to circuit and pcb design, and I prefer manual layout (two layers only) over importing a schematic into the pcb software. I find that:

- . package placement is easier without the ratsnest, but with the ability to have just a selection of interconnections showing.
- . the autorouter is great at achieving 85% faster than me, but often fails to complete and I have to undo sooo much that I haven't used autorouting in the last five years.
- . I have developed a fairly thorough and successful checking process that hasn't let a layout/connection error through in those five years.

We recently did a board that has over 1000 parts, including a uP and two FPGAs, 8 layers, parts on both sides. Hand checking that would

Re: PCB Layout Designers

probably take two people a week or so, one calling out connections and the other tracing them. PADS will do a full connectivity check on this board in about 2 seconds, and a full design-rule check in under 10.

We do the full schematic thing for even the tiniest boards. One nice thing is that you can ECO a schematic and export the changes to the PCB and keep things in sync. You can also resequence the ref designators on the board and back-annotate the schematic. We formally release the schematic and the PCB files together, and we have a rule that they **must** fully cross-check.

We never autoroute, and almost always go with the pins as originally assigned on the schematic, ie no pin or gate swapping. We also often pass the design around from person to person. An engineer may do some critical placement as a model, our layout guy does the real work, then another engineer or two may have a final lick at it, checking critical clock nets and such.

John

I treat my autorouter like a dog on a chain...
(What's smarter ...An autorouter or a dog?)
The dog can pull on the chain in all different directions.
I use a short chain and never let the dog loose..
Training helps...
I've had better results by tweeking the autorouting settings.
So... I just beat my autorouter with a rolled up newspaper when it's bad.
(I wouldn't do that to a dog..just the autorouter..)

I take my autorouter for short walks only..always on the chain..When I don't like where the dog is going..I just pull on the chain..

Some autorouter users I can imagine just let all the untrained dogs out and the pcb turns into a mess..
Right...like they think they got the Hal 9000 doing the routing.. :)
D from BC

.