PCBs aren't so hard to make and needn't break the bank

BY JAMES TURNER // APRIL 2010

Breadboarding a new circuit is a key skill and an important step in many projects—especially early on, when you need to move wires around and substitute components. But that very flexibility also makes it easy to knock wires out. Eventually, if your project is a keeper, you're going to want something with a bit more permanence.

Printed circuit boards (PCBs) solve all those shortcomings. But most people don't even consider translating a one-off project into a PCB design. For one thing, PCB fabrication has traditionally been expensive, viable only in commercial quantities. (One alternative is to do it yourself with etches and silk screens, a messy and time-consuming process.) Also, there are technical constraints involved with PCB designs that are daunting to the casual hobbyist. But it turns out that nowadays you can produce a professional PCB very inexpensively.

I only recently delved into the mysteries of PCB fabrication, for an upcoming IEEE Spectrum article on building a robotic digital microscope. Part of the project is a variable LED illumination system, and to dim LEDs you need to design a pulse-wide-modulation circuit. With a little research, I found a design based around a 555 timer chip, a power transistor, a couple of diodes, two resistors, and three capacitors. Not a huge number of components, but enough that I was dreading getting them all hooked up on a breadboard. Since I already had the design in hand, I was pretty confident it would work out of the box. That made it a perfect candidate for doing it as a PCB.

I had looked into custom PCBs a while back. I had even downloaded Eagle, a PCB design program from CadSoft Computer. I liked Eagle (and eventually returned to it for this project), but back then the cost of a single board—US $75 and up—stopped me. This time I discovered BatchPCB. BatchPCB consolidates a bunch of individual projects onto a single large board and then cuts them apart when they come back from the factory. Small boards can be fabricated for under $20 this way. There's a three-week turnaround, but I wasn’t in a hurry. I dove into designing my LED dimmer board.

A basic review of PCBs is in order here. A PCB consists of a thick, rigid insulating layer with conductive traces on the top and bottom. Because the middle insulates, the traces on the top and bottom can run over each other. To get a signal from one side to the other, you drill a hole called a via, which is like a trace running through the board vertically. You also drill holes for ICs, resistors, diodes, and other discrete pieces. The traces lead to the holes, and the components have solder pads, allowing them to be soldered in place.

Some components may be connectors to bring signals to the board, and you can also leave larger holes with solder pad rings around them to solder external wires directly. You can even add layers with traces in between the top and the bottom, although this can get pricey. At BatchPCB, a two-layer board costs $2.50 per square inch (about $0.40 per square centimeter), while a four-layer board costs $8 for the same area (about $1.24/cm²).

The first step in creating a custom PCB is laying out the schematic view. Place your components (which usually come from a component library included with your design software) onto a canvas, and then connect the pins with lines representing electrical connections. You may find yourself faced with multiple choices for the same part number. That's because many components come in different packages, such as a DIP (dual in-line package) chip or a surface-mount chip. For hobby PCBs, you almost always want to go with the big, clunky DIPs and SIPs (system-in-packages), because they're easily found at hobby venues and are easier to solder than surface-mounted devices, which are meant for commercial applications. Although the various packaging options may look the same in the schematic view, they will appear very different when you go into the layout view to actually design the board.
MIKE DIAMOND 04.07.2010
I have been using a similar service for prototype PCB's for several years now. They provide a free (minimalist) schematic capture and layout software. Their least expensive package is 3 circuit boards for $65 + tax. Based on fast turn around PCB prototypes we almost never do a wired prototype anymore. They are Expresspcb. Another competing service which also provides layout software is pcbexpress. A comment on wirewrap: For analog prototypes, wire wrap does not work very well, and in the time I could do a wire wrap prototype I could probably lay out the circuit board. Also, I would rather “cut and jump” circuit board traces than unwire and rewire a prototype during debugging.

HARRY H. ARENDS 04.07.2010
Great article but Eagle does include a autorouter in all versions. I just looked at the website and could not find a phrase saying it isn’t.

LORAN STRINGER 04.02.2010
Looks like wire wrap would have been a good solution, I may be showing my age.

ALLEN MOORE 04.01.2010
Minor clarification: SIP = Single Inline Package, SiP = System in Packages. I believe you meant the former.