

Re: Has anyone produced a board using Kicad?

## Re: Has anyone produced a board using Kicad?

---

*Source:* <http://sci.tech-archive.net/Archive/sci.electronics.cad/2006-05/msg00045.html>

---

- *From:* [fpga\\_toys@xxxxxxxxxx](mailto:fpga_toys@xxxxxxxxxx)
  - *Date:* 19 May 2006 22:29:59 -0700
- 

DJ Delorie wrote:

fpga\_toys@xxxxxxxxxx writes:

I wish the gEDA integration was as good as Kicad at times,

It's a common request, and we're making *\*some\** progress with it. Dan just offered his first pass at a "mode menu" for gschem, which lets you do pcb-specific things right from the gschem menu. PCB also has a listener port for remote control. He's working on tying things together in useful ways (like cross-selecting).

It really needs to be the same tool, one netlist, one symbol library (with referencing a schematic footprint and a pcb footprint, with common pin naming/annotation), and two physical windows (one for the schematic domain, and the other for the PCB domain), with a common working file which contains the physical "tracks" and object placement for both.

That way when you hand route the pcb assigning pins, new parts and nets show up as unplaced symbols and rats on the schematic window. And when you add schematic objects and connections they show up as unplaced footprints and rats on the pcb window.

.