The Power of Starting Small

One ham's first foray into board etching provides a lot of knowledge and confidence.

Matt Severin, N8MS

I enjoyed ARRL's® new DIY (Do It Yourself) campaign and short video (**www.youtube.com/watch?v=vlDwVhx7miQ&feature=plc**), which was released at the end of December 2011. It prompted me to start surfing the web to find an idea for my next project. At *MAKE* Magazine's website (**makezine.com**). I found many interesting small projects ranging from robots to pumpkin throwing trebuchets. I found my inspiration when I saw footage of a preteen girl explaining how to etch a circuit board.

Next, I needed a circuit to etch. After a little more surfing I settled on creating a theremin (the instrument that makes *Star Trek*-like sounds when you wave your hands above it — search the Internet for it if you've never seen one; they are very interesting). My version would be much simpler, modeled after a RadioShack weekend project called "Light Theremin" (**www.radioshackdiy.com/project-gallery/snap-circuits-optical-theremin**) that uses variations in light intensity to control the sound. I also decided to tweak the design and replace the 555 timer with a Picaxe microcontroller (**www.picaxe.com**). My students are participating in a new balloon project that uses the Picaxe microcontroller and this seemed like a good opportunity for me to learn more about this device too.

After building my prototype on a breadboard, I downloaded *Express PCB* (**www.express-pcb.com**) and started laying out the components, pads and traces to make my circuit board. I printed an image of the circuit layout and transferred the image to an overhead transparency using a copy machine. I placed the transparency on the copper side of the copper-clad printed circuit board (PCB) and used an iron to transfer the inked image to the copper surface. I carefully peeled the transparency away leaving the circuit layout on the PCB. The ink lines covering the copper surface prevent the etchant from dissolving the copper below them. I dropped the board into a bath of PCB etchant solution and waited for it to dissolve the unprotected copper.

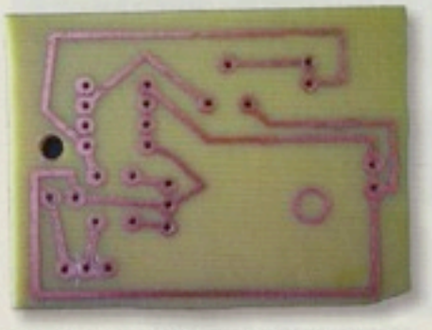

Figure 1 — Here is the etched and drilled PC board. Note the weak trace at the top. The broken area would need to be bridged with a short wire.

Once all of the unprotected copper had dissolved, I rinsed the remaining etchant from the board and checked the copper traces for continuity. Of course, there were gaps where the copper of the trace had been etched away (see Figure 1). I went back to the drawing board to thicken all of the lines and make sure all were solid and dark enough to guarantee there would be no gaps in the traces on the second attempt.

The Component Side

After redoing the board, it was time to mount the components. I drilled holes for the parts using a flex-shaft drill with a .040 inch jewelry bit. (The smallest drill bit in most drill sets is much too big for this type of project.) With the holes drilled I added the components and soldered them into place. Once I had the power section of my circuit on board, I checked the voltages to make sure everything was up and running correctly — it wasn't.

It only took a few seconds to realize I had missed a ground trace, so with a new hole and a few additional copper wires soldered to the board, I was able to bypass my mistake. The theremin powered up as expected and the rest of the project assembled without incident (see Figure 2).

I enjoyed building this project from start to finish. While my wife agrees the completed theremin is just a little box that makes an annoying sound, it has given me the confidence to try other projects. I now know that I won't be limited by the lack of a circuit board.

DIY Pointers

I leave you with a few lessons learned from this project. Hopefully, they will help you avoid some pitfalls:

Figure 2 — Here is the completed theremin and battery all in an Altoids case.

1. Double check your circuit before you print. This saves the time and effort needed to bypass mistakes later.

2. When etching your circuit board, the traces and pads need to be *fat*. *ExpressPCB* uses a default trace width of 1/16 of a millimeter, which is too thin for home production methods. In my experience, a width of ¼ of a millimeter is necessary.

3. The etchant solution, ferric chloride, turns things a funny brown color — wear gloves.

4. While an Altoids tin is a cool project box, eating all the mints in one day is not recommended.

Photos by the author.

Matt Severin, N8MS, an ARRL member, is the principal of Dowagiac Middle School in southwest Michigan serving over 550 students in grades 6 through 8. Matt has incorporated Amateur Radio into his classroom since 1999 when he first earned his Amateur Radio license. He has worked with students ranging from grade 3 to 12 and has found ways to integrate Amateur Radio at all levels. Matt has been licensed for 12 years and holds an Amateur Extra class license. He can be reached at 7555 E Main St, Eau Claire, MI 49111-9664, **n8ms@arrl.net**.

PCB - Easy and Cheap

- Why only now
- How much does it really Cost
- How to get started

How is a PCB Made

- Gerber files to mask
- Drill board material
- Expose Boards
- Plate traces and holes
- Etch board
- Solder mask and silk screen

Multi-layer Boards

- Normal is 2 sided
- 4 layer - multiple boards bonded together
- Power and Ground on inner layers
- Vias
- Buried Vias
- Inner Layers can have traces
- Copper pour

Layout Software

- KICad
- Design Spark PCB
- Eagle
- Board House Supplied
- Commercial

Design Steps

- Schematic Capture
- Generate Net List
- Assign Footprints
- Place Components
- Route traces
- Generate Gerber Files

Schematic Capture

- Might as well put schematic in program
- Component Libraries
- Component Editors
- Place Symbols
- Interconnect Components

Assign Footprints

- Through Hole or SMT
- Choose Size
- Type

Place Components

- Draw Board Outline
- Move Components from Rats Nest
- Orient Components

Route Board

- Set Rules
- Choose Number of Layers
- Draw Traces
- Autorouters
- Design Rule Checks

Generate Gerbers

- Just “Print” Gerber Files
- Generate NC Drill File
- Zip up and Send

PCB Resources

Programs

1. KiCad - http://www.kicad-pcb.org/display/KICAD/KiCad+EDA+Software+Suite
   Links to download, documentation, and tutorials.
   Video tutorials on UTube.

2. Designspark PCB
   http://www.designspark.com/page/designspark-pcb-home-page
   Free - but must become member of DesignSpark.
   Lots of components from manufactures.
   They want to supply your parts.

3. Eagle (Easily Appicable Graphical Layout Editor)
   http://www.cadsoftusa.com/?CMP=KNC-G-BRND-EC
   Limited free version - non-profit, 1 sheet schematic, 2 layers, 4.3 x 3 inches
   Probably $600 for next better edition.
   Owned by CadSoft.

4. Diptrace
   http://www.diptrace.com
   Competitor to Eagle - Non-profit license tops out at $650

5. Board House Proprietary
   http://www.4pcb.com/free-pcb-layout-software/index.html?gclid=CKGq5-jW47UCFc5AMgod42YAbQ
   Only get Gerber files after first order to Advanced Circuits.

   http://www.expresspcb.com/expresspcbhtm/free_pcb_layout_software.htm
   No Schematic Capture.

   http://bayareacircuits.com/pcb-creator/?gclid=CI3Mn9rX47UCFexaMgodc1QAmQ
   $350 to “upgrade” to version to export Gerber Files.

6. High End Commercial Products - You can’t afford them
   http://www.mentor.com/products/pcb-system-design/layout-routing/expedition-pcb/
   Won’t show the price on their website.

   http://blog.zuken.com/index.php/start-cadstar-for-free-today
   CadStar - again no price on web site.