All-Clad: An Electronics Prototyping Canvas

Aaron Faivre | 03/17/2025

Introduction:

After designing a circuit, a prototype must be built to verify its functionality. In the age of cheap, quick turn-around printed circuit board (PCB) services, building a circuit "from scratch" is not always worth the time. Despite this, sometimes a circuit designer wants to verify their design *now*.

I often find myself in the category of the impatient, excited designer. I have used various methods to build prototypes over the years. Eight years ago, I was milling PCBs on my homemade CNC milling machine. In the past couple years, I returned to building perfboard prototypes. I found that I could verify a design faster using perfboard then a CNC mill.

I sought to improve my prototyping process. I wanted a method that was quick and kept the "spirit" of an electronics prototype. For a prototype, production speed and functionality are the two most important qualities. After giving it some thought, I arrived at my now preferred prototyping method. It is a perfboard with a special pattern, which I have dubbed "All-Clad."

On the spectrum between a PCB and a perfboard prototype, an all-clad prototype lies somewhere in the middle. With all-clad, a prototype can be made in as little as an hour. This paper will briefly outline my method for developing perfboard prototypes and then provide a tutorial on making an all-clad prototype.

Traditional Perfboard:

Figure 1 show an example of a perfboard prototype. The “traces” are pieces of wire which have been soldered directly to the underside board:

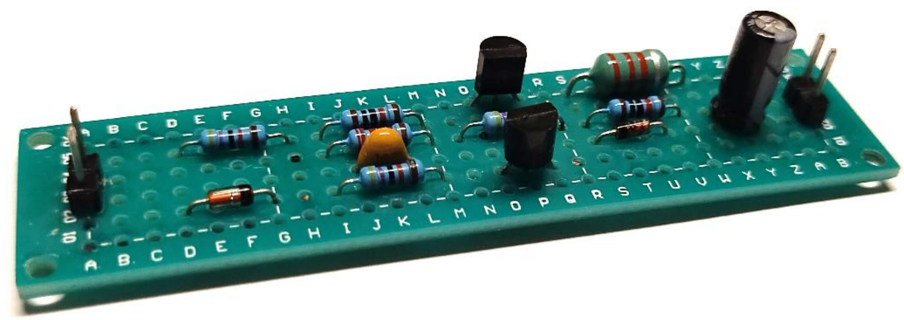

Figure 1 - Perfboard prototype

Constructing perfboard prototypes can be a tedious, error-prone process. Though with good technique, a working prototype can be fabricated in short order. I found double-sided perfboard to be hard to work with, as solder tends to fall through the holes. For this reason, I prefer single-sided perfboard.

When building a perfboard prototype, I first layout the board in a computer-aided design (CAD) package such as KiCad. I found this easier then laying out the circuit at the workbench. Prior to layout, I draft and import a template of the perfboard. I then lay out the circuit over the template. Figure 2 show a perfboard prototype designed in KiCad:

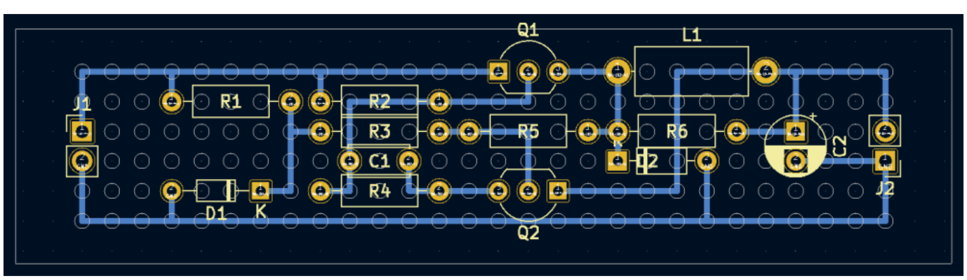

Figure 2 - Perfboard prototype, CAD layout

I then transfer the pattern to a perfboard using a permanent marker. Here, I distinguished between “pads” from “traces” using red and black ink:

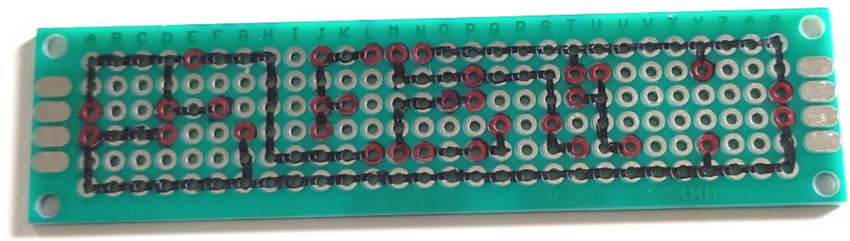

Figure 3 - Perfboard prototype, transferred layout

The traces are formed using solid-core wire which is bent into u-shaped “staples.” The staples are temporarily held down with tweezers and then soldered into place. After soldering, the staples are trimmed flush to the top side of the board.

If a trace is no more than three holes-across, it can be easily formed using a solder bridge (solder-blob) instead of using a staple. The components are then soldered in place. The connections between traces and components are made using solder bridges. This process is fairly messy, and results in many “cold” solder joints:

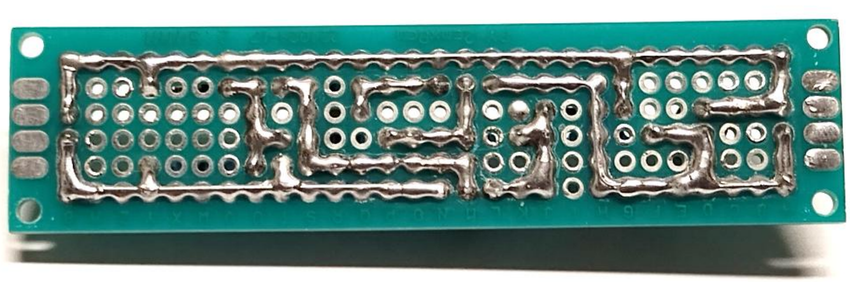

Figure 4 - Perfboard prototype, bottom (rotated)

All-Clad:

While inking the traces and pads for a perfboard by hand, I have often thought how convenient it would be if the process ended there. This is what I came up with:

Figure 5 - All-Clad perfboard

This perfboard pattern is composed of through-plated solder pads which are shorted together through a grid-pattern on the top and bottom of the board. A permanent marker is used to draw a circuit pattern directly onto the board. The board is then submerged in an etchant, such as Ferric Chloride ($FeCl_3$.). The remaining copper forms the traces and pads of the circuit board.

The goal of this pattern is to provide a prototyping method which requires little tooling. An all-clad prototype can be designed on a computer, then drawn directly onto the board without leaving the office.

Since the initial pattern was made by a professional PCB manufacture, the traces are straight, and the holes are drilled with great accuracy. This is in contrast to traditional hand-drawn boards with their characteristic wavy-traces. Also, the prototype may be double-sided, which can be difficult to achieve using other prototyping methods.

The rest of this paper will focus on developing an all-clad prototype. This technique is ideal for verifying subcircuits. The circuit chosen for this tutorial is a single-ended primary-inductor converter (SEPIC). This is a variant of boost-buck converter used for DC-DC conversion.

This SEPIC circuit is one of my designs. I will not discuss the circuit here, but I do plan on documenting the design in another article. Every component was chosen to have lead spacings of multiples of 2.54mm (0.1”), with the exception of the capacitors, which have 5mm lead spacings. This ensures that every part will fit into the grid.

Design and Layout:

A template for a 1 x 4” all-clad perfboard is first imported into KiCad as a .dxf file. The components are then placed on the board and traces are drawn according to the ratsnest. To make this easier, the grid-snap setting can be set to 2.54mm (100 mills).

This board is single-side, but vias may be placed to join traces on the top and bottom of the board. The finished layout is shown in Figure 6. A 3D rendering of the prototype is shown in Figure 7.

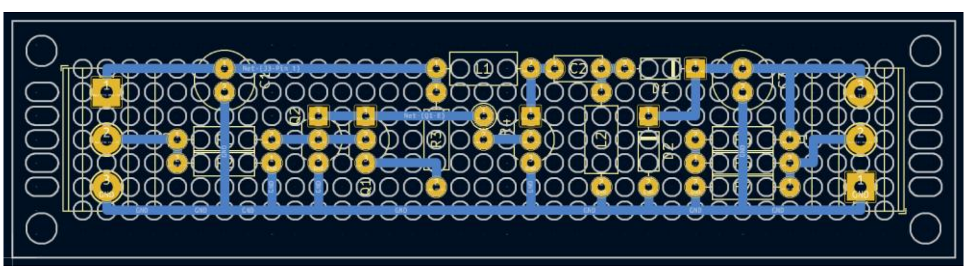

Figure 6 - SEPIC, layout

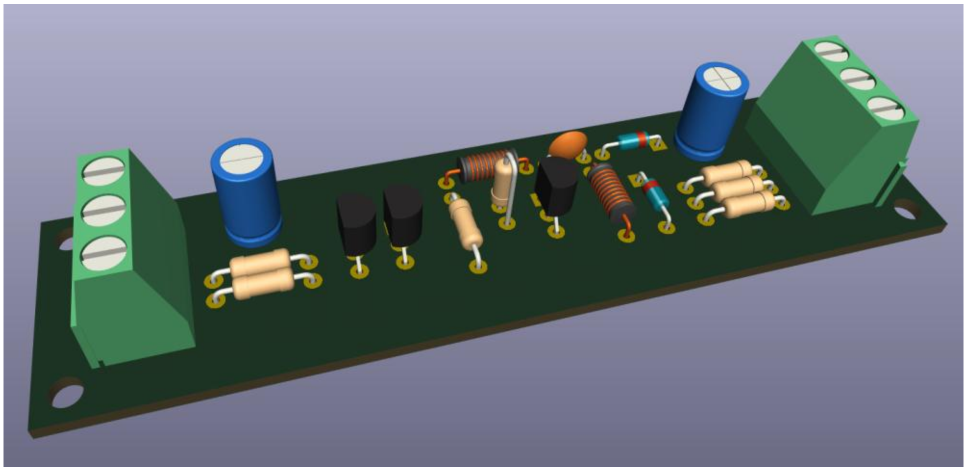

Figure 7 - SEPIC, 3D rendering

Drawing and Etching:

Enable “flip board view” to show see the board from the bottom-perspective. If this is not done, the mirror-image of the layout will be transferred to the board.

The “Ultra Fine Point” permanent marker by Sharpie is ideal for transferring pads and traces onto the board. For long traces, such as the ground-bus shown at the bottom of Figure 6, a less precise marker is faster to use, such as the “Fine Point” permanent marker by Sharpie. To fix small mistakes, scratch away the dried ink with a scribe. For large mistakes, wipe the board with a cloth soaked in isopropanol (90%).

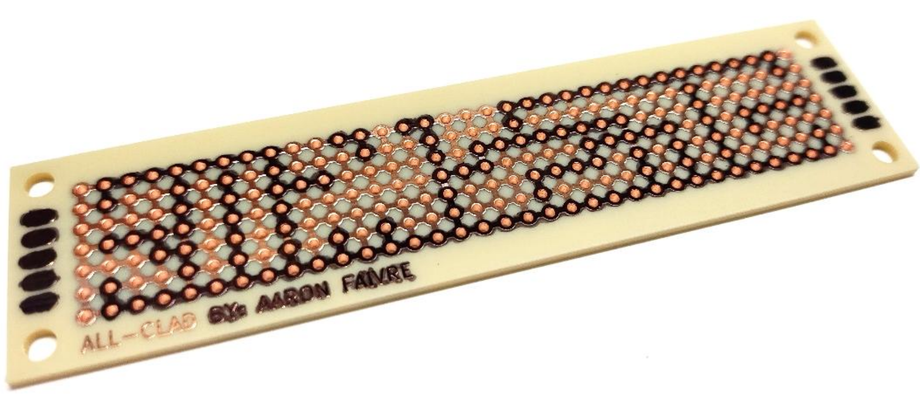

Figure 8 - SEPIC, transferred layout

All the holes are through-plated, with the exception of the four mounting holes. For this reason, only draw on the surface of the board; avoid getting ink within the holes. I had ordered the first batch of boards without through-plated holes, but unfortunately the fabricator put them in anyway. Luckily, it is not too challenging to keep ink out of the holes.

To etch the board, follow the etchant manufacturer's instructions and adhere to all safety precautions outlined in the safety data sheet (SDS). As of writing this, I have only etched circuit boards using $FeCl_3$. I etch circuit boards in a Pyrex dish with a locking lid. I will tie fishing line to the corner of the board and hang it outside the dish.

The fishing line makes it easier to pull the board from the solution. Frequently agitate the solution to speed up the etching process. At room temperature, it usually takes me 20 minutes to etch a board using $FeCl_3$. After etching, rinse the board thoroughly with water.

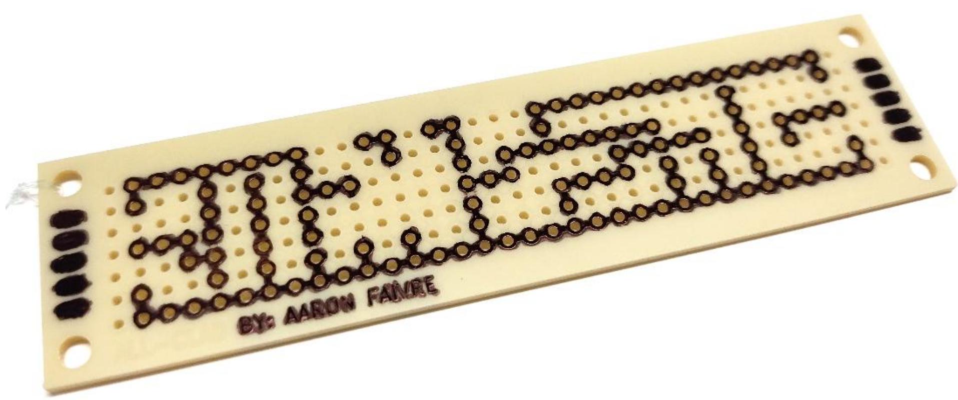

Figure 9 - SEPIC, etched with layout markings

After etching the board, the ink must be removed. To remove the ink, either scour the surface with a wet abrasive pad, or dissolve the ink using a clothe soaked in isopropanol (90%).

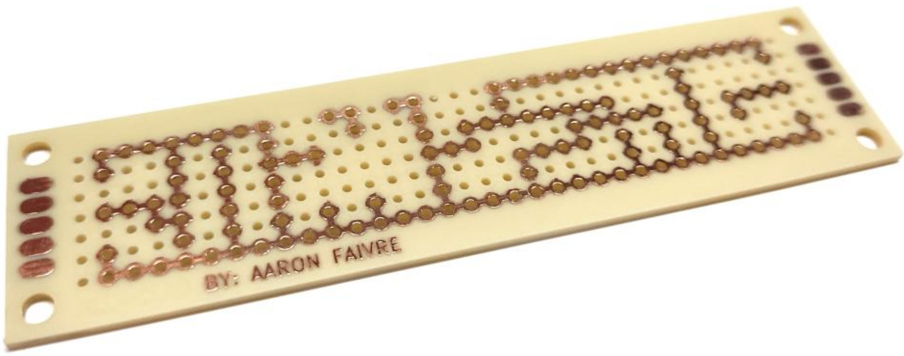

Figure 10 - SEPIC, etched, cleaned

Double-sided boards can be made using the same process. When etching double-sided boards, ensure the through-plating will not short-circuit the top and bottom traces. If the board is etched sufficiently long, the through-plating will simply wash away. Stray through-plating will cast dark shadows, as shown in Figure 11. For single-side boards, leftover through-plating is not a concern.

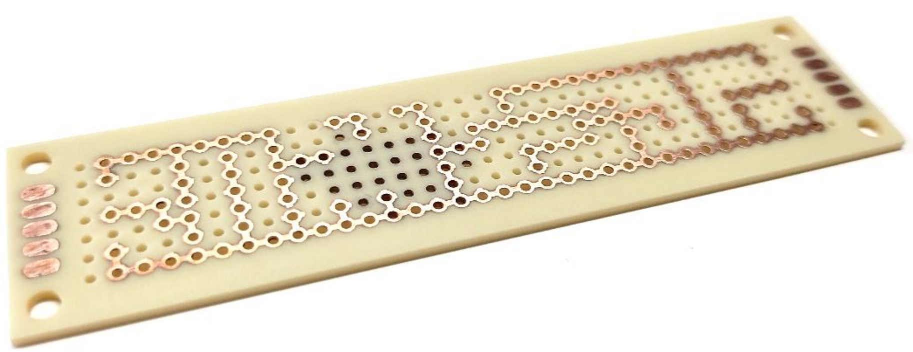

Figure 11 - Under-etched through-plated holes

Soldering and Cleanup:

Soldering an all-clad prototype is no different than any other circuit board. Prior to soldering, inspect the holes for any through-plating which may not have been etched away. If there is a chance through-plating will cause a short between traces between top and bottom traces, the hole may be reamed out using a small drill bit.

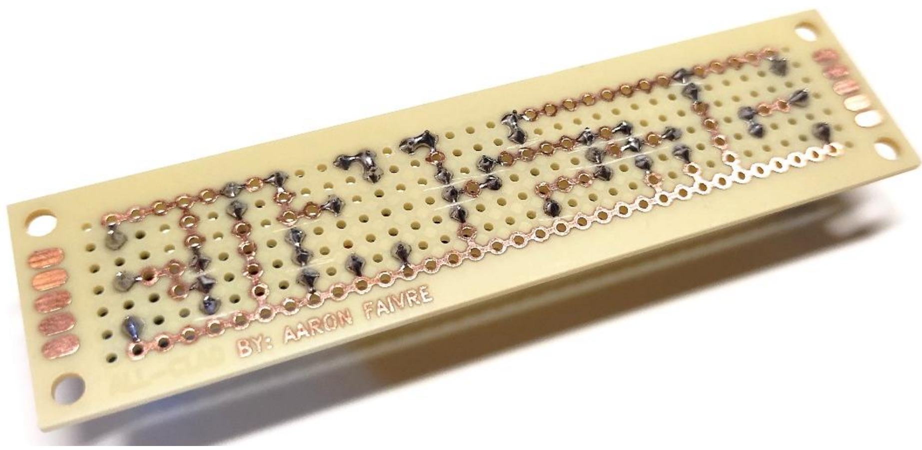

Figure 12 - SEPIC, soldered and scored

To create a connection between traces on the top and bottom of the board, solder a solid-core wire into the hole. This process is made easier by using two self-locking tweezers. Hold the wire with the first tweezer and use the second tweezer to hold the first against the board. Solder both sides using this method and trim the excess wire.

If there are any connections that appear too close for comfort, use a scribe to carve a small valley between the two pads, as shown in Figure 12. Brush away the dust, and remove any flux with isopropanol (90%).

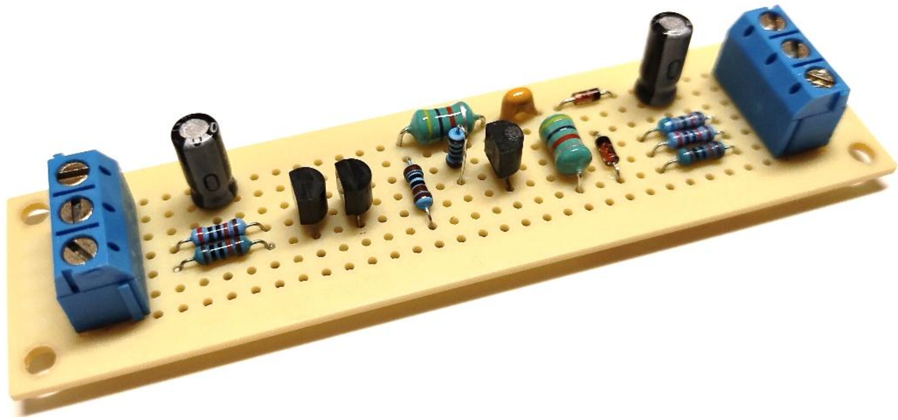

Figure 13 - SEPIC, finished all-clad prototype

Conclusion:

I made this perfboard pattern to increase the speed of my electronics-design workflow. I have used other prototyping methods in the past, such as PCB milling, point-to-point soldering, wire-wrap, breadboard, and perfboard.

I found the hassle of calibrating and programming a CNC machine to mill PCBs was not worthwhile. Much care had to be taken to perfect the process, and accidents still happened (tool-crashes, workpieces-shifting, lost-steps, power-outages etc.). I found building perfboard prototypes was a more reliable method than milling PCBs, though time consuming. Assembling a prototype on a breadboard is simple, but breadboards are not always suitable for verifying high-frequency circuits.

I wanted a process even easier than using perfboard, and "all-clad" was my solution. The solution seems simple and obvious and has likely been done before. If it has, I have not been able to find prior art. If anyone reading this has seen an example of this perfboard pattern being used, please email me at: aaron.c.faivre@protonmail.com. If there are any improvements, modifications, or other ways to use the pattern, I would be pleased to hear them!