The MUPPET Show

Chuck Adams—K7QO chuck.adams.k7qo@gmail.com

No, this article is not about Jim Henson and his creation of 1955. This is about a method of construction using PCB material without having to glue pads to the board or drill holes. It combines Manhattan building, ugly construction and the making of printed circuit boards without a lot of extra work. I made up the term MUPPET from Manhattan, Ugly and Precise Placement Experimental Technique. I have used all the building techniques individually, but I find this combination to be a lot easier and saves me more time in the building process.

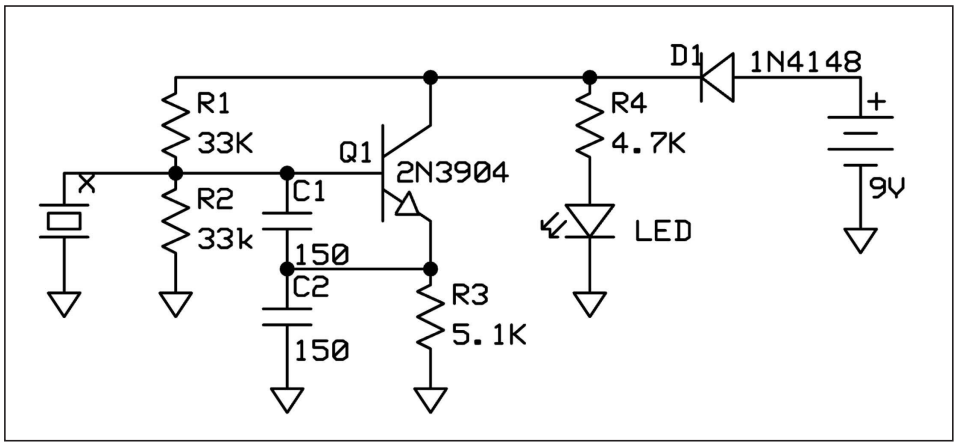

Figure 1—A Colpitts crystal oscillator is a simple project to start with.

Getting Started

Let's first start with a review of some of the material in Chapter 23 of the latest *ARRL Handbook*.

Manhattan style construction was first used in student labs at Stanford. It was brought light at a building contest in Dayton at the QRP group meeting called Four Days in May (FDIM). Jim Kortge, K8IQY, won the building contest with a beautiful transceiver that was constructed using only 2N2222s for the active components. The physical layout looked like a map of the streets of Manhattan Island and a number of the larger components with their vertical height look like sky scrapers, thus the name. You make circular or rectangular 'pads' of PCB material that are super glued to a PCB subtrate. Then component leads are soldered to the pads and/or to the ground plane in the case of ground connections.

Wes Hayward, W7ZOI, and Roy Lewallen, W7EL, and many others promoted in *QST* articles the ugly or ground plane construction. They emphasised point to point wiring without a formal structure.

And of course there is the printed circuit board, PCB, with a regular layout and drilled holes for component insertion and soldering. For the more complex circuits, the drilling of the holes may take considerable time in addition to the expense of a Dremel tool, a drill press and the smaller and more expensive drill bits.

All of the above, plus the newer surface mount technologies, offer the home builder a large number of options for construction of circuits.

PCB Layout and Etching

I have been working for many years now using the so called Toner Transfer technique for doing single sided PCBs. I first started, like every one else, using a common household iron to fuse the laser printer toner to the PCB. Then you etch away the exposed areas using a two part hydrogen peroxide with one part muriatic acid solution. The muriatic acid is the common hydrogen chloride acid using in swimming pool maintenance and found at your local Home Depot or Lowes. Runs about four bucks per gallon.

I can etch a 6" × 6" single sided PCB with 1/2 cup of hydrogen peroxide to 1/4 cup of muriatic acid in about ten minutes. Then using common backing soda to neutralize the etching solution for disposal.

The *Handbook* has a description of my technique for etching the board. Since then I have simplified the procedure and greatly reduced the materials needed.

Here's a quick summary of what you need.

- Steel Wool for cleaning boards
- Hammermill Color Laser Gloss paper
- 750 ml Pyrex dish
- Hydrogen Peroxide 32 oz bottle
- Muriatic Acid, one gallon jug
- PCB material
- Shear or Cutter
- Laser Printer, I use a cheap Samsung printer
- Clear Enamel spray paint
- Laminator
- Plastic 1/4 measuring cup and a plastic spoon.
- Baking Soda for neutralizing the used mixture.

Pick a Project to Build

Using any one of a number of PCB CAD programs to carefully layout a board. As you will see, you have to do a layout using any building technique you choose, but here you do not need to use circular or square pads and super glue in Manhattan construction. Lots of time is spent in waiting for the glue to cure. No more drilling of holes and the time and energy that process requires for PCBs using PTH components. PTH is parts through holes or what you and I call components with leads.

The board layout is done on the top layer of the board. You may have to reverse or mirror the output to paper using software readily available and then iron on or use a laminator to transfer the toner to the PCB.

To transfer the toner to the PCB, I use a cheap laminator made by GBC and model is called BadgeMates. I found it at one of the big office supply stores. The qrp-tech group has found a source for under fifteen bucks online and another one for under ten dollars, US. Check WA8MCQs recent columns for possible information. I preheat the PCB by running through the laminator a couple of times. Then placing the printer output on the board carfully with the toner to the PCB I run the the combo through about 5 more

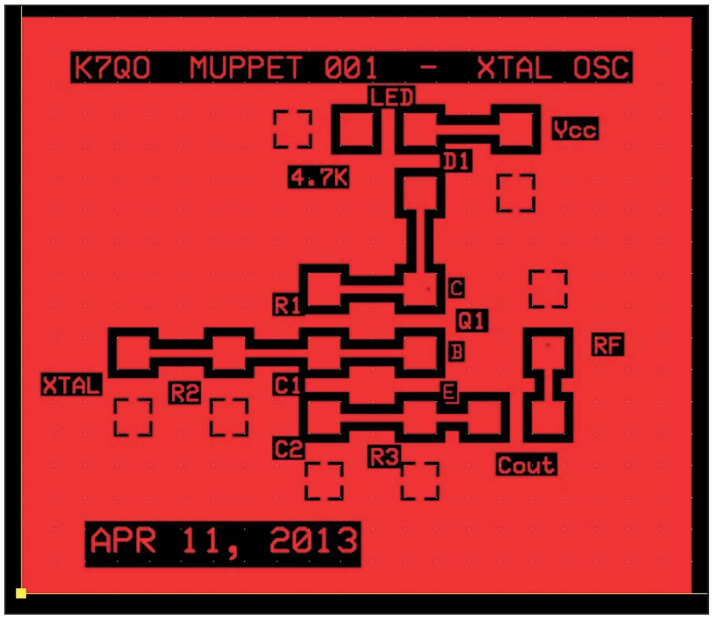

Figure 2—Oscillator board layout.

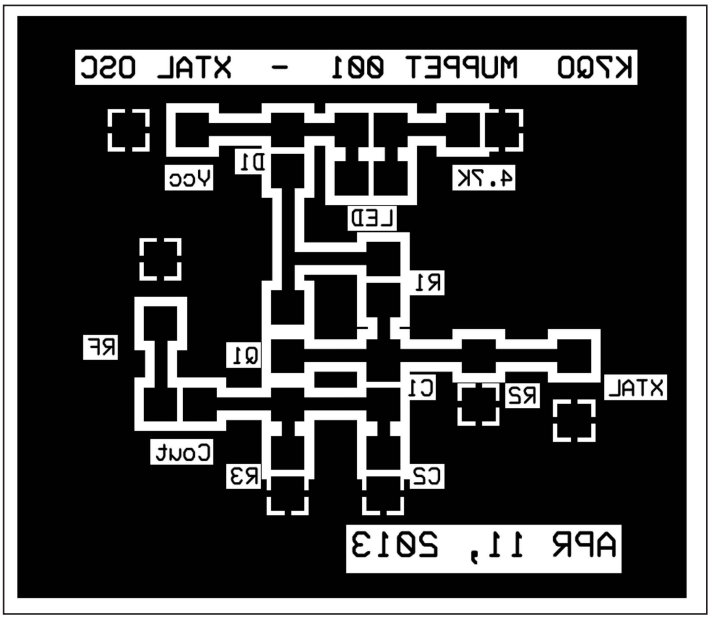

Figure 3—Reversed layout for printing (not to scale).

times to insure that the toner is fused to the copper layer. I get a perfect board every time. If you make a mistake during this process, then soak the board in water to remove the paper. Then just use steel wool to remove the toner and start over. You haven't lost any thing at this point other than one sheet of paper and some time.

I soak the paper and board in the pyrex dish for 10 minutes or so and then gently remove the paper without removing any of the toner.

Mix the 2:1 mixture in the pyrex dish. The peroxide (2 parts) goes in first and then the muriatic acid (1 part). Use eye protection and use rubber gloves. Do this in a well ventilated area, preferably outside.

Soak the board in the mixture while GENTLY stirring with a plastic spoon. Be careful not to spill the mixture on anything. You will see the liquid turn from clear to a light green (copper oxide) and then the copper will be removed from all the exposed areas and the toner will protect the copper layer areas to be kept.

Clean with water and neutralize the mixture with plain baking soda for disposal. Use common sense. Then remove the toner using steel wool. Acetone could be used, but it is a nasty chemical and getting more expensive as we speak.

Now you have a nice clean board. I take a spray a very very very thin clear layer of enamel and let it air dry. Do not bake it or otherwise speed up the drying process as you will cure the layer and make it difficult to solder through. I find that the heat of the soldering iron and the solder melts away the enamel and makes a nice solder connection. The board will remain oxide free for years to come and also protected during the construction process. Oh... Did I remind you to use a very very thin layer? You can do a lot of boards with just one can of spray paint. I probably use up more paint clearing the nozzle than on the PCBs.

Start Simple

You have to walk first. So start with a few simple projects to get a feel for the techniques involved. Let's walk through a simple crystal oscillator. Figure 1 is the schematic of a simple Colpitts crystal oscillator. Nothing fancy.

I have added a reverse voltage protection diode, 1N4148, just in case you want to hook up the battery backwards and add simple LED to show that the circuit is on and running when powered up correctly. Also the LED serves to remind us the circuit is powered up so that we don't go off and leave it on the bench and run down the battery.

For the beginning muppet boards, I used 0.10" squares for the mounting of component leads and components. Use 0.040" PCB traces for power and signal lines. You can put the ground connections anywhere on the copper plane without pads, but I like to use the thermal pads with traces from the pad to the ground plane to remind me where the ground leads go. It also requires less time and heat to solder to and the solder stays nicely within the pad area.

Another handy thing to do is to add text to help place and label parts. Speeds things up for me during the construction phase and requires only a little extra time during the layout process.

For ease of construction I have use 0.20" spacing for the parts leads. We are not trying to fit this onto a postage stamp. Later, for more difficult projects where you do need the space, you can reduce the distance and the pad sizes. With the laminator I can do 0.008" traces (8 mils) with no failures.

Figure 2 is the computer display image of the crystal oscillator PCB layout using ExpressPCB software.

I print this off reversed (not to scale), as shown in Figure 3.

Using the laminator, we fuse the toner to the PCB and then remove the paper with soaking in water and then we etch the board. Afterwards, remove the toner with steel wool, spray the board with a light coat of clear enamel and then solder the parts into place.

The resulting crystal oscillator will look like the photo in Figure 4.

One trick is to take some vector board to aid in bending the leads and cutting them to a standard length. Look at the finished board to figure out the dimensions that I used.

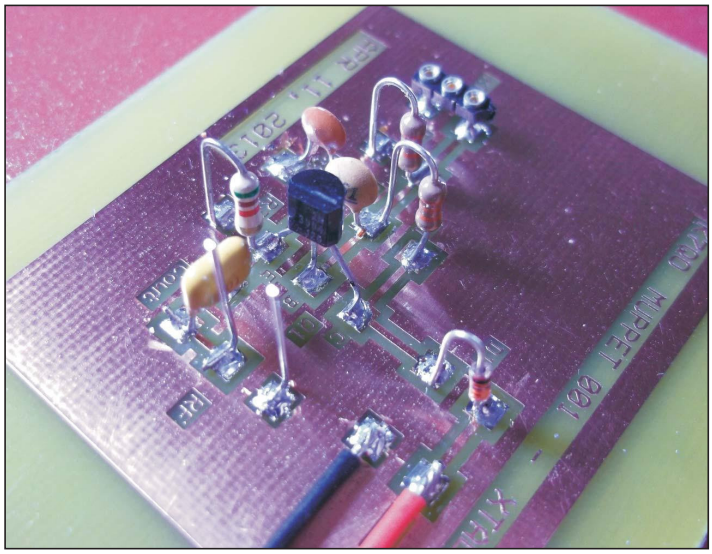

Figure 4—Photo of the finished oscillator.

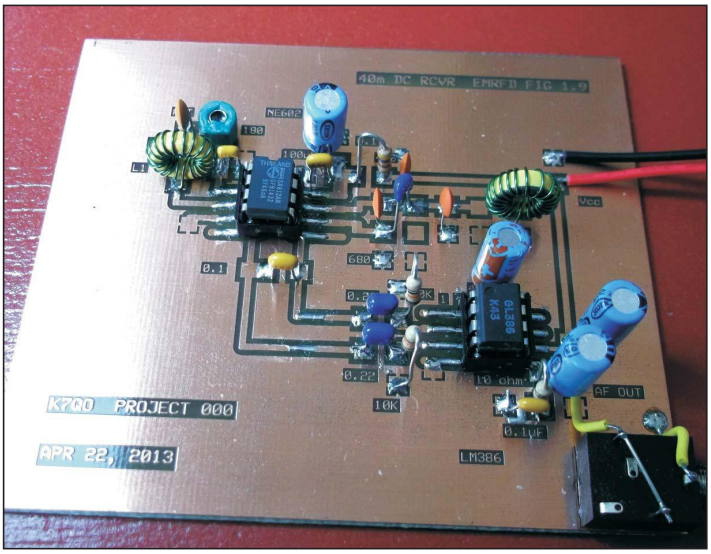

Figure 5—Another example: DC receiver from *EMRFD*.

Wrap Up

One of the most expense items for doing these projects was the Enco shear (Model #130-5700). It was obtained when on sale for less than $90 US. But, not only is it used for the sizing of PCB material for the circuit boards, but doubles as a cutter for PCB material to make enclosures. Saves a great deal of money and well pays for itself the more you get into building and experimenting with RF.

I have recently been going back and doing some simple projects for a training exercise for an online Yahoo group. First, starting with some home brew test equipment for those that just do not have the budget to purchase an entire HP laboratory, new or surplus.

I would recommend using *Experimental Methods in RF Design* (*EMRFD*) by Wes Hayward, W7ZOI, et al from the ARRL store and try out some of the designs and experiments in the excellent book. I do not get anything for this recommendation, by the way.

Dig into your library and look for interesting projects on the Web, too.

Of course, implied in all this is the well stocked parts supply or go to the usual mail order places for parts that you need.

Enjoy the MUPPET show and we want to see your results in the pages of *QQ*.

I've included a final photograph (Figure 5) of a direct conversion receiver built in short order from the *EMRFD* book.

About the Author

Chuck Adams, K7QO, started out in amateur radio as a teenager in the mid 1950's during the great sunspot cycle of all time. The bands were open 24 hrs. Those were the days of taking old TV sets and radios and scrounging for the needed parts. We were so poor the cockroaches went next door to eat, so salvage was the only game in the small town of Wink TX.

Then came the college days and hard work during the summers to finance school. It was only during a stint at TX A&M University for a PhD in physics that he renewed his interest in ham radio at W5AC.

The past couple of decades have been dedicated to QRP work and education. A member of the QRP ARCI Hall of Fame and author of the K7QO Code Course. Now spending a lot of time in the lab with trying to use up all the parts in stock before they pull the big switch in the shack.