Chapter 5

The MUPPET Show

After years of building, experimenting and writing about Manhattan and PCB construction techniques, I have just come up with the term MUPPET board. MUPPET standing for the words Manhattan, Ugly and Precise Placement Experimental Technique. You can think of the following as allowing you to use all of the building techniques together as a hybrid and could even add SMT (surface mount technology) into the mix.

I use printed circuit board material as the basis for the building of a circuit. But unlike making a regular printed circuit board that has to be drilled with a large number of holes to place components, the MUPPET board allows you to place and solder components upon the top of the board without having to drill large numbers of holes. So in that respect it is like Manhattan construction, but without the tedious task of placing pads using super glue to hold them down. It is like ugly construction in that components are soldered to the copper layer, but if I think ahead enough I can save a lot of wires from one part of the board to another by doing traces, i.e. lines of copper material that form a flat wire from point A to point B.

But what is handy about the MUPPET board is that you form the pad area from the copper layer and create traces for signal and power paths to other parts of the circuit. You can leave a number of areas of the PCB plain (unetched) for room to construct modifications or additions at a later time. Think of this additional area as an area to ‘tinker’ in.

You can choose to be creative with either Manhattan or Ugly construction in these areas to improve or modify a project. Then you can do a new board layout using all that you have learned for a final polished project, if you so desire. Or not, if you are the only person on the planet that will ever see the insides. And you don't even have to put a lot of projects in enclosures, especially for small test equipment or a build that tests a section of a larger circuit.

The way a MUPPET board works is that I can precisely place components and solder them to pads or areas using the leads of each component. I do not have to place Manhattan pads using super glue and wait for the glue to cure. If you work it out, a great deal of time is invested in doing pads for the Manhattan style of construction. And even for the drilling of a large number of holes for a regular PCB layout with the copper foil below the board and the components on the top side of the PCB.

You have to plan ahead on any layout, no matter what technique. You will paint yourself into a corner if you don't. I use ExpressPCB software to do the board layout. I do not do schematic capture, so this is just using experience to guide me in my layouts. Comes from doing so many Manhattan Projects over the years.

Let me demo the technique for you. The first muppet project$^{(tm)}$ will be a simple board. This is a just a test board for you to do and check out the procedure. You have to walk before you can run.

Here is a list of the items that I use. You may substitute where you see fit. This is not a do it just my way technique. You may think of something that I have not. Let me list them and then give details where needed.

- PCB material
- Steel Wool #0000 or 3M Heavy Duty Stripping Pads
- Software for board layout and a computer to run it on
- A Laser Printer
- Hammermill Laser Gloss Paper
- Laminator or Common Clothes Iron
- Small plastic clamp for hot PCB material
- Hydrogen Peroxide (2%)

- Muriatic Acid (14.5% HCl)
- Pyrex dish
- Plastic 1/4 cup for measuring the etching fluids
- Clear enamel spray paint
- and all the components that make up the project.

5.1 Printed Circuit Board Material

I use printed circuit board material that I get on the Internet from Ebay. The vendors name is abcfab. Here is a link to his online store on ebay.

abcfab Store

I like to go for the board by the pound sales, but if you are just starting out, then go for the FR-4 material, flame–retardant, and I like the 0.060" thickness and single sided, if you can find it in stock and like the pricing. Also, if you are going to do PCB enclosures for projects, at the same time get a supply of larger double sided (DS) PCB material. This makes great enclosures, as I will show you in another chapter devoted entirely to making enclosures.

If you don't like the above supplier (he does have a 100 per cent rating), then find a supplier for the PCB material and get what you think you can use. If you already have a supply, then all the more better.

5.2 Steel Wool or 3M Heavy Duty Stripping Pads

These two items I find in the Home Department of Wal*Mart. Both are about the same price. I think I am going to prefer the 3M stripping pads for a reason that I will demonstrate in a photo to follow in this section and a discussion in the enclosure chapter.

The steel wool also does a nice job.

Figure 5.1: 3M Heavy Duty Stripping Pads.

Figure 5.2: Fine Steel Wool.

What we need either or both of the items for is for cleaning the PCB before we do anything with it. The board material has oxidized as copper is a metal that easily tranishes and oxidizes. Also, some board material may have been handled with bare hands and finger prints and oils will tarnish the surface badly.

Just take the board you are going to work with and lightly rub it until you get a nice clean surface. I then wash with tap water and rapidly dry it before it reacts with the crud in the water. Water out of the warm tap will work better than the cold. Not because of temperature but because the warm tap water is purer than the cold water. Trust me on this.

Here are photos of three boards, the same size and from the same batch received from an order with abcfab.

Figure 5.3: Unmodified Board.

Figure 5.4: Board Cleaned with 3M Pad.

Figure 5.5: Board Cleaned with Steel Wool.

Figure 5.6: Cleaned Boards Side by Side.

As you can see above, the 0000 steel wool does a smooth job. I am a fan of the 3M pads and I think that a famous online builder, AA7EE, used a course material for removing tarnish from his boards before coating with clear enamel spray paint.

AA7EE's Handiwork on PCB Material

Another thing about the rougher surface. In the toner transfer method we are going to melt the toner and 'fuse' it to the PCB surface to protect it against the etching fluid. Which surface do you think the toner will stick to more tightly than the other? My guess is the plowed corn field material. I will sacrifice both boards to scientifically determine which is better. Bill Nye is very jealous of this experiment. He did not do it first.

Two things about this cleaning procedure. The first being that you get a clean substrate that will be easier to etch and also easier to use the toner transfer technique, see the second section after this one, to mask off the areas we do not want to remove from the FR-4 material.

I do my cleaning in the garage on the old trusty and sometimes dusty workbench. I use a sheet of newspaper to do the cleaning on as it creates fine copper dust that I do not want in any electronic gear and if you use steel wool then it leaves steel fibers all over the place. You can fold up the newspaper afterwards and put it in the trash and you have a clean area. Neatness counts in this business or you are going to pay for not getting organized by destroying a project.

5.3 Copper Foil Layout

This first demo will show how we lay out a the foil pattern to be used in a construction project. Let's do a simple board. This board will have a number of straightline traces of varying width to show just how fine (narrow) traces can be made on a PCB.

Also a couple of pads to put a 1/4W resistor on and demonstrate how I make the leads match.

Use what ever program you want to layout a simple pattern shown in the next image. I use ExpressPCB as it had a simple learning curve and has no physical limits on the layout and it is free. Here is the URL to find the program.

ExpressPCB Download URL

I run the program under wine using debian linux 7.0, which was just released in early May 2013.

Make, using your program of choice, the following pattern. I did mine on the top layer of the board. I also made a ground plane filled area with 0.030" spacing between the plane and the traces and pads. Use 0.10" square pads at the end of the traces. I also have text showing the width of the line for the single trace between two pads. Make this to fit a board size you have. Mine is 4"x4" for this test.

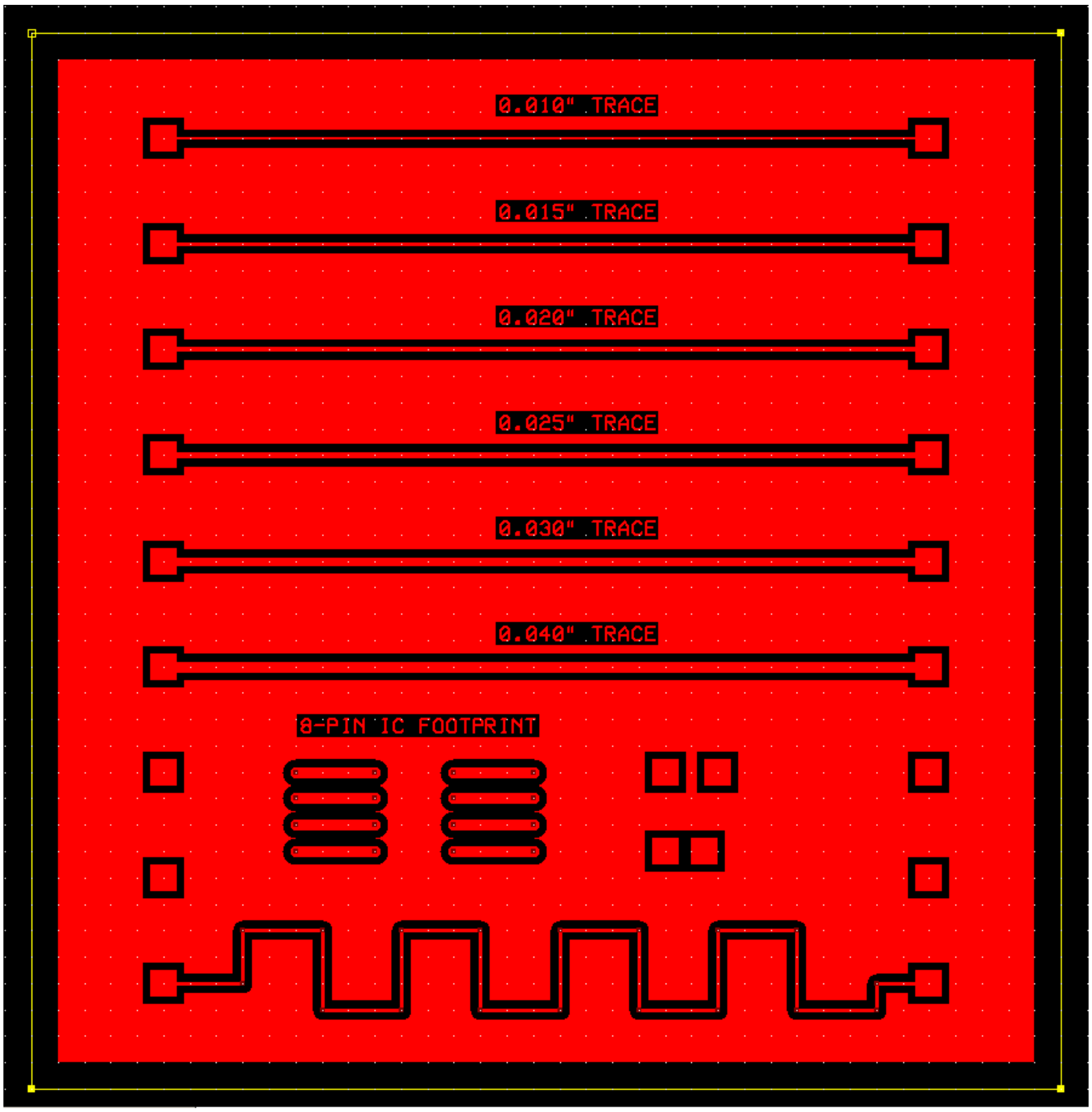

Figure 5.7: ExpressPCB Test Board Layout.

5.4 Printing PCB Layout

Now that you have a board laid out in software, it is time to convert it to a hardcopy. Now you have to put on your thinking cap. If you print out the top layer to a laser printer, you will look at it and you can read the text and 99.99 percent of todays printers will do a 1:1 ratio on the print and it will be the right size.

But you do not want to print the layout just now. You have to have one of two things. A printer with an option to mirror or reflect the image about the vertical, so that it is reversed.

Or, if you are running something like ExpressPCB using wine under Linux, then all you have to do is setup a PDF printer. Google on how to do that.

```
sudo apt-get install cups-pdf
```

for debian based linux systems.

Then when you do a print from the menu option in ExpressPCB, select to print the top layer only and send it to the PDF printer. It will generate a file named PDF.pdf or ExpressPCB.pdf. You have to have LaTeX installed under Linux with the latex-options package that contains a program called pdfflip that will reverse the image and create a file named PDF-flipped.pdf or ExpressPCB-flipped.pdf. Now send the flipped file to the real physical printer and you will get the image that you need to do the toner transfer.

I use Hammermill Color Laser Gloss paper with the printer. This paper is heavy weight and it is silky shiny smooth. The laser printer heats the toner and ‘fuses’ it to the paper surface. We will be reheating the toner, while it is contact with the copper layer of the PCB material to get a Oreo type sandwich made up of paper–toner–copper. A photo of the paper is shown a few pages further into this document.

Here is the PDF output from ExpressPCB flipped. This is what we will use to laminate to the PCB material. The file that I point to in the next section will print so that the board is four inches square.

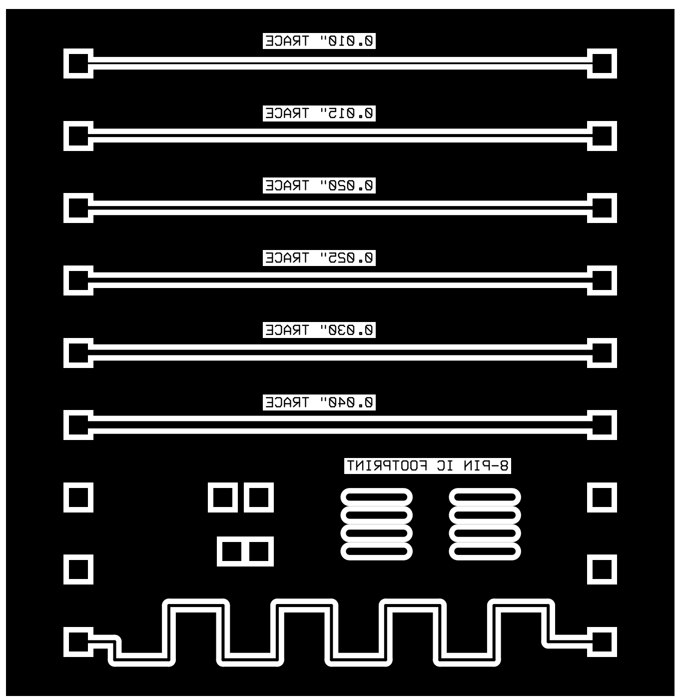

Figure 5.8: Test Board Layout for Printing. Not to scale.

5.5 Laminator to Transfer Toner to PCB

Now you have a piece of paper with the layout on it. You have two ways to transfer the toner to the PCB copper layer. Iron it on or use a laminator. If you are so strapped for cash that you can not or will not purchase a laminator to tranfer the toner to the PCB then you are on your own. Google for iron on toner transfer and get the instructions. I've done it and I would not recommend it now. I get 100 per cent success with the laminator. Much cleaner and faster. IMHO.

I use a GBC BadgeMates laminator that I bought on sale at one of the big box office suppliers. It has been a while and I think it was Staples.

Here is a picture. If you have the capability, here is a video that I did to show it in use.

K7QO Use of Laminator Video Snippet

The laminator quit making the funny little noise after the video. I'm really not worried about it and I think it is going to outlast me. I will let you know when it fails, if ever.

Following the steps in the video, I have now transfered the test pattern to both the boards, soaked in water and removed the paper. You do not have to be retentive about the removal of the paper. Just make sure the bare areas of the board are clean. There is some chemical in the paper that dissolves in the water and I just like to make sure that it does not interfere with the areas to be etched.

5.6 Your First Board

I'd like for you to try the following board. It requires a 4"x4" piece of single sided material. It took me less than 10 minutes to etch it with a board that has 1 oz copper density on it. I'll put the PDF file at:

PDF of test board.

Download it and print it on Hammermill Color Laser Gloss paper and iron or laminate it to a clean board. Here is a photo of the paper that I use. People search far and wide and I have done the same and I think this is the best paper for the job at hand. It has a smooth surface and it removes well with soaking in water for about 10–15 minutes.

Figure 5.9: Hammermill Color Laser Gloss Paper.

Then you CAREFULLY mix two parts hydrogen peroxide (I use two 1/4 cups into a pyrex 750ml rectangular dish from Wal*Mart) and THEN put in a single 1/4 cup of muriatic acid. Be very careful with this chemical. Remember your chemistry class where they taught you to put the other chemicals in first and the acid goes LAST every time. Here is a photo of the pyrex dish, measuring cup and spoon that I use.

Figure 5.10: Pyrex Dish, Measuring Cup and Spoon.

GENTLY stir with a plastic spoon for a few seconds to mix things up. Do this in the garage or outdoors or in a well ventillated place with the air moving away from you. Do not breath the fumes at all. Keep away from your body and your eyes, ears and mouth.

Then put the board into the mixture with the toner side up. You should see the copper turn dark almost immediately. That is the oxygen combining with the copper to oxidize it. Now just gently keep moving the liquid around with the plastic spoon. Do not try anything fancy and make a mess or cause serious injury. Just take your time. You may also note a bunch of bubble coming from the small amount of paper remaining attached to the toner.

It takes me about 6-10 minutes of gently stirring the liquid over the board without touching the toner at all. After the board has etched, then remove the board with rubber gloves on and clean with warm tap water.

Dry the board with a paper towel and set aside for a moment. Come back and put almost a 1/4 cup of baking soda into the mixture to neutralize it. Flush this. Flush twice to really dilute the stuff.

I take the board and then using the fine steel wool and a lot of elbow grease I remove the toner from the copper layer. Try not to damage the copper, but do get it clean

Then wash and dry and then I put a very very thin layer of clear enamel on the board and let it air dry for a few hours. This is the tough part. I know you want to get to melting solder almost immediately.

Here is what your finished board should look like. Take lots of pictures and send them to your relatives to show how smart you are to be able to make electronic boards.

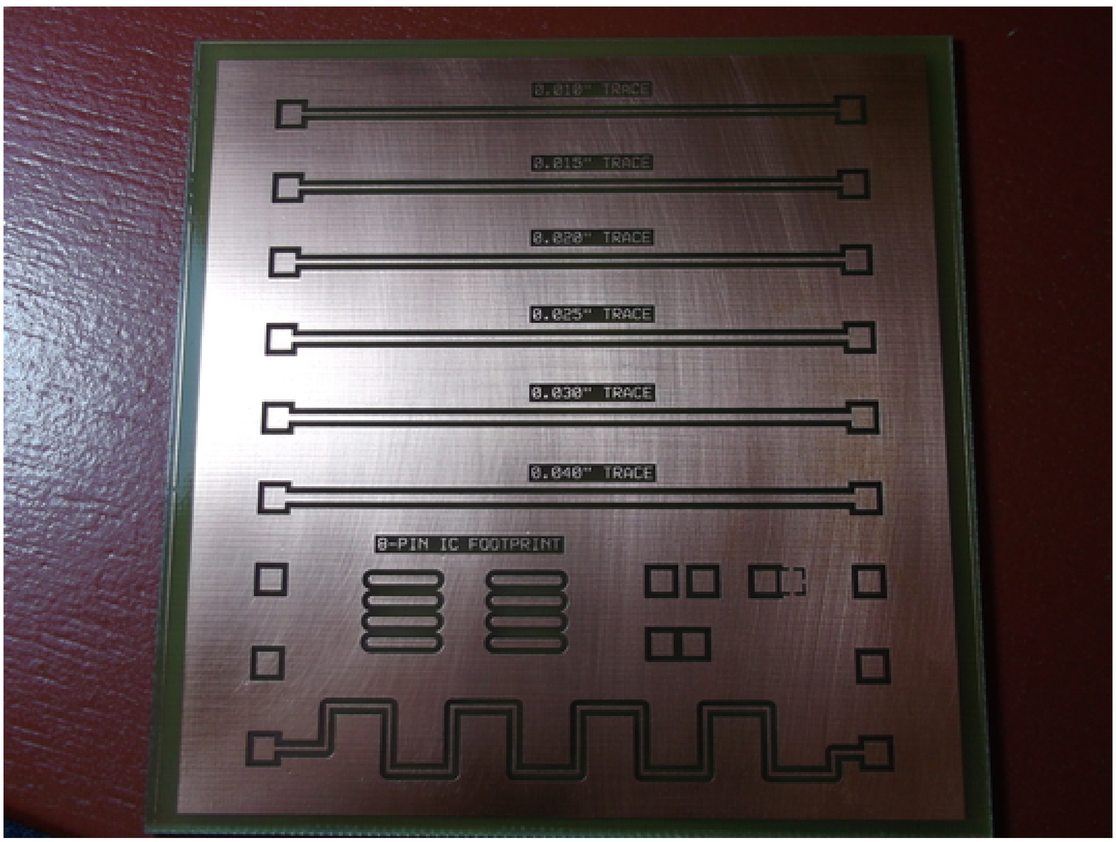

Figure 5.11: K7QO MUPPET 000 Board.

5.7 Test Soldering to PCB

I want you to see how soldering to a muppet board works.

I use a Weller 25W soldering iron, just like the ones you will find in the tools department of Home Depot. Costs about $20. I have an old Ungar tip on mine, but the stock one will work just as well. I have built over 200 kits with this same iron and the tip. Too bad that Ungar does not make the tip any longer. I'd buy a dozen, not that I need a spare.

First. Using the iron and some solder, tin each of the square pads at the end of each of the straight traces. Don't use much. Just a thin layer will do. I use solder that has two per cent silver. It comes out the shiniest for me. I recommend a thin solder for better control.

Here is what I got for my board. Take a DMM and measure the resistance of each trace. Write this down in your lab notebook that you should be keeping. The resistance will depend upon the thickness of the copper layer and the width. You want the math? You should get a small value for each trace. If you get an infinite resistance, then the trace has a break in it. Use a large magnification spy glass to search for the defect if you encounter one. It's not critical here, but it will be the demise of a project if it occurs later. Analyze what went wrong to cause the break, if and when it occurs.

After EVERY board that I etch, and this is why I use single sided board, I put it up to the light and I carefully examine every square mm of the board for defects. It is much easier to find problems now than later. Been there. Done that. Just have fun doing this. It is not a race. You may wind up taking days to find a simple error when the projects get bigger.

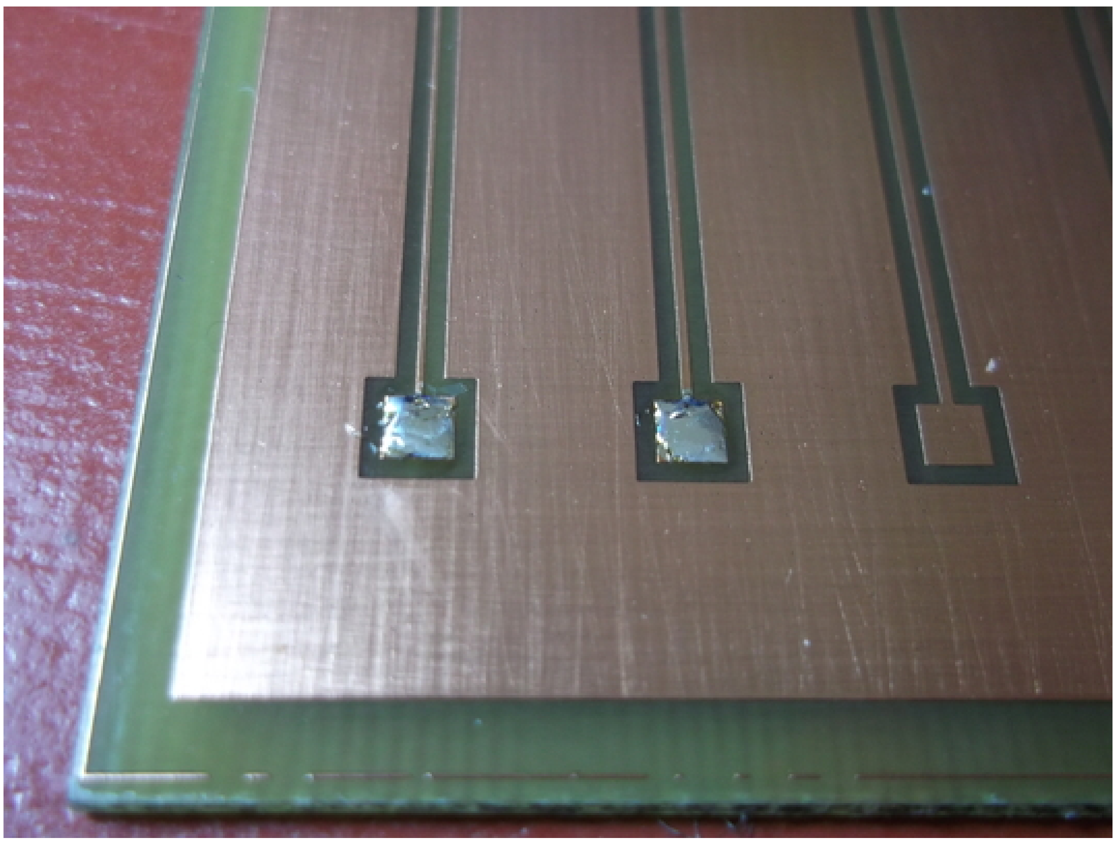

Figure 5.12: K7QO MUPPET 000 Board with soldered pads.

Here is board with just some of the pads at the end of the straight line traces soldered. You pre–tin the pads to solder components to or wires to during construction of a project. Just a itsy bitsy teeny weeny bit of solder is all it takes. Solder is so expensive now, that I know I don't have to convince you to use a little and not a lot. This is not an aircraft carrier we are building.

Now, with your DMM set in the lower range of resistance measurement, measure the resistance of each of the straight line traces at the end pads. If you get an infinite value on the 0.010" trace, then you have to find and correct some step in the process to eliminate the error(s). The resistance in my probe leads was greater than that of any of the traces and I estimate about 0.1 ohms for the traces.

5.8 Resistor Placement on Muppet Board

I'll show you how to mount a resistor and the steps that I use. You can easily determine how to do other parts. There are some parts, like the SBL-1 and relays, that are going to be a royal pain to figure out. It's not all easy, that's what makes it fun. If it was easy, everybody would be doing it.

Get a scrap piece of vector board or make a fixture out of a piece of scrap PCB material that is 0.060" thick. I use the board edge to bend the lead to form a hair pin like structure.

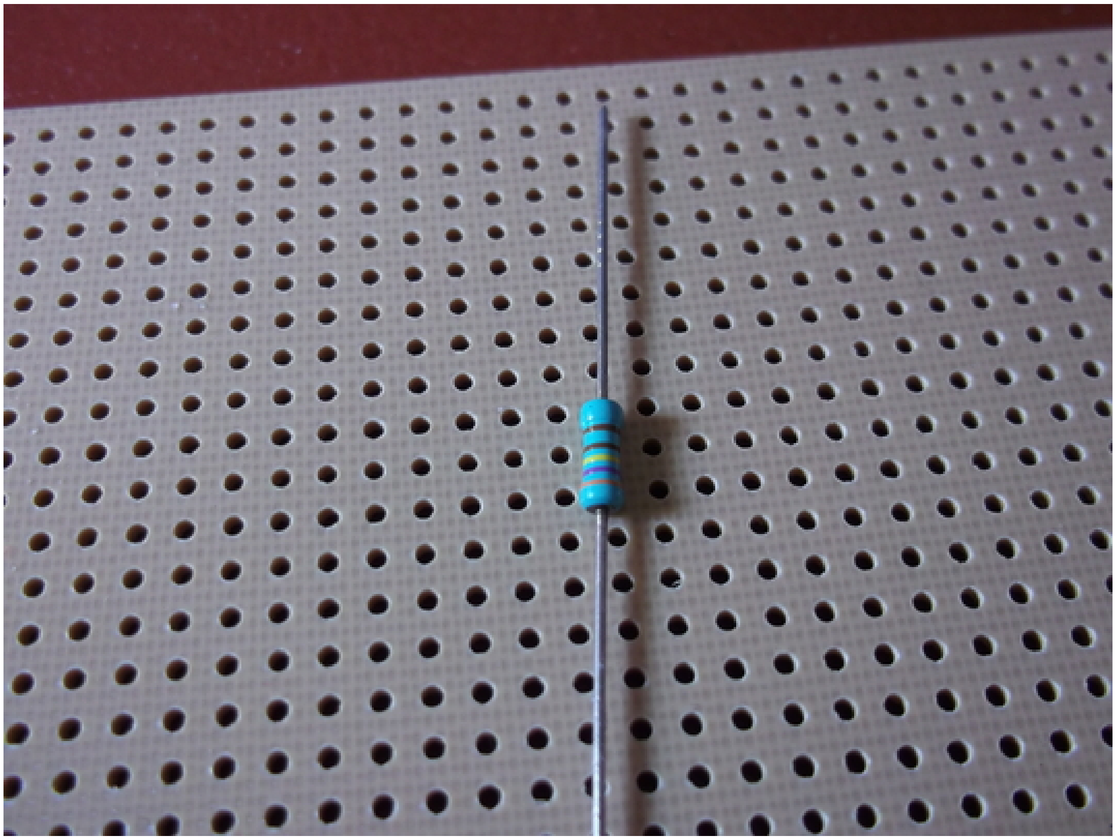

Figure 5.13: Vector Board and Resistor.

Above we see a section of what we call vector board. It has 0.10" spaced holes. And, since I laid out the practice boards with pads spaced at 0.20", we can preform component leads to fit.

I went to the parts draw and found an envelope with 100 4.73K ohm resistors that I got at some place in some galaxy some time ago. I know of no reason to have 4.73K as we usually use 4.7K in most circuits. So let's sacrifice this puppy for the good of mankind. And I'll show you that it's not really a sacrifice, as I have not lost the part and can reuse it if need be.

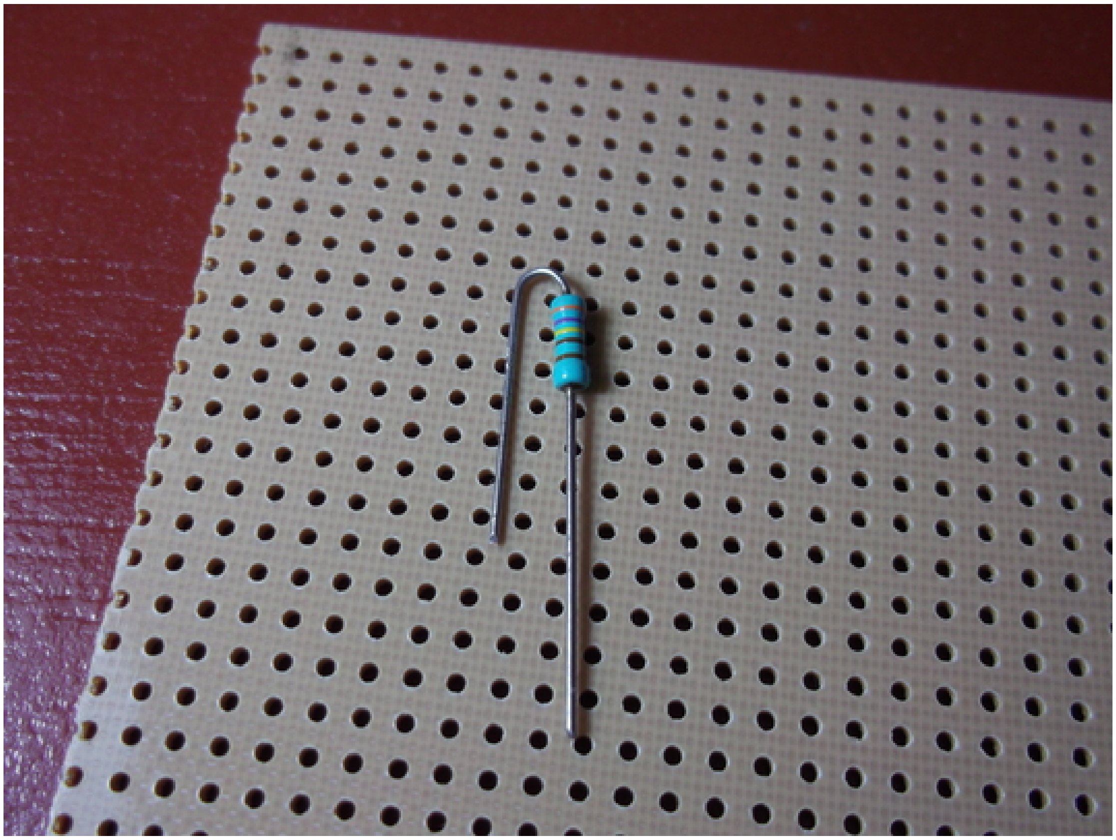

Figure 5.14: Bent Leads on a Resistor.

Here is what the part looks like after we bend its little legs. I just hold the part flat against the top of the board with the top of the resistor aligned the with the top row of holes adjacent to the edge (the top edge in this photo and not the raggedy edge on the left) and then bend over the top. You can get every resistor bent to the same dimensions every time. I'm OCD with ADD. I like perfect but not for very long. I just like the looks and it isn't that much trouble to bend them using the fixture.

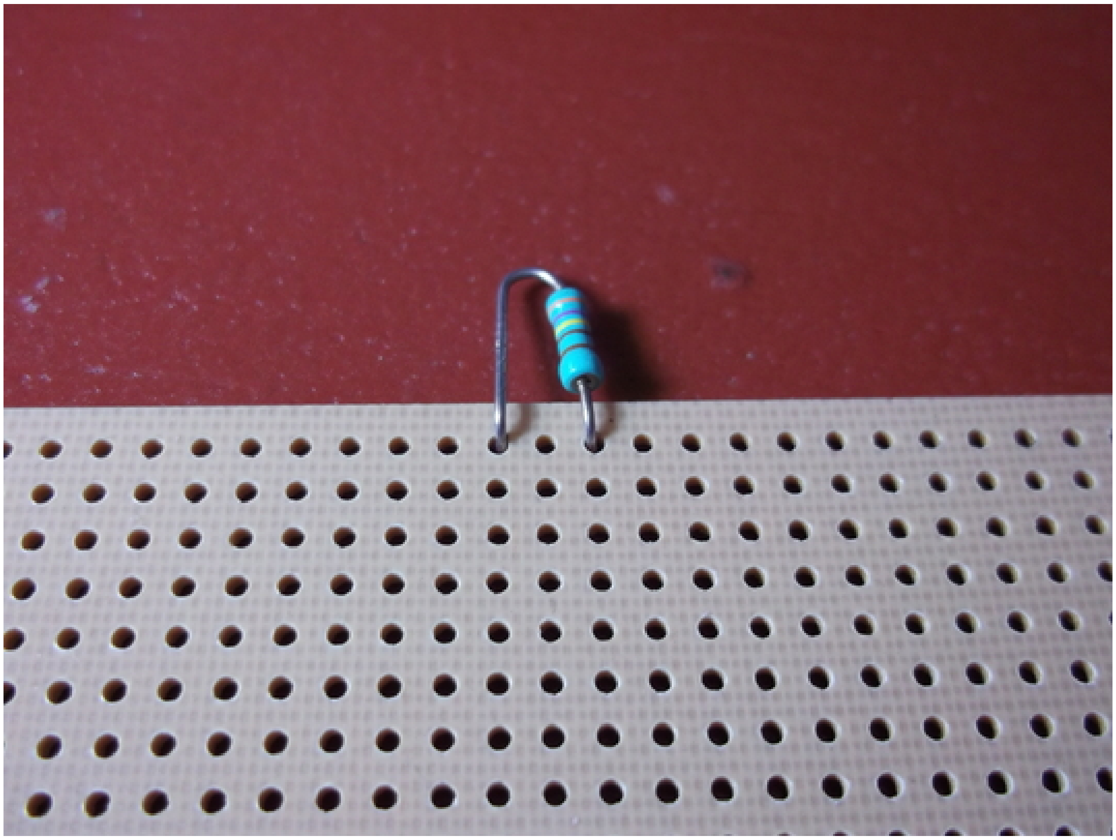

Figure 5.15: Preform resistor leads for PCB mounting.

Now, insert the resistor into the two holes that are the same distance apart as the pads where you want to place it. Here I am using the 0.20" spacing that is the same spacing I set the first two practice pads to the right of the IC pads. WHILE you have the resistor in the holes and bent, use your diagonal cutters to cut the two leads on the other side of the board flush with the board. This forms two 'feet' that will soldered to the pad. Gives you some length to form a good physical structure to hold the part in place and give a low resistance path for signals and currents to pass through.

This may take some practice to get a feel for just what pressures to use, etc.

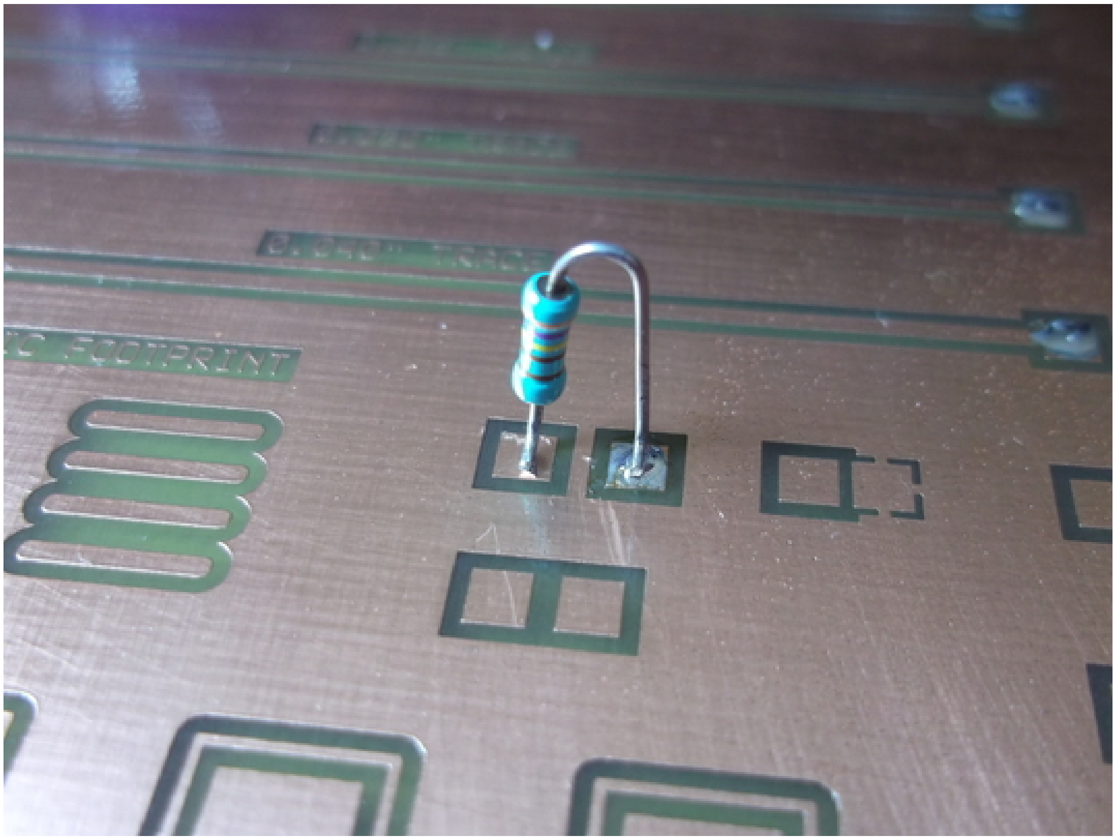

Figure 5.16: Resistor in place with one lead soldered.

Now, tin the two ’feet’ of the resistor (lightly) and ONE and only one of the two pads. Just a smidge of solder will do. You should get something like you see above. I tin only one of the pads so that the second leg will be flush with the pad before I apply solder.

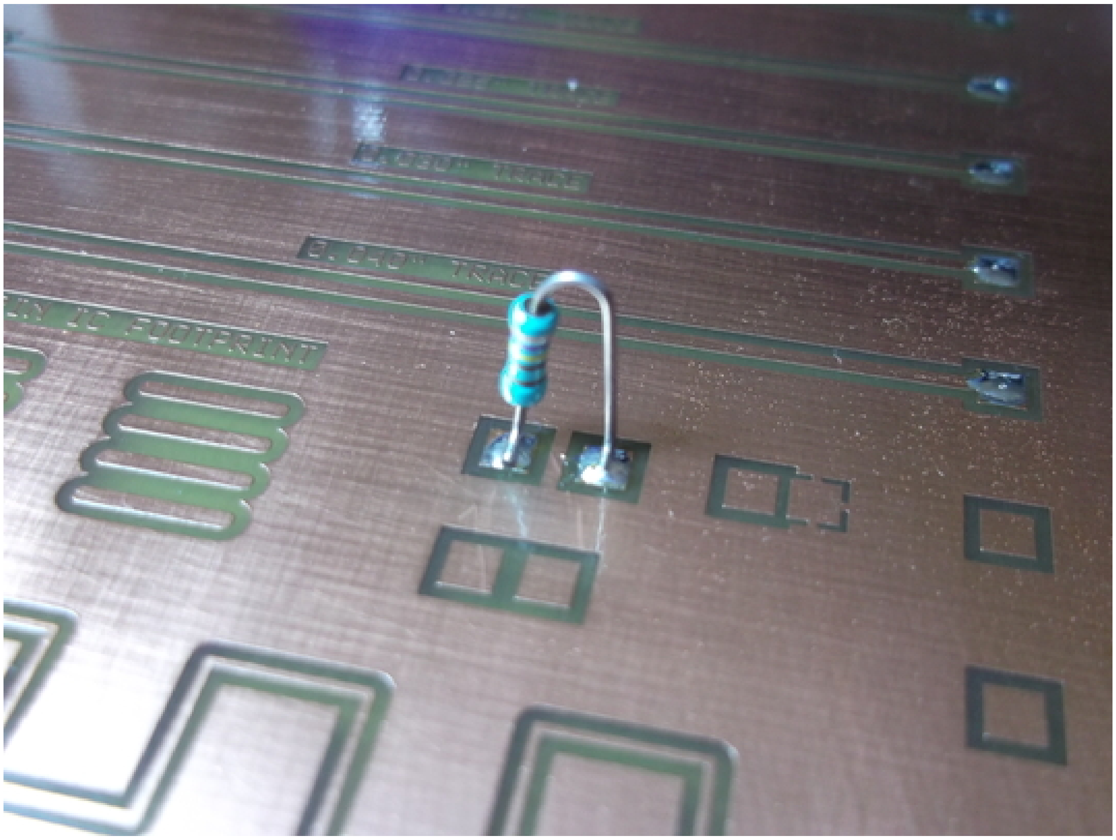

Figure 5.17: Resistor soldered in place.

Above is the final result. Let me also note. You can have the feet of the resistor closer to the inside edges to allow one or more other feet from other parts to also be soldered to the same pad. This you will need to do for more complex circuits. I'll show you two approaches to this in the next chapter on how to do a crystal oscillator.

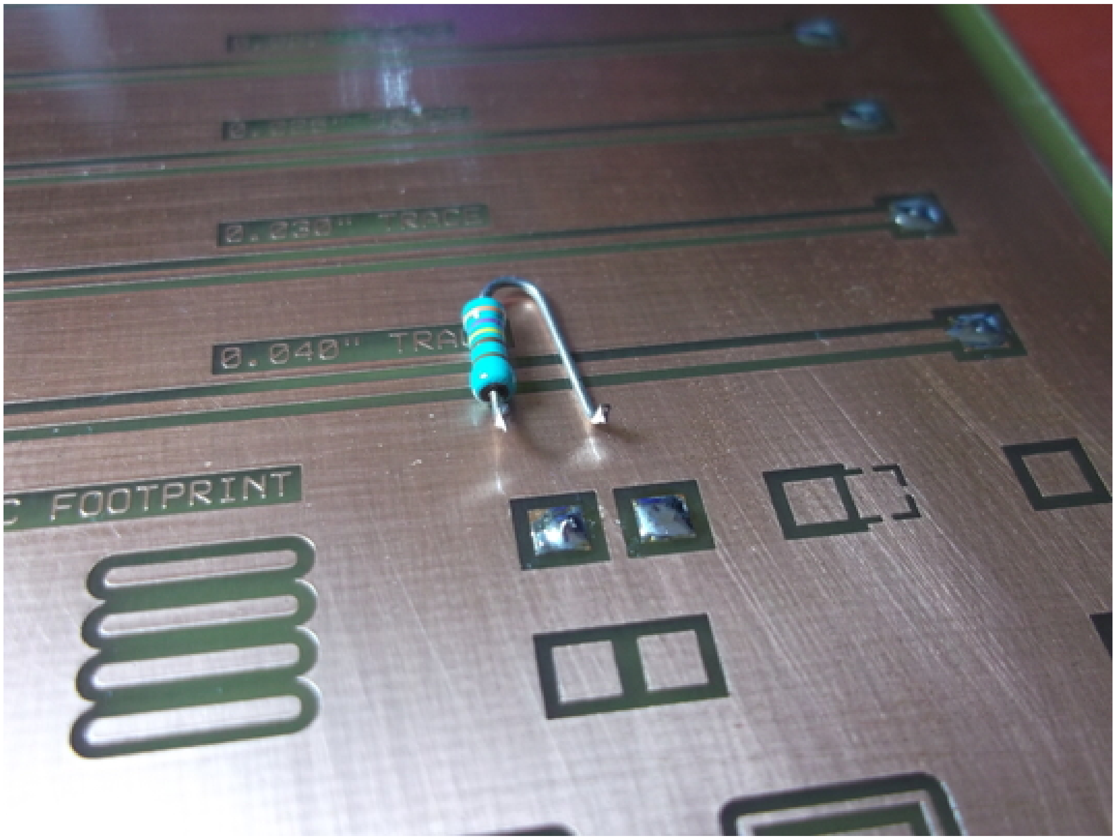

Figure 5.18: Resistor desoldered from its location.

Above you see that I have unsoldered the resistor from the two pads. I have not harmed the resistor nor have I ruined the PCB. You can easily replace a part with one of another value if you think the circuit, after testing, needs some other value. You can keep the old part for later use. just gotta figure out where and how to store them for later.

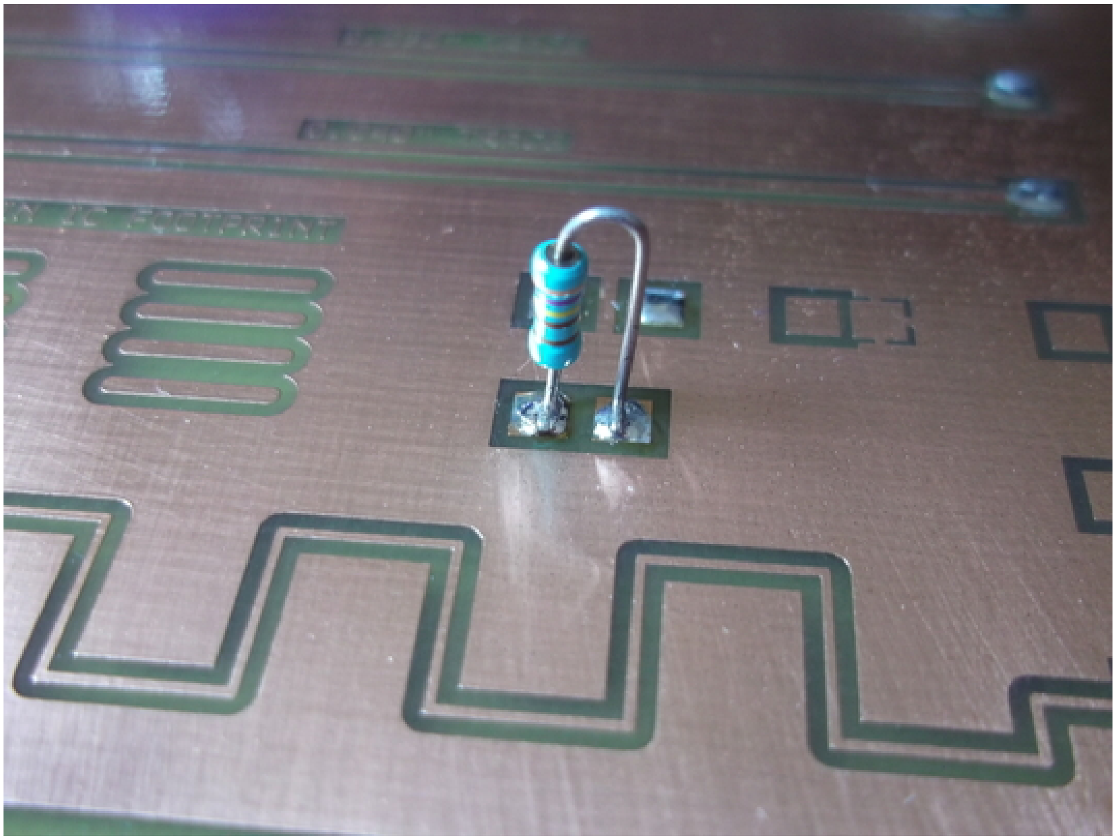

Figure 5.19: Resistor soldered into closer pads.

Using the same resistor. I go back and using the vector board I then form the leads closer together. This then allows me to solder the resistor to the two pads that are closer together. This is practice later when you want to make more densely populated boards for more complex circuits. You do not want the real estate to wind up the size of some Texas counties.

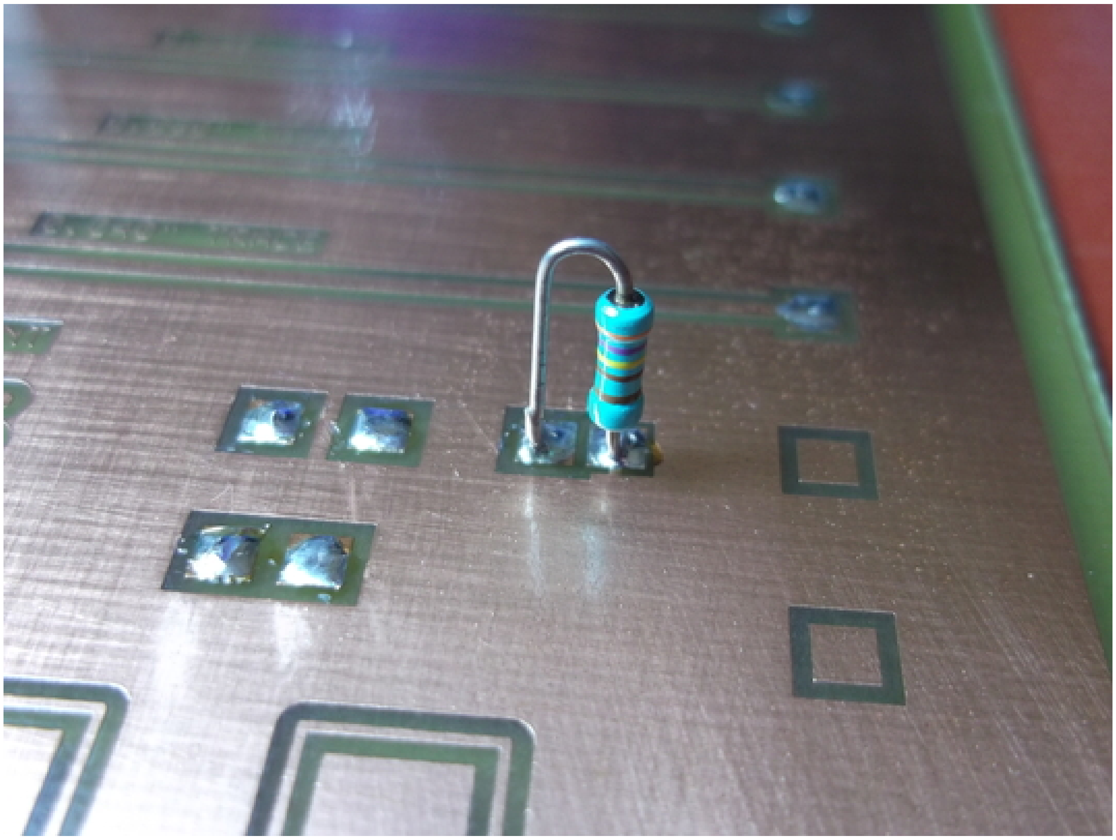

Figure 5.20: Resistor soldered to two pads. One at ground potential.

I tinned the ground pad first, which I do with ‘thermal relief pads’ in ExpressPCB. I do thermal relief pads for two reasons. One, to remind me that the pad is there and to double check for matched sets of pads before etching and during the board layout process. Second, to allow me to pre tin the pad and not have to heat up the entire board and take a lot of time in soldering. This, if you experiment with it, you will find to be a great help. IMHO.

And here is the MOST IMPORTANT thing that I can teach you here. Put the short lead of any component to the ground pad. That pad is at ground potential and you would have no reason to ever measure a voltage at that pad. By putting the lead that is not at ground potential on top, you can easily get a DMM or RF probe tip to it to measure a voltage or probe with a scope for a signal during a power on condition. Or to check for a short to ground WITH THE PROJECT POWERED OFF.

Also, plan ahead to do make things easier to measure in board layouts to make debugging and test measurements easier.

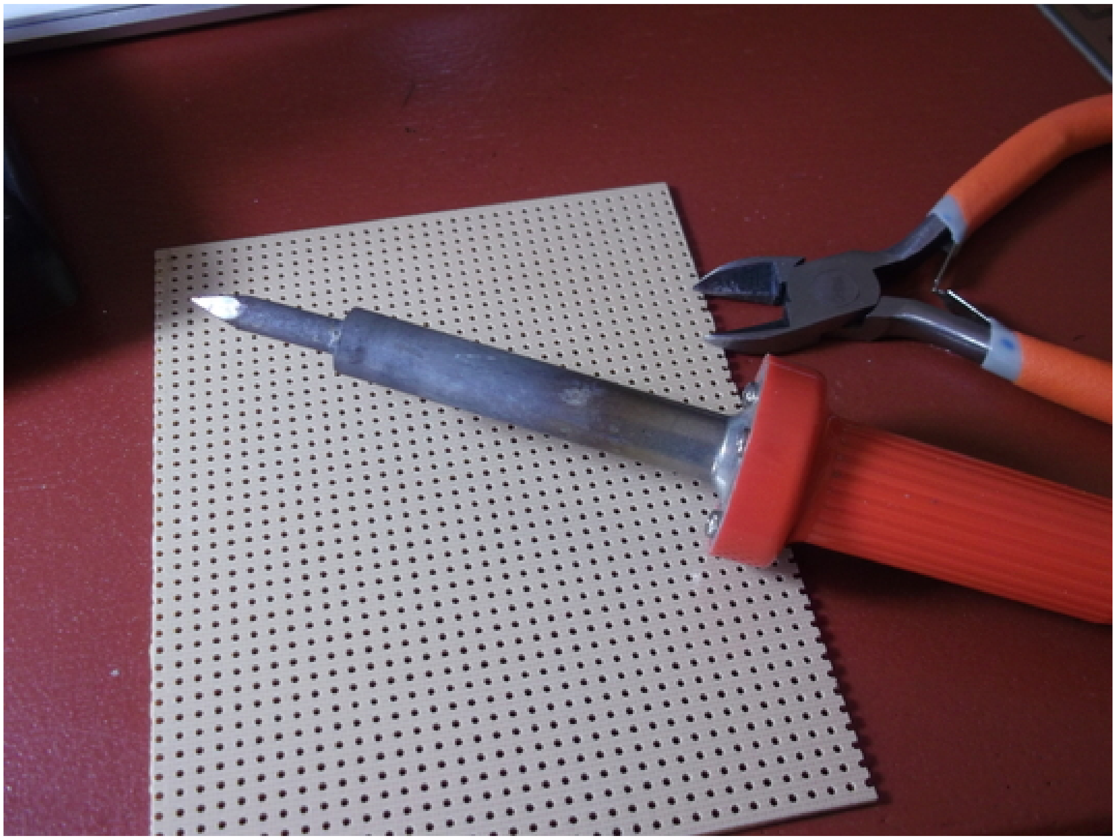

Figure 5.21: Weller iron with Ungar tip.

Here is an old old Weller SP-23 with the Ungar tip. Been using this thing forever. Money saved on a soldering station more than paid for one or more kits.

Figure 5.22: Solder iron stand.

I found this soldering iron stand (originally from Radio Shack) at a swap meet for the measely price of one buck. I like it as it doesn't cool down the iron and makes it handy to pick up.

Figure 5.23: Wash cloth.

And this will piss off a lot of guys as a bad idea. I keep my iron clean by just a swipe across this cloth rag. It does two things for me. It does NOT cool down the iron as with damp sponge critters and it does a good job of getting crud off the tip. Your mileage may vary. Cheap solution to an old problem.

Something I forgot to mention, but now is a good time. Did you wonder, when you saw my vector board fixture for bending leads, why did Chuck leave one side of the board cut at an angle and ragged? Here is why. Two pictures worth 2,000 words.

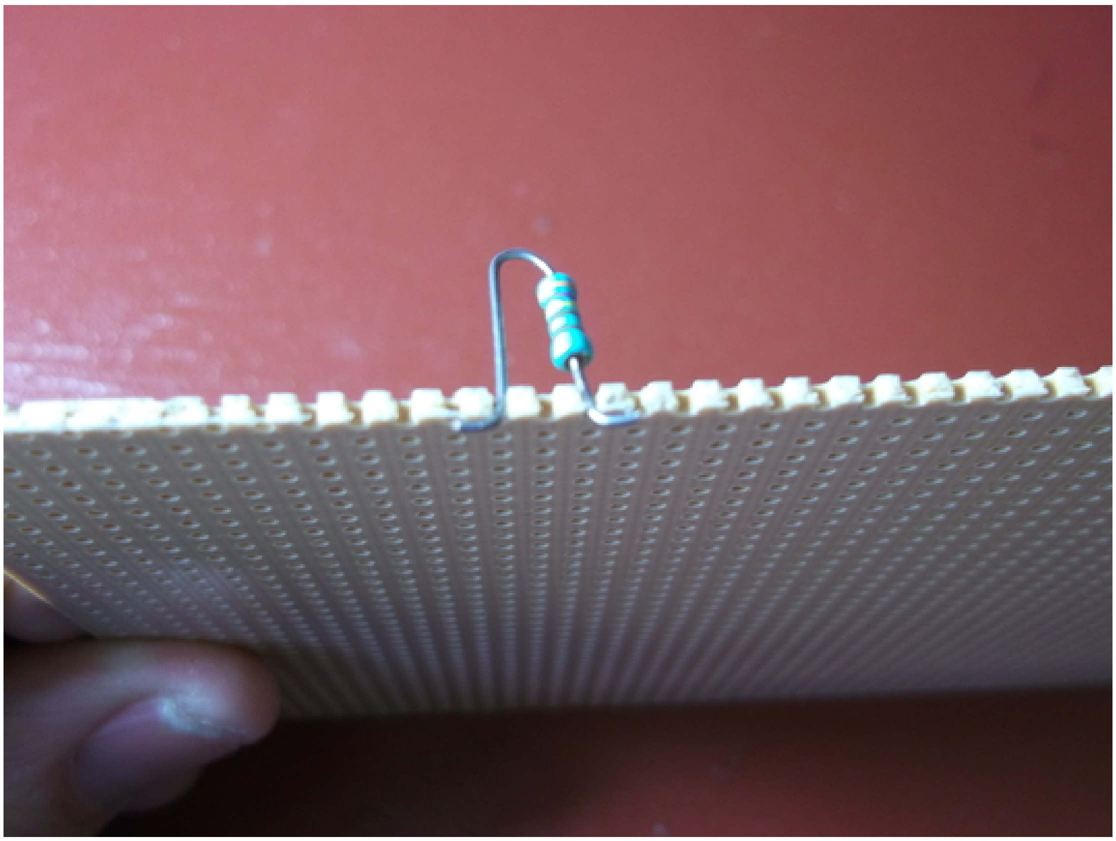

Figure 5.24: Edge bending of resistor leads.

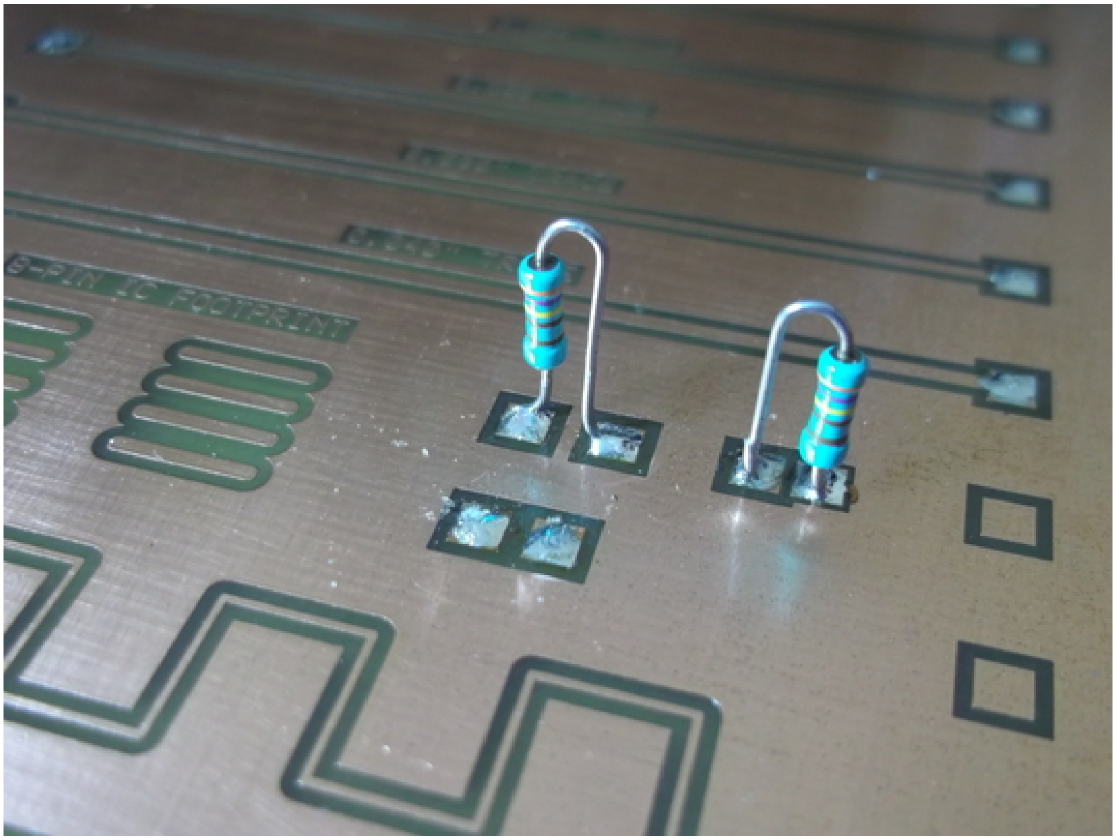

Figure 5.25: Resistor soldered into place with leads outwards.

OK. Enough of the newbie lessons for today. Let's now get on to some real building and experimenting that will result in some useful stuff for the workbench and shack (if separate).

5.9 Soldering IC Sockets and IC components

For IC components, like NE602A mixers and LM386 audio amplifiers, I like to use sockets. It is just my personal favorite just in case I want to abandon the project or need a part for some other device from time to time. I have not had one fail on me, even after years of use in a transceiver. And, I have not had troubles with connections between the socket and the IC. Some people complain from time to time on the Web and I wonder what they did to get the problems.

On the practice muppet board I purposely made an 8–pin set of traces to solder a socket to and demostrate the technique that I use.

I first use solder and the soldering iron to pre–tin the socket pins that I have bent out from the bottom of the plastic that makes up the socket. Here is a photo of what it looks like. Note that I have pre–tinned pin number one on the board. I always do pin number one first to help remind me which way the socket is to be orientated when I go to start soldering it to the board. I have an OCD and an ADD problem. I like to be perfect, but not for long.

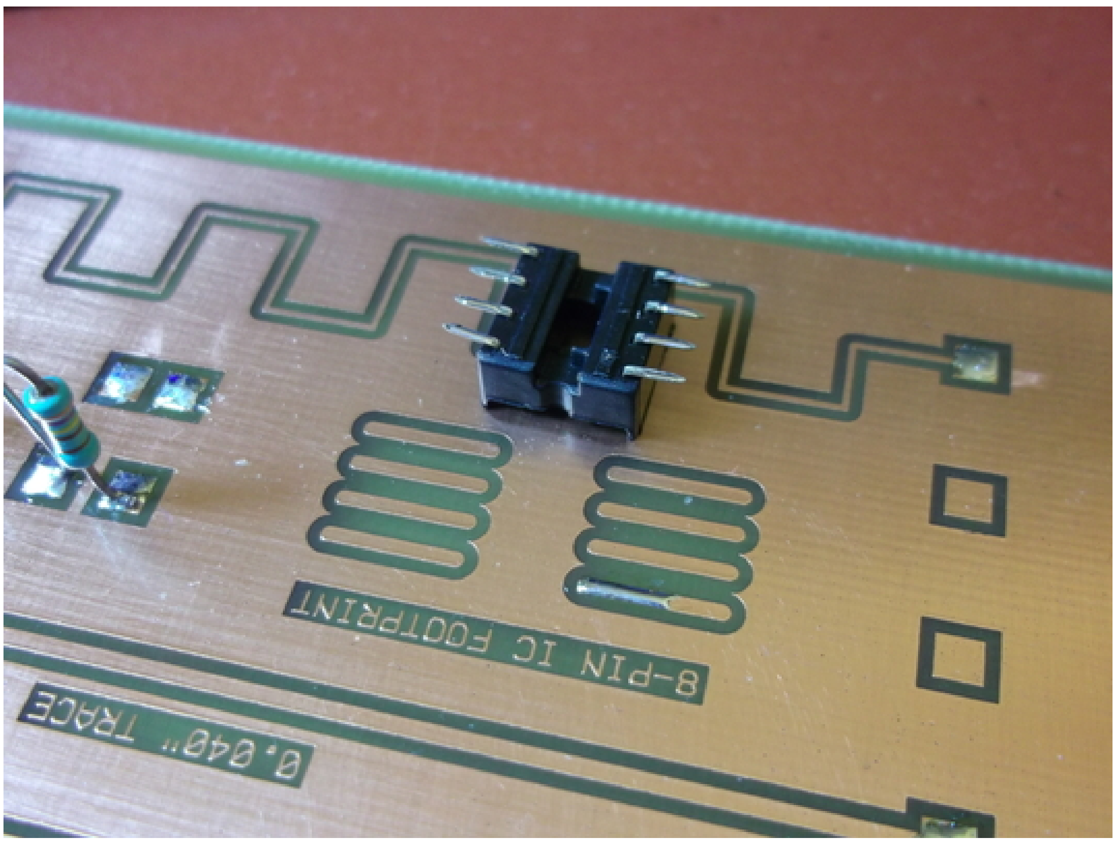

Figure 5.26: Pre–tinned leads outwards from the socket.

Then I carefully center the socket over the traces and heat the leg and the pad to place the socket. I double check to insure that all the leads are centered on all the traces.

I then solder the opposite leg to its corresponding trace and then double check that all the remaining legs of the socket are centered on their traces. Double check to make sure the socket is oriented with the half–moon adjacent to the number one pin on the PCB correctly to remind you the direction and orientation of the IC chip when you insert it into the socket during later assembly. They do not like to be powered up backwards. It could destroy a part in not time at all.

Here is photo of the two pins soldered and you'll note that in this example that the solder joint on pin number one isn't quite right, but it did hold the pin down. Which is all that is important at this stage of the game.

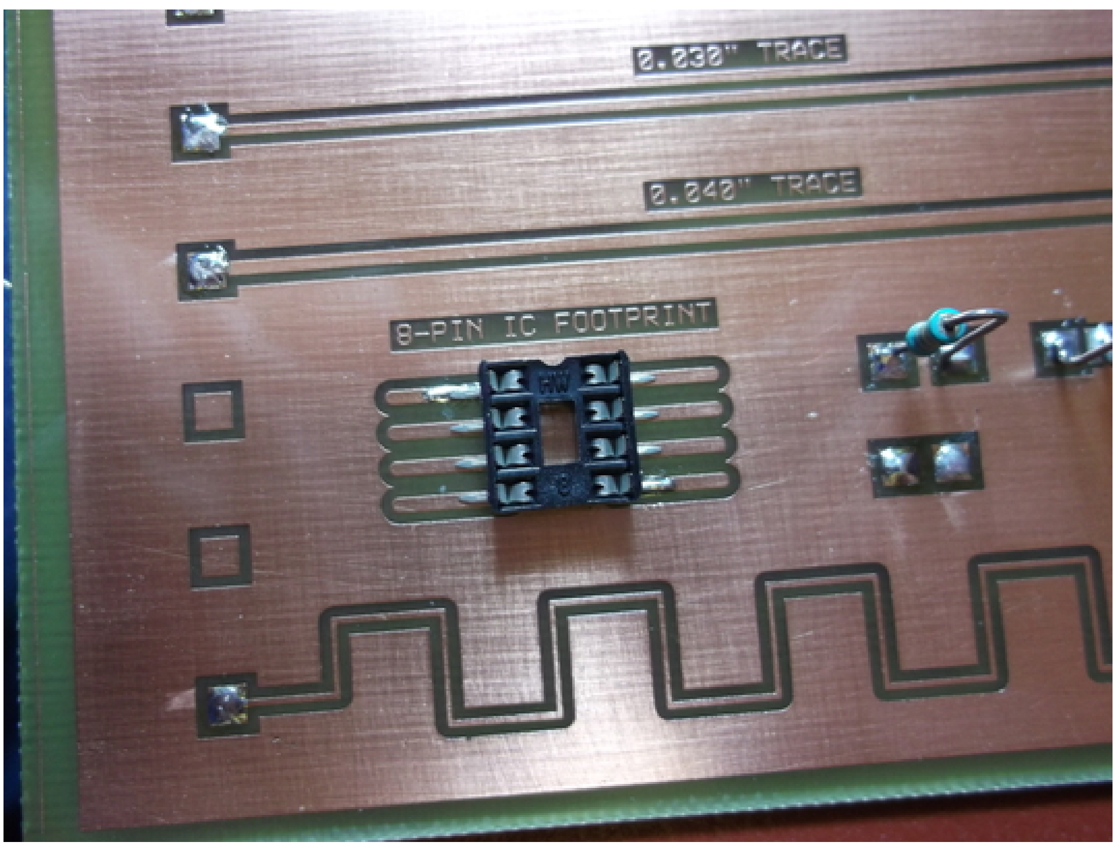

Figure 5.27: Socket with two pins soldered into place.

Now, go ahead and solder the remaining pins and double check and touch up any connections that don't look just right. Use the solder in small amounts here. After assembly there should never be any extreme forces ever applied after the IC is inserted. This device is not going to the Moon or to Mars. I don't think.

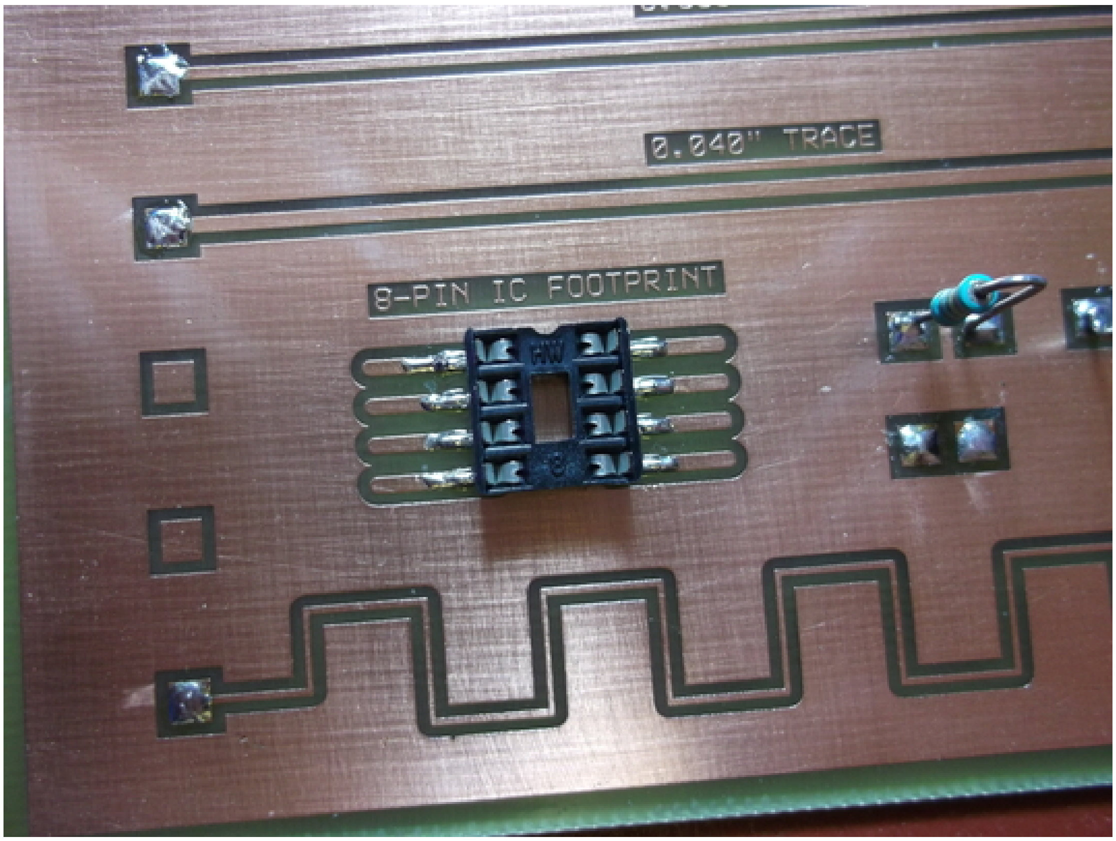

Figure 5.28: Socket with all pins soldered into place.

That's all there is to do to do a nice job. BTW, I bought 500 of the 8–pin sockets from a China source online for five American bucks, including free shipping, so there are very very cheap. Work well and will allow me to replace parts easily.

I'm thinking that later, I will build a test fixture for some ICs that use these sockets and allow me to exchange parts easily and quickly. Time will tell.